

Automation Engine 14

Complete Manual

Contents

1. User Guide	16
1.1 About Automation Engine.....	16
1.1.1 Copyright Notice.....	16
1.1.2 Getting Help.....	18
1.2 What is Automation Engine.....	18
1.2.1 The Automation Engine Pilot.....	19
1.2.2 Data Management in Automation Engine.....	23
1.2.3 File Processing in Automation Engine.....	25
1.3 Getting Started with Automation Engine.....	27
1.3.1 Starting Automation Engine.....	27
1.3.2 Performing Basic Configuration.....	32
1.3.3 Uploading Files using the Pilot.....	35
1.3.4 Launching a Task on a File.....	36
1.3.5 Checking the Status of the Task.....	37
1.3.6 Cancelling a Launched Task.....	38
1.3.7 Relaunching the Task.....	38
1.3.8 Generating Output.....	38
1.4 Server Administration.....	38
1.4.1 Status Overview of the server.....	39
1.4.2 Setting up the server.....	41
1.4.3 Backing up the server.....	41
1.4.4 Restoring the server.....	42
1.4.5 Managing databases.....	43
1.4.6 Managing SQL servers.....	44
1.5 Uploading Files to Automation Engine.....	45
1.5.1 Uploading Files using the Pilot.....	45
1.5.2 Uploading Files from a FTP Server.....	45
1.5.3 Using Access Points to Upload Files.....	47
1.5.4 Using Shuttle to Upload and Process Files.....	47
1.6 Launching Tasks on Files.....	67
1.6.1 Creating a Task Ticket.....	67
1.6.2 Launching a Task on a File.....	76
1.6.3 Checking the Status of the Task.....	78
1.6.4 Relaunching the Task.....	79
1.7 Launching Workflows on Files.....	79
1.8 Viewing Files in the Automation Engine Viewer.....	79
1.8.1 Introduction.....	79
1.8.2 Comparing Files.....	81
1.9 Integration with Other Esko Applications.....	83

1.9.1 Review and Approval of Page Impositions using WebCenter.....	83
1.9.2 Integration with Nexus.....	103
1.9.3 Integration with Odystar.....	106
1.9.4 Integration with Enfocus PitStop.....	111
1.9.5 Integration with Third Party Applications using Hot Folders.....	111
1.10 Integrating with Third Party Imposition Systems.....	112
1.10.1 Workflows: The bigger picture.....	112
1.10.2 Importing and Creating Impositions.....	113
1.10.3 Exporting Impositions.....	114
1.10.4 Workflows with Selection and the Convert Selection tasks.....	114
1.11 PDF Processing in Automation Engine.....	115
1.11.1 PDF Normalization in Automation Engine.....	115
1.11.2 Retaining XMP data in PDF tasks.....	121
1.12 Scripting.....	122
1.12.1 Automation Engine Script Runner.....	122
1.12.2 Getting Started with Scripting.....	123
1.12.3 Use case: Scripting.....	125
1.13 Example of Building and Using a Packaging Workflow.....	127
1.13.1 Build a Workflow.....	127
1.13.2 Use Shuttle in the Workflow.....	136
2. Reference Guide.....	146
2.1 Copyright Notice.....	146
2.2 What is Automation Engine.....	147
2.2.1 The Automation Engine Pilot.....	148
2.2.2 Data Management in Automation Engine.....	166
2.2.3 File Processing in Automation Engine.....	168
2.3 Different Ways of Working with Automation Engine.....	182
2.3.1 Working with Jobs.....	182
2.3.2 Working with Products.....	182
2.3.3 Working with Pages.....	182
2.3.4 Working with Proofs.....	189
2.3.5 Working with Plates.....	192
2.4 Server Administration Web Tool.....	207
2.4.1 Main.....	208
2.4.2 Downloading A Client App.....	208
2.4.3 Server Administration.....	209
2.5 The Automation Engine Shuttle.....	230
2.5.1 Launching the Automation Engine Shuttle.....	231
2.5.2 Working with the Automation Engine Shuttle.....	232
2.6 The Workflow Editor.....	239
2.7 The Tasks and Workflow Controls.....	239
2.7.1 Add Dynamic Marks.....	239
2.7.2 Add Ink Eaters.....	241

2.7.3 Add Marks To Imposition.....	245
2.7.4 Add SmartMarks.....	253
2.7.5 Add to Pages View.....	254
2.7.6 Add To Products.....	254
2.7.7 Adjust PDF Screening.....	254
2.7.8 Apply ArtPro Action List.....	257
2.7.9 Apply Creep.....	258
2.7.10 Archive Job.....	260
2.7.11 Assign Pages To RunList.....	260
2.7.12 Calculate Ink Key Settings (InkPlanner).....	262
2.7.13 Check Dynamic VDP Database.....	263
2.7.14 Check Job Parameters.....	264
2.7.15 Check Print Rules (PRC).....	264
2.7.16 Check Product Parameters.....	265
2.7.17 Check Technical Inks.....	265
2.7.18 Compare PDF.....	265
2.7.19 Convert CMYK Colors.....	266
2.7.20 Convert Colors with Equinox.....	267
2.7.21 Convert IMP File to Normalized PDF.....	269
2.7.22 Convert Selection.....	270
2.7.23 Copy or Move File.....	270
2.7.24 Create CAD Sheet.....	271
2.7.25 Create Design (DesignWizard).....	277
2.7.26 Create IMP File.....	279
2.7.27 Create Imposition from JDF Layout.....	282
2.7.28 Create Imposition from RunList.....	283
2.7.29 Create Job task.....	285
2.7.30 Create or Modify WebCenter Project.....	285
2.7.31 Create PAF / JPG / XML (LinkEdge).....	285
2.7.32 Create PDF File from JDF RunList.....	288
2.7.33 Create PDF File from RunList.....	289
2.7.34 Create Report (ReportMaker).....	290
2.7.35 Create RunList.....	298
2.7.36 Create RunList from PDF.....	300
2.7.37 Create Sheet Report from IMP File.....	302
2.7.38 Create Tiles.....	303
2.7.39 Create Wrapper File.....	303
2.7.40 Create XML File.....	308
2.7.41 Data Collector.....	308
2.7.42 Data Splitter.....	308
2.7.43 Delete File.....	308
2.7.44 Download WebCenter Document.....	308
2.7.45 Enrich Black.....	308

2.7.46 Expand Dynamic VDP data.....	310
2.7.47 Export ArtPro to Normalized PDF File.....	312
2.7.48 Export IMP File to GRI File.....	313
2.7.49 Export IMP File to JDF Layout.....	316
2.7.50 Export IMP File to JDF Post-Press.....	317
2.7.51 Export IMP File to PDF File.....	320
2.7.52 Export IMP File to PDF File Chain.....	320
2.7.53 Export IMP File to PostScript File.....	321
2.7.54 Export Imposition to JDF Layout.....	321
2.7.55 Export Imposition to PDF File.....	322
2.7.56 Export Nested Layouts.....	322
2.7.57 Export Step and Repeat to JDF Layout.....	322
2.7.58 Export to 3D.....	323
2.7.59 Export to ArtPro File.....	327
2.7.60 Export to EPS / DCS File.....	327
2.7.61 Export to Normalized PDF File.....	329
2.7.62 Export to Normalized PDF File (Embed All).....	330
2.7.63 Export to PDF File.....	331
2.7.64 Export to PDF File Chain.....	340
2.7.65 Export to PostScript File.....	341
2.7.66 Extract Inks.....	350
2.7.67 Fork.....	351
2.7.68 Image to MJM File.....	351
2.7.69 Image to Screened Separations.....	352
2.7.70 Image to Unscreened Separations.....	353
2.7.71 Import Image.....	353
2.7.72 Import JDF Layout.....	355
2.7.73 Import JDF Step & Repeat.....	355
2.7.74 Import JDF Stripping.....	358
2.7.75 Integrate via External Hot Folder.....	362
2.7.76 Integrate with WebCenter.....	364
2.7.77 Interact with Database.....	364
2.7.78 Interact with Web Service.....	364
2.7.79 Join.....	364
2.7.80 Join XML Files.....	364
2.7.81 Link Product To Job.....	364
2.7.82 Make Plate on CDI.....	364
2.7.83 Manage Color.....	371
2.7.84 Manage Milestone.....	375
2.7.85 Manage Product Status.....	375
2.7.86 Map Data.....	375
2.7.87 Mark as Proof.....	376
2.7.88 Mark File (link).....	376

2.7.89 Merge PDF Files.....	376
2.7.90 Modify Workflow Parameter Values.....	377
2.7.91 Normalize PDF File.....	377
2.7.92 Normalize PostScript / PDF / Illustrator 8.0 File.....	379
2.7.93 Optimize and Clean (PSFix).....	390
2.7.94 Optimize PDF Document.....	394
2.7.95 Optimize PDF Separations.....	399
2.7.96 Outline Fonts.....	405
2.7.97 Preflight via External Application.....	405
2.7.98 Preflight with PitStop.....	407
2.7.99 Prepare for Inspection.....	411
2.7.100 Prepare for Viewer.....	415
2.7.101 Prepare Graphics for Nesting.....	417
2.7.102 Prepare Station.....	417
2.7.103 Print.....	419
2.7.104 Print IMP File.....	420
2.7.105 Print on Digital Press.....	420
2.7.106 Print on Digital Press (PPML).....	431
2.7.107 Process PDF Spreads.....	431
2.7.108 Proof (FlexProof).....	435
2.7.109 Proof IMP File (FlexProof).....	444
2.7.110 Proof to File - Unscreened.....	444
2.7.111 Publish Imposition on WebCenter Chain.....	445
2.7.112 Publish On WebCenter.....	449
2.7.113 Relocate Job.....	449
2.7.114 Resize PDF for Output.....	450
2.7.115 Resolve OPI.....	452
2.7.116 Restore Job.....	453
2.7.117 Rip IMP File to Screened and Unscreened Tiff File.....	453
2.7.118 Rip to CDI LEN File.....	453
2.7.119 Rip to Screened and Unscreened DCS File.....	454
2.7.120 Rip to Screened and Unscreened PDF File.....	457
2.7.121 Rip to Screened and Unscreened PostScript File.....	459
2.7.122 Rip to Screened and Unscreened Tiff File.....	461
2.7.123 Router.....	469
2.7.124 Run Script.....	469
2.7.125 Select File.....	470
2.7.126 Select Mark File.....	471
2.7.127 Select Referenced File.....	471
2.7.128 Send E-mail.....	471
2.7.129 Send PostScript File to Device.....	473
2.7.130 Send to Nexus.....	473
2.7.131 Send to Odystar.....	474

2.7.132 Set Priority.....	475
2.7.133 Sort.....	475
2.7.134 Split Pages.....	475
2.7.135 Split XML.....	476
2.7.136 Step & Repeat CAD based.....	476
2.7.137 Step & Repeat Tabular.....	478
2.7.138 Step & Repeat Tabular (Dynamic).....	492
2.7.139 Step & Repeat Template based.....	497
2.7.140 Step & Repeat Template based (Dynamic).....	499
2.7.141 Submit to Gang Run.....	500
2.7.142 Submit XML to Gang Run.....	500
2.7.143 Track Edit Session.....	507
2.7.144 Trap with PowerTrapper.....	511
2.7.145 Unzip.....	519
2.7.146 Upload via FTP.....	519
2.7.147 Verify Certification.....	521
2.7.148 Version PDF.....	522
2.7.149 Wait.....	524
2.7.150 Wait for Action (Checkpoint).....	524
2.7.151 Wait for Files.....	524
2.7.152 Wait for Files (via JDF).....	524
2.7.153 Wait for Product Status.....	525
2.7.154 Zip.....	525
2.7.155 Deprecated and Obsolete Tasks.....	526
2.8 The Automation Engine Viewer.....	578
2.8.1 Introduction.....	578
2.8.2 Supported File Types.....	580
2.8.3 Viewer Statuses in the Files View.....	581
2.8.4 The Viewer Window.....	582
2.8.5 Working with the Viewer.....	608
2.9 The Tools.....	616
2.9.1 To Do List.....	616
2.9.2 SmartNames.....	621
2.9.3 Explore.....	621
2.9.4 Task History.....	621
2.9.5 View Cleanup Rules.....	626
2.9.6 Merger Monitor.....	627
2.9.7 Configure.....	628
2.9.8 Queue Monitor.....	688
2.9.9 Containers.....	691
2.9.10 Access Points.....	696
2.9.11 Customers.....	696
2.9.12 Users.....	698

2.9.13 Connection Doctor.....	707
2.10 SmartNames.....	708
2.11 The ArtPro Action List Editor.....	708
2.11.1 Automating ArtPro Processing.....	708
2.11.2 Working with ArtPro Action Lists.....	708
2.11.3 Handling Fonts in ArtPro Files.....	711
2.11.4 Available Actions.....	712
2.12 The PitStop Profile Editor.....	782
2.12.1 Setting Up the PitStop Profile Editor.....	782
2.12.2 Using SmartNames in PitStop Profile Editor.....	784
2.13 The RunList Editor.....	786
2.13.1 Setting Up the RunList Editor.....	786
2.13.2 Creating a RunList.....	787
2.13.3 Viewing the RunList Files.....	792
2.13.4 Changing the Files' Extension.....	792
2.13.5 Sorting the RunList's Files.....	793
2.13.6 Submitting the RunList to a Workflow.....	793
2.14 The Script Runner.....	794
2.14.1 Scripting for Script Runner on Mac OS.....	794
2.14.2 Scripting for Script Runner on Windows.....	806
2.14.3 Using ExtendScript on Script Runner (On MacOS/Windows).....	817
2.14.4 Appendix : Script Samples.....	823
3. Jobs.....	828
3.1 Job Basics.....	828
3.1.1 The Job Concept.....	828
3.1.2 Types of Jobs.....	829
3.1.3 The Job Views.....	830
3.1.4 The Job Menu.....	834
3.2 Creating a Job.....	835
3.3 Job Setup Parameters.....	837
3.3.1 Job tab.....	838
3.3.2 Customer tab.....	839
3.3.3 Inks tab.....	840
3.3.4 Parameters tab.....	841
3.3.5 Bar Codes tab.....	844
3.3.6 Step&Repeat tab.....	847
3.3.7 Categorization tab.....	847
3.3.8 Advanced tab.....	848
3.3.9 Automation tab.....	849
3.3.10 Page Assignment tab.....	849
3.3.11 Imposition tab.....	849
3.4 About Job Folder Location.....	850
3.4.1 General Note on Data Structure.....	850

3.4.2 FAQs on Job Folder Location.....	851
3.4.3 Good Practices regarding Job Folder Location.....	852
3.5 Uploading Files to Jobs.....	853
3.6 Milestones: Tracking Job Status.....	854
3.6.1 Concept.....	854
3.6.2 The Milestones View.....	855
3.6.3 Adding Milestones to a Job.....	858
3.6.4 Manage (Custom) Milestones.....	860
3.6.5 Working with Milestone Actions.....	861
3.6.6 Using Milestone Statuses to Filter Jobs.....	863
3.6.7 Using Milestones in Workflows: the Manage Milestone Task.....	864
3.6.8 Informing Other Systems of the Milestones Statuses.....	866
3.7 About Removing Jobs.....	867
3.8 Tasks you can Launch on Jobs.....	869
3.8.1 The Relocate Job Task.....	869
3.8.2 The Archive Job Task.....	871
3.8.3 The Restore Job Task.....	873
3.8.4 The Check Job Parameters Task.....	874
3.9 Automated Creation of Jobs.....	875
4. Products.....	876
4.1 Products Basics.....	876
4.1.1 The Products Concept.....	876
4.1.2 Global Products and Job Products.....	879
4.1.3 Products can have Multiple Parts.....	880
4.2 Products Data Structure.....	881
4.2.1 Examples of Data Zone Depth.....	883
4.3 Products Views and Menu.....	886
4.3.1 The Products Gallery View.....	886
4.3.2 The Product(s) in a Job View.....	890
4.3.3 The Product Menu.....	892
4.4 Product (Part) Properties.....	893
4.4.1 Product Setup.....	894
4.4.2 Product Part Setup.....	895
4.5 Creating Products.....	897
4.5.1 Creating Products Manually.....	897
4.5.2 Uploading As a New Product.....	898
4.5.3 Creating Products Automatically.....	899
4.6 Linking Products to a Job.....	903
4.7 Launching Tasks on Products.....	904
4.8 Working with Products in Editors.....	905
4.9 Tasks specific for Products.....	908
4.9.1 The Add to Products task.....	908
4.9.2 The Link Product to Job task.....	909

4.9.3 The Manage Product Status task.....	910
4.9.4 The Wait for Product Status task.....	910
4.9.5 The Check Product Parameters task.....	912
4.10 Publishing Products to WebCenter.....	913
4.11 Backing Up and Archiving Products.....	913
4.12 Removing Products.....	914
5. SmartNames.....	916
5.1 Concept.....	916
5.2 General Principles.....	918
5.2.1 Using a SmartName.....	918
5.2.2 The SmartNames View.....	920
5.2.3 SmartName Types (Overview).....	923
5.2.4 SmartName Categories.....	924
5.2.5 SmartName Scopes.....	926
5.2.6 SmartName Tags.....	927
5.2.7 Creating a SmartName (General).....	929
5.2.8 Testing SmartNames.....	931
5.2.9 SmartName Relations and Dependencies.....	935
5.2.10 Exporting and Importing SmartNames.....	938
5.3 System (Value) SmartNames.....	939
5.3.1 Default System SmartNames.....	939
5.3.2 Creating System Value SmartNames.....	944
5.4 String Extract SmartNames.....	948
5.4.1 Creating a String Extract SmartName.....	948
5.5 CSV Text Extract SmartNames.....	951
5.5.1 Creating a CSV Text Extract SmartName.....	951
5.5.2 Example.....	952
5.6 Xpath Query SmartNames.....	953
5.6.1 Creating an Xpath Query SmartName.....	953
5.6.2 The XPath Builder.....	954
5.7 XMP path Query SmartNames.....	958
5.7.1 Creating an XMP path Query SmartName.....	959
5.7.2 Examples.....	959
5.8 Conditional SmartNames.....	961
5.8.1 Creating a Conditional SmartName (generic).....	961
5.8.2 Setting up a Conditional Block.....	962
5.8.3 Examples.....	964
5.9 Database Query SmartNames.....	968
5.9.1 Concept.....	968
5.9.2 Creating a Database Query SmartName.....	969
5.10 Script SmartNames.....	970
5.10.1 Creating a Script SmartName.....	971
5.10.2 Examples.....	971

5.11	Formatting SmartNames.....	972
5.11.1	Text Formatting.....	972
5.11.2	Number Formatting.....	974
5.11.3	Date and Time Formatting.....	975
5.12	Using Regular Expressions.....	978
5.12.1	A Selection of Useful Characters in Regular Expressions.....	978
5.12.2	Using Parentheses to Extract.....	980
5.12.3	Examples from Users.....	981
5.13	SmartNames of Parameter Values.....	982
6.	Workflows.....	986
6.1	Workflow Basics.....	986
6.2	Building Workflows using the Workflow Editor.....	987
6.2.1	The Workflow Editor Window.....	987
6.2.2	The Workflow Editor Menus.....	988
6.2.3	The Workflow Editor Tool Bar.....	990
6.2.4	Building a Workflow from Scratch.....	990
6.2.5	Building a Workflow using Custom Tickets.....	993
6.2.6	Visual Optimizations of a Workflow.....	995
6.2.7	Subworkflows.....	998
6.2.8	Inserting Wait Points.....	1001
6.2.9	Routing in Workflows.....	1006
6.2.10	Exchanging Workflows.....	1009
6.2.11	Good Practices for Building Workflows.....	1010
6.3	Workflow Controls.....	1014
6.3.1	Modify Workflow Parameter Values.....	1015
6.3.2	Select Referenced File.....	1018
6.3.3	Set Priority.....	1021
6.3.4	Sort.....	1021
6.3.5	Router.....	1024
6.3.6	Mark File and Select Marked File.....	1029
6.3.7	Data Collector.....	1029
6.3.8	Data Splitter.....	1030
6.3.9	Fork and Join.....	1032
6.4	Launching and Diagnosing Workflows.....	1033
6.4.1	The Parameter Inspector.....	1033
6.4.2	Launching a Workflow on a File.....	1034
6.4.3	Using Favorite Workflow Tickets.....	1036
6.4.4	Building and Launching a Workflow On The Fly.....	1041
6.4.5	General Workflow Task Options.....	1042
6.4.6	Relaunching Your Workflow with Different Settings.....	1043
6.4.7	Diagnosing a Workflow in the Tasks Pane.....	1043
6.4.8	Diagnosing a Workflow in the Workflow Editor.....	1047
6.4.9	Pausing or Cancelling Your Workflow.....	1050

6.5 Using Public Parameters in Workflows.....	1051
6.5.1 Concept.....	1051
6.5.2 Creating Public Parameters in a Workflow.....	1052
6.5.3 Managing Public Parameters in a Workflow.....	1053
6.5.4 Making Presets to Simplify the Users' Choices.....	1055
6.5.5 Example of Further Customizing the Public Parameters Dialog.....	1058
6.5.6 Launching a Workflow with Public Parameters.....	1062
6.6 Using Workflow Parameters.....	1065
6.6.1 Concept.....	1065
6.6.2 Most Typical Use Cases for Using Workflow Parameters.....	1067
6.6.3 Creating Workflow Parameters Manually.....	1069
6.6.4 Loading Workflow Parameters from XML.....	1070
6.6.5 Using Workflow Parameters.....	1071
6.6.6 Using Workflow Parameter Values from XML.....	1072
6.6.7 Changing the Value of a Workflow Parameter.....	1074
6.6.8 Modifying Workflow Parameters During a Workflow.....	1075
6.7 Sample Workflows.....	1076
6.8 Automated Launching of Workflows.....	1078
6.9 Migrating Old Task Chains.....	1078
7. Working with Gang Run Printing.....	1081
7.1 What is Gang Run Printing?.....	1081
7.1.1 Nesting.....	1081
7.1.2 Ganging.....	1082
7.1.3 Tiling.....	1083
7.1.4 Manual or Automated Gang Run Printing.....	1084
7.2 Working with Gang Run Printing.....	1084
7.2.1 Preparing and Submitting your Graphics (Input Workflow).....	1085
7.2.2 Creating Nested Layouts on the Nesting Server.....	1087
7.2.3 Getting Production Files out of your Nested Layouts (Output Workflow).....	1090
7.3 The Gang Run Printing View.....	1091
7.3.1 Working with Substrates.....	1091
7.3.2 Working with Graphics Submitted for Printing.....	1098
7.4 The Gang Run Printing Tasks.....	1100
7.4.1 Prepare Graphics for Nesting.....	1100
7.4.2 Create Tiles.....	1104
7.4.3 Submit to Gang Run.....	1109
7.4.4 Export Nested Layouts.....	1116
8. RIP'ing with Imaging Engine.....	1123
8.1 Copyright Notice.....	1123
8.2 What's New in Imaging Engine 14.1 Prerelease Version X?.....	1124
8.3 What is Imaging Engine?.....	1124
8.4 Using Imaging Engine for Proofing and RIP'ing.....	1125

8.5 What Does Imaging Engine Do?.....	1125
8.5.1 Color Management.....	1125
8.5.2 Rasterizing Vector Graphics to Bitmaps.....	1126
8.5.3 Screening.....	1127
8.5.4 Dot Gain Compensation.....	1142
8.5.5 RIP'ed File Formats.....	1147
8.6 Imaging Engine Installation and Configuration.....	1150
8.6.1 Licensing.....	1150
8.6.2 Installation.....	1150
8.6.3 Configuration in the Automation Engine Pilot.....	1154
8.7 The Imaging Engine Tasks.....	1156
8.7.1 Image to Screened Separations.....	1156
8.7.2 Image to Unscreened Separations.....	1175
8.7.3 Proof.....	1193
8.7.4 Proof to File - Unscreened.....	1204
8.7.5 Working with Separations.....	1219
9. Integrating with External Systems.....	1228
9.1 Introduction.....	1228
9.1.1 Why Integrate?.....	1228
9.1.2 External System sending commands To Automation Engine.....	1229
9.1.3 Automation Engine sending commands To an External System.....	1229
9.1.4 Some general advice.....	1229
9.1.5 Need any help?.....	1230
9.1.6 What about JDF?.....	1230
9.2 Integration Concepts.....	1230
9.2.1 Access Points Triggering a Workflow on Automation Engine.....	1231
9.2.2 Automation Engine tasks Interacting directly with the External System.....	1232
9.2.3 Automation Engine Mapper tasks Transforming Data.....	1233
9.2.4 Automation Engine sending data Back to the External System.....	1235
9.3 Access Points.....	1236
9.3.1 Folder Access Point.....	1238
9.3.2 FTP Access Point.....	1240
9.3.3 Mail Access Point.....	1243
9.3.4 Database Access Point.....	1245
9.3.5 Web Service Access Point.....	1249
9.4 Data Transforming Tasks.....	1255
9.4.1 Create XML File task.....	1255
9.4.2 Split XML Task.....	1257
9.4.3 Join XML Files Task.....	1260
9.4.4 Map Data task.....	1264
9.5 Tasks Interacting with other Systems.....	1274
9.5.1 Interact with Database task.....	1274
9.5.2 Interact with Web Service task.....	1276

9.6	Tasks often used in Integration Workflows.....	1280
9.6.1	Create Job task.....	1280
9.6.2	Add To Products task.....	1287
9.6.3	Link Product To Job task.....	1287
9.7	Examples.....	1287
9.7.1	Creating a Job using XML.....	1287
9.7.2	Creating Multiple Jobs from One XML.....	1295
9.7.3	Create Multiple Jobs from a CSV file.....	1300
9.7.4	Creating Jobs using a Database Access Point.....	1310
9.7.5	Launching a workflow with parameters from an XML.....	1319
9.7.6	Getting info from a Web Service into a Workflow.....	1330
9.7.7	Updating a Product Status in an External System via Interact with Database.....	1339
10.	Integrating with WebCenter.....	1346
10.1	Introduction.....	1346
10.2	WebCenter Sites.....	1347
10.3	The WebCenter View in Automation Engine Pilot.....	1349
10.4	WebCenter Integration Tasks.....	1350
10.4.1	Create or Modify WebCenter Project task.....	1350
10.4.2	Publish on WebCenter task.....	1353
10.4.3	Download WebCenter Document task.....	1361
10.5	Advanced WebCenter Integration.....	1362
10.5.1	Integrate with WebCenter task.....	1363
11.	Troubleshooting.....	1366
11.1	Troubleshooting the Automation Engine Server.....	1366
11.1.1	How to check the status of the AE databases.....	1366
11.1.2	How to run network diagnostics tests on the AE server.....	1367
11.2	Troubleshooting the Automation Engine Pilot.....	1369
11.2.1	Connection Doctor.....	1369
12.	IT requirements and FAQs.....	1374
12.1	Introduction.....	1374
12.2	Technical Details.....	1374
12.2.1	Automation Engine in its environment.....	1374
12.2.2	Automation Engine installation.....	1375
12.2.3	Network Details.....	1377
12.2.4	Database server.....	1378
12.2.5	File Server.....	1380
12.2.6	Mail Server.....	1380
12.2.7	Users and Groups.....	1381
12.2.8	The Automation Engine Pilot.....	1381
12.3	General Guidelines.....	1382
12.3.1	Scalability and Performance.....	1382
12.3.2	Snapshots/Cloning.....	1387

12.3.3 Remote Desktop Services.....	1387
12.3.4 Sharing the Automation Engine server with Other Applications.....	1387

1. User Guide

1.1 About Automation Engine

Maximizing Productivity

Automation Engine doesn't only enable you to keep track of your files throughout your workflow, it also optimizes and automates this workflow to increase your productivity.

Automation Engine:

- avoids double entry and possible operator errors,
- processes files faster (through digital standardization of production procedures, hot folders automation, smart notifications...).
- frees operators from administrative tasks so they can concentrate on value-adding graphical tasks,
- allows operators to share their expertise through tickets,
- keeps jobs, related files and related customer information organized to facilitate communication and treatment of additional orders.

All this maximizes your efficiency and the quality of your deliverables, while lowering the cost of prepress-related actions.

Integrating with your Existing Systems

Automation Engine can integrate seamlessly with your graphical editors, RIPs, file servers, business systems...

It supports the industry standards (PDF, XML, XMP, SQL queries, Web Services, JDF...).

The Automation Engine client application (the "Pilot") can be installed on any Mac or Windows machine in your network, so all your operators can view files, get information or launch processing tasks as needed.

Growing with You

Automation Engine is very modular and can be tailored to your needs, while always offering growth possibilities.

From a background assistant to Adobe® Illustrator® to a central prepress server to a complete server platform linking to your administrative systems, Automation Engine can grow with you to let you handle larger accounts, accepting more jobs without increasing your costs.

Find more general product information on <http://www.esko.com/AutomationEngine>.

1.1.1 Copyright Notice

© Copyright 2015 Esko Software BVBA, Gent, Belgium

All rights reserved. This material, information and instructions for use contained herein are the property of Esko Software BVBA. The material, information and instructions are provided on an AS IS basis without warranty of any kind. There are no warranties granted or extended by this document. Furthermore Esko Software BVBA does not warrant, guarantee or make any representations regarding the use, or the results of the use of the software or the information contained herein. Esko Software BVBA shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or inability to use the software or the information contained herein.

The information contained herein is subject to change without notice. Revisions may be issued from time to time to advise of such changes and/or additions.

No part of this document may be reproduced, stored in a data base or retrieval system, or published, in any form or in any way, electronically, mechanically, by print, photoprint, microfilm or any other means without prior written permission from Esko Software BVBA.

This document supersedes all previous dated versions.

PANTONE[®], PantoneLIVE and other Pantone trademarks are the property of Pantone LLC. All other trademarks or registered trademarks are the property of their respective owners. Pantone is a wholly owned subsidiary of X-Rite, Incorporated. © Pantone LLC, 2014. All rights reserved.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software are copyright © 1996-2002 The FreeType Project (www.freetype.org). All rights reserved.

Portions of this software are copyright 2006 Feeling Software, copyright 2005-2006 Autodesk Media Entertainment.

Portions of this software are copyright ©1998-2003 Daniel Veillard. All rights reserved.

Portions of this software are copyright ©1999-2006 The Botan Project. All rights reserved.

Part of the software embedded in this product is gSOAP software. Portions created by gSOAP are Copyright ©2001-2004 Robert A. van Engelen, Genivia inc. All rights reserved.

Portions of this software are copyright ©1998-2008 The OpenSSL Project and ©1995-1998 Eric Young (ey@cryptsoft.com). All rights reserved.

This product includes software developed by the Apache Software Foundation (<http://www.apache.org/>).

Adobe, the Adobe logo, Acrobat, the Acrobat logo, Adobe Creative Suite, Illustrator, InDesign, PDF, Photoshop, PostScript, XMP and the Powered by XMP logo are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Microsoft and the Microsoft logo are registered trademarks of Microsoft Corporation in the United States and other countries.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. 1986 2003. All Rights Reserved.

JDF and the JDF logo are trademarks of the CIP4 Organisation. Copyright 2001 The International Cooperation for the Integration of Processes in Prepress, Press and Postpress (CIP4). All rights reserved.

The Esko software contains the RSA Data Security, Inc. MD5 Message-Digest Algorithm.

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Sun Microsystems in the U.S. and other countries.

Part of this software uses technology by BestTM Color Technology (EFI). EFI and Bestcolor are registered trademarks of Electronics For Imaging GmbH in the U.S. Patent and Trademark Office.

Contains PowerNest library Copyrighted and Licensed by Alma, 2005 – 2007.

All other product names are trademarks or registered trademarks of their respective owners.

Correspondence regarding this publication should be forwarded to:

Esko Software BVBA

Kortrijksesteenweg 1095

B – 9051 Gent

info.eur@esko.com

1.1.2 Getting Help

You can also find the PDF version of this manual on the Automation Engine on <http://help.esko.com/products/Automation%20Engine> .

To get an overview all of the keyboard shortcuts you can use in the Automation Engine Pilot, go to **Help > Keyboard Shortcuts...**

1.2 What is Automation Engine

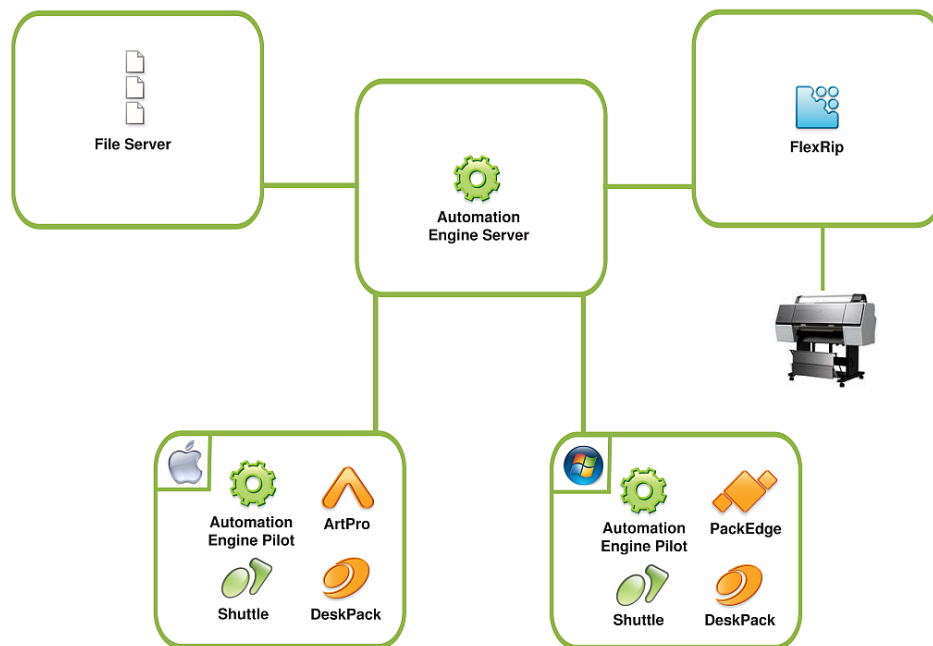
Automation Engine Architecture

Automation Engine is a **client/server** workflow software, running on **Mac and Windows**. This means that one machine runs the Automation Engine server, to which you can connect from all Mac and Windows machines in your network (using the Automation Engine **Pilot**).

Your Automation Engine server can connect to **graphical editing software** like ArtPro, PackEdge, Adobe Illustrator (using the DeskPack plug-ins)...

It can also connect to your **RIP** software to ensure professional output.

You can store your **files** either on the Automation Engine server, on a dedicated file server, or on different machines over the network.



Automation Engine Processing

Automation Engine processes your files using single **Tasks** (for example a Trap or RIP task), or **Workflows** (more powerful and flexible series of tasks that can take complete care of the file from arrival to final output).

Files and Jobs in Automation Engine

In Automation Engine, your files can be part of **Jobs** (customer orders that contain files and related customer information), that are located in **Containers** (local or remote shared folders that Automation Engine can access).

Depending on how you want to work with Automation Engine, you can link your jobs to customer **Products**, focus on job **Pages, Proofs, Plates...**

Automation Engine Integration

Automation Engine can not only connect to editing and RIP software, but also to **WebCenter** (Esko's collaborative web platform) and other Esko workflow software (**Nexus** and **Odystar**). Automation Engine can also be integrated with external systems.

1.2.1 The Automation Engine Pilot

The Automation Engine server software runs on a Windows server. Every Automation Engine action is performed through the Automation Engine client, called the **Pilot**.

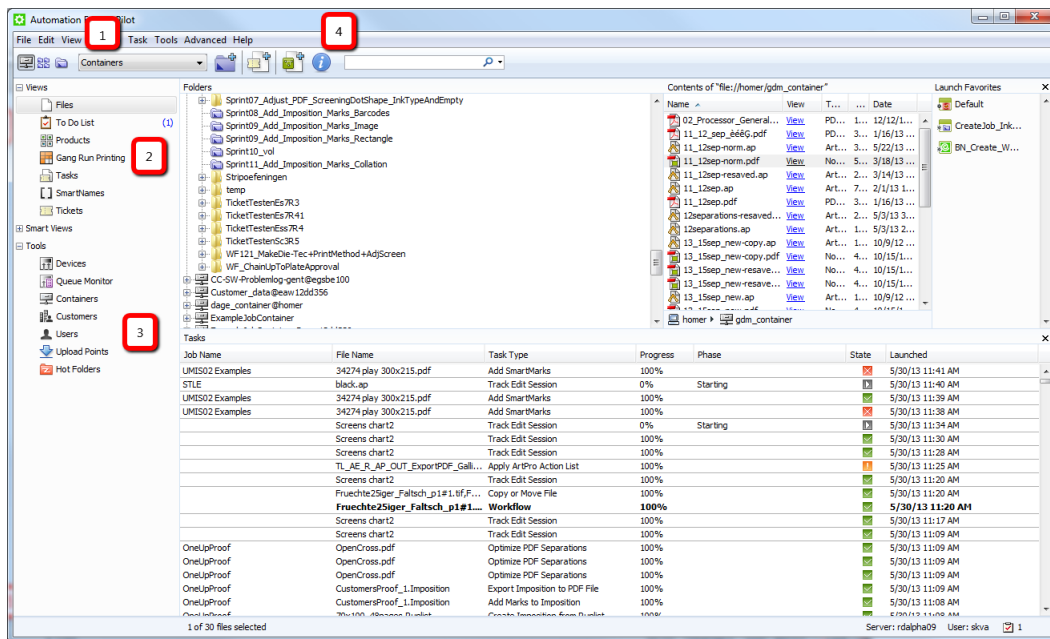
The Pilot can be installed on any Mac or Windows machine on your local network.

Depending on the user access rights you will set, operators can use the Pilot to:



- organize Jobs and files,
- create and modify tickets,

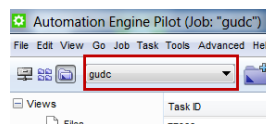
- launch and monitor tasks,
- control workflows and devices,
- administrate users and perform general configuration,
- etc.


The Pilot's Main Window







1. These icons show different modes in the Pilot:

- **Containers** : This shows all the data that Automation Engine can access, in the different Containers. Learn more in [Containers](#) on page 24.
- **Job Overview** : This lists all jobs existing in the Automation Engine jobs database. Learn more in [Jobs](#) on page 828.
- **Last used Job** : This shows the last job you worked on, and the job's data (files and folders).



- The **Views** highlight different parts of the Automation Engine functionality. Certain views are only accessible in defined workflows. Learn more in [Views](#) on page 22.
- The **Tools** area contains tools useful to administrate Automation Engine. Learn more in [The Tools](#) on page 616.
- The **Create Job**  button allows you to create a job.


- The **New Task**  button creates a new Ticket from the pilot.
- You can create a new workflow from the pilot using .
- Using the  button, you can access the information regarding the files.

Note: In the **Job Overview** mode, only **Create Job**  is available.

The contents of the central panes depend on the **View** you are in.

Tab support

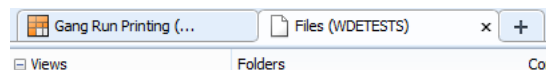
In any Automation Engine view, a new tab can be created by using one of the following methods:

- Click **File > New Tab** .
- Use the keyboard shortcut **Control-T** (Windows) or **Command-T** (Mac).
- If the tab bar is visible, you can also create a new tab by clicking the plus button in the tab bar .
- If the tab bar is visible, you can also create a new tab by right-clicking on the tab bar and choosing **New Tab**.

Note: The new tab will open with the same view that was active.

An opened tab can be closed by one of the following methods:

- Click on the close button in a tab (visible by hovering over the tab).




- Use the keyboard shortcut **Control-W** (Windows), or **Command-W** (Mac).
- Click on **File > Close Tab** .
- Right-click on the desired tab and choose **Close Tab**.

Note: To close all other tabs, save the desired one, you can also choose **Close Other Tabs** from this menu.

Full Screen Mode on Mac

When running the Automation Engine Pilot on Mac OSX or newer, you can enter and exit the Full Screen Mode by using one of the following methods:

- Click on the **Full Screen** button  in the top right corner of the main window
- Click on **Views > Enter Full Screen** .
- Use the keyboard shortcut **Command + Control + F**.

Views

The Pilot's **Views** highlight different parts of the Automation Engine functionality. You can use them to work with your data in different ways (see [Different Ways of Working with Automation Engine](#) on page 182).

Note:

- Depending on the Data View mode you are using (**Containers**, **Job Overview** or **Last used job**), you will have access to different views. Only those that make sense are shown.
- Depending on your [access rights](#), you may not be able to see some of the views.

Files

The **Files** view allows you to manage your jobs, files, favorite tickets, tasks... Learn more in [The Files View](#) on page 152 for more information.

To Do List

The **To Do List** view lists all actions that need user intervention. Learn more in [The To Do List View](#) on page 153 and [To Do List](#) on page 616.

Pages

The **Pages** view is only used in page workflows. It is extremely useful when working with page impositions. For more information, see [Working with Pages](#) on page 182.

Products

The **Products** view shows the items you added to your Products database. This concept is useful to customer who have re-prints. Products represent production data that you linked to a Job. It is the Job order that has the specifications how to output the product this time, with or without many other Product items.

Products are what you keep long term, until you are sure you will not have to produce them again. Jobs however are Work In Progress, temporary by definition. A Job should be removed when it is finished. Learn more in the dedicated chapter [Products](#) on page 876.

Proofs

The **Proof** view is only used in page workflows. It shows the proof sets and the proof details. Learn more in [Working with Proofs](#) on page 189.

WebCenter

This view enables Pilot users to see what documents are inside a WebCenter project, without having to use a browser. Read more in separate chapter [Integrating with WebCenter](#) on page 1346, in its section [The WebCenter View in Automation Engine Pilot](#) on page 1349.

Plates

The **Plates** view is only used in page workflows. It shows details about all plates associated with a job. It also includes advanced features for managing plates for jobs with multiple variants. Learn more in [Working with Plates](#) on page 192.

CDI

The **CDI** view shows all the files sent to the Digital Flexo Suite and used to image a flexo plate on a Cyrel Digital Imager. See [The CDI View](#) on page 159.

Gang Run Printing

The **Gang Run Printing** view in the Pilot shows an overview of all graphics waiting to be nested and printed on your available substrate sheets. It has two parts:

- The list of all substrates.
- The table of graphics waiting to be nested and printed on the selected substrate.

For more information, please see [The Gang Run Printing View](#) on page 1091 in the separate chapter [Working with Gang Run Printing](#) on page 1081.

Tasks

The **Tasks** view shows the running tasks, the tasks waiting for execution and the finished tasks. Learn more in [The Tasks View](#) on page 160. This list of tasks can be automatically cleaned up by using [Task Cleanup Rules](#) on page 682.

Devices

The **Devices** view lists all output devices connected to your Automation Engine server. It allows you to monitor, start and stop your devices and device queues.

Milestones

The **Milestones** view shows the milestones set for your job (in **Last used job** mode), or shows the milestones set for all jobs (in **Job Overview** mode). Learn more in [Milestones: Tracking Job Status](#) on page 854.

Tickets

The **Tickets** view shows a list of all tickets available on your server (both the default and your custom tickets). Learn more in [Tickets](#) on page 26.

Jobs

This view is only visible in **Job Overview** mode. It lists all or a filtered list of your jobs in the Automation Engine job database. Learn more in [The Job Views](#) on page 830.

SmartNames

This view lists all SmartNames defined on the server. It is not visible in **Job Overview** mode. Learn more in [The SmartNames View](#) on page 920.

1.2.2 Data Management in Automation Engine

Automation Engine works with [Containers](#) on page 24, [Jobs](#) on page 82, [Products](#) on page 876, files and folders.

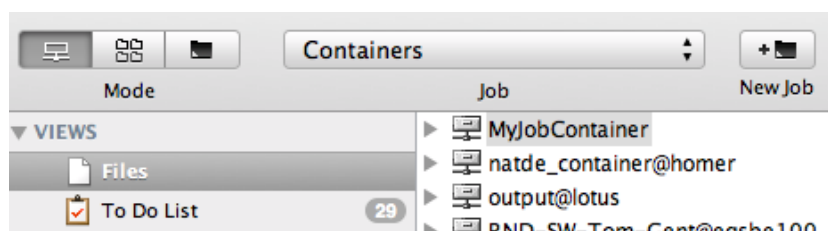
Customers with page (imposition) workflows can use the extra modes [Pages](#), [Proofs](#) and [Plates](#).

Containers

A **Container** is a network data share that contains the data that Automation Engine can work with. You can make (part of) an existing network share a Container or you can have Automation Engine create such network shares. These Containers can be located either on your Automation Engine server or another computer in the same LAN.

By default, you have an empty **ExampleJobContainer**, on the disk drive that Automation Engine was installed on.

You can have many Containers. You can see and browse through them in the Pilots' **Containers** mode.




To create a Container, see [Creating a Container](#) on page 691.

The Job Concept

What is an Automation Engine Job?

An Automation Engine Job represents an amount of work to be done for a certain order in your business system. This can be a classic order management system but it could also come from a web application that needs Automation Engine to execute some preprocess tasks.

A Job is an entry in the Automation Engine Jobs database and has a unique name. In the Pilot, Jobs are represented by this blue folder icon .

In Automation Engine, a Job consists of 2 things:

- **data:** files and folders, stored under the Job's **Job Folder**, in a Container.
- **metadata:** the parameters of that job order, usually how the order management system describes this job. They can be general parameters like the job's name, order ID, customer and due date but can also be parameters describing what to do with the input files to get to the targeted result.

Jobs can be created manually by an Automation Engine user or automatically via some integration with an external system.

A Job represents Work In Progress (WIP). This means that you should remove Jobs that are 'done'. Removing a Job from the Automation Engine Jobs database does not mean that you have to delete the related data files.

Most Automation is based on Jobs

A lot of the automation that Automation Engine brings comes from the fact that tasks can automatically use parameters from their Job context. Job parameters automatically become SmartNames and the (workflow) task tickets can use those SmartNames to pick up their values.

Note: Customers that do not use Jobs at all find a similar level of automation by using **Workflow Parameters**. Learn more in [Using Workflow Parameters](#) on page 1065.

Launching tasks on Jobs

Jobs can be managed manually in the Pilot, but several actions are also possible by launching tasks.

- The [Create Job task](#) on page 1280 task allows to create and modify Jobs.
- The [The Check Job Parameters Task](#) on page 874 task allows to check and correct barcode and ink parameters used in designs.
- The [Archive Job](#) task and the [Restore Job](#) task allow to remove Jobs, store their data on a different network place and, if needed later, restore them back, including their metadata.
- The [Relocate Job](#) task allows to move a Job's Job Folder to a different place. For example to another Container that has more disk space.

Learn more about these tasks in [Tasks you can Launch on Jobs](#) on page 869.

Like all tasks, these tasks can also be triggered by an external system. Learn more in [Integrating with External Systems](#) on page 1228.

1.2.3 File Processing in Automation Engine

In Automation Engine, you can process your files using single **Tickets** or **Workflows** (a sequence of Tickets).

Launching those tickets or workflow results in (a series of) **Tasks**.

A **Ticket** contains all the parameters for that task.

Tasks

What is a Task?

A task is an action executed on a file, a folder or a Job on the Automation Engine server.

Types of Tasks

There are seven types of tasks:

1. **Conversion** tasks such as **Export to PDF, Export to PostScript...**
2. **File Creation** tasks such as **Create Wrapper File, Create PAF / JPG / XML...**
3. **File Editing** tasks such as **Contourize & Clean, Trap...**
4. **Verification** tasks such as **Check Job Parameters, Check Print Rules, Preflight...**
5. **Administration** tasks such as **Prepare to Archive, Upload via FTP, Zip...**
6. **RIP** tasks.
7. **Device Output** tasks.

Task Settings

You can define settings for the tasks, and save these settings as **Tickets** to reuse them (see [Tickets](#) on page 26).

Task Progress

When a task is launched, you can monitor its progress in the Pilot. See [Checking the Status of the Task](#) on page 37.

Workflows

You can build **Workflows** and launch them on your files using the **Workflow Editor**. Learn more in [Building Workflows using the Workflow Editor](#) on page 987 .

You can also launch them from the **Pilot**, from **Shuttle** (see [Using Shuttle to Upload and Process Files](#) on page 47) or from within most Esko Editors. They can also be launched automatically, triggered by external signals (files arriving, commands from external systems, etc.).

Learn more in the dedicated chapter [Workflows](#) on page 986.

Tickets

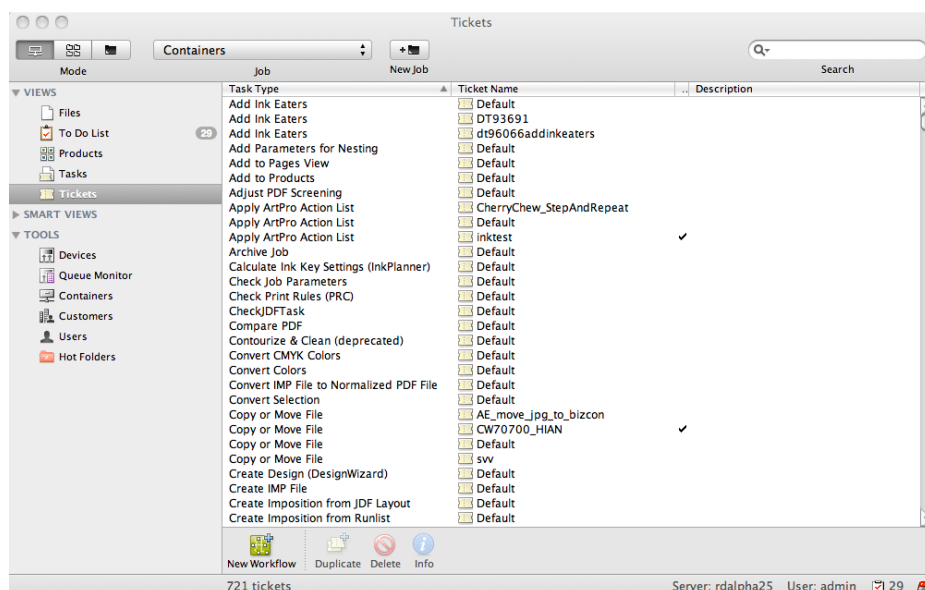
What is a Ticket?

The settings used when executing a task or workflow on a file can be saved as a ticket.



Default and Custom Tickets

The Pilot comes with a default ticket for each task type, but you can also create custom tickets. You cannot overwrite or delete default tickets.

You can see all the available tickets (default and custom) in the Pilot's **Tickets** view.










Global and Job Tickets

Global tickets () can be used on any file you can access through the Pilot. Job tickets () are only available when you work in the Job that you saved them for.

Public Tickets

You can make tickets public if you want to submit files to them from Shuttle. There are several ways to make a ticket public (see [Making Your Ticket Public](#) on page 49 in the User Guide).

Public tickets have a mark in the **Public** column.

Task Type	Ticket Name	Public	Description
Zip	 Default		
Workflow	 WVC_Approval		
Workflow	 WaitForRunListFiles		
Workflow	 TrapWF		
Workflow	 Trap_nestedWF	✓	
Workflow	 SimpleTrap		
Workflow	 Send2Ody		

Note: Depending on your [access rights](#), you may only see public tickets, and public parameters inside the tickets.

1.3 Getting Started with Automation Engine

1.3.1 Starting Automation Engine

To start working with Automation Engine you need to:

- Start the Automation Engine server software, see [Starting the server](#).
- Start the Automation Engine Pilot, see [Starting the Pilot](#).

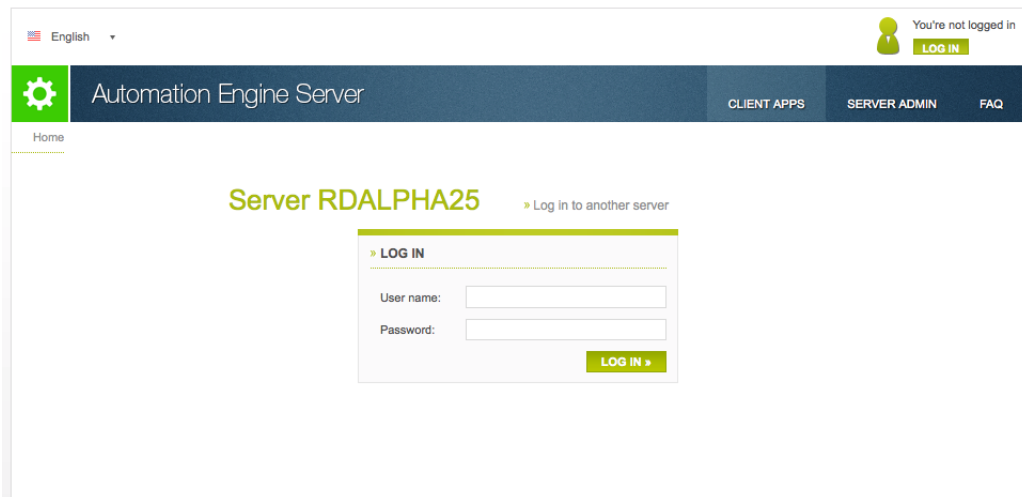
Starting the Automation Engine Server Software

Automation Engine features a new web tool for server administration. This means you can access the server administration tool from any workstation via <http://servername> or <http://servername:9999/>.

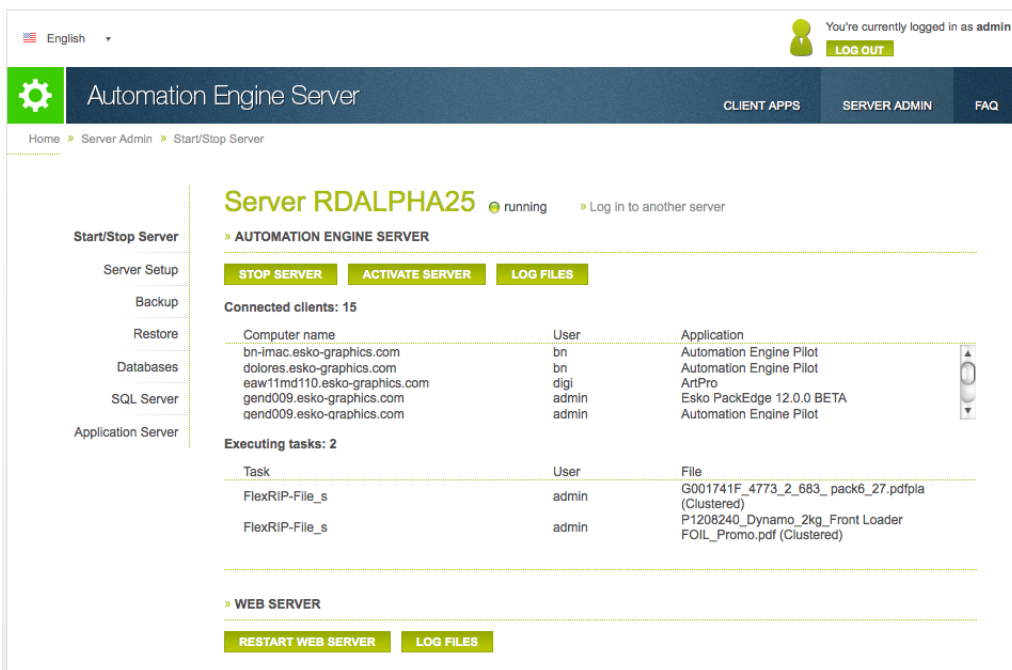
On this web server administration tool you will also find the client application(s) such as the Automation Engine Pilot, a diagnostics tool, a back-up tool, ...

To start the Automation Engine server:

1. Open your web browser and go to <http://servername> or <http://servername:9999/>.



2. Log in with your user name and password.
3. In the Server administration > Start/Stop Server window, you have access to all administration tasks for the Automation Engine server.



For more information on working with the server, see [Server Administration](#).

Installing Client Applications

The Automation Engine Client applications like the Pilot, Shuttle and Diagnostics are available in the **Client Apps** section of the Server Admin Web Tool. Access the Server Admin Web Tool via `http://servername` or `http://servername:9999/`.

Click the **Download** button next to **Client Applications**. This download includes the Pilot, Shuttle and Diagnostics.

A shortcut for Pilot and Shuttle will be created during installation.

Server EAW12DD320 ● running [» Log in to another server](#)

» MAIN APPLICATIONS

» **Client Applications** DOWNLOAD

This download includes the Pilot, Shuttle and Diagnostics.

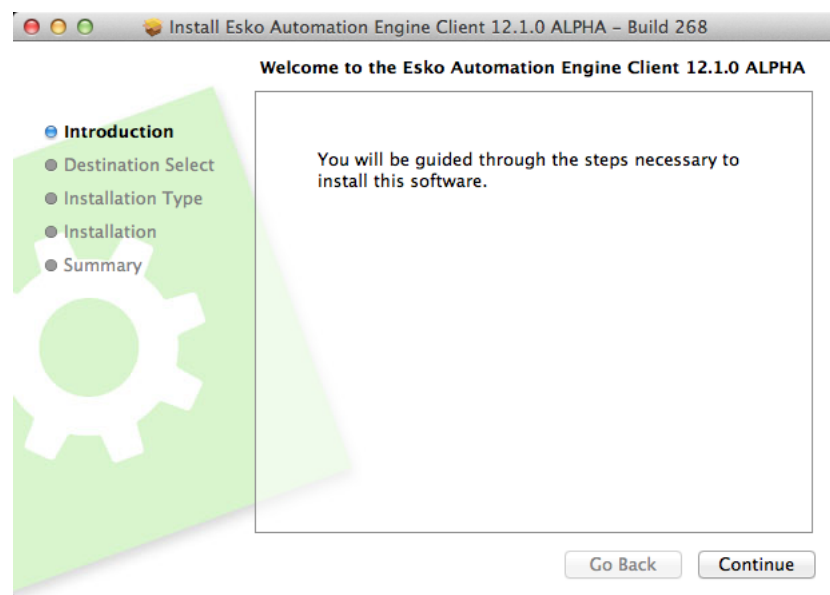
The **Pilot** is the primary client tool for your prepress workflow automation.

The **Shuttle** is a small client tool that enables you to easily submit files to Automation Engine workflows and monitor their progress.

The **Diagnostics** tool is used to monitor the status and activity of the Automation Engine Server.

Installing Client Applications on Mac

1. Go to the Client Applications page of the Server Admin Web Tool. Click the **Download** button next to **Client Applications**.



2. Open the downloaded installation file Automation Engine Client 12.1.dmg.
3. Follow the installation instructions.

Client applications will be available in the `Applications` folder.

Installing Client Applications on Windows

1. Go to the Client Applications page of the Server Admin Web Tool. Click the **Download** button next to **Client Applications**.

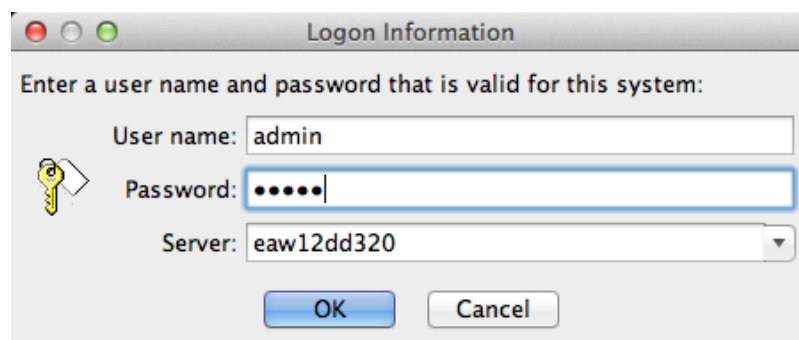
2. Open the downloaded installation file (**Automation Engine Client 12.1.exe**).
3. Follow the installation instructions.

Automation Engine Client applications will be available **Start > Esko > Automation Engine Client 12.1** .

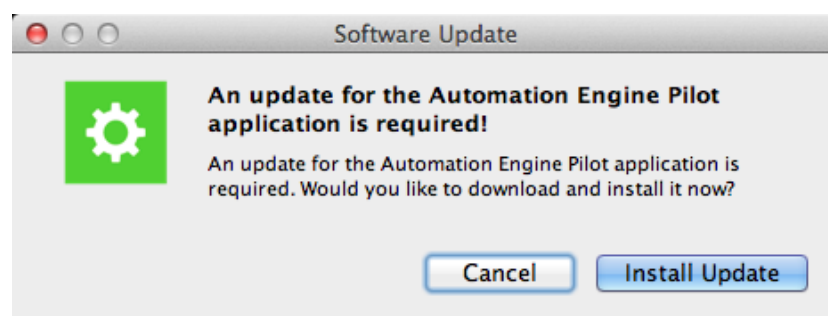
Updating the Automation Engine Pilot

The Pilot and the Automation Engine Shuttle display automatic update alerts. This makes sure that the latest version of these applications are available.

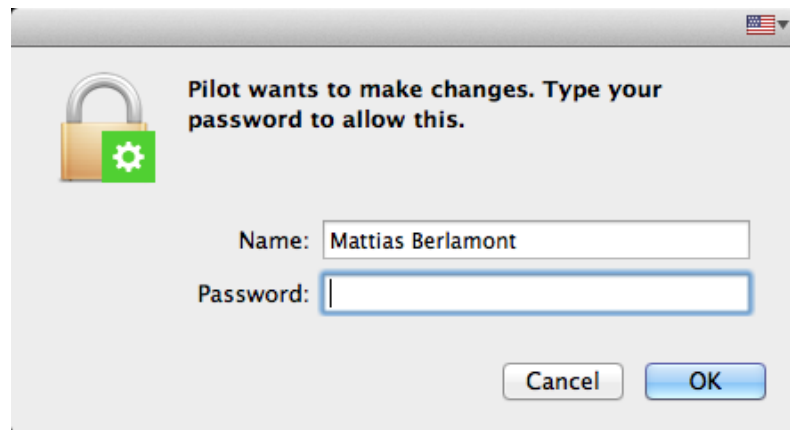
- While starting the Automation Engine Pilot for the first time, in the **Logon Information** window, enter **User name**, **Password** and the **Server** name. Click **OK**.



- If the version of the Pilot matches that of the Automation Engine Server, the main Pilot window will open.
- If there is a major version mismatch among the Pilot and the Automation Engine Server, you cannot login to the Automation Engine Server. You will be alerted with a warning.
- If there is a build mismatch among the Pilot and the Server, the **Software Update** dialog will pop up.



- Click **Install Update**. The Pilot will stop and the **Updating Automation Engine Pilot** dialog will appear.
On Mac, you need to enter the credentials of an administrator use to update the Pilot application.



The **Updating Automation Engine Pilot** dialog indicates the download progress.

- At the end of the download, this window displays `Ready to install` message. Click the **Install and Relaunch** button to install the update.

After the update is installed, the Pilot application will be relaunched and you will be asked to enter credentials in the **Logon Information** window again.

Note: You can use the same procedure to update Automation Engine Shuttle.

Starting the Pilot

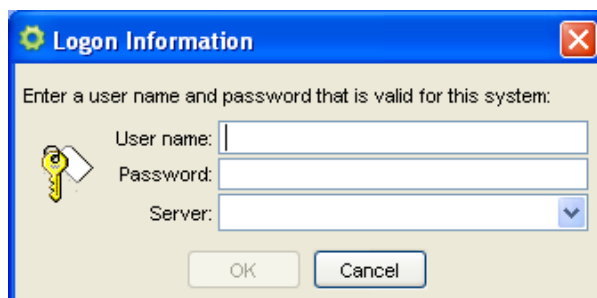
On your Automation Engine client machine:

1. Do one of the following:

-

double-click the Pilot desktop icon ,

- go to **Start > All Programs > Esko > Automation Engine Client 10.0 > Pilot**.
2. In the **Logon Information** dialog that opens:



- a) Enter a **User name** and **Password** valid for your Automation Engine server.

Note:

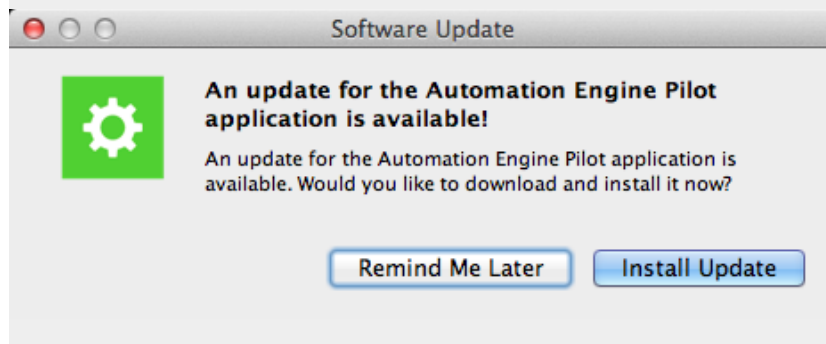
Before you define any user, you can log in using `admin` as both user name and password.

Once you have created users (see [Creating Users](#) on page 32), you can log in with one of your users' credentials.

- b) In **Server**, enter either your Automation Engine server's name or its IP address.
- c) Click **OK**.

Note:


If the build version of the Pilot is different from that of the Automation Engine Server or when an update for the Automation Engine Pilot is available, the **Software Update** dialog will pop up. To download and install the update of the Pilot, click **Install Update**. You can postpone it by clicking the **Remind Me Later** button.



This opens the Pilot's main window.

1.3.2 Performing Basic Configuration

Creating Users

1. In the **Pilot**, go to **Tools > Users**.
2. In the **Users** window, do one of the following:
 - go to **File > New User**,
 - use the **CTRL + N** shortcut,
 - click the  icon.

The **New User** dialog opens.

3. Enter the user's **Name**, **Full Name**, **Password**, **Description** and valid **E-mail address**.
4. If you want the user to be able to administer Automation Engine, check the toggle.
This will give all user access rights to that user.
5. Click **OK** to add your new user to the **Users** list.

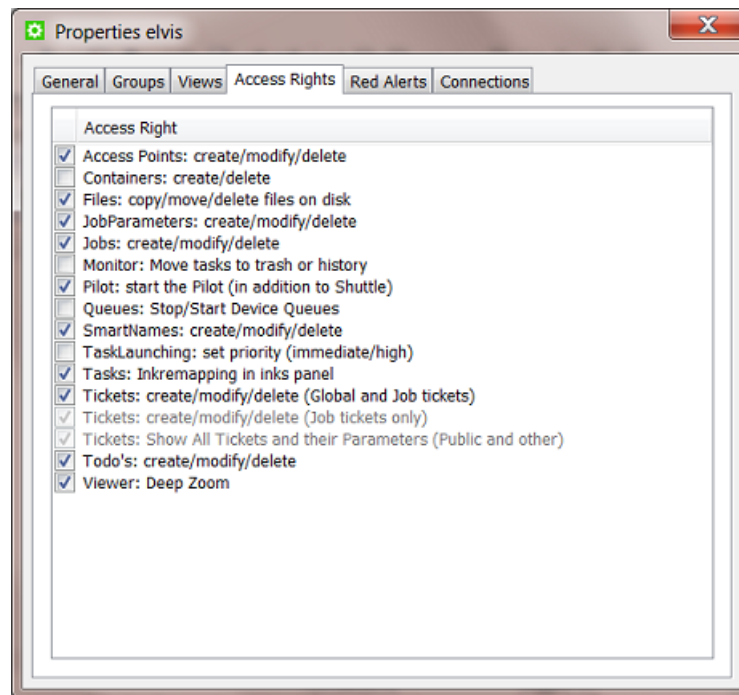
**Attention:**

When creating a user, Automation Engine requests permission from the Windows OS. Therefore, some changes (for example, changing the password) require the use of Windows User Administration tools. These changes must also meet the existing domain policies (for example, the length and complexity of the password).

Defining User Access Rights

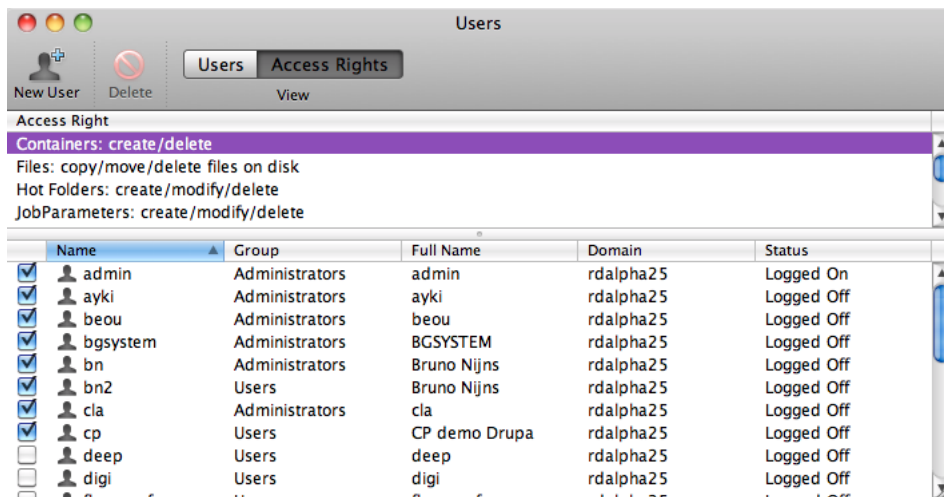
Not all users have the same responsibilities, so you can assign them different access rights in Automation Engine.

- To give access rights to a single user:
 - a) in the **Users** window, double the user you want to give access rights to,
 - b) in the **Access Rights** tab of the **Properties** window, check the access rights you want this user to have.



Note: Automation Engine Administrators automatically have all access rights. You cannot change this.

- To give a particular access right to several users:
 - a) in the **Users** window, click the **Access Rights** button to switch to the **Access Rights** view,
 - b) select the access right you want to give to your users,
 - c) check the users you want to give this access right to.



Configuring Output Devices

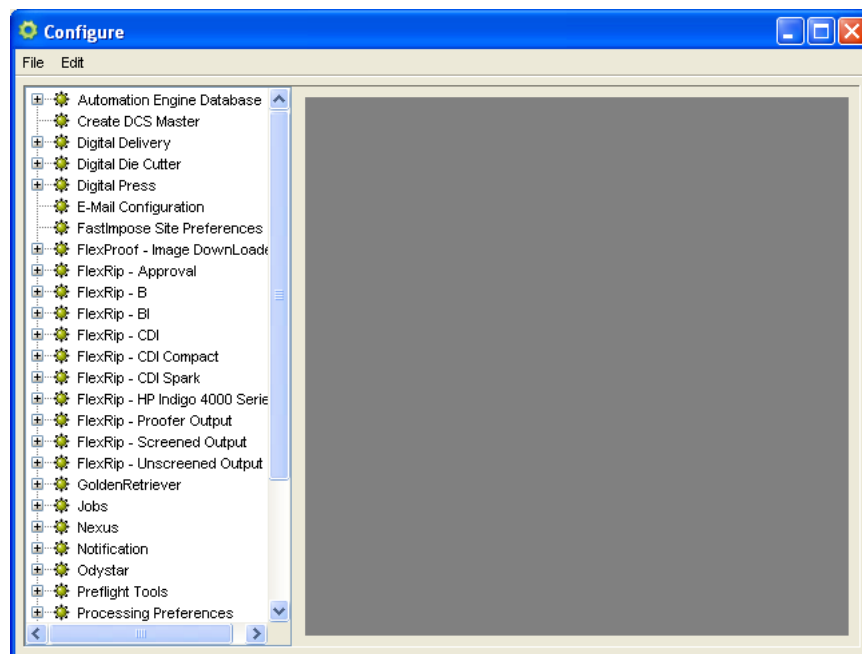
Automation Engine can output files to a variety of output devices (CDI, HP Indigo, Kodak Approval, etc.).

Automation Engine uses Esko's FlexRip engine to RIP the file in preparation for output.

To connect Automation Engine to an output device, you need to connect it to FlexRip.

1. In the Pilot, go to **Tools > Configure**.

This opens the **Configure** dialog.



2. Select the option corresponding to your output device in the list at left (for example, select **FlexRip - Proofer Output** to output to a proofer).

3. Go to **File > New** or use the **Insert** key (on Windows).

This shows tabs with parameters at right.

4. In the **General** tab:

- a) To output to a RIP, enter the name or IP address of your RIP server in **RIP Node**.

For a FlexProof output, enter the name / IP address of your proof server in **Proofer Node**.

- b) Enter the EskoLink port number used by the Dispatcher in **EskoLink Portnr.** (only for FlexRip outputs).

Note: You can find that port number by going to **Info > Ports** in the Dispatcher.

- c) Click the **Synchronize with Rip / Synchronize with ProofNode** button.

After a few seconds, your RIP/proof server is connected to Automation Engine, and you can see its name in the **Type** field.

5. In the **Queue** tab:

- a) Give your queue a name in the **Queue** field.

- b) Choose a **Queue Job Limit**:

- choose **1** to have the RIP handle one file at a time (this ensures the files are ripped in the order they enter the RIP),
- choose **2** to have the RIP handle two files at once (the files may not be ripped in the order they enter the RIP if for example you send a big then a small file).

6. Go to **File > Save** to save the configuration.

You can now see the devices you are connected to in the Pilot's **Devices** view.

Views	Name	Type	Rip Node	Port	Dispatcher State	Queue Name	Queue State
Tools	Proofer Output1	FlexRipProof -> Generic Proofer output	gend035	3060	Running	FlexProofE	Started

1.3.3 Uploading Files using the Pilot

In the Pilot, in the **Container** mode or in the **Last used Job** mode, select the folder in which you want to upload your file(s).

Choose one of these methods:

- drag and drop files or folders from outside the Pilot onto a folder in the Pilot. Or use copy/paste shortcuts.

- go to **File > Upload...** . In the browser dialog that opens, browse to your file and click **Upload**.

1.3.4 Launching a Task on a File

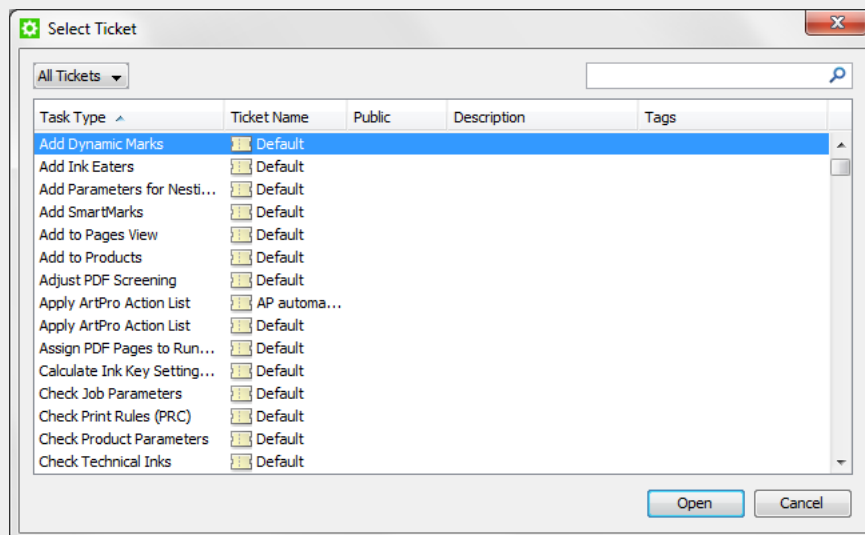
There are several ways to launch a task on a file (from the Pilot, from Shuttle, via Access Points or via JDF/JMF). Here is how to launch a task from the Pilot's **Files** View.

1. In the Pilot, select the **Files** view from the **Views** bar (or use **Go > Files**).
2. In the **Containers** mode, browse to your chosen file.
3. Right-click it, select **New Task** then **More Tickets...**

Note: If you have launched tasks previously, the most recent task tickets you used are listed above **More Tickets...**

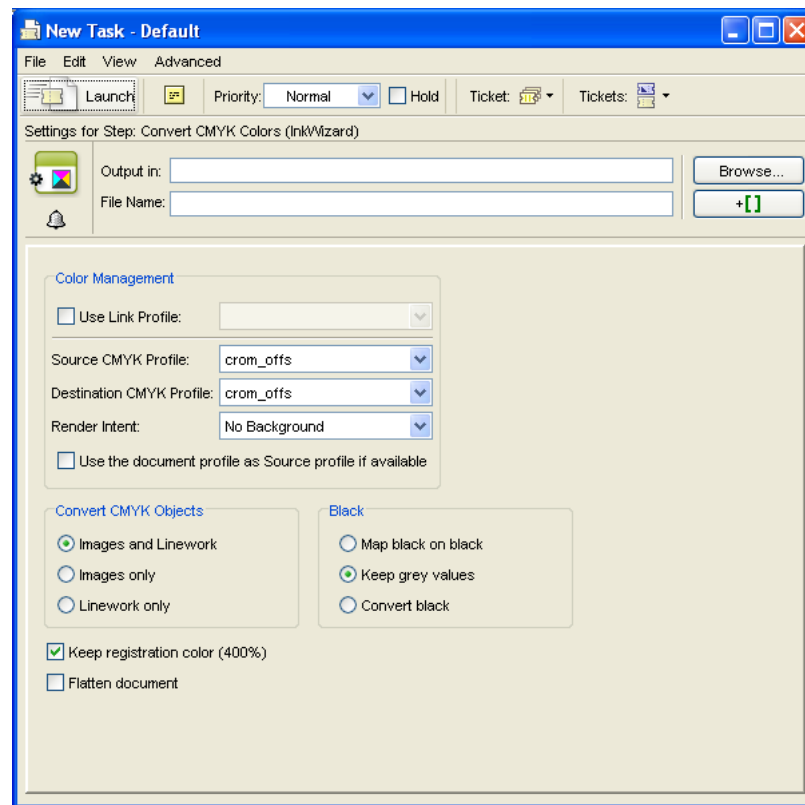
4. In the **Select Ticket** dialog that opens, select the task ticket you want to use.

Note: This dialog contains all the tickets suitable for the file type you selected. You can choose either a **Default** ticket, or a custom ticket (saved under a different name) if you have already created one.



5. Click **OK**.

This opens the task's ticket. An example:



6. Fill in the settings you want to use in the ticket.

Note: For a description of all of the ticket's settings for each task, see [The Tasks and Workflow Controls](#) on page 239 in the Reference Guide.

7. Click the **Launch** button.

You can monitor the tasks progress in the **Files View's Task** pane or in the **Tasks View**. See [Checking the Status of the Task](#) on page 37.

1.3.5 Checking the Status of the Task

After launching a task on one or several file(s), you can check its processing status in the Pilot's **Tasks** pane. You can see the **Tasks** pane in the **Files**, **Product** or **Tasks** views.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
07_RussianText_OTF.pdf		Copy or Move File	100%		✓	05/01/10 18:24


You can see the progress (in percentage) and the processing state. The state can be:

- ▶: starting
- ✓: success
- ⚠: warning
- ✗: failure
- ⏸: processing cancelled

You can also double-click the task entry in the **Tasks** pane to open your task ticket and check its parameters.

Note: Depending on your [access rights](#), you may only see the public parameters.

1.3.6 Cancelling a Launched Task

You can cancel a launched task by right-clicking on the task and choosing **Cancel**. When it was successfully cancelled, the task will show the cancelled state with this icon: 

Note: Tasks that are already too far in their processing or are no longer under control of Automation Engine (for example FlexRip tasks) can no longer be cancelled

1.3.7 Relaunching the Task

- To relaunch the task without changing the settings:
 - a) Select the task to relaunch in the **Tasks** pane.
 - b) Go to **Tasks > Relaunch**.
- To change the settings and relaunch the task:
 - a) Select the task to relaunch in the **Tasks** pane.
 - b) Go to **Tasks > Open**.
 - c) Change the task settings as necessary.
 - d) Click the **Launch** button.

1.3.8 Generating Output

To output a file, use a task related to the output device you have configured (see [Configuring Output Devices](#) on page 34).

For example, use the **Proof (FlexProof)** task if you have configured a proofing device using **FlexRip - Proofer Output**.

Note: The task related to your output device will only be available once you have correctly configured your output device.

1.4 Server Administration

Automation Engine features a new web tool for server administration. This means that you can access your server administration tool from any workstation via `http://servername` or `http://servername:9999/`.

You can use this web tool to download the Automation Engine Client apps such as

- the Pilot
- the Script runner

- the ArtPro Action List Editor
- the Automation Engine 10 Backup Tool

The Server Admin Web Tool also includes a Server Admin section which can be used to administer the Automation Engine Server and its settings, create and restore backups and manage databases, SQL Servers and the Application Server.

Server administration consists of :

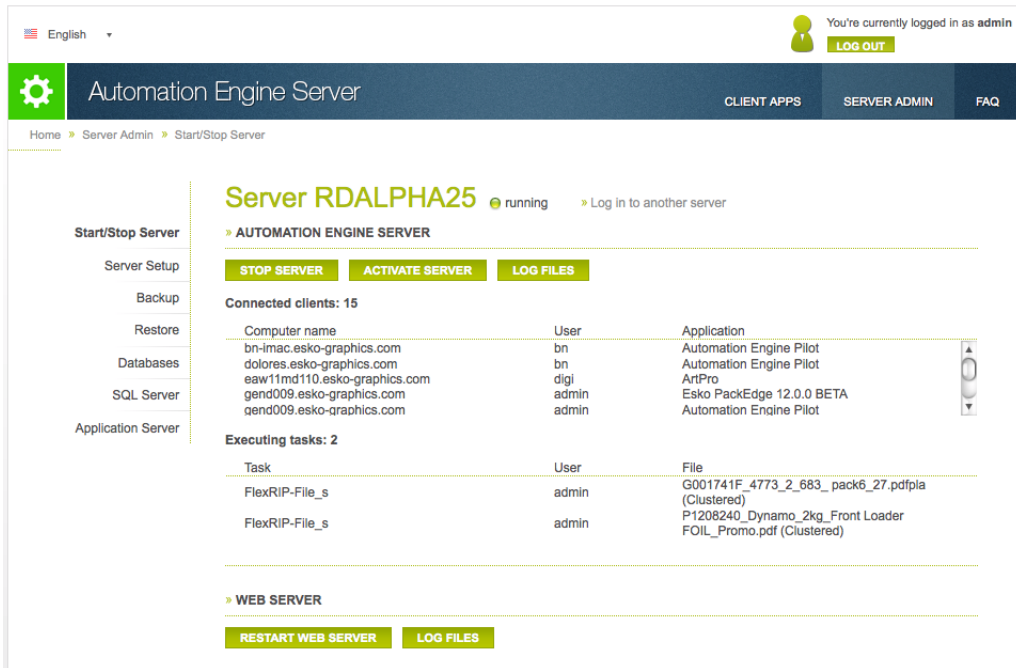
- [Status Overview of the Server](#)
- [Setting up the server](#)
- [Backing up the server](#)
- [Restoring the server](#)
- [Managing databases](#)
- [Managing SQL servers](#)

Read detailed information in Automation Engine Reference Guide.

1.4.1 Status Overview of the server

The **Status Overview** shows an overview of the connected clients and the number of running tasks. You also get an overview of the **Server Version**, **Current role** and **Server databases**. In the **Status Overview** you can:

- Start and stop the Automation Engine server.
- Activate the server after install.
- Restart the web server.
- Download the server administration log files for support purposes.
- Start, stop or activate the local application server.
- Download Application Server log files for support purposes.
- Deploy components on the application server.
- Log in to another server.



Automation Engine Server

Home > Server Admin > Start/Stop Server

Server RDALPHA25 running [Log in to another server](#)

AUTOMATION ENGINE SERVER

[STOP SERVER](#) [ACTIVATE SERVER](#) [LOG FILES](#)

Connected clients: 15

Computer name	User	Application
bn-Imac.esko-graphics.com	bn	Automation Engine Pilot
dolores.esko-graphics.com	bn	Automation Engine Pilot
eaw11md110.esko-graphics.com	digi	ArtPro
gend009.esko-graphics.com	admin	Esko PackEdge 12.0.0 BETA
gend009.esko-graphics.com	admin	Automation Engine Pilot

Executing tasks: 2

Task	User	File
FlexRIP-File_s	admin	G001741F_4773_2_683_pack6_27.pdfpla (Clustered)
FlexRIP-File_s	admin	P1208240_Dynamo_2kg_Front Loader FOIL_Promo.pdf (Clustered)

WEB SERVER

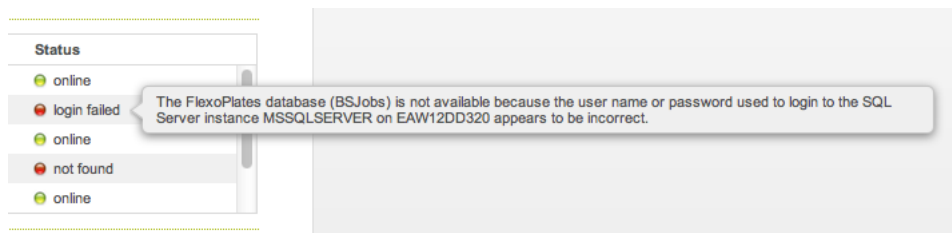
[RESTART WEB SERVER](#) [LOG FILES](#)

The **Status Overview** page contains more information from 12.1.1 onwards. Read the following.

For each **Server database**, the **SQL Server Instance** and the **SQL Server** location is indicated at the **Status Overview** page. The **Status** column indicates the following statuses :

- **Online** : The SQL Server instance where the server database is located is running. This also implies that the user name and password used to login to the SQL Server instance are correct. The server database is online and available for access.
- **Not Accessible**: The SQL Server instance where the server database is located is running. Valid credentials(user name and password) are used. But the server database on the SQL Server instance is not accessible.
- **Not Found** : The SQL Server instance where the server database should be located is running and the credentials used to login to the SQL Server instance are valid. But the server database could not be found on the SQL Server instance.
- **Login Failed**:: The SQL Server instance where the server database should be located is running. The server database is not available because the credentials used to login to the SQL Server instance are invalid.
- **Offline**: The server database is not available because the SQL Server instance where it is located is currently not running.
- **Unknown**: The status of the server database on the SQL Server instance is unknown.

Tip: When you move your cursor over the status of a server database, a pop-over message will indicate the detailed status information.



Status

- online
- login failed
- online
- not found
- online

The FlexoPlates database (BSJobs) is not available because the user name or password used to login to the SQL Server instance MSSQLSERVER on EAW12DD320 appears to be incorrect.

1.4.2 Setting up the server

In the Server Setup window you can change the server role: master, assistant or idle. You can also:

- View and remove Assistant Servers.
- Change the system account and/or password.
- Change the processing capabilities.
- Redirect central resources to a central file server.

Server RDVMAEDEMO ● running ⌂ [Log in to another server](#)

» SERVER SETUP

Server version 14.0.0 - build 421

Current role

Assistant servers 0

System account

User name

Password

Processing capabilities

- Can run Tasks
- Can run File Streaming Sessions
- Max. sessions
- Can run Gang Run Printing Sessions
- Max. sessions

Central resource server

Click the lock to make changes.

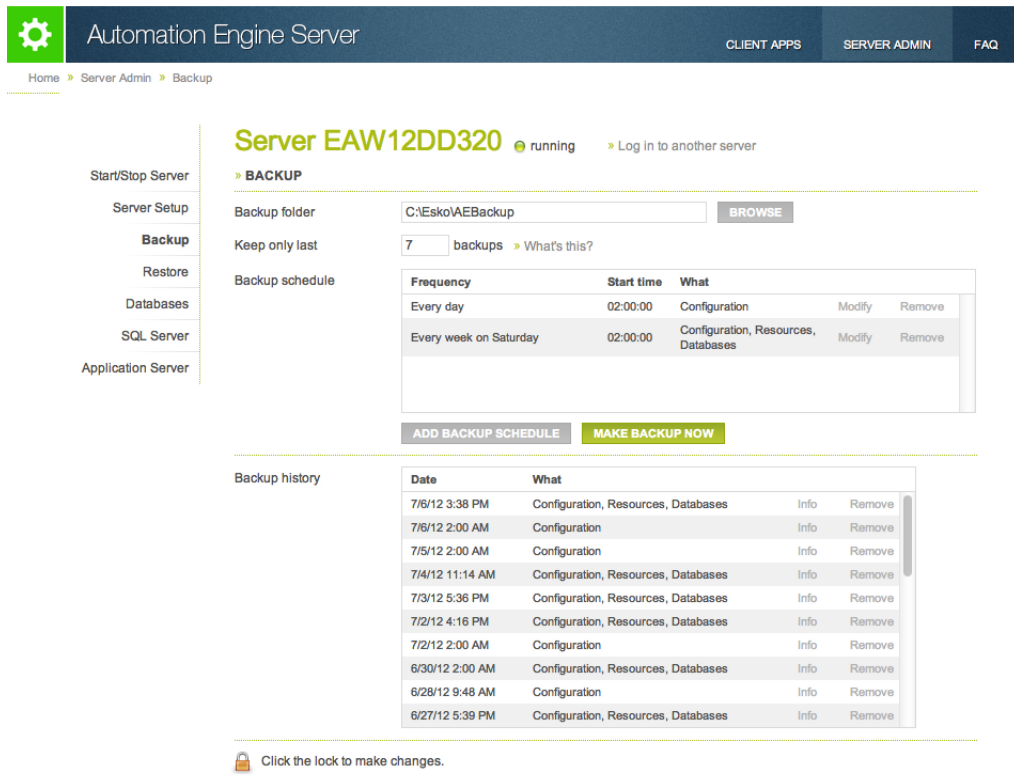
1.4.3 Backing up the server

In the Server Backup window you can manage the following backup settings:

- Specify a **backup folder** on the server.
- Specify the number of backups to be kept in **Keep only last**.
- Set a **backup schedule** or manage existing schedules.
- See the **Backup history** and details of each backup.

Tip: Click on **info** to get more information about the selected backup. A **Backup info** window will list the server, date, time, size and composition of the backup.

- **Remove** a backup.



The screenshot shows the 'Automation Engine Server' interface. The top navigation bar includes 'CLIENT APPS', 'SERVER ADMIN', and 'FAQ'. The breadcrumb trail is 'Home > Server Admin > Backup'. The main content area is titled 'Server EAW12DD320' and shows it is 'running'. A sidebar on the left lists navigation options: 'Start/Stop Server', 'Server Setup', 'Backup', 'Restore', 'Databases', 'SQL Server', and 'Application Server'. The 'Backup' section is active, showing configuration for 'Backup folder' (C:\Esko\AEBBackup), 'Keep only last' (7 backups), and 'Backup schedule'. The schedule table lists two entries: 'Every day' at 02:00:00 and 'Every week on Saturday' at 02:00:00. Below the schedule are buttons for 'ADD BACKUP SCHEDULE' and 'MAKE BACKUP NOW'. A 'Backup history' table shows a list of backup events with columns for 'Date', 'What', 'Info', and 'Remove'. At the bottom, there is a lock icon and the text 'Click the lock to make changes.'

Server EAW12DD320 running [Log in to another server](#)

Backup configuration:

- Backup folder: C:\Esko\AEBBackup [BROWSE](#)
- Keep only last: 7 backups [What's this?](#)


Backup schedule:

Frequency	Start time	What	Modify	Remove
Every day	02:00:00	Configuration	Modify	Remove
Every week on Saturday	02:00:00	Configuration, Resources, Databases	Modify	Remove

[ADD BACKUP SCHEDULE](#) [MAKE BACKUP NOW](#)

Backup history:

Date	What	Info	Remove
7/6/12 3:38 PM	Configuration, Resources, Databases	Info	Remove
7/6/12 2:00 AM	Configuration	Info	Remove
7/5/12 2:00 AM	Configuration	Info	Remove
7/4/12 11:14 AM	Configuration, Resources, Databases	Info	Remove
7/3/12 5:36 PM	Configuration, Resources, Databases	Info	Remove
7/2/12 4:16 PM	Configuration, Resources, Databases	Info	Remove
7/2/12 2:00 AM	Configuration	Info	Remove
6/30/12 2:00 AM	Configuration, Resources, Databases	Info	Remove
6/28/12 9:48 AM	Configuration	Info	Remove
6/27/12 5:39 PM	Configuration, Resources, Databases	Info	Remove

 Click the lock to make changes.

1.4.4 Restoring the server

In the Restore window you can view and restore backups. You can also:

- Restore, duplicate or migrate your server configuration
- Restore resources or databases only

Note: Take into account that the server should be stopped before you can restore a backup.

The screenshot shows the Automation Engine Server interface. At the top, there is a language dropdown set to 'English' and a user profile for 'admin' with a 'LOG OUT' button. The main header includes the 'Automation Engine Server' logo and navigation tabs for 'CLIENT APPS', 'SERVER ADMIN', and 'FAQ'. The breadcrumb trail is 'Home > Server Admin > Restore'. The left sidebar contains a tree view with categories: Start/Stop Server, Server Setup, Backup, Restore, Databases, SQL Server, and Application Server. The main content area is titled 'Server RDALPHA25' and shows it is 'running'. Below this is a 'RESTORE' section with a 'Backup history' table. The table has columns for 'Date', 'What', and 'Restore'. Below the table is a 'SELECT BACKUP FOLDER...' button. At the bottom, there is a lock icon and a message: 'Click the lock to make changes. The Automation Engine Server should be stopped to restore a backup.'

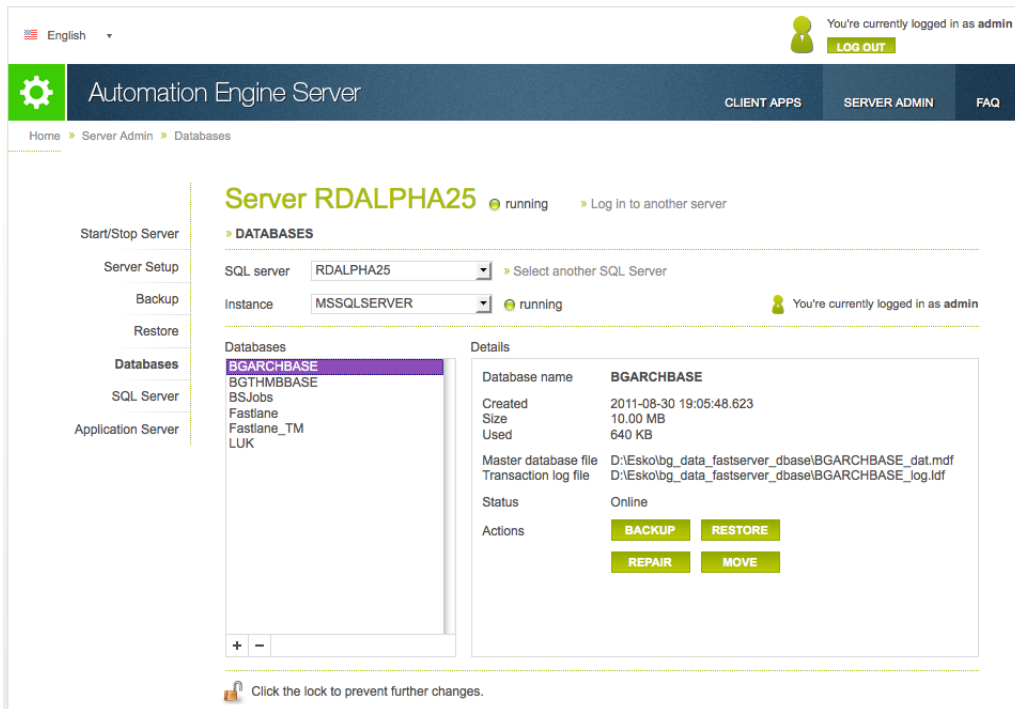
Date	What	Restore
5/24/12 4:51 PM	Configuration, Databases, Resources	Restore
4/5/12 3:57 PM	Configuration, Databases, Resources	Restore
3/16/12 12:47 PM	Configuration, Databases, Resources	Restore
3/15/12 10:37 AM	Configuration, Databases, Resources	Restore
3/15/12 10:35 AM	Configuration, Databases, Resources	Restore
2/17/12 12:51 PM	Configuration, Databases, Resources	Restore

1.4.5 Managing databases

In the **Databases** window you can manage all databases that are on any accessible SQL server in your network. Depending on the permissions on the selected SQL instance, you can:

- View the SQL server status
- Login and view the properties and status for each database
- Create or remove databases
- Back-up or restore databases
- Repair or move databases

Note: Take into account that, even if you are logged in to the Server Administration, you also need to log in to the SQL Server instance separately.



The screenshot shows the 'Automation Engine Server' interface. The main heading is 'Server RDALPHA25' with a status of 'running'. Below this, there's a 'DATABASIS' section. On the left, a sidebar lists various server management options: Start/Stop Server, Server Setup, Backup, Restore, Databases (selected), SQL Server, and Application Server. The 'Databases' section shows a list of databases: BGARCHBASE (highlighted), BGTHMBBASE, BSJobs, Fastlane, Fastlane_TM, and LUK. To the right, the 'Details' for the selected 'BGARCHBASE' database are shown:

Database name	BGARCHBASE
Created	2011-08-30 19:05:48.623
Size	10.00 MB
Used	640 KB
Master database file	D:\Eskolbg_data_fastserver_dbase\BGARCHBASE_dat.mdf
Transaction log file	D:\Eskolbg_data_fastserver_dbase\BGARCHBASE_log.ldf
Status	Online
Actions	<input type="button" value="BACKUP"/> <input type="button" value="RESTORE"/> <input type="button" value="REPAIR"/> <input type="button" value="MOVE"/>

At the bottom of the details section, there is a lock icon and the text: 'Click the lock to prevent further changes.'

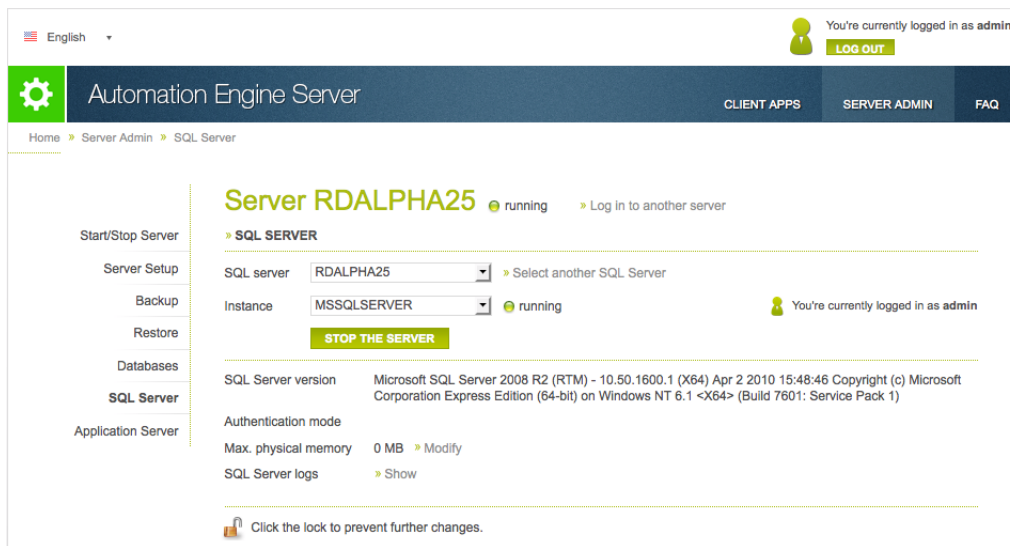
In this page, you can manage the databases on any SQL Server in the network.

1.4.6 Managing SQL servers

In the **SQL Server** window you can manage any accessible SQL server in your network. You can:

- View the SQL server status
- Stop and restart an SQL server
- Login and change the properties: modify the maximum physical memory of the SQL server
- Look at the SQL server logs

Note: Take into account that, even if you are logged in to the Server Administration, you also need to log in to the SQL Server instance separately.



1.5 Uploading Files to Automation Engine

You can work with files that are already stored in your **Containers**, or upload new files to Automation Engine. You can upload files onto the **Pilot**, or by using the **Shuttle** application and plug-in. **Access Points** can also upload data but they will also already trigger a workflow task.

1.5.1 Uploading Files using the Pilot

In the Pilot, in the **Container** mode or in the **Last used Job** mode, select the folder in which you want to upload your file(s).

Choose one of these methods:

- drag and drop files or folders from outside the Pilot onto a folder in the Pilot. Or use copy/paste shortcuts.
- go to **File > Upload...** . In the browser dialog that opens, browse to your file and click **Upload**.

1.5.2 Uploading Files from a FTP Server

1. In **Containers** mode or when in a **Job**, right-click the folder in which you want to upload your file, and select **Upload from FTP Site...**

This opens the **Upload from FTP Site** dialog, where you can enter the details necessary for the FTP connection.

2. In **Host**, enter the name of the FTP server that hosts your files.

Tip: To use a port other than the default port, add **:** and the port number after the host name. For example: `myftpserver:1085`

3. Enter the **User Name** and **Password** you use to connect to the FTP server.

Make sure you use a valid **User Name** and **Password** corresponding to an account that exists on the FTP server.

4. In **File Transfer**, choose between **Binary** and **ASCII**.

Binary will always work, but if you have an ASCII file, the transfer will go a bit faster when selecting **ASCII**.

5. By default, the transfer uses Active **Connection Mode** (where the FTP server opens the data connection).

If your system is behind a firewall that blocks incoming FTP server connections, select **Passive** to use Passive Connection Mode (where the client initiates the connection).

6. Select **Secure FTP** if you are connecting to a secure FTP server (over FTPS, not SSL or SSH).

Tip: If you are experiencing problems using secure FTP, try connecting over regular FTP, and ask your IT administrator to check the security settings of your FTP server.

7. If desired, you can delete the files from the FTP site once they are uploaded to the Pilot (select **Remove remote data after transfer**).

If you do this, you can choose to **Backup remote data before removal**. Select the **Backup Folder** in which to copy the files.

8. When you are done, click **Connect**.

Upload from FTP Site

Connect To FTP Server

Host: ftp.belgianbrew.com

User Name: johnndoe

Password: ●●●●●●

File Transfer: Binary

Connection Mode: Passive
 Secure FTP

Remove remote data after transfer.
 Backup remote data before removal.

Backup Folder:

1.5.3 Using Access Points to Upload Files

Access Points are documented in the dedicated chapter [Integrating with External Systems](#) on page 1228.

1.5.4 Using Shuttle to Upload and Process Files

You can upload and process files from any Mac or Windows client computer using the Shuttle application.

The Automation Engine Shuttle

The Automation Engine Shuttle is a client tool for easy submission of files to Automation Engine workflows. Shuttle also allows monitoring of the workflows.

Download Shuttle from the Main section of the Server Admin Web Tool. You can access it by browsing to `http://<name of your server>` or `http://<name of your server>:9999`. Read more in [Installing Client Applications](#) on page 28

If the Shuttle application is installed on your computer, the system will check if you have the appropriate version and download updates if necessary.

As explained in [Shuttle](#) on page 681, you first need to configure your Shuttle in **Tools > Configure > Shuttle**.

What is Shuttle?

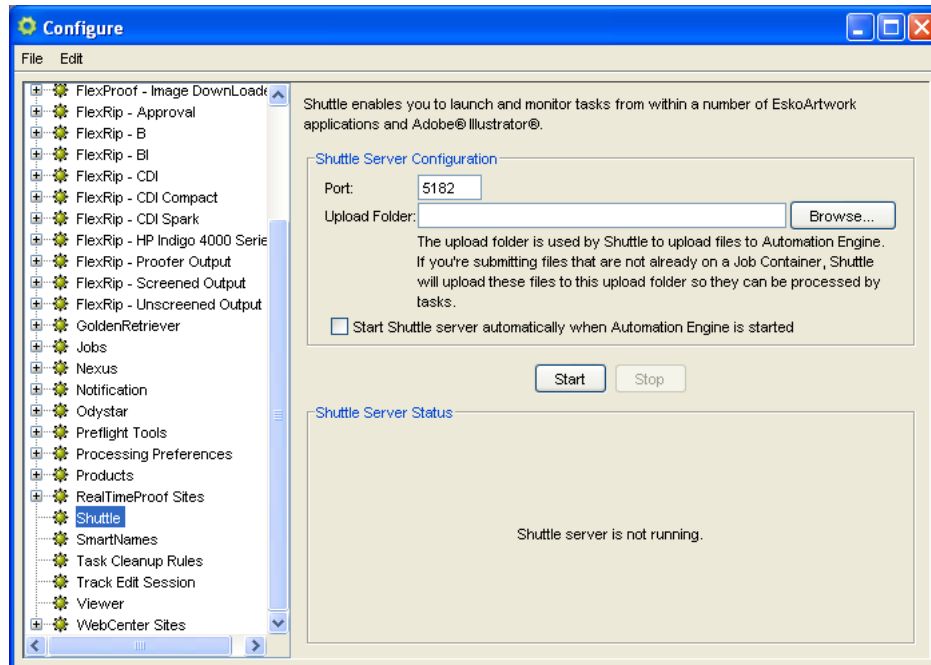
Shuttle is a small stand-alone application that enables you to submit files to Automation Engine tasks or workflows, Odystar canvases, and Nexus workflows, and monitor their progress. You can use Shuttle to connect to several servers.

The Shuttle functionality is also integrated in ArtPro, Neo, PackEdge, Plato, FastImpose and in Adobe Illustrator as a DeskPack plug-in.

Shuttle Setup

Automation Engine Setup for Shuttle

1. In the **Pilot**, go to **Tools > Configure**.
2. Select **Shuttle** in the left pane.



3. Enter the **Port** to use for communication between your Automation Engine server and the Shuttle clients.

Note: By default this is port 5182, but if this port is already used by other processes, you should enter the next available port.

4. Choose an **Upload Folder** using the **Browse...** button.

This is where Shuttle clients will upload files that are not located in a container (except the Shuttle plug-in, that will always upload files to the **DeskPackContainer**).

Note: The **Upload Folder** must be located on a container.

5. Select **Start Shuttle server automatically when Automation Engine is started** if you plan to make frequent use of the Shuttle technology.
6. Click the **Start** button to start Shuttle server.

Shuttle server makes Automation Engine's workflows, tasks and files accessible to the Shuttle clients.

7. Save your settings.

Note: If you need to make changes to this Setup, you must **Stop** the Shuttle server first.

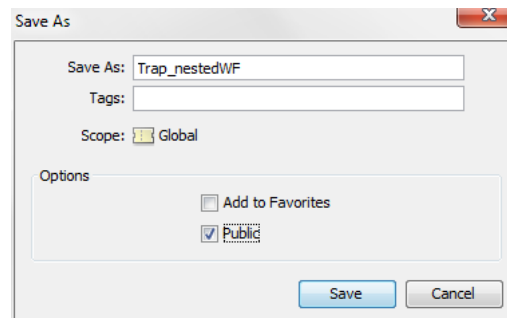
Making Your Ticket Public

Important: Only Public tickets or workflows can be launched from the client applications mentioned [here](#).

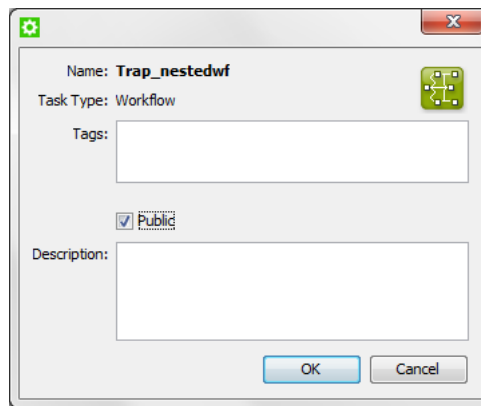
Note: Default tickets can not be made Public.

To make a ticket **Public**, you can either:

- Select the **Public** option when creating the ticket.



- Select your ticket in the **Ticket** view, click  and select **Public**.



- Right-click your ticket in the **Tickets** view and select **Make Public**.

Note: Public tickets have a mark in the **Public** column.


Task Type	Ticket Name	Public	Description
Zip	Default		
Workflow	WVC_Approval		
Workflow	WaitForRunListFiles		
Workflow	TrapWF		
Workflow	Trap_nestedWF	✓	
Workflow	SimpleTrap		
Workflow	Send2Ody		

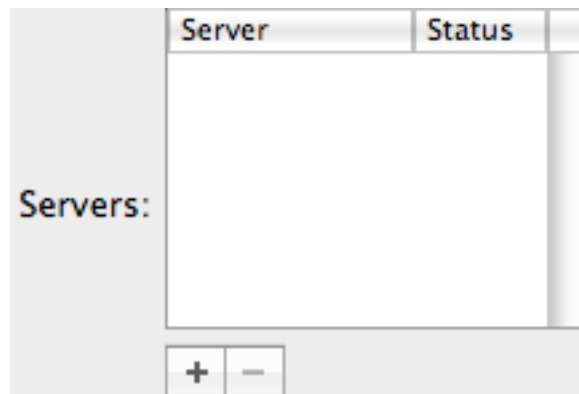
Shuttle Clients Connecting to Any Server

The Automation Engine Shuttle, the Shuttle plug-in, and the Shuttle clients in ArtPro, Neo and FastImpose Standalone can connect to a number of Odystar, Nexus and Automation Engine servers.

1. Launch your application.
2. Open the **Preferences**.

In...	go to...
Illustrator	<ul style="list-style-type: none"> • Illustrator > Preferences > Esko > Shuttle Preferences... on Mac • Edit > Preferences > Esko > Shuttle Preferences... on Windows
ArtPro	ArtPro > Preferences... or use Command + K
Neo	Neo > Preferences... or use Command + , (comma)
FastImpose Standalone	Tools > Shuttle Preferences...

3. In the **Servers** tab (**Advanced** tab for Neo), click the  button at the bottom of the **Servers** list.



The following dialog will prompt.

4. Enter your **User Name and **Password**.**

- When connecting to an Automation Engine server, enter the **User Name** and **Password** you use to log in to the Pilot (the password can be blank if your server is configured to accept this).
You will have the same access rights (for example, the right to set processing priorities or not) in Shuttle as you have in the Pilot.

Note: If you don't know which **User Name** and **Password** to use, contact your system administrator.


- 5. To connect to a server *not* broadcasting over the Bonjour network (Automation Engine servers or non-Bonjour-enabled Nexus servers):**
- a) Select **Manual** in the **Browse Method** list
 - b) Enter your server's name or IP address in **Server Address**
 - c) Fill in the **Server Port** used by your server

Note:

When connecting to an Automation Engine server, enter the port you defined in Configure (see [Automation Engine Setup for Shuttle](#) on page 48).

- 6. Click the **Add** button.**

This adds a connection to your server in the **Servers** list.

- To remove the connection to a server, select it in the **Servers** list and click the  button.
- To check or edit a server's settings, double-click it in the **Servers** list.

Note:

If you are using the Shuttle plug-in with other Deskpack plug-ins, and you have set up a server connection with the **Server Connection Assistant**, you will see this connection in bold in the Shuttle plug-in's **Servers** list (after restarting Illustrator).

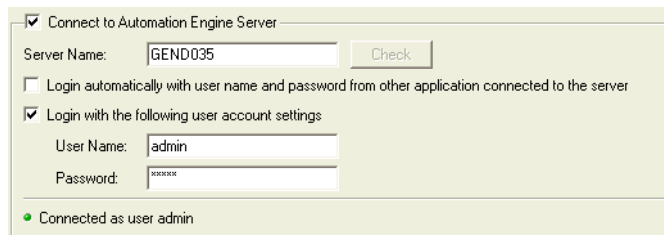
You will be able to change the user name and password if necessary, but you will not be able to remove this connection from the Shuttle plug-in.

See the Adobe Illustrator Client documentation for more information on the Server Connection Assistant.

Shuttle Clients Connecting only to Automation Engine

The Shuttle clients in PackEdge, Plato and FastImpose Server can only connect to one Automation Engine server.

1. Launch your application.
2. Open the **Preferences**.
 - In PackEdge or Plato, go to **Edit > Preferences** or use **Ctrl+Alt+Shift+P**.
 - In FastImpose Server, go to **Tools > Options...**
3. In the **Server&Resources** tab (the **Server** tab for FastImpose Server):
 - a) Select **Connect to Automation Engine Server**.
 - b) Enter the Automation Engine **Server Name** and click **Check**.
You should see a message saying the server is up and running.
 - c) Select **Login with the following user account settings**, and fill in the **User Name** and **Password** you use to login to the Pilot.
If the connection is successful, you will see a green dot and the message **Connected as user ...**



Tip:

If you are running an Automation Engine Pilot on the same computer, or another Shuttle client connecting to it (either PackEdge, Plato or FastImpose Server), you can select **Login automatically with user name and password from other application connected to the server.**

This way, if you are already logged in to the Pilot or the other client application, you won't need to log in when starting PackEdge / Plato / FastImpose Server.

- 4. Restart your application.

Launching the Automation Engine Shuttle

When you launch the Shuttle application from your installed programs, a pop up window with the Shuttle Launch panel and a Task Monitor will appear.

The screenshot displays the Automation Engine Shuttle interface. At the top, a window titled "Repeat and Export to Normalized PDF File" contains a task monitor table. Below the table is a workflow selection dropdown, a "Drop Files Here" area with a downward arrow icon, and a "Launch..." button.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
OmNom-eat.ai	mabm	Run Script	100%		✗	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✓	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	11/6/12 1:45 PM
OmNom.ai	mabm	Wait for Actio...	0%	User intervention needed	▶	10/30/12 5:15 PM
OmNom.ai	mabm	Workflow	0%	Running	▶	10/30/12 5:15 PM
Screen Shot 2012-09...		Wait for Actio...	0%	User intervention needed	▶	10/30/12 5:14 PM
Screen Shot 2012-0...		Workflow	0%	Running	▶	10/30/12 5:14 PM
arrow.png		Workflow	100%		✗	10/30/12 10:56 AM
arrow.png		Workflow	100%		✗	10/30/12 10:56 AM
arrow.png		Workflow	100%		✗	10/30/12 10:56 AM
arrow.png		Workflow	100%		✗	10/30/12 10:56 AM
Screen Shot 2012-09...		Wait for Actio...	0%	User intervention needed	▶	10/30/12 9:16 AM
Screen Shot 2012-0...		Workflow	0%	Running	▶	10/30/12 9:16 AM
Screen Shot 2012-09...		Workflow	100%		✗	10/30/12 9:16 AM
Screen Shot 2012-0...		Workflow	100%		✗	10/30/12 9:16 AM
OmNom-eat.ai	mabm	Run Script	100%		✗	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✓	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Run Script	100%		✗	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✓	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✗	10/22/12 12:27 PM

Workflow: Repeat and Export to Normalized PDF File

Drop Files Here

Launch...

- To open a new window, go to **File > New Window** . You can use dedicated Shuttle windows for each of your Workflows.
- Shuttle saves the settings of the open windows while closing. These settings are restored during the next launch.
- You can use **View** menu to hide or show the The Task Monitor, Shuttle Launch panel and the Status Bar.
 - **View > Tasks** hides/shows the Task monitor.
 - **View > Launch Panel** hides/shows the Launch Panel.
 - **View > Status Bar** hides/shows the Status Bar indicating the progress.

Working with the Automation Engine Shuttle

You must login to the Automation Engine Server to use the Shuttle.

Note: You have access to Shuttle if you have access to Pilot. But your access to Pilot may be restricted even when you are allowed to use Shuttle. This requires a user **access right (Pilot: start the Pilot (in addition to Shuttle))**. Read more in [Access Rights](#) on page 702.

The following menus are available in the Shuttle application on Mac and Windows:

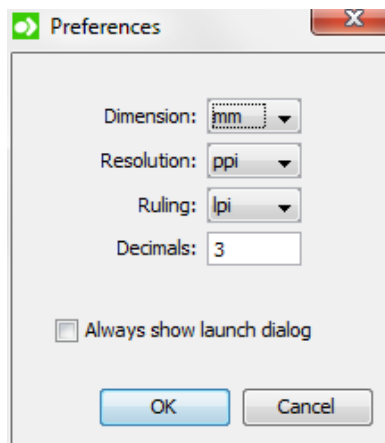
- **File** menu: Submit one or more files and launch the selected Workflow, open new Shuttle windows and close Shuttle windows.
- **Edit** menu: cut, copy, paste, delete a Task or Workflow, select all Tasks and Workflows in the Task Monitor.
- **View** menu: show or hide the Task Monitor, the Shuttle Launch panel and the Status Bar.
- **Task** menu: open or show the output file of the selected Task or Workflow, release or cancel the Task or Workflow, add annotations and show detailed information.
- **Window** menu: minimize, zoom and switch between Shuttle windows.
- **Help** menu: See the **About** window or open the Online Help (Esko Help Center).

Setting the Preferences for Shuttle

- On Mac, select **Preferences...** in the **Application** menu.
- On Windows, go to **Edit > Preferences...**

In the **Preferences** window, set the units for **Dimension, Resolution, Ruling** and the number of **Decimals** to use.

You can also select the **Always show launch dialog** option. This will show the **Public Parameter** dialog upon starting a new task, even if there aren't any public parameters. This allows you to alter these types of tasks before they launch.



The Task Monitor

When you are logged in, **The Task Monitor** displays your Automation Engine Workflows launched from both the Shuttle and the Pilot.

To filter the displayed Workflows, you can choose a job context filter.

- Go to the drop-down list in the toolbar and select the desired job. All recently used jobs are listed in this list.
- If the desired job is not present in the list, choose **Other...**. The **Open** dialog will prompt where you can choose the desired job.
- Choose the menu item "All Jobs" to clear the current filter.

Note: You cannot view, open or edit the Tickets of the Tasks or the Workflows in the Task Monitor. This is only possible in the Automation Engine Pilot.

Intervening in a workflow

You can intervene when a workflow is halted for user intervention provided the Workflow user intervention settings designate you for intervention. To intervene,

- Click on **User intervention needed** link of a workflow.
- In the resulting **To Do** window, choose to **Forward** or **Release**.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
OmNom-eat.ai	mabm	Run Script	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✔	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08...
OmNom-eat.ai	mabm	Run Script	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✔	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom.ai	mabm	Wait for...	0%	User intervention	▶	10/30/12 5:15...
OmNom.ai	mabm	Workflow	0%	Running	▶	10/30/12 5:15...
Screen Shot 20...		Wait for...	0%	User intervention	▶	10/30/12 5:14...
Screen Shot 20...		Workflow	0%	Running	▶	10/30/12 5:14...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
Screen Shot 20...		Wait for...	0%	User intervention	▶	10/30/12 9:16...
Screen Shot 20...		Workflow	0%	Running	▶	10/30/12 9:16...

Tip: You cannot hide the columns displayed in the Task Monitor. However, you can change the order and the width of the columns.

Context menu options

Right-click on a workflow/task to access the context menu with the following options:

- **Open Output File** opens the output file of the Task or Workflow with the appropriate application.
- **Show Output File in Finder/Explorer** shows the folder that contains the output file in Mac Finder or in Windows Explorer.
- **Release** allows you to release the Task or Workflow if it is on Hold.
- **Cancel** allows you to cancel the Task or Workflow.
- **Delete** allows you to delete the Workflow from the list of Workflows the Task Monitor.
- **Annotation...** shows a dialog that allows you to add or modify an annotation.
- **Details** shows the Details dialog with detailed information about the Task or Workflow.

Alternatively, these options are also available via the **Task** menu of the Shuttle application.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
shuttle12.1_cont...		Wait for Acti...	100%		▶	11/7/12 2:52 PM
shuttle12.1_con...		Workflow	100%		✘	11/7/12 2:52 PM
shuttle12.1_ae_s...		Wait for Acti...	100%		▶	11/7/12 2:52 PM
shuttle12.1_ae_...		Workflow	100%		✘	11/7/12 2:52 PM
shuttle12.1_todo...		Apply ArtPr...	100%		✘	11/7/12 2:20 PM
shuttle12.1 tod...		Workflow	100%		✘	11/7/12 2:20 PM
OmNom.ai	mabm	Wait for Acti...	100%		▶	10/30/12 5:15...
OmNom.ai	mabm	Workflow	100%		▶	10/30/12 5:15...
Screen Shot 201...		Wait for Acti...	100%		▶	10/30/12 5:14...
Screen Shot 201...		Workflow	100%		▶	10/30/12 5:14...
arrow.png		Workflow	100%		▶	10/30/12 10:5...
arrow.png		Workflow	100%		▶	10/30/12 10:5...
arrow.png		Workflow	100%		▶	10/30/12 10:5...
arrow.png		Workflow	100%		▶	10/30/12 10:5...
Screen Shot 201...		Wait for Acti...	100%		▶	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		▶	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		✘	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		✘	10/30/12 9:16 AM

Open Output File

Show Output File in Finder

Release

Cancel

Delete

Annotation...

Details

The Shuttle Launch Panel

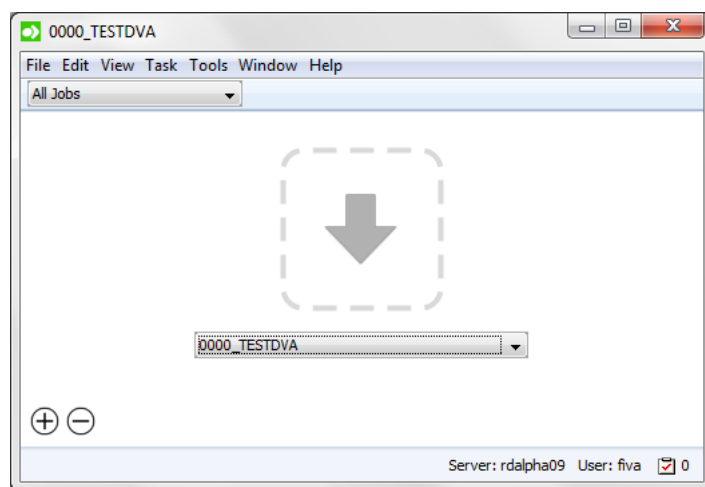
You can drop one or more files as input files for the selected Workflow/Task through the Shuttle Launch Panel.

- Browse to the desired workflow using the **Workflow** drop-down list. This will be updated when new workflow tickets are created. Shuttle launches the selected workflow when files are placed in the Shuttle Launch panel.



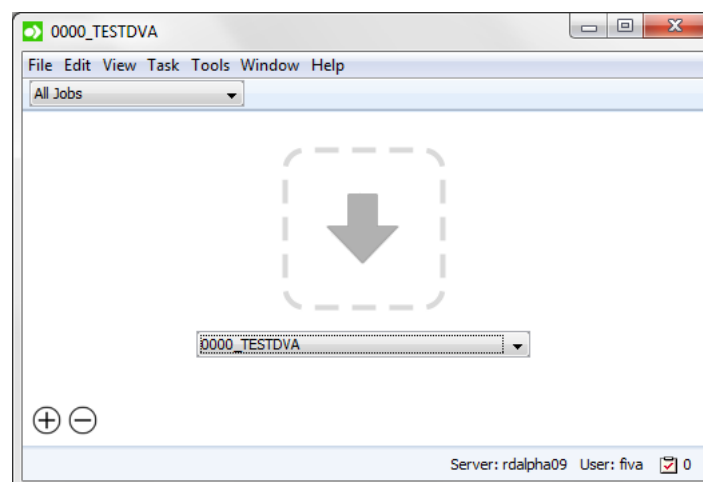
Attention: The drop-down list only shows the Workflow Tickets that are public.

- The selected workflows are saved when you quit Shuttle. Shuttle restores the selection of workflows during the next launch.



Launching a Workflow from the Automation Engine Shuttle

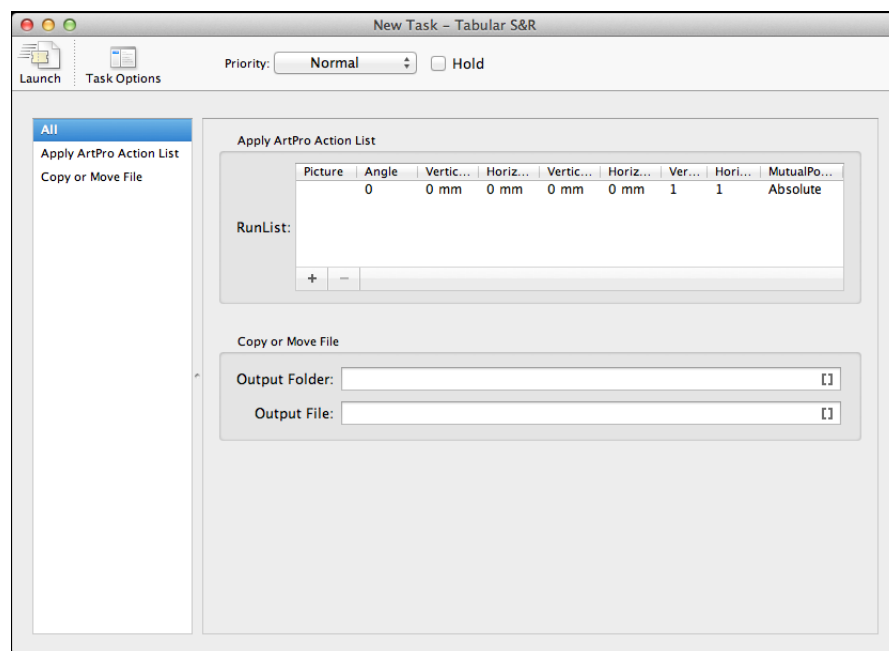
1. If the Launch Panel is not visible in the Shuttle window, go to **View > Launch Panel** .



2. Select the desired **Workflow** with the drop-down menu. Shuttle will launch this **Workflow** when files arrive at the launch panel.
3. Submit one or more files in one of the following ways.
 - Drag and drop the desired file(s) into **Drop Files Here** area.
 - Go to **File > Launch...** and select the files.
 - Use the keyboard shortcut `Command-L` on Mac or `Ctrl-L` on Windows and select the file(s).

Note:

- The selected file(s) will be uploaded to the root folder of the job.
 - If a file was previously launched on the chosen workflow, a dialog will prompt to give you the option to reuse the public parameter values from the previous launch. If you choose to reuse these public parameter values, they will be used as default parameter values. You can still edit these values if necessary.
 - If a file with the same name is already present in the root folder of the job, a dialog is prompted warning you of overwriting the file. You can then choose to either **Replace** the existing file, or **Cancel** the launch to rename the file.
4. If your workflow has Public Parameters, you can fill them in the **New Task** window which will prompt. Learn more in [Using Public Parameters in Workflows](#) on page 1051. It also possible to set the **Task Options** in this window.



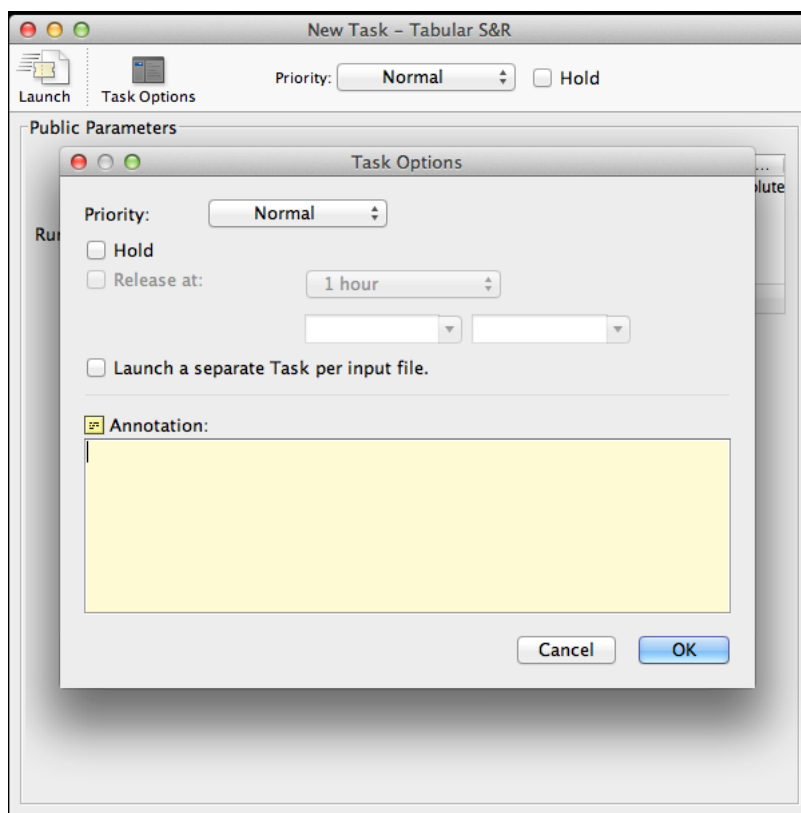
- a) You can set **Task Options** in the **New Task** window, click the **Task Options** button. In the resulting window, you can:
 - set the **Priority** of the Task or Workflow as **Low**, **Normal**, **High** or **Immediate** by selecting from the drop-down.

- select **Hold** if you want to hold the launch of the Workflow. The Workflow will not be launched until you **Release** via the right-click menu in the Task Monitor or via the **Task** menu.

Alternatively, hold the workflow until a set period of time after which it will be automatically released. To do this, select **Release at** and specify the period or date and time.



Note: You can bypass the automatic release setting by selecting **Release** via the right-click menu in the Task Monitor or via the **Task** menu.

- Select **Launch a separate Task per input file** for launching a separate Task or Workflow for each input file.
- You can also set an **Annotation** for the Task or Workflow in this window.





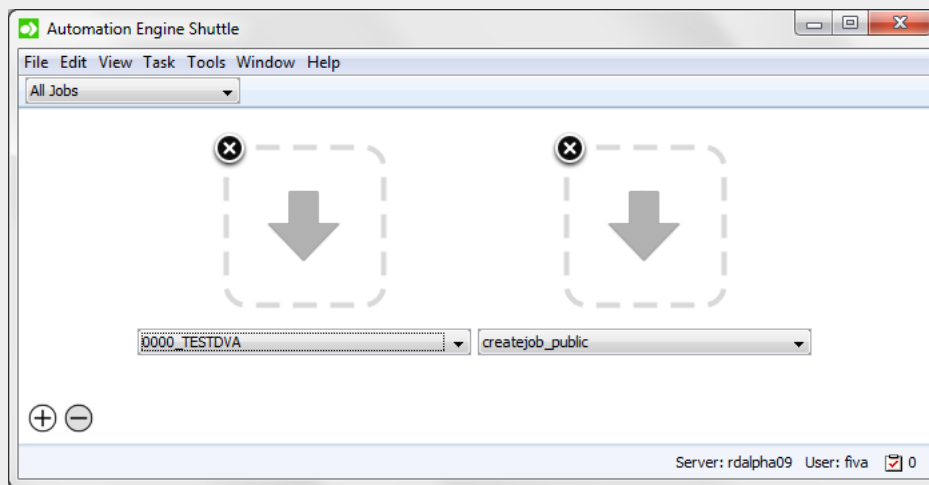
5. Click the **Launch** button to launch the workflow.
The task will immediately appear in the Task Monitor.

Note:

You can upload multiple files for multiple workflows. Click on the **Plus** button   to add extra drop zone slots.

To remove these slots, proceed as follows:

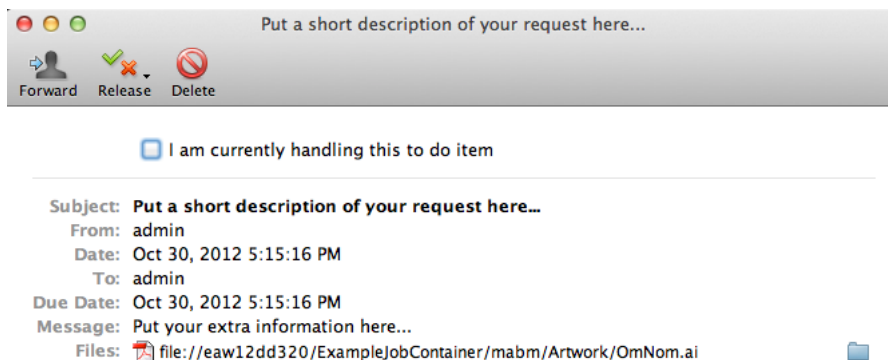
- Click on the **Minus** button  . This will place the user interface in Editing mode
- Click on the desired **Delete** button on the top left corner of the drop zone.
- To leave the Editing mode, click on the **Minus** button, or press **Escape**.



User Intervention from the Shuttle

If you have setup a **Wait for Action** step in the workflow, the process halts for user intervention (for a **To Do** action). **User intervention needed** link will appear in the Task monitor when the workflow is halted.

1. Click on the **User intervention needed** link.
The **To Do** dialog will prompt.



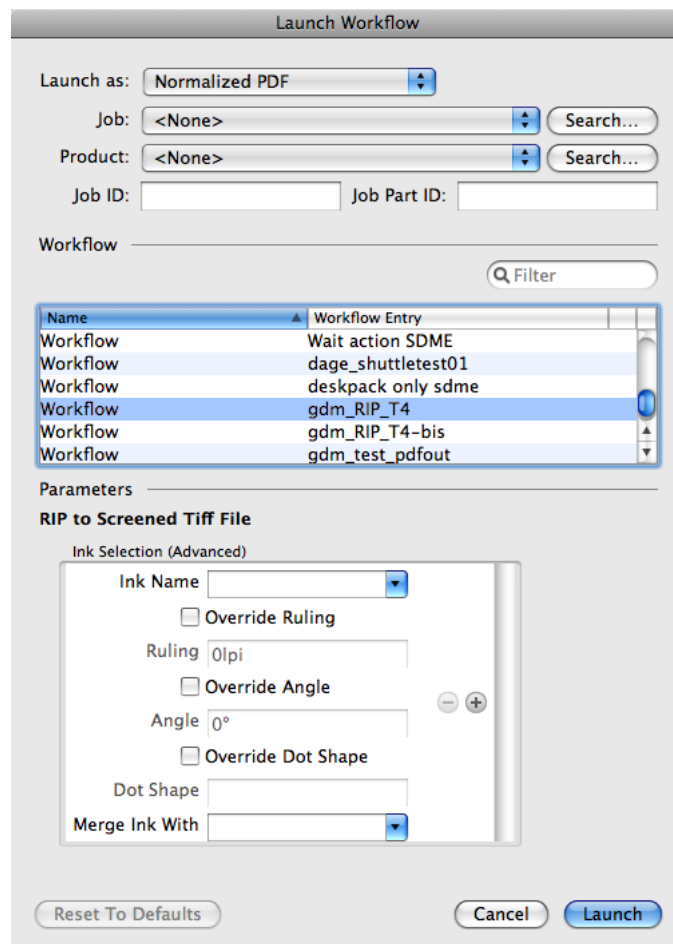
2. You can select **I am currently handling this to do item** to mark that you are handling this **To Do**.
3. You can choose to **Forward**, **Release** or **Delete** the **To Do**.

- You can let the workflow continue by clicking **Release**. Select the output state that you require in the release. The **To Do** window will close and the Workflow will continue processing.
- You can forward the **To Do** to another user. Click **Forward** and the workflow will not continue until the other user handles the **To Do**.
- You can also delete the **To Do** by clicking on **Delete**. This will end the processing of the workflow.

Launching Files from ArtPro

1. To launch your current file to Shuttle, go to **File > Launch Workflow...**

This opens the **Launch Workflow** window.

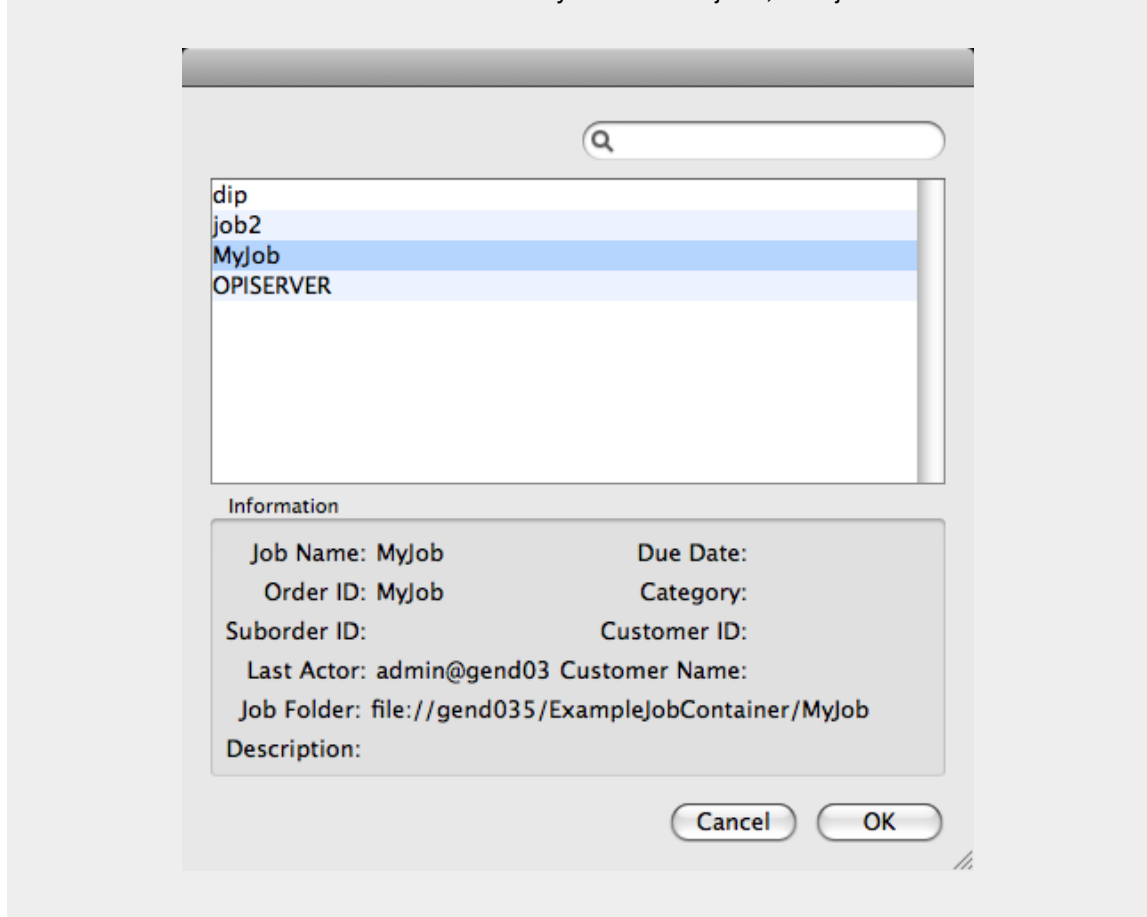


2. Choose what file type you want to launch your file as.

If your workflow is...	you can launch the file as...	with...
an Automation Engine workflow accepting Normalized PDF files as input	Normalized PDF	either Normalized PDF or Normalized PDF (embedded)
an Automation Engine workflow accepting PDF files as input	PDF	any of the available PDF flavors
an Automation Engine workflow accepting ArtPro files as input	ArtPro	with the included images (TIFF, EPS...) or without (only the ArtPro and the ArtPro CT file)

3. If you want to add your file to an Automation Engine job, select it in the **Job** list or fill in the **Job ID** and/or **Job Part ID**.

Note: Use the **Search...** button to see a list of your server's jobs, with job information and a filter.



4. If you want to add your file to an Automation Engine product, select it in the **Product** list or fill in the **Job ID** and/or **Job Part ID**.

Note: Use the **Search...** button to see a list of your server's products.

5. Select the workflow you want to launch your file to.
6. If your workflow has public parameters, you can fill them in in the same window.
7. Click the **Launch** button.

Launching Files from PackEdge or Plato



Attention: You must save your file on an Automation Engine container before you can launch it into a workflow from PackEdge or Plato.

1. Go to **File > Launch Workflow...**

This opens the **Launch Workflow** window.

Name	Workflow
Create SoftProof for WebCenter	Export to PDF File
Mark as Proof	Export to PDF File
Public	Add to Products

The default parameter set is used now.

Parameters _____

Add to Products

Product Name [File]

Product ID [File]

Reset To Defaults Launch Cancel

2. Select the workflow you want to launch on your file in the workflow list.
3. If your workflow has public parameters, you can fill them in in the same window.

Note: For more information on public parameters, see [Using Public Parameters](#) on page 68.

4. Click **Launch** to launch the workflow on your file.

Tip:

To access Shuttle easily in the future (without using the menus):



1.

Go to **Window > Toolbars > Shuttle** to open the Shuttle palette.



2. Drag it into your application's toolbar.

3.

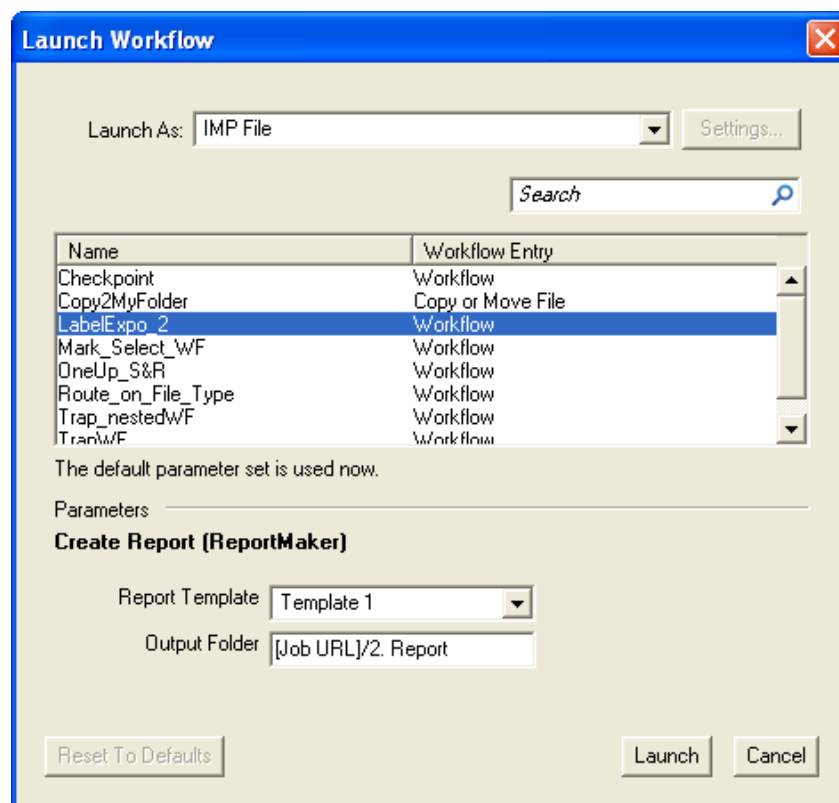
Click  to open the **Launch Workflow** window, or  to open the **Shuttle** window.

Launching Files from FastImpose

When working with FastImpose Server, you must save your file on an Automation Engine container before you can launch it.

1. Go to **File > Launch Workflow...**

This opens the **Launch Workflow** window.



2. Choose what file type you want to launch your file as.

- When working with FastImpose Server, you can only launch your file as an IMP file.
- When working with FastImpose Standalone, you can launch your file either as a JDF or a PDF file.

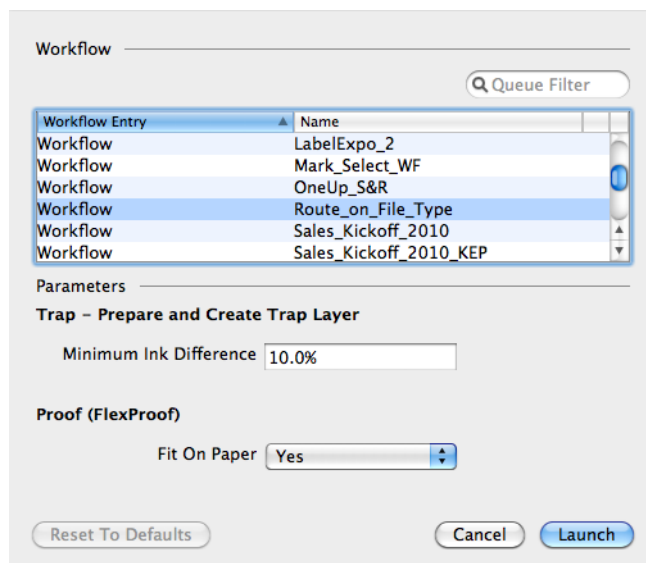
Click the **Settings...** button to set **JDF** or **PDF Export Preferences** (see the FastImpose manual for more information).

3. Select the workflow you want to launch on your file in the workflow list.
4. If your workflow has public parameters, you can fill them in in the same window.
5. Click **Launch** to launch the workflow on your file.

Launching Files from Neo

1. Go to **File > Launch Workflow...**

This opens the **Launch Workflow** window.



2. Select the workflow you want to launch on your file in the workflow list.
3. If your workflow has public parameters, you can fill them in the same window.

For more information on public parameters, see:

- [Using Public Parameters](#) on page 68,

Note: If you want to go back to the default parameters values after changing them, use the Reset To Defaults button.

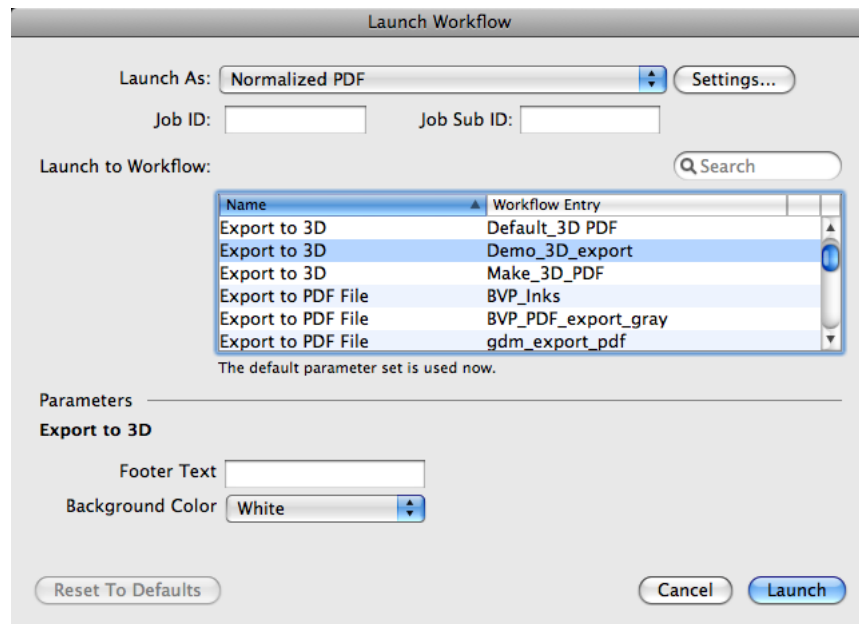
4. Click **Launch** to launch the workflow on your file.

Launching Files from the Shuttle Plug-in (Adobe Illustrator)

To launch a file into a workflow through the Shuttle plug-in, it needs to be the current file in Adobe Illustrator (open and at the front).

1. Go to **File > Launch Workflow...** or use **Option + Command + Z**.

This opens the **Launch** window.



2. When launching to Automation Engine, choose **Normalized PDF** in **Launch As**.

This copies the Normalized PDF version of your file onto the **DeskPack Container**.

Note: This uses the PDF Export plug-in to convert your Illustrator file to Normalized PDF. To refine the conversion, click **Settings...** and edit the **PDF Export Preferences**.

3. If you want to link your file to an existing Automation Engine job, fill in the **Job ID** and/or **Job Sub ID**.

Note:

- You must enter a **Job ID** / **Job Sub ID** that already exists on your Automation Engine server. The Shuttle plug-in cannot create jobs.
- The Normalized PDF that you launch with a **Job ID** / **Job Sub ID** will be **linked** to that job, so that tasks and workflows processing that file can use the job parameters.
- If you are launching a file that was already in an Automation Engine job, the **Job ID** and/or **Job Sub ID** will be filled automatically.

4. Select the workflow you want to launch your file to in **Launch To Workflow**.
5. If your workflow has public parameters, you can fill them in, in the same window.
6. Click **Launch** to launch your file.

1.6 Launching Tasks on Files

1.6.1 Creating a Task Ticket

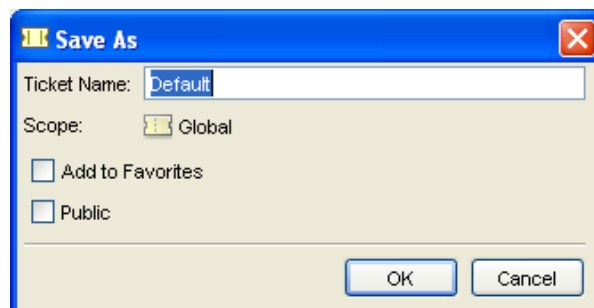
Creating a Custom Ticket from the Tickets View

To create a ticket with your own parameters, that you will be able to reuse:

1. Open the **Default** ticket of your chosen task.

Note: To create a custom **workflow** ticket, see [Workflows](#) on page 986.

2. Fill in your parameters.
3. Go to **File > Save As...** or use **Ctrl + Shift + S** (Windows) / **Shift + Command + S** (Mac).
4. In the **Save As** pop-up:



- a) Enter the name to give your custom ticket
- b) Add the ticket to your **Favorites** if you like
- c) Make the ticket **Public** if you want to be able to use it from Shuttle (send files to that task using your ticket's parameters)

Note: The **Scope** is always **Global** when creating a custom ticket from the **Tickets** view.

Creating a Custom Ticket when Working with Files

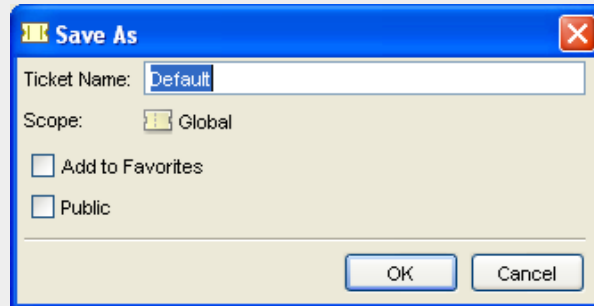
You can create a custom ticket when launching a task on a file too:

1. Right-click your file, select **New Task** then **More Tickets...**
2. In the **Select Ticket** dialog that opens, select the **Default** ticket for the task you want to use.
3. Fill in the settings you want to use in the ticket.
4. Go to **File > Save As...** or use **Ctrl + Shift + S** (Windows) / **Shift + Command + S** (Mac).
5. In the **Save As** pop-up:

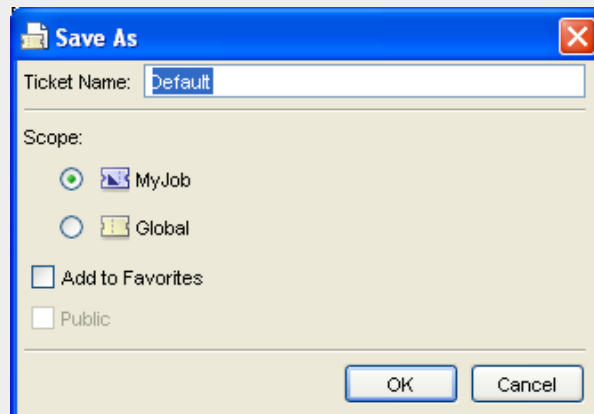
- a) Enter the name to give your custom ticket
- b) Add the ticket to your favorites if you like
- c) Make the ticket public if you want to be able to use it from Shuttle (send files to that task using your ticket's parameters)

Note:

- If you are not in a job context, the **Scope** is always **Global**.



- If you are in a job context, you can choose between **Job** and **Global Scope**.
When choosing the job scope (called after your job), you **cannot** make the ticket public.



Using Public Parameters

Public Parameters are Ticket parameters that users can define when launching files from these client applications connected with the Automation Engine server:

- Automation Engine Shuttle
- Automation Engine Pilot
- ArtPro, Neo
- PackEdge, Plato, FastImpose
- DeskPack (Adobe Illustrator)

Note: Only Public Tickets can be launched from these client applications.

For each public parameter, you can either:

- enter a list of predefined values that the user must choose from,
- let the user enter a value himself.

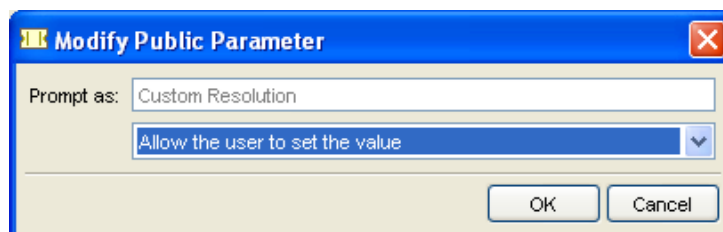
Making a Parameter Public

Note: You need to define which parameters are public before files are submitted to your workflow.

Note: Not all parameters of all tickets can be made public.

In the ticket containing the parameter to make public:

1. Right-click the parameter and select **Make Parameter Public**.
2. Right-click it again and select **Modify Public Parameter...**
3. In the **Modify Public Parameter** dialog that opens:
 - a) If necessary, change the way the parameter will be called in the client application in **Prompt as**.
 - b) Choose either:
 - **Allow the user to set the value** if you want client application users to be able to freely enter a value for that parameter.



- **Let the user select a predefined value** if you want client application users to choose from a list of values that you define.

If that parameter already offers a list of values to choose from in the ticket, they will be displayed here. You can **Add** or **Remove** values.

Note: client application users will see (and choose from) the **Predefined Settings**. They will not see the associated **Values**.

- c) If you have chosen **Let the user select a predefined value:** in **Default Predefined Setting**, choose the setting that will be selected by default. If the user doesn't select another setting, this one will be used for processing.

4. Save your ticket to also save all settings related to public parameters.

This is how your public parameter will look like when it prompts to the users of the client application:

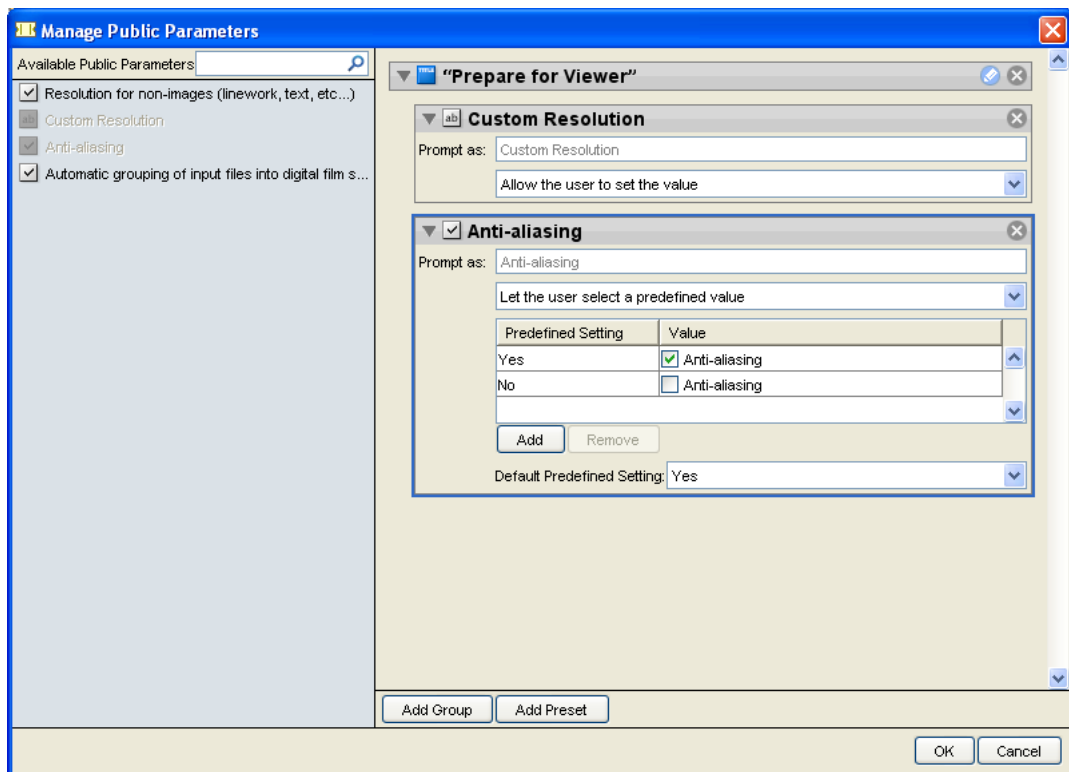
If you chose Allow the user to set the value	If you chose Let the user select a predefined value

Managing Public Parameters in Your Ticket

Once you have made parameters public in your ticket, you can have an overview of your public parameters and edit them at the ticket level.

1. In your ticket, go to **Advanced > Manage Public Parameters...**



The **Manage Public Parameters** dialog that opens contains all of your ticket's public parameters.

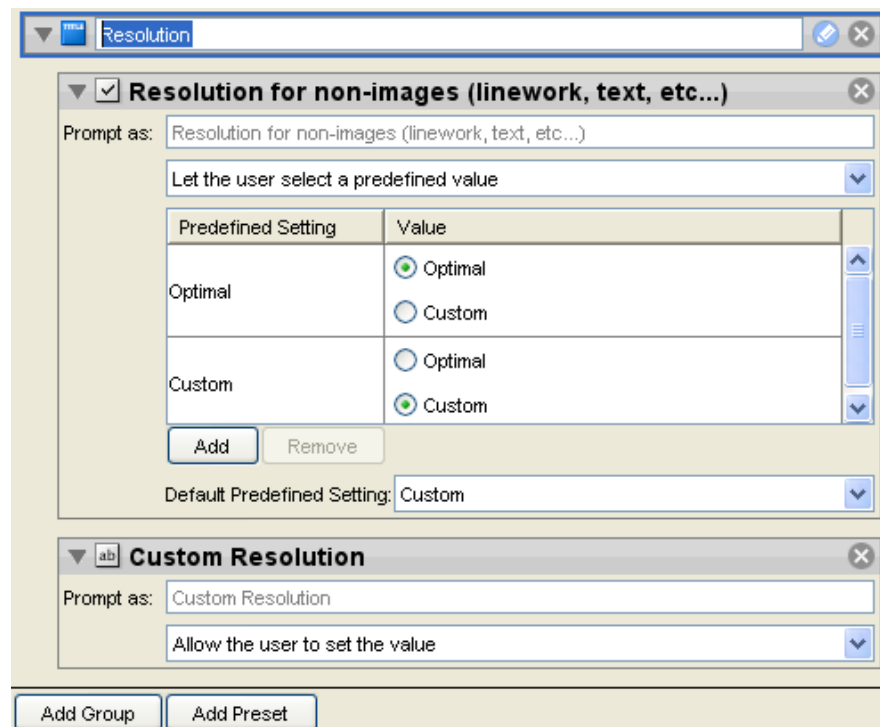


The left pane shows all parameters that can be made public for that ticket (the ones you have made public are greyed out).

The right pane shows the parameters you have made public, with their settings (predefined values, etc.).

2. You can:

- drag parameters from left to right to make them public;
- click a public parameter's cross  if you don't want it to be public anymore;
- rearrange the order of your public parameters (drag them up or down);
- change public parameters' settings (**Prompt as**, **Predefined Settings**, etc.);
- group public parameters under a custom header (click **Add Group**, click  to edit the header, and drag parameters from the left);



- use presets to simplify users' choices (see [Making Presets to Simplify the Users' Choices](#) on page 72).

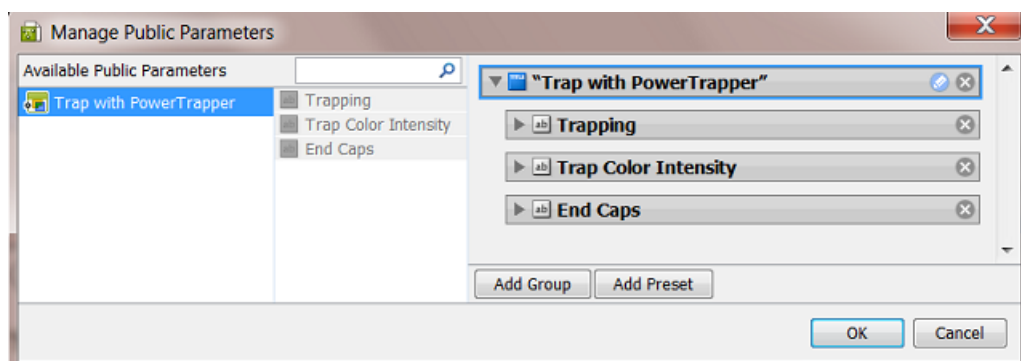
Making Presets to Simplify the Users' Choices

Presets are custom combinations of parameters. They simplify the choice users have to make. They limit the amount of decisions that users have to make.

Example

For example, you want users to use different trapping settings when printing offset versus dry offset. You can define those trapping settings in a preset, so that users will only have to choose between 'offset' and 'dry offset'.

We start from a **Manage Public Parameters** dialog where the trapping step has 3 public parameters:



1. Click on to remove the (default) **Group** named "Trap with PowerTrapper"

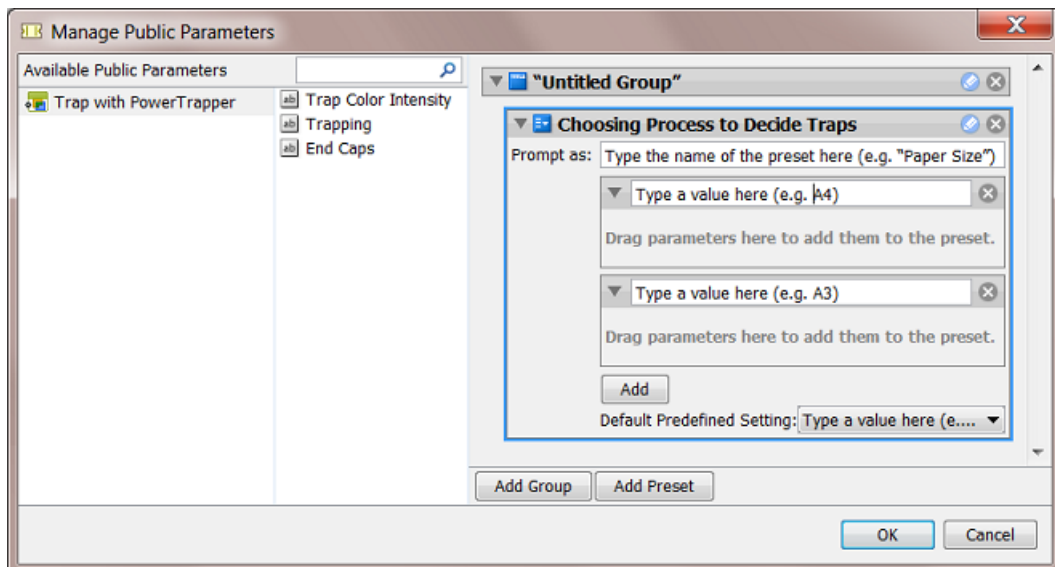
2. Press the **Add Group** button.

A new "Untitled Group" is created. Click on  to change the name.

3. Press the **Add Preset** button.

A new "Preset" is created. Click on  to change the name.

By default, two example values will appear. the preset was renamed to 'Choosing Process to Decide Traps'.

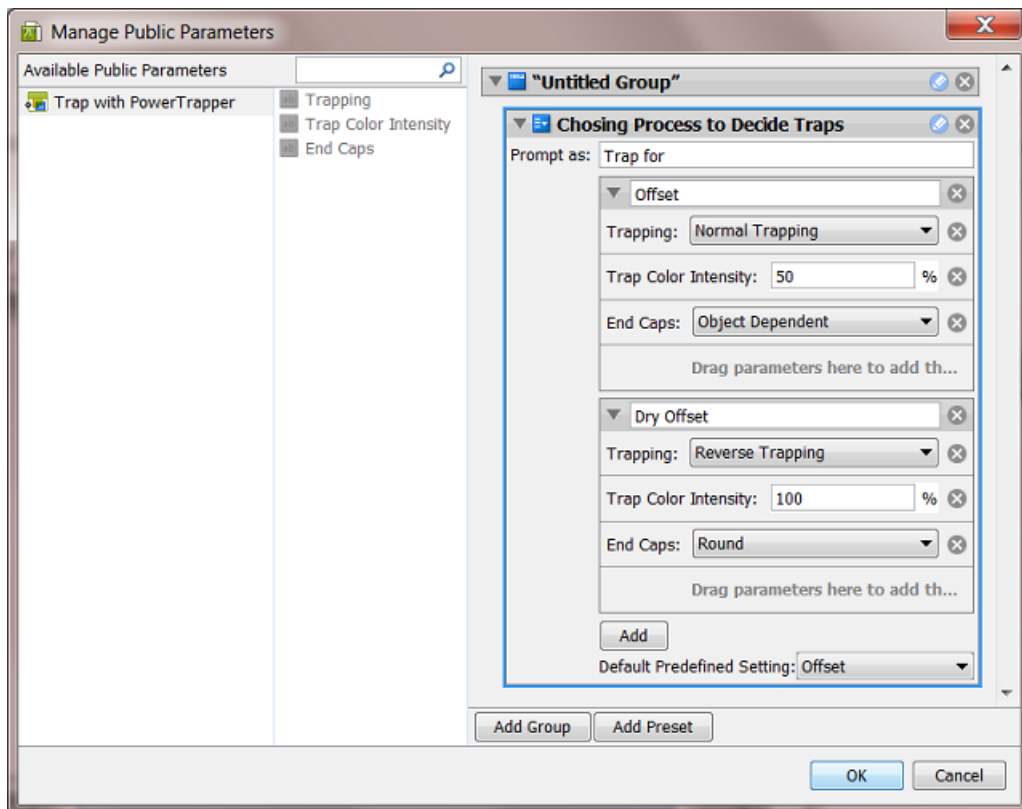
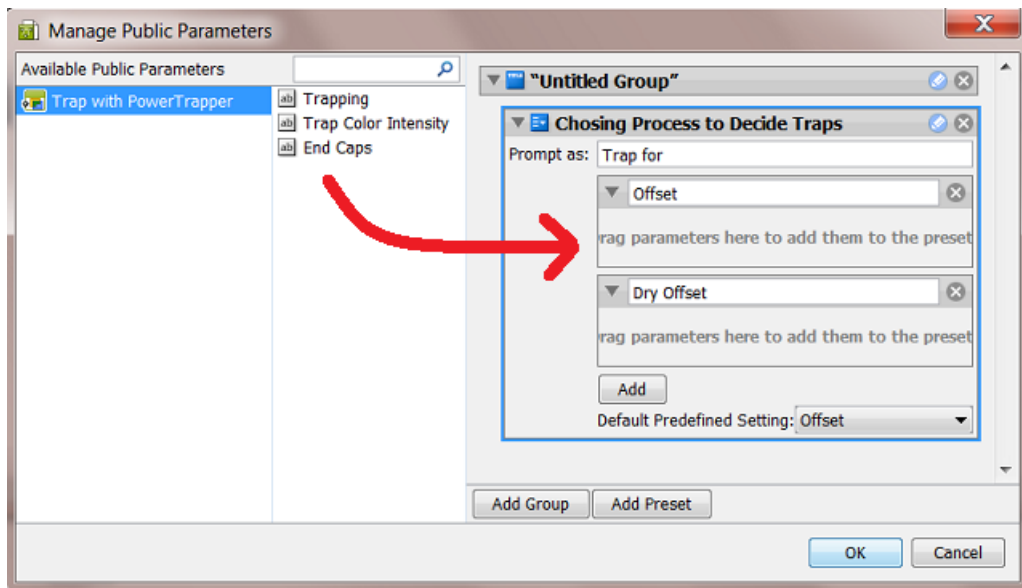


4. Define what the users will see:

- a) in **Prompt as**, enter the text that users will see in the client application (replacing 'Type the name of the preset here (e.g. "Paper Size")').
- b) enter the values that the users will choose from (replacing 'Type a value here (e.g. A4)'). We here added 'Offset' and 'Dry Offset'.

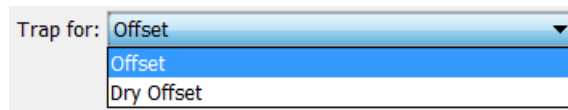
Click **Add** if you want to add extra values (not the case in our example).

5. Drag the parameters from the dialog's middle pane onto one preset item. This will make them automatically appear under each preset item (here both under *Offset* and *Dry Offset*).



6. Define the parameters values for each preset item.
See the above screenshot for a good example.
7. Click **OK** to confirm and close the dialog.
8. Save your ticket.

Result: this is how the resulting dialog will look like to the users of the client applications:



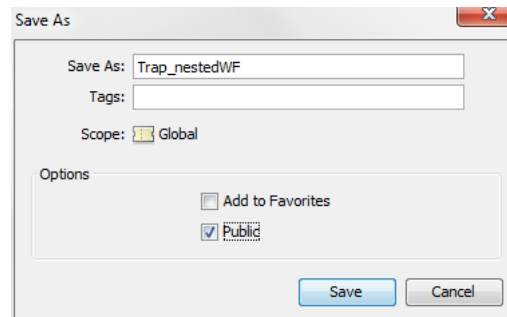
Making Your Ticket Public

Important: Only Public tickets or workflows can be launched from the client applications mentioned [here](#).

Note: Default tickets can not be made Public.

To make a ticket **Public**, you can either:

- Select the **Public** option when creating the ticket.



- Select your ticket in the **Ticket** view, click  and select **Public**.



- Right-click your ticket in the **Tickets** view and select **Make Public**.

Note: Public tickets have a mark in the **Public** column.

Task Type	Ticket Name	Public	Description
Zip	Default		
Workflow	WC_Approval		
Workflow	WaitForRunListFiles		
Workflow	TrapWF		
Workflow	Trap_nestedWF	✓	
Workflow	SimpleTrap		
Workflow	Send2Ody		

SmartNames

SmartNames are documented as a separate chapter in the "Complete Manual". Find it here: [SmartNames](#) on page 916.

1.6.2 Launching a Task on a File

There are several ways to launch a task on a file (from the Pilot, from Shuttle, via Access Points or via JDF/JMF). Here is how to launch a task from the Pilot's **Files** View.

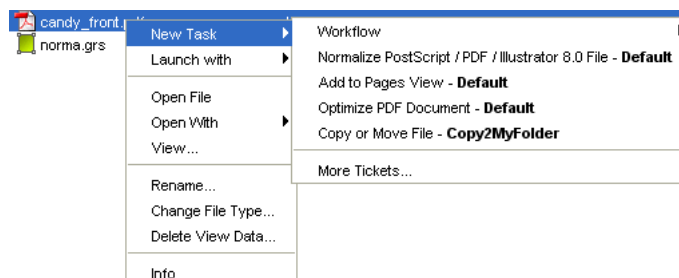
1. In the Pilot, select the **Files** view from the **Views** bar (or use **Go > Files**).
2. In the **Containers** mode, browse to your chosen file.
3. Right-click it, and select either:
 - **New Task** to fill in the ticket's settings before launching the task.
 - **Launch with** to launch the task without changing the ticket's settings.

Note:

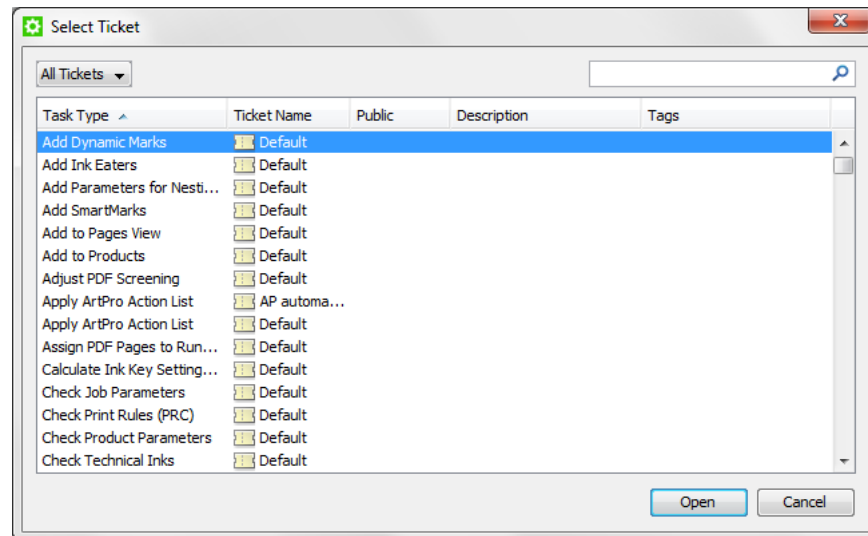
If you don't have the **Tickets: Show All Tickets and their Parameters (Public and other)** access right (see [Access Rights](#) on page 702 in the Reference Guide), you will only see **Launch with**.

In this case, you can only select public tickets, and only fill in public parameters in the ticket.

4. Select the ticket to use.
 - If you have launched tasks previously, the most recent task tickets you used are listed.

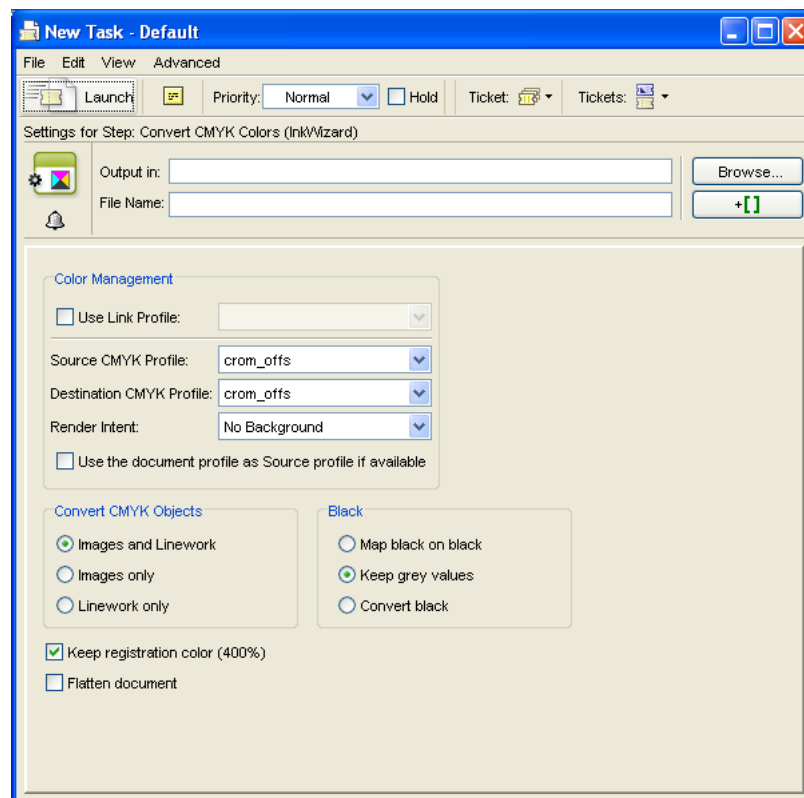


- If you didn't, or if you need a different ticket, select **More Tickets...**
In the **Select Ticket** dialog that opens, select the task ticket you want to use and click **OK**.



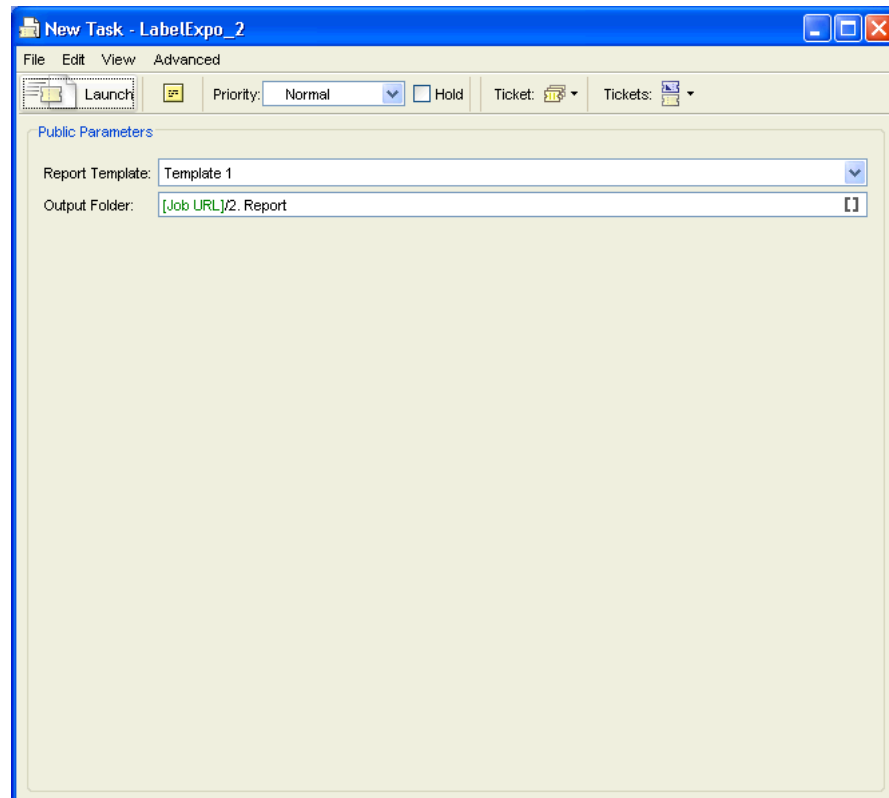
Note: This dialog contains all the tickets suitable for the file type you selected. You can choose either a **Default** ticket, or a custom ticket (saved under a different name) if you have already created one.

5. • If you chose **New Task**, the task ticket opens. Fill in the settings you want to use in the ticket and click the **Launch** button.



- If you chose **Launch with** and you have the **Tickets: Show All Tickets and their Parameters (Public and other)** access right, the task is launched on the file as soon as you selected the ticket. You don't need to fill in any settings.
- If you chose **Launch with** and you don't have the **Tickets: Show All Tickets and their Parameters (Public and other)** access right, the task is launched on the file as soon as you selected the ticket if there are no public parameters.

If the ticket contains public parameters, it will open for you to fill them in (you will only see the public parameters). Click **Launch** when you are done.



Note: For a description of all of the ticket's settings for each task, see [The Tasks and Workflow Controls](#) on page 239 in the Reference Guide.






You can monitor the task's progress in the **Files View's Task** pane or in the **Tasks View**. See [Check the Status of the Task](#).

1.6.3 Checking the Status of the Task

After launching a task on one or several file(s), you can check its processing status in the Pilot's **Tasks** pane. You can see the **Tasks** pane in the **Files**, **Product** or **Tasks** views.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
07_RussianText_OTF.pdf		Copy or Move File	100%		✓	05/01/10 18:24

You can see the progress (in percentage) and the processing state. The state can be:

- : starting
- : success
- : warning
- : failure
- : processing cancelled

You can also double-click the task entry in the **Tasks** pane to open your task ticket and check its parameters.

Note: Depending on your [access rights](#), you may only see the public parameters.

1.6.4 Relaunching the Task

- To relaunch the task without changing the settings:
 - a) Select the task to relaunch in the **Tasks** pane.
 - b) Go to **Tasks > Relaunch**.
- To change the settings and relaunch the task:
 - a) Select the task to relaunch in the **Tasks** pane.
 - b) Go to **Tasks > Open**.
 - c) Change the task settings as necessary.
 - d) Click the **Launch** button.

1.7 Launching Workflows on Files

Workflows are documented in a separate dedicated chapter [Workflows](#) on page 986.

1.8 Viewing Files in the Automation Engine Viewer

1.8.1 Introduction

What is the Automation Engine Viewer?

The Automation Engine Viewer is a comprehensive, powerful and integrated Viewer and QA tool. It can easily be integrated in your Automation Engine workflow, to view, check and compare graphic production data. It can be launched directly from within the Automation Engine Pilot, on any Mac or Windows computer in your network. Anywhere you can install and log on to an Automation Engine Pilot you can start the Automation Engine Viewer (DTP, prepress, QA department, CSR desk...).

You can use as many Automation Engine Viewers as you have Pilot connections. Opening the Automation Engine Viewer does not allocate an extra Pilot connection. It is even possible to have two

or more Automation Engine Viewers on one workstation by opening a second Automation Engine Pilot (Pilot - File menu - New window) and opening a file in the Automation Engine Viewer from there. If you also use the Esko Bitmap Viewer, you will notice that the interface and shortcuts are nearly identical.

The **Viewer** offers you

- Reliability: Showing the data in a correct way. At best in the same way the RIP / Proofer will see them.
- Detail: Unlimited zoom, high detail measuring, measuring traps and exact distances (die objects).
- Speed: Opening files swiftly, even when it concerns large files. Fast zooming.

Supporting many file formats, Comparing and more

The **Viewer** supports many different file types: PDF, AI, ArtPro, images and even bitmap RIP data. For a full list, see [Supported File Types](#) on page 580.

You can even compare with other files, even when they are a different file type and even when they have a different size! Learn more in [Comparing Files](#) on page 81.

Basic use

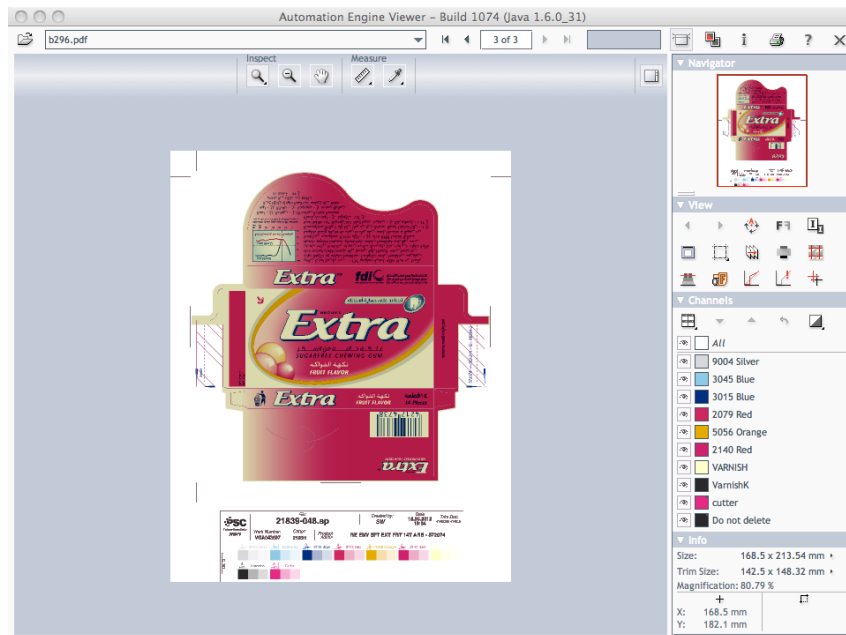
To open files in the **Viewer**,

- click the word **View** next to your file.

Name /	View	Type	Size	Date
 Celebrity.pdf	View	PDF File	41MB	07/04/10...
 CMYK.pdf	View	Normalized...	32KB	05/01/10...
 Dolly_May.pdf	View	PDF File	2.6MB	06/04/10...

- right-click your file and select **View...**
- select your file and go to **File > View** .

Here 's a first look at the Viewer:



You can zoom in on the file, measure distances and densities, view separations individually or together, etc.

Note: The Viewer has its own local On Line Help. Start it from the question mark icon in the top bar.

The concept of Prepared View Data

Reliability is a must. Detail can also not be compromised, it is always offered. The challenge lies in opening very large files quickly. That is why the Automation Engine Viewer offers 2 view modes: with or without Prepared View Data.

- In case you do not prepare View Data, the server will stream the data to the Automation Engine Viewer. When a large file has to be opened or compared or when the server is very busy, a delay can be noticed. Deep zoom detail mode is immediately available.
- In case you do opt to prepare View Data in advance, then the Viewer will show you the prepared View Data instantly. Zooming in will of course require object data streaming from the server, but then for a smaller area. For more information about creating View Data, see the task [Prepare for Viewer](#) on page 415 in the Reference Guide.

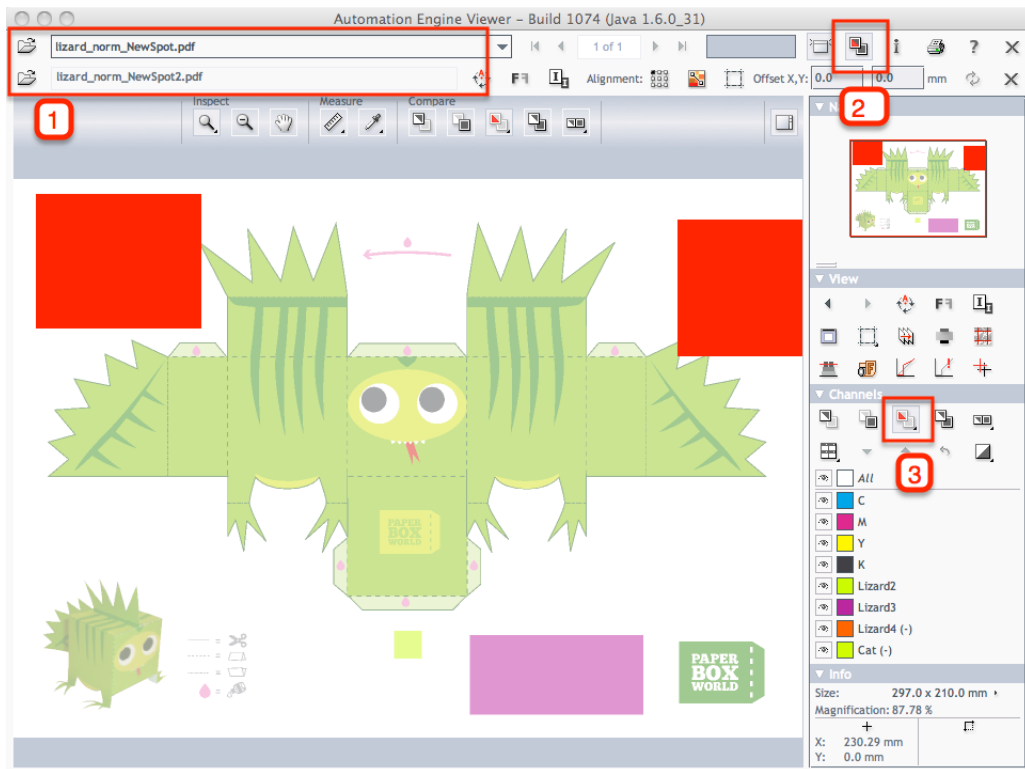


Attention: If the **User Access Right** for **deep zoom** is not enabled (read more in [Defining User Access Rights](#) on page 33), then

- you can only view 'prepared' files.
- your zoom is restricted.

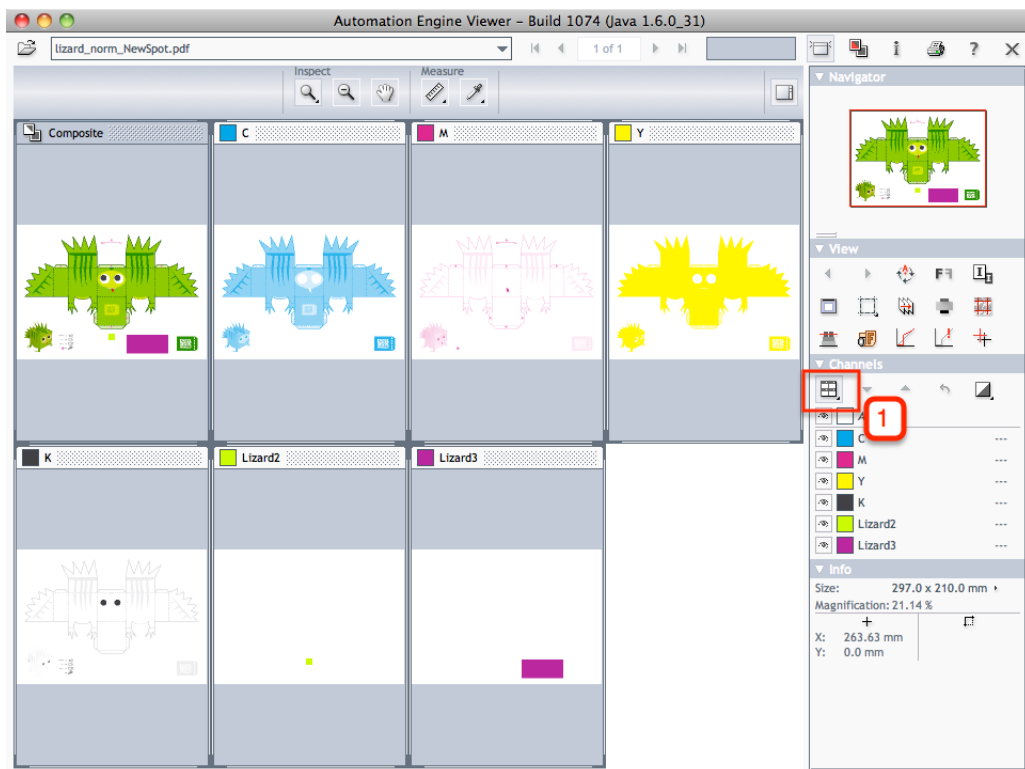
1.8.2 Comparing Files

The **Viewer** is also a powerful compare tool. It allows to compare two files and have a close look at the differences, if needed per separation, including changes in images (internal or referenced images). Various tools help you analyse those differences.



- 1: The names of the two files that you are comparing
- 2: **Enable difference view** - to activate the Compare tool
- 3: **View both - highlight the difference**: the differences between the two files is highlighted in red. Right-click this button to see the other options for compare that are available.

While comparing you can also use other tools like for example "view separations in separate windows", measure, (de)select, rotate, etc...



1: Show channels (Automatic tiling) - Right-click this button to see the other options that are available to see the different channels.

1.9 Integration with Other Esko Applications

Note: Integrating Automation Engine with WebCenter is documented in [Integrating with WebCenter](#) on page 1346, a dedicated chapter in the *Complete* documentation of Automation Engine.

1.9.1 Review and Approval of Page Impositions using WebCenter

This chapter is specific on publishing **page impositions** to WebCenter. For all other generic documentation on integrating Automation Engine and WebCenter, we refer to [Integrating with WebCenter](#) on page 1346.

WebCenter Setup

In **WebCenter**, you need to set up a **Project Template** with **Approvers**.

You will use this template in **Automation Engine** when sending impositions: it will allow you to create new WebCenter **Projects** (already containing your **Approvers**) as needed, directly from Automation Engine.

Note: Alternatively, you can set up a WebCenter **Project** with **Approvers** in WebCenter, and send your impositions to it from Automation Engine. But then you will only be able to create new **Projects** from **WebCenter** and not from **Automation Engine**.

1. Create a new **Project**.
2. Add **Members** to the project.
3. Choose **Approvers** from within the Project's members.
4. Save the Project as a **Template**.

Creating a New Project

1. Log in to WebCenter as a user who is a **Project Manager**.
2. Click **Projects**.

The **Favorite Projects** page is shown.

3. Click **Create New Project** at the top right of the page.
4. Choose **Start from a Blank Project** and click Continue.
5. Enter the name for the Project in the **Project Name** field.
 - To make another user or user group the Project Manager for this project, select his or her name, or the group name, from the **Project Manager** list.
 - If desired, enter a description in the **Description** field.
 - To make the Project active, set the **Status** as **Active**.
 - To specify an optional due date, click the calendar control at the end of the **Due On** field and select the desired due date. To set the due time, click the drop-down list box to the right of the calendar control.

Note: This is not the **Approval Due Date**. You will set the Approval Due Date when setting the **Approval Cycle**.

- To use a custom thumbnail, click **Change** and enter the complete path and filename in the associated field or **Browse** for it. To use the default thumbnail, leave that choice selected.

WebCenter by ESKO - Version 12.0 - Build r80

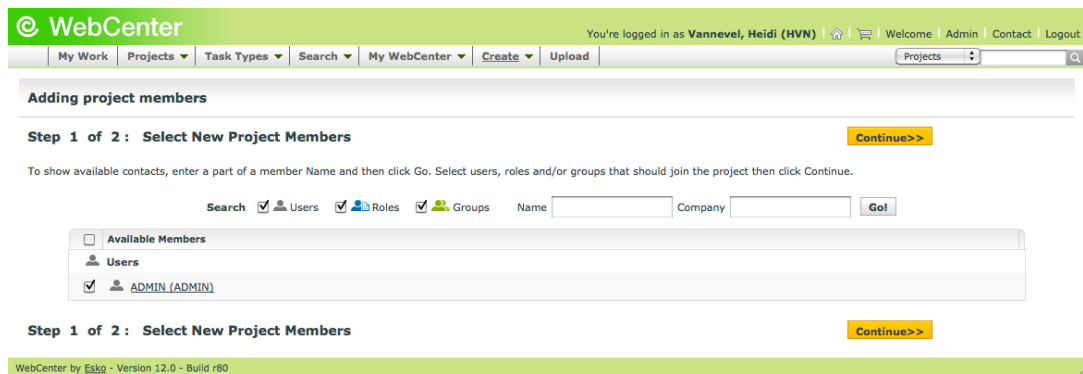
6. Click **Create Project**.
The Project is created.

Adding Members to a Project

1. Once the project is created, click **Members** > **Add Members**.

WebCenter by ESKO - Version 12.0 - Build r80

- This shows a list of the current WebCenter members.
2. Select the users or groups to add as members and click **Continue**.



WebCenter You're logged in as Vannevel, Heidi (HVN) | Welcome | Admin | Contact | Logout

My Work | Projects | Task Types | Search | My WebCenter | Create | Upload | Projects

Adding project members

Step 1 of 2 : Select New Project Members [Continue>>](#)

To show available contacts, enter a part of a member Name and then click Go. Select users, roles and/or groups that should join the project then click Continue.

Search Users Roles Groups Name Company [Go!](#)

Available Members

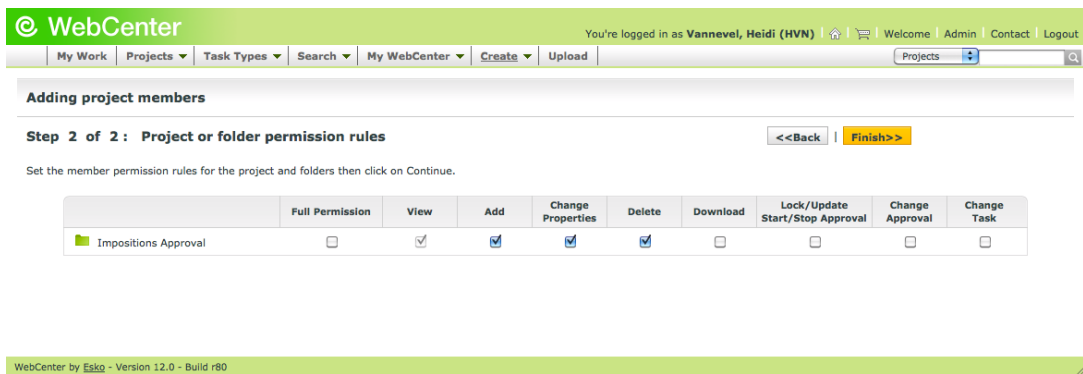
- Users
- ADMIN (ADMIN)

Step 1 of 2 : Select New Project Members [Continue>>](#)

WebCenter by Esko - Version 12.0 - Build r80

3. Set the members' permissions as desired. For Approval tasks, members need at least **View** permissions (selected by default).

Note: The permissions you set here apply to all members you have just added.



WebCenter You're logged in as Vannevel, Heidi (HVN) | Welcome | Admin | Contact | Logout

My Work | Projects | Task Types | Search | My WebCenter | Create | Upload | Projects

Adding project members

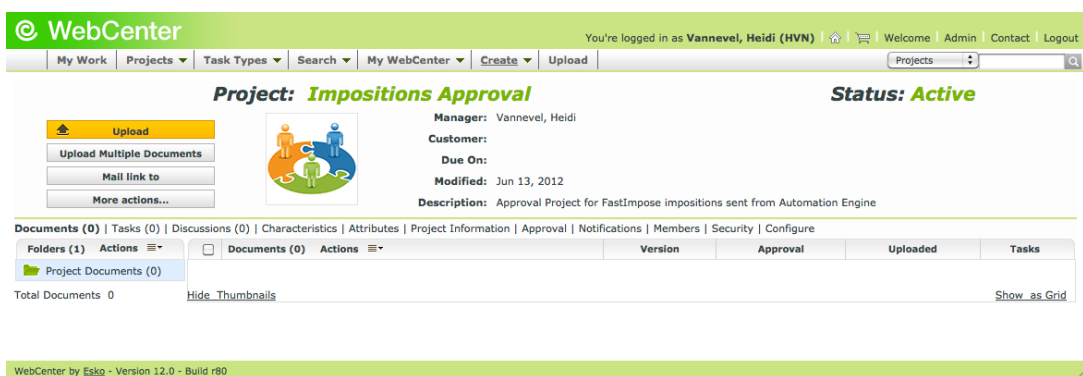
Step 2 of 2 : Project or folder permission rules [<<Back](#) | [Finish>>](#)

Set the member permission rules for the project and folders then click on Continue.

	Full Permission	View	Add	Change Properties	Delete	Download	Lock/Update Start/Stop Approval	Change Approval	Change Task
Impositions Approval	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

WebCenter by Esko - Version 12.0 - Build r80

4. Click **Finish**.
5. Once the members are added, click **Go back to the project**.



WebCenter You're logged in as Vannevel, Heidi (HVN) | Welcome | Admin | Contact | Logout

My Work | Projects | Task Types | Search | My WebCenter | Create | Upload | Projects

Project: **Impositions Approval** **Status: Active**

Manager: Vannevel, Heidi
Customer:
Due On:
Modified: Jun 13, 2012
Description: Approval Project for FastImpose impositions sent from Automation Engine

Documents (0) | **Tasks (0)** | **Discussions (0)** | Characteristics | Attributes | Project Information | Approval | Notifications | Members | Security | Configure

Folders (1) **Actions** **Documents (0)** **Actions**

	Version	Approval	Uploaded	Tasks
Project Documents (0)				

Total Documents 0 [Hide Thumbnails](#) [Show as Grid](#)

WebCenter by Esko - Version 12.0 - Build r80

Adding Approvers to the Project

Approvers can be either regular users or they can be user groups.

When you add a group as an approver, you must indicate whether:

- **all the members of the group** (select the group entry labeled as `whole group`) must approve the document before it gets the approved status, or
- whether the approval of a **single group member** (select the group entry labeled as `one of group`) will set the document to approved status.

Note: You can only change (add or remove) Approvers when the **Approval Cycle** is stopped.

1. In the Project Page, click **Approval**.

WebCenter by Esko - Version 12.0 - Build r80

2. Click **Setup Approval**.

3. Open the **Add approver** list, and select the individual users or the groups you want to add.

Tip: Use Ctrl+Click to select multiple members.

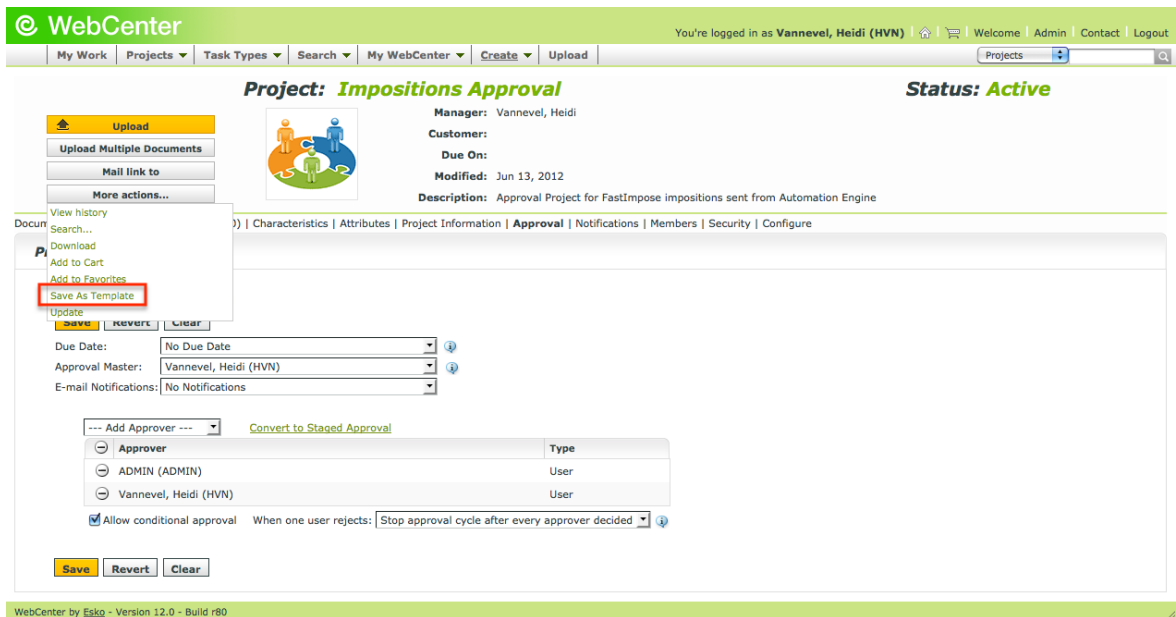
Note: The Project Manager is not automatically made an Approval User and must be explicitly added to the list of Approvers.

WebCenter by Esko - Version 12.0 - Build r80

Saving a Project as a Template

Note: To save a Project as a Template, you need have Administrator permissions.

1. If the user you are logged in with is not an Administrator:
 - a) Log out.
 - b) Log back in with an Administrator user.
 - c) Go back to your Project page.
2. In the **More actions** menu on the left, click **Save as Template**.



The screenshot shows the ESKO WebCenter interface for a project named "Impositions Approval". The status is "Active". The user is logged in as "Vannevel, Heidi (HVN)". The "More actions..." menu is open, and "Save As Template" is highlighted. The project details include: Manager: Vannevel, Heidi; Customer: (blank); Due On: Jun 13, 2012; Description: Approval Project for FastImpose impositions sent from Automation Engine. The approver table is as follows:

Approver	Type
ADMIN (ADMIN)	User
Vannevel, Heidi (HVN)	User

3. Enter the **Template Name** (different from your **Project Name**) and a **Description**.

To use a custom thumbnail, click **Specify New Thumbnail** and enter the complete path and filename in the associated field or **Browse** for it. To use the default thumbnail, leave that choice selected.

WebCenter You're logged in as **Vannevel, Heidi (HVN)** | Home | Welcome | Admin | Contact | Logout

My Work | Projects | Task Types | Search | My WebCenter | Create | Upload | Projects

Create a new template from project

Template name, description and thumbnail **Finish>>**

Type in the template's name and description and select a thumbnail.

Template Name: * [Clear this form](#)
Tip: (* indicates required field)

Description:

Specify New Thumbnail

Use Default Thumbnail

WebCenter by Esko - Version 12.0 - Build r80

4. Click **Finish**.

The Template is created.

Configure Workflow Approval Clients

When and Automation Engine are not on the same LAN, you can use the **workflow approval client** to establish a bi-directional communication. The Automation Engine Pilot needs to have the **workflow approval client** credentials to send messages to .

This is configured through **Admin > Configuration > Workflow Approval Clients** in , and, in the case of Automation Engine, in **Tools > Configuration > Web Sites** in the Automation Engine Pilot.



Attention:

Configure the account first so that it can be tested from within the workflow approval server. and the workflow approval server must use the same user name and password!

This is completely separate from a normal user. Workflow approval clients may not log in to .

Two manual configuration steps must also be performed outside of to ensure proper communication between the workflow approval client and systems that are not on the same LAN.

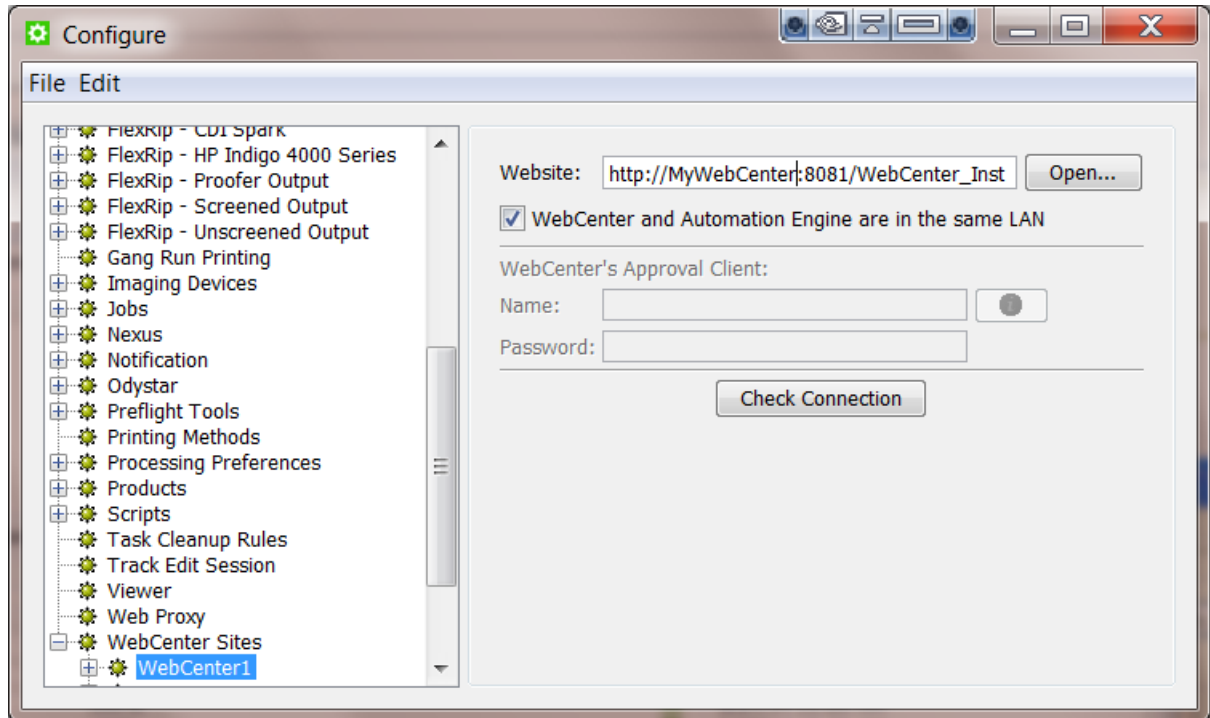
Create a Workflow Approval Client

1. Click **Admin > Configuration > Workflow Approval Clients**.
2. Click **New Workflow Approval Client**.
3. Enter the **User Name** and **Password** for the workflow approval client user.
4. Click **Create**.

Note: If you have several Automation Engine, Nexus, or Odystar sites communicating with , you need to create a separate Workflow Approval Client for each site.

WebCenter Sites

To configure links from Automation Engine to WebCenter sites, go to the item **WebCenter sites** in the **Configure** tool (**Pilot > Tools > Configure**).



Website:

Specify the web address of the WebCenter site. After you entered the WebCenter address, click **Open...** to check if the WebCenter site is up and running. If you are redirected to the login page, it means that the address is correct and that the site is up and running.

Tip: Be aware that this address is also the address that will be included in the notification e-mails when adding links to documents (approval cycle start e-mail). You should therefore make sure that the address can be accessed from the outside. Make sure to not use an internal address, but one that you want your users to use when *they* use WebCenter.

WebCenter and Automation Engine are in the same LAN

Check this toggle if this is true. This will most likely be the case. Un-check this toggle if WebCenter and Automation Engine are remote.

Note: How can you verify that WebCenter and Automation Engine are in the same LAN? WebCenter and Automation Engine are in the same LAN when you can connect from WebCenter's application server, this is the system where JBOSS is installed, to an Automation Engine Job Folder with the credentials of the user 'BGSsystem'.

WebCenter's Approval Client:

If WebCenter and Automation Engine are not in the same LAN, you first need to create an Automation Engine **Approval Client** in WebCenter. This is necessary to give Automation Engine permissions to listen to the approval messages that are stored in WebCenter.

Tip: If the **Approval Client** does not exist yet, follow these steps:

1. In WebCenter, log in as `Administrator`.
2. Go to the **Admin** section. In **Configurations**, select **Workflow Approval Clients**.
3. Click the link "**New Workflow Approval Client**".
4. Enter a **User name** and a **Password** for the Automation Engine client. Mind that this is also the user name and password that have to be entered when configuring the WebCenter site in Automation Engine.

Fill out the client **Name** and **Password** and click the '**Check Connection**' button. This will check if that user can log on successfully.

The **Info** button displays the following information: *"In case Automation Engine and WebCenter are not in the same local area network (LAN), the retrieval of the approval results is secured by a name and password. The name and password specified in Automation Engine must match the name and password specified in WebCenter by the WebCenter administrator. When clicking the 'Check' button, the WebCenter version will be checked and the name and password will be validated by WebCenter."*

Check Connection

When you click this button, you should, after a few seconds, get a window saying "**Check OK**". The software version of that WebCenter will also be mentioned.

Note: The button will check the validity of the user name or password to connect to that WebCenter but **not** of your WebCenter's **Approval Client**.

Note: When you connect to a secured WebCenter site (for example one that uses HTTPS://), then you have to export its security certificate and import it on the Automation Engine server. If this is not done, publishing from Automation Engine to WebCenter will result in an error. Read more on how to do this in the chapter **Install SSL Certificates on the Workflow Production Server** of the WebCenter documentation.

Tip: After configuring a new WebCenter site, it is recommended to close your Pilot and start it again in order to see the updated list of WebCenter sites in your tickets.

Sending an Imposition from Automation Engine to WebCenter

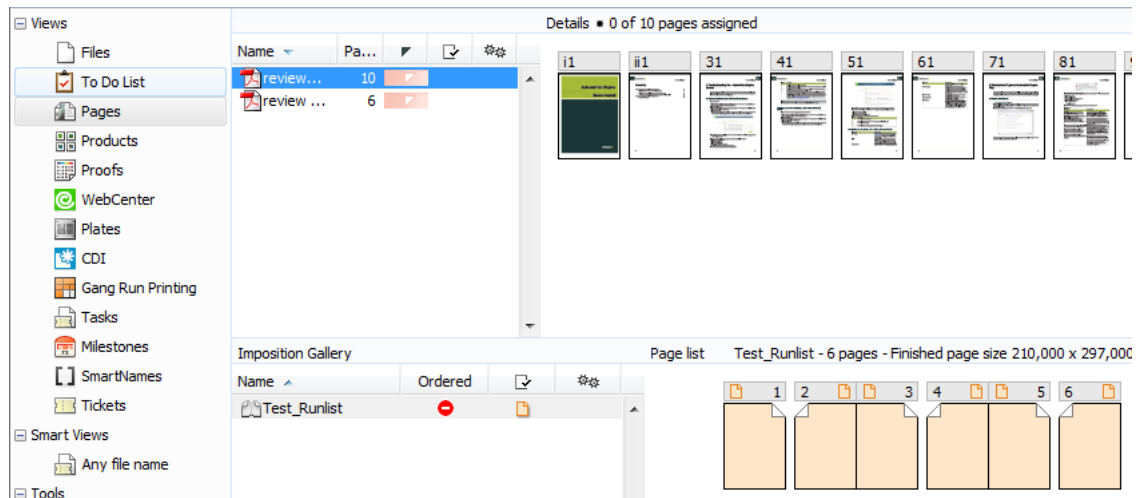
Preparing the Imposition

1. In the **Files** view, browse to the FastImpose imposition file (`*.imp`) you want to upload.

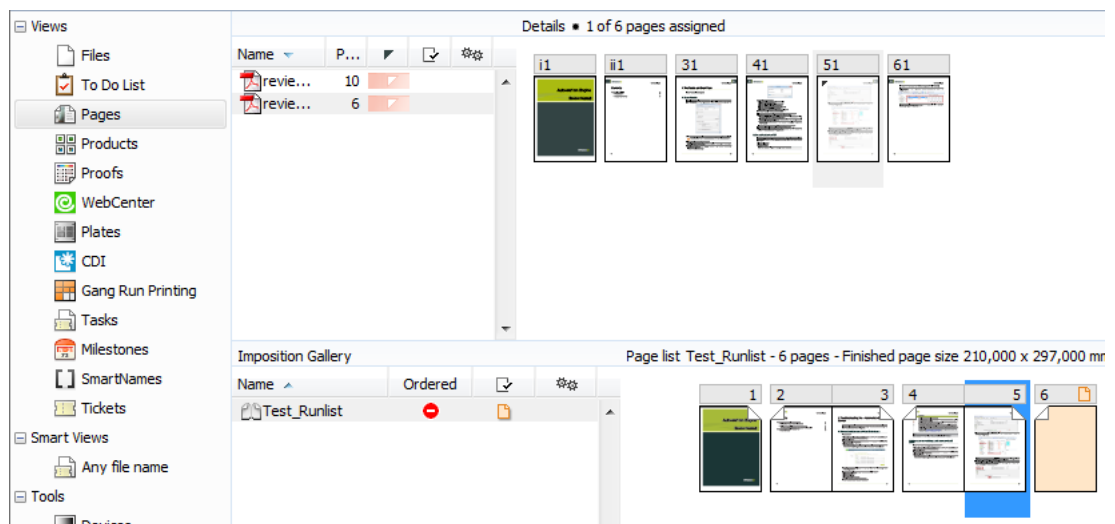
Note: The imposition file must be part of a job.

2. Right-click it and select **Add to Imposition Gallery** from the context menu.

When you switch to the **Pages** view, you can see your file in the **Imposition Gallery**.



3. If the imposition doesn't show the pages preview (as in the previous example), add them as follows:
 - a) Click on the first page's file name in the **Page Gallery**. This shows the page's preview in the **Details** area at right.
 - b) Drag this preview onto the first page of the imposition in the **Page list** area.
 - c) Repeat this for the other pages of your imposition.



Sending the Imposition to WebCenter

If your imposition contains several books, you must send the books one by one to WebCenter. Each book will be shown as a Page List in WebCenter.

1. In the **Imposition Gallery**, right-click the imposition (or the book) and select **New Task > Prepare Imposition for WebCenter Workflow**.

If the **Prepare Imposition for WebCenter Workflow** task does not appear in the context menu, choose **More Tickets** and select the task in the **Select Ticket** dialog.

2. Modify the settings of the **Prepare Imposition for WebCenter** step (the first step in the workflow) if needed.
3. In the **Send Imposition to WebCenter** step, modify the settings of the **Destination** tab (you can use SmartNames where indicated by the SmartNames button):
 - a) Select your WebCenter **Site** from the list.
 - b) Enter the name to give to your **Project**. Automation Engine will create this project in WebCenter, from the **Project Template** you specify below.

Note: You can also send your imposition to a **Project** already existing in WebCenter. In that case, you don't need to enter a **Project Template**. Note that this Project must have Approvers defined in WebCenter.

- c) If you want to send your imposition to a subfolder in the project, enter that **Folder's** name. If this folder doesn't exist in the template (or project), Automation Engine will create it.
- d) Enter the name to give to your imposition **Document**. This is the name that will be shown in WebCenter. For example you can use `[ImpositionName]_[BookName]_[LayerName]`.
- e) Enter the **Project Template** to use to create your WebCenter project.

Important: This **Project Template** must be defined in WebCenter. See [WebCenter Setup](#) on page 83.

- f) Enter the **User Name** and **Password** used to connect to your WebCenter site. You can use an **Encrypted** password or **SmartNames**.

Steps:

Settings for Step: Send Runlist to WebCenter

Destination Approval

Site: RDVMNX01

Project: [Operator]_Proof

Folder:

Document Name: [ImpositionName]_[RunlistName]_[LayerName]

Document Description:

Project Template: TL_AE_Template

Site Logon

User Name: admin

Password: Encrypted:

Use SmartNames:

4. Modify the settings of the **Approval** tab:
 - a) Enable **Start the Approval Cycle**.

- b) Select a **Due date** and time.

5. Click the Launch button in the top left corner to upload the imposition to WebCenter.

Note:

If you don't start the Approval Cycle from Automation Engine, you will still be able to start it from WebCenter, and view the approval results in Automation Engine.

For more information, see the WebCenter manual.

Approving the Imposition in WebCenter

In WebCenter 7.1, you can:

- Approve / reject documents sent from **Odystar**, **Nexus** or **Automation Engine** as regular documents (see [Approving a Document](#) on page 95).
- Approve / reject documents **FastImpose** impositions added to the **Pages View** in **Automation Engine** as regular documents or **Page Lists** (see [About Page Lists](#) on page 95 and [Approving a Page List](#) on page 96).

When approving a Page List, you can:

- Approve / reject not only the whole multipage imposition but each individual page.
- Update and have version control not only on the whole multipage imposition but on each individual page.

Note: Only **FastImpose** impositions that have been added to the **Pages View** in Automation Engine are recognized as **Page Lists** in WebCenter. Multipage files uploaded from within WebCenter, or from Odystar, Nexus, or even other Automation Engine documents will never be treated as Page Lists. Instead, they are considered regular multipage PDF documents.

When approving a Document, you can:

- If you are an **Approval User**: **Approve** the document, **Reject** it, or leave it **Pending** to approve / reject it in the future.
- If you are a **Project Manager**: **Approve** or **Reject** the document, leave it **Pending**, or **Force Approve** / **Force Reject** it if you need to overrule the approval status granted by the approval users.

Note:

For more information about Forced Approval or Rejection, see the WebCenter manual.

Approving a Document


Documents waiting for your approval are shown on My To Do List under My Work. If you are an Approval User, do the following to Approve or Reject a document:

Note:

You can also approve or reject documents in the **Viewer**.

1. Click **Approve/Reject** at the end of the **Approval** field in the document header.

Document - dollyBook 1



Document: [dollyBook 1](#) (3.7KB) **Pages:** 4

Version: [1](#) **Binding:** Left Edge

Description:

Project: [Imposition MrChen 110409](#)

Approval: Cycle Started - Status Pending [Approve/Reject](#)

2. Choose the desired **Approval Status** from the drop-down list: *Approved* indicates approval; *Pending* indicates future Approval or Rejection, and *Rejected* indicates disapproval.
3. Once you choose **Approved** or **Rejected**, you may not change the status later; a view-only version of the page will be shown showing the choice you picked. Only a **Pending** status allows future changes.

If desired, you can add a comment in the **Comment** field. Submitted approvals and comments are shown on the **Action History** page.

4. Click **Finish** to change the status of the document.

To cancel the changes, navigate to another page without clicking Finish.

About Page Lists

What is a Page List document?

Page Lists are a special type of multipage PDF documents in WebCenter that represent imposed books or magazines. As such they are an important tool for commercial printers.

Page Lists are uploaded from a Automation Engine production server using the **Pages View** in the Automation Engine Pilot.

Note: Only **FastImpose** impositions that have been added to the **Pages View** in Automation Engine are recognized as **Page Lists** in WebCenter. Multipage files uploaded from within WebCenter, or from Odystar, Nexus, or even other Automation Engine documents will never be treated as Page Lists. Instead, they are considered regular multipage PDF documents.

Page List documents have two major advantages over other multipage PDF files:

- You can **upload either the complete document, or specific pages** only (for example, pages 6-12);
- **individual pages can be approved or rejected** in WebCenter (for example, you can approve pages 3, 5 and 7-16 of a 16-page Page List).

Note: A multipage PDF file can only be approved or rejected as a whole.

Working with Page Lists

Page List documents are **displayed as a single document** in document lists. Click the Page List name to view its pages and details in the dedicated Page List views.

Note: Page Lists cannot be downloaded.

Approving a Page List

In order to approve your imposition as a Page List in WebCenter, you need to first login to WebCenter as an approver, then navigate to the document (Page List) to approve, in the project created from Automation Engine.

You can also access this Page List from the link in the approval notification email, if you received one.

Make sure you have the correct permissions to approve documents in WebCenter. For more information, please see your WebCenter manual.

You can choose from a number of different methods to approve the pages in a Page List:

- **Use the WebCenter Viewer:**

- In the Project, click on the Page List's thumbnail image. The **Viewer** opens, displaying the Page List.
- In the **Approval** area in the sidebar on the right of the Viewer window, select the appropriate approval state (Approved, Rejected, Pending) from the list.

The **Approval** dialog opens in which you can add approval comments.

- Enter your comments, provide your password if required and click **Commit**.

Note: You can approve / reject one page at a time, or all of the document's pages (using **Apply to all pages**).

- **Use the Page Gallery view:**

a) In the Project, click on the Page List's name.

The **Document Details** page appears.

b) Select **Page Gallery** from the menu.

c) Select the pages you want to approve and then click **Approve/Reject**. You can also use **Approve/Reject all Pages**.

[Approve/Reject](#)



[Approve/Reject all Pages](#)

d) Enter your comments, provide your password if required and click **Commit**.

- **Use the Page List view:**

a) In the Project, click on the Page List's name.

The **Document Details** page appears.

b) Select **Page List** from the menu.

c) Select the pages you want to approve and then select **Approve/Reject** from the gearbox menu. You can also use **Approve/Reject all Pages**.

[Hide Thumbnails](#)

[Approve/Reject all Pages](#)

<input type="checkbox"/>	Status	Index	Folio	Sheet Name	Sheet Side	View	Page Version	List Version	Uploaded Date	Approve/Reject	Annotation
<input type="checkbox"/>		1	1	sheet1	2		1	1	Apr 10, 2009 at 3:10 PM	Approve/Reject	
<input checked="" type="checkbox"/>		2	2	sheet1	1		1	1	Apr 10, 2009 at 3:10 PM	Approve/Reject	
<input checked="" type="checkbox"/>		3	3	sheet1	1		1	1	Apr 10, 2009 at 3:10 PM	Approve/Reject	
<input checked="" type="checkbox"/>		4	4	sheet1	2		1	1	Apr 10, 2009 at 3:10 PM	Approve/Reject	

d) Enter your comments, provide your password if required and click **Commit**.

- **Use the Approve/Reject link on the Document properties page to approve all the pages at once.**

Document - dollyBook 1



Document: [dollyBook 1](#) (3.7KB)

Pages: 4

Version: 1

Binding: Left Edge

Description:

Project: [Imposition MrChen_110409](#)

Approval: Cycle Started - Status Pending

[Approve/Reject](#)

a) Enter your comments, provide your password if required and click **Commit**.

Sending a RunList / PageList to WebCenter

You can send a RunList from Automation Engine to WebCenter without having to create an Imposition. The resulting **PageList** will have pages in the right order. These pages are sent to Automation Engine after approval.

In PageLists, information such as Sheet info is not available.

Use this workflow if you want to send pages to WebCenter in the right order to get them approved.

Note: Unlike sending impositions, you still need to position them afterwards.

The uploaded PageList in WebCenter takes the Sheet Name **NoImposition**. The Sheet Name and the Sheet Side will always be 0.

Preparing a RunList

Context for the current task

Run the **Create RunList** task on a multipage PDF file. Remember to enter a name for your RunList. If the **Name** field is left blank, the task uses the file name.

Read more about the task settine in [Create RunList from PDF](#) on page 300.

A RunList is created and shown in the **Imposition Gallery**. Switch to the **Pages** view to see the RunList and its pages.

Sending A RunList to WebCenter

1. In the **Pages View**, right-click the RunList and select **New Task > Publish Imposition on WebCenter** .
2. Modify the settings of the **Publish Imposition on WebCenter** chain task.
Read more in [Publish Imposition on WebCenter Chain](#) on page 445
3. Click the Launch button in the top left corner to upload the RunList to WebCenter.

Note:

If you don't start the Approval Cycle from Automation Engine, you will still be able to start it from WebCenter, and view the approval results in Automation Engine.

For more information, see the WebCenter manual.

The RunList will be uploaded to WebCenter. If approval is required the approval feedback is visible in page and proofs views.

Note: In the **pages view**, the approval status is indicated in a color on top of the page. When you hover over the page, the details of the approval becomes visible.

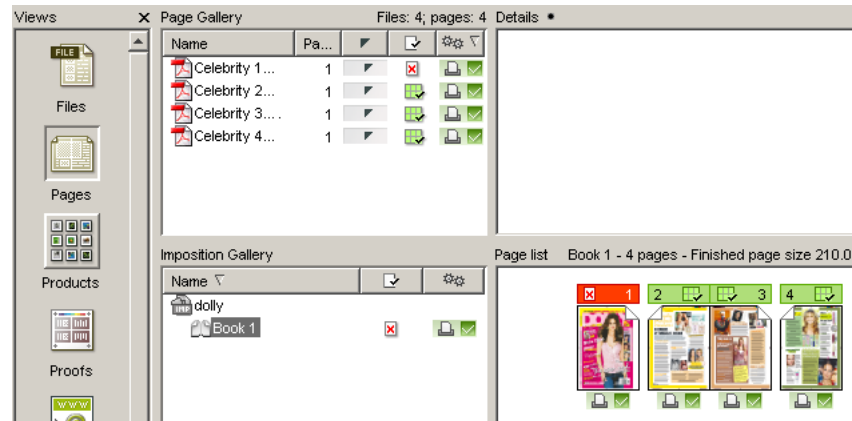
Checking the Approval Results in Automation Engine

- To view the status of documents in a WebCenter Approval Cycle in Automation Engine, you can use either:

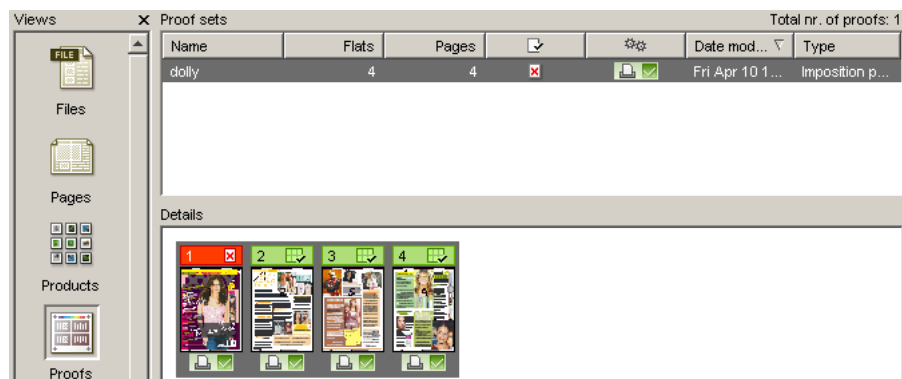
- the **Pages** View,
- the **Proofs** View,
- the **WebCenter** View.

In the Pages and Proofs Views

- Go to the **Pages** or **Proofs** View to view the approval status of **each page** of the imposition.
- In the **Pages** View:



- In the **Proofs** View:



Note:

When you try to expose a sheet that contains a rejected page, you will get an error in the **Expose** task and an error status in the **Plates** View.

If you really need to expose the sheet anyway, you can overrule the WebCenter approval status from Automation Engine (see [Overruling the WebCenter Approval](#) on page 102).

In the WebCenter View

In the **WebCenter** View, you can view the approval status of **the whole imposition**.

Important: You need to first set up the WebCenter View before you can view all the approved / rejected documents in the WebCenter project you created.

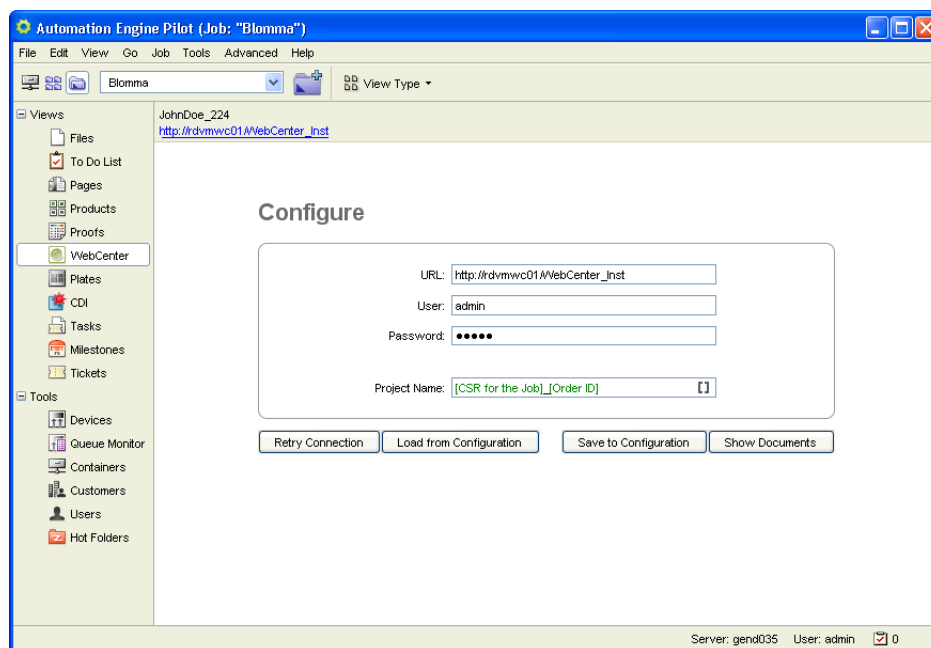
1. Right click the **WebCenter** View's background and choose **Configure**.
2. Enter the **URL** of your WebCenter site.
3. Enter the **User** name and **Password** used to connect to your WebCenter site.
4. In **Project Name**, enter the name of the WebCenter project you sent your imposition to. You can use SmartNames.

Important: Make sure this matches the **Project Name** of the project you created (or used) when sending your imposition from Automation Engine to WebCenter. See [Sending the Imposition to WebCenter](#) on page 92.

5. To check that the information you entered is correct and that the WebCenter server is running, click the **Retry Connection** button at the bottom of the view.

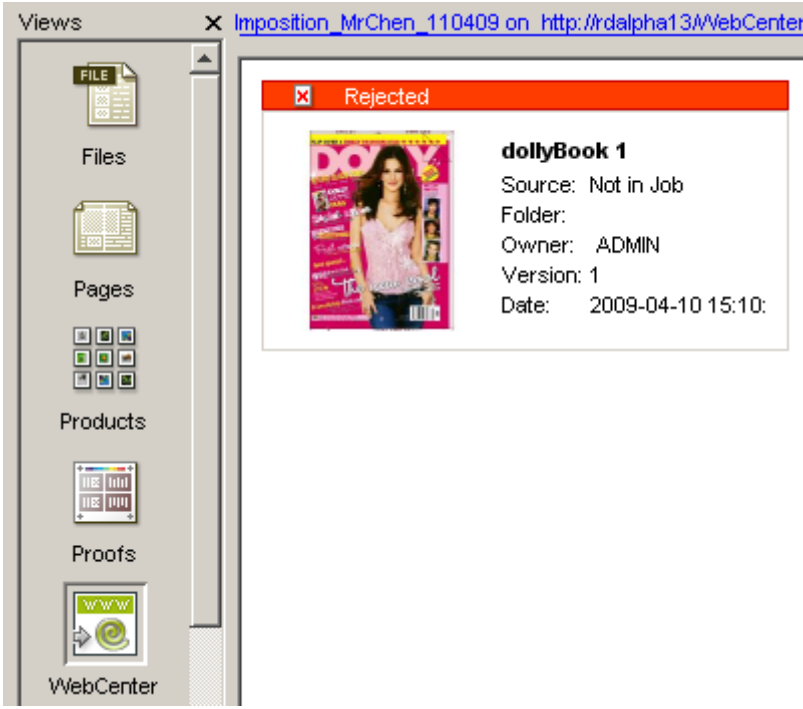
You can also use the link at the top of the view to open the web page of the WebCenter project.

6. Once you have checked that your configuration is correct, use the **Save to Configuration** button to save it.



Once it is saved, you will be able to return to this configuration using the **Load from Configuration** button after any unwanted change.

7. Click the **Show Documents** button to view the documents in your WebCenter project.



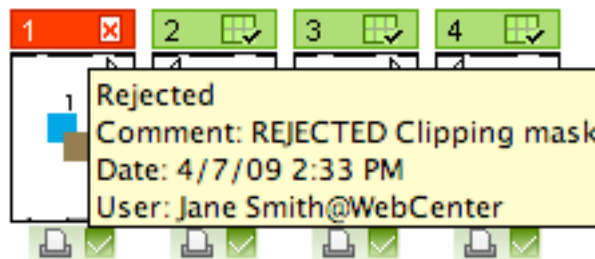
Note: If at least one page of the imposition is rejected, the status of the whole imposition is **Rejected**.

Possible Document Statuses

This icon...	means that...
	the page and the original imposition layout have been approved by all approvers. Note: The imposition layout can be approved even if some of its pages are rejected.
	the page has been approved by all approvers, but it may have changed the imposition layout, which needs to be checked (for example for a new page replacing a rejected one).
	the page or the whole imposition has been rejected by at least one approver.
	the page or the whole imposition has yet to be approved or rejected by at least one approver.



When you hover on the top of a page in the **Pages** or **Proofs** View, you can see:

- the approval status,
- any comment entered,
- the date and time of the approval,
- the WebCenter user who approved/rejected the page.



Overruling the WebCenter Approval

After your imposition has been approved or rejected in WebCenter, you can still overrule the approval status of some of the pages in Automation Engine, if necessary.

1. In **Automation Engine Pilot**, go to either the **Pages** or the **Proofs** View and locate your imposition document.
2. You can either:
 - Select a page of your document and use the  and  buttons in the View's toolbar to approve/reject the document without entering a comment.
 - Right-click on a page of your document and choose **Approve/Reject...** to open the **Approve/Reject** dialog where you can enter a comment.
3. In the **Approve/Reject** dialog, select **Manual** as **Approval Type**.



4. Choose to either:
 - **Approve**,
 - **Approve with warning**,

- **Reject** the document.
5. Enter a **Comment** if you wish and click **OK**.

Note: You can hover on the top of a page to view approval information, whether it has been approved/rejected automatically in WebCenter, or manually in Automation Engine.

1.9.2 Integration with Nexus

Why run Nexus workflows from the Automation Engine Pilot?

You can use all the power of Nexus workflows from within a Automation Engine Job. This allows you to bring both Automation Engine and Nexus workflows in line, running them from within a single controlling environment: the Automation Engine Pilot.

How does it work?

The **Send to Nexus** ticket gives you access to the workflows that are available on a specific Nexus server.

Any public parameters that have been defined in the Nexus workflow (NeXML) will show up on the **Send to Nexus** ticket in Automation Engine. In addition, these public parameters are SmartName-enabled, so you can use Job metadata to drive the Nexus workflows.

On the Nexus side the **Copy to BackStage** activity allows you to set up workflows that send their resulting files back to Automation Engine.

Behind the scenes, Automation Engine and Nexus communicate via the SOAP protocol, exchanging files and notifying each of other of their progress.

NeXML

Setting up public parameters in Nexus requires knowledge of NeXML. Please refer to the NeXML Manual for more information.

Setup on the Nexus Side

To enable the Automation Engine Pilot to launch Nexus workflows, you must perform a number of setup tasks on the Nexus server.

Mounting Your Containers on the Nexus Server

The Automation Engine containers have to be mounted on the Nexus server.

- If the Nexus server runs on a Windows OS, the container should be mapped to a drive letter on the operating system.
- If the Nexus server runs on a Mac OS, the container has to be mounted over the SMB protocol.

Setting up the Nexus SOAP Server and Workflows

1. In the Modules list, double-click the **NexusSoapServer** module.
2. Enter a port number in the **Listening Port** field of the **NexusSoapServer Module Defaults** dialog, and click **OK**.

The Nexus SOAP server is now active.

Note:

The option **User Management**, which determines the workflows that a particular user can access, is ignored by the Automation Engine Pilot. As a result, Automation Engine Pilot operators automatically have access to all Nexus workflows.

3. Create valid Nexus workflows with the **Input Type: Hot Folder**.
4. Configure the Hot Folder queues for the workflows. Make sure the option **Show in Shuttle** is selected.

Only Hot Folders that have **Show in Shuttle** enabled are published to the Nexus SOAP server and become accessible to the Automation Engine Pilot.

5. Create the workflows with the **Nexus Workflow Editor**.
6. Make sure you add the **Copy to BackStage** activity at the end of the workflow.

Your Nexus workflow is now available for use with the Automation Engine Pilot.

Setup on the Automation Engine Side

1. In the Pilot, go to **Tools > Configure** and select the **Nexus** category.
2. Click **File > New**.
3. Enter the name of the Nexus server in the **Host** field.
4. Enter the **Port** on which the Nexus server is accessible.
5. Click **Test Connection** to verify the settings you entered.

Note: You can also test whether the Nexus SoapServer is online by browsing to the following url in your favorite web browser: `http://NexusServerName:5182`.

Running a Nexus Workflow from the Pilot

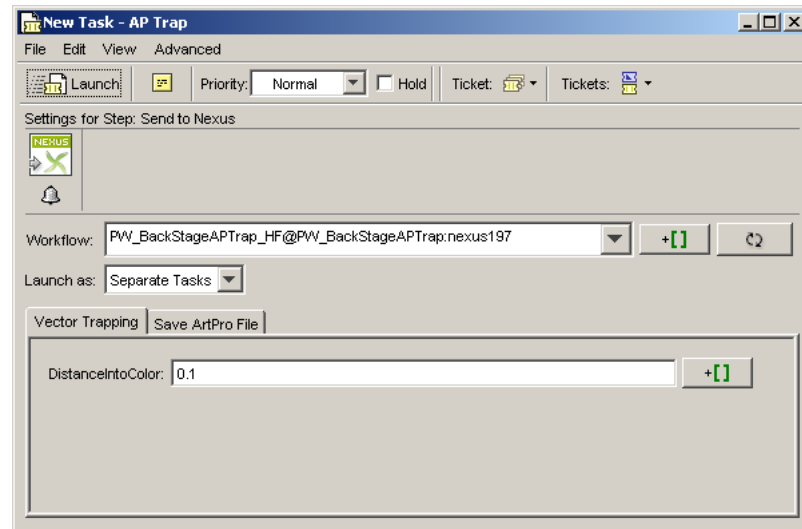
To run a Nexus workflow from within the Automation Engine Pilot, you can use the **Send to Nexus** ticket.

Additional integration features allow you to check the progress of the workflow in the Automation Engine Task Monitor, to jump to the workflow in Nexus Manager, and more.

Understanding the Send to Nexus Ticket

The **Send to Nexus** ticket consists of two parts:

- the top section of the ticket dialog is always the same, and allows you to:
 - select a particular Nexus workflow from the **Workflow** list;
 - indicate whether you want to launch a new, separate workflow in Nexus for every file you select in the Pilot (**Separate Tasks**), or whether you want to send all the selected files to one Nexus workflow (**One Single Task**).
- the bottom section of the ticket dialog contains fields for the **public parameters** for the workflow you selected. These parameters can be different for every Nexus workflow (they are defined using NeXML).



Note: You can also manage your Nexus workflow's public parameters (not one by one but as a group) in your task ticket. See [Managing Public Parameters in Your Ticket](#) on page 70.

Launching a Nexus Workflow from the Pilot

1. In the Pilot, select the file(s) you want to process with your Nexus workflow.
2. Launch the **Send to Nexus** task.
3. Select the **Workflow** you want to run.
4. If you selected multiple files:
 - select **One Single Task** to send all the files to a single workflow in Nexus.
 - select **Separate Tasks** to launch a separate Nexus workflow for every selected file.
5. Provide the required **public parameters** for the different activities in the workflow.

Note: You can use **SmartNames** in all the public parameter fields.

6. Click **Launch** to send the files to the Nexus workflow.

The **Send to Nexus** task appears in the **Task List**, where you can see the progress of the Nexus workflow. Note that progress is not displayed in real time.

The processed files from Nexus are returned to Automation Engine as the output of the **Send to Nexus** task. In other words, you can include **Send to Nexus** as a step into a more complete workflow.

Switching Between Automation Engine and Nexus

- In the **Automation Engine Pilot**: in the Task List, right-click on the Send to Nexus task and choose **Go To Job in Nexus Manager** to view the job in Nexus Manager (this command is only available on Mac OS).
- In **Nexus Manager**, double-click the **Copy to BackStage** activity to view the job in the Automation Engine Pilot.

Note: In order to jump from the Nexus Manager 9.0 to the Automation Engine Pilot, you must connect to Nexus Manager with a blank user name and the password `awsnexus`.

1.9.3 Integration with Odystar

Why Run Odystar Workflows from the Automation Engine Pilot?

You can use all the power of Odystar workflows from within a Automation Engine Job. This allows you to bring both Automation Engine and Odystar workflows in line, running them from within a single controlling environment: the Automation Engine Pilot.

How Does it Work?

The **Send to Odystar** ticket gives you access to the workflows that are available on a specific Odystar server. You can use the **Send to Odystar** ticket as a single task or as part of a more complete Automation Engine workflow.

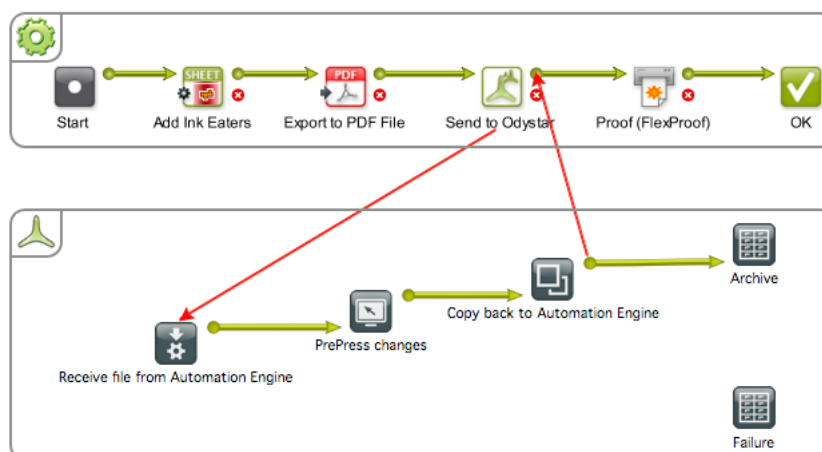
Any public parameters that have been defined in the Odystar workflow will show up in the **Send to Odystar** ticket in Automation Engine. In addition, these public parameters are SmartName-enabled, so you can use Job metadata to drive the Odystar workflows.

On the Odystar side the **Copy File** gateway allows you to set up workflows that send their resulting files back to Automation Engine.

Behind the scenes, Automation Engine and Odystar exchange files and notify each of other of their progress.

Example

- The Automation Engine workflow (top) sends the files to Odystar for additional processing before outputting them to a proofer.
- The Odystar workflow (bottom) applies PrePress changes to the files before copying them back to Automation Engine, where they go through the rest of the workflow (Proof step).



Note:

Odystar accepts Normalized PDF files as input, but its changes may render the files non-Normalized. We suggest that you do either (or both) of the following:

- send regular (not Normalized) PDFs to Odystar (add an **Export to PDF File** step before **Send to Odystar**),
- re-normalize the files you get back from Odystar (add a **Normalize PostScript / PDF / Illustrator 8.0 File** step after **Send to Odystar**).

Setup on the Odystar Side

To enable the Automation Engine Pilot to launch Odystar workflows, you must perform a number of setup tasks on the Odystar server.

Mounting Your Containers on the Odystar Server

- Mount the Automation Engine containers on your Odystar server using the SMB protocol.

Note: Make sure you have Read and Write permissions for those mounted volumes from your Odystar server machine.

▼ Sharing & Permissions:
You can read and write

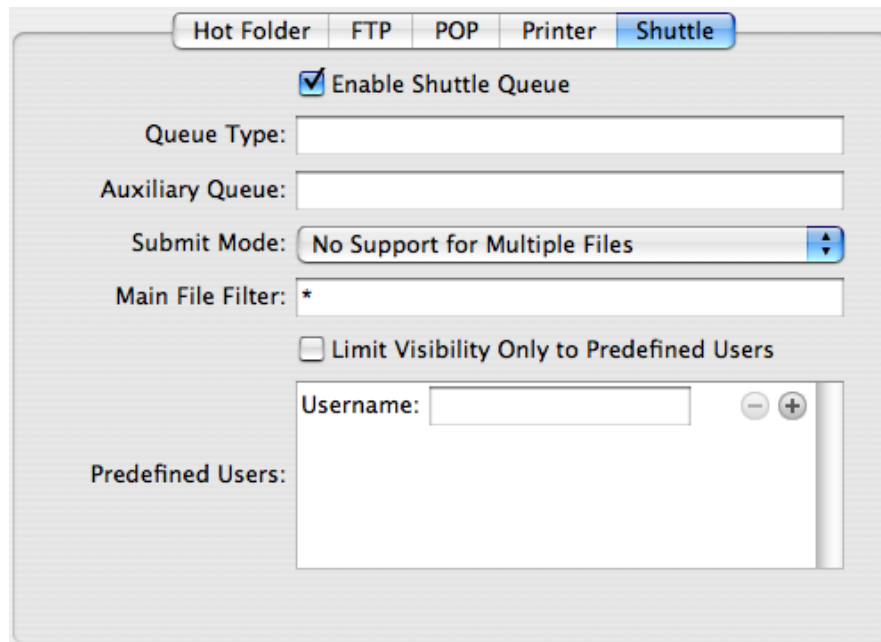
Setting up the Hub and Your Odystar Canvases

1. On your Odystar server, launch the **Hub** application. You can find it in Odystar's **Server Software** folder.
2. Go to **Hub > Preferences**.
You can also use the shortcut **Command + , (comma)**.
3. In the **Shuttle** tab, enter the **IP Address** of the server running your Hub, and the port to use for communication between Automation Engine and the Hub (in **Shuttle Port**).

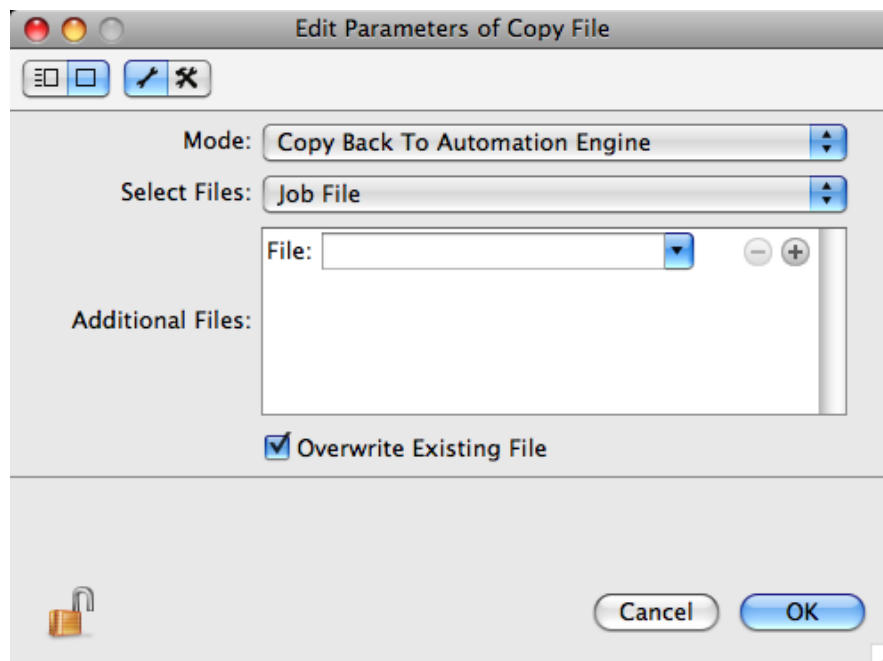
Server:	
IP Address:	10.31.132.171
Shuttle Port:	5182
Bonjour Name:	Odystar Hub - Mac Intel 1

Note: By default this is port 5182, but if this port is already used by other processes, you should enter the next available port.

4. Create your Odystar workflow using the **Inspector**.
5. Double-click the **Receive** Process Folder of your Odystar workflow to open its parameters.
6. In the **Shuttle** tab, click **Enable Shuttle Queue**.



7. Make sure you add a **Copy File** gateway, with the **Copy Back To Automation Engine** mode selected, at the end of your workflow.



Your Odystar workflow is now available for use from the Automation Engine Pilot.

Setup on the Automation Engine Side

1. In the Pilot, go to **Tools > Configure** and select the **Odystar** category.

2. Click **File > New**.
3. Enter the name or IP address of the Odystar server in the **Host** field.
4. Enter the **Port** on which the Odystar server is accessible.
5. Click **Test Connection** to verify the settings you entered.

Note: You can also test whether the Odystar Hub is online by browsing to the following url in your favorite web browser: `http://OdystarServerNameorAddress:5182`.

Running an Odystar Workflow from the Pilot


To run an Odystar workflow from within the Automation Engine Pilot, you can use the **Send to Odystar** ticket. You can use it as a single task or as part of a more complete Automation Engine workflow.

You can also check the progress of the Odystar workflow in the Automation Engine Task Monitor.

Understanding the Send to Odystar Ticket

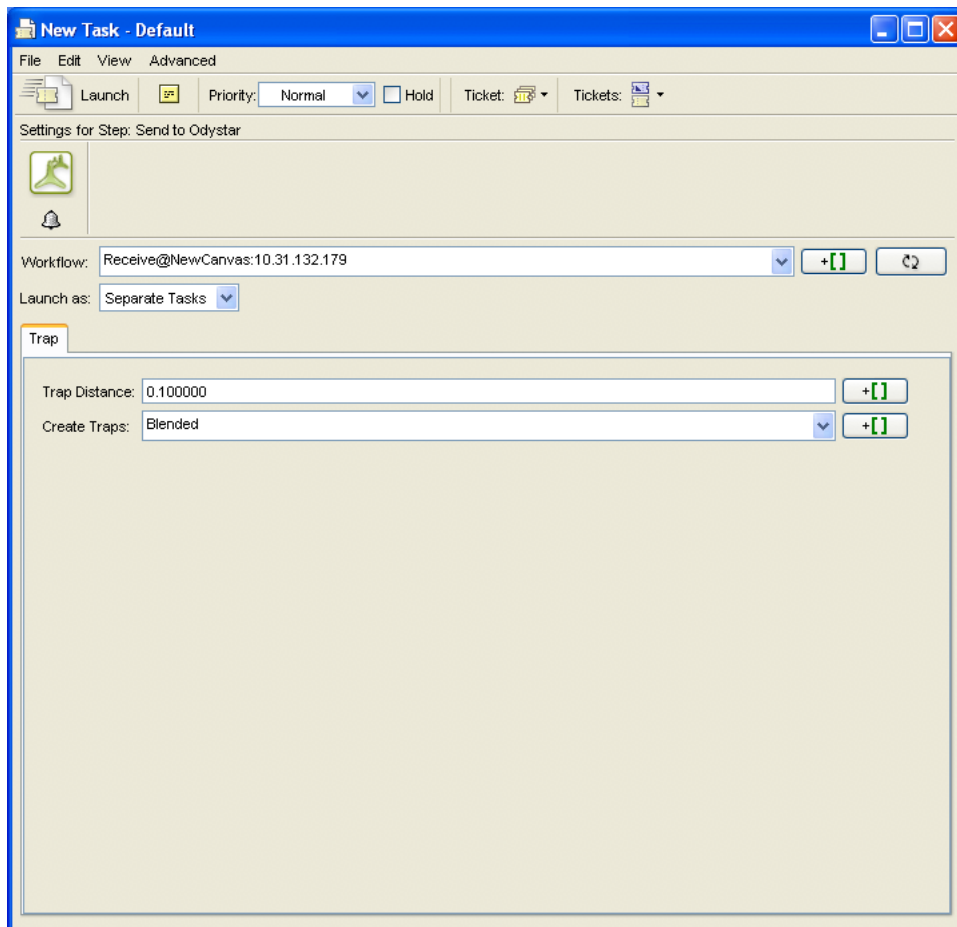
The **Send to Odystar** ticket consists of two parts:

- the top section of the ticket dialog is always the same, and allows you to:
 - select a particular Odystar workflow from the **Workflow** list;

Tip: If you can't see the workflow you are looking for, use the refresh button .

- indicate whether you want to launch a new, separate workflow in Odystar for every file you select in the Pilot (**Separate Tasks**), or whether you want to send all the selected files to one Odystar workflow (**One Single Task**).
- the bottom section of the ticket dialog contains fields for the **public parameters** for the workflow you selected. These parameters can be different for every Odystar workflow (they are defined in the Inspector).

Note: You can also manage your Odystar workflow's public parameters (not one by one but as a group) in your task ticket. See [Managing Public Parameters in Your Ticket](#) on page 70.



Launching an Odystar Workflow with the Send to Odystar Task

1. In the Pilot, select the file(s) you want to process with your Odystar workflow.
2. Launch the **Send to Odystar** task.
3. Select the **Workflow** you want to run.
4. If you selected multiple files:
 - select **One Single Task** to send all the files to a single workflow in Odystar.
 - select **Separate Tasks** to launch a separate Odystar workflow for every selected file.
5. Provide the required **public parameters** for the different gateways in the workflow.

Note: You can use **SmartNames** in all the public parameter fields.

6. Click **Launch** to send the files to the Odystar workflow.

The **Send to Odystar** task appears in the **Task List**, where you can see the progress of the Odystar workflow. Note that progress is not displayed in real time.

Note: The files processed by Odystar are returned to Automation Engine as the output of the **Send to Odystar** task (or as the output of the **Send to Odystar** workflow step when including the **Send to Odystar** ticket in an Automation Engine workflow - see the workflow [example](#)).

1.9.4 Integration with Enfocus PitStop

You can integrate a separate install of Enfocus PitStop with Automation Engine, to be able to preflight PDF files from the Pilot using the task **Preflight via External Application**.



Caution: Depending on your licenses, check if your Automation Engine already has the PitStop software built in. This then enables the **Preflight with PitStop** task.

To preflight a file using a separate software install of Enfocus PitStop:

1. Go to **Tools > Configure** and set up the connection to PitStop (see **Preflight Tools** in the Reference Guide for more details).
2. Launch the **Preflight via External Application** task on your file. Learn more in [Preflight via External Application](#) on page 405.

1.9.5 Integration with Third Party Applications using Hot Folders

The **Integrate via External Hot Folder** task allows you to integrate external prepress applications into your workflow tickets. The only requirement is that the external system works with hot folders.

1. In the Integrate via External Hot Folder ticket, you define the location of the hot folder used by the external application, as well as the Result subfolders in which the external application will output processed jobs (the **OK Folder**), failed jobs (**Error Folder**) or jobs that produced warnings (**Warnings Folder**).
2. When you process a job with this ticket, Automation Engine copies the files to the external hot folder.
3. The external application picks up the files in its hot folder and processes them. The processed files are saved in the subfolder that matches the processing result: OK, Warning or Error.
4. Automation Engine monitors the OK, Warning and Error folders for any files that have been processed.
5. If it detects a file in the OK folder, Automation Engine copies it to the Output folder defined in the Integrate via External Hot Folder ticket, and continues its workflow. Depending on your settings, Error or Warning files might also be copied into the Output folder, or they might be deleted.

The screenshot shows a software interface window titled "New Task - Integrate via External Hot Folder(job)". The window has a menu bar with "File", "Edit", "View", and "Advanced". Below the menu bar are several controls: a "Launch" button, a "Priority" dropdown set to "Normal", a "Hold" checkbox, a "Ticket" dropdown, and a "Tickets" dropdown. The main area is titled "Settings for Step: Integrate via External Hot Folder". It contains several sections:

- Output:** "Output in:" field with value "[Job URL]/Processed" and a "Browse..." button; "File Name:" field with value "[File]_Processed" and a "+[]" button.
- Hot (External):** "Folder:" field with value "file://myserver/PitStop_HotFolder/Input" and a "Browse..." button; "File Name:" field with value "[File]" and a "+[]" button.
- Result:**
 - "OK Folder:" field with value "file://myserver/PitStop_HotFolder/Success" and a "Browse..." button; "OK File Name:" field with value "[File]" and a "+[]" button.
 - "Warning Folder:" field (checkbox is unchecked) and "Warning File Name:" field, both with "Browse..." and "+[]" buttons.
 - "Error Folder:" field (checkbox is checked) with value "file://myserver/PitStop_HotFolder/Error" and a "Browse..." button; "Error File Name:" field with value "[File]_Error" and a "+[]" button.
 - "Error when no result after" dropdown set to "1 hour", with two empty dropdowns below it.
 - Radio buttons for "Delete error files" (selected) and "Move error files to output folder".

Note: In the Error area of the ticket, you can define an amount of time after which the external processing should be considered as failed. To do this, enable the option **Error when no result after ...** and define a time period (for example, you could enter one hour, a number of days, or even a specific date).

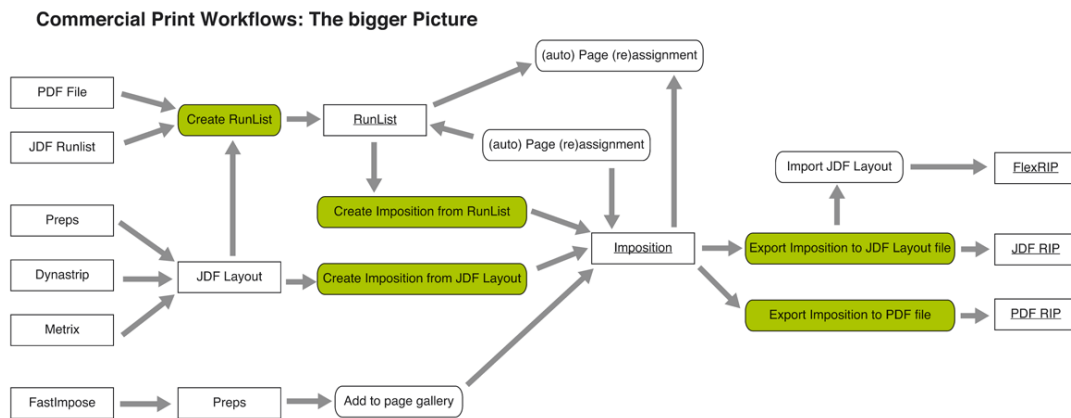
1.10 Integrating with Third Party Imposition Systems

Automation Engine enables working with non-Esko imposition systems. These tasks import or export industry standard file types to or from the Impositions in the **Pages** and **Plates** views.

You can check or change (approval) statuses, assign pages and select elements for further processing or output in the **Pages** and **Plates** views in Automation Engine.

1.10.1 Workflows: The bigger picture

You can find a schematic overview of how impositions can be created, processed and exported below. The tasks highlighted in green are new in Automation Engine 12.



You can import populated or unpopulated impositions from applications that produces JDF layout file as its output. Some of the applications that can write a JDF Layout file are Kodak Preps, Litho Technics Metrix, Ultimate ImposStrip , Dynagram Inpo2 , Dynagram Dynastrip, Plato and Pandora.

1.10.2 Importing and Creating Impositions

You can import the JDF Layout and use the **Create Imposition from JDF Layout** task to bring the **Imposition** to the **Pages** and **Plates** views in Automation Engine.

If you start with a PDF file or a JDF Runlist, you can use the **Create RunList** task to create a Runlist in Automation Engine. Subsequently, you can use the **Create Imposition from RunList** task to create Imposition from this RunList.

It is possible to separate the creation of a RunList and the creation of an Imposition from that Runlist in time. In some cases printers prefer to let the RunList be approved and corrected before they select the layout and create the Imposition.

You can achieve this by using two workflows.

1. Assemble the RunList and send it for approval.



2. Create the Imposition from the RunList and export it:



Read more about the new importing and creating tasks in the Reference Guide:

[Create Imposition from JDF Layout](#) on page 282

[Create RunList from PDF](#) on page 300

[Create Imposition from RunList](#) on page 283

General Remarks

Note: File Compatibility

- JDF layout files: tests have been done with Preps, Metrix, ImpoStrip, InpO2, DynaStrip, Plato and Pandora.
- PDF files: Normalized as well as non Normalized files can be used in the Imposition workflows. However, third party rips and workflows do not support external images.

Note: Thumbnails in the **Pages** and **Plates** views are only shown when the pages in the files contain thumbnails. If you have no thumbnails in your PDF files, you can speed up the system. You can also save some screen space by hiding the details pane and by switching to **list** mode.

1.10.3 Exporting Impositions

You can use **Export Imposition to JDF Layout** task to send an Imposition or part of an Imposition to another workflow system or to a Rip that is compatible with JDF layout.

- The output is JDF layout containing a JDF runlist.
- You can create a runlist for the **Join Pages** task using this output. This allows you to create a multipage PDF file to send to your customer for approval.
- It can be used to drive the FlexRip when followed by the **Import JDF Layout** task.



You can send an Imposition or part of an Imposition to a PDF Rip (for example Nexus Rip) using **Export Imposition to PDF** task. The output will be a PDF file containing the Imposition.



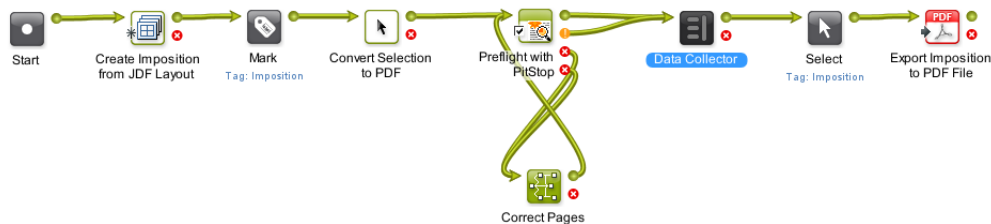
More info regarding these exporting tasks in:

[Export Imposition to JDF Layout](#) on page 321

[Export Imposition to PDF File](#) on page 322

1.10.4 Workflows with Selection and the Convert Selection tasks

Page, RunList, Layer, PageSlot, Imposition and Plate are abstract file types that do not correspond directly to a file. They can be used as entities in a workflow to route the elements you want to process through the workflow. For example, you can import an Imposition and preflight all the PDF files referred from that Imposition. The workflow below accepts a JDF Layout as its input, selects the PDF files referred in the JDF file and waits until all of them pass the preflight check before producing outputs as sheets to a PDF file. You need to convert the Selection to a PDF before Preflight.



Read more about **Convert Selection** task in [Convert Selection](#) on page 270 the Reference Guide.

1.11 PDF Processing in Automation Engine

1.11.1 PDF Normalization in Automation Engine

Normalization in Automation Engine 10 and older

In Automation Engine 10 and older versions, PDF files had to be normalized at the start of a workflow before any further processing could take place.

With the release of Automation Engine, 10.1 new PDF processing tasks were introduced that can handle incoming PDF files without the need for normalization. However, these tasks could not process normalized PDF files. On top of this the PDFs produced by these tasks needed to be normalized before they can be handled by other tasks. As a consequence, it was not straightforward to build workflows that used a combination of these tasks.

Improvements from Automation Engine 12 onwards

From Automation Engine 12 onwards, the collection of PDF processing tasks that do not require normalization was extended. To avoid that the user needs to carefully consider which tasks require normalized PDF as input when building workflows 2 features were introduced in Automation Engine 12:

- Tasks that **do not require normalized PDF** (the new ones added in Automation Engine 12 and the ones that were added in Automation Engine 10.1) can now also handle normalized PDF, and keep the files normalized.
- Tasks that **require PDF files to be normalized will now do this automatically** without the need to introduce explicit normalization tasks in a workflow.

What Makes a Normalized PDF Special?

Overview

Normalized PDF differs from regular PDF 1.6 in the following areas:

- External references
- Color space
- Document level metadata

- Object level metadata

External references

A Normalized PDF is not always self-contained. It can refer to images or other PDFs that are not embedded in the PDF itself. Making use of references instead of embedding images and/or graphics can have several advantages in the workflow. The main advantage is that it allows for “late binding” for example an image can be color corrected. After the color correction all PDFs that refer to this image will automatically contain the corrected version of the image without the need to open and rewrite all those PDFs. Working with Normalized PDFs with external referenced can also lead to faster processing especially when the Normalized PDF refers to large amounts of external image data.

Although referring to external files is a standard PDF feature (for example used in the PDF/X-5), most PDF applications do not support this. To guarantee that a PDF can be processed correctly by PDF application (like Adobe Acrobat) it has to be made self-contained (embedding all the referred components like images). A self-contained Normalized PDF is 100% PDF compatible.

Color space

A Normalized PDF can only contain objects in the final printing color space. This final printing color space can have any number of separations but in most cases this will be CMYK and some spot colors. The PDF object model allows objects to be defined in RGB and CIE based color spaces. This is not possible in Normalized PDF. During normalization such objects are color converted to CMYK.

The fact that Normalized PDF can't contain objects in RGB or CIE based color spaces its biggest limitation. On the positive side it makes the workflow more predictable. It avoids late and often somewhat hidden color conversions for example in the RIP that can lead to problems in printing.

Document level metadata

A Normalized PDF contains XMP metadata. The schema of the XMP data is Esko specific, but the specification is public. The XMP metadata serves 2 purposes. It contains info about the file that can be extracted by 3th party applications and used for example for quality control or asset management. The XMP metadata are also used by Esko applications like Automation Engine to retrieve information from a PDF in an efficient way (for example the number of separations).

Object level metadata

A Normalized PDF contains metadata for specific objects like barcodes. Object level metadata are added to allow modification of those objects using an Esko application. Object level metadata are stored using a proprietary mechanism and format.

Which Tasks Require PDF Normalization?

The table below lists the Automation Engine tasks...

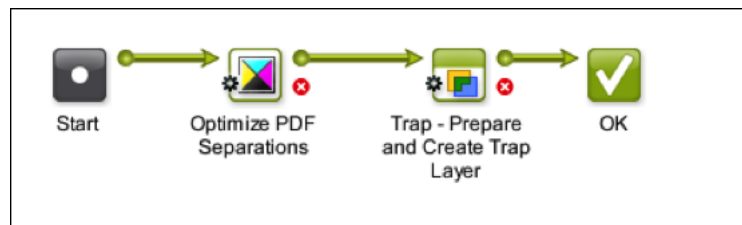
- that require normalization and will *normalize regular PDFs automatically* (**Normalized PDF Tasks**),
- that do not require normalization and work natively with regular PDFs (**PDF tasks**).

Note: The exact list of tasks available on your Automation Engine Server depends on the licenses you have, so some tasks in the list below may not be visible in your Pilot.

Normalized PDF Tasks	PDF Tasks
Add Ink Eaters	Adjust PDF Screening
Check Job Parameters	Compare PDF
Check Print Rules (PRC)	Create PDF File from RunList

Normalized PDF Tasks	PDF Tasks
Convert CMYK Colors	Extract Inks
Convert Colors with Equinox	Optimize PDF Document
Create Design (DesignWizard)	Optimize PDF Separations
Create PAF/JPG/XML (LinkEdge)	Preflight with PitStop
Create Report (ReportMaker)	Process PDF Spreads
Enrich Black	Resize PDF for Output
Export to 3D	Resolve OPI
Export to ArtPro File	Split Pages
Export to EPS/DCS File	Version PDF
Export to PDF File	
Export to PostScript File	
Export to VRML File	
FastVariants (all tasks)	
Manage Colors	
Optimize and Clean (PSFix)	
Outline Fonts	
PowerTrapper	
Prepare Station	
RIP (all tasks)	
Step & Repeat (all tasks)	
Trap (all tasks)	

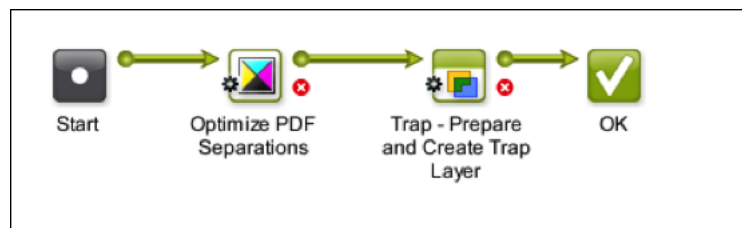
How do PDF Tasks Work with Normalized PDF Files?



Task	Task Type	Description
Optimize PDF Separations	PDF task	<p>The task checks whether the Normalized PDF input file contains any external references to images or other Normalized PDFs:</p> <ul style="list-style-type: none"> • if there are external references, the Optimize PDF Separations task checks if the modifications it is instructed to make will affect the external references. If any of the external references are affected, the task will first make the Normalized PDF input file self-contained (by embedding all the external references).

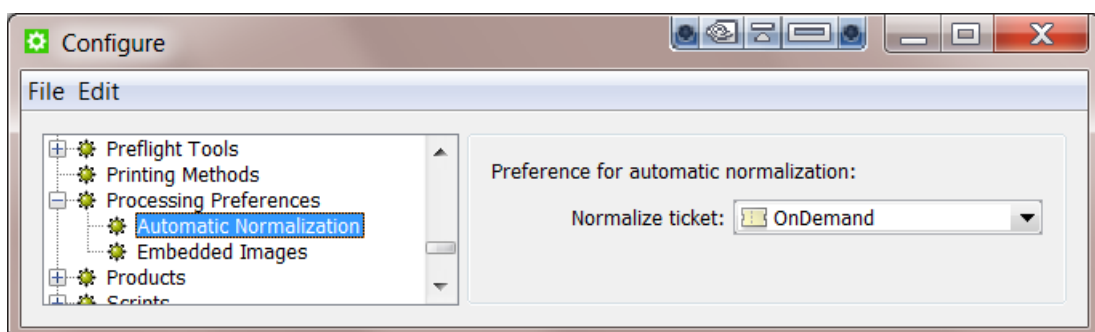
Task	Task Type	Description
		<ul style="list-style-type: none"> if there are no external references, or if the external references are not affected by the Optimize PDF Separations task will process the Normalized PDF directly. <p>The task also updates the XMP document metadata.</p>
Trap - Prepare Trap Layer	Normalized PDF task	The task traps the Normalized PDF file.

How Do Normalized PDF Tasks Work With Regular PDF Files?

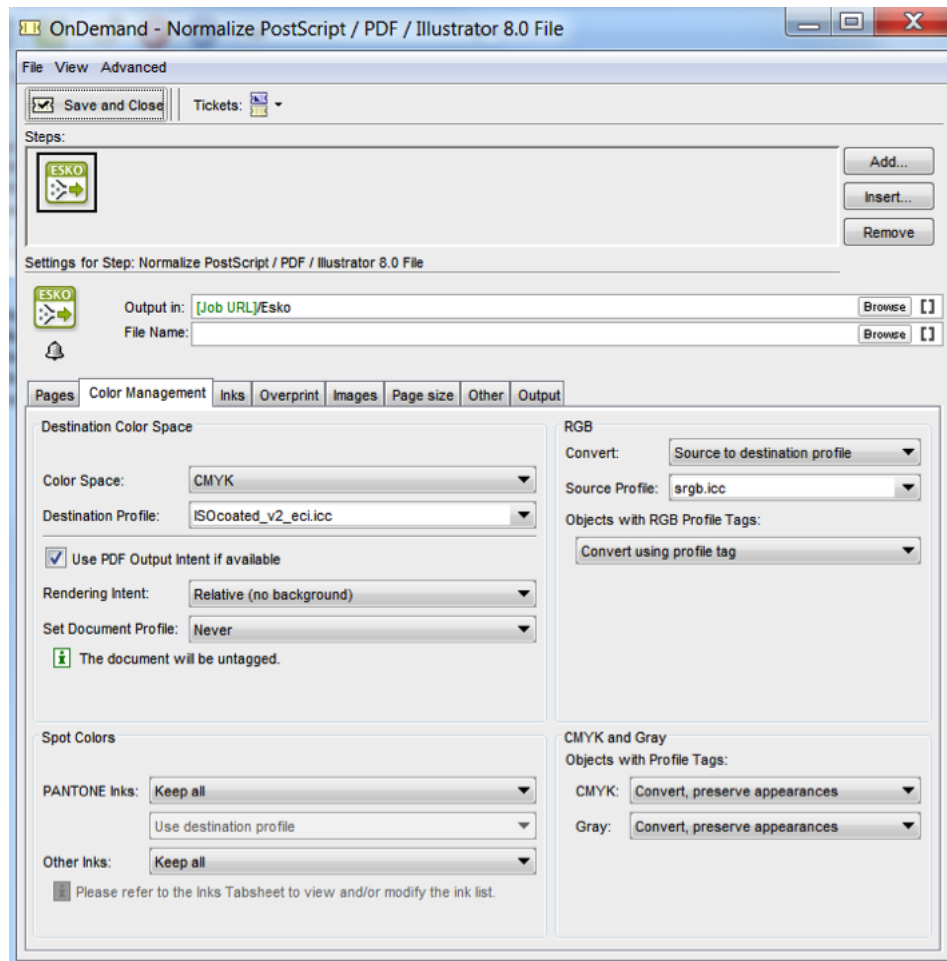


Task	Task Type	Description
Optimize PDF Separations	PDF task	The PDF is processed directly, because Optimize PDF Separations is a PDF task.
Trap - Prepare Trap Layer	Normalized PDF task	The task detects that the input file is a regular PDF document, and automatically normalizes it. After that, the file is trapped and a self-contained Normalized PDF is produced.

The ticket used for Automatic Normalization can be configured using the Configure window in the Automation Engine Pilot (choose **Tools > Configure** and navigate to **Processing Preferences > Automatic Normalization**).



Automation Engine is pre-configured to use the “Normalize PostScript/PDF/Illustrator 8.0 File” ticket called “OnDemand”. From this ticket only the settings in the “Color Management” tab are taken into account. All other settings of the auto normalization are forced to fixed default settings. For example, automatic normalization will always embed all images.

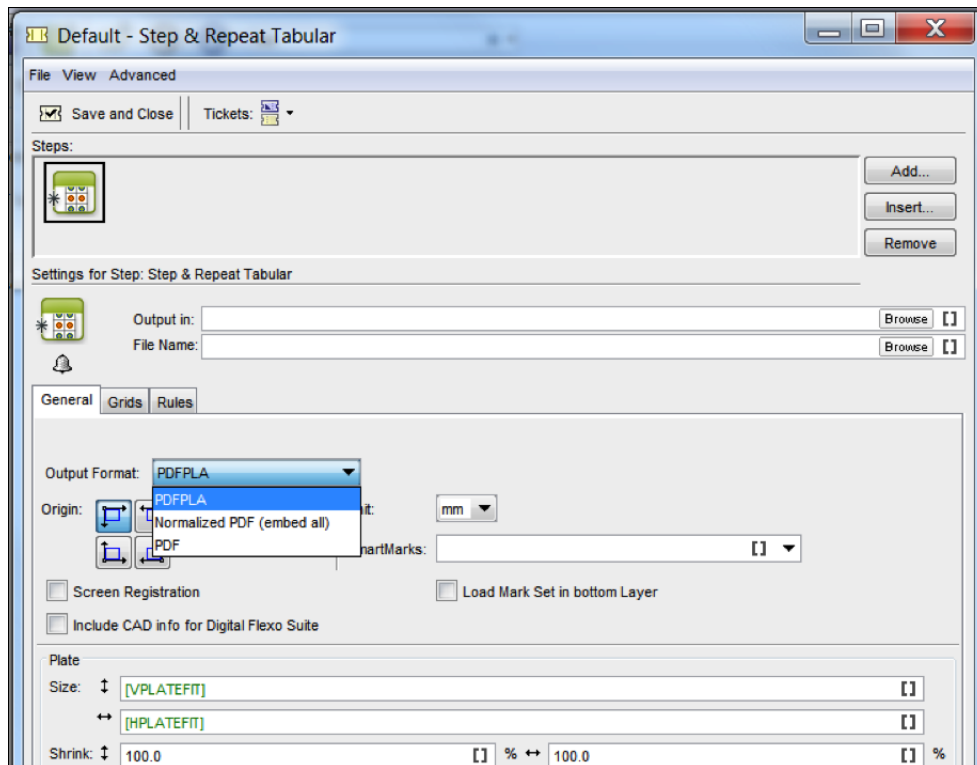


PDF Normalization and Step and Repeat

Output formats

Depending on your choice in the **Output Format** list of the **Step & Repeat** ticket, the format of the resulting document can be:

- PDFPLA
- a self-contained Normalized PDF
- a regular PDF



PDFPLA

When you choose this option, the output of the step and repeat task will be a PDFPLA file. A PDFPLA is a Normalized PDF with external references to the 1-ups. It can be loaded and modified in Plato. It can be RIP'ed directly by FlexRip. To send it to a non-Esko system, the PDFPLA needs to be exported to a PDF using the **Export PDF** task.

Note:

PDFPLA output can only be generated if all the inputs are Normalized PDFs. The task will fail if a 1-up at the input is not normalized on beforehand.

This format is the **recommended output format** when the primary purpose of the step and repeat file is to **send it to FlexRip**.

Normalized PDF (embed all)

When you choose this option, the output of the step and repeat task is a self-contained Normalized PDF. The 1-ups are embedded in the PDF using PDF forms so that the file size remains compact. The file can not be modified in Plato. It can be RIP'ed directly by the FlexRip. It can also be sent to a non-Esko system.

Normalized PDF inputs will be embedded in the output. PDF inputs will be auto normalized and then embedded in the output.

This format is **recommended when further processing is needed on the output**, for example by the Optimize PDF Separations task before sending it to FlexRip.

PDF

When you choose this option, the output of the step and repeat task is a PDF. The 1-ups are embedded in the PDF using PDF forms so that the file size remains compact. The file can not be modified in Plato. FlexRip can not RIP this file directly: it will first automatically normalize the step and repeat file. The file can be sent to a non Esko system.

Normalized PDF inputs will be embedded in the output. PDF inputs will be embedded in the PDF "as is".

This format is **recommended when the primary purpose** of the step and repeat file is to send it to a **third-party, non-Esko RIP**.

1.11.2 Retaining XMP data in PDF tasks.

It is possible to retain XMP metadata from third party software while processing native and normalized PDF files in the following tasks (with PDF as input or output):

- Optimize PDF Separations
- Optimize PDF Document
- Adjust PDF Screening
- Resize PDF for Output
- Process PDF Spreads
- Resolve OPI
- Extract Inks
- Prepare Graphics for Nesting
- Preflight with PitStop (changes to XMP metadata are possible through specific PitStop Actions)
- Normalize PDF File
- Normalize PostScript / PDF / Illustrator 8.0 File
- Export to PDF File
- Trap with PowerTrapper
- Prepare Station
- Check Job Parameters
- Convert CMYK Colors
- Convert Colors
- Create PAF / JPG / XML (LinkEdge)
- Enrich Black
- Export to Normalized PDF (Embed All)
- Trap - Prepare and Create Trap Layer
- Outline Fonts
- Check Print Rules (PRC)
- FastVariants - Split (Deprecated)
- FastVariants - Remove White Boxes (Deprecated)

Following tasks can retain XMP data when specific settings are enabled.

- Split Pages: when the option **Keep XMP Data** is switched on, all XMP metadata is copied to each and every output file.
- Version PDF: when the "Combine Base with Version Files" action is chosen, only the XMP metadata of the base file is kept.

1.12 Scripting

You can write Scripts (small programs) to automate the execution of certain tasks during a workflow. Writing and using such scripts is called scripting. You need the Script Runner application to link scripting with your Automation Engine workflows. You can automate actions from Adobe applications (e.g. Illustrator, Photoshop, InDesign) and third party tools (for example Alwan) using Scripts.

Some of the instances where scripting is useful are given below:

- to ensure file format integrity (standardization) for workflow inputs.
- to use standardized PDF as input during the Preflight process.
- to automate Adobe Illustrator, Photoshop, InDesign and InDesign Server via ExtendScript on Mac OS and Windows.

Supported Script Types

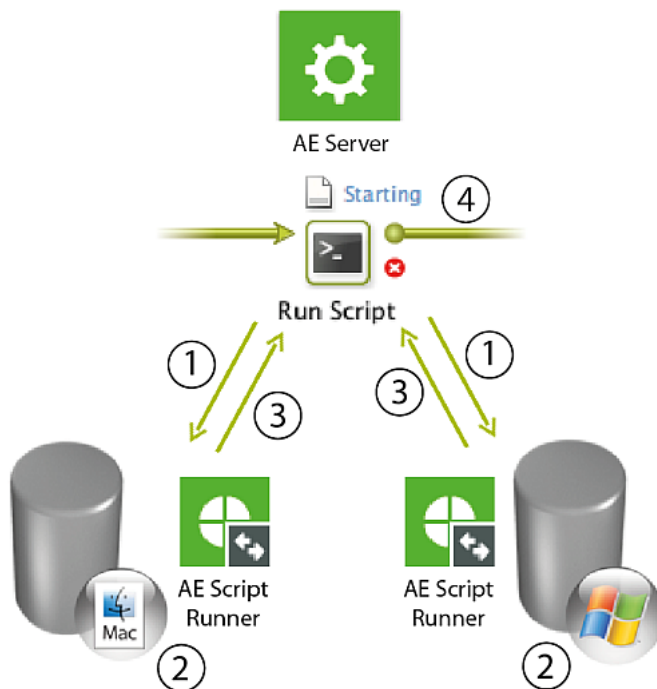
- AppleScript (on Mac OS)
- ShellScript (on Mac OS)
- Batch files (on Windows)
- Windows Script (VBScript, JScript) (on Windows)
- ExtendScript (on Mac OS and Windows)

More info on Scripting :[Getting Started with Scripting](#) on page 123

More info on Scripting in workflows: [Use case: Scripting](#) on page 125

1.12.1 Automation Engine Script Runner

The Automation Engine(AE) Script Runner is a standalone AE server component which runs scripts on behalf of AE. You can add customization to your workflow by adding a **Run Script** task. This task will run on the Automation Engine server while the execution of the script will be done on the Script Runner application which can be installed on Windows or Macintosh. When you launch the workflow, the following will take place:



1. The **Run Script** task sends a request to run the specified script.
2. The Script Runner processes the request accordingly and runs the script.
3. The Script Runner sends the results back to the server.
4. The workflow will continue with the outputs from this task.

1.12.2 Getting Started with Scripting

Scripting helps you customize some steps in your work flows. You can achieve this by adding a **Run Script** task to the work flow where you want customization. The Automation Engine will use Script Runner to run scripts which are stored either locally or on a server. When you launch the workflow in Automation Engine, the Script Runner runs the script which contains the main function incorporating the inputs from the Run Script task, output folder and some optional script parameters from the **Run Script** ticket. The workflow in Automation Engine continues with the contents of the output folder which contains the outputs from the script. To achieve this, you need to do the following steps :


1. Download Automation Engine Script runner. You can download it via the web access to the Automation Engine Server. **Client Apps > Tools** .


Server RDALPHA25 ● running [Log in to another server](#)


Pilot


Tools


» TOOLS

- 
» Automation Engine Diagnostics
LAUNCH
 Monitor the status and activity of the Automation Engine Server.

- 
» Automation Engine 10 Backup
LAUNCH
 Use this tool when you want to make a backup of an Automation Engine 10 Server to duplicate its configuration on a new Automation Engine 12 Server.

- 
» Automation Engine Script Runner
DOWNLOAD
 Allows you to automate tasks in applications like Illustrator, Photoshop, InDesign using scripting via the Run Script task in the Automation Engine Pilot.

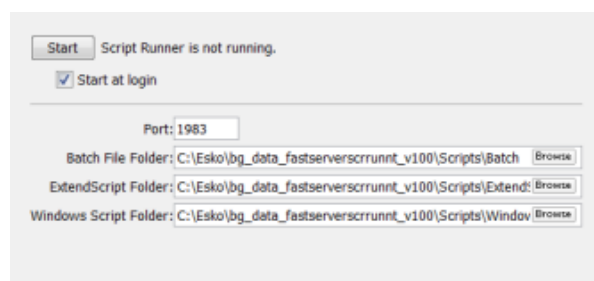
- 
» Automation Engine ArtPro Action List Editor
DOWNLOAD
 Create and edit ArtPro Action List files to automate tasks in ArtPro via the ArtPro Action List task in the Automation Engine Pilot.

- 
» Automation Engine RunList Editor
DOWNLOAD
 Create and edit RunLists to collect several files into one job folder, or to make one final PDF out of several smaller PDFs (e.g. for imposition).

- On Mac OS, open the 'dmg' file after downloading the Script Runner which contains the installer package. Double-click to start the installation.
 - On Windows, double-click the downloaded installer to start the installation.
2. Install the Script Runner by following the instructions in the Installshield Wizard/Assistant and make sure it is running in your computer.
- On Windows, open **Start > All Programs > Esko > Automation Engine Script Runner > Preferences** .
 - On Mac OS, open **Applications > Automation Engine Script Runner > Esko > Automation Engine Script Runner > Script Runner Preferences** .

In the Script Runner **Preferences** window, you can:

- check if the Script Runner is actually running
- start / stop the Script Runner
- enable(disable) Start at login
- view and change the port the Script Runner is communicating with
- view and change the default folders for scripts



3. Configure the Script Runner on your computer. Read more in [Scripts](#) on page 681.

4. Write and save scripts. You can find sample scripts in the subfolders of /Library/Scripts/Esko on Mac OS and C:\Esko\bg_data_fastserverscrunnt_v100\Scripts on Windows.
 - You can store the scripts on your computer on a default folder (specific for script type). This is ideal when you are developing and testing a script or when you do not intend to give access to other Script Runners on the network.

Note: The default location :

on Mac OS : /Library/Scripts/Esko.

on Windows : C:\Esko\bg_data_fastserverscrunnt_v100\Scripts.

Automation Engine does not back up these scripts.

- Alternatively, you can store your scripts in the Automation Engine data folder. This option is suitable when you want to make your scripts available to all configured Script Runners.

Note: The default location :

C:\Esko\bg_data_fastserver_v100\Scripts.

Automation Engine backs up these scripts.

5. Run the Script using Run Script task in a workflow. Read more about this in [Run Script](#).

Note: Refer to the Reference Guide for more on Scripting

1.12.3 Use case: Scripting

Note:

Sample scripts are solely intended to demonstrate techniques for accomplishing common tasks. Additional script logic and error-handling may need to be added to achieve the desired results in your specific environment.

It is up to the user to verify that his intended use of the offered automation functionality is compliant with any third party license agreement and/or other restrictions applicable to any non-Esko products.

Please read [Getting Started with Scripting](#) on page 123 before conducting the steps below.

In this example, we are demonstrating a script which will perform a Photoshop action and produce a JPEG output. The script 'EskoPSDoActionAndSaveasJPG.jsx' is available from the sample scripts subfolder (ExtendScript) in /Library/Scripts/Esko on Mac or C:\Esko\bg_data_fastserverscrunnt_v100\Scripts on Windows.

1. You have to save the **Set** and **Action** in Photoshop . In this example we are using **Default Actions** as **Set** and **Molten Lead** as **Action**.
2. You can open the script with ExtendScript Toolkit to edit the script if required. However, in this sample, we do not need to edit the script.

```

ExtendScript Toolkit CS4
main

/**
 * Sample scripts are provided as-is with no warranty of fitness for a particular purpose.
 * These scripts are solely intended to demonstrate techniques for accomplishing common tasks.
 * Additional script logic and error-handling may need to be added to achieve the desired results in your specific environment.
 *
 * It is up to the user to verify that his intended use of the offered automation functionality is compliant with any third party license agreement and/or other
 */

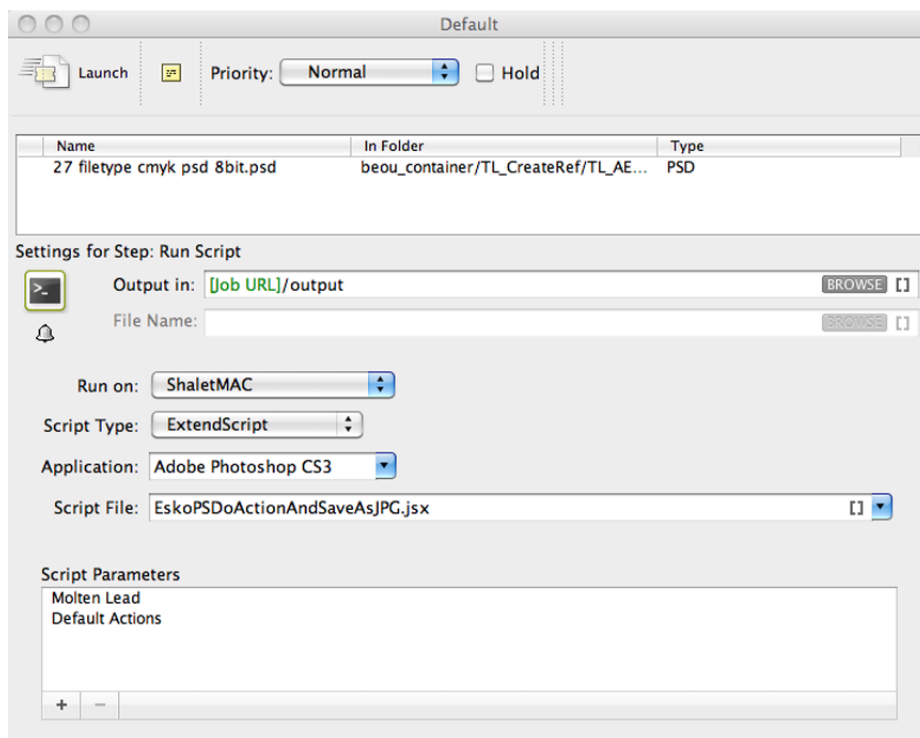
function main(inputs, outputFolder, params)
{
    for (i=0;i<inputs.length;i++)
    {
        var fileRef = new File(inputs[i]);
        var docRef = app.open(fileRef);

        app.doAction(params[0], params[1]);

        outputFile = new File(outputFolder + "/" + fileRef.name);
        jpgSaveOptions = new JPEGSaveOptions ();
        jpgSaveOptions.embedColorProfile = true;
        jpgSaveOptions.formatOptions = FormatOptions.STANDARDBASELINE;
        jpgSaveOptions.matte = MatteType.NONE;
        jpgSaveOptions.quality = 1;
        docRef.saveAs(outputFile, jpgSaveOptions, true, Extension.LOWERCASE);
        docRef.close(SaveOptions.DONOTSAVECHANGES);
    }
    return "OK"
}

```

3. Edit the settings of the **Run Script** task as follows:



- Run on:** is the name of the computer where your **Automation Engine ScriptRunner** is installed. The name in this example is `ShaletMac`. You can read more about configuring and naming Script Runner in [Scripts](#) on page 681
- Script Type:** choose `ExtendScript`.
- Script File:** choose `EskoPSDoActionAndSaveasJPG.jsx`.

- d) **Script Parameters:** Add the name of your Photoshop **Action** name followed by your **Set** . In this example, Molten Lead; Default Actions
 - e) Save the task.
4. Launch the task:
- a) Select the Photoshop file.
 - b) Right-click the file and browse for the ticket.
 - c) Click **Launch**.
- You will see that while the script starts to run on the Automation Engine Server, PhotoShop opens, performs the actions, closes and continues the script. The result of this sample script is finally an adjusted jpeg.

1.13 Example of Building and Using a Packaging Workflow

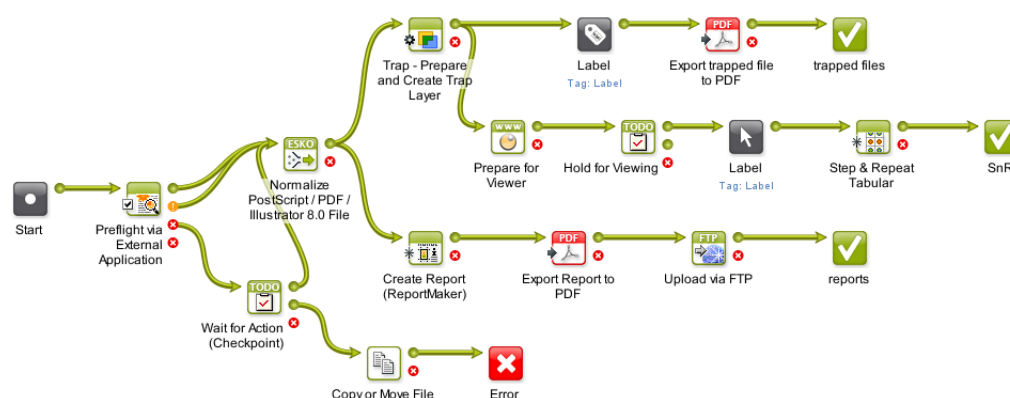
1.13.1 Build a Workflow

Note: Working with Workflows is documented in this dedicated chapter [Workflows](#) on page 986.

You can build workflows from zero or by using custom tickets and you create nested workflows.

You can use [SmartNames](#) on page 916 to use predefined values. You can also choose to have the prepress manager build workflows that daily users will use, or let the daily users build workflows on the fly while processing files. See [Workflows](#) on page 986.

The following steps will help you build this example workflow:



Preflight your file

Preflight via External Application

The **Preflight via External Application** step allows you to preflight a PDF file by integrating external (non- Automation Engine) preflight software into Automation Engine.



Attention: To define the preflight settings for Automation Engine go to **Tools > Configure** and select [Preflight Tools](#)

1. From **Start** connect to the **Preflight via External Application** workflow step.



2. Double-click on the workflow step to edit its parameters. For more information go to [Preflight via External Application](#).

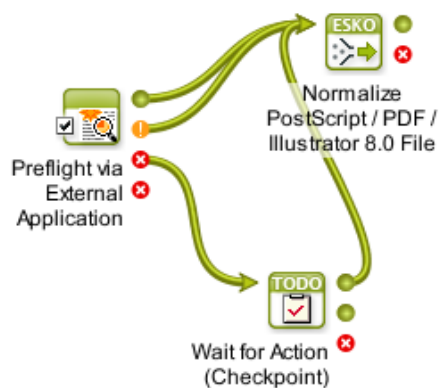
When Preflighting in a workflow, the **Preflight via External Application** workflow step has four output pins:

Note:

- **Preflight OK:** your files will go through this pin if the preflight gave an OK status.
- **Preflight with Warnings:** your files will go through this pin if the preflight gave a warning.
- **Preflight with Errors:** your files will go through this pin if the preflight gave an error.
- **Error:** your files will go through this pin in case of a processing error unrelated to the preflight status.




3. Connect each pin to the appropriate workflow step (for example, files which are “OK” or “Warning” can be normalized depending on your preflight setup, while files with “Error” need to be checked by an operator).

**Wait for Action (Checkpoint)**

Use the **Wait for Action (Checkpoint)** task to make your workflow stop at certain predefined points so that you have time to check and possibly correct something. See [Wait for Action](#).

This Workflow is built so that you can route files based on your expertise. You will get a notification

(**To Do List** ) , when the Preflight detects an error. The processing will be paused until you decide whether to approve the file or not, based on the preflight error report.

1. Add the **Wait for Action (Checkpoint)** step to your workflow.
2. Double-click the step to open its settings.
3. Create the notification that will be sent:
 - a) choose which user you want to send it to,
 - b) enter a **Subject**,
 - c) add a **Message** if desired.

To: John Doe

Subject: Check the traps please!

Message (optional): Show proof to manager if ok.

4. Choose a **Due date**. Either:

- choose one of the predefined due dates (from **Immediate** to **Next Sunday**),
- choose **Other...** in the **Due date** list and pick the due date yourself.

Due date: Other ... 01-May-2010 09:00

Output states:

Name
Completed
Aborted

May 2010

M	T	W	T	F	S	S
26	27	28	29	30	1	2
3	4	5	6	7	8	9
10	11	12	13	14	15	16
17	18	19	20	21	22	23
24	25	26	27	28	29	30
31	1	2	3	4	5	6

Today

5. In **Output states**, define the output possibilities you want for your checkpoint.

By default, this step has two outputs: **Completed** and **Aborted**.

To add an output:

- click **Add**,
- enter the output's name.

Add as many outputs as you like.

6. If necessary, use the **Move Up** and **Move Down** buttons to display your outputs in a different order.

Note: You can **Remove** the default outputs if you don't need them.

Output states:

Name	Description (optional)
Completed	Accept the job
Aborted	Reject the job

The receiver of this task needs to select one of the output states.
The file(s) will be released to the selected output state.

Buttons: Add, Remove, Move Up, Move Down

7. Click **OK** to apply your settings.

Tip: After connecting the **Output states** you can:

- Add an extra **Output state** to the checkpoint and connect it to the “**Preflight via External Application**” step, to preflight your file again.
- [Relaunch](#) the file from the Task pane or directly in the canvas. Right click and choose **Relaunch**

Copy or Move File

This task copies or moves the input file or the marked files to a new location. To mark a file, please insert the [Mark](#) file task in your workflow chain. If you don't want to evaluate an existing file, select the "Fail" task when the file that is to be copied or moved already exists in the output folder. See [Copy Move File](#)

Normalize your File

The **Normalize PostScript / PDF / Illustrator 8.0 File** generates a Normalized PDF file based on the preflighted file. See [Normalize PostScript / PDF / Illustrator 8.0 File](#)

- **Transparencies** and **Layers** check the box to import PDF files as in Scope 3 (and lower). All transparencies and layers will be flattened. If not checked, a normalized PDF file will be created that contains all typical PDF features (like transparencies and layers).

Workflow overview



Trap your file

Note: This example still used the older trapping technology that pre-dated the more recent task **Trap with PowerTrapper**.

Trap - Prepare and Create Trap Layer

The trapping process is a safeguard against possible (even likely) fluctuations in the printing process. Minor discrepancies in registration, slight paper shifting can cause inks to mis-register. See [What is Trapping?](#)

There are three different Trap tasks;

Trap – Create Trap Layer: This task traps the input file according to the settings specified in the Trap Color Pairs file associated with the input file. The output of this task is a trapped Normalized PDF file. This task contains only the output options of the Trap task. All other trapping parameters are stored in the TCP file located in the same folder as the input file.

Trap – Prepare and Create Trap Layer: this step selects candidate trap color pairs and executes the trapping. The trapping areas are added in a separate layer on top of the original job.

Trap – Prepare Trap Pairs: This scans through the job and suggests a list of candidate trap color pairs.

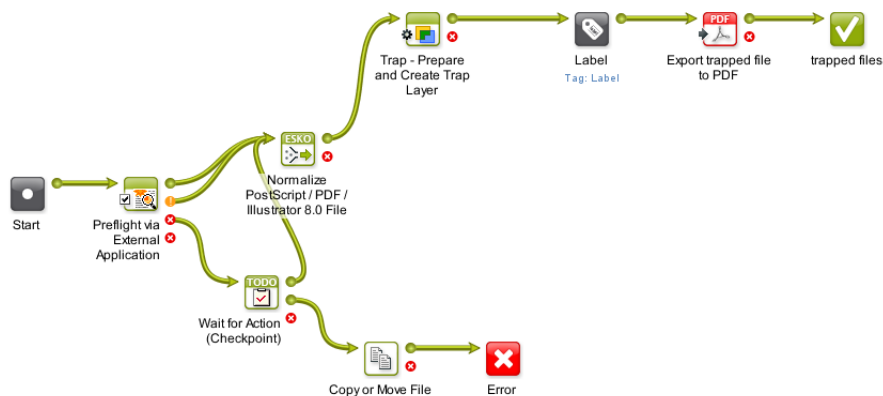
Export Trapped File to PDF

Export your Normalized PDF to generic PDF for storage or approval. See [Export to PDF File](#)

Tip:

- To define the name and location of the output file that will be generated, use **SmartNames**.
- You can send the output file via mail.
- You can use PDF as SoftProof.

Workflow Overview



View your file

Prepare for Viewer

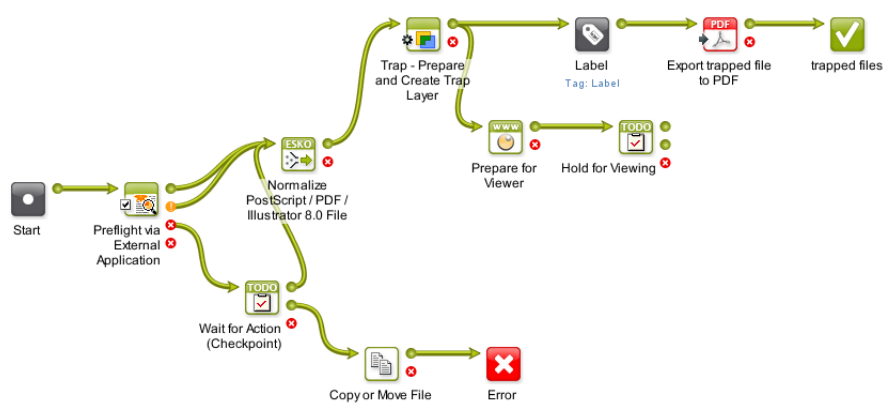
The **Prepare for Viewer** task prepares view data for the **Automation Engine Viewer**. This view data will help the Viewer to quickly open the file (the Viewer will use the prepared View Data instead of asking the server to make or stream it). See [Prepare for Viewer](#).

Hold for Viewing

Choose a decision maker with experience to approve the trapped file. See [Wait for Action](#). The **Automation Engine Viewer** is a powerful integrated Viewer and QA tool. The Automation Engine Viewer offers:

- Reliability: Showing the data in a correct way. At best in the same way the RIP/ Proofer will see them.
- Detail: Unlimited zoom, high detail measuring, measuring traps and exact distances (die objects).
- Speed: Opening files swiftly, even when it concerns large files. Fast zooming.
- See [Automation Engine Viewer](#).

Workflow Overview



Tip: You could add an extra **Output state**, "Send to Proof" to the checkpoint, connecting to [Proof\(FlexProof\)](#) to print HardProof.

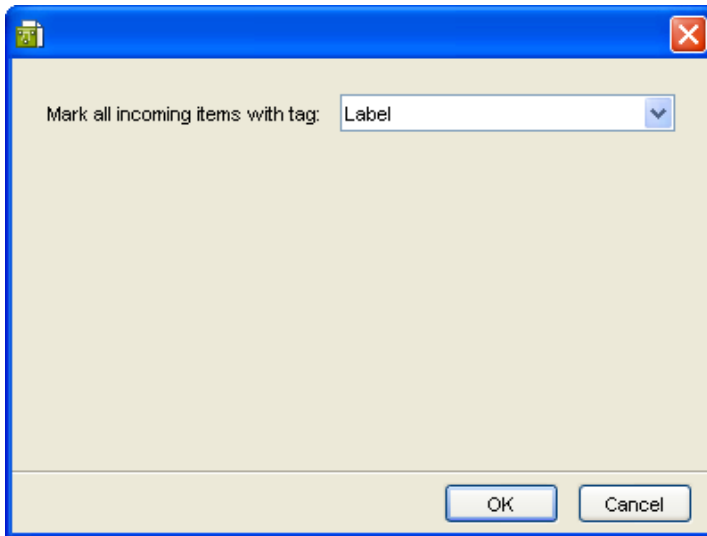
Generate Step & Repeat

Mark



Use this to tag files that you will need to select later in the workflow (with the **Select** workflow control). The file will be tagged as "Label" and after approved in **Hold for Viewing**, the **Select** will pickup the original file.

- Give the tag a name.

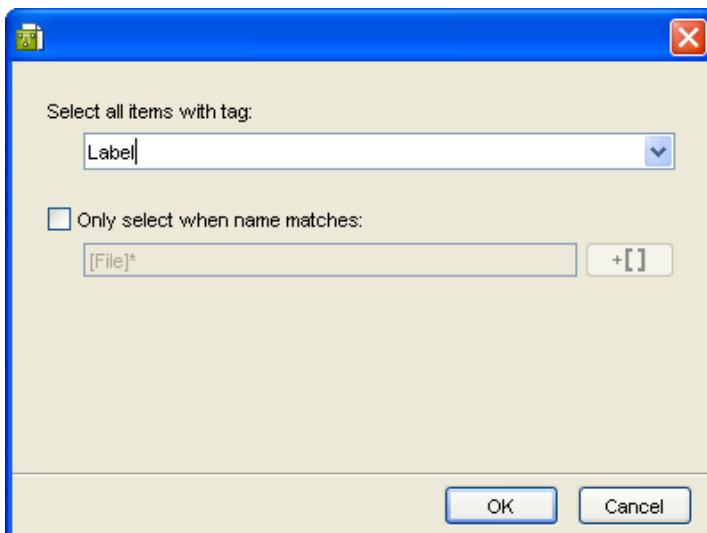


Select



Use this ticket to select files that you have tagged earlier in the workflow (with the **Mark** workflow control).

- Enter the tag of the file.

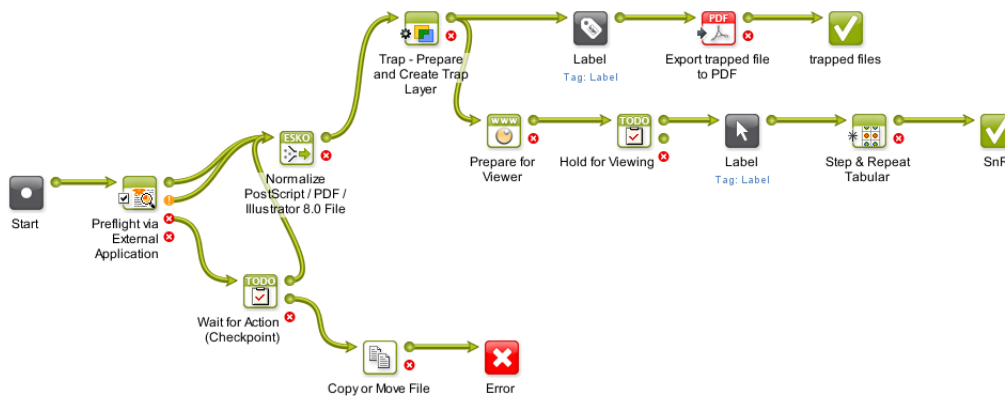


For an example see [Mark and Select](#).

Step & Repeat Tabular

This Workflow Step executes all Step and Repeat parameters (number of one-ups vertically and horizontally, gap between one-ups, alignment, SmartMarks...) after which the server will output the sheet layout. The ticket's input fields also accept SQL queries, so that information stored in a database can be used to drive the sheet layout creation. See [Step & Repeat Tabular](#).

Workflow Overview



Create Report

Create Report (ReportMaker)

The *ReportMaker* task is used to automate the creation of design print cards (in French: fiche technique, in German: Drueck Karten).

The idea is that you generate reports containing all kind of information: file name, used inks, name of the operator, an image of the file. All this information is defined in a report template, then this template is used to generate a specific report. For more information about templates, see [ReportMaker Templates](#).

Note: You need PackEdge 3.0 or higher to make a ReportMaker template.

Export Report to PDF

Export to PDF makes it possible to specify different settings, for example **PDF version**, **Color Management**...

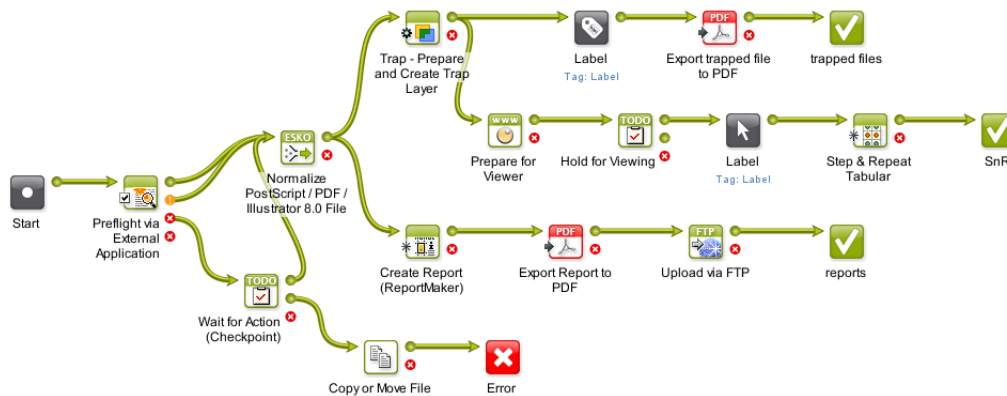
Tip: You can define the **File Name** and the, **Output location**, of the report file that will be generated, by using SmartNames.

Upload via FTP

Upload via FTP is an administration task which allows you to send files to a server. You need an FTP account on that server. For details or how to connect to an FTP server, see [Upload via FTP](#).

Tip: To interact with customers you can also use steps like **Publish on Web** and **Create WebCenter Project**.

Workflow Overview



1.13.2 Use Shuttle in the Workflow

What is Shuttle

Shuttle is a small stand-alone application that enables you to submit files to Automation Engine tasks or workflows, and monitor their progress. You can use Shuttle to connect to several servers. The Shuttle functionality is also integrated in ArtPro, Neo, PackEdge, Plato, FastImpose and in Adobe Illustrator as a DeskPack plug-in. You can download Shuttle from Esko's Download Center (<https://mysoftware.esko.com/>). For more information about Shuttle see the **Shuttle User Guide**.

Set up Shuttle for Automation Engine

To use Shuttle and submit files to your Automation Engine workflow, you need to set up your server, your Shuttle client, and make sure they can exchange files via a shared folder.

Workflow Server Setup

Automation Engine's Configure tool. See **Automation Engine Setup for Shuttle, Automation Engine User Guide**.

Shuttle Client Setup

Set up your Shuttle client in the applications you use. See **Shuttle Clients Setup, Automation Engine User Guide**.

Shared Folder Setup

You also need to make sure that the relevant servers shared folders are mounted on the Shuttle clients machines, with the appropriate Read and Write permissions. For Automation Engine, the shared folder is either an Automation Engine Container, or an Upload Folder you set up in Configure. See **Automation Engine Setup for Shuttle, Automation Engine User Guide**.

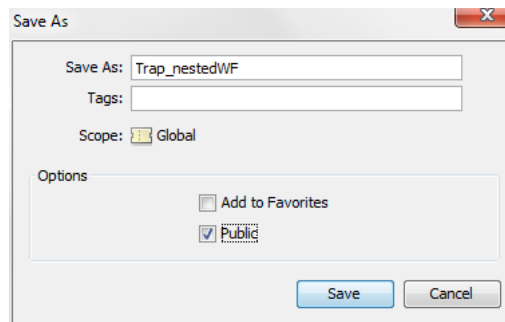
What are Public Parameters

Learn more about Public Parameters in [Using Public Parameters in Workflows](#) on page 1051.

Making Your Tickets Public

With Shuttle, you can submit files to any Automation Engine task, task chain or workflow that has a **custom ticket**. To do this, you need to make the ticket public. You can either:

- Select the **Public** option when creating the ticket.



- In the **Ticket** view, right-click your ticket and select **Public Ticket**.

Note: Public Tickets have a mark in the **Public** column.

Task Type	Ticket Name	Public	Description
Zip	Default		
Workflow	WC_Approval		
Workflow	WaitForRunListFiles		
Workflow	TrapWF		
Workflow	Trap_nestedWF	✓	
Workflow	SimpleTrap		
Workflow	Send2Ody		

Using Public Parameters

Public Parameters are ticket parameters that users can define when they launch tasks using Shuttle (applications).

For each public parameter, you can either:

- enter a list of predefined values that the user needs to choose from,
- let the user enter a value himself.

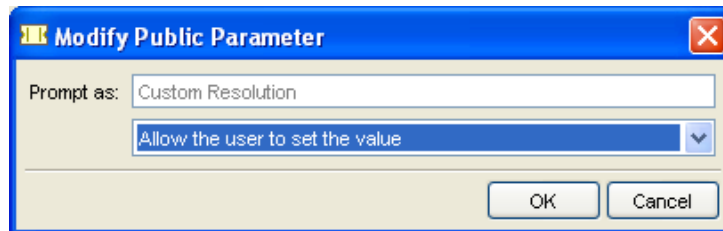
Making a Parameter Public



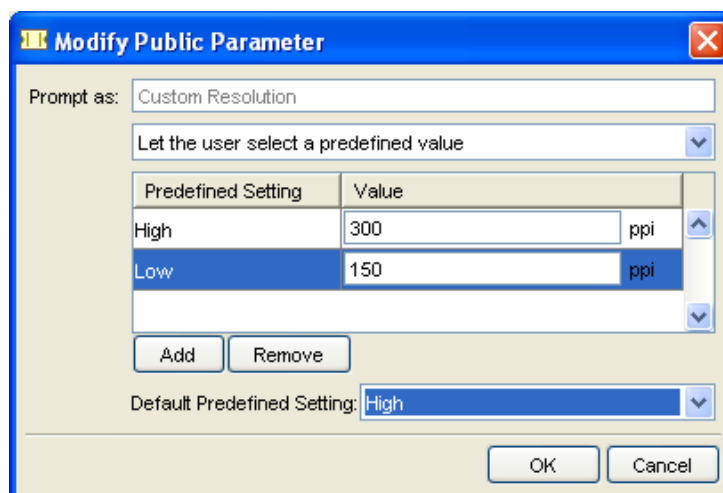
Attention: You need to define which parameters are public before files are submitted to your workflow.

In the ticket containing the parameter to make public:

1. Right-click the parameter and select **Make Parameter Public**.
2. Right-click it again and select **Modify Public Parameter...**
3. In the **Modify Public Parameter** dialog that opens:
 - a) If necessary, change the way the parameter will be called in Shuttle in **Prompt as**.
 - b) Choose either:
 - **Allow the user to set the value** if you want Shuttle users to be able to freely enter a value for that parameter.



- **Let the user select a predefined value** if you want Shuttle users to choose from a list of values that you define.





If that parameter already has a list of values to choose from in the Ticket, they will be displayed here. You can **Add** or **Remove** values.

Note: Shuttle operators will see (and choose from) the **Predefined Settings** but not the associated **Values**.

- c) If you have chosen **Let the user select a predefined value:** in **Default Predefined Setting**, choose the setting that will be selected by default when submitting files from Shuttle (if the operator doesn't select another setting, this one will be used for processing).
4. Don't forget to save your ticket.

This is how your public parameter will look to Shuttle operators:

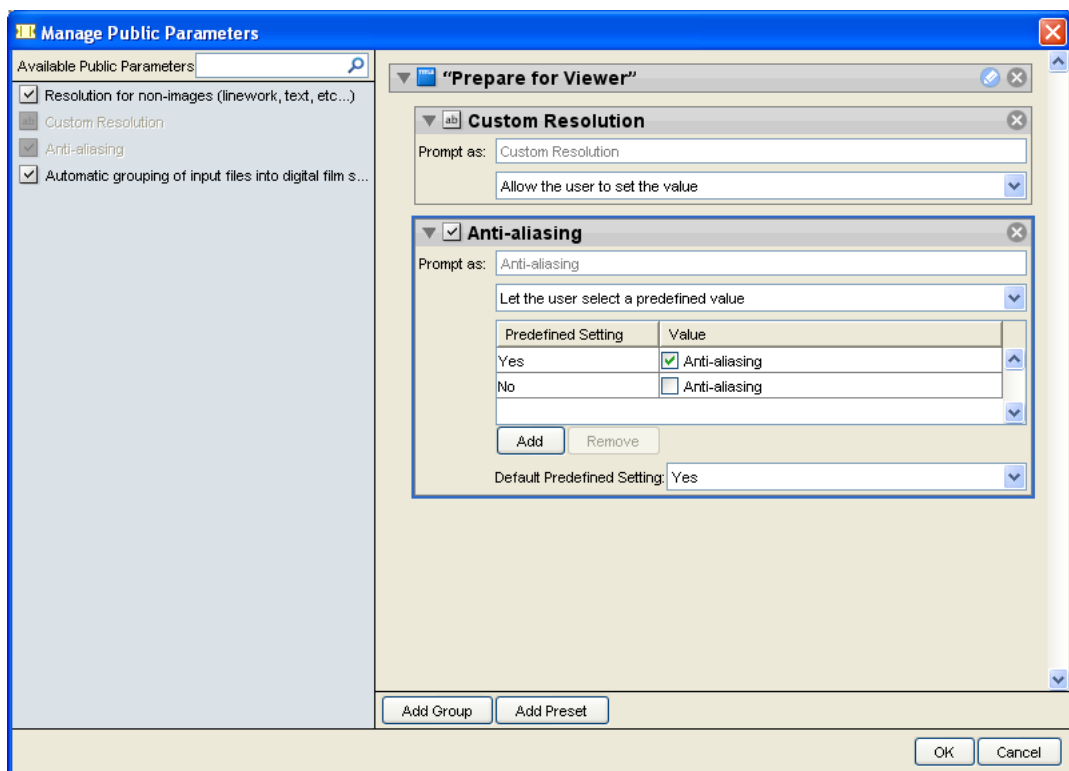
If you chose Allow the user to set the value	If you chose Let the user select a predefined value
	

Managing Public Parameters in Your Ticket

Once you have made parameters public in your ticket, you can have an overview of your public parameters and edit them at the ticket level.

1. In your ticket, go to **Advanced > Manage Public Parameters...**


The **Manage Public Parameters** dialog that opens contains all of your ticket's public parameters.




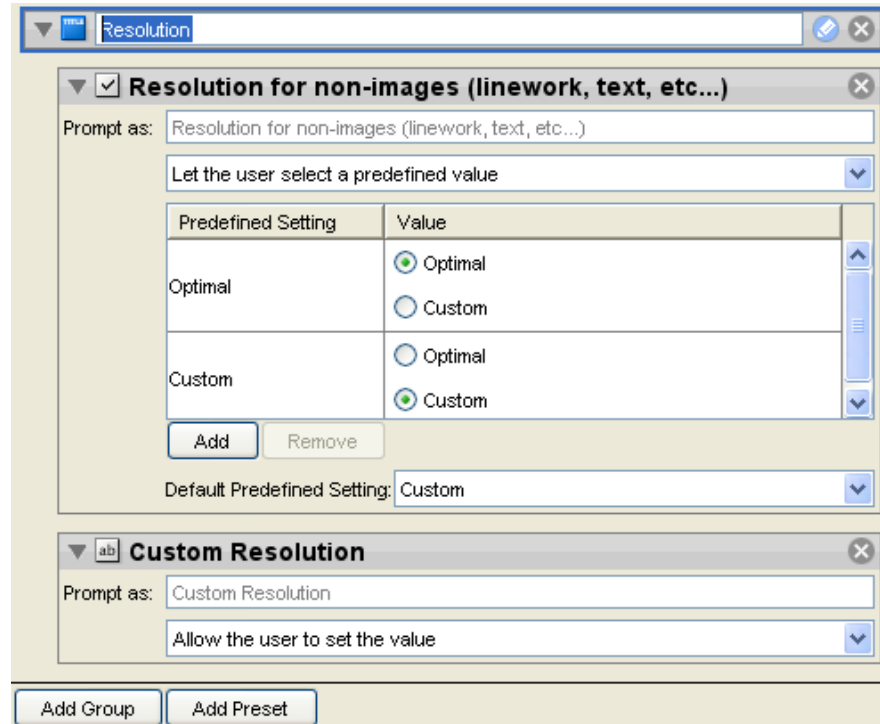
The left pane shows all parameters that can be made public for that ticket (the ones you have made public are greyed out).

The right pane shows the parameters you have made public, with their settings (predefined values, etc.).

2. You can:

- drag parameters from left to right to make them public;
- click a public parameter's cross  if you don't want it to be public anymore;
- rearrange the order of your public parameters (drag them up or down);

- change public parameters' settings (**Prompt as**, **Predefined Settings**, etc.);
- group public parameters under a header (click **Add Group**, click  to edit the header, and drag parameters from the left);



Using Presets to Simplify Operators' Choices

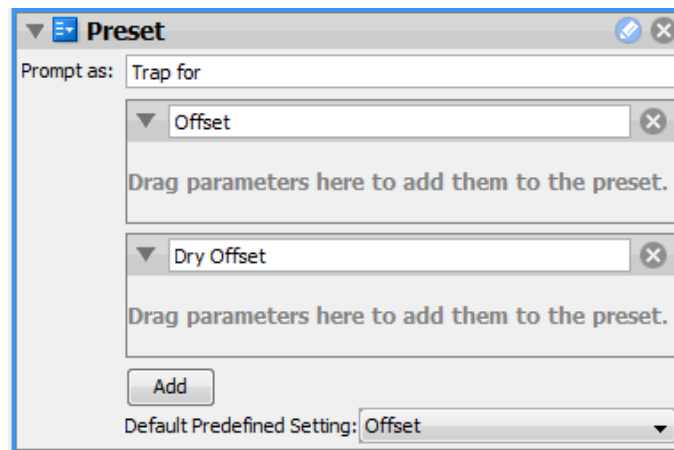
You can use presets to minimize the amount of public parameters your operators have to choose from, and make it simpler for them.

For example, you want operators to use different trapping settings for printing with offset or dry offset. You can define those trapping settings in a preset, so that operators will only have to choose between offset and dry offset.

In the **Manage Public Parameters** dialog:

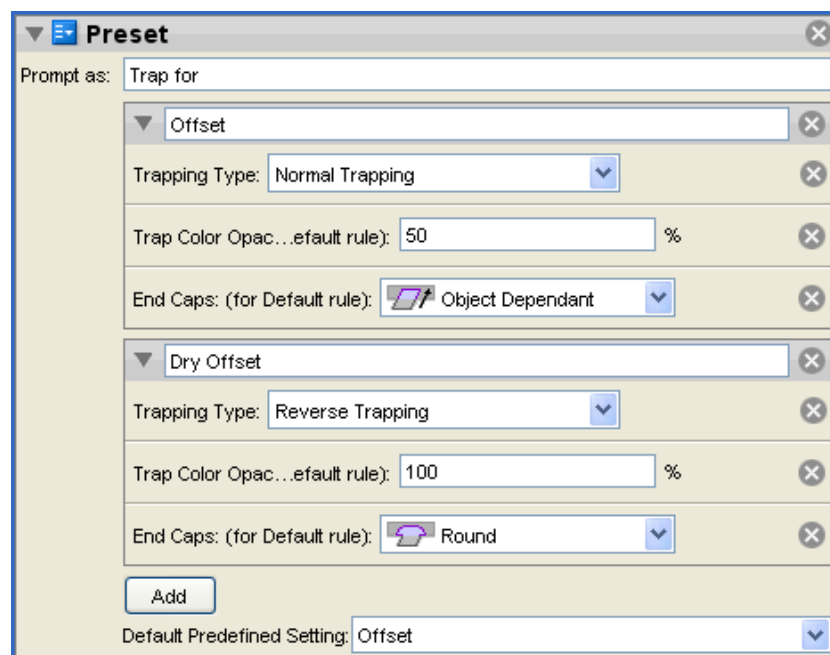
1. Click the **Add Preset** button.
2. Enter what your operators will see:
 - a) in **Prompt as**, enter the name to give the preset parameter in Shuttle,
 - b) enter the values that operators will choose from (replacing `Type a value here`).

Click **Add** if you need to add extra values.



3. Drag parameters from the dialog's left pane into the preset.

You can drag as many parameters as you want. They will appear under each value (here both under *Offset* and *Dry Offset*).

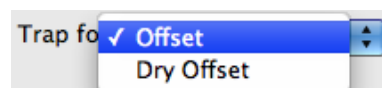


4. Define the parameters values for each preset value.

For example, define normal trapping, 50% opacity and object dependant end caps for *Offset*, and reverse trapping, 100% opacity and round end caps for *Dry Offset*.

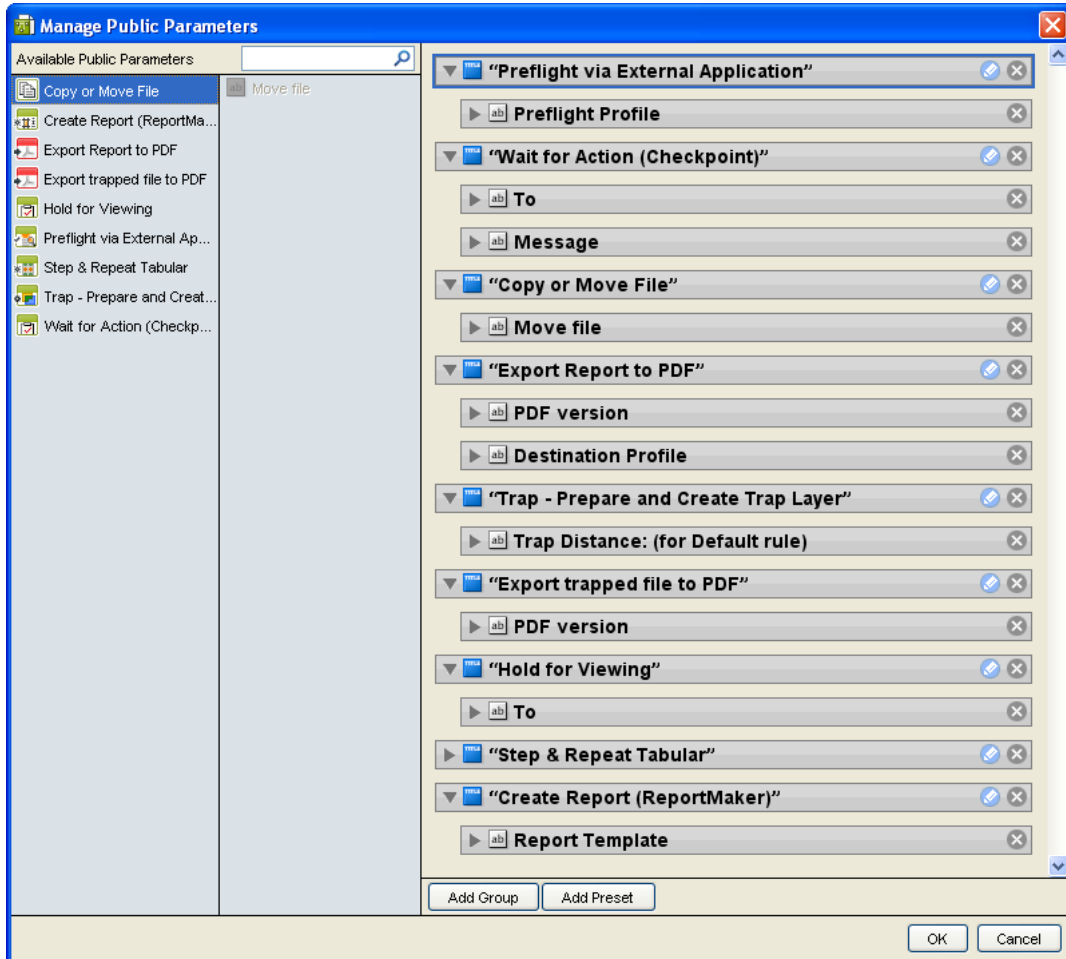
5. Don't forget to save your ticket.

This is how your preset will look to Shuttle operators:



Public Parameters in Sample Workflow: Packaging

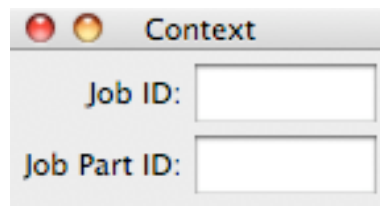
When choosing which parameters to make public you can keep in mind end production, different customers, workflow capability etc. Define the parameters so that you can change the production file without having to change the workflow ticket.



Launching Files into a Workflow

Launching Files from Shuttle Standalone

1. To assign a **Job ID** and / or **Job Part ID** to the files to process in your workflow, go to **Window > Context** and fill in the **Job ID** and / or **Job Part ID**.

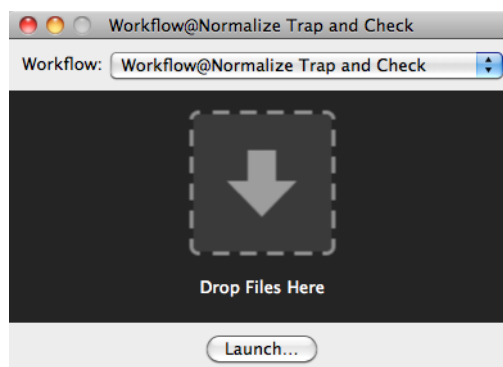


Do this when:

- you want to add your file(s) to an Automation Engine job,

Note: If the file you are launching is already located in an Automation Engine job, it will automatically be processed within that job's context (you don't need to enter the Job ID).

2. Open the Launch window using:
 - **File > New Launch Window...** ,
 - **Option + Command + N** (Mac) or **Alt + Control + N** (Windows).
3. Select the **Workflow** you want to launch on your file.



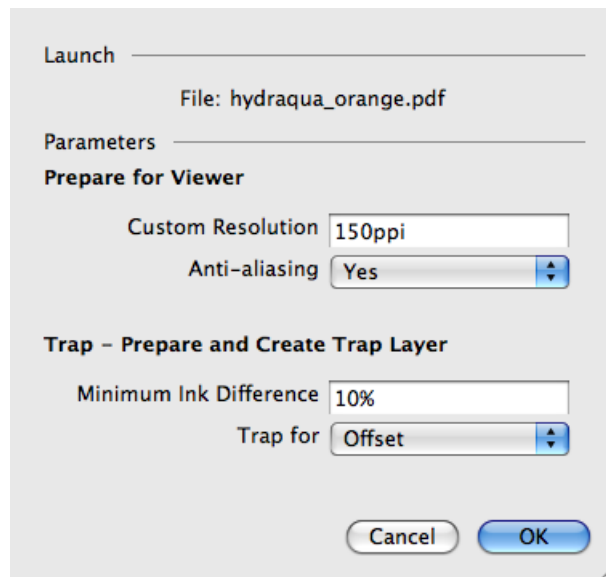
- For Automation Engine workflows, you will see public tickets.

Note:

You can open several Launch windows, and select a different workflow in each.

If you quit and restart Shuttle, your Launch windows will be remembered.

4. Launch your file in one of the following ways:
 - Drag and drop your file on the **Drop Files Here** arrow.
 - Click the **Launch...** button and browse to the file you want to launch.
 - Go to **File > Launch...** and browse to the file you want to launch.
 - Use **Command + S** (Mac).
5. If your workflow has public parameters, you will see a pop-up where you can fill them in.

**Note:**

For more information on public parameters, see:

- [Using Public Parameters](#) on page 68,

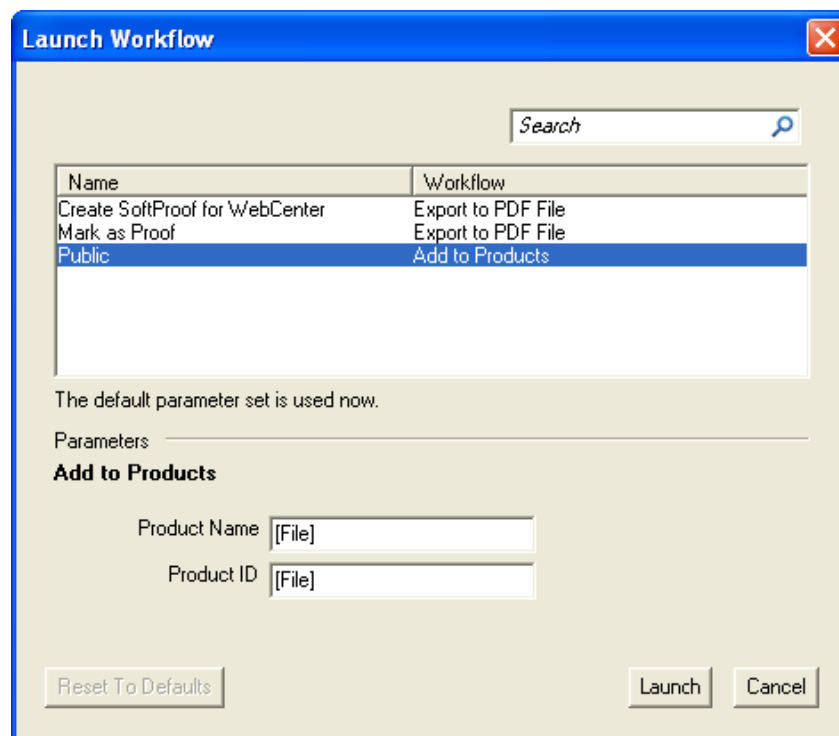
Launching Files from PackEdge



Attention: You must save your file on an Automation Engine Container before you can launch it into a workflow from PackEdge.

1. Go to File > Launch Workflow...

This opens the **Launch Workflow** window.



2. Select the workflow you want to launch on your file in the workflow list.
3. If your workflow has public parameters, you can fill them in in the same window.

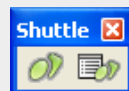
Note: For more information on public parameters, see [Using Public Parameters](#) on page 68.



4. Click **Launch** to launch the workflow on your file.

Tip:

To access Shuttle easily in the future (without using the menus):

1. Go to **Window > Toolbars > Shuttle** to open the Shuttle palette.



2. Drag it into your application's toolbar.
3. Click  to open the **Launch Workflow** window, or  to open the **Shuttle** window.

2. Reference Guide

2.1 Copyright Notice

© Copyright 2015 Esko Software BVBA, Gent, Belgium

All rights reserved. This material, information and instructions for use contained herein are the property of Esko Software BVBA. The material, information and instructions are provided on an AS IS basis without warranty of any kind. There are no warranties granted or extended by this document. Furthermore Esko Software BVBA does not warrant, guarantee or make any representations regarding the use, or the results of the use of the software or the information contained herein. Esko Software BVBA shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or inability to use the software or the information contained herein.

The information contained herein is subject to change without notice. Revisions may be issued from time to time to advise of such changes and/or additions.

No part of this document may be reproduced, stored in a data base or retrieval system, or published, in any form or in any way, electronically, mechanically, by print, photoprint, microfilm or any other means without prior written permission from Esko Software BVBA.

This document supersedes all previous dated versions.

PANTONE[®], PantoneLIVE and other Pantone trademarks are the property of Pantone LLC. All other trademarks or registered trademarks are the property of their respective owners. Pantone is a wholly owned subsidiary of X-Rite, Incorporated. © Pantone LLC, 2014. All rights reserved.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software are copyright © 1996-2002 The FreeType Project (www.freetype.org). All rights reserved.

Portions of this software are copyright 2006 Feeling Software, copyright 2005-2006 Autodesk Media Entertainment.

Portions of this software are copyright ©1998-2003 Daniel Veillard. All rights reserved.

Portions of this software are copyright ©1999-2006 The Botan Project. All rights reserved.

Part of the software embedded in this product is gSOAP software. Portions created by gSOAP are Copyright ©2001-2004 Robert A. van Engelen, Genivia inc. All rights reserved.

Portions of this software are copyright ©1998-2008 The OpenSSL Project and ©1995-1998 Eric Young (ey@cryptsoft.com). All rights reserved.

This product includes software developed by the Apache Software Foundation (<http://www.apache.org/>).

Adobe, the Adobe logo, Acrobat, the Acrobat logo, Adobe Creative Suite, Illustrator, InDesign, PDF, Photoshop, PostScript, XMP and the Powered by XMP logo are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Microsoft and the Microsoft logo are registered trademarks of Microsoft Corporation in the United States and other countries.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. 1986 2003. All Rights Reserved.

JDF and the JDF logo are trademarks of the CIP4 Organisation. Copyright 2001 The International Cooperation for the Integration of Processes in Prepress, Press and Postpress (CIP4). All rights reserved.

The Esko software contains the RSA Data Security, Inc. MD5 Message-Digest Algorithm.

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Sun Microsystems in the U.S. and other countries.

Part of this software uses technology by Best™ Color Technology (EFI). EFI and Bestcolor are registered trademarks of Electronics For Imaging GmbH in the U.S. Patent and Trademark Office.

Contains PowerNest library Copyrighted and Licensed by Alma, 2005 – 2007.

All other product names are trademarks or registered trademarks of their respective owners.

Correspondence regarding this publication should be forwarded to:

Esko Software BVBA

Kortrijksesteenweg 1095

B – 9051 Gent

info.eur@esko.com

2.2 What is Automation Engine

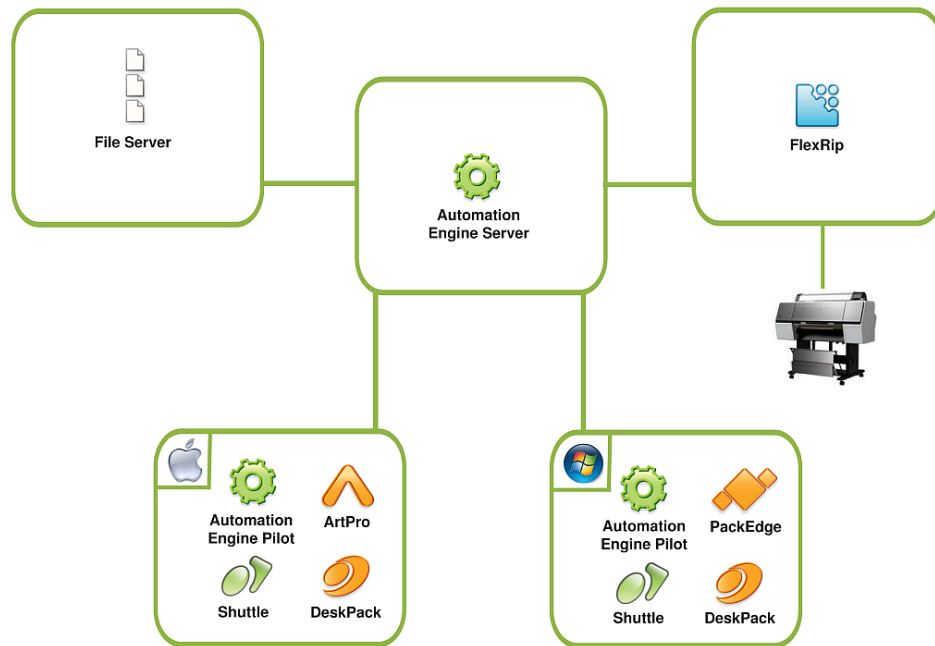
Automation Engine Architecture

Automation Engine is a **client/server** workflow software, running on **Mac and Windows**. This means that one machine runs the Automation Engine server, to which you can connect from all Mac and Windows machines in your network (using the Automation Engine **Pilot**).

Your Automation Engine server can connect to **graphical editing software** like ArtPro, PackEdge, Adobe Illustrator (using the DeskPack plug-ins)...

It can also connect to your **RIP** software to ensure professional output.

You can store your **files** either on the Automation Engine server, on a dedicated file server, or on different machines over the network.



Automation Engine Processing

Automation Engine processes your files using single **Tasks** (for example a Trap or RIP task), or **Workflows** (more powerful and flexible series of tasks that can take complete care of the file from arrival to final output).

Files and Jobs in Automation Engine

In Automation Engine, your files can be part of **Jobs** (customer orders that contain files and related customer information), that are located in **Containers** (local or remote shared folders that Automation Engine can access).

Depending on how you want to work with Automation Engine, you can link your jobs to customer **Products**, focus on job **Pages, Proofs, Plates...**

Automation Engine Integration

Automation Engine can not only connect to editing and RIP software, but also to **WebCenter** (Esko's collaborative web platform) and other Esko workflow software (**Nexus** and **Odystar**). Automation Engine can also be integrated with external systems.

2.2.1 The Automation Engine Pilot

The Automation Engine server software runs on a Windows server. Every Automation Engine action is performed through the Automation Engine client, called the **Pilot**.

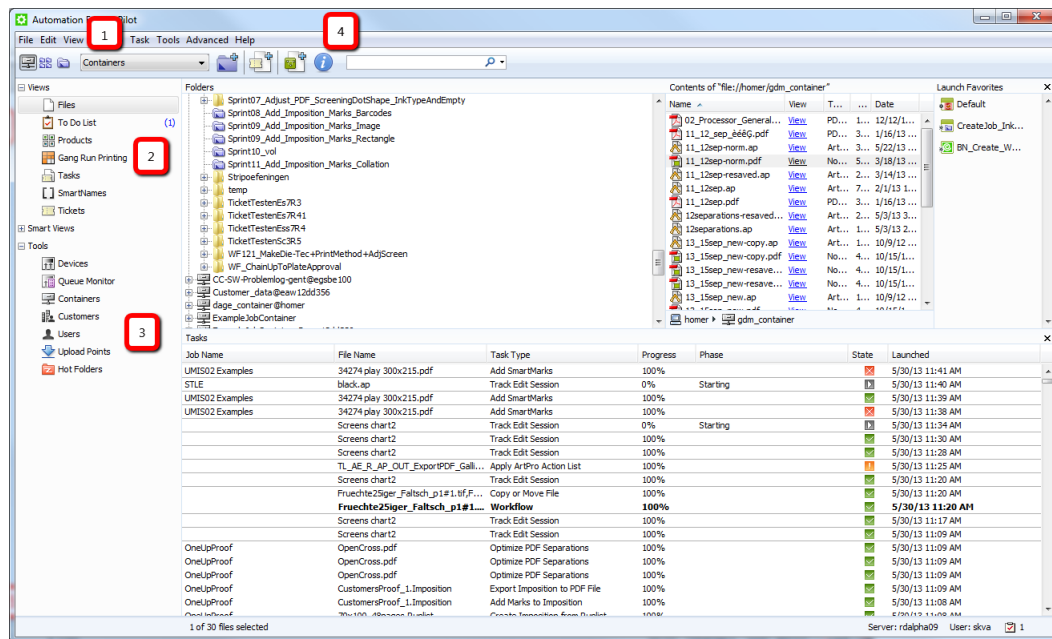
The Pilot can be installed on any Mac or Windows machine on your local network.

Depending on the user access rights you will set, operators can use the Pilot to:



- organize Jobs and files,
- create and modify tickets,

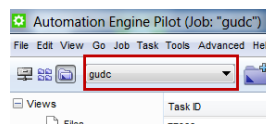
- launch and monitor tasks,
- control workflows and devices,
- administrate users and perform general configuration,
- etc.


The Pilot's Main Window







1. These icons show different modes in the Pilot:

- **Containers** : This shows all the data that Automation Engine can access, in the different Containers. Learn more in [Containers](#) on page 24.
- **Job Overview** : This lists all jobs existing in the Automation Engine jobs database. Learn more in [Jobs](#) on page 828.
- **Last used Job** : This shows the last job you worked on, and the job's data (files and folders).



- The **Views** highlight different parts of the Automation Engine functionality. Certain views are only accessible in defined workflows. Learn more in [Views](#) on page 22.
- The **Tools** area contains tools useful to administrate Automation Engine. Learn more in [The Tools](#) on page 616.
- The **Create Job**  button allows you to create a job.


- The **New Task**  button creates a new Ticket from the pilot.
- You can create a new workflow from the pilot using .
- Using the  button, you can access the information regarding the files.

Note: In the **Job Overview** mode, only **Create Job**  is available.

The contents of the central panes depend on the **View** you are in.

Tab support

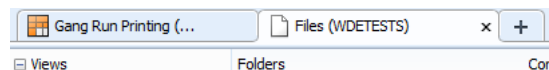
In any Automation Engine view, a new tab can be created by using one of the following methods:

- Click **File > New Tab** .
- Use the keyboard shortcut **Control-T** (Windows) or **Command-T** (Mac).
- If the tab bar is visible, you can also create a new tab by clicking the plus button in the tab bar .
- If the tab bar is visible, you can also create a new tab by right-clicking on the tab bar and choosing **New Tab**.

Note: The new tab will open with the same view that was active.

An opened tab can be closed by one of the following methods:

- Click on the close button in a tab (visible by hovering over the tab).




- Use the keyboard shortcut **Control-W** (Windows), or **Command-W** (Mac).
- Click on **File > Close Tab** .
- Right-click on the desired tab and choose **Close Tab**.

Note: To close all other tabs, save the desired one, you can also choose **Close Other Tabs** from this menu.

Full Screen Mode on Mac

When running the Automation Engine Pilot on Mac OSX or newer, you can enter and exit the Full Screen Mode by using one of the following methods:

- Click on the **Full Screen** button  in the top right corner of the main window
- Click on **Views > Enter Full Screen** .
- Use the keyboard shortcut **Command + Control + F**.

Views

The Pilot's **Views** highlight different parts of the Automation Engine functionality. You can use them to work with your data in different ways (see [Different Ways of Working with Automation Engine](#) on page 182).

Note:

- Depending on the Data View mode you are using (**Containers**, **Job Overview** or **Last used job**), you will have access to different views. Only those that make sense are shown.
- Depending on your [access rights](#), you may not be able to see some of the views.

Files

The **Files** view allows you to manage your jobs, files, favorite tickets, tasks... Learn more in [The Files View](#) on page 152 for more information.

To Do List

The **To Do List** view lists all actions that need user intervention. Learn more in [The To Do List View](#) on page 153 and [To Do List](#) on page 616.

Pages

The **Pages** view is only used in page workflows. It is extremely useful when working with page impositions. For more information, see [Working with Pages](#) on page 182.

Products

The **Products** view shows the items you added to your Products database. This concept is useful to customer who have re-prints. Products represent production data that you linked to a Job. It is the Job order that has the specifications how to output the product this time, with or without many other Product items.

Products are what you keep long term, until you are sure you will not have to produce them again. Jobs however are Work In Progress, temporary by definition. A Job should be removed when it is finished. Learn more in the dedicated chapter [Products](#) on page 876.

Proofs

The **Proof** view is only used in page workflows. It shows the proof sets and the proof details. Learn more in [Working with Proofs](#) on page 189.

WebCenter

This view enables Pilot users to see what documents are inside a WebCenter project, without having to use a browser. Read more in separate chapter [Integrating with WebCenter](#) on page 1346, in its section [The WebCenter View in Automation Engine Pilot](#) on page 1349.

Plates

The **Plates** view is only used in page workflows. It shows details about all plates associated with a job. It also includes advanced features for managing plates for jobs with multiple variants. Learn more in [Working with Plates](#) on page 192.

CDI

The **CDI** view shows all the files sent to the Digital Flexo Suite and used to image a flexo plate on a Cyrel Digital Imager. See [The CDI View](#) on page 159.

Gang Run Printing

The **Gang Run Printing** view in the Pilot shows an overview of all graphics waiting to be nested and printed on your available substrate sheets. It has two parts:

- The list of all substrates.
- The table of graphics waiting to be nested and printed on the selected substrate.

For more information, please see [The Gang Run Printing View](#) on page 1091 in the separate chapter [Working with Gang Run Printing](#) on page 1081.

Tasks

The **Tasks** view shows the running tasks, the tasks waiting for execution and the finished tasks. Learn more in [The Tasks View](#) on page 160. This list of tasks can be automatically cleaned up by using [Task Cleanup Rules](#) on page 682.

Devices

The **Devices** view lists all output devices connected to your Automation Engine server. It allows you to monitor, start and stop your devices and device queues.

Milestones

The **Milestones** view shows the milestones set for your job (in **Last used job** mode), or shows the milestones set for all jobs (in **Job Overview** mode). Learn more in [Milestones: Tracking Job Status](#) on page 854.

Tickets

The **Tickets** view shows a list of all tickets available on your server (both the default and your custom tickets). Learn more in [Tickets](#) on page 26.

Jobs

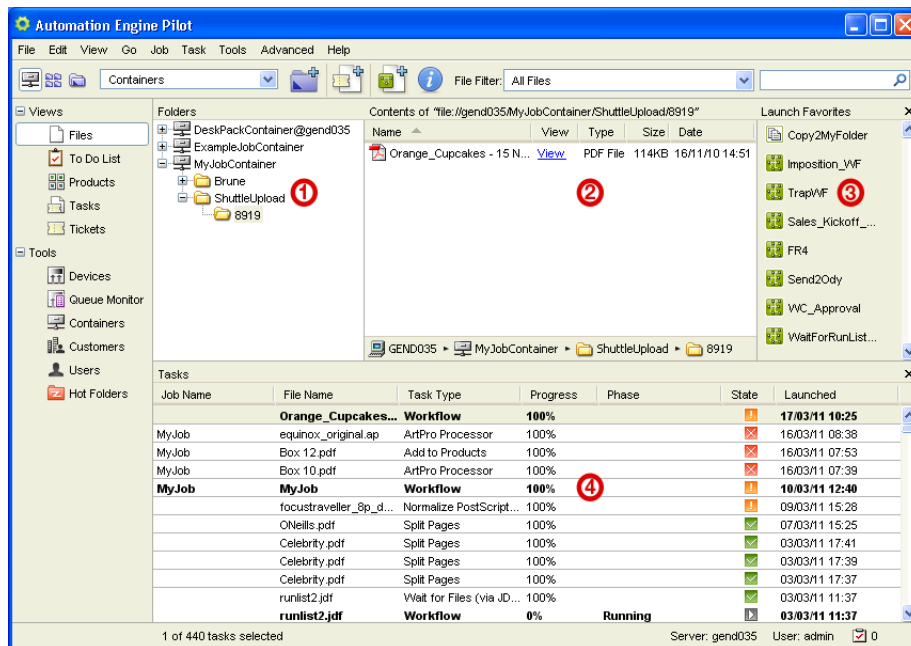
This view is only visible in **Job Overview** mode. It lists all or a filtered list of your jobs in the Automation Engine job database. Learn more in [The Job Views](#) on page 830.

SmartNames

This view lists all SmartNames defined on the server. It is not visible in **Job Overview** mode. Learn more in [The SmartNames View](#) on page 920.

The Files View

This view is available in **Containers** and **Job** modes.



1. The **Folders** pane shows your containers, folders and job folders (in **Containers** mode) or a job folder and its subfolders (in **Job** mode).
2. The **Contents** pane shows the contents of the folder you selected in the **Folders** pane. It only shows the contents of a job folder in **Job** mode.
3. The **Launch Favorites** pane allows you to add tickets to your favorites and find them again quickly to launch them. For more information, see [Favorite Tickets](#) on page 179.
4. The **Tasks** pane shows all the tasks launched for a job (if a job is selected), or all tasks launched on your server.

The To Do List View

This view is available in **Containers** and **Job** modes.

The **To Do List** view lists all actions that need user intervention. Among these actions are Messages, Checkpoints, Red Alerts, Milestone Actions.

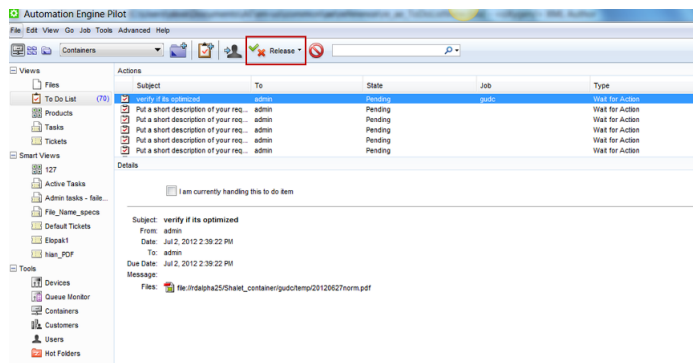
Use this view to view, modify, forward and delete these actions, and all related files.

Note: The **To Do List** view is job context dependent:

- in container mode you can see all To Dos,
- in a job context you only see the To Dos of that job.
-

you can release the jobs from this view by clicking on



**Tip:**

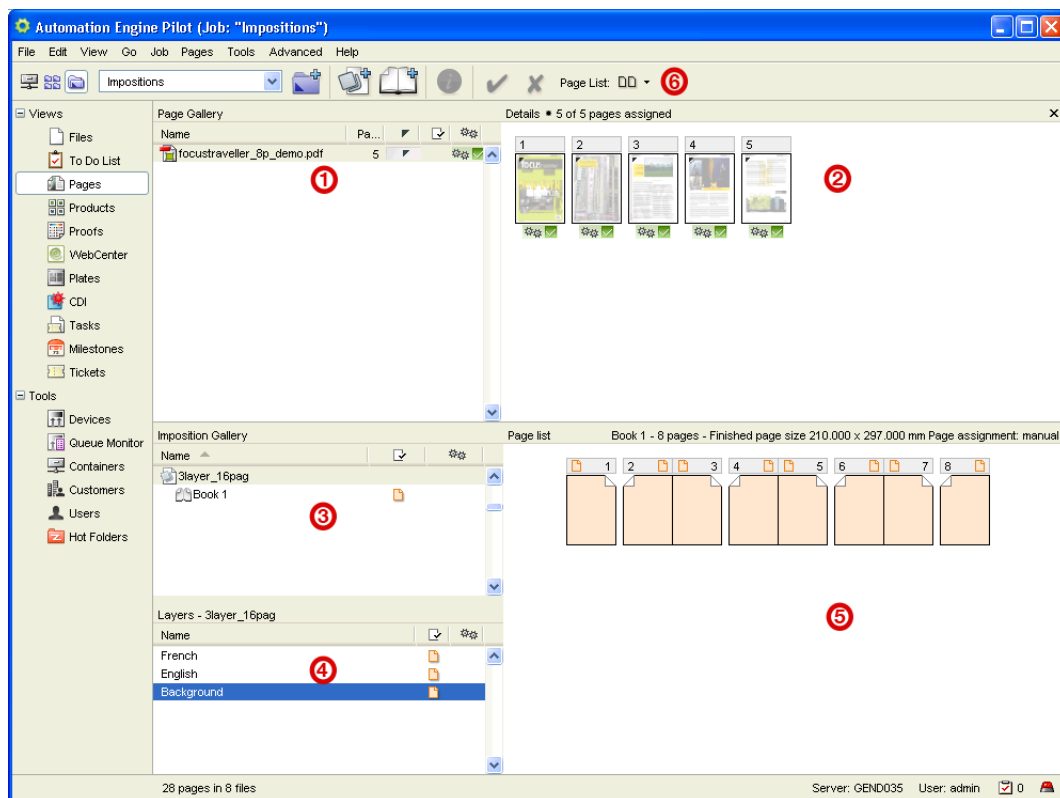
- Initially, the entries in the To Do List are not sorted. You can sort them by clicking the appropriate column heading.
You can also show / hide or reorganise columns by going to **View > Select Columns in 'To Do List'...**
- You can refresh the list using **Ctrl + R** or **Edit > Refresh**.

For more information, see [To Do List](#) on page 616.

The Pages View

This view is only available in **Job** mode. It is only used in page workflows.

The Pages View is divided into several panes. The top part of the view (the **Page Gallery** and the **Details** pane) displays both assigned and unassigned pages, the bottom part displays the impositions, layers and assigned pages.

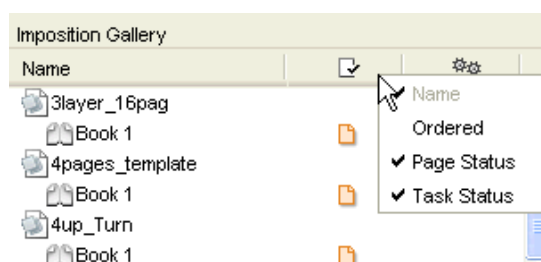


1. The **Page Gallery** pane lists all incoming files, indicates the number of pages and the assignment status.
2. The **Details** pane displays thumbnails of the individual pages of the selected file.
3. The **Imposition Gallery** pane shows the imposition structure with its different books. It also allows you to create books that later will be used in an imposition.
4. The **Layers** pane shows the different layers of the selected imposition. You can rename layers, approve them, and set them empty or blank.

Note: The **Layers** pane is only visible when the selected imposition or book contains more than one layer.

5. The **Page List** shows you which pages are assigned and which are not. For each page status information can be asked.
6. Click the **Page List** button to switch between thumbnail view or list view.

You can view both the task status and the page status information. Simply right-click the columns **Page Gallery** and / or **Imposition Gallery** pane(s) and select both **Task Status** and **Page Status**.





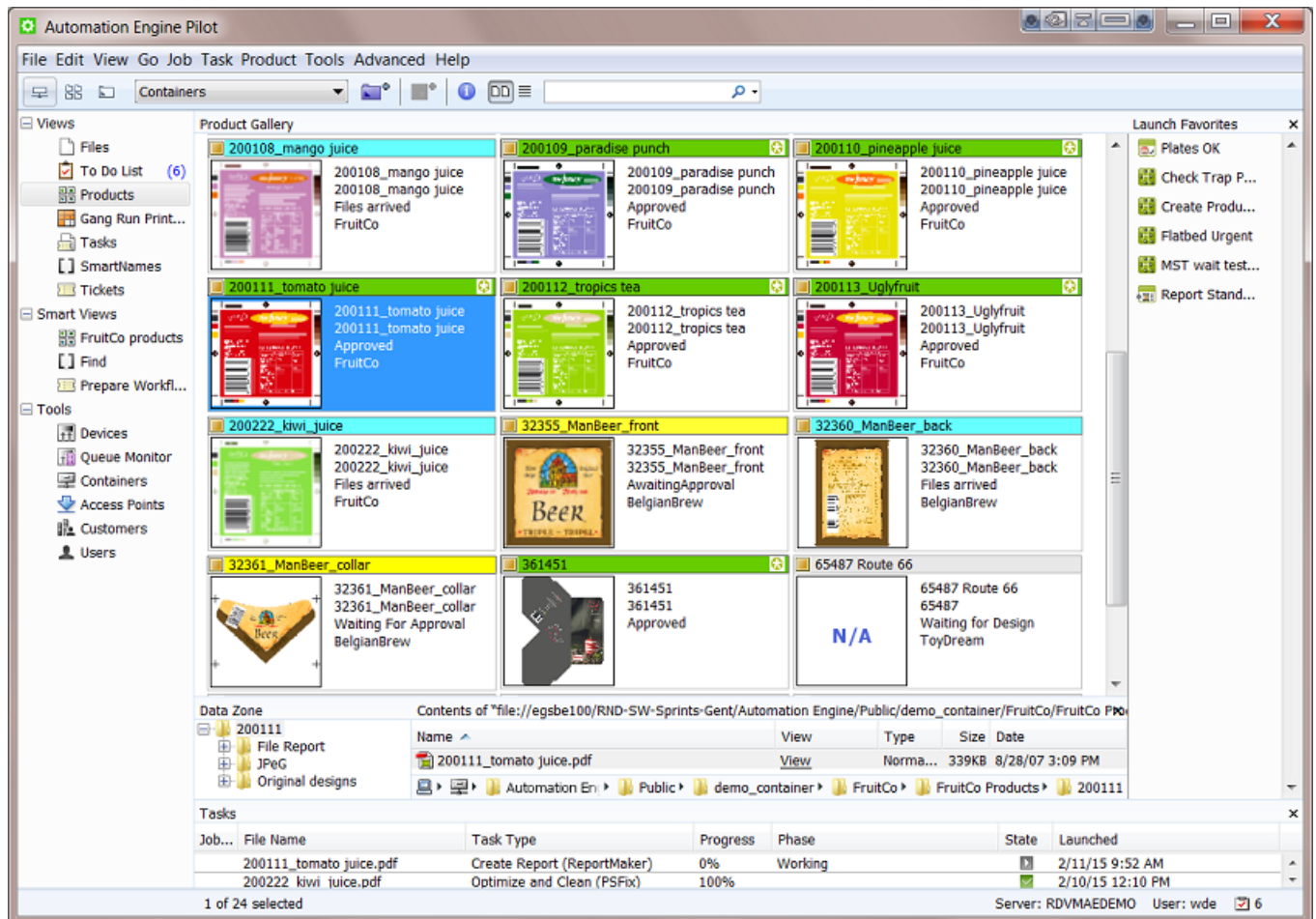
See [Working with Pages](#) on page 182 for more information.

The Products View

This view is available in **Containers** and **Job** modes.

The **Products** view is a user-friendly, searchable view on the Products database. You can use it to create new Products, search them and link them to Jobs.

You can see the Products in a thumbnail view  or a list view .



Working with Products is documented in a separate chapter [Products](#) on page 876.

The Proofs View

This view is only available in **Job** mode. It is only used in page workflows.

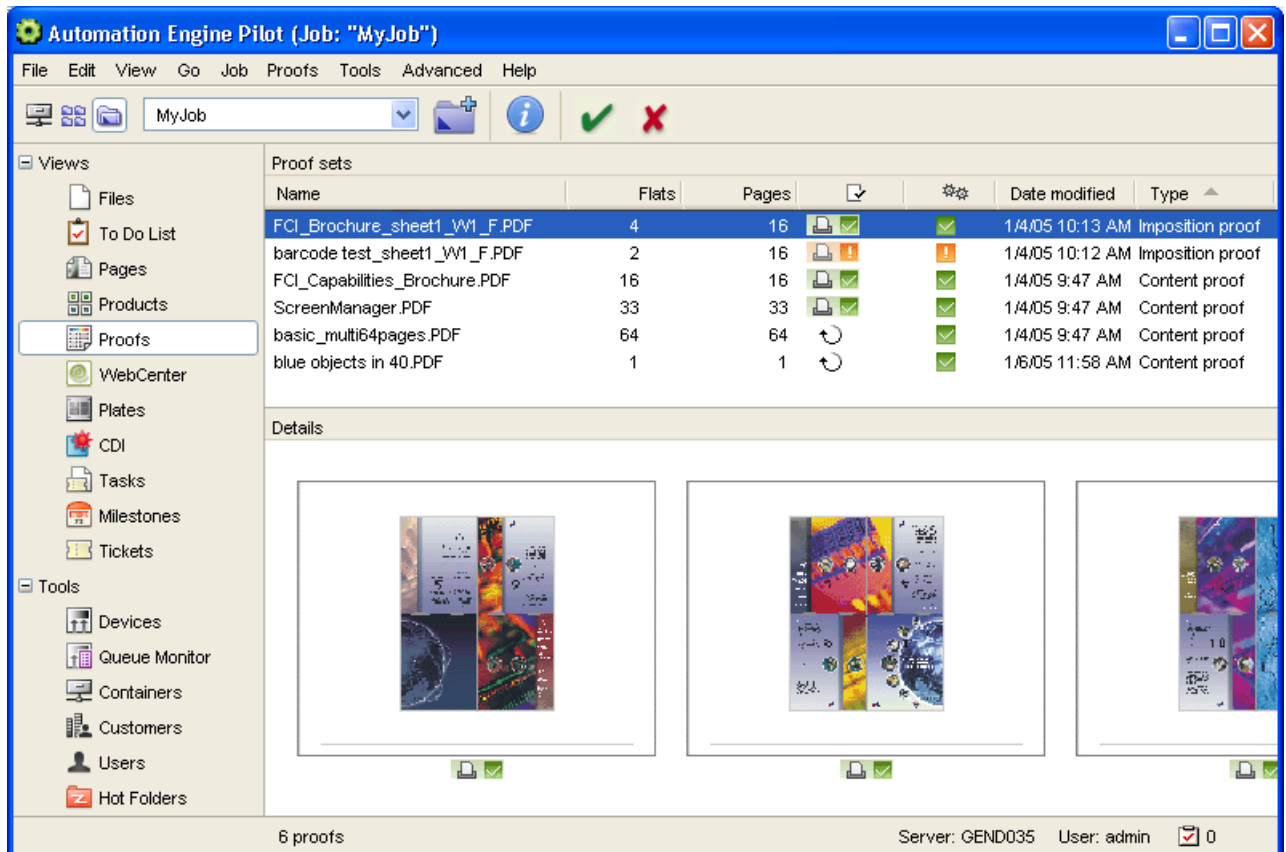
The Proof View consists of two panes that give a different view on the proof:

1. The **Proof Sets** pane, where the proof sets are listed.
2. The **Details** pane is a visual presentation of all proofs of the set.

The Proof Sets pane shows you the names of the proof sets, both hardcopy proofs and softproofs, as well as the number of flats, pages, status, the date on which they were last modified and information

about the kind of proof that has been made (**Content proof** if a file is proofed, or **Imposition proof** if an imp file is proofed).

Thumbnails of the proofs can be viewed with the corresponding application, for example Acrobat for PDF proofs.



The Info button will open a properties file of that page (proof).

See [Working with Proofs](#) on page 189 for more information.

The WebCenter View

This view is only available in **Job** mode.



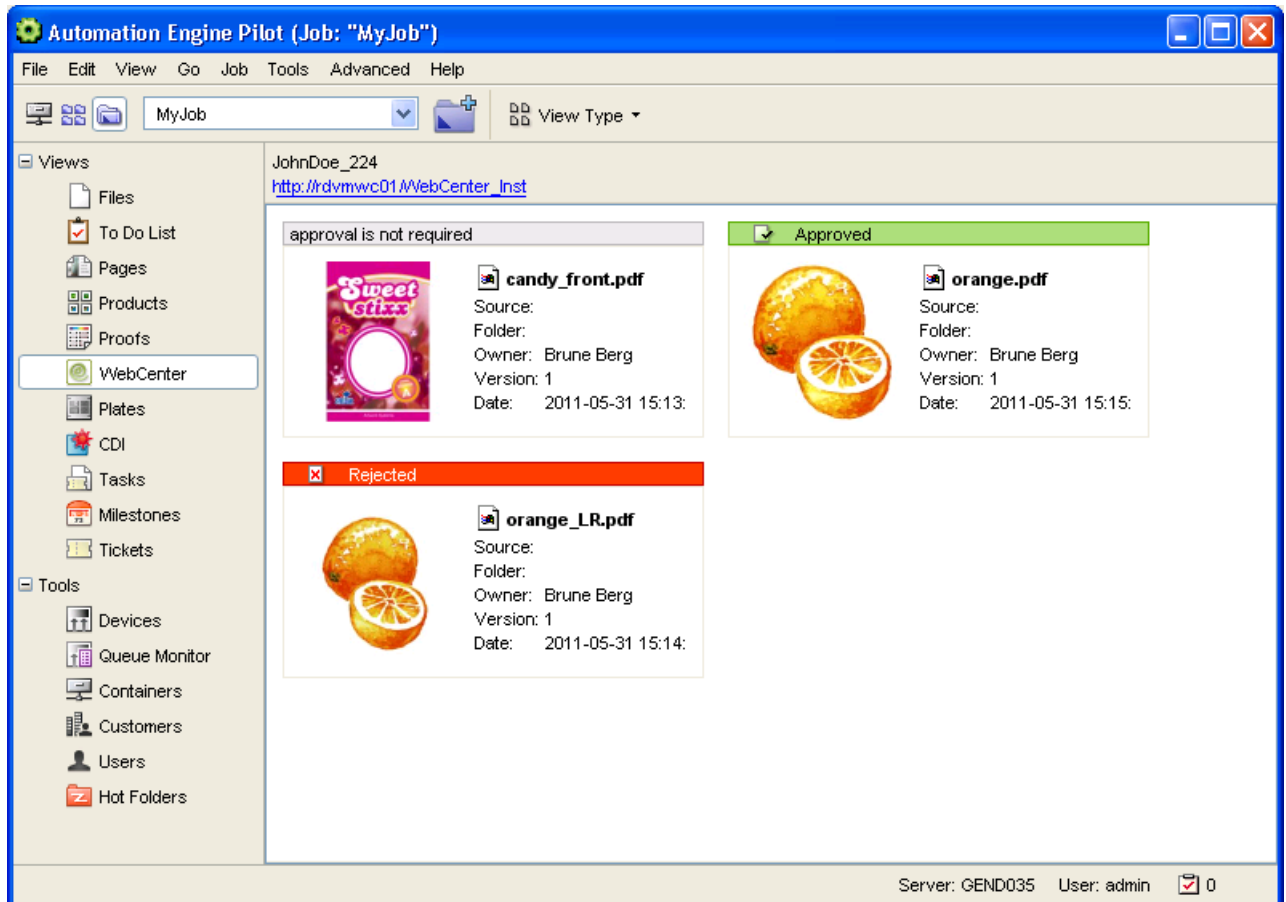
Attention: You need to have configured the connection to your WebCenter site before you can use this view. See [WebCenter Sites - OLD](#).

Jobs that have been published on WebCenter (with the [Publish on Web task](#) or the **Publish imposition on WebCenter Chain**) will appear in the WebCenter View.

Those jobs can have different statuses:

- When a job arrives on the WebCenter site and approval is required, its status will **pending**.
- Files which do not require approval are marked with **approval is not required**.
- After a file is approved or rejected in WebCenter, you can see its **approved** or **rejected** status in the WebCenter view.

The status will be updated in Automation Engine as soon as the job is approved or rejected in WebCenter.



Tip:

- Press F5 to update the WebCenter view to the latest information.
- Click the link in the top-left corner to open WebCenter in a browser window.

See [Review and Approval of Page Impositions using WebCenter](#) on page 83 in the User Guide for more information.

The Plates View

This view is only available in **Job** mode. It is only used in page workflows.

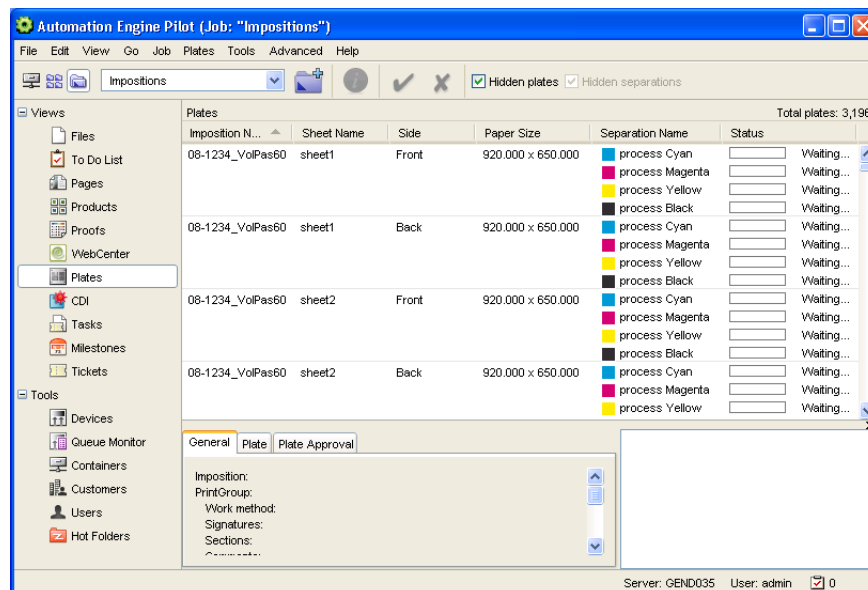
Note: In a page workflow, the job's inks are also used for creating "expected" separations (and corresponding plates) in the **Plates View**. This means that an expected separation will be created for each ink and for each sheet. If the assigned pages contain inks that are not listed in the inks list, "unexpected" separations will be created.

- The Plates View makes it easier to manage the most important deliverables that prepress provides to the press room, plates.

- It gives an overview of the total number of plates that have to be output.
- It allows you to track the progress of a job at a very detailed level.
- It allows you to detect discrepancies between the expected plate output and the supplied data very easily.

For example, when a sheet side is supposed to have only a black plate, but one of the supplied pages for that sheet side contains CMYK, this can be easily seen in the Plates overview.

The same is true for the inverse situation (for example when an expected Pantone plate is not being realized).



See [Working with Plates](#) on page 192 for more information.

The CDI View

This view is available in **Job Overview** and **Job** modes.

The **CDI** view shows all the files sent to the Digital Flexo Suite (using the [Make Plate on CDI task](#)) and used to image a flexo plate on a Cyrel Digital Imager.

You can view the flexo plates created for all jobs (in **Job Overview** mode) or for a particular job (in **Job** mode).

You can double-click an entry in **Job Overview** mode to see all the plates and len files corresponding to that job in **Job** mode.

You can also select an entry to view its **Details** at the bottom of the window.

The screenshot shows the 'Automation Engine Pilot' software interface. The main window displays a table of 'FlexoPlates' with columns for Order ID, Customer Name, Manufacturer, Type, Due Date, State, Launched, Used Area, Waste, and Error. The table contains several rows, with the row for Order ID 2011_45898 highlighted. Below the table, the 'Job Details' section shows information for job 'jobke3', including Order ID, Due Date, Customer ID, and Customer Name. A small thumbnail image of a FlexoPlate is visible on the right side of the job details section.

Order ID	Customer N...	Manufacturer	Type	DueDate	State	Launched	UsedArea (sq...	Waste (sqcm)	Error
TestBook	BelgianBrew ...	Flint	ACE_114	30/10/09 17:03	Done	18/02/11 15...	4,391.58	21,108.80	
jobk5		Flint	ACE_114	04/02/11 15:48	Done	03/02/11 15...	789.90	24,710.48	
2011_4578	BakeMark De...	Flint	ACE_114	15/02/11 11:48	Done	14/02/11 11...	4,911.01	20,589.37	
		Flint	ACE_114	17/02/11 09:44	Done	16/02/11 09...	516.29	12,233.90	
2011_4578	BakeMark De...	Flint	ACE_114	18/02/11 11:01	Done	17/02/11 11...	4,911.01	20,589.37	
2011_45898	Frozen Fish I...	Flint	ACE_114	25/03/11 14:26	Done	24/03/11 14...	789.90	9,249.62	
2011_45898	Frozen Fish I...	Flint	Ace_114	07/04/11 10:54	Done	06/04/11 10...	2,065.18	2,954.58	
2011_452111	Chen Consulti...	Flint	Ace_114	06/04/11 10:57	Done, with errors	07/04/11 10...			4
2005_00001	BB King	Dupont	ACE_114	20/04/11 11:00	Done	31/05/11 16...	1,109.00	7,675.58	
2005_00001	BB King	Dupont	ACE_114	20/04/11 11:00	Done, with errors	31/05/11 16...			1
2005_00001	BB King	Dupont	ACE_114	20/04/11 11:00	Done	31/05/11 15...	1,109.00	7,675.58	

Job Details:

Job Name: **jobke3**
Order ID: 2011_45898
Due Date: 25-Mar-2011 14:26:48
Customer ID: 15103
Customer Name: Frozen Fish International GmbH
Folder: file://begeleppem/examplejobcontainer/Jobs/jobke3LenFiles/1223

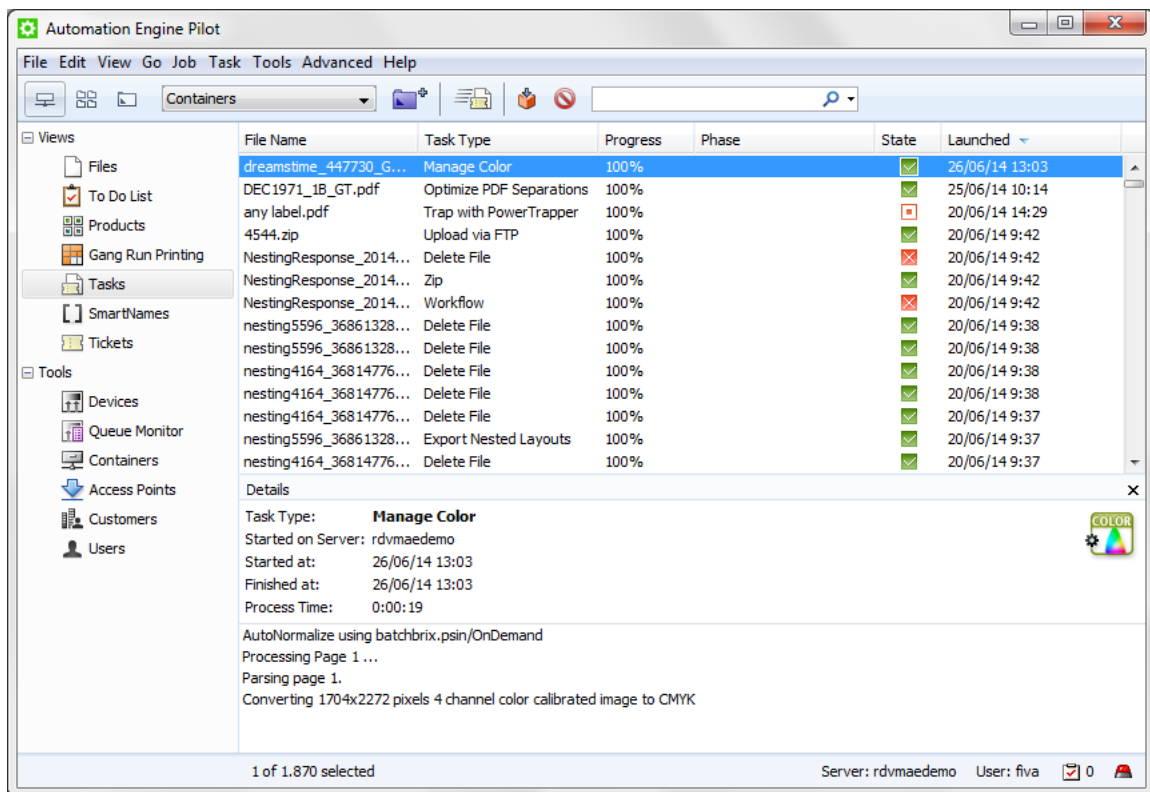
1 of 40 selected Server: begeleppem User: admin

See [Make Plate on CDI](#) on page 364 for more information.

The Tasks View

This view is available in **Containers** and **Job** modes.

- In **Containers** mode, it shows all tasks and workflows.
- In **Job** mode, it shows all tasks and workflows run for that job.



Tasks

For each entry in the **Tasks** list, you can see the **Job Name**, the **File Name**, the **Task Type**... You can show / hide or reorganise columns by going to **View > Select Columns in 'Tasks'**...

Details

Select an entry in the **Tasks** list to show its **Details** below. They include extra information, like when processing was **Started** and **Finished**, and some processing details.

Note: For more information on how to run tasks and workflows on files, see [Launching Tasks on Files](#) on page 67 and [Workflows](#) on page 986 .

Display Related Tasks

If you want to filter all tasks in this view to only the ones you are interested in, you can select a few desired tasks and choose to **Display** the **Related Tasks**. This way it's easier for you to follow up your tasks, spot user interactions and possible errors in your workflows.

To display all tasks that are related to one or more workflows, proceed as follows:

- Select one or more tasks, belonging to the desired workflows.
- Right-click on one of the chosen tasks.
- Choose **Display Related Tasks**.

A filter on the Tasks list will occur, focusing on the chosen workflows. In this filter, only tasks belonging to the chosen workflows are shown. This is indicated in the scope bar as **Focused on X workflows**.

Focused on 2 workflows Search: All File Name Task Type	
File Name	Task Type
4544.zip	Upload via FTP

To reveal all tasks once more, click on **Done** in the right-hand side of the **Tasks** view.

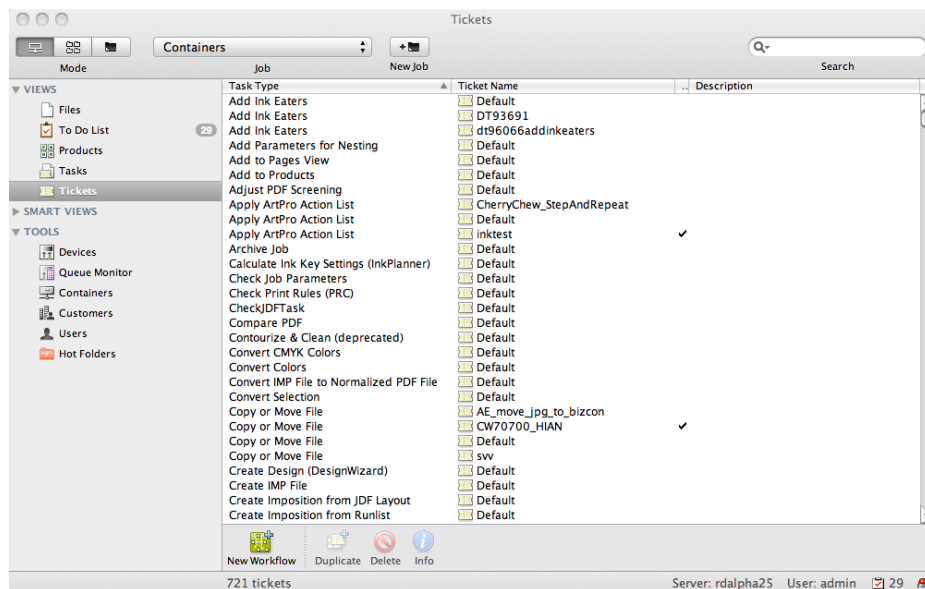
The Milestones View

The [Milestone Views](#) are documented in the dedicated chapter on [Jobs](#) on page 828.

The Tickets View

This view is available in **Containers** and **Job** modes.

You can see all the available tickets in the Pilot's **Tickets** view.



You can see:

- default and custom tickets,
- global and job tickets,
- public tickets.

See [Tickets](#) on page 26 for more information about tickets.

The Jobs View

[The Job Views](#) on page 830 are documented in the dedicated chapter on [Jobs](#) on page 828.

The SmartNames View

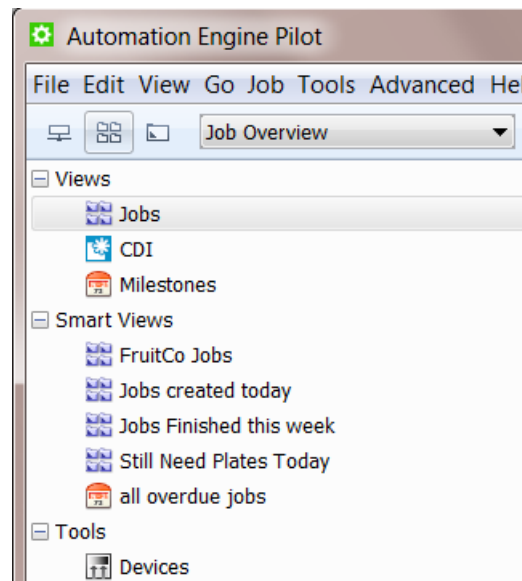
[The SmartNames View](#) on page 920 is documented in the dedicated chapter on [SmartNames](#) on page 916.

The Gang Run Printing View

The *The Gang Run Printing View* on page 1091 is documented in the chapter dedicated to *Working with Gang Run Printing* on page 1081.

Smart Views

Smart Views allow to save a filter as a custom view.



You can do this for all filters for the items available under **Views**. You can create Smart Views for all View types: lists of Tickets, Tasks or Jobs, Products, Milestones etc. Smart Views also makes it easier to and to navigate between views.

How to create a Smart View is explained below.

You can modify a Smart View's filter setting by double clicking the Smart View name.

You can remove a Smart View by right-clicking it and choosing **Remove from Views Bar**.

Creating Smart Views

As an example, follow these steps to create a Smart View that will filter all tasks based on these criteria:

- Belong to a chosen operator/administrator
 - Launched during the past two days
 - Failed
1. In the Automation Engine Pilot, go to the **Containers** view and then click **Tasks** to see an overview of all tasks.
 2. Press **Ctrl + F** to open the **Search** bar and see the extra filter options that can help you to limit your search results.

Job No.	Task ID	File Name	Task Type	Progress	Phase	State	Launched	Operator
EA09	64107	Testfile.pdf	FastVariants - Split (deprecated)	100%			5/16/12 11:01 AM	bn@rdalpa25
	64106	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 10:53 AM	admin@rdalpa25
	64105	G_VR_CheckPrintRulesPRC.pdf	Copy or Move File	100%			5/16/12 10:52 AM	admin@rdalpa25
	64104	11_12sep.pdf	Check Print Rules (PRC)	100%			5/16/12 10:49 AM	admin@rdalpa25
	64103	11_12sep.pdf	Normalize PDF File	100%			5/16/12 10:49 AM	admin@rdalpa25
	64102	Color_HPI-CMYKOVG_v2-Normalized.pdf	Check Print Rules (PRC)	100%			5/16/12 10:47 AM	admin@rdalpa25
EA03	64101	527087.mfg.527087_Lut.ap	Apply AnPro Action List	100%			5/16/12 10:38 AM	admin@rdalpa25
EA03	64100	527087.mfg.527087_Lut.ap	Workflow	100%			5/16/12 10:38 AM	admin@rdalpa25
EA03	64099	C_DO_PrintDigitalPress.pdf	Print on Digital Press (Track) on...	100%			5/16/12 10:38 AM	admin@rdalpa25
EA03	64098	527087.mfg.527087_Lut.ap	Apply AnPro Action List	100%			5/16/12 10:37 AM	admin@rdalpa25
EA03	64097	527087.mfg.527087_Lut.ap	Workflow	100%			5/16/12 10:37 AM	admin@rdalpa25
EA03	64096	527087.mfg.527087_Lut.ap	Apply AnPro Action List	100%			5/16/12 10:36 AM	admin@rdalpa25
EA03	64095	527087.mfg.527087_Lut.ap	Workflow	100%			5/16/12 10:36 AM	admin@rdalpa25
EA09	64094	Serig TRANQUERA JOVEN 133 X 35 13	Prepare for Viewer	100%			5/16/12 10:34 AM	bn@rdalpa25
EA09	64093	Serig TRANQUERA JOVEN 133 X 35 13	FlexRip/T4 on Awantra-rip1-srv	100%			5/16/12 10:33 AM	bn@rdalpa25
EA09	64091	Serig TRANQUERA JOVEN 133 X 35 13	RP to Screened TIFF File on Awa...	100%			5/16/12 10:33 AM	bn@rdalpa25
EA09	64090	TRANQUERA JOVEN 133 X 35.pdf	Optimize and Clean (PSDF)	100%			5/16/12 10:32 AM	bn@rdalpa25
EA03	64089	527087.mfg.527087_Lut.ap	Apply AnPro Action List	100%			5/16/12 10:24 AM	admin@rdalpa25
EA03	64088	527087.mfg.527087_Lut.ap	Workflow	100%			5/16/12 10:24 AM	admin@rdalpa25
EA07	64087	0062640_00 SR-64086.ap	Send to Nexus	100%			5/16/12 10:20 AM	admin@rdalpa25
EA07	64086	0062640_00 SR.ap	Apply AnPro Action List	100%			5/16/12 10:14 AM	admin@rdalpa25
EA09	64085	Test size arpo.pdf	Workflow	100%			5/16/12 9:55 AM	bn@rdalpa25
EA09	64084	Test size arpo.pdf	Workflow	100%			5/16/12 9:54 AM	bn@rdalpa25
EA09	64083	Test size arpo.pdf	Workflow	100%			5/16/12 9:53 AM	bn@rdalpa25

3. Set these filters:

- Set the first filter to **Operator** + **is** + <the operator whose task you want to see>. You can browse for the operator/administrator of your choice by clicking the **Browse** button in the Search field (1).
- Open a second filter line by clicking the **+**-button (2) in the first filter line.
- Set the following parameters for this second filter: **Launched date** + **is** + **within last** + **2** + **days**

Job No.	Task ID	File Name	Task Type	Progress	Phase	State	Launched	Operator
	64122	G_VR_CheckPrintRulesPRC.pdf	Rules (PRC)	100%			5/16/12 11:25 AM	admin@rdalpa25
	64121	G_VR_CheckPrintRulesPRC.pdf	Rules (PRC)	100%			5/16/12 11:24 AM	admin@rdalpa25

4. Click the **+**-sign in the second filter line while holding down the **Alt**-key.

It changes into a **...** - sign and allows you to further specify that second filter line by adding one or more criteria to it.

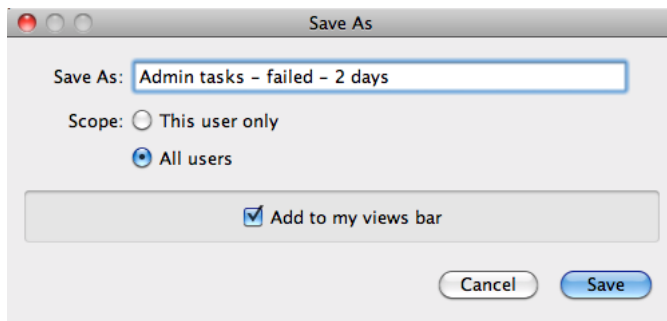
5. Add these filters:

- All of the following are true
- State is 'cancelled'

Job No.	Task ID	File Name	Task Type	Progress	Phase	State	Launched	Operator
	64122	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 11:25 AM	admin@rdalpa25
	64121	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 11:24 AM	admin@rdalpa25
	64120	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 11:24 AM	admin@rdalpa25
	64119	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 11:24 AM	admin@rdalpa25
	64118	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 11:22 AM	admin@rdalpa25
	64106	G_VR_CheckPrintRulesPRC.pdf	Check Print Rules (PRC)	100%			5/16/12 10:53 AM	admin@rdalpa25
	64104	11_12sep.pdf	Check Print Rules (PRC)	100%			5/16/12 10:49 AM	admin@rdalpa25
	64102	Color_HPI-CMYKOVG_v2-Normalized.pdf	Check Print Rules (PRC)	100%			5/16/12 10:47 AM	admin@rdalpa25
	64076	boost.grid	Track Edit Session	100%			5/15/12 4:43 PM	admin@rdalpa25
	64075	boost.grid	Track Edit Session	100%			5/15/12 4:33 PM	admin@rdalpa25
	64072	boost.ap	Track Edit Session	100%			5/15/12 4:31 PM	admin@rdalpa25

6. Click **Save** to save these filter settings in a Smart View and give it an appropriate name.

- Select the option **All users** to make this Smart View accessible for other users all well.
- Select the option **Add to my views bar** to add this smart view to the Views bar.



The Smart View 'Admin tasks - failed - 2 days' is now part of the **Smart Views** list.

Pilot Preferences

Go to **Edit > Preferences** to open the Pilot's Preferences.

Tickets

- **Show default tickets:** select/deselect this option to show/hide the default tickets.
- **Use bold ticket names in task launching menu:** select this if you want to see the ticket names in bold when doing **New Task** or **Launch with** on a file.
- **Show only Public Parameters in Tickets:** this preference is only available if you have the **Tickets: Show All Tickets and their Parameters (Public and other)** access right (see [Access Rights](#) on page 702).

It is selected by default. Deselect it to only see public tickets and public parameters in the Pilot (you will see the Pilot as if you didn't have the **Tickets: Show All Tickets and their Parameters (Public and other)** access right). You can select that preference again at any time to go back to the full Pilot.

This is useful for administrators who want to check how users without the full Pilot access right will see the tickets and their parameters.

- **Units:** select the units you want to work with for **Dimension** (mm, cm, mil, inch, point), **Resolution** (ppi, pppm, ppcm) and **Ruling** (lpi, lpmm, lpcm).
- Enter the number of **Decimals** you want the system to display.

Files

The **Preview Size** allows you to define the size of the preview in the **Info** dialog for example (in pixels).

Open Normalized PDF files with

Indicate which application should become the default application when opening Normalized PDF files from within the Pilot.

Normalized PDF files have a .PDF extension, which implies that the OS will not make a distinction between Normalized PDF files and regular PDF files.

By default PDF files will be opened by the OS with Adobe Acrobat Reader if it is installed. However, when working with the Pilot, you probably will want to see Normalized PDF files opened in an Esko

editor. As making this distinction is not possible on OS level, you will have to specify here which application you prefer to use.

So, if a PDF file is double-clicked in the Pilot, and it happens to be a Normalized PDF file, the application selected in the Files tab will be used.

On OS level (for example in Windows Explorer or Macintosh Finder), Adobe Acrobat Reader will be the application that is started up when you double-click a Normalized PDF file.

This preference will be taken into account:

- In the **Files** View, when double-clicking the file or selecting **Open** from the pop-up menu.
- In the **Task** View, when selecting **Open Output File** from the pop-up menu.
- In the **To Do List**, when clicking a URL link to a PDF.

Tip: Still, the Pilot also offers **Open with** which allows you to indicate the non-default choice.

Pilot

You can indicate the **Number of recently used jobs in the job selector**. By default this is set to 10. Changes will be applied when you log back into your Pilot.

Jobs

Multi-Job Removal

Select your preferences when you remove multiple jobs at once:

- **Remove Jobs even when they are in use by other users.**
- **Remove Jobs even when they contain unfinished tasks.**

2.2.2 Data Management in Automation Engine

Automation Engine works with [Containers](#) on page 24, [Jobs](#) on page 828, [Products](#) on page 876, files and folders.

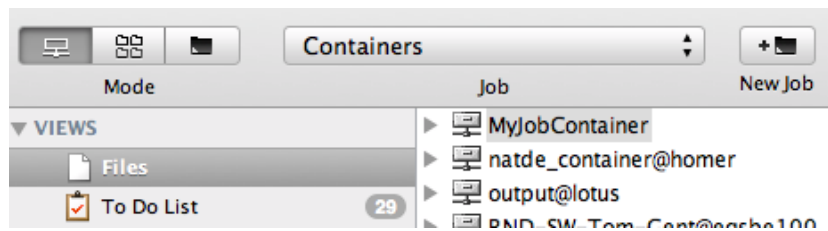
Customers with page (imposition) workflows can use the extra modes [Pages](#), [Proofs](#) and [Plates](#).

Containers

A **Container** is a network data share that contains the data that Automation Engine can work with. You can make (part of) an existing network share a Container or you can have Automation Engine create such network shares. These Containers can be located either on your Automation Engine server or another computer in the same LAN.

By default, you have an empty **ExampleJobContainer**, on the disk drive that Automation Engine was installed on.

You can have many Containers. You can see and browse through them in the Pilots' **Containers** mode.




To create a Container, see [Creating a Container](#) on page 691.

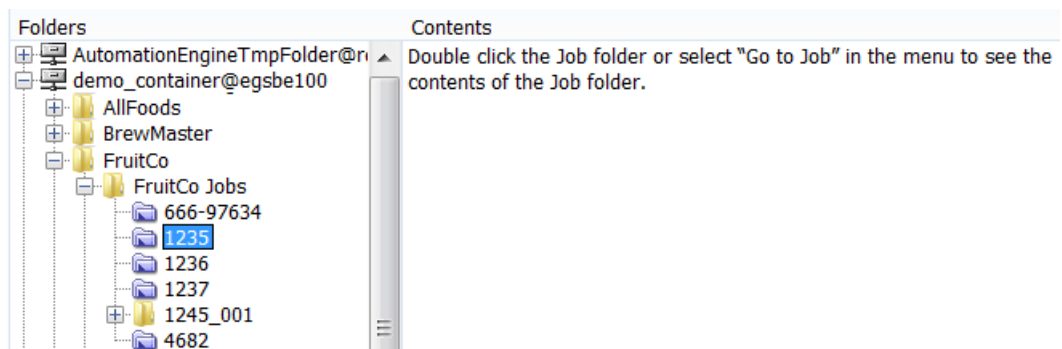
Jobs

What is a Job?

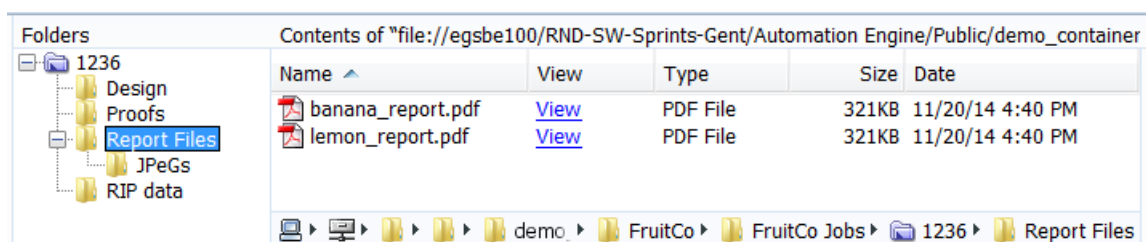
A Job represents a to do for Automation Engine. A Job contains all the files and information related to a specific job-order. A Job has a unique name. A Job represents a job order and is also WIP: Work In Progress. This means that in Automation Engine you should remove Jobs that are finished. Still, the Jobs database can hold up to 100.000 Jobs.

It has a blue icon  and is stored in the Automation Engine Job Database.

In the **Containers** mode, you can see your Job in its Container.



In the **Last used Job** mode, you can see your Job's subfolders and files.



Job Metadata (Job Setup Parameters)

You can attach an extended set of metadata to every Job: customer information, inks to use, bar codes, impositions, categorizations or any other custom parameter...). This metadata is stored in the Automation Engine database, and can be used when launching tasks on the Job. Ideally, these

parameters are not set here manually but came in automatically when the Job was created by your business system.

Job Automation

Automation Engine is very Job-centric. Workflows can take their settings from the Job setup. They can even be launched automatically when the Job reaches a certain status (for example: a new page has been delivered, the sheet is complete, the **Milestone** changed to 'Approved' etc...).

Note: When you choose not to work with this Jobs concept, you can still get a similar level of automation by using alternative tools like [Workflow Parameters](#).

Learn more in the dedicated chapter [Jobs](#) on page 828.

2.2.3 File Processing in Automation Engine

In Automation Engine, you can process your files using single **Tickets** or **Workflows** (a sequence of Tickets).

Launching those tickets or workflow results in (a series of) **Tasks**.

A **Ticket** contains all the parameters for that task.

Tasks

What is a Task?

A task is an action executed on a file, a folder or a Job on the Automation Engine server.

Types of Tasks

There are seven types of tasks:

1. **Conversion** tasks such as **Export to PDF, Export to PostScript...**
2. **File Creation** tasks such as **Create Wrapper File, Create PAF / JPG / XML...**
3. **File Editing** tasks such as **Contourize & Clean, Trap...**
4. **Verification** tasks such as **Check Job Parameters, Check Print Rules, Preflight...**
5. **Administration** tasks such as **Prepare to Archive, Upload via FTP, Zip...**
6. **RIP** tasks.
7. **Device Output** tasks.

Task Settings

You can define settings for the tasks, and save these settings as **Tickets** to reuse them (see [Tickets](#) on page 26).

Task Progress

When a task is launched, you can monitor its progress in the Pilot. See [Checking the Status of the Task](#) on page 37.

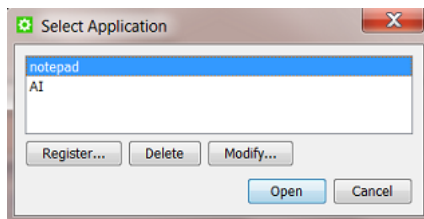
Task Menu

The Task Menu contains the following items:

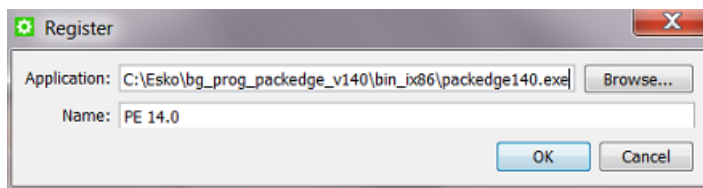
- **Open Task** [CTRL + O]: opens the selected task(s), allowing you to relaunch them (with the same, or with modified settings).
- **Relaunch** [CTRL + L]: relaunches the selected task(s) without opening them first.
- **Open Output File** [CTRL + Shift + O]: opens the output file that was generated by the task with the associated application.
- Right-click a file to select '**Open Output File with...**'. If no application has been registered, the default application for the selected file type will be used.

To register and select an application:

1. Click 'Register'



2. Browse to the location on the hard disk to select the application's .exe file.

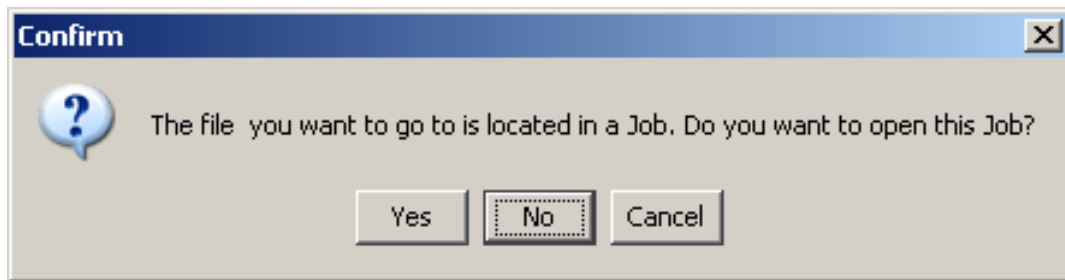


3. Click 'OK'. The path and the official name of the application appear in the Register dialog box.
 4. Click 'OK'. You can now select this application to open a file.
- **Go to Input / Output File**: selects the input or output files of the selected task. The Automation Engine Pilot will automatically switch to the Files View and select the file in the appropriate folder.

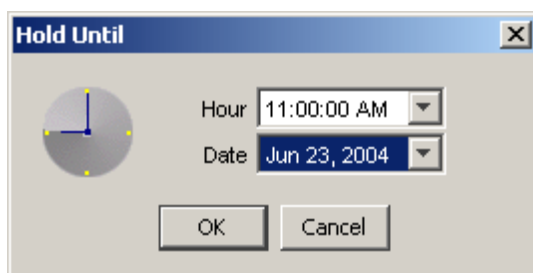
It is possible that the folder containing that file is no longer available in the current containers, because it has been deleted or because the container is no longer available. In this case, an error message will appear:



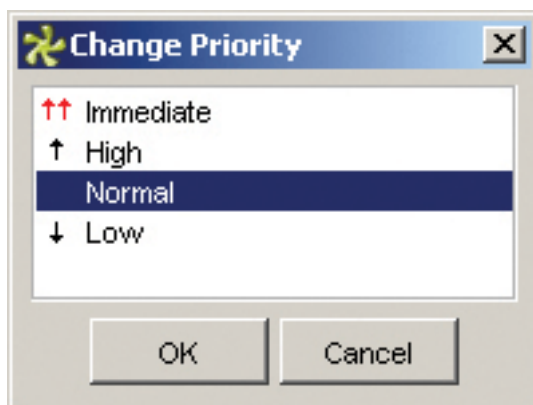
If the file is found, the container and path are opened in the folder selector, and the file is selected if it is still available. If the file is located inside another Job than the current Job, a message is shown asking you if the Job should be opened or not:



- **Hold**[CTRL + K]: holds a task until it is 'released' manually.
- **Hold Until...** [CTRL + Shift + K]: holds a task until a predefined point in time.



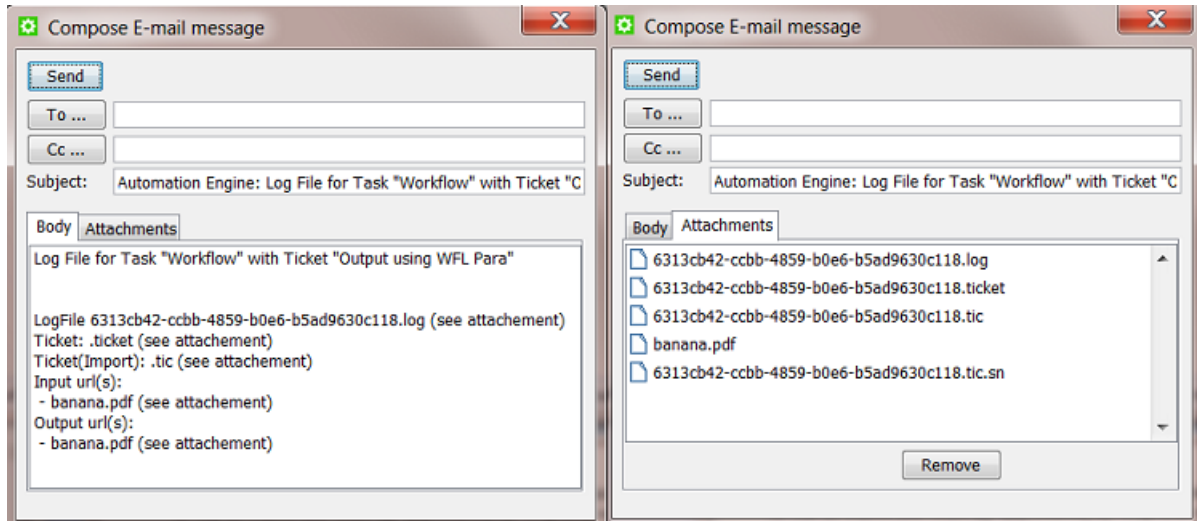
- **Release** [CTRL + G]: releases a task that's 'On Hold'.
- **Cancel** [CTRL + Y]: cancels the launch of a task.
- **Priority...** [CTRL + U]: allows you to change the task's priority.



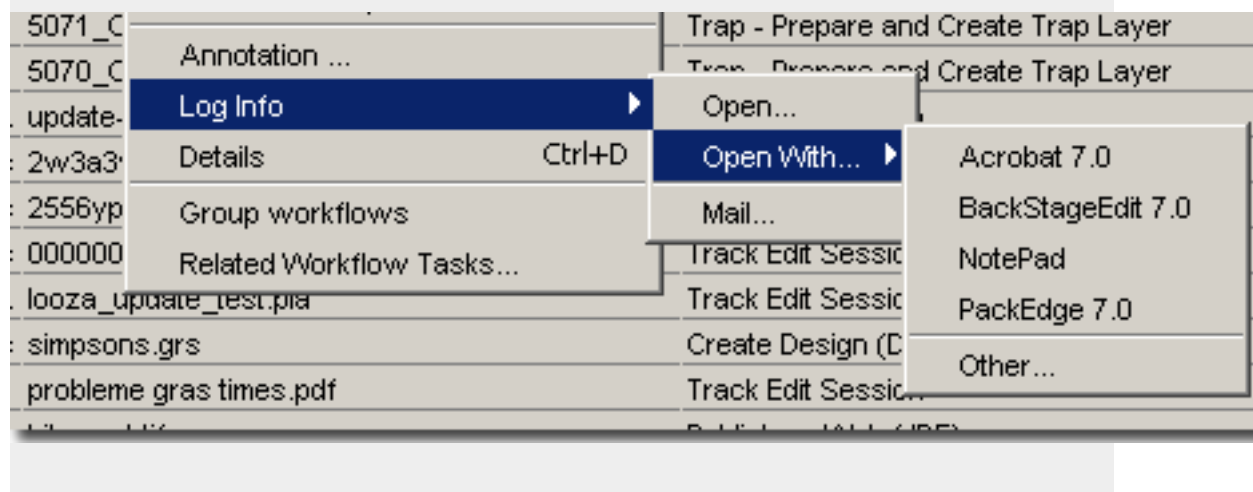
- **Move to History** [CTRL + M]: moves the selected task(s) to the [Task History](#).
- **Annotation...** adds or modifies an annotation. When the annotation pops up, click 'Save' to save modifications.
- **Log Info...** offers you several possible ways to handle the log file. The log file contains all information that the system displayed in the 'Details panel' during the execution of the task.
 - **Open** opens the log file in the Pilot's log window [Ctrl + Shift + L].
 - **Open with** allows you to open the log file with your favorite text editor. This will only work if the Automation Engine client has access to the share `AutomationEngineLogging` on the

Automation Engine Server. Once the file is opened in your favorite editor you can save it locally for future reference.

- **Mail** allows you to compose a mail message with the log file, the ticket and the input and output files attached.



Tip: Click 'Open with' to open the log file with your favorite text editor. Keep in mind that this is only possible if the client machine has access to the share on the Automation Engine Server that contains those log files (in the share AutomationEngineLogging).



- **Group workflows [CTRL + B]:** workflows will be grouped in the Task list.
- **Related Workflow Tasks...:** the system displays a dialog in which all related tasks are listed. A specific task can be selected from the list.
- **Show/Hide Details:** shows or hides the Details pane.

Workflows

You can build **Workflows** and launch them on your files using the **Workflow Editor**. Learn more in [Building Workflows using the Workflow Editor](#) on page 987 .

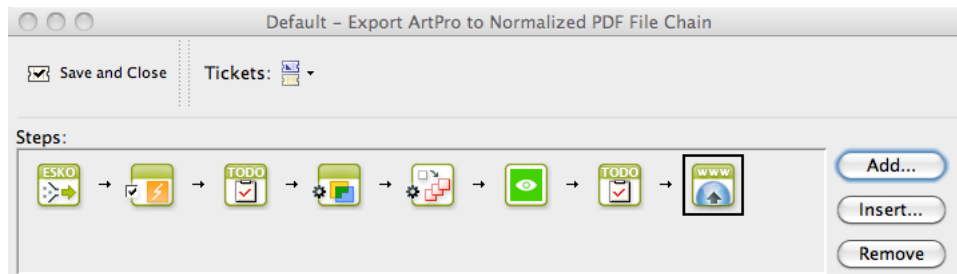
You can also launch them from the **Pilot**, from **Shuttle** (see [Using Shuttle to Upload and Process Files](#) on page 47) or from within most Esko Editors. They can also be launched automatically, triggered by external signals (files arriving, commands from external systems, etc.).

Learn more in the dedicated chapter [Workflows](#) on page 986.

Task Chains (old workflows)

Task Chains are old BackStage workflows. They are still supported in Automation Engine but will need to be migrated.

A Task Chain is a succession of tasks, linked together in a Task Chain Ticket.



You can build new workflows from these old Task Chains, while keeping all your tickets settings (see [Migrating Old Task Chains](#) on page 1078).

Tickets

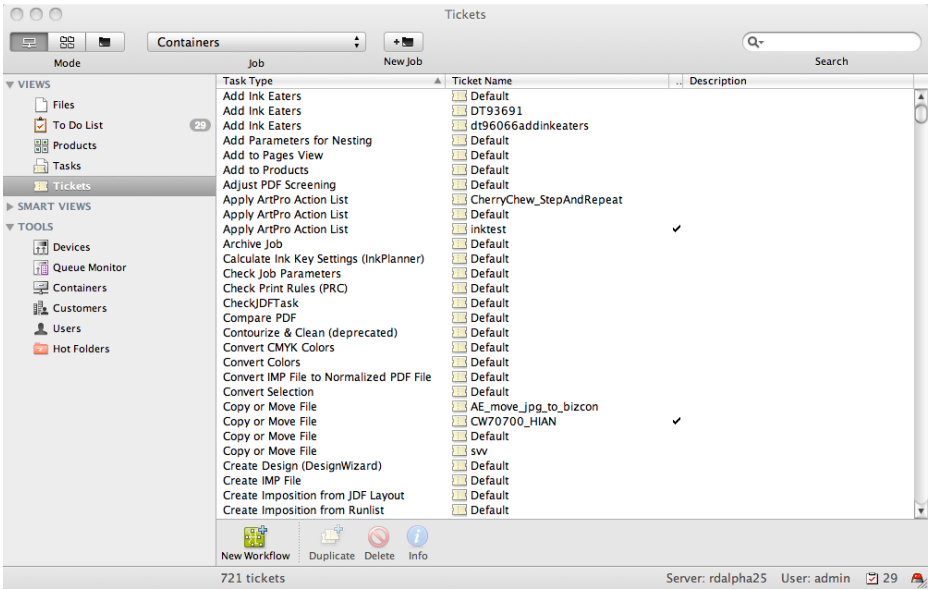
What is a Ticket?

The settings used when executing a task or workflow on a file can be saved as a ticket.

Default and Custom Tickets

The Pilot comes with a default ticket for each task type, but you can also create custom tickets. You cannot overwrite or delete default tickets.

You can see all the available tickets (default and custom) in the Pilot's **Tickets** view.



Global and Job Tickets

Global tickets (🌐) can be used on any file you can access through the Pilot. Job tickets (🏢) are only available when you work in the Job that you saved them for.

Public Tickets

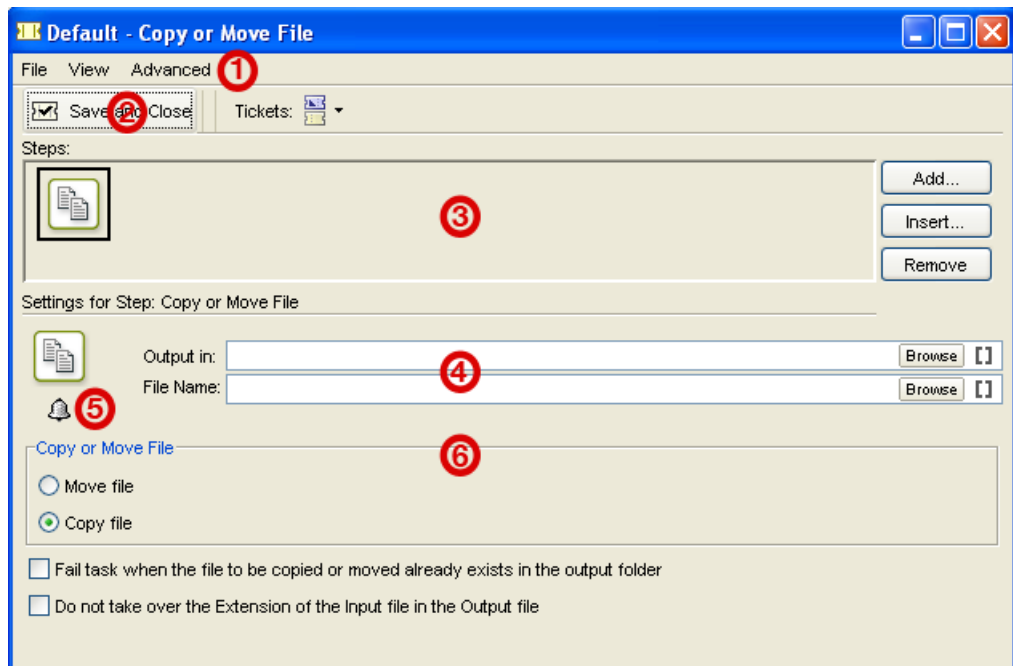
You can make tickets public if you want to submit files to them from Shuttle. There are several ways to make a ticket public (see [Making Your Ticket Public](#) on page 49 in the User Guide).

Public tickets have a mark in the **Public** column.

Task Type	Ticket Name	Public	Description
Zip	Default		
Workflow	WC_Approval		
Workflow	WaitForRunListFiles		
Workflow	TrapWF		
Workflow	Trap_nestedWF	✓	
Workflow	SimpleTrap		
Workflow	Send2Ody		

Note: Depending on your [access rights](#), you may only see public tickets, and public parameters inside the tickets.

Inside a Ticket



1. The menus:

- Use the **File** menu to create a **New** ticket, **Save** the ticket, save it under another name (**Save As**), or **Close** it.
- Use the **View** menu to **Show** different kind of tickets (when creating a workflow): **All**, **Public**, **Global** or **Job Tickets**.
- Use the **Advanced** menu to **Manage Public Parameters**.

2. Use the **Save and Close** button when you are done editing the current ticket.

Note: If you were editing a default ticket, the system will ask you to save the ticket under another name.

3. **Steps:** The above screenshot is outdated. Learn more about workflows in [Workflows](#) on page 986.

4. Most of the tickets offer the possibility to define an **Output** path and a **File Name**.

Use the **Browse** button to browse to the output path.

Use the SmartNames icon  to add SmartNames to the output path and / or file name.

For more information, see this separate chapter in the *Complete Manual*: [SmartNames](#) on page 916.

5. You can add a **Notification** to the task by clicking .

For more information, see [Notifications](#) on page 175.

6. The ticket's **settings** differ for all tasks.

For detailed documentation of all tasks and tickets, please see [The Tasks and Workflow Controls](#) on page 239.

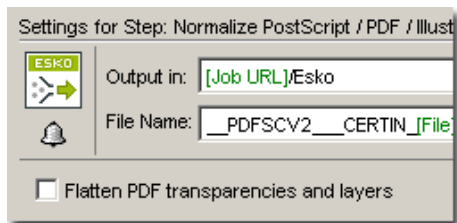
Notifications

Each task ticket can trigger a **Notification**. The system will so notify a user by sending an E-mail when a task finishes, or notify an external system by feeding it an XML or a TXT file with the same kind of content.

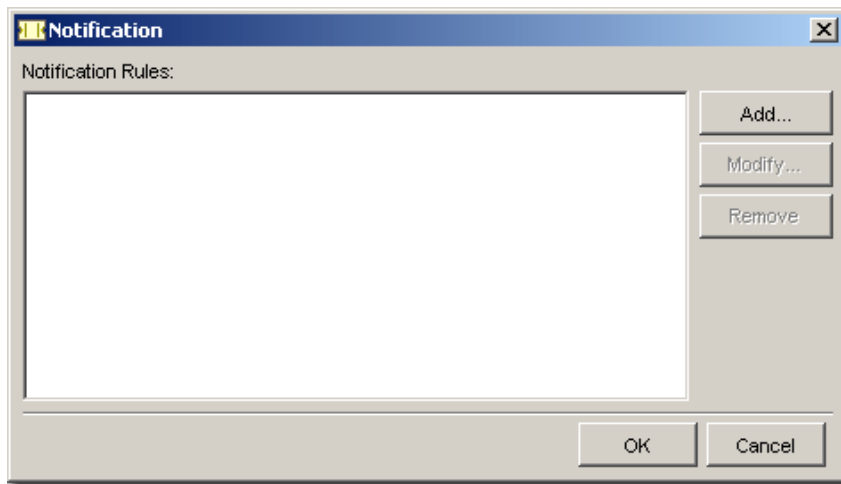
Note: In workflows, when your goal is to send an E-mail, consider using the separate task [Send E-mail](#) on page 471 instead.

To add a notification:

1. Click the notification button  in a ticket.

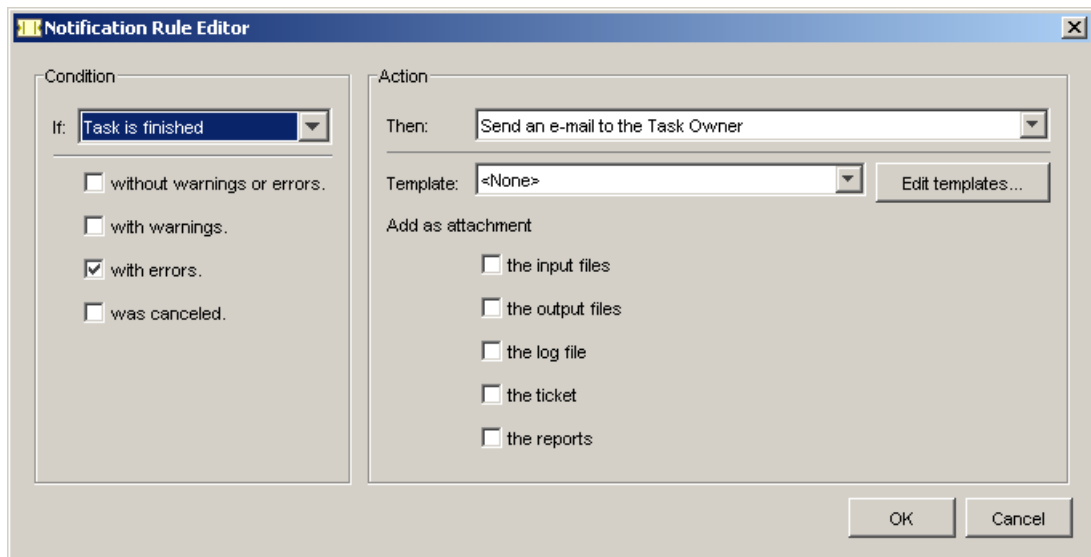


The 'Notification' dialog pops up.



2. Click **Add**.

The **Notification Rule Editor** pops up.



3. Define the Notification.

Define the condition for the notification **and** the action.

A notification may consist of an E-mail sent to the task owner (or to any other valid address), it may also be an entry in a XML/text file, or an XML/text file in a certain folder.

Note: When selecting **Send an e-mail**, you can **add more than 1 E-mail address** in the 'To' field (including SmartNames) by using ' ; ' as a separator.

You can add the input files, the output files, the task's log file and or the ticket to the notification.

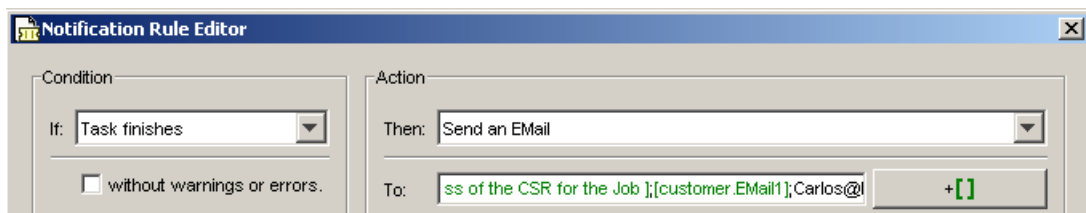
The notification can be based on a template. Hit the **Edit templates ...** button to create and modify templates. Learn more in [Edit Notification Templates](#).

Note: The Template list and **Edit templates** button will only be visible if templates have been created before.

Note that **SmartNames** can be used in the **To** field, for example to refer to E-mail addresses of the Job's CSR or up to 3 customer contacts.

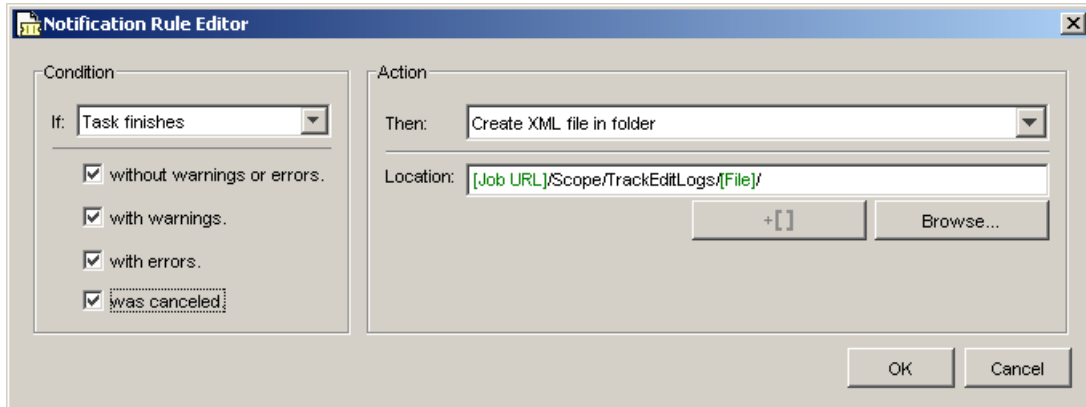
4. Click OK. The notification rule is now added to the **Notification** dialog.

E-mail notifications of tasks that end in a warning or error will contain the **basic problem description** (the messages that appear in the task's details pane). This avoids having to analyze more complex log files.



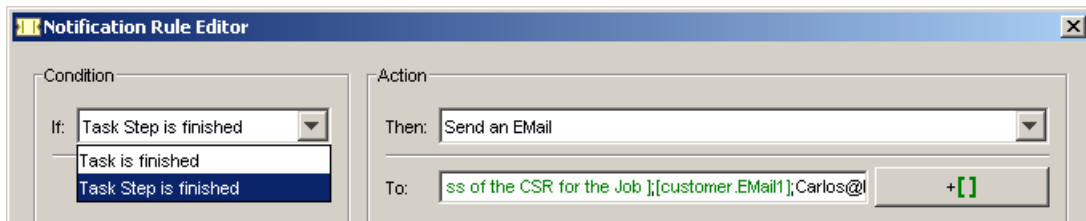
Customizing the Contents

It is possible to save XML notifications on Smart enabled paths. For example saving XML notifications inside the job folder or on a dedicated central folder:



Early Notification




The condition **Task Step is finished** makes it possible to receive a notification message at the end of the step on which it was set, before the rest of the workflow has finished.




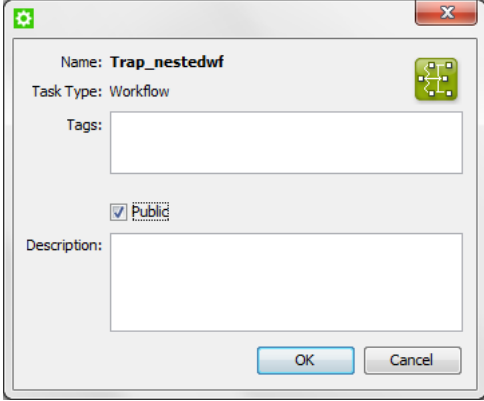
Tip: Use the [Configure tool](#) to configure general notification rules for tasks and devices, and to define the places where the XML or text files will be saved. This will save you from having to do this inside the task ticket(s).

Working with Tickets

From the **Tickets** view, you can:

- Create a new task ticket: double-click an existing task ticket (for example a default one), fill the settings in as desired, and save it under a new name.
- Create a new workflow ticket: click  and build your workflow (Learn more in [Workflows](#) on page 986).
- Copy a ticket: select the ticket to copy, click  and enter a name for the new ticket.
- Delete a ticket: select the ticket to delete and click .
- Make a ticket public: right-click it and select **Public Ticket**.

- Export a ticket: right-click the desired ticket and choose **Export Ticket**. For more info, see [Exporting Tickets](#) on page 178.
- Import a ticket from an other Automation Engine. Go to **Tickets > Import Tickets** . For more info, see [Importing Tickets](#) on page 179.
- Enter information about a ticket: select the ticket, click  and enter a **Description**.



The **Description** you enter here will be visible in the **Description** column of the **Tickets** view.

You can also make the ticket **Public** from here.

- Filter tickets: you can either filter by file type (to only show tickets valid for that file type), or by entering text in the search field.



Note: Depending on your [access rights](#), you may not be able to create, edit or delete tickets. Setting priority may also be limited.

Exporting Tickets

Follow these steps to exchange (workflow) tickets with an other Automation Engine:

1. In the Pilot, go to the **Tickets** view.
2. Select one or more tickets.
3. In the menu, go to **Tickets > Export Ticket** .
Alternatively, you can right-click the (workflow) ticket and choose **Export Ticket**.
4. Choose the location and file name to save the exported data (as a ZIP file).
5. Press **Export** to start the Export action.

The resulting zip file contains the selected ticket(s) and the following dependent resources, if used in the ticket(s):

- SmartNames
- Referred Tickets
- ArtPro Action lists

- PitStop Profiles and Action Lists
- SmartMark sets (as .paf files)
- ppd files
- ReportMaker templates (as .paf files)
- Rule sets for the **Check Print Rule** task

- Tiling templates for the **Create Tiles** task

- Step&Repeat Templates for the **Step & Repeat Template based** task

- CAD file for the **Prepare Station** task

- Template for the **Create Imposition from RunList** task

- Compare with file for the **Compare PDF** task

- Script for the **Run Script** task

- .xml file for the **Create Design (DesignWizard)** task

Exporting tickets has the following limitations:

- Certain resource information is not included, such as the DGC files and CMS related files like Color Strategies.
- Workflows that include (blue) Job Tickets by reference, can not be exported. An error message will appear.
- Plate templates for the **Import JDF Stripping** task are not included.

- Template for the **Create IMP File** task is not included.

- In the **Expand Variable Data to PPML File** task, .xml or .csv database files per grid are not included.

Importing Tickets

To import Tickets that were made on an other Automation Engine, proceed as follows:

1. In the Pilot, go to the **Tickets** view.
2. In the menu, go to **Tickets > Import Tickets** .
3. Select the desired ZIP file that contains the Ticket(s) and their dependant files and click **Import**.
If certain resources would be overwritten by importing the chosen ZIP file, a dialog will ask you whether those resources should overwrite the existing resources. Click **OK** to accept the new files.

After the import is completed, the newly imported tickets will be selected in the Tickets view (when no filters are set up).

Note: Tickets are imported as Global Tickets, while retaining their original name.

Favorite Tickets

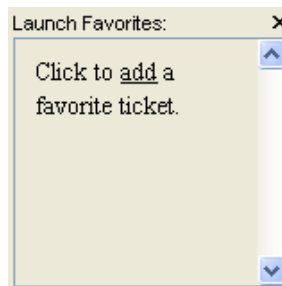
You can add tickets of single tasks or workflows to your favorites, to be able to launch them quicker in the future.

Favorite tickets are linked to:

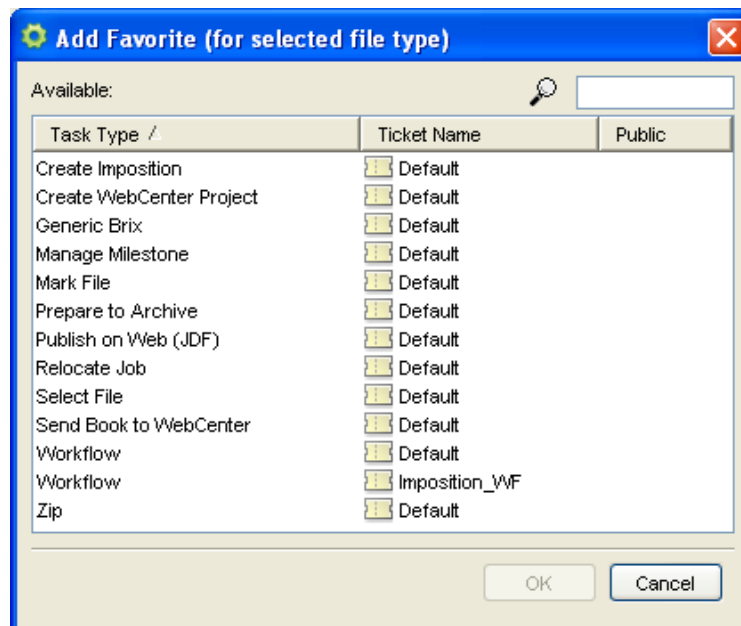
- the Pilot user account (so every user can have his/her own favorites),
- the file type selected in the Pilot (so the favorite tickets you can add or see always match the file type you selected).

Adding a Ticket to your Favorites from the Files View

1. In the **Containers** or **Last used job** mode, select a file (remember that you make favorites per file type)
2. Right-click in the **Launch Favorites** pane and choose **Add Favorite...** .



3. In the **Add Favorite** dialog that opens, select the ticket you want to add as a favorite and click **OK**.



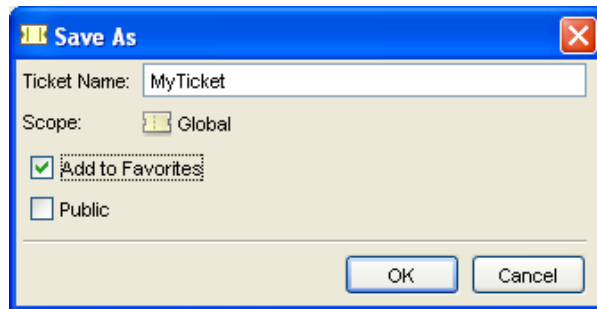
Note: This dialog contains all (single and workflow) tickets suitable for the file type selected in the Pilot.

Adding a Ticket to your Favorites from the Tickets View

You can either:

- Right-click your chosen ticket and select **Add to Favorites**.

- When creating a ticket, select the **Add to Favorites** option in the **Save As** dialog.



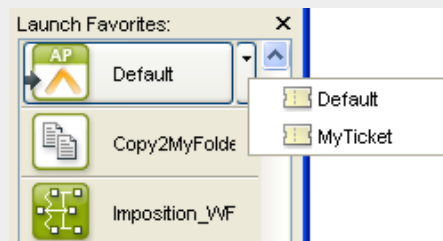
This adds the ticket as a favorite for all file types that can be processed with this ticket.

For example a ticket for the **Copy or Move File** task will be added as a favorite for all file types.

Using a Favorite Ticket

1. Select the file to process in the **Containers** or **Last used job** mode.
The **Launch Favorites** pane shows all the favorite tickets for this file type.
2. Click the favorite ticket you want to use.

Note: If you want to use another ticket of the same task type (for example another **Export to ArtPro File** ticket), click the arrow beside the favorite ticket and choose your desired ticket.



3. The ticket opens. If necessary, review or edit the ticket's parameters.
4. Click **Launch**.

Tip: To launch the ticket's on the file without opening the ticket or workflow, press **Alt** when clicking the ticket.

2.3 Different Ways of Working with Automation Engine

Note: The concepts of **Pages**, **Plates** and **Proofs** only serve in page workflows (commercial printing).

2.3.1 Working with Jobs

The Jobs concept and how to work with it is explained in a dedicated chapter [Jobs](#) on page 828.

2.3.2 Working with Products

The **Products** concept and how to work with it is explained in a dedicated chapter [Products](#) on page 876.

2.3.3 Working with Pages

Note: The concepts of **Pages**, **Plates** and **Proofs** only serve in page workflows (commercial printing).

About Pages

The Pages View is extremely useful when working with impositions. The Pages View allows you to:

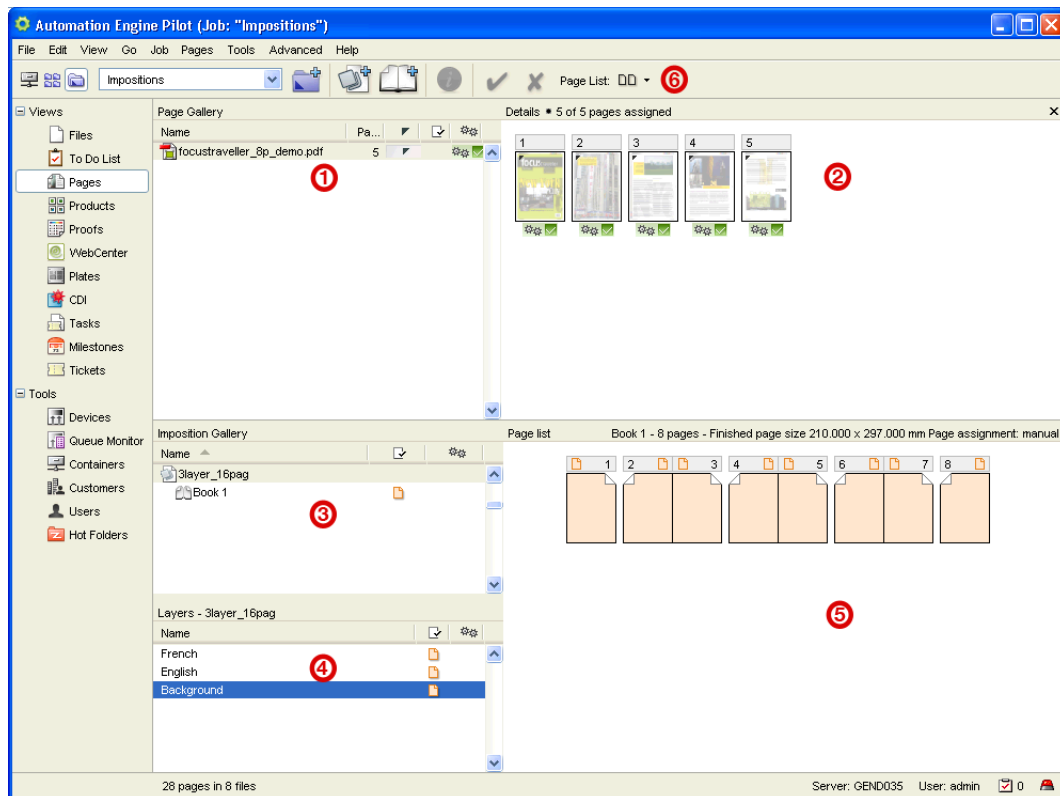
1. View, check and (re-)assign pages to imposition or books.
 - If the Imposition file contains all pages, the Pages View can be used to simply **check** the presence and correct order of the pages.
 - If there are pages missing, you can **assign** them by either selecting the (multi-page) file in the Page Gallery or individual pages in the Details pane, and drag them to the correct location in the Page List.
2. Generate proofs and follow up approval states.

Both PDF and IMP files can be exported, exposed, proofed, etc. from the Pages View. The same tasks as in the Files View can be executed on both file types.
3. Upload and normalize PDF files.
4. Create impositions after templates.
5. Create books for later use in a new imposition.

The Pages View

This view is only available in **Job** mode. It is only used in page workflows.

The Pages View is divided into several panes. The top part of the view (the **Page Gallery** and the **Details** pane) displays both assigned and unassigned pages, the bottom part displays the impositions, layers and assigned pages.

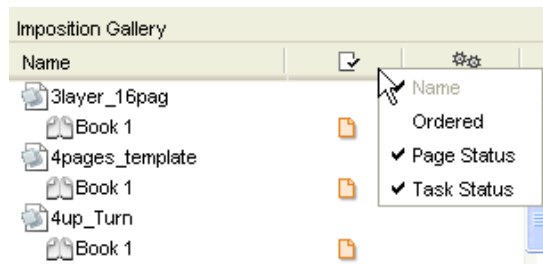


1. The **Page Gallery** pane lists all incoming files, indicates the number of pages and the assignment status.
2. The **Details** pane displays thumbnails of the individual pages of the selected file.
3. The **Imposition Gallery** pane shows the imposition structure with its different books. It also allows you to create books that later will be used in an imposition.
4. The **Layers** pane shows the different layers of the selected imposition. You can rename layers, approve them, and set them empty or blank.

Note: The **Layers** pane is only visible when the selected imposition or book contains more than one layer.

5. The **Page List** shows you which pages are assigned and which are not. For each page status information can be asked.
6. Click the **Page List** button to switch between thumbnail view or list view.

You can view both the task status and the page status information. Simply right-click the columns **Page Gallery** and / or **Imposition Gallery** pane(s) and select both **Task Status** and **Page Status**.



See [Working with Pages](#) on page 182 for more information.

Adding Imposition Files to the Pages View

When you go to the **Pages** View for the first time, you will see that all panes are empty.

To send an **imposition** file from the **Files** View to the **Pages** View:

- Select the **IMP** file and launch the **Add to Imposition Gallery** task on it.
The imposition file will appear in the **Pages** View.
If the imposition contains pages, the corresponding files will appear in the **Page Gallery**.


Tip:

When an IMP file is already present in the **Imposition Gallery** of the **Pages** View, launching the **Add to Imposition Gallery** task resynchronizes the IMP file.

This can be useful when problems with non-matching SmartIDs occur.

Creating an Imposition


To create a new imposition:

1. Right click the **Imposition Gallery** and select **New Imposition**, or click the **New Imposition**  button.
This opens the **New Imposition** dialog.
2. Enter the **Imposition Name**.
3. Select the appropriate **Imposition Template** from the list, or browse to the correct location.
4. By default, **Use Layer Name(s), Book Name(s) and Finished Page Sizes from Template** is selected, which makes the **Width** and **Height** parameters unavailable.
Deselect this option if you want to overrule the template sizes.
Additional information on the number of books and pages of the selected template is displayed.
5. If you want to automate the **Page Assignment**, insert the regular expression to use. You can include SmartNames.
6. By giving a particular **Page Offset**, the pages will be assigned to the correct page slots.
7. Select the correct **Use case** and **Variant separation** if necessary.
8. Click **Create**.

The imposition will appear in the **Imposition Gallery**.

Creating a Book

To create a new book:

1. Right click the **Imposition Gallery** and select **New Book**, or click the **New Book**  button.
This opens the **New Book** dialog.
2. Enter the **Book Name**.
3. Indicate the **Number of Pages** the new book should contain.
4. Set the correct **Finished Page Size**.
5. If you want to automate the **Page Assignment**, insert the regular expression to use. You can include SmartNames.
6. By giving a particular **Page Offset**, the pages will be assigned to the correct page slots.
7. Indicate the correct **Folio** numbering, if necessary.
8. Select the correct **Use case** and **Variant separation** if necessary.
9. Click **Create**.

The book will appear in the **Imposition Gallery**.

The properties of the book can be consulted or changed at any time by right-clicking the book and selecting **Book Setup...**

Tip: Use the [Create Imposition](#) task to turn your book into an imposition.

Adding (PDF) Content Files to the Pages View

When you go to the **Pages** View for the first time, you will see that all panes are empty.

To add a **PDF** file to the Pages View:

- Select a PDF in the **Files** View and choose **Add to Page Gallery** from the pop-up menu. The PDF will appear in the **Page Gallery**, unnormalized.

Note: The **Add to Page Gallery** option is only available if the selected file is not yet present in the **Page Gallery** of the **Pages** View.

- Right-click the **Page Gallery** and select **Get Data and Normalize** to browse to the correct PDF file.

The PDF file will be copied to the root of the Job. After normalization your pages will automatically be added to the **Page Gallery**.

Note: You can upload files that are already present in the job folder. In this case, the files will be left in their current location and will not be copied to the root folder of the Job.

Note:

You will only be able to create a limited number of workflows that work in combination with the **Add to Page Gallery**:

When a PDF file is selected you can use:

- **Copy or Move File > Normalize PostScript / PDF / Illustrator 8.0 File > Add to Page Gallery**
- **Normalize PostScript / PDF / Illustrator 8.0 File > Add to Page Gallery**
- **Add to Page Gallery**

When a Normalized PDF File is selected you can use:

- **Copy or Move File > Add to Page Gallery**
- **Add to Page Gallery**

Page Assignment

Pages can be assigned to the Page List in different ways:

- **Drag** a (multi-page) file from the Page Gallery and **drop** it on the correct page in the Page List. Reader's spreads are automatically recognized and assigned correctly.
- Select one or multiple pages in the Details Gallery and **drop** them on the correct page(s) in the Page List.
- Selecting **Replace Pages** in the context menu of the **Page List** allows you to upload and normalize a PDF file which will be assigned automatically when the Normalize task finishes.
- If you don't want to replace the pages that have already been assigned, you can use **Insert Pages**. In this case, the new pages will be assigned, moving the 'old' pages to the next pages slots.
- Pages can also be assigned automatically, via a **naming convention**.

Note: Within one Page List, pages can also be dragged to another location.

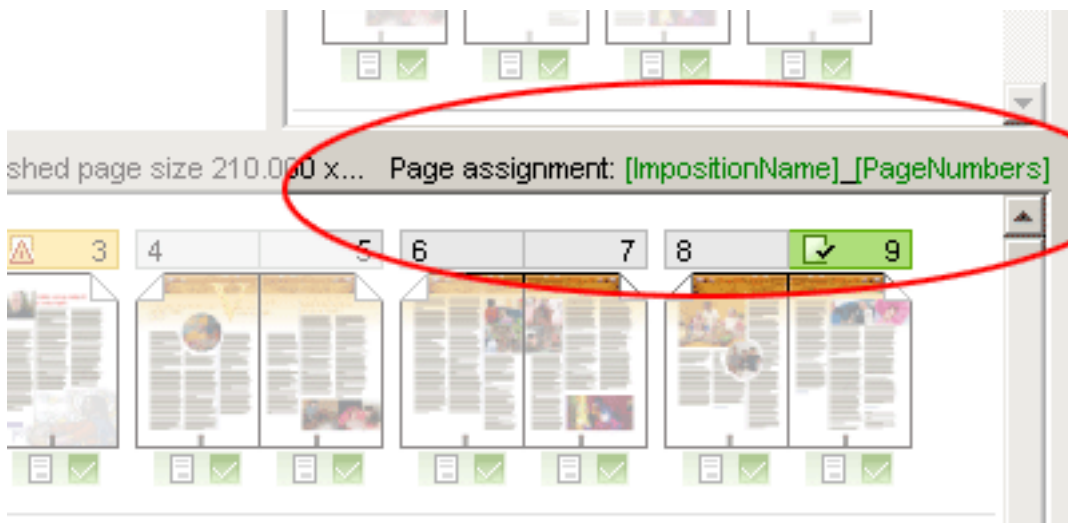
Page Assignment via Naming Convention

In case the page number in the page list is encoded in the name of the incoming PDF files, Automation Engine can automate the assigning of these pages by using a **regular expression**. This regular expression can be composed from ordinary characters, SmartNames, wildcard characters and placeholder(s) for the page index.

In case a regular expression for a certain book is defined, each incoming file that is added to the job containing this book is checked against this regular expression. If the file name conforms to this expression, it is assigned to the corresponding page slot(s).

The regular expression can be specified on *Job*, *imposition* and *book* level.

If automatic page assignment is enabled, it is indicated in the Page List of the Pages View.



The specified **regular expression** is automatically matched to an incoming file when its pages are created.

- The pages are assigned on the condition that the regular expression fully matches the *file name. Moreover,
 - when the regular expression develops into just one page, all pages of the file will be assigned from this desired page onwards.
 - if there are several situations in which the regular expression matches the file name, the first successful one (which leads to a page assignment) will be taken.
 - if the imposition has more than one book or layer and the regular expression does not specify any, the first book or layer will automatically be selected.

Note: Pages can only be assigned when the desired page slots are empty, not when they are blank. Previously assigned pages will not be unassigned / reassigned during page assignment via naming conventions.

You can also specify a **Page Offset**. By default, the page offset is 0, which means that the first page will be assigned to the first empty page slot.

These are the available wildcard characters:

- ? stands for any character
- * stands for any sequence of characters (even empty)


These are the available SmartNames that can be used to create the rule:

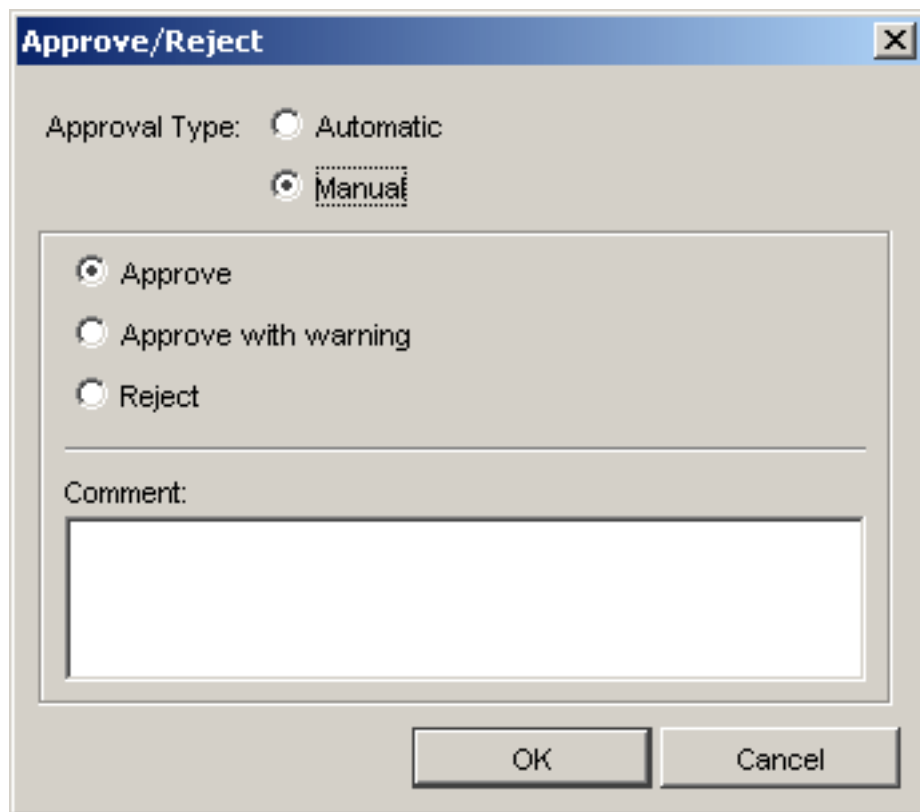
- [BookIndex]: index of the book inside imposition (going from 1)
- [BookName]: name of the book
- [Folios]: folio number
- [ImpositionName]: name of the imposition
- [JobName]: name of the job
- [LayerIndex]: index of the layer inside book (going from 1)
- [LayerName]: name of the layer
- [PageNames] .. A range of page names (folios, for example A,C-E,G)

- The same rules as for [PageNumbers] are applicable, but now with folios.
- No prefix like in page numbers is assumed (for example, p12 means page with folio "p12", 01 means a page with folio "01").
- [PageNumbers]: a range of page numbers (numbers are going from 1, for example 1-5,7,9-11)
 - A page number can be just a number or a number preceded by "p". For example P23 stands for the 23rd page of the imposition.
 - Page numbers can be separated by one or more commas (",") or ("-") to indicate a range. For example 10, p23-p37 .
 - Page ranges can be separated by one dash ("-"). For example 10-12 indicates pages 10,11,12.
 - If just one page is specified and the file has more than one page, the pages are assigned from the desired page onwards.
 - If two pages are specified (for example p10_p13) but the file has the number of pages equal to the number of pages in a range between the given page numbers (for example, 4 pages for page numbers p10_p13), the given page numbers are assumed as a page range.
- [PageSlotId]: page slot id (product id)




Approving and Rejecting Pages

You can approve or reject pages or files in different ways:

- select the file or page and click the **approve**  or **reject**  button in the tool bar.
- select the file or page, click right and select **Approve/Reject...**. Then select the appropriate status in the pop-up dialog.



If you select **Automatic** as approval type, the approval state will automatically show the result of the approval task (**Publish on Web**).

- Choose **Manual** if you want to set the approval status yourself.
 - **Approve**  approves the selected object.
 - **Approve with warning**  makes it possible to add a comment in which you can indicate what modifications should be made or what should be kept in mind. The comment can be seen in the tool tip when moving the cursor over the warning icon in the Page List.
 - **Reject**  rejects the selected object.

Note: Pages can be proofed as such (content proofs) or in their imposition context. Content proofs are more flexible, imposition proofs are more reliable.

2.3.4 Working with Proofs

Note: The concepts of **Pages**, **Plates** and **Proofs** only serve in page workflows (commercial printing).

About Proofs

Proofs are preferably made with the ImposeProof! functionality.

The Proofs View

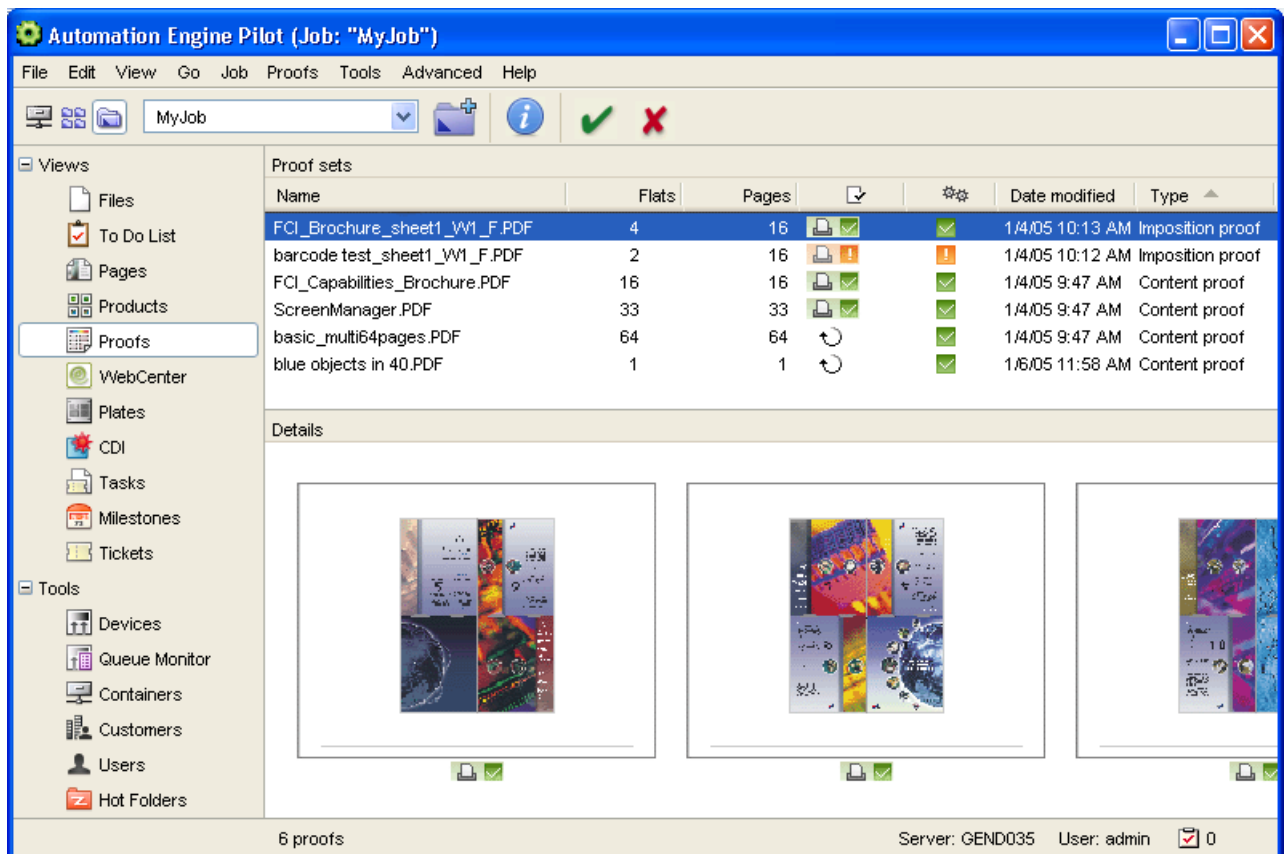
This view is only available in **Job** mode. It is only used in page workflows.

The Proof View consists of two panes that give a different view on the proof:

1. The **Proof Sets** pane, where the proof sets are listed.
2. The **Details** pane is a visual presentation of all proofs of the set.

The Proof Sets pane shows you the names of the proof sets, both hardcopy proofs and softproofs, as well as the number of flats, pages, status, the date on which they were last modified and information about the kind of proof that has been made (**Content proof** if a file is proofed, or **Imposition proof** if an imp file is proofed).

Thumbnails of the proofs can be viewed with the corresponding application, for example Acrobat for PDF proofs.



The Info button will open a properties file of that page (proof).

See [Working with Proofs](#) on page 189 for more information.

Marking Files As Proofs

Proof sets appear in the **Proofs** View after being 'marked as proof'. The **Mark as Proof** ticket has to be connected to an export ticket, for example the **Export to PDF File** ticket.

Perform the following steps to make proofs appear in the **Proofs** View:

1. Select an imposition in the **Pages** (or **Files**) View.
2. Click right and select the **Export Imposition to PDF File Chain** ticket via New Task.
3. Specify the appropriate settings.

Approving Proofs

Setting the approval status of the proof also sets the page state of the corresponding page. This is visible in the Pages View.

Approval statuses can be set manually by an operator or via JDF. Please refer to the presentation on JDF for more information.

Pages can be proofed as such (content proofs) or in their imposition context. Content proofs are more flexible, imposition proofs are more reliable.

The main properties of content proofs:

- Once a page has been proofed it can be moved or copied to any location in the imposition without losing its proofing and approval status.
- The order of the pages in the imposition is not checked.
- The placing (offset and scaling) of the page in its “window” is not checked.

The main properties of imposition proofs:

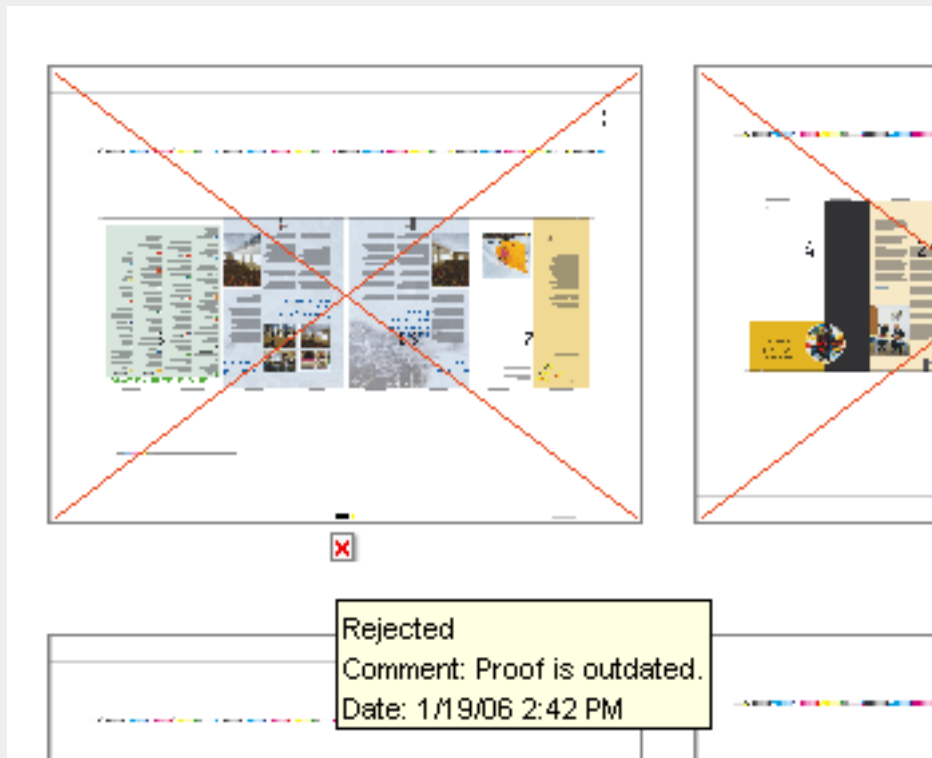
- Imposition proofs can only be made after the pages have been assigned to the imposition.
- Sheet Imposition proofs require large output devices and are harder to check since the pages are not in reading order.
- ImposeProof imposition proofs put the pages in reading order. This makes it easier to check the order and does not require large output devices.
- The safety of imposition proofs can sometimes hinder flexibility since the signature of the imposition is also stored and checked. Many times impositions are changed without a consequence for the pages.
- Reassigning a page degrades the existing imposition proofs to content proofs.
- Changing the impositions degrades the existing imposition proofs to content proofs.
- Approval on an imposition proof on a page slot, implies content approval on the page assigned to that slot and thus on all page slots to which this page is assigned.

Note:

The validity of the proofs is being maintained (based on Smart IDs).

When a page is changed, for example renormalized, the related proofs are rejected and get a note "Proof is outdated".

The link between the page and the proof is broken, which means that the proof is no longer valid for that page.



2.3.5 Working with Plates

Note: The concepts of **Pages**, **Plates** and **Proofs** only serve in page workflows (commercial printing).

About Plates

The Plates View is a view that shows a whole range of details about all plates that are associated with a job. It includes advanced features for managing plates for jobs with multiple versions.

- It visualizes the corresponding flat for a selected plate along with all relevant sheet layout information (plate size, paper size, etc.).
- It makes it possible to expose or proof a set of plates with an ink & sheet selection preset according to the chosen plates.
- It shows progress and status information about each plate.
- It shows both the expected and the realized plates, so differences between planning and production can be detected.

The Plates View

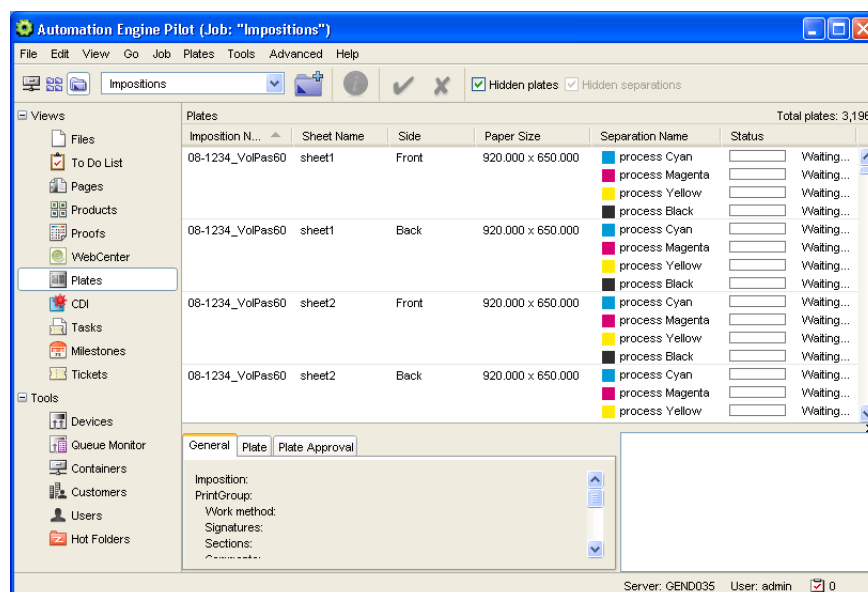
This view is only available in **Job** mode. It is only used in page workflows.

Note: In a page workflow, the job's inks are also used for creating "expected" separations (and corresponding plates) in the **Plates View**. This means that an expected separation will be created for each ink and for each sheet. If the assigned pages contain inks that are not listed in the inks list, "unexpected" separations will be created.

- The Plates View makes it easier to manage the most important deliverables that prepress provides to the press room, plates.
- It gives an overview of the total number of plates that have to be output.
- It allows you to track the progress of a job at a very detailed level.
- It allows you to detect discrepancies between the expected plate output and the supplied data very easily.

For example, when a sheet side is supposed to have only a black plate, but one of the supplied pages for that sheet side contains CMYK, this can be easily seen in the Plates overview.

The same is true for the inverse situation (for example when an expected Pantone plate is not being realized).




See [Working with Plates](#) on page 192 for more information.


Task Status information


The Status column is used for progress tracking. It indicates the status of the plate and offers a wide variety of statuses going from **Waiting for pages** till **Plate approved**.


Every time an action takes place the status column will provide you with the latest information.

Delivering:


 **Waiting for pages...** **Waiting for pages** appears when no pages have been assigned to this sheet side or when the separation has not yet been delivered.

 **Delivered (8 pages)** **Delivered** appears if all pages of a flat are delivered. This status only appears when the output prerequisite in the Imposition tab of the Job Setup dialog is set to **When all pages are approved**.


 **Delivering (5/8 pages)...** **Delivering** gives an indication of how many pages have already been assigned to the sheet, and how many are still missing.

 **Not delivered** **Not delivered** appears if a separation was expected, but is not present in this particular sheet and if there are no empty pages remaining for this sheet. Expected separations are those that are defined in the Inks tab of the Job Setup dialog. By default, this is CMYK. This status only appears when all pages of a flat have been delivered.

Approving:

 **Approving pages...** **Approving Pages** can only appear as a status if the output prerequisite in the Imposition tab of the Job Setup dialog is set to **When all pages are approved**. When the option is set to 'When no page is rejected', the status automatically switches from 'Delivering' to 'Ready for ripping'.

 **Ready for ripping** **Ready for ripping** will appear if all pages of the sheet have been approved.

In case not all pages have been approved, but ripping has been started, the following status icon will be shown:  **Ripped**

Ripping:

 **Ripping...** **Ripping**

 **Ripped** **Ripped** implies that the rip task ended well.


 **Ripped with warning** **Ripped with warning**

 **Ripping failed!** **Ripping failed**

Plate approval:

 **Plate approved** **Plate approved**

 **Plate approved with warning** **Plate approved with warning**

 **Plate rejected** **Plate rejected:** In case new pages have been assigned to a rejected plate and the plate is ripped again, the following status icon will be shown:

 **Plate rejected**

Adding Plates

An imposition cannot be added to the Plates View only. The Plates View is always linked to the Pages View. When an imposition is added to the Imposition Gallery of the Pages View, it is automatically added to the Plates View. Likewise, it is removed from the Plates View when the imposition is removed from the Pages View.

Modifications in the Pages View (like adding and removing pages, approving or rejecting pages...) will usually have an effect on the Plates View. Modifications in the Plates View will not affect the Pages View.

In a JDF workflow the Pages and Plates Views can be populated at the time of job creation if the JDF file contains the relevant information.





Inks and Plates

Adding an imposition to the Imposition Gallery of the Pages View will also create the entries for the Plates View.















How the imposition was made is not important in this respect. It can be made interactively in FastImpose, by executing the Create Imposition task or by creating a New Imposition in the Imposition Gallery of the Pages View.

All the columns of the Plates View will automatically be filled in.

When no pages have been assigned to a sheet side yet the Pages View will show a list of plates that are expected. This list is based on the inks defined in the Job Setup of that job. When no inks are defined in the Job Setup, CMYK is assumed.

kiddybips	sheet1	Front	file:70x102...	1,030.000 x 79...	790.000 x 759....		Process Cyan	<input type="checkbox"/>	Waiting for pages...
							Process Magenta	<input type="checkbox"/>	Waiting for pages...
							Process Yellow	<input type="checkbox"/>	Waiting for pages...
							Process Black	<input type="checkbox"/>	Waiting for pages...
kiddybips	sheet1	Back	file:70x102...	1,030.000 x 79...	790.000 x 759....		Process Cyan	<input type="checkbox"/>	Waiting for pages...
							Process Magenta	<input type="checkbox"/>	Waiting for pages...
							Process Yellow	<input type="checkbox"/>	Waiting for pages...
							Process Black	<input type="checkbox"/>	Waiting for pages...

When all the pages of a sheet side have been delivered, the list of plates for that sheet side will represent the inks of the pages on that sheet side.

kiddybips	sheet1	Front	file:70x102...	1,030.000 x 79...	790.000 x 759....		Process Cyan	<input checked="" type="checkbox"/>	Ready for ripping
							Process Magenta	<input checked="" type="checkbox"/>	Ready for ripping
							Process Black	<input checked="" type="checkbox"/>	Ready for ripping
							Designer Green	<input checked="" type="checkbox"/>	Ready for ripping
							Process Yellow	<input checked="" type="checkbox"/>	Ready for ripping
							Designer kiddy green	<input checked="" type="checkbox"/>	Ready for ripping
kiddybips	sheet1	Back	file:70x102...	1,030.000 x 79...	790.000 x 759....		Process Cyan	<input checked="" type="checkbox"/>	Ready for ripping
							Process Magenta	<input checked="" type="checkbox"/>	Ready for ripping
							Process Yellow	<input checked="" type="checkbox"/>	Ready for ripping
							Process Black	<input checked="" type="checkbox"/>	Ready for ripping
							Designer 25-23-32-10	<input checked="" type="checkbox"/>	Ready for ripping
							Designer Green	<input checked="" type="checkbox"/>	Ready for ripping
							Designer luiers cover	<input checked="" type="checkbox"/>	Ready for ripping
							Designer Xbedsprei	<input checked="" type="checkbox"/>	Ready for ripping

Variant Setup

Decisions on which separations are created, which unplanned plates will be created and how the separations are mapped to plates depend on "use case rules". Use case rules are applied when an imposition is synchronized for the first time and again when the user explicitly applies the selected use case to the job. A use case is selected on job level but can be overwritten on imposition level.

This section contains the following topics:

- [Variant setup on Job level](#)
- [Variant setup on Imposition or Book level](#)
- [Reapplying use cases](#)

You can select one of these "Use Case"s:

USE CASE 1 : No plate reuse between print runs (no background layer)

In this case, there is a separation for every job ink, layer and side of a sheet. Each such separation creates a plate. All layers contain full-colored variants.

The English and French versions share the same imposition characteristics but they are printed separately without the reuse of common plates. The files are supplied separately in CMYK.

Result: the imposition has 2 layers for 2 variants. The number of plates created is 32: 4 flats x 4 inks x 2 layers. This is the default setting.

Delivered separations:

French: CMYK

English: CMYK

Plates:

Press run 1:

CF	MF	YF	KF
----	----	----	----

Press run 2:

CE	ME	YE	KE
----	----	----	----

(8 plates)

USE CASE 2 : Background layer contains variant (n variants = n layers)

In the example below, the variant ink is black. This means that C, M, Y (or all inks other than the variant ink) are taken from the background. The variant ink is taken only from variant layers.

The bottom layer acts as a background layer and contains the common elements of all variants. Plates will be created for the non-variant inks of this layer and can be reused. For every other (foreground) layer, a plate will be created for the variant ink.

Files are supplied as full CMYK.

Result: there are 2 layers for 2 variants. The number of plates created is 20.

Delivered separations:

French: C M Y K

English: K

Plates:Press run 1:

C	M	Y	KF
---	---	---	----

Press run 2:

re-used	KE
---------	----

(5 plates)

Tip:

You can opt for this use case if you don't have a FastVariant licence. In this case there's no need to split the incoming files. The English and French PDF file can be assigned to the corresponding layers of the imposition file, and as background layer one of the language files can be reused.

USE CASE 3 : Background layer is merged with variant layers (n variants = n+1 layers)

The English and French versions are printed with the same CMY plates. When the run length of the first variant is reached, only the black plate is changed. The files are supplied as one file with the common CMYK elements (images, logos...) This file goes into the background layer. Per variant there is one file that contains only the changing black variants (usually just text). These file will be assigned to the language layers that lay on top of the background layer.

Result: there are 3 layers for 2 variants. The number of plates created is 20: (4 inks + 1 ink) x 4 flats

Delivered separations:

French: K

English: K

Background: C M Y K

Plates:Press run 1:

C	M	Y	K + KF
---	---	---	--------

Press run 2:

re-used	K + KE
---------	--------

(5 plates)

USE CASE 4 : Background layer is printed separately (n variants = n+1 layers)

This case will be used when working with variants on a 5-color press. A complete preprint is done for the images and the black of the text is printed separately for both languages.

The bottom layer acts as background layer and contains the common elements of all variants. For all inks of this layer plates will be created, which can be reused. For every other (foreground) layer, a plate will be created for the variant ink.

Files are supplied as full CMYK, plus an additional ink for the text. However, this additional ink needs to be combined with the black ink.

Result: there are 2 layers for 2 variants. The number of plates created is 20 and not 24 as would have been the case when 5 inks were used.

Delivered separations:

French: K

English: K

Background: CMYK

Plates (5 color press):

Press run 1:

C	M	Y	K + KF
---	---	---	--------

Press run 2:

re-used	K + KE
---------	--------

(6 plates)

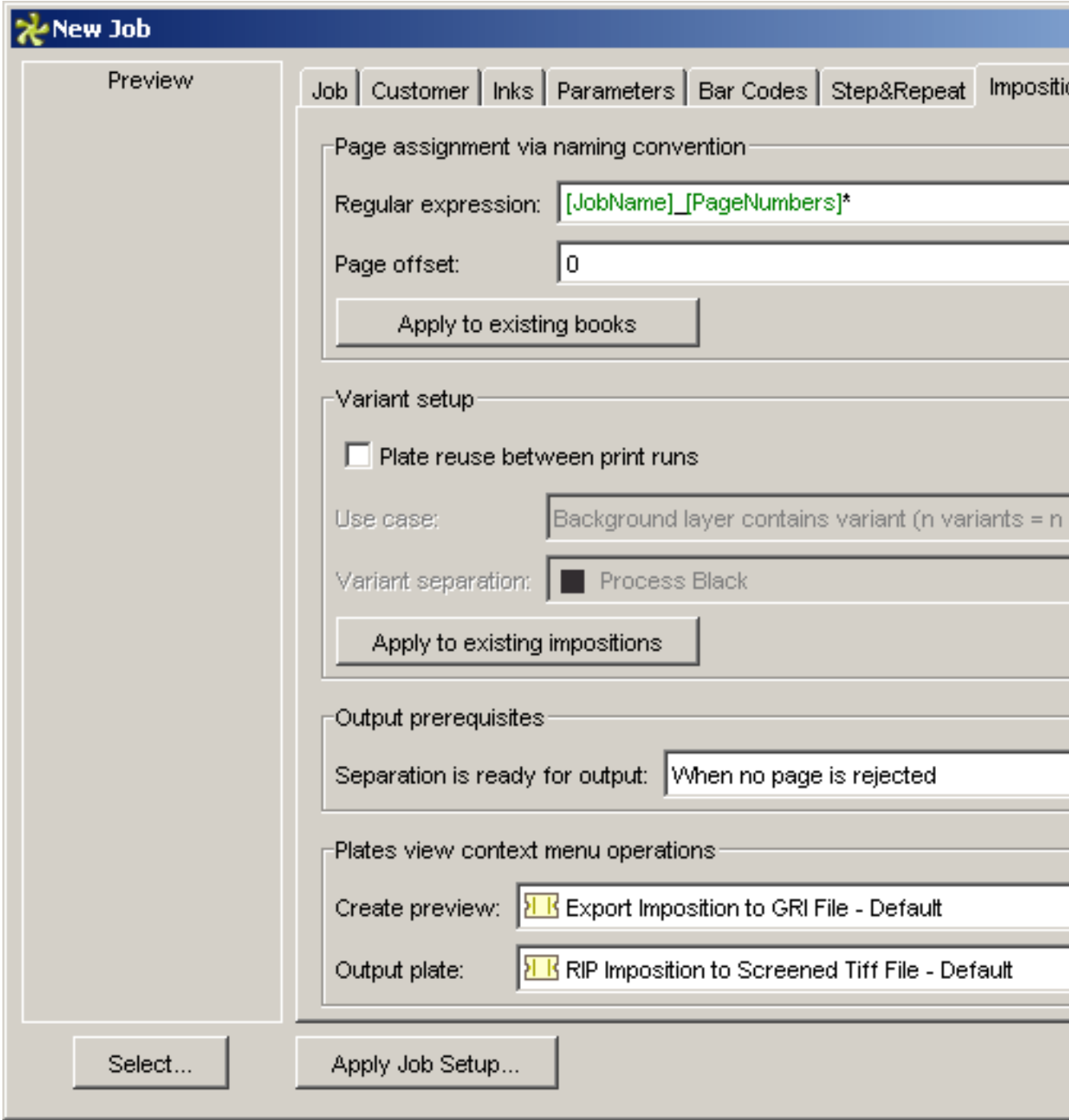
The default selection of use cases is "No plate reuse between print runs", which is defined on Job level (Job Setup # Imposition tab). The same use case is selected when there is no definition on Job level. The list of variant separations is actually the list of job inks, which corresponds with the Ink Settings parameter of a job. In case a job contains no ink settings, Process CMYK inks are present.

Note: When creating multilayer jobs in FastImpose, please make sure that the background layer is always the bottom layer in order to avoid incorrect output when handling variants in Automation Engine.

Variant Setup on Job level

The Variant Setup option on Job level is available in:

New Job



Check the “Apply to Existing Impositions” if you want to apply the use case to all impositions in the job.

Note: The option “Apply to Existing Impositions” is only available if there is an applicable imposition in the job.

Job Setup

Job Setup

Preview

Job | Customer | Inks | Parameters | Barcodes | Step&Repeat | Imposition

Page assignment via naming convention

Regular expression: [PageNumbers]*

Page offset: 0

Apply to existing books

Variant setup

Plate reuse between print runs

Use case: Background layer contains variant (n variants = n)

Variant separation: Process Black

Apply to existing impositions

Output prerequisites

Separation is ready for output: When all pages are approved

Plates view context menu operations

Create preview: Export Imposition to GRI File - Default

Output plate: Export Imposition to GRI File - Default

Select...

Apply Job Setup...

Check the “Apply to Existing Impositions” if you want to apply the use case to all impositions in the job.

Note: The option “Apply to Existing Impositions” is only available if there is an applicable imposition in the job.

Variant Setup on Imposition or Book level

When creating an imposition or book in the Pages View, one of the following dialogs will pop up:

Note: Selecting a use case in the Book or Imposition Setup dialog overrules the selection on Job level (not just for a separation or variant, but for the whole imposition). This way, you can have two impositions following different use cases in one job.

New Imposition

Imposition Name:

Imposition Template:

Use Layer Name(s), Book Name(s) and Finished Page Sizes from Template

Layer:

Book:

Number of pages: 16 pages

Width: mm

Height: mm

Page Assignment:

Page Offset:

Variant setup

Plate reuse between print runs

Use case:

Variant separation:

New Book

Book Name:

Number of Pages:

Finished Page Size

Width: mm

Height: mm

Page Assignment via Naming Convention

Regular Expression:

Page Offset:

Folios

Prefix:

Pages:

Suffix:

Variant setup

Plate reuse between print runs

Use case:

Variant separation:

Reapplying Use Cases

Use cases can be reapplied in two ways. Settings can be changed on book or on imposition level.

Note: Selecting a use case in the Book or Imposition Setup dialog overrules the selection on Job level (not just for a separation or variant, but for the whole imposition). This way, you can have two impositions following different use cases in one job.

1. On Book level

A use case can be reapplied on book level if the page list has not yet been turned into an imposition file. In this case the imposition level is not present in the Pages View. The Variant Setup option can be found in the Book Setup dialog.

New Book

Book Name:

Number of Pages:

Finished Page Size

Width: mm

Height: mm

Page Assignment via Naming Convention

Regular Expression:

Page Offset:

Folios

Prefix:

Pages:

Suffix:

Variant setup

Plate reuse between print runs

Use case:

Variant separation:

- Process Black
- Process Cyan
- Process Magenta
- Process Yellow
- Process Black
- Pantone 286
- Pantone 342
- Pantone 375
- Pantone 382

Note: If the imposition has an IMP file associated with it, the Variant Setup is not present in the Book Setup dialog. As the use case has to be common for all books in the imposition, the use case is to be defined on the imposition level.

2. On Imposition level

If the imposition level is present in the Pages View, the Variant Setup option can be found in the Imposition Setup dialog.

Imposition Setup

Imposition Name:

Imposition Template:

Use Layer Name(s), Book Name(s) and Finished Page Sizes from Template

1. Layer:

2. Layer:

Background:

Book:

Number of pages: 4 pages

Width: mm

Height: mm

Page Assignment:

Page Offset:

Variant setup

Plate reuse between print runs

Use case:

Variant separation:

Manipulating Plates

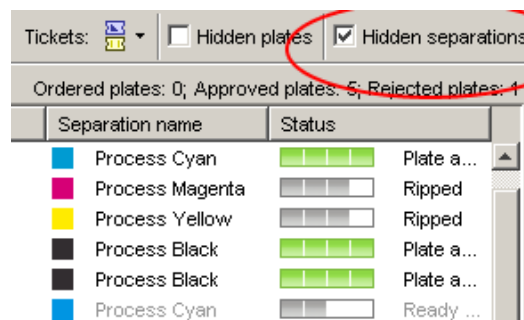
1. Separations need to be exposed together

It is not uncommon that two separations need to be exposed together, because the supplied file is not correct. The file can be corrected interactively, but when it is simply a matter of combining two separations on the same plate, it can also be done here.

Example: a file contains a designer ink “Text” which has to be exposed together with “Black”. This can be done by selecting one of the plates and selecting “Add Separation...” from the pop-up menu.

2. There are superfluous separations

Sometimes a file contains inks for a special purpose that do not need to be exposed onto a plate for example a designer ink called “dieshape”. Such a plate can be hidden by selecting “Hide” from the pop-up menu. In the top right corner of the Plates View there is a toggle with which hidden separations can be shown.



This operation can be reversed. When the “Hidden separations” toggle is on, select the hidden plate (separation?), right click and select “Create Plate...” from the pop-up menu to turn the separation into a plate.

3. Separations are named incorrectly

Some MIS systems do not name spot colors (correctly). In the JDF file, a spot color could be named “Spot1”, but in reality a designer ink is required. The pages, however, do contain the correct designer ink. This leads to a situation in which there is an ordered plate which is not realized (Spot01) and a plate/separation which is there, but not ordered (Designer Xbed).

Plate Making...	Sig#1Sheet#1	Front	1000x100...	1,000.000 x...	640.000 x 450...	Process Cyan	✓	Ready for rip...
						Process Magenta	✓	Ready for rip...
						Process Yellow	✓	Ready for rip...
						Process Black	✓	Ready for rip...
						Process Spot01	✗	Not delivered
Opt2292-6_...	Sig#1Sheet#1	Front	1000x100...	1,000.000 x...	640.000 x 450...	Designer Xbed...	✗	Ready for rip...
Plate Making...	Sig#1Sheet#1	Back	1000x100...	1,000.000 x...	640.000 x 450...	Process Cyan	✓	Ready for rip...
						Process Magenta	✓	Ready for rip...
						Process Yellow	✓	Ready for rip...
						Process Black	✓	Ready for rip...
						Process Spot01	✓	Not delivered

2.4 Server Administration Web Tool

A Server Admin Web Tool is installed with every Automation Engine Server from Automation Engine 12 onwards. You can open the Server Admin Web Tool by entering `http://<name of your server>` or `http://<name of your server>:9999` in the address field of your browser.

You can download and launch Client Apps and conduct Server Administration via this tool.

System Requirements

Windows :

- Internet Explorer (8.0 or higher required), Firefox, Chrome or Safari.
- Windows XP or Windows Server 2003 or higher and Java JRE 1.7 or higher

Mac

- Safari, Chrome or Firefox.
- Mac OS X 10.5 or higher and Java JRE 1.6 or higher

2.4.1 Main

The default start page of the Server Admin Web Tool is the **Client Apps > Main** page. You can access it by browsing to `http://<name of your server>` or `http://<name of your server>:9999`.

Main

You can download the main applications (the **Pilot**, the **Shuttle** and the **Diagnostics** tool) from this page.

- **Automation Engine Pilot** : The Pilot is the primary client tool for your prepress workflow automation. Read more in [The Automation Engine Pilot](#) on page 19.
- **Automation Engine Shuttle** : The Automation Engine Shuttle is a client tool for easy submission of files to Automation Engine workflows. Shuttle also allows monitoring of the workflows. Read more in the Automation Engine User Guide.
- **Automation Engine Diagnostics** : You can use this app to monitor the status and activity of the Automation Engine Server. It can be launched and used both on Mac and Windows.

You can click on **Tools** to access the other Client Apps.

Note: The Client Apps are only available on Automation Engine Servers that are configured as Master server.

2.4.2 Downloading A Client App

The Client Apps section is the place to launch or download the Automation Engine Client Apps. These apps work both on Mac and Windows. The Server Admin Web Tool will automatically detect the OS from which you access it and provide the correct versions of the Client Apps that can be launched or downloaded.

To download Click the **Download** button next to the Client App you want to download. The Server Admin Web Tool will detect the Operating System and will download the appropriate installers.

Automation Engine Script Runner

You can automate tasks in applications like Illustrator, Photoshop and InDesign using Script Runner. Read more in the Automation Engine User Guide [Automation Engine Script Runner](#) on page 122 and in [The Script Runner](#) on page 794.

Automation Engine ArtPro Action List Editor

Create and edit ArtPro Action List files to automate tasks in ArtPro. This tool is available for download on both Mac and Windows. Read more in [The ArtPro Action List Editor](#) on page 708

Automation Engine RunList Editor

This tool helps in creating and editing RunLists to collect several files into one job folder, or to make one final PDF out of several smaller PDFs (for example for imposition). This tool is available for download on both Mac and Windows. Read more in [The RunList Editor](#) on page 786

Automation Engine PitStop Profile Editor

This tool helps in creating and managing PitStop PDF **Profiles** and **Action Lists**. This tool is available for download on both Mac and Windows. Read more in [The PitStop Profile Editor](#) on page 782

2.4.3 Server Administration

Automation Engine features a new web tool for server administration. This means that you can access your server administration tool from any workstation via <http://servername> or <http://servername:9999/>.

You can use this web tool to download the Automation Engine Client apps such as

- the Pilot
- the Script runner
- the ArtPro Action List Editor
- the Automation Engine 10 Backup Tool

The Server Admin Web Tool also includes a Server Admin section which can be used to administer the Automation Engine Server and its settings, create and restore backups and manage databases, SQL Servers and the Application Server.

Server administration consists of :

- [Status Overview of the Server](#)
- [Setting up the server](#)
- [Backing up the server](#)
- [Restoring the server](#)
- [Managing databases](#)
- [Managing SQL servers](#)

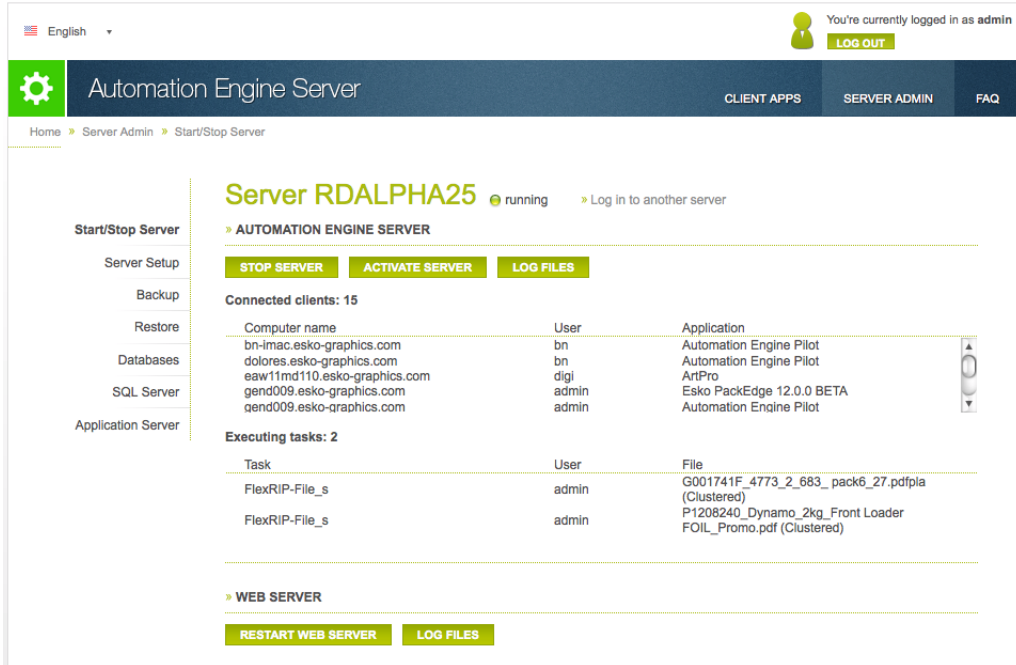
Read detailed information in Automation Engine Reference Guide.

Status Overview of the server

The **Status Overview** shows an overview of the connected clients and the number of running tasks. You also get an overview of the **Server Version**, **Current role** and **Server databases**. In the **Status Overview** you can:

- Start and stop the Automation Engine server.
- Activate the server after install.
- Restart the web server.
- Download the server administration log files for support purposes.
- Start, stop or activate the local application server.

- Download Application Server log files for support purposes.
- Deploy components on the application server.
- Log in to another server.



The screenshot shows the 'Automation Engine Server' interface. The main heading is 'Server RDALPHA25' with a 'running' status and a 'Log in to another server' link. Below this, there are buttons for 'STOP SERVER', 'ACTIVATE SERVER', and 'LOG FILES'. A section titled 'Connected clients: 15' contains a table with columns for Computer name, User, and Application. Below that, 'Executing tasks: 2' is shown with a table of tasks, users, and files. At the bottom, there is a 'WEB SERVER' section with buttons for 'RESTART WEB SERVER' and 'LOG FILES'.

Computer name	User	Application
bn-imag.esko-graphics.com	bn	Automation Engine Pilot
dolores.esko-graphics.com	bn	Automation Engine Pilot
eaw11md110.esko-graphics.com	digl	ArtPro
gend009.esko-graphics.com	admin	Esko PackEdge 12.0.0 BETA
gend009.esko-graphics.com	admin	Automation Engine Pilot

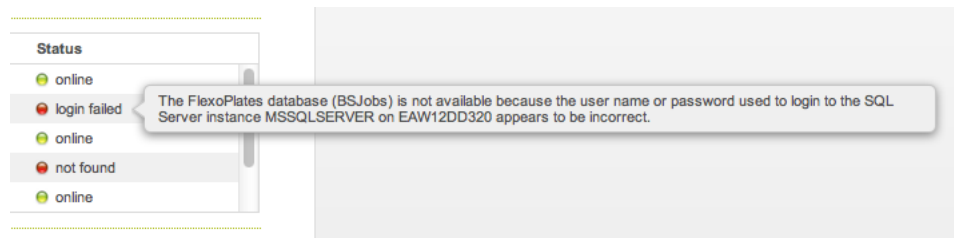
Task	User	File
FlexRIP-File_s	admin	G001741F_4773_2_683_pack6_27.pdfpla (Clustered)
FlexRIP-File_s	admin	P1208240_Dynamo_2kg_Front Loader FOIL_Promo.pdf (Clustered)

The **Status Overview** page contains more information from 12.1.1 onwards. Read the following.

For each **Server database**, the **SQL Server Instance** and the **SQL Server** location is indicated at the **Status Overview** page. The **Status** column indicates the following statuses :

- **Online** : The SQL Server instance where the server database is located is running. This also implies that the user name and password used to login to the SQL Server instance are correct. The server database is online and available for access.
- **Not Accessible**: The SQL Server instance where the server database is located is running. Valid credentials(user name and password) are used. But the server database on the SQL Server instance is not accessible.
- **Not Found** : The SQL Server instance where the server database should be located is running and the credentials used to login to the SQL Server instance are valid. But the server database could not be found on the SQL Server instance.
- **Login Failed**:: The SQL Server instance where the server database should be located is running. The server database is not available because the credentials used to login to the SQL Server instance are invalid.
- **Offline**: The server database is not available because the SQL Server instance where it is located is currently not running.
- **Unknown**: The status of the server database on the SQL Server instance is unknown.

Tip: When you move your cursor over the status of a server database, a pop-over message will indicate the detailed status information.



FAQ on Starting the Server

How to start or stop the server?

Click on either the **Start server** or the **Stop server** button. Starting or stopping the Automation Engine Server can take up to one minute. The Server Admin Web Tool will indicate the status during this process.

You can refresh manually or visit other pages while it is processing.

How to activate the server?

Click on the **Activate server** button. When activating the server, a log window will indicate the server activation progress.

How to stop or activate a server that still has connected clients?

If you try stopping or activating a Master or Assistant server with connected clients and/or executing tasks, you will be asked to close all the Automation Engine applications. If you fail to do so, the open applications will be interrupted and the server will be automatically stopped or activated within five minutes.

Note: Stopping/ Activating process will make it impossible to login via a new client or start a new Task on this Automation Engine Server

How to Cancel the Stop and Activate Processes?

You can cancel the **Stop** or **Activate** process by clicking on **Cancel** . You will be able to login again with a Client and start new Tasks.

You can also **Force stop** or **Force activate** while waiting for the open applications to close. The server will then break the connection with clients and will cancel all the executing Tasks.

How to download the Automation Engine Server log files?

When you click the **Log Files** button, a ZIP file containing the Automation Engine Viewer log files and the Application Server log files will be downloaded. The ZIP file contains a folder named `egappsrvlogs` with Viewer log files (`ImageList*.log`) and a folder `jbosslogs` with Application Server log files.

How to restart the Web Server?

You can restart the Tomcat Web Server using **Restart Web Server** button from the **Web Server** section. The Start/Stop Server page will be automatically refreshed when the Tomcat Web Server is restarted. You can also refresh manually to see if the Tomcat Web Server is restarted.

How to download the Web Server log files?

For Tomcat Web Server log files, click the **Log Files** button in the **Web Server** section. A ZIP file will be downloaded to your computer containing the log files of the Tomcat Web Server and the log files of the Server Admin Web Tool.

Log files for Client Apps are not found in the Server Admin Web Tool, because they are executed on the computer itself.

Gang Run Database

You can see the Gang Run Printing database in the **Status Overview** page when it is a separate database (separated from the Jobs database). The Gang Run database will be listed under **Server databases** section.

Server EAW12DD320 ● running ↻ [» Log in to another server](#)

» **STATUS OVERVIEW**

Server status ● running ↻

STOP SERVER **ACTIVATE SERVER** **LOG FILES**

Connected clients: 0
Executing tasks: 0

Server version 14.0.0 - build 17

Current role Master (0 Assistant servers)

Server databases

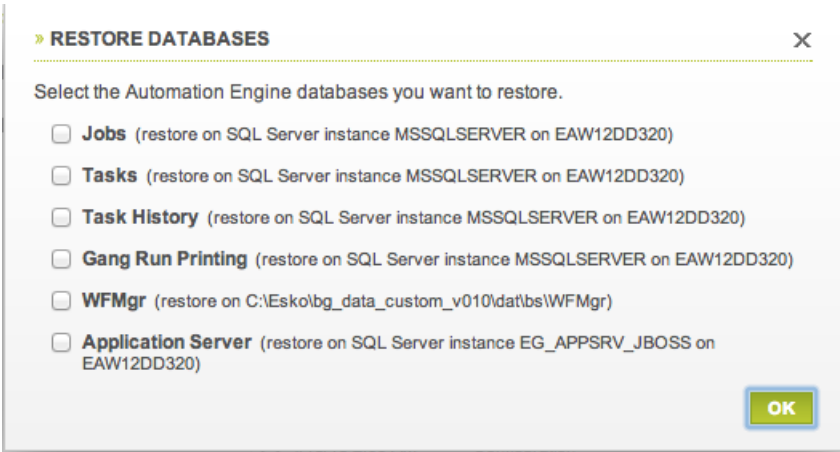
Database name	SQL Server instance	Status
FastLane_JBoss	EG_APPSRV_JBOSS on EAW12DD320	● online
Fastlane	MSSQLSERVER on EAW12DD320	● online
Fastlane_TM	MSSQLSERVER on EAW12DD320	● online
GangRunPrinting	MSSQLSERVER on EAW12DD320	● online

Application Server ● running ↻

STOP **ACTIVATE** **LOG FILES** **DEPLOY**

Web Server **RESTART WEB SERVER** **LOG FILES**

When the Gang Run Printing database is a separate database, you can restore it independently. Use the **Restore** page of the Server Admin Web Tool.

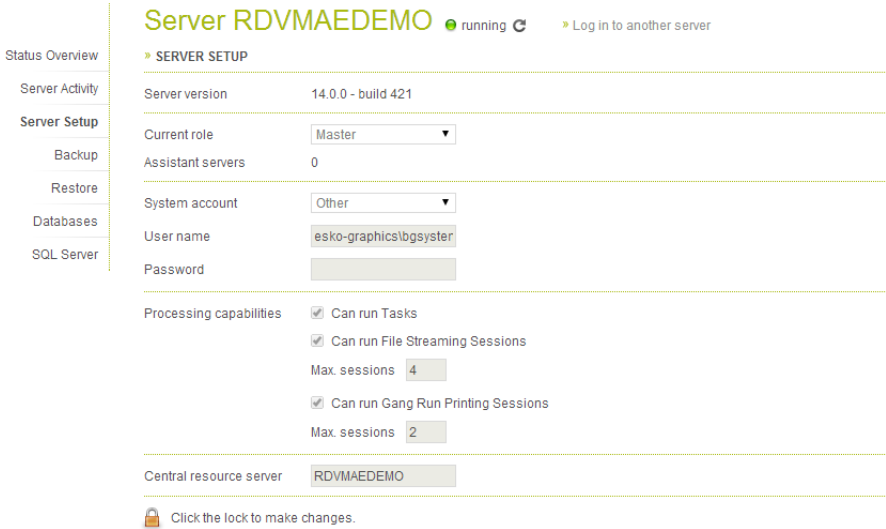


Read more about restoring databases in [Restoring Database](#) on page 228.

Setting up the server

In the Server Setup window you can change the server role: master, assistant or idle. You can also:

- View and remove Assistant Servers.
- Change the system account and/or password.
- Change the processing capabilities.
- Redirect central resources to a central file server.



Server Roles

You can configure an Automation Engine server installation as

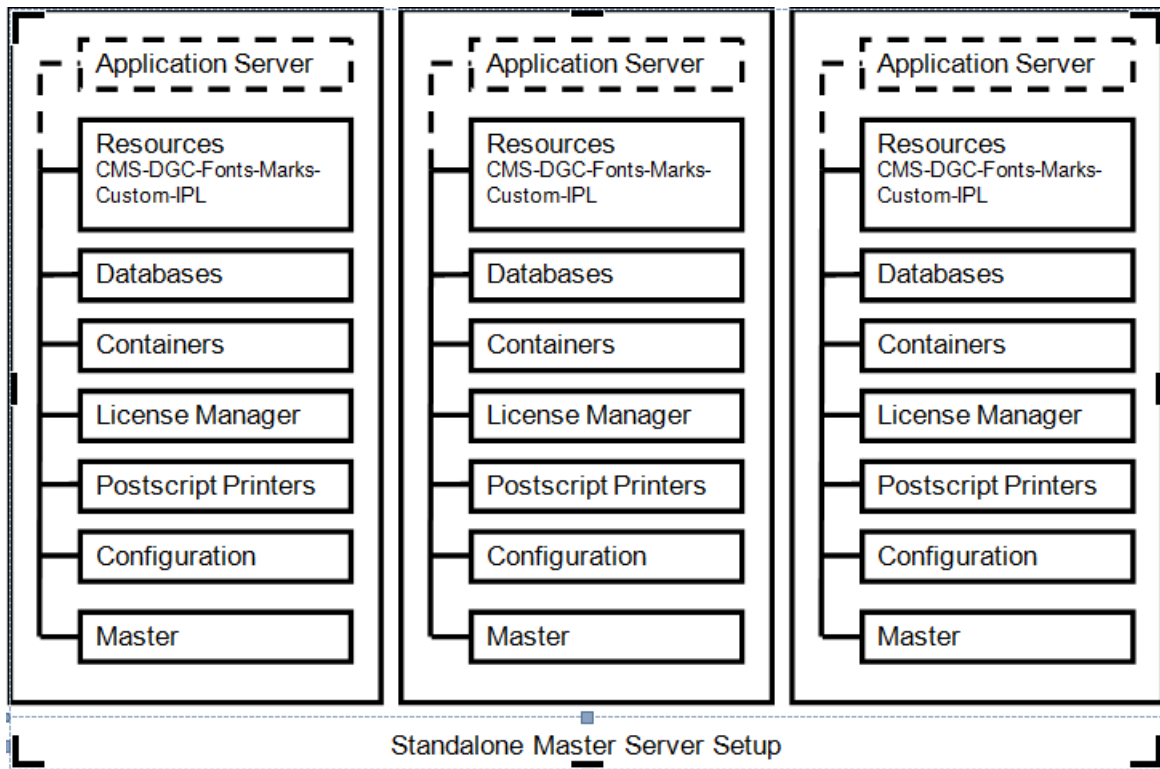
- Automation Engine Master
- Automation Engine Assistant
- Data Server (Idle).

These act as a set of cooperating servers to distribute the workload and provide recovery when a system goes down.

Automation Engine Server Standalone Setup

The simplest configuration is a stand-alone Automation Engine server Setup. You can configure more than one such setups. In this setup,

- each system has its own set of resources (such as CMS data, DGC data, font data, custom data, marks data, ink planner data, task history database). Therefore, they function independently without sharing the workload.
- each system has separate local license management
- you can create containers on shares of other systems.

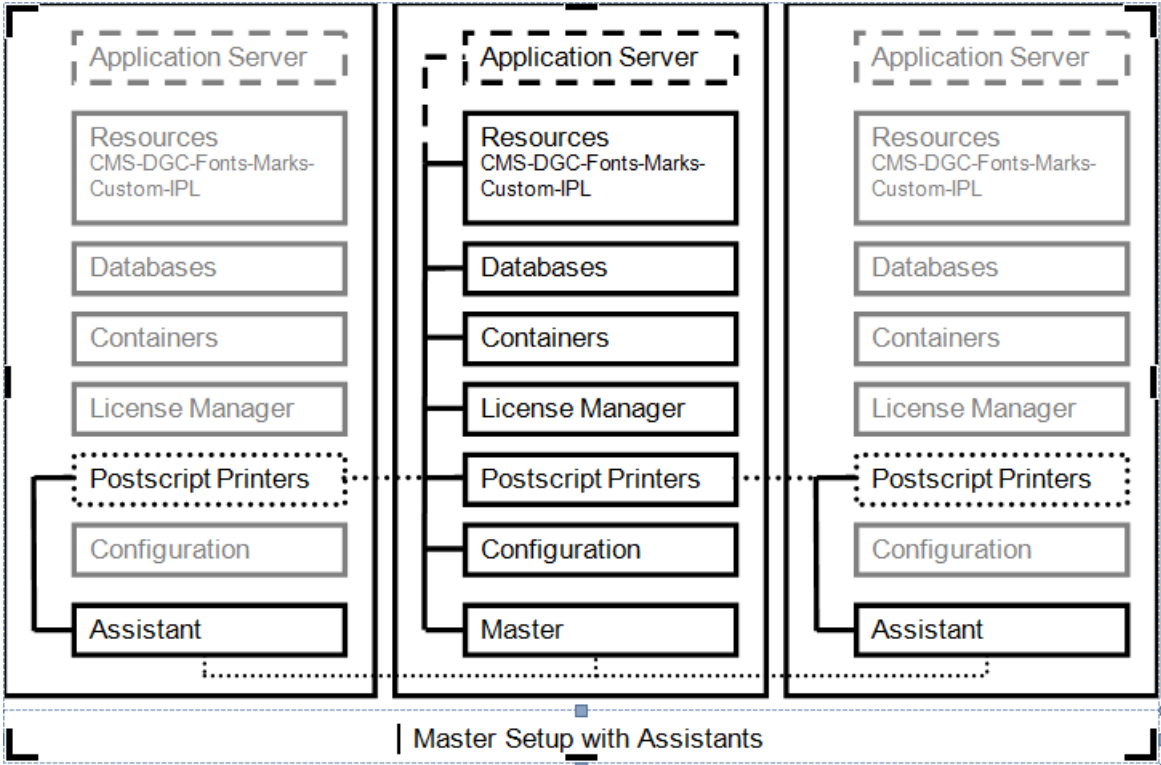


Automation Engine Server setup with Assistant system(s)

This consists of Automation Engine Master server system and one or more Automation Engine Assistant systems. In this setup,

- the Assistants use Master resources for performing tasks.
- Automation Engine Clients connect to the Master computer (but not to the Assistants).
- the workload on the different Automation Engine systems is allocated to the system with the smallest load. If one of the Assistant systems goes down, it is automatically removed from the configuration and the other systems take over the workload.

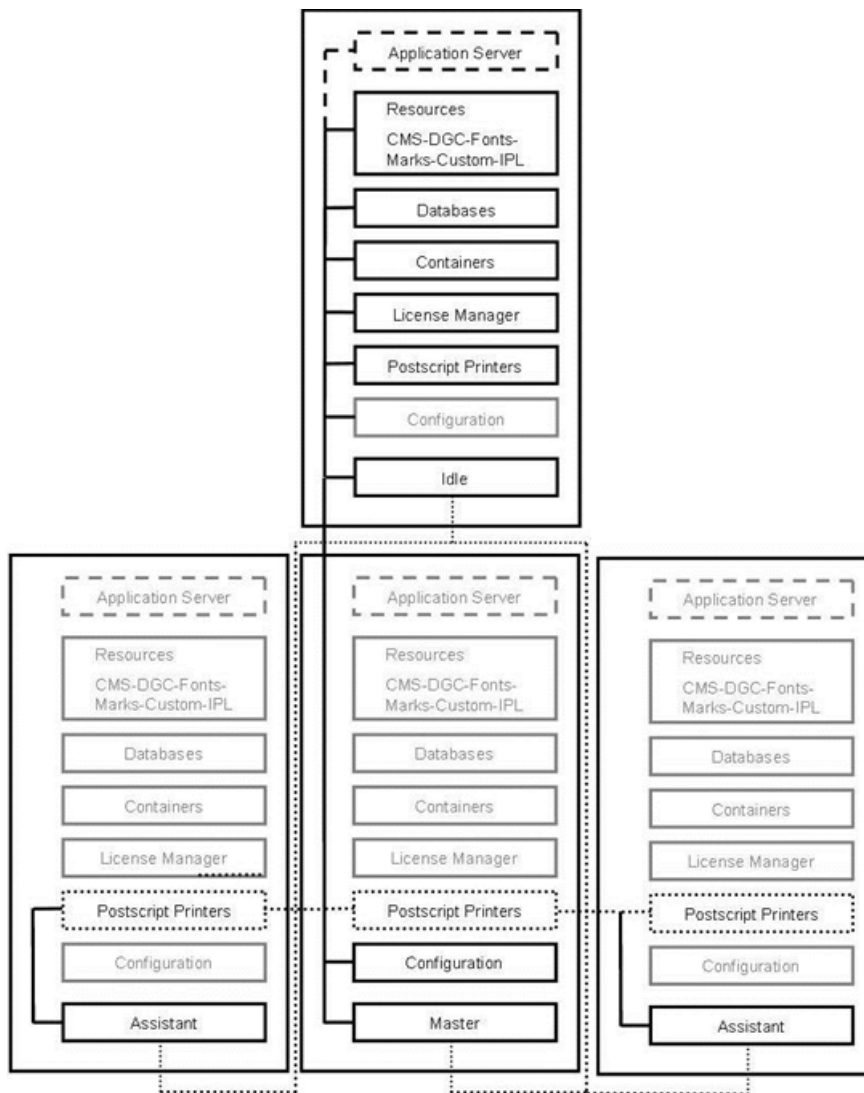
If the Master system goes down, the complete system shuts down.



- Note:**
- The Assistants take a copy of the complete configuration of the Master system. This helps in taking over the Master role.
 - The Assistants should have an Application Server and database engine installed locally for taking up the Master role.

Automation Engine Server setup with Assistant system(s) and Data Server

You can configure an Automation Engine Data Server (Idle) which can be accessed when the Master goes down along with the Assistants. A Data Server system can be set up for all the resources required to facilitate the take over of the Master role by one of the Assistant systems.



Master Setup with Assistants and Data Server

Note: You need to save regular backups of the Application Server database on the Data server system. These can be imported into the Assistant system's database when the Master goes down.

Tip: You can install the Application server software on another system instead of the Master.

For more information on how to schedule backups read [Creating a Back-Up Schedule](#) on page 219

General Remarks

- You should define and configure the PostScript printers on all the systems (Automation Engine Master and Assistant) if the **Print Task** is used for outputting on PostScript printers.

We recommend installing the PostScript printers locally on the:

- Master system (in Automation Engine Master- Assistant Setups)
- or on the Data Server (Automation Engine Data Server- Assistant server setups)

Configure them on the network so that you can access them on all other systems as network printer.

- Automation Engine Viewer and Automation Engine Server for CP applications require the Application Server software. We recommend the Application Server Software to be installed on:
 - the stand-alone systems (they use own local installation of Application Server software and local database engine)
 - the Master Server (in Automation Engine Master- Assistant Setups).

Note: The tasks running on the Assistants always connect to the Application Server on the Master.

- the Data Server (Automation Engine Data Server- Assistant server setups)

Licensing Requirements for Different Server Setups

You require specific licenses for installing or converting systems to Automation Engine Assistant and/or Master server systems. These limit the number of Automation Engine Assistants and/or Master systems.

These licenses are installed on the Automation Engine Master (in Master- Assistant Server setups) and in the Automation Engine Data Server (in Assistant- Data Server Setups). These systems act as remote license manager systems for the Automation Engine Assistant or Master systems.

System Account

The Automation Engine Server service (BGMD), the Automation Engine Web Service (egwebsrv) and the Application Server service (EG Application Server) run on service account (user: **BGSYSTEM** - ask the default password to Esko). You can change the username and password of this account.

Changing the System Account

- You can change the username to an existing Windows user. This user should be a member of the Windows Administrators group of the Master computer.
- Modifying the system account will modify the service account of the:
 - Automation Engine Server service (BGMD)
 - Automation Engine Web Service (egwebsrv)
 - Application Server service (EG Application Service) of the Master, Assistants and Data Server in one operation.

You can use either local or domain username. However, the Application Server only supports the local BGSsystem user. If you specify another local user or a domain user as system account, the service account of the Application Server will NOT be modified automatically.

Tip: You can use Windows tools (i.e. Service Manager) to modify the service account of the Application Server.

- When the Application Server is not installed on the Master, Assistant or Data Server, the modification of the system account is to be performed manually.
- If you have configured a Container or Resource a computer other than the Master, Assistant or Data Server, you have to modify the system account manually.

Processing Capabilities

Can Run Tasks

You can select this setting to allocate the Task workload to this Server. If you have a Master Server with many Assistants where the Master is heavily loaded, you can use this setting to allocate Task workload to the Assistants. Do not enable this setting to run only File Streaming Sessions.

Can Run File Streaming Sessions

Select this option to set the Server to run File Streaming Sessions. This setting determines Automation Engine Viewer's deep zoom abilities via File Streaming (implemented by a process called **sdbserver.exe**) which is started when you view a file which has no prepared view data or when you zoom deeper than the resolution of the prepared view data.

The maximum sessions setting is the maximum number of simultaneous File Streaming sessions on the Server.

Important: We recommend that only experienced system administrators or Esko Customer Service engineers modify the maximum number of File Streaming sessions. Erroneous modification of the maximum File Streaming sessions can seriously endanger the proper functioning of the setup.

Can Run Gang Run Printing Sessions

Select this option to set the Server to run **Gang Run Printing Sessions**.

A **Gang Run Printing Session** is the background process which calculates the optimal nested layouts (PDFPLA) of Gang Run Jobs for one particular substrate at the time. Since this background process can be time consuming, you can configure the Automation Engine server to run multiple of these background processes simultaneously for separate substrates.

When this option is selected, enter the maximum number of Gang Run Printing Sessions the server can run in the **Max. sessions** field.



Attention: Automation Engine's default maximum number of Gang Run Printing Sessions is 1. Increasing this number may lead to the Automation Engine Server slowing down.

See the [Gang Run Printing](#) documentation for more information about Gang Run Printing.

Redirect Central Resources

The Master uses the locally installed resources (fonts, CMS database, DGC curves etc) by default . You can use **Redirect Central Resources** task (only on Master) to redirect this to a Data server system.

You can redirect:

- `bg_data_cms_v010` – the Color Engine database
- `bg_data_custom_v010` - a resource containing various data like ArtPro action lists, Print Rule Checker profiles, etc..
- `bg_data_dgc_v010` - the Curve Pilot database contain dot gain curves
- `bg_data_fonts_v040` – the Central fonts repository
- `bg_data_ipldata_v010` – the Ink planner data
- `bg_data_marks_v010` – the SmartMarks repository

Note: You cannot redirect individual resources.

Backing up the server

In the Server Backup window you can manage the following backup settings:

- Specify a **backup folder** on the server.
- Specify the number of backups to be kept in **Keep only last**.
- Set a **backup schedule** or manage existing schedules.
- See the **Backup history** and details of each backup.

Tip: Click on **info** to get more information about the selected backup. A **Backup info** window will list the server, date, time, size and composition of the backup.

- **Remove** a backup.

The screenshot shows the 'Automation Engine Server' interface. The main heading is 'Server EAW12DD320' with a status of 'running'. The 'BACKUP' section includes:

- Backup folder:** C:\Esko\AEBBackup (with a BROWSE button)
- Keep only last:** 7 backups (with a link 'What's this?')
- Backup schedule:** A table with columns: Frequency, Start time, What, and actions (Modify, Remove).

Frequency	Start time	What	Modify	Remove
Every day	02:00:00	Configuration	Modify	Remove
Every week on Saturday	02:00:00	Configuration, Resources, Databases	Modify	Remove

Buttons: ADD BACKUP SCHEDULE, MAKE BACKUP NOW


Backup history: A table with columns: Date, What, Info, Remove.

Date	What	Info	Remove
7/6/12 3:38 PM	Configuration, Resources, Databases	Info	Remove
7/6/12 2:00 AM	Configuration	Info	Remove
7/5/12 2:00 AM	Configuration	Info	Remove
7/4/12 11:14 AM	Configuration, Resources, Databases	Info	Remove
7/3/12 5:36 PM	Configuration, Resources, Databases	Info	Remove
7/2/12 4:16 PM	Configuration, Resources, Databases	Info	Remove
7/2/12 2:00 AM	Configuration	Info	Remove
6/30/12 2:00 AM	Configuration, Resources, Databases	Info	Remove
6/28/12 9:48 AM	Configuration	Info	Remove
6/27/12 5:39 PM	Configuration, Resources, Databases	Info	Remove

Click the lock to make changes.

Creating a Back-Up Schedule

You can create a schedule to Back-Up the server configurations.

1. Go to the **ServerAdmin** pages. Task step.
2. Click on **Back-Up** in the left column.
3. Click the lock  to make changes. You can specify where the Back-Up folder should be stored.
4. Click on the **Add Back-Up Schedule** button. A pop-up window will open asking for details.

You can also choose **Make Back-Up Now** to immediately start backing up.

5. Define when and what to back up and how many backups should be stored in the given fields. Fill in the **Frequency**, **Start time**, **Back-Up What** and **Keep only last** fields.

FAQ on Backing-Up the Server

Is the configuration of the Automation Engine backed up automatically?

No. You need to set up a Back-Up schedule to achieve that. Read more about it in [Creating a Back-Up Schedule](#) on page 219

Can an Automation Engine Back-Up be made while the system is running ?

Yes. It can also be made when Automation Engine system is not running.

What is included in the Automation Engine Back-Up ?

The Back-Up always includes the basic configuration like tickets, workflows, SmartNames, configuration settings etc.. Additionally, you can include DGC-files, Marks, CMS data, Fonts and the data stored in the Databases in the Back-Up. This is called a Restore point which you can restore later.

Where are the backups stored ?

In the Server Back-Up window you can specify a Back-Up folder on the server. By default, the location of this folder is a path local to the Automation Engine server.

Tip: A UNC path is accepted as the location. You need to make sure that the corresponding share is accessible.

Tip:

You can keep a local path as destination Back-Up folder and save a copy on a DVD,/USB stick. The Back-Up consists of a subfolder named like `[Full]Backup__yyyy-MM-dd-HHMM`.

Will I eventually run out of disk space when I schedule a regular Back-Up ?

You can configure this by setting how many backups you want to retain. Read more in [Creating a Back-Up Schedule](#) on page 219.

Note: The restore points (Back-Up including basic configuration, resources and databases) are never deleted. Therefore, you need to save restore point backups on DVD/USB and delete them from the hard disk.

Restoring the server

In the Restore window you can view and restore backups. You can also:

- Restore, duplicate or migrate your server configuration
- Restore resources or databases only

Note: Take into account that the server should be stopped before you can restore a backup.

The screenshot shows the Automation Engine Server interface. The top navigation bar includes 'English', 'Automation Engine Server', 'CLIENT APPS', 'SERVER ADMIN', and 'FAQ'. The user is logged in as 'admin'. The main content area is titled 'Server RDALPHA25' and shows a 'RESTORE' section. A table lists backup history with columns for Date, What, and Restore. A lock icon is present, indicating that the server should be stopped to restore a backup.

Date	What	Restore
5/24/12 4:51 PM	Configuration, Databases, Resources	Restore
4/5/12 3:57 PM	Configuration, Databases, Resources	Restore
3/16/12 12:47 PM	Configuration, Databases, Resources	Restore
3/15/12 10:37 AM	Configuration, Databases, Resources	Restore
3/15/12 10:35 AM	Configuration, Databases, Resources	Restore
2/17/12 12:51 PM	Configuration, Databases, Resources	Restore

Click the lock to make changes.
The Automation Engine Server should be stopped to restore a backup.

How to Restore the Server

You can use **Restore** when you want to:


- reverse to a previous backup in order to correct some mistakes by going back in time.
You can use **Restore Configuration** to restore the Automation Engine backup that was made previously.
- build a testing scenario (alpha, beta or shadow production). You can take the production configuration settings from the production computer by creating and copying a backup.
After installing full Automation Engine on testing computer, you can restore the configuration in the testing computer using **Duplicate Configuration** setting. You can therefore have production and testing systems run in parallel.
- move to another computer. For example, upgrading from Automation Engine 10.1 on a separate computer or to have a new computer where the Automation Engine will run henceforth.

Note: You can create an Automation Engine 12 backup from an AE10.1 system. Read more in

You can choose **Migrate Configuration** after the full installation on the new computer. This will restore the old production settings on the new computer.

- give the Master's configuration to an Assistant when the Master goes down.
You can restore the Master backup on the Assistant using **Migrate Configuration** restore action.

Note: The Master backup can be found on the Assistant disk (C:\Esko\AEBBackup).

- Go to the ServerAdmin pages and click on **Start/Stop Server** in the left column.
- Click on **Restore** in the left column.
- Click the lock  to make changes.
- Click **Restore** after selecting the corresponding entry in the Backup History list. For all other cases, click the **Select Backup Folder** button and browse to the folder that contains the backup to be restored.
- Finally, choose **OK** to start the restore action.

FAQ on Restoring the Server

Can I use a backup of one system to transfer its Workflow tickets to another Automation Engine system ?

Yes, You need a dedicated Restore action. Consult Esko for this.

Can I use a backup of one system to just transfer its SmartNames to another AE system ?

Yes, You need a dedicated Restore action. Consult Esko for this.

Do I need to restore backup when upgrading from Automation Engine 10 to Automation Engine 12 ?

No. When the same computer is used to run Automation Engine 12 , it will use the same configuration data and databases.

Can I upgrade my BackStage 7Rx directly to Automation Engine 12 ?

No. You need to upgrade BackStage 7Rx to Automation Engine 10.1. You can upgrade this to Automation Engine 12.

Can I restore data from a backup of BS7Rx or AE 10 into an AE 12 ?

No. The format of BS 7Rx and Automation Engine 10.x backups is different from the format of the Automation Engine 12 backup.

How do I make Automation Engine 12 based backup of my Automation Engine 10.1 system ?

In the Client Apps view, click on **Tools** in the left column. Click on the **Launch** button next to Automation Engine 10 Backup.

In the window that pops up, fill in the destination folder, decide whether to include the databases and click on **Backup Now**.

Managing databases

In the **Databases** window you can manage all databases that are on any accessible SQL server in your network. Depending on the permissions on the selected SQL instance, you can:

- View the SQL server status
- Login and view the properties and status for each database
- Create or remove databases
- Back-up or restore databases
- Repair or move databases

Note: Take into account that, even if you are logged in to the Server Administration, you also need to log in to the SQL Server instance separately.

The screenshot shows the 'Automation Engine Server' interface. The main heading is 'Server RDALPHA25' with a status of 'running'. Below this, the 'DATABASES' section is active. A dropdown menu shows 'RDALPHA25' as the selected SQL server. The 'Instance' dropdown shows 'MSSQLSERVER' with a 'running' status. A list of databases is shown on the left, with 'BGARCHBASE' selected. The 'Details' panel for 'BGARCHBASE' shows the following information:

Database name	BGARCHBASE
Created	2011-08-30 19:05:48.623
Size	10.00 MB
Used	640 KB
Master database file	D:\Eskolbg_data_fastserver_dbase\BGARCHBASE_dat.mdf
Transaction log file	D:\Eskolbg_data_fastserver_dbase\BGARCHBASE_log.ldf
Status	Online
Actions	<input type="button" value="BACKUP"/> <input type="button" value="RESTORE"/> <input type="button" value="REPAIR"/> <input type="button" value="MOVE"/>

At the bottom of the interface, there is a lock icon and the text: 'Click the lock to prevent further changes.'

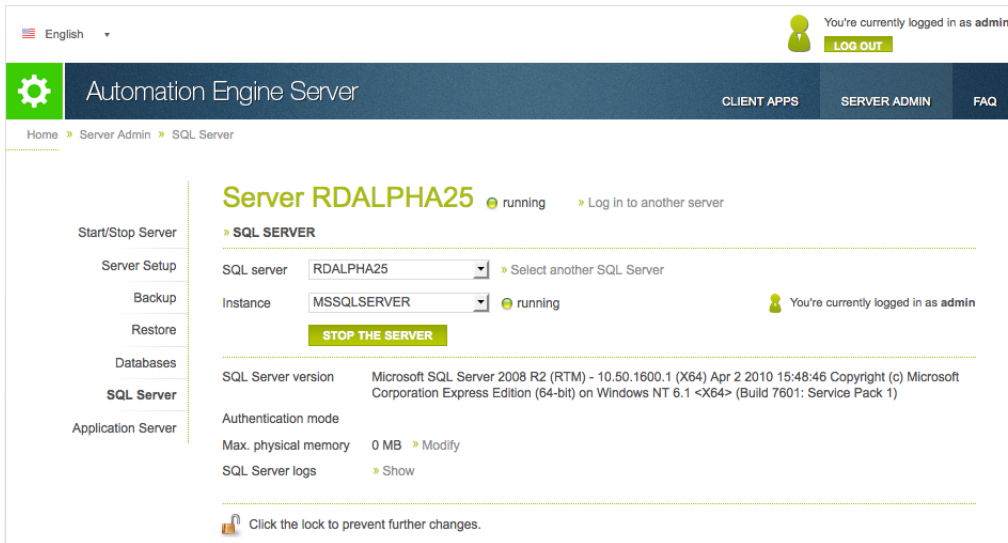
In this page, you can manage the databases on any SQL Server in the network.

Managing SQL servers

In the **SQL Server** window you can manage any accessible SQL server in your network. You can:

- View the SQL server status
- Stop and restart an SQL server
- Login and change the properties: modify the maximum physical memory of the SQL server
- Look at the SQL server logs

Note: Take into account that, even if you are logged in to the Server Administration, you also need to log in to the SQL Server instance separately.



The screenshot shows the 'Automation Engine Server' interface. At the top, there's a language selector (English) and a user profile for 'admin' with a 'LOG OUT' button. The main header includes 'Automation Engine Server' and navigation tabs for 'CLIENT APPS', 'SERVER ADMIN', and 'FAQ'. The breadcrumb trail is 'Home > Server Admin > SQL Server'. The main content area is titled 'Server RDALPHA25' and shows it is 'running'. Below this, there's a 'SQL SERVER' section with a dropdown menu for 'SQL server' (currently 'RDALPHA25') and a 'Select another SQL Server' link. Underneath, there's an 'Instance' dropdown (currently 'MSSQLSERVER') and a 'running' status indicator. A 'STOP THE SERVER' button is prominently displayed. Further down, the 'SQL Server version' is listed as 'Microsoft SQL Server 2008 R2 (RTM) - 10.50.1600.1 (X64) Apr 2 2010 15:48:46 Copyright (c) Microsoft Corporation Express Edition (64-bit) on Windows NT 6.1 <X64> (Build 7601: Service Pack 1)'. Other details include 'Authentication mode', 'Max. physical memory 0 MB' with a 'Modify' link, and 'SQL Server logs' with a 'Show' link. At the bottom, there's a lock icon and the text 'Click the lock to prevent further changes.'

How to Select and Log-In to SQL Server Instances

- You can select a SQL Server from the list of all SQL Servers that could be detected in your local network.


The local SQL Server will be selected as default if there is one.

- To choose one not in the list, you can click **Select another SQL Server**, enter the name of the SQL Server and click **OK**.

You can see the list of SQL Server Instances and their statuses beneath the SQL Server list. You can select the Server Instance on which you want to manage the databases.

Note: If the SQL Server Browser service is not running or your access is blocked to the Server Service, the status of the Instance will be unknown.


Note: If the list of SQL Server Instances is not available, you can select a SQL Server Instance by entering its port number. The port number of the default SQL Server Instance will already be filled in.

- After selecting the SQL Server Instance, click the lock  at the bottom of the page. Enter your credentials to login.

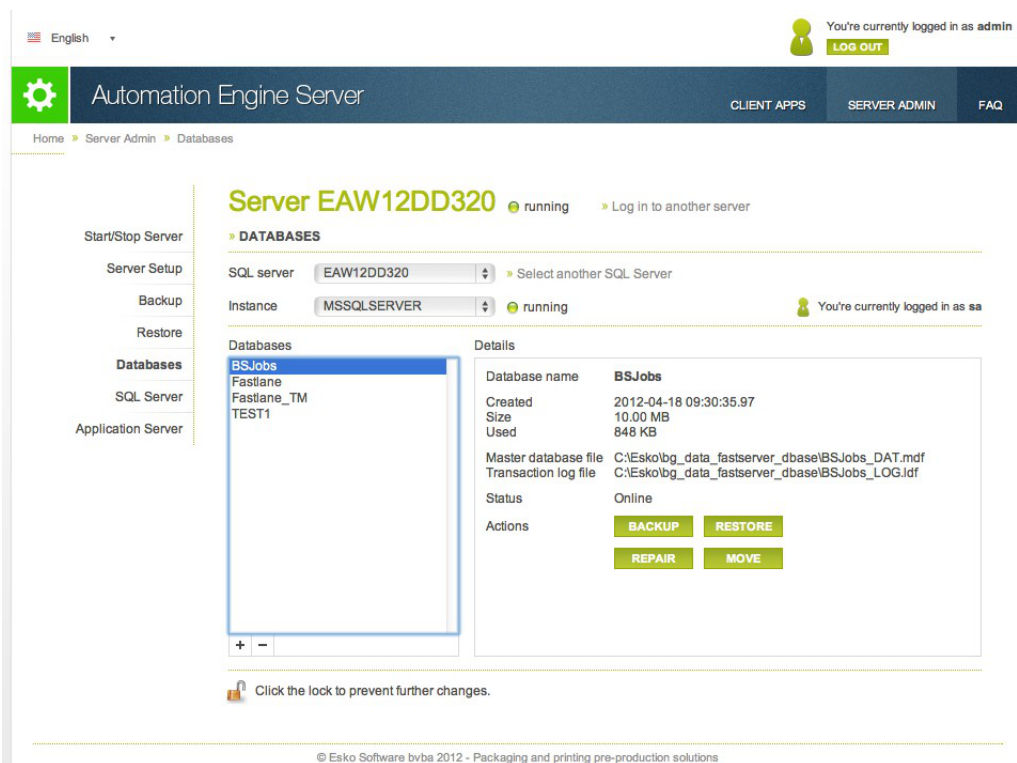
You should login to the SQL Server Instance to be able to manage the databases of a SQL Server Instance.

Note: All SQL instances configured with Automation Engine has two users by default: `sa` and `admin`. The `sa` user has all permissions for managing databases.

The `admin` user has a limited set of permissions. Therefore, if you are logged in as `admin`, you may get warnings while performing certain database actions.

- You can log out from the SQL Server Instance by clicking on the lock .
- You can view the databases installed on an SQL Server Instance by selecting it and logging in. The list of databases will load after logging in.

Detailed database information will be available to the right of the selected database.



FAQ on SQL Server Instances

How to start or stop a SQL Server Instance?

Select the SQL Server Instance you want to start or stop and click on **Start the server** or **Stop the server**.

You are not required to login to the SQL Server Instance to Start or Stop the Server.

How to view the properties of a SQL Server Instance ?

You should select a SQL Server Instance and login to it. The properties will be loaded automatically after you successfully logged in. Read more in [How to Select and Log-In to SQL Server Instances](#) on page 224

How to modify the maximum physical memory of a SQL Server Instance?

Click on the **Modify** link next to the **current value of the maximum physical memory**. The **Change max. physical memory** dialog will open. Enter a value and click **OK**.

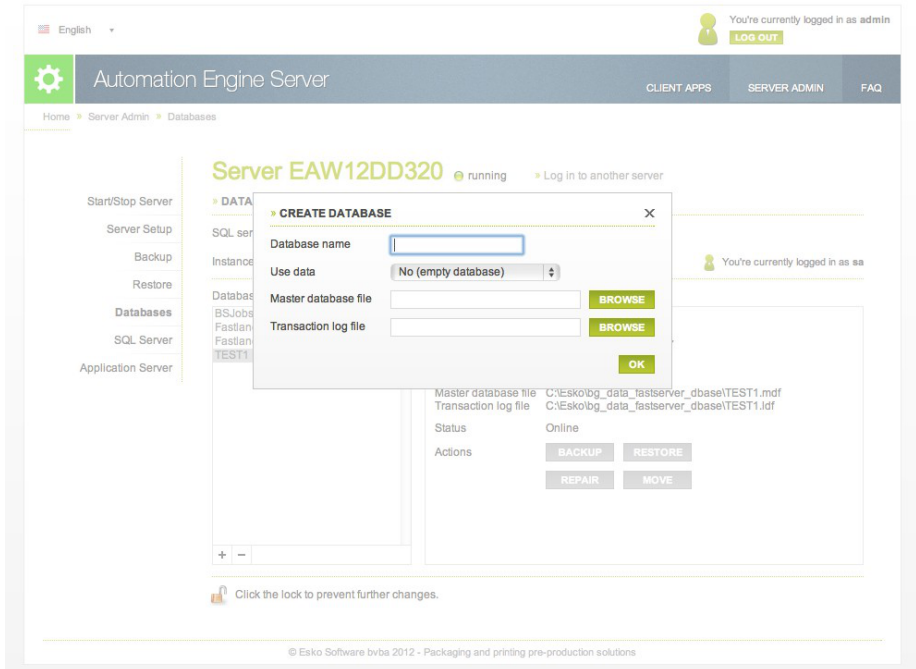
How to view SQL Server logs?

Click on the **Show** link next to **SQL Server logs**. You can see the SQL Server logs in the following dialog.

Note: You will get a warning when you attempt to view the SQL Server logs if you have limited permissions.

Creating an Empty Data base

1. Click on + button. The **Create database** dialog will follow.



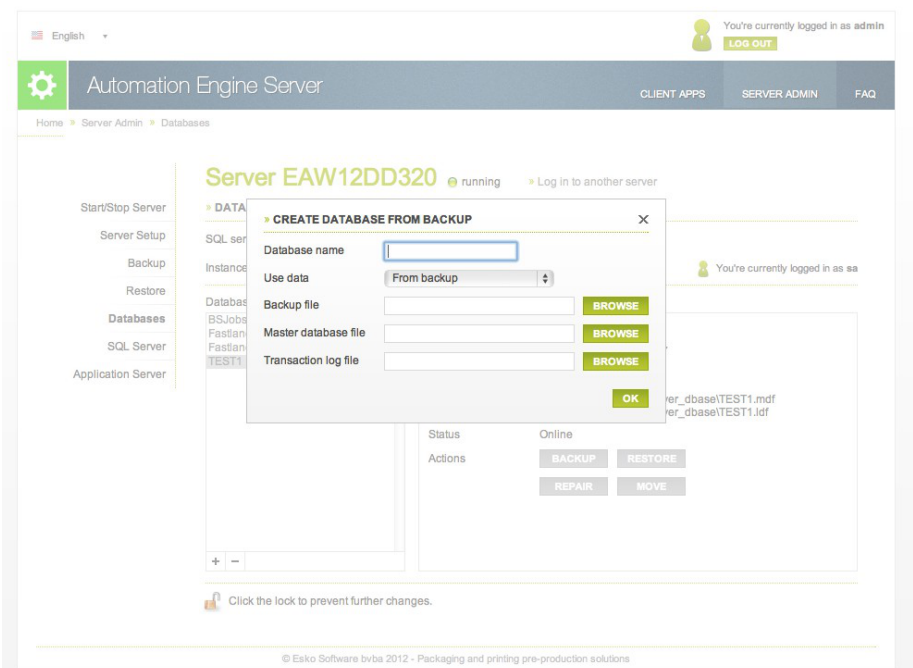
2. Fill in **Database name** with a name that is not currently used.
3. If you want to create a new empty database, leave the **Use data** setting to **No (empty database)**.
4. Enter the location and the file name for the **Master database file** and the **Transaction log file**. Alternatively, use the **Browse** to a specific location for these files.

Note: The file names can be either a local or UNC path. If the selected SQL Server is not the local SQL Server, the **Browse** will be disabled. In such cases, fill in the UNC path.

5. Click **OK** to create an empty database.

Creating A Database from Back-Up

1. Click on + button. The **Create database** dialog will follow.



2. Change the **Use data** setting to **From backup** .
3. Enter the location and the file name of the **Backup file** or use the **Browse** button to select it.

Note: The file names can be either a local or UNC path. If the selected SQL Server is not the local SQL Server, the **Browse** will be disabled. In such cases, fill in the UNC path.

4. The information from the backup file will be loaded and the **Database name**, **Master database file** and **Transaction log file** textfields will be filled in. If not, fill them in manually. Check the fields to ensure that the database will be re-created correctly.
5. Click **OK** to create a database from the backup file.

Creating a Database from Existing Database Files

1. Click on + button. The **Create database** dialog will follow.
2. Change the **Use data** setting to **From existing database files** .
3. Enter the location and the file name of the **Master database file** and **Transaction log file** or use **Browse** .

Note: The file names can be either a local or UNC path. If the selected SQL Server is not the local SQL Server, the **Browse** will be disabled. In such cases, fill in the UNC path.

4. Click **OK** to create a database from the backup file.

Deleting a Database

You can delete a database if that is not used by Automation Engine and the Automation Engine Server. If it is running, trying to delete will prompt a warning.

1. Select a database from the list and click the - button.
2. Choose either or both of the options.

- **delete database files from disk**
 - **close existing connections**
3. Click **OK** to confirm.

Database Back Up

Context for the current task

- Select a database and click on **Backup** . The **Backup database** dialog will open.
- Enter the location and the file name for the **Backup file**. use **Browse** to choose a back-up file.

Note: The file names can be either a local or UNC path. If the selected SQL Server is not the local SQL Server, the **Browse** will be disabled. In such cases, fill in the UNC path.

- Click **OK**.

If the selected SQL Server does not have access to the UNC path you entered, the Server Admin Web Tool will show a warning: "Backup file for database X could not be created. Please make sure the path for the backup file is correct.".

You can ensure access to the UNC path share by following the procedure outlined below:

1. In Windows Explorer, right-click the shared folder and then click **Properties**.
2. Select the **Security** tab.
3. Click on **Advanced** . The Advanced Security Settings dialog will open.
4. Click on **Change Permissions** under the **Permissions** tab.
5. Click on **Add** .
6. Click on **Object Types** button and make sure the **Computers** checkbox is checked. Click **OK**.
7. In the textfield, give the SQL Server computer name for which you want to add permissions. Click on **Check Names** . Click **OK**. The Permission Entry dialog will open.
8. Make sure the **Allow** checkbox next to **Create files / write data** is checked. Click **OK**.
9. Click **OK** and close the open dialogs.

Restoring Database

You can restore a database by using **Restore** function. You can will the location and file name in the **Restore database** dialog. Click **OK** to restore the backup.

You can restore a database that does not exist (anymore) on the selected SQL Server Instance databases list. Click on + to create a database from a backup file or from existing database files. Read more in [Creating A Database from Back-Up](#) on page 226 and in [Creating a Database from Existing Database Files](#) on page 227

Note:

If the database you are trying to restore is used by Automation Engine and the Automation Engine Server is running, it will not be possible to restore the database and you will get a warning.

Moving or Renaming a Database

Context for the current task

1. Select a database from the list and click on **Move** . The **Move database** dialog will open. .

2. Enter the new location and the new file name for the **Master database file** and the **Transaction log file** or use the **Browse** button to select it.

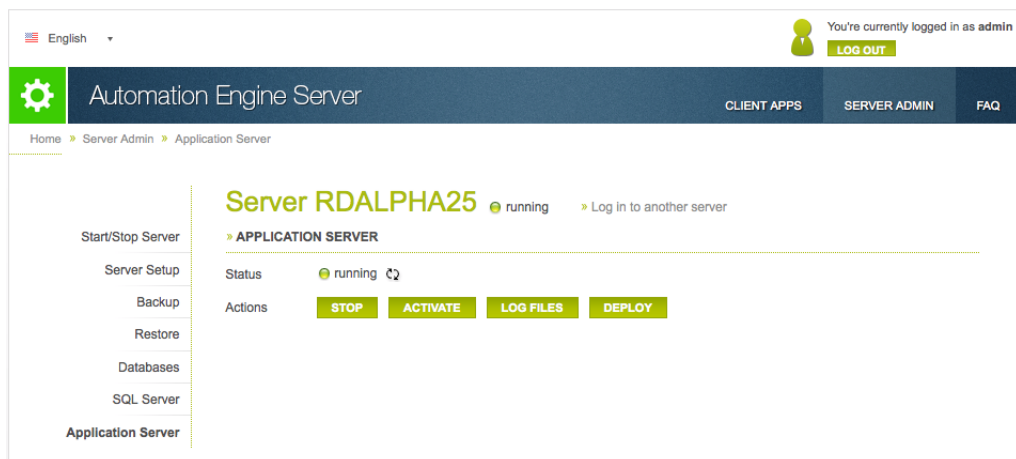
Note: The file names can be either a local or UNC path. If the selected SQL Server is not the local SQL Server, the **Browse** will be disabled. In such cases, fill in the UNC path.

3. Click **OK** to move/rename a database.

Application Server

You can manage the local Application Server which is installed on the same computer as the Automation Engine via the Server Admin Web Tool. Go to `Home/Server Admin/Application Server`. In the **Application Server** window you can view the status of any accessible application server in your network. You can:

- Start, stop or activate the local application server
- Download log files for support purposes
- Deploy components on the application server
- Log on to another application server



Among the actions available at this page, **Deploy** is only available when the server is configured as Master Server. All the other actions are available when Application Server is installed.

Managing Application Server

Activating the Application Server

You can **Activate** the Application Server for repairing a serious malfunction of the Application Server and when executing Deploy does not solve the malfunctioning of the Application Server.

Retrieving the Automation Engine Viewer Log Files

When you click the **Log Files** button, a ZIP file containing the Automation Engine Viewer log files and the Application Server log files will be downloaded. The ZIP file contains a folder named `egappsrvlogs` with Viewer log files (`ImageList*.log`) and a folder `jbosslogs` with Application Server log files.

Tip: The log files of the **Pages and Proofs View** are contained in the log files of the Application Server.

Deploy

When you click **Deploy**, Automation Engine Components are copied into the `EG_APPSRV_DEPLOY` and `EG_APPSRV_BINDEPLOY` share of the computer indicated in the Deploy dialog. When you start the deploying, you indicate which computer Automation Engine will use for connecting to the Application Server.

Note: Deploy can only be performed when the Master Server is running.

When to use Deploy?

You can use **Deploy** when:

- you want to use another computer as Application Server.
- the Application Server installation/reinstalled, repaired or updated.
- fixing incomplete or failed deployments.

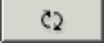
The Automation Engine components are deployed on the Application Server during the installation of Automation Engine. If the deployment failed or was not executed, you can use Deploy to complete the procedure.

Note: Incomplete deployments can cause the following issues :

- The Viewer window fails to open when clicking on **View** in the Pilot.
- The Pages and Proofs View fails to function.

Troubleshooting

If you get an error message “Failed to connect server: host:8080” while clicking **View** in the Automation Engine Pilot:

- Verify that the Application Server is started. Check the status and click the refresh icon  next to the status. Restart the Application Server and open the **View** again.
- It is possible that the deployment is incomplete. Click the **Deploy** button and verify that the deployment succeeds.

2.5 The Automation Engine Shuttle

The Automation Engine Shuttle is a client tool for easy submission of files to Automation Engine workflows. Shuttle also allows monitoring of the workflows.

Download Shuttle from the Main section of the Server Admin Web Tool. You can access it by browsing to `http://<name of your server>` or `http://<name of your server>:9999`. Read more in [Installing Client Applications](#) on page 28

If the Shuttle application is installed on your computer, the system will check if you have the appropriate version and download updates if necessary.

As explained in [Shuttle](#) on page 681, you first need to configure your Shuttle in **Tools > Configure > Shuttle** .

2.5.1 Launching the Automation Engine Shuttle

When you launch the Shuttle application from your installed programs, a pop up window with the Shuttle Launch panel and a Task Monitor will appear.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
OmNom-eat.ai	mabm	Run Script	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✔	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom.ai	mabm	Wait for Actio...	0%	User intervention needed	▶	10/30/12 5:15 PM
OmNom.ai	mabm	Workflow	0%	Running	▶	10/30/12 5:15 PM
Screen Shot 2012-09...		Wait for Actio...	0%	User intervention needed	▶	10/30/12 5:14 PM
Screen Shot 2012-0...		Workflow	0%	Running	▶	10/30/12 5:14 PM
arrow.png		Workflow	100%		✘	10/30/12 10:56 AM
arrow.png		Workflow	100%		✘	10/30/12 10:56 AM
arrow.png		Workflow	100%		✘	10/30/12 10:56 AM
arrow.png		Workflow	100%		✘	10/30/12 10:56 AM
Screen Shot 2012-09...		Wait for Actio...	0%	User intervention needed	▶	10/30/12 9:16 AM
Screen Shot 2012-0...		Workflow	0%	Running	▶	10/30/12 9:16 AM
Screen Shot 2012-09...		Workflow	100%		✘	10/30/12 9:16 AM
Screen Shot 2012-0...		Workflow	100%		✘	10/30/12 9:16 AM
OmNom-eat.ai	mabm	Run Script	100%		✘	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✔	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/24/12 11:37 AM
OmNom-eat.ai	mabm	Run Script	100%		✘	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✔	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✔	10/22/12 12:27 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	10/22/12 12:27 PM

Workflow: Repeat and Export to Normalized PDF File

Drop Files Here

Launch...

- To open a new window, go to **File > New Window** . You can use dedicated Shuttle windows for each of your Workflows.
- Shuttle saves the settings of the open windows while closing. These settings are restored during the next launch.
- You can use **View** menu to hide or show the The Task Monitor, Shuttle Launch panel and the Status Bar.
 - **View > Tasks** hides/shows the Task monitor.

- **View > Launch Panel** hides/shows the Launch Panel.
- **View > Status Bar** hides/shows the Status Bar indicating the progress.

2.5.2 Working with the Automation Engine Shuttle

You must login to the Automation Engine Server to use the Shuttle.

Note: You have access to Shuttle if you have access to Pilot. But your access to Pilot may be restricted even when you are allowed to use Shuttle. This requires a user **access right (Pilot: start the Pilot (in addition to Shuttle))**. Read more in [Access Rights](#) on page 702.

The following menus are available in the Shuttle application on Mac and Windows:

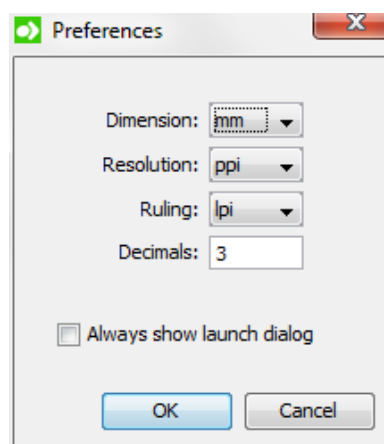
- **File** menu: Submit one or more files and launch the selected Workflow, open new Shuttle windows and close Shuttle windows.
- **Edit** menu: cut, copy, paste, delete a Task or Workflow, select all Tasks and Workflows in the Task Monitor.
- **View** menu: show or hide the Task Monitor, the Shuttle Launch panel and the Status Bar.
- **Task** menu: open or show the output file of the selected Task or Workflow, release or cancel the Task or Workflow, add annotations and show detailed information.
- **Window** menu: minimize, zoom and switch between Shuttle windows.
- **Help** menu: See the **About** window or open the Online Help (Esko Help Center).

Setting the Preferences for Shuttle

- On Mac, select **Preferences...** in the **Application** menu.
- On Windows, go to **Edit > Preferences...**

In the **Preferences** window, set the units for **Dimension**, **Resolution**, **Ruling** and the number of **Decimals** to use.

You can also select the **Always show launch dialog** option. This will show the **Public Parameter** dialog upon starting a new task, even if there aren't any public parameters. This allows you to alter these types of tasks before they launch.



The Task Monitor

When you are logged in, **The Task Monitor** displays your Automation Engine Workflows launched from both the Shuttle and the Pilot.

To filter the displayed Workflows, you can choose a job context filter.

- Go to the drop-down list in the toolbar and select the desired job. All recently used jobs are listed in this list.
- If the desired job is not present in the list, choose **Other...**. The **Open** dialog will prompt where you can choose the desired job.
- Choose the menu item "All Jobs" to clear the current filter.

Note: You cannot view, open or edit the Tickets of the Tasks or the Workflows in the Task Monitor. This is only possible in the Automation Engine Pilot.

Intervening in a workflow

You can intervene when a workflow is halted for user intervention provided the Workflow user intervention settings designate you for intervention. To intervene,

- Click on **User intervention needed** link of a workflow.
- In the resulting **To Do** window, choose to **Forward** or **Release**.

Repeat and Export to Normalized PDF File						
File Name	Job Name	Task Type	Progress	Phase	State	Launched
OmNom-eat.ai	mabm	Run Script	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✔	11/7/12 9:08 AM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/7/12 9:08...
OmNom-eat.ai	mabm	Run Script	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✔	11/6/12 1:45 PM
OmNom-eat.ai	mabm	Workflow	100%		✘	11/6/12 1:45 PM
OmNom.ai	mabm	Wait for...	0%	User intervention	▶	10/30/12 5:15...
OmNom.ai	mabm	Workflow	0%	Running	▶	10/30/12 5:15...
Screen Shot 20...		Wait for...	0%	User intervention	▶	10/30/12 5:14...
Screen Shot 20...		Workflow	0%	Running	▶	10/30/12 5:14...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
arrow.png		Workflow	100%		✘	10/30/12 10:5...
Screen Shot 20...		Wait for...	0%	User intervention	▶	10/30/12 9:16...
Screen Shot 20...		Workflow	0%	Running	▶	10/30/12 9:16...

Tip: You cannot hide the columns displayed in the Task Monitor. However, you can change the order and the width of the columns.

Context menu options

Right-click on a workflow/task to access the context menu with the following options:

- **Open Output File** opens the output file of the Task or Workflow with the appropriate application.
- **Show Output File in Finder/Explorer** shows the folder that contains the output file in Mac Finder or in Windows Explorer.
- **Release** allows you to release the Task or Workflow if it is on Hold.
- **Cancel** allows you to cancel the Task or Workflow.

- **Delete** allows you to delete the Workflow from the list of Workflows the Task Monitor.
- **Annotation...** shows a dialog that allows you to add or modify an annotation.
- **Details** shows the Details dialog with detailed information about the Task or Workflow.

Alternatively, these options are also available via the **Task** menu of the Shuttle application.

File Name	Job Name	Task Type	Progress	Phase	State	Launched
shuttle12.1_cont...		Wait for Acti...	100%		⊗	11/7/12 2:52 PM
shuttle12.1_con...		Workflow	100%		⊗	11/7/12 2:52 PM
shuttle12.1_ae_s...		Wait for Acti...	100%		⊗	11/7/12 2:52 PM
shuttle12.1_ae_...		Workflow	100%		⊗	11/7/12 2:52 PM
shuttle12.1_todo...		Apply ArtPr...	100%		⊗	11/7/12 2:20 PM
shuttle12.1_tod...		Workflow	100%		⊗	11/7/12 2:20 PM
OmNom.ai	mabm	Wait for Acti...	100%		⊗	10/30/12 9:16 AM
OmNom.ai	mabm	Workflow	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Wait for Acti...	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		⊗	10/30/12 9:16 AM
arrow.png		Workflow	100%		⊗	10/30/12 9:16 AM
arrow.png		Workflow	100%		⊗	10/30/12 9:16 AM
arrow.png		Workflow	100%		⊗	10/30/12 9:16 AM
arrow.png		Workflow	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Wait for Acti...	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		⊗	10/30/12 9:16 AM
Screen Shot 201...		Workflow	100%		⊗	10/30/12 9:16 AM

Context Menu:

- Open Output File
- Show Output File in Finder
- Release
- Cancel
- Delete
- Annotation...
- Details

The Shuttle Launch Panel

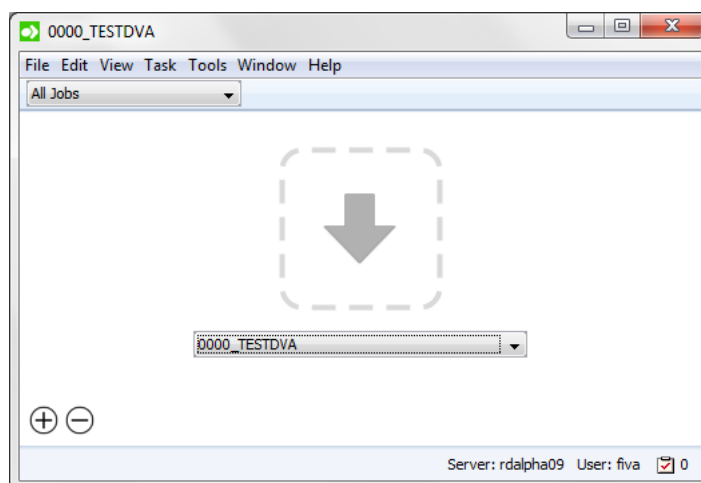
You can drop one or more files as input files for the selected Workflow/Task through the Shuttle Launch Panel.

- Browse to the desired workflow using the **Workflow** drop-down list. This will be updated when new workflow tickets are created. Shuttle launches the selected workflow when files are placed in the Shuttle Launch panel.



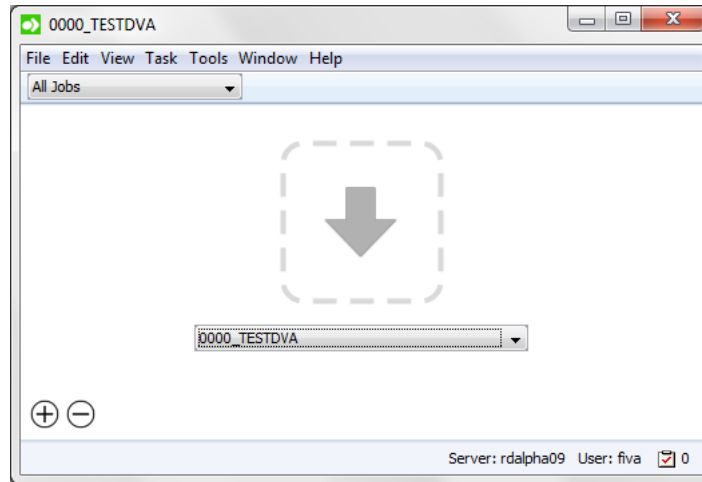
Attention: The drop-down list only shows the Workflow Tickets that are public.

- The selected workflows are saved when you quit Shuttle. Shuttle restores the selection of workflows during the next launch.



Launching a Workflow from the Automation Engine Shuttle

1. If the Launch Panel is not visible in the Shuttle window, go to **View > Launch Panel** .

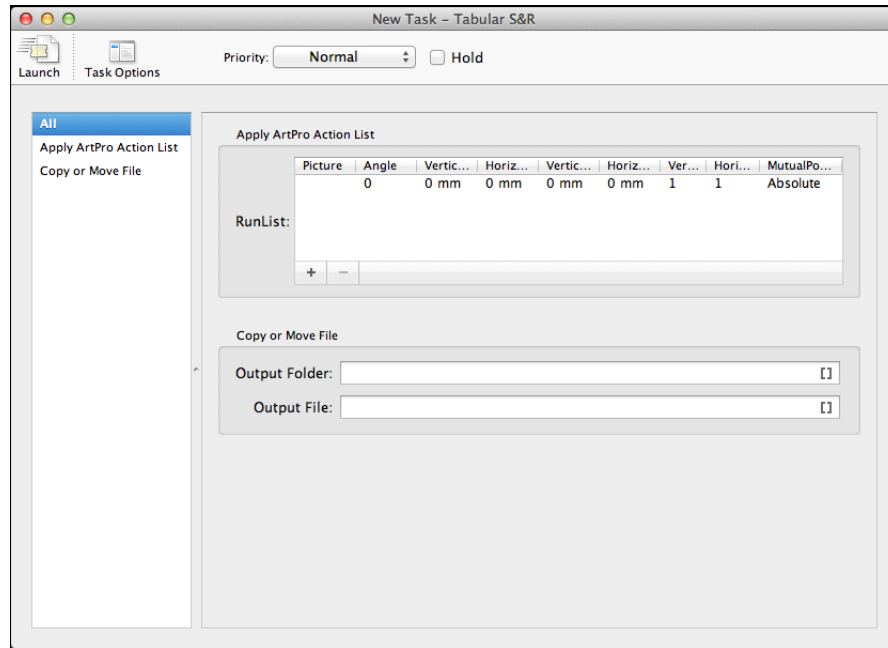


2. Select the desired **Workflow** with the drop-down menu. Shuttle will launch this **Workflow** when files arrive at the launch panel.
3. Submit one or more files in one of the following ways.
 - Drag and drop the desired file(s) into **Drop Files Here** area.
 - Go to **File > Launch...** and select the files.
 - Use the keyboard shortcut `Command-L` on Mac or `Ctrl-L` on Windows and select the file(s).

Note:

- The selected file(s) will be uploaded to the root folder of the job.
- If a file was previously launched on the chosen workflow, a dialog will prompt to give you the option to reuse the public parameter values from the previous launch. If you choose to reuse these public parameter values, they will be used as default parameter values. You can still edit these values if necessary.
- If a file with the same name is already present in the root folder of the job, a dialog is prompted warning you of overwriting the file. You can then choose to either **Replace** the existing file, or **Cancel** the launch to rename the file.

4. If your workflow has Public Parameters, you can fill them in the **New Task** window which will prompt. Learn more in [Using Public Parameters in Workflows](#) on page 1051. It also possible to set the **Task Options** in this window.



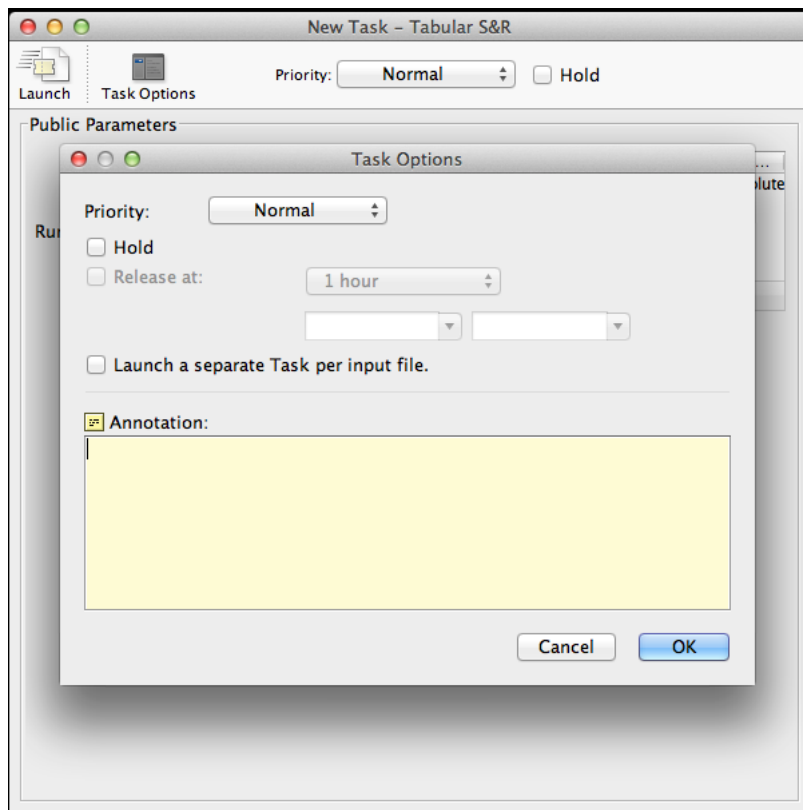
a) You can set **Task Options** in the **New Task** window, click the **Task Options** button. In the resulting window, you can:

- set the **Priority** of the Task or Workflow as **Low**, **Normal**, **High** or **Immediate** by selecting from the drop-down.
- select **Hold** if you want to hold the launch of the Workflow. The Workflow will not be launched until you **Release** via the right-click menu in the Task Monitor or via the **Task** menu.

Alternatively, hold the workflow until a set period of time after which it will be automatically released. To do this, select **Release at** and specify the period or date and time.



Note: You can bypass the automatic release setting by selecting **Release** via the right-click menu in the Task Monitor or via the **Task** menu.

- Select **Launch a separate Task per input file** for launching a separate Task or Workflow for each input file.
- You can also set an **Annotation** for the Task or Workflow in this window.





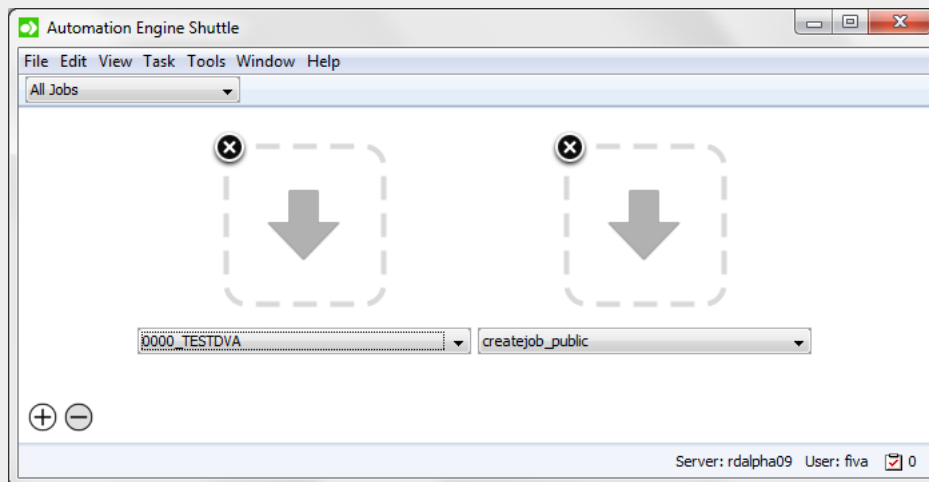
5. Click the **Launch** button to launch the workflow.
The task will immediately appear in the Task Monitor.

Note:

You can upload multiple files for multiple workflows. Click on the **Plus** button   to add extra drop zone slots.

To remove these slots, proceed as follows:

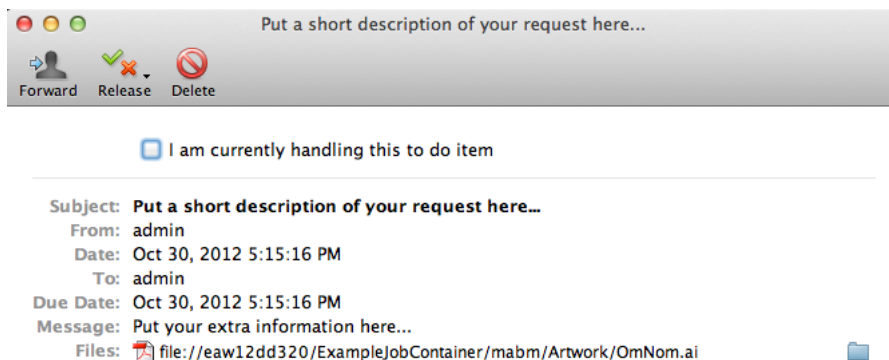
- Click on the **Minus** button  . This will place the user interface in Editing mode
- Click on the desired **Delete** button on the top left corner of the drop zone.
- To leave the Editing mode, click on the **Minus** button, or press **Escape**.



User Intervention from the Shuttle

If you have setup a **Wait for Action** step in the workflow, the process halts for user intervention (for a **To Do** action). **User intervention needed** link will appear in the Task monitor when the workflow is halted.

1. Click on the **User intervention needed** link.
The **To Do** dialog will prompt.



2. You can select **I am currently handling this to do item** to mark that you are handling this **To Do**.
3. You can choose to **Forward**, **Release** or **Delete** the **To Do**.

- You can let the workflow continue by clicking **Release**. Select the output state that you require in the release. The **To Do** window will close and the Workflow will continue processing.
- You can forward the **To Do** to another user. Click **Forward** and the workflow will not continue until the other user handles the **To Do**.
- You can also delete the **To Do** by clicking on **Delete**. This will end the processing of the workflow.

2.6 The Workflow Editor

The Workflow Editor is documented in the separate chapter on [Workflows](#) on page 986.

2.7 The Tasks and Workflow Controls

This chapter is a full (alphabetical) list of all available Tasks and Workflow Controls and their description. Some of the tasks and tools mentioned are optional.

Some of the items refer to another page where they are documented as part of a dedicated chapter related to that technology.

At the end of this complete list, we mention the deprecated and obsolete tasks.

2.7.1 Add Dynamic Marks

Use this task to apply marks from a pre-defined mark set to the input PDF file. The mark sets are created using the Dynamic Marks plug-in for Adobe Illustrator.

This task produces a non-normalized PDF. This PDF does contains all the information from the input file (for example object based screening).

Note: In situations where a normalized PDF is required in steps of the workflow after the **Add Dynamic Marks** task, you can add the **Normalize PostScript/ PDF/ Illustrator file** task to the workflow. If you use this extra task, ensure the **Use rulings, angles and dot gain compensation from file** option is enabled on the **Inks** tab to preserve the screening parameters.

Task Settings

Replace existing marks

Mark Set	Add To
1 x LabelRow	All

Processing: Calculate once and copy (faster)
 Calculate for each page separately (more accurate)

Replace existing marks: Select this option to remove existing marks from the input file while adding the new mark set. If left unchecked, the old marks will be preserved.

Mark Set: This table lists the mark sets added to the task and the pages those mark sets apply to. To add or remove marks from the table, or edit the existing marks, see the following section: **Mark Set Control** for more info.

Processing: The time this task will take to process the marks. Choose from the following options:

- **Calculate once and copy (faster):** When choosing this method, the mark content is calculated only once for the first and second page. After that, this information is copied to all other pages. The content from the first page is copied to the odd pages, the content from the second page is copied to all even pages. This method is also called "caching".

Note: Some marks cannot be cached because the content depends on the page number (for example text mark with a sheet number variable). In this case, the task will fail to avoid a wrong output. If this happens a message will be displayed in the task details.

Note: Do not combine caching together with text marks that contain current time. If so, the time stamp from the first page will be copied to all odd pages and the time stamp from the second page will be copied to all even pages.

- **Calculate for each page (more accurate):** Use this option when mark content calculation for each page is necessary. This option will slow down the progress of the task, but it will generate more accurate results.

Mark Set Control

You can use the following options to control the **Mark Set** table:

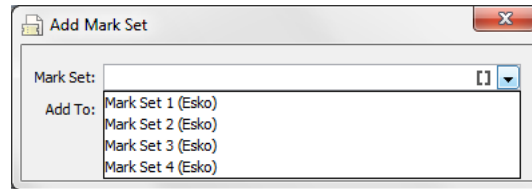
- **Add:** Click this button underneath the table to add a new **Mark Set**. The **Add Mark Set** dialog will prompt:

Add Mark Set

Mark Set: [] ▾

Add To: All pages ▾

- **Mark Set:** specify the name of the **Mark Set**, using one of the following methods:
 - Select the name from the existing **Mark Set** names, available via the selector on the right side.



- You can type a name manually.
- You can use SmartNames.
- **Add to:** specify on what page(s) the mark set should be applied. Choose one of the following options :
 - **All pages:** The mark set will be applied on all pages.
 - **Even pages :** The mark set will be applied on all even pages.
 - **Odd pages:** The mark set will be applied on all odd pages.
 - **First page:** The mark set will be applied on the first page only.
 - **Last page:** The mark set will be applied on the last page only.
- **Remove:** To remove a mark list, select the desired mark list and click the **Remove** button underneath the **Mark Set** table.
- **Edit:** To edit a mark set, double-click the desired mark set in the table. The **Add Mark Set** dialog will prompt, allowing you to edit the details.

2.7.2 Add Ink Eaters

Ink Eater areas are the non-printed areas that are not part of the printing jobs. These can be left-overs that will be cut off or the white spaces next to and in between one-ups. Ink Eater areas help to achieve a constant ink consumption over the width of the plate. This makes it easier for press operators to set the ink keys, and results in a faster setup time for the press.

Ink Eater areas helps to avoid excess ink buildup by stabilizing "peaks" and "lows" in the ink key histogram. This prevents spilling of excess ink onto the printed zones of the printing jobs. You can flatten the ink key histogram by filling non-printed zones with a gradient. A gradient(resolution) is used to avoid hard edges in the compensation object. You can adjust the settings for minimum ink density and minimum ink key value. You can get warnings when the maximum total area coverage (TAC) is too high depending on the settings of this task. These areas are then compensated(filled with ink) depending on the strategy settings you give in the task.

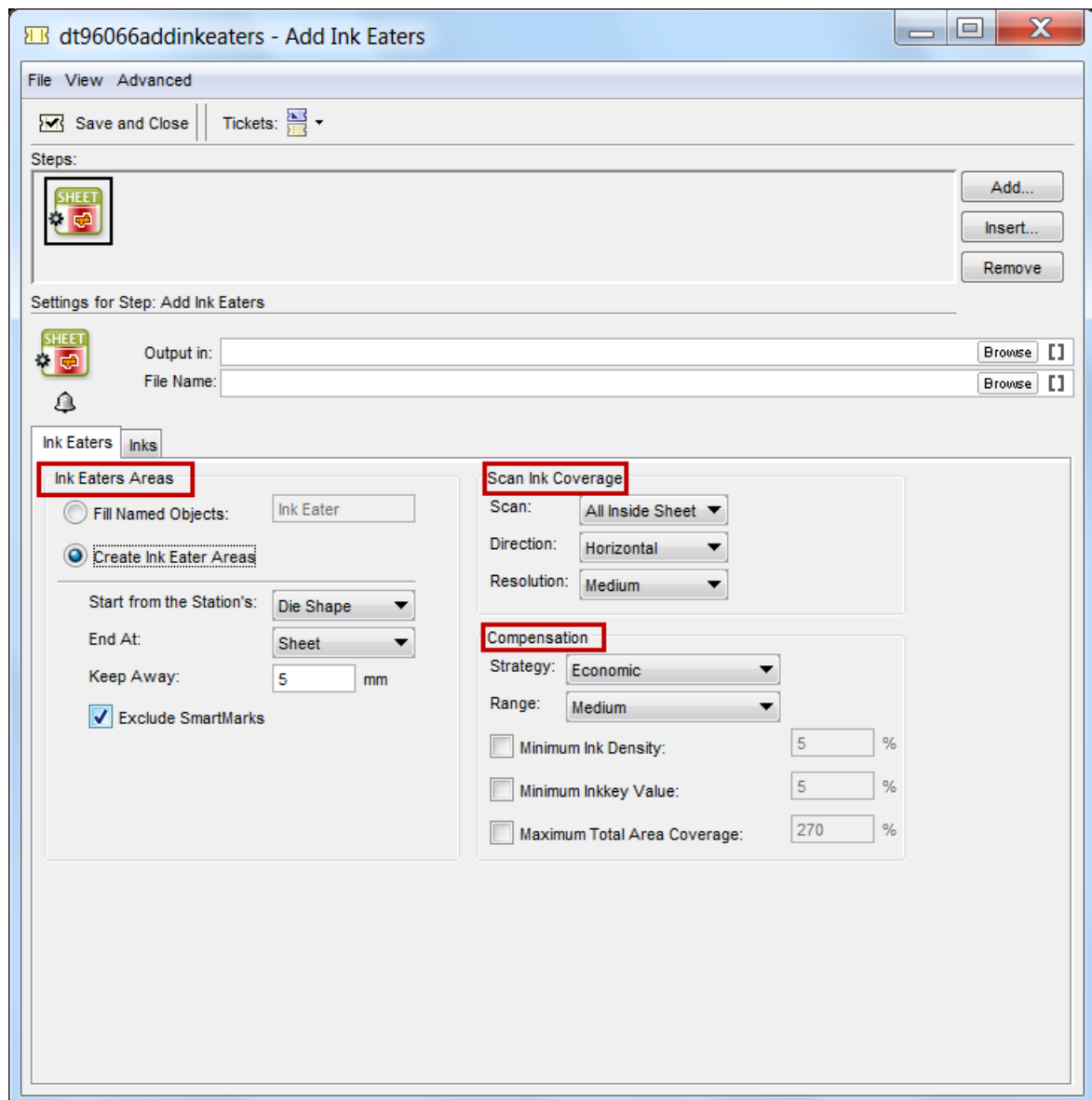


Using this task you can :

- create Ink Eater areas or choose a predetermined Ink Eater area
- fill Ink Eater areas with predefined settings.

Select the **Ink Eater** tab to create ink eater area and to define settings for filling Ink Eater zones.

Ink Eater tab



Ink Eaters Areas

You can choose predefined and named objects as Ink Eater by choosing **Fill Named objects**. This is useful when you have defined the Ink Eater areas in another software such as Plato.

Create Ink Eater Areas

You can choose from four options in the **Start from Station's** field to create Ink Eater areas:

- **Die Shape:** when the Die Shape is pre-defined (for example in Plato) and when it can be used as edge for the Ink Eater object.
- **Trim Box:** when the edge of the Ink Eater object is defined by the Trim Box of the one-up on the plate.
- **Media Box:** when the edge of the Ink Eater object is defined by the Media Box of the one-up on the plate. Media Box is larger and it contains the Trim Box.
- **Die Annotation:** You can mark an object as a die in a software such as Plato. When you choose this option, these objects will be used to define the edge of Ink Eater area.

You can restrict the creation of Ink Eater areas by choosing the shape as above. The selected shapes will be saved as Ink Eater areas. To set the offset for the Ink Eater object in relation to the Die Shape or Trim Box, you can fill in the distance in the **Keep Away field** . You can exclude SmartMarks in the Ink Eater object by enabling **Exclude SmartMarks** option.

Scan Ink Coverage

When you have selected a contour object or a group of contour objects for ink coverage, you can use the options under Scan Ink Coverage for filling the Ink Eaters

- **Scan:** You can specify the area the scan has to cover by selecting either of the two choices. **All Inside Sheet** (Sheet View) will scan within the sheet and will exclude objects that are outside the sheet. **All Inside Plate** (Sheet View) option will scan all objects on the sheet and plate. If you have objects that extend or are outside the plate, they are not included.
- **Direction:** You can specify the direction for ink coverage. **Horizontal** means that the ink zones are vertical on the plate and the ink histogram is on the horizontal base of the document. The direction of the compensation gradient in the ink eater area is horizontal.
- **Resolution:** This indicates the number of intermediate steps of the gradient to be used as compensation.

Note:

You can achieve **global compensation** by giving a low resolution value (for example 5 gradient steps). This gives a very smooth compensation gradient along the length of the press. The ink histogram zones are very wide with this setting.

You can achieve **very local compensation** by setting up a high resolution value (for example 60 gradient steps). This results in a gradient with more banding.

Use **Very Low** for 5 steps, **Low** for 15 steps, **Medium** for 30 steps, **High** for 45 steps and **Very High** for 60 steps.

Compensation

You can choose the ink use strategy for filling the Ink Eater zones from the options given below.

- **Economic** : you can minimize the ink use by choosing this strategy to fill Ink Eater areas.
- **Best performance:** you can have the best performance in print by using the maximum ink to fill the Ink Eater zones.
- **Intermediate:** you can optimize the ink use and to get medium performance, you can use this strategy.

You can specify the Ink key settings for filling the Ink Eater areas by defining the following settings.

- **Minimum Ink Density** You can set the minimum ink density of the ink separation in the ink eater area by enabling this option and specifying the percentage.
- **Minimum Ink key Value:** This indicates the minimum ink key value for the ink eater area. The ink density of the gradient is adjusted to make sure the ink key value is higher than this settings.
- **Maximum Total Area Coverage:** You can enable the option and enter a maximum total area coverage (for example, 270%) to display warnings in those areas of the histogram where the total ink coverage exceeds your limits.

Inks tab

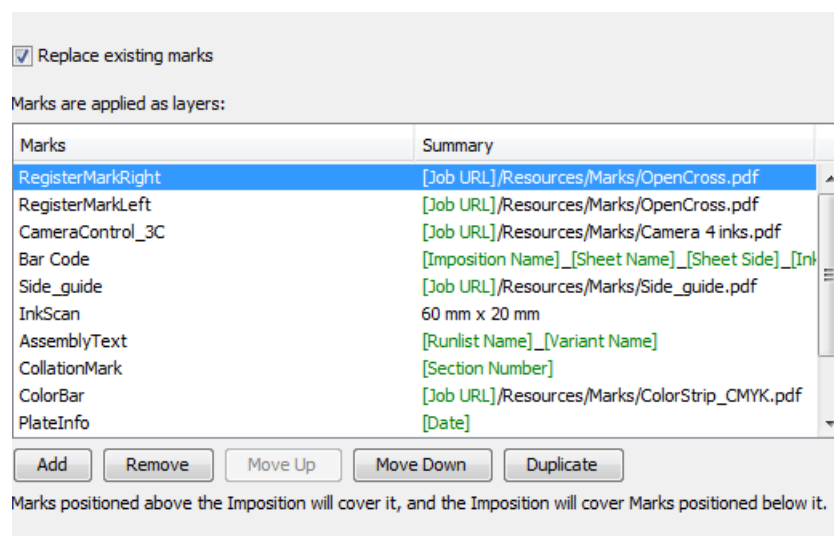
You can exclude inks such as **Non-process**, **Opaque** and **Varnish** inks by enabling these options in the **Inks** tab. You can keep the original separations by enabling **Keep similar inks in their original separations**

2.7.3 Add Marks To Imposition

Use this task to add **mark** definitions to impositions, plates or pages. This task can add marks at all these levels:

- **page**
- **sheet**
- **section**

For example, you can add sluglines to include text at sheet level or page level and collating marks for the entire section.



The added marks will be included while exporting the output using the **Export Imposition to PDF file** task or the **Export Imposition to JDF file** task .

Check **Replace existing marks** to remove the existing marks before adding the new marks.

Note: Existing marks here include the marks defined in the JDF template.

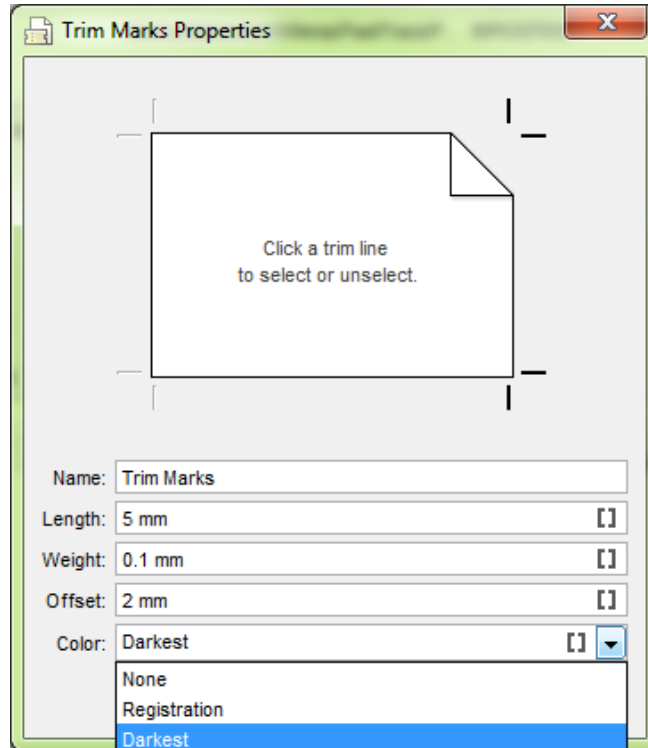
Use **Add** to compose the list of **marks**.

You can choose from the following mark definitions:

- **Trim Mark:** Info about adding a **Trim mark** in [Adding Trim Marks](#) on page 246,
- **Rectangle:** Read about adding a rectangle mark in [Adding Rectangle Marks](#) on page 247
- **Text:** Read about adding text marks such as collating marks in [Adding Text Marks](#) on page 247
- **Image:** Info about adding an image mark in [Topic title](#)
- **Bar Code**

When you have selected the type of smart mark, a **Properties** dialog will open where you can define appropriate settings.

Adding Trim Marks

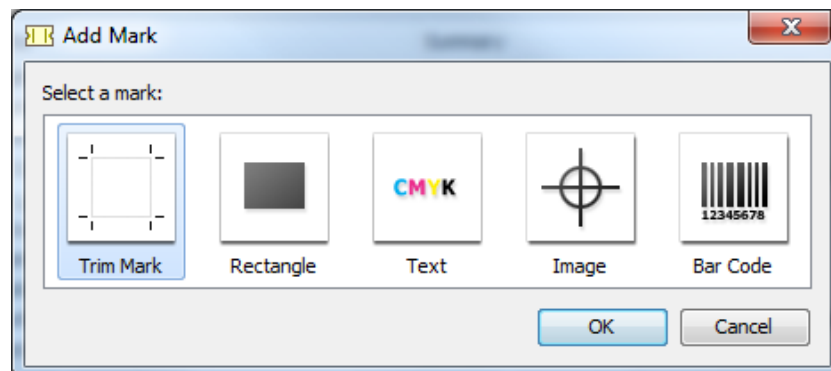


1. In **Add marks to imposition** task, click on **Add**.


Check **Replace existing marks** to remove the existing marks before adding the new marks.

Note: Existing marks here include the marks defined in the JDF template.

The **Add mark** dialog will open up.



2. Click on the **Trim Mark** icon.
The **Trim Mark** window will open up.

3. Click the trim marks button  located adjacent to the four corners around the page image to activate/deactivate the individual marks.

Note: This right hand side of the page image is pointing outwards.

4. Give a **Name** which will be displayed in the **Add marks to Imposition** task panel.
5. Enter **Length** and **Weight** of the individual trim mark.
6. Set **Offset** which is the distance between the trim box of the page and the trim mark.
7. Choose **Color** for the marks. You can choose from:
 - **None** : This is useful when enabling or disabling the trim marks using SmartNames.
 - **Registration**: to choose all inks
 - **Darkest**: choose this when you want a **Color** darker than **Process Black**.
 - **Black or Darkest**
 - **Knockout**

Note: When creep is applied, set trim marks will move depending on the creep settings of the individual pages.

Adding Rectangle Marks

You can create **Rectangle Marks** at:

- **page** level
- **sheet** level
- **section** level.

Most of the settings of adding a **rectangle mark** is similar as in [Adding Text Marks](#) on page 247.

In the rectangle smart option, you can specify the intensity for the chosen color of the mark and its background. This is useful when building control strips using rectangles.

1. In **Add marks to imposition** task, click on **Add**.

Check **Replace existing marks** to remove the existing marks before adding the new marks.

Note: Existing marks here include the marks defined in the JDF template.

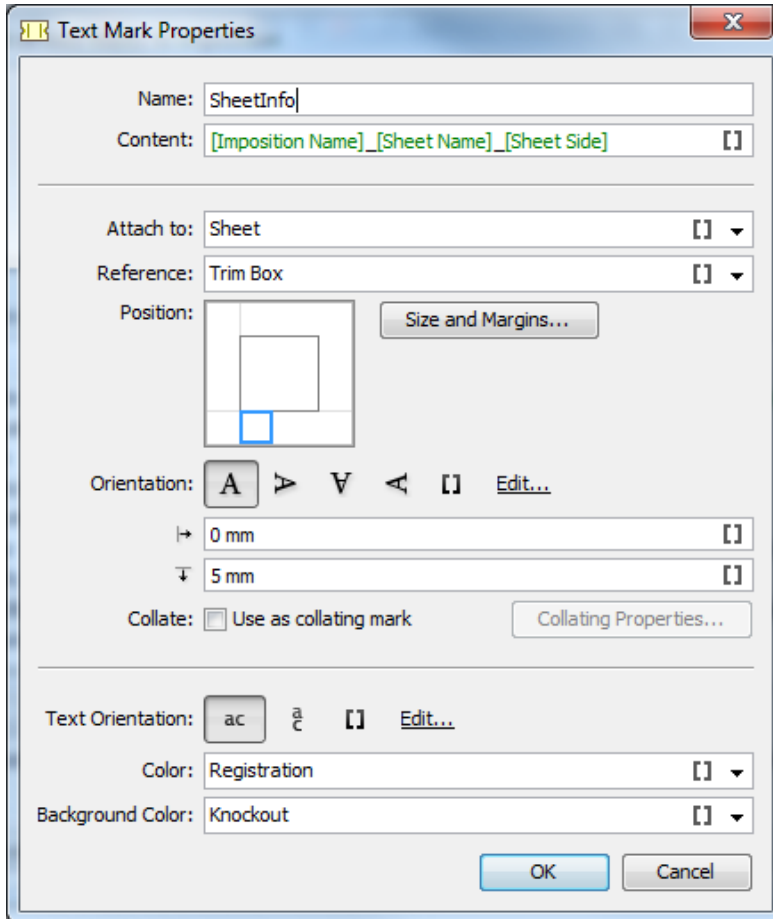
The **Add mark** dialog will open up.

2. Click on **Rectangle** icon.
Rectangle Mark Properties window will open where you can specify the mark settings.
3. Specify the settings as in [Adding Text Marks](#) on page 247.
4. Specify an intensity for the **Color** and Background.

Adding Text Marks

You can create a text mark at:

- **page** level
- **sheet** level
- **section** level.



1. In **Add marks to imposition** task, click on **Add**.

Check **Replace existing marks** to remove the existing marks before adding the new marks.

Note: Existing marks here include the marks defined in the JDF template.

The **Text Mark Properties** dialog will open up.

2. Give a **Name** which will be displayed in the **Add marks to Imposition** task panel.
3. Give the specific **content** by:
 - typing it in
 - using SmartNames
 - by browsing to the file with the intended text.

Use the **InkNames** SmartName in the **content** to identify the individual printing plates. The text mark will be set for each plate and the SmartName will be replaced with the individual Ink name.

4. Select the level of the smart Mark. In the **Attach to** field, select from **Sheet**, **Page** or **Section**.
5. Select the **Reference** to your desired level. You can choose from
 - Paper

- Plate
- Trim Box
- Bleed Box

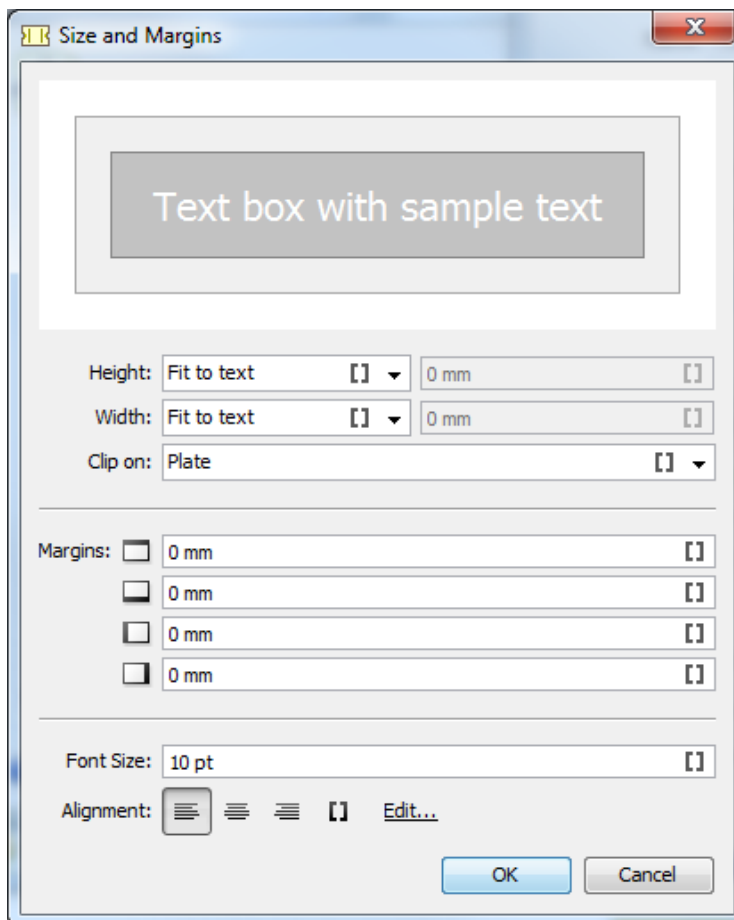
The **reference** is the rectangle you will use as a reference to position your mark.

6. Set the **Position** tool to the precise location of the **reference**.

The **Position** tool allows you to snap the blue rectangle (your mark) relative to the inner gray rectangle (your **reference** rectangle).

7. Click on **Size and Margins** to define the size and margins of the text mark.

You can specify the final dimensions of the mark in this window. The resulting mark will be positioned and rotated using the parameters from the **Text Mark Properties** dialog.



- Specify a **height** and **width** for the text mark. You can either choose **Fixed** and specify the dimensions or choose **Fit to text**.
- Set the **Clip on** level to limit the boundaries of your mark. You can choose to clip the mark at **Plate**, **Paper** or **Mark** levels.
- Select the **Margins**. These are extra margins on top of the text box **height** and **width**.

Note: These margins are taken into account for positioning the mark and are included when creating background color.

- Set **Alignment** to position the text within text box. relative positioning when the SmartName [InkName] is used and multiple text lines are set on top of each other.

The **Size and Margins** window will open up.

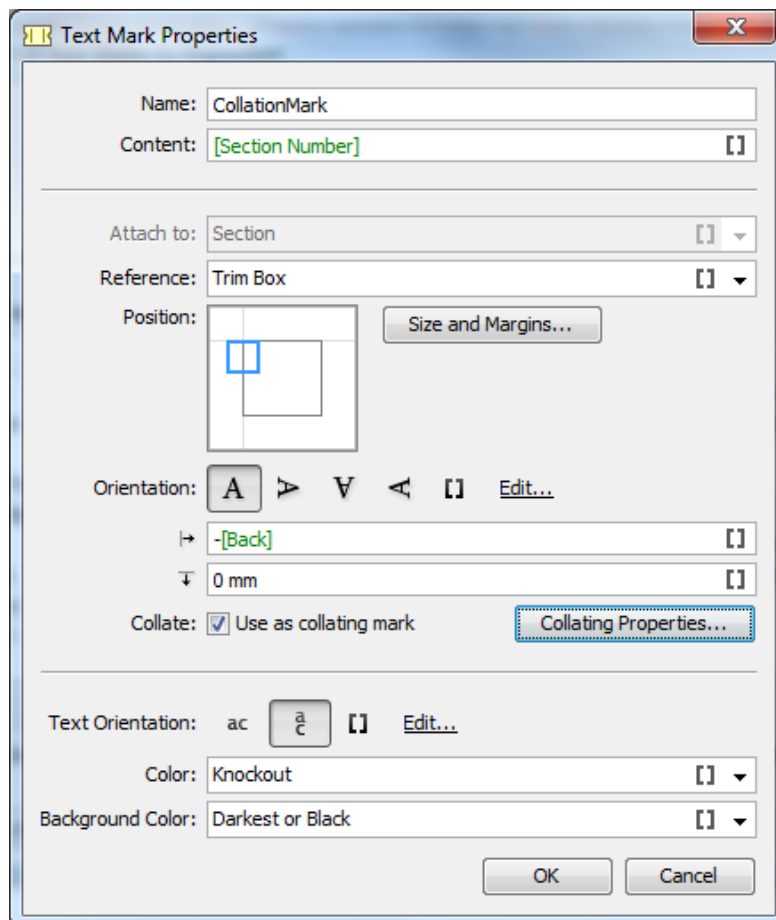
8. Choose the **Orientation** of the text and set the dimensions. Use the orientation mark to rotate your mark.

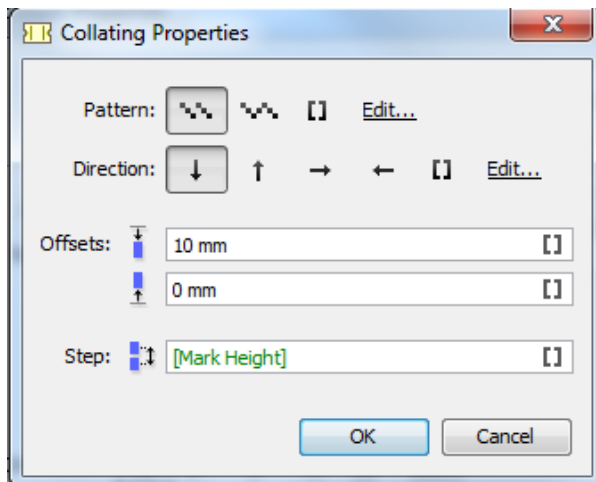
Specify a vertical and horizontal offset after positioning the mark relative to the reference rectangle.

9. If you have chosen **Section** in Step 4, you can use the text mark as a collating mark. Enable **Use as collating mark**.
 - a) Click on **Collating Properties**.
The **Collating Properties** window will open.
 - b) Select the **Pattern** and **Direction** of the collating mark from the symbols given. Alternatively, use SmartNames.
 - c) Specify the **Offset** which is the distance between two collating marks.
 - d) Specify the **Step** which is the height of the collating mark.

Note: You can pick any mark except a trim mark to be used as collating mark.

Use the following settings for a standard collation mark.





- **Position:** Place the top center of the mark attached to the top left of the **Reference** trim box.
- Set `-back` as horizontal offset. Enter a hyphen (- and use SmartName **Back** from the SmartName category **Mark Information**.
- Set a **Fixed** height and width for your mark in the **Size and Margins** window.
- In the **Collating Properties**, use "markheight" as the collating step.
 - Use SmartName **Markheight** from the **Mark Information** category as **Step**
 - Choose the saw tooth **pattern**.
 - Select **down** as stepping **direction**
- Use vertical text **Orientation**.
- Select **Registration** as background color.
- Select **Knockout** as the mark color.

10. Select the **Color** which is the selection of the color for the text in the mark.

Note: This value is ignored if you use the SmartName [InkName] in the **Content** field.

11. Select a **Background Color** which is the color of the rectangle within which the text mark is placed.

Background color is the color of the rectangle surrounding your slug line, you can influence the dimensions of this rectangle using the **Size and Margins** dialog.

Adding Image Marks

You can create image marks at:

- **page** level
- **sheet** level

- **section** level.

Most of the settings of adding a **image mark** are the same as in [Adding Rectangle Marks](#) on page 247.

1. In **Add marks to imposition** task, click on **Add**.

Check **Replace existing marks** to remove the existing marks before adding the new marks.

Note: Existing marks here include the marks defined in the JDF template.

The **Add mark** dialog will open up.

2. Click on **Image** icon.
Image Marks Properties dialog will open up.
3. Specify the settings as in [Adding Text Marks](#) on page 247.
In the **Image Marks Properties** window, you can choose **Mapped** as a **Color**.
4. Specify an intensity for the **Color** and **Background Color**.

Adding Bar Code Marks

You can create **Bar Code marks** at:

- **page** level
- **sheet** level
- **section** level.

1. In **Add marks to imposition** task, click on **Add**.

Check **Replace existing marks** to remove the existing marks before adding the new marks.

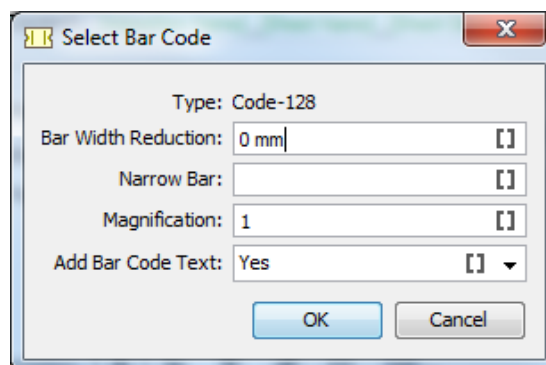
Note: Existing marks here include the marks defined in the JDF template.

The **Add mark** dialog will open up.

2. Click on **Bar code** icon.
Bar Code Marks Properties dialog will open up.
3. Specify most of the settings as in [Adding Text Marks](#) on page 247.

Note: If you use **Registration** as the **Color**, the mark will appear in all the plates you have defined in the plates via **Plates View**.

4. Click on **Select Bar code**. This will result in **Select Bar Code** window.



5. Specify Bar Code Parameters.

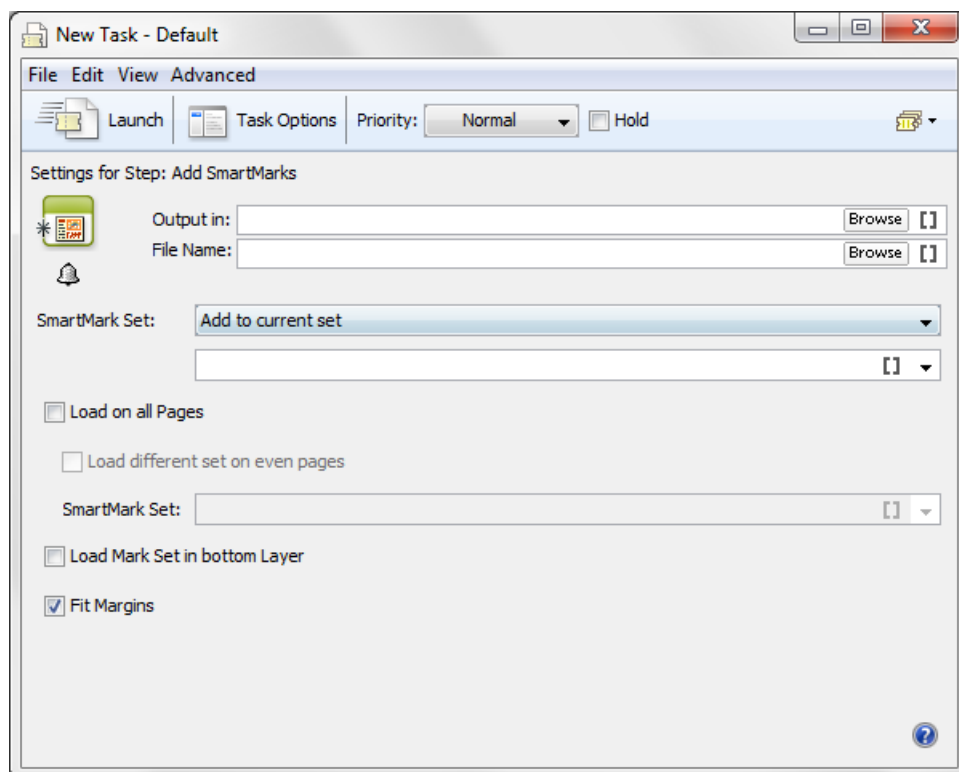
- **Bar Width Reduction:** Use this option to compensate for the ink spreading on the press..
- **Narrow Bar:** Fill this in to change the default size of the bars.
- **Magnification:** Use this to make your barcode bigger or smaller than its default size.
- **Add Bar Code Text:** Select **No** to remove the text located below the bar code.

Note: You cannot choose the Bar Code type. The default is **Code-128**

2.7.4 Add SmartMarks

Add SmartMarks

With the Add SmartMarks task you can apply a SmartMarks set to a PDF file. A SmartMark set is a group of marks that can be created in PackEdge, Plato or i-cut Layout. Consult the PackEdge, Plato or i-cut Layout documentation to learn how to create SmartMarks sets.



Applying a SmartMark set can be done in 2 ways:

- **Add to current set:** will merge the SmartMark set with any existing SmartMarks in the file.
- **Replace current set:** will replace any existing SmartMark sets

Note:

- Since only a normalized PDF file can contain SmartMarks, **Replace current set** is not relevant for native PDF files.
- When applying a SmartMark set on a native PDF file, the resulting file of the task will be native PDF too. Any applied SmartMark set will be converted into normal PDF objects. For more information on normalized versus native PDF you can refer to [the User Guide](#).

Load on all Pages: Selecting this option loads the selected SmartMark set on all pages of a multi page input file.

Load different set on even pages: Selecting this allows you to apply a different SmartMark set on the even pages of a multi-page input file. For example: load a different marks set on the backside of a layout.

Load Mark Set in bottom layer: This option allows you to choose if the marks set should be applied on top of the graphics or if the graphics should be on top of the marks.

Fit Margins: Depending on the setup of your SmartMark set, it can happen that the marks generated are (partially) positioned outside the PDF's media box. Selecting this option will extend the media box to all the visible content outside the trim box.

2.7.5 Add to Pages View

The **Add to Imposition Gallery** task brings IMP files from the Files View into the Pages View. This task will especially be used in workflows or when working with Folder Access Points.

The following workflows work in combination with the **Add to Pages View** ticket.

On a **PDF** file:

- Copy or Move > Normalize PDF File > Add to Pages View
- Normalize PDF File > Add to Pages View
- Right-click the file in the Files View and select 'Add to Page Gallery' from the context menu.

On a **PDFSC** file:

- Copy or Move > Add to Pages View
- Right-click the file in the Files View and select 'Add to Page Gallery' from the context menu.

Note:

This task is only used in page workflows.

2.7.6 Add To Products

This task is documented in the chapter [Products](#) on page 876. Find a direct link [here](#).

2.7.7 Adjust PDF Screening

You can apply rule-based modifications of screening parameters using this task. This helps to limit the changes to specific requirements. For example, you can set just one rule. However the task is

flexible enough to let you make a complex set of rules for changing the screening parameters. See the following example screenshot which has criteria defined to change the screening parameters of a CMYK document (of corresponding separations).

You can preview the **Separation Details** when a file is selected. This will display the results for the specified set of rules in the current file.

This task:

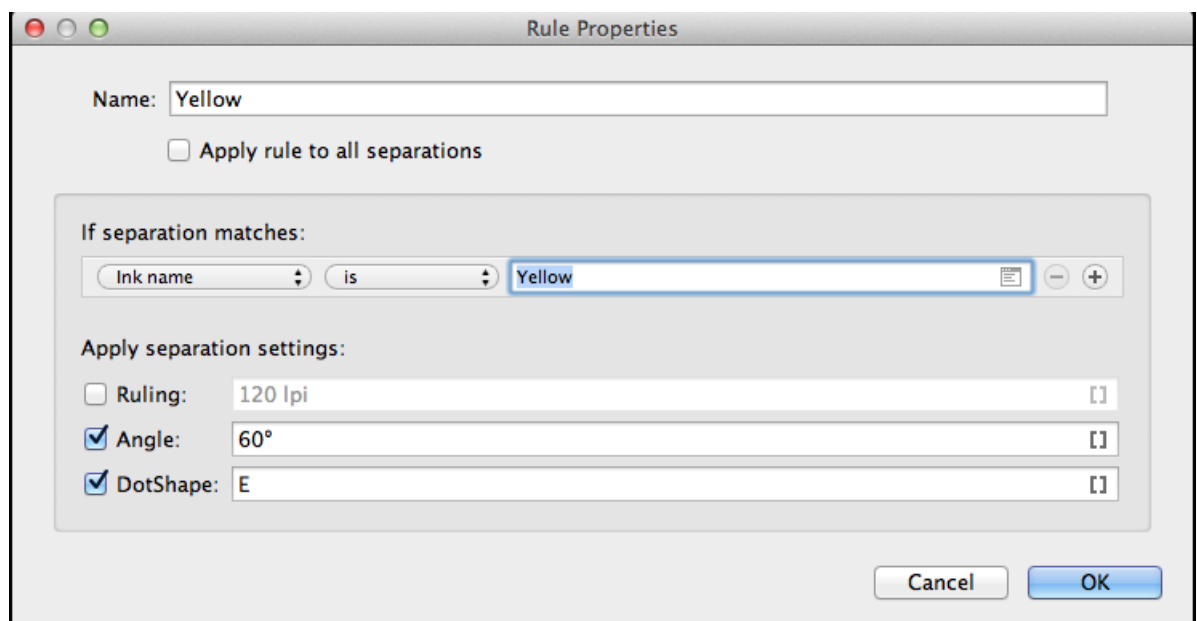
- limits the changes to what is needed. For example, when you only want to change the ruling for all separations.
- can use the criteria like inkbook, printing method or ink type. For example, you can set rule such as : when printing method is **flexo**; set **DotShape** to "F".
- can use SmartNames. You can set SmartNames for screening parameter values and specify the condition accordingly. For example, if the name is "Cyan" or "Magenta" or "Yellow" or "Black"; set **DotShape** to [mydotshape] .
- add complex and elaborate set of rules. For example, in the case of Object Based Screening, use a rule such as : when name is "Cyan" and angle is between "0" and "45" ; set angle to 30 and **DotShape** to "R"

Note:

The screening parameter **DotShape** does not exist in native PDF. Instead there is a **SpotFunction Name**, which can have a limited number of values like **SimpleDot**, **Ellipse**, **Square** which are mapped to the Esko DotShapes "C", "E", "S" respectively. Other **SpotFunctionName** values are mapped to "C".

The task **Normalize AI/Illustrator/PDF File** does not take account of the DotShapes (SpotFunction Name Values) specified in Native PDF files. You need to adjust the PDF Screening parameters during or after normalization.

1. In the task settings, click on **Add**. This will start a dialog where you can enter the rules for screening parameters.



2. Enter a **Name** for the rule. For example, you can type in the same name as the ink name.

3. Select **Apply rule to all separations** if the rule applies to all separations.
4. Select the condition for the criterion under **If separation matches**. You can choose from **Ink name**, **Ink book**, **Ink type**, **Printing method**, **Ruling**, **Angle** and **Dot shape**. Set the condition specific and give related text in the field. If needed, add more criteria by clicking on **+**.

If separation matches:

Ink name	contains	<input type="text"/>	<input type="checkbox"/>	-	+
Ink book	begins with	<input type="text"/>	<input type="checkbox"/>	-	+
Ink type	is	<input type="text"/>	<input type="checkbox"/>	-	+
Ruling	equals	<input type="text"/>	<input type="checkbox"/>	-	+
Ruling	equals	<input type="text"/>	<input type="checkbox"/>	-	+
Dot shape	contains	<input type="text"/>	<input type="checkbox"/>	-	+
Angle	equals	<input type="text"/>	<input type="checkbox"/>	-	+
Printing method	is	<input type="text"/>	<input type="checkbox"/>	-	+
Ink name	contains	<input type="text"/>	<input type="checkbox"/>	-	+





5. Select and specify the **Apply Separation settings** as per the requirement. You can specify **Ruling** and /or **Angle** and /or **DotShape**.
6. Click **OK**.
7. Repeat steps 1-6 to add more rules.

Rules will be applied to matching separations, in the following order:

Rule Name	Ruling	Angle	Dot Shape
Cyan	120 lpi	45°	C
Magenta	120 lpi	30°	C
Yellow	120 lpi	60°	C
Black	120 lpi	90°	C
<all separations>	120 lpi	45°	R

+ -

Separation Details: Preview

Ink Name	Ruling	Angle	Dot Shape
 Cyan	120 lpi	45°	C
 Magenta	120 lpi	30°	C
 Yellow	120 lpi	60°	C
 Black	120 lpi	90°	C

Original screening settings are displayed in grey, modified settings are displayed in black.

? Save As... Manage Public Parameters... Cancel OK

For each of the separations the rules are accounted for in the set order. In the current example (see the previous screenshot), when a separation (Cyan, Ruling of 135 lpi, Angle of 60 degrees, no

dot shape) is present in the document, the first rule is considered. Since it complies with the set conditions, a new angle is set (45 deg). In the remaining set of rules, only the last one is relevant. According to the last rule, the task sets the dot shape and the ruling. but does not modify the already set angle.

If a file is selected, you can preview the applied rules under **Separation Details**. To be able to do this, enable **Preview**.

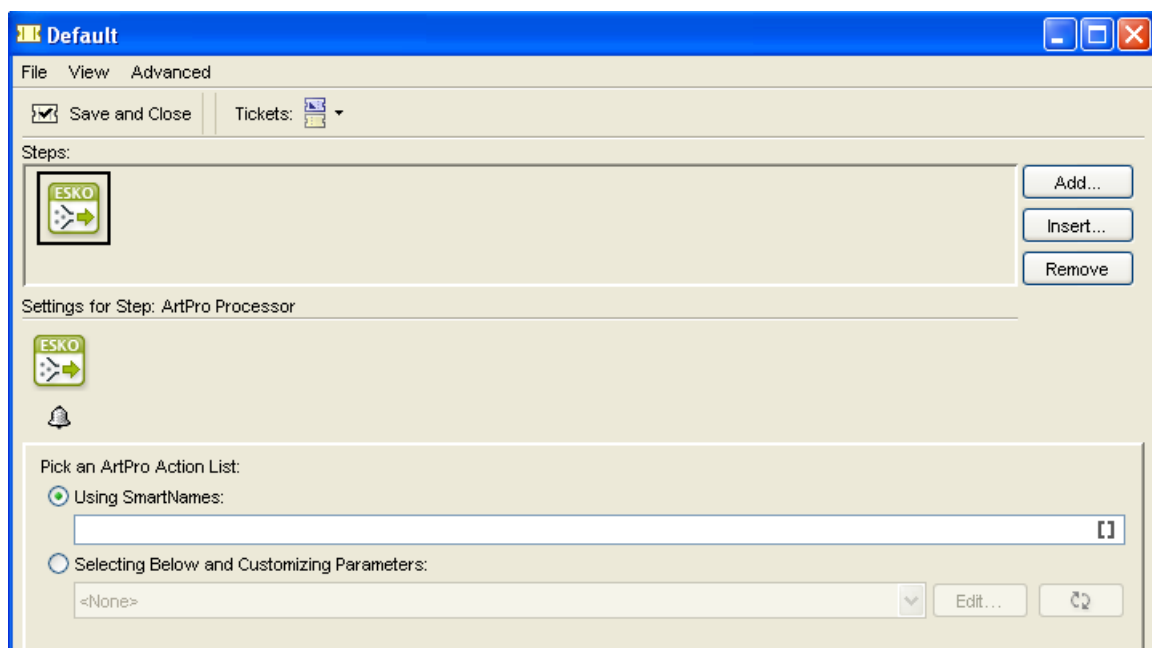
8. Click **OK** to save the task.

2.7.8 Apply ArtPro Action List

In Automation Engine the **Apply ArtPro Action List** task automates tasks in ArtPro and converts Nexus workflows into one or more Automation Engine workflows.

The Processor uses an **ArtPro Action List** to define the list of actions which are to be executed. This file can be created and edited by using the **ArtPro Action List Editor**. An **Edit** button in the **Apply ArtPro Action List** task panel launches the **ArtPro Action List Editor** with the selected Action List.

Complicated NexusProcessor workflows with file dependencies and "Select File" activities are broken into a set of processor tasks with relatively small Action Lists. These tasks are combined in workflows using Router, Select File etc. All Action Lists are placed in a custom folder on the server.



The Processor window consists of a section in which users can choose ArtPro Action List items.

Click **Using SmartNames** radio button to define ArtPro Action List items using SmartNames (please see **Using SmartNames** for more information about SmartNames).

Or click the **Selecting Below and Customizing Parameters** radio button and pick one from the list. Click the **Edit** button to open the selected action list in the **ArtPro Action List Editor**.

Click the **Refresh** button to update the panel. This not only updates the list of Action List items but also updates the parameters of the selected item. However, it does not change any parameter values.

Click the **Refresh** button while holding the **Shift** key to revert the parameter values in the panel to the values that were saved in the Action List.

Using SmartNames in Apply Artpro Action List Task

This task has access to Automation Engine SmartNames linking to Artlink parameters in ArtPro files. Use SmartNames from the following categories:

- File Information
- Global (custom created SmartNames)
- Job Information
- Job barcodes
- Job parameters
- Miscellaneous
- Path information
- Product information
- Product Parameters
- Product Part Parameters

Note:

- The SmartNames are resolved when you open the ArtPro file.
- You cannot use SmartNames for which a task is required to resolve the values (for eg. TaskID). Use SmartNames based on the file location or file name.
- You cannot use a Workflow Parameter directly as an Artlink parameter.

Tip: You can create a Global SmartName based on the Workflow parameter and link this as Artlink Parameter Value. Read more in ArtPro User Guide.

- When an ArtPro file is saved in a Job and linked to a Job Parameter, the information is only kept within that Job. When the same file is saved outside that Job, this Job Parameter info is lost.
- When a Global Product is linked to multiple Jobs and Job SmartNames are used, any of these values can be stored in the file. The ArtPro Processor does not update the parameter to the current Job.

2.7.9 Apply Creep

Use this task to define compensation for 'creep'. When a publication is folded, the bulk of the paper causes the inner pages to extend beyond the outer pages. Creep defines by how much the edges of each inserted spread or signature extend beyond the edges of the one enclosing it.

When used correctly **Apply creep** task will ensure that the pages are correctly aligned after folding. This task works for Perfect Bound as well as Saddle Stitched impositions.

- Set creep for each new Sheet in the case of Perfect Bound impositions.
- Apply creep across all sheets in the case of Saddle Stitched impositions.

You can apply either outer creep or inner creep depending on the bleed.

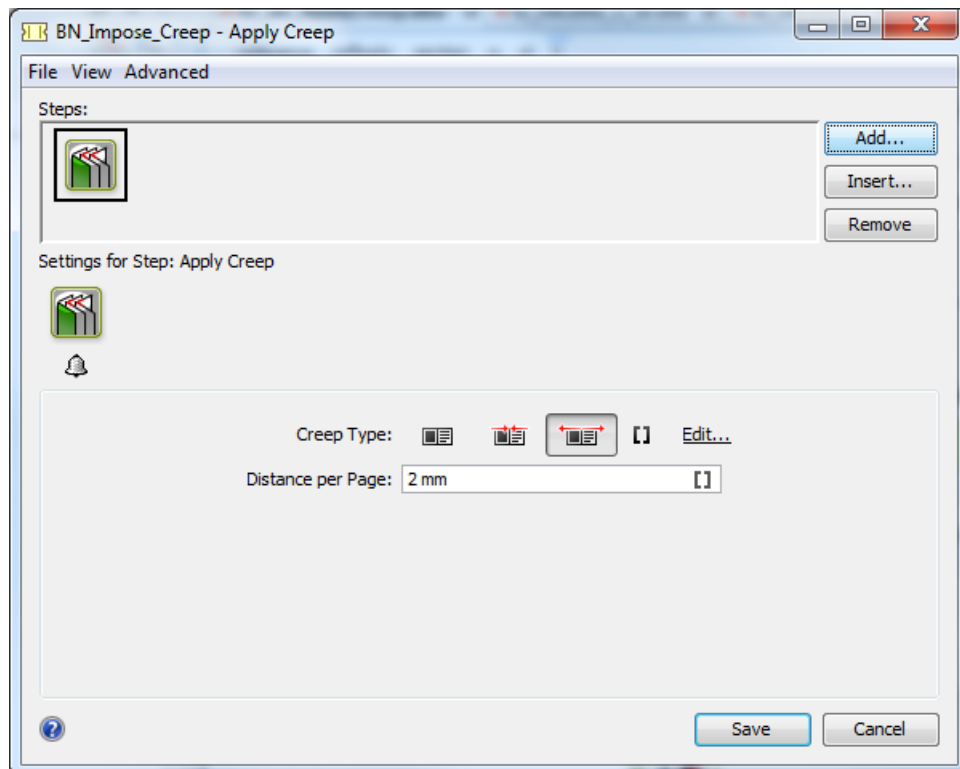
- Use outer creep when there is enough bleed in the back of your pages. This will add bleed to the back of outer pages.
- Use inner creep when there is not enough bleed in the back of your pages. This removes small strips from the back of inner pages for proper alignment.

Creep is applied when exporting an Imposition or Plate to PDF or JDF using the **Export Imposition to PDF file** or **Export Imposition to JDF file**




tasks. You can inspect the results of your Creep settings by using the "Open in Browser" function on an Imposition in the pages view or a Plate in the plates view.

Using the Task

1. Go to **Plates View** and select the Imposition. If you are selecting Pages, go to **Pages View** and select Page Slots.



2. Specify a **Creep Type**. Choose from:

- None 
- Inner 
- Outer 

Alternatively, choose the relevant SmartName.

Note: You need to create SmartNames with values `None`, `Inner` and `Outer`.

You can create exceptions to the top level **Creep Type** for particular Sheets or Pages (like no creep). To do that:

- Select the Page where you want exception to the general **Creep Type**.

- Right-click and start a new **Apply Creep** task.



Attention: If you apply higher level creep after setting lower level creep settings made earlier will be overruled. Therefore, first create the higher level creep and then apply exceptions to lower levels.

3. Set the **Distance per page** depending on the expected shift.

Note: You can calculate it as $\text{Pi}/2$ times the thickness of the paper.

Note: The distance values gets converted to mm. For example: '0.01 in' the ticket will be returned as '0.254 mm'.

2.7.10 Archive Job

This task is documented in the [separate chapter on Jobs](#). Find a direct link [here](#).

2.7.11 Assign Pages To RunList

Use this task to assign individual pages (for example revised pages, new pages) to an existing Imposition or RunList. You can add pages to a designated slot in a folio or index.

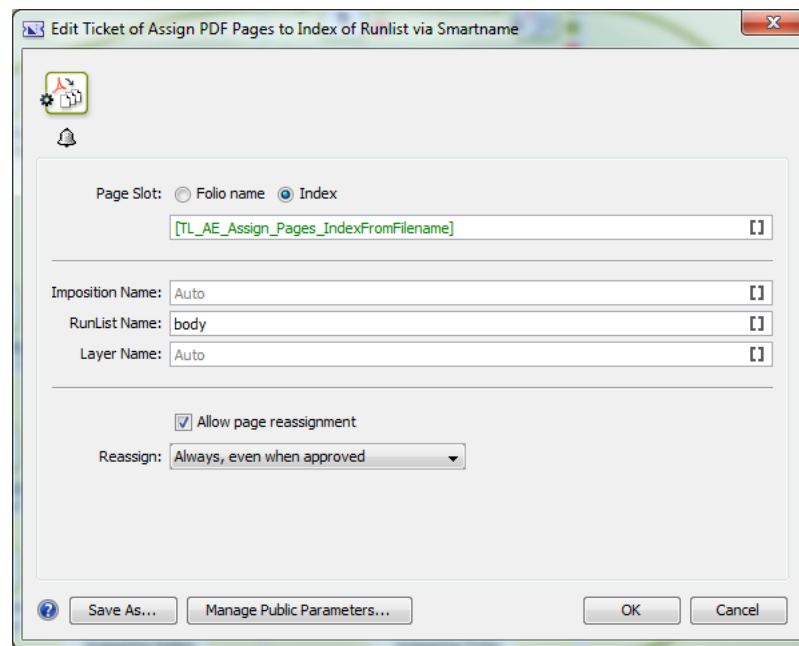
The task uses :

- PDF files
- Pages

Tip: Define SmartNames for Impositions, RunLists, PageSlots and Layers to be used in the **Assign Pages to RunList** task. This task is best suited in a workflow setting to update specific pages. For example, you can assign pages to an existing pagelist in WebCenter. Use this to automate the re-assigning of corrected pages and the corresponding WebCenter approval workflow.

You can either assign a single Page or all the pages using this task.

This task has two output pins for assigned and unassigned page slots when used in a workflow.



Page Slot

- Define the designated slot where the pages should be placed
 - **Folio**
 - **Index**
- Fill in the input field. Use a SmartName to indicate the page to be mapped. When you select a single page, it will be mapped to the corresponding page in the existing RunList.
- When left blank, the page index or folio of the pdf file is mapped onto the corresponding page slot in the runlist.

Imposition: Give the name of the Imposition where you want the specific page or all pages to be reassigned.

If you leave this on **Auto**, all Impositions for the current job or workflow (when you work without job context) will be considered for assigning the page/all pages.

RunList: Give the name of the RunList in this field in which you want the specific page or all pages to be reassigned.

If you leave this on Auto, all RunLists for the current job or workflow (when you work without job context) will be considered.

Layers: If you have Layers, give the name of the Layer in this field in which you want the specific page or all pages to be reassigned.

If you leave this on Auto, all Layers for the current job or workflow (when you work without job context) will be considered.

Allow page Reassignment: Check this to allow to replacement of existing pages and blanks. The task assigns only empty page slots when this toggle is unchecked.

Reassign: Use this control to protect pages against unwanted replacement. Choose from the drop-down options:

- **Always, even when approved.**

- **Always, except when approved**
- **When rejected or approved with warning.**
- **When rejected.**
- **Never.**

Note:

- You cannot assign pages to a locked Imposition.
- Split your spreads before using this task.

2.7.12 Calculate Ink Key Settings (InkPlanner)

The **InkPlanner** ticket is used to create CIP3 files (mainly PPF), which contain all necessary data for the printing and further processing of a job.

Note: The InkPlanner ticket cannot be launched separately. It always needs to be connected to an **Expose to FlexRip** ticket. It cannot be connected to Proof or Approval tickets.

Global settings

Select the **Plate Setting File** (*.ini), which contains specific information for the CIP3 generation, like image position, rotation, compensation, etc.

Select the **Press Configuration File** (*.dat), which contains data about the press and the InkPlanner output.

Note: You need to have created the Plates Setting and Press Configuration files with the CIPress Setup tool before using this ticket.

The name entered in the **Output File Name** will be used for making files created by InkPlanner. This InkPlanner output will be stored in the 'output directory' specified in 'Press Configuration File' (*.dat).

Page Side: select Automatic, Front, or Back from the list.

Note: The option selected here has priority over the page side option selected in the Plate Settings file.

Tip: If an imposition file is processed, the page side option should be set as **Automatic**. The page side settings typical of the imposition file will then be processed.

CIP3

Use the input fields provided to enter administrative data in the output file. This information will be useful when further processing CIP3 files.

- **Job Name:** (required) If nothing is specified, the default job name will be used.
- **Sheet Name:** (optional)
- **JobCode:** (optional) is the code used for the job identification.
- **Make:** (optional) refers to the make of expose device.
- **Artist:** (optional) refers to the person who created the sheet.

- **Customer:** (optional) refers to the target customer.
- **Software:** (optional) refers to the software used to create the sheet.
- **Paper Weight:** (optional) the weight of the paper to be used.
- **Paper Thickness:** (optional) the thickness of the paper to be used.

Note:

If you do not enter anything in the input fields, the corresponding input fields in the CIP3 tab of the Plate Settings tab in the CIPress Setup tool will be checked.

Anything that is entered in there will be taken into account.

2.7.13 Check Dynamic VDP Database

The **Check Dynamic VDP Database** task validates a VDP database before running the **Expand Dynamic VDP Data** task. Although, the same checks are conducted as in the **Expand Dynamic VDP** task, the **Check Dynamic VDP Database** runs faster because it does not generate any output.

This task analyzes all entries from the database using rules stored in Variable Data Template (VDT file).

Note:

When used in a workflow and the check succeeds, the task will produce a database CSV file which can be sent to the next workflow step.

If the check fails, the task delivers a report file listing the detected issues.

This task requires the following

- a CSV database as input,
- a VDT template as mandatory parameter.

Task Settings

- **CSV File Encoding:** Specify the desired character encoding.

Note: The default encoding is **UTF-8**. This is also the recommended encoding.

- **Field Separator:** specify the separator or let the task identify automatically via **Detect Automatically**. The following separators can be used:
 - **,** (Comma)
 - **;** (Semicolon)
 - **Space**
 - **Tab**
- **Field Names:** Field names map data chunks from the database entry to its corresponding VDP template items. You can choose from:
 - **From CSV file:** If this is selected, the task will extract **Field Names** from the first record of CSV file by default.



Attention: The **Field Names** can only contain the following characters:

- lower-case ASCII letters ("a-z")
 - upper-case ASCII letters ("A-Z")
 - digits ("0-9")
 - underscore ("_")
- **From Header file:** When selected, provide a **Header File** containing **Field Names**.
 - **Header File:** You can browse to the CSV Header File or use a SmartName to access the file. The **Header File**:
 - should contain a single line specifying the **Field Names**.
 - should have **Field Names** corresponding to each column present in the CSV file.
 - and the CSV file must use the same delimiter and encoding.
 - **Template:** browse to the Variable Data Template file containing rules to check database validity. The template is created and exported using the Dynamic VDP DeskPack Plug-in for Adobe Illustrator.

2.7.14 Check Job Parameters

This task is documented in the chapter dedicated to [Jobs](#) on page 828. Find a direct link [here](#).

2.7.15 Check Print Rules (PRC)

Print Rule Checker inspects jobs, completely or partially, for potential printing problems and violations of standard printing conventions. Insufficient line thickness, wrong halftone densities, the use of too many inks, and various other production-related issues are brought up instantly. The PRC task will add the results of the checking pass to the Normalized PDF file. The Normalized PDF file can be inspected in detail using the graphic editor. In the editor the user will see a full report of the checking pass and has tools to quickly navigate to the parts of the job where rules have been violated. Additionally, a report file with a list of all the errors that were found in the job can be generated.

General

Ruleset: default.prc

PRC Report

Generate Report

Report name: [File].txt

+[] Browse.....

When rules are violated

Task ends with warning



Task fails

The **Print Rule Checker** consists of one tab.

Select the **PRC Ruleset** from the drop down list. You can choose a standard **Ruleset**, or you can use a customized one. You can customize the **PRC rules** in the graphic editor.

PRC Report. Select the toggle to make sure the system generates a **PRC** report. Enter a name for the report and select the place where you want the report to be stored. You can use SmartNames, learn more on [SmartNames](#) on page 916).

Decide what the state of the task will be when the rules are violated.

- The task ends with a warning .
- The task fails .

2.7.16 Check Product Parameters

This task is documented in the chapter [Products](#) on page 876. Find a direct link [here](#).

2.7.17 Check Technical Inks

The **Check Technical Inks** task checks whether the input file (Normalized PDF, Plato File or PDF File) contains technical objects in knockout.

When technical objects are in knockout, the task will fail or end with a warning according to the task settings.

Read more in Esko white paper Working with Technical Inks in Suite 12 assembly 3.

2.7.18 Compare PDF

Use this task to make a comparison between two PDF files. You can define what you want to compare and how you want the comparison results to be viewed and saved.

Note: You can use SmartNames in most fields of this dialog. For information on SmartNames, see [SmartNames](#) on page 916.

1. In the **Compare with** field, browse for the reference file with which you want to compare your PDF file.
2. Select the **Compare Region** or media box within which you want to make the comparison. This allows you to restrict the comparison to the contents of one of the media boxes.
3. Set the **Compare Resolution**. This is the resolution you want to RIP the files to before comparing them (TIFF files are created per separation and those are compared).
This value should be between 50 and 300 ppi
4. Set the **Noise Tolerance**. The **Compare Resolution** defines in what *detail* you compare the files. This **Noise Tolerance** however defines how *strict* you want to compare. What do you consider as a difference? If you do not want the most strict form, you can here define a different **Noise Tolerance**. The distance you set here is the maximum (horizontal or vertical) distance between isolated pixels for them to be ignored. An isolated pixel is a pixel that is different in both files but

that has all surrounding pixels equal in both files. The higher you put the value, the more *tolerant you are against noise*: the more you allow noise, the less strict the comparison will be. The default is '0 mm', no differences allowed (most strict).

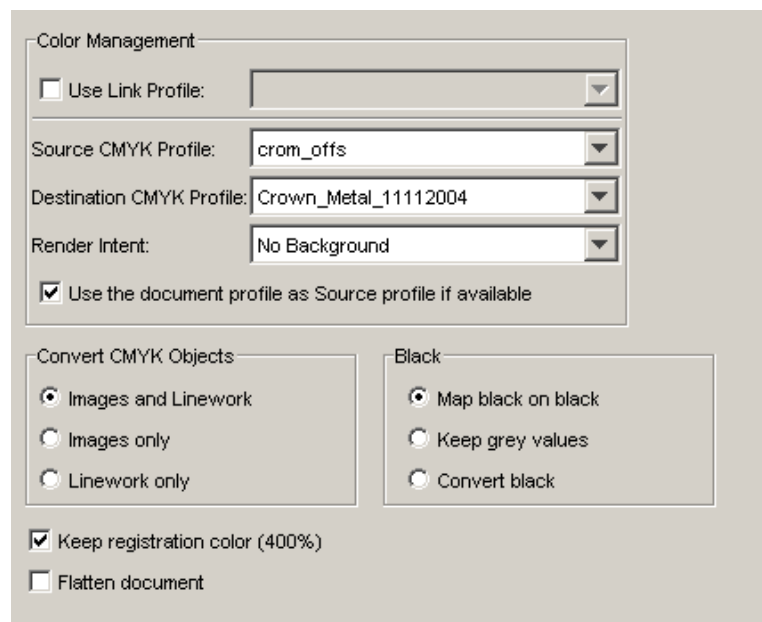
For example: The comparison shows 2 isolated pixels that are 10 mm apart and also 2 isolated pixels that are only 2 mm apart. If you now set the **Noise Tolerance** to '5 mm', then the set of isolated pixels that are 10 mm apart will be ignored in the comparison.

5. If necessary, add the separation(s) that should be ignored. For example for multilingual magazines you may want to ignore the separation(s) that contain(s) text.
6. In the **Output in** field browse for the folder where you want to save the file containing the differences between the two PDF files.
7. Select the **Create Differences layer** option to save the differences between the two files in a separate layer. Then browse for a Layer file or make one automatically using SmartNames.

Note: If page boxes are different and/or separations do not match, the task will report a mismatch but it will not create a difference layer.

8. Select the **Report differences to file** option to save the differences in a separate XML report. When files don't match, this report contains the coordinates of the mismatched pixels for each separation. Then browse for a report file or make one automatically using SmartNames.

2.7.19 Convert CMYK Colors



This task allows you to convert your file from one CMYK Profile to another.

Use Link Profile Link profiles describe a transformation from 1 device dependent color space to another. This is actually a concatenation of a forward transformation of 1 profile and a backward transformation of another profile. This means that all other options will be grayed out as the Link Profile takes complete care of the color management.

Source CMYK Profile Select the source CMYK Profile.

Destination CMYK Profile Select the source CMYK Profile

Render intent

- **No Background:** Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- **Perceptual:** This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.
- **Absolute (Background):** Use this option if you want background simulation and you want an exact color match, including the paper simulation.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

Use the document profile as source profile if available If a document profile is defined, then that one will be used as a source profile. The one chosen from the Source Profile dropdown list will be overruled. If no document profile is present while processing, the default source profile will be used.

Convert CMYK Objects. Choose what objects need to be converted.

- Images and Linework (everything)
- Images only
- Linework only

Black

- **Map black on black:** All colors will be converted, but the conversion will treat the black separation separately. As a result, the black separation will only slightly change (to compensate for possible differences in dot gain). Colors that contain only black, will remain black-only. Use this setting if you want to convert a file from one color profile to another, but you do not want to change the UCR/GCR level of the file.
- **Keep Gray:** Gray colors (colors that only have a certain percentage of black and no cyan, magenta or yellow) will be kept unchanged. All colors that contain some cyan, magenta or yellow are converted according to the profile. Use this setting if you want to use the UCR/GCR settings from the destination profile, but you would like to keep black text and other gray objects untouched.
- **Convert: black:** All CMYK colors will be converted according to the destination profile. The output file will have the black-generation and UCR/GCR settings from the destination profile. Because of this, the black separation can possibly change a lot. Use this setting if you are using this task to change the UCR/GCR level of a file. Note that black text might get converted into a rich composite black, depending on the destination profile.

Keep Registration Color In case you use marks in your file, and if you do not want them to be converted, then check this toggle.

Flatten document Select to flatten the entire document. Objects may be flattened that should not have been resolved, and in some occasions new CTs are generated during resolving. However, flattening may eliminate some unexpected results in color conversions.

2.7.20 Convert Colors with Equinox

The **Convert Colors with Equinox** task enables you to convert your PDF files' colors using **Equinox**.

Equinox is Esko's award winning **extended gamut printing** solution. Extended gamut printing is using a standardized set of 5, 6 or 7 inks for printing, instead of CMYK plus different spot colors for each job.

Equinox functionality is available in Automation Engine, *ArtPro*, *PackEdge*, the *Color Engine Pilot* and as a plug-in for Adobe Photoshop.

For more information about Equinox, please see the [the Equinox product page](#) and [the Equinox documentation](#).

Use this task to convert jobs' colors for extended gamut printing in the following cases:

- You don't have ArtPro or PackEdge and work with regular PDF files.

With this task, you don't need to convert your file to an ArtPro file or a Normalized PDF file before conversion, but you can get the same results as if you did.

- You do have ArtPro or PackEdge, but you want to automate the conversion.

For example, you can:

1. First experiment with Equinox conversion in ArtPro or PackEdge, and make small adjustments to the conversion until it is exactly the way you want it.
2. Then use the resulting color strategy in the **Convert Colors with Equinox** task to automate the conversion of all your job's variants, or "SKUs" (Stock Keeping Units), for example different languages and sizes.

This task takes PDF and Normalized PDF files as input. When working with regular PDFs, it will automatically normalize them using the **Normalize ticket** set in the **Configure** tool (see [Processing Preferences](#) on page 674).

For more information about Normalized PDFs and automatic normalization, see [PDF Normalization in Automation Engine](#) on page 115.

Note:

This task:

- flattens all the complex blend modes in your file, to make sure the colors are converted accurately.
- keeps your file's metadata (for example CAD information) intact.

Parameters

Tip: You can make all of this task's parameters public if desired.

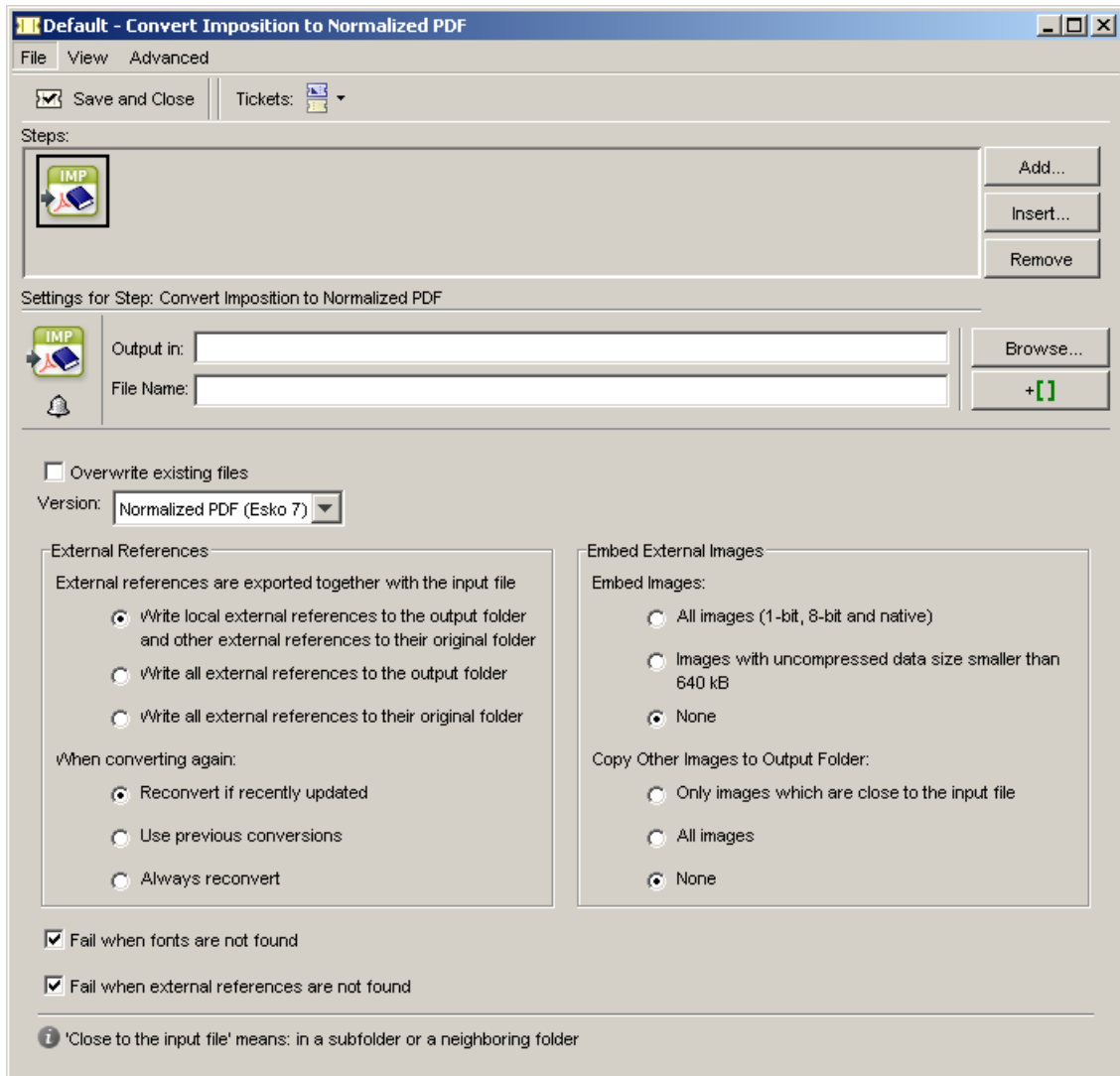
1. Select the **Color Strategy** you want to use for the conversion (you can use SmartNames).
This must be an **Equinox Color Strategy** that you have created in the Color Engine Pilot.
2. Select **Hide Technical Inks** if you want to keep technical inks out of the conversion.
3. Select which types of objects you want to **Convert**:
 - **Images and Linework,**
 - **Linework only,**

- **Images only.**

2.7.21 Convert IMP File to Normalized PDF

Use this task to convert your GRS impositions to Normalized PDF impositions.

This convert the GRS imposition's template, marks and content to Normalized PDF, and adapts the imposition around it.



Overwrite existing files

This option determines what the task will do if it finds some of the imposition's files (mark files for example) already converted to Normalized PDF.

- Select the option to overwrite the existing Normalized PDF files with the new ones generated by the task.
- Leave it deselected to keep the existing Normalized PDF files.

Note: The remaining options are similar to those of the **Export to Normalized PDF File** task (see [Export to Normalized PDF File](#) on page 329).

2.7.22 Convert Selection

You can use this task in a workflow to automate file conversion. For example, you can convert some of the layers from an Imposition from the **Pages** view to a PDF format for proofing before proceeding with the next task.

You can convert formats such as

- PDF
- Page
- Runlist
- Layer
- PageSlot
- Imposition
- Plate

into any of these output formats

- PDF
- Page
- Runlist
- Layer
- PageSlot
- Imposition
- Plate

This task is useful in a workflow when you need a certain file format to continue the flow.

You can choose the output format of the task in the **Output Format** field.



2.7.23 Copy or Move File

This task copies or moves the input file or marked files to a new location.

To mark a file, please insert the [Mark File](#) workflow control in your workflow. Learn more in [Mark File and Select Marked File](#) on page 1029.

Check the box if you want the task to **fail when the file to be copied or moved already exists in the output folder**.

You can also choose to **not take over the Extension of the Input file in the Output file**.

Tip: The **Copy or Move File** task can also be launched on folders.

2.7.24 Create CAD Sheet

The **Create CAD Sheet** task allows you to fully automate your Step and Repeat process based on information from the business system (MIS, ERP). No more manual intervention is then required to produce the plates or sheets. The **Create CAD Sheet** task places graphic files on the stations defined in an MFG, DXF or CFF2 CAD file. For the graphic files, the task can use an XML file, or one or more PDF files.

If the input file is an XML file, this XML file contains all information on what file(s) to use, the quantity, bleed, etc. to build the Plate / Sheet.

If the input file is a PDF file, this PDF is placed on all stations. If the input is multiple PDF files, the files will be distributed equally.

See also [Distributing the one up graphics on the stations](#) on page 275 and [Positioning a one up graphic onto a station](#) on page 275

XML input

The task looks for a <JOB> element in the XML file.

```
<?xml version="1.0" encoding="UTF-8"?>
- <JOB>
  <FileName>\\homer\beou_container\beou_CAD_demo\Input\demo\anybox.pdf</FileName>
  <Quantity>7</Quantity>
  <Bleed>2.0</Bleed>
</JOB>
```

It's also possible to combine multiple graphics in the XML file. In that case, multiple <JOB> elements are gathered in one <JOBS> element

```
<?xml version="1.0" encoding="UTF-8"?>
- <JOBS>
  - <JOB>
    <FileName>\\homer\beou_container\beou_CAD_demo\Input\demo\otherbox.pdf</FileName>
    <Quantity>2</Quantity>
    <Bleed>2.0</Bleed>
    <Page number="1"/>
  </JOB>
  - <JOB>
    <FileName>\\homer\beou_container\beou_CAD_demo\Input\demo\anybox.pdf</FileName>
    <Quantity>7</Quantity>
    <Bleed>2.0</Bleed>
    <Page number="1"/>
  </JOB>
</JOBS>
```

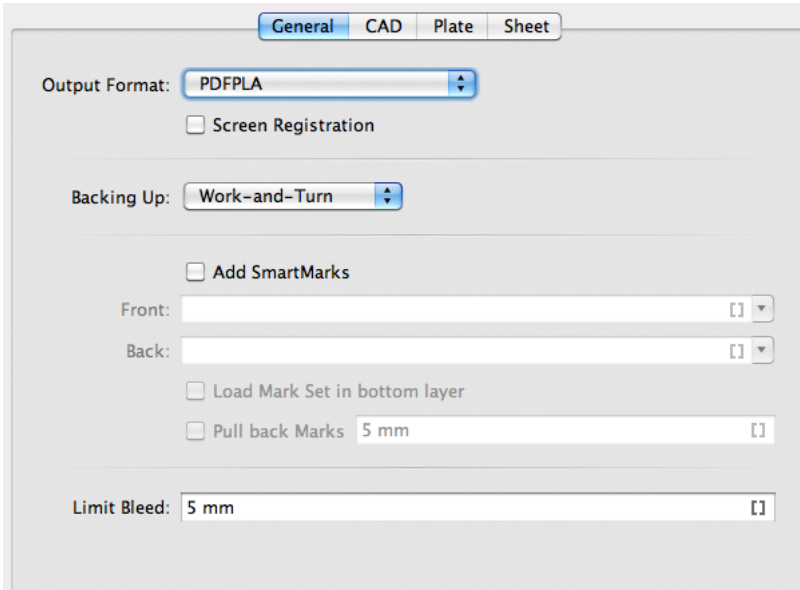
A <JOBS> element can contain the following sub elements:

XML Element	Description
Filename (mandatory)	URL of the graphics
Quantity (mandatory)	Number of times the graphics in FileName needs to be placed. Set the value to 'Maximum' to fill the CAD file with as many graphics as possible.
DieShape	Ink name of the die line of the graphics.
Bleed	Setting the bleed mask in mm to be applied starting from the die line.

XML Element	Description
Page@number	When no page number is defined, the first page of a multipage PDF is used. Set the page number if you want to use a different page.

Ticket settings

General



The screenshot shows the 'General' settings panel. At the top, there are tabs for 'General', 'CAD', 'Plate', and 'Sheet'. The 'General' tab is selected. Below the tabs, the settings are as follows:

- Output Format:** A dropdown menu set to 'PDFPLA'.
- Screen Registration:** An unchecked checkbox.
- Backing Up:** A dropdown menu set to 'Work-and-Turn'.
- Add SmartMarks:** An unchecked checkbox.
- Front:** A dropdown menu.
- Back:** A dropdown menu.
- Load Mark Set in bottom layer:** An unchecked checkbox.
- Pull back Marks:** A text input field set to '5 mm'.
- Limit Bleed:** A text input field set to '5 mm'.

The **Output Format** can be PDFPLA, Normalized PDF or PDF.

The **Screen Registration** parameter allows to choose if the RIP should reset the screening origin for every one-up or keep the same screening origin for the complete document

Use the **Backing Up** when working with a back and front. You can choose to Turn (rotate over a vertical axis) or Tumble (rotate over a horizontal axis)

Note: In order to make a CAD sheet with a Front and a Back, multi page PDF files are needed, the first page to be placed on the front, the second on the back.

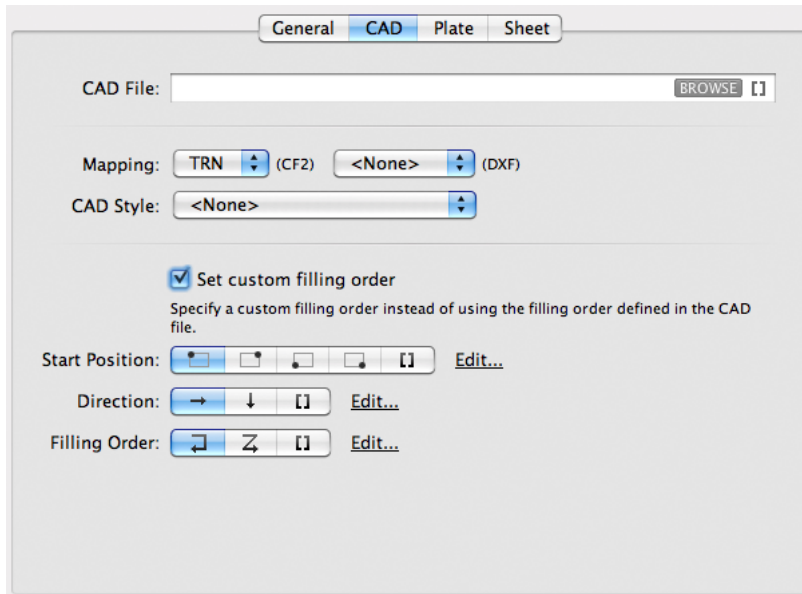
Add **SmartMarks** by selecting a SmartMark set from the drop down list. You can also use SmartNames.

Check the '**Load Mark Set in bottom layer**' box if you want to make sure the marks are positioned in the bottom layer, to make them appear 'behind' the graphics.

Use **Pull back Marks** to keep marks away from the graphics. You create a marks free zone starting from the die line over the given distance. This is only available if Load Mark Set in bottom layer is enabled.

The **Limit Bleed** toggle: the bleed can be limited to a specific distance. The bleed mask is created by spreading the die shape.

CAD



Both **Mapping** and **CAD Style** are options you can define in the Editors.

Mapping allows you to select a set of Mapping rules. Mapping rules allow to map DXF layer names, DXF line styles and DXF color numbers onto line types (ArtiosCAD style). Use the mapping rules to indicate which DXF elements will be imported as cut, crease or bleed lines. The mapping rules are scanned from top to bottom, as soon as a DXF element matches a rule, the action corresponding with this rule (ignore or import as cut, crease, bleed) will become effective.

A **CAD Style** changes the appearance of the imported CAD lines.

By default, the filling order as defined in the CAD file is used. However, by setting **Set custom filling order**, you can specify a custom filling order.

The filling order is defined by selecting a start position, direction and filling order. For each of those, you can also use a SmartName.

The sequence of the designs is determined by taking the center points of the designs and sorting these in rows and columns. Keep in mind that designs with different sizes or layouts with staggered designs can result in more rows or columns than expected.

Plate

General CAD **Plate** Sheet

Plate Size: ↔ 1066.8 mm []
 ↓ 812.8 mm []

Sheet Position on Plate:

Sheet Offset: ↳ 0 mm []
 ⊕ 0 mm []

The sheet size can be automatically extracted from the CAD template but neither Artios MFG nor CFF2 have a notion of plate. As a result the plate size and position of the sheet on the plate have to be specified in the Plate tab.

Plate Size: enter -if necessary- the plate's horizontal and vertical dimensions.

Position the sheet on the plate by dragging and dropping the blue square (the sheet) to the right position on the grey square (the plate).

Define horizontal and vertical **Sheet Offset**

Tip: All edit fields are SmartName-enabled. Make sure the SmartName returns a value according to the field, for example a SmartName in the Plate Size field should return a number

Sheet

General CAD Plate **Sheet**

Sheet Size: Set custom sheet size
 Specify a custom sheet size instead of using the sheet size defined in the CAD file.
 ↔ 1066.8 mm []
 ↓ 812.8 mm []

Blank Position: Set custom position
 Specify a custom position instead of using the position defined in the CAD file.

Blank Position on Sheet:

Blank Offset: ↳ 0 mm []
 ⊕ 0 mm []

If you want to use a custom sheet size instead of the sheet size defined in the CAD file, enable **Set custom sheet size** and enter the dimensions for the sheet.

You can also specify a custom **Blank Position** instead of using the position defined in the CAD file by enabling **Set custom position**. Using the Blank position, you can position the bounding box of all cut lines on the sheet.

Position the Blank on the sheet by dragging and dropping the blue square (the sheet) to the right position on the grey square (the sheet).

Define horizontal and vertical **Blank Offset**

Positioning a one up graphic onto a station

To decide how to position a PDF onto the CAD file, the task will try the following options (in order):

1. Check if the input PDF contains a die cut object, i.e. a linked ARD file. If the name of this ARD file matches the name of the design in the MFG file, the graphics will be placed

Note: This option is only available for Normalized PDF files.

2. Position the trim box of the PDF on the bounding box of the cut and crease lines. If the trim box and bounding box match, the graphic can be placed.
3. Position the die line of the PDF defined in the XML file, on the bounding box of the cut and crease lines. If it matches, the graphic can be placed

Note: This option is only available for XML input.

If the PDF trim box or die line bounding box does not match any of the bounding boxes of the design in the CAD file, then the file will not be processed and an error will be given.

Distributing the one up graphics on the stations

Distributing the one up graphics on the stations defines what PDF goes on what position in the CAD file.

If the input file of the task is **XML**, the task will look for an element **Quantity**, defining the amount of stations to be filled with the specified graphic, for example `<Quantity>10</quantity>`.

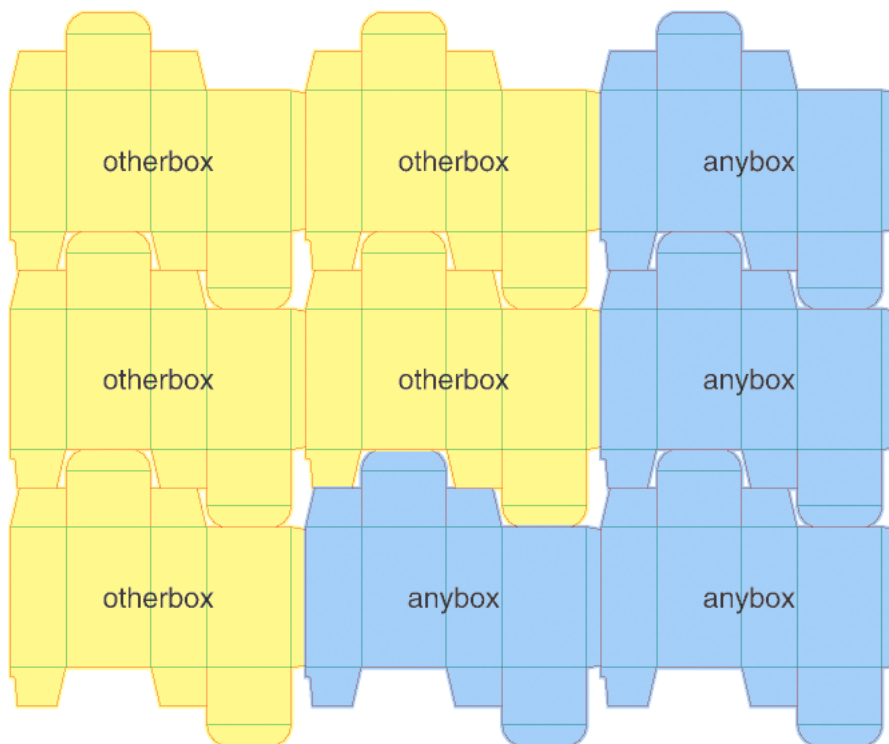
Note: If the total number of quantities doesn't match the number of stations, the task will stop with an error.

When using the element `<quantity>Maximum</quantity>`, the task will place the referred PDF file as many times as possible.

If more than one PDF file has the quantity element set to Maximum, the task will equally distribute the referred PDF files. You can also make a combination of files with a fixed quantity, and one or more files set to Maximum.

If the input for the task is PDF, the task will consider all PDFs to be set to "Maximum" quantity, so the provided input files will be distributed evenly onto the provided CAD file.

If the number of available stations cannot be divided by the number of input files (for example 9 stations and 2 PDF files), the task will automatically distribute the remaining stations.



Note: When the CAD file contains different designs (for example different type of boxes), the graphics will only be distributed on the corresponding (fitting) designs (see [Positioning a one up graphic onto a station](#) on page 275)

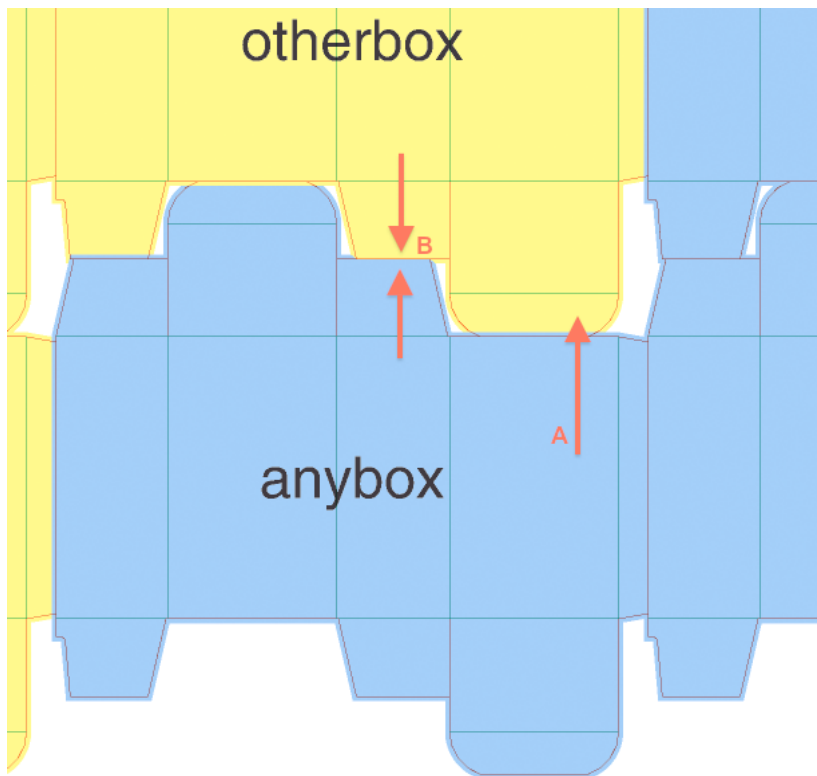
Bleed and overlap

When the one up contains bleed, this bleed will be used around the diecut, limited by the **Limit Bleed** value.

Note: Everything outside the dieline is considered to be bleed. Bleed zones without content will knockout when overlapping. Overlaps can only work well if a correct bleed limit is applied.

If the bleed of two stations would overlap, you can have the following situations:

- There is a gap between the panels of the two stations. In this case, centerline overlapping is applied, so the bleeds from both neighboring stations touch on the center-line of the gap.
- The station panels are touching:
 - the biggest panel will overlap the smaller panel (situation A in the example below)
 - if the panels are the same size, center-line overlapping will apply (example B in the example below)



Note: When deciding if a station is touching another station, minor tolerances are taken into account to cope with small inaccuracies in the diecut file.

2.7.25 Create Design (DesignWizard)

Note: This Task requires a **PackEdge** editor

This task is intended for the automatic generation of template-based designs with different content. The template is made in PackEdge.

The objects that vary between the different versions are generated by SmartMarks.

DesignWizard will simply regenerate the SmartMarks using different data sets. These data sets can come as field in a database (accessible from the Automation Engine server through ODBC) **OR** as fields in XML files.

Tip: SmartName support for path to XML file makes it possible to create more generic tickets.

Settings for Step: Create Design (DesignWizard)

Output in: [Job URL]/Variations_[Date] Browse...

File Name: [jp.Product_Name]_drupa +[]

SmartMarks: mark1_small +[]

XML File: [Job URL]/FunkyJuices/master/Funky_Juices.xml +[] Browse...



These labels vary in color, text and bar code. External references are also varied (images, logos). The task gets the correct input from an XML file:

```

- <dataroot>
  - <Berri>
    <Design_Number>799477</Design_Number>
    <Product_Name>apple juice</Product_Name>
    <Litre_Size>2 LITRES</Litre_Size>
    <B_Number>B21-j136</B_Number>
  - <Ingredients>
    INGREDIENTS: RECONSTITUTED APPLE JUICE (99.9%), FOOD ACID (330),
    FLAVOUR PRESERVATIVES (202, 211), VITAMIN C (300), NATURAL
    COLOUR (150d).
  - <Packed_In>
    NO ARTIFICIAL COLOURS & NO ARTIFICIAL FLAVOURS PACKED IN
    AUSTRALIA FROM IMPORTED AND AUSTRALIAN FRUIT JUICES.
    CONSUME WITHIN 5-7 DAYS OF OPENING
  - <Servings>10</Servings>
  - <Serving_Size>200mL</Serving_Size>
  - <Energy_Per_Serve>360kJ</Energy_Per_Serve>
  - <Energy_Per_100>180kJ</Energy_Per_100>
  - <Protein_Per_Serve>Less than 1g</Protein_Per_Serve>
  - <Protein_Per_100>Less than 1g</Protein_Per_100>
  - <Fat_Total_Per_Serve>Less than 1g</Fat_Total_Per_Serve>
  - <Fat_Total_Per_100>Less than 1g</Fat_Total_Per_100>
  - <Fat_Saturated_Per_Serve>0g</Fat_Saturated_Per_Serve>
  - <Fat_Saturated_Per_100>0g</Fat_Saturated_Per_100>
  - <Carbs_Total_Per_Serve>21.2g</Carbs_Total_Per_Serve>
  - <Carbs_Total_Per_100>10.6g</Carbs_Total_Per_100>
  - <Carbs_Sugar_Per_Serve>20.6g</Carbs_Sugar_Per_Serve>
  - <Carbs_Sugar_Per_100>10.3g</Carbs_Sugar_Per_100>
  - <Dietary_Fibre_Per_Serve>Less than 1g</Dietary_Fibre_Per_Serve>
  - <Dietary_Fibre_Per_100>Less than 1g</Dietary_Fibre_Per_100>
  - <Sodium_Per_Serve>8mg</Sodium_Per_Serve>
  - <Sodium_Per_100>4mg</Sodium_Per_100>

```

Using SmartNames in a DesignWizard template

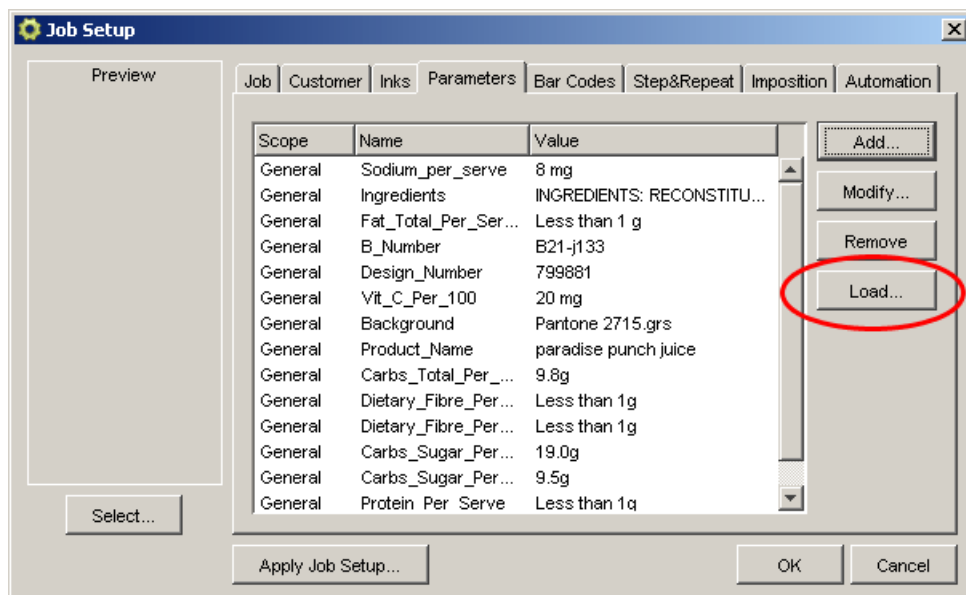
DesignWizard assumes the texts are available as either:

- fields in a database (accessible for the Automation Engine server through ODBC)
- fields in XML file(s)

In the case of a live connection to a database, you need to define a set of SmartNames through the Automation Engine *Configure* panel.

In the case of XML files, the XML file is expected to have a similar format as when saving a database query into an XML file. This means the data can either be split over 1 file per instance (i.e. each XML file contains all the data for 1 flavor) or there can be multiple sets in 1 XML file.

Taking such an XML file, you can easily convert all the tags from this XML file into job parameters. This is done via the **Load button** in the Parameters tab of the *Job Setup* dialog:



Once this is done, all these job parameters are also available as SmartNames (inside this job).

As a result, building the templates is in this case similar to using global SmartNames (defined in the Automation Engine Configure panel).

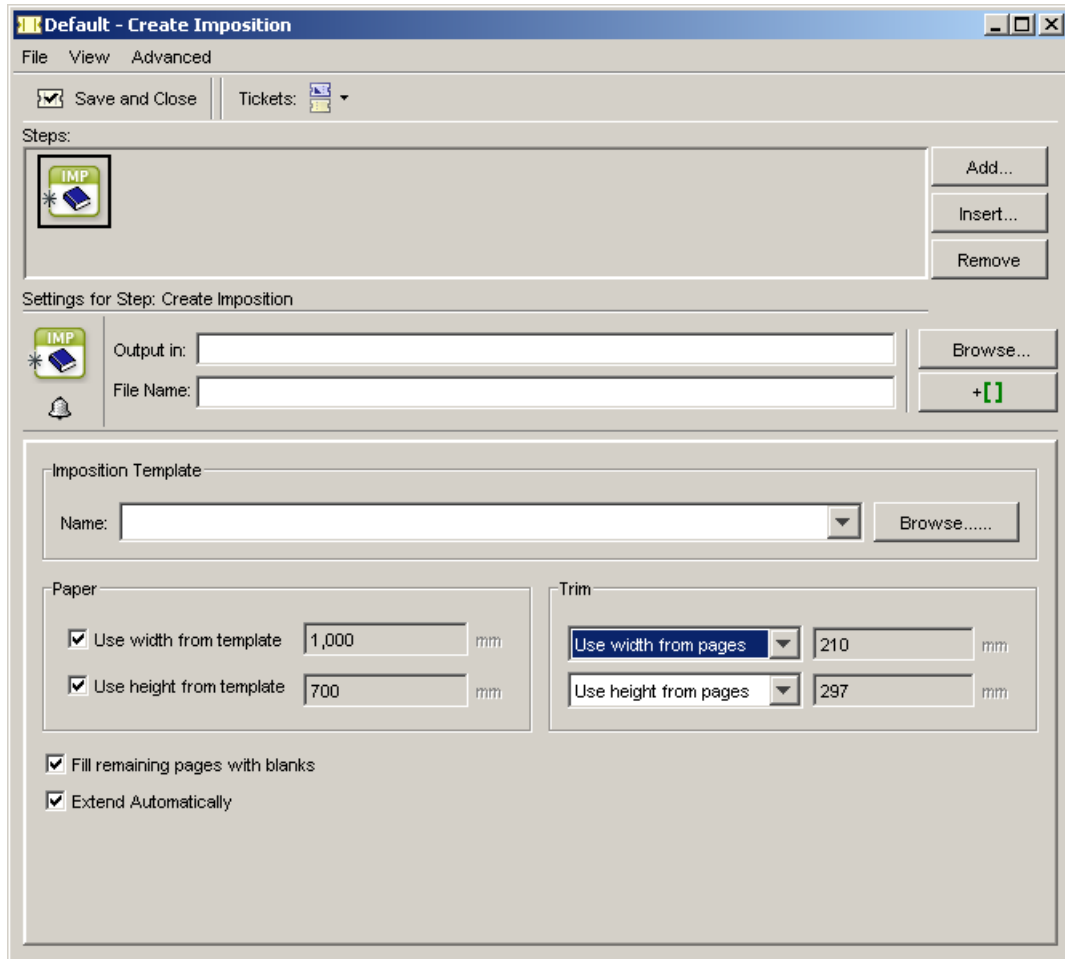
2.7.26 Create IMP File

The **Create Imposition** ticket makes it possible to create an imposition file simply by selecting an imposition template.

The **pages** selected in the Files View will be automatically assigned to the template.

In the Pages View, the input will be a **book** to which you attach an imposition template.

Settings



- Select an **Imposition Template** you previously created in FastImpose. The pages will automatically be assigned to that template.
An imposition file can also be used as a template. The pages assigned to that imposition will then be overwritten.
- **Paper:** you can either use the width / height from the template you selected, or specify an adapted paper size.
- **Trim:** you can either use the width / height from the pages, from the template, or specify an adapted trim size.
- **Fill remaining pages with blanks:** In case not enough pages have been selected to fill all pages of the template, the empty pages at the end will be filled with blank pages if this option has been selected. The task will not shrink the template.
- **Extend Automatically:** select this option to automatically generate extra sheets in the template if it doesn't have enough sheets for your imposition.

Note:

To do this, your template must have **at least one master sheet**, and **no more than one exception sheet**.

The task will generate the number of extra master sheets needed, and use the template's exception sheet if it fits the last sheet of your imposition better.

Possible Workflows

What follows is an overview of all possible situations that can occur when creating impositions.

Note: You can start by selecting either a number of files, or an entire folder. In both cases, situations 1 to 3 can occur. As far as the incremental workflow is concerned (see below), only situation 1 can occur when selecting a folder.

Possible situations (and their message in the log window):

1. If the number of pages selected **equals** the number of pages in the selected template, the task will be processed correctly.
2. If **fewer** pages are selected than there are pages in the template, we call that a page '**underflow**'.
 - The task will end with a warning.
Log message: Page x in page list remains empty. Page underflow in AutoImpose: the template contains x pages, x pages remain empty.
 - The task will be processed correctly if the **Fill remaining pages with blanks** toggle is selected.
Log message: Page x in page list is made blank. Page underflow in AutoImpose: the template contains x pages, x blanks are added.
3. If **more** pages are selected than there are pages in the template, we call that a page '**overflow**'. Since the excess pages are not clipped, the task will end with an warning.
Log message: Page overflow in AutoImpose: x excess pages, the template contains only x pages.

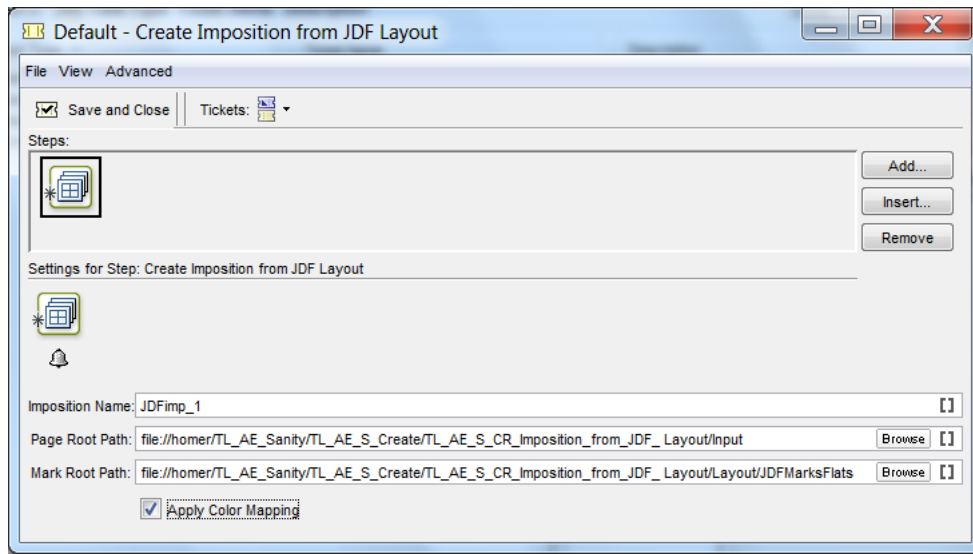
You can also opt to assign your pages in different steps. Do not forget to check the '**Fill page list incrementally**' toggle.

1. Select for example pages 1 & 4. The pages will be assigned to pages 1 & 4 of the template (and not to 1 & 2).
 - The task will end with a warning if the 'Fill remaining pages with blanks' toggle has not been selected.
Log message: Page underflow in AutoImpose: the template contains 4 pages, 2 pages remain empty. Page 2 in pagelist remains empty. Page 3 in pagelist remains empty.
 - The task will be processed correctly if the 'Fill remaining pages with blanks' toggle has been selected.
Log message: Page underflow in AutoImpose: the template contains 4 pages, 2 blanks are added. Page 2 in the pagelist is made blank. Page 3 in the pagelist is made blank.
2. Suppose you want to overwrite page 1 of an imposition called 1234. Select page 1 again in Pilot and launch with the 1234.imp file as template.

Log message: Adding file x to the page list. Page number 1 not free in template, overwritten by page (path), placed on page 1.

2.7.27 Create Imposition from JDF Layout

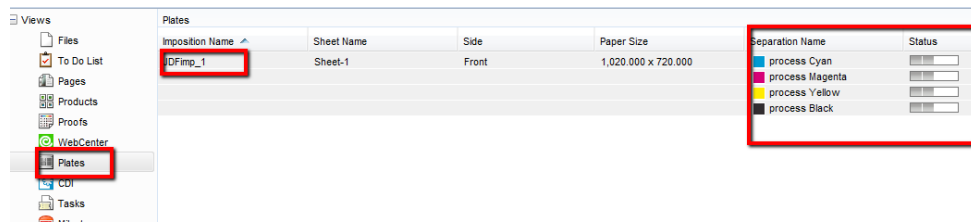
You can import a complete imposition including layout and page assignments into the **Pages** and **Plates** view using this task. This creates an IMP file from a JDF Layout file.



- **Imposition Name:** Here you can assign a name to the Imposition which will be visible in the Imposition Gallery.
- **Page Root Path:** You can specify the path to the pages referred in the JDF layout here.
- **Mark Root Path:** This path will be used as root to look for the marks referred from the JDF Layout File.
- **Apply Color Mappings:** Enable this option when you want to use the same ink mapping scheme which is defined in the JDF file. This mapping can be viewed in **Plates** View.

Note:

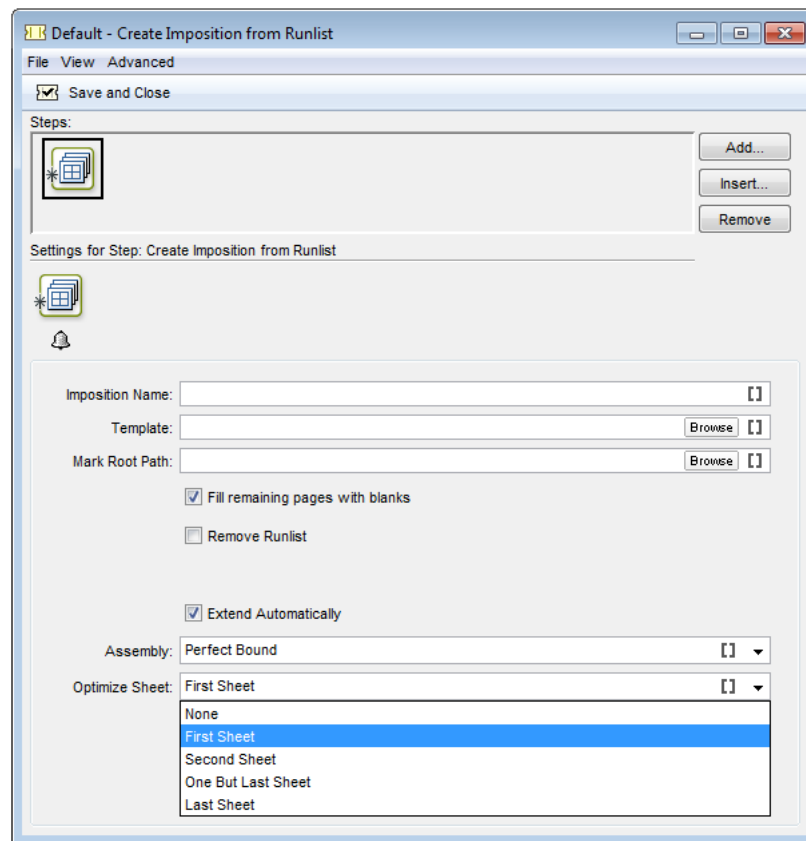
- This option works only when the input JDF file contains a color mapping.
- The mapped inks will be visible in the **Plates** View after applying color mapping. However, the inks in the referenced input PDF files will remain the same as before.
- When you enable **Apply Color Mapping** in the **Export Imposition to PDF file** ticket, the color mapping from the **Plates** View will be exported to the output PDF file.



2.7.28 Create Imposition from RunList

You can use this ticket in a workflow to create an Imposition from a RunList. This creates an IMP file from a RunList file.

Settings



- **Imposition Name:** You can use this name to identify the Imposition in the **Imposition Gallery**
- **Template:** You can browse to your JDF layout file here.

Note: You can use SmartNames like `nPages`, `pageHeight` and `pageWidth` to make the selection of the Layout Template dependent on the RunList.

- **Mark Root Path:** You have to enter the path to the file with marks referred from the JDF layout.

- **Fill remaining pages with blanks:** You can enable this if you want to add blank pages when the layout contains more pages than the RunList.
- **Remove RunList:** You can enable this if you want to remove the RunList after it is included in the Imposition.
- **Extend Automatically:** Enable this to expand the number of sheets in the template to accommodate for all pages in the runlist.
- **Assembly:** Choose from : **Saddle Stitch**, **Perfect Bound**, **Cut & Stack** and **Automatic**. You can use SmartNames to select an **Assembly** provided your SmartNames are prepared using the localized assembly name.



Attention:

- All sheets in the template should contain equal number of pages.
 - If **Assembly** is set to **Automatic** , the template should contain a minimum of 2 sheets to establish the numbering scheme. Numbering should be consistent across the template when there are more sheets than 2.
 - If the imposition contains printgroups of multiple sheets, the renumbering will be at the printgroup level as that is the logical foldable unit.
 - If the runlist has layers, individual layers will be handled independantly while renumbering. Similiarly, independant flats will be added.
- **Optimize Sheet:** When **Extend Automatically** is enabled, you can select and optimize sheets. When this option is set, the task removes the back side of the selected sheet and uses the front side for printing (Work & Turn or Work & Tumble printing depending on your template numbering).
 - Select from : **First Sheet**, **Second Sheet**, **One but last Sheet** and **Last Sheet**.

Note: The works only when the assembly is **Perfect Bound** or **Saddle stitched**.

Note: Use **Optimize Sheet** only when the sheets are filled up to 50 percent or less.

You can view the results of this task in the **Imposition Gallery**.

Imposition Gallery

Page list

Runlist 1 - 32 pages - Finished page size 210.000 x 297.000 mm Page :

1	2	3	4	5	6	7	8	9	10	11	
1	2	3	4	5	6	7	8	9	10	11	
12	13	14	15	16	17	18	19	20	21	22	23
12	13	14	15	16	Blank	Blank	Blank	Blank	Blank	Blank	Blank
24	25	26	27	28	29	30	31	32			
Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank			

2.7.29 Create Job task

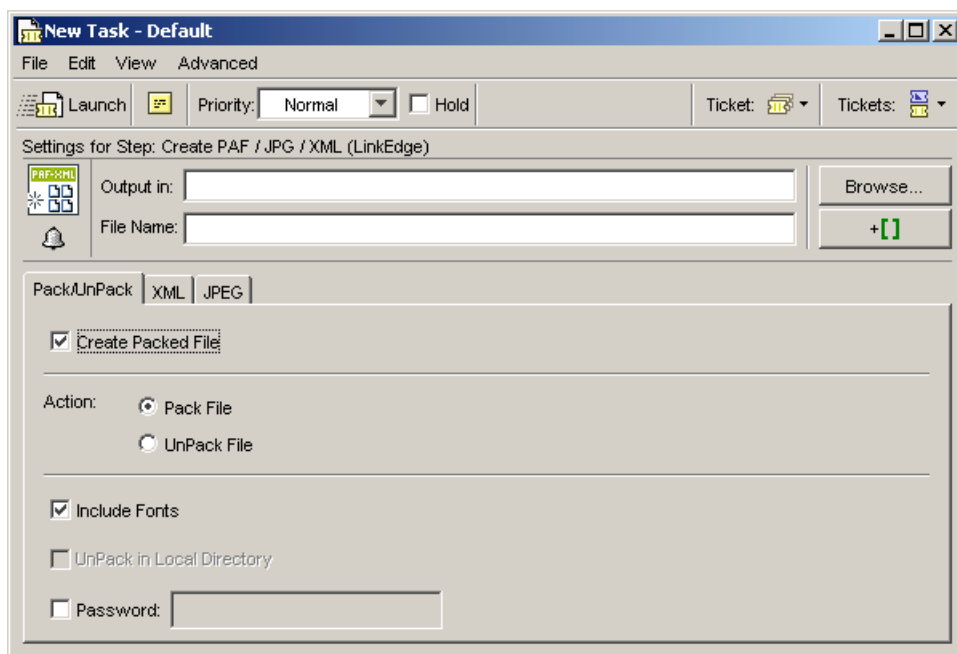
This task is documented in the chapter [Integrating with External Systems](#) on page 1228. Find a direct link [here](#).

2.7.30 Create or Modify WebCenter Project

This task is documented in the chapter [Integrating with WebCenter](#) on page 1346. Find a direct link [here](#).

2.7.31 Create PAF / JPG / XML (LinkEdge)

LinkEdge started as an interactive tool to link 2 sites both using PackEdge. This tool packed the Esko native data (with all its references) and allowed to unpack it again at the receiving site. However, this is more than just 'Esko-Zip'. The (un)Packing also communicates with the Esko editing kernel and resource folders like central or local font folders. Automation Engine then started a background server version of this historical tool. It was logic to also add extra file representation data like thumbnails and metadata (XMP-XML).



Pack/UnPack

Select **Create Packed File** to create a PAF file (a packed file) from a Normalized PDF or Plato PLA file (and legacy file formats GRS and GRQ).

Set the Pack/Unpack Options:

Select an action:

- Pack File, this creates a .PAF or a .PDFSC file
- Unpack File
- You can **include** the **Fonts**
- Tick the **Unpack in Local Directory** toggle if you want to be sure that all elements of your packed file will be unpacked in the same directory. Be it the default directory or the one you defined as output directory. If you deselect this option, LinkEdge will try to restore the original file structure.

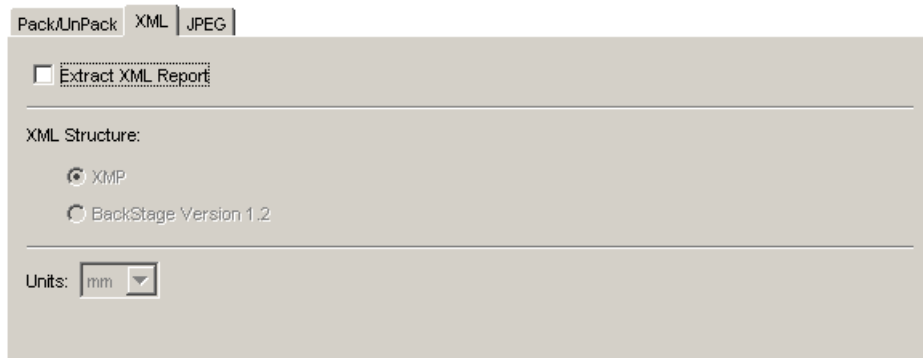
Suppose you have packed a Normalized PDF (D:/Jobs/2345 banana/2345_50ml.pdfsc) and it has one referenced image (residing in a different folder: D:/Jobs/2345 banana/images/banana.eps). If you were to unpack using the Unpack in Local Directory option, the LinkEdge task would drop all files in the same directory. If you were to Unpack without the option, then the LinkEdge task would put the CT in a CT folder.

- Use the **Password** toggle to protect the Packed file. You will be prompted to enter the password when you want to unpack.

XML

The system is also able to generate an XML Report, which contains information about the file: CTs, inks, angles, LPI, ...

XML Structure: choose between standard XMP and the old BackStage 1.2 version (also based on XMP, but contains less information), and define the units.



Note: Detailed information on Esko XMP tags can be found on the technology pages on www.esko.com.

JPEG

Tick the **Extract JPEG Preview** toggle if you want a preview file. This is useful for archive systems that work with thumbnails.

Set the various Image settings:

- Horizontal and Vertical resolution in dpi
- Set the width and height in pixels
- Choose a compression from the dropdown list
- Select the anti aliasing toggle (use the dropdown list to define the quality of the anti aliasing)
- Decide if you want the JPEG to contain data from outside your borders. Check 'Account for margins' to select the margins of the job as the true boundary. Normally the borders of the job are taken as the true boundary of the job. Sometimes however you may also want to convert everything that is situated within the margins too.
- Check the option 'Use varnish inks in output' if you want the inks which have the attribute 'varnish' in the input file to be included in the JPEG file.
- Page range: choose between 'All pages' or enter a page number / range.

Note: Detailed information on Esko XMP tags can be found on the technology pages on www.esko.com.

2.7.32 Create PDF File from JDF RunList

This task uses JDF RunLists to combine individual PDF (one-up) files into one multipage PDF file. You send the JDF RunList to the task, and define where to find the individual PDFs.

You can create JDF RunLists using the [Split Pages](#) on page 475 task, or [The RunList Editor](#) on page 786.

The RunList contains the file names of the files to be joined, their joining order and their original location.

One Up Location: Define the location of the one-up PDFs. Choose one of the following options:

- Select **Defined in JDF** if you want to use the location defined in the JDF RunList,
- Select **Custom Location** if you want to specify the one-ups' location yourself.

Note: When you choose **Custom Location**, fill in the location of the **Custom Folder**. You can browse to it, enter a path or/and use SmartNames.

Replace * in File Names By: If you created your RunList manually using the RunList Editor, you may have wildcards (*) in the file names of the PDFs to be joined. For example, the file names in your RunList may look like this: *_01.pdf, *_02.pdf, etc.

By filling in the option **Replace * in File Names By**, you can define the part of the file name that the * matches.

Note:

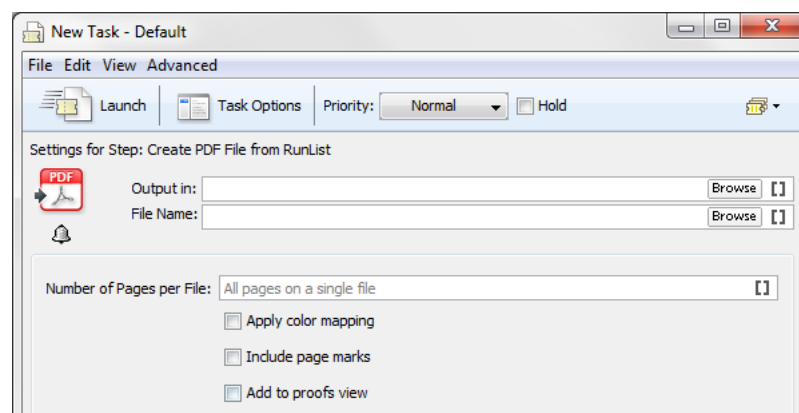
The output PDF will not be normalized if:

- at least one input file is not normalized,
- at least two input files have different media or trim boxes.

The XMP information of the individual PDFs will be merged into the multipage PDF.

2.7.33 Create PDF File from RunList

The **Create PDF File from RunList** task can be used to create a multipage PDF file containing (a selection of) the pages of a RunList. This is typically done for content proofing purposes.



You have the following options in this task:

- **Number of Pages per File:** This defines the maximum number of pages after which a new output file will be created. This number is a maximum. When multiple layers are selected, each layer will start with a new file.
- **Apply Color Mapping:** This will add and/or remove inks to/from the RunList pages to ensure that your proof will look the same as your sheet output.

Note: This option will typically be used in combination with the **Apply Color Mapping** option in the **Export Imposition to PDF File** or the **Export Imposition to JDF Layout** tasks.

- If your RunList contains marks on page level (like trim marks, page numbers, etc), you can include these marks by selecting the **Include page marks** option.
- **Add to proofs view:** You can optionally add your proof to the Proofs view after it's created. This will allow you to keep an overview of your proofs and ensure that the content of your proofs is consistent with the content of the pages in your Imposition.

Note:

In case an imposition was defined for the RunList, the trim box from the PageSlots in the imposition will be used. If no imposition was defined, the trim box of the RunList will be used instead.

In case a media box was defined for the RunList, this media box will be used. In all other cases, the media box from the included pages will be used.

Input and output formats

Possible inputs for this task can be one of the following:

- An IMP file
- An imposition
- A RunList
- A PageSlot
- A layer
- A plate

The output for this task will be a PDF file.

2.7.34 Create Report (ReportMaker)

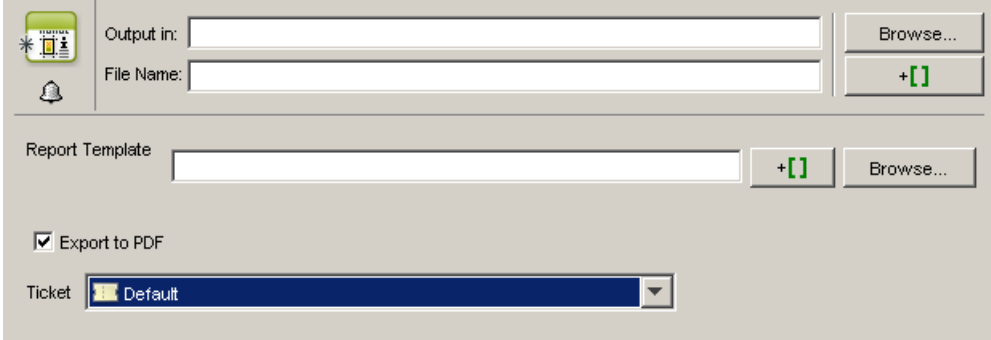
Note: This Task requires a PackEdge license.

The **ReportMaker** task is used to automate the creation of design print cards (in French: fiche technique, in German: Drueck Karten).

The idea is that you generate reports containing all kinds of information: job name, used inks, name of the operator, an image of the job...

All this information is defined in a report template, then this template is used to generate a specific job. For more information about templates, see [ReportMaker Templates](#) on page 291.

Settings



The screenshot shows the 'Settings' dialog for the ReportMaker task. It features the following elements:

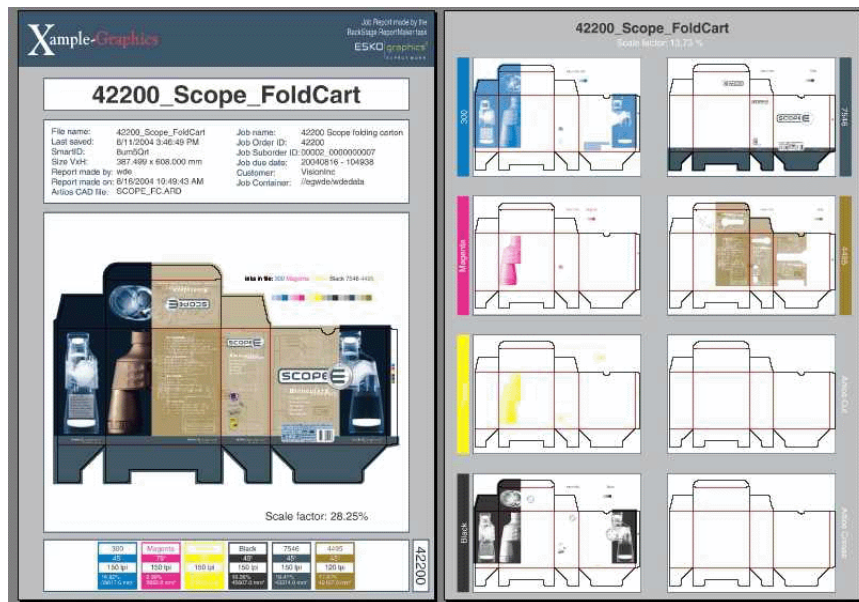
- Output in:** A text input field with a 'Browse...' button to its right.
- File Name:** A text input field with a '+[]' button to its right.
- Report Template:** A text input field with a '+[]' button and a 'Browse...' button to its right.
- Export to PDF:** A checked checkbox.
- Ticket:** A dropdown menu currently showing 'Default'.

Select a **Report Template** (you can browse to it or use SmartNames to define the path).

If you want your report to be a regular PDF instead of a Normalized PDF, select **Export to PDF**, and choose a **Ticket** for the PDF export. For example, use this to make a report containing a 3D object (see [Creating a Report Containing a 3D Object](#) on page 297).

Example

This is an example of how a report may look when viewing it in Acrobat. The format and content is decided by the **Report Template**, also the amount of pages.



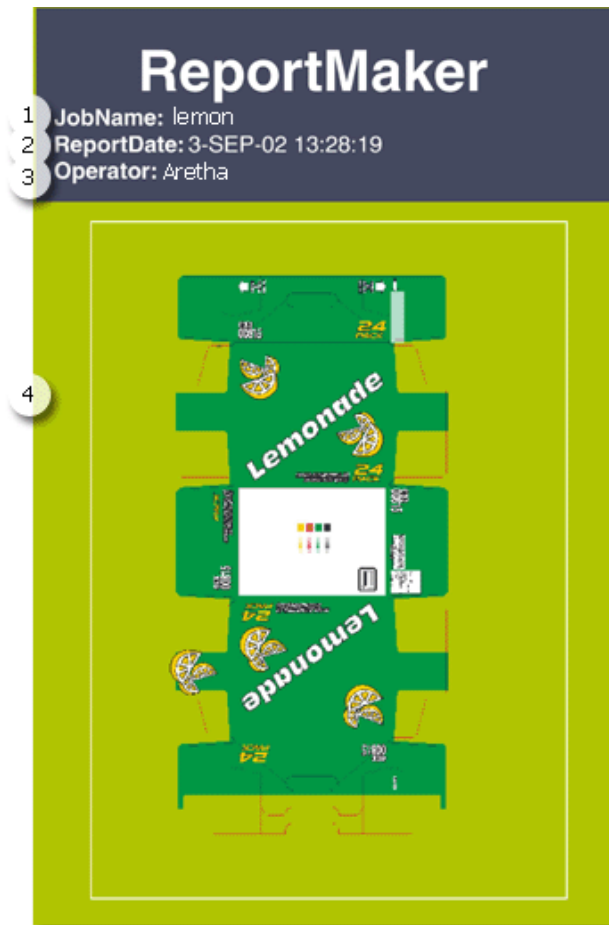
ReportMaker Templates

You need PackEdge 3.0 or higher to make a ReportMaker template.

The ReportMaker Template - how the resulting print card will look - is a Normalized PDF which you can design yourself.

You are placing an image of your input file before you even know what your input file will be (since it's a template and should be generic), but PackEdge offers a solution.

Let's make a template that will give us this result:



1. The name of the Job
2. The date/time of the report
3. The operator who made the report
4. An image of the job (possibly scaled to fit the report)

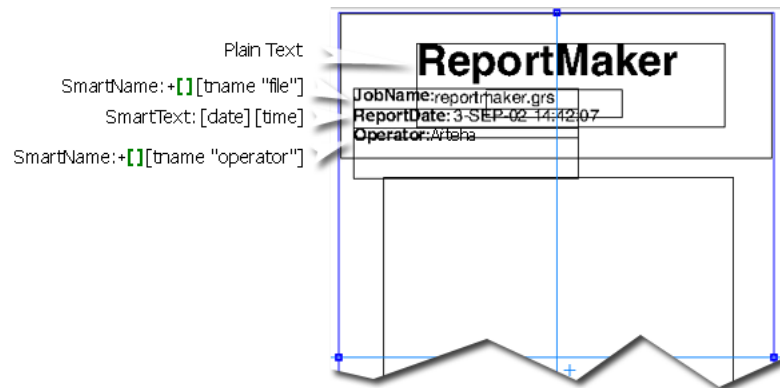
The name of the job, the date and time, the operator are all Text Marks that are placed in the file and they are resolved when the ReportMaker task is executed.

In fact, you can place as many SmartMarks as you see fit.

Note:

SmartMarks used in a ReportMaker template can give you useful information:

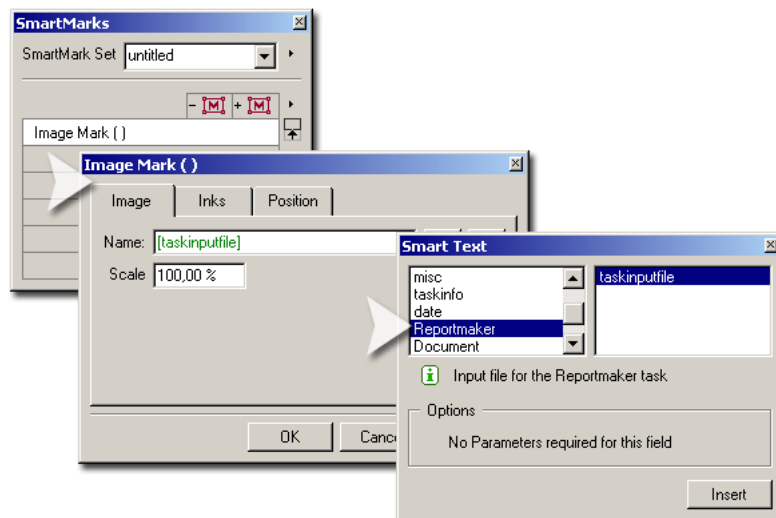
- If you use ink-related SmartMarks, they will reflect the inks of the input job, not of the generated report.
- Ink coverage will give you the ink coverage of the unscaled version of the input job (the report may contain a scaled version of the input job).



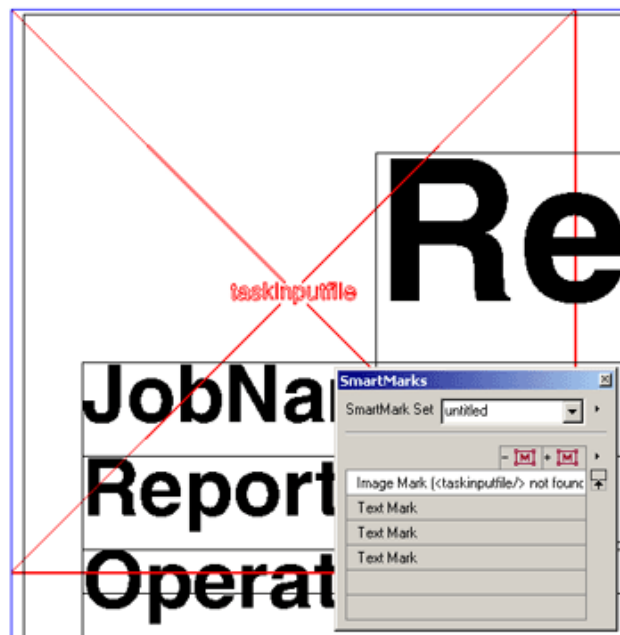
The image of the input file is a special kind of ImageMark.

1. Select a new Image Mark
2. Use Smart Text [taskinputfile] to define the name of your Image Mark.

taskinputfile is the one smart text that's able to define the Image Mark in such a way that it will place the input file in the Report at the time of outputting.



3. The Linework Editor will place the Image Mark, it will look like a square because at that time the editor has no idea how big the input file will be.

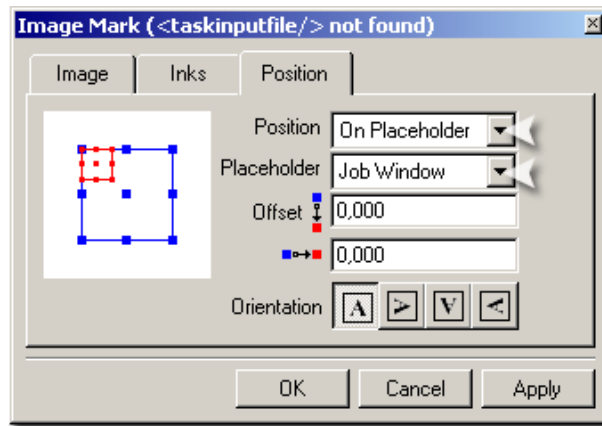


4. You can use Named Objects to further define the behavior of the Image Mark during output. You can make any object a placeholder, in this example we'll make the white contour a placeholder

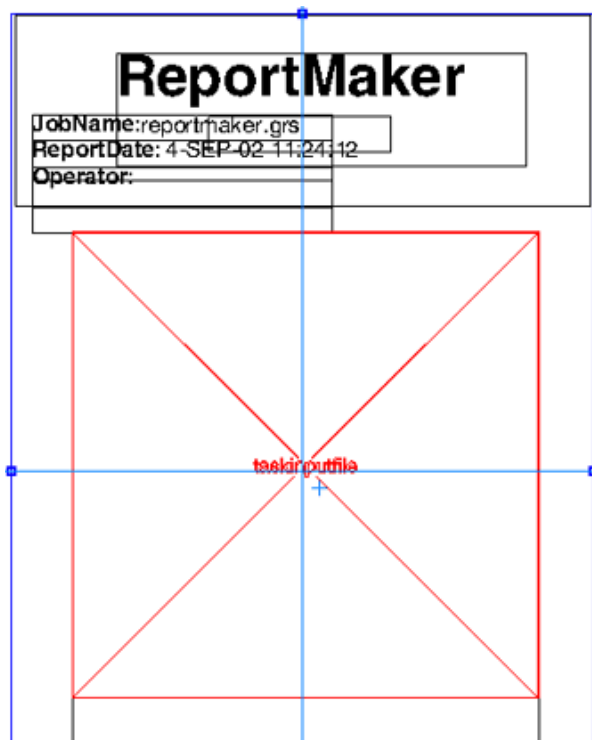


Select the object you want to make a contour, select 'Placeholders...' from the Production Menu, add a new one and give it a name (in this example: Job Window).

- Now you can define the Image Mark according to the Placeholder. Choose On Placeholder from the Position dropdown list. Choose the correct placeholder from the Placeholder dropdown list.



You'll see that the editor places the Image Mark in the Placeholder.

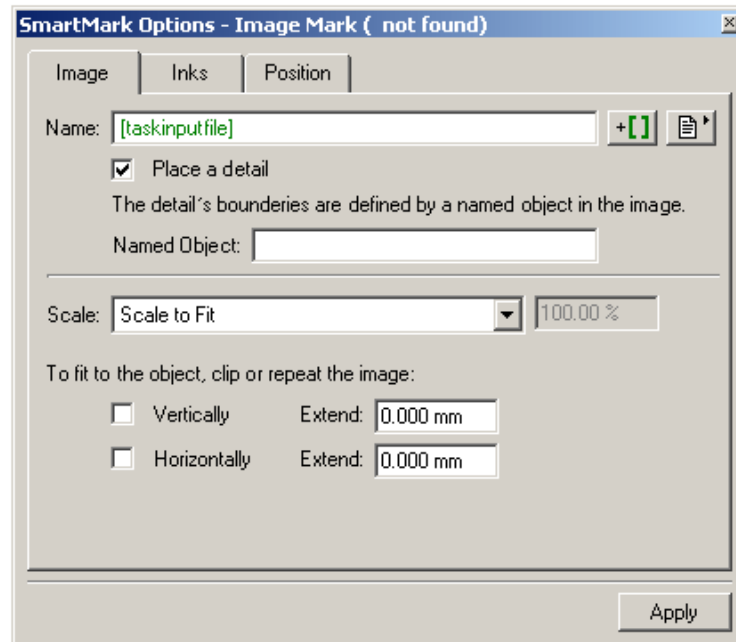


- Bear in mind that the task inputfile Image Mark has the same functionality of any other SmartMark, it behaves the same, it can be defined in the same way, it can be scaled, the inks can be defined...

Viewports in ReportMaker

A ReportMaker report can show you several 'viewports' of a file. This is ideal to zoom in on smaller objects (ingredient copy, small logos, bar codes...). It also proves to be very useful if the design contains only few graphic objects, as could be the case for example on corrugated boxes.

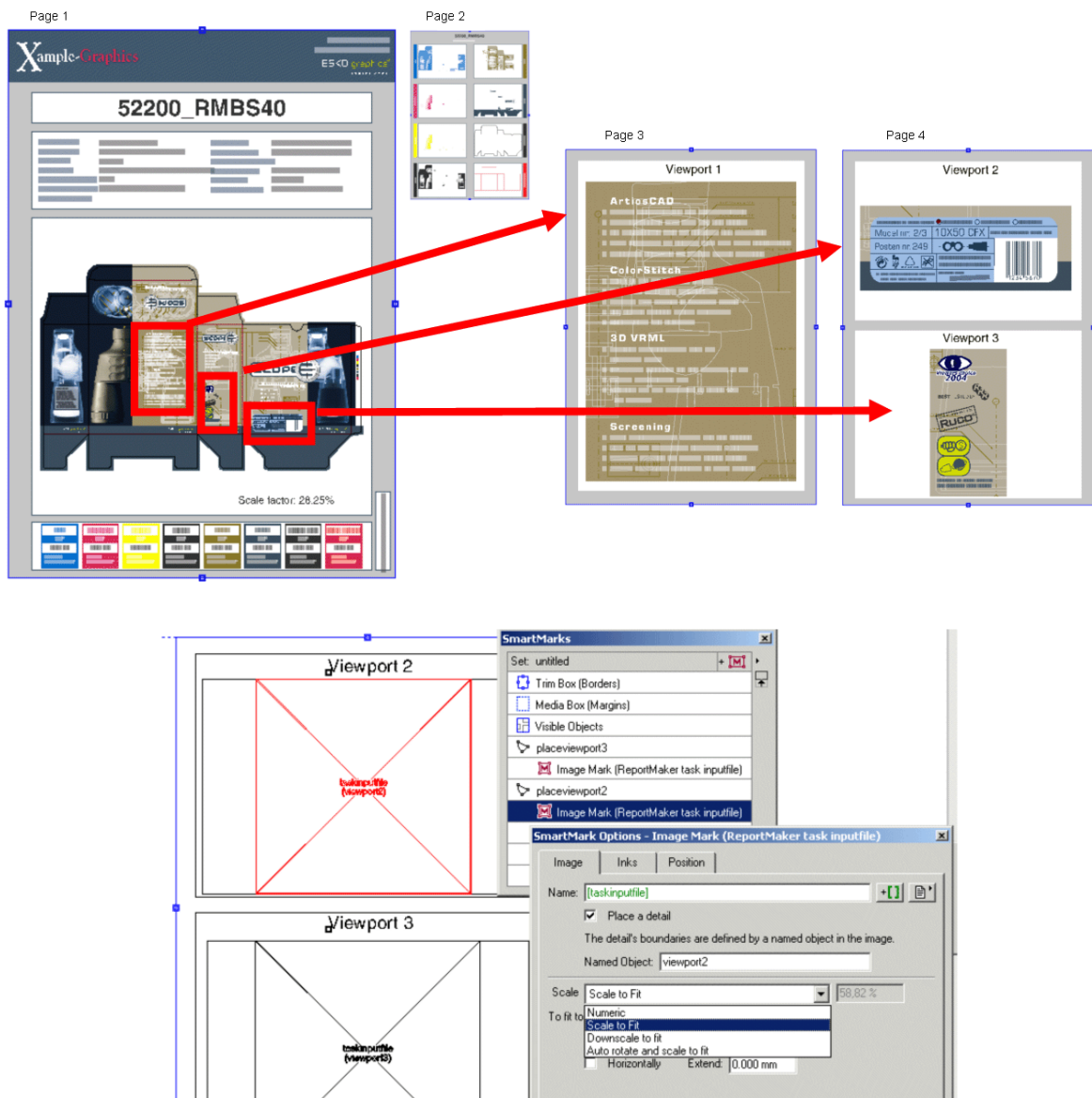
When creating ReportMaker templates in PackEdge, you can specify a named object to attach the task input file to. This task input file will make sure that the file will be pasted in the named object, which will act as a 'window'.



1. Select an object or draw a box and give it a name.
2. Select the named object and open the SmartMarks dialog box. Make sure that you add the new mark to the named object.
3. Select the SmartMark type 'Image Mark' which allows to indicate that you want to place only a detail of the [taskinputfile] (under 'reportmaker').

You can zoom in by taking the bounding box of the named object and applying the specified scale factor on this bounding box. For example 'Scale to Fit' will calculate the scale factor without taking into account the size of the input file but the size of the named object in the input file.

If the named object is a contour object, the contour object will be used as a clip mask on the input file. If the named object is a different object (CT, group of objects...) a rectangular clip mask (with size equal to the bounding box of the named object) will be applied.



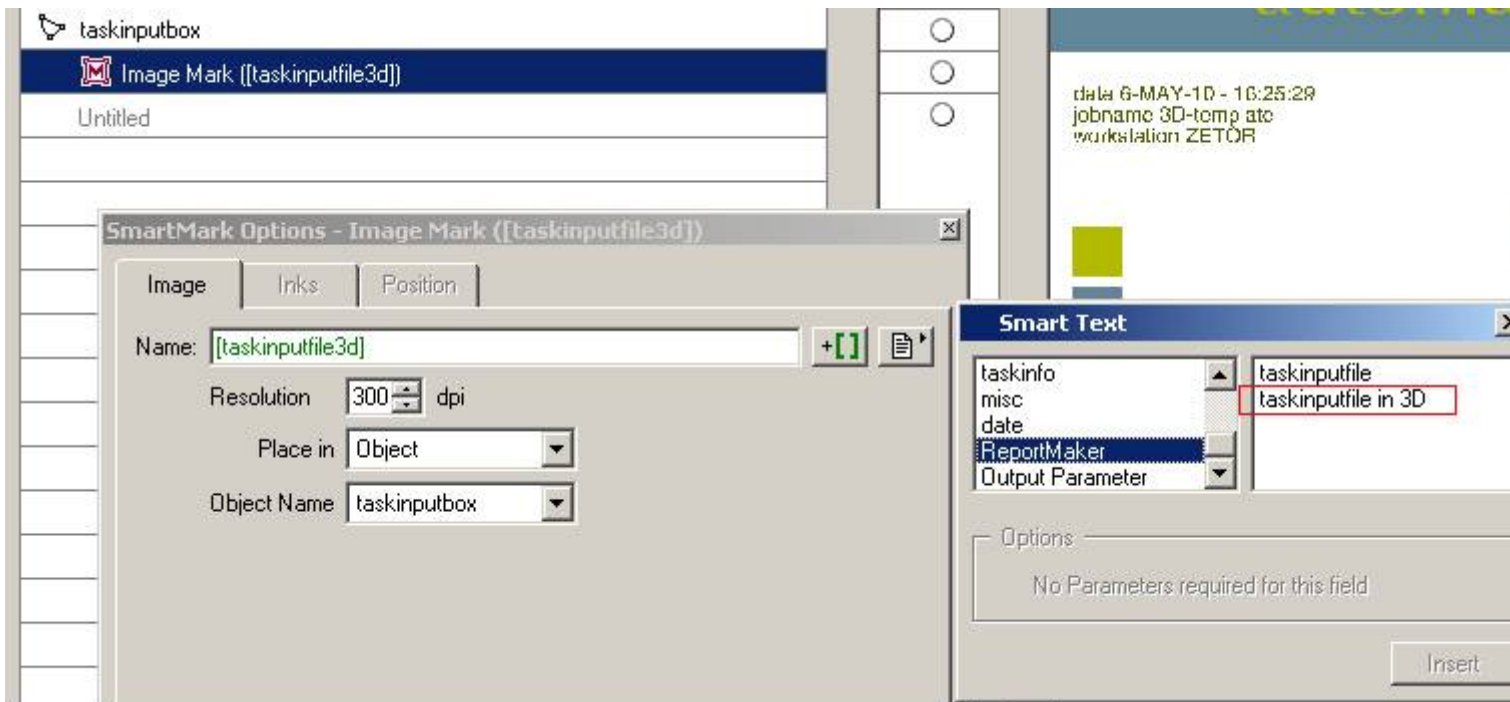
Creating a Report Containing a 3D Object

To create a report containing a 3D object, you need:

- a report template with image marks allowing 3D objects,
- a Normalized PDF file with a link to a CAD (.ard, .dae or .zae) file.

1. In PackEdge:

- a) create a template,
- b) add an image mark where you want your 3D object to be,
- c) use the SmartNames button to use Smart Text for your image mark name,
- d) in the **Smart Text** pop-up, choose **ReportMaker** then **task inputfile in 3D**,
- e) save your template.



2. In Automation Engine:

- a) launch the **Create Report (ReportMaker)** task on your Normalized PDF with a linked CAD file,
- b) in **Report Template**, browse to the 3D-enabled report you created in PackEdge,
- c) select **Export to PDF**,
- d) choose a **Ticket** for the PDF export.
- e) click **Launch**.

You will get a report containing a 3D object corresponding to your CAD file.

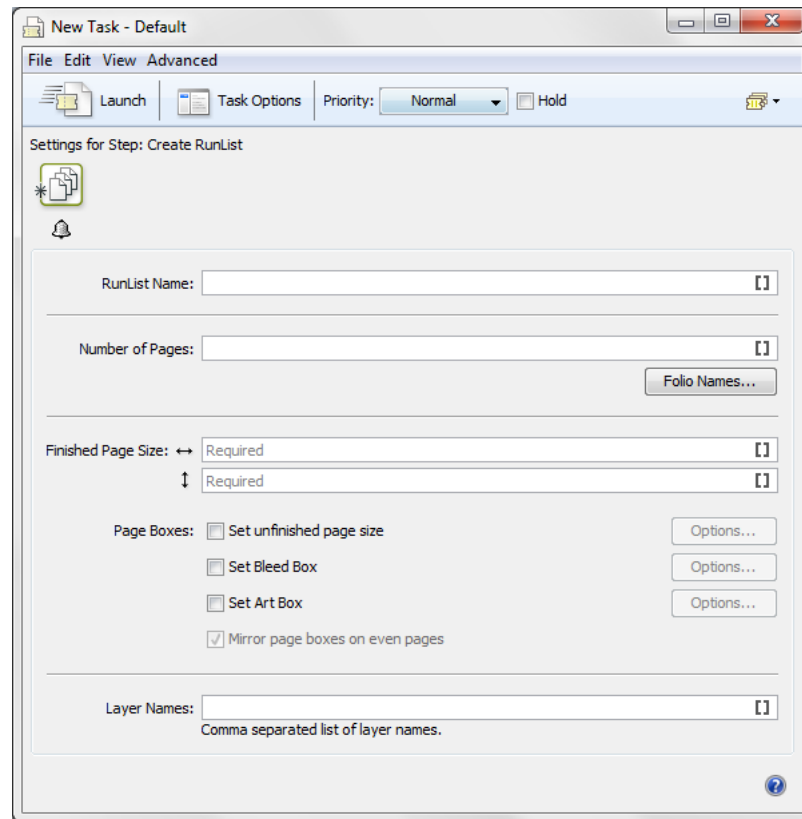
You can open this report and view the 3D object on all computers running Adobe Reader 8.0.1 or later.

2.7.35 Create RunList

With the Create RunList task, you can create a new RunList from one or more uploaded file(s).

When you use this task in a job context, the created RunList will be added to the imposition gallery in the pages view for this job.

When this task is used in a workflow context, the created Runlist can be used as long as the workflow is running. After the workflow has completed, the created RunList will be removed.

**Note:**

By default, the source files are not visible in the task window. To see these files, click **View > Show Files > .**

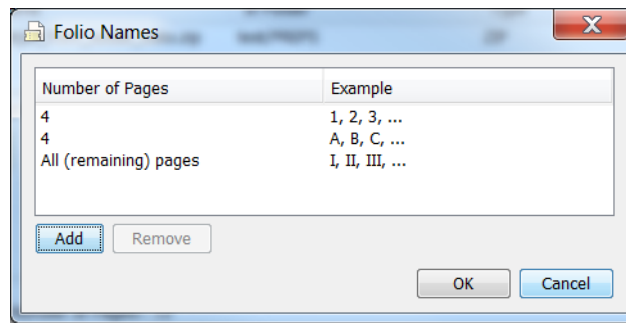
If you want to add more source files, click: **File > Select File .**

1. Specify the **Runlist Name**. This name will be visible in the **Pages** view of the job when the task successfully finishes. If no name is specified, the name of the first source file will be used for this.



Caution: If a RunList with this name already exists for this job, it will be overwritten.

2. Specify the **Number of Pages** in your RunList.
If you want to specify the number of pages by specifying the number of pages in the folio ranges (see below) , then leave this field blank.
3. If you want to use certain folios for the pages in your RunList, you can build these by clicking on the **Folio Names** button.
The **Folio Names** dialog will prompt.



4. You can add a new folio range as follows:
 - a) Click on the default **Folio Range** and click **Remove**.
 - b) Click on the **Add** button to add a new **Folio Range**.
The **Folio Name** dialog will prompt.
 - c) Choose the desired **Number Format** for your folio.
 - d) Choose the **Number of Pages** the folio will run.
 - e) Choose the number of the first folio in the field **Start at**.
 - f) Determine the **Prefix** and **Suffix** of the Folio.
 - g) Click **Add** to add the new folio to the **Folio Names** dialog list.
5. If all desired Folio ranges have been added, click **OK**.
6. Specify the vertical and horizontal **Finished Page Size** (the trim size) of your RunList.
7. You can choose to (de)select one or more of the following checkboxes:
 - **Unfinished Page Size**: This option allows you to set the unfinished page size to be used by proofing and impositioning later on in your workflow.
 - **Bleed Box**: This option allows you to set the page bleed box to be used by proofing and impositioning later on in your workflow.
 - **Art Box**: This option allows you to set the art box to be used for page positioning later on in your workflow.

For each of these options, you can click on **Options...** to select the offset values.
8. If you're using single sided printing, deselect the **Mirror Page Boxes** checkbox.
9. You can specify the **Layer Names** if you want to create your RunList in multiple layers. For example, to handle language variants with plate reuse.
For each comma-separated name, a new layer will be added to the RunList.

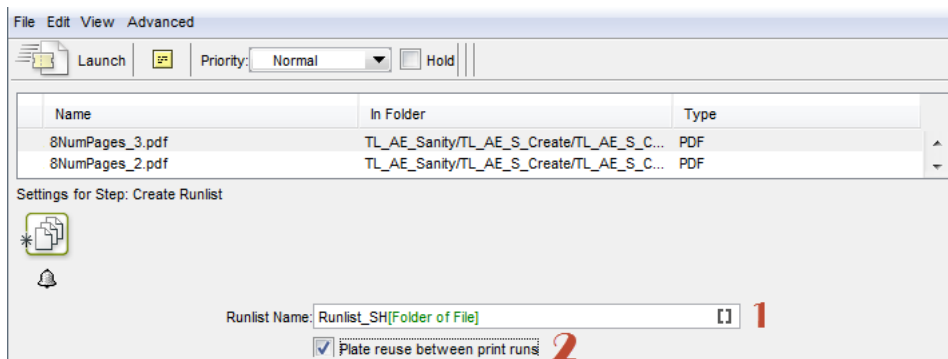
2.7.36 Create RunList from PDF

Using this task, you can create a **RunList** from one or more selected PDF files. A **RunList** collects the pages from one or more PDF files sequentially to make one final JDF RunList (for example for an imposition). It contains the names of the files to be collected or joined, their collecting / joining order and their original location.

Note:

The source file(s) can be both normalized and non-normalized PDFs.

Using the **Create RunList from PDF** task will copy each page into the RunList. However, if you want to add **Folio Names** to the pages, change the **Finished Page Size** or change the **Page Boxes**, use the **Create RunList** task instead.

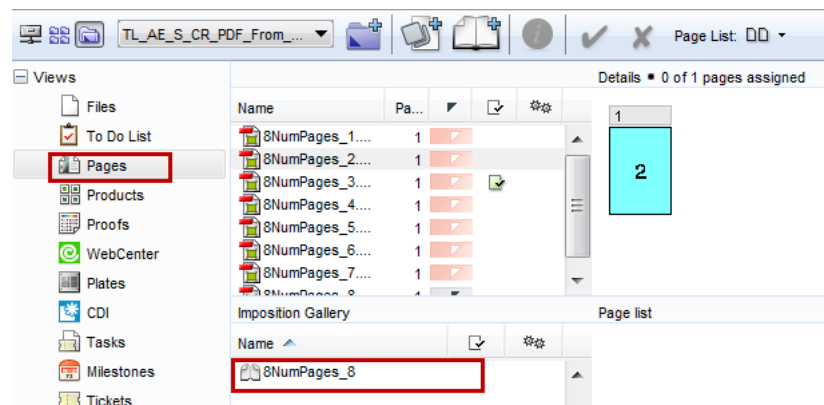


Note:

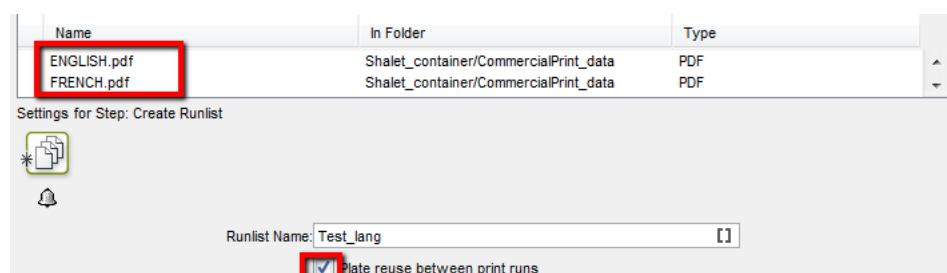
By default, the source files are not visible in the task window. To see these files, click **View > Show Files >** .

If you want to add more source files, click: **File > Select File** .

1. Give a suitable name in the **RunList Name** field. You can use SmartNames for this. If this task is completed, the result can be seen in the **Pages** view when working in a job context.

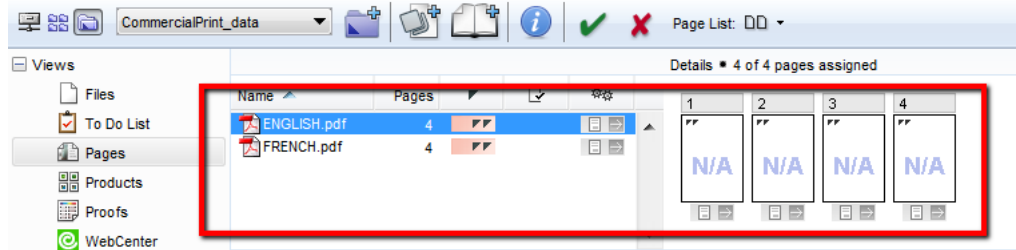


2. If you have language variants, enable **Plate reuse between runs**. This enables the reuse of the image plates among the language variants. This option will consider each input file as a separate variant, if the number of pages in these files are equal.



You should use this in combination with the **Variant Separation** settings in **Job Setup**.

This will produce a RunList (with multiple layers, if plate reuse has been enabled). Plates will be reused for common images in these language variants.

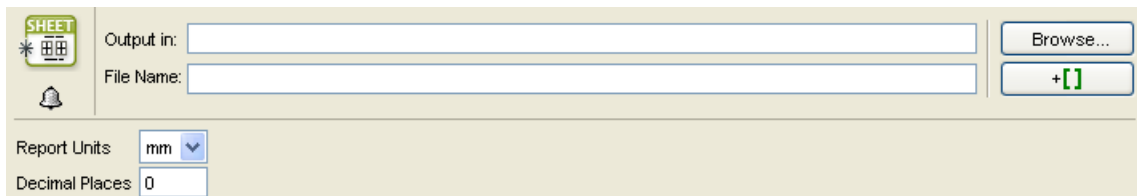


When you create an Imposition from a multilayer RunList, the system will take this into account when creating the plates in the **Plates** view.

2.7.37 Create Sheet Report from IMP File

Use this task to output your imposition report as a PDF.

Note: The report is created in FastImpose, when you make the imposition (you choose a report template).

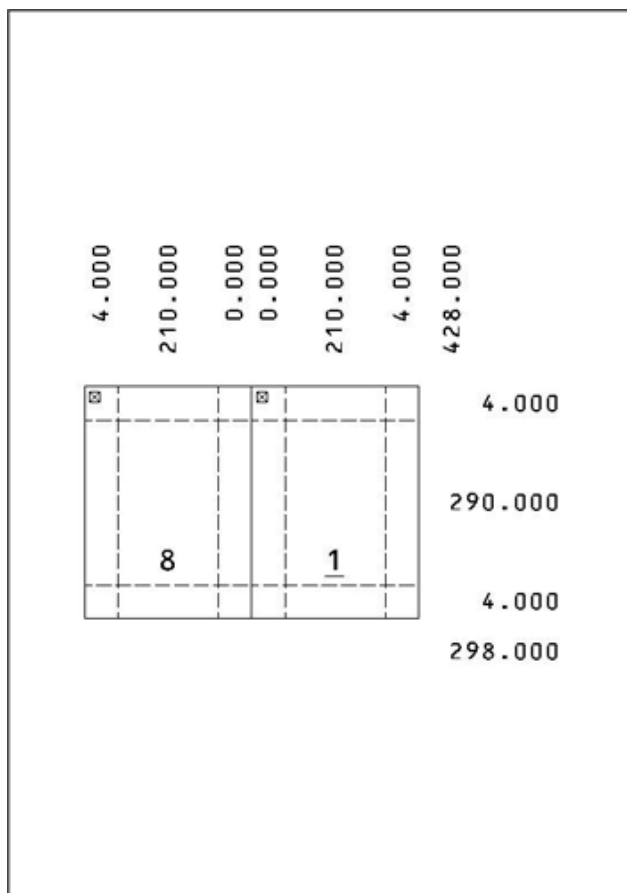


Output in:
 File Name:
 Report Units:
 Decimal Places:

Report Units: You can choose to show the imposition's dimensions in either millimeters or inches in the report.

Decimal Places: Choose how many decimal places you want in the dimensions.

The report below uses millimeters and 3 decimal places.

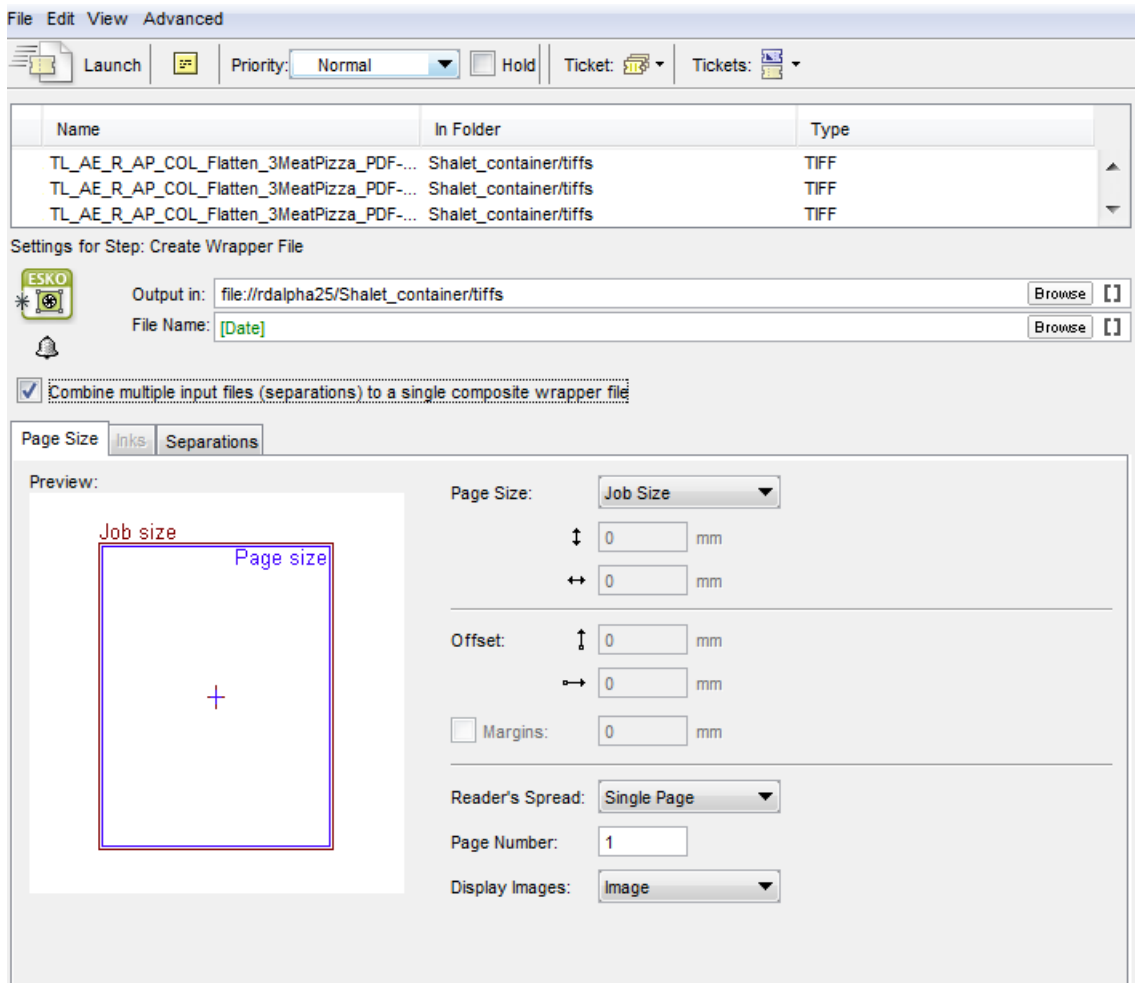


2.7.38 Create Tiles

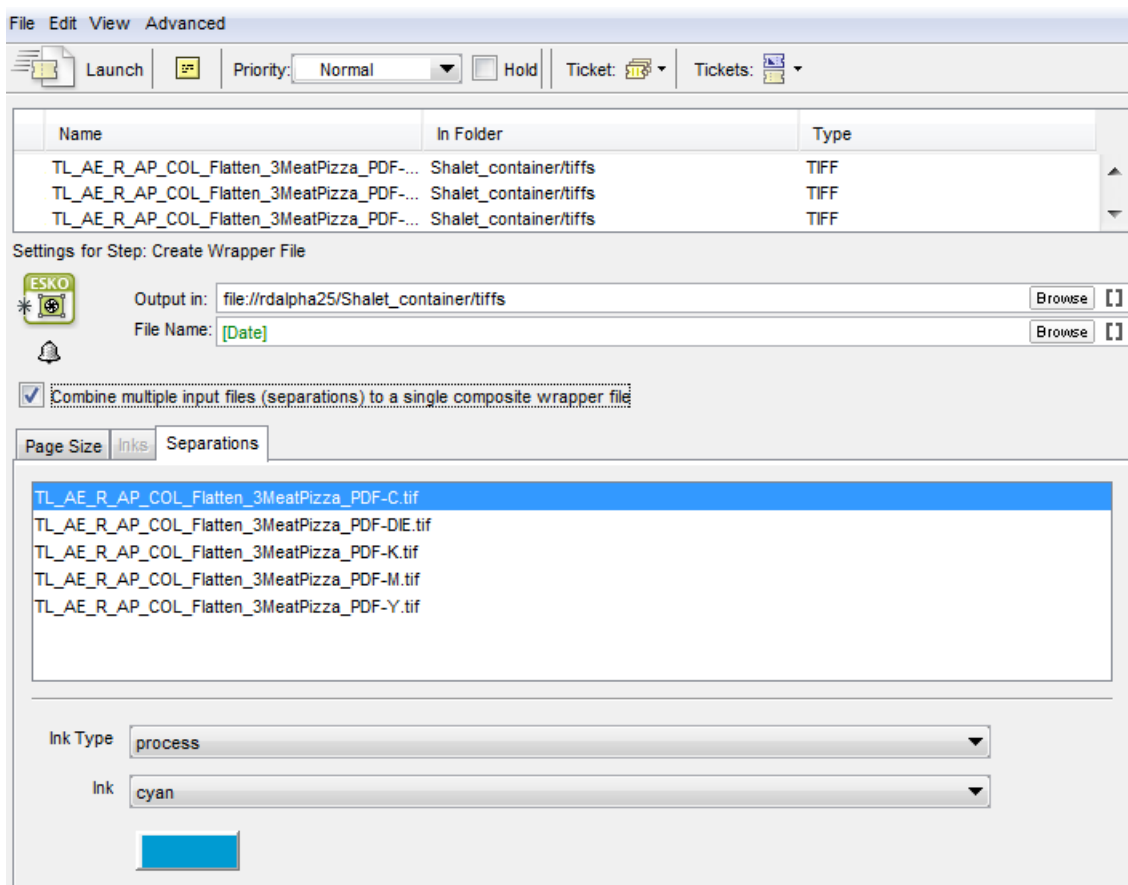
For detailed information about the **Create Tiles** task, please see [Create Tiles](#) on page 1104 in the Gang Run Printing chapter.

2.7.39 Create Wrapper File

The Create Wrapper File task creates a Normalized PDF that encapsulate a given image, for example a screened TIFF copydot file. The Normalized PDF encapsulating the image can then be processed further via tasks that can handle Normalized PDF data such as FlexRip tasks.



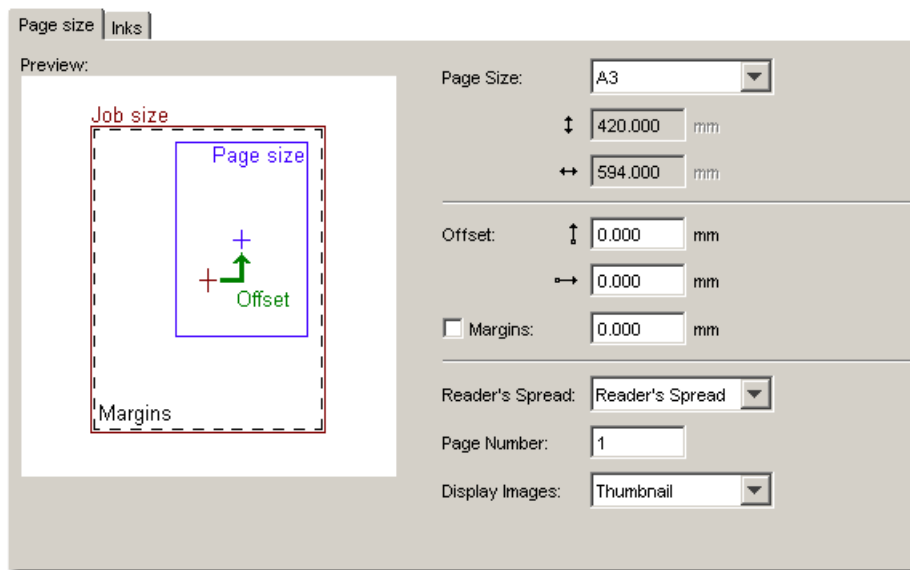
You can bind a set of stand-alone files containing bitmap or copydot data that contains ink separations of one image using the the **Separations** tab. The output file will be a normalized PDF document.



The Create Wrapper File task supports the following file types:

- TIFF
- TIFF_IT
- PSD
- JPG
- BG_CT
- BG_LP
- BG_JS
- EPS
- DCS
- Copydot

Page Size tab



Page size

The default page size is **Job size**. When Job size is selected, the page size found in the selected file will be taken into account when importing. The **Custom** option allows you to enter page size values manually. The dropdown list displays all possible options (for example Custom, A5, A3, Letter, Tabloid, Scanner Trim Size...). Click the arrow in the dropdown list to switch between millimeters and inches.

Note:

All the Eskoscan scanners have a feature to indicate the trim size of the page when making a scan. Scanners equipped with software release JPS9.0 build 99 or higher will write that information into the master DCS file.

When choosing the option 'Scanner Trim Size' in the 'Job Size' list, the borders of the resulting file will be automatically positioned where intended. The placement of that Normalized PDF file in FastImpose or Plato will not require any manual correction.

Offset: Enter the appropriate values to give the page a horizontal and/or vertical offset. When no offset is given, the center of the image will be positioned on the center of the page. If the page is smaller than the image (width, height, or both), then margins will be set that enclose the image completely. The size of the margins will not be fixed. Margins indicate the differences between the image size and the user-defined page size.

Margins: Tick the margins toggle and define the margins' dimensions if you want to give the margins of the page a fixed size (for example 3 mm for bleed). Click the arrow in the dropdown list to switch between millimeters and inches.

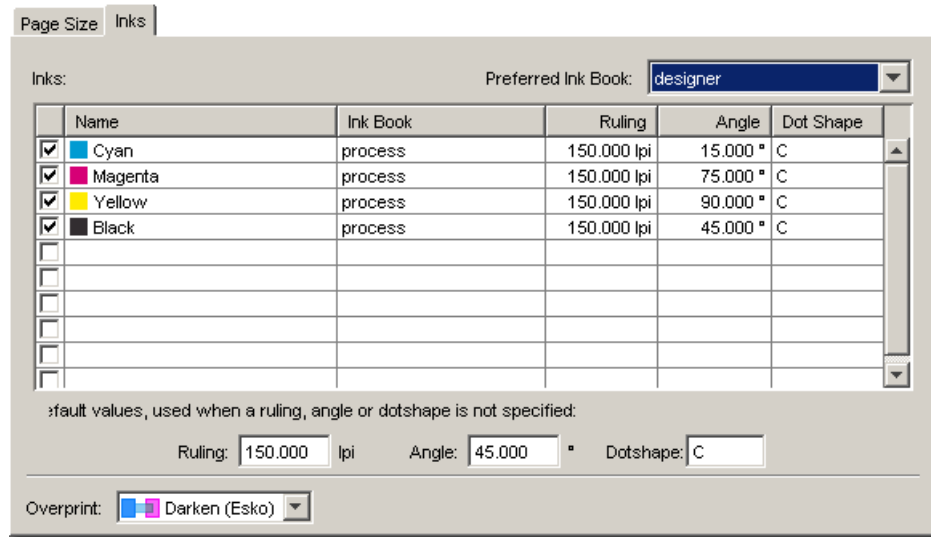
Reader's Spread None of the file formats that can be wrapped into a Normalized PDF file have the notion of reader's spread. It can be useful, for example when using the Normalized PDF files in FastImpose, to give an indication of whether the file is a single page or a reader's spread.

- **Single Page** forces the file as single page output.
- **Reader's Spread** forces the file into a reader's spread format. The file will be set as double pages.
- **Auto** checks the size of the image and when the image width equals or exceeds two times the selected page width, the reader's spread option is set. Otherwise it is treated as a single page.

Page number: Select the correct page number from the Normalized PDF that you want to output.

Display images: This option allows you to change the Normalized PDF preview settings. Select None, Thumbnail or Image from the dropdown list.

Inks tab



The Inks tab contains a fixed list of inks (default, cyan, magenta, yellow, and black) to which you can add other inks manually.

Tip:

Choose the ink book you use most frequently in the **Preferred Ink Book** list. Doing so ensures that when you type an ink name in the table, Automation Engine searches for the ink in your preferred ink book first. For example, if you type the ink name "154" and your preferred ink book is set to Pantone Colors Uncoated, Automation Engine will automatically suggest the ink PANTONE 154 Uncoated.

If an image contains definitions for its inks (for example a CMYK Tiff or a PhotoShop DCS2) and an ink of the image matches an ink selected in the inks tab, then the angles, screen ruling, etc... in the Normalized PDF file will be the ones from the inks tab. All inks in the image that do not match the selected ink in the inks tab will be processed according to the values for angle, screen ruling and dotshape in the Default options.

For images that contain **no ink definition**:

- If only one ink is selected in the inks tab, the Normalized PDF file generated will contain that specific ink with angles, lpi and dotshape as specified in the inks tab.
- If multiple inks are selected and K is part of that selection, the Normalized PDF file generated will contain K with angles, lpi and dotshape as defined in the inks tab.
- If multiple inks are selected and K is not part of that selection, then the Normalized PDF file generated will contain the first ink from the list with angles, lpi and dotshape as defined in the inks tab.
- If no ink, or the wrong ink (which cannot be found in the Pantone or Designer book) is selected, then a Normalized PDF file will be generated which contains K with default parameters.
- If the wrong ink is selected, together with the default ink, then a Normalized PDF file will be generated with parameters from the default ink.

Overprint: Select the appropriate overprint from the dropdown list. Choose between Opaque and Darken.

2.7.40 Create XML File

This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.41 Data Collector

This Workflow Control is documented in the chapter "Workflows". See [Data Collector](#) on page 1029.

2.7.42 Data Splitter

This Workflow Control is documented in the chapter "Workflows". See [Data Splitter](#) on page 1030.

2.7.43 Delete File

This task is mostly used at the end of a workflow ticket, where the original input may be deleted since the workflow has created other data that are more worth keeping..

Tip: This task can also be combined with the **Wait** task. For example: first wait 2 hours, then delete the marked file.

Tip: To delete marked files, you can perform the following actions: **Mark File** + any other action + **Select Marked File** + **Delete File**.

2.7.44 Download WebCenter Document

This task is documented in the chapter **Integrating with WebCenter**. Find a direct link [here](#).

2.7.45 Enrich Black

This task automatically creates rich black (super-black) objects. A darker, deeper color is obtained by adding a layer of overprinting objects covering all the pure black areas in a color of your choice. The task can also be used to enrich other inks than process black.

In the example below 80% Cyan is added (as separate layer of overprinting objects) to all areas containing pure black, resulting in an enriched black:



Two tabs:

- General
- Scan (identical to the Scan tab in the **Trap - Prepare and Create Trap layer** task).

General Tab

General | Scan

Add Inks

Cyan 80 %

Magenta 0 %

Yellow 0 %

Black 0 %

Choke Distance: 0,2 mm

Corners: Mitered 2

To

Black

Minimal Density: 100 %

Add Inks




A color containing up to four inks can be added to all colors containing an ink of your choice. Typically 80%C or a more neutral combination of CMY such as 60%C 40%M 40%Y are used for rich black generation. Alternatively PANTONE ink names may be entered in these input fields. When the percentage next to an ink name is set to 0%, the ink will not be used.

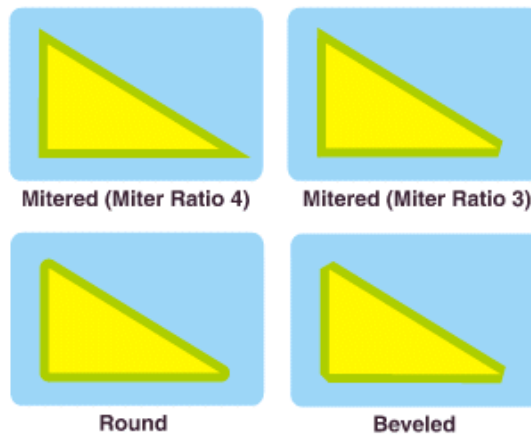
Choke Distance

Defines the distance between the edge of the area in the defined color and the edge of the original (black) object. If this distance is not zero, the generated area in the defined color will be smaller than the original (black) area to prevent registration errors on the press.

Corners

This option allows you to change the way sharp corners in the trap areas will be handled.

- **Mitered:**  If the distance from the base of the trap to the corner point is larger than the miter ratio multiplied by the trap distance then the corner will be beveled, otherwise it will remain sharp. See also miter ratio option. (default)
- **Round:**  A round cap will be placed at all corners.
- **Beveled:** 



Miter Ratio

Determines when a sharp corner is beveled. If the Distance from the base of the trap to the corner point is larger than the miter ratio multiplied by the trap distance then the corner will be beveled, otherwise it will remain sharp. This option is only available when mitered corners are selected (default value 2).

To Ink

Specifies the ink to which the selected color will be added. This ink is typically Process Black. An alternative process or PANTONE ink name may be entered in this input field, to enrich colors containing that specific ink.

Minimal Density

Specifies the minimum percentage of 'To Ink' needed in an area before the specified color will be added to that area. The default percentage is 100%, which means that the selected color will be added to areas containing 100% of the 'To Ink'.

2.7.46 Expand Dynamic VDP data

The **Expand Dynamic VDP Data** task creates PDF/VT files from a database containing variable data, using a VDP template (VDT file).

This task requires the following:

- a CSV database as input, and
- a VDT template as mandatory parameter.

Task Settings

Database File:

- **Database File Encoding:** Specify the character encoding here. Select the correct encoding to ensure accurate interpretation of database entries.

Note: The default encoding is **UTF-8**. This is also the recommended encoding.

- **Field Separator:** specify the separator or let the task identify automatically via **Detect Automatically**. The following separators can be used:
 - , (Comma)
 - ; (Semicolon)
 - Space
 - Tab
- **Field Names:** Field names map data chunks from the database entry to its corresponding VDP template items. You can choose from:
 - **From CSV file:** If this is selected, the task will extract **Field Names** from the first record of CSV file by default.



Attention: The **Field Names** can only contain the following characters:

- lower-case ASCII letters ("a-z")
 - upper-case ASCII letters ("A-Z")
 - digits ("0-9")
 - underscore ("_")
- **From Header file:** When selected, provide a **Header File** containing **Field Names**.
 - **Header File:** You can browse to the CSV Header File or use a SmartName to access the file. The **Header File**:

- should contain a single line specifying the **Field Names**.
- should have **Field Names** corresponding to each column present in the CSV file.
- and the CSV file must use the same delimiter and encoding.

Dynamic VDP:

- **Template:** Here, browse to the VDP Template file containing artwork and the placeholders necessary to generate the task output.


Note: The VDP template can be created and exported using Dynamic VDP DeskPack Plug-in for Adobe Illustrator.

- **Image Path:** If the database file has image references, provide the **Image path**.

2.7.47 Export ArtPro to Normalized PDF File

The Export ArtPro to Normalized PDF File can generate a Normalized PDF file based on an ArtPro file.

Settings for Step: Export ArtPro to Normalized PDF File


 Output in:

File Name:

Preferred Ink Book:

Search for missing images in:

Embed all images

Add non-printable layers

Add fully transparent objects

Folder Mapping:

Client	Server

Preferred Ink Book Select the ink book to look in first when processing inks that are not in the predefined ink books, or for inks that may be in several ink books.

For each ink in the ArtPro file, the task will:

- first look in the predefined ink books (such as the process and PANTONE ink books),
- then in the **Preferred Ink Book** you define here,
- if it still doesn't find the ink, it will check other ink books you created,
- if the ink is not present anywhere in the color database, it will be saved as an unregistered ink.

Search for missing images in: If the ArtPro file references some images that the task can't access, you can copy those images to a folder on one of your Automation Engine containers, and browse to that folder here.

Embed all images Embeds all images referenced in the ArtPro document into the PDF file.

Add non-printable layers

If switched on, layers set to non printing are exported into the PDF file. With this option off, these layers are not output, so the PDF file contains only layers with the Print option on.

Add fully transparent objects

If this option is off, objects that are completely transparent will not be included in the PDF file. If it is on, they are included.

Folder Mapping:

When the ArtPro file contains links to images that reside on a Mac OS-based file server, you can mount the drive of the Mac OS server on the Automation Engine server and use this Folder Mapping table to keep the references to the images working in the Normalized PDF.

To map folders from the ArtPro file to the Normalized PDF file:

1. Click **Add**.
2. In the **Client** column, type the drive name on the Mac OS system.
3. In the **Server** column, type the drive letter that is used to mount the Mac OS drive on the Automation Engine server.

For example, in our ArtPro file, the path to an externally linked image is:
`Intel_Mac:images:image1.`

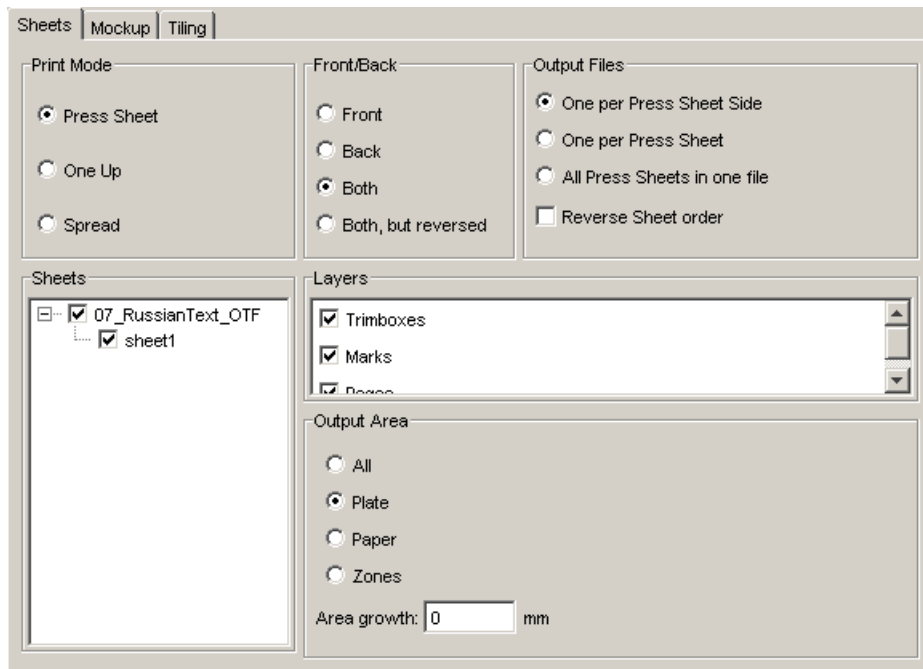
We mount the `Intel_Mac` drive of the Mac OS file server on our Windows-based Automation Engine system, using the `G:\` drive letter.

To allow Automation Engine to automatically locate the images on the Mac OS file server when exporting the ArtPro file, we use the Folder Mapping feature to map `Intel_Mac` to `G:\`.

2.7.48 Export IMP File to GRI File

In the unlikely event that some editing is required, the Export Imposition to GRI File task can generate a .gri file that can be opened in PackEdge.

Sheets



Print Mode

- Select the **Press Sheet** button to create a .gri file and select the appropriate radio button in the Output Area field to specify the output format.
- Select **One Up** to create output of a user-definable size, containing one page per output page.
- Select **Spread** to create output of a user-definable size, containing a printer's spread per output page.

Front/Back

This is only available when Press Sheet is selected. Select the appropriate option to indicate which side of the sheets you want to expose: **Front**, **Back**, **Both** or **Both, but reversed**.

Output Files

You can choose how many output files to generate.

When Press Sheet is selected, you can generate:

- **One** (output file) **per Press Sheet Side**,
- **One per Press Sheet**,
- **All Press Sheets in one file**.

You can also choose to **Reverse Sheet order** to generate the sheets in reverse order.

When either One Up or Spread is selected, you can generate:

- **One** (output file) **per Proof Sheet Side**,
- **One per Proof Sheet**,
- **One per Section**,
- **One per Booklet**,
- **One per Book**.

Sheets

This is only available when Press sheet is selected. Select the sheets you want to expose by checking the appropriate boxes. If you want to select all sheets, check the file.

Layers

- Check the **trim boxes** box to expose the trim boxes layer, so that you will be able to see where the sheets will be trimmed.
- Check the **Marks** box to expose the layer containing all marks.
- Check the **Pages** box to expose the layer containing the pages.

Output Area

- **All** outputs all elements, even those that are situated outside the plate.
- **Plate** outputs all elements within the plate format.
- **Paper** outputs all elements within the paper format.
- **Zones** outputs all elements within the signature format.

Area growth: enter a positive or negative value in the input box to enlarge or reduce the output area formats mentioned above. Select the appropriate unit from the dropdown list.

Pages

This is only available when One Up or Spread is selected.

- Check the **All** toggle to select all pages.
- Enter the correct **Range** of pages.
 - **All pages in range** prints only the pages indicated.
 - **Include front and back** prints the selected pages and the corresponding front and back sides.

Paper

This is only available when One Up or Spread is selected.

Specify the appropriate paper width and height. When nothing is entered, the size of the file is used when sent to an output device.

Other Settings

When a one-up is selected, you can enter a **BackBleed** value to be able to check the bleed in the spine of the book for one-up proofs.

Checking the **Show ImposeProof numbers** box puts a page number on the recto side of the page.

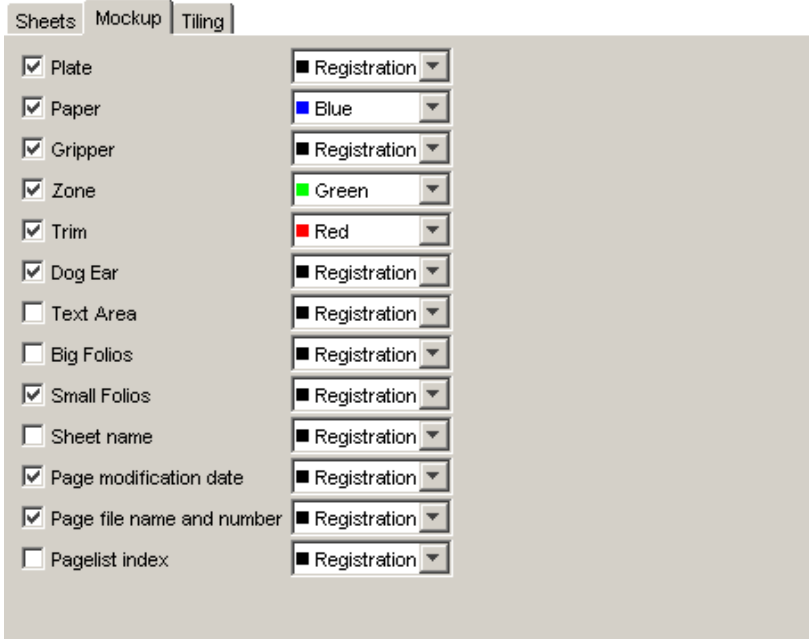
Check the **Skip administrative cover** box if you do not want to have a cover printed, containing administrative information about the job.

Checking the **Optimize for OutRight** toggle makes sure that all pages are incorporated in the file.

Note: All data that is outside the clipping masks will be clipped before being processed. This results in better performance and smaller files.

Mockup

Check the elements you want to be visible on output and select the appropriate mockup color from the dropdown list to enhance visibility.

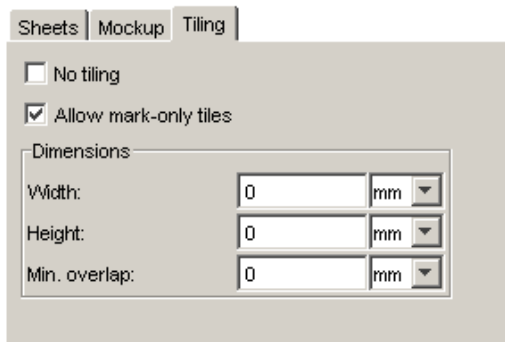


Tiling

If the job is too big to fit the selected paper/plate size, it may be interesting to opt for a tiled output. The job will then be exported to a series of smaller pages/plates (tiles).

Select **No tiling** if no tiled output is required.

Deselect all toggles to obtain a tiled output, omitting tiles that contain only marks. To include these kind of tiles, select **Allow mark-only tiles**. Enter width and height of the tiles, and a certain amount of overlap.



2.7.49 Export IMP File to JDF Layout

The JDF Export task converts IMP files to JDF files, referring to PDF pages.

This task combines elements of the Export to PDF File tasks and the Export Imposition to GRI File task.

JDF version: JDF 1.2 or JDF 1.3

Pages format: PDF or Normalized File. If you select 'Normalized File' all PDF related tabs will be greyed out.

This task consists of six tabs:

- The Output tab is identical to the [Output tab](#) of the Export to PDF File task.
- The Rasterize tab is identical to the [Rasterize tab](#) of the Export to PDF File task.
- The [Compression tab](#) is identical to the Compression tab of the Export to PDF File task.
- The [Advanced tab](#) is identical to the Advanced tab of the Export to PostScript task.
- The [Sheets tab](#) is identical to the Sheets tab of the Export Imposition to GRI task.
- The [Mockup tab](#) is identical to the Mockup tab of the Export Imposition to GRI task.

2.7.50 Export IMP File to JDF Post-Press

The Export to JDF PostPress task creates JDF for cutting and saddlestitching, starting from an imposition file. The resulting JDF file will be interpreted directly by cutting machines (like for example Wohlenberg or POLAR-Mohr) and binding machines (like for example the Muller Martini Amrys SaddleStitcher or Heidelberg StitchMaster). The JDF capabilities of finishing equipment vary significantly depending on the supplier and the machine model. Contact your supplier for more information.

The aim is to output in JDF all information that can be extracted from prepress and that serves to optimize the production workflow downstream.

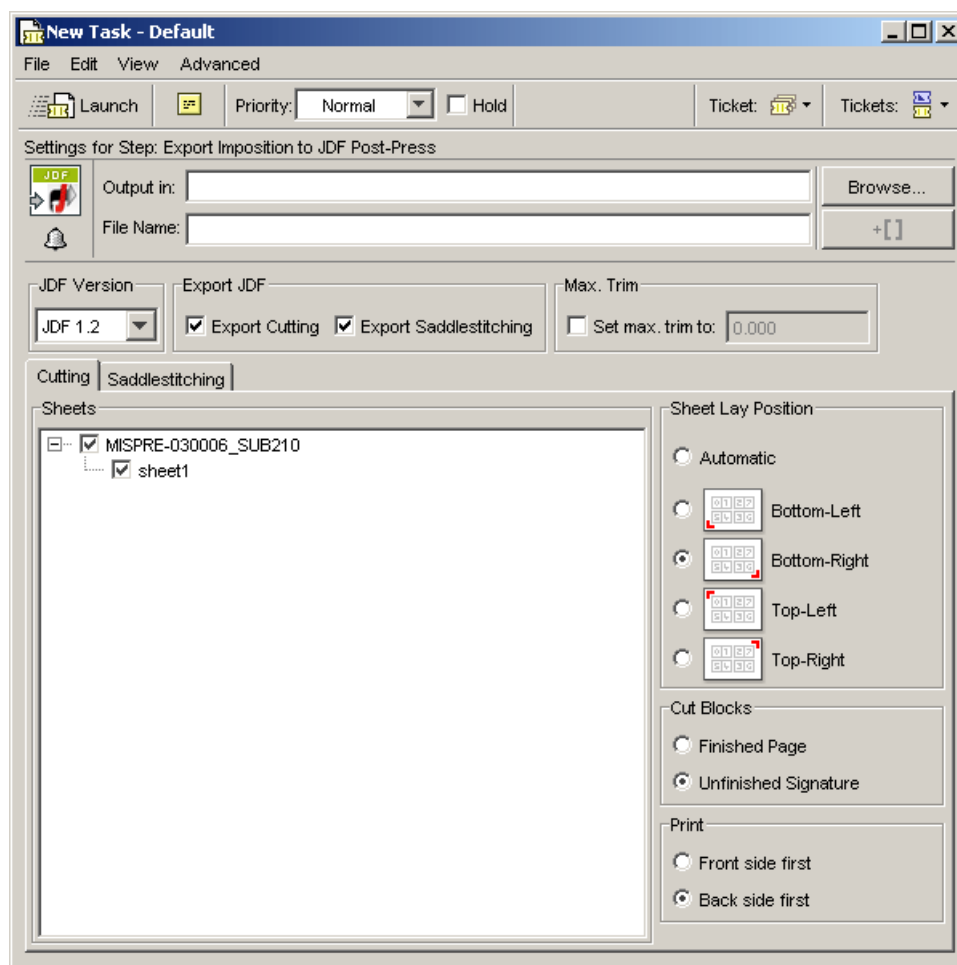
JDF Version: Select whether you want to generate a JDF version 1.2 or 1.3 file.

Export JDF: Indicate what data should be included in the JDF file: cutting, saddlestitching or both.

Maximum Trim: If the distance between the edge of the paper and the cut line is smaller than or equals the trim indicated then no cutting will take place.

Cutting

The JDF file generated by the task contains a set of Cut Blocks that the cutting machine needs to cut out of a sheet of paper.



Sheets: allows you to indicate which sheets must be exported.

Sheet Lay: determines how the paper is placed in the feeder. You should indicate the reference edge of the paper relative to the front side of the sheet as seen in FastImpose.

- **Automatic:** determines the Sheet Lay by looking at the location of the Side Guide mark. If there is exactly one side guide mark found its location is used. When there is no Side Guide or more than one Side Guide, the Sheet Lay cannot automatically be determined, and consequently an error will be generated.

- **Not Automatic:** allows the user to select one of the four corners of the paper as Sheet Lay.

Cut Blocks: indicate where the cutting machine will have to cut, on the finished page size of each cell of the signature or on the unfinished signature. Cutting on the finished page size of each signature cell is useful when individual elements are combined as flat work, for example postcards or posters.

Print: indicate which side will be printed first, front or back.

Saddlestitching

The JDF file generated by the task contains the number of signatures that need to be bound, their size, overfold and number of pages. It also contains information for stitching and trimming.

Saddlestitch

- **Staple Shape:** Crown, Overlap, Butted, ClinchOut, Eyelet
- **Stitch Width**
- **Wire Gauge**
- **Wire Brand**
- The **Position** boxes allow you to indicate the positions of the stiches. Positions are measured starting from the trimmed side of the Registered Jog Edge. Stitches will be centered on these positions.

Registered Jog Edge

The Jog Edge is the reference edge of the folded signatures. This is the side that is used to align the different signatures to.

- **Automatic:** the smallest of the FastImpose Head and Foot is used as Jog Edge.
When the head and foot trims are equal, the FastImpose head is used as Jog Edge.
- **FastImpose Head**
- **FastImpose Foot**

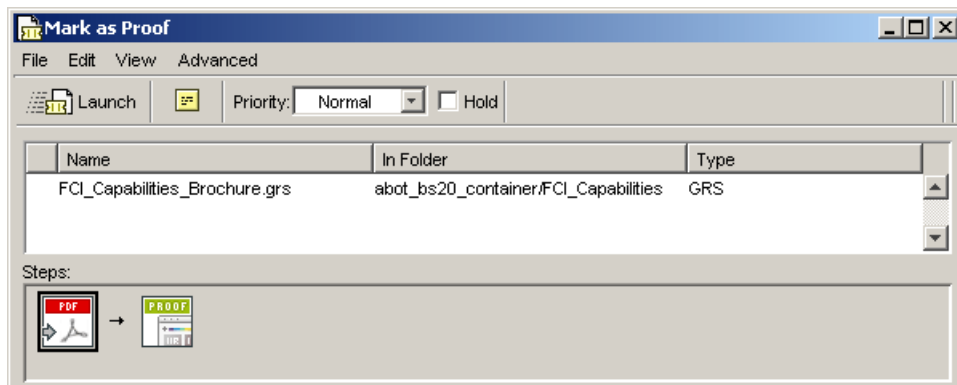
2.7.51 Export IMP File to PDF File

This task combines elements of the Export to PDF File task and the Export Imposition to GRI File task. See [Export to PDF File](#) on page 331 and [Export IMP File to GRI File](#) on page 313.

2.7.52 Export IMP File to PDF File Chain

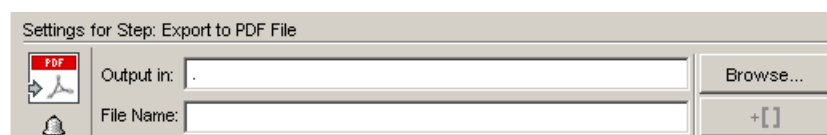
This task chain ticket consists of the [Export IMP File to PDF File](#) on page 320 ticket, linked to the [Mark as Proof](#) on page 376 ticket. The result of the action is a PDF file that can be checked in the Proofs View.

Note: This task is used only in page workflows.



Depending on where the output file name is specified, in the PDF step or in the Proof step, a different naming mechanism for the Proof Sets is triggered.

PDF step



When a task is launched from the Pages or Plates View, the default proof file name is **proof_[File]**.

Mark as Proof step



The default proof set name is: **Output file name. extension.**

Use SmartMarks to build up the name or enter it manually.

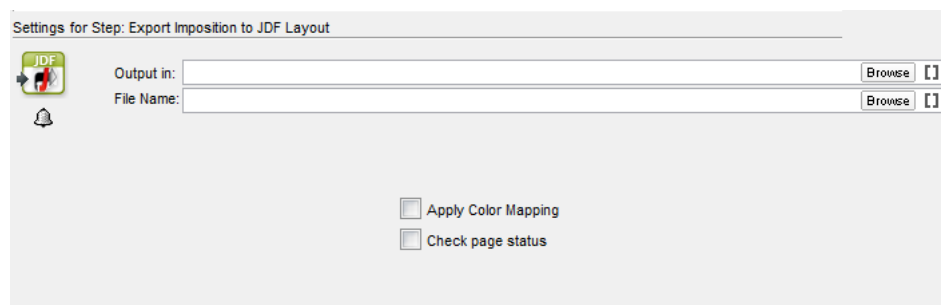
In the Details pane you'll see the original name of the file, including extra information on trim boxes, side, etc.

2.7.53 Export IMP File to PostScript File

This task combines elements of the [Export to PostScript File task](#) and the [Export Imposition to GRI File task](#).

2.7.54 Export Imposition to JDF Layout

Using this task, you can export Imposition, RunList, Layer, Plate and Page to JDF Layout and JDF Runlist format. This task will send an Imposition or a part of an Imposition to a workflow that requires JDF Layout as input. The JDF Layout contains a JDF RunList which you can use in the **Join Pages** task to produce a multipage PDF book for getting approval.



Apply Color Mapping: You can enable this to export the layout file with the same color mapping as defined in the **Plates** view.

Check page status: You can use this option to check if the approval status of the pages is compliant with the **Job setup** settings. The task will fail if it is not compliant.

2.7.55 Export Imposition to PDF File

This task exports Imposition, RunList, Layer, Plate and Page formats to PDF format.

Settings for Step: Export Imposition to PDF File

PDF icon, Bell icon

Output in: Browse []

File Name: Browse []

Number of Flats per File:

Page Label:

Apply Color Mapping

Check page status

Number of Flats per File: You can use this option to specify how you want to split the output file. One sheet side of an Imposition is considered as a Flat.

- If you enter 0 , the output will be one multipage file.
- If you enter 1 , the output will be two PDF files containing front(recto) and back(verso) sides separately. For each sheet in the Imposition, there will be two PDF files like this.
- If you enter 2 , the output will contain front(recto) and back(verso) side together in one PDF file with two pages. This will be the outcome for each sheet in the Imposition .

Page Label: The page labels as they will appear in the output file(s).

Note: You can make use of SmartNames from the categories **Page Information** and **Plate Information** here.

Apply Color Mapping: You can enable this to export the layout file with the same color mapping as defined in the **Plates** view.

Check page status: You can use this option to check if the approval status of the pages is compliant with the **Job setup** settings. The task will fail if it is not compliant.

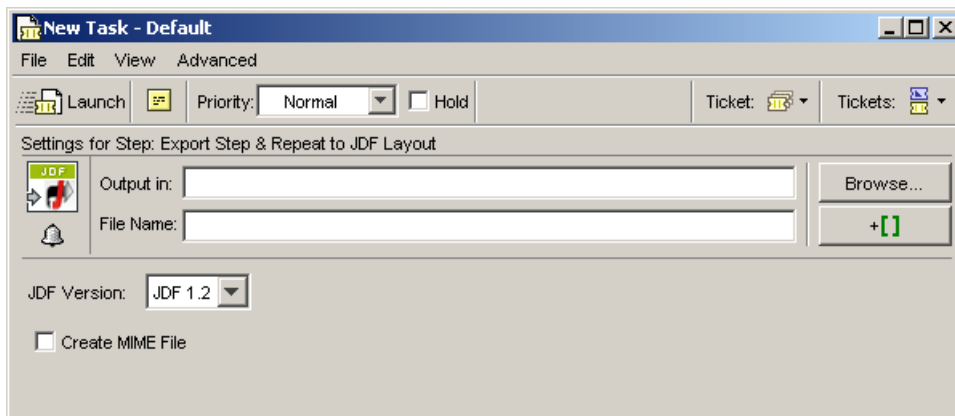
2.7.56 Export Nested Layouts

For detailed information about the **Export Nested Layouts** task, please see [Export Nested Layouts](#) on page 1116 in the Gang Run Printing chapter.

2.7.57 Export Step and Repeat to JDF Layout

This task makes it possible to export a JDF Layout file. Stations, masks and marks are exported to PDF 1.5 files. A JDF file containing references to all these files will be generated. You can select whether you want to generate a JDF version 1.2 or 1.3 file.

When selecting the 'Create MIME file' toggle, the JDF and the referenced PDF files are put together in a MIME packed file.



2.7.58 Export to 3D

This task produces 3D output files which you can view in 3D viewers. 3D viewing helps in

- Quality control
- Better viewing of directions, positioning and visibility of the elements

With this task, you can export a Normalized PDF or PDFSTA (Station file) with a linked structural design (CAD) file into a 3D file. The task accepts multiple input files with support to define one scene. For example, you can give multiple input files belonging to one **Scene** which you can load in the task separately.

- two input PDFs containing the same structural design
- one input PDF with three pages referring to one part of a scene
- three input single page PDFs referring to one part of a scene

This task can create output in the following formats:

- 3D Collada file : this can be uploaded to WebCenter and viewed in the integrated 3D Viewer.
- Tiff image
- PDF with 3D Design: view this as 3D in Adobe Reader or Acrobat Professional (version 8.0.1 or later).
- U3D : it is a common exchange format for 3D objects. In Acrobat Professional 8.0.1 or higher you can place U3D files in PDF files.



Attention: Your input file must have a linked CAD file or the task will fail.

About Structural Design Files

A structural design file (2D and 3D) contains all the cut and crease lines for a specific type of package. A structural design file may also contain folding angles that define how the package is to be folded into its final shape. The structural design file does not contain the graphics, only the structural lines.

To bring the structural design and the graphic design together, a structural design file can be placed or opened in either:

- Adobe Illustrator using the Esko Data Exchange plug-ins,

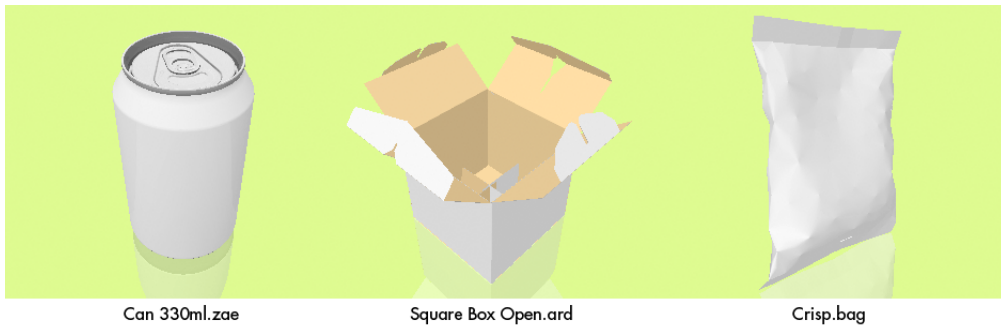
Note: You can get the Data Exchange plug-ins for free at <http://www.esko.com/>.

- ArtPro,
- PackEdge.

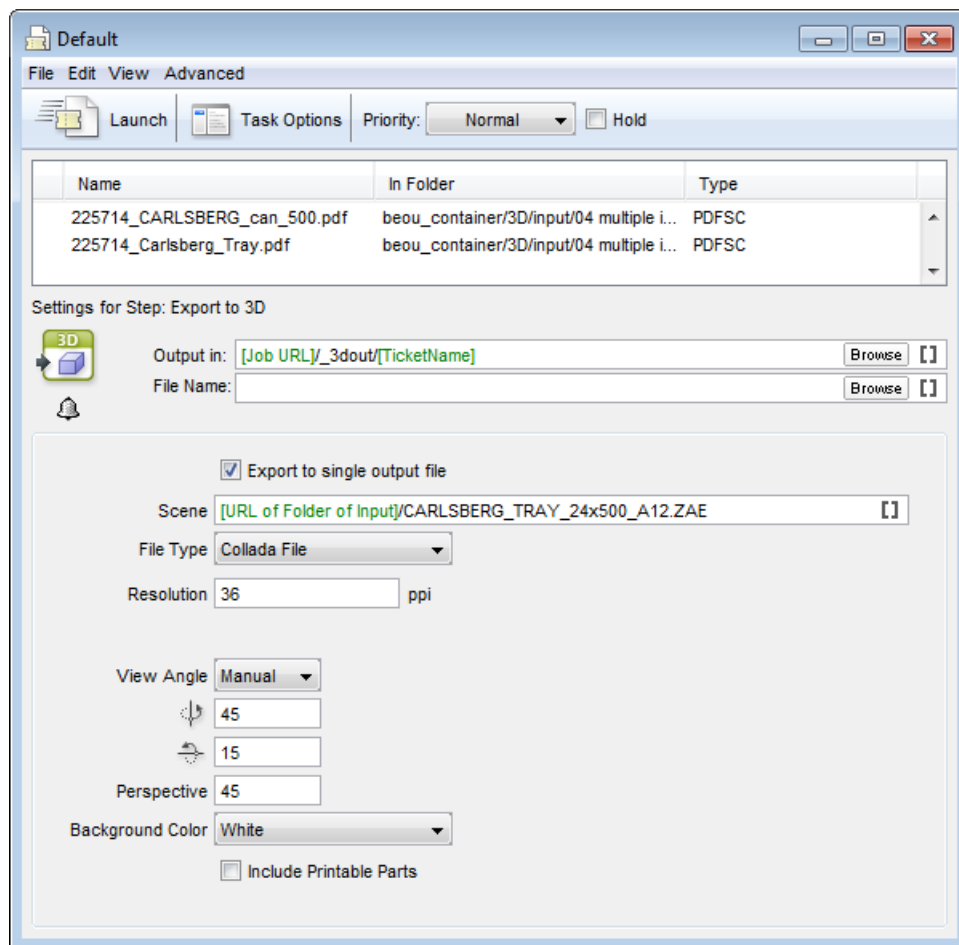
Supported structural design file formats:

- ArtiosCAD or Studio Toolkit for Boxes (.ard files) for boxes
- Score! (.vlm or .scb files) for boxes
- Studio Toolkit for Flexibles (.bag files)
- Collada files with a printable part (.dae or .zae files)

A printable part is an Esko extension to the Collada format. Collada files with one or more printable parts can be created in Studio Designer, Studio Toolkit for Shrink Sleeves, Studio Toolkit for Labels and ArtiosCAD.



Export to 3D Settings



This task will work of multiple file belonging to a **Scene**.

Note: You can create a **Scene** using Esko ArtiosCAD. The Scene will contain links to multiple .ARD files which contains Structural Designs. For example

Exporting to Collada

Choose **Collada File** to export the 3D design to a Collada file.

Specify the **Resolution** of the image.

Define the **View Angle** from which the package should be rendered. You can either inherit the angle from the file (using the view angle the file was saved with in Illustrator, Studio or PackEdge), or specify it manually by entering the angles numerically.

Adapt the **Perspective**. A high value has the effect of a camera with a wide angle lens, close to the object. A low value has the effect of a camera with a zoom lens, further away from the object. The default perspective is 45 degrees.

Note: The object will always fit completely inside the exported image.

Choose a **Background Color: White, Black or Transparent**. For example, choose a transparent background if you want to use the file for a product placement.

Exporting to TIFF

Choose **Tiff Image** to export the 3D design to an RGB TIFF image.

You can define the **Width / Height** of the image that you want to export. The TIFF image will always be square and will contain the whole package.

You can also specify the **Resolution** of the image.

You also need to specify the **View Angle** from which the package should be rendered. You can either inherit the angle from the file (using the view angle the file was saved with in Illustrator, Studio or PackEdge), or specify it manually by entering the angles numerically.

You can also change the **Perspective**. A high value has the effect of a camera with a wide angle lens, close to the object. A low value has the effect of a camera with a zoom lens, further away from the object. The default perspective is 45 degrees.

Note: The object will always fit completely inside the exported image.

Choose a **Background Color: White, Black or Transparent**. For example, choose a transparent background if you want to use the file for a product placement.

Exporting to PDF with 3D Design

Choose **PDF File with 3D Design** as **File Type**.

You can define the **Width, Height** and **Resolution** of the PDF file.

The **Show Footer** toggle allows you to switch the **Footer Text** on or off. Instead of the default Esko text you can personalize the footer text to include your own message. You can also use SmartNames.

As when exporting to TIFF, you can use the **View Angle** saved in the file or enter it manually, and change the **Perspective**.

You can choose a **White** or **Black Background Color**.

Exporting to U3D File

Choose **U3D File** as **File Type**.

You can set the U3D file's **Resolution, View Angle** and **Perspective**.

You can choose a **White** or **Black Background Color**.



Using Prepare Station with Export to 3D

In a workflow, you can connect the **Prepare Station** and the **Export to 3D** tasks to get both a one-up file for your Step and Repeat, and a 3D preview of the final result (box, bag, can or bottle).

- As input, you need your design in Normalized PDF format (you can export Normalized PDF from Illustrator using the DeskPack plug-ins), with a link to your CAD file (see [Export to 3D](#) on page 323 for a list of supported CAD files).
- As output, you will get a PDFSTA one-up, collada (ZAE) and a 3D file in TIFF, PDF or U3D format.

For more information, see [Prepare Station](#) on page 417 and [Export to 3D Settings](#) on page 324.

2.7.59 Export to ArtPro File

This task allows you to export a Normalized PDF file to the ArtPro file format.

Version: you can export your file as:

- an **ArtPro 10.x/9.x/8.x** file,
- an **ArtPro 7.5/7.0** file.

ArtPro 7.5/7.0 files do not contain version-specific information or blend modes (the file will be flattened).

Copy linked files next to output file: select this to copy the linked images next to the ArtPro file created.

2.7.60 Export to EPS / DCS File

The EPS Export ticket has five tabs:

- **Output**
- Color Management is the same as the *Color Management tab* of the PostScript Export task.
- Rasterize is the same as the *Rasterize tab* of the PostScript Export task.
- Compression is the same as the *Compression tab* of the PostScript Export task.
- Advanced is the same as the *Advanced tab* of the PostScript Export task.

Note: The names of multiple DCS files consist of the name of the original EPS DCS file followed by _C, _M, _Y, _K (for process ink plates) or _1, _2, _3, ... (for all other kinds of inks). The files have .x as extension.

Output

PostScript version: choose between PostScript 3 or PostScript level 2.

Output type: choose between an EPS, a Single DCS and a Multiple DCS file.

EPS: This will create an EPS file with an RGB preview. When the selected PostScript version is PostScript 3 and the color management toggle in the Processing Tab is switched off, this will be a deviceN EPS, in all other cases a CMYK composite EPS will be created.

Single DCS file: This will create one EPS DCS file containing multiple separations. The DCS file always contains an RGB preview used when placing the EPS DCS file. In addition a composite grayscale or color CMYK preview may be added used for composite printing.

Multiple DCS file: This will create a master EPS DCS file referring to multiple separations (one file per ink). The names of these files consist of the output filename followed by the name of the ink. The master DCS file always contains an RGB preview used when placing the EPS DCS file. In addition a composite grayscale or color CMYK preview may be added used for composite printing.

- EPS
- Single DCS file, no composite
- Single DCS file, grayscale composite
- Single DCS file, color composite
- Multiple DCS file, no composite
- Multiple DCS file, grayscale composite
- Multiple DCS file, color composite

Preview Resolution

The default resolution is set on 72 ppi. This option can change the resolution of the RGB preview used when placing the EPS DCS file. A higher value will produce a finer, more detailed preview but will also increase the files size of the EPS DCS. The minimum value is 72 ppi.

Resolution

The default resolution is set to 4.000 ppi. If you do not know where the EPS will be placed, you might want to make a high resolution file. Mind, however, that the higher the resolution, the bigger the file size of the EPS will become.

Page Number: To select the page of the job you want to export.

Scale: allows you to scale using percentages.

Add **SmartMarks** sets by selecting one from the drop-down list.

Account for job margins: selects the margins of the job as the true boundary. Normally the borders of the job are taken as the true boundary of the job. Sometimes, however, you may want to convert everything that is situated within the margins too.

Hide Trap Layer in output: allows you to export your file with or without the trap layer.

2.7.61 Export to Normalized PDF File

This task converts a GRS file to a Normalized PDF file. A GRS is an old Esko file format.

Note: This task is only available when you are working in a Normalized PDF job folder.

ESKO

Output in: Browse...

File Name: +[]

Version: Normalized PDF (Esko 7)

External References
External references are exported together with the input file

- Write local external references to the output folder and other external references to their original folder
- Write all external references to the output folder
- Write all external references to their original folder

When converting again:

- Reconvert if recently updated
- Use previous conversions
- Always reconvert

Embed External Images

Embed Images:

- All images (1-bit, 8-bit and native)
- Images with uncompressed data size smaller than 640 kB
- None

Copy Other Images to Output Folder:

- Only images which are close to the input file
- All images
- None

Fail when fonts are not found

Fail when external references are not found

Close to the input file' means: in a subfolder or a neighboring folder

External References

External references have to be converted, you can define what the system needs to do.

- 'Write local external references to the output folder and other external references to their original folder'.

Upon exporting, external references in subfolders or neighboring folder will be written to the output folder. External references in for example central folders will be written in their original folder.

- Write all external references to the output folder.
- Write all external references to their original folder.

When converting again, you can opt to:

- Reconvert if recently updated: only updated references will be reconverted, all other conversions will be used as they were.
- Use previous conversions: the existing conversions will be used, potential updates will not be considered.
- Always reconvert: all external references will be converted at all times.

Embed External Images

Referenced external files in GRS format can be converted to referenced files in Normalized PDF file format. Indicate which images you want to have converted.

1. All images
2. Images with uncompressed data size smaller than 640 kB (as specified via Configure > Processing Preferences > Embedded Images). This option is identical to the option in the *Inks tab* of the Normalize PostScript / PDF / Illustrator 8.0 File task.
3. None

Indicate what should happen with the external images. You can have them copied to the output folder, or you can opt to leave them in their original folder.

Fail when fonts are not found

When the fonts used in the file are not found the task will fail, when the toggle is not selected, the task will end with a warning.

Fail when external references are not found

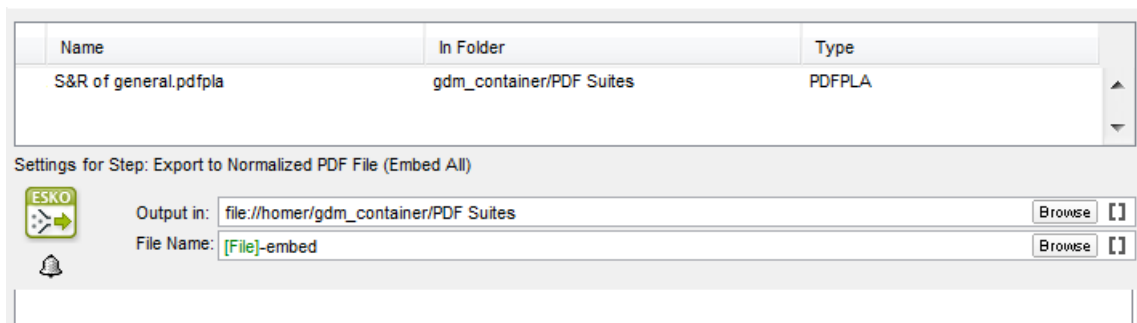
The task will fail when not all external files were found.

2.7.62 Export to Normalized PDF File (Embed All)

This task embeds all the external references, images and fonts to a Normalized PDF file. You can use this task to include external references and links in a PDFPLA or Normalized PDF file.

review:	Data Type:	External References:				Units: ▾ mm	Resolution: ▾ ppi		
		File Name	Type	Available	H Resol...	V Resol...	Page Nr	Usage	Occurre...
	ArtWork								
	External References	general.pdf	PDFSC				1	station	4
	Inks								
	Levers								

You can specify the output file name and location in the ticket settings.

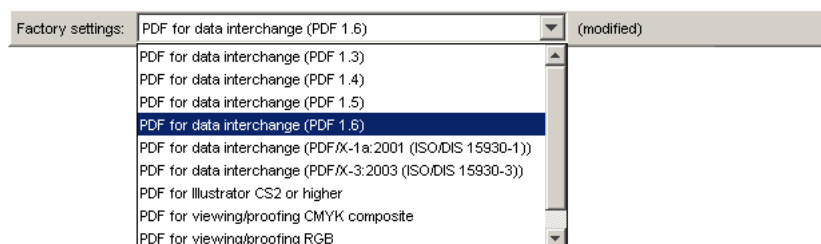


The output will be one Normalized PDF file with all the external references, fonts and images embedded.

Note: In the **Export to Normalized PDF** task you can embed the images, but the external references in a normalized PDF file is exported together to the specified location. Read more in [Export to Normalized PDF File](#) on page 329

2.7.63 Export to PDF File

We advise to use one of the following factory settings. Using a factory setting does not mean that you cannot modify the settings:



These settings are a quick way to define your ticket. There are seven settings, select one from the drop down list:

- **PDF for data interchange (PDF 1.3):** High compatibility but slow and less editable.
 - Acrobat 4 and lower
 - Ripping
- **PDF for data interchange (PDF 1.4):** Interchange with modern applications, well editable. For example Acrobat 5 and more recent.
- **PDF for data interchange (PDF 1.5):** Interchange with modern applications, well editable for example Acrobat 6 (and higher).
- **PDF for data interchange (PDF 1.6):** Interchange with modern applications, well editable for example Acrobat 7 (and higher).
- **PDF for data interchange (PDF/X-1a:2001 (ISO 15930-1)):** PDF/X-1a is a standardized subset of PDF designed specifically for reliable prepress data interchange. PDF/X-1a ensures complete exchange using CMYK and spot color data. Uses: ripping and imposition
- **PDF for data interchange (PDF/X-3:2003 (ISO 15930-3)):** PDF/X-3 is a standardized subset of PDF designed specifically for reliable prepress data interchange. PDF/X-3 ensures complete exchange suitable for coloured managed workflows. Uses: ripping and imposition

- **PDF for Illustrator CS2 or higher:** PDF file meant for editing in Adobe Illustrator CS2 (or higher). Note that this mode improves the editability in Adobe Illustrator, but does not guarantee full editability.
- **PDF for viewing / proofing (CMYK Composite):** Useful when using hard copy proofers, or when the result doesn't have to be editable or when the result is used for softproofing. It is not useful to serve as final output.
- **PDF for viewing / proofing (RGB):** Useful for softproofing (smaller file, and it's faster to generate).
- **PDF for viewing/proofing DeviceN:** Interchange file for proofing/viewing. Adobe Acrobat Reader 7.0 or later is required to view overprint settings correctly.

If you change the predefined settings, the user interface will indicate that the Factory Settings are modified.



Note:

If you define a ticket using the Factory Settings, and you save it, the Factory Settings option disappears.

If you overwrite a Default ticket, the Factory Settings will be gone as well.

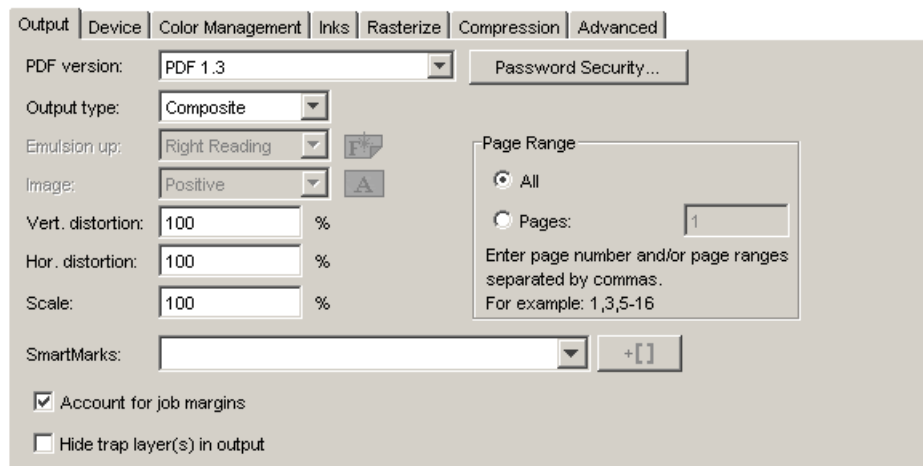
Note:

Even though Acrobat 7 Professional now offers the possibility to define an ink as Opaque ("Ink Manager") for Trapping reasons it has no influence on the preview.

The PDF Export ticket has seven tabs:

- **Output,**
- **Device,**
- **Color Management,**
- **Inks** (similar to the Inks tab in the Export to PostScript File task, see [Inks / Separations](#) on page 346),
- **Rasterize** (similar to the Rasterize tab in the Export to PostScript File task, see [Rasterize](#) on page 347),
- **Compression,**
- **Advanced.**

Output



PDF Version

Select a PDF version from the list.

- **PDF for Illustrator:** Select this version if you want to open the generated PDF 1.4 file in Adobe Illustrator CS2 or higher. Because Adobe Illustrator only supports a (undocumented) subset of PDF 1.4 features some objects and/or colorspaces must be split up, transformed and/or pre-ripped in order to correctly represent them in Illustrator.
- **PDF/X-1a:2001 (ISO 15930-1)**
- **PDF/X-3:2003 (ISO 15930-3)**
- **PDF 1.6: the Acrobat 7 file format**
- **PDF 1.5: the Acrobat 6 file format**
- **PDF 1.4: the Acrobat 5 file format**
- **PDF 1.3: the Acrobat 4 file format**

Password security

Allows you to make your file password protected. For more information, please see [Password Security](#) on page 334.

Output type

- **Composite:** Generates a composite file, which if no color management is used produces a deviceN pdf for data interchange. DeviceN means colors can be combinations of the standard CMYK process colors and/or spot colors, guaranteeing perfectly separable PDF files which can also be proofed on a composite PostScript 3 device. The generated PDF file will proof correctly and separate correctly. Color management for spot color objects must be handled by the device.
- **Separated:** Activate this option to generate separations. Multiple pages will be generated, corresponding to the inks in the job.

Scale

Define the scale in percentages. You might want to scale your job to make it easier to softproof via e-mail.

Select **SmartMarks** from the dropdown list.

Put marks with ink coverage on a hardcopy proof (or view it in Acrobat).

Account for job margins

To select the margins of the job as the true boundary. Normally, the borders of the job are taken as the true boundary of the job. Sometimes, however, you may want to convert everything that is situated within the margins too.

Hide trap layers in output

Toggle this option on if you do not want the trap layers included in your output.

Page Range

- Output **All** pages.
- Output a number of **Pages**. Enter page numbers or page ranges separated by commas, for example 1,3,5-8.

Password Security

The options in the Password Security window depend on the PDF version you chose in the Output tab.

Click the Password Security button in the Output tab (Export to PDF task), the Password Security window opens:

Compatibility: the compatibilities offered here depend on the PDF version you chose in the Output Tab.

Encryption level: Low (40-bit RC4)

Require a password to open the document: enter a password.

Use a password to restrict permissions: enter a password.

Printing Allowed

- Select 'None' to prevent users from printing the file.
- Select 'low resolution (150 dpi)' to give users permission to print the file at a resolution of 150 dpi only.(Acrobat 5.0 and later)
- Select 'High resolution' to give users permission to print the file at high resolution only.

Changes allowed

- Select 'None' to prevent users from doing anything with the file, including filling in signature and form fields.
- Select 'Inserting, deleting and rotating', lets users insert, delete, and rotate pages, as well as create bookmarks and thumbnail pages. This option is available only if a high encryption level is selected. (Acrobat 5.0 and later)
- Select 'Any except extracting pages' to let users do anything to the document except extract contents.

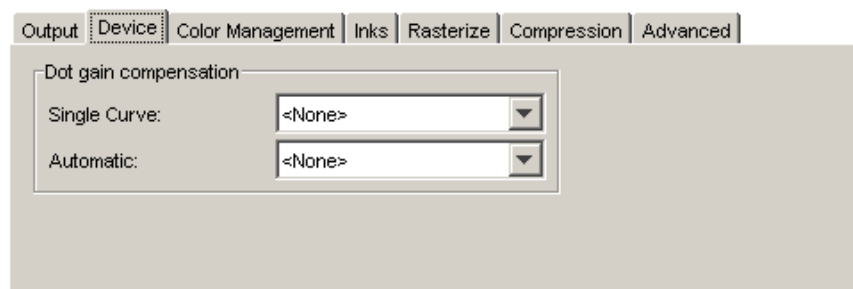
Enable copying of content: Lets the user select and copy the contents of the PDF document. This option is available only if the Compatibility option is set to Acrobat 5.0 and later.

Enable Text Access: (Acrobat 5.0 and later) Enables visually impaired users to read the document with screen readers. This option does not allow users to copy or extract the document's contents. This option is available only if the Compatibility option is set to Acrobat 5.0 and later.

Device

Make your export PDF device specific by adding a description of the dot gain compensation curves inside that PDF.

Note: These are not calculated, but they are stored as PDF standard transfer curves. Opening the PDF in Acrobat will not show any difference.



Dot Gain Compensation

You can select Dot Gain Compensation (DGC) curves made with IntelliCurve or IntelliCurvePro.

- Single curve: Allows you to select one DGC curve (.dgc file) that will be applied to all your data.
- Automatic: Allows you to select a Screen-Based DGC table (.scredgc file) or a DGC Strategy (.icpro file). The former allows you to apply dot gain compensation selectively depending on the dot shape, screen ruling, angles and even Contone or Linework properties. The latter does the same but also offers ink-based dot gain compensation.

Note:

You can also apply both single curve and automatic DGC simultaneously, resulting in a cumulation of both effects.

Color Management

The screenshot shows the 'Color Management' dialog box with the following settings:

- Color management
- Color management: CMYK
- Use Link Profile: <Automatic>
- Source profile: crom_offs
- Destination profile: crom_offs
- Rendering intent: Absolute (background)
- Map black on black
- Use the document profile as Source profile if available
- Embed the Destination profile as an intermediate color space
- Embed the document profile as PDF Output Intent

Convert to

- CMYK: all inks are converted to CMYK values.
- RGB: all inks are converted to RGB values.
- Gray: all inks are converted to Gray values.

Use link profile: Link profiles describe a transformation from 1 device-dependent color space to another. This is actually a concatenation of a forward transformation of 1 profile and a backward transformation of another profile. This means that all other options will be grayed out as the Link Profile takes complete care of the color management.

Source profile: If any CMYK objects are to be converted, choose a source profile. The list shows all CMYK print profiles in the color database. The source profile defines the target that the destination profile is supposed to match.

Destination profile: Choose a color profile for your destination color space. The list shows all print profiles in the color database. These profiles are either measured in Color Engine Pilot or imported ICC-profiles.

Rendering intent

- Relative (No Background): Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- Perceptual: This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact

color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.

- **Absolute (Background):** Use this option if you want background simulation and you want an exact color match, including the paper simulation.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

Black Mapping: When toggled on, the CMY-part of the composite colors are converted separately from the black component. As a result, the type of black separation (Skeleton, UCR, GCR...) is preserved by the conversion. (It is still possible that the black percentage changes slightly due to different dot gain in the color profiles.) When toggled off, the converted CMYK colors are completely re-separated into the black behavior that is built into the destination profile.

Use the document profile as source profile if available: If a document profile is defined, then that one will be used as a source profile. The one chosen from the Source Profile dropdown list will be overruled.

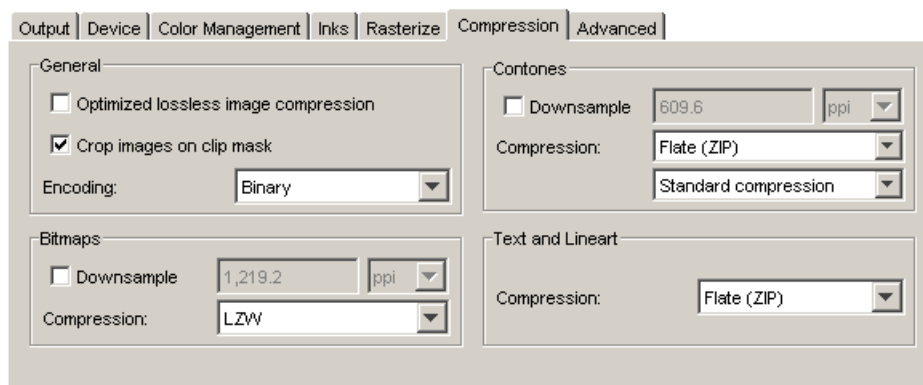
Embed the Destination profile as an intermediate color space

If this toggle is off, the colors in the PDF file will not be tagged with an ICC color profile. The document is device dependent and its colors will only be reproduced accurately if the PDF file is output on the device that is specified in the Destination Profile. This choice is preferable if we are sure the PDF file will be sent to the device specified in the Destination Profile.

If this toggle is on, the color in the PDF file will be tagged with an ICC color profile. The document then becomes device independent, and its colors will be reproduced accurately even on devices with a color space different from the one specified in the Destination Profile. In this case, the Destination Profile is an intermediate colorspace between the Source Profile and the color profile of the device that will eventually process the PDF file. This method is preferable when the PDF file will be sent to an unknown device.

Embed the document profile as PDF Output intent: The output intent is an ICC profile that specifies the intended output device. If this option is toggled on, the document profile will be saved as the PDF output intent. For certain PDF flavors (PDF-X/3) an output intent is mandatory and this option is always on. If no document profile was set explicitly, the default document profile (crom_offs) is used instead. If the document profile is a Color Engine Pilot profile, it will first be converted to an ICC profile.

Compression



General

Optimized lossless image compression

Selecting an overall lossless image compression method seldom results in the smallest overall file size. This option will use the smallest lossless compression method for every image separately (JPEG2000, Flate, FlateDiff, LZW, LZWDiff, CCITT3, CCITT4 or RLE) depending on the image type and the allowed lossless compression methods for the chosen PDF version. When activating this option the compression methods for Contones and Bitmaps are disabled and the encoding is set to Binary. Please do not overrule this option unless you have good reasons to choose another encoding method, otherwise the resulting file will become larger.



Attention: Activating this option will slow down PDF export considerably, so use it only for file size critical jobs.

Check **Crop images on clip mask** to crop the image(s) on the clip mask. All information located outside the clip mask will be lost.

Encoding

- Binary: The image data is represented as binary data. This results in compact files that however are not editable in a common ASCII editor. Nor can these files be sent over a communication line using a communication protocol that relies on control characters in the data stream itself.
- Hex: In this way, the PostScript file remains editable with an ASCII editor at all times. They also can be sent over a communication line using whatever communication protocol is available. The drawback, however, is that hexadecimal image data may take up to twice as much size as binary data.
- Ascii 85: Encodes binary data in an ASCII base-85 representation. This encoding uses nearly all of the printable ASCII character set. The resulting expansion factor is 4:5, making this encoding more efficient than hexadecimal.

Bitmaps

Downsample The images referenced in your design are subsampled approximately to the specified value. Activating this option, allows you to reduce the size of the generated PDF file. Select the unit for subsampling from the dropdown list: lpm or ppi. Please be aware that 1-bit copydot Images cannot be downsampled.

Compression

- None
- CCITT group 3 and CCITT group 4
- Run Length
- LZW
- LZW Diff
- Flate (ZIP)
- Flate (ZIP) Diff

Contones

Downsample The images referenced in your design are subsampled approximately to the specified value. Activating this option, allows you to reduce the size of the generated PDF file. Select the unit for subsampling from the dropdown list: lpm or ppi.

Compression

- None
- LZW
- LZW Diff
- JPEG Compressions

- JPEG2000
- Flate (ZIP)
- Flate (ZIP) Diff

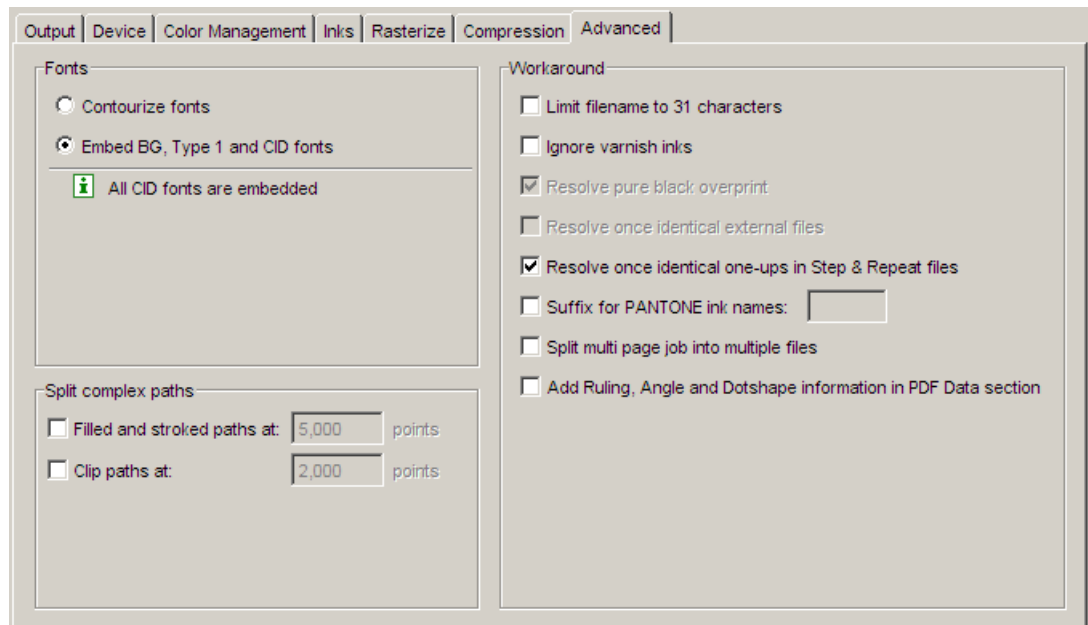
When selecting JPEG, an extra dropdown list becomes activated. It allows you to select the quality of the JPEG compression that needs to be used: from Limited over Standard to Extreme. When selecting JPEG2000, the dropdown list offers the following compression types: lossless, limited, low, standard, high or extreme compression.

Text and Linart

Compression

- None
- LZW
- Flate (ZIP)

Advanced



Fonts

- **Contourize fonts**

Note: Protected CID fonts will be referenced by name.

- **Embed BG and Type1 and CID fonts:** Allows you to include the used BG and Type 1 fonts of your design in your PostScript file.

Split Complex Paths

Note: These are workarounds for typical PostScript level 1 path limitations. Only use these when **absolutely** necessary.

- **Filled and stroked paths at X points** Use this option to split linework with more than x points into several smaller linework parts. This option decreases the complexity of the linework. A suitable value for the selected device type is calculated for this option.
- **Clip paths at X points** Use this option to split linework masks for images with more than x points into several smaller linework parts. This option decreases the complexity of the masks.

Workaround

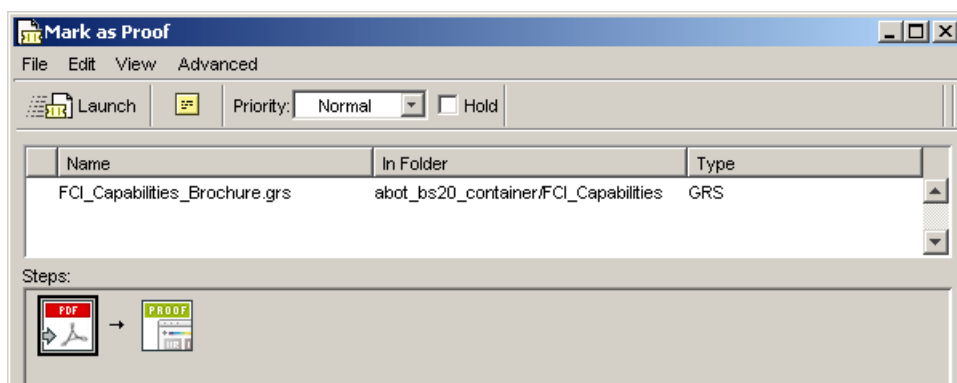
- **Limit filename to 31 characters** Use this toggle if you want to make sure your imported files can be seen/read from/by all Apple operating systems.
- **Ignore varnish inks** Varnish inks will not be included while outputting.
- **Resolve pure black overprint.** Overprints are normally resolved, but by default, pure black is left untouched. If you want Rich Black to be resolved too, select this option. This option is only available when Color Management in the Processing tab is activated.
- **Resolve once identical external files** Instead of resolving the complete file and all its external references, identical external files will be resolved only once.
- **Resolve once identical one-ups in Step & repeat file** Instead of resolving the complete Step & Repeat file, identical one-ups will be resolved only once.
- **Suffix for PANTONE ink names** Enter a suffix for PANTONE ink names. The default is C.
- **Split multi-page jobs into multiple files** Every page of a multi-page job will be output as a separate PDF file.
- **Add ruling, angle and dotshape information in PDF Data section**
- **Add Ink Coverage data in XMP**

Uncheck this to not save the ink coverage data in the XMP section of the exported PDF file (calculating the ink coverage might be time consuming and slow the PDF export down).

2.7.64 Export to PDF File Chain

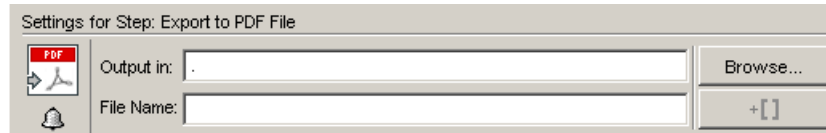
Note: This task is used only in page workflows.

This task chain ticket consists of the [Export to PDF File](#) on page 331, linked to the [Mark as Proof](#) ticket. The result of the action is a PDF file that can be checked in the Proofs View.



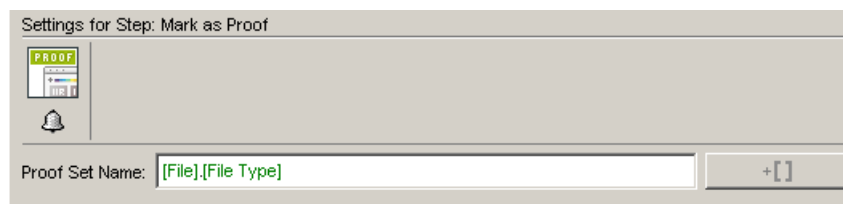
Depending on where the output file name is specified, in the PDF step or in the Proof step, a different naming mechanism for the Proof Sets is triggered.

PDF Step



When a task is launched from the Pages or Plates View, the default proof file name is `proof_[File]`.

Mark as Proof Step



The default proof set name is: **Output file name. extension.**

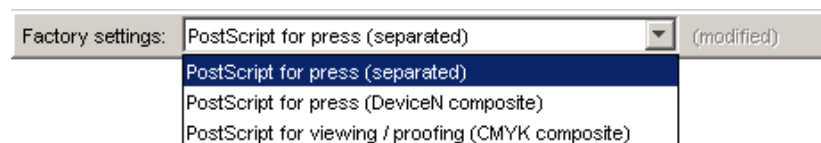
Use SmartMarks to build up the name or enter it manually.

In the Details pane you'll see the original name of the file, including extra information on trim boxes, side, etc.

2.7.65 Export to PostScript File

The **Factory settings** supplied are a quick way to define your ticket.

We advise you use one of these settings (but you can modify them if needed).



- **PostScript for press (separated)**
- **PostScript for press (DeviceN composite):** This output mode (only supported by PostScript 3 devices) uses the PostScript DeviceN colorspace to produce high-fidelity colors.

These colors are combinations of the standard CMYK process colorants and/or spot colors, guaranteeing perfectly separable PostScript code that can be proofed on a composite PostScript 3 device.

- **PostScript for viewing / proofing (CMYK composite)**

If you change the predefined settings, the user interface will indicate that the Factory Settings are modified.



Note: If you define a ticket using the Factory Settings, and you save it, the Factory Settings option disappears. If you overwrite a Default ticket the Factory Settings will also be gone.

Output

Device type: To select the output device you want your design to be exported to or printed to, the corresponding PPD is chosen and will be used.

Output type: To export or print your design as CMYK Composite, Separated or Composite.

- CMYK Composite: Activate this option to generate a composite PostScript file for proofing.

Note: Remark that a CMYK composite PostScript file will proof correctly but separations will not be correct since all spot color are converted to CMYK.

- Separated: Activate this option to generate separations. Multiple pages will be generated, corresponding to the inks in the job.
- Composite: this output type can only be selected for PostScript 3 devices since they support deviceN colorspace to produce high-fidelity colors.

These colors are combinations of the standard CMYK process colorants and/or spot colorants, guaranteeing perfectly separable PostScript code which can also be proofed on a composite PostScript 3 device.

The generated PostScript file will proof correctly and separate correctly. Color management for spot color objects must be handled by the device.

Emulsion Up: To obtain Right Reading (= emulsion up) or Wrong Reading (=emulsion down).

Image: To specify Positive or Negative print mode.

Note: Image and Emulsion up becomes unavailable (appears dimmed) when CMYK Composite Output was chosen.

Enter the **Vertical** and/or **Horizontal** distortion in percentages.

Select **SmartMarks** from the dropdown list.

Account for job margins: To select the margins of the job as the true boundary.

If you deselect this toggle, the borders of the job are taken as the true boundary.

When you want to output everything that is placed within the margins too, leave the option activated.

Hide trap layer(s) in output: Trap layers will not be included in the output.

Page range

- Output All pages.
- Output a number of pages. Enter pagenumbers or page ranges separated by commas. For example 1,3,5–8.

Scale & Tile

- Enter a **Scale** factor (percentage).
- **Fit job on page** scales the job so it fits the selected paper size.
- **Shrink oversized jobs to paper size.**
- Select '**Tile Job**' to obtain a tiled output if the job is too big to fit on the selected paper size.

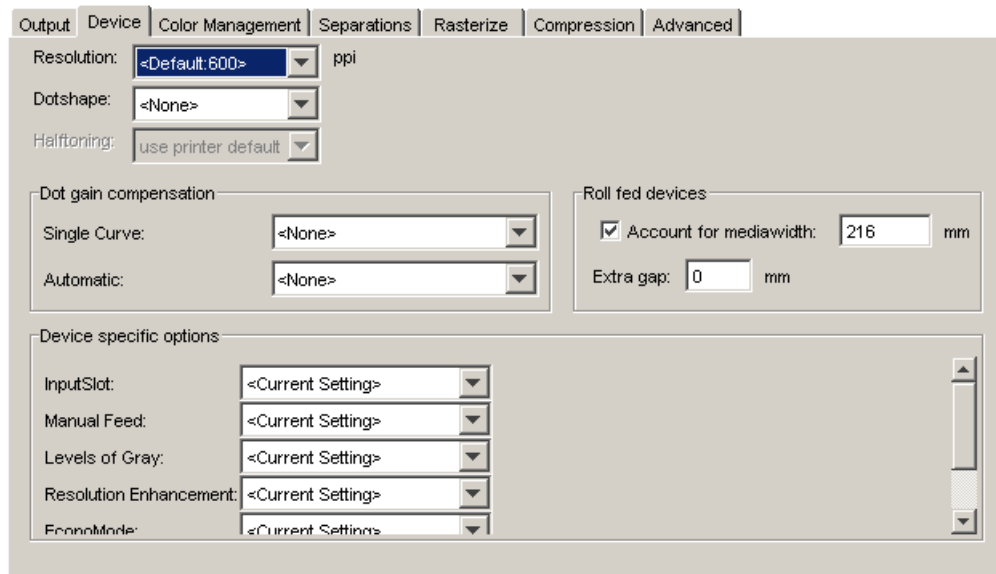
The tiling option which can be used to export a large job to a series of smaller pages (tiles) allows the user to cycle through the tiles which will be produced and see just how many tiles there are.

Overlap X mm/inches: To specify a tiling overlap. The value you enter here, is the amount of overlap in all four directions between adjacent tiles. It defaults to 5 mm.

Page Setup

- Select a **paper size** from the drop-down list. The list that appears here is based on the page sizes supported by the selected ppd.
- **Orientation:** Auto, None, 90° cw, 90°ccw, 180°. The orientation options are disabled when 'Tile job' is selected.
- **Alignment:** Bottom left, Top left, Top right, Bottom right, Centered. The alignment options are disabled when 'Tile job' is selected.
- Enter the number of copies.

Device



The **Resolution** depends on your output device and on the purpose of the PostScript file. For proofing for example, a lower resolution might be sufficient.

Select the **Dot shape**.

Halftoning: If you select CMYK Composite or Composite as Output type, you can choose to include screen information when outputting (lpi and angle).

- **Use Printer Defaults:** the output device's default screen information will be used.
- **Take over from job**
- Some devices offer other dotshapes if they are available in the PPD.

Dot Gain Compensation: You can select Dot Gain Compensation (DGC) curves made with IntelliCurve or IntelliCurvePro to be added as transfer curve.

Note: A screen-based DGC file was the format written by the two DGC Editors (DGC Editor and Screen-based DGC Editor). These two editors have been replaced by IntelliCurve, which writes DGC Strategies. However, a screen-based DGC file can still be applied.

- **Single curve:** Allows you to select one DGC curve (.dgc file) that will be applied to all your data.
- **Automatic:** Allows you to select a Screen-Based DGC table (.scredgc file) or a DGC Strategy (.icpro file). The former allows you to apply dot gain compensation selectively depending on the dot shape, screen ruling, angles and even Contone or Linework properties. The latter does the same but also offers ink based dot gain compensation.

Note: You can also apply both single curve and automatic DGC simultaneously, resulting in a cumulation of both effects.

Note: When a DGC curve is selected, the job will be flattened to ensure the correct Dot Gain Compensation is applied to transparent objects. This may result in a larger file.

Roll Fed Devices: Account for Media Width allows the conversion program to find the best printing orientation. For roll-fed devices, mostly imagesetters, a Media Width (film width) can be specified.

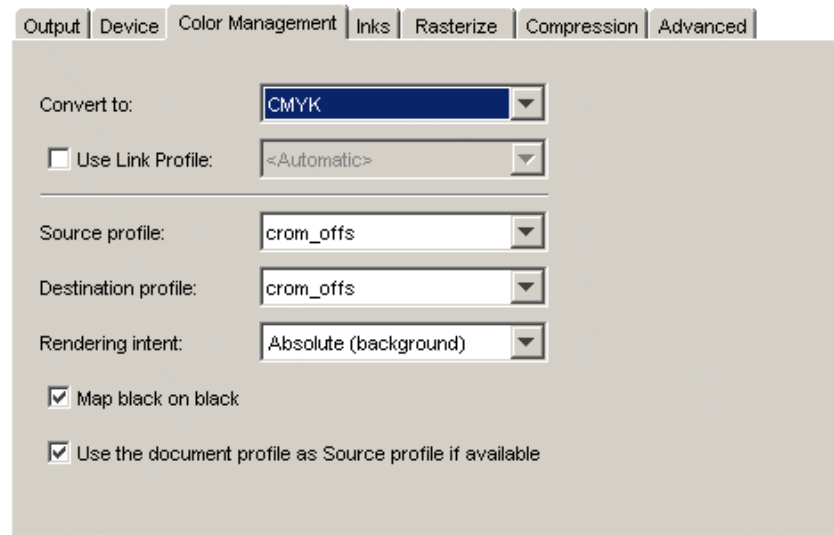
The conversion program finds the best printing orientation in order not to waste paper or file if Rotation is set to Auto.

The Media Width value is expressed in mm.

The option '**Extra Gap**' allows you to specify extra width for the film or paper feed. The Extra Gap value is expressed in mm. By default, no Extra Gap is set.

For **Device specific options** please refer to the manual of your output device.

Color Management



Convert to

- CMYK: all inks are converted to CMYK values
- Gray: all inks are converted to Gray values.

Use link profile: Link profiles describe a transformation from 1 device dependent color space to another. This is actually a concatenation of a forward transformation of 1 profile and a backward transformation of another profile. This means that all other options will be grayed out as the Link Profile takes complete care of the color management.

Source profile: If any CMYK objects are to be converted, choose a source profile. The list shows all CMYK print profiles in the color database. The source profile defines the target that the destination profile is supposed to match.

Destination profile: Choose a color profile for your destination color space. The list shows all print profiles in the color database. These profiles are either measured in Color Engine Pilot or imported ICC-profiles.

Rendering intent

- Relative (No Background): Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- Absolute (Background): Use this option if you want background simulation and you want an exact color match, including the paper simulation.

Map black on black: When toggled on, the CMY-part of the composite colors is converted separately from the black component. As a result, the type of black separation (Skeleton, UCR, GCR...) is preserved by the conversion. (It is still possible that the black percentage changes slightly due to

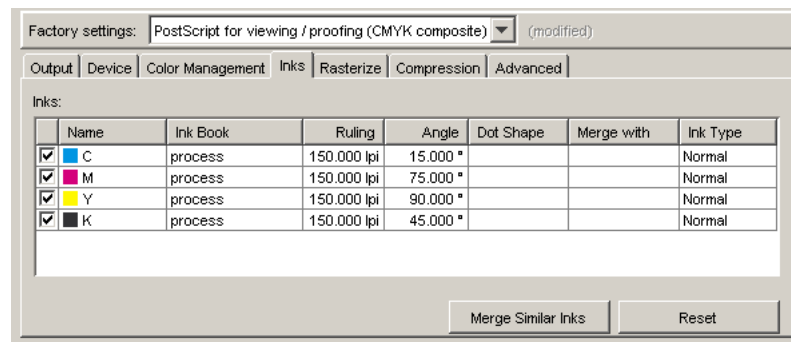
different dot gain in the color profiles.) When toggled off, the converted CMYK colors are completely re-separated into the black behavior that is built into the destination profile.

Use the document profile as source profile if available: If a document profile is defined, then that one will be used as a source profile. The one chosen from the Source Profile dropdown list will be overruled.

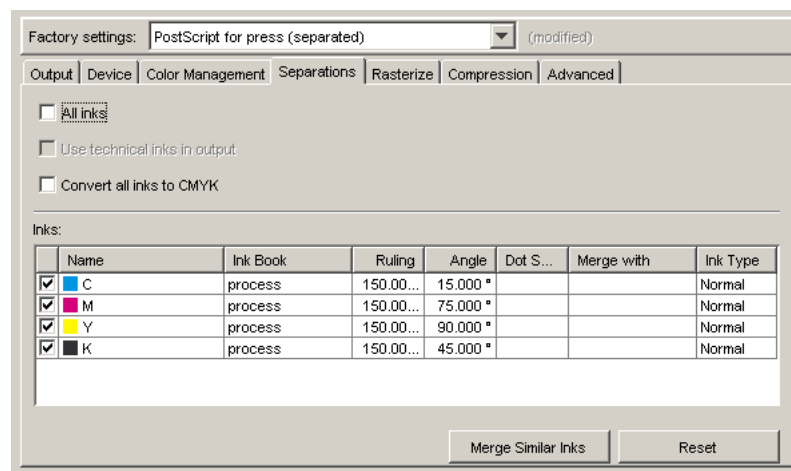
Inks / Separations

The **Inks/Separations** tab depends on the output type you selected in the [Output tab](#).

- If you choose CMYK Composite or Composite, the **Inks** tab will be visible.



- If you choose separated output, the **Separations** tab will be visible.



All Inks: The standard behavior of the default ticket is to offer all inks in the job to be exposed. Deactivating **All Inks** opens an ink-list panel where you can select specific inks. The list contains ink names, ruling, angle, dot shape and ink type.

Note: The Inks panel only shows the inks that are present in the input file(s) for the first task in your workflow. When no files are selected (for example, when building workflows in the Tickets view, you will not be able to “preselect” inks.

Use technical inks in output: Normally technical inks are hidden when you save a job in an Esko editor. Check the box to output the visible (non hidden) technical inks.

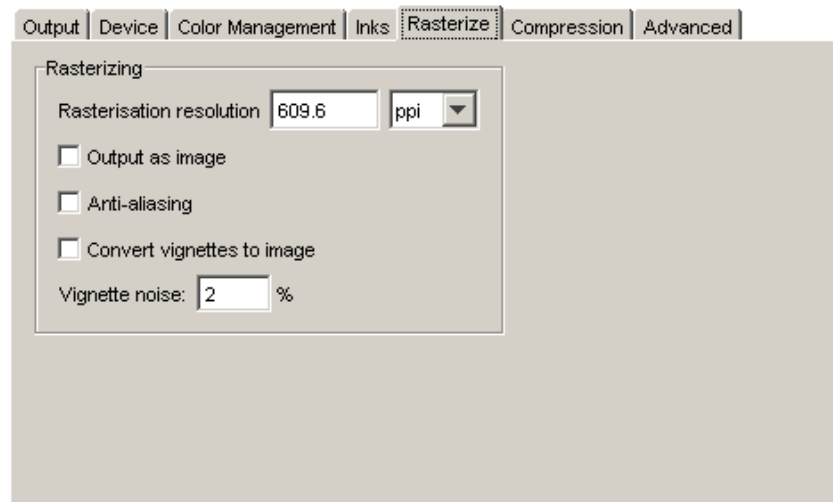
Convert all inks to CMYK: inks will be converted to CMYK.

Use **Merge with** to output inks on top of each other in the same output file. This is common when you use different rulings for 1 ink.

For example, let's assume that the job contains two Cyans, and that they have a different ruling. You might want to put them on the same separation.

You can do so by clicking in the **Merge with** column and selecting the ink it has to be merged with.

Rasterize



Rasterizing resolution: Enter the resolution for the objects that will be resolved.

Output as image: When this toggle is activated an image is made of the job. The resolution entered above will be taken into account.

Anti-aliasing: is relevant for the line-art objects in your design, that need pre-processing and for which images are generated. Anti-aliasing smoothens the staircase effect you get in the generated images of lineart objects that need pre-processing.

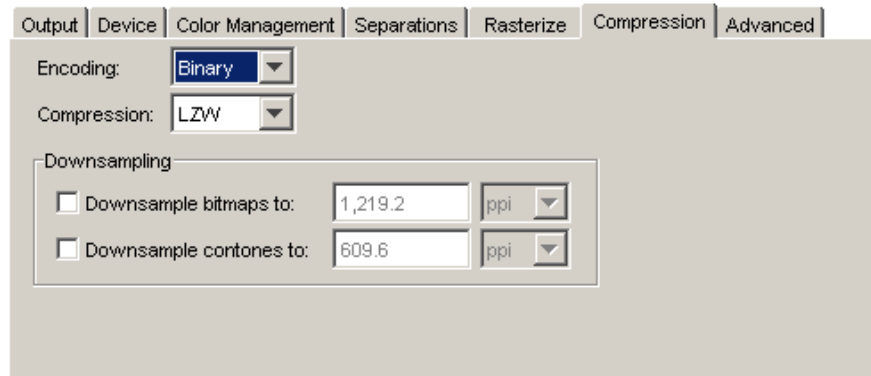
Tip: This option increases the readability of small text that is pre-processed to image.

Convert vignettes to image: To avoid banding in the vignettes when exporting. Exporting vignettes to a PostScript Format, may result in blends and banding due to restrictions in PostScript. However, to obtain a nice result in your export file, activate Convert Vignettes to CT. All the vignettes present in your design will be converted to images.

Vignette noise: When exporting to PostScript, some vignettes or parts of vignettes will be converted to images. This happens when the vignettes overlap other objects with translucency or overprint between the vignette and the other objects.

When the option 'Convert Vignettes to Image' is activated all vignettes are converted to images regardless of overlap with other objects. Vignettes that are converted to images may have tone jumps in the final result. To reduce this banding effect, noise is applied on the image. The amount of noise can be specified in the field Vignette Noise.

Compression



Encoding:

- **Binary:** The image data is represented as binary data. This results in compact files that however are not editable in a common ASCII editor. Nor can these files be sent over a communication line using a communication protocol that relies on control characters in the data stream itself.
- **Hex:** In this way, the PostScript file remains editable with an ASCII editor at all times. They also can be sent over a communication line using whatever communication protocol is available. The drawback, however, is that hexadecimal image data may take up to twice as much size as binary data.
- **Ascii 85:** Encodes binary data in an ASCII base-85 representation. This encoding uses nearly all of the printable ASCII character set. The resulting expansion factor is 4:5, making this encoding more efficient than hexadecimal.

Compression: None, LZW or Flate (PostScript3 devices).

Downsampling The images referenced in your design are subsampled approximately to the specified value. Activating this option allows you not only to reduce the size of the generated PostScript file, but also a quicker generation of the PostScript file.

Select the unit for subsampling from the dropdown list: lpm or ppi. You can subsample both bitmaps and contones.

Please be aware that 1-bit copydot Images cannot be downsampled.

Advanced

Fonts

- **Contourize fonts**

Note: Protected CID fonts will be referenced by name.

- **Embed BG and Type1 fonts:** Allows you to include the used BG and Type 1 fonts of your design in your PostScript file.

Split Complex Paths

Note: These are workarounds for typical PostScript level 1 path limitations. Only use these when really necessary.

- **Filled and stroked paths at X points:** Use this option to split linework with more than x points into several smaller linework parts. This option decreases the complexity of the linework. A suitable value for the selected device type is calculated for this option.

- **Clip paths at X points:** Use this option to split linework masks for images with more than x points into several smaller linework parts. This option decreases the complexity of the masks.

Workaround

- **Limit filename to 31 characters:** Use this to make sure your imported files can be seen/read from/by old Apple operating systems.
- **Ignore varnish inks :** Varnish inks will not be included when outputting to PostScript.
- **Resolve pure black overprint:** Overprints are normally resolved, but by default, pure black is left alone. If you want Rich Black to be resolved too, select this toggle.

This toggle is only available when the Color Management in the Processing tab is activated.

- **Resolve once identical external files:** Instead of resolving the complete file and all its external references, identical external files will be resolved only once.
- **Resolve once identical one-ups in Step & Repeat files:** Instead of resolving the complete Step & Repeat file, identical one-ups will be resolved only once.
- **Suffix for PANTONE ink names:** Enter a suffix, the default is C.
- **Suppress hairlines at clips :** Due to the PostScript limitation of the maximum number of points allowed for a path, complex paths will be cut into smaller pieces.

When the PostScript file is printed/exposed on some older PostScript RIPs, hairlines can become visible. To avoid this problem, activate **Suppress hairlines at clips**.

- **Split multi page job into multiple files:** Every page will be output as a separate file.
- **Image to paper edge:** This moves the origin of the converted job to the lower left corner of the paper or film. Activate this option to align your job to the edge of the paper or film to use the full area.

Note: Exporting to a paper PostScript device: For most of these devices, there is always a border on the output page on which nothing can be imaged. Parts of your design that are located in that border area will be lost.

- **Avoid tabular number problem:** When digits are written as tabular numbers, exporting to some PostScript RIPs may result in bad positioning of the numbers. The tabular numbers are not correctly placed at the text base line.

To avoid this problem, activate **Avoid tabular number problem**.

- **Disable path clipping:** By default, paths are clipped by the borders of the job, but if there are instances where you do not want the paths to be clipped, check this box.
- **Map spot color separations to CMYK:** This option is intended to export a job containing spot colors to pre-separated PostScript for systems which are limited to handling process color separations only (for example a Scitex Brisque Workflow).

This option is available when **Pre-Separated** is selected.

Spot color separations can only be handled on such systems by remapping them to a specific process separation. When this box is checked, spot colors will be exported as a C, M, Y or K separation according to a simple rotating name principle (C->M->Y->K->C->...).

Spot colors which are named exactly after process colors remain unchanged.

- **Omit "Process" For CMYK in %% PlateColor comment:** This option is intended to export a job to a device that does not understand "Process".









If your different separations all have the K (Black separation) Angle and Rulings when output to your Image Setter, you must activate this option.

- **Split separated output into multiple files:** Instead of writing one file containing all separations, the system will write one file per separation.
- **Force Black Overprint:** Select this if you want all 100% black objects to overprint in your PostScript file.

2.7.66 Extract Inks

You can keep ink separations according to the Printing Methods present in the PDF file. The output PDF contains only those inks which are corresponding to the printing methods specified in this task.

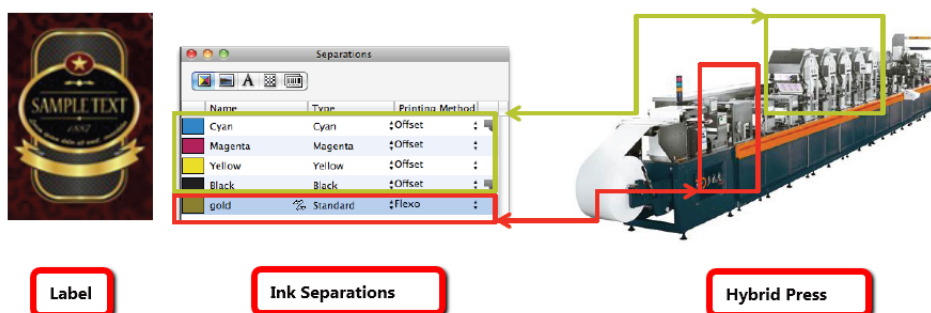
Note: This task extracts inks(keeps inks) in a file according to the printing method/criteria you specify. Therefore, it is essential that it contains the information on the printing methods.

Name	Type	Printing Method
 PANTONE Red 032 C	Standard	↕Flexo
 Yellow	Yellow	↕Flexo
 Cyan	Cyan	↕Flexo
 PANTONE 356 C	Standard	↕Flexo
 Black_Contone	Black	↕Flexo
 Black_Line	Standard	↕Flexo
 PANTONE 199 C	Standard	↕Screen
 Keyline	Technical	↕Unknown

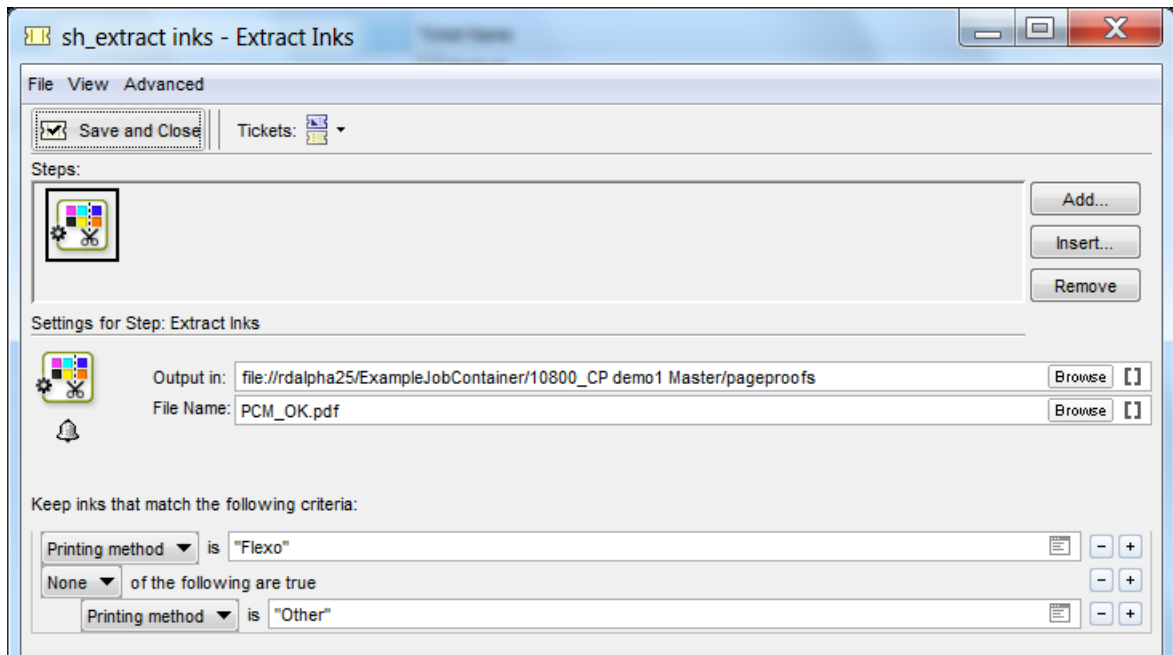
Note:

You can set the printing methods in a PDF file via Automation Engine **Job setup** or in Plato/PackEdge. In the **Inks** panel of the **Job Setup** dialog in Automation Engine, the printing method can be set for each separation. Read more in [How Can the Printing Method of a Separation Be Defined?](#)

You can use **Extract Inks** task while setting up separate processing workflows for individual separations with specific printing methods. This is especially useful in hybrid press industry where press- specific inks can be kept in the output PDF. Another instance where this task is useful is when you want to generate separate reports corresponding to inks. Read more on tasks relating to hybrid press industry in [How Can You Differentiate the Processing of Separations for Different Printing Methods?](#)



You can specify **Printing method** in the field about which the information is pre-defined in the file. You can have multiple printing methods specified by adding fields using +. If you use a combination of **Alt** and ... , you can exclude printing methods.



2.7.67 Fork

This Workflow Control is documented in the chapter "Workflows". See [Fork and Join](#) on page 1032.

2.7.68 Image to MJM File

The Image to MJM File task will pack the JDF that describes the ImageSetting process together with the data files (ripped data) into one .mjm file.

This file can then be easily transported to another site for exposing or archiving purposes.

At the remote site, the exposure can be started by sending this .mjm file to a JDF/JMF enabled device controller using the task as described in [Make Plate on CDI](#) on page 364.

If an .mjm file is selected as input file, the task panel will be empty. This is normal as all crucial information is stored in the .mjm file itself.

JDF ImageSetting

Priority

Priority: Normal

Due Date: Job Due Date Jan 1, 2007 12:00 PM

Plate

Manufacturer: [Plate.Manufacturer] + []

Brand: [Plate.Brand] + []

MediaType: Plate + []

PlateTechnology: FlexoAnalogSolvent + []

Thickness: [Thickness] + []

2.7.69 Image to Screened Separations

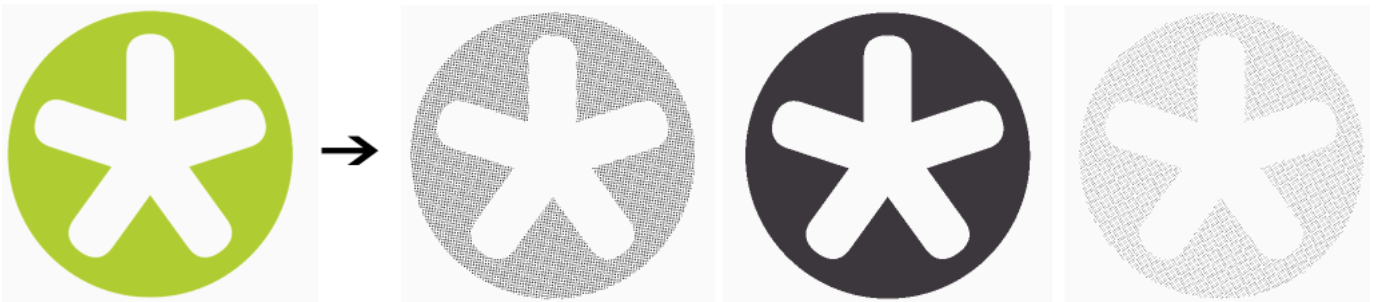
Use this task if you need *screened* output to send to a platesetter or filmsetter (for offset or flexo for example).

Task Output

The **Image to Screened Separations** task generates one file per separation, in either *PDF, TIFF or LEN format*.

For example, you can send screened LEN separations to the CDI.

In the example below, processing a file containing the lime green Esko star generates screened files for the Cyan, Yellow and Black separations.



Task Parameters

In the **Device** field, select the imaging device *that you configured*.

Then use the selector pane at left to show more settings to fill in. See:

- *General* on page 1157,
- *Transformations* on page 1159,
- *PDF Objects* on page 1160,
- *Document Inks* on page 1161,

- [Output](#) on page 1161,
- [Separations](#) on page 1168,
- [Summary](#) on page 1175.

2.7.70 Image to Unscreened Separations

Use this task if you need unscreened output to send to a device that will take care of the screening, or a device that doesn't need screened input (for gravure for example).

Task Output

The **Image to Unscreened Separations** task generates one file per separation, in either [PDF](#) or [TIFF format](#).

In the example below, processing a file containing the lime green Esko star generates unscreened files for the Cyan, Yellow and Black separations.



Task Parameters

In the **Device** field, select the imaging device [that you configured](#).

Then use the selector pane at left to show more settings to fill in. See:

- [General](#) on page 1176,
- [Transformations](#) on page 1177,
- [PDF Objects](#) on page 1178,
- [Document Inks](#) on page 1179,
- [Output](#) on page 1179,
- [Separations](#) on page 1187,
- [Summary](#) on page 1193.

2.7.71 Import Image

The Image Import task is able to convert pixel images to native CT or LP file format. This task can optionally convert RGB images to CMYK for easier processing in workflow. You can also convert CMYK CTs with Inkswitch or BlackSmith.

Mask/Path Settings

Use these toggles to control the conversion of pixel masks and clipping paths. They can be removed or kept during conversion.

- Select the **Keep Pixel Masks toggle** to keep the original pixel mask.
- **Keep the original Clipping Path**, the resulting file will contain the original clipping path.

Preview Settings

- Create a Thumbnail image
- Create a Preview

Keep Indexed

If the colors in the image are reduced (indexed) and you want to keep it like that, use this toggle.

RGB Conversion Method

You may want to convert the image you want to import to a CMYK colorspace. There are two ways to convert to CMYK.

- **Device Link Profile**

This allows you to select a **Process Link**, made in Color Engine Pilot. This option will give you the best result.

Link profiles describe a transformation from 1 device dependent color space to another.

This is actually a concatenation of a forward transformation of 1 profile and a backward transformation of another profile.

This means that all other options will be grayed out as the Link Profile takes complete care of the color management.

- **Device Profile**

This enables you to select a source and a destination (proofer) process.

Use this option if you did not make a Process Link in Color Engine Pilot but you still want a better color result. The process links will be made for you.

RGB Profile: Source profile used for conversion.

CMYK Profile: Destination profile used for conversion.

Rendering intent: Possibilities are:

- **Perceptual:** This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.
- **Relative Colorimetric:** Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

- **Absolute Colorimetric:** Use this option if you want background simulation and you want an exact color match, including the paper simulation.

Note: For BGCMS profiles Perceptual, Relative Colorimetric and Saturation are in fact identical.

2.7.72 Import JDF Layout

The JDF Imposition task handles JDF files generated by JDF compliant imposition programs. It imports the JDF layout into Esko native files.

The JDF file is a low level JDF file, which means that the delivered imposition is a flat imposition and thus no longer editable. Automation Engine will handle this kind of JDF file as a GRx file.

The JDF file contains two parts:

1. A page list, referring to PDF or PostScript files.
2. Geometrical layout info.

The JDF Imposition task converts all PDF and PostScript files and creates 1 GRO file per sheet side.

This ticket's tabs are identical to the Normalize PostScript/PDF/Illustrator 8.0 File ticket's tab, and the Sheets tab differs a little from the Sheets tab of the Export Imposition to GRI File task, but has the same functionality.

Sheets

Indicate of which pages you want to import the JDF layout. For the front as well as the back sheets, you can either select all pages or choose for a range of pages.

The screenshot shows the 'Sheets' tab in the Automation Engine interface. The tab is selected and shows options for 'Front sheets' and 'Back sheets'. Each section has a radio button for 'All sheets' (selected) and a radio button for 'Range (example: 1-5,7,9-11)' with an empty text input field below it.

2.7.73 Import JDF Step & Repeat

With this task, you can generate a Step & Repeat (PDFPLA) file from a JDF file you received, probably from a business (planning) system.

You can launch this task on:

- a JDF file containing a link to a CAD file and PDF station files, to perform CAD-based Step & Repeat,
- a JDF file containing a link to PDF station files, to perform tabular Step & Repeat.

Note: The JDF ticket you use should always contain a list of the input files.

General

SmartMarks

Select your desired SmartMark (created in **PackEdge**) in the list, or enter a SmartMark name. You can use SmartNames to define your SmartMark name.

Load Mark Set in Bottom Layer

Select this if you want to make sure the marks are positioned in the bottom layer, "behind" the graphics.

Limit Bleed to

Use this option to limit the bleed to a specific distance.

Note: The bleed mask is created by spreading the die shape.

Plate Setup

Use Sheet Size from JDF

You can either use the sheet size defined in your JDF ticket, or define a **Sheet Size** yourself.

When defining a sheet (or a plate) size, choose the **Unit** to use: either **mm** or **inch**.

Use Plate Size from JDF

You can either use the plate size defined in your JDF ticket, or define a **Plate Size** yourself.

When defining a plate size manually, you also need to define the **Sheet Offset** and the **Sheet Position on Plate**.

Sheet Offset

Define by how much the sheet should be offset from the plate corner, horizontally and vertically.

Sheet Position on Plate

Position the sheet on the plate by dragging and dropping the small red rectangle (the sheet) to the right position on the big blue rectangle (the plate).

Note: Take into account the point of **Origin** from which the repetition will be built. This is defined in the JDF and is always bottom left.



Shrink

Enter horizontal and vertical shrink values to compensate for printing distortion (especially in Flexo). The default value is 100 both horizontally and vertically (no distortion compensation).

Screen Registration

Select this option to restart the screening for each unit (use this when you need each unit of the Step & Repeat to have the exact same screening, for example in a workflow aimed at printing stamps).



Attention: This setting is only supported by **FlexRIP / Imaging Engine**. It will be lost if you use another PDF RIP on your file.

Tip: You can use SmartNames in all the input fields of this tab. Just click in a field then click the **Insert SmartNames** button.

CAD Options

Mapping

If you are working with DFX, you can map your DFX layers, line styles and color numbers to ArtiosCAD line styles (for example cut lines, crease lines...). You need to create a mapping in Plato or PackEdge, then select it here.

For more information, please see the Plato / PackEdge documentation.

CAD Style

Use this to map the ArtiosCAD line styles of your CAD file (cut lines, crease lines...) to Plato / PackEdge inks. You need to create a CAD style in Plato or PackEdge, then select it here.

For more information, please see the Plato / PackEdge documentation.

Include CAD info for Digital Flexo Suite

If you are using the Digital Flexo Suite, select this option to write technical cutting and creasing information into the header of the output file.

This information will be embedded in the LEN files output by the FlexRIP, and used in the Digital Flexo Suite to help visualizing the output.

Use Blank Position from JDF

You can either use the blank position defined in your JDF ticket, or position the blank on the sheet yourself.

Note: The blank is a representation of the single design that will be stepped & repeated. It is equivalent to the one-up without the artwork (or the bounding box of the one-up).

Position Blank on Sheet

Position the blank on the sheet by dragging and dropping the small red rectangle (the blank) to the right position on the big blue rectangle (the sheet).

Note: Take into account the point of **Origin** from which the repetition will be built. This is defined in the JDF and is always bottom left.

Blank to Sheet Offset

Define by how much the blank should be offset from the sheet corner, horizontally and vertically.

Note: When defining the blank to sheet offset, choose the **Unit** to use: either **mm** or **inch**.

2.7.74 Import JDF Stripping

One of the typical components of a business system (MIS-ERP) is production planning. This is where it is determined how an order (a job) will be produced.

Taking into account for example the presses available, folding and cutting machines, a production plan will be worked out.

This may not necessarily be the cheapest way, but it will certainly be the most efficient way at that particular moment (for example because a certain press is in maintenance, another - less economical press - will be used.)

The JDF file that is sent to the prepress department contains a lot more information than only imposition information.

This valuable imposition information, like for example sheet and signature information will be retrieved from the JDF file, in other words, the file will be stripped.

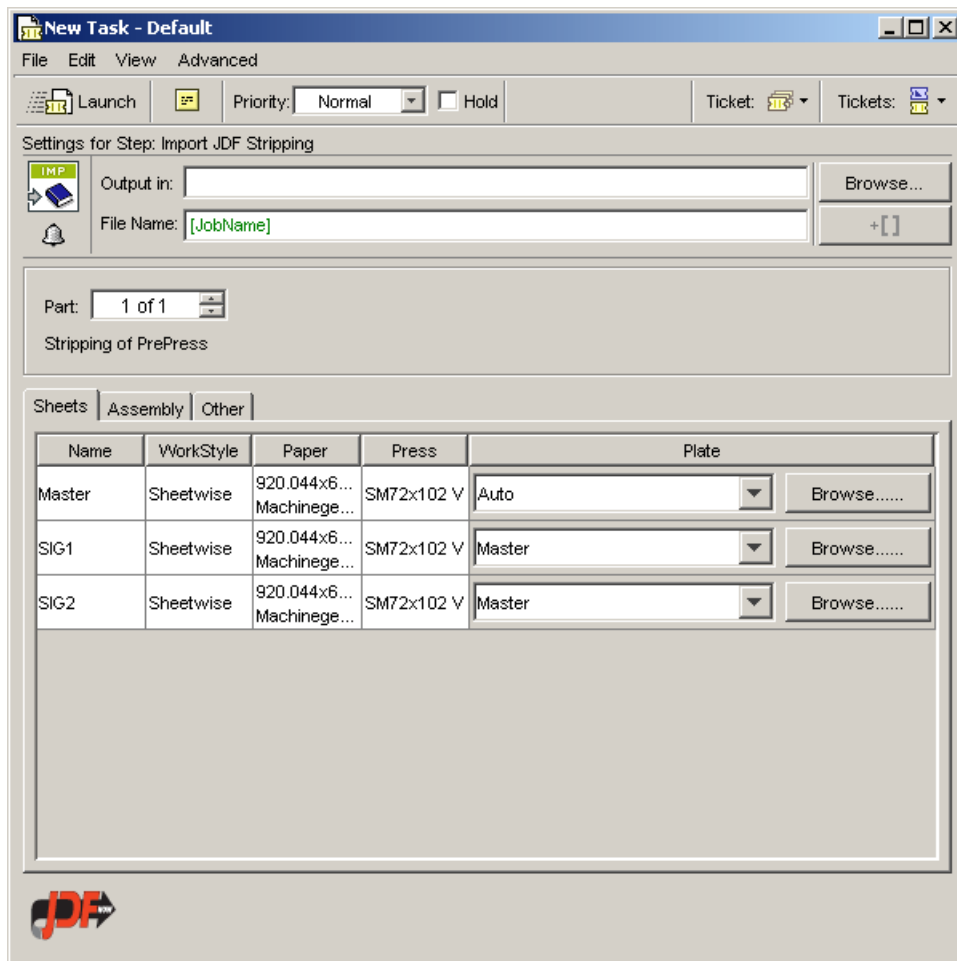
A JDF file can contain several **parts**, for example cover, pages, insert... Each part has to be stripped separately. You can also choose to strip all parts in one go. Select 'All parts from the list'.

The result of the stripping task is an empty imposition (IMP) file.

All signatures and positioning are decided upon, as well as a number of other valuable imposition settings that are not visible in the Stripping task, like for example finished and unfinished page properties.

Note that these parameters are fixed in the JDF file and can therefore not be modified interactively.

The **JDF Stripping** task retrieves all relevant imposition information from the JDF file.



Sheets

- **Name** indicates the name of the sheet.
- **WorkStyle** indicates the work style for the particular sheets: Sheetwise, Work&Turn, Work&Tumble, etc.
- **Paper** shows the dimensions of the paper on which will be printed.
- **Press** indicates the type of press on which the sheet will be printed.

All of the above settings are retrieved from the business system (JDF file from an MIS / ERP).

- **Plate** settings have to be determined by the operator.

If no specific plate is selected, the values for the Master are taken into account, which is **Auto** in the example above.

Auto refers to the value specified for Auto in the [Other tab](#).

Master indicates that the same plate as the one indicated on Master level will be used.

In the example below, the plate SM72x102 V_tumble will be used for both sheets.

Plate	
SM72x102 V_tumble	Browse.....
Master	Browse.....
Master	Browse.....

Assembly

Assembly	
Type	Template
SaddleStitch	Auto

- **Type:** is determined by the business system (MIS / ERP).
- **Template:** The operator has to map the correct template to the assembly type.

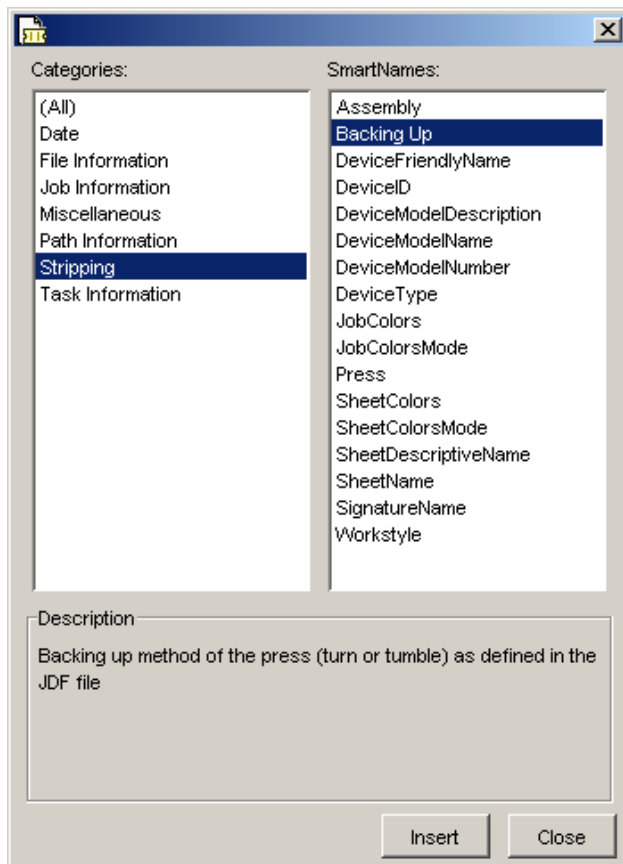
Other

Other	
Auto Rules	
Plate template:	[Press]_ [Backing Up] +[]
Assembly template:	[Assembly] +[]
Positioning of signatures	
Vertical:	Horizontal:
<input checked="" type="radio"/> Expert	<input checked="" type="radio"/> Expert
<input type="radio"/> Center in paper	<input type="radio"/> Center in paper
<input type="radio"/> Use bottom of paper as origin	<input type="radio"/> Use left side of paper as origin
<input type="checkbox"/> Search for ProductID of plates	

Auto Rules

Here you can indicate which plate or assembly template should be used when 'Auto' is selected in the [Sheets tab](#).

Clicking the SmartNames icon +[] opens the list of available SmartNames. Click the SmartNames category to see the SmartNames that have already been created. For each SmartName, a description is given in the description area at the bottom of the dialog.



Positioning of signatures

Indicate how the signature should be positioned on the paper.

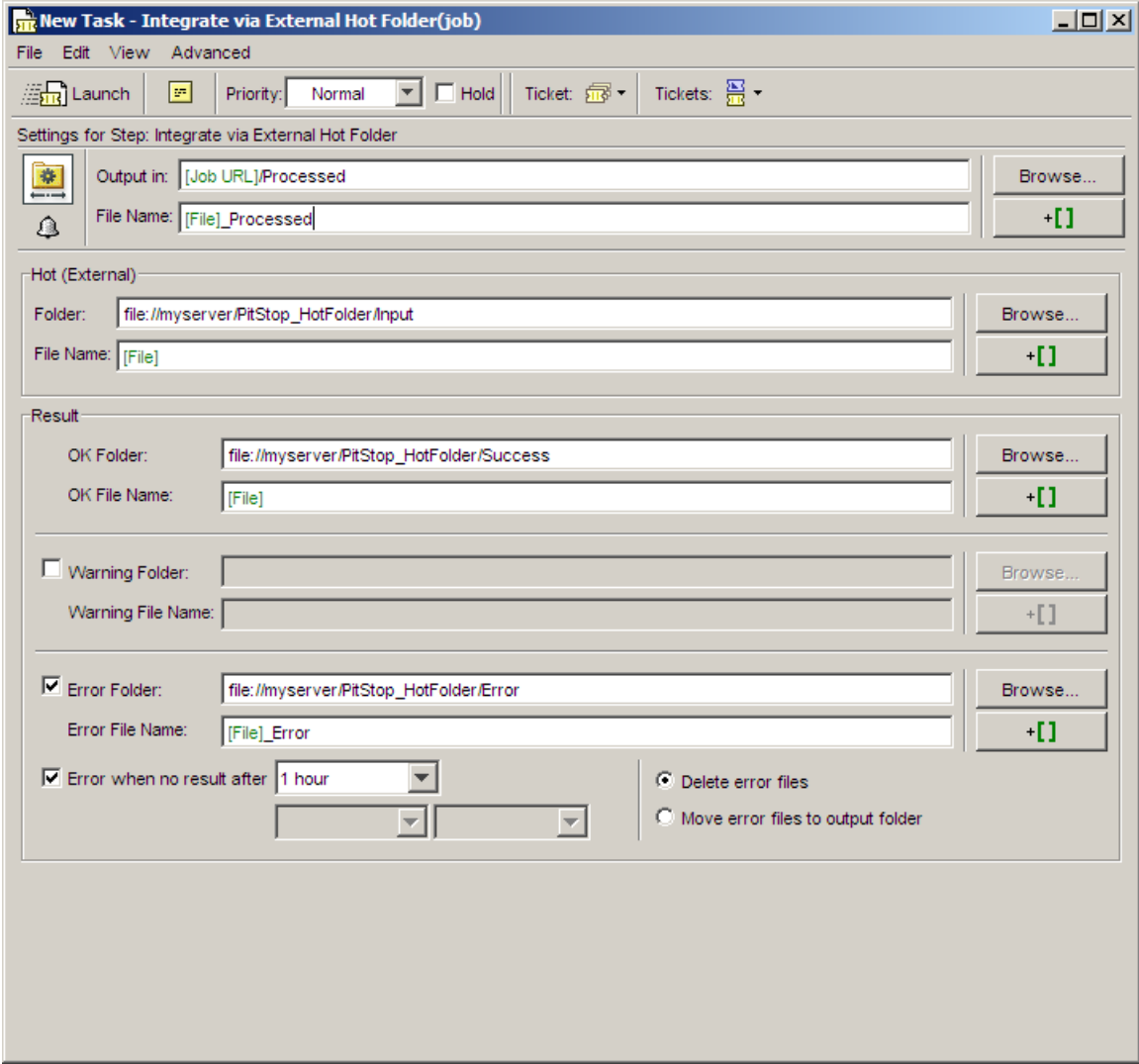
- **Vertical**
 - Expert: positions the signature so that the bleed margin coincides with the gripper in a turn workflow. In case of a tumble workflow, the signature will be positioned in the center of the paper.
 - Center in paper
 - Use bottom of paper as origin
- **Horizontal**
 - Expert: In case of a horizontal positioning, the Expert option is identical to the Center in paper option.
 - Center in paper
 - Use left side of paper as origin

Search for productID of plates: check this toggle if you want to have the JDF file searched for Plate Product IDs. You can then use this Plate Product ID as a text or barcode mark on the plate.

2.7.75 Integrate via External Hot Folder

The **Integrate via External Hot Folder** task allows you to integrate external prepress applications into your workflow tickets. The only requirement is that the external system works with hot folders.

1. In the Integrate via External Hot Folder ticket, you define the location of the hot folder used by the external application, as well as the Result sub-folders in which the external application will output processed jobs (the **OK Folder**), failed jobs (**Error Folder**) or jobs that produced warnings (**Warnings Folder**).
2. When you process a job with this ticket, Automation Engine copies the files to the external hot folder.
3. The external application picks up the files in its hot folder and processes them. The processed files are saved in the sub-folder that matches the processing result: OK, Warning or Error.
4. Automation Engine monitors the OK, Warning and Error folders for any files that have been processed.
5. If it detects a file in the OK folder, Automation Engine copies it to the Output folder defined in the Integrate via External Hot Folder ticket, and continues its workflow. Depending on your settings, Error or Warning files might also be copied into the Output folder, or they might be deleted.



Note:

In the **Error** area of the ticket, you can define an amount of time after which the external processing should be considered as failed. To do this, enable the option **Error when no result after ...** and define a time period (for example, you could enter one hour, a number of days, or even a specific date).

2.7.76 Integrate with WebCenter

This task is documented in the chapter **Integrating with WebCenter**. Find a direct link [here](#).

2.7.77 Interact with Database

This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.78 Interact with Web Service

This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.79 Join

This Workflow Control is documented in the chapter "Workflows". See [Fork and Join](#) on page 1032.

2.7.80 Join XML Files

This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.81 Link Product To Job

This task is documented in the chapter [Products](#) on page 876. Find a direct link [here](#).

2.7.82 Make Plate on CDI

The **Make Plate on CDI** task can send ripped data (.LEN, .tiff files) directly to the Digital Flexo Suite (DFS) (version 10.1 and later).

The Digital Flexo Suite drives a Cyrel Digital Imager that will image a flexo plate using the ripped data from the **Make Plate on CDI** task.

Note: the integration between Automation Engine and the DFS uses JDF. This does *not* require you to buy any JDF-license from an optional integration module!

Advantages of Using the Make Plate on CDI Task

- You can enforce the **plate type** through *job parameters* (that can be filled in from a business system) to reduce errors.
- It automates your plate-making *workflow*: ripped files are automatically pushed to the Digital Flexo Suite operator (no need to FTP those files, or browse through a file server to get them).
- You can get *feedback* about the waste per plate.
- It builds up a *Flexo Plate History*, which makes it easier to remake a plate if necessary.

Settings

Destination

Enter the **Address** of the computer on which the Digital Flexo Suite software is installed. It will typically take the following shape: `http://<computer name>:4410/JDFP/JMF`.

Priority

Choose the **Priority** you want to give to the task: **Low**, **Normal** or **High**.

Choose a **Due Date**: either choose **Job Due Date**, or choose **Other...** and pick the due date yourself.

Plate

Tip: You can use job parameter SmartNames to retrieve the plate information from the job.

Enter the plate **Manufacturer** and **Type**.

Select the **Media Type** you want to use (**Plate** or **Sleeve**).

- If you want to expose a **Plate**, enter its **Thickness**.
- If you want to expose a **Sleeve**, enter its **Width** and its **Circumference**.

Enter the **Relief Thickness**.

XML Notifications


You can use the standard Automation Engine notification mechanism to create an XML file containing:

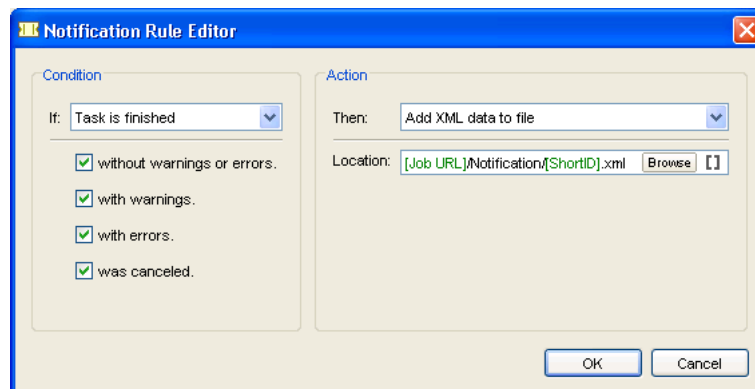
- a history of your Job's different statuses in the Digital Flexo Suite and on the CDI,
- information useful for billing (total area used on the plate, waste...).

Once the task is finished, this XML file will be saved in the location you defined. Your MIS or ERP system can pick it up and use that information.

Note: Depending on the capabilities of that external system, other techniques might be preferred. Learn more in [Integrating with External Systems](#) on page 1228.

Defining an XML Notification

1. Click  in the **Make Plate on CDI** ticket.
2. Click **Add** in the **Notification** dialog.
3. In the **Notification Rule Editor**, define your notification as follows:



Note: Using `[Job URL]/Notification/[ShortID].xml` as **Location** saves your XML file:

- in a subfolder of the Job Folder called `Notification`,
- with the run ID of the task (`[ShortID]`, visible in the Tasks pane) as name.

See [Notifications](#) on page 175 for the basics of defining notifications.

4. Click **OK** in the **Notification Rule Editor** and the **Notification** dialog.

See [Feedback in the XML Notification File](#) for an example of a file generated by an XML notification.

The Make Plate on CDI Task in a Workflow

The **Make Plate on CDI** task is commonly used in combination with a RIP ticket (for example **RIP to CDI LEN File**) in a workflow.



1. The workflow is launched on the one-up.
2. The one-up is stepped and repeated.
3. The Step and Repeat file is ripped.
4. View Data is created for the ripped file.
5. The operator gets a To Do asking him to view the ripped file and check if it's OK.
6. If the file is OK, the Make Plate on CDI task sends the ripped file to the Digital Flexo Suite.

After this the Digital Flexo Suite operator:

1. gets the file automatically in the Digital Flexo Suite,
2. places it on a virtual plate using the Merger application,
3. exposes the flexo plate when it is full enough.

The Automation Engine operator gets feedback from the Digital Flexo Suite at each point. See [Feedback](#) on page 367 for details.

Feedback

When using the **Make Plate on CDI** task, you can receive feedback in two ways:

- XML notifications that can be used by your MIS - ERP system.
- Live status feedback in the Pilot's **Tasks** and **CDI** views.

In the XML Notification File

In the [XML file](#) generated by the **Make Plate on CDI** task notification, you can see:

- A description of how a separation was processed by the Digital Flexo Suite.

For example:

```

<MESSAGES>
<COMMENT Type="INFO">Fetching (XMP) bitmap info for strawberry_123.len</COMMENT>
<COMMENT Type="INFO">Initializing ImageSetting Process for strawberry_123.len.</COMMENT>
<COMMENT Type="INFO">Logging JDF file: file://begeloppem/AutomationEngineTmpFolder/14951_temp_21782062_imagesetting.jdf</COMMENT>
<COMMENT Type="INFO">Doing SubmitQueueEntry towards http://begeloppem:4410/JDFP/JMF.</COMMENT>
<COMMENT Type="INFO">03:18:38 Remote QueueEntry ID: CDI1_@_20110620151838_@_171</COMMENT>
  
```

```
<COMMENT Type="INFO">03:18:39 CDI state change(NEW): NEW job in Merger
JDF list</COMMENT>
<COMMENT Type="INFO">03:24:26 CDI state change(MERGE): Job in Merge
editor</COMMENT>
<COMMENT Type="INFO">03:24:58 CDI state change(CDI): Job Sent to the CDI
Queue</COMMENT>
<COMMENT Type="INFO">03:25:10 CDI state change(RUN): Job Running on the
CDI</COMMENT>
<COMMENT Type="INFO">03:25:29 CDI state change(DONE): Job Completed on
CDI</COMMENT>
<COMMENT Type="REPORT">&lt;HTML&gt;03:25:52 Received the
processed JDF File: &lt;A href="file://begeloppem/
AutomationEngineTmpFolder/14951_temp_21782135_imagesetting.jdf"&gt;c:
\fastlane\data_v100\Tmp\14951_temp_21782135_imagesetting.jdf&lt;/A&gt;
&lt;/HTML&gt;</COMMENT>
</MESSAGES>
```

with time stamps that your MIS / ERP system can use:

```
<StateInfo>
<CDI.NEW>2011-06-20T13:18:39.50Z</CDI.NEW>
<CDI.MERGE>2011-06-20T13:24:26.44Z</CDI.MERGE>
<CDI.CDI>2011-06-20T13:24:58.70Z</CDI.CDI>
<CDI.RUN>2011-06-20T13:25:10.39Z</CDI.RUN>
<CDI.DONE>2011-06-20T13:25:29.63Z</CDI.DONE>
</StateInfo>
```

- Statistics and valuable accounting information.

For example:

```
<PlateInfo>
<Plate.PlateJobId>20jun15_24_49_stry_Kstr945str1@76298443584</
Plate.PlateJobId>
<Plate.MediaType>Plate</Plate.MediaType>
<Plate.TotalArea>263537.4</Plate.TotalArea>
<Plate.UsedArea>26356.48514455161</Plate.UsedArea>
<Plate.TotalUsedArea>79069.45543365483</Plate.TotalUsedArea>
<Plate.Manufacturer>Flint</Plate.Manufacturer>
<Plate.Brand>ACE_114</Plate.Brand>
</PlateInfo>
```

Monitoring the Waste per Plate

Waste per plate can be monitored using the `PlateJobId` (unique to each plate).

You can calculate the total amount of waste per plate as follows: `TotalArea - TotalUsedArea`

Monitoring the Cost per Separation

To calculate the plate cost per separation, multiply the `UsedArea` value by the cost per square unit.

To take into account the plate waste, you can use this: $(UsedArea/TotalUsedArea) \times Plate\ Waste$.

In the Tasks View

After running the **Make Plate on CDI Task**, you will get detailed feedback in the **Tasks** view (and the **Tasks** pane of the **CDI** view in **Job** mode).

You will see the following statuses:


- **NEW job in Merger JDF list,**

- **Job in Merge editor** (while the Digital Flexo Suite operator is placing the job files on the virtual plate),
- **Job Sent to the CDI Queue**,
- **Job Running on the CDI**,
- **Job Completed on CDI**.

Once the plate is made, you will see a green check  (or a red cross  if there was an error).

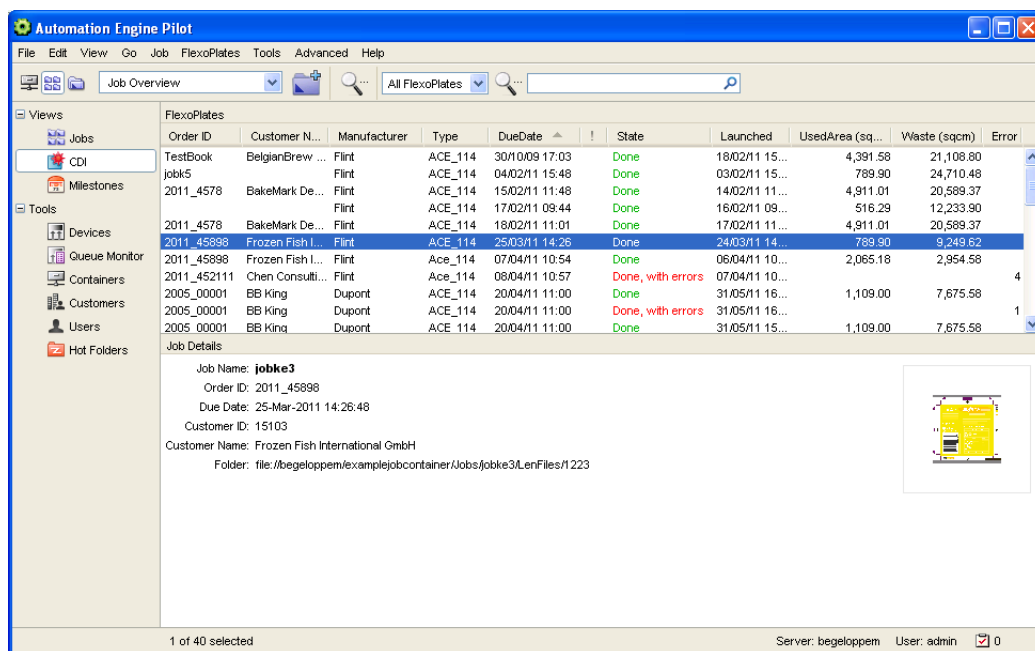
In the CDI View

The **CDI** view shows all the files sent to the Digital Flexo Suite and used to image a flexo plate on a Cyrel Digital Imager.

You can view the flexo plates created for all jobs (in **Job Overview** mode) or for a particular job (in **Job** mode). You can also use the filters to only view certain plates (or click  to define your own filters).

You can double-click an entry in **Job Overview** mode to see all the plates and LEN files corresponding to that job in **Job** mode.

You can also select an entry to view its **Details** (in **Job Overview** mode), or the related task (in **Job** mode) at the bottom of the window.



For each plate, you can see valuable:

- administrative information (**Order ID, Customer Name...**),
- technical information (plate **Manufacturer, Type** and **Thickness**, imaging **Device**),
- production information (**Launched** date, **Due Date, Priority...**),
- accounting information (**UsedArea** and **Waste**).

Tip: Go to **View > Select Columns in 'FlexoPlates'...** to pick the information to show or hide.

You can also see the plates' **status in real time**.

Statuses

In **Job Overview** mode, you will see the following statuses:

- **Queued on CDI,**
- **Done** or **Done, with errors,** or **Done, with cancellations.**

You can also see the number of plates that are ready in the **Done** column. The status will be **Queued on CDI** until all plates are ready.

In **Job** mode, you will see:

- **NEW job in Merger JDF list,**
- **Job in Merge editor** (while the Digital Flexo Suite operator is placing the job files on the virtual plate),
- **Job Sent to the CDI Queue,**
- **Job Running on the CDI,**
- **Job Completed on CDI,**
- **Exposed,**
- **Error** or **Cancelled** if applicable.

Checking the Files

In the **CDI** view, you can:

- Open the LEN file(s) in the Viewer: right-click the entry (in **Job** or **Job Overview** mode) and select **View**.

You can then show / hide separations using the Viewer's **Channels** panel.

- Go to the LEN file(s) in the **Files** view:

- in **Job Overview** mode, right-click the entry and select **Go To Folder**.

This opens the job folder and the subfolder containing the LEN files.

- in **Job** mode, right-click the entry and select **Go To File**.

This opens the job folder and the subfolder containing the LEN files, and selects the LEN file.

Re-exposing the Plate

If one or several of the plates need(s) to be re-exposed (it turned out wrong, or broke coming out of the CDI or after a number of print runs), it can easily be sent to the Digital Flexo Suite again.

1. Use the filters to find the right (set of) plate(s).
2. Right-click the desired plate entry and select **Go To Workflow** to open the workflow used to process the file in the workflow editor.
3. Right-click the workflow step where you want to relaunch processing from, and select **Relaunch**.

For example,

- if the resolution of the LEN files is wrong, relaunch from the **RIP**.
- if the plate broke, relaunch from **Make Plate on CDI**.

**Attention:**

Make sure you have kept the LEN files if you are relaunching the workflow after the **RIP** step!

We recommend you either:

- Save them somewhere consistent and easy to find in your job folder structure, for example using:

Output File	
Path	[Job URL]\LenFiles\[ShortID] []
Name	[File]_[ShortInk] []

- Save them in an external location if you have the habit of quickly deleting jobs (but if you delete the job without deleting its folder and files, you can still store them in your job folder).

2.7.83 Manage Color

With this task, you can:

- convert a CMYK file to another CMYK profile,
- reparate a CMYK file so that it contains less CMY and more K, while keeping the image looking the same,
- both convert and reparate your CMYK file.

This task takes PDF and Normalized PDF files as input. When working with regular PDFs, it will automatically normalize them using the **Normalize ticket** set in the **Configure** tool (see [Processing Preferences](#) on page 674).

For more information about Normalized PDFs and automatic normalization, see [PDF Normalization in Automation Engine](#) on page 115.

Converting to another CMYK Profile

You can convert a file to another CMYK profile if for example you want to print it on a different substrate, on a different press, or even with a different printing process (offset, digital printing...).

In **Actions**, select **Convert color space** to convert your input file to another CMYK profile. This displays extra parameters below it.

1. Choose what to use as a **source profile** when converting your input file:
 - a) In **Source CMYK Profile**, select the source profile to use for the conversion in case you are not using your input file's profile.

This can be an Esko or an ICC profile and you can use a SmartName (resolving to the full profile name) to set it. By default this is `ISOcoated_v2_eci.icc`.
 - b) If your input file has a CMYK profile and you want to use it for the conversion, select **Use document profile if available**.

When processing a file without a profile, the task will use the **Source CMYK Profile** you selected.

2. Choose what **Destination CMYK Profile** to convert your file to.

This can be an Esko or an ICC profile and you can use a SmartName (resolving to the full profile name) to set it. By default this is `ISOcoated_v2_eci.icc`.

3. Choose the **Rendering Intent** to use when converting the colors.

This can be:

- **Perceptual:** This rendering intent shrinks the image's gamut into the destination gamut. It modifies all the colors, but relatively to each other, so it keeps an overall harmony.

It does not perform any background simulation: it maps the source profile's white point to the destination paper's white (so no ink is used when printing white).

Use this option if you don't care too much about an exact color match but want a visually pleasing result.

- **Relative Colorimetric:** this converts the colors that fall inside the destination gamut to the destination profile, and converts each color falling outside the destination gamut to the closest in-gamut color.

It does not perform any background simulation: it maps the source profile's white point to the destination paper's white (so no ink is used when printing white).

Relative Colorimetric is a good choice for photographic imagery, because it preserves color relationships without sacrificing color accuracy.

It generally gives less good results when converting images with gradations or blend modes.

- **Saturation:** This rendering intent saturates the in-gamut colors, and converts each out-of-gamut color individually to the closest in-gamut color.

This means that close out-of-gamut colors may be converted to the same in-gamut color (the image will then lose several shades).

Use this option if you care more about saturated colors than color accuracy, for example if you are working with cartoons / comics, business graphics, or you want to improve weak images.

- **Absolute Colorimetric:** this converts the colors that fall inside the destination gamut to the destination profile, and converts each color falling outside the destination gamut to the closest in-gamut color.

This rendering intent simulates the source white within the image (background simulation), so that if for example the source profile is designed for a newspaper, its white will be slightly grey, and the whole image will have a slight grey undertone.

This rendering intent is mostly used for proofing (on-screen or on a proofing device), to reproduce what the image's appearance will be on the final paper.

If accidentally printed on the final paper with this rendering intent, the image will look dull and darker!

Absolute Colorimetric is the most accurate rendering intent if the image's color profile contains correct white point information: it can then give an exact color match (for in-gamut colors).

However, it maintains color accuracy at the expense of preserving relationships between colors: when converting to a smaller gamut, two colors that are different in the source gamut may be converted to the same color in the destination gamut.

Note:

- When converting to an ICC destination profile, you can choose from the four rendering intents listed here.
- When converting to an Esko destination profile, you can choose either **Relative Colorimetric** or **Absolute Colorimetric**.

You can also set the rendering intent with a SmartName, resolving to 2 (for **Perceptual**), 3 (for **Relative Colorimetric**), 4 (for **Saturation**) or 5 (for **Absolute Colorimetric**).

4. Select what objects in your file you want to convert in **Apply on**. This can be either:

- **Images and Linework**,
- **Linework only**,
- **Images only**.

You can also set this with a SmartName, resolving to `images_and_linework`, `linework_only` or `images_only` (SmartNames are case insensitive).

5. Select **Keep registration color** if you use marks in your file and if you don't want them to be converted to the new profile (you want them to stay 100% of each CMYK color).

6. Select **Flatten document** if you are experiencing unexpected results in color conversions (if after conversion the color of some blend modes or other objects on top of each other looks different).

Note: We recommend you only use this in case of conversion problems, as it will remove any transparencies in your file.

Reseparating your CMYK File

Reseparating a file is replacing some amount of Cyan, Magenta and Yellow in the file (that would give grey) by a certain amount of Black, giving the same grey color.

Objects that are only Cyan, Magenta or Yellow (or only two of these colors) are kept as they are.

This can help you save on CMY inks, have a more stable output (less risks of color casts in neutral areas) or have more detail in the shadow areas.

In **Actions**, select **Reseparate** to reseparate a CMYK file so that it contains less CMY and more K, while keeping the image looking the same.

This displays extra parameters below it.

1. Choose the profile to use when reseparating your input file:

a) In **CMYK Profile**, select the profile to use in case you are not using your input file's profile.

This can be an Esko or an ICC profile and you can use a SmartName (resolving to the full profile name) to set it. By default this is `ISOcoated_v2_eci.icc`.

b) If your input file has a CMYK profile and you want to use it for reseparating, select **Use document profile if available**.

When processing a file without a profile, the task will use the **CMYK Profile** you selected.

2. Select what type of reseparation you want to do in **Purpose**:

- **Ink Saving** replaces as much CMY as possible by Black.

This is a good choice if you want to save on CMY inks.

- **GCR** replaces CMY by Black in the grey and chromatic areas.

This is a good choice if you want to avoid color casts in the grey areas, and to have more detail in the shadow areas.

This is selected by default.

- **UCR** only replaces CMY by Black in the neutral areas.

This is a good choice if you want to avoid color casts in the grey areas.

You can also set this with a SmartName, resolving to `ink_saving`, `GCR` or `UCR` (SmartNames are case insensitive).

3. In **Range**, choose how much of the CMY inks that can be replaced by Black you want to actually replace.

This depends on the **Purpose** you chose:

- For **Ink Saving**, this is how much CMY inks in **all areas** of the file you want to replace.
- For **GCR**, this is how much CMY inks in the **grey and chromatic areas** of the file you want to replace.
- For **UCR**, this is how much CMY inks in the **grey areas** of the file you want to replace.

Note: You can set all percentage values in this task with SmartNames.

4. In **Total Ink**, select the maximum amount of ink (CMYK) that you want to print.

If you don't limit the amount of ink, you can get a maximum of 400% (100% Cyan + 100% Magenta + 100% Yellow + 100% Black), which tends to cause problems on most substrates (slow drying, web growth, or even a risk of tearing).

Additionally, limiting the maximum amount of ink used can make printing jobs less expensive.

The maximum percentage of ink you should use depends on your printing process and substrate. For example for newsprint, you may only want to have a maximum of 240%, while you may be able to use 280 to 320% on a glossy substrate.

The default **Total Ink** percentage set in the task is 310%.

5. Define the minimum and maximum amount of Black that you want to have in your output file:

- a) In **Start K**, define from which (minimum) final Black value you want to start reseparating.

If replacing CMY by Black in a certain color gives a Black percentage lower than this, the CMY won't be replaced.

This is to avoid having really small Black dots, which can be prone to high dot gain, making them less stable than the equivalent CMY dots.

- b) In **Max K**, define until which (maximum) final Black value you want to start reseparating.

If replacing CMY by Black in a certain color gives a Black percentage higher than this, the CMY won't be replaced.

This is to avoid having too dark shadows and losing detail.

6. Select what objects in your file you want to reseparate in **Apply on**. This can be either:

- **Images and Linework**,
- **Linework only**,
- **Images only**.

You can also set this with a SmartName, resolving to `images_and_linework`, `linework_only` or `images_only` (SmartNames are case insensitive).

7. Select **Keep registration color** if you use marks in your file and if you don't want them to be reseparated (you want them to stay 100% of each CMYK color).

8. Select **Flatten document** if you are experiencing unexpected results after reseparating a file (if after reseparation the color of some blend modes or other objects on top of each other looks different).

Note: We recommend you only use this in case of reseparation problems, as it will remove any transparencies in your file.

Converting and Reseparating your File

When converting and reseparating your file(s) at the same time, you need to:

1. Select the **Source profile**, **Destination profile** and **Rendering Intent** for the [conversion](#).
2. Select what type of [reseparation](#) you want to do and set the appropriate options.
3. Select on which objects to perform **both** the conversion and the reseparation, whether to keep objects in **registration color** untouched, and whether to **Flatten** your file.

Note:

- When using one **Manage Color** task to both convert and reseparate, the conversion will try to keep the black channel unchanged, so as not to affect the reseparation.
- When using two **Manage Color** tasks after each other, one for conversion and one for reseparation, this will not be the case, so the output will be different.

If you want to perform both conversion and reseparation on your file(s), we recommend you use one **Manage Color** task for both instead of a task for each in your workflow.

2.7.84 Manage Milestone

The **Manage Milestone** task is documented [here](#) in the chapter on [Jobs](#) on page 828.

2.7.85 Manage Product Status

This task is documented in the chapter on [Products](#) on page 876. Find a direct link [here](#).

2.7.86 Map Data

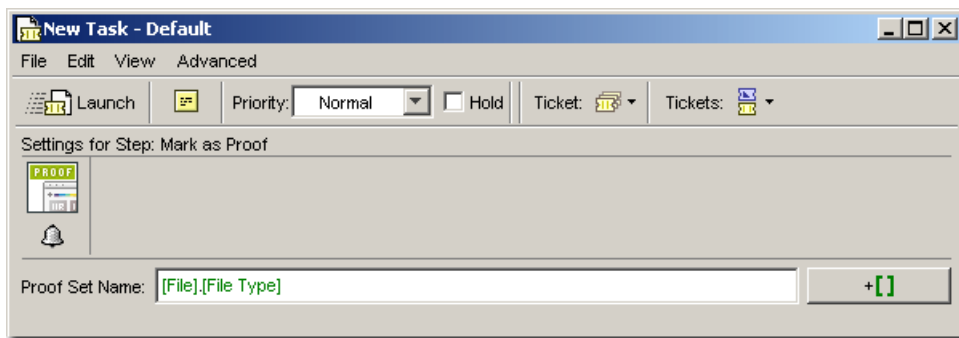
This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.87 Mark as Proof

Note: This task is used only in page workflows.

The Mark as Proof ticket can only be used in a workflow, for example in combination with the Proof (Imposition) to PDF File ticket. It is used to make the proofs appear in the Proofs View. Automation Engine offers a ready-made [Export Imposition to PDF File workflow](#) ticket, which contains the export ticket, linked to the Mark as Proof ticket.

The Proof Set Name will by default be indicated as [File].[File Type], but can be adapted with SmartNames or hard text.



2.7.88 Mark File (link)

This Workflow Control is documented in the chapter "Workflows". See [Mark File and Select Marked File](#) on page 1029.

2.7.89 Merge PDF Files

The task **Merge PDF Files** can be used to combine multiple PDF files into one single PDF.

The task can accept multiple native PDF files to produce a single native PDF file, or a multiple normalized PDF Files to produce a normalized PDF File.

Take the following notes into account when using this task:

Note:

- Normalized and native PDF files cannot be mixed together.
- The page sizes of the normalized PDF files must be the same.
- The output PDF will only be normalized if all the input files are normalized files and have the same media and trim boxes. In case there is a mismatch in size, the task will fail.
- For normalized PDF files, the XMP information of the first file will be copied into the resulting PDF file. Then it will be updated with information from the other normalized PDF files.
- In workflow context, all pages of all incoming PDF files are added in the same order as the inputs of the task in job context. To control the order of the files, use the **Sort** Workflow Control.

2.7.90 Modify Workflow Parameter Values

This Workflow Control is documented in the chapter "Workflows". See [Modify Workflow Parameter Values](#) on page 1015.

2.7.91 Normalize PDF File

Colors

Use the **Colors** tab to define the conversion settings needed when converting the document's (non-process) colors during Normalization.

Note: The conversion is done by Esko's Color Engine.

Objects With Profile Tags: In this parameter, choose how to convert non process color images that have an embedded ICC profile. Choose either:

- **Ignore Tag** to use the **Default RGB Profile** or **Default Gray Profile** set in this tab to convert the RGB or gray images. This disregards the images' own profiles, and is the default option.
- **Convert Using Profile Tag** to use the images' own profiles to convert them to CMYK.

Rendering Intent

- **Relative (No Background):** Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- **Perceptual:** This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.
- **Absolute (Background):** Use this option if you want background simulation and you want an exact color match, including the paper simulation.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

Default CMYK Profile: Select the default profile to use for CMYK objects without an embedded color profile. It will be used as a source profile for colorspace conversions.

You can choose any CMYK profile in the Color Engine database.

Default RGB Profile: Select the default profile to use for RGB objects without an embedded color profile. It will be used as a source profile for colorspace conversions.

You can choose any RGB profile in the Color Engine database.

Default Gray Profile: Select the default profile to use for gray objects without an embedded color profile. It will be used as a source profile for colorspace conversions.

You can choose:

- any gray profile in the Color Engine database,
- **Convert to Black Channel**, if you want your grayscale object to only have a value in the Black separation when outputting it to CMYK.

Output Color Profile: Enter the colorspace of the output device (for example proofer or press) in this parameter.

You can choose any CMYK profile in the Color Engine database.

Separations

Use the **Separations** tab to define ink specifications for the input file. These ink specifications will be used when you edit that file with an Esko editor.

Note: The settings in this tab only affect Normalized PDF output. They are ignored if the file is output as a regular PDF.

1. Choose your **Preferred Ink Book**.

When you type an ink name in the table, Automation Engine will first look up its color information (CMYK, RGB, or LAB values), or ink type (Technical, Varnish, Opaque...) directly in the PDF.

If it cannot find it there, it will look it up the **Preferred Ink Book** you define here.

For example, if you type 154 as ink name, and your preferred ink book is `Pantone Colors Uncoated`, Automation Engine will automatically suggest the `PANTONE 154 Uncoated` ink.


2. Enter the name of the first ink to define in the **Ink Name** column.
3. The **Ink Book** column shows the first ink book containing this ink (or your **Preferred Ink Book** if it contains the ink). To select another ink book, click the ink book name.
4. If the **Ruling**, **Angle** and **Dot Shape** information are defined for this ink, they will be shown automatically.

If they aren't, you can enter them manually.

Note: You can also specify **Default Values** to use **when a ruling, angle or dotshape is not specified**.

Default values, used when a ruling, angle or dotshape is not specified:

Ruling: lpi Angle: ° Dotshape:

 If there are inks specified in the list, but no default values are set, factory settings will be used for ruling (120 lpi) and angle (45°).

5. If the **Ink Type** shown is not correct, you can click it to change it.
6. Do the same for the remaining inks you want to define.

7. If desired, you can change the ink order using the **Move Up** and **Move Down** buttons.

2.7.92 Normalize PostScript / PDF / Illustrator 8.0 File

Note:

- The PostScript / PDF interpreter extracts fonts embedded in PostScript, Illustrator and PDF files. Only Type 1 fonts embedded in a PostScript / PDF file will be extracted.
The extracted fonts will be installed locally automatically.
- For copyright reasons extracted fonts can **only** be used to view and output the particular PostScript, PDF or Illustrator job containing the font.
Any other use of that font (for example editing the text) requires a license of that font.

Tip: When using an XMP XPath SmartName in the **Output** field of a **Normalize PostScript / PDF / Illustrator 8.0 File** ticket, and the XMP XPath cannot be resolved (because there is an error in the SmartName or, for some reason, the queried data cannot be found), the output file is saved in a / xmp_unresolved subfolder.

Pages

Pages | Color Management | Inks | Overprint | Images | Page size | Other | Output

All Pages

Range (Example: 1-5,7,9-11)

Split Up a Multipage File

Maximum Pages per File

First Pagenumber in Output

Compose a Group of CT Images into 1 CT Image

Merge Separated File to Composite Page

Optimize Merged File

Password

Select **All Pages** or specify a **Range** of pages.

Split up a multi-page file

Check the box to divide the original multi-page PostScript file into smaller parts. You define the amount of pages to open (of the specified pages or range). What you need to know:

- Each part is stored as a separate Normalized PDF file. It is either stored locally or stored in the Output Directory.

- Each part is stored with the name of the imported PostScript file followed by _001, _002, _0001_001... or it gets the Output Name.

Compose a Group of CT Images into 1 CT Image

Some ArtPro files have tiled CTs, those are CTs which are actually divided into a number of little CTs (tiles). Some jobs are so large that there are thousands of these little CTs.

Select this toggle to merge the tiles into one CT to improve the editability of the Normalized PDF and to optimize further processing.

This option is selected by default, it doesn't matter if you import a file with no tiled CTs.

Note: When errors occur when a PostScript file is imported (for example a font is not found), the **Merge Tiled CTs to one CT** will not be performed.

Merge Separated file to Composite

Merges separated single-page and multi-page PostScript or PDF files to a composite file.

Optimize Merged File

This option optimizes the merged file. This is useful when you need to edit the imported file. This option becomes available when 'Merge Separate File to Composite Page' is activated.

Note: When errors occur when a PostScript file is imported (for example a font is not found), the **Optimize Merged PostScript File** will not be performed.

Password: You can enter a password for protected PDF files.

Color Management

The screenshot shows the 'Color Management' dialog box with the following settings:

- Destination Color Space:** Color Space: CMYK, Destination Profile: crom_offs, Use PDF Output Intent if available: , Rendering Intent: Absolute (background), Set Document Profile: Never. Note: The document will be untagged.
- RGB:** Convert: Source to destination profile, Source Profile: calibrator, Objects with RGB profile tags: Ignore tag.
- Spot Colors:** PANTONE Inks: Keep all, Method: Use destination profile, Designer Inks: Keep all.
- CMYK and Gray:** Objects with profile tags: CMYK: Ignore tag, preserve percentages, Gray: Ignore tag, preserve percentages.

Please refer to the Inks TabSheet to view and/or modify the ink list.

Destination Color Space

Here you define what colorspace the opened/imported document will have.

Colorspace:

- CMYK
- Grayscale

Destination Profile: choose a color profile from the list. Currently only CMYK and Grayscale profiles are supported, you can choose between standard ICC profiles and Esko profiles made in Color Engine Pilot.

Use PDF Output intent if available: in some PDF files, the PDF Output intent is defined (for example PDF/X-3). (The output intent is the intended output color space.)

If you want to use the color space as defined in the output intent, select this option. If you selected it for a file without PDF Output intent, the chosen **Destination Profile** will be used.

Rendering intent

- **Relative (No Background):** Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- **Perceptual:** This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.
- **Absolute (Background):** Use this option if you want background simulation and you want an exact color match, including the paper simulation.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

Set Document Profile

A Document Profile is the color space of the process inks used in the document.

- **Always:** you can 'tag' a Normalized PDF file with its Document Profile. This means that the file 'knows' what its Document Profile is, which may make further color conversion throughout a workflow more correct.

If you selected the **Use PDF Output intent if available** option, the Document Profile will be the same as the PDF Output intent (if the imported file had one).

- **From PDF File:** the Document Profile is taken from the PDF file, if there is an output intent defined in the PDF file (this option is only available when the **Use PDF Output Intent** option is selected). If there is no Output Intent defined in the PDF file, then the Document Profile is not set (untagged).
- **Never:** the Document Profile is not set.

Spot Colors

PANTONE inks

- Keep All: No PANTONE inks will be converted to process inks.
- Convert All: All PANTONE inks will be converted to process inks.
- Keep list: All PANTONE inks will be converted **except for** those listed in the Inks tab.

Method

- Use Destination Profile: PANTONE and Designer inks will be converted using the selected Destination Profile. This gives the most accurate color reproduction.
- PANTONE Solids to Process (Euroscale). PANTONE inks will be converted to process inks according to a fixed conversion table provided by PANTONE. The result is independent of the selected Destination Profile. This assumes the Euroscale print conditions.
- PANTONE Solids to Process (SWOP). PANTONE inks will be converted to process inks according to a fixed conversion table provided by PANTONE. The result is independent of the selected Destination Profile. This assumes the American SWOP print conditions.

Designer Inks:

- Keep All: No Designer inks will be converted to process inks.
- Convert All: All Designer inks will be converted to process inks.
- Keep list: All Designer inks will be converted **except** those listed in the Inks tab.

Note:

If the 'Flatten PDF transparencies and layers' box is checked and you select 'Convert all' or 'Keep list', the color conversion may give unexpected results when overprint is involved.

RGB

RGB will always be converted to process inks during import.

RGB Method: Select one from the list

- Source to destination profile: the user needs to specify both the Source Profile (RGB) **and** the Destination Profile (CMYK or Grayscale).
- With Link profile: the user specifies a Color Engine Pilot Device Link Profile. this Link Profile implicitly determines the Source and Destination profiles.

Source Profile / Link Profile: Select one from the list.

Objects with RGB Profile tags: The RGB objects in a PostScript or PDF file may be tagged with a Color Profile.

- **Convert using the RGB Profile tags:** The user may choose to use that Color Profile as a Source Profile (overruling the Source / Link Profile selected above).

Tip: In strictly controlled workflows, the embedded tags are probably accurate, so this option may be preferable.

- Ignore tags: the embedded Color Profile will be ignored, the Source / Link Profile defined above will be used during conversion of RGB objects.

CMYK and Gray

The CMYK objects in a PostScript or PDF file may be tagged with a Color Profile. The color percentages of untagged objects will always be preserved. For tagged objects, users have a choice:

- **Ignore tags, preserve percentages:** the Color Profile tag will be ignored, the percentages will be those of the object's color.
- **Convert, preserve appearance:** the objects' color profile will be taken into account; this means that their color values will change.

The Gray objects in a PostScript or PDF file may be tagged with a Color Profile. The color percentages of untagged objects will always be preserved. For tagged objects, users have a choice:

- **Ignore tags, preserve percentages:** the Color Profile tag will be ignored, the percentages will be those of the object's color.
- **Convert, preserve appearance:** the objects' color profile will be taken into account; this means that their color values will change.

Inks

Pages Color Management **Inks** Overprint Images Page size Other Output

Use rulings, angles and dot gain compensation from file if available

Force ink settings from Job setup

Apply Screen Swap Tab... <None>

Map PANTONE Inks to: Coated

Use ClassicColors ink book

Search Custom Ink Names: Only in the ink book 'designer'

First in: and then in 'designer'

Inks:

<input checked="" type="checkbox"/>	Ink Name	Ink Book	Ruling	Angle	Dot Shape	Ink Type	Printing ...
<input checked="" type="checkbox"/>	Cyan	process	150 lpi	15 °		Normal	
<input checked="" type="checkbox"/>	Magenta	process	150 lpi	75 °		Normal	
<input checked="" type="checkbox"/>	Yellow	process	150 lpi	90 °		Normal	
<input checked="" type="checkbox"/>	Black	process	150 lpi	45 °		Normal	

Default values, used when a ruling, angle or dotshape is not specified:

Ruling: 150 lpi Angle: 45 ° Dotshape:

Use rulings, angles and dot gain compensation from PostScript file if available: If the PostScript file contains rulings, angles and dot gain compensation, these will be used when this toggle is selected. Other settings in this tab will be overruled.

Force ink settings from Job setup: If inks were defined in your job setup, you can use this toggle to make sure that this task uses these parameters.

For more information, please refer to the section on [the Inks tab in Job Setup Parameters](#).

Apply Screen Swap Table: Select a screen swap table from the list.

Map PANTONE Inks to: Enable this option to map all Pantone inks found in the source file to the Pantone ink book of your choice.

For example, if you enable this option and select the Uncoated ink book, any Matte or Coated Pantone inks will be automatically mapped to the nearest equivalent Pantone Uncoated ink.

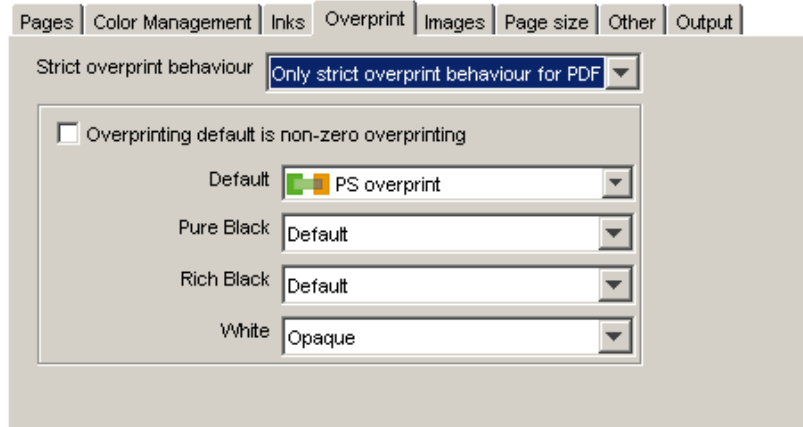
Search Custom Ink Names: Determines how Automation Engine searches for inks in its ink books when you type in an ink name in the Inks table.

Choose **Only in the ink book 'designer'** to search for ink names from the "designer" ink book only; or choose **First in <your ink book> and then in designer** to search in another ink book of your choice first, and only use the designer ink book if no match is found there.

Inks table: double-click the fields to edit them. Select one or more inks to limit the effect of the task to those separations.

Default values, used when a ruling, angle or dotshape is not specified: Enter the default values for ruling angle and dot shape, the system will take these values when they are not specified elsewhere (i.e. in the PostScript file, or in job ink parameters...).

Overprint



Strict overprint behavior

CertIn (the Esko input module) will honor the overprint parameter 100% according to the rules mentioned in the "PostScript Language Reference" (in case of PostScript / EPS /AI import) and "PDF Reference Manual" (in case of PDF).

This option **must** be set when the user wants to process test files like the Altona test suite or other test suites which verify overprint behavior correctness.

This option controls how zero percentage inks are handled when reading composite files.

When **Strict overprint behavior** is switched on all zero percentage inks of a color are kept and correspondingly affect the overprint behavior.

When **Never strict overprint behavior** is selected, all zero percentage inks are removed.

- **Never strict overprint behavior:** all zero percentage inks are removed
- **Always strict overprint behavior:** CertIn will honor the overprint parameter 100% according to the rules mentioned in the "PostScript Language Reference" (in case of PostScript / EPS /AI import) and "PDF Reference Manual" (in case of PDF import).
- **Only strict overprint behavior for PDF:** CertIn will honor the overprint parameter 100% according to the rules mentioned in the "PDF Reference Manual" (in case of PDF).

Overprinting default is non-zero overprinting

Prevents overprinted objects with zero CMYK values from knocking out CMYK objects beneath them.

Default: To set the overprint mode that the conversion needs to apply when importing the file.

- **Opaque:** Ignores any overprint of your job.
- **BG Overprint:** Converts all overprint present in your PostScript job to BG Overprint.
- **PS Overprint:** All PS overprint present in the job is imported correctly. (This is the default setting.)

Pure Black: To set the overprint mode for objects containing 100% black only. Possible Pure Black overprint modes in the list:

- **Default:** As it is defined in the job.
- **Force Opaque:** Forces Opaque for objects in 100% black and knocking out all underlying inks.
- **Force PS Overprint:** Forces PS overprint for objects in 100% black.
- **Force BG Overprint:** Forces BG overprint for ALL objects in 100% black. The black color is printed on top of other colors, instead of erasing the color underneath.

Note:

There are two problems:

- Sometimes ink saturation can occur.
100% black text on a 100% red (this is, 100% magenta and 100% yellow) background, results in three inks printed at 100%.
- Black is best handled as knockout for flexo, since it is better to add a small trapping region than to make the complete black region overprint.

Rich Black

Set the paint mode for objects containing 100% black and at least one other ink.

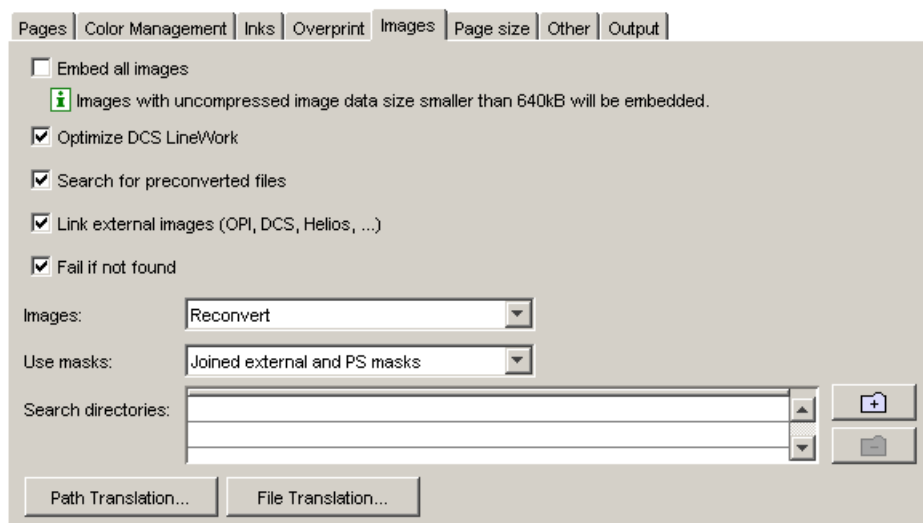
- **Default:** As it is defined in the job.
- **Force Opaque (masking):** Sets the overprint mode to opaque for all objects containing 100% black and at least one other ink.
- **Force PS Overprint:** Keeps the PS overprint for objects in 100% black and at least one other ink (overprint objects stay in overprint and opaque objects stay opaque).
- **Force BG Overprint:** Sets BG Overprint for all objects in 100% black and at least one other ink.

White

Possible white paint modes to select from the list:

- **Opaque:** Forces all objects in white to opaque.
- **Keep PS Overprint:** Keeps the PS overprint for objects in white (overprint objects stay in overprint, opaque objects stay opaque).

Images



Embed all images Toggle this on if you want to embed all 1-bit, 8-bit and native images. If the box is not checked, then only images with uncompressed image data smaller than 640 kB will be embedded.

Optimize DCS LineWork Without linework DCS optimization, the result after conversion of linework DCS files can be difficult to edit afterwards. However it will be correct when viewed in extended preview or RIPped. 'Optimize DCS Linework' will do an additional operation to combine the elements on each DCS plate to a composite and a more editable result.

Search Preconverted Files This option forces the system to look for the GRx, CT or LP equivalent for the original EPS file. If this equivalent is not found, it will use the EPS instead.

Link External Images (OPI, DCS, Helios, ...) Force the system to look for referenced images.

Fail if not Found Activate this option if you do NOT want the system to continue importing / normalizing your PostScript file when it cannot find an image. When this option is not activated the imported (E)PS file will appear on screen but without the missing images.

Images Determines how images are treated when the task is run:


- Use Most Recent: forces the system to use the existing images of a previous conversion of the same job.
- Reconvert: forces the system to regenerate the images on-the fly.

Use Masks Available in the list are:

- **PS Mask:** Forces the system to take the PS mask present in the EPF file. The mask available in the external image is discarded. If no PS mask is present in the file, no mask will be taken.
- **External Mask: if any, PS Mask otherwise:** Forces the system to take the mask present in the external image. The PS mask is discarded. If the external image has no mask, but there is a PS mask available in the EPF file, this PS mask will be taken.
- **External Mask: if any, no mask otherwise:** Forces the system to take the mask present in the external image. The PS mask is discarded. If the external image has no mask, no mask will be taken.
- **No Masks:** The PS mask and the mask of the external image are discarded.
- **Joined External and PS Mask:** Forces the system to join the available PostScript mask in the EPF file and the available mask in the external image.

Search directories. To specify a list of directories to look for referenced images.

Use the 'Add'  and 'Remove'  buttons to optimize the Search Directories list.

Clicking  opens the directory selector.

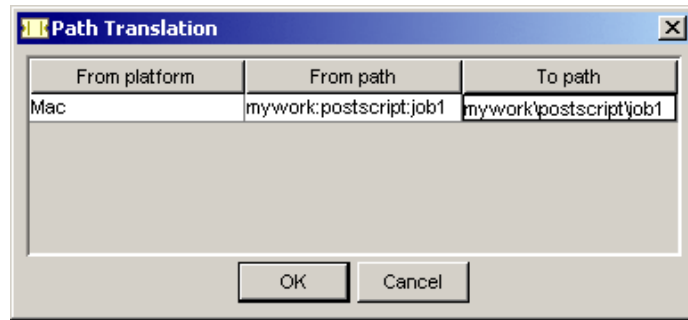
You can browse to the correct directory and click OK to add the directory to the Search Directory list.

Note: You can also use SmartNames in the Search directories.

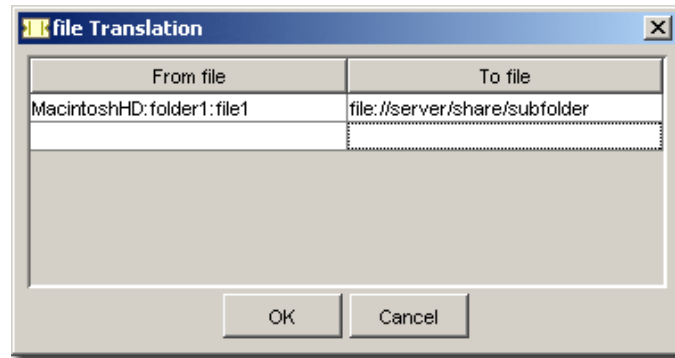
Path Translation To define an understandable link for paths from a platform to another platform. The Path Translation dialog opens, it has three columns:

- **From platform:** Use the list to select a platform from which the files are to be retrieved: NT, Macintosh, Unix or VMS.
- **From path:** specify the syntax of the path(s) of the platform you want to retrieve files from. For example `Mywork:PostScript:job1`
- **To Path:** define the syntax of the paths so that the destination platform understands. For example `Mywork/PostScript/job1`

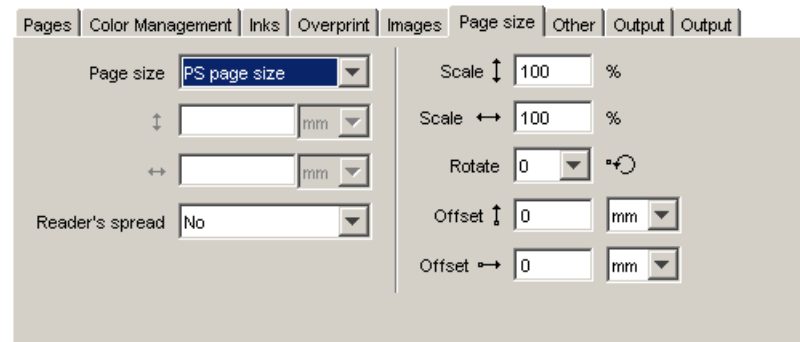
For example, when going from Macintosh to Windows NT, the path translation would look like this:



You can do the same for files: **File Translation**.



Page Size



The default **page size** is PS Page Size. PS Page Size is the page size found in the selected PostScript file for import. The list displays all possible options (Custom, A5...).

Reader's Spread: To determine how the PostScript file, which has reader's spread format, must be opened.

- **Single Page:** Forces the reader's spread in the PostScript file into single page output.
- **Double Page:** Forces PS File into reader's spread format. When the PostScript file contains no reader's spread info, the single pages will be set as double pages.

Scale vertical ↑ : To enter the vertical scale percentage. 100 % is the original size of the PostScript file.

Scale horizontal ↔ : To enter the horizontal scale percentage. 100 % is the original size of the PostScript file.

Rotate ↻ : Forces the system to rotate the PostScript File with the specified angle. The origin for rotation is bottom left.

Possible angles in the list:

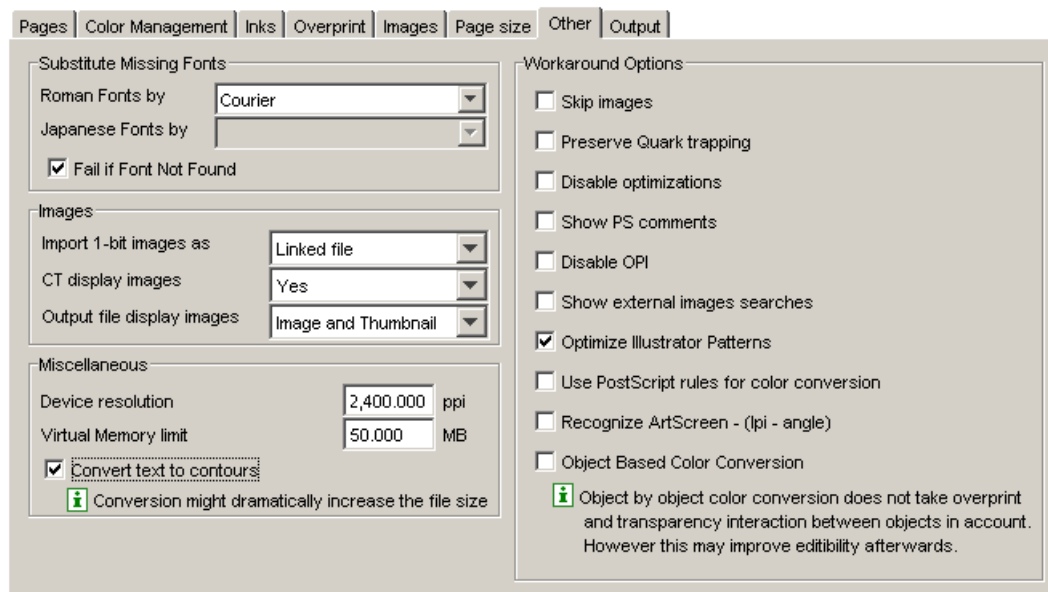
- **0**: No rotation is to be performed.
- **90**: Forces to rotate over 90 degrees counterclockwise
- **180**: Forces to rotate over 180 degrees.
- **270**: Forces to rotate over 270 degrees counterclockwise.

Offset ↕ ↔ : The original offset is Bottom Left.

Specifying a positive vertical/horizontal offset moves your PostScript file upwards/to the right in relation to the bottom/right border.

Specifying a negative vertical/horizontal offset moves your PostScript file downwards/left in relation to the bottom/left border.

Other



Substitute Missing Fonts

Roman Fonts by: Choose a default replacement font for the missing fonts for this job on your system.

Japanese Fonts by: Choose a default replacement font for the missing fonts for this job on your system.

Fail if Font Not Found: The task will fail if the font is not found.

Images

Import 1-bit images as:

- **Intern:** Images become internal black and white bitmaps.
- **Extern:** Images are imported as external referenced LP files.
- **Contours:** Images are contoured during import of the (E)PS file.

CT display images: Choose **Yes** to create a display image for the generated CT files on-the-fly.

Output file display images: Select the type of display from the list: **None**, **Thumbnail** or **Image and Thumbnail**.

Miscellaneous

Device resolution: The resolution used internally by CertIn during the normalization.

Normally the produced output is not dependent of this internal resolution but in some rare cases like when converting RIP test strips or device specific tests it is important that this value is the same resolution of the final RIP device which will be used to print output.

Virtual Memory limit: The default value for the virtual memory is 50MB. The limit is set on 1 gigabyte.

Convert text to contours

Note: Conversion might increase the file size.

Workaround Options

Skip Images: To open the (E)PS file without linking to external images or converting embedded images.

Preserve Quark trapping: Forces the system to keep the trap strokes made by QuarkXpress.

Disable optimizations: Vignettes recognition and optimization for Photoshop Multitone images are performed (whenever possible) when this option is not activated.

Sometimes after updating your DTP applications, Esko can no longer perform the optimization and the conversion of the (E)PS file might fail. Switching this option ON will allow you to import the (E)PS file.

Show PS comments: All PS comments present in the (E)PS file are displayed on screen.

Disable OPI: This option is valuable when you are importing (E)PS files of QuarkXpress containing high resolution Tiffs files for which the option 'Include tiff, omit ...' or 'Include all' was activated.

In this case, when importing such a file and if **Link External Images** is activated, the matching for the TIFF files will surely fail. It fails because at the time the (E)PS file was generated OPI commands were also inserted for the TIFF files.

Therefore the system will try to link to images for the Tiff files. To overcome this, activate Disable OPI.

Show external images searches: Displays the path(s) in which the system has found and linked to external images.

Optimize Illustrator Patterns: Illustrator writes patterns in a different way than PostScript does, default behavior is that Illustrator patterns will be optimized.

Normally this speeds up the further processing (less linework in your file) sometimes, though, the optimization itself may take some time.

Use PostScript rules for color conversion: This toggle makes sure that all color management will happen using PostScript rules, instead of BGCMS. Most options in the [Color Management tab](#) will be grayed out.

Recognize ArtScreen (lpi-angle): Check this box and the application will try to recognize screens that are created in ArtPro applications and swap them to Esko screens.

If a (E)PS or PDF file with an unknown ArtScreen is imported, an error will be generated.

For more information, please refer to the ScreenFilter user manual.

Object Based Color Conversion: Object by object color conversion does not take overprint and transparency interaction between objects into account, and may therefore produce undesirable results. However, it may greatly improve editability.

Output

As a CT folder is no longer automatically created in every new job folder, you can specify your own favorite local images folder here.

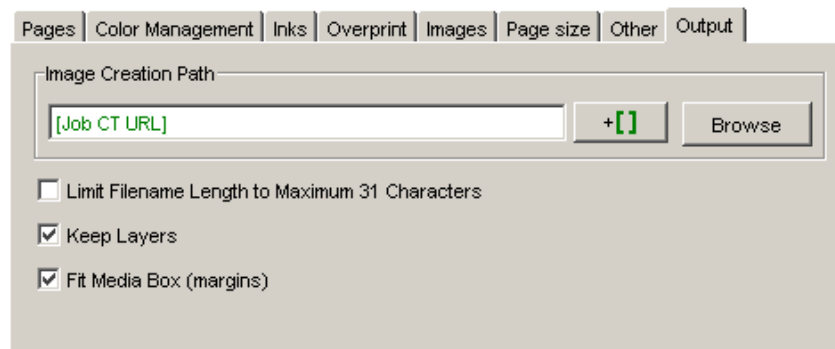


Image Creation Path

Browse to a location or define one with SmartNames.

If, during normalization, extra image data is created, you can indicate here where these images should be written.

If no output directory is specified, then CT images generated during the conversion will be saved in the current working directory.

Limit Filename Length to Maximum 31 characters

Use this toggle if you want to make sure your imported files can be seen/read from/by all Apple operating systems (CTs that are kept on a Windows share but might be edited on a Mac for example).

Keep Layers

Keeps the Illustrator layer information. This will only work for Illustrator 8 files. It will not work for native Illustrator 9 files (or higher).

When normalizing PDF 1.5 files (or higher), the 'Keep Layer' option will also preserve the PDF layers.

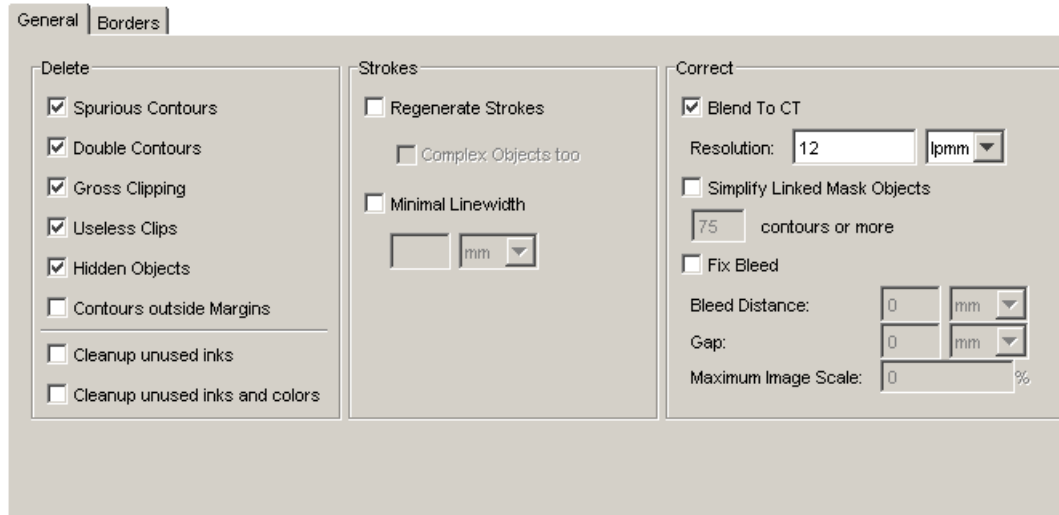
Fit Media Box (margins)

Allows you to automatically fit the margins.

2.7.93 Optimize and Clean (PSFix)

PSFix is a type of technology that will optimize and clean up ex-PS/PDF data: removing redundant objects is a typical action. The optimization and clean-up will speed up further processing. PSFix is typically executed right after a conversion to Normalized PDF.

General



Delete

- **Spurious Contours** are:
 - Single points that only have a fill attribute.
 - Single points that have a stroke attribute with but or square caps.
 - Contours that contain two points, that have a mask attribute, but do not mask out any object. These contours can be created by accident or generated by PostScript printer drivers.
- **Double Contours**: removes double contours by recombining the fill and stroke attribute of two identical objects into one object again. Double contours are created when importing PostScript files, in which case an object with a fill and a stroke attribute is split into two objects with each one attribute.
- **Gross Clipping**: selects and deletes the objects whose bounding boxes fall completely outside the bounding box of the clipping path. Gross Clipping can also be applied on composed text and externally referenced objects.
- **Useless Clips**: Selects and deletes useless clipping paths, objects that are totally clipped away or objects that are totally covered by other objects.
- **Hidden Objects**: Selects and deletes objects that are covered completely with another object, because the designer did not want to delete them or could not delete them on a certain page, as they were components of a master page.
- **Contours outside margins**: Selects and deletes contours outside the margins.
- **Clean up unused inks**: Automatically removes all unused inks.

Strokes

- **Regenerate Strokes**: searches and selects simple objects that contain 20 points or less and converts them to the equivalent strokes. Select the **Select Complex Objects too** option if you want the system to search all objects that might be degenerated strokes, instead of limiting the search to objects of less than 20 points.
- **Minimal Line width** defines the minimal width for strokes.

Correct

- **Blend to CT** converts all recognized blends in the job to externally referenced CT objects.

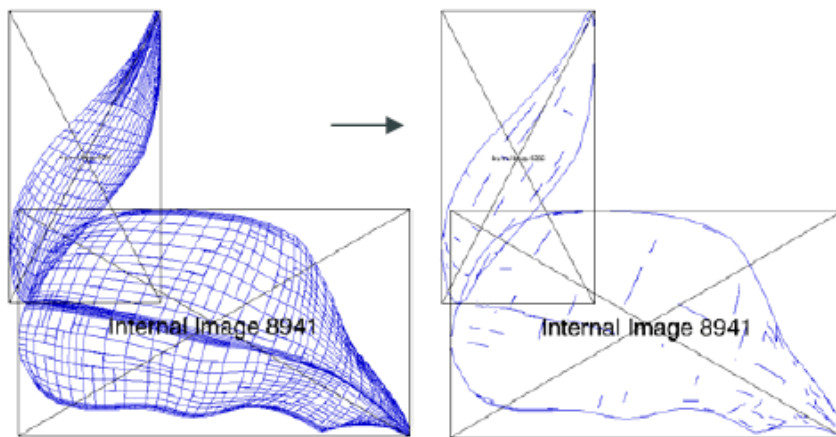
In PostScript files, blends are constructed by a large amount of contours that lie close to each other and that have slightly different colors. The more accurate the blend, the more contours and colors. This leads to very large jobs and decreases the processing speed.

For each detected blend, **Blend to image** creates 1 image.

Note: After conversion to different images, the display of the blend can change. However, the conversion does not affect the printed result.

Resolution defines the resolution of the blend that was converted into a CT.

- **Simplify Linked Mask Objects** will try to reduce the number of linked contours for complex mask objects. This option is only useful for linked mask objects with a huge number of contours (>50). When used with a lower number, PSFix will become less powerful and it will have no effect whatsoever on the complexity of the mask object.



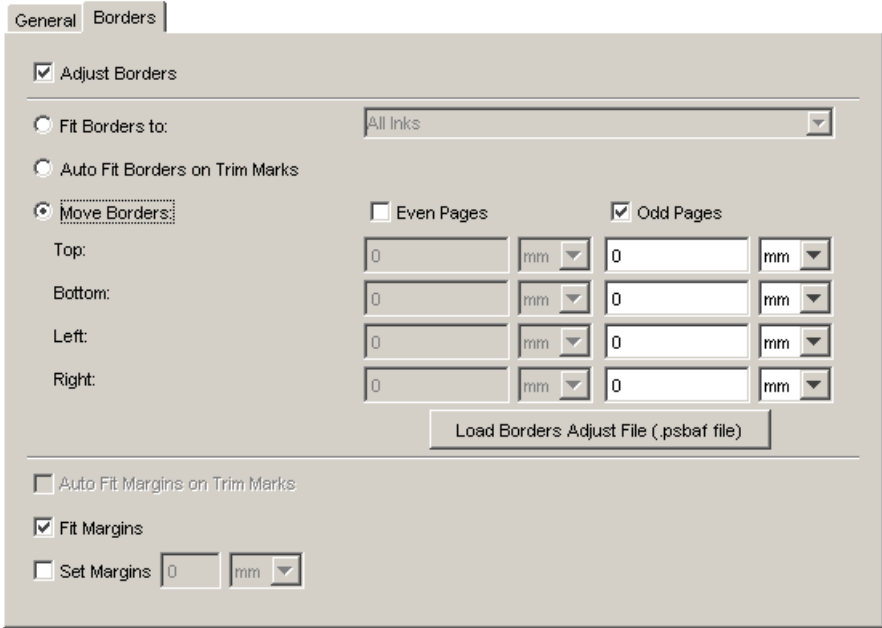
- **Fix bleed:** It is a common occurrence for pages to be supplied without any bleed. This means that either a new page has to be sent or the page has to be manually corrected in PackEdge. Especially the latter can be a time-consuming process. **Optimize and Clean** (PSFix) now has the capability of fixing this problem automatically, by adding bleed where there is none, or not enough. There are three parameters to control the process:
 - **Bleed Distance** is the amount of bleed that is required. For any object that does not have this amount of bleed the correct bleed will be created.
 - **Gap:** any object that is inside the borders, and that is closer than this distance to the borders will be considered to require bleed. Typically, this value will be small. It makes sure that objects which do not really touch the borders because of inaccuracies in the design, are taken into account for bleed creation anyhow.
 - **Maximum Image Scale:** in the case of linework objects all the bleeds that are created are added to an additional layer called "Bleedlayer". This layer can be removed at all times to return to the original file. This is not possible with images, however. Images are modified in their original location. There are two possible ways in which images can be handled. When an image is larger than its clip mask, it is possible to create bleed by extending the clip mask of the image; hidden pixels will become visible and the missing bleed problem is fixed. When the image is not larger than its clip mask, the only way in which bleed can be created is by scaling the image. When doing so, the position of the image inside the page does not change, but it is disproportionately scaled to create the correct bleed. To avoid an unacceptable distortion of

the image this parameter determines how much scaling of an image is allowed. If no scaling is desired, then the value can be set to 0.

In order for the bleed-fixing to work properly it is imperative that the borders of the file are set correctly. **Optimize and Clean (PSFix)** also has functions to correct borders and it now also has a new function to set the borders of a file automatically to the trim marks.

When no bleeds have been corrected the exit status of the Automation Engine task will be OK (green). When bleeds have been corrected, the exit status will be a warning (orange), and when the maximum scale factor was not enough to create bleed for an image an error will be generated (red).

Adjust Borders



The screenshot shows the 'Adjust Borders' dialog box. The 'Borders' tab is selected. The 'Adjust Borders' checkbox is checked. The 'Fit Borders to:' dropdown is set to 'All Inks'. The 'Auto Fit Borders on Trim Marks' checkbox is unchecked. The 'Move Borders:' radio button is selected, and the 'Odd Pages' checkbox is checked. There are four rows of input fields for 'Top', 'Bottom', 'Left', and 'Right' borders, each with a numeric value of 0 and a unit dropdown set to 'mm'. A button labeled 'Load Borders Adjust File (.psbaf file)' is positioned below these fields. At the bottom of the dialog, the 'Auto Fit Margins on Trim Marks' checkbox is unchecked, the 'Fit Margins' checkbox is checked, and the 'Set Margins' checkbox is unchecked with a numeric value of 0 and a unit dropdown set to 'mm'.

Select the Adjust Borders to make the options available.

To move any of the four borders numerically, to measure manipulations applied to borders and to save the borders manipulation to an option file. This can be done separately for odd and even pages.

Fit borders to : select from the dropdown list:

- All inks
- Any ink in your file

'**Auto fit borders on trim marks**' tries to detect the presence of trim marks. When valid trim marks are recognized the borders will be set accordingly. When not, the borders will remain unaffected. This feature is useful when the trim size of the supplied files is unknown and unreliable (for example missing or wrongly defined trim box in a PDF file) and the user wants to determine it automatically. Even when the desired trim size is known to the user, it can still be interesting to see if the automatically determined border size corresponds to the desired one: a difference between the two is likely to point to an error. Users very often neglect to work with accurately set borders, because the intelligent placement of files in FastImpose and Plato does not require them to do so. However, if you want to use the function to fix wrong bleeds it is a requirement to have the borders defined correctly.

You can define the borders: choose **Even** and/or **Odd** and fill out the specifications for **Top**, **Bottom**, **Left** and **Right**.

You can load a Borders Adjust file (.psbaf file), you can create such a file in the interactive editor.

Set Margins: enter a value for the margins.

2.7.94 Optimize PDF Document

The **Optimize PDF Document** task enables you to make some prepress-specific changes to a PDF document.

If you want to output the files in a different location, you can define that location in **Output in**. You can enter a path manually, browse to a location, or use Smart Names.

If you want to give the output file a different name, enter it in **File Name** field. You can enter a name manually, browse to an existing file to replace it, or use Smart Names.

Document

Select **Preserve fonts** if you want to make sure external systems receiving the PDF file can print or display the fonts correctly. You can choose:

- **Embed Fonts:** this embeds all used fonts.
- **Embed & Rename:** this embeds Base 14 Fonts fonts under a different name (their name in the Esko PDF Library), and all other fonts under their existing name. This prevents Acrobat from replacing the document font with a local font of the same name when using live text.

Note: Base 14 fonts are: Courier, Courier Bold, Courier Oblique, Courier Bold-Oblique, Helvetica, Helvetica Bold, Helvetica Oblique, Helvetica Bold-Oblique, Times Roman, Times Bold, Times Italic, Times Bold-Italic, Symbol and Zapf Dingbats.

- **Vectorize:** this converts all text to outlines.

Apply document's rotation flags: A rotation flag is a property of a PDF page that specifies by what angle the contents should be rotated when previewing or printing the file.

Some older applications/output devices ignore or mishandle the rotation flag. If you are sending your file to such an application/device, select this option to execute the rotation specified in the rotation flag (that is, actually rotate the file's contents).

Select **Outline strokes** if you want to convert the document's strokes to outlines. You can choose either:

- **All:** this converts all strokes to regular outlines.
- **Dash Only:** this converts dashed strokes (only) into individual outline objects. This can significantly improve trapping.

If you want to change the strokes' width in your document, select **Increase line width** and enter the **Minimum Line Width** to use. All strokes widths below this value will be enlarged to it.

Rotate Job: Use this parameter to change the rotation of the contents of your file by **0°**, **90°**, **180°** or **270°**.

- If **Apply document's rotation flags** is off, this changes the rotation flag's value but doesn't rotate the file's contents.
- If **Apply document's rotation flags** is on, this rotates the file's contents.

Adjust Page Box: This option allows to change some or all of the document's page boxes, or add page boxes in your document. See [Adjust Page Box](#) on page 395 for details.

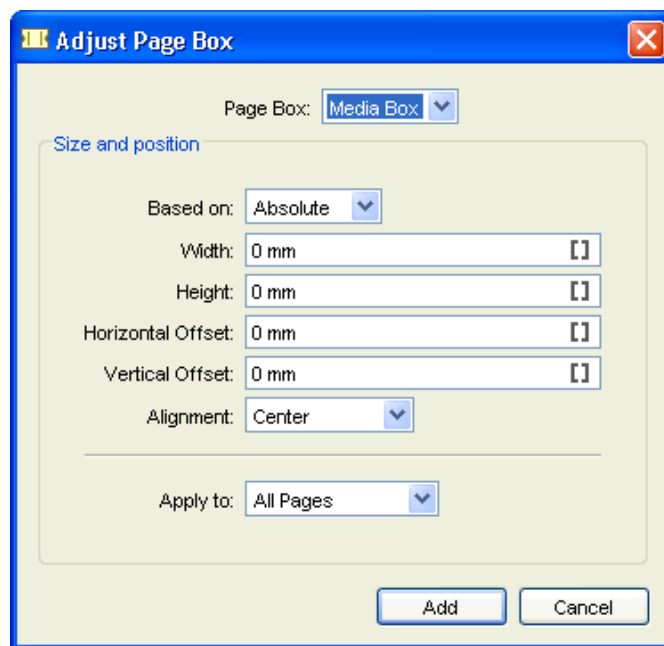
Outline Page Box: This option allows to outline some or all of the document's page boxes. Use this if you want to show the page boxes on proof (for example, you can outline the trim box to show where the artwork will be).

See [Outline Page Box](#) on page 397 for details.

Adjust Page Box

To add or change a page box:

1. Click the **Add** button. This opens the **Adjust Page Box** pop-up.



2. Select the **Page Box** to add or change.
3. Choose what you want the new page box size to be **Based on:**
 - Choose **Absolute** if you want to define the page box size independently of the document's contents or existing page boxes.
 - Choose **Content** if you want to base the page box size on the document's contents. This takes clipping paths and masks into account.
 - Choose **Media Box**, **Crop Box**, **Trim Box**, **Bleed Box**, or **Art Box** if you want to base the page box size on another one of the document's page boxes.
4. Determine the new **Width** and **Height**.
 - If you have chosen to use an **Absolute** size, enter the page box's **Width** and **Height**.
 - If you have chosen to base the page box size on the document's **Content** or on another page box, you can adjust the width and height in **Increase Width** and **Increase Height**.

Note:

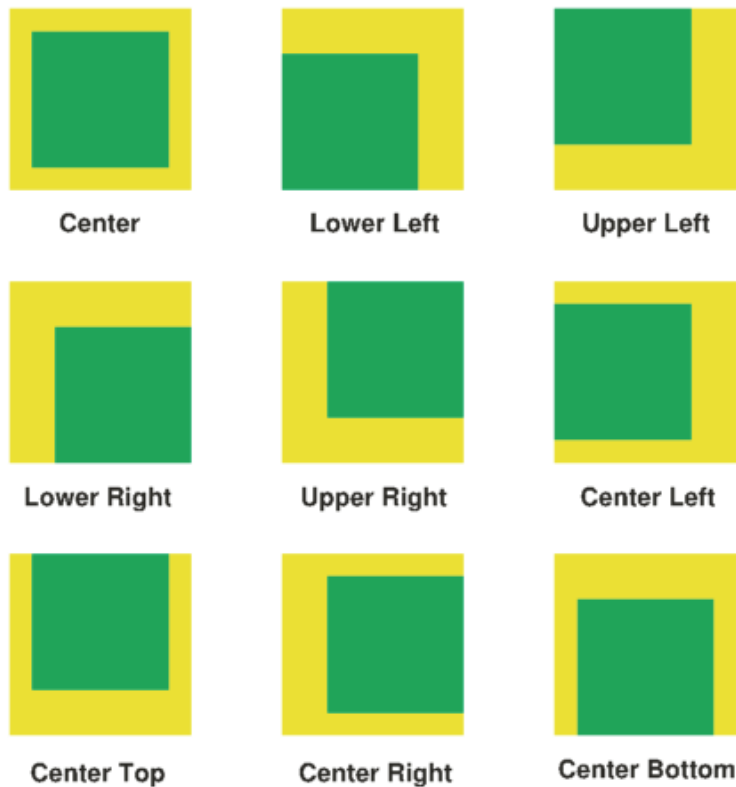
- Use a negative value to reduce the page box size instead of extending it.
- You can use Smart Names.

5. If desired, enter an **Horizontal Offset** and / or a **Vertical Offset**.

- A positive horizontal offset will move the page box to the right, a negative one to the left.
- A positive vertical offset will move the page box to the top, a negative one to the bottom.

You can use Smart Names.

Note: This is applied after the **Alignment!**

6. Choose the page box's **Alignment** (relatively to the old page box):

7. Apply your changes to either:

- **All Pages,**
- **Odd Pages Only,**
- **Even Pages Only.**

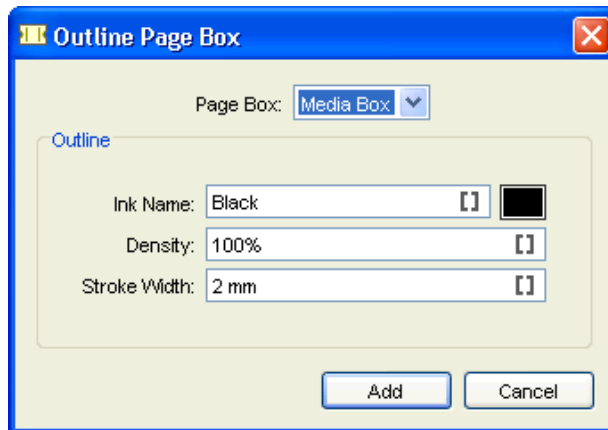
8. Click the **Add** button.

Note: You can **Remove** a page box you have added.

Outline Page Box

To outline a page box:

1. Click the **Add** button. This opens the **Outline Page Box** pop-up.



2. Select the **Page Box** to outline.
3. Select the **Ink** to use.
You can type in the **Ink Name**, use Smart Names, or click the color patch to open a color picker.
4. Enter the ink **Density** to use.
You can use Smart Names.
5. Specify the **Stroke Width** to use.
You can use Smart Names.
6. Click the **Add** button.

Note: You can **Remove** a page box outline you have added.

Layers

Use the **Layers** tab to rename, remove, or add layers, as well as handle the PDF's layer printability.

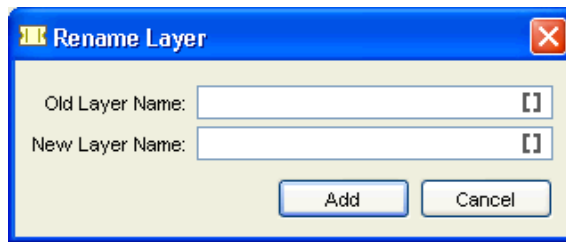
Note:

- The layer functionality is executed in the order shown in the user interface (**Action 1, Action 2...**).
- You can use wild cards (*) in the layer names.

Action 1: Rename Layers

Use this parameter to rename layers.

Click **Add** to open the **Rename Layer** pop-up, then enter the **Old Layer Name** and the **New Layer Name**. You can use Smart Names and / or wild cards (to replace part of the layer name).

**Tip:**

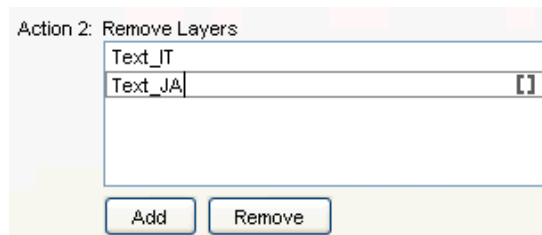
You can also merge layers by renaming a layer to an existing layer's name, or by renaming several layers to the same name.

The layers to merge have to be directly on top of each other in the file.

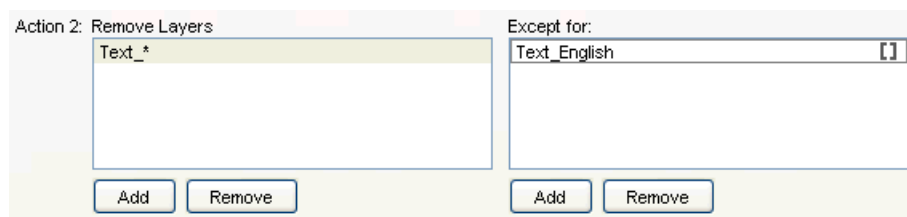
Action 2: Remove Layers

Use this to remove layers from the file.

Click **Add** and enter the names of the layer you want to remove. You can use Smart Names and / or wild cards.



Except for: When using a wild card to remove layers, use this parameter to keep certain layers, even if they match the name entered in **Remove Layers**.

**Action 3: Add Layers**

Use this to add empty layers to all pages of your PDF.

Click **Add** and enter the name of the layer to add. Repeat as necessary.

Update Layer Printability: Select how to handle layers' printability in your document. You can choose either:

- **Don't Change Layer Printability,**
- **Make All Layers Printable,**
- **Remove Non Printable Layers.**

Images

Downsample Images

Select this parameter if you want to downsample images' resolution. This makes the parameters underneath it available.

- Choose what resolution you want to **Downsample bitmap** (black & white or one-bit) **images to, if their resolution is above** a certain resolution.
- Choose what resolution you want to **Downsample contone** (grayscale or color) **images to, if their resolution is above** a certain resolution.

You can use Smart Names.

Crop Images

Select this parameter to crop images clipped by a clipping path to the rectangular bounding box of the clipping path. This makes the **Split images area...** parameter available.

Split images area when at least ... % of the images is clipped away: This parameter defines how the task will crop the image, depending on how much of it is clipped away (for example, for images that have multiple clipping paths on them).

- If the part of the images clipped away is less than what you enter here, the images will stay together and be cropped to the bounding box of the combined clipping paths.
- If the part of the images clipped away is greater than what you enter here, the images will be split into multiple clipped images, each cropped to the rectangular bounding box of its clipping path.

The default percentage is 50%.

Recompress Image Data

This option allows you to re-compress images in the PDF file. Selecting it makes the parameters underneath available.

- For **Color**, **Gray Scale** and **Soft Mask Images**, you can choose between the **JPEG** and **ZIP** compressions.
For **JPEG** compression, you can choose between **Minimum**, **Low**, **Medium** or **High Quality**.
- For **Bitmap Images**, you can choose between the **CCITT Group 3**, **CCITT Group 4**, **ZIP**, and **Run Length** compressions.

Note: DeviceN images will always be compressed to ZIP files when **Recompress Image Data** is selected.

2.7.95 Optimize PDF Separations

The **Optimize PDF Separations** task enables you to make changes to a PDF document's separations.

If you want to output the files in a different location, you can define that location in **Output in**. You can enter a path manually, browse to a location, or use SmartNames.

If you want to give the output file a different name, enter it in **File Name**. You can enter a name manually, browse to an existing file to replace it, or use SmartNames.

In **File Type**, choose if you want to output a PDF or a Normalized PDF file.

Colors

Use the **Colors** tab to define the conversion settings needed to convert the document's (non-process) colors.

Note: The conversion is done by Esko's Color Engine.

Embed Profiles Mode: This parameter handles color profiles embedding. You can select:

- **Do not change:** This doesn't make any changes to color profiles embedding.
- **Embed:** This embeds all color profiles.
- **Embed & Overwrite:** This re-embeds all color profiles, overwriting profiles already embedded.
- **Remove All Embedded:** This removes all embedded profiles.

Note: The task uses the embedded profile(s) as source profiles for conversions.

Convert RGB, LAB, Gray To CMYK: Enable this parameter to convert all non-process color spaces (except spot colors) to CMYK.

Note: This will generate errors and/or warnings when trying to convert non-separable blend modes.

Fail When Converting Color Spaces: Color converting objects that have transparency effects (like blend modes) can generate visually unexpected results. You can:

- leave this option selected if you want the task to stop with an error when this problem occurs.
- deselected this option if you want the task to only generate a warning for this problem.

Default CMYK Profile: Select the default profile to use for CMYK objects without an embedded color profile. It will be used as a source profile for Color Space conversions.

Note: You can choose any CMYK profile in the Color Engine database.

Default RGB Profile: Select the default profile to use for RGB objects without an embedded color profile. It will be used as a source profile for Color Space conversions.

Note: You can choose any RGB profile in the Color Engine database.

Default Gray Profile: Select the default profile to use for gray objects without an embedded color profile. It will be used as a source profile for Color Space conversions. You can choose one of the following:

- Any gray profile in the Color Engine database.
- **Convert to Black Channel**, if you want your gray-scale object to only have a value in the Black separation when outputting it to CMYK.

Output Color Profile: Enter the Color Space of the output device (e.g. proofer or press) in this parameter.

Note: You can choose any CMYK profile in the Color Engine database.

Update spot color equivalents: This option handles CMYK values for spot color inks. You can:

- enable this option to overwrite the CMYK values found in the file with the CMYK values stored in the ink book.
- leave this option deselected to use the CMYK values found in the file for spot color inks.

Spot Color Lookup: Use this parameter to select where to look for the spot colors' information. Choose one of the following methods:

- **Look in Ink Books First:** the task first looks in the ink book you defined in the **Preferred Ink Book** option (see [Optimize PDF Separations](#) on page 399) .
If it doesn't find the spot colors' information in that ink book, it then looks for it in the job.
- **Look in Ink Books Only:** the task only looks in the ink book you defined in the **Preferred Ink Book** option.
If it doesn't find the spot colors' information in that ink book, it ends in error.
- **Use Colors in File:** the task looks for the spot colors' alternate color values (generally in the CMYK color space) in the file, and uses those values.

Preferred Ink Book: Choose the desired Ink Book that will be used in this task.

PANTONE Color Lookup: Choose the settings for the PANTONE colors conversion. This can be one of the following:

- **Use destination profile:** Use the color profile defined in the **Output Color Profile** to determine CMYK equivalents of the PANTONE Color.
- **HP Indigo CMYK 1.1:** Use CMYK equivalents for the HP Indigo Press.
- **Pantone Color Bridge Coated:** Use the CMYK equivalents defined in the Pantone Color Bridge Coated guide.

Inks

Use the **Inks** tab to define how to handle inks. Use the following parameters to rename, convert or remove inks.

Note: The functionality is executed in the order shown in the user interface (**Action 1, Action 2...**).



Caution: All ink-related parameters are case-sensitive.

Action 1: Rename Inks

Use this parameter to rename inks.

Click **Add** to open the **Remap Ink** pop-up, then enter the **Old Ink Name** and the **New Ink Name**.

Tip: You can also merge inks by remapping an ink to an existing ink's name, or by remapping several inks to the same name.

Note: If your original ink has a PANTONE Suffix, add it to the **Old Ink Name**, even if you are changing this suffix. The reason for this is that when you use both the **Remap Inks** and **Pantone Suffix** parameters, the task renames the inks first.

Pantone Suffix: With this parameter, you can change the suffix (for example CV, CVC, CU...) of PANTONE colors.

- **Do not change:** This keeps the suffix as it is.
- **Convert:** This changes the suffix to the one you select in the **To** list, for all PANTONE colors.
- **Convert If Found In Ink Books:** this changes the suffix to the one you select in the **To** list, for PANTONE colors found in the ink books you defined (see [Optimize PDF Separations](#) on page 399).

Note: This can also merge inks (for example if you have PANTONE 484 U and PANTONE 484 C, and you changing all the suffixes to C).

Action 2: Convert Inks

Use this parameter to convert spot colors to CMYK. You can use wild card characters (*) in the ink's name.

Action 3: Remove Inks

Use this to remove inks from the file.

Click **Add** and enter the names of the ink you want to remove. You can use SmartNames and/or wild card characters.

Except for: When using a wild card character to remove inks, use this parameter to keep certain inks, even if they match the name entered in **Remove Inks**.

For example, you can remove all the inks except the die cut ink, to send your file to the die-cutter.

The screenshot shows a dialog box titled 'Action 3: Remove Inks'. It has two main input areas. The first is labeled 'Remove Inks' and contains a single asterisk (*) in a text box. Below it are two buttons: 'Add' and 'Remove'. The second area is labeled 'Except for:' and contains the text 'diecut' in a text box. Below it are also two buttons: 'Add' and 'Remove'.

Preserve Separation Interaction: Use this parameter to define what to do when the separations you remove contain objects knocking out other separations.

- **None:** this only deletes knockout objects that are only defined in the removed separation(s).
Use this when removing separations that don't influence the final result (for example die cut separations).
- **Standalone:** this converts knock-out objects that are only defined in the removed separation(s) to white knock-out objects defined in one of the remaining separations. The remaining separations keep looking the same.
This method is recommended for most use cases.
- **For Versioning:** this converts knock-out objects that are only defined in the removed separations to white overprinting objects.
Those are defined in a color space made from the remaining separations of the file, and overprinting on all other separations.
This ensures that when re-adding separation layers for versioning, the final result looks the same as the original file.

Note: This may create objects in DeviceN color spaces, that some applications can't handle. Only use this method if you want to do versioning on your file.

Remove unused separations: Enable this to remove unused separations (separations containing no objects) from the file.

Ink Properties

The **Ink Properties** tab allows to apply ink types and printing methods to the inks in native and normalized PDF files. These properties can be used to steer behavior of tasks later on in the workflow, for example the **Extract Inks** task for printing methods.

Ink Name	Ink Type
DieCut	Technical
High Gloss	Varnish

Ink Name	Printing Method
PANTONE Process Yellow C	Screen

Apply Ink Type : You can define the **Ink Type** for each **Ink Name**. The task will accept Ink Names containing wild cards (asterisks).

Use **Add** to add an entry to the **Apply Ink Type**. You can type in the **Ink name** and **Ink type** or use corresponding SmartNames.

Note:

- The Ink Name may contain wild cards (asterisks). In such instances, the inks matching the pattern of the **Ink Name** will have the specified properties set. If one ink matches more than one pattern, the first one is taken.
- When matching, the letter case is ignored.

Ink Type can be:

- **Normal, Opaque, Varnish, Technical.** Limitations for **opaque** can be imposed by CMS.

Note: process colors (CMYK) can only be **normal**

Use **Add** to add an entry to the **Apply Ink Type**. You can type in the Ink name and Ink type or use corresponding SmartNames.

Apply Printing Method : Set the **Printing Method** corresponding to **Ink name** here. Choose from predefined printing methods:

- **Offset, Gravure, Flexo, Letterpress, Screen, Inkjet, Toner, Thermal Transfer, Other, Unknown.**

Note: You can use SmartNames to specify **Printing Method**.

Overprints

To Knockout

Use this tab to change the Overprint versus Knockout behaviour of inks in a PDF file.

All overprint objects: Select this to disable all overprints in your file.



Attention: This can change the file's appearance.

White text: Use this to set white text to knockout.

White vector: Use this to set white line art to knockout.

Technical objects: Use this to set technical objects to knockout.

Varnish objects: Use this to set varnish objects to knockout.

Opaque ink vector: Use this to set line art in opaque ink to knockout. Opaque inks are defined in the ink book, using the Color Engine Pilot.

Opaque ink text: Use this to set text in opaque ink to knockout. Opaque inks are defined in the ink book.



Caution: When an ink is explicitly defined as "opaque" in the ink book, you cannot make it "normal" with this task. You will get a warning if you try.

To Overprint

All knockout objects: This parameter changes all knockout objects to overprinting objects, and modifies their color space(s), so that the file keeps the same appearance.



Attention:

- Only use this parameter in versioning situations.
- This parameter affects how you can place the file afterwards. If you place this overprinting file on top of another file, all the bottom file's channels not present in the top file will be visible.

Black text: Use this to set black text to overprint. You can choose to make:

- **All** black text overprint, or
- black text **Smaller Than** a defined value overprint.

Black vector: Use this to set black line art to overprint.

Technical objects: Use this to set technical objects to overprint.

Varnish objects: Use this to set varnish objects to overprint.

Opaque ink vector: Use this to set line art in opaque ink to overprint. Opaque inks are defined in the ink book.

Opaque ink text: Use this to set text in opaque ink to overprint. Opaque inks are defined in the ink book.

2.7.96 Outline Fonts

This task will contourize ALL fonts in a file (and in all files it refers to). All live and composed text will be contourized. 'Internal' text (executed SmartMarks for example) is already contourized.

The task will save the output as a copy of the original job, and will add the suffix `_c`, this behavior can be overruled by using SmartNames.

Tip:

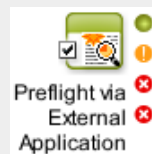
When you processed a file that did not contain any fonts, the details pane of the [Tasks View](#) will tell you so.

2.7.97 Preflight via External Application

The **Preflight via External Application** task allows you to preflight a PDF file by integrating external (non-Automation Engine) preflight software into Automation Engine.

Using the Task in a Workflow

Note: When preflighting in a workflow, the **Preflight via External Application** workflow step has four output pins:



- **Preflight OK:** your files will go through this pin if the preflight gave an OK status, or if you chose **Task ends OK** for **When warnings appear** in the ticket (and the preflight gave warnings). See [Task Parameters](#) on page 406.
- **Preflight with Warnings:** your files will go through this pin if the preflight gave an error or a warning (depending on what you chose in the ticket).
- **Preflight with Errors:** your files will go through this pin if the preflight gave an error or a warning (depending on what you chose in the ticket).
- **Error:** your files will go through this pin in case of a processing error unrelated to the preflight status.

Supported Preflight Software

Third party preflight products that Automation Engine can link up to are:

- Callas PDFInspektor (batch version): versions 2, 3 and 4

- Enfocus PitStop Server: versions 3, 4, 8, 9 and 10

Note: The preflight software should be able to run in the background.

Task Parameters

Tool: select a supported preflight tool from the list.

Note:

Changes made in the Enfocus PitStop Server application's preferences will not affect the Automation Engine Preflight task.

To change the Report Language, Profiles and Action Lists folders, use **Configure** (go to **Tools > Configure > Preflight Tools**).

Executable File: indicate the path to the preflight engine executable or dll (PDF Inspektor only).

Profile: browse to, or select a preflight profile from the list. Note that a profile can also contain action lists.

Tip: You can also create these preflight profiles with Adobe Acrobat Professional.

Action: use the buttons at right to **Add** or **Remove** actions from the list (only from PitStop Server 4). You can also reorganize the list using **Move up** and **Move down**.



Attention: Tickets created with versions of BackStage prior to BackStage 70R5 will be loaded with one single action in the list.

Report: define the name and location of the preflight report file that will be generated. You can use SmartNames.

When rules are violated: choose what status you want the task to have when preflighting gives errors:

- **Task ends with a warning**
- **Task fails**

When warnings appear: choose what status you want the task to have when preflighting gives warnings:

- **Task ends OK**
- **Task ends with a warning**
- **Task fails**

Preflight Report

The **Preflight Report** file generated by the task contains the **Errors** and **Warnings** found by the preflight, and any **Fixes** it implemented.



Preflight Report



Errors were found on page 1 and in general document properties

- Document contains layers
- Document uses 0 separations, should be equal to 1
- Transparent objects have been found (1x on page 1)
- Resolution of color or grayscale image is less than 250 dpi (1x on page 1)
- RGB color is used (1x on page 1)



Warnings

- Compression ratio of image is more than 10 (1x on page 1)



Fixes

- Removed marked content (2x on pages 1-2)
- Removed form fields (1x on page 1)

If you generated a PDF report, you can open it in the Viewer (or the application associated with this file type).

2.7.98 Preflight with PitStop

The **Preflight with PitStop** task allows you to preflight a PDF file in Automation Engine, without needing external preflight software.

This task uses the latest PitStop preflight technology.

Using the Task in a Workflow

When preflighting in a workflow, the **Preflight with PitStop** workflow step has four output pins:

- **Finished OK:** your files will go through this pin if the preflight gave an OK status (with no warnings or errors).
- **Finished with Warnings:** your files will go through this pin if the preflight gave warnings (but no errors).
- **Finished with Errors:** your files will go through this pin if the preflight gave errors.
- **Error:** your files will go through this pin in case of a processing error unrelated to the preflight status.

To customize what to do when you get warnings from the preflight:

1. Right-click the workflow step and select **Configure Outputs...**
2. In the pop-up that opens, select what to do under **Warning handling:**
 - select **OK on Warning** to send the files that have preflight warnings through the **Finished OK** output pin,
 - select **Error on Warning** to send them through the **Finished with Errors** output pin,

- select **Dedicated handling of Warning** to send them through the **Finished with Warnings** output pin.

3. Click **OK**.

Tip:


You can also use the **Configure Outputs...** pop-up to:


- Send the successfully preflighted files further in the workflow (select **Add an extra output pin where the task's inputs will be when no errors occur** and connect that pin to the workflow step of your choice).
- Use the preflight reports in your workflow (select **Add an extra output pin where the task's reports will be put** and connect that pin to the workflow step of your choice).


Settings

Preflight Tab

Settings for Step: Preflight with PitStop

 Output in:

 File Name:



Preflight Certified Report Color

Action Lists:

Profile:

When rules are violated: Task ends with warning
 Task fails

When warnings appear: Task ends OK
 Task ends with warning
 Task fails

Action Lists: Define which Enfocus action list(s) you want to use on your file. Use the **Add** or **Remove** buttons to add/remove **Action Lists**. You can reorganize the list using **Move up** and **Move down**.



Attention: Tickets created with versions of BackStage prior to BackStage 7R5 will be loaded with one single action in the list.

Profile: You can browse to, or select a preflight profile from the list. Note that a profile can also contain action lists.

Tip: You can create these preflight profiles with Adobe Acrobat Professional.

Define how you want the task to behave depending on the preflight results:

- In **When rules are violated**, choose what happens when you get preflighting errors:
 - **Task ends with a warning**
 - **Task fails**
- In **When warnings appear**, choose what happens when preflighting gives warnings:
 - **Task ends OK**
 - **Task ends with a warning**
 - **Task fails**

Certified Tab

Settings for Step: Preflight with PitStop

Output in: BROWSE [X]

File Name: BROWSE [X]

Preflight | **Certified** | Report | Color

Certify: Make Certified [v]

User: [CSR for the Job] [X]

E-Mail: [Email Address of the CSR for the Job] [X]

Session Comment: Preflighted by [Application Name and Version] on [Date] [X]

Allow to Roll Back

You can certify a PDF file in this task (for example, when a Pre-Media company wants certified files to be sent to the Converter). This certification can be verified using **Verify Certification** task, for example in a converter's workflow (Read more in [Verify Certification](#) on page 521).

- Set **Certify** to:
 - **Make Certified:** to certify the file.
 - **Keep Certified:** to preserve the existing certified/uncertified status of the file.
 - **Make Uncertified:** to remove all the certified information from the PDF file.

If your choice is **Make Certified** or **Keep Certified**, you need to set the following items:

- **User:** fill in the user name to be added to the certified PDF.
- **E-Mail:** fill in the E-mail that needs to be added to the certified PDF.
- **Session Comment:** use this space for comments.
- **Allow to Roll Back:** Enable (default) this, if you want the users to be able to roll back to previous sessions of the file (for example to compare versions). If you disable it, users cannot roll back to previous versions. But the Certified history will be in the file.

Report Tab

- **Generate a Report** : Enable this (default) if you want to generate a report of the preflighting session.
- **Settings for the generated report** :
Choose the settings according to your report requirements:
 - **Units** : select from **mm**, **cm**, **Point** and **Pica**.
 - **Language** : choose from the list of eight major languages.
 - **Type** : Set your preferred file type. Choose from **PDF**, **PDF + XML** and **XML**.

Read more in [Preflight Report](#) on page 410

Color Tab

You can correct color space errors (for example convert RGB colors to CMYK) in this tab. Select **Enable Color Management** and choose the **Grayscale**, **RGB**, **CMYK** and **Lab** ICC profiles for color conversions.











Preflight Report

The **Preflight Report** file generated by the task contains the **Errors** and **Warnings** found by the preflight, and any **Fixes** it implemented.

Report: define the name and location of the preflight report file that will be generated. You can use SmartNames.







Errors were found on page 1 and in general document properties

-   Document contains layers
-   Document uses 0 separations, should be equal to 1
-   Transparent objects have been found (1x on page 1)
-   Resolution of color or grayscale image is less than 250 dpi (1x on page 1)
-   RGB color is used (1x on page 1)

Warnings

-   Compression ratio of image is more than 10 (1x on page 1)

Fixes

-   Removed marked content (2x on pages 1-2)
-   Removed form fields (1x on page 1)

If you generated a PDF report, you can open it in the Viewer (or the application associated with this file type).

2.7.99 Prepare for Inspection

You can set measurement points in from the design which can be later monitored through an automated /sem-automated inspection system. This will take a JDF-Layout file to create a JDF-Inspection file.

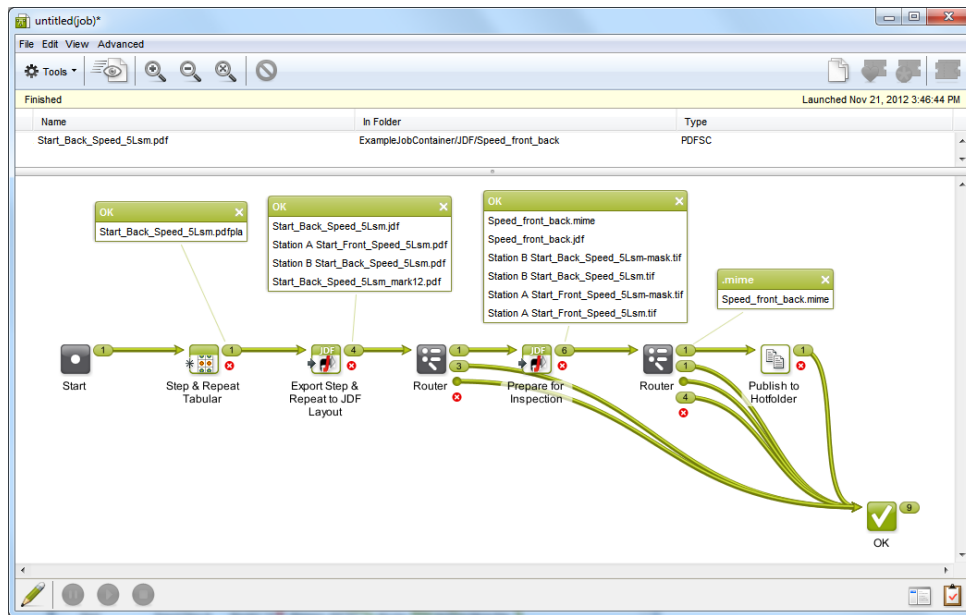
The JDF-Inspection file contains the layout of the sheet/sheets and for each one-up information on the following:

- the contour of the retained area (**ClipPath**)
- the location of the barcodes (**IdentificationField**)
- the location of the text areas (**IdentificationField**)
- the location and values of Spectrometer measurements (**CIELabMeasurement**)

The **Prepare for Inspection** task generates a preview of the one-up PDF files in RGB TIFF format, and a rendering of the ClipPath as a monochrome mask in TIFF format. The task packs the results as a MIME file which can be delivered to the inspection system as a single self-contained file. Read more about the task settings in [Settings](#) on page 413

Sample Workflow using Prepare for Inspection

This sample workflow starts with a PDFPLA Step and Repeat file. As a first step, you can export the file as a JDF Layout file. This creates a JDF file and self-contained PDF files for each one-up.



Prepare JDF for Inspection task uses this JDF layout to create the MIME file containing the JDF file and the TIFF previews. Next task publishes this MIME file to a hotfolder to be picked up by the designated inspection system.

You can use the **Router** tasks to select the correct file as input file.

Preview and Cut Mask Samples

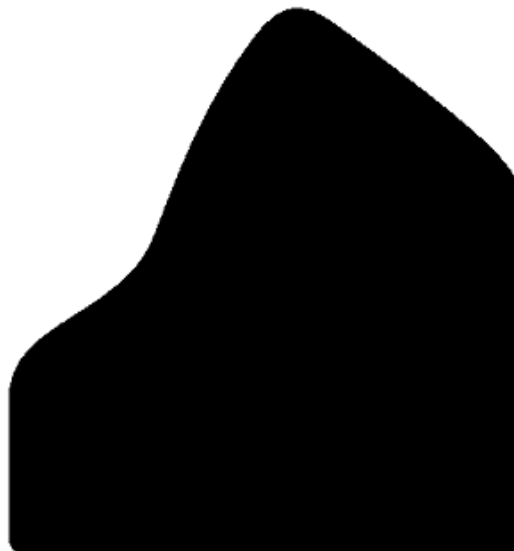
Speed Label One-Up:



Preview Sample: Note that the cut contours are not visible in the Preview.



Cut Mask Sample:

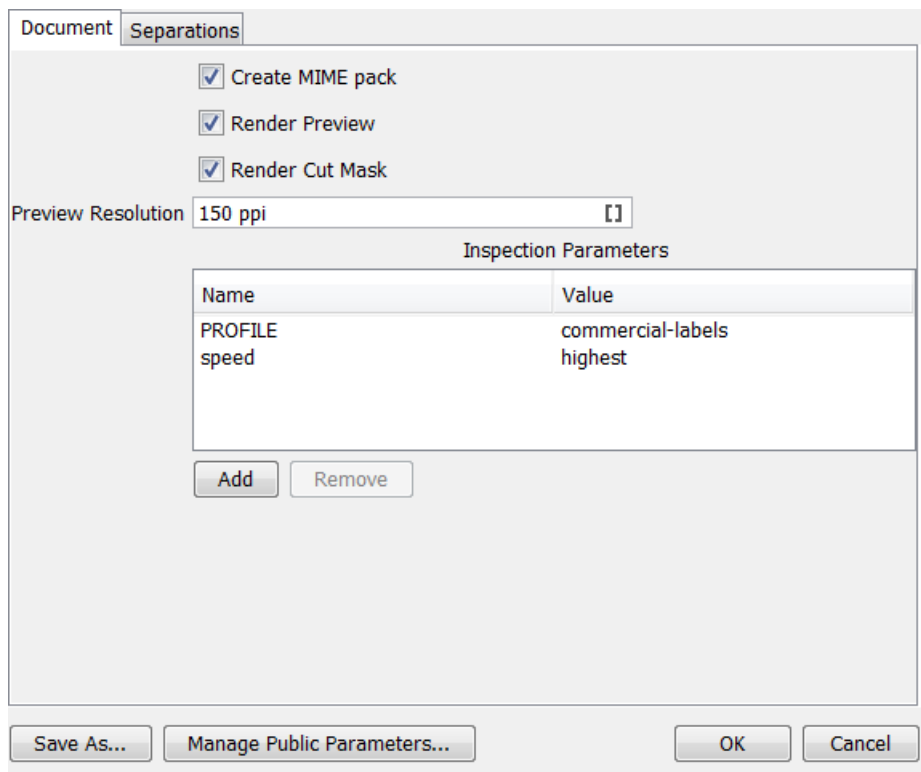


Settings

Document Tab

Specify the output requirements here. You can output at each step of the processing separately as a Preview, Cut Mask and or the entire results as MIME pack. You can enable/disable **Create MIME pack**, **Render Preview** and the **Render cut mask** options.

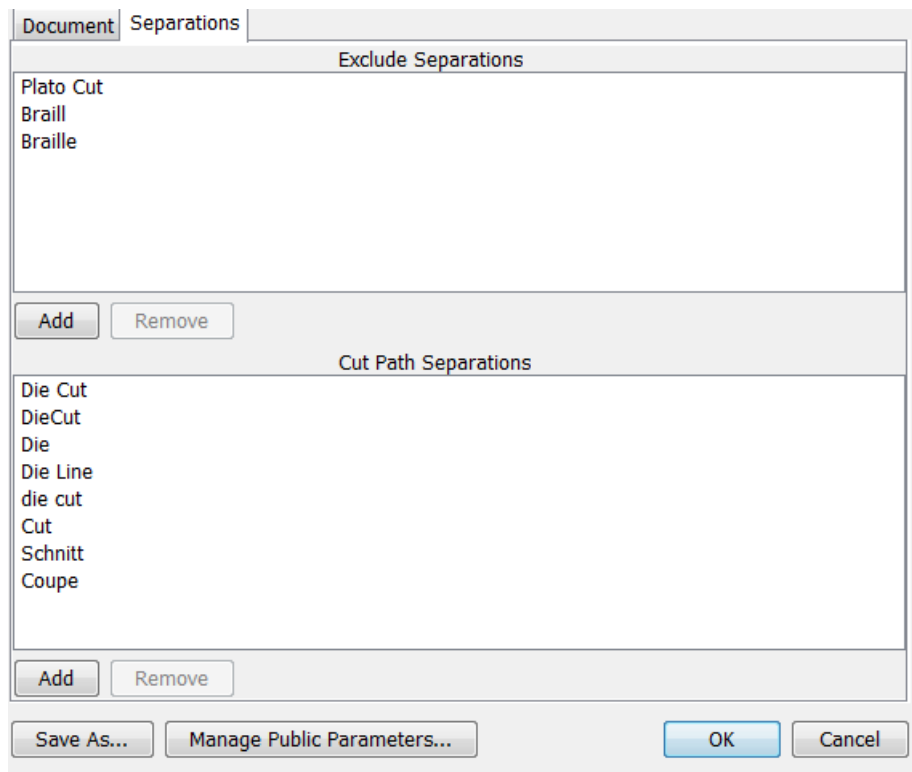
Configure the rendering resolution via setting the **Preview Resolution** manually or via SmartNames.



Add extra SmartName/ free text parameters such as a "profile" dictating the settings for the inspection system (for example, "commercial labels"). Specify these free-text parameters for inspection system under **Inspection Parameters**. Use **Add** or **Remove** to add or delete these parameters.

Separations Tab

Use this tab to indicate the selection of colors to be extracted (the "Die Cut" separations) and to specify the non-printable (the technical ink separations) areas to be excluded.



Specify the Separations to exclude from the preview under **Exclude Separations**. Enter the separations to be rendered as cut mask under **Cut Path Separations**.

2.7.100 Prepare for Viewer

About View Data

The Prepare for Viewer task prepares View Data for the Automation Engine Viewer.

This View Data will help the Viewer to quickly open the file (the Viewer will use the prepared View Data instead of asking the server to make or stream it).

Note: When you start zooming in, there will be a point where the View Data is no longer sufficient and when the server will be asked to stream 'new' data anyway. This point depends on the size of your file and the zoom-in that you request.

How to Prepare View Data

You can prepare View Data manually, or in a (semi-)automatic way.

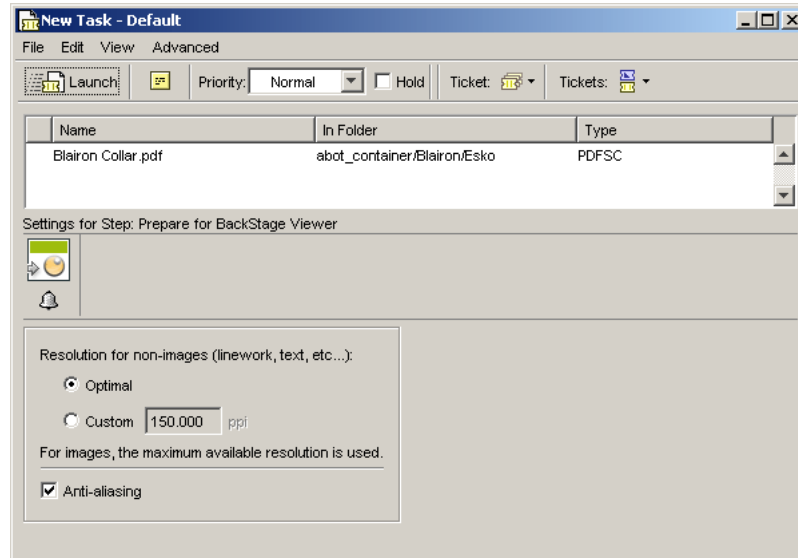
- Select a file and launch this task manually.
- Include this task in a workflow ticket (semi-automatic).
- Create View Data by activating '**Speed up for Viewing**' on a folder.

When should I prepare View Data?

Only create View Data when it is certain that the file reached its final state, preferably before the QA operator opens the file. What you could do is add a checkpoint to indicate when the View Data is ready to your QA operator.

Using the Task

The Prepare for Viewer ticket, which allows you to generate View Data semi-automatically, has the following options:



- **Resolution for non-images: Optimal** allows Automation Engine to use its own logic to generate the optimal speed/quality compromise, **Custom** allows you to indicate a desired resolution manually.
- Anti-aliasing
- **Automatic grouping of input files into digital film sets:** will group digital film separations into sets that can be combined for viewing a composite image.

For example, when the input files are 123_C.len, 123_M.len, 123_Y.len, 123_K.len, then these will be grouped into one digital film set and rendered in a composite view.

If you disable this option, all input digital films are grouped and displayed in a composite view, regardless of the file names.

Fully Automatic View Data Generation

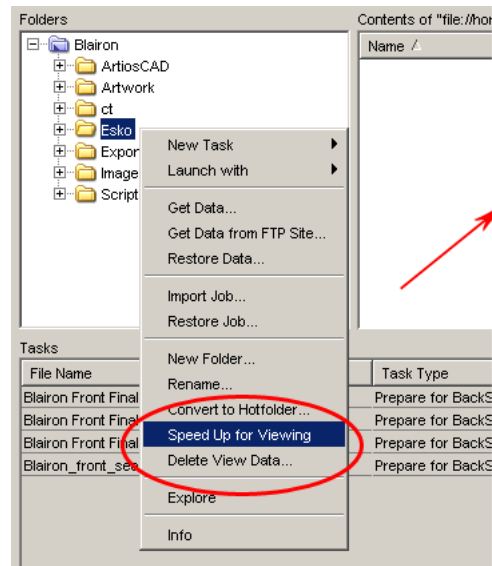
If you decide to use the Prepared View Data mode, you will want the data to be prepared as automatically as possible.

This can be done by setting an attribute to a specific folder of choice: **Speed Up for Viewing**. The folder will get a specific icon 📁.

When **Speed Up for Viewing** is activated, Automation Engine will automatically launch the Prepare for Viewer task on all Esko PDF and Esko PLA files in the folder which are new or have been changed (indicated by a changed SmartID). You can [Configure](#) to indicate other file types.

The **Speed Up for Viewing** attribute of the folder can be inherited from other Jobs and it can simply be removed again. It will be removed automatically when the Job is removed from the Automation Engine database, manually, via the Archive task or via JDF.

Note: Extra smart integration with Esko editors is guaranteed. When a file is being edited in PackEdge or Plato, you definitely do not want to have View Data re-created every time you click **Save**. Of course, the 'Prepare View Data' task only starts and updates your View Data when you indicate that you have finished the editing session.



2.7.101 Prepare Graphics for Nesting

For detailed information about the **Prepare Graphics for Nesting** task, please see [Prepare Graphics for Nesting](#) on page 1100 in the Gang Run Printing chapter.

2.7.102 Prepare Station

This task is used to prepare one ups for use in Step and Repeat. The task's input is a Normalized PDF file.

If the orientation of the PDF does not match the orientation used in the one-up CAD, the automatic repetition will fail. This is why a one-up ARD/CFF2 file is loaded on top of the PDF.

The position of the one-up CAD will use the die shape (defined by an ink) as a reference. The die shape and the one-up CAD will be compared in order to derive the rotation angle. If the rotation angle is not identical, the one-up CAD will be placed rotated on top of the PDF file.

The Prepare Station task can create either Station files or Normalized PDF files. The kind of output that will be generated depends on whether or not the **Create Station File** toggle is activated.

Check the 'Create Station File' box to create a **Station File** (STA or PDFSTA file).

- The input file is always placed externally.
- Check the option '**Extract Die using Ink**' to create a die by extracting all contours in a specific separation. If the box is not checked then the trim box of the station will be used as die shape. This is the default situation.
- '**Load CAD**' is only active when the option 'Create Station File' is selected. It allows you to place a CAD file. Supported CAD files include ARD files and 1-up CFF2 files.

Note: In case a layout CFF2 file is selected, the task will generate an error.

You can browse to the appropriate CAD file or use SmartNames. If needed, you can select a CAD mapping file which will determine how the CFF2 line attributes are mapped to the ArtiosCAD line types. This will happen only when a CFF2 file was selected. Select the appropriate CAD style file to determine in which separations the different CAD lines will appear.

- The **bleed** can be limited to a specific distance. The bleed mask is created by spreading the die shape.
- **Graphics** on the **backside** of the station can be loaded (also through an external reference).
- To load a **SmartMark set**, select a set from the dropdown list or enter the set via SmartNames. Note that only one set will be loaded for both front and back.

Note:

Keep the following in mind when positioning graphics on the back side:

- If the image PDF contains objects in the 'die ink', the graphics for the back side are positioned so that these object (in the die ink) are centered on the die of the front side.
- In the other case, the graphics on the back are positioned centered on the original borders (i.e. unfitted) of the front side graphics.

When placing a CAD file, the following sequence of operations takes place:

- The graphics file is placed in a station.
- If the option 'Extract die ink' is selected, the die shape is extracted from the graphics. If not, die shape is set to trim box.
- The CAD file is loaded and the outline of the CAD file is compared to the outline of the die shape generated in the above mentioned step. The graphics is rotated in such a way to create a maximal

intersection of the CAD outline and the outline of the die shape. If a rotation is applied, the log file will contain a message.

- The CAD is taken as the die shape for the station.

If the 'Create Station File' box is not checked, a **Normalized PDF** file will be created:

- The input file is opened (so it is not placed externally).
- Borders can be fitted to objects in a specific ink.
- Margins can be limited to a specific value.
- SmartMarks can be placed around the graphics.

2.7.103 Print

Select a device from the drop down list. The devices that appear in the list are the ones that are configured on the server.

Note:

The devices should be configured on the server using the user BGSYSTEM. The Print Task will not be available for devices configured by other users.

The configured devices must be PostScript printers.

Factory settings: These settings are a quick way to define your ticket.

There are two settings, select one from the drop down list and don't forget to click Apply:

- PostScript for press (Separated)
- PostScript for Press (DeviceN Composite), only available with PostScript 3 devices.

- PostScript for viewing/proofing (CMYK Composite)

If you change the predefined settings, the user interface will indicate that the Factory Settings are modified.



Note:

If you define a ticket using the Factory Settings, and you save it, the Factory Settings option disappears. If you overwrite a Default ticket the Factory Settings will be gone as well.

Please refer to [Export to PostScript File](#) on page 341.

2.7.104 Print IMP File

This task is a combination of the [Print task](#) and the [Export Imposition to GRI File task](#).

2.7.105 Print on Digital Press

Working with a Digital Front End Server

Automation Engine can create print jobs on an Esko Digital Front End Server (DFE) using the files in your Automation Engine containers. It will monitor those print jobs until they are printed on the digital press.

In Automation Engine:

- use the **Print on Digital Press** task to create print jobs on the DFE server. If Automation Engine is connected to a DFE server lower than version 4.0, the **Print on Digital Press** Task can only create static print jobs. When connected to a DFE server of version 4.0 or higher, this task can create both static and variable print jobs. Learn more below in [Print on Digital Press Task Settings](#) on page 421.
- use the **Print on Digital Press (PPML)** task to create variable print jobs on the DFE server from Automation Engine from Esko PPML input file (see [Print on Digital Press \(PPML\)](#) on page 431),
- the **Print on Digital Press (Track)** task is started automatically to track DFE print jobs created from within the DFE server. This means you can monitor print jobs created from sources other than Automation Engine. Because this is an automatic task, you don't need to fill in any task settings.

Each of those tasks shows detailed feedback (for each step of the DFE printing process) in Automation Engine's **Tasks** pane.

Setup

Before using those tasks, you must connect your Automation Engine server to your DFE server in **Configure**.

If your DFE server drives several digital presses, you also need to select the press to connect to from Automation Engine.

See [Digital Press](#) on page 635 for details.

Supported File Types

You can send the following file types to a digital press:

For static print jobs:	For variable print jobs:
Either: <ul style="list-style-type: none"> • Normalized PDF • PDF • Plato PDF 	An Esko PPML file, with links to several Normalized PDF files (for the fixed background and the variable data). An Esko PPML is a PPML file generated by DeskPack, PackEdge or Plato.
	Note: Make sure that the files referenced by the PPML file (the fixed PDF and the variable PDFs and images) are in an Automation Engine container, so the Digital Front End server can access them.
	PDF/VT (for example generated by Adobe Illustrator, while using the Dynamic VDP Plug-in).

Note:

- If you are using an MIS - ERP system to store print job parameters, you can add the JDF ticket created by your MIS system to the input of the **Print on Digital Press** task.
- If you want to send other file types to the digital press, you should convert them to Normalized PDF first.

Using the Print on Digital Press Ticket in a Workflow

Unlike most workflow steps, the **Print on Digital Press** or **Print on Digital Press (PPML)** step does not create output files to be transferred to another Automation Engine workflow step, but sends data to the Esko DFE server.

This means it is always the final task in a workflow.

In the workflow editor, you can see the number of files it has successfully processed on the pin called **Inputs** at the end of the step.



Print on Digital Press Task Settings

The **Print on Digital Press** task allows you to create a VDP Print Job from a One-Up Variable Data Template (.VDT) file. The Variable Data Database parameters (expansions of the variable data) will happen on the DFE. This is typically used when running the Expansion in Adobe Illustrator or Automation Engine takes too much processing resources.

Note:

- A VDT file (.vdt) is a variable data template file that contains **VDP objects** (placeholders for the variable data) and must be expanded to a PDF/VT file on the L&P Print Server.
- A VDP file can be created in the Dynamic VDP DeskPack plug-in.
- A VDT file must be self-contained: For example, it must NOT contain external references to other resources than Variable Images.

Settings

Settings for Step: Print on Digital Press

Device: HP-Indigo 20000 Digital Press

Print Job Ticket:

General | Files | Step and Repeat | Transformation | Color Management

Print Job Name:

Substrate: (<None>)

Copies:

Lead-in Sheets: (<None>)

Lead-out Sheets: (<None>)

Die Cut Layout Input Ink:

Die Cut Layout Print Ink:

Comments:

Device: This list contains all the digital press devices that you connected to in [Configure](#). Choose the digital press you want to send your file to.

Print Job Ticket: Choose the print job ticket you have created on the DFE server to process and print static print jobs.

General settings: see [General Tab](#) on page 423.

Files settings: see [Files tab](#) on page 424.

Step and Repeat settings: see [Step and Repeat Tab](#) on page 425.

Transformation settings: see [Transformation Tab](#) on page 430.

Color Management settings: see [Color Management Tab](#) on page 430.

Note:

When using the **Print on Digital Press** task, make sure the DFE server is running.

If your task is stuck, check if the Digital Press connection is still OK in **Configure**.

You can send the following file types to the **Print on Digital Press** task:

General Tab**Note:**

For all parameters, you can choose **[From Print Job Ticket]** to use the parameter value defined in the DFE **Print Job Ticket** you selected.

The value defined in the **Print Job Ticket** is then shown beside the parameter.

Copies: (1)

If you added a JDF ticket to the task input, you can choose **[From JDF Ticket]** to use the parameter value defined in your JDF ticket.

Print Job Name: You can specify a name. You can type in a name or select a SmartName / Public Parameter / Workflow Parameter.

Substrate: Contains a list of substrates available at the press. The substrate specified in the ticket is selected by default.

Copies: Enter the number of copies to be printed by the press.

Finishing Marks: Choose if finishing marks have to be applied (you have more options if Step and Repeat is enabled).

Lead-in Sheets: Enter the number of lead-in sheets that needs to be printed at the start of the Print Run, or enter <None> in case you don't want to use Lead-in.

Lead-out Sheets: Enter the number of lead-out Sheets that needs to be printed at the end of the Print Run, or enter <None> in case you don't want to use Lead-out.

Die Cut Layout Input Ink: Specify the ink of the input file of the task that will be used as contents of the Lead-in/Lead-out Sheet. This is typically a Technical Separation (for example Cutter)

Die Cut Layout Output Ink: Specify the ink that will be used to print the Lead-in/Lead-out Sheets. This must be one of the inks of the selected **Ink Set** in the **Color Management** tab.

Comments: Choose **Custom** to enter print job related comments here. They will be included into the file sent to the press, and be visible on the press (if your press model supports it).

Files tab

Settings for Step: Print on Digital Press

Device: esko-gen8-1 HP-Indigo 20000 Digital Press

Print Job Ticket: Default

General Files Step and Repeat Transformation Color Management

Duplex Settings: Custom

Duplex print

Backing Up: Turn

One-Up Settings: Custom

One-Ups:	Database File	Image Path
Front ^		
Graphic Input File 1 - Milk.pdf		

Add Remove

Duplex Print Job

To create a duplex Print Job, select **Duplex print** in the **Files** Tab.

Backing Up:

- Select **Tumble** if you want the roll to be re-inserted into the Digital Press with rewinding.
- Select **Turn** if you want the roll to be re-inserted into the Digital Press without rewinding.

The **One-Up Settings** list allows you to **Custom** specify an input file for both the Front and the Back side of the Print Job.

Note:

For all the above parameters goes the following: if you added a JDF ticket to the task input, you can choose **[From JDF Ticket]** to use the parameter value defined in your JDF ticket.

VDP Print Job

Also in the Files Tab, you can specify the Variable Data Database parameters. To make a job variable, a source of variable information is needed. To do this, proceed as follows

1. Choose **Custom** in the **One-Up Settings**.
2. On the bottom of this tab, click **Add**.

This is set in the **Select Database File** dialog. You can either use a **CSV File** or a **Simple Database**.

A **CSV** file is used in case the job should contain complex information such as names, serial numbers or links to images. When using a **Simple Database**, you only use Simple Counters, and

don't need a CSV file. This is suited for jobs that only require a simple sequence of numbers. You can define how many records need to be generated in the **Number of records** field.

For more information, see the DFE User Documentation.

Note: Creating a VDP Print Job from a VDT file is supported on the **HP L&P Print Server 4.0** and later for **Dynamic Workflows**.

Step and Repeat Tab

1. To make the Step and Repeat options available, Select **Custom** in the **Step and Repeat Settings** and select **Step and Repeat** at the top of the dialog.

Note: When using the PowerLayout DeskPack plug-in to send jobs to the DFE, you have already defined your Step and Repeat grid, so you don't need to set Step and Repeat options here.

2. Choose to either define a tabular Step and Repeat, or to Step and Repeat using a template.
 - If you want to do a tabular Step and Repeat, select **None** in the **Template Field**. Then fill in the parameters in the **Grid Details** tab (for more info, see [Grid Details](#)).
 - If you want to Step and Repeat from a template, select one in the **Template** field.

Note:

- If you are working with multiple input files, you need to use a Step and Repeat template. You cannot define a tabular Step and Repeat.
- Step and Repeat using a template is only supported when using a **Dynamic job ticket**.
- The Step and Repeat template and marks can be created in **DeskPack** (Powerlayout Standalone & Dynamic Marks). For more information how to create these templates, see the Deskpack documentation.

If your template has multiple grids and you have selected multiple one-ups at print job creation, the application will automatically assign the first grid to the first one-up, etc. You can change this by:

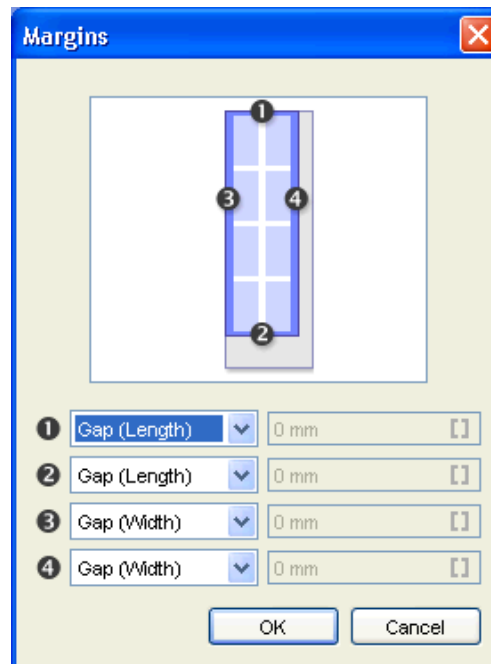
1. Selecting a grid by either clicking it in the **Sheet Preview** or selecting it using the arrows in the **Grid** parameter,
2. Selecting the one-up to go with this grid in the **One-up** parameter,
3. Repeating this for each grid and one-up you want to associate.

Note: If the number of one-ups doesn't match the number of grids in the template, or if the size of a one-up doesn't match the size defined in the template's grid, you will see a warning.

The **Grid Details** tab will automatically be filled with the data of the selected grid from the template.

3. In **Sheet Preview**, you will see a preview of the Step and Repeat after creation of the print job, based on the parameters you filled in at right.
4. If you are working with a variable job, fill in the parameters in the **Variable Data** tab (for more info, see [Variable Data](#)).
5. Click on **Marks and Margins** to set the marks and the margins for your Step and Repeat.

Note: You can set marks and margins even if you are not using Step and Repeat in your print job (if the **Step and Repeat** option at the top of the dialog is deselected).



- a. Choose if you want to add **Finishing Marks** to your print job. Finishing marks are used to align the substrate for cutting.

You can choose marks you made in Dynamic Marks or the default marks (**1 x LabelRow, 1 x Sheet** or **2 x LabelRow**). These marks are all visible when Step & Repeat is active, or from within PowerLayout. When Step & Repeat is inactive, only "1 x Sheet" is shown.

Note: If you are using a template, you cannot add finishing marks here. The marks from the template will be applied.

- b. If you are working on a duplex print job, you can also add **Second Pass Marks** (also called "reinsertion marks"). These marks are used to align the substrate when reinserting it back into the machine for printing its back.

The default marks available are: **Reinsertion Marks (Bottom Right) 10x5**, **Reinsertion Marks (Bottom Right)**, **Reinsertion Marks (Upper Left) 10x5** and **Reinsertion Marks (Upper Left)**.

Note: If you are using second pass marks in combination with finishing marks, make sure that your margins are big enough to accommodate both sets of marks.

- c. Set **Margins**: you can define each gap (**1, 2, 3** and **4**) using the following:
- **Gap (Length or Width)**: the size of a gap between one-ups.
 - **Gap (Length or Width) / 2**: half the size of a gap between one-ups. This is typically used for top and bottom margins, as the bottom margin of one sheet plus the top margin of the next sheet will be equal to one gap.
 - **Custom**: a custom number (formulas are not allowed).

Note:

If you selected a mark whose size you defined in the Configuration tool, the corresponding margin will show *Defined by Mark*. This means that the margin is automatically large enough to provide space for this finishing mark.

For example, if you selected the default mark **1 x LabelRow**, which is set in the Configuration tool to need 8mm of right margin, the right margin (**4**) will be set to *Defined by Mark* and be 8mm wide.

6. At the bottom of the dialog, you will see the **Grid Size** if you defined a tabular Step and Repeat, or the **Template Size** if you used a template. This includes the margins you defined in the **Marks and Margins** dialog.
7. After creation of the print job, the **Actual Repeat Area** shows the area actually used by the repetition (not including the margins). The width and length values are calculated based on the Step and Repeat parameters, including gaps. You cannot edit them.

Grid Details

You can define parameters here if you are building a tabular Step and Repeat from a single one-up (you have selected one input file at print job creation).

If you are using a template, this tab will automatically be filled with the data from the template (which is then non-editable).



Repeat one-up using its: Choose whether to use the dimensions of the one-up's **Trim Box** or **Media Box** to calculate the repetition grid.

You can see the **One-up Size** at right.



Clip on Trim box with bleed of: If the Trim Box / Media Box of the one-up is too large for the desired repetition, you can clip it to the desired value to avoid overlapping between one-ups.

Note: This value should not be larger than half of the gap defined in the corresponding direction.



Orientation: Choose how to orientate the one-up.

Available Repeat Area: Define the width  and length  of the repeat.

- The width can be set to **Substrate Width** (the width of the substrate you chose in the **Printing Properties**) or to a **Custom** value.
- The length can be set to the **Maximum Repeat Length** or to a **Custom** value.

Repetition: This defines the number of horizontal  and vertical  steps in the repetition grid. You can choose either:


- **Maximum:** The server will calculate the maximum amount of repetition steps possible within the **Available Repeat Area**.
- **Custom:** Enter the repetition amount of your choice.

Gaps: The values you enter here (for a horizontal  and a vertical  gap) will be used as a distance between the one-ups' borders in the repetition grid.

Note:





- The amount of gaps is equal to the number of repetition steps, in order to provide seamless printing.
- The length of the sheet is equal to the length of the vertical repetition, plus the length of the vertical gaps, plus the top and bottom margins.

Variable Data

This tab is only available when you are working on a variable print job (for this you need to have selected either the **Default** ticket  or a custom ticket created from the **Default** ticket when creating the print job).

Use this tab to define how to distribute your variable data over the grid of one-ups you built in the **Grid details** tab, or over the one-ups repeated according to the **Template** you chose.

Start: Choose where to start printing your variable data (where to print the first record):

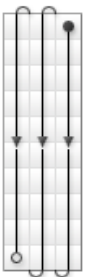
-  top left,
-  top right,
-  bottom left,
-  bottom right.

On the preview at right, you can see a black dot where the first record is.

For example this distribution starts on the top left:



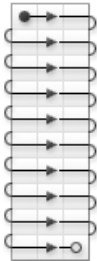
while the distribution below starts on the top right:



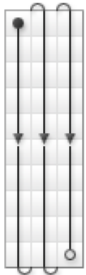
Choose also if the flow of variable data should run horizontally or vertically .

The preview at right reflects your choice.

Horizontal flow:



Vertical flow:

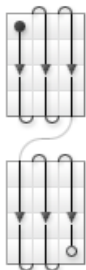


Note: The **Printing direction** marked with an arrow shows the direction the paper flows through the press.

Fill over: choose if you want to distribute the variable data over the **Complete Job** or over **Blocks**.

If you choose **Blocks**, the variable data flow will run over **Blocks** of a certain size, then run over the next **Block** when the previous one is complete.

You can see an example of **Blocks** in the preview at right when selecting **Blocks**.



Block Size: select the size of the **Blocks** you want to use. You can express the size in **Grid Rows**, **Sheets**, **Feet** or **Meters**.

If you have chosen to **Fill over** the **Complete Job**, then the **Block Size** will be **1.00 Job** by default.

You can set it to another unit if you still want to print in **Blocks**, but without having the variable data flow respect the **Block**. This is useful if you want to add gaps (see below).



Add gap between blocks: this is available if you have chosen to **Fill over Blocks**, or to fill over the **Complete Job** but still have **Blocks** (as explained above).

It adds white space in between the **Blocks**.

Gap size: this is available when you have selected **Add gap between blocks**. Define the gap size in **Grid Rows, Sheets, Feet** or **Meters**.

Transformation Tab

Note:

For all parameters, you can choose **[From Print Job Ticket]** to use the parameter value defined in the DFE **Print Job Ticket** you selected.

The value defined in the **Print Job Ticket** is then shown beside the parameter.

If you added a JDF ticket to the task input, you can choose **[From JDF Ticket]** to use the parameter value defined in your JDF ticket.

Vertical Scale: Use this to scale the print job vertically (in the length).

Horizontal Scale: Use this to scale the print job horizontally (in the width).

Rotation: Choose the rotation of the print job (**None, 90° CW, 180°** or **90° CCW**).

Note: The **Rotation** option is disabled when **Step and Repeat** is enabled.

Mirror: Choose whether to mirror the contents of the print job or not.

Color Management Tab

Note:

For all parameters, you can choose **[From Print Job Ticket]** to use the parameter value defined in the DFE **Print Job Ticket** you selected.

The value defined in the **Print Job Ticket** is then shown beside the parameter.

If you added a JDF ticket to the task input, you can choose **[From JDF Ticket]** to use the parameter value defined in your JDF ticket.

Ink Set: Choose the set of process inks that will be generated by the RIP (this depends on the model of the press used).

Use: Define if you want to use color management when ripping the print job (to convert the file separations to the press ink set), and what kind of color management.

- Choose **<None>** if you don't want to use any color management settings for the conversion.
- Choose **Color Strategy** to select a Color Strategy to use for the conversion.

When you select **Color Strategy** in the **Use** list, you can choose the **Color Strategy** to use for color conversion of the print job.

2.7.106 Print on Digital Press (PPML)

Use the **Print on Digital Press (PPML)** task to create variable print jobs on the DFE server from Automation Engine.



Caution: From version 4.1.1 onwards, the DFE will no longer support PPML as an input format. Therefore this task can only be used with DFEs prior to version 4.1.1.

Settings

Device: This list contains all the digital press devices that you connected to in [Configure](#). Choose the digital press you want to send your file to.

Variable Print Job Ticket: Choose the print job ticket you have created on the DFE server to process and print variable print jobs.

General settings: see [General Tab](#) on page 423.

Transformation settings: see [Transformation Tab](#) on page 430.

Color Management settings: see [Color Management Tab](#) on page 430.

Note:

When using the **Print on Digital Press (PPML)** task, make sure the DFE server is running.

If your task is stuck, check if the Digital Press connection is still OK in **Configure**.

2.7.107 Process PDF Spreads

PDF Spread options: process modes

There are three process modes:

- **Split Reader Spread:** to create single pages from an incoming spread
- **Make Reader Spread:** to create spread pages from incoming single pages.
- **Make Printer Spread:** to create spread pages that can be used to create an imposition mock-up from incoming single pages

The settings for each process mode are described in the following sections.

Settings for Split Reader Spread

Spread position:

- **Reference Box:** Select a page box to choose how much of the page to include in, or retrieve from a spread. For example select Crop box to crop everything outside that box. Reference boxes include the Media Box, Crop Box, Bleed Box, Trim Box, and Art Box.
- **Reference Box Offset:** Enter an exact offset distance from the reference box specified in Reference Box field. The offset distance will extend beyond the box boundaries. This parameter is commonly used when creating spreads of asymmetric files.
- **Bleed Mode:** Control the placement and size of bleeds. There are three options:

Remove Bleed	Only the content of the reference box will be retained, any bleeds present in the incoming job will be removed.
Bleed from Bleed Box	The bleed will be the Bleed Box but only if the reference box is the Trim Box.
Bleed from Bleed Distance	This option will add the specified bleed distance to the chosen reference box. Enter the bleed distance in millimeters. A bleed distance will extend beyond the borders of the Trim Box and therefore will set the boundaries of the Media Box when output.

Spread structure:

- **Flow Direction:** Determine whether pages are arranged in numerical order from either left to right or right to left. When set at left to right direction, page one will be on the left hand side of a two-page spread, with the second page on the right. If the flow direction is right to left, page one will be on the right hand side of the spread, with page two on the left hand side.
- **First Spread Contains** Control how pages will be organized in the output job:

Page 1 and 2	To divide a spread into single pages. If you use this mode in conjunction with the “Mixed Spreads” option, you can split spread pages in a job while leaving single pages intact. Note: Note: The reference page for a splitting job using Mixed Spreads should be a spread page, not one of the single pages.
First page only	If you place the first page of the job on the right-hand side of the

	<p>first spread page; the left-hand side remains blank.</p> <p>For jobs with an even number of pages, the last page will be also be placed on a spread page, on the left-hand side, with an accompanying blank page.</p> <p>The left / right positioning of pages on spreads is determined by the selection made in the Flow Direction field. The description above assumes a left to right flow direction.</p>
Last and first page	<p>When making a spread in Last and First Page mode, the last and first pages of the input job are placed side by side on the first spread page.</p> <p>In a left to right flow direction setting, the last page will be on the left and page one on the right of the first spread page output.</p> <p>In “Split Reader Spread” mode the task will simply reverse the make mode. A job with spread pages will have the spread pages split but the page on the left side of the first spread will be output as the last single page of the output job.</p>
Even number of pages	<p>To prevent incoming jobs with an even number of pages from losing any pages while being split.</p> <p>The parameter produces different results depending on which of these spread types it is combined with.</p> <ul style="list-style-type: none"> • Select the Even Number of Pages option together with the First Page Only spread type to split the output job. The left side of the first spread and the right side of the last spread will not be output. • A combination of the Last and First Page spread type and the Even number of Pages parameter will split a job successfully.

Note: This option is active only in the Split Reader Spread process mode and with the First Page Only and the Last and First Page spread types.

Mixed spreads:

Use this option for incoming jobs with a variety of paper sizes for which you need to define the Reference page.

For example, if a job contains both an A4 page as page one and a two-page spread as page two, enter page two as the reference page and indicate that this page has two pages on it. All the spread pages are correctly detected and split correctly while leaving the single pages intact.

- **Auto Detect Spread Reference:** to take the most frequently featured page size as the Reference page
- **Reference Page Number:** to specify an exact page as the Reference Page. Use the actual page number in the series of pages, not the page's number label
- **Reference Page contains X One-Ups:** enter the number of single pages to be positioned on the Reference Page.

Settings for Make Reader Spread

Select this process mode to create spread pages from incoming single pages.

The Spread position and Spread Structure options are the same as for the process mode Split Reader Spread.

Settings for Make Printer Spread

Select this process mode to create spread pages that can be used to make an imposition mock-up from incoming single pages.

The Spread position and Spread Structure options are largely the same as for the process mode Split Reader Spread.

Processing PDF Spreads

Use this task to manage PDF Spreads. You can create spread files, split spread files, specify bleed distances and control the order and positioning of the pages in spreads. This task can handle jobs that contain a combination of single pages and spread pages.

1. Use the **Output in** field to define where to place the PDF spreads.
This can be the job's original location, or another location.
2. Define the **Job Name** to use.
By default, this field contains the **[JobName]** SmartName, meaning that the original job name will be used for the PDF spreads.
3. Choose one of the following **Actions**:
 - Split Reader Spread
 - Make Reader Spread
 - Make Printer Spread

4. Choosing **Split Reader Spread** allows you to create single pages from an incoming spread. Set the following parameters:
 - the Reference Box: Media, Crop, Trim, Bleed or Art box
 - the Reference Box Offset
 - The Bleed Mode and Bleed distance, when relevant
 - The Spread Structure: the flow direction and the contents of the First Spread
 - If you have mixed page sizes in the PDF spread, choose your Reference page and specify the number of one-ups on that Reference page.
5. Choosing **Make Reader Spread** allows you to create spread pages from incoming single pages. Set the following parameters:
 - the Reference Box: Media, Crop, Trim, Bleed or Art box
 - the Reference Box Offset
 - The Bleed Mode and Bleed distance, when relevant
 - The Spread Structure: the flow direction and the contents of the First Spread
6. Choosing **Make Printer Spread** allows you to create spread pages that can be used to create an imposition mock-up from incoming single pages. Set the following parameters:
 - the Reference Box: Media, Crop, Trim, Bleed or Art box
 - the Reference Box Offset
 - The Bleed Mode and Bleed distance, when relevant
 - The Spread Flow Direction
7. Click **OK** to save your settings for this task.

2.7.108 Proof (FlexProof)

In the **Device** list, select the **FlexRip Proofer Output** you added in **Configure**.


General


Transformations

Enter a percentage in **Scale V** and/or **Scale H** to scale the design vertically or horizontally. The scaling direction is always applied 'as the file is' (so the vertical and horizontal dimensions are as you see them in the editor).

Rotate: Choose the requested rotation from the list. The rotation is counter clockwise.

You can also select **Fit On Paper** to fit the design on the paper of your proofing device. This makes the **Scale** and **Rotate** options unavailable.

Positive-negative : Define whether you want a positive or negative output file.

Mirroring : choose whether you want mirroring or not.

Original Size (incl. Margins)

The system provides the original Job size, margins included.

Output Size (incl. Margins)

The size shown here is the real output size, that is the original job size influenced by the scaling factors (and/or distortions) and the chosen rotation.

Marks

If desired, you can add **SmartMarks** to your file. SmartMarks are created in an Esko editor.

Dot Gain Compensation

Single Curve: Select the preferred Dot Gain Compensation Curve.

Automatic: Select the preferred screen based Dot Gain Curve.

Inks

All Inks. The standard behavior of the default ticket is to offer all inks in the job to be exposed. Deactivating 'All Inks' opens the ink list where you can select specific inks. The list contains ink names, ruling, angle, dot shape and ink type.

Use technical inks in output. Normally technical inks are hidden when you save a job in an Esko editor. Check the box to output the visible (non hidden) technical inks.

Keep similar inks in their original separations. Check the box if you want to keep similar inks in their original separations.

Keep inks separated. Do not merge the inks.

Use **Merge with** to output inks on top of each other in the same output file. This is common when you use different rulings for 1 ink.

Let's assume that the job contains two Cyans, and that they have a different ruling. You might want to put them on the same separation. You can do so by clicking in the 'Merge with' column and selecting the ink it has to be merged with.

Click **Merge Similar Inks** to prompt the system to look for the same inks with a different ruling or angle. The image shows how two Cyans (with a different ruling) were detected and put together. You can reset any modifications by clicking Reset.

Inks:

	Name	Ink Book	Ruling	Angle	Dot Shape	Merge with	Ink Type
<input checked="" type="checkbox"/>	M	process	150.000 ...	75.000 °			Normal
<input checked="" type="checkbox"/>	Y	process	150.000 ...	90.000 °			Normal
<input checked="" type="checkbox"/>	K	process	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	215	PANTONE Colors...	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	C	process	120.000 ...	15.000 °			Normal
<input checked="" type="checkbox"/>	C	process	150.000 ...	15.000 °		C	Normal
<input type="checkbox"/>	Die	designer	120.000 ...	45.000 °			Technical

Merge Similar Inks Reset

Advanced

Hold In Rip Queue This options offers to hold the task in the rip-queue until someone manually releases this task. For more information, please see the [Tasks View](#).

Choose between:

- **Don't change Y ruling:** this is the default, the Y ruling is not adapted.
- **Change Y ruling:** change the Y ruling, possibilities are: -15%, -7%, 0% (changes nothing), + 7% and +15%.
- **Change Y ruling stochastic:** same as the 'Change Y ruling' settings, but the dotshapes is adapted to the Monet dot.

Pre - Rip

The Pre-Rip tab contains options that are only relevant when the job really needs pre-ripping. The Pre-ripping mentioned here is the 'simplification' that can be necessary before the job-objects can be offered to the Rip. The decision to do this is taken automatically. And even when this is done, the options are best left to their default settings. Special objects (in Normalized PDF format) that require the need for this 'simplification for rip' pre-ripping are:

- 'Edge vignettes' (vignettes with alpha channels effect on the edge of the object)
- Barco Blends (=non contourized GRO-blends)
- Warped CTs (warping done on Placed CTs)

Pre-ripping results in making a CT of these special objects that the rip can handle them. The resolution of such a CT and the anti-aliasing can be set here.

Specify the output **resolution** for the generated CT(s). Default output resolution is 12 lpmm or 304 ppi.

Anti-aliasing is relevant for the lineart objects in your design, that need pre-processing and for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of lineart objects that need pre-processing.

Vignette Noise When exporting to some EPS vignettes or parts of vignettes will be converted to CTs. This happens when the vignettes overlap other objects with translucency or overprint between the vignette and the other objects. When the option Convert Vignettes to CT is activated all vignettes are converted to CT regardless of overlap with other objects. Vignettes that are converted to CT can result in tone jumps in the final result. To reduce this banding effect, noise is applied on the CT. The amount of noise can be specified in the field Vignette Noise.

The **Blend to CT** toggle converts the Barco Graphics blend objects into a CT when exporting your job.

Use the **Smooth Blend** toggle to set the steps of the blend objects to 255 steps when exporting your job. As long as Convert Blend Objects to CT is not activated, Smooth Blend appears dimmed. When Convert Blend Objects to CT is activated and Smooth Blend also, the generated CT will ignore the original steps of the blends of your Barco Graphics design. The amount of steps is set at the maximum which is 255. The goal is to obtain a fluid blend effect.

Output

Configuration Set

Indicates the name of the selected configuration file. These configuration files represent specific behavior of the output device. The list of configurations is filtered according to the CMS license that the selected dispatcher holds. For example if it has a 4 channel license, no multi channel configurations will be visible.

Output Folder

Leave this empty to use the output path as defined in the Dispatcher (EskoLink queue – Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Output File

Leave this blank to get an output file name based on the input file. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Output Type

Choose between **Composite** and **Separate** and select the appropriate **Output Color**.

Output Color

In case of **Composite** output, you can choose between **Colored** and **Greyscale**. In case of **Separate** output, you can choose between **Colored** and **Black & White**.

- **Colored:** all inks are converted to CMYK values
- **Greyscale:** all inks are converted to grey values.
- **Black and White:** if you want a Black & White proof.

Anti-Aliasing is relevant for the lineart objects in your design, that need pre-processing and for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of lineart objects that need pre-processing.

Enter the **Number of Copies** you need.

Check **Keep job files for relaunch purposes** to be able to proof jobs a second time without having to rip them again.

Note:

When using this option, the files created for the proof (and the proof job on the Esko Proof Client) will not be automatically cleaned up.

You will have to delete them manually.

White Coloring

Attention: You will only have this option if one of your **Configuration Sets** connects to an Epson Stylus Pro WT7900 proofer.

When you use an extra white ink to print on foil (transparent or metallic), this option allows you to customize your white ink (for example, to get a cooler or warmer white).

You can choose between:

- **No Coloring:** this uses the proofer's white ink to print objects in the white separation, without applying coloring to this ink.
- **Custom values:** this allows you to color the proofer's white ink manually. Enter the CMYK values to add to the white inks.
- **Values from Color Management:** this uses a Color Strategy to color the proofer's white ink.

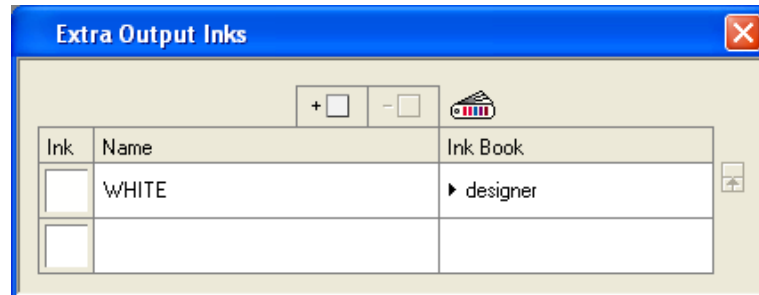
This Color Strategy must have a white **Extra Output Ink** (called `white`) and map the white separation used in the file to this white output ink.

Color Managing White Ink for an Epson WT7900 Proof

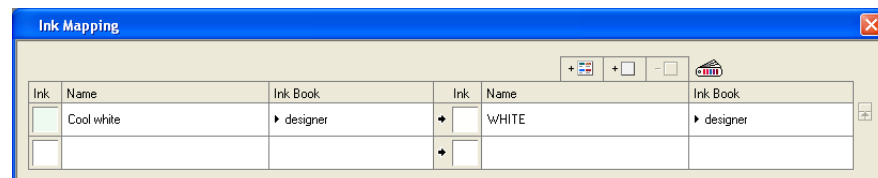
For example, you want to print a file with a `cool white` ink on your Epson Stylus Pro WT7900 proofer.

1. In Color Engine Pilot's **Designer** book:
 - a) add your file's white ink (here, `cool white`),

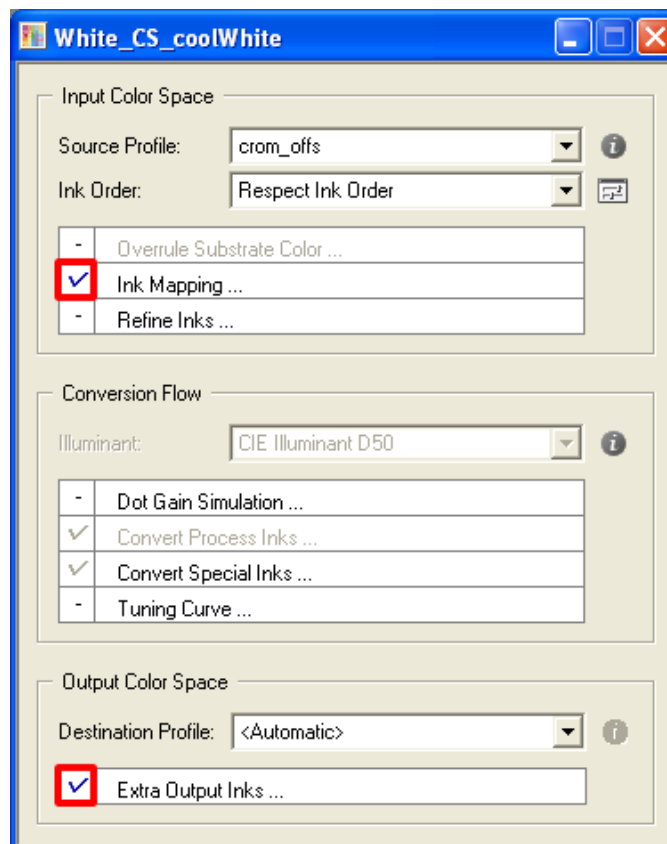
- b) add a generic white ink. Call it `white`.
2. Create a Color Strategy that will color manage your file's white ink:
- a) add your generic `white` ink as an **Extra Output Ink**,



- b) map your file's white ink (`cool white`) to your `white` output ink,



- c) make sure **Extra Output Ink** and **Ink Mapping** are selected in your Color Strategy.



3. In Automation Engine's **Proof (FlexProof)** ticket, go to the **Output** tab and:
 - a) select the **Configuration Set** corresponding to your Epson Stylus Pro WT7900 proofer,
 - b) in **White Coloring**, select **Values from Color Management**.
4. In the ticket's **Color** tab:
 - a) select **Manage Color**,
 - b) select **Color Strategy**,
 - c) choose the Color Strategy with white ink mapping you created in Color Engine Pilot.

Color

Simple Color Workflow

The screenshot shows the 'Color' tab in the Automation Engine interface. At the top, there are tabs for 'General', 'Inks', 'Advanced', 'Pre-Rip', 'Output', 'Color', and 'Verification'. The 'Color' tab is active. Below the tabs, there is a 'Manage Color' checkbox which is checked, and a dropdown menu set to 'for Normal Proof'. A diagram shows a 'Press' (Source Profile) connected to a 'Proofer' (Destination Profile) via a dashed line with a circled '1' in the middle. Below the diagram, there are several configuration options:

- Color Strategy:** A dropdown menu with 'preview' selected.
- Source Profile:** A dropdown menu with '<None>' selected.
- Use the Document Profile as Source Profile if Available**
- Destination Profile:** A dropdown menu with '<None>' selected.
- Rendering Intent:** A dropdown menu with 'Relative Colorimetric' selected.
- Black Mapping:** A dropdown menu with 'No' selected.
- Generate an error if the job contains unregistered inks** **Unregistered inks will not be color accurate**

Check **Manage Color** to apply color management to your job.

If you choose to apply color management **for Normal Proof** the settings of the requested color management are, by default, chosen by a Color Strategy which can be selected from the list.

If the color strategy toggle is disabled, the color management settings are entered with profile settings.

Then, **Use the Document Profile as Source Profile if Available** is set but as this document profile may not be available, a backup of the **source profile** must still be chosen.

Choose the Process you want to simulate on the proof here. This process can be a registered ICC profile or an Esko Profile.

There are some default Esko Profiles on the system, for example crom_offs (Cromalin Offset). If you have measured an Esko Profile yourself, for example for your own Flexo press, you will be able to select it here.

Destination Profile

This is the profile for the proofer. Choose the proofer profile that describes the colors of your proofer here (for example the characteristics of your proofer).

Both Esko Proofer profiles and ICC profiles of proofers are available in this list.

Also defaults are listed, for example `ddcw_glossy`, a description for a Digital Cromalin/WaterProof on glossy paper.

If you profiled your proofer using Color Engine Pilot, you will be able to select your Proofer profiles here as well.

Note:

The list of color strategies and destination profiles is filtered according to the CMS license that the dispatcher holds.

If the Dispatcher has no CMS license, only “preview” and “linear” can be selected as color strategies, whereas additional strategies are available with a 4-channel or multi-channel license.

Rendering intent

Rendering intent decides how you will visualize the simulated profile on your proofer. Since there are many differences between the simulated process (for example a gravure press) and the proofer, the CMS software gives some options to give you the best result for your purpose. Possibilities are:

- **Relative (No Background):** Use this option if you do not want any background simulation. The white point of the source will be mapped on the white point of the destination process. This rendering intent takes into account the ability of the human eye to adapt to the surrounding white.
- **Perceptual:** This will result in no background simulation (white of reference colorspace is mapped on white of destination colorspace). Use this option if you don't care too much about an exact color match but you want the pictures at least to look nice on your proofer. It will give a visually pleasing result. If the gamut of the source is wider than the proofer, the entire gamut is shrunk so that it fits inside the gamut of the proofer.
- **Absolute (Background):** Use this option if you want background simulation and you want an exact color match, including the paper simulation.
- **Saturation:** Use this option if you care especially about saturated colors. This rendering intent will map saturated colors on saturated colors.

Note:

For Esko profiles Perceptual, Relative and Saturation are in fact identical.

Black Mapping

This option is only available if you choose an Esko destination profile (not if your destination profile is an ICC profile). Black will be mapped onto black or, in other words, the black separation will only be printed with black ink.

In case you want to simulate a very dark black, which is darker than the black of your proofer, you might want to switch this option off. In this case all 4 proofer inks will be used to simulate the high density of the black.

Generate an error if the job contains unregistered inks

This toggle is by default unchecked. You can enable this option when **Manage Color** is on.

When this toggle is checked, the task will indicate an error if unregistered job inks are being color managed. The resulting **Error** will indicate a message:

- The job contains unregistered inks when launched as a stand-alone task.
- Proofing aborted because the file contains unregistered inks in the **Log** window when launched as a workflow task.

This helps to avoid inaccurate results from unregistered inks. When unregistered inks are not present in the job or the unregistered inks are deselected in the inks-tab, the task will launch without indicating an error.

Note: When you select an Equinox source profile (or use a Color strategy with Equinox source profile) with **Manage color** toggle on, this task will perform a default check. This check verifies if all the selected job inks are specified in the Equinox source profile. If they are not specified, the task will end in **error** with a message such as :
Proofing aborted because the following input inks are not found in the source profile: cut glue

Color (Emulation Proof)

You need an extra license to see the Manage Color **for Emulation Proof** option. If unchecked, then the workflow remains as described in the simple color workflow. Otherwise, extra options are available:

Here, CMS options can be set twice:

- In the topmost section of the tab, you set the color management options for emulating the conventional press on a digital press (conversion from the **source profile** to the **emulation profile**).
- In the second section, you set the color management options for the output proofing device (conversion from **emulation profile** to **destination profile**).

The preferred option is to use a Color Strategy for both conversions. Select the emulation Color Strategy first, so that the list of color strategies for the output settings can be filtered to have the same source profile as the emulation profile selected in the emulation step.

Alternatively, you can manually select the source profile, emulation profile and destination profiles with their respective rendering intents.

Tip:

If you select a multichannel printing profile in the Emulation Profile list (such as the seven-channel IndiChrome Plus profile), you can use the **Output Inks...** button to enable or disable the additional inks (for example, indiChr Orange, indiChr Green, and indiChr Violet).

This allows you to use a profile like IndiChrome Plus to generate output without one or more of the additional inks (for example, without the IndiChrome Orange ink).

Generate an error if the job contains unregistered inks

This toggle is by default unchecked. You can enable this option when **Manage Color** is on.

When this toggle is checked, the task will indicate an error if unregistered job inks are being color managed. The resulting **Error** will indicate a message:

- The job contains unregistered inks when launched as a stand-alone task.
- Proofing aborted because the file contains unregistered inks in the **Log** window when launched as a workflow task.

This helps to avoid inaccurate results from unregistered inks. When unregistered inks are not present in the job or the unregistered inks are deselected in the inks-tab, the task will launch without indicating an error.

Note: When you select an Equinox source profile (or use a Color strategy with Equinox source profile) with **Manage color** toggle on, this task will perform a default check. This check verifies if all the selected job inks are specified in the Equinox source profile. If they are not specified, the task will end in **error** with a message such as :

Proofing aborted because the following input inks are not found in the source profile: cut glue

Verification

Note: You can only set **Verification** options if you have chosen an Esko Proof **Configuration Set** in the **Output** tab.

Print Verification Strip: To print a verification strip, select this option and fill in the options below (that become available once **Print Verification Strip** is selected).

Strip Preset: a certain number of presets come with your color database. They contain information about the strip and the measurement device to use, the tolerances in color difference, etc.

You can select a preset here, that will fill in the options below, or select **Custom** and fill the options in yourself. You can also start from a preset and change the other options as desired (the **Strip Preset** field will then show **Custom**).

For more information about the strip presets available, see XXX.

In **Strip Type**, choose the strip to print with your job: this can be **EskoStrip** or **Esko + Solid Color Strip**. The Esko + Solid Color Strips contain the same patches as the EskoStrips, plus a 100% patch for every spot color present in the job. When printing a color managed strip, you should use an Esko + Solid Color Strip.

For more details about the available strips, see the FlexProof/E documentation.

Apply Color Management to Strip: If you selected **Manage Color** in the **Color** tab, the **Apply Color Management to Strip** option is available. Select it if you want to color manage your strip.

Verify strip (with Verification tool): If you have a Verification license and you are color managing your job, this option is available. To verify your strip, select this option and choose what to use as a **Reference**.

You can choose either:

- **Manually load reference values** if you want to verify the color managed strip against a standard (for example Ugra/Fogra) or a previous measurement.
You will need to load this standard manually in Esko Verification.
- **Automatically load reference values from source profile** if you want to verify the color managed strip against the source ICC profile (set in the Color Strategy used or defined in the **Color** tab).

dE Formula: choose the Delta E formula to use when verifying your strip.

You can select between **dE CIE76**, **dE CMC**, **dE 94** and **dE 2000**. If your company is already using one of those standards, we recommend that you choose the same one. If not, you should choose the most recent one, dE 2000.

Click **Edit Tolerances...** to set custom tolerances for the measurements.

You can fill in:

- **General dE Limits:** the Delta E values that the color patches and paper should not exceed.
- **Primary color dE Limits:** the maximum Delta E values allowed for CMYK colors.
- **Hue difference Tolerance Limits:** the maximum Delta H values for CMYK, CMYKRGB and grayscale colors.

Delta H is the difference in Hue between the expected color and the printed color. XXX

- **Tone Value Tolerance Limits**
- **Chromaticness Tolerance Limits**

If you selected a **Strip Preset**, the tolerances may be automatically defined.

The verification will pass if the Delta E values are within the tolerances, or fail if they are bigger.

Strip Layout

2.7.109 Proof IMP File (FlexProof)

This task is a combination of the [Proof \(FlexProof\) task](#) and the [Export Imposition to GRI File task](#).

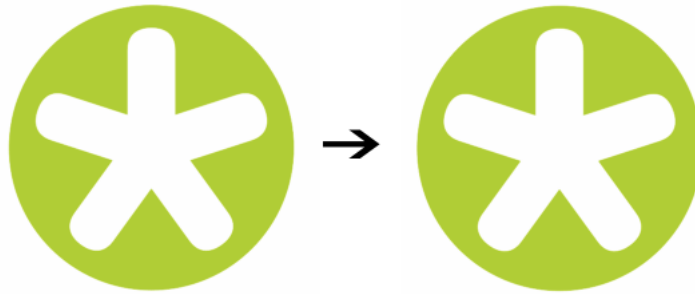
2.7.110 Proof to File - Unscreened

Use this task if you need unscreened but color managed output, in a file that you can send to a proofer.

Task Output

The **Proof to File - Unscreened** task can generate composite [PDF or TIFF files](#) (one file with all the separations).

In the example below, processing a file containing the lime green Esko star generates one multi-channel unscreened file containing the Cyan, Yellow, Magenta and Black channels, where the Magenta channel is empty.



Task Parameters

In the **Device** field, select the imaging device *that you configured*.

Then use the selector pane at left to show more settings to fill in. See:

- [General](#) on page 1204,
- [Proofer](#) on page 1206,
- [Transformations](#) on page 1207,
- [PDF Objects](#) on page 1208,
- [Document Inks](#) on page 1208,
- [Color Management](#) on page 1209,
- [Output](#) on page 1209,
- [Summary](#) on page 1218.

2.7.111 Publish Imposition on WebCenter Chain

You can publish your Imposition to WebCenter using this task chain. You can then approve/reject pages in WebCenter. The input of this task can be an Imposition or a RunList. If you want to approve the RunList before adding a layout you can use this task to publish a Runlist on WebCenter.

The screenshot displays the WebCenter interface for a document titled "Document: shr_Runlist 1_Layer 1". The document details include:

- Document: shr_Runlist 1_Layer 1 (19.9KB)
- Pages: 32
- Version: 3
- Description: A pagelist from FastTrack
- Project: operator_Proof
- Approval: Cycle Started - Pending [Stop Cycle](#) [Approval Info](#)

 Below the document details is the "Page Gallery" section, which shows a grid of 32 page thumbnails. The first thumbnail (page 1) is highlighted with a green checkmark and a green box around the "Approve Selected Pages" button. The second and third thumbnails (pages 2 and 3) are highlighted with a red box around the "Reject Selected Pages" button. The page numbers 1 through 32 are visible below the thumbnails.

Your approval in the WebCenter is registered in the Automation Engine **Proofs** view. You can proceed with these proofs in the Automation Engine workflows.

Name	Flats	Pages	File	Date modified	Type
shrt	32	32		Wed Aug 01 10:42:53 CEST	Content proof
shrt	32	32		Wed Aug 01 10:08:50 CEST	Content proof
shrt	32	32		Wed Aug 01 10:40:30 CEST	Content proof

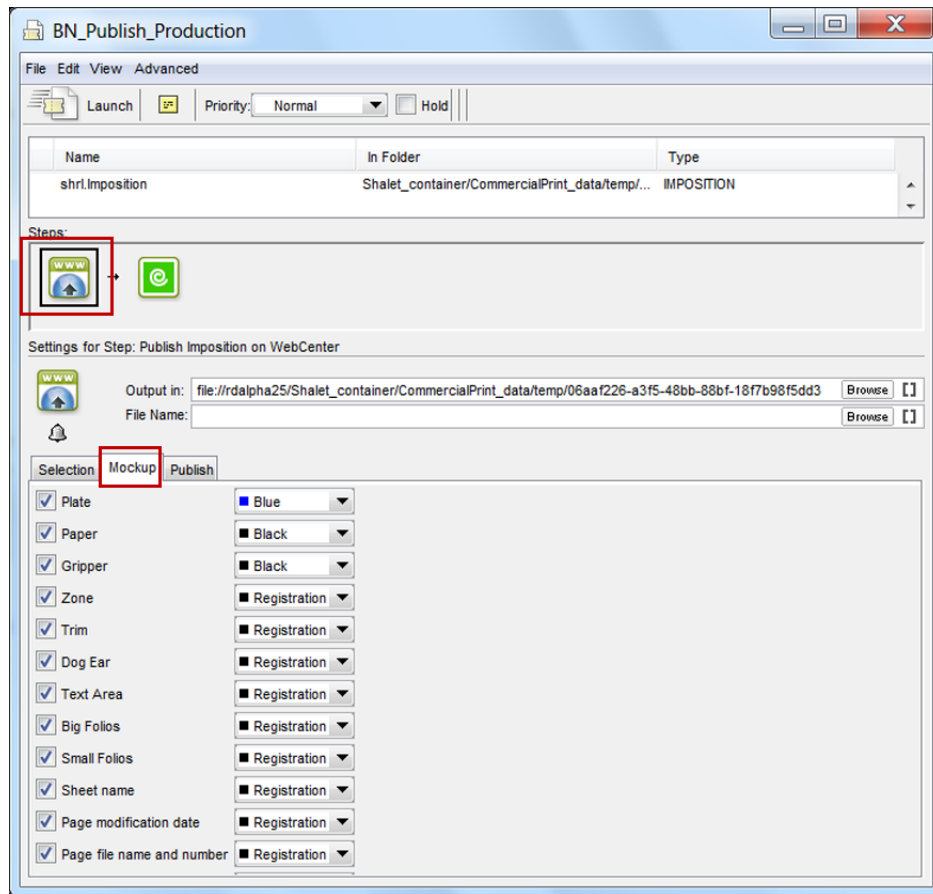
This ticket has two subtasks(steps):

- Publish imposition on WebCenter. Read more in [Publish Imposition on WebCenter](#) on page 446
- Send RunList to WebCenter. Read more in [Send RunList to WebCenter](#) on page 448

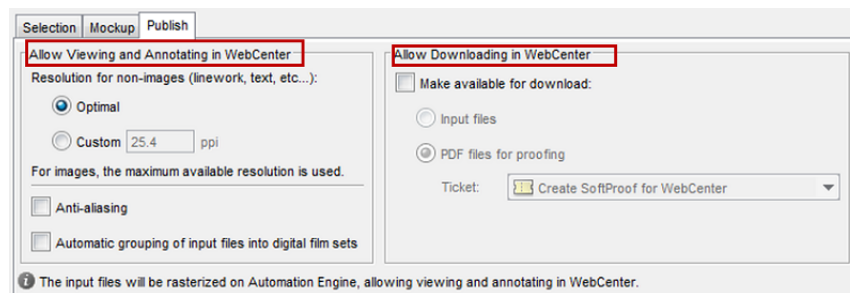
Publish Imposition on WebCenter

Mockup

In this tab, you can check the elements you want to be visible on output and select the appropriate mockup color from the drop down list to enhance visibility.



Publish



Allow Viewing and Annotating in WebCenter

The files will be processed by Automation Engine and uploaded to WebCenter according to the settings for resolution and image processing.

Resolution for non-images for linework and text etc.. can be set to either **Optimal** or **Custom**.

- **Optimal:** If you select this, Automation Engine will calculate the best resolution based on the size of the file.
- **Custom:** You can specify the required resolution.

For images, the maximum resolution available will be used.

Anti-aliasing: You can enable this image correction option to relevant objects. Anti-Aliasing smoothes the staircase effect you get in the generated CTs of line art objects that need pre-processing.

Allow Downloading in WebCenter

You can select which kind of files you want to make available for downloading in WebCenter here:

- **Input files**
- **PDF files for proofing.** You have to choose which **Ticket** should be used to generate the PDF files.

Note: If you select **PDF files for proofing** the viewing quality will be limited to the resolution of the PDF files for viewing.

Send RunList to WebCenter

Destination

You can select the WebCenter settings.

Steps:

Settings for Step: Send Runlist to WebCenter

Destination Approval

Site: RDVMNX01

Project: [Operator]_Proof

Folder:

Document Name: [ImpositionName]_[RunlistName]_[LayerName]

Document Description:

Project Template: TL_AE_Template

Site Logon

User Name: admin

Password: Encrypted:

Use SmartNames:

- **Site:** You can indicate to which site you want to upload the documents. The sites available are the ones defined in [Configure](#).
- **Project:** You can indicate the name of the WebCenter project to which the imposition will be published.

Tip: Use [JobName] as project name to map the jobs in Automation Engine to the projects in WebCenter.

Note: The project does not have to exist in WebCenter at upload time. If the project does not exist, it will be created in WebCenter. The person that does the upload needs to be Project Manager in WebCenter to be able to create a project in WebCenter over the upload.

- **Folder:** You can indicate in which folder the documents should be stored.

- **Document Name:** You can set the name of the published document and this setting allows to have documents published with a name which is different from the name of the published file.
- **Document Description:** You can enter a description about the document which will be available in WebCenter.
- **Project Template:** Here you can enter the Webcenter Project Template. You should have created this project template in WebCenter prior to using this ticket.
- **User Name:** Enter your WebCenter user name to gain access to the site. User name and password are used by WebCenter to check if the Automation Engine operator has the right to publish in the specified project.
- **Password:** Enter your WebCenter password to gain access to the site.

Note: You can use SmartName in most of the fields.

Approval

The Approval tab allows you to select if and when the approval cycle should be started on WebCenter.

Note: Make sure you have an approval cycle set up in WebCenter! You can do this by setting up an approval cycle in a project template, and creating your the project from this template.

Select the **Start the Approval Cycle** option if you want the approval cycle to start automatically as soon as the file is uploaded to WebCenter.

If the user you logged in with in the Destination tab has the **Change Approval Setup** permission in WebCenter, you can also set the approval **Due date**. You can choose:

- **today,**
- **tomorrow,**
- **today + x days** (then define the number of days in the **x =** field that appears),
- **job due date,**
- **other...** (then enter the due date or select it in the calendar).

You can also select the hour at which the cycle should be completed (except when using **job due date**).

2.7.112 Publish On WebCenter

This task is documented in the chapter **Integrating with WebCenter**. Find a direct link [here](#).

2.7.113 Relocate Job

This task is documented in the [separate chapter on Jobs](#). Find a direct link [here](#).

2.7.114 Resize PDF for Output

This task allows to resize PDF pages and/or their contents for output to a proofer or a RIP.

Note: The task uses the **Dimension** unit defined in the [Preferences](#). You can use SmartNames for the values.

Resize to a specific page size: Select this parameter to adjust the PDF markup for a given paper size.

Leave it deselected if you only want to adjust the page contents (page boxes, etc.).

Selecting or deselecting this parameter affects which parameters you see in the **Page Content** and **Page Size** areas. See [Resizing PDF Pages](#) on page 450 and [Resizing PDF Contents](#) on page 451 for details.

Resizing PDF Pages

When **Resize to a specific page size** is selected, you can resize the PDF pages and their contents using the following parameters:

1. In the **Page Content** area, select the **Page Box to Use** for positioning, scaling, etc.
2. Select how to **Position** this page box and its contents on the page:
 - **Don't Fit** (default): the file will be placed "as is".
 - **Center on Page:** the task will center the job in the selected page box.
 - **Scale to Fit (Reduce Only):** the task will scale the job down if necessary, so it fits in the selected page box.
 - **Scale to Fit (Reduce & Enlarge):** the task will scale the job up or down if necessary, so it fits in the selected page box.
 - **Use Tiling:** the task will keep the original file size, and tile it over several pages as necessary.
 - **Use Proportional Tiling:** the task will tile the file over several pages as necessary, but make all tiles the same size.
 - **Gripper on Top:** the task will leave space for the gripper at the top of the page.
 - **Gripper at the Bottom:** the task will leave space for the gripper at the bottom of the page.

When selecting **Use Tiling** or **Use Proportional Tiling**, you can enter the size of the **Tile Overlap** (where two tiles are partly on top of each other).

When selecting **Gripper on Top** or **Gripper at the Bottom**, you can enter the amount of space needed for the gripper in **Gripper Size**.

3. Select **Mirror page** if you want to mirror the page contents and the page boxes horizontally.
4. If desired, use the **Scale** parameter to define a scaling to apply to the artwork before placing it on the page.

Note: The **Scale**, **Horizontal Distortion** and **Vertical Distortion** parameters are not available when selecting **Scale to Fit (Reduce Only)** or **Scale to Fit (Reduce & Enlarge)** in the **Position** parameter.

5. If desired, enter a **Horizontal Distortion** and/or a **Vertical Distortion** to apply to the artwork before placing it on the page.

Distortion is a stretch of the jobs' original dimensions. It is specifically important for Flexo jobs.

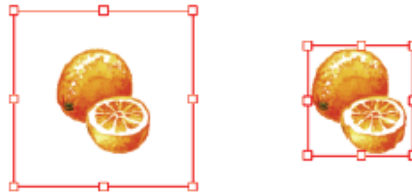
When mounting a Flexo plate on the press, it is stretched out in one direction. By making the output slightly smaller than needed, we compensate for this distortion. The dimensions of the original file remain unchanged.

You can either enter the distorted output values (for example 97% or 103%), or the positive/negative distortion values (for example -3% or +3%). Leave the values at 100% (default) to apply no distortion.

6. Select **Clip on selected page box** if you want to clip away all artwork outside the page box you selected in **Page Box to Use**.

Note: The artwork outside the selected page box is not deleted, just clipped away by a clipping path (it is invisible but still present in the file).

7. Select **Crop to fit** if you want to crop your Page Box to the content of your original file.



8. In the **Page Size** area, select the page size you intend to print to in **New Page Size**.

This page size is used by the **Scale** and **Position** parameters.

- If you select a predefined page size, you will see an **Orientation** parameter, where you can select the orientation of the paper you intend to use. Choose either **Portrait** or **Landscape**.
- If you select **Custom** as a page size, you will see a **Page Width** and a **Page Height** fields where you can enter the dimensions of the custom paper you intend to use.

9. Enter the **Page Margins** required by the output device.

The **Page Margins** value you enter will be used for the four sides of the page. These margins cannot be used for exposing.

Resizing PDF Contents

If you don't select **Resize to a specific page size**, you can resize/reposition the PDF contents using the following parameters:

1. In the **Page Content** area, select the **Page Box to Use** for positioning, scaling, etc.
2. Select **Center content on page** if you want to center the page contents on the page.
3. Select **Mirror page** if you want to mirror the page contents horizontally.
4. If desired, use the **Scale** parameter to define a scaling to apply to the artwork before placing it on the page.

Note: The **Scale**, **Horizontal Distortion** and **Vertical Distortion** parameters are not available when selecting **Scale to Fit (Reduce Only)** or **Scale to Fit (Reduce & Enlarge)** in the **Position** parameter.

5. If desired, enter a **Horizontal Distortion** and/or a **Vertical Distortion** to apply to the artwork before placing it on the page.

Distortion is a stretch of the jobs' original dimensions. It is specifically important for Flexo jobs.

When mounting a Flexo plate on the press, it is stretched out in one direction. By making the output slightly smaller than needed, we compensate for this distortion. The dimensions of the original file remain unchanged.

You can either enter the distorted output values (for example 97% or 103%), or the positive/negative distortion values (for example -3% or +3%). Leave the values at 100% (default) to apply no distortion.

6. In the **Page Size** area, use the **Set Orientation** parameter to alter the orientation of every page without resizing it. You can choose either:
 - **Keep**,
 - **Portrait**,
 - **Landscape**.
7. Use the **Rescale page boxes** and **Rescale page size** parameters to define what happens when you scale the page contents:
 - a) Select **Rescale page boxes** to scale the page boxes together with the contents.
Deselect it to only scale the page contents and not the page boxes.
 - b) Select **Rescale page size** to scale the page size together with the contents.
Deselect it to only scale the page contents and not the page size.

2.7.115 Resolve OPI

Use this task to resolve the OPI-links (links to external high-resolution images) in incoming PDF files that contain the low-resolution images and links to the corresponding high-resolution images.

Note: You can use SmartNames in most fields of this dialog. For information on SmartNames, see [SmartNames](#) on page 916.

Repository tab

- Add the location(s) of the external folder(s) that contain(s) the OPI-files.
- If necessary, add the location(s) of the excluded folders. These folders will not be searched.

Note: Use this option if all images can be found on the same volume and you want to exclude all folders containing low-resolution files.

- Select the option **Update OPI links only** to only update the links to the high-resolution images and not to replace the links by embedding the high-resolution images.

Mapping tab

Add the extensions that need to be replaced while resolving the OPI-links:

- In the **From** field enter the original extension. If left blank, the extension in the **To** field will be applied to all linked files.
- In the **To** field, enter the new extension to give the linked files, or a comma-separated list of extensions to use as replacement for the original extension.

Note: If you enter multiple extensions, Automation Engine will search for a file matching one of them. Use the backslash (\) to preserve a character (for example to use a comma as part of an extension and not as a separator).

2.7.116 Restore Job

This task is documented in the [separate chapter on Jobs](#). Find a direct link [here](#).

2.7.117 Rip IMP File to Screened and Unscreened Tiff File

This task is a combination of the [Rip to Screened Tiff File task](#) and the [Export Imposition to GRI File task](#).

2.7.118 Rip to CDI LEN File

This task has a lot of settings in common with the [Rip to Screened Tiff File task](#).


Select the **device** on which you want to expose.

This task has five tabs:

- General (see the [Rip to Screened Tiff File ticket](#).)
- Inks (see the [Inks tab in a Rip to Screened Tiff File ticket](#).)
- Advanced (see the [Advanced tab in a Rip to Screened Tiff File ticket](#).)
- Pre-Rip (see the [Advanced tab in a Rip to Screened Tiff File ticket](#).)
- [Output](#)

Output

Note:

Output settings that have been specified in the Dispatcher Queue Settings, are preceded by a blue icon .

Configuration Set

Name of the selected configuration file. These configuration files represent specific behavior of the output device.

Output File

Path: leave this empty to use the output path as defined in the Dispatcher (EskoLink queue - Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Name: leave this blank to get an output file name based on the input file. The output name of a multipage file will look as follows: <file>_<page>_<sink>. Non-multipage file will lose the <page> indication. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Resolution

Define the resolution of the output file.

- **Perimeter:** When a predefined configuration set is chosen, this perimeter field will be grayed out and will show the perimeter as it was set in the device queue (see [Configure tool](#)). When no 'conf' was selected. You will be able to edit this field
- **Pixelclock:** Choose here the appropriate Pixelclock of your CDI. This field will most probably be fixed as it does read what CDI type your device queue was based on (in the [Configure tool](#)). 'VariablePLL' CDIs will not show the Pixelclock field nor the Perimeter field.
- **Actual resolution:** This informational field shows the result of the desired resolution and the effect of the perimeter on that desired resolution. It is the combination of these two, the so called 'actual resolution' that will be requested to the Rip. The FlexRip will then (see the log file) round off any non-integer number.

Compression

Indicates the compression method that will be used.

2.7.119 Rip to Screened and Unscreened DCS File

This task has a lot of settings in common with the Rip to Screened Tiff File task.

Select the **device** on which you want to expose.

This task has five tabs


- General (see the [Rip to Screened Tiff File ticket.](#)) There's the obvious distinction between the Screened and unscreened DCS file. The unscreened ticket will not show the screening options.
- [Inks](#)
- Advanced (see the [Rip to Screened Tiff File ticket.](#))
- Pre-Rip (see the [Rip to Screened Tiff File ticket.](#))
- [Output](#)

Note:

The fields in gray are the presets that are defined in the Dispatcher Queue settings.

Output

Note:

Output settings that have been specified in the Dispatcher Queue Settings, are preceded by a blue icon .

Configuration Set

Name of the selected configuration file. These configuration files represent specific behavior of the output device.

Output File

Path: is defined in the Dispatcher (EskoLink queue - Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Name: leave this blank to get an output file name based on the input file. The output name of a multipage file will look as follows: <file >_<page>_<sink>. Non-multipage file will lose the <page> indication. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Select a **Resolution**.

Single File DCS: when this toggle, all separations will be ripped to separate files first. Afterwards, they will all be included in the final DCS file. This means that in the end only one file is created. If you don't select this toggle a Master file will be created, referring to plate files. The number of plate files to which is referred equals the number of separations.

Compression

Choose the compression for the output TIFF file: *Flate*, LZW or CCITT Group.

The *LZW* compression is most commonly used.

CCITT group compressions are more effective but take longer to generate (and to decompress on the receiving side).

The compression to use in your workflow depends on the TIFF downloader on the device you are creating the TIFFs for.

Define the **Preview** resolution.

- For placement only (in a DTP application)
- Proofable Preview (composite printing from a DTP application)

Inks

	Name	Ink Book	Ruling	Angle	Dot Shape	Merge with	Ink Type
<input checked="" type="checkbox"/>	C	process	150.000 lpi	15.000 °			Normal
<input type="checkbox"/>	Die	designer	120.000 lpi	45.000 °			Technical
<input checked="" type="checkbox"/>	307	PANTONE Color...	150.000 lpi	45.000 °			Normal
<input checked="" type="checkbox"/>	215	PANTONE Color...	150.000 lpi	45.000 °			Normal
<input checked="" type="checkbox"/>	K	process	150.000 lpi	45.000 °			Normal
<input checked="" type="checkbox"/>	Y	process	150.000 lpi	90.000 °			Normal
<input checked="" type="checkbox"/>	M	process	150.000 lpi	75.000 °			Normal

All Inks. The standard behavior of the default ticket is to offer all inks in the job to be exposed. Deactivating 'All Inks' opens the ink list where you can select specific inks. The list contains ink names, ruling, angle, dot shape and ink type.

Use technical inks in output. Normally technical inks are hidden when you save a job in an Esko editor. Check the box to output the visible (non hidden) technical inks.

Keep similar inks in their original separations

Keep inks separated

Use **Merge with** to output inks on top of each other in the same output file. This is common when you use different rulings for 1 ink.

For example, Let's assume that the job contains two Cyans, and that they have a different ruling. You might want to put them on the same separation. You can do so by clicking in the 'Merge with' column and selecting the ink it has to be merged with.

Click **Merge Similar Inks** to prompt the system to look for the same inks with a different ruling or angle. The image shows how two Cyans (with a different ruling) were detected and put together. You can reset any modifications by clicking Reset.

Inks:

	Name	Ink Book	Ruling	Angle	Dot Shape	Merge with	Ink Type
<input checked="" type="checkbox"/>	M	process	150.000 ...	75.000 °			Normal
<input checked="" type="checkbox"/>	Y	process	150.000 ...	90.000 °			Normal
<input checked="" type="checkbox"/>	K	process	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	215	PANTONE Colors...	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	C	process	120.000 ...	15.000 °			Normal
<input checked="" type="checkbox"/>	C	process	150.000 ...	15.000 °		C	Normal
<input type="checkbox"/>	Die	designer	120.000 ...	45.000 °			Technical

Merge Similar Inks Reset

Check the box **Highlight modified values** to have all modified values highlighted and preceded or followed by an asterisk.

Inks:

	Name	Ink Book	Ruling	Angle	Dot
<input checked="" type="checkbox"/>	C	process	150.000 lpi	15.000 °	
<input type="checkbox"/>	Die	designer *	* 120.00...	* 45.000 °	
<input checked="" type="checkbox"/>	307	PANTONE Col... *	150.000 lpi	* 45.000 °	
<input checked="" type="checkbox"/>	215	PANTONE Col... *	150.000 lpi	45.000 °	
<input checked="" type="checkbox"/>	K	process *	150.000 lpi	45.000 °	
<input checked="" type="checkbox"/>	Y	process *	150.000 lpi	* 90.000 °	

2.7.120 Rip to Screened and Unscreened PDF File

This task has a lot of settings in common with the Rip to Screened Tiff File task.

Select the **device** on which you want to expose.


This task has five tabs

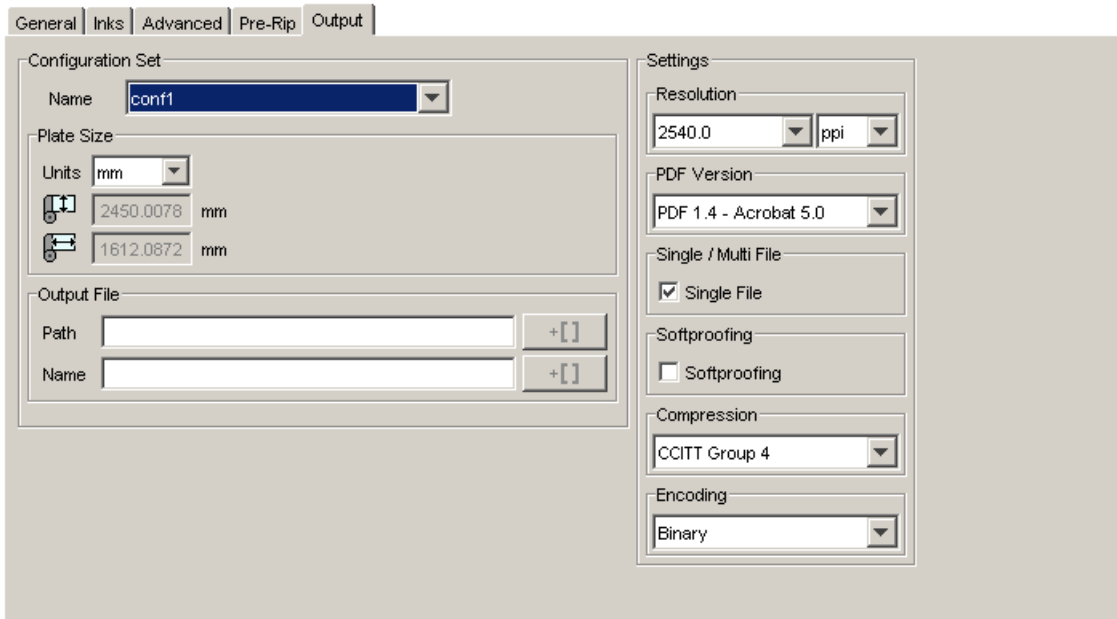
- General (see the [Rip to Screened Tiff File ticket.](#)) There's the obvious distinction between the screened and unscreened PDF file. The unscreened ticket will not show the screening options.
- Inks (see the [Rip to Screened Tiff File ticket.](#))
- Advanced (see the [Rip to Screened Tiff File ticket.](#))
- Pre-Rip (see the [Rip to Screened Tiff File ticket.](#))
- [Output](#)

Note: The fields in gray are the presets that are defined in the Dispatcher Queue settings.

Output

Note:

Output settings that have been specified in the Dispatcher Queue Settings, are preceded by a blue icon .


Configuration Set

Name of the selected configuration file. These configuration files represent specific behavior of the output device.

Plate Size

This shows the plate size of the configuration file you selected.

Output File

Path: leave this empty to use the output path as defined in the Dispatcher (EskoLink queue – Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Name: leave this blank to get an output file name based on the input file. The output name of a multipage file will look as follows: <file >_<page>_<sink>. Non-multipage file will lose the <page> indication. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Select a **Resolution**.

Anti-aliasing When ripping to Unscreened PDF files the output tab offers the option to use Anti-aliasing. Anti-aliasing is relevant for the line-art objects in your design, that need pre-processing and

for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of line-art objects that need pre-processing.

PDF Version

Select a PDF version from the list.

- PDF for Illustrator: Select this version if you want to open the generated PDF 1.4 file in Adobe Illustrator CS2 or higher. Because Adobe Illustrator only supports a (undocumented) subset of PDF 1.4 features, some objects and/or colorspaces must be split up, transformed and/or pre-ripped in order to correctly represent them in Illustrator.
- PDF/X-1 a:2001 (ISO 15930-1)
- PDF/X-3 (ISO 15930-2)
- PDF 1.4: the Acrobat 5 file format

Single / Multi file

When selecting 'single' file output, all separations will be ripped to separate files first. Afterwards, they will all be included in the final PDF or PostScript file. This means that in the end only one file is created. Selecting 'multiple' file output creates a Master file, referring to plate files. The number of plate files to which is referred equals the number of separations.

Softproofing

When soft proofing is enabled FlexRip/P writes a composite PDF file that combines the different separations in a single PDF page. This PDF file can be used for colored viewing in Acrobat.

The combination of 'Single file' with 'Softproofing', can lead to four situations:

- All separations are written in a different page, and all pages are written in a separate output file. (no soft proofing, multi file)
- All separations are written in a different page, and all pages are concatenated in a single file. (no soft proofing, single file)
- Separations are kept together per page, and all pages are written in a separate output file. (soft proofing, multi file)
- Separations are kept together per page, and all pages are concatenated in a single file. (soft proofing, single file)

Compression: choose between CCITT (group 3 and group 4), Flate, RLE, LZW, JPEG and 'No compression'.

Encoding

- Ascii 85: Encodes binary data in an ASCII base-85 representation. This encoding uses nearly all of the printable ASCII character set. The resulting expansion factor is 4:5, making this encoding more efficient than hexadecimal.
- Binary: The image data is represented as binary data. This results in compact files that however are not editable in a common ASCII editor. Nor can these files be sent over a communication line using a communication protocol that relies on control characters in the data stream itself.
- Hex: In this way, the PostScript file remains editable with an ASCII editor at all times. They also can be sent over a communication line using whatever communication protocol is available. The drawback, however, is that hexadecimal image data may take up to twice as much size as binary data.

2.7.121 Rip to Screened and Unscreened PostScript File

This task has a lot of settings in common with the Rip to Screened Tiff File task.

Select the **device** on which you want to expose.

This task has five tabs


- General (see the [Rip to Screened Tiff File ticket.](#)) There's the obvious distinction between the screened and unscreened PS file. The unscreened ticket will not show the screening options.
- Inks (see the [Rip to Screened Tiff File ticket.](#))
- Advanced (see the [Rip to Screened Tiff File ticket.](#))
- Pre-Rip (see the [Rip to Screened Tiff File ticket.](#))
- **Output**

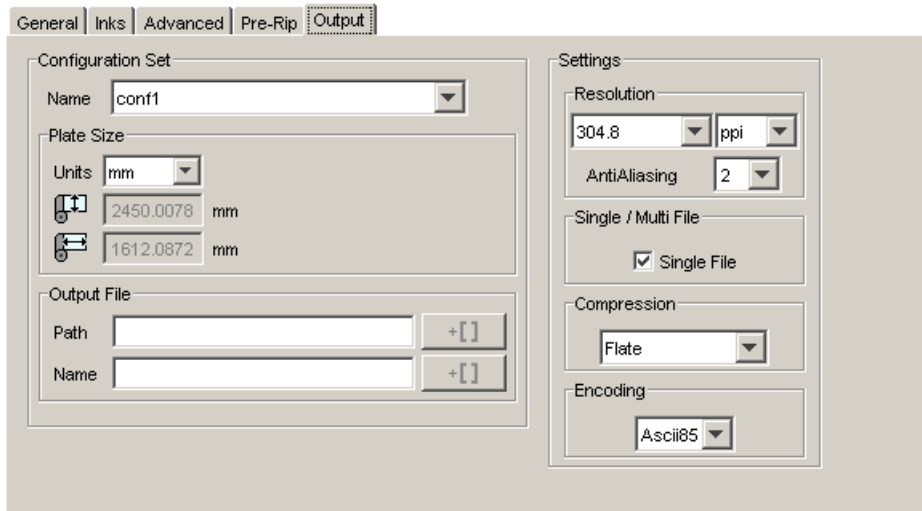
Note:

The fields in gray are the presets that are defined in the Dispatcher Queue settings.

Output

Note:

Output settings that have been specified in the Dispatcher Queue Settings, are preceded by a blue icon .



The screenshot shows the 'Output' tab of a software interface. It is divided into two main sections: 'Configuration Set' and 'Settings'.
Configuration Set:
 - Name: conf1 (dropdown)
 - Plate Size: Units: mm (dropdown); Dimensions: 2450.0078 mm and 1612.0872 mm (with icons for width and height).
 - Output File: Path and Name fields, each with a '+[]' button.
Settings:
 - Resolution: 304.8 (dropdown) ppi (dropdown)
 - AntiAliasing: 2 (dropdown)
 - Single / Multi File: Single File
 - Compression: Flate (dropdown)
 - Encoding: Ascii85 (dropdown)

Configuration Set

Name of the selected configuration file. These configuration files represent specific behavior of the output device.

Plate Size

The plate size of the above selected configuration file is shown.

Output File

Path: leave this empty to use the output path as defined in the Dispatcher (EskoLink queue – Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Name: leave this blank to get an output file name based on the input file. The output name of a multipage file will look as follows: <file>_<page>_<sink>. Non-multipage file will lose the <page> indication. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Select a **Resolution**.

Anti-aliasing When ripping to Unscreened PostScript files the output tab offers the option to use Anti-Aliasing. Anti-aliasing is relevant for the line-art objects in your design, that need pre-processing and for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of line-art objects that need pre-processing.

Single / Multi file When selecting 'single' file output, all separations will be ripped to separate files first. Afterwards, they will all be included in the final PDF or PostScript file. This means that in the end only one file is created. Selecting 'multiple' file output creates a Master file, referring to plate files. The number of plate files to which is referred equals the number of separations.

Compression Choose between *CCITT* (group 3 and group 4), *Flate*, *RLE*, *LZW*, JPEG and 'No compression'.

Encoding

- **Ascii 85:** Encodes binary data in an ASCII base-85 representation. This encoding uses nearly all of the printable ASCII character set. The resulting expansion factor is 4:5, making this encoding more efficient than hexadecimal.
- **Binary:** The image data is represented as binary data. This results in compact files that however are not editable in a common ASCII editor. Nor can these files be sent over a communication line using a communication protocol that relies on control characters in the data stream itself.
- **Hex:** In this way, the PostScript file remains editable with an ASCII editor at all times. They also can be sent over a communication line using whatever communication protocol is available. The drawback, however, is that hexadecimal image data may take up to twice as much size as binary data.

2.7.122 Rip to Screened and Unscreened Tiff File

Select the **device** on which you want to expose.

Note: The fields in gray are the presets that are defined in the Dispatcher Queue settings.

General

Transformations

Scale V and **Scale H**: Define here in % the requested vertical or horizontal scaling. Scaling direction is always applied 'as the file is' (so the V and H are the V and H as you see them in the editor).

Rotate: Choose the requested rotation from the list. Rotation is counter clockwise.

Shrink values from File: GRQ (QuickStep) jobs that have shrink values defined inside the GRQ file will show these values here. You can choose to deactivate the toggle and still edit (overrule) the values.




Attention:

If you use both scaling and distortion (shrink) options, then both these scaling values will be executed!

For example: if you have distortion V 98% set here and in the above options Scale V 95%, then the output-size for V will be 98% of 95% (which is 93.1 %).

So the job will be scaled first and then distorted.

Positive-negative : Define whether you want a positive or negative output file.

Mirroring : Choose whether you want mirroring or not.

Original Size (incl. Margins): The system provides the original Job size, margins included.

Output Size (incl. Margins): The real output size. The size shown here is the original job size influenced by the scaling factors (and/or distortions) and the chosen rotation.

Select a **Mark** Set from the dropdown list. These marks were created (in a central database) by the SmartMarks tool in the LineWork editor (from the **Production** menu).

Screening

Note: The Screening options will not be shown in the Rip to Unscreened Tiff File ticket.

- **Type:** choose between **Esko Classic (Precalculated)**, **DynaCell**, **DynaCell HighlineXM**, **HighlineXM** and **Seamless**.
- **Dotshape:** The selection of JOB (Dotshape(s) from job)' will show in orange when the job does not have the predefined dots (in its ink menu).



Attention: You can expose with an 'orange' dot shape, but the Rip's default dotshape will be used then.

Press Dot Gain Compensation

- **Single Curve:** Select the preferred DotGain Compensation Curve.
- **Automatic:** Select the preferred Screen based Dot Gain Curve.
- **Platemaking DGC:** displays the platemaking DGC defined (if any) in the FlexRip Configurator.

Tip: The Dot Gain Compensation will be applied to Smart Marks as well. The Curve will be applied and compensated for smart marks. To prevent this, rip smart marks separately as copy-dot files.

Inks

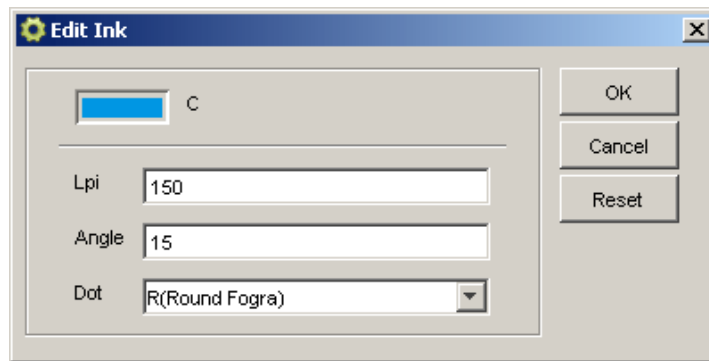
	Name	Ink Book	Ruling	Angle	Dot S...	Merge with	Ink Type
<input checked="" type="checkbox"/>	C	process	150.00...	15.000 °			Normal
<input checked="" type="checkbox"/>	M	process	150.00...	75.000 °			Normal
<input checked="" type="checkbox"/>	Y	process	150.00...	90.000 °			Normal
<input checked="" type="checkbox"/>	K	process	150.00...	45.000 °			Normal
<input checked="" type="checkbox"/>	215	PANTONE Color...	150.00...	45.000 °			Normal
<input checked="" type="checkbox"/>	307	PANTONE Color...	150.00...	45.000 °			Normal
<input type="checkbox"/>	Die	designer	120.00...	45.000 °			Technical

Inks on separate TIFF files This toggle appears when you're outputting to a merging device. Toggle it on to make sure that different inks end up on different films.

All Inks The standard behavior of the default ticket is to offer all inks in the job to be exposed. Deactivating 'All Inks' opens the ink list where you can select specific inks. The list contains ink names, ruling, angle, dot shape and ink type.

If you are granted the **Ink remapping in inks panel** access right (see [Access Rights](#) on page 702), you can edit the inks.

Double-clicking the color swatch opens the Edit Ink dialog:



You can edit Lpi, Angle and Dotshape. Click 'OK' to apply the changes, 'Cancel' to close the dialog without applying the changes and 'Reset' to reset all values to default ones.

Use technical inks in output Normally technical inks are hidden when you save a job in an Esko editor. Check the box to output the visible (non hidden) technical inks.

Keep similar inks in their original separations

Keep inks separated

Use **Merge with** to output inks on top of each other in the same output file. This is common when you use different rulings for 1 ink.

For example, let's assume that the job contains two Cyans, and that they have a different ruling. You might want to put them on the same separation. You can do so by clicking in the 'Merge with' column and selecting the ink it has to be merged with.

Click **Merge Similar Inks** to prompt the system to look for the same inks with a different ruling or angle. The image shows how two Cyans (with a different ruling) were detected and put together. You can reset any modifications by clicking Reset.

Inks:							
	Name	Ink Book	Ruling	Angle	Dot Shape	Merge with	Ink Type
<input checked="" type="checkbox"/>	M	process	150.000 ...	75.000 °			Normal
<input checked="" type="checkbox"/>	Y	process	150.000 ...	90.000 °			Normal
<input checked="" type="checkbox"/>	K	process	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	215	PANTONE Colors...	150.000 ...	45.000 °			Normal
<input checked="" type="checkbox"/>	C	process	120.000 ...	15.000 °			Normal
<input checked="" type="checkbox"/>	C	process	150.000 ...	15.000 °		C	Normal
<input type="checkbox"/>	Die	designer	120.000 ...	45.000 °			Technical

Merge Similar Inks Reset

Check the box **Highlight modified values** to have all modified values highlighted and preceded or followed by an asterisk.

Inks:					
	Name	Ink Book	Ruling	Angle	Dot
<input checked="" type="checkbox"/>	C	process	150.000 lpi	15.000 °	
<input type="checkbox"/>	Die	designer *	* 120.00...	* 45.000 °	
<input checked="" type="checkbox"/>	307	PANTONE Col... *	150.000 lpi	* 45.000 °	
<input checked="" type="checkbox"/>	215	PANTONE Col... *	150.000 lpi	45.000 °	
<input checked="" type="checkbox"/>	K	process *	150.000 lpi	45.000 °	
<input checked="" type="checkbox"/>	Y	process *	150.000 lpi	* 45.000 °	

Advanced

General
Inks
Advanced
Pre-Rip
Output

Hold In Rip Queue

Yellow Ruling: Don't change Y ruling

CT Resampling: Don't resample

Hold In Rip Queue This option offers to hold the task in the rip-queue until someone manually releases this task. For more information, please see the [Tasks View](#).

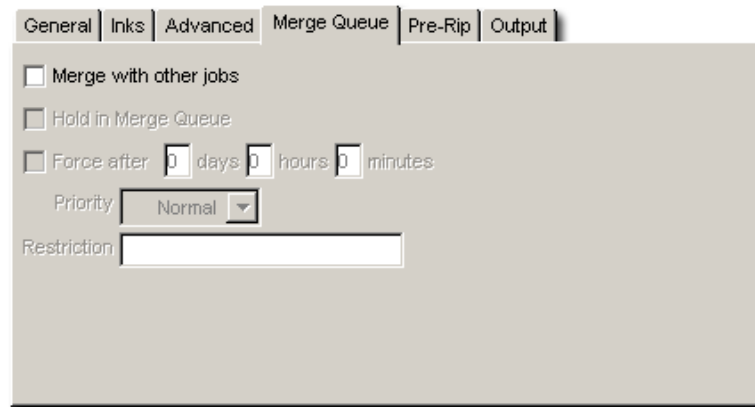
Yellow Ruling choose between:

- **Don't change Y ruling:** this is the default, the Y ruling is not adapted.
- **Change Y ruling:** change the Y ruling, possibilities are: -15%, -7%, 0% (changes nothing), + 7% and +15%.
- **Change Y ruling stochastic:** same as the 'Change Y ruling' settings, but the dotshapes is adapted to the Monet dot.

CT Resampling: choose between:

- **Don't resample:** this is the default. No resampling is done. This avoids most image pixel shift.
- **Resample to highest CT resolution from job:** this choice fixes the remaining image shift problems, with the least upscaling possible without data loss.
- **Resample to automatic chosen resolution:** this option reduces broken dot problems, by interpolating images to at least 1000 ppi
- **Resample to output resolution:** this is the slowest option, resampling all CT's to the output resolution

Merge Queue



If the device queue of a FlexRip/T4 Dispatcher allows the use of the Handlers's background Merger software (see setting in the [Configure tool](#)), you will get this extra 'Merge ' options tab.

Merge with other jobs Choose here if you want that merger program to merge these separations with separations from other jobs or not. 'Merge with other Jobs' is **deactivated** by default.

When activating the 'Merge with other jobs', the other merge options will be offered.

Hold in Merge Queue This will 'hold' the separations in the merger database. This means they are 'not to be considered for merging'. Use the [Merger Monitor](#) to see this database and to release the entries that were put on 'hold in merger'.

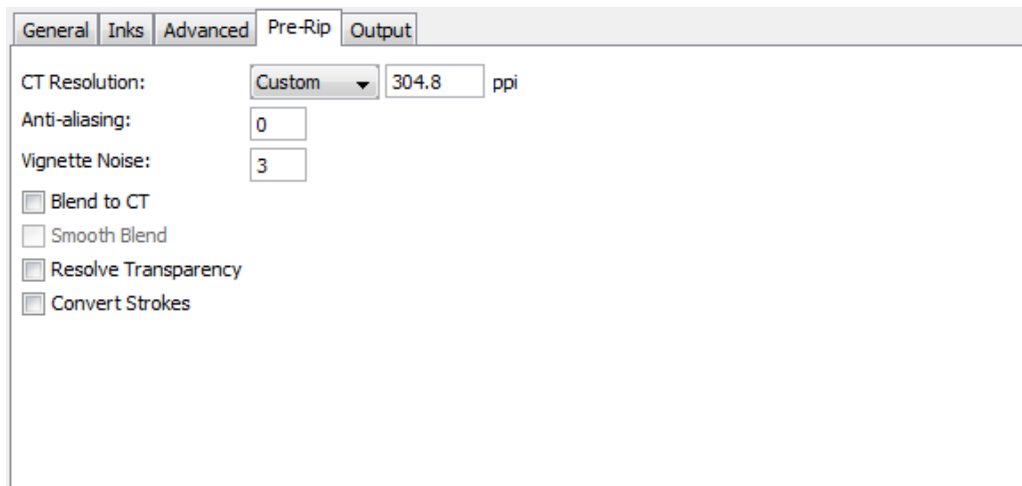
Force after... Use this toggle to have the separations be forced out of the merger database after a chosen time. This prevents having to wait too long before the 'minimum fill %' has been reached.

Priority decides on the priority that entries will have in the merger database. Entries with a high priority will be chosen sooner for merging with other entries.

Restriction The separations will only merge with other separations of jobs that have received the same restriction value from the operator. This value can be anything, whatever makes sense for your workflow. Some examples:

- **fruit** This word 'fruit' could be the restriction: these separations will only be merged with other separations that were also sent with the same restriction : the word 'fruit'. Practical reason could be that you wish to bundle the 'banana', 'cherry' and 'apple' job all on the same film.
- **Bob.** The word 'Bob' could be operator Bob's restriction to only get films with separations of 'his own jobs'.

Pre - Rip



The Pre-Rip tab contains options that are only relevant when the job really needs pre-ripping. The Pre-ripping mentioned here is the 'simplification' that can be necessary before the job-objects can be offered to the Rip. The decision to do this is taken automatically. And even when this is done, the options are best left to their default settings. Special objects (in Normalized PDF format) that require the need for this 'simplification for rip' pre-ripping are:

- 'Edge vignettes' (vignettes with alpha channels effect on the edge of the object)
- Barco Blends (=non contourized GRO-blends)
- Warped CTs (warping done on Placed CTs)

Pre-ripping results in making a plain CT of these special objects that the rip can handle them. The resolution of such a CT and the anti-aliasing can be set here.

CT Resolution

- **Custom:** The CTs created during pre-ripping will be at the resolution given by the user. This is the default choice (with a value of 304.8 ppi) and guarantees the behavior of previous Esko Software Suite versions. This option allows for maximum flexibility in CT resolution setting but is independent of output resolution.
- **Output:** The CTs created during pre-ripping will be at the same resolution as the output resolution of the rip, with a maximum of 1000ppi. When selecting 'Output', the text field containing the CT resolution shows what that resolution will be, but remains un-editable. This option allows for the rip output to be optimal, but may slow down pre-ripping and may require more disk space.
- **Automatic:** The CTs created during pre-ripping will be at a resolution as close as possible to 300 ppi, depending on the output resolution. This CT resolution is calculated in such a way that the output resolution of the rip is an integer amount of times this CT resolution. When selecting 'Automatic', the text field containing the CT resolution shows what that resolution will be, but remains un-editable. This option is a compromise between optimal rip output and fast pre-ripping using minimal disk space. Examples: If the output resolution of the rip is set to 609.6 ppi, then the CT resolution will be set to 304.8 ppi. If the output resolution is 600 ppi or 1200 ppi, then the CT resolution will be set to 300 ppi.

Specify the output **resolution** for the generated CT(s). Default output resolution is 12 lpm or 304 ppi.

Anti-aliasing is relevant for the line-art objects in your design, that need pre-processing and for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of line-art objects that need pre-processing.

Vignette Noise Smoothens the tone jumps or banding effect in vignettes by adding noise/grain (in %). The amount of noise can be specified in the noise field. The higher the value, the more noise (grain) is added.

The **Blend to CT** toggle converts the Barco Graphics blend objects into a CT when exposing your job.


Use the **Smooth Blend** toggle to set the steps of the blend objects to 255 steps when exposing your job. As long as Convert Blend Objects to CT is not activated, Smooth Blend appears dimmed. When Convert Blend Objects to CT is activated and Smooth Blend also, the generated CT will ignore the original steps of the blends of your Barco Graphics design. The amount of steps is set at the maximum which is 255. The goal is to obtain a fluid blend effect.

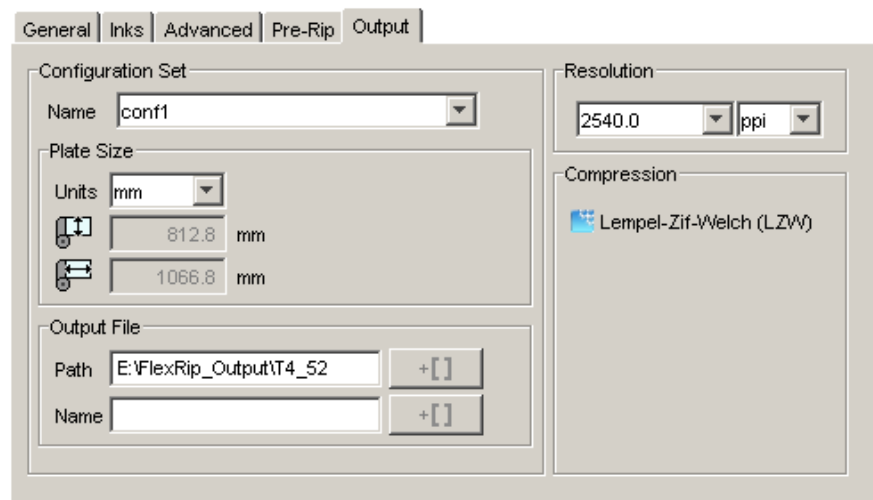
Ripping jobs with lots of transparency can be rather slow. By using the **Resolve Transparency** toggle you can resolve transparencies up-front.

Using the **Convert Strokes** toggle, strokes in the job will be converted to regular objects. By doing so, the Miter limit calculation will be done in exact the same way as it is done in for example PackEdge.

Output

Note:

Output settings that have been specified in the Dispatcher Queue Settings, are preceded by a blue icon .



Configuration Set

Name of the selected configuration file. These configuration files represent specific behavior of the output device.

Plate Size

The plate size of the above selected configuration file is shown.

Output File

Path: leave this empty to use the output path as defined in the Dispatcher (EskoLink queue – Output tab). If your EskoLink queue has no value for the output place; then you can define output folders here.

Name: leave this blank to get an output file name based on the input file. The output name of a multipage file will look as follows: <file>_<page>_<sink>. Non-multipage file will lose the <page> indication. Edit this to overrule this standard behavior. Only plain letters and numbers will be allowed. Other, unsupported characters (symbols) will be replaced by an _.

Select a **Resolution**.

Anti-aliasing When ripping to Unscreened PostScript files the output tab offers the option to use Anti-Aliasing. Anti-aliasing is relevant for the line-art objects in your design, that need pre-processing and for which CTs are generated. Anti-Aliasing smoothens the staircase effect you get in the generated CTs of line-art objects that need pre-processing.

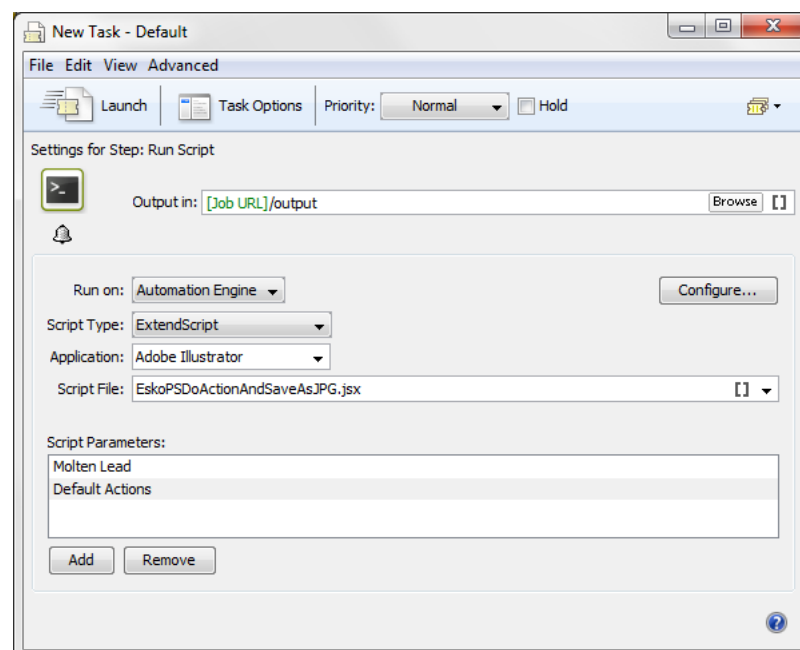
Compression Choose the compression for the output TIFF file. The LZW compression is most commonly used. CCIT group compressions are more effective but take longer to generate (and to decompress on the receiving side). Which compression to use in your workflow depends on the TIFF-downloader of the device you are creating the TIFFs for.

2.7.123 Router

This Workflow Control is documented in the chapter "Workflows". See [Router](#) on page 1024.

2.7.124 Run Script

You can use this task to run a script on the input file. You can run AppleScript, Shell Script in Mac while you can Batch File and Windows Script in Windows. You can use scripting to automate Adobe Illustrator, Photoshop, InDesign and InDesign Server via ExtendScript in Mac and Windows.

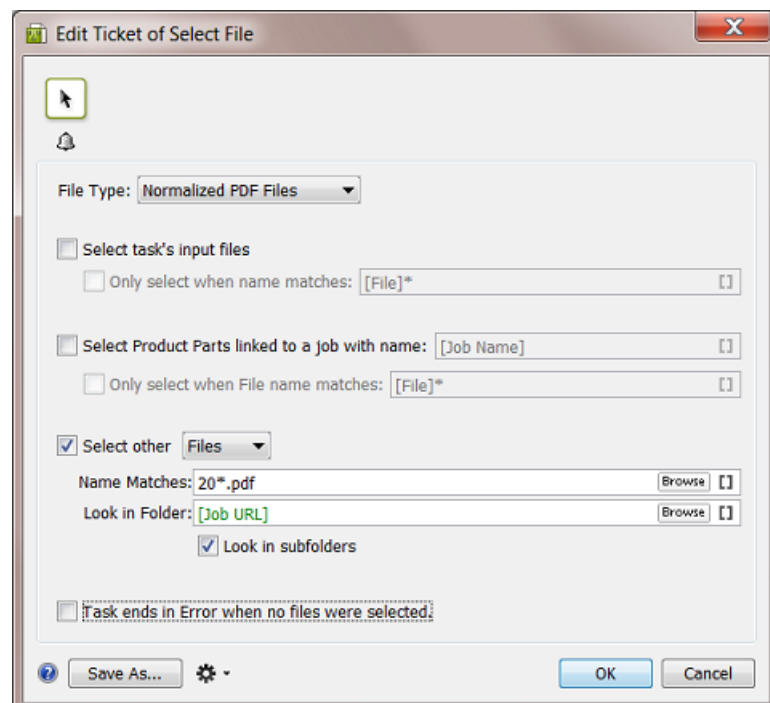


1. **Run on:** is the name of the computer where your **Automation Engine ScriptRunner** is installed. You can read more about configuring and naming Script Runner here [Scripts](#) on page 681
See [Scripts](#) on page 681 for details on how to configure a Script Runner to your Automation Engine server.
2. Select the **Script Type**. The script types are different, depending on your OS:
 - a) For Mac OS: You can choose from **AppleScript**, **Shell Script** and **ExtendScript**.
 - b) For Windows OS: You can choose from **Batch File**, **ExtendScript**, **Windows Script** and **Windows Script (32-bit)**.
The type **Windows Script** will run your script using a 64-bit Scripting Host on a 64-bit Windows OS and a 32-bit Scripting Host on a 32-bit Windows OS.
3. Select the **Script File**. You can select one from the drop-down list, enter a path or/and use SmartNames.
4. If the script you want to run needs one or more parameter(s), enter them in **Script Parameters:** click **Add** and type the parameters.

2.7.125 Select File

The **Select File** workflow control allows you to select one or a group of files on which you launch the next workflow step

Learn more about workflows and see some examples in the chapter [Workflows](#) on page 986 (specifically in [Workflow Controls](#) on page 1014).



Note: You can use SmartNames, and * as wild card character in all input fields.

Note: In fields that end with "name matches", you can also use regular expressions. Learn more in [Using Regular Expressions](#) on page 978.

1. Define what file types you want to select in the **File Type** list. You can:
 - choose to select **Any** file type,
 - pick one of the default file types,
 - choose **Select File Types** and pick them in the **Select File Types** dialog.
2. Select which files you want to select:
 - **Select task's input files** will select all input files (of the selected **File Type** if you specified one).
Use the **Only select when name matches** option if you want to filter the input files to select based on their name.
 - **Select Product Parts linked to a Job with name** will select Product Parts linked to a job of your choice. Specify the job name in this option.
Use the **Only select when File name matches** option if you want to filter the Product Parts based on the name of the file they contain.
See the chapter [Products](#) on page 876 more information about Products and Product Parts.
 - **Select other** will select other files or folders (that have not been processed by the workflow).
 1. Choose to select **Files** or **Folders**.
 2. Use the **Name Matches** option to select those files or folders based on their name. This is not case sensitive.
 3. Define where the files or folders to select are located in **Look in Folder**. Choose to **Look in sub folders** or not.
3. If desired, you can choose to make the **Task end in Error when no files were selected**.

2.7.126 Select Mark File

This Workflow Control is documented in the chapter "Workflows". See [Mark File and Select Marked File](#) on page 1029.

2.7.127 Select Referenced File

This Workflow Control is documented in the chapter "Workflows". See [Select Referenced File](#) on page 1018.

2.7.128 Send E-mail

You can use this task to have Automation Engine send an E-mail automatically at a certain point in your workflow.

This E-mail can contain:

- custom text,

- job information (you can use SmartNames in all of the ticket's fields),
- the task input files.

Note: The mail server settings must be correctly set in **Configure** for this task to work. See [E-Mail Configuration](#) on page 636.

1. Enter the E-mail address(es) you want the E-mail to be sent **To**. If you want to send the E-mail to the customer, add the SmartName (E-mail address) from SmartName category **Customer E-mail address**. You can use SmartNames to add the E-mail address of the Task owner from **Task information** category in the SmartName field.

Note: If you use SmartNames, make sure they resolve to a valid E-mail address.

2. If desired, add extra E-mail address(es) in **Cc**. The recipients' mail address will be displayed to the receiver.
3. If desired, add extra E-mail address(es) in **Bcc**. The recipients' mail address will not be displayed to the receiver.

Similar to the **To** field, you can also use SmartNames (that are valid E-mail addresses) in **Cc** and **Bcc**.

4. Enter the desired E-mail **Subject**.
5. If desired, you can change the sender of the mail in **From**. By default the E-mail address of the **Task Owner** will be used.

The **Send E-Mail** task's ability to send E-mails on behalf of the task owner (or any other user address that's filled in as **From**) is subject to the mail server configuration. When in doubt, please consult your IT administrator.

6. Enter your **Message**.
7. By default, the **Send E-mail** task uses the system's local encoding.

If you notice unreadable characters when reading an Automation Engine E-mail in your E-mail program (for example for Asian characters), you can change the encoding to UTF-8.

Add this to the Message field:	to...
<code>#set (\$SubjectCharSet="utf-8")</code>	set the encoding of the Subject to UTF-8.
<code>#set (\$ContentType="text/plain;charset=utf-8")</code>	set the encoding of the Message to UTF-8 and use plain text.
<code>#set (\$ContentType="text/html;charset=utf-8")</code>	set the encoding of the Message to UTF-8 and use HTML.
<code>#set (\$AttachmentsCharSet="utf-8")</code>	set the encoding of the file name of the attachments to UTF-8.

8. If you want to include the task input files, select **Attach input files**.
9. You can set **public parameters** if you want to be able to fill in E-mail settings when submitting files to your workflow from Shuttle.

Learn more in [Using Public Parameters in Workflows](#) on page 1051.

10. Save your ticket if you want to use it as an **E-mail template** later.

Note:

Self-closing tags are not supported (example
 or <address/>). When setting up HTML E-mails, you have to use a separate opening tag and closing tag (example <address></address>), or the empty variant (example:
).

2.7.129 Send PostScript File to Device

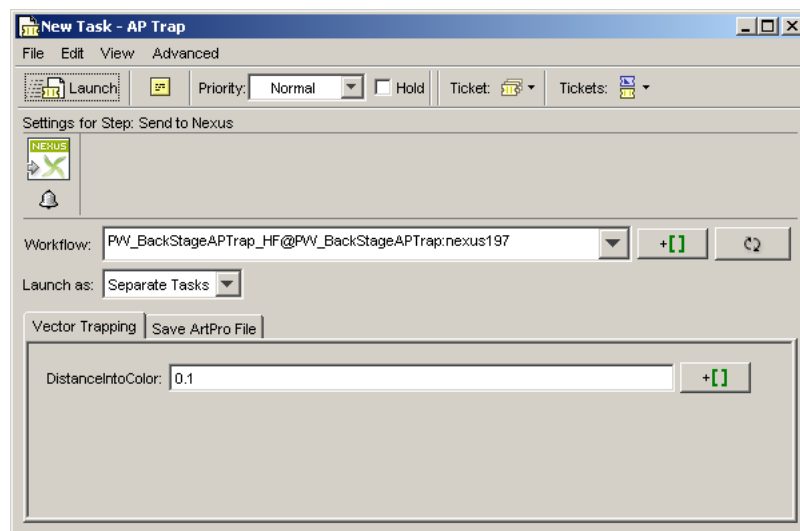
This task outputs a PostScript file to a PostScript device.

Use the drop down list to select the PostScript device you want to output to. The devices are PostScript devices defined on the server.

2.7.130 Send to Nexus

The **Send to Nexus** ticket consists of two parts:

- the top section of the ticket dialog is always the same, and allows you to:
 - select a particular Nexus workflow from the **Workflow** list;
 - indicate whether you want to launch a new, separate workflow in Nexus for every file you select in the Pilot (**Separate Tasks**), or whether you want to send all the selected files to one Nexus workflow (**One Single Task**).
- the bottom section of the ticket dialog contains fields for the **public parameters** for the workflow you selected. These parameters can be different for every Nexus workflow (they are defined using NeXML).




For more information, please see [Integration with Nexus](#) on page 103 in the User Guide.

2.7.131 Send to Odystar

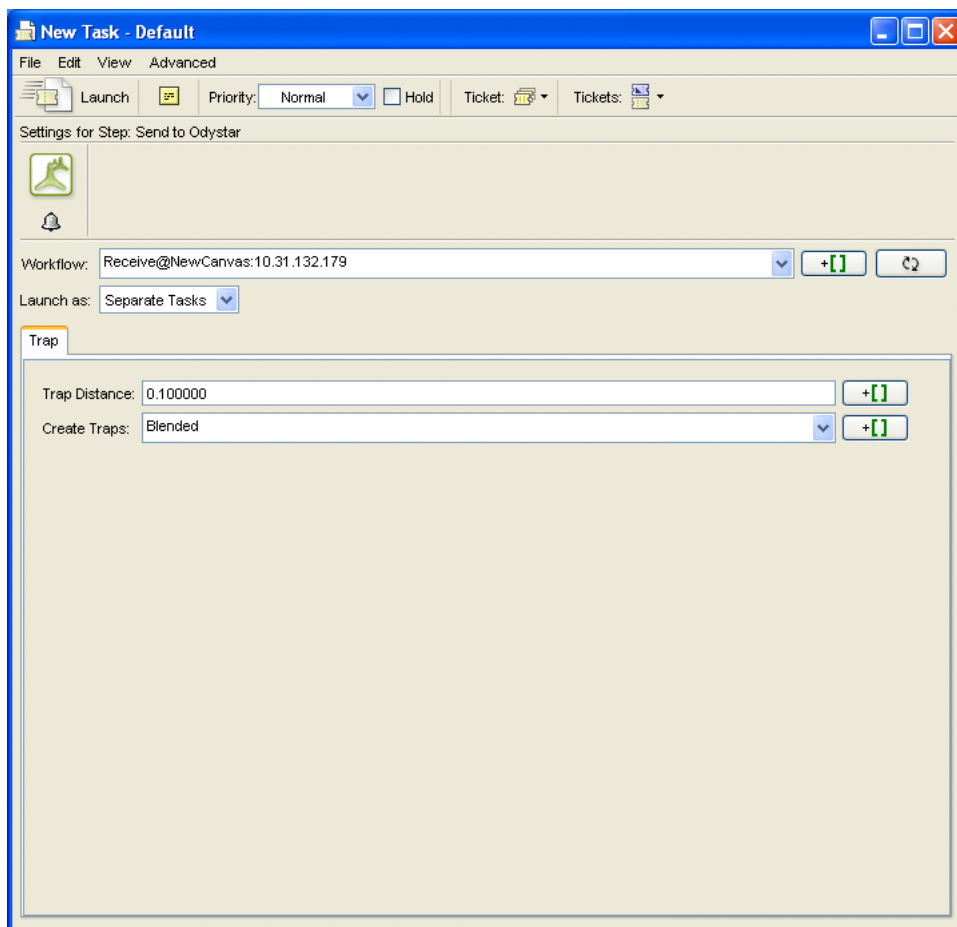
The **Send to Odystar** ticket consists of two parts:

- the top section of the ticket dialog is always the same, and allows you to:
 - select a particular Odystar workflow from the **Workflow** list;

Tip: If you can't see the workflow you are looking for, use the refresh button .

- indicate whether you want to launch a new, separate workflow in Odystar for every file you select in the Pilot (**Separate Tasks**), or whether you want to send all the selected files to one Odystar workflow (**One Single Task**).
- the bottom section of the ticket dialog contains fields for the **public parameters** for the workflow you selected. These parameters can be different for every Odystar workflow (they are defined in the Inspector).

Note: You can also manage your Odystar workflow's public parameters (not one by one but as a group) in your task ticket. See [Managing Public Parameters in Your Ticket](#) on page 70.



2.7.132 Set Priority

This Workflow Control is documented in the chapter "Workflows". See [Set Priority](#) on page 1021.

2.7.133 Sort

This Workflow Control is documented in the chapter "Workflows". See [Sort](#) on page 1021.

2.7.134 Split Pages

The **Split Pages** task can split a multi-page PDF into single page PDFs, or smaller multi-page PDFs, and generate a RunList that contains references to all output PDFs.

Output in: select where you want to output the generated PDFs and the RunList. By default, this is the location of the original file (**[File URL]**).

File Name: Enter the name of the generated RunList. By default, this is **[File]_RunList.jdf**.

Split Mode: Choose how you want to split the pages of the input file:

- **Split Pages:** the task will split the job into individual 1-page files, and generate a RunList.
- **Split in Groups:** the task will split the job into several files having the same number of pages, and generate a RunList. You can enter this number of pages in the **Group Size** parameter.
- **Extract Pages:** the task will extract the pages you specify in the **Page Ranges** parameter, and generate a RunList.

Note: Depending on the **Split Mode** you choose, you will see different parameters.

Group Size: This parameter is available when you choose the **Split in Groups Split Mode**.

Enter the number of pages you want each output file to have.

Note: If the task cannot output files having the same number of pages (for example if you send a 5 pages PDF to be split into files of 2 pages), the last output file will have less pages.

Pages File Name: Use this parameter to set the name of the output files.

You can use SmartNames in this field, including two extra SmartNames: **[Page Label]** and **[Page Number]** (the page label and page number included in the PDF).

Page Ranges: This parameter is available when you select **Extract Pages** in the **Split Mode** parameter.

Enter the page ranges to extract: either single pages or page ranges with a hyphen (for example 1-3 for pages 1 to 3). Separate the page ranges by commas.

Output as one multi page document: This parameter is available when you select **Extract Pages** in the **Split Mode** parameter.

Select it if you want to output all of the page ranges you entered as one single document.

Output remaining pages: This parameter is available when you select **Extract Pages** in the **Split Mode** parameter.

Select it if you want to output all the pages you didn't include in the **Page Ranges** parameter as one multi-page PDF.

Remaining Pages File Name: This parameter is available when you select the **Output remaining pages** parameter.

It allows you to set the name of the multi-page PDF containing all the pages you didn't include in the **Page Ranges** parameter.

You can use SmartNames in this field (including the **[Page Label]** and **[Page Number]** SmartNames).

Copy XMP data: Select this parameter if you want to copy the XMP data of the original file to the output files.

2.7.135 Split XML

This task is documented in the chapter **Integrating with External Systems**. Find a direct link [here](#).

2.7.136 Step & Repeat CAD based

This batch step and repeat task is based on a die-layout, it places graphic files on the different designs in the CAD layout.

This task supports MFG files (Artios Manufacturing files) and CFF2 files.

General

Location Stations Define the folder where the task can find the stations.

When the task is unable to find the correct files, set the task's status to:

- Provide Info
- Generate a warning, the task will continue.
- Generate an error, the task will stop.

Add **SmartMarks** by selecting a SmartMark set from the drop down list. You can also use SmartNames.

Check the '**Load Mark Set in bottom layer**' box if you want to make sure the marks are positioned in the bottom layer, to make them appear 'behind' the graphics.

The **Limit Bleed** toggle: the bleed can be limited to a specific distance. The bleed mask is created by spreading the die shape.

Plate Setup

The sheet size is automatically extracted from the CAD template but neither **Artios MFG** nor CFF2 have a notion of plate. As a result the plate size and position of the sheet on the plate have to be specified in the **Plate Setup** tab.

Plate Size: enter -if necessary- the plate's horizontal and vertical dimensions.

The **Fit Plate to Sheet** toggle makes sure that the plate has the same size as the sheet. Deselecting this option enables you to manually define the plate size and the sheet offset.

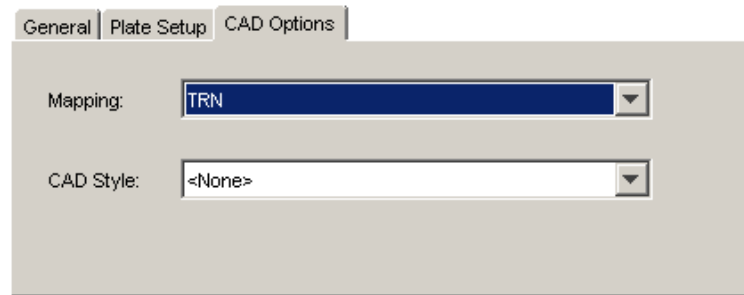
Define horizontal and vertical **Sheet Offset**

Position the sheet on the plate by dragging and dropping the red square (the sheet) to the right position on the blue square (the plate).

Enter horizontal and vertical **shrink** values.

Tip: All edit fields are SmartName-enabled.

CAD Options



Both **Mapping** and **CAD Style** are options you can define in the Esko Editors.

Mapping allows you to select a set of Mapping rules. Mapping rules allow to map DXF layer names, DXF line styles and DXF color numbers onto line types (ArtiosCAD style). Use the mapping rules to indicate which DXF elements will be imported as cut, crease or bleed lines. The mapping rules are scanned from top to bottom, as soon as a DXF element matches a rule, the action corresponding with this rule (ignore or import as cut, crease, bleed) will become effective.

A **CAD Style** changes the appearance of the imported CAD lines.

2.7.137 Step & Repeat Tabular

This task will execute all step and repeat parameters (number of one-ups vertically and horizontally, gap between one-ups, alignment, SmartMarks...), and output the sheet layout.

The ticket's input fields also accept SQL queries, so that information stored in a database can be used to drive the sheet layout creation.

The main benefit of this process is the single entry point for data: the typical production-critical data required for step & repeat is usually known upfront and stored in a project or workflow management system.

The operator does not have to enter these values from a printed sheet, which results in serious performance increases and reduction of human errors.

Tabular typically means 'next to each other'. In this task, data can be entered in three different ways:

- Manually entered numeric values,
- Esko formulas like [VPLATE] (the Esko legacy QuickStep formulas),
- User - defined SQL queries that retrieve the values from external (DIE) databases.

General

Origin: define the point of origin from which the repetition is to be built. Choose from four possible origins.

(top left), (top right), (bottom left), (bottom right).

Unit: select a unit (mm or inch).

SmartMarks: Select SmartMarks from the dropdown list if needed.

Insert SmartNames button Click this button to open the SmartNames dialog. The button is only enabled if the selected input field accepts SmartNames.

Tip: Double clicking or right-clicking a field also opens the 'Insert SmartNames dialog'.

Check the '**Load Mark Set in bottom layer**' box if you want to make sure the marks are positioned in the bottom layer, to make them appear 'behind' the graphics.

Plate

Size: enter the horizontal and vertical plate formula, value or SmartName(s). **Shrink:** if necessary, enter a shrink value, formula or SmartName.

Sheet

Size: enter the horizontal and vertical sheet formula, value or SmartName(s) (see [Formulas](#) on page 487 for details).

Offset: enter an offset value.

Sheet Position on Plate: position the sheet on plate by moving the red rectangle (the sheet) on the blue square (the plate).

Screen registration: relevant only to Esko FlexRip users.

When activated, the origin of the screening will be reset for each one-up on the layout. This guarantees that the exact same screening will be applied for each one-up.

When deactivated, screening will be done continuously throughout the complete job.

Tip:

Right-click a field to open an extra dialog:

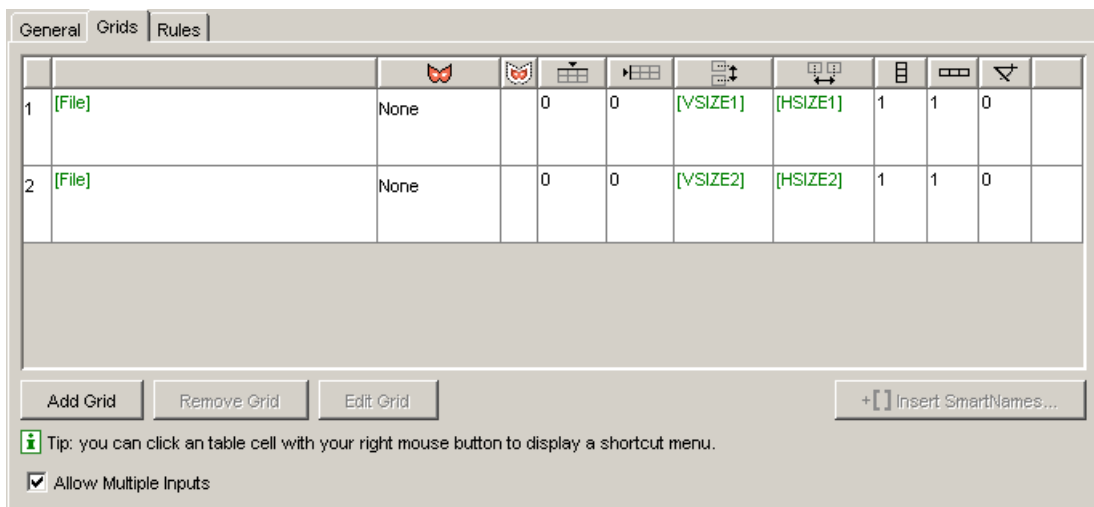
- Cut
- Copy
- Paste
- Clear Contents
- Insert Formula (choose between a global or a grid formula). See [Formulas](#) on page 487.
- Insert SmartNames

Note: Some fields only accept SmartNames defined with 'Step and Repeat' where as others also allow global SmartNames. See [SmartNames](#) for more information.

Grids

A repetition chart may be composed of several grids. If a repetition chart is composed of two jobs, there will be two grids. Each grid has its own number.

Grid Tab Columns












1. **Grid number** You can click it to select the corresponding grid.
2. **File Name** This shows the name of the one-up assigned to this grid.
3. **Grid Mask** To define the mask area around a job.

Depending on where the repetition is positioned in the layout, a mask may or may not be required. Use the dropdown list to choose between 'none' or 'borders'. The dropdown list appears when you click the field.

4. **Grid Mask Bleed** To extend a selected grid mask with a bleed.

Specify the spread distance in the Grid Mask Bleed input box.

Mask is spread but masks will not overlap.

5. **Vertical offset to sheet**  To define the top or bottom shift of the grid in relation to the sheet, depending on the setup of the origin.
6. **Horizontal offset to sheet**  To define the left or right shift of the grid in relation to the sheet, depending on the setup of the origin.
7. **Vertical step**  To define the vertical distance between two steps of the grid. The distance is calculated from center point to center point of the job.
8. **Horizontal step**  To define the horizontal distance between two steps of the grid. The distance is calculated from center point to center point of the job.
9. **Vertical count**  To define the number of times you want to repeat your job vertically.
10. **Horizontal count**  To define the number of times you want to repeat your job horizontally.
11. **Angle**  To define the angle to rotate your job when repeated.
12. **Head Turn / Stagger** This column indicates if you have defined a head turn or staggering for your grid (for example,  for vertical stagger, or  for column-based head turn). See [Grid Settings](#) on page 481 for details on staggering and head turns.

Managing Your Grids

Add Grid: Use this button to add an extra grid.

Note: When you select multiple files in the pilot, the system adds enough grids to accommodate all the jobs you selected (the ticket is automatically updated).

Remove Grid: To remove a grid, select it and click this button.

Edit Grid: Select a grid and use this button to define advanced grid settings (staggering, head turns...). See [Grid Settings](#) on page 481.

Other Options

Insert SmartNames...  opens the 'Insert SmartNames' dialog, which allows you to insert 'Formulas' in the selected field.

Note: Some fields (Grid Mask and Grid Mask Bleed) cannot contain formulas.

Allow Multiple Inputs

This option determines what the task will do when you launch it on multiple files.

- Select it to generate one Step and Repeat file for all of the input files. Each input file will be assigned a different grid in the Step and Repeat.
- Deselect it to treat each input file separately, and generate a different Step and Repeat file for each input file.

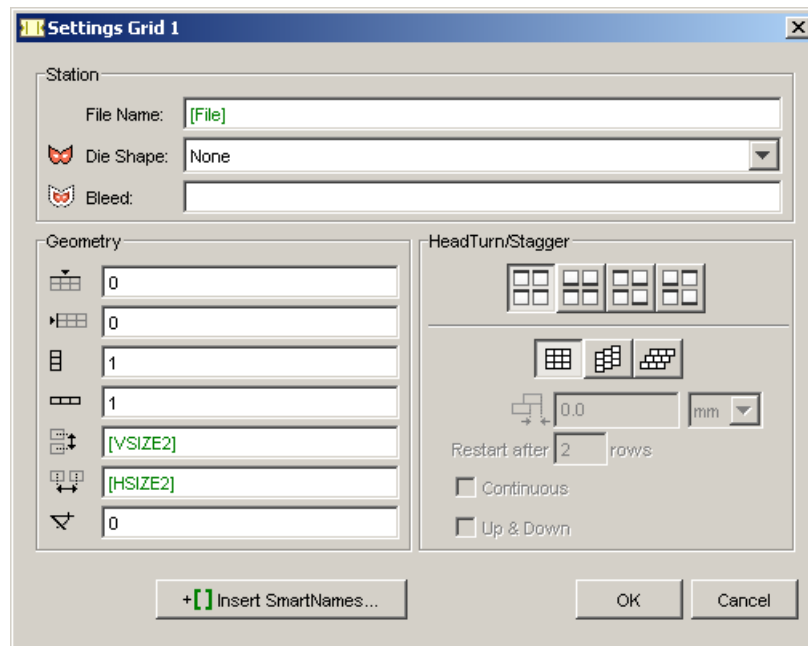


Attention:

This option is only available when you open the **Step & Repeat Tabular** ticket from the **Tickets** view.

To use this option on your files, save your Step & Repeat ticket as a custom ticket first.


Grid Settings




In the **Settings Grid x** dialog, you can change settings defined in the ticket (**File Name**, **Die Shape**, **Bleed**, and **Geometry** options), but also define advanced settings (**Head Turn/Stagger**).

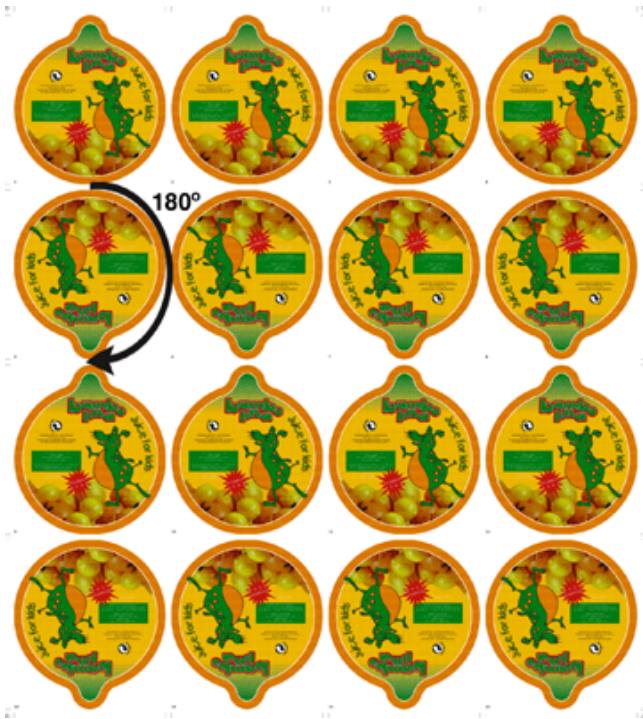
Head Turn


Use this parameter to rotate certain stations in the grid by 180 degrees.

- Use the first button () to keep all stations as they are, without rotating them (default).




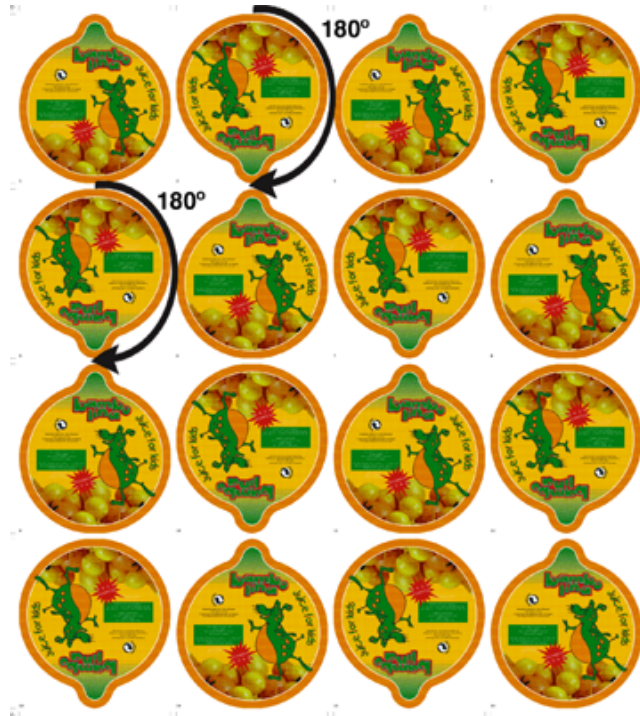
- Use the second button () to rotate objects in alternating rows by 180°.



- Use the third button () to rotate objects in alternating columns by 180°.






- Use the fourth button () to rotate objects both in alternating columns and in alternating rows by 180°.



Stagger

Use the Stagger option to stagger (move) certain stations of your grid horizontally or vertically.

- Use the first button () to keep all stations as they are, without staggering them (default).
- Use the second button () to define a vertical stagger for your grid.
- Use the third button () to define an horizontal stagger for your grid.

To define a horizontal or a vertical stagger:

1. Click the horizontal or the vertical stagger button.
2. Enter a stagger value and choose a unit in the fields below the stagger buttons (mm, cm, mil, inch or percentage of the width or height).

The example below shows a 4 x 6 Step & Repeat with a 50% vertical stagger.



3. Adjust your staggering with the **Restart after**, **Continuous** or **Up & Down** options if you wish.

Restart after

You can choose to restart the staggering after a certain number of rows or columns. This means that:

- for vertical staggering: the top station of that column is positioned at the same height as the top station of the first column (it has the same vertical offset from the top of the sheet).

For example, the staggering below restarts after 3 columns.



- for horizontal staggering: the first (leftmost) station of that row is positioned at the same horizontal offset as the first station of the first row.

Continuous

Select **Continuous** if you want to add extra stations to fill the blank space.

The example below is a 4 x 6 Step and Repeat (with staggering restarting after 3 columns), but the **Continuous** option added some stations vertically.



Up & Down

Use **Up & Down** together with **Restart after** to go back in the other direction gradually instead of at once.

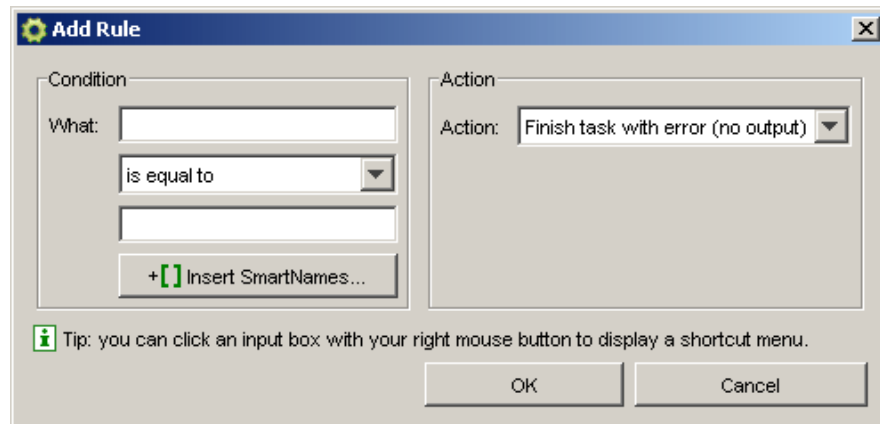
The example below restarts after 3 columns, with the **Up & Down** option selected.



Rules

General		Grids	Rules
Condition	Action		
<input checked="" type="checkbox"/> [NROFINKS] is less than 8	<input checked="" type="checkbox"/> Finish task with error (no output)		
<input type="button" value="Add ..."/> <input type="button" value="Modify ..."/> <input type="button" value="Remove"/>			

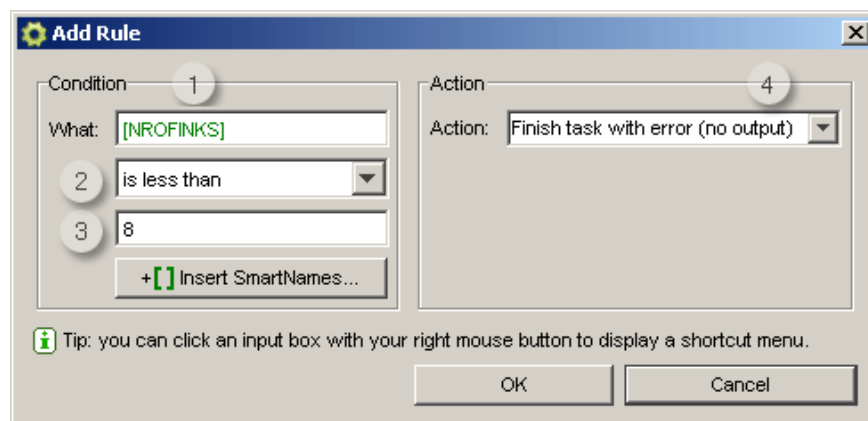
You can modify the task's behavior by defining rules. Click 'Add' to open the 'Add Rule' dialog:



Define the 'Condition' and the 'Action'.

For example, we want to define a rule that states that all jobs with more than 8 inks shouldn't be output.

First you define the condition, then you define what happens if the condition is not met.



1. Enter the parameter you want to use as check.
2. Choose an 'expression' from the dropdown list.
3. Enter the value for the parameter.
4. Choose the Action for when the conditions is not met from the dropdown list: OR the task finishes with a warning OR the task finishes with a warning (there's no output).

Formulas

Some of the fields work with formulas, which act as parameters and may represent different things.

There are two different kinds of formulas: Global formulas (that define the whole repetition) and Grid formulas (that define individual grids).

Note: A repetition chart may be composed of several grids. If a repetition chart is composed of two jobs, there will be two grids. Each grid has its own number.

1. Global formulas:

- VPLATEFIT: vertical plate size that fits around a repetition sheet and SmartMarks.
- HPLATEFIT: horizontal plate size that fits around a repetition sheet and SmartMarks.
- VPLATE: vertical plate size.
- HPLATE: horizontal plate size.
- VSHEETOFFSET: vertical offset of the sheet relative to the Plate.
- HSHEETOFFSET: horizontal offset of the sheet relative to the Plate.
- VSHEETFIT: vertical sheet size that fits around the repetition.
- HSHEETFIT: horizontal sheet size that fits around the repetition.
- VSHEET: vertical sheet size.
- HSHEET: horizontal sheet size.
- SHEETTOP: top position of the sheet relative to the Plate.
- SHEETLEFT: left position of the sheet relative to the Plate.
- SHEETBOTTOM: bottom position of the sheet relative to the Plate.
- SHEETRIGHT: right position of the sheet relative to the Plate.
- NROFINKS: number of non-technical inks (Technical inks will not be printed).
- NROFGRIDS: number of grids


2. Grid Formulas:

x = the grid number.

- VLABELx: vertical size of non-rotated job.
- HLABELx: horizontal size of non-rotated job.
- VSTEPx: vertical step size, from center point to center point.
- HSTEPx: horizontal step size, from center point to center point.
- VSIZEx: vertical size of bounding box of rotated job.
- HSIZEx: horizontal size of bounding box of rotated job.
- VCOUNTx: number of vertical repetitions in grid x.
- HCOUNTx: number of horizontal repetitions in gridx.
- ANGLE: the angle of the rotation.
- GRIDVSIZEx: vertical size of the grid (top-bottom).
- GRIDHSIZEx: horizontal size of the grid (left-right).
- TOPx: top position of the grid relative to the sheet.
- LEFTx: left position of the grid relative to the sheet.
- BOTTOMx: bottom position of the grid relative to the sheet.
- RIGHTx: right position of the grid relative to the sheet.
- BLEEDx: distance to spread the grid mask with.

This example illustrates how one could use formulas to define an auto fill repetition: a defined sheet will be automatically filled.

General Grids Rules

Origin:  Unit: mm SmartMarks: <None>

Plate

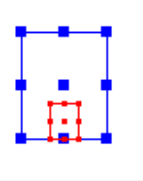
Size:

Shrink: %
 %


Sheet

Size:











Offset:

Sheet Position on Plate: 


+ [] Insert SmartNames...

 Tip: you can click an edit box with your right mouse button to display a shortcut menu.







General Grids Rules

File Name										
1 banana_y oghurt.grs	Borders	3.0	$(([VSHEET]/[VCOUNT1] - [VLABEL1])/2)$	$(([HSHEET]/[HCOUNT1] - [HLABEL1])/2)$	$[VSHEET]/[VCOUNT1]$	$[HSHEET]/[HCOUNT1]$	$[VSHEET]/[VLABEL1]$	$[HSHEET]/[HLABEL1]$		0

Add Grid Remove Grid + [] Insert SmartNames...

 Tip: you can click an table cell with your right mouse button to display a shortcut menu.

The formulas break down like this:

1. First you define the vertical  (and horizontal ) count of the grid: the vertical count is the vertical size of the sheet divided by vertical size of the job, hence: $VSHEET / VLABEL1$. (The horizontal count: $HSHEET / HLABEL1$.)
2. Then you are able to define the vertical  (and horizontal ) step: vertical gap is the vertical sheet size divided by the vertical count, hence: $VSHEET / VCOUNT1$. (The horizontal step: $HSHEET / HCOUNT1$.)
3. Now for the vertical  (and horizontal ) offset: you want to center the repetition on the sheet. You can do that by defining the vertical offset as half the vertical step minus the vertical size of the job, hence: $(VSTEP1 - VLABEL1)/2$. (For the horizontal offset: $(HSTEP1 - HLABEL1)/2$.)

Database Query: an Example

Let's say you have a database in which you keep data about the Die you use:

	Die	StepV	StepH	Teeth	Teethsize	DrumV	DrumH	Cilinder	GapH	GapV
▶ +	MES603	4	2	109	3.75	408.75	250	160.925196850	6	10
+	MES604	4	2	106	3.75	319.45482562	215	125.769616386	10	20
*		1	1			0		0	0	0

You might want to use this data to set up a **Tabular Step and Repeat** Ticket.

It's possible to define SmartNames in such a way that they query a database, and the **Tabular Step and Repeat** Task is able to use SmartNames.

Note:

Use the **Configure** tool to define SmartNames (see [SmartNames](#) on page 916).

Edit SmartName

Name: DB_gapV

Scope: Step and Repeat

Action

Type: Database Query

Data Source Name: XampleGraphics

SQL Command: SELECT Dies.GapV
FROM Dies INNER JOIN Jobs ON Dies.GapV = Jobs.Dies
WHERE (((Jobs.[Job ID])=[First5]));

Example

Value: 10

OK Cancel

You can define SmartNames that query all the values of your database, using SQL commands, in this example, we defined the following:

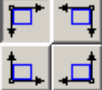
- DB_DrumH: Retrieves the value for the horizontal size of the drum.
- DB_DrumV: Retrieves the value for the vertical size of the drum.
- DB_GapH: Retrieves the value of the gap's horizontal size.
- DB_GapV: Retrieves the value of the gap's vertical size.
- DB_Teeth: Retrieves the number of teeth.
- DB_StepH: Retrieves the value of the die's horizontal step.
- DB_StepV: Retrieves the value of the die's vertical step.

CustomerName	Global	Database Query
PhoneNumber	Global	Database Query
DB_DrumH	Step and Repeat	Database Query
DB_DrumV	Step and Repeat	Database Query
DB_teeth	Step and Repeat	Database Query
DB_gapH	Step and Repeat	Database Query
DB_gapV	Step and Repeat	Database Query
DB_StepH	Step and Repeat	Database Query
DB_StepV	Step and Repeat	Database Query

You can use the SmartMarks in the **Step and Repeat** Ticket: Use the **Insert SmartNames...** button, or use the right mouse button shortcut.

The **General** tab:

General | Grids | Rules

Origin:  Unit: mm SmartMarks: <None>

Plate

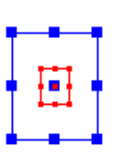
Size:


Shrink: %
 %

Sheet

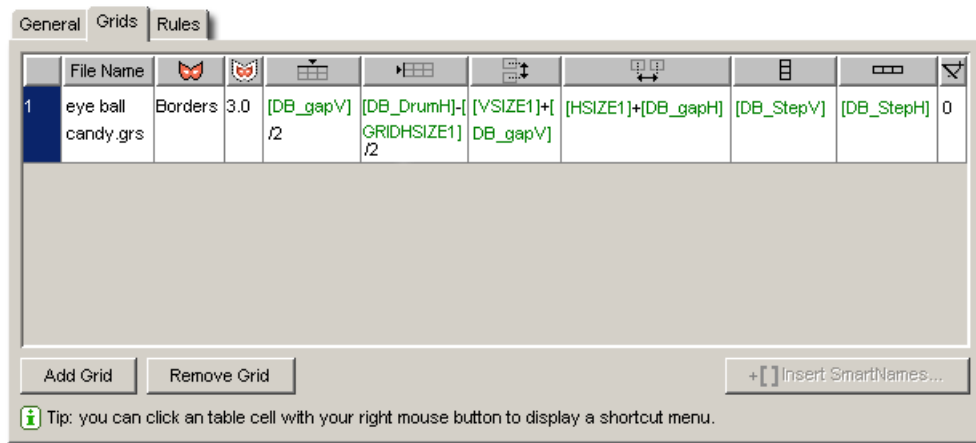
Size:

Offset:

Sheet Position on Plate: 

 Tip: you can click an edit box with your right mouse button to display a shortcut menu.

The **Grids** tab:



The ticket is now defined by using nothing but values queried from the database. Your database controls the output.

2.7.138 Step & Repeat Tabular (Dynamic)

This dynamic version of the **Step & Repeat Tabular** task allows you full interactive control over the grids and positioning, much as if you were doing this in an interactive application (like for example **PowerLayout**, the stand-alone plug-in for Adobe Illustrator).

Use this **Step & Repeat Tabular (Dynamic)** task when you *do not have a template* that was created with PowerLayout (if you do have such a template, then see the [Step & Repeat Template based \(Dynamic\)](#) on page 499).

Clicking on one of the **Task Options** will show you their available settings. Almost all of these are identical to those in the PowerLayout tool: See also [documentation of PowerLayout standalone](#). As usual, many of these options can be set by using **SmartNames**.

On **Job level**, you decide the **Sheet and Plate** options, the **Filling Order** of the items and you set the **Marks**. On **Grid level**, you decide the amount of grids and their own specific filling order.

- **Sheet and Plate** options: See: [Sheet and Plate Options](#) on page 493.
- **Filling Order** options: See: [Filling Order](#) on page 493.
- **Marks** options: See: [Marks](#) on page 496.
- **Grid x** options: See [Setting specific Grid Options](#) on page 496.
- **Grid selection** : Use these buttons to **Add** or **Remove** Grids, .

Note: Grids are numbered sequentially. When three **Grids** are present, and you delete **grid 2**, then **grid 3** will automatically be renamed to **grid 2**.

Note: Preview: The **Preview** displays the position of all grids. The currently active grid will be highlighted in orange. The others will be displayed in blue.

Input

This task needs a **group** of input files for each grid placed in the template. The group of input files defines a *pool* of PDF pages. Pages will be pulled from the pool sequentially to populate the cells of a grid. The way the pages from this pool are positioned in the grid is defined by the filling order.

There are 3 ways to define such a group:

- a single PDF file.

Note: Grid Filling: If a group consists of a single PDF file with only a single page, then a special behavior is triggered, called **Grid Filling**. This means that every cell of the grid will be filled with the specified static one-up print item.

- multiple PDF files, if they follow the **naming convention "filename-number"**. When a number is appended behind the file name using a hyphen, the files with the same base name are grouped together.

For example, banana-001.pdf, banana-002.pdf, banana-003.pdf will be grouped together.

- one XML file for each grid. See the dedicated page describing the [Dynamic Step & Repeat RunList XML Specification](#).

Sheet and Plate Options

Use this tab to specify the **Sheet and Plate** settings of your output file. The **sheet size** will be the area where your one-up grids will be positioned within the **Plate**.

You can define the following:

- **Plate Size:** Define the horizontal and vertical size.

Note: The plate size will correspond to the Media Box of the generated step and repeat PDF.



Caution: Take the printer's size limits into account when defining these values.

- **Shrink:** Set the size options for additional page transformation. Shrink is used to compensate for plate distortion, later in the output process.
- **Sheet Size:** The inner printable area on your plate.

Note: The sheet size will correspond to the Trim Box of the generated step and repeat PDF.

- **Position on Plate:** The blue square in the image represents the position of the sheet on the plate. You can click or drag this blue square in the desired alignment.
- You can adjust by using **Offset** values to fine-tune the position. The values are calculated versus the light grey lines.



Filling Order

Note: This task is also used to manage the layout of variable data.







Note: We here name the one-up print items as 'items'.

Use the **Filling Order** options to specify how the items should be distributed over the grid. You can specify the following:

- a starting point
- whether the grid should be filled in row/column order

- gaps (empty grid stations).

This tab has the following options:

- **Scope settings:** Choose:
 - **Same Settings for all Grids** to apply these settings on this tab to all grids present in the template. Choosing this option will automatically disable setting filling order on grid-level.
 - **Individual Settings for Each Grid** for specifying different settings individually for each grid. Choosing this option will automatically disable further settings here. You now have to set the filling order on grid-level.
- **Start Content filling:** Choose where the first print item is printed. The next options decide how the other items will follow.
 - : Top Left. SmartName text version: TL
 - : Top Right. SmartName text version: TR
 - : Bottom left. SmartName text version: BL
 - : Bottom right. SmartName text version: BR
 - Also specify whether the grid is to be filled by rows  or by columns .

Note: See the preview of your selected options on the right hand side. The **Printing direction** marked with an arrow shows the direction that the paper/substrate flows through the press.

Note: in that preview, a black dot shows the first record. For example:

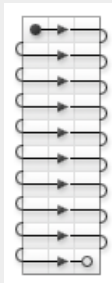
- The following distribution starts on the Top left with vertical flow:



- The following distribution below starts on the top right with vertical flow:



- The following distribution starts on the top left with horizontal flow:



- **Fill over:** choose if you want to distribute the print items over the **Complete Job** or over **Blocks**. Blocks can be used to subdivide a long print job in such a way that rolls of a specified number of items (for example labels) can be delivered to the end customer.
- If you choose **Blocks**, the variable data flow (one-ups flow) will run over blocks of a certain size, then run over the next block when the previous one is complete.
You can see an example of blocks in the preview at right when selecting **Blocks**.



Block Size: Define the size of these blocks. You can express the size in **Job**, **Sheets**, **Rows** or set a SmartName.

- If you have chosen to **Fill over the Complete Job**, then the **Block Size** will be 1 Job by default. You can set it to another unit if you still want to print in blocks, but without having the items (variable data) flow respect the block. This is useful if you want to add gaps (see below).



- **Gap between blocks:** this option is available if you have chosen to **Fill over Blocks**, or to fill over the **Complete Job** but still have blocks (as explained above).

This option adds white space in between the blocks.

Gap size: this is available when you have selected yes for **Gap Between Blocks**.

Marks

Use this tab to add **Dynamic Marks**.


Read more about adding/removing/editing Dynamic Marks in the description of this dedicated task: [Add Dynamic Marks](#) on page 239.

Setting specific Grid Options

- Specify the **Cell Height** and **Cell Width** for the one-up print item (cell).

Tip: Instead of manually entering data, you can use SmartNames to fill this and many other the Grid Options.

- Set the grid **Orientation** and **Clip Bleed**.

The **Initial Orientation** of the one-up can be set using the buttons .

The orientation of the one-up within the Repetition can be changed further by the **Head Turn** settings.

The **Clip Bleed at** allows to create a clipping mask around the one-up, to clip bleed to the desired distance.

Read more about bleed in the manual of PowerLayout Standalone, in [Working with bleed](#).


Note: When **Clip Bleed** is deselected, no clipping will occur. This can cause overlapping of one-ups. This can be useful in some special layouts.

- Specify the **Stagger** direction and **Stagger Options** options :

Staggering allows to shift the 2nd, 3rd, ... row or column down or to the right.



Select the **Direction** in which you want to stagger the one-ups. From left to right, the choices are: No Stagger, Stagger Columns, and Stagger Rows.

Next to the **Stagger Distance** button , you can define the distance for staggering. By clicking the Stagger Distance button you can switch between entering the distance as a fixed value in the current units, or as a percentage of the one-up size.

The stagger will be repeated for a number of rows or columns as defined in **Restart after n columns/rows**. After that, the next one-up will be at the begin position again.

If **Add partial labels** is enabled, the gaps created by staggering will be filled with partial labels (print items) .

See some examples of staggering in the manual of PowerLayout Standalone, in [Staggering Examples](#).

- Specify the **Position** of the layout grid using the Positioning tool and **Offset** distances.

You can choose to **Attach to a Sheet or Plate** as the reference.

The way the Repetition is attached, can be chosen by clicking and dragging the blue square (representing the Repetition) to the desired position on the grey square (representing the **Attach To** reference). Each square has 9 reference points (top left center and right, middle left center and right, and bottom left center and right), so you have 81 different ways of positioning the Repetition.

In the example underneath, the top center of the (bounding box of the) Repetition will be placed on the left bottom corner of the (bounding box) of the reference defined in the 'Attach to' field.



In the offset fields, you can move the Repetition based on the defined position. The direction of the offset (up, down, left or right) depends on the positioning, and is shown by the arrows.

2.7.139 Step & Repeat Template based

This task will make a layout based on an indicated template layout.

The layout will be exactly the same as the template as new design files are typically 'same size' variants of the ones in the layout template.

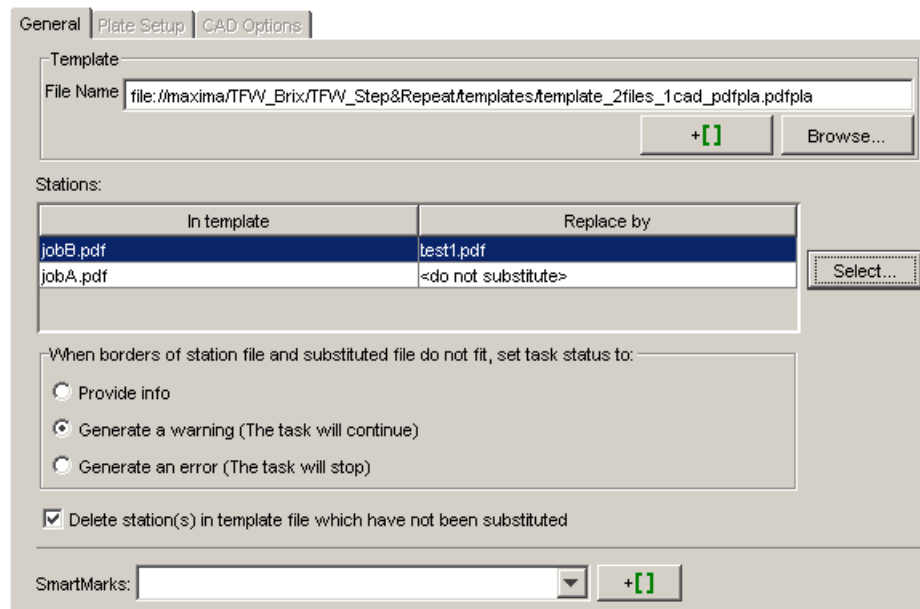
It can be used for all print segments: labels, boxes, flexibles etc..

The template that is indicated in the ticket can be:

- a layout file: PDFPLA, even legacy layout files like PLA or GRQ.
- a Normalized PDF one-up file (that contains external references) - even legacy GRS.
- depending on the version, also specific JDF can be used (containing right 'stripping' info).

The output file will be a PDFPLA file. If you do not specify a name, it will simply receive the name of the file you selected in the Pilot, with a '_rep' attached to it. So if you want to step and repeat **2345banana.pdf** and you do not define an output name, it will be output as **2345banana_rep.pdfpla**.

1. Select a normalized PDF file in the Pilot.
2. Open the 'Step and Repeat - Template based' ticket.
3. Select a Template. Enter a path or click the Browse button.
4. The ticket will present a list of stations (externally referenced files), according to the template you selected.



The Template in the example contained two stations, job A and job B. Job B will be replaced by the file you selected in the Pilot (in this case Test1.grs). The other Station has not been substituted yet. You can do that by clicking the browse button and defining a Normalized PDF file.

Obviously, you cannot step and repeat more Esko native files than there are dummy stations in the Template. But you don't have to substitute all dummy stations. There is an option that automatically deletes the unsubstituted dummy stations in a template.

5. You can add SmartMarks. Use the list to select a Mark.
6. When the borders of a station file and substituted file do not fit, you can define the status of the task.
 - Provide info.
 - Generate a warning. The task will continue.
 - Generate an error. The task will stop.



Attention: If you use a Template which contains legacy QuickStep macros, these macros will be lost. You need to replace these macros with SmartMarks.

2.7.140 Step & Repeat Template based (Dynamic)

Use the **Step & Repeat Template based (Dynamic)** task to place specified one-ups (print items) into grids based on a PDF template.

The template containing the grids is created and exported from Adobe Illustrator using the PowerLayout Standalone plug-in. For more info, see the [manual of PowerLayout standalone](#).

Selecting the template

In **Task Options**, in the **Job** tab, select the desired **Template** file that contains the information about the grids.

Once the template is selected, all other information in this task will be automatically filled in with information from the template. You will be unable to change any of the information in the task, but you can click on any of the Option tabs to see how it has been filled in. For more information on these options, see [Step & Repeat Tabular \(Dynamic\)](#) on page 492.



Attention: Filling order is not applied globally. Therefore, if you specify the filling order for each grid separately, ensure that your number of grids in this ticket match the number of grids in the template file.

Input

This task needs a **group** of input files for each grid placed in the template. The group of input files defines a *pool* of PDF *pages*. Pages will be pulled from the pool sequentially to populate the cells of a grid. The way the pages from this pool are positioned in the grid is defined by the filling order.

There are 3 ways to define such a group:

- a single PDF file.

Note: Grid Filling: If a group consists of a single PDF file with only a single page, then a special behavior is triggered, called **Grid Filling**. This means that every cell of the grid will be filled with the specified static one-up print item.

- multiple PDF files, if they follow the **naming convention "filename-number"**. When a number is appended behind the file name using a hyphen, the files with the same base name are grouped together.

For example, banana-001.pdf, banana-002.pdf, banana-003.pdf will be grouped together.

- one XML file for each grid. See the dedicated page describing the [Dynamic Step & Repeat RunList XML Specification](#).

2.7.141 Submit to Gang Run

For detailed information about the **Submit to Gang Run** task, please see [Submit to Gang Run](#) on page 1109 in the Gang Run Printing chapter.

2.7.142 Submit XML to Gang Run

The **Submit XML to Gang Run** task allows to submit a batch of jobs, described in an XML file, to a **Gang Run Printing** queue. Such an XML is usually exported by the MIS (the business system).

The ticket itself has no interface options. Everything needs to be defined in the input XML file.

Technical summary

The input XML that describes the 'nesting request' needs to be very specific. Read more in [Specification of the XML and using the task](#) on page 500

This task also outputs a 'return XML'. This is specified by the `ReturnURL=` element in the input XML. Once the nesting request has been processed fully, it will then write a `NestingResponse_[date].xml` file on the folder specified by the ReturnURL. This response can always be linked to the original request via the `id=` attribute. This file can be picked up directly by the original submitter/the MIS by monitoring that ReturnURL folder. In this way, the MIS gets a response to its request.

When adding the attribute `production="TRUE"` to the input XML, the return XML will also include references to the actual production files.

The Return XML allows these extra use cases

- Feedback to the business system to help **Cost Calculation** and **Accounting**. When all jobs for a given batch have been nested, Automation Engine can write an XML file back that can be used for cost accounting. Read more in [Feedback XML for Cost Calculation or Accounting](#) on page 502.
- Using the return XML to inform **Production systems/devices**. The XML file generated by Automation Engine can be enriched with references to the actual production files. The XML can then also be used for organizing production. Read more in [Information to Production via XML with File References](#) on page 505.
- The return XML with those references to the production files allows many **workflow extensions**. Those referenced files can be used as input to many other tasks. For example to feed that data to WebCenter or other (web)systems. Read more in [Workflow Extensions](#) on page 506.

Specification of the XML and using the task

Specification of the XML

The Production Order:

Let's take the example where your company needs to do a cost calculation for customer 123456 to produce the following items, before noon at 5th of June of 2014:

- 10 items of AmericanCars_7 on Dibond-4mm, 1100x2400 mm

- 12 items of AmericanCars_2 on Dibond-4mm, 600x1100 mm
- 12 items of AmericanCars_1 on Forex-3mm, any size

The Nesting Request in form of XML:

Your MIS could generate this XML file:

```
<?xml version="1.0" encoding="UTF-8"?>
- <NestingRequest Id="2014-84103">
  <ReturnURL>file://aeserver01/Data1/SnDResponses</ReturnURL>
  <CustomerId>123456</CustomerId>
  <Substrate>Dibond-4mm</Substrate>
  <SubstrateSize>1100x2400</SubstrateSize>
  <Quantity>12</Quantity>
  <DieShape>Cut</DieShape>
  <GroupingKey1>Finishing-Method5</GroupingKey1>
  <GroupingKey4>2014-84103</GroupingKey4>
  <DueDate>2014-06-05T12:00:00+01:00</DueDate>
- <JOBS>
  - <JOB>
    <FileName>file://aeserver01/Data1/Jobs/SnDJob1/sndprepared/AmericanCars_7.pdf</FileName>
    <Quantity>10</Quantity>
  </JOB>
  - <JOB>
    <FileName>file://aeserver01/Data1/Jobs/SnDJob1/sndprepared/AmericanCars_2.pdf</FileName>
    <SubstrateSize>600x1100</SubstrateSize>
  </JOB>
  - <JOB>
    <FileName>file://aeserver01/Data1/Jobs/SnDJob1/sndprepared/AmericanCars_1.pdf</FileName>
    <Substrate>Forex-3mm</Substrate>
  </JOB>
</JOBS>
</NestingRequest>
```

Launching the task

Select the generated XML file and launch the *Submit XML to Gang Run* task. As usual, this task can also be triggered by for example a **Folder Access Point**.

The task has no options. Everything must be specified in the XML file itself.

As a result of running the task, the corresponding jobs enter into the Gang Run Queues.

Nesting calculations are started, with or without taking possible other jobs in the queues into account, all depending on the specified possible grouping keys.

When all jobs for a certain *NestingRequest* have been placed, a *NestingResponse output* XML file will be written on the requested location. This corresponds to the **ReturnURL** element, as specified in the XML file.

Only a file based url is supported at this time.

The *NestingResponse output* XML file can be picked up as soon it becomes available.

The MIS could monitor the folder (*SnDResponses*) on the share(*Data1*) on the computer (*aeserver01*), waiting for a *NestingResponse* file with the valid id (*2014-84103*).

Remarks on the Specification of the XML

- **NestingRequest** element: of course, such an element is required.
- **Id Attribute**: required. It will come back in the *NestingResponse* output XML file.

- **ReturnURL element:** optional. Specifies the folder via a file:-based URL where the *NestingResponse* output XML file will be written once all jobs have been nested.
- **DueDate element:** optional (ISO 8601 format).
- **JOBS element:** required. Contains a list of JOB elements.
- **JOB element:** at least 1 JOB must be present.
- **FileName:** required. A file:-based URL indicating the 1 up item involved.
- **Substrate, SubstrateSize, Quantity elements:** these can be specified under the *NestingRequest* element. However, if these elements occur under the JOB element, they overrule the values appearing under the *NestingRequest* element.
- As can be seen in above example, 1 *NestingRequest* can hold multiple jobs, possibly on multiple substrates, possibly on any, 1 specified or multiple substrate sizes. This allows to keep an order, which can consist of multiple components, as 1 *NestingRequest*.

Some more examples

Using the same graphic on the back but flipped:

```
<?xml version="1.0"?>
- <JOB>
  <FileName>file://AEServer01/Data1/labels/111_kriek belle vue.pdf</FileName>
  <Quantity>3</Quantity>
  <Scale>100</Scale>
  <DieShape>BORDERS</DieShape>
  <Bleed>2.0</Bleed>
  <SameGraphic>1</SameGraphic>
  <FlipGraphic>1</FlipGraphic>
  <Page number="1"/>
</JOB>
```

Using a specific graphic for the back and filling the sheet:

```
<?xml version="1.0"?>
- <JOB>
  <FileName>file://AEServer01/Data1/labels/111_kriek belle vue.pdf</FileName>
  <Quantity>Maximum</Quantity>
  <Scale>100</Scale>
  <DieShape>BORDERS</DieShape>
  <Bleed>2.0</Bleed>
  <BackGraphic>file://AEServer01/Data1/labels/111_kriek belle vue_back.pdf</BackGraphic>
  <Page number="1"/>
</JOB>
```

Feedback XML for Cost Calculation or Accounting

How to get a return XML

The *NestingResponse* output XML file is placed in the folder that was specified in the **ReturnURL** in the *NestingRequest*. The *id* of the *NestingRequest* is taken over in the *NestingResponse*. It becomes available as soon as all jobs of the *NestingRequest* have been placed.

Example of a Return XML

```

<?xml version="1.0" encoding="UTF-8"?>
- <NestingResponse id="2014-84103" Created="2014-05-20T10:01:06.206+02:00">
  - <Sheets>
    - <Sheet id="419801a2-a743-4628-8119-3abaf1418154" Created="2014-05-20T10:01:00.360+02:00">
      <Substrate>Forex-3mm</Substrate>
      <SubstrateSize>900x900</SubstrateSize>
      - <Jobs>
        - <Job id="adf105b0-ce2d-4fbc-a96e-ae1efa56558e">
          <Quantity>12</Quantity>
          <UsedSheets>6.0</UsedSheets>
          <UsedSurface unit="m2">4.314980209851265</UsedSurface>
          <TotalSurface unit="m2">4.859999999999999</TotalSurface>
          <Waste unit="m2">0.5450197901487349</Waste>
        </Job>
      </Jobs>
    </Sheet>
    - <Sheet id="b802d6f3-31a6-466e-88f3-95e101ae5704" Created="2014-05-20T10:01:06.380+02:00">
      <Substrate>Dibond-4mm</Substrate>
      <SubstrateSize>600x1100</SubstrateSize>
      - <Jobs>
        - <Job id="b6258add-e501-464c-bb85-398e92d88fe6">
          <Quantity>12</Quantity>
          <UsedSheets>12.0</UsedSheets>
          <UsedSurface unit="m2">5.350619541508913</UsedSurface>
          <TotalSurface unit="m2">7.92</TotalSurface>
          <Waste unit="m2">2.569380458491087</Waste>
        </Job>
      </Jobs>
    </Sheet>
    - <Sheet id="f36b931f-4064-4aad-8bed-4b6640332a7e" Created="2014-05-20T10:01:02.166+02:00">
      <Substrate>Dibond-4mm</Substrate>
      <SubstrateSize>1100x2400</SubstrateSize>
      - <Jobs>
        - <Job id="2d05b634-e4a4-4906-97d9-6cc6759d505c">
          <Quantity>10</Quantity>
          <UsedSheets>2.0</UsedSheets>
          <UsedSurface unit="m2">4.458849617924094</UsedSurface>
          <TotalSurface unit="m2">5.279999999999999</TotalSurface>
          <Waste unit="m2">0.8211503820759057</Waste>
        </Job>
      </Jobs>
    </Sheet>
  </Sheets>
  - <Jobs SubmitDate="2014-05-20T10:00:55.654+02:00">
    - <Job id="adf105b0-ce2d-4fbc-a96e-ae1efa56558e">
      <FileName>file://aeserver01/Data1/Jobs/SnDJob1/AmericanCars_1.pdf</FileName>
      <Page>1</Page>
      <Quantity>12</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-06-05T12:00:02.626+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey1>Finishing-Method5</GroupingKey1>
      <GroupingKey4>2014-84103</GroupingKey4>
    </Job>
    - <Job id="b6258add-e501-464c-bb85-398e92d88fe6">
      <FileName>file://aeserver01/Data1/Jobs/SnDJob1/AmericanCars_2.pdf</FileName>
      <Page>1</Page>
      <Quantity>12</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-06-05T12:00:02.626+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey1>Finishing-Method5</GroupingKey1>
      <GroupingKey4>2014-84103</GroupingKey4>
    </Job>
    - <Job id="2d05b634-e4a4-4906-97d9-6cc6759d505c">
      <FileName>file://aeserver01/Data1/Jobs/SnDJob1/AmericanCars_7.pdf</FileName>
      <Page>1</Page>
      <Quantity>10</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-06-05T12:00:02.626+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey1>Finishing-Method5</GroupingKey1>
      <GroupingKey4>2014-84103</GroupingKey4>
    </Job>
  </Jobs>
</NestingResponse>

```

Feedback to Production via Folder Organisation

Concept

The *NestingResponse* example as shown in [Feedback XML for Cost Calculation or Accounting](#) on page 502 holds 2 parts: Sheets and Jobs.

- The **Sheets** element describes which (part of) sheets were used to nest the given files. It refers to the files by a `<Job id="">` element.
- The **Jobs** element lists all jobs that had to be nested.

In that example, you see that 3 sheets were generated:

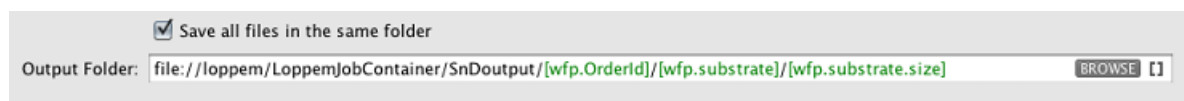
- the first sheet holds 2 items of the AmericanCars_1, and must be produced 6 times (Forex-3mm 900x900)
- the second sheet holds 1 item of the AmericanCars2, and must be produced 12 times. (Dibond-4mm 600x1100)
- the third sheet holds 5 items of the AmericanCars_7, and must be produced twice. (Dibond-4mm 1100x2400)

Contrary to the next chapter [Information to Production via XML with File References](#) on page 505, the actual files for production (print/cut/...) are not mentioned in this *NestingResponse* output XML file. In fact, at the moment this *NestingResponse* report XML file is written, these files must still be generated, by the workflow that is linked to the substrate in the Gang Run Queues.

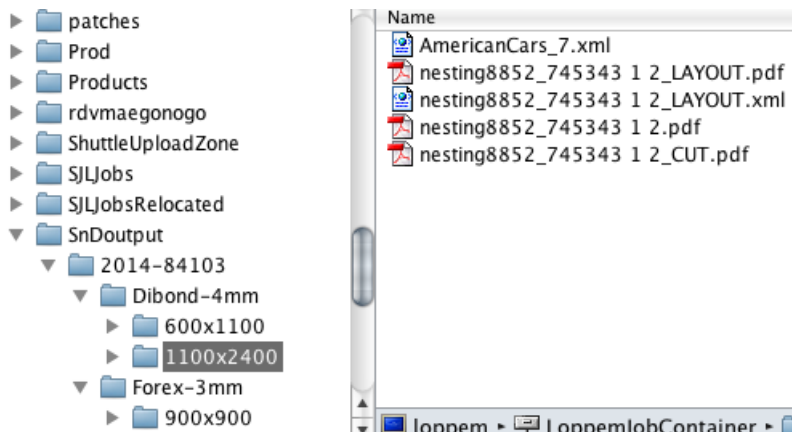
Of course, there are ways to link this *NestingRequest* to the actual production files. To achieve this, it is useful to know that certain **workflow parameters** are set in the workflow that is launched on the generated Nested Layout files: *substrate*, *substrate.size*, *substrate.quantity*, *CustomerId* and all the *GroupingKey*'s that are set up as grouping key.

Example

- In Pilot-Configure, set up *GroupingKey4* as grouping key, under the name *OrderId*.
- For the workflow that the Gang Run launches on the generated nested layouts, set up the **Output Folder** in the **Production Files** tab of the **Export Nested Layouts** ticket as follows:



As a result, the production files are organized in the following folder structure, in which it is easy to find the appropriate files related to the initial *NestingRequest*:



Information to Production via XML with File References

How much material is needed to produce the order?

This information is part of the *NestingResponse* output XML file. In our example:

- 6 sheets of Forex-3mm 900x900 (AmericanCars_1), waste 0,54 m2 (11%)
- 12 sheets of Dibond-4mm 600x1100 (AmericanCars_2), waste 2,56 m2 (32%)
- 2 sheets of Dibond-4mm 1100x2400 (AmericanCars_7), waste 0,82 m2 (16%)

Extending the XML with references to the production files

The *NestingResponse* output XML can also be extended to contain references to the actual production files. To do this, add the attribute **production="TRUE"** to the *ReturnURL* element. See an example:

```
<?xml version="1.0"?>
- <NestingRequest Id="21185">
  <ReturnURL production="TRUE">file://AEServer01/Data1/SnDResponses</ReturnURL>
  ...
</NestingRequest>
```

As a result, each */NestingResponse/Sheets/Sheet* element will be extended with a *FileList* element.

For each production file, there will be a *File* element attached to the *FileList* element.

Note: What is considered to be a 'production file'? Production files are the output files of the workflow that is launched/configured per **Substrate** in the **Gang Run Printing** View.

Example

A *NestingResponse* Output XML with references to production files:

```

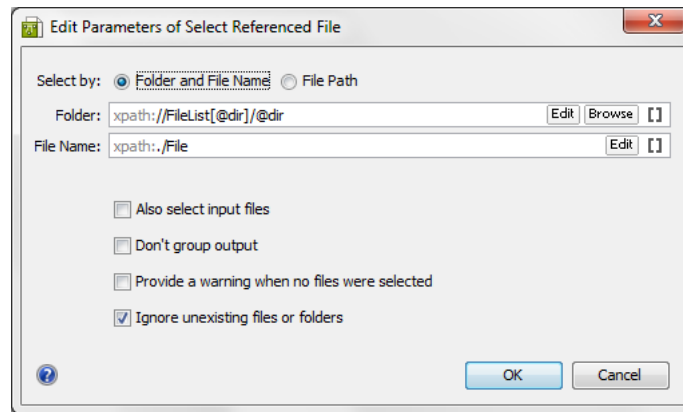
<?xml version="1.0" encoding="UTF-8"?>
- <NestingResponse id="21418" ReturnURL="file://aeserver01/Data1/SnDResponses" Created="2014-05-27T11:45:21.191+02:00">
  - <Sheets>
    - <Sheet id="2fdb7110-98a5-4ab5-bc64-676d16aa6a3d" Created="2014-05-27T11:45:21.160+02:00" WorkflowId="035db4f8-a7fe-4c36-b2ec-66d">
      <Substrate>Dibond-4mm</Substrate>
      <SubstrateSize>600x1100</SubstrateSize>
      - <Jobs>
        - <Job id="86301612-5c10-4b0e-8aa0-489a55237d83">
          <Quantity>12</Quantity>
          <UsedSheets>12.0</UsedSheets>
          <UsedSurface unit="m2">5.350619541508913</UsedSurface>
          <TotalSurface unit="m2">7.92</TotalSurface>
          <Waste unit="m2">2.569380458491087</Waste>
        </Job>
      </Jobs>
      - <FileList Created="2014-05-27T11:45:39.563+02:00">
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/600x1100/nesting17624_265965%201%2012.pdf</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/600x1100/nesting17624_265965%201%2012_LAYOUT.xml</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/600x1100/nesting17624_265965%201%2012_LAYOUT.pdf</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/600x1100/AmericanCars_2.xml</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/600x1100/nesting17624_265965%201%2012_CUT.pdf</File>
      </FileList>
    </Sheet>
    - <Sheet id="831fdb7f-fdd-4289-a5ba-866ffb153c59" Created="2014-05-27T11:45:21.294+02:00" WorkflowId="728ca7ee-c071-482d-8a98-cbd">
      <Substrate>Dibond-4mm</Substrate>
      <SubstrateSize>900x1100</SubstrateSize>
      - <Jobs>
        - <Job id="f11e2e1c-4d35-4ef8-b357-c99d8ca3b6f5">
          <Quantity>12</Quantity>
          <UsedSheets>6.0492</UsedSheets>
          <UsedSurface unit="m2">4.9683519386757915</UsedSurface>
          <TotalSurface unit="m2">5.989269327278993</TotalSurface>
          <Waste unit="m2">1.020917388603202</Waste>
        </Job>
        - <Job id="43e38513-391d-4388-b911-2eb2abb74ad1">
          <Quantity>12</Quantity>
          <UsedSheets>5.9508</UsedSheets>
          <UsedSurface unit="m2">4.8866099617074426</UsedSurface>
          <TotalSurface unit="m2">5.8907306727210065</TotalSurface>
          <Waste unit="m2">1.004120711013564</Waste>
        </Job>
      </Jobs>
      - <FileList Created="2014-05-27T11:45:39.563+02:00">
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/nesting17624_265965%202%2012.pdf</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/nesting17624_265965%202%2012_LAYOUT.xml</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/nesting17624_265965%202%2012_LAYOUT.pdf</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/AmericanCars_3.xml</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/AmericanCars_4.xml</File>
        <File>file://aeserver01/Data1/SnDoutput/21418/Dibond-4mm/900x1100/nesting17624_265965%202%2012_CUT.pdf</File>
      </FileList>
    </Sheet>
  </Sheets>
  - <Jobs SubmitDate="2014-05-27T11:45:13.094+02:00">
    - <Job id="86301612-5c10-4b0e-8aa0-489a55237d83">
      <FileName>file://aeserver01/Data1/SnDFiles/AmericanCars_2.pdf</FileName>
      <Page>1</Page>
      <Quantity>12</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-05-27T11:45:21.206+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey4>21418</GroupingKey4>
    </Job>
    - <Job id="f11e2e1c-4d35-4ef8-b357-c99d8ca3b6f5">
      <FileName>file://aeserver01/Data1/SnDFiles/AmericanCars_4.pdf</FileName>
      <Page>1</Page>
      <Quantity>12</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-05-27T11:45:21.206+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey4>21418</GroupingKey4>
    </Job>
    - <Job id="43e38513-391d-4388-b911-2eb2abb74ad1">
      <FileName>file://aeserver01/Data1/SnDFiles/AmericanCars_3.pdf</FileName>
      <Page>1</Page>
      <Quantity>12</Quantity>
      <Scale>100.0</Scale>
      <DueDate>2014-05-27T11:45:21.206+02:00</DueDate>
      <CustomerId>123456</CustomerId>
      <GroupingKey4>21418</GroupingKey4>
    </Job>
  </Jobs>
</NestingResponse>

```

Workflow Extensions

Concept

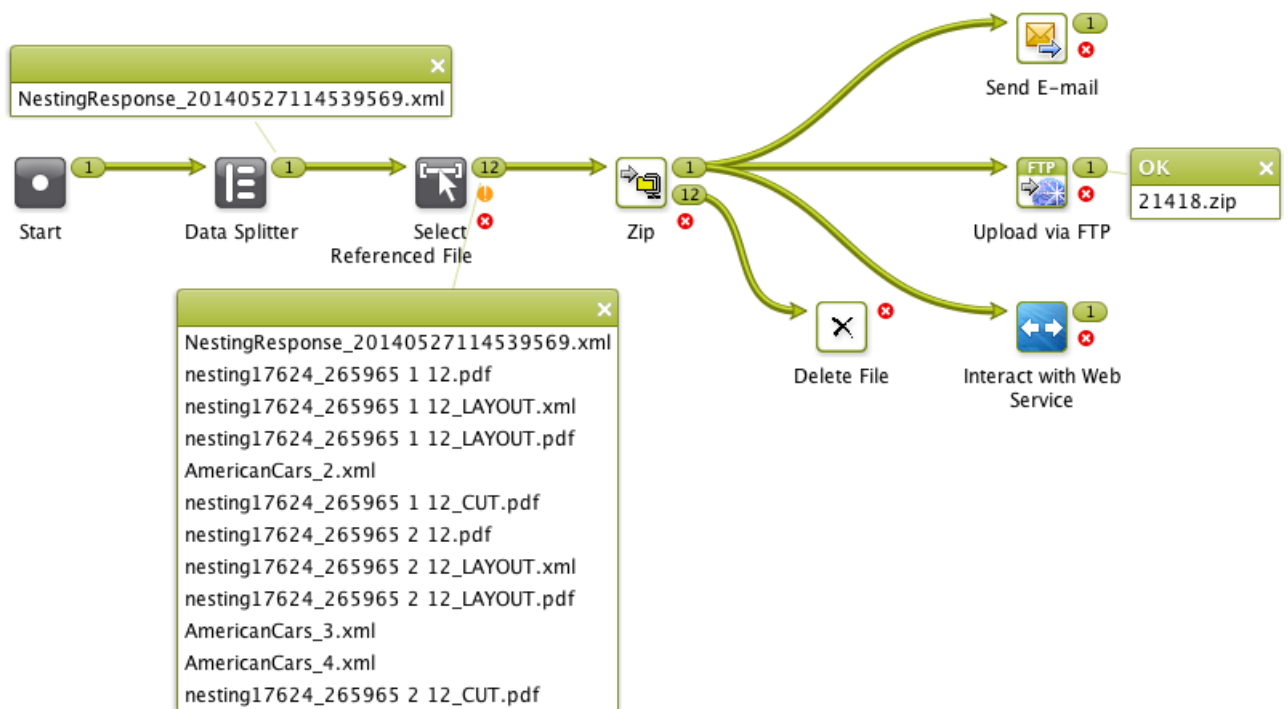
The production files in the *NestingResponse output* XML can be picked up in an Automation Engine Workflow via the workflow control **Select Referenced File**.



Read more about this workflow control in [Select Referenced File](#) on page 1018.

Example workflow

This enables a lot of extensions to the workflow. The selected files can for example be ZIP'ed and sent out to external systems or partners. Some users then also continue by deleting these files locally.



2.7.143 Track Edit Session

Automation Engine can track and log interactive editing actions. Decide in Configure if and for which editors you want automatic tracking to happen.

When you open a file that resides in an active Job (folder) in an Esko editor, the Track Edit Session task will automatically start logging the interactive editing actions. This tracking will stop when the file is closed.

As this tracking is actually an Automation Engine task, this also enables:

- Notifications (for example XML for reporting to other systems).
- Automatic storage in the Task History.

File Name	Task Type	Progress	Phase	State	Launched
x-9007_Flower.pdf	Track Edit Session	0%	Editing with PackEdge 6.0...		11/28/06 11:26 AM
x-9007_Flower_CS2....	Track Edit Session	100%			11/17/06 3:42 PM

Note: This task cannot be launched interactively. The Track Edit Session task will appear in the Task View if the task has been configured in the [Configure Track Edit Session panel](#). Keep in mind that the editor needs to be restarted to activate the settings.

Note: Track Edit Session also works when editing your file with ArtPro, but in this case it will only record session times, not editing details.

Viewing the Edit Session Information in the Pilot

In the Pilot, the Tasks View offers more information about the Track Edit Task.

File Name	Task Type	Progress	Phase	State	Launched
FCI_Capabilities_Brochure.grs	Track Edit Session	100%			11/28/06 11:26 AM

Details	
Progress: <input type="text"/>	*** Start interactive PackEdge 6.0 ***
Current Phase:	Time 11/28/06 11:26 AM
Started on Server: RDALPHA16	User Antonio@rdalpha16
Started at: 11/28/06 11:26 AM	BackStage server RDALPHA16
Finished at: 11/28/06 11:40 AM	Loading file file://maxima/abot_container/FCI_Capabilities/FCI_Capabilities_Brochure.pdf
Process Time: 0:14:17	PS-Fix: modified 146 contour(s)
	Trap with local ticket Default
	Retrap
	Created barcode EAN 13 with code 1234567890128
	Saving file file://maxima/abot_container/FCI_Capabilities/FCI_Capabilities_Brochure.pdf
	*** End interactive PackEdge ***
	*** End interactive PackEdge 6.0 ***
	Time 11/28/06 11:40 AM

The logging of the editing actions will appear:

- In the Pilot's message window of that task.
- In the task's log files.

- In notifications (TXT or XML or E-mail format). This can happen by setting up a general notification rule for all Track Edit Session tasks.
 - It is possible to save all notifications locally in a job folder or centrally in one or several files.
 - It is possible to name TXT / XML logs via SmartNames. Typically this is the name of edited file.

How to Store the Editing Information System-Wide?

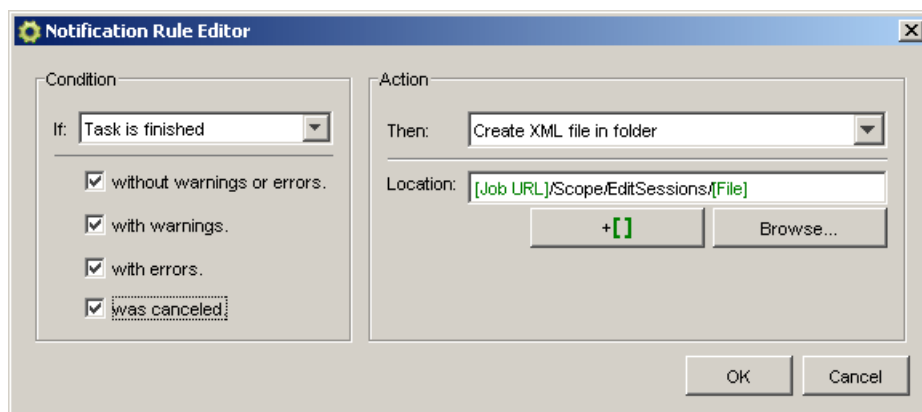
Many users want to store all this editing information in a system-wide way. This can be done easily through the standard Notifications tool (see [Defining a Notification Rule](#) on page 667).

The Track Edit session task starts automatically, so unlike other task types, you will not create your own Track Edit Session ticket to add the notification in there. In this case, all you need to do is setting a general notification rule for this task type. This is done in the Configure panel.

How can this edit log be stored automatically?

- Either you save all edit logs inside the job folder. You will then have to indicate a path for the XML/TXT under the [JOB URL] and decide if you want 1 (XML/TXT) log per file or per date or if you want all logs in 1 file.
- Or you save all edit logs per customer or even system-wide. You will then have indicate a general path to a higher level 'above' all Jobs.

1. Go to **Configure > Notification > Rules > .**
2. Select the Track Edit Session task in the **Tasks** tab.
3. Click **Add**. The **Notification Rule Editor** opens.
4. For an XML file, select the **Create XML File in folder** as **Action**.
5. Define the folder's **Location** and click **OK**.



Which Edit Actions are Logged?

Among the items logged are:

- Which user opens which file from which workstation?
- Start and stop timings of this editing session
- Was the file saved? When and with what name?
- Loading, saving, placing, exporting functions
- All changes to inks
- All changes to (Esko) bar codes objects: creating, modifying, type change
- All changes to layers

- All changes to pages
- Starting, stopping a session. Loading, saving, placing, exporting files
- Changes of page size, of margins, of reader's spread
- Losing the connection
- HIFI conversions
- Ink mapping
- Trap edits
- External references: creating, replacing
- Layers edits: creating, renaming
- PSFix: Process, Remove Spurious Contours, Remove Double Contours, ...
- InkWizard: destination profile, CMYK profile, ...
- Die object edits: adding, turning, flipping, rotating

This is an example of what information can be available in a Track Edit Session log.

```

*** Start interactive PackEdge 7.0 ***
    Time 13/01/06 17:19
    User wde@backstage1
    Automation Engine server BACKSTAGE1
        Loading file file://backstage1/wdedata/52355/
Scope/52355_Leffe_back.pdf
    Remove layer OMBRE
    Trap with local ticket Default
    "Modified ink angle for C from 22 to 7 degrees"
    "Created ink PANTONE 123 C (100 lpi, 7 degrees, dot ?, type Normal)"
    "Created ink C (170 lpi, 75 degrees, dot ?, type Normal)"
    "Created ink M (170 lpi, 15 degrees, dot ?, type Normal)"
    "Created ink Y (170 lpi, 90 degrees, dot ?, type Normal)"
    "Created ink ROUGE (170 lpi, 45 degrees, dot ?, type Normal)"
    "Created ink BISTRE (170 lpi, 45 degrees, dot ?, type Normal)"
    "Created ink ILLOGLOSS (170 lpi, 45 degrees, dot ?, type Opaque)"
    "Created ink K (170 lpi, 45 degrees, dot ?, type Normal)"
    "Created ink EP (170 lpi, 45 degrees, dot ?, type Technical)"
    "Created ink roc (170 lpi, 45 degrees, dot ?, type Technical)"
        Placing file file://backstage1/wdedata/52355/
Scope/52355_Leffe_collar.pdf
    Changed page margins to top 3.998701, bottom 9.998703, left 4.001198,
right 66.0012 mm
    Changed page size to width 110.0024 mm and height 73.99741 mm
    PS-Fix: modified 2 contour(s)
    Applied Job inkmapping
    "Modified ink angle for C from 7 to 75 degrees"
    "Modified ink angle for M from 37 to 15 degrees"
    "Modified ink angle for Y from 82 to 90 degrees"
    "Removed ink C (170 lpi, 75 degrees, dot ?, type Normal)"
    "Removed ink M (170 lpi, 15 degrees, dot ?, type Normal)"
    "Removed ink Y (170 lpi, 90 degrees, dot ?, type Normal)"
    Changed page margins to top 7.25, bottom 7.25, left 7.25, right
8.361092 mm
        Exporting file file://backstage1/wdedata/52355/
Scope/52355_Leffe_back.tif
        Saving file file://backstage1/wdedata/52355/
Scope/52355_Leffe_back2.pdf
    *** End interactive PackEdge 7.0 ***
    Time 13/01/07 17:22
    Notification:Wrote XML file:E:\wdedata\52355\Scope\EditSessions
\52355_Leffe_back.

```

2.7.144 Trap with PowerTrapper

The advantage of using a PowerTrapper is that you do not have to prepare the PDF file or create the trap layer in advance. In addition, you can trap the input file without an intermediate file. The output of this task is a Normalized PDF file.

You can choose your trapping settings in the **Distance & Direction**, **Color & Shape**, **Processing**, **Rules** and **Output** tabs.

Distance & Direction

You can set the trapping distance and direction at the tab **Distance & Directions**.

Settings for Step: Trap with PowerTrapper

Output in: Browse []

File Name: Browse []

Distance & Direction | Color & Shape | Processing | Rules | Output

Trapping: Normal Trapping 1

Trapping Distance: 0.200 mm [] 2

Minimum Ink Difference: 10.00% 3

Image Trapping: Automatic 4

Pull Back: Automatic 5

Trapping Distance: Into black 0.200 mm []

Into spot 0.200 mm []

Into image 0.200 mm [] 2

Pull back 0.200 mm []

Image Trapping: Trap images to images 4

Pull Back: Also pull back light inks 5

Pull back images and gradients

Trap Inks Sequence: Treat all spot colors as opaque inks 6

Overshoot Mode: Automatic 7

1. Trapping

You can choose between two trapping modes:

- **Normal Trapping:** This selects adjoining color pairs, which are likely to cause registration problems like ugly light gaps, and unwanted halo effects. Lighter colors are generally trapped into darker colors to minimize the visual effect of the trap.
- **Reverse Trapping:** This prevents overprint of adjoining color areas by trapping these color pairs with a white knockout (erasing) trap. The white knockout trap is put on top of the lighter of the two colors. This is useful to prepare a job for Dry Offset printing (example: metal beverage cans). This avoids overprinting of inks.

2. Trapping Distance

Enter the width you want to give your traps. The unit used is the General unit defined in Illustrator's Unit & Display Performance Preferences. The default trap width is **0.2 mm** (or equivalent in your chosen unit).

You can choose more trapping distance specifications such as: **into black**, **into spot**, **into image** and **into pullback** further down in this tab.

3. Minimum Ink Difference

Trapping is applied to two adjoining colors when either:

- The two colors each consist of at least one ink but they have no common inks.
- The two colors have common inks but there is a difference of at least the **Minimum Ink Difference** percentage for two or more inks making up these colors.

These differences in ink percentages must be in opposite directions.

For example, with the default minimum ink difference percentage of 10%, the following two colors are trapped:

1st color: C 5% M 10% Y 50% K 15% (+10% K)

2nd color: C 5% M 10% Y 85% K 5% (+35% Y)

Increase the **Minimum Ink Difference** to trap colors that differ more, lower it to trap colors that are similar.

4. Image Trapping

- **Automatic** (default): this determines the most suitable trap direction automatically, by comparing the average lightness of the CT image with the lightness of the adjoining area, then traps the lighter color into the darker one.
- **Always Towards Images**: this always traps line art under adjoining CT images.
- **Always Towards Line Art**: this always traps CT images under adjoining line art.

If you want images to be trapped with each other, enable **Trap images to images**.

5. Pull Back

You can use ink pull back to keep away all but the darkest ink of a color area composed of multiple inks bordering to a very light background. It prevents individual inks from the composed color to become visible on the background due to registration errors during the printing process.

- **Automatic** (default): this creates pull backs on rich black or rich spot colors, when the color left in the pull back area is not very different from the composed color.
- **Only Pull Back Rich Black**: this only creates pull backs on rich black.
- **Do Not Pull Back**: this never creates any pull back.

There will be automatic pull back for the darkest color. You can enable **Also pull back light inks** option to create pull backs even when the pullback ink (ink remaining in the pull back area) is visibly different from the composed color. Enable **Pull back images and gradients** to create pull backs on images and gradients.

6. Treat all spot colors as opaque inks

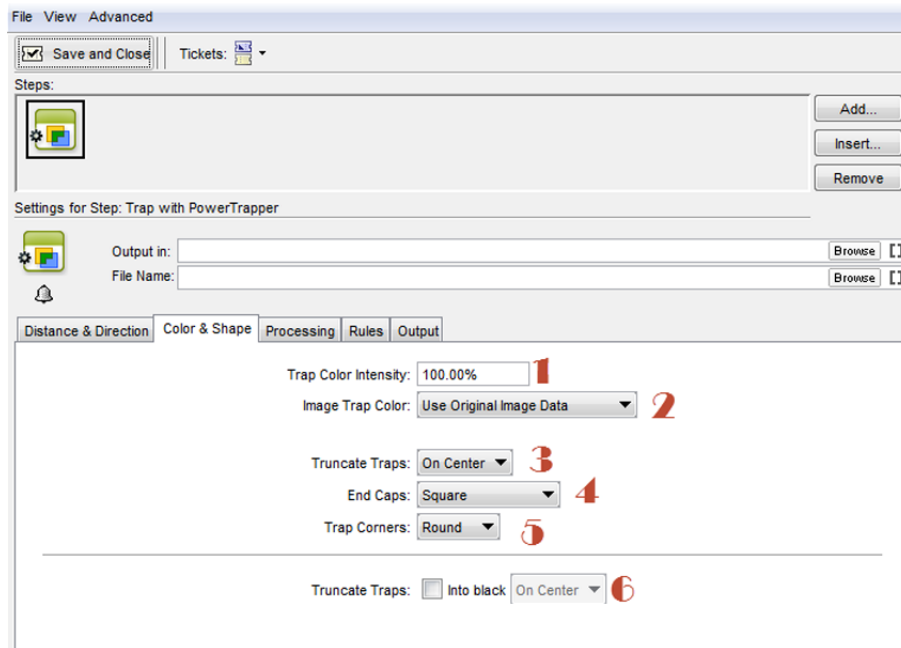
You can enable this option to set the trapping direction by ink sequence. PowerTrapper uses the color luminance to define the trapping direction by default except for opaque inks. PowerTrapper uses ink sequence to determine trapping direction in the case of opaque inks. If the **Treat all spot colors as opaque inks** option is enabled, color luminance is used only for CMYK. For opaque and spot inks, the trap direction is determined by the ink sequence.

7. Overshoot Mode

You can choose between the following three overshoot modes:

- **Automatic** (default): The centerline behavior is according the rules determined in the color pair list.
- **Never on Dark Areas**: The centerline is forced not to get beyond the center of dark areas.
- **Never**: The centerline is forced not to get beyond the center of any area.

Color & Shapes



1. Trap Color Intensity

By default, **PowerTrapper** uses the full color of the object to spread in the trap (100% trap color intensity).

You can create a trap of a lighter color by reducing the **trap color intensity** percentage.

2. Image Trap Color

Choose how to trap your images:

- **Use Original Image Data** (default): When spreading an image, **PowerTrapper** uses the underlying image data (when the image is clipped) to create the trap.

Note: An empty trap is created, when there is no underlying image data (when the image is not clipped).

- **Extend Image Data:** **PowerTrapper** automatically extrapolates the image along its top, right, bottom and left edges to ensure that the trap looks as expected even if there is no clipped image data outside the visible image.

- **Use an Approximate Flat Color:** Instead of using existing or extrapolated image data in the trap, **PowerTrapper** uses a solid color. A suitable color is determined by averaging the image's colors along its boundary with the object to trap.

3. Truncate Traps

PowerTrapper can truncate the trap so that it does not stick out on another color. There are two truncation modes:

- **On Center** (default): when a portion of a trap comes too close to the contour of another object, **PowerTrapper** limits it to half the distance between the trapped object and the other object.
- **On Edge:** when a portion of a trap comes too close to the contour of another object, it is truncated on the other object's edge.

4. End Caps

This option specifies how to shape the ends of an open trap.

- **Square** (default): this option ends the trap at right angle to the adjoining object.
- **Round:** this rounds the ends of the trap. This option is typically only used in combination with white knockout (reverse trapping).

Note: We recommend that you do not to use **Round End Caps** together with **Truncate Traps On Center**, as this can generate some artefacts (the round caps are truncated).

- **Object Dependent:** with this option, the trap is a logical continuation of the contour of the spread object.

5. Trap Corners

This option allows you to change how the traps' sharp corners are handled.

- **Round** (default): a round cap will be placed at all corners.
- **Beveled:** this cuts sharp corners off.
- **Mitered:** this option works with a miter ratio.

The miter ratio serves to limit the length of the sharp corner (the distance from the base of the trap to the corner point).

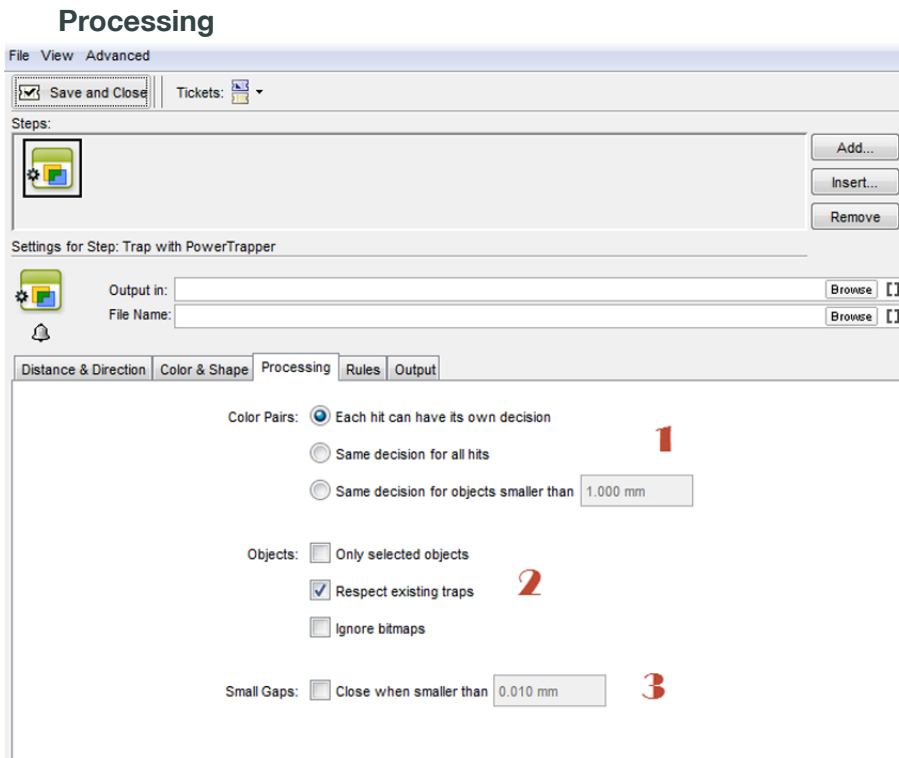
The default miter ratio value is 4. This means that if the length of the sharp corner is more than 4 times the **Trapping Distance**, then the corner are cut off (beveled). If it is less than 4 times the **Trapping Distance**, the corner are left as it is.

6. Advanced Settings

Click the collapsible button to show the advanced settings.

Truncate Traps

If you want to truncate traps into black differently, select **Into black** and choose the truncation mode (**On Center** or **On Edge**).



1. Color pairs

PowerTrapper can trap all hits (occurrences) of a color pair the same way or recalculate the trap direction for different hits.

- Each hit can have its own decision (default): this recalculates the trap direction for each hit of a color pair.
- Use the same decision for all hits: this traps all hits of the same color pair in the same direction.
- Use the same decision for objects smaller than: this uses the same trap direction for hits of a color pair belonging to small objects, and recalculates the trap direction for hits belonging to bigger objects.

Use the field next to the option to determine the maximum size of small objects. The unit used is the **General** unit defined in Illustrator's **Unit & Display Performance Preferences**.

The default size is 1 mm (or equivalent in your chosen unit).

2. Objects

In some cases **PowerTrapper** can safely ignore certain object types to speed up the trapping.

- **Only selected objects**
- **Respect existing traps:** if your document has already been trapped, select this option to keep the existing trap layer out of the trapping process.
- **Ignore bitmaps:** this keeps bitmap pixel data from being trapped. Bitmap data is contourized before trapping which may slow down trapping depending on the bitmap's contents. Bitmaps with lots of small isolated islands typically take a longer time to trap.

Note: When you indicate an object type to ignore, it is ignored completely as if it did not exist in the job. Also, the ignored objects does not truncate trapping areas resulting from other objects below them.

3. Small Gaps

Some input jobs contain very small unintentional gaps between neighboring objects, preventing the correct trapping of these objects. Although it is better to clean up such jobs before trapping, **PowerTrapper** can ignore these gaps automatically.

To use this option, select **Close when smaller than** and enter the maximum size small gaps can have.

The unit used is the **General** unit defined in Illustrator's **Unit & Display Performance Preferences**.

The default size is 0.01 mm (or equivalent in your chosen unit).

Note: If you set a big gap size, trapping may become slower and small parts of the job may be ignored during trapping.

Rules

You can define custom trapping settings for specific trapping pairs using the **Rules** tab.

1. When to use rules?

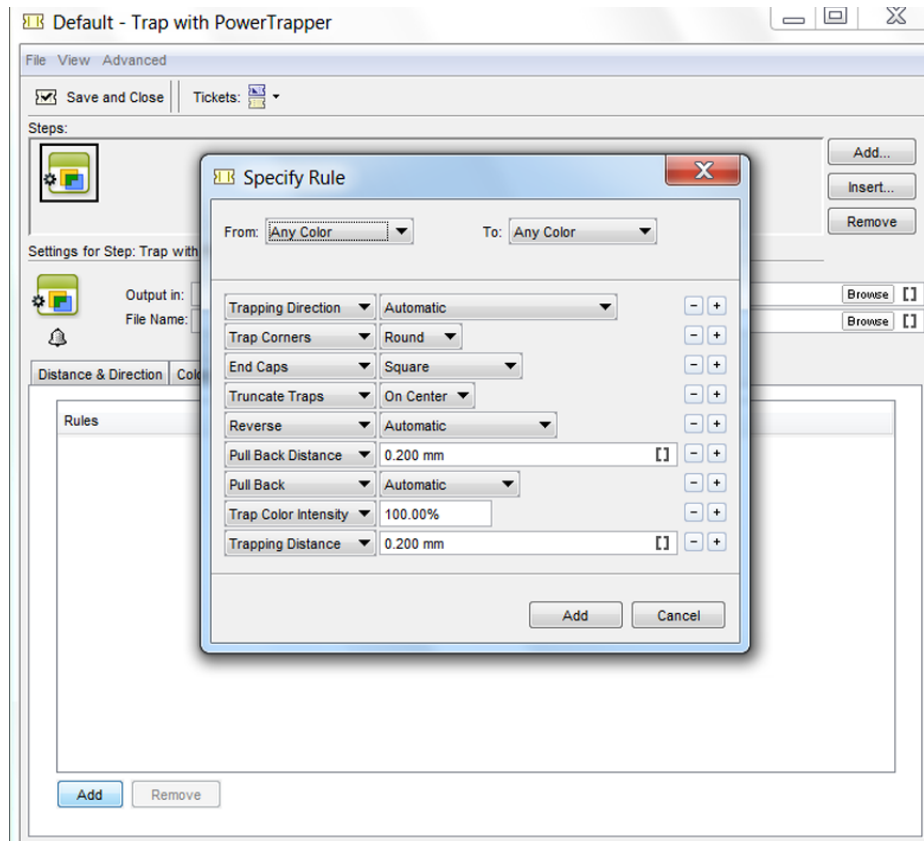
In most cases the general trapping settings entered in the **Distance & Direction** gives you excellent results. You can refine them using rules. If you add a rule that is not coherent with your inputs in the other tabs, PowerTrapper will overwrite them with the rules

Note: This is an advanced feature and we recommend it for experienced users.

You can specify exceptions to the general trapping settings for certain color pairs using **Add Rule**. Some of the instances where you can use it are :

- to set a different trap distance for a specific spot ink. Sometimes you need a larger trap distance for spot inks if there is no other ink to help cover leaks on the press.
- to prevent trapping or ink pull back in specific cases. Depending on the type of the adjoining colors, trapping and / or pull back may be undesirable.
- to force a specific trap direction. PowerTrapper Standalone determines the trap direction based on the relative lightness of the adjoining colors. If the colors have a similar lightness level, it selects an arbitrary direction. You can use a rule to make it always use a particular direction for a color pair.
- to specify a different shape and/ or truncation mode for certain color pairs. For example, you can choose to use "On Center" truncation when trapping under a 100% black line and "On Edge" truncation on all other color pairs.

2. Creating a rule



- Click the **Add Rule** button at the bottom of the **Rules** tab.
- In the pop-up that opens, choose the trapping pair to apply the rule to using the **From** and **To** lists. You can create a rule to trap from/to: any color, a particular ink (process, spot or opaque), a particular object (image, gradient or empty background) or the registration color.

Note: When trapping a particular ink, you can choose the ink name from which **Minimum Density** the rule is applied and if the color must be pure (no other separation in the color object) to apply the rule.

- Choose the contents of the rule.

Options	Description
Trapping Direction	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Standalone calculate the trapping direction. • Force Trapping from 'From' to 'To' always spreads the "From" color/object on the "To" color/object. • Do Not Trap 'From' to 'To' never traps the "From" and "To" colors/objects together.

Options	Description
Trapping Distance	Use this to define a special trapping distance for the trapping pair.
Trap Color Intensity	Use this to define a special trap color intensity for the trapping pair.
Pull Back	<ul style="list-style-type: none"> • Automatic lets PowerTrapper decide if there should be a pull back on the trapping pair. • Always Pull Back always creates a pull back on the trapping pair. The darkest ink of the pair is the pullback ink (remaining ink in the pull back area). • Do Not Pull Back never creates a pull back on the trapping pair.
Pull Back Distance	Use this to define a special pull back distance for the trapping pair.
Reverse	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Standalone decide if the trapping pair should have a reversed trap. • Always Trap Reversed always generates a reversed trap for the trapping pair. • Do Not Trap Reversed never generates a reversed trap for the trapping pair.
Truncate Traps	Choose if traps generated for the trapping pair should be truncated on center or on edge.
End Caps	Choose how to shape the ends of open traps for the trapping pair (square, round or object dependent).
Trap Corners	Choose how to shape the trapping pair's trap corners (round, beveled or mitered).

- If necessary, use the + button to add contents to your rule. You can use the - button to remove contents.
- After defining the rules, click **OK** button. You can now see your rule at the **Rules** tab.

When trapping your document, PowerTrapper traps the trapping pairs that match rules according to those rules and the rest of your document according to the main trapping settings. If necessary, you can add another rule or remove a rule previously created. You can also edit a rule by double-clicking it in the **Rules** tab.

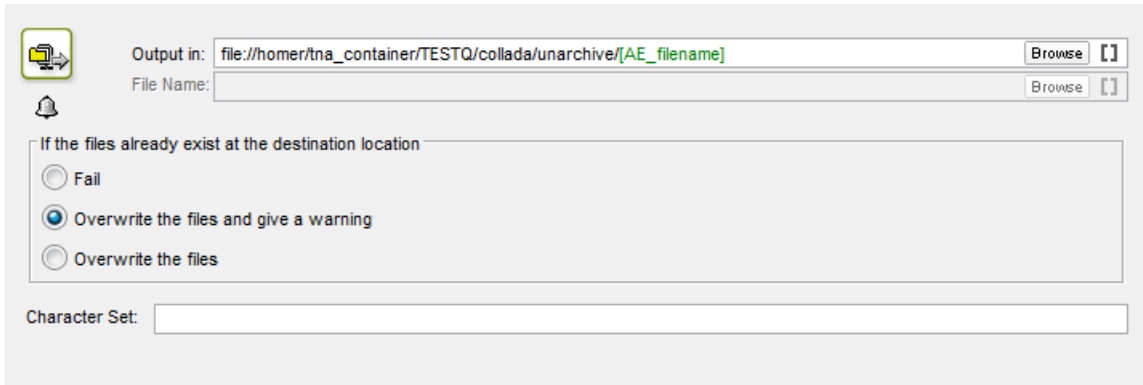
Output

You can specify the output at **Output** tab. You can enable **Trap layer only** or **Load trap color pairs if present** or enable both

See also: [What is trapping?](#)

2.7.145 Unzip

You can use this task to extract files from a zipped archive. (Please refer to the [Zip](#) on page 525.)



Output in: file://homer/tna_container/TESTQ/collada/unarchive/[AE_filename] Browse

File Name: Browse

If the files already exist at the destination location

Fail

Overwrite the files and give a warning

Overwrite the files

Character Set:

You can select to one of these options if the files already exist in the destination folder:

- **Fail**
- **Overwrite the files and give a warning**
- **Overwrite the files**

You can also specify character set in the given field to overrule the default character set to avoid problems due to special characters while unzipping.

2.7.146 Upload via FTP

Use the FTP task to send your files to a server. You need to have an FTP account on that server.

1. Click **New** to define a location. The **Edit Ftp Location** dialog pops up.

2. Enter the **Connection** information:

- **Host:** the name of the server you want to send your files to.

Tip: To use a port other than the default port, add : and the port number after the host name. For example: `myftpserver:1085`

- **Folder:** if you do not specify a path, the files will be stored on the default directory. If you do enter a directory, the files will end up in the directory you specified.

Tip: Using SmartNames to define the destination folder may help you to organize the arriving data.

- **Connection Timeout:** enter the connection timeout in milliseconds. You only need to change this if you are working on a slow network and keep getting time out errors.
- **Secure FTP:** select this if you are connecting to a secure FTP server (over FTPS, not SSL or SSH).

Tip: If you are experiencing problems using secure FTP, try connecting over regular FTP, and ask your IT administrator to check the security settings of your FTP server.

3. Enter the **User name and password** you use to connect to the FTP server:

- **User Name:** enter a valid user name for the server.
- **Password:** a valid password for the account.

Tip:

SmartNames can be used to select user name and password.

This might be useful as user name and password are often centrally stored and can so be SQL queried. And when user names and passwords change, your tickets do not have to be changed.

4. Define settings for uploading your **File** on the FTP server:

- **File Transfer:** choose between Binary and ASCII.

Binary will always work, but if you have an ASCII file, the transfer will go a bit faster when selecting ASCII.

- **If File Exists:**

- **Overwrite and give warning:** the file will be overwritten, but the tasks finishes in the state *Warning*.
- **Overwrite:** this will just overwrite the file and the tasks finishes OK.
- **Generate a unique filename:** this will append the task ID to the file name, and store the file at the destination under its unique name.
- **Generate an error:** the file will not be sent, the task will fail.

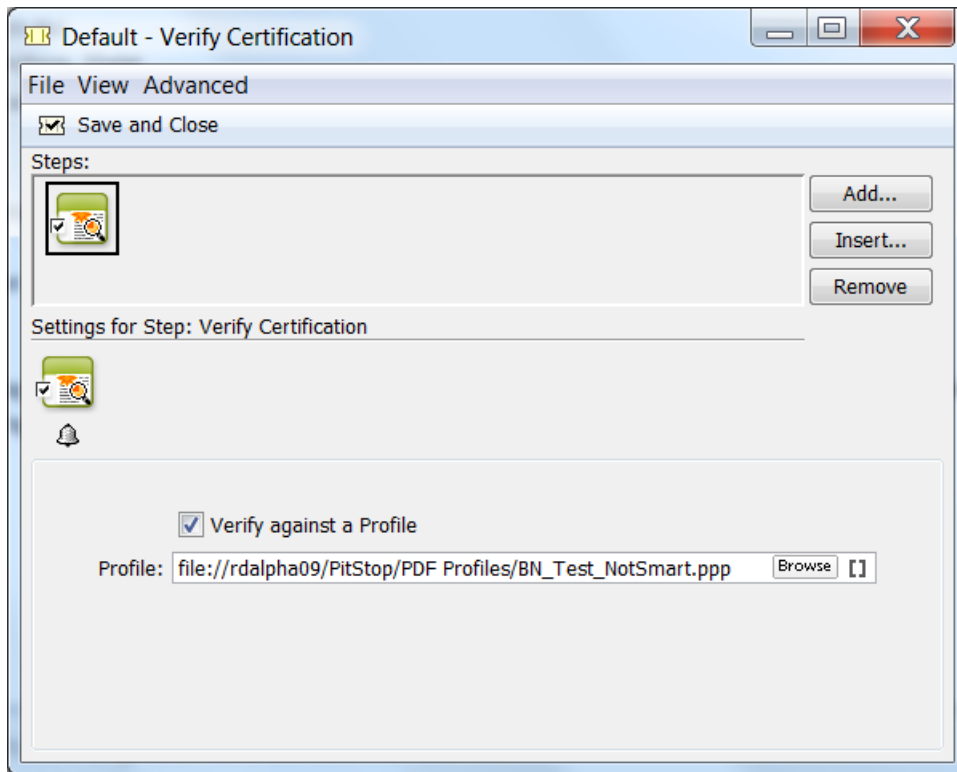
- **Connection Mode:** By default, the transfer uses Active **Connection Mode** (where the FTP server opens the data connection).

If your system is behind a firewall that blocks incoming FTP server connections, select **Passive** to use Passive Connection Mode (where the client initiates the connection).

2.7.147 Verify Certification

A PDF file can be certified when it is being preflighted against a PDF profile (using **Preflight with PitStop** task. The preflight information is stored in the certified PDF file. This include preflight profile, the preflight report and the information on whether the PDF has subsequently been modified. If your PDF is not re-certified after subsequent modification, it is no longer certified.

You can use **Verify Certification** task to verify if the PDF file is certified. Use this task to verify against a **Profile**. It checks whether the PDF file was certified to comply with the selected profile settings.

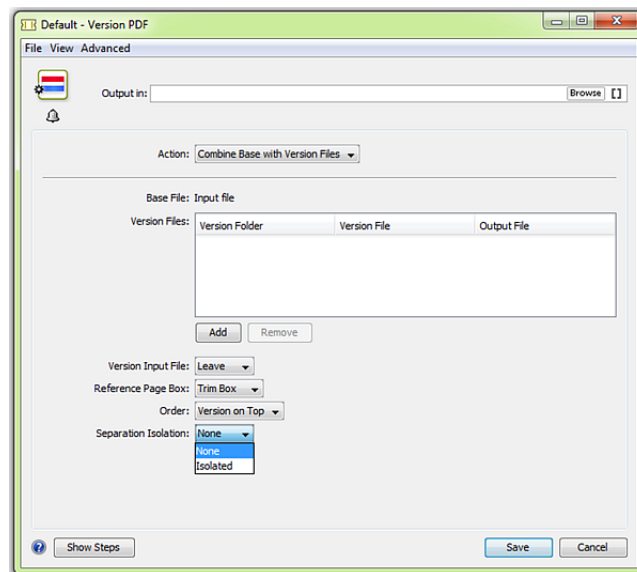


The task results in **Verification OK** if the profile that was used to certify the PDF is identical or stricter than the selected profile in the task.

2.7.148 Version PDF

When working with various versions of a single PDF, such as multiple language versions or versions containing minor differences, you can use this task to save on output material by:

- Combining a base file with multiple version files.
- Extracting separations or layers from the base file and replacing them by variants.



Combine base file with version files

By combining a base file (or input file) with a number of version files you can create various flavours for the same document and save on output material.

For example: your base file can contain graphics and the CMY-separations. Your version files contain text in different languages in the K-separation. By combining the images in your base file with the various language versions, you only need to make the plates containing the separations for the graphics once.

Proceed as follows:

1. Under the **Version Files** dialog, click **Add**.
2. In the **External File** dialog, specify the following:
 - The location of the **Version Folder**, which contains the version files.
 - The name(s) of the **Version File**(s) including the file extension.
 - The name of the **Output File**, which is a combination of the base file and a version file.
3. Choose what you want to do with the **Version Input File** (base file or input file):
 - If you want it to remain in the same place, choose **Leave**.
 - If you want it to be deleted, choose **Delete**.
 - If you want it to be moved to another folder, choose **Move to**. If you choose this option, determine the new location with the **Browse** button.
4. Select the desired **Reference Page Box**. Selecting the correct media box here can prevent errors, such as double time stamps in the text of a page mark.
5. Determine the **Order** of the Base File and the Version file.
6. Choose if you want **Separation Isolation**. The value "Isolated" can be used to use the "Form Isolation" PDF construct for combining the files, to reduce the interaction between the content of the files. This is helpful when the files contain information for different separations, rather than artwork to be placed on top.

Note: You can use **SmartNames** and wild cards to specify folder or file locations and names.

Note: You cannot combine Normalized PDF files with other PDF files.

Extract separations from input file

Extract a separation from the input file and map it to another separation to limit the total number of separations for output.

An example: Map a 'black' Pantone color that is used for text to 'Key color' (Black in CMYK).

Extract layers from input file

Extract layers from the input file and split the input file into a fixed master and a variant part based on the graphical object database. Often black is the variable ink.

You can also work with layers to extract the language layer from the input file and replace it with the corresponding translation(s).

Note: Work with wild cards to optimize your workflow: for example to only extract the language layer, use 'lang_*' if you know that all language versions follow this name rule. This will allow for the lang_EN, lang_FR and lang_ES layers to be automatically extracted from the base file.

2.7.149 Wait

This task is documented in the chapter "Workflow", in the section [Inserting Wait Points](#) on page 1001.

2.7.150 Wait for Action (Checkpoint)

This task is documented in the chapter "Workflow", in the section [Inserting Wait Points](#) on page 1001.

2.7.151 Wait for Files

This task is documented in the chapter "Workflow", in the section [Inserting Wait Points](#) on page 1001.

2.7.152 Wait for Files (via JDF)

Use this task when you are working with a JDF RunList, but the files referenced in the RunList are not ready yet.

Send the JDF RunList to this task, and it will check if the files are ready at a regular interval. When it finds all the files, it ends in OK and passes them on to the next workflow task.

You can set where to look for the files, and what to do if the files aren't found after a certain time.

Example Workflow

You can use this task when working an MIS system:

1. The MIS system creates jobs via JDF, and sends the JDF RunList to your Automation Engine workflow.
2. The **Wait for Files (via JDF)** task checks if all the files referenced in the RunList are where they are supposed to be.
3. When the files are ready, the task finds them and passes them on to the next workflow task.
4. The rest of the workflow starts on the files.

Parameters

Look in Folder

By default, the task looks for the files at the location defined in the RunList. However, you can set a custom location in this parameter.

You can use SmartNames here: for example, use [URL of Folder of Input] if you know the files will be copied in the same folder as the RunList.

Stop waiting when time exceeds ...

Select this if you want the task to finish after a certain time if the files referenced in the RunList haven't been found.

You can set the time in minutes, hours, days or weeks. To set a custom time, select **other...** then select a date and time from the drop-down menus below.

If you don't select this parameter, the task will look indefinitely (or until it finds the files).

Task ends in error when not all files were found

This parameter is only available when using the **Stop waiting when time exceeds ...** parameter.

Select this if you want the task to finish with an **Error** state when:

- the time you set in the **Stop waiting when time exceeds ...** parameter has passed,
- the task hasn't found the files referenced in the RunList at that point.

2.7.153 Wait for Product Status

This task allows to hold a workflow until one or more **Products** have reached a certain status.

This task is documented in the chapter on [Products](#) on page 876. Find a direct link [here](#).

2.7.154 Zip

This task creates a Zip file from the selected file(s) or folder.

Choose the task's behavior in case the zip file already exists:

- Generate an error.
- Replace the zip file.

See also the [Unzip Task](#).

2.7.155 Deprecated and Obsolete Tasks

Why Are Tasks Deprecated or Obsolete?

Several tasks have been declared “obsolete” or “deprecated” since Automation Engine 12. This document explains what that means and why certain tasks should no longer be used..

Deprecated tasks are old types of Automation Engine tasks that are either no longer used or that were replaced with newer tasks with similar functionality. By declaring tasks "deprecated" Esko also warns its customers that it plans to take these tasks out of a next version of the Automation Engine.

This gives you the chance

- to replace these tasks by newer equivalents
- to contact Esko support with the request to keep a task in the software in case their workflow relies on tasks that have no equivalent.

The next step is to declare a task "obsolete" - it is removed from the next version of the software.

Obsolete Tasks since Automation Engine 12.1

Task	Reason for making this task obsolete
Add Parameters for Nesting	<p>Obsolete due to the changes in i-cut Automate module. Use the Submit to Gang Run task as a replacement.</p> <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <p>Note: The Submit to Gang Run does not create an XML file (formerly typically dropped into a hotfolder). This information is written directly to the gang run database.</p> </div> <p>Modify your workflows with the new task. Also convert nesting mechanisms via a hotfolder into a Substrate queue in the Gang Run Printing view. See the Gang Run Printing documentation for more information.</p>
Create Nested Layouts	<p>This task has become obsolete since the i-cut Automate module has been reworked. The functions and settings of this task are now divided over the Substrate settings in the Gang Run Printing view and the Export Nested Layouts task. In the substrate settings you will find all the sheet specific settings and the nesting parameters. The 'Export Nested Layouts' task holds all the output settings.</p>

<p>Create Preview</p> <p>Normalize EPF File</p>	<p>Note: The nesting no longer happens in the task, but as part of a background process triggered by the nesting server.</p> <p>Modify the workflows and convert nesting mechanisms via a hotfolder into a Substrate queue in the Gang Run Printing view. See the Gang Run Printing documentation for more information.</p> <p>This task was used to prepare a thumbnail image for images. These display images were used by editors in later steps of the workflow. Today editors make these display images when needed without performance penalties, so there is no longer any need to generate these display images in advance.</p> <p>EPF is an outdated data format written by DeskPack Illustrator versions prior to DeskPack 7 R6. Since version DeskPack 7 R6 the Normalized PDF format is used to send data to other Esko applications, including Automation Engine.</p>
---	--

Note: Adjust the workflows based on the obsolete tasks with new features or tasks.

Deprecated Tasks in Automation Engine 12.1

Figure 1: Deprecated tasks are marked in the Pilot by the suffix “(deprecated)” in the Task Type. List of deprecated tasks in Automation Engine 12.1

Add Parameters for Nesting (deprecated)	Default
Adjust PDF Screening (deprecated)	Default
Contourize & Clean (deprecated)	Default
Create Master DCS File (deprecated)	Default
Create Nested Layouts (deprecated)	Default
Create Nested Layouts (deprecated)	DT106327cnl
Engrave on Daetwyler (deprecated)	Default
Engrave on Hell (deprecated)	Default
Expand Variable Data to PPML File (DeskPack only) (deprecated)	Default
Export to GRS File (deprecated)	Default
Export to Normalized File (DeskPack Only) (deprecated)	Default
Export to VRML File (deprecated)	Default
FastVariants - Check (deprecated)	Default
FastVariants - Remove White Boxes (deprecated)	Default
FastVariants - Split (deprecated)	Default
Fork (deprecated)	Default
Normalize TIFF/IT File (deprecated)	Default
Proof (Image Downloader) (deprecated)	Default
Rip to WS4000 series (deprecated)	Default
Trap - Create Trap Layer (deprecated)	Default
Trap - Prepare and Create Trap Layer (deprecated)	Default
Trap - Prepare Trap Pairs (deprecated)	Default

“Deprecated” means that it is being considered to remove these tasks from a future version of the software. The exact version has not been decided yet. Customers are recommended to start planning how they can switch to newer tasks with equivalent functionality.

These tasks are not available for new customers. Existing customers that had the tasks, will still have access to them after upgrade to Automation Engine 12.

The reasons for deprecating these tasks are listed in the table below.

Task	Reason for making this task deprecated
Adjust PDF screening	The user interface as well as the algorithm of this task have been totally reworked. The old task has been kept so existing workflows will not break after an upgrade.
Contourize and Clean	This task was used to turn scanned bitmap images in vector art. We believe there are better tools available in desktop applications like Adobe Illustrator.
Create Master DCS File	This task was used to create a DCS wrapper for a given set of images. The DCS could then be placed in an application like PackEdge or Adobe Illustrator. The replacement for this is to use the Create Wrapper task. This task will create a PDF. DCS is an old format. This PDF is a modern equivalent that can be used in the same way as the DCS.
Create Nested Layouts	This task has become obsolete since the i-cut Automate module has been reworked. The functions and settings of this task are now divided over the Substrate settings in the Gang Run Printing view and the Export Nested Layouts task. In the substrate settings you will find all the sheet specific settings and the nesting parameters. The 'Export Nested Layouts' task holds all the output settings. Note that the nesting itself no longer happens in the task itself, but as part of a background process triggered by the nesting server. It is advised to rework workflows that are based on the old tasks. That includes converting nesting mechanisms via a hot folder into a Substrate queue in the Gang Run Printing view. See the Gang Run Printing documentation for more information.
Export to GRS File	The GRS format was the native format for Esko applications. Many years ago the GRS format was replaced by Normalized PDF as the native format for Esko applications. The ability to export a Normalized PDF to GRS was meant to send files to older Esko applications. We believe this is no longer needed.
Export to VRML File	The Export to VRML task was used to export 3D files in the VRML format. This is replaced by 3D PDF generated by the "Export 3D" task.

FastVariants (3 tasks)	These tasks are being replaced by “Version PDF” and “Compare PDF”
Fork	This task was used to connect task chains into more complex workflows. This functionality is replaced with the extended workflow functionality introduced in Automation Engine 10. The since then available Fork workflow control also works differently. Learn more in Fork and Join on page 1032.
Normalize TIFF/IT	This task was used to create a Normalized PDF starting from a TIFF/IT. We believe TIFF/IT is no longer used.
Trap – Create Trap Layer, Trap – Prepare and Create Trap Layer and Trap – Prepare Trap Pairs	All the functions of these tasks have been integrated in the Trap with PowerTrapper task. Modify your workflows with Trap with PowerTrapper task.



Attention: Replace any deprecated tasks you might have in a workflow with the updated tasks.

Normalize TIFF / IT File (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

The TIFF-IT import task is used in two situations: to edit linework, or to import a TIFF-IT file containing copydot or blends.

A TIFF-IT job can consist of four data files:

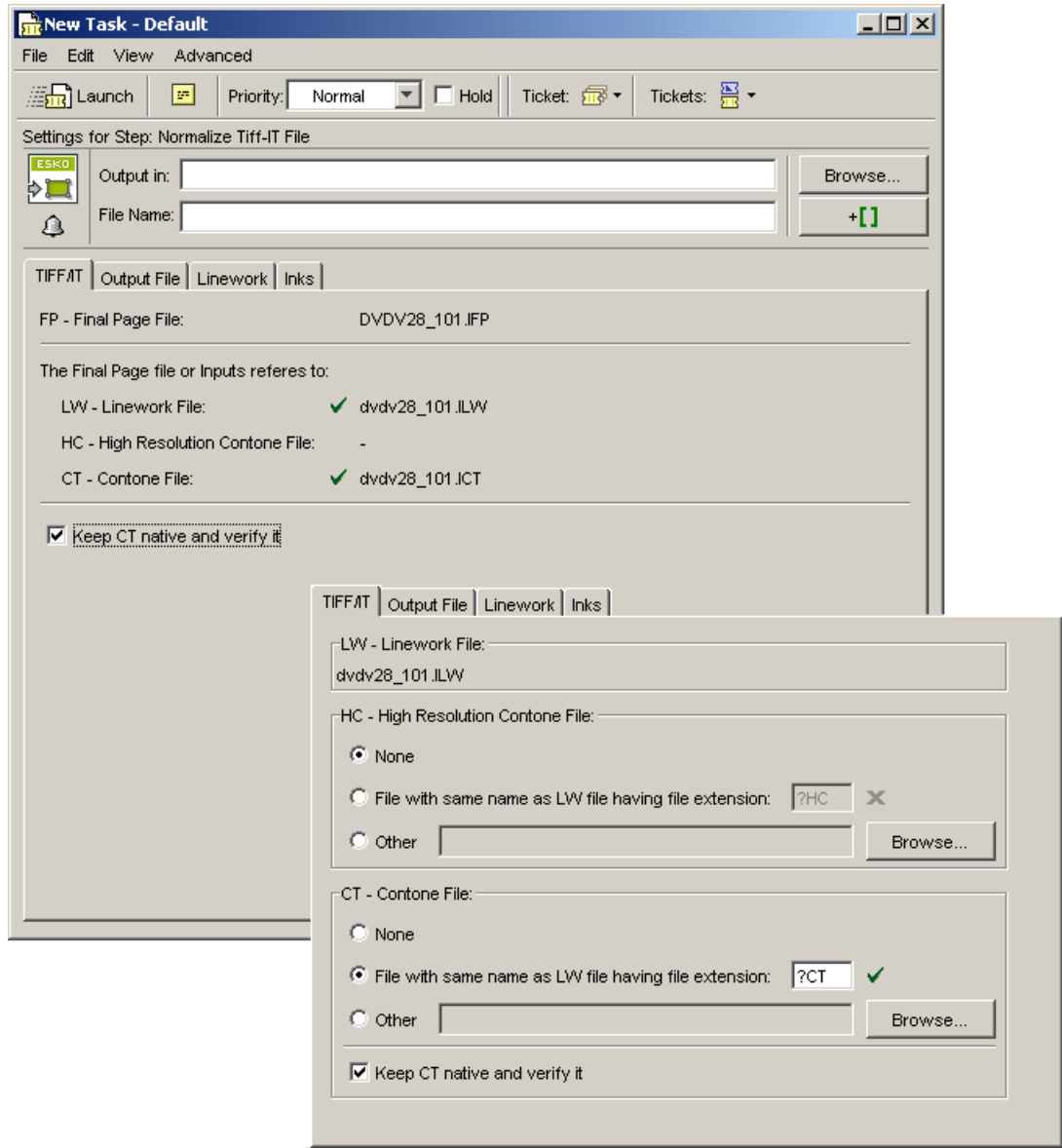
- an FP file (final page) which has links to the data files with the graphic content. This file is optional.
- an LW file (linework). This is a palette high-resolution image that contains the linework part of the job. This file is optional or required, depending on whether or not it is referred to in the FP file.
- an HC file (true-color high-resolution image), containing the linework part of the job. It is possible, but unusual that both the LW and the HC are used at the same time. This file is optional or required, depending on whether or not it is referred to in the FP file.
- a CT file (true-color CMYK contone). This file is optional or required, depending on whether or not it is referred to in the FP file.

The TIFF-IT Import task converts a TIFF-IT job to a single PDF referring to an external CT, plus optionally to a number of LPs with copydot and/or high-resolution CTs with blend areas.

The external CT may stay in TIFF format or can be normalized to the BG CT format. LPs with copydot and CTs with blends will be always BG ones. The rest LW/HC part of the job (after removing copydot and/or blends) gets contourized and is stored within the PDF.

When the job is launched, the topmost image is passed as the 'input' file and references to other images are made by entries in the task ticket. The server part of the task recognizes a job as being a set of free images and processes them accordingly.

TIFF-IT



The options on this tab change according to the type of job selected:

1. For jobs with a Final Page it shows the image files and indicates their presence (see first screen shot above).
2. For jobs without a Final Page, the TIFF/IT tab shows the name mode selector, one for each possible dependent file (LW, HC and CT file) (see second screen shot above).

The different modes are:

- Select **None** if the dependent file is omitted (missing).

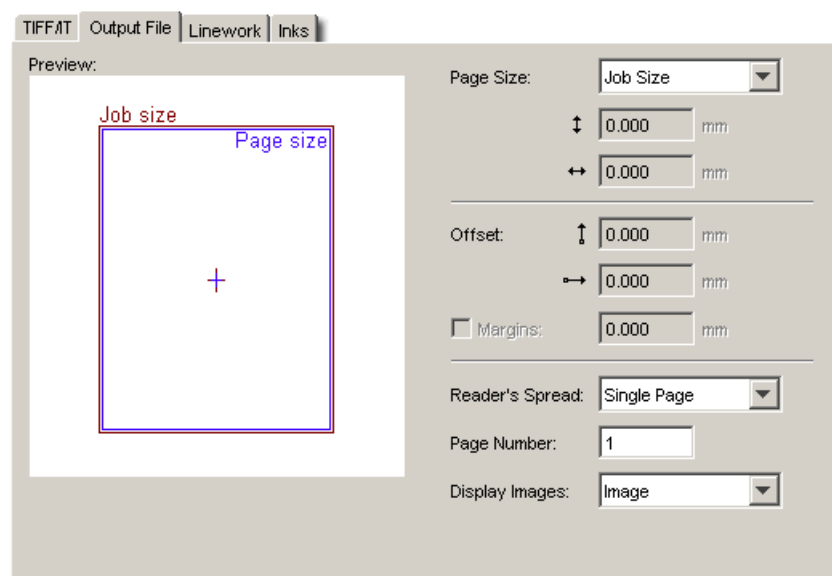
- **As LW with extension:** Enable this toggle if you want the name of the dependent file to be derived from the primary one (by changing the extension).
- **Other:** Use the Browse button to indicate the path of the dependent file.

The special notation ('?') allows unique handling of jobs having both the more frequent extension ILW/IHC/ICT and the less frequent one LW/HC/CT.

✓ or ✗ indicates whether or not the image file exists.

Keep native and verify Certain file types like copydot, tiff and eps files are already in a reliable format. These files will be verified and left in their native format. On-the-fly preview files will be created.

Output File



The default **page size** is Job size. When **Job size** is selected, the page size found in the selected file will be taken into account when importing. The **Custom** option allows you to enter page size values manually. The dropdown list displays all possible options (Custom, A5, A3, Letter, Scanner Trim Size, Tabloid...).

Reader's Spread

None of the file formats that can be wrapped into a GRS has the notion of reader's spread. It can be useful, for example when using the GRS files in FastImpose, to give an indication of whether the file is a single page or a reader's spread.

- **Single Page** forces the file as single page output.
- **Reader's Spread** forces the file into a reader's spread format. The file will be set as double pages.
- **Auto** checks the size of the image and when the image width equals or exceeds two times the selected page width, the reader's spread option is set. Otherwise it is treated as a single page.

Offset

Enter the appropriate values to give the page a horizontal and/or vertical offset. When no offset is given, the center of the image will be positioned on the center of the page. If the page is smaller

than the image (width, height, or both), then margins will be set that enclose the image completely. The size of the margins will not be fixed. Margins indicate the differences between the image size and the user-defined page size.

Margins

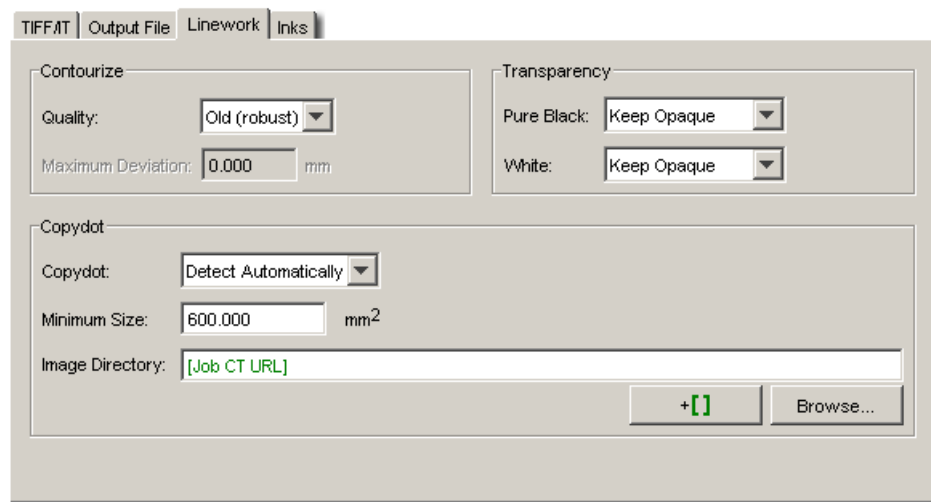
Tick the margins toggle and define the margins' dimensions if you want to give the margins of the page a fixed size (for example 3 mm for bleed). Click the arrow in the dropdown list to switch between millimeters and inches.

Select the **page number** from the file that you want to output.

Display images

This option allows you to change the preview settings of the file. Select None, Thumbnail or Image from the dropdown list.

Linework



Contours:

- **match pixels exactly (robust mode):** Contourization is done using an older, but proven algorithm.
- **match pixels exactly (fast mode):** Contourization is done using a faster algorithm.

Tip:

In the unlikely event that contourization fails, try again in the robust mode.

- **are straightened:** This option straightens borders of continuous single color areas (for example the borders of black characters on white paper).

Overprint Settings:

- **Pure Black:** Set the paint mode for objects containing 100% black and at least one other ink. Possible Rich Black paint modes are:
 - **Keep Opaque** (masking) sets the overprint mode to opaque for all objects containing 100% black and at least one other ink.
 - **Force BG Overprint** sets BG Overprint for all objects in 100% black and at least one other ink.

- **White:** Possible White paint modes:
 - **Keep Opaque** forces all objects in white to Opaque.
 - **Make transparent** will remove all white objects.

Copydot:

- Detect where the job contains copydot: the data will first be pre-read and analyzed for copydot and/or blends. The areas found will be handled appropriate way.
- Job does not contain copydot: all data will be contourized.
- Job contains only copydot: all copydot-like data will be separated to LPs without detection.

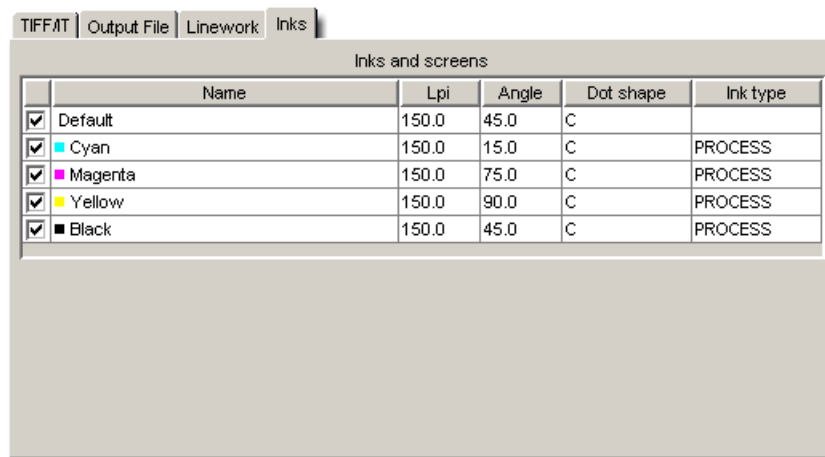
Note: The implementation includes storing the results of the copydot/blend detection as part of the metadata of the HC/LW file. If the job will be re-processed, then re-reading won't be necessary.

Minimum size (square units): Six square units is the default value.

Tip: If your job terminates with an error, check the log message. It may be that the selected minimum size is too small. Select a higher value and relaunch the job. Error message: "ERROR: Premature end of data splitting. Exception of class ACQF_EDataSelectHCWithTransparency with message: "A copydot or blend area was detected which is transparent. The result of splitting can therefore not be guaranteed to be correct. Please try doubling or tripling the "Copydot Minimum size" option value."

Image Directory: all .lp files are by default stored in the ct directory of the job folder. Use the Browse button to select a different directory to store .lp files.

Inks



The Inks tab contains a fixed list of inks (default, cyan, magenta, yellow, and black).

Note: You cannot add or modify inks in the Normalize TIFF/IT File task.

If an image contains definitions for its inks (for example a CMYK Tiff or a PhotoShop DCS2) and an ink of the image matches an ink selected in the inks tab, then the angles, lineature, etc... in the output file will be the ones from the inks tab. All inks in the image that do not match the selected ink in the inks tab will be processed according to the default values for angle, lineature, etc...

If the image contains an ink that is not listed in the inks tab, its angles, lpi, etc... the default values will be taken into account.

Values (angle, lpi, dotshape, ...) you define in the default line of the inks tab are used to define the angle, lineature, etc... of new inks added to the inks tab. The default line has no effect on the output file.

If the wrong ink is selected, together with the default ink, then the output file will be generated with brix parameters from the default ink.

Add Parameters for Nesting (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

Use this task to specify nesting parameters for a particular graphic (cut path, quantity needed, due date...).

If you want to combine several graphics onto a sheet, you should send them all to the same **Add Parameters for Nesting** task.

You can send PDF (single page or multipage), TIFF, PSD or JPEG files to this task. Progressive JPEG compression is not supported.

It outputs an XML file containing all of the nesting parameters for the graphic(s). You can then send that XML file to the [Create Nested Layouts task](#).

Note: You can use SmartNames to retrieve the parameters needed from the job setup.

1. In the **Front** tab, specify the **Quantity** of that graphic that you want to print.

You can select **Maximum Fill** if you want to print as many copies as will fit in one sheet (if you are not sure how many copies fit exactly and you don't want to waste paper).

2. If the input file is a downscaled version of your graphic, enter the **Scale** to use.
For example, enter 1000% to compensate a graphic that was downscaled to 10%.
3. If the graphics in your files have a bleed outside the cut path, you can clip it away by entering the bleed distance into **Bleed Clipping Mask**.
4. Enter/choose the ink used for the cut path in the **Cut path** table (the drop-down list will contain all inks used in the input file).

You can **Add** several inks (for example if you have customers from different countries using different ink names like *cut*, *découpe*, *taglio*...) and **Move** them **Up** or **Down** the list. The order is important as the task will use the first ink it finds in the list.

Note: If you have specified a cut path ink name in the **Prepare Graphics for Nesting** task, you only need to enter that one ink name.

You can use SmartNames to retrieve ink names from the job setup.

Select **Create cut path from trim box if no ink is found** if you want to use the trim box to create a cut path when a file doesn't contain any of the inks you specified.

5. Specify a **Due Date** for the nesting (when the sheet with the nested graphics needs to be printed).
6. In the **Back** tab, you can specify what to print on the back of each repetition of the graphic:

- **No graphics.**
- **Even pages of multipage file:** if you choose this, the odd pages of the file will be printed on the front, and the even pages on the back (this only works with multipage PDFs).
- **Graphics from front side:** choose this if you want to print the same graphic on the front and on the back.

If your graphic has an irregular shape, you can select **Flip horizontally** so both graphics face the same direction and have the exact same contour for cutting.

If your graphic contains text, you shouldn't flip it as this will make the text unreadable.

- **Other graphics:** you can **Browse** to the graphic you would like to print on the back side.

Tip: Give the graphics to print on the back a consistent name, and use SmartNames to retrieve them (for example, [File]_back.[Extension of File]).

Adjust PDF Screening (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

Use this task to change the screening parameters in the input PDF. You can use it to define specific screening parameters for some or all inks in your file, and/or assign default screening parameters to inks with no screening parameters specified in the file.





Note: This task doesn't change the ink order in the PDF.

1. In the **Inks** table, double-click the first row in the **Ink Name** column, enter the first ink for which you want to define (new) screening parameters and press **Enter**.
2. The **Ink Book** column, you will see all the ink books containing this ink name in the CMS database. Select the correct ink book for your file.
3. Enter the **Ruling**, **Angle** and **Dot Shape** to use for this ink.
This uses the **Ruling** unit defined in the [Preferences](#).
4. Repeat for all inks needing (new) screening parameters.

Tip:

To define the same parameter value for all inks in the table, double click the header of the parameter column, enter that value and press **Enter**.

For example, to give all inks a ruling of 150 dpi, enter 150 in the **Ruling** column header and press **Enter**.

Inks:			
Ink Name	Ink Book	150.0	Angle
 Cyan	process	150 lpi	15 °
 Magenta	process		
 Yellow	process		
 Black	process		

5. At the bottom of the task panel, enter the **Default Ruling, Angle** and **Dot Shape values** to use for inks you entered in the **Inks** table without those values.

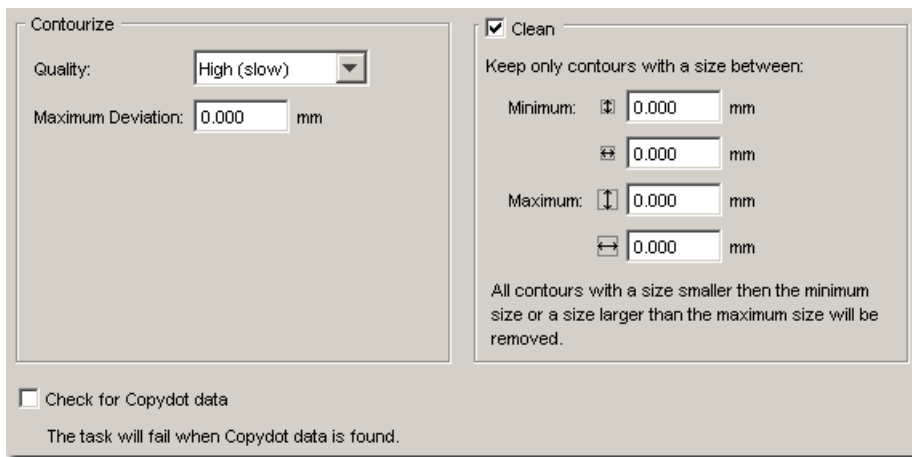
Note: If you don't fill in default values, inks without a ruling defined will get a ruling of 120 lpi (or equivalent in your ruling unit), and inks without an angle defined will get an angle of 45°.

Contourize & Clean (Deprecated)

Note:

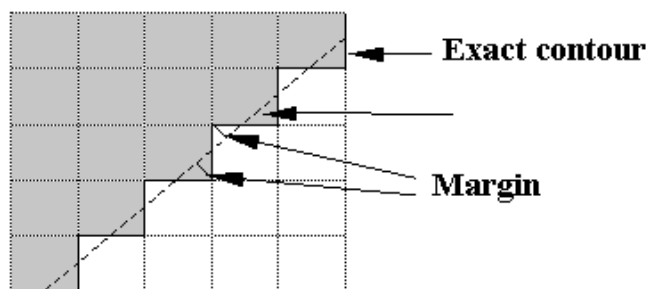
This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

Contourizing is reading image pixel data and writing them as vector data. This reduces the amount of image data and can straighten borders of continuous single color areas (for example the borders of black characters on white paper).



Contourize

- **Low Quality** (fast speed): The fastest way of contourization with low data amount reduction ability; the contours follows pixels as rectangles.
- **Medium Quality** (medium speed): Does not reduce the data very efficiently, but it is very fast; it straightens the borders of continuous single colored areas.
- **High Quality** (slow speed): Is much slower than Medium Quality contourization, but it reduces the amount of data a lot. You can define the margin the system has to take into account when it contourizes (enter the number of pixels).



When the **Copydot data** toggle is selected, the system will check for Copydot data. If the system finds Copydot data the task will be terminated. If the toggle is not selected, the job will be processed no matter what.

Note:

The Background color is not contoured.

If you want to clean your job, select the **Clean** toggle first. You can set the cleaning parameters, you can also select the unit of those parameters (pixel, mm, inch and mil).

You define the contours you want to keep, a zero will be ignored.

- Minimal Horizontal and Minimal Vertical size. You fill in the minimal size of the contours you want to keep, all the others (smaller ones) will be cleaned.
- Maximal Horizontal and Maximal Vertical size of the contours you want to keep, all the contours that are larger will be cleaned. Contours with a vertical size larger than maxv and a horizontal size larger than maxh will be removed. A value of 0 in one of the fields will disable the check on maximum size. A scan can have a black border on the edges, just like an image scan will often show the edge of the slide it was scanned from. That black border will be made up of two contours with a large size, a size larger than the 'logical' size of the page. You can therefore remove it by specifying the maximum sizes as equal to the page size.

Create Master DCS File (Deprecated)

Note:

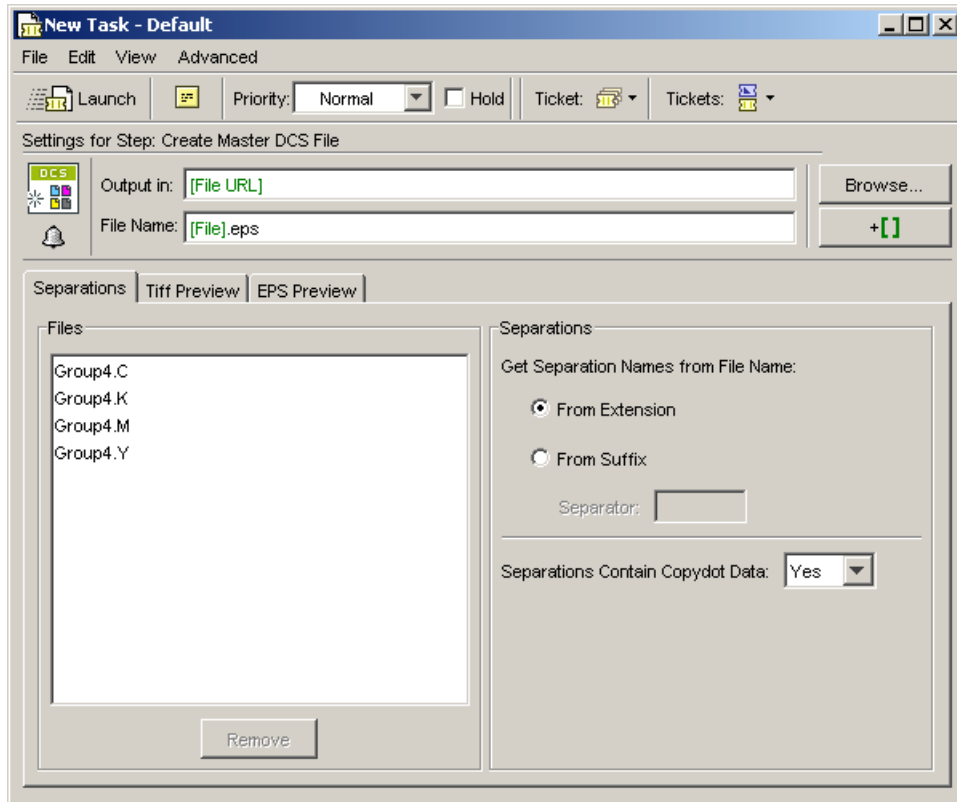
This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

The Create Master DCS task is intended to bind a set of stand-alone files containing bitmap or copydot data (particular ink separations of one image).

Output consists of a file in DCS 2.0 format with references to the separate plate files selected by the user.

This task is useful when separation plate files have been delivered without a master DCS file or when the master DCS file is no longer valid (due to wrong references or wrong preview).

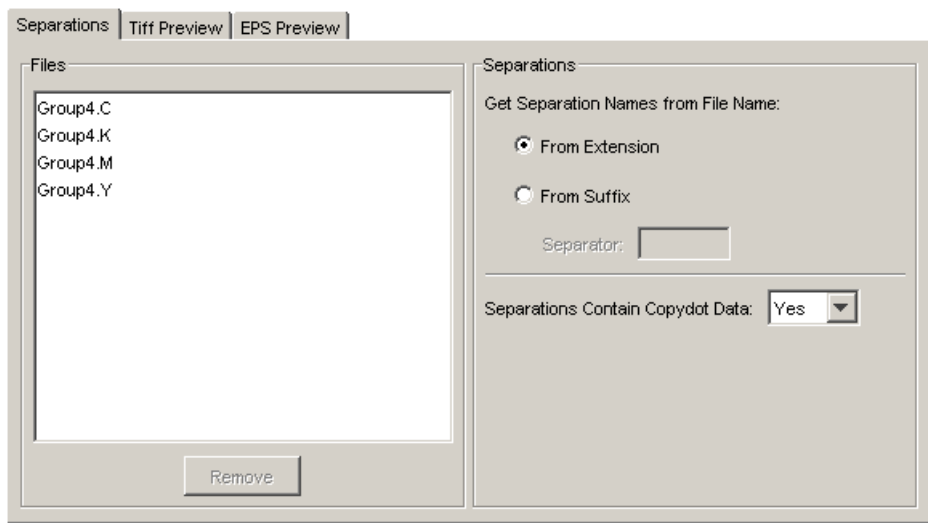
The Create Master DCS task supports the TIFF, JPEG and EPS file types, and can also be applied to a Folder.



Separations

The Separations tab consists of three groups:

- Files
- Separations
- CopyDot



Files

The **Remove** button is intended for removing files from the list. It is most useful if you select the entire Folder first.

Separations

This option allows you to control how ink names will be parsed from the file name.

- **From extension:** Enable this toggle to indicate that the short ink name is an extension of the file name. For example 6787.c
- **From suffix:** Enable this toggle to indicate that the short ink name is a part of the file name.

In the **Separator** field you can enter the character which is inserted between the basic name of a file and the short ink name. For example '\$' in 6787\$c.tif

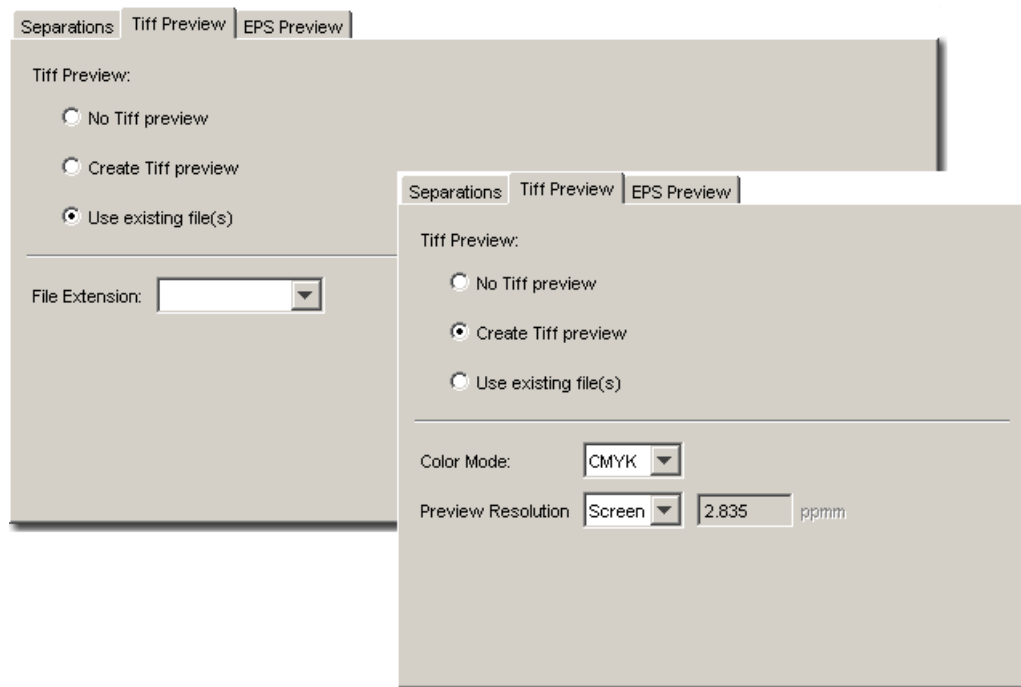
CopyDot

- **Yes:** Enable this toggle to indicate that all separations are Bitmap or Colored. The CopyDot marker is added.
- **No:** Enable this toggle to indicate that separations don't contain CopyDot data. The CopyDot marker is not added.
- **Detect:** Enable this toggle to indicate that all separations are Bitmap or Colored and the application detects CopyDot data in at least one of them. The CopyDot marker is added.

Tip: Using the Detect option has an effect on the speed, while the real separations data is read during detection.

Note: Files containing CopyDot data must be marked in order to be processed correctly in the workflow.

Tiff Preview



Select whether or not you want to create a new preview or use an existing one. Depending on your selection, different options will appear:

Use existing file(s)

When a **file** is selected, and you tick the Use File radio button, the **Browse** button will be activated.

When a **folder** is selected however, you will have to pick the correct **extension** from the list.

Create Tiff preview

Select the appropriate preview mode:

- CMYK
- RGB

Resolution

Set the resolution of the created preview image. Values are expressed in dots per inch (dpi).

- **Screen:** Enable this toggle to create a preview at a resolution of 72 ppi.
- **Medium:** Enable this toggle to create a preview at a resolution of 150 ppi.
- **High:** Enable this toggle to create a preview at a resolution of 300 ppi.
- **Other:** Enable this toggle to create a preview at the resolution you specified in the ppi input box.

Note: The values mentioned about are only valid if you work in ppi. The value specified here depends on the resolution [preference settings](#).

EPS Preview

The EPS Preview tab displays the same options as the [Tiff Preview tab](#).

Create Nested Layouts (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

Use this task to nest a set of graphics onto one or more sheets, and create production files (a PDF print file and a cutting file) for digital printing and finishing.

Send the XML files generated by the [Add Parameters for Nesting task](#) to this task. You don't need to send the graphics as each XML file contains a link to the related graphic.

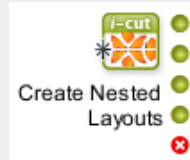
You can also use (compatible) XML files produced by an MIS system.

Alternatively, you can send already stepped and repeated (PDFPLA) files to this task to generate production files. This will then skip the nesting part and only output a print file and a cutting file.

Note: You can use SmartNames to retrieve the parameters needed from the Job setup.

Note:

In a workflow, this task has four output pins (**Print Files**, **Cutting Files**, **Done Graphics** and **Remaining Graphics**), so you can send each type of file to the right place more easily.



If you select **Create report**, the task will have a 5th output pin for the report.

Sheet Setup Parameters

1. In the **Sheet Setup** tab, define the sheet size. You can either:

- Use a **Sheet Template** that you have defined in i-cut Layout.

Sheet templates contain height, width and substrate information.

Tip:

If you have several templates using the same substrate but in different sizes, you can also select the **substrate name** as sheet template (shown in blue in the list).

In this case, the task will try nesting the graphics on the different substrate sizes, and will pick the best size for the job (or a combination of sizes).

- Define a **Sheet Height** and a **Sheet Width**.
- Enter the **Substrate** that will be used for the sheet.

You can also **Fit** the sheet size to the size of the nesting (you can fit either the **Sheet Height**, the **Sheet Width** or the **Width & Height**).

For example, for roll-fed printing, the sheet height will typically be the height of the roll, and the width will be fitted to the size of the nesting.

2. Optionally, you can add **Margins** around the nesting (a **Top**, **Bottom**, **Left** and **Right Margin**).
The margins won't be used for printing/cutting, but may contain some bleed.
3. If desired, you can add **SmartMarks** (that you created in i-cut Layout or PackEdge) to your sheet.
4. If printing Front and Back, define how the sheet will be turned for back printing in **Backing Up**: either **Work-and-Turn** or **Work-and-Tumble**.

Nesting Parameters

1. Select the **Nesting Type** to use for the layout:

- **Minimal Waste** tries to reduce the waste in material to a minimum,
- **Minimum Layouts** reduces the number of production files to a minimum (as for each new layout, a new print file and cutting file are generated),
- **Guillotine Cut** prepares layouts to be cut on a Guillotine table, reducing the number of cuts, and making sure all cuts are possible on a guillotine cutter.

2. Select the **Search Method** to use for calculating the layout.

Depending on if you are running a test or processing graphics for production, you can set the layout search from **Very Fast** to **Slow (Very Accurate)** (the faster the search, the less accurate the result can be).

3. Optionally, select **Respect the graphics order** if you want to nest the graphics in the order in which they are coming in.

Tip:

You can put a [Sort workflow control](#) in front of the **Create Nested Layouts** task to sort the files in a particular order.

Alternatively, you can put a **Sort** in front of the **Add Parameters for Nesting** task to have that order already reflected in the XML file (when using an XML file for several graphics).

4. If you have selected **Minimal Waste** or **Minimum Layouts**, you can set:
 - the **Fill Starting Point on Sheet** (where the first graphic will be placed): at the **Top Left, Top Right, Bottom Left** or **Bottom Right**,
 - the **Primary Fill Direction** (in which direction the sheet will be filled): **Horizontal** or **Vertical**.
5. If you have selected **Guillotine Cut**, you can set:
 - the direction of the **First Cut: Horizontal, Vertical** or **Don't Care**,
 - a **Maximum Overrun** to get less different layouts: if you are printing different quantities of your various graphics, some of the graphics with a lower quantity might get printed in excess to keep the number of layouts to a minimum. Enter the maximum number of excess graphics that can be printed in **Maximum Overrun**.

If you only want to print the exact quantity you specified for each graphic, keep the **Maximum Overrun** to 0.
6. If desired, you can add a **Gutter** in between cut paths (specify the width).
7. In **Rotation**, specify if the task can rotate the layout in case a better fit is possible. You can choose:
 - **Allow any angle**,
 - **Allow 90 degree**,
 - **Allow 180 degree**,
 - **No rotation**.
8. Optionally, you can center the nested graphics on the sheet: in **Align nesting center graphics on sheet**, select either:
 - **No** if you don't want to center them,
 - **Vertical** to center the graphics vertically,
 - **Horizontal** to center the graphics horizontally,
 - **Vertical and Horizontal** to center them in both directions.
9. Enter a **Minimum Fill** if you only want to output the sheet if it's filled to a certain level.

Note:

Do not put 100% here as only the graphics inside the cut path count as fill. The margins, gutters etc. all count as waste.

By default, the minimum fill is set to 75%.

10. If there is a due date linked to your graphics (see [Add Parameters for Nesting \(Deprecated\)](#) on page 534), select what to do if the minimum fill hasn't been reached and the nesting hasn't been output by that date in **Action when Due Date Expires**:

- **Set Task Status to Error.**
- **Force Nesting and Save to PDFPLA** to output the nesting as a PDFPLA instead of as production files.

You can then edit the PDFPLA manually in i-cut Layout (for example to print it on an offcut), or feed it back to the **Create Nested Layouts** task together with other graphics to fill the sheet a little more.

- **Force Nesting** to output the sheet anyway.

Output Parameters

The task will output:

- PDF print files,
- cutting files (in your desired format),
- nesting reports (optionally),
- XML files containing references to any non-nested graphics and their quantity.

1. In **Save Print Files**, select how to output the PDF print files:

- **as Single Files** to have one single page PDF file per layout,
- **as 1 Multipage File** to have one multipage PDF file containing all the layouts.



Attention: If the dimensions of the different layouts vary (for example when using substrate-based nesting or the **Fit** option), the task cannot create a multipage output.

Note:

If multiple copies of the same layout need to be printed:

- For single page output, the number of copies will be part of the PDF file name.
- For multipage PDF output (and if the number of copies per page is less than 10), the same page will be inserted multiple times, using forms to prevent the file size from growing.

2. In **Save Cutting Files**, choose the cutting file format you want to generate (**i-cut**, **HPGL**, **PDF**, **MFG** or **ACM**).

Note: When generating an ACM file (for a Kongsberg table), you can select **Print Side Up**, to cut with the print side visible.

Tip:

- If you have several cutting tables and you don't know yet which one you will use to cut the sheet, you can generate several file formats: choose **Select File Types...**, select all the file types you want to generate and click **OK**.
- If you want to check your cutting files for quality control, you can also output a PDF file and inspect it in Acrobat.

3. If desired, you can **Add Extra Structural Inks** (for fold lines for example) that will be added to the cutting file and excluded from the print file.

Tip: Use commas to separate the list. You can use wildcards (*).

4. If you want, you can create a nesting report (for example to use as feedback for a MIS system).
Select **Create layout report** if you want a report detailing the different graphics used in each layout.

Select **Create report per graphic** if you want a report detailing in which layout(s) each graphic is nested.

5. In **Output Name**, enter the base name to use for the output files.

For example, if you enter `Promotional_Coasters`, the first layout will be called `Promotional_Coasters_1.pdf`, the second `Promotional_Coasters_2.pdf`, etc.

If you don't enter anything, the name of the first input file will be used.

6. Define where to save the production files, the report(s) and the XML file(s) referencing any non-nested graphics:
 - If you want to save all output files in the same folder, select **Save all files in the same folder**, and browse to that **Output Folder**.
 - Otherwise, make sure **Save all files in the same folder** isn't selected, and browse to your chosen folder for the **Print Files**, **Cutting Files**, **Report Files** and **Remaining Files** (XML file(s) referencing any non-nested graphics).

Expand Variable Data to PPML File (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

The Expand Variable Data to PPML File task performs expansion of the specified database entries into the corresponding graphical output and comes in two variations; one for the PackEdge workflow and one for the DeskPack workflow.

The **Expand Variable Data to PPML File** is used for the PackEdge and Plato workflow. The input file is a PDFPLA file. The ticket allows you to set all variable data parameters.

In the **Expand Variable Data to PPML File (DeskPack only)** ticket, only the output location and a file name can be specified. This task is only used for expansion executed from within stepX.

All workflows will start with this task. In most cases, the expansion task is followed directly by the [RIP to WS4000 Series task](#), so creating a workflow ticket for both tasks is recommended.

However, independent use of the expansion task is also possible (in case you only care about the expansion output, but do not want to run the RIP task afterwards).

It is compulsory to specify an output folder for the 'Expand Variable Data to PPML File (DeskPack only)' task. Not doing so could generate a task error.

However, since you are allowed to use Smart Text in the specification, it is possible to create a path similar to this: `[joburl]/tmp_expansion/[date]_[time]`.

The contents of this folder must be maintained manually as the files are not deleted automatically.

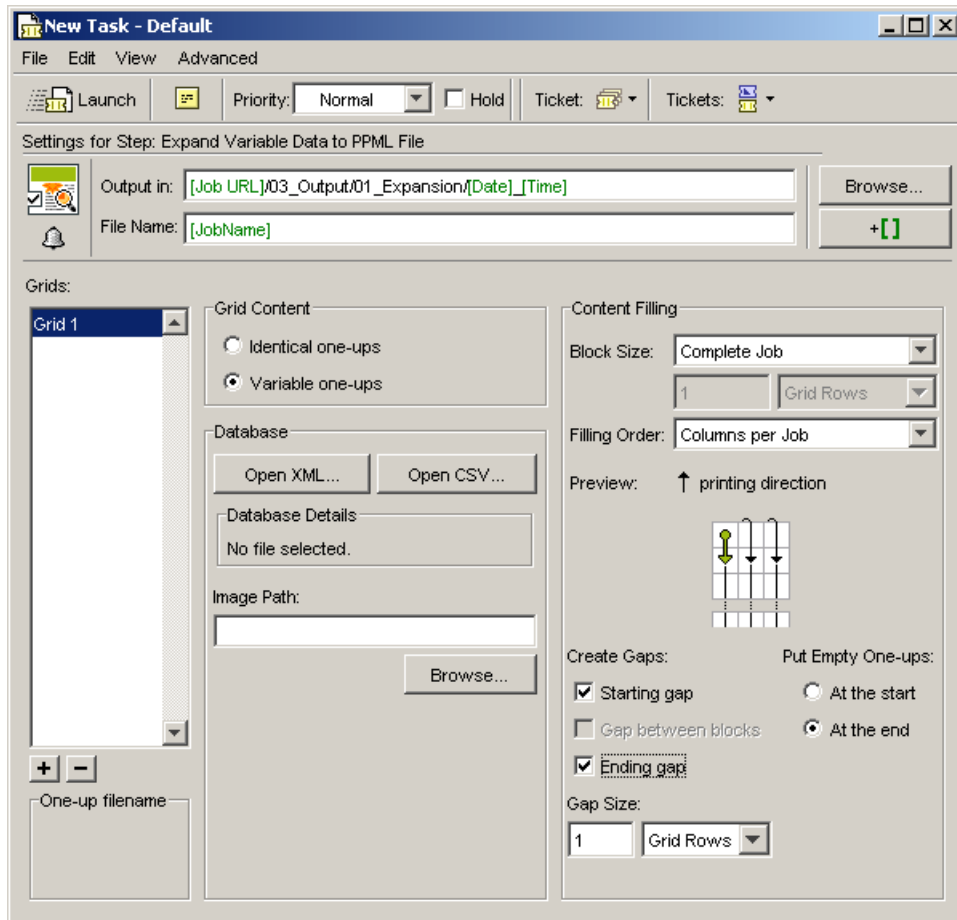
The task takes the following input files:

- a design file containing variable elements (Normalized PDF file)
- a repetition file generated by the stepX plug-in (PDFPLA file)
- a database file (XML or CSV)

It can generate the following output files:

- one file with constant (unchanging) graphic elements (Normalized PDF file)
- several files with variable graphic elements (Normalized PDF files)

All these files are linked together by a standard PPML file.



Grids

The VDP parameters must be specified for each grid individually. To specify the parameters for a grid, select the grid on left. If the repetition layout contains more than one grid, use the + button to add the required number of grids and specify parameters for every grid.

Grid Contents

The expansion task can handle two types of grid contents. 'Identical One-ups' does not include any variable elements. It will fill the empty space on a substrate that is unused by variable data. 'Variable One-ups' carries the design with variable data elements. These two types of grid contents have different parameters. The main difference lies in the absence of a database for identical one-ups.

Following enumeration gives you an overview of the differences:

Identical one-ups:

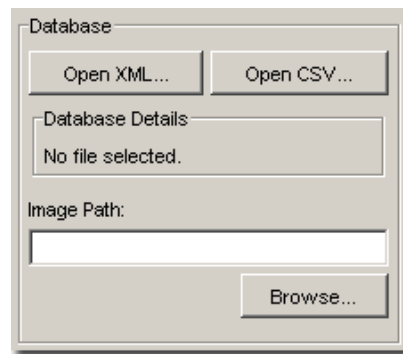
- Total number of one-ups for repetition
- Block size (optional)
- Gap size
- Gap at the beginning of the job (optional)
- Gap at the end of the job (optional)

Variable one-ups:

- XML database – file URL
- CSV database – one or two file URLs + other options (delimiter character...)
- Image Path URL – when Image Marks are used (optional)
- Block size: job, sheet or custom
- Gap at the beginning of the job (optional)
- Gap between blocks (optional)
- Gap at the end of the job (optional)
- Gap size
- Ordering
- Empty one-ups

Database

This parameter defines the database which will be used for expansion. The database is specified either by an XML or a CSV file.



XML Database This type of database is a file in the eXtensible Markup Language format. Clicking 'Open from XML...' opens a file browser, which allows you to select the XML file. Please note that the XML file must be stored within an Automation Engine container.

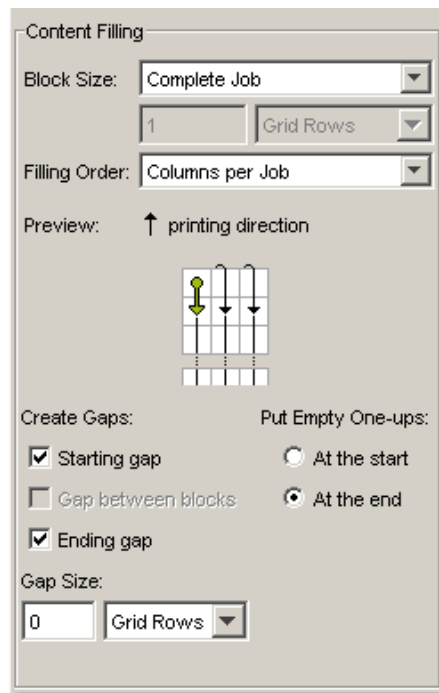
CSV Database A CSV is a Comma Separated Value file. The 'Open from CSV...' button opens a dialog box, where the operator has to specify a URL to the database file, its Encoding and Field Separator options. If field names are not stored in the first line of the database file, you can specify a separate file with the required information by using the Field Names part of the dialog box. An overview of the settings will be displayed in the expansion task window.

Image Path The location specified here is used to fill in the 'vdp_image_path' File SmartName, which might be necessary if the job uses Image Marks. It must contain a valid path which will be used to locate the images in the variable data.

For a description of the Image Path parameter, please refer to Variable Data Printing workflow document..

Content Filling

This part of the dialog box is used to specify how the database values will flow through the variable job.



Block Size

Following options are available:

Complete Job This option defines that database will flow through the job without any interruptions.

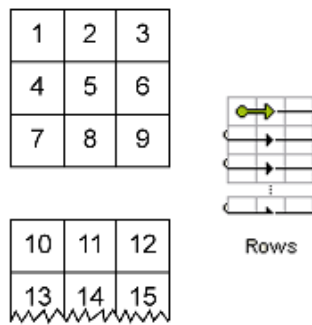
One Frame The size of a frame is equal to size of the document defined in stepX or Plato.

Custom The size of a block is custom-defined. Units are grid rows, frame size, meters or feet.

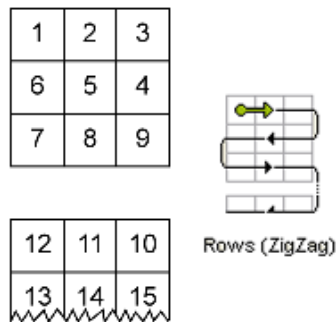
Filling Order

Following options are available:

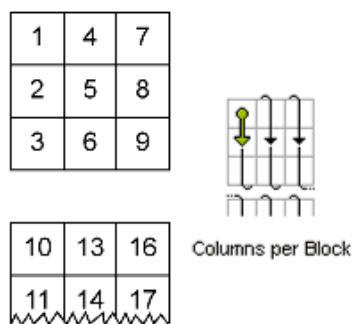
- **Rows:** The numbering is restarted when the end of a row is reached and continues from the left cell of the next row. The starting position is the top left cell of each block.



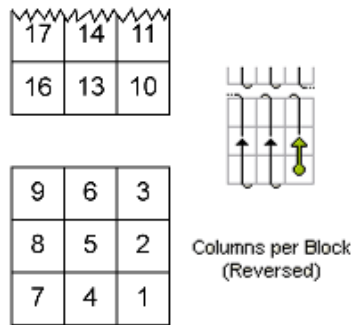
- **Rows (ZigZag):** The numbering is restarted when the end of a row is reached and continues from the cell located below the last cell. The starting position is the top left cell of each block.



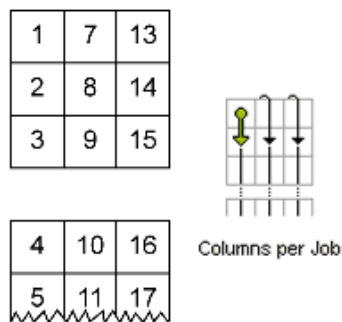
- **Columns per Block:** The numbering is restarted when the end of a column in a block is reached and continues from the top cell of the next column. The starting position is the top left cell of each block. Note: If the Block Size is set to 'Complete Job', then this filling order is equal to 'Columns per Job'.



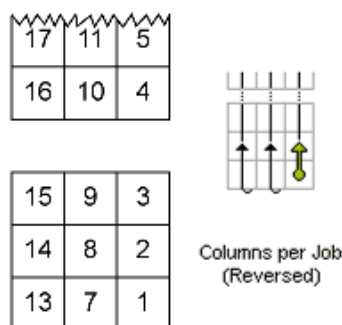
- **Columns per Block (Reversed):** The numbering is restarted when the end of a column in a block is reached and continues from the bottom cell of the next column. The starting position is the bottom right cell of each block. Note: If the Block Size is set to 'Complete Job', then this filling order is equal to 'Columns per Job (Reversed)'.



- **Columns per Job:** The numbering is restarted when the last cell of a column in the job is reached and continues from the top cell of the next column. The starting position is the top left cell of the job.



- **Columns per Job (Reversed):** The numbering is restarted when the last cell of a column in the job is reached and continues from the bottom cell of the next column. The starting position is the bottom right cell of the job.



Create Gaps

Some jobs need to be interrupted after a certain amount of labels (defined by Block Size) and continue after this interruption to the next gap or to the end of the job. No one-ups are printed in the gap area, so the press operator can easily identify the gap as a white area. However, all elements outside the one-up area, such as die cut marks, are printed.

Here you can specify where you want the gaps to be generated. Following options are available:

Starting gap: This gap will be printed first from the press point of view.

Gap between blocks: These gaps will be generated in between Blocks. Note: When Block Size is set to 'Complete job', this toggle is useless.

Ending gap: This gap will be printed last from press point of view.

The 'Gap Size' option specifies the size of the gaps in grid rows, frames, meters or feet.

Put Empty One-Ups

In general, the database size is not an exact multiple of the number of cells in a block. One of the blocks cannot be filled completely with database entries. It will be padded with empty one ups. The toggles 'At the start' or 'At the end' specify where to put the empty one-ups from Filling Order point of view.

Export to GRS file (Deprecated)

Exports a Normalized PDF file to a GRS file.

External references External references have to be converted, you can define what the system needs to do:

- Write external references which are close to the input file to the output folder. All other external references will be written in their original folder.
- Write all external references to the output folder.
- Write all external references to their original folder.

When converting again, you can opt to:

- Reconvert if recently updated: only updated references will be reconverted, all other conversions will be used as they were.

- Use previous conversions: the existing conversions will be used, potential updates will not be considered.
- Always reconvert: all external references will be converted at all times.

External images Referenced files in Normalized PDF job file format will be converted to referenced external files in the GRS file. Indicate which images you want to have converted.:

1. Only images which are close to the input file.
2. All images
3. None

Fail when fonts are not found: when the fonts used in the file are not found the task will fail, when the toggle is not selected, the task will end with a warning

Fail when external references are not found: the task will fail when not all external files were found.

Export to Normalized PDF File (DeskPack Only)(Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

This Task is only useful in a DeskPack workflow, please refer to the 'Adobe Illustrator Client' manual.

Export to VRML File (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

VRML stands for **V**irtual **R**eality **M**odeling **L**anguage. This task can make a 3D model of an Esko native file that has a link to an ARD (Artios CAD) file. The result of the task is:

- A VRML file: a .wrl file. wrl stands for 'world' as in virtual world.
- One or more JPEG files, to build up the visualization of all the panels and all the sides of the 3D simulation.

Note:

- The Esko native file must contain a link to an ARD file.
- The ARD file must contain folding information.

Tip: You will need a VRML viewer to view the result of the task. There are three types of VRML viewers: a plug in, a helper application or a stand-alone application. Several freeware VRML viewers are available on the Internet.

Tip: Supports recto/verso graphics. If the input file has 2 pages, the second page will be placed on the inside of the CAD file.

Note:

Linking a Normalized PDF file to an Artios CAD file can be done:

- using PackEdge,
- using Esko's ArtiosCAD plug-ins in Adobe Illustrator.

You can set **resolution** in ppi, ppm or ppcm and define the JPEG **compression**. Options range from: JPEG Limited, JPEG Low, JPEG Standard Low, JPEG Standard (default), JPEG Standard High, JPEG High and JPEG Extreme.

Note: There is a limit to the kind of resolution that the task will process, internally the output resolution is restricted, so entering a higher resolution does not necessarily result in high resolution output. Except for when the option high resolution images is selected.

JPEG Images:

- Separate Image on each face: creates a separate JPEG file for each face.
- High resolution images: use 'High Resolution Images when there is small text on the design, or other graphics which require a high level of detail. A separate file is created for each face, but if more detail is available, the files are larger.
- Single image: makes a single large JPEG file for each design instead of a single JPEG for each face. Use this option when using the VRML file in other 3D graphics applications.

Creases:

- No creases
- Creases in board color: creases are included in the VRML, they have the same color as the board, the lightning in the VRML rendition shows them as slightly contrasting.
- Creases in pink: turns the creases pink for easy visibility.
- Graphics on creases: wraps the graphics around the creases, this is the most sophisticated option, but it increases the file size by 50 percent.

Animation:

- No animation: creates an unmoving view of the design.
- Forward: runs the animation continuously.
- Forward and Back runs the animation forward, then backward, and repeats.
- With toolbar inserts a toolbar into the VRML file allowing you to manually step through the file, as well as toggle design transparency.

Note: This works only when the ARD file contains frames.

Output units: As there is no measurement ability in the browser window, the options in this group control the granularity of the VRML plug-in's zoom tools. Selecting 'Meters' provide coarse zoom while selecting 'millimeters' provides fine zoom. 'Centimeters' is the default, and should be selected when choosing 'With toolbar' in the 'Animation' group.

Include graphics: this toggle controls the export of graphics. Toggle on for rich detail, toggle off for the smallest VRML files with the most robust performance.

Include corrugations: this option draws corrugations on the edges of corrugated boards. This depends on the ARD file's substrate definition.

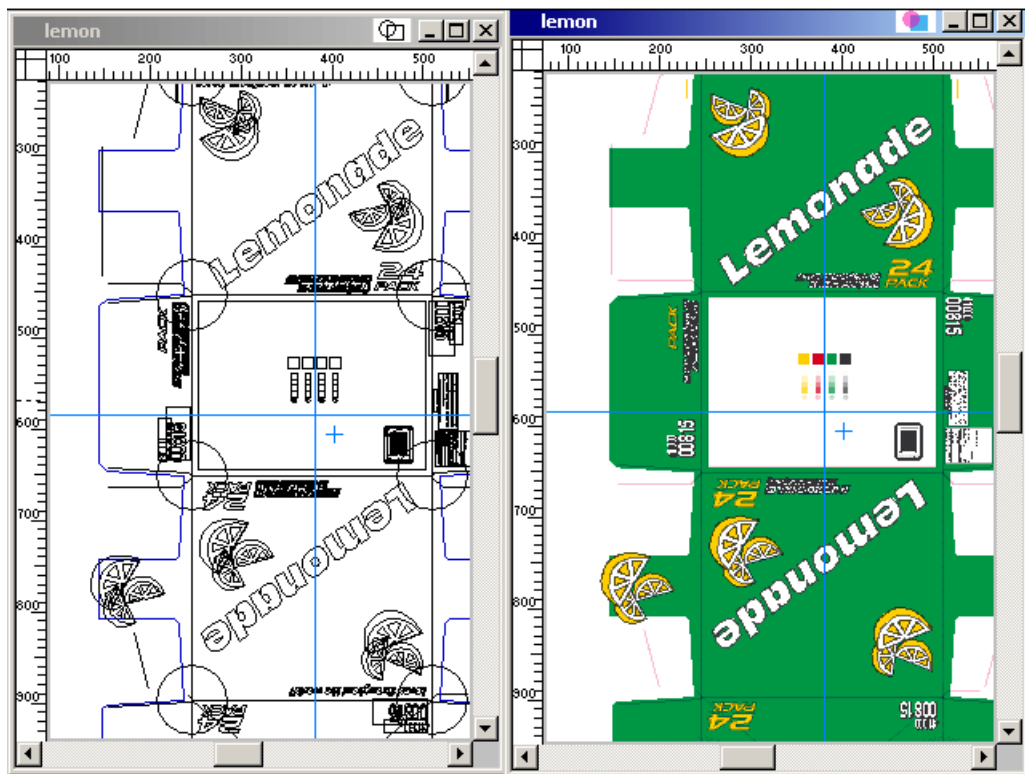
Use compression: this option controls whether the VRML geometric information file is compressed or not. A smaller file is faster to download, but some VRML browser plug-ins do not support compression. The JPEG image files are already compressed according to the JPEG Compression (at the top of the task).

View Zoom Factor: sets the initial field of vision in the VRML file. If you find that animations are bigger than the browser window, try setting the value to a smaller number. You can enter zoom factors between 40 and 250 percent.

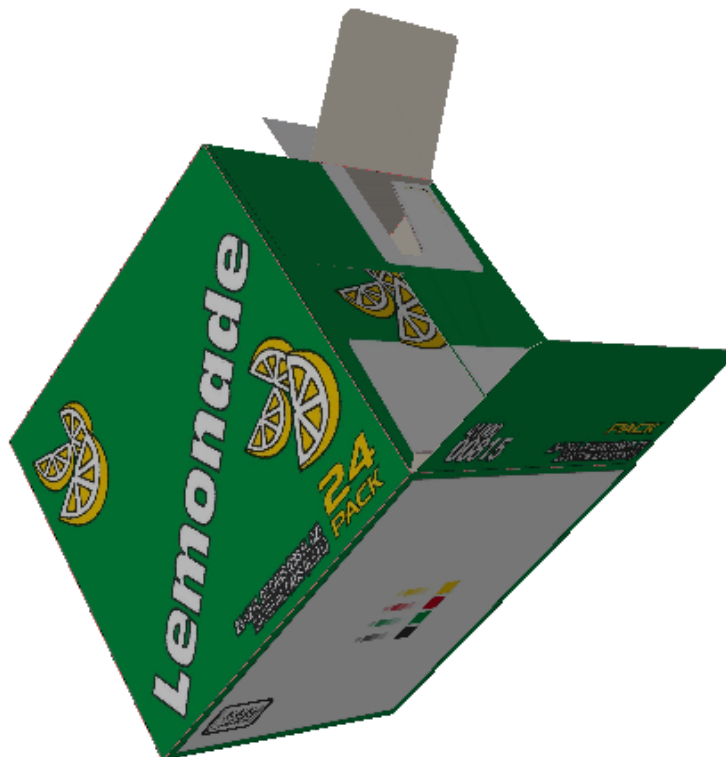
Background color

- No background: no background will be defined, the preferences set in the VRML plug-in will be used.
- Define values for Red, Green, Blue.

For example, this job ([Lemon.pdf](#)) contains a link to an ARD file which contains folding information.



If we run this file through the VRML task, we get a wrl file that looks like this when we view it in a VRML browser:



FastVariants - Check (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

FastVariants is the perfect solution to avoid problems in a double-burn workflow, when one of the plates on the press is replaced for a new run, and the other(s) are identical.

Two double-burn workflows are possible:

1. A neutral version + K variant for each language separately.
2. Different variants as full CMYK.

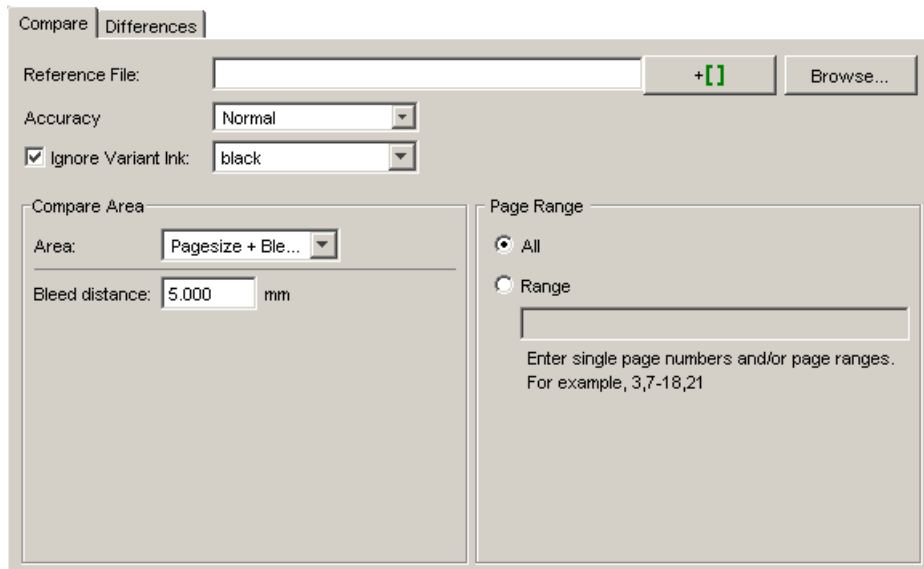
Use the FastVariants - Check task to compare the non-variable inks of two files (CMY in most cases). It checks if there are differences, and shows you where they are situated. If the result of the Check task is OK, the two files are double-burnable and you can use the FastVariants - Split task.

In case the FastVariants - Check task is run on impositions, it compares the page assigned in the background (imposition) layer with the master page of the assigned page in the variant (imposition) layer. By master page is meant the appropriate page from the master (or original if there is no master) layer.

The FastVariant task can also be run from the **Pages View**. A few remarks:

- The variant ink selection follows the use case.
- The FastVariants task also works on page selection.
- When two incoming files or layers are selected, the Check task checks these (appropriately normalized) files.
- Layers under incoming files are by default named as follows:
 - 'original' stands for the originally normalized file.
 - 'master' stands for the master file (result from the Split task).
 - 'variant' stands for the variant file (result from the Split task).
 - 'difference' stands for the difference file (result from the Check task).

Compare



You can either compare files or folders. Depending on what you selected, the Compare option will display '**Reference File**' or '**Reference Folder**'. In case a folder is selected, the FastVariants - Check tool compares the folder file per file on the condition that both folders contain an equal number of files and that the number of pages in the selected files matches the number of pages in their reference files.

Use the **Browse** button to select the correct reference folder or reference file. This is the file or folder you want to compare the selected file or folder with.

You can set the **accuracy** to normal, high, or very high. Depending on this setting, the checking will take more or less time.

Enable the **Ignore variant ink** toggle and select the appropriate ink from the dropdown list. FastVariants will now compare the selected files or folders and will report all changes, except for those in the variant ink. If a folder is selected, it is possible that not all inks appear in the list. If this is the case, enter the ink name manually.

Note:

This option is not relevant when comparing several versions of the same file.

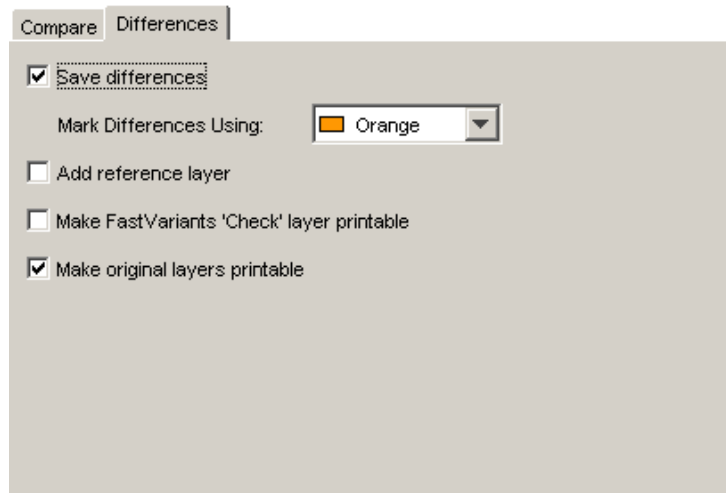
In the **Compare Area** you can indicate what area should be taken into account when the file is being checked. If you set the area to **Job Size**, the whole job will be compared, including margins. **Custom** allows you to set the size and the offset of the area you want to have checked. **Page size + bleed** checks everything that is within the borders of the file. You can specify an extra bleed distance.

Note:

Selecting the correct compare area is important to avoid irrelevant differences being reported, like for example a time stamp in the text of a page mark.

Page Range allows you to indicate the range of pages that need to be checked. When multiple files or layers are selected, the Checker task checks these files.

Differences



Enable the **Save differences** toggle to activate this part. When differences are found, but not saved, no separate layer will be created.

Note: Specify an output path and file name if you do not want the original input to be overwritten.

Select a color from the dropdown list to **mark the differences**.

If the **Add reference layer** toggle is checked, a separate layer with an external reference will be added to the job. When the reference layer is present you can toggle the differences in PackEdge and check the differences visually.

Check the box '**Make FastVariants 'Check' layer printable**' to be able to see the layers containing differences when proofing. If the toggle is not selected, however, then layers containing differences will still be visible in PackEdge, but as they have the 'non-printable' mark, they will not be on the proof.

Make original layers printable

Make original layers printable. Set this mark for the original image.

Note: Make sure that one of the print options is set.

FastVariants - Remove White Boxes (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

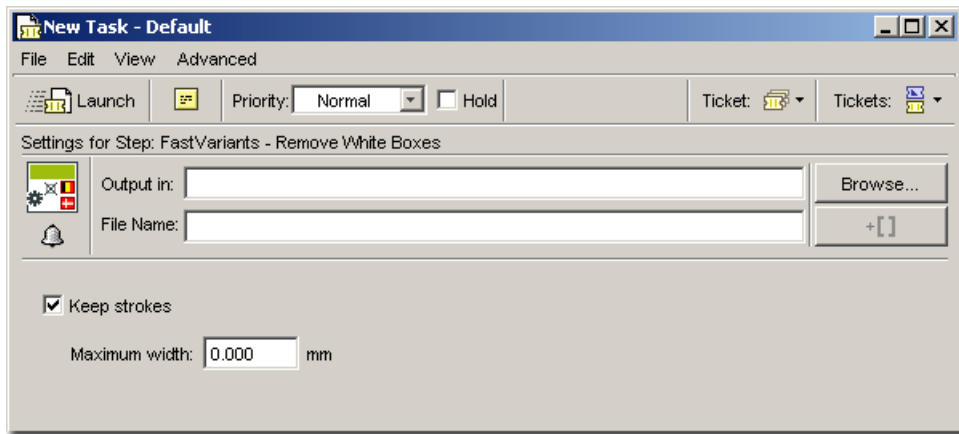
It is common in double-burn workflows that the different versions are supplied as separated PostScript files containing only the variant ink, usually K.

These files can be easily combined with the neutral version by superimposing the two, for example by using layers in FastImpose.

It is usually the case, however, that these separated PostScript files contain white opaque boxes that cause elements to disappear from the neutral file, when the two are put on top of each other.

The FastVariants - Remove White boxes task is specially designed to remove these obtrusive white opaque boxes.

White opaque text will never be removed. White opaque strokes can be kept when the toggle has been enabled.



Enable the **Keep strokes** toggle to keep strokes with a maximum width of x mm or inches. All strokes smaller than the selected width will be kept.

FastVariants - Split (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

Use the FastVariants Splitter task to split the input file into a fixed master and a variant part based on the graphical object database. The splitting is being performed on the basis of the variant parameter ink. In most cases, black will be the variable ink.

The term **Master** refers to the fixed part of the job, **Variant** to the variable part of the job.

When this task is run on impositions in the Pages View, it splits an (original) page assigned in a variant (imposition) layer into a variant and a master page. It replaces the original page by the variant page in the variant (imposition) layer and assigns the master page to the background (imposition) layer, provided that this one is empty.

The FastVariant task can also be run from the **Pages View**. A few remarks:

- The variant ink selection follows the use case.
- The FastVariants task also works on page selection.
- When two incoming files or layers are selected, the FastVariants - Check task checks these (appropriately normalized) files mutually.
- layers under incoming files are by default named as follows:
 - original: original normalized file.
 - master: master file (as a result of the Split task).
 - variant: variant file (as a result from the Split task).
 - difference: difference file (as a result from the Check task).

The screenshot shows a software interface with the following sections:

- Output:** A dropdown menu for 'Variant Ink' is set to 'black'.
- Extract Master:** A checked checkbox. Below it is a text field containing 'm_[File]', a green '+ []' button, and a 'Browse...' button.
- Extract Variant:** A checked checkbox. Below it is a text field containing 'v_[File]', a green '+ []' button, and a 'Browse...' button.
- Page Range:** Two radio buttons: 'All' (unselected) and 'Range' (selected). Below them is a text input field. A note below the field reads: 'Enter single page numbers and/or page ranges. For example, 3,7-18,21'.

Select the **Variant Ink** from the dropdown list. Enable the **Extract Master** toggle if you want to keep the fixed (master) part of the job and have the variant part removed. Enable the **Extract Variant** toggle if you want to keep the variant part of the job and have the fixed (master) part removed. Enable **both** toggles if you want to split the (reference) file into a Master part and a Variable part. Enter an appropriate **file names** for the both **Master** and **Variant**.

Page Range allows you to indicate the range of pages that need to be split.

Fork (Deprecated)

Note: This task is deprecated from Automation Engine. You can use the advanced routing functionality of the Workflow Editor instead. Learn more in [Routing in Workflows](#) on page 1006 in the chapter "Workflows".

Note: The currently available **Fork** Workflow Control is a different feature. Learn more in [Fork and Join](#) on page 1032.

A workflow ticket consists of a number of sequential tasks. Each task is executed after the former was finished, each task involves processing the output file resulting from the former process.

It is not possible to define a workflow ticket in such a way that more than one task involves simultaneous processing of the output file of the former. However, by inserting a Fork task in the workflow ticket these kinds of workflows can be built.

The Fork Task launches other tasks via the tickets as specified (based on SmartNames or on plain ticket names).

When such a task cannot be launched because the name cannot be resolved or the ticket cannot be found, then an internal warning flag is set. Only when this happens for a line that is flagged as critical, the Fork task will end with an error.

The Fork task allows you to launch multiple tasks at once. This makes a few new types of workflows possible.

- Workflows in which the order of the tasks is not important.

I want to create, from my normalized PDF, a small export PDF, a JPEG and a ReportMaker print card PDF. I do not care which one is made first, I just need all 3 of them.

- Workflows in which the required mix of file formats is not possible to combine in 1 single workflow ticket.

My workflow ticket should include both PostScript output and output to FlexRip, but first the PostScript output.

The ticket(s) that a Fork task will launch can now be '**Smart-selected**'.

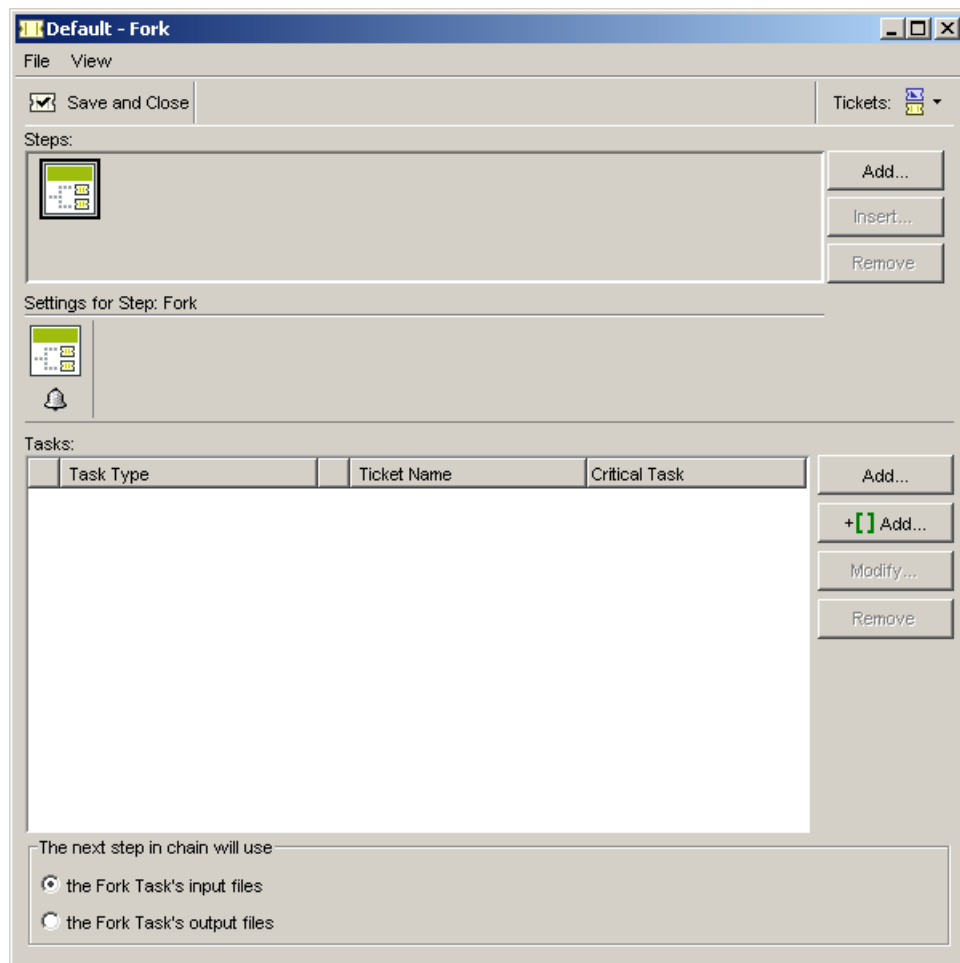
For example If the Export to PDF ticket contains a SmartName that indicates the customer name (as filled in in the Job Setup), then the fork ticket can generate different output, depending on the settings for that customer.

Even the task type can be 'Smart-selected' which enables Automation Engine to intelligently select the correct ticket, based on the type of action that needs to be performed, for example:

- reach an HP or an Epson proof?
- reach a CDI Compact or a CDI Spark?
- create a PDF proof or launch a WebCenter approval cycle?

Defining a Fork Task

1. Select the Tickets View in the Views bar, or in the Go menu. Select the Fork task.

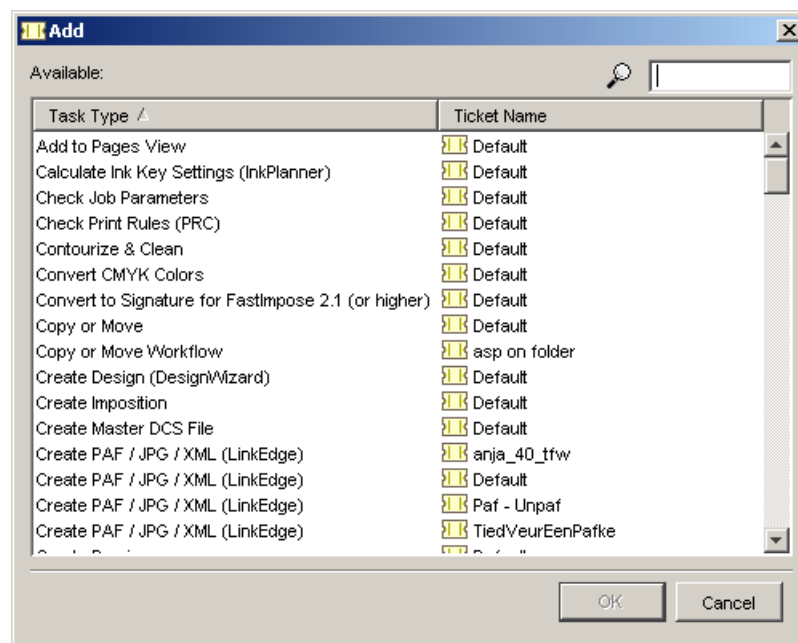


2. Click the task's 'Add' button:



It is possible to add a definition of a ticket by using SmartNames. You can for example define the task type or the ticket name as a SmartName. This can help you to limit the number of tickets to choose from when launching a task.

3. The Add dialog pops up. Depending on which 'Add' button you clicked, different Add dialog boxes will appear. If you do not want to use SmartNames, you can simply select a task type from the list.



If you do use SmartNames, you also select the task type from the list and indicate whether this new fork ticket is a Job ticket or a global ticket. Enter a new ticket name.

Let us assume you have two customers and one needs PDF 1.5 files and the other PDF 1.6 files. You can then create a fork ticket called [export_for_customer] like this:

4. The selected ticket is added to the fork list.

Task Type	Ticket Name	Critical Task
<input checked="" type="checkbox"/>	Export to PDF File	<input checked="" type="checkbox"/>
<input checked="" type="checkbox"/>	Export_for_[Job Customer Name]	<input checked="" type="checkbox"/>

When you have two tickets in your ticket list: `export_for_LabelPrinter` and `export_for_FoldingCarton` and you launch the task “`Export_for_[Job Customer Name]`” in a job where the customer name = `FoldingCarton`, then this task will select the “`Export_for_FoldingCarton`” ticket.

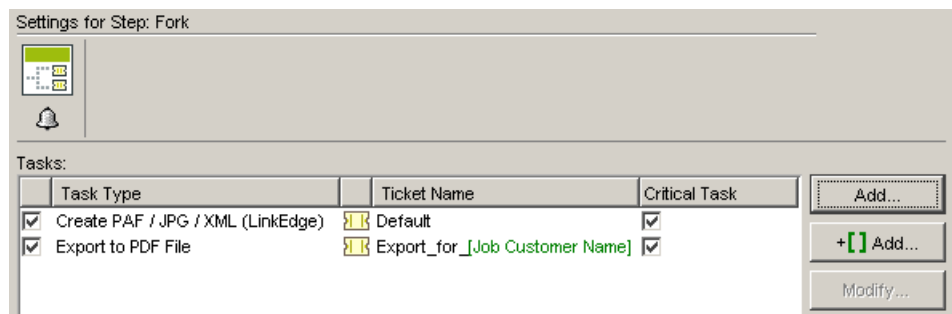
When you launch the same fork task in a job where the customer name = `LabelPrinter`, it will use the “`Export_for_LabelPrinter`” ticket.

This means that using SmartNames in a Fork ticket does not limit the number of tickets as you still need both “`Export_for_...`” tickets (you even need an extra fork ticket).

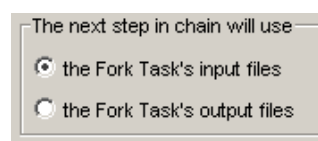
It does offer the advantage that the operator can always launch the same task “`Export_for_customer`”. So the risk of mistakes will be reduced.

5. Add as many tickets to the fork you want.

Hit the 'Remove' button to remove the selected ticket from the list.



6. If you use the Fork task in a workflow ticket, you can choose whether the next step in the workflow will use the fork's initial input files, or the fork's output files.



Mark File (Deprecated)

Note: This task is deprecated since Automation Engine 10. You can use the [Mark](#) workflow control in the Workflow Editor instead.

This task marks the input file. Use it in a chained task ticket, in combination with the [Copy or Move File](#) and / or the [Delete File](#) task.

Process Remote Proofs (Deprecated)

This is a deprecated task. It allowed to proof the files you received on your local printer with the right color management and linearisation etc.. Some customers receive such files via FTP or as e-mail attachments.

Trap - Create Trap Layer (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

This task traps the input file according to the settings specified in the TCP file associated with the input file.

The output of this task is a trapped Normalized PDF file.

This task contains only the output options of the Trap task. All other trapping parameters are stored in the TCP file located in the same folder as the input file.

The Create Trap Layer task has only one tab, and it's the same of the Prepare and Create Trap Layer task's [output tab](#).

See also: [What is trapping?](#)

Trap - Prepare and Create Trap Layer (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

This task offers you all the settings of the PowerTrapper: full automatic trapping.

It selects candidate trap color pairs and executes the trapping. The trapping areas are added in a separate layer on top of the original job.

What is Trapping?

The trapping process is a safeguard against possible (even likely) fluctuations in the printing process. Minor discrepancies in registration, slight paper shifting can cause inks to misregister.

When two abutting colored objects don't meet exactly, you can end up with a printed result like the one below:



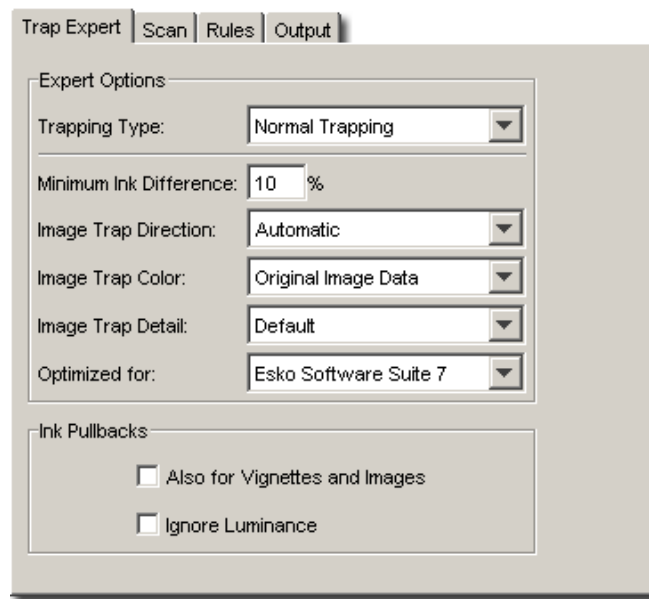
The overlap creates a dark area and the gaps cause an undesirable light area. To prevent these potential discrepancies, a trap can be implemented, causing the original colors to slightly overlap each other:



In general trapping is necessary when there are two abutting color areas that don't share a common ink.

Placing a fine line where they join, made of a color that contains components of both abutting colors, can effectively trap such areas.

Trap Expert



Select the automatic trap expert that will examine the job and select candidate color pairs for trapping. Depending on the expert and its associated parameters, different color pairs may be selected for trapping.

Two Trapping Types:

- **Normal Trapping:** This trapping expert selects abutting color pairs, which are likely to cause registration problems like ugly light gaps, and unwanted halo effects. The expert automatically chooses the most appropriate trap direction based on the relative luminance of the adjoining colors. Lighter colors are generally trapped into darker colors to minimize the visual effect of the trap.
- **Reverse Trapping:** This trap expert prevents overprint of abutting color areas with different inks by selecting these color pairs to be trapped with a white knockout (erasing) trap. The white knockout trap is put on top of the lighter of the two colors. This expert is useful to prepare a job for Dry Offset printing (for example metal beverage cans) where overprinting of inks is not allowed as the inks would contaminate each other on the blanket.

Minimum Ink Difference: Trapping is applied to two adjoining colors when one of the following two rules hold:

1. The two colors each consist of at least one ink but they have no common inks.
2. The two colors have common inks and there is a difference larger than the minimum ink difference percentage for at least two different inks that make up these colors. These differences in ink amount must be in opposite directions (see also example 3 below).

The minimum ink difference defined in the second rule can be altered by changing the value in the 'Minimum ink Difference' input field. The default minimum ink difference percentage is 10%.

Image Trap Direction

- **Automatic:** Determine the most suitable trap direction automatically by comparing the average lightness of the CT image with the lightness of the abutting area and trap the lighter color into the darker one. (default).
- **Trap Images into Linework:** Always trap CT images under adjoining line work.
- **Trap Linework into Images:** Always trap line work under adjoining CT images.

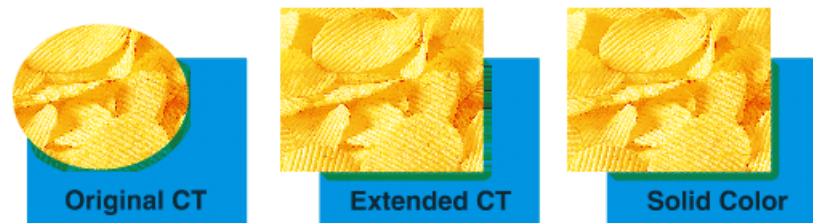
Image Trap Color Specifies the color of CT traps in those cases where the CT should be spread into an adjoining color area.

- **Original Image Data:** Use the trap shape as a clipping mask on a copy of the original CT image placed in overprint. This only works if there is additional image data outside the original CT mask (default).
- **Extended Image:** Same as above but the CT image is automatically extended (extrapolated) along its top, right, and bottom and left edges to ensure the trap gets the expected color even if there is no image data outside the original CT mask.
- **Solid Color:** Use a solid color as trap color instead of a (variable) CT. A suitable solid color is computed by averaging the CT color along the boundary between the abutting color areas.

Image Trap Detail

This option is only used in a DeskPack workflow: image traps are rendered to linework.

- **Coarse:** is faster to process but will generate coarser results.
- **Default:** is a trade off between speed and detail.
- **Fine:** nice results, but larger files and slower to process.



Optimized for: The “Optimized for” option has three possible values that each correspond to a different way to represent the traps in this particular case.

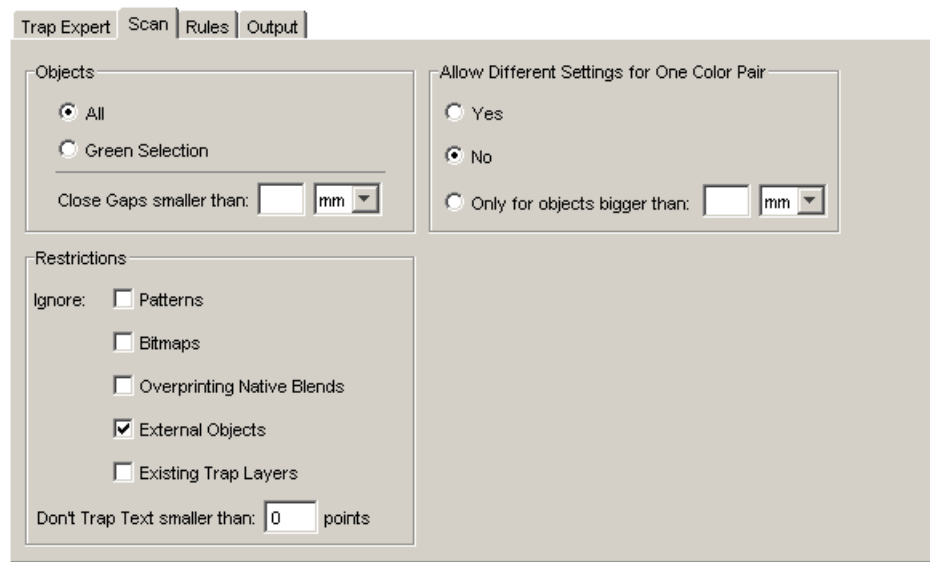
- **Scope 2 and lower** When this option is selected, the trap will be created as a stack of objects. There will be as many objects in the stack as there are colors/objects contributing to the transparency blend. Each object in the stack will have the shape of the trap and the color of one of the transparent objects involved and will be put in blend mode Darken. Note that this does not result in the correct trap color in all cases. The reason is that a stack of trap objects each with blend mode Darken does not accurately represent the result of the original blend of objects that can each have a blend mode different than Darken. The only advantage of using this mode is that it will not create any new CT images and that it is 100% compatible with GRS and older versions of the trapping software (Scope 2 and lower).
- **Scope 3** When this option is selected, the trap will be created as a new internal raster image that is obtained by rasterizing the original stack of objects/colors that contribute to the transparency blend. The raster image is put in blend mode Darken and the trap shape is used as a clip mask on the image. The advantage is that (apart from resolution limitations) the trap color is always correct. A disadvantage is that new images are created to represent the traps, and that increases the document's filesize. Although these images are stored as internal (embedded) images, such images will still become externally linked if the file is later exported to the legacy GRS format. Therefore, if a GRS-based workflow is still used and/or needed, the use of this option is not advised.
- **Esko Software Suite 7 and higher** When this option is selected, the trap will be created as a transparency group containing a stack of trapping objects corresponding to the colors and blend modes of the original objects taking part in the transparency blend. The group as a whole is put in blend mode Darken. Note that this is different from option 1 “Scope 2 and lower”, where each of the objects in the stack were put in blend mode Darken. This option uses the new group

transparency feature. This always results in the correct trap color without the need to create any new raster images. It is advised to always use this option unless there is a frequent need to export documents to GRS or to share files with sites that use Scope 2 or 3 versions. In those cases this option shouldn't be used since the transparency group construct will have to be flattened on export to GRS or import in the older Scope system, which could result in performance issues.

Ink Pullbacks

- **Also for vignettes and images:** normally the system only takes solid colors in account when it comes to ink pullbacks. If you want to, you can force the system to perform ink pullbacks on everything, also on objects overlapping vignettes and images.
- **Ignore luminance:** the system takes certain aesthetic precautions while trapping. It takes luminance into account when calculating what ink (if any) to pull back. You can disable this behavior by selecting this toggle.

Scan



During the scan phase the trapping software looks for areas in the job where trapping could be applied.

To do so, the objects to be examined will be converted to a flattened graph. In the flattened graph, the areas are searched for places where two regions are approximating each other.

For each region, the accumulated color is taken into account. This accumulated color is the result of applying all the paint and transparency settings of the objects contained in the given region.

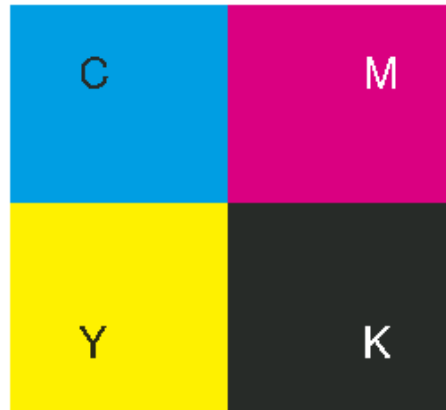
The accumulated colors of the two regions are then checked for a chance to trap.

If there is a chance between the two colors to trap, a color pair is generated, and the place of proximity of the two regions is stored as a 'color pair hit' with the given color pair.

There are two different kinds of color pair hits: **direct** hits and **indirect** hits.

- A **direct hit** of a color pair for a common edge between two regions occurs where one of the regions has one color and the other region has the other color of the color pair.

In this case, there is a direct contact between the two colors of the region. In the picture below there is a direct hit for the color pairs (C, M); (Y, C); (M, K) and (Y, K).



- An **indirect hit** of a color pair occurs at places where the two regions are geometrically close to each other, but not with a common edge.

This is the case at places where the regions touch each other just at one point, where four or more regions touch each other.

In the picture above there is an indirect hit for the color pairs (C, K) and (Y, M) at the point where the four boxes touch each other.

The scan tab contains options to fine-tune this scan action.

Objects:

- **All:** Scan all objects in the job when looking for trap color pairs (default).
- **Green Selection:** Only scan objects that have been selected in the LineWork editor (in the green user selection). If this option is active but no green selection is found in the job, the entire job will be trapped instead.
- **Close Gaps smaller than x mm:** Some input jobs contain very small unintentional gaps between neighboring objects, preventing the correct trapping of these objects.

Although it is **better to clean up such jobs before trapping**, the trapping engine contains an algorithm that tries to ignore these gaps automatically.

When you enter a value (in **mm**, **cm** or **inches**), the algorithm will try to ignore gaps having a width smaller than the value you entered. However, this gives no guarantee that all small gaps will be ignored.

The gap range is 0.0mm (meaning the algorithm is not used) to 0.1mm. The default value is 0.005mm.

Note: If you set the distance too high, the scanning process will become slower, and small parts of the job may be removed.

Restrictions: In some special cases the trapping software can safely ignore certain object types. This knowledge can substantially speed up the trapping. The following object types can be ignored:

- **Patterns:** Ignores native fill and stroke patterns. Prior to trapping, fill patterns are converted to contours. Fine patterns may result in a huge number of contours. In some cases this can cause memory shortage and/or long trapping times.
- **Bitmaps:** Ignores bitmap pixel data. This type of data is contoured before trapping, which may slow down the trapping action depending on the contents of the bitmap(s). Bitmaps (LPs) with lots of small isolated islands will typically take a longer time to trap.

- **Overprinting Native Blends:** This option is only used in a DeskPack workflow. Certain native Adobe Illustrator blends cause performance issues, since they have little or no actual effect on the trapping process you may ignore them.
- **External Objects:** Ignores references to external Normalized PDF, legacy GRS, LC, and LP files. If this toggle is off then the external references are degraded to internal objects prior to trapping. By default external references are not trapped because in some cases, when the input job contains many references to external files, this can cause memory shortage and/or long trapping times.
- **Existing Trap Layers:** Ignores any existing trap layers contained in the job. This option is useful to re-trap a job that was already trapped (with different trapping parameters).

Note: When you indicate an object type to ignore, it will be ignored completely as if it did NOT exist in the job. Also, the objects that are ignored will NOT truncate trapping areas resulting from other objects. Use these options with care and only to speed up the trapping of specific types of jobs.

- **Don't trap text smaller than x points**

Small type must be kept as sharp as possible. Trapping may blur the sharp edges of small characters making them harder to read.

The Trapping engine has an option to prevent trapping of text below a specific point size (1 point equals 1/72 of an inch). Edges of text characters below the given point size will not be selected for trapping.

This option only affects the trapping of native text objects not that of contoured text (the default value 0 points, meaning trap all text).

Allow Different Settings for One Color Pair: The trap expert can generate a different trap action for different hits of the same color pair.

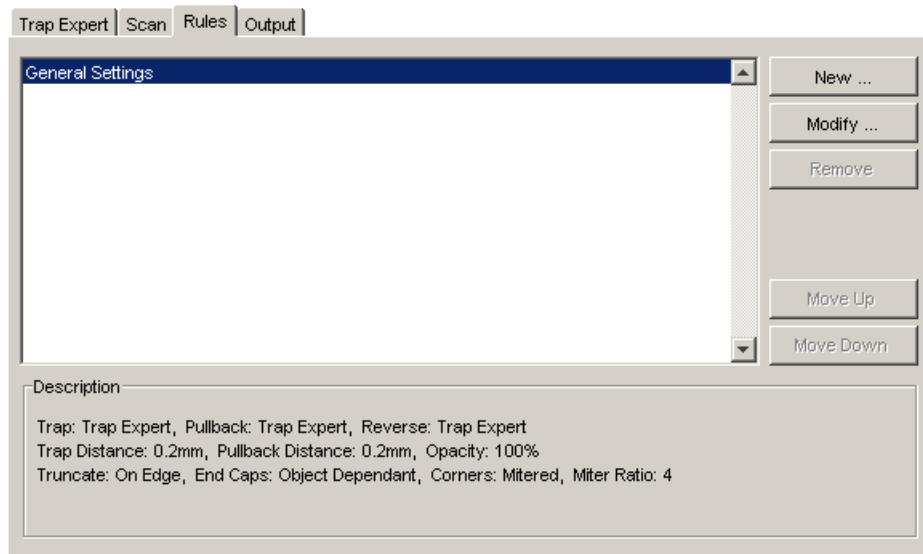
Select **Yes** to allow different settings for the different hits of one color pair. This is the default value.

Select **No** to force the trap expert to apply one identical action to all the hits of the same color pair.

Select **Only for objects bigger than** to force the trap expert to apply one identical action to the hits of one color pair belonging to small objects, and to allow different settings for the hits belonging to the bigger objects.

Use the size parameter to select the size of small objects.

Rules Tab



The rules tab allows users modify the default trap distance and shape settings and to specify one or more custom trap color pair rules. Custom rules allow users to forbid specific types of trapping or to specify a different set of trap settings (distance and shape) depending on the type of the abutting colors of the trap pairs.

Note: Custom trap color pair rules are an advanced feature that should only be used when trapping with the general settings does not give you the desired result.

How do Trap Color Pair Rules Work?

While scanning the job, the selected trap expert builds an internal list of abutting color pairs that are suitable candidates for trapping.

Each of these color pairs has an associated set of trap parameters like the trap distance (width of the trap area) and other parameters controlling the shape of the trap.

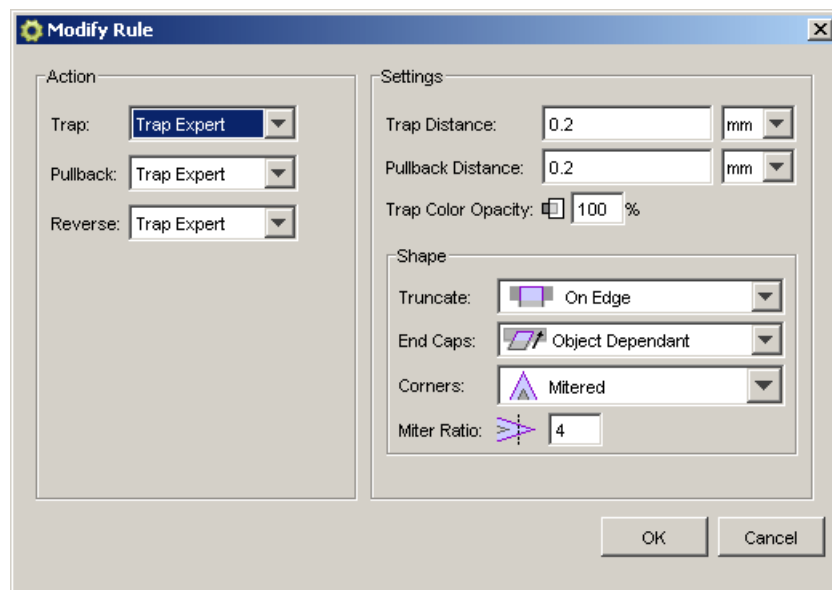
By default a single set of default parameters ('General Settings') is associated with each of the color pairs. If the general settings are not appropriate for a specific type of color pair (for example CT to line work trapping), a rule can be added to specify a different set of parameters for this type of color pair.

For example a rule can be created to trap spot colors into spot colors with a specific distance different from the default.

In that case only the pairs of colors containing PANTONE inks will be trapped with that distance. All other color pairs will be trapped with the default distance specified in the default settings.

Modifying General Settings

Select the default settings in the rule list by clicking on 'General Settings' and press the modify button. The modify panel will appear:

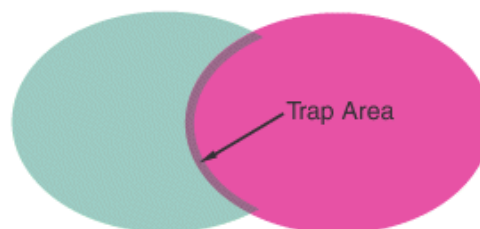


Action Group

This panel allows you to specify the actions that will be performed for each color pair (trap and/or pullback) and the settings that will be used to create the trap area (distance, color and shape).

The radio buttons in this group specify the action performed on each candidate trap color pair. Two actions are possible: **Trap** and **Ink Pullback**:

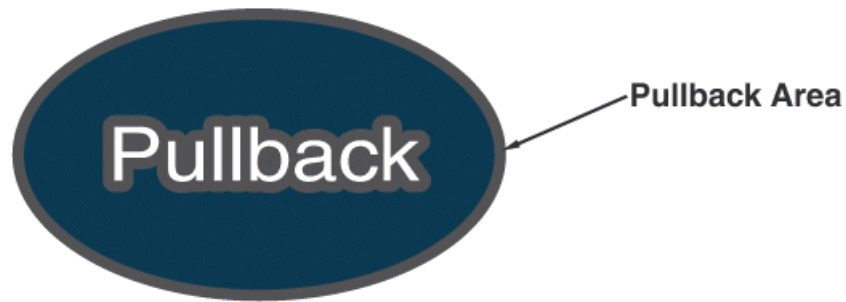
- A thin **trap** area will be created between the abutting colors where the colors will overprint each other. Traps are also known as Frames or Grips.



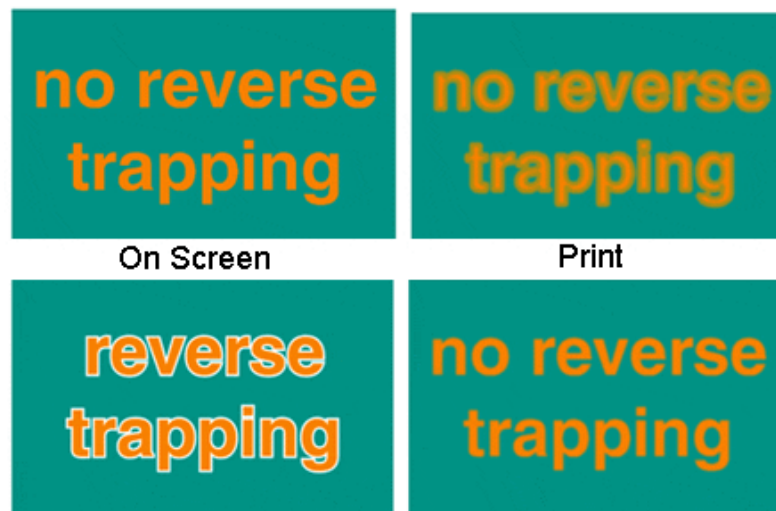
Options:

- Force: Force a trap area for each color pair.
- No: Forbid creation of trap areas for each color pair.
- Trap Expert: The selected trap expert automatically decides for each of the color pairs if they are to be trapped or not. (default).
- Ink **Pullback** is an action that keeps away all but the darkest inks of a color area composed of multiple inks that borders to a very light background. This action prevents that individual inks from the composed color will become visible on the background due to registration errors during the printing process. Ink pullback is also known as Cutbacks or Keepaways.

Object with 100%C,100%K on an empty background. 100%C is pulled back, a fine line of 100%K remains:

**Options:**

- **Force:** Force a pullback area for each color pair where this is possible. At least one of the two colors should be composed of multiple inks.
- **No:** Forbid creation of pullback areas for each color pair.
- **Trap Expert:** The selected trap expert automatically decides for each of the color pairs they are to be pulled back or not. (default).
- **Reverse** prevents overprint of abutting color areas with different inks by selecting these color pairs to be trapped with a white knockout (erasing) trap.



**Options:**

- **Force:** Force a Reverse area for each color pair where this is possible.
- **No**
- **Trap Expert:** The selected trap expert automatically decides for each of the color pairs (default).




Settings Group

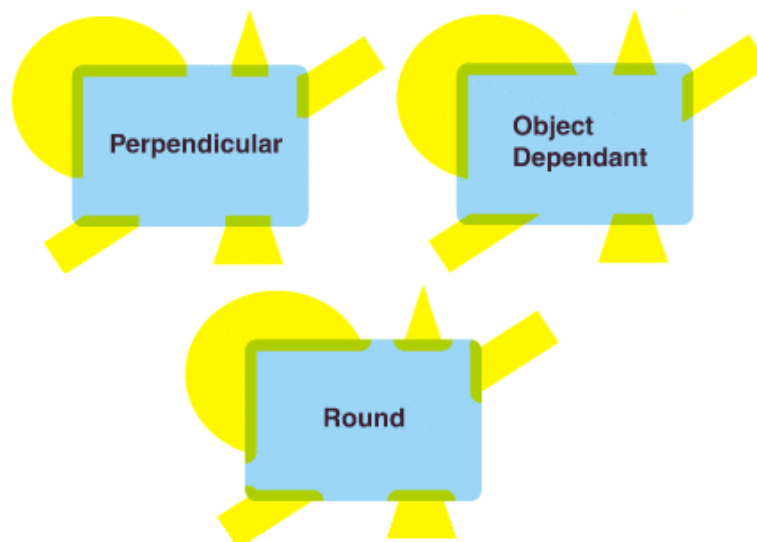
- **Trap Distance:** Specifies the width of the trap area in millimeters or inch.
- **Pullback Distance:** Specifies the width of the pullback area in millimeters or inch. If a color pair is both trapped and pulled-back, this distance will be used.
- **Trap Color Opacity:** Instead of spreading the full color, the trap color can be reduced to a percentage of the original color. This options can be used to create a lighter trap color. (default 100%, spread full color)

Shape




- **Truncate** The trap area is truncated to avoid that it exceeds the edge(s) of the color area under which is trapped. There are two different truncation modes:
 - On Edge:  The trapping area is truncated on the object edge, even if a larger distance has been defined. This is to avoid that the trapping area of an object exceeds the edge of the object under which is trapped. This is the default truncation mode. (default)
 - On Center:  The system recalculates the trapping distance for the areas where a trapping area comes too close to the contour of an object. In that case, the trapping area stops halfway the object. In all the other areas, the complete trapping distance is applied.

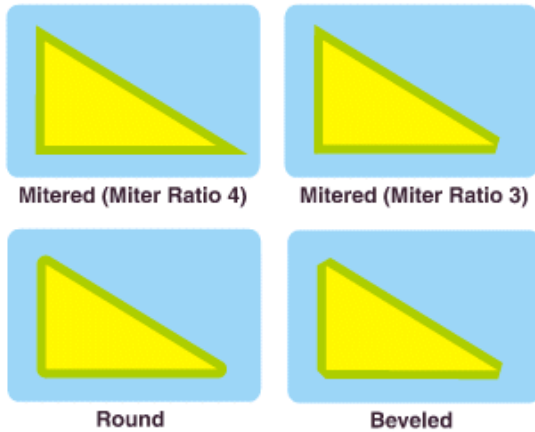



- **End Caps** This option specifies how the ends of an open trap area are constructed.
 - Perpendicular:  When this mode is selected, the trapping areas are placed at right angles relative to the boundaries of the abutting color areas.
 - Object Dependent:  When this mode is selected the trapping areas are a logical continuation of the contour of the object that is spread. (default)
 - Round:  When this mode is selected the trapping areas are ended with circular arcs. This mode is typically used in combination with white knockout or ink pullback.



- **Corners** This option allows you to change the way sharp corners in the trap areas will be handled.

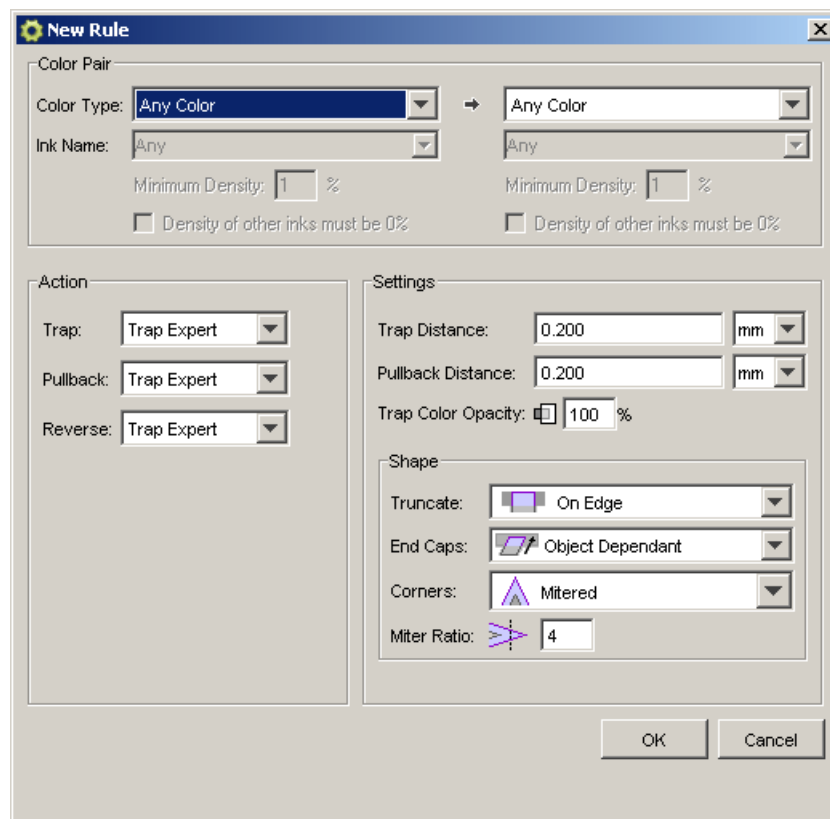
- Mitered:  If the distance from the base of the trap to the corner point is larger than the miter ratio multiplied by the trap distance then the corner will be beveled otherwise it will remain sharp. See also miter ratio option. (default)
- Round:  A round cap will be placed at all corners.
- Beveled: 



Miter Ratio  Determines when a sharp corner is beveled. If the Distance from the base of the trap to the corner point is larger than the miter ratio multiplied by the trap distance then the corner will be beveled otherwise it will remain sharp. This option is only available when mitered corners are selected (default value 4).

Adding a New Rule

To add a new rule, press the 'New' button on the 'Rules' tab. The 'New Rule' panel will appear:



The top section of this panel defines the type of color pair for which this rule applies. A rule will only 'match' with color pairs that have the same type of trap and boundary color as specified in this color pair template. The bottom section specifies the trapping actions associated with this type of color pair and the parameters of the trap area.

Color Pair Group Contains a description (template) of the color pair. On the left is the description of the trap color and on the right is the description of the color that is being trapped into (the boundary color).

Both color templates are composed of the same fields:

Color Type

- Any Color: Matches to any color (wildcard).
- Process Ink: Any color containing at least one process ink (CMYK).
- Spot Ink: Any color containing at least one PANTONE ink or at least one designer ink.
- Image: Any contone Image.
- Vignette: Matches to any vignette color
- Empty Background: The empty background. Can be used on the left side to match with color pairs for which ink pullback will be performed.
- Registration Color: The color that is composed of all inks used in the job used at 100%.
- Opaque Ink: Any color containing at least one opaque (metallic) ink.

Ink Name When the color type is Process or Spot, an optional ink name may be specified. In that case the color template will only match with colors containing the ink with that specific name (default 'Any').

Densities When the color type is Process, PANTONE, Designer or Opaque, additional constraints on the densities of the specified ink (type) may be imposed.

Minimum Density: the color template will only match with colors that contain at least the given percentage of the specified ink.

Density of other inks must be 0%: If this toggle is on, the color template will only match with colors that do not contain any other (type of) inks than the one specified. If the toggle is off, the template will also match to colors containing other inks. (default off)

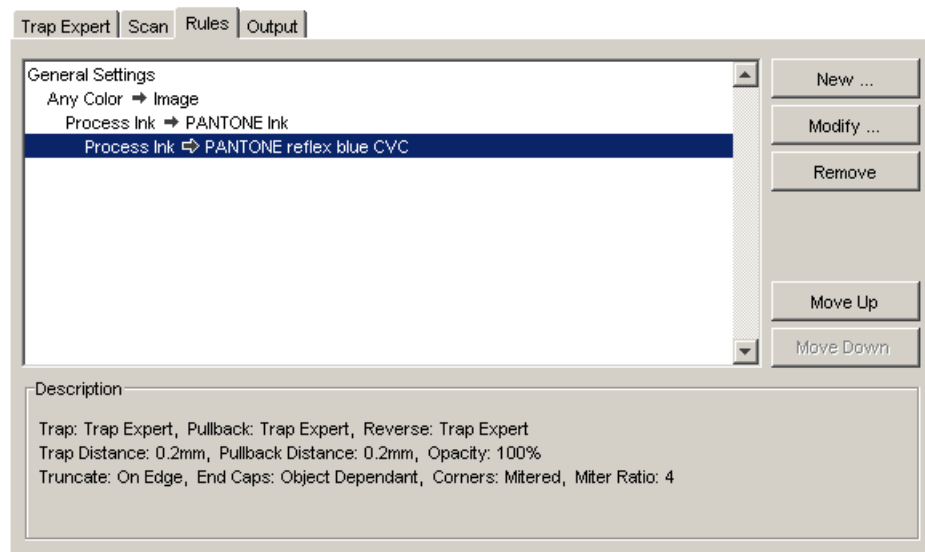
The **Action** and **Settings** groups are the same as those in the '*Modify Rule*' panel. Here you can specify which actions to take for the specified type of trap color pair and what parameters to use to create the trap areas.

Finally press OK to add the new rule to the list of rules in the rules tab. Under the rules list a textual summary is give of the settings associated with the selected rule. Differences with the default settings are indicated with blue text.

Rule Priority

Rules have a priority associated with their position in the rules list. The system always evaluates the list of rules in a bottom-up fashion. The system first tries to match each color pair to the last rule in the list. If this rule's color pair template matches the color pair, then the evaluation stops and the settings associated with the rule are used for the color pair. If the rule does not match then the next rule (the one above the last one in the list) is tested.

This process continues until a match is found or the top of the list is reached. If no matching rule was found, the default settings are applied to the color pair. The "Default Settings" are always at the top of the list and contain the parameters associated with all color pairs that do not match to a more specific rule.



For the list above the color pairs will first be tested for a match with the rule 'Process Ink >PANTONE reflex blue CVC', then with the broader rule 'Process Ink >PANTONE Ink' then with 'Any Color >Image', until a match is found. If no match is found the default settings are applied to the color pair.

- **Move Up / Down buttons:**These buttons can be used to change the position and the associated priority of the selected rule in the list. The 'General Settings' cannot be moved. They always remain at the top of the list.
- **Remove button:** This button removes (deletes) the selected rule from the list of rules. The 'General Settings' cannot be removed. They always remain at the top of the list.

When to Use Rules?

Trap color pair rules are an advanced feature of the Trapping technology, intended for experienced users. The Trapping engine will deliver high quality trapping results for most jobs without any customized rules if suitable default settings are selected.

There are cases, however, in which the automatic algorithm based on the relative darkness of the colors to be trapped is not perfect. In other cases a single set of trap parameters (distance, shape, color) independent of the type of trap colors may be too restrictive. Customized rules offer a mechanism to specify exceptions.

Below a number of typical examples (use cases) are given to demonstrate the use of color pair rules.

1. Different widths for different types of traps. Depending on the type of the abutting colors a different trap distance may be required. Sometimes larger trap distances are used for spot inks since there are usually no other inks to help cover leaks on the press. Process tints typically only need a very small trap. A larger distance may be required when trapping into a dark color (for example containing 100% black).

To set a specific distance when trapping a CT into another color, add a new rule: 'CT' > 'Any Color' with the required trap distance. If colors trapping into a CT should also be of this distance then a second rule 'Any Color' > 'CT' should be added.

2. Prevent Trapping or Ink Pullback in specific cases. Depending on the type of the abutting colors, trapping and/or pullback of ink may be undesirable. In such cases a rule may be created with the 'Trap' and/or 'Pullback' action set to 'No'.

To allow ink pullback only on colors containing 100%K: Set 'Pullback' to 'No' in the 'General Settings' and add a rule 'Empty Background' > '100% Process Black' with 'Pullback' set to 'Trap Expert'. This rule will allow ink pullback on composite colors containing 100% process black but forbid pullback on any other composite color.

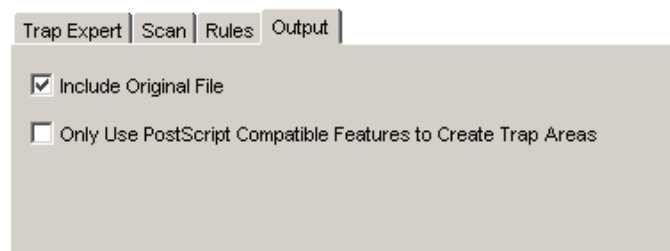
3. Force a specific Trap direction. The trapping engine determines the trap direction based on the relative lightness of the abutting colors. If the colors are of (almost) equal lightness, then the algorithm selects an arbitrary direction. In some cases this direction can be selected with a rule.

By default the color 100%C is trapped into the color 100%M. To change the trap direction, add a new rule '100%M' ® '100%C' and set the trapping action to 'Yes'. This will force the pair to trap in the specified direction. Activate the toggles: 'Other Inks must be 0%' to prevent other color pairs containing cyan and magenta from being affected by this rule.

4. Specify a different Shape and/or Truncation mode for certain color pairs.

Only apply 'On Center' truncation when trapping under a 100% black key line and 'On Edge' truncation on all other color pairs. Use round corners on some types of traps and mitered corners on others.

Output



Include Original File

By default the trapping engine generates a copy of the original GRS job containing one additional layer which contains all the trapping areas. When this option is off, the resulting GRS will only contain the new layer with trapping areas. This layer can later be placed on top of the original job later using the Place option in the LineWork editor. This allows you to add objects that don't need trapping (for example bar codes and registration marks) in the original job and add the trapping layer as a final step. (default: On)

Only Use PostScript Compatible Features to Create Trap Areas

By default the trapping engine uses proprietary paint operators (BG Overprint and in some cases Reverse Overprint) to obtain the correct colors in the trap areas. Use of these special overprint operators gives the best results in an Esko workflow (for example output using an Esko RIP). However, this default output model is less suited when the trapped job should finally be output on a PostScript device or as a PDF file. In that case it is best to activate this option to prevent long conversion times when exporting the trapped file to PostScript or PDF.

Note: Trapping times may be longer when this option is active. In some areas where multiple traps overlap each other, the resulting color may be undesirable. Therefore it is advised to use this option in combination with Centerline truncation (as it prevents overlap of trap areas).

Trap - Prepare Trap Pairs (Deprecated)

Note:

This task is deprecated. Read more in [Deprecated Tasks in Automation Engine 12.1](#) on page 527

This task scans the file and suggests a list of candidate trap color pairs.

The 'suggest' task has the same options as the [Trap task](#) except that it has no output options. The output of this task is a trap color pair information file. This file contains all information gathered by the automatic trap expert during the scan phase. This file has the same name as the input job and has the extension '.tcp'. It is placed in the same folder as the input file. The settings associated with the color pairs in the TCP file can be edited in the LineWork editor. Jobs that have been scanned by this task (and as a consequence have an associated TCP file) can be submitted to the 'Create trap layer' task.

2.8 The Automation Engine Viewer

2.8.1 Introduction

What is the Automation Engine Viewer?

The Automation Engine Viewer is a comprehensive, powerful and integrated Viewer and QA tool. It can easily be integrated in your Automation Engine workflow, to view, check and compare graphic production data. It can be launched directly from within the Automation Engine Pilot, on any Mac or Windows computer in your network. Anywhere you can install and log on to an Automation Engine Pilot you can start the Automation Engine Viewer (DTP, prepress, QA department, CSR desk...).

You can use as many Automation Engine Viewers as you have Pilot connections. Opening the Automation Engine Viewer does not allocate an extra Pilot connection. It is even possible to have two or more Automation Engine Viewers on one workstation by opening a second Automation Engine Pilot (Pilot - File menu - New window) and opening a file in the Automation Engine Viewer from there. If you also use the Esko Bitmap Viewer, you will notice that the interface and shortcuts are nearly identical.

The **Viewer** offers you

- Reliability: Showing the data in a correct way. At best in the same way the RIP / Proofer will see them.
- Detail: Unlimited zoom, high detail measuring, measuring traps and exact distances (die objects).
- Speed: Opening files swiftly, even when it concerns large files. Fast zooming.

Supporting many file formats, Comparing and more



The **Viewer** supports many different file types: PDF, AI, ArtPro, images and even bitmap RIP data. For a full list, see [Supported File Types](#) on page 580.

You can even compare with other files, even when they are a different file type and even when they have a different size! Learn more in [Comparing Files](#) on page 81.

Basic use

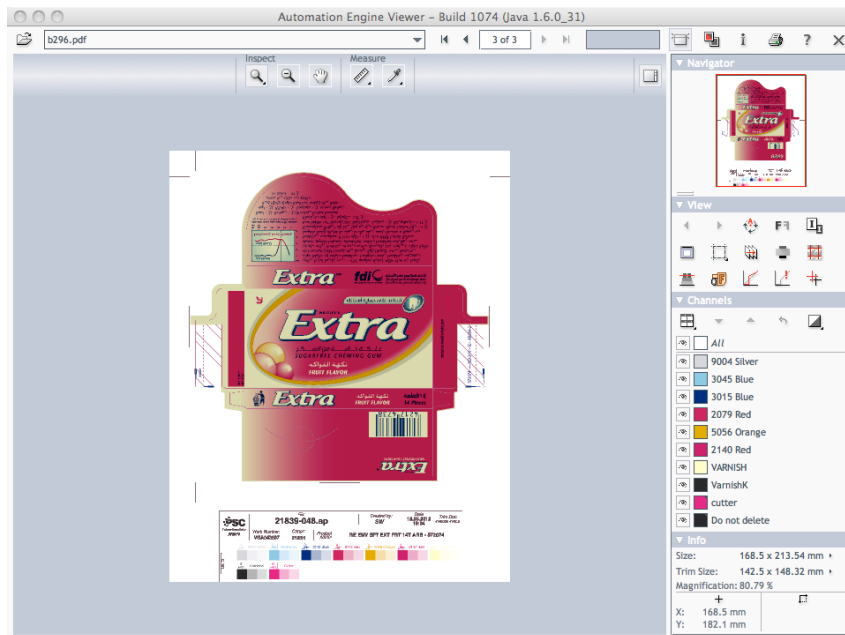
To open files in the **Viewer**,

- click the word **View** next to your file.

Name /	View	Type	Size	Date
 Celebrity.pdf	View	PDF File	41MB	07/04/10...
 CMYK.pdf	View	Normalized...	32KB	05/01/10...
 Dolly_May.pdf	View	PDF File	2.6MB	06/04/10...

- right-click your file and select **View...**
- select your file and go to **File > View** .

Here 's a first look at the Viewer:



You can zoom in on the file, measure distances and densities, view separations individually or together, etc.

Note: The Viewer has its own local On Line Help. Start it from the question mark icon in the top bar.

The concept of Prepared View Data

Reliability is a must. Detail can also not be compromised, it is always offered. The challenge lies in opening very large files quickly. That is why the Automation Engine Viewer offers 2 view modes: with or without Prepared View Data.

- In case you do not prepare View Data, the server will stream the data to the Automation Engine Viewer. When a large file has to be opened or compared or when the server is very busy, a delay can be noticed. Deep zoom detail mode is immediately available.
- In case you do opt to prepare View Data in advance, then the Viewer will show you the prepared View Data instantly. Zooming in will of course require object data streaming from the server, but then for a smaller area. For more information about creating View Data, see the task [Prepare for Viewer](#) on page 415 in the Reference Guide.



Attention: If the **User Access Right** for **deep zoom** is not enabled (read more in [Defining User Access Rights](#) on page 33), then

- you can only view 'prepared' files.
- your zoom is restricted.

2.8.2 Supported File Types

The Automation Engine Viewer supports the following **file types**:

Graphic files, both with and without prepared View Data

<p>Esko normalized data:</p> <ul style="list-style-type: none"> • Esko PDF • Esko PDFPLA • Legacy Esko GRS • PLA • Legacy Esko STA • EPF (DeskPack) 	<p>Non-normalized graphic files:</p> <ul style="list-style-type: none"> • PDF • PostScript • EPS • ArtPro • AI (Adobe Illustrator) • For Adobe Illustrator 'EPS' file types, the same restrictions are valid as in PackEdge.
<p>Images, both with and without prepared View Data</p> <ul style="list-style-type: none"> • Esko CT • Image EPS • DCS/EPS • PSD • TIF (< 600 ppi) • JFIF • LP 	
<p>Post-RIP data only with prepared View Data</p> <ul style="list-style-type: none"> • LEN • TIF (> 600 ppi) • DCS 2.0 (> 600 ppi) 	

Note:

When opening files that are not yet normalized, the Viewer needs to read in the data. This is equal to a normalization step.

The Viewer Settings in Configure make it possible to indicate which Normalization ticket is preferred to execute this hidden normalization (see [Viewer](#) on page 685).

If you use View Data, the Prepare for Automation Engine Viewer task will include a normalization step.

If you do not use View Data, the file will be normalized right after choosing 'Open'.

Tip: If you want to compare an Esko normalized file with its original non-normalized design, you can set up a workflow where you always keep the first file version after the normalization - before starting to edit - and compare that file with the resulting edited version.

2.8.3 Viewer Statuses in the Files View

The View column can display a number of different statuses. Select a file to find out its status.

1. A dash (-) indicates that the file is not viewable, for example text, Word documents, pdf files... File extensions are taken into account to decide whether or not a file is viewable.
2. [View](#) 'View' indicates that the file **could** be viewable. It is possible based on the file extension, but no view files have been generated as the calculation could provoke a performance slowdown.
3. [View \(Prepared\)](#) 'View (Prepared)' indicates that the file can be viewed as prepared view data has been generated, either by launching a file with the **Prepare for Viewer** task (see [Prepare for Viewer](#) on page 415), or by copying or moving files in a folder that is marked as **Speed up for Viewing**. Up-to-date view files are available.
4. [View](#) 'View' indicates that the file can be viewed. However, there are no view files available, and they will be calculated on the fly if you want to view.
5. [View \(Prepared\)](#) 'View (Prepared)' indicates that the file could be viewed, but viewing is disabled because the view files are out of sync with the original file.

Digital Film Files

For digital film files, the situation is slightly different.

A) For **.len** files, it is mandatory that view files are prepared so as to be able to view.

1. [View \(Prepared\)](#) 'View (Prepared)' indicates that the files have up-to-date view files and can be viewed.
2. - (Need View Data) indicates that for the selected .len file no view data has been generated and it cannot be viewed.
3. - (Prepared) indicates that the .len files have view data, but that the view data is not in sync with the original file and can therefore not be viewed.

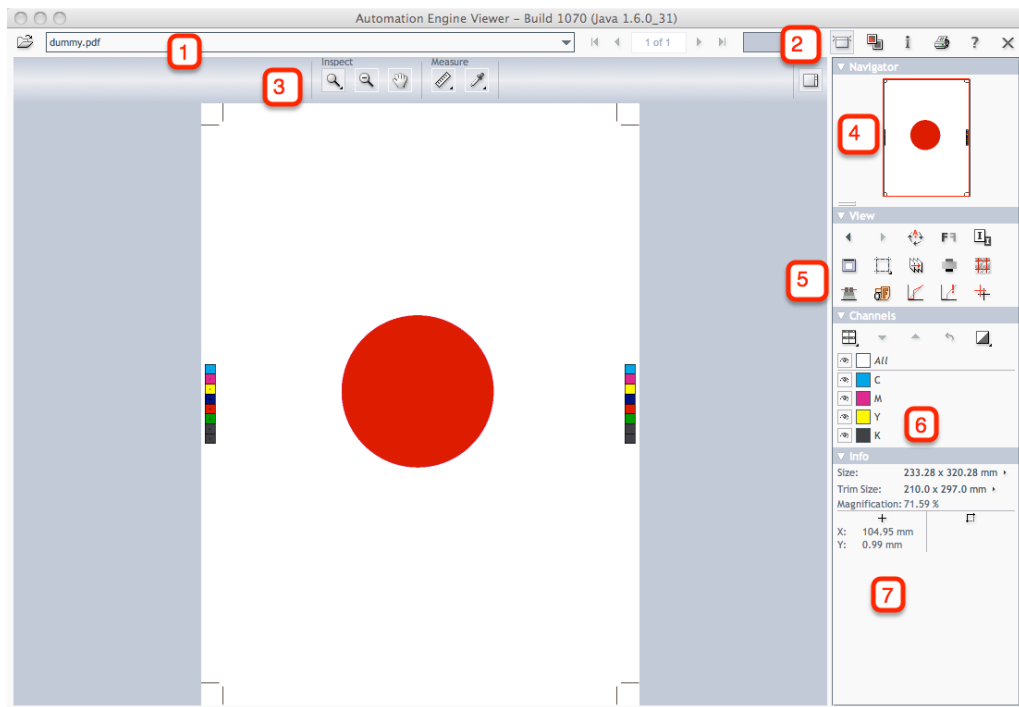
B) For **.tif** files, behavior depends on the resolution of the file.

A TIFF with low resolution (<600 ppi) can be viewed without having view files.

A TIFF file with a high resolution (>600 ppi) must have up-to-date view files to be able to view it.

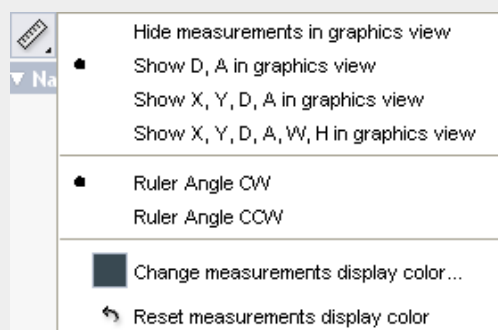
2.8.4 The Viewer Window

The Viewer comes with a number of powerful tools that make it easy to navigate, view and compare files.





1. [The View Selector](#) on page 587
2. [The Toolbar](#) on page 591
3. [The Top Tools Panel](#) on page 591
4. [The Navigator Panel](#) on page 596
5. [The View Panel](#) on page 596
6. [The Channels Panel](#) on page 606
7. [The Info Panel](#) on page 607

Note: Some tools have an arrow in their lower-right corner. Right-click those tools to show more options.



Customizing the Viewer Window

You can hide some of the Viewer's tools to get a bigger viewing area, and show them again when you need them.



- Click  in the toolbar to hide or show [The Top Tools Panel](#) on page 591.
- Click  on the right to hide or show the side panels (Tools, Navigator, View, Channels...).

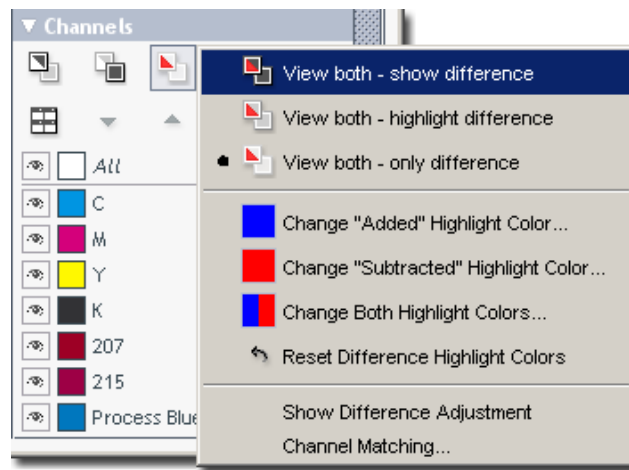
Viewing Differences in the Separations

In case you compare files that have different separations, then all differences will be shown in both common and unique separations.

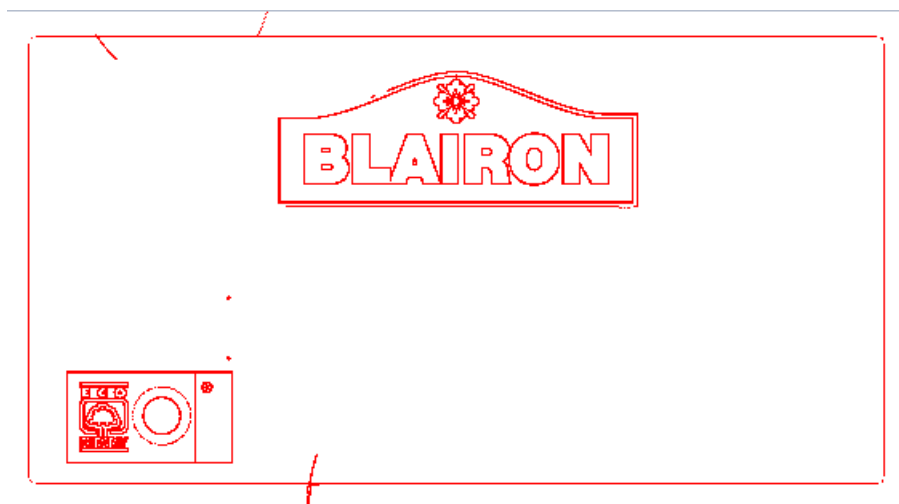
When the compare tool is activated, the **Channels** group will display an additional number of icons.

Move over the icon with the cursor and the tool tip will indicate what it stands for.

Right-click the following icons   to get additional options that allow you to change view modes, highlight colors and difference levels.



In the example below, only the difference is shown.



Comparing Files of Different Sizes

Comparing files with different sizes may seem like a challenge as they are not aligned in the Viewer, but the Viewer has some tools that make it very easy.

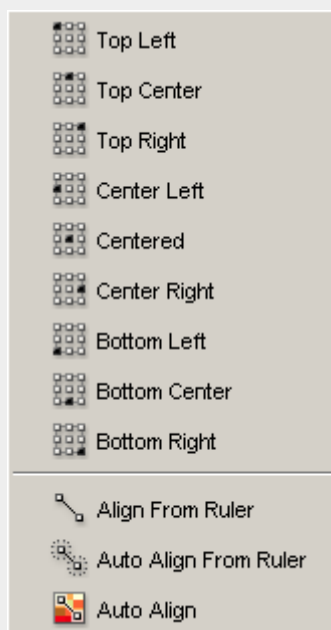


You can align both files by:



- Rotating one file (by 90 degrees)
- Mirroring one file
- Inverting one file
- Aligning both files on one corner
- Aligning with the help of a ruler (see [Align from Ruler](#) on page 586, [Auto Align from Ruler](#) on page 587 and [Auto Align](#) on page 587)

Note: You can find the ruler tools under the alignment button.

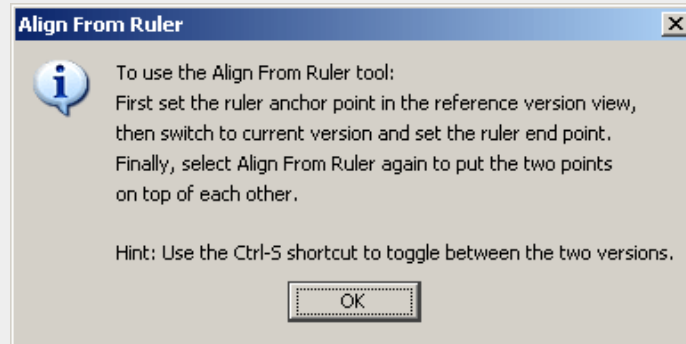




- Indicating the offset manually (typing in the fields)

Align from Ruler

The ruler tool works by defining two points (one in each file) that have to be aligned.

Tip: Click the Align from Ruler tool to show additional information.



1. Select the reference version of the file by clicking the  icon.
2. Click the alignment icon  and select **Align From Ruler** in the list.
3. Indicate the ruler anchor point.



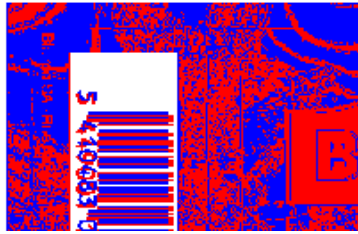
4. Switch to the other version by clicking the  icon (or use **Ctrl + S**).

The anchor point, here in the red circle, will be in a different location in the second file version.

5. Indicate the point that should be positioned on the ruler anchor point (here in the blue circle).



6. Click the **Align From Ruler** tool again. The files are now aligned.



Note: Depending on how precisely you defined the second ruler point, you may have to zoom in and repeat the alignment procedure to align your files with more precision.

Auto Align from Ruler


Auto Align from Ruler is used the same way as the Align from Ruler tool, but is more accurate at high resolution.

If the details of the alignment points match, the viewer will be able to increase the alignment precision (and have a good alignment with fewer iterations).

Auto Align

Auto Align does not use the ruler, but rather the currently visible view of the two compared images to attempt an automatic alignment.

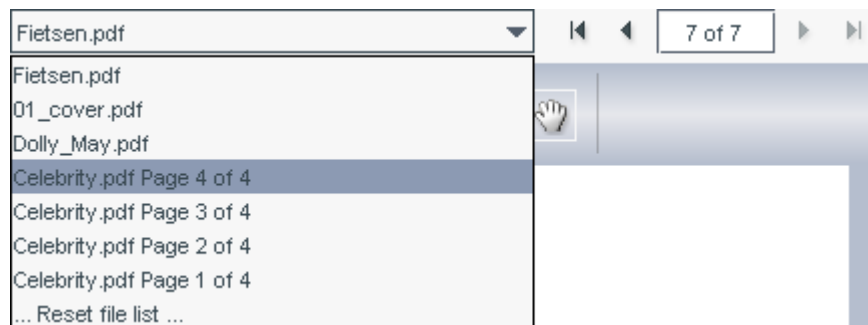
Using this tool repeatedly with one or more zoom-ins can be a very convenient and quick way to align two images.

For convenience, a separate tool button  is available to perform an Auto Align operation without having to open the Alignment pop-up menu. Clicking this button will perform an Auto Align from Ruler if a ruler selection is active, otherwise an Auto Align (from view).

The View Selector

The View Selector allows you to go through both:

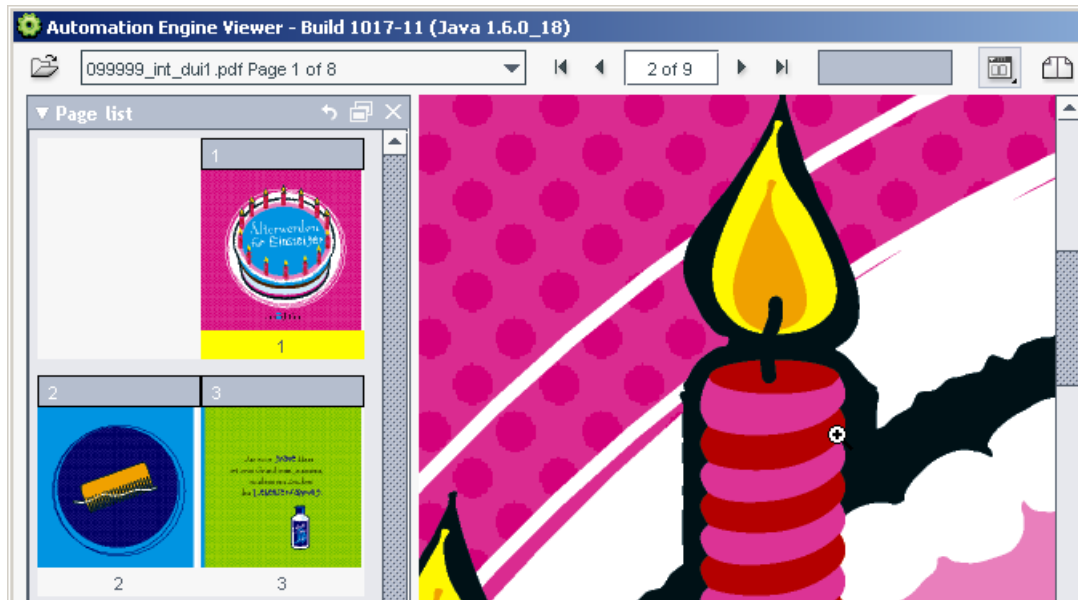
- the recently viewed files,
- the pages of a (recently or currently viewed) multi page file.




Select the file or page you want to view in the list, or use the arrows.

Page Lists

When opening a multi-page document, the Viewer offers a special preview and navigation panel called the Page list panel. The Page list panel occupies the left part of the Viewer window.



The Page list panel can be shown (or hidden) by clicking the  button in the top right part of the file selection bar. This button is not visible if only single page documents are opened in the viewer.

The major part of the Page list panel is occupied by the page list thumbnail view. This view shows thumbnails of the all the pages in the current multi-page document paired two and two according to the file's binding definition.

The size of the thumbnails can be changed by right-clicking the Page list view button in the file selection bar.




You can switch the main view to a specific page by clicking the corresponding page thumbnail.

Below the thumbnail view is a navigation bar that applies only to the currently selected multi-page document.






Reader Spreads

You can use the Reader spread button  in the file selection bar, to show your file in reader spread view (pages side-by-side). The reader spread view shows pairs of pages as they will appear in the final document.

The pages are assembled by stitching two pages together on the pages' trim-boxes. If one of the two pages is missing (if the active page is the first or the last), the active page will be shown with an empty (white) area next to it.

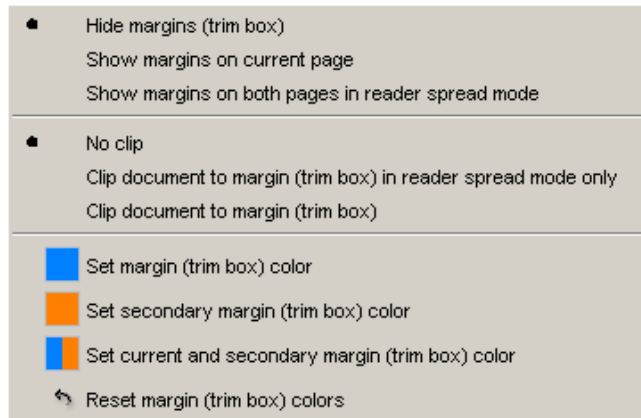


Note: In reader spread view, clicking   displays the previous / next reader spread (set of two pages).

There is a single active page in reader spread view. You can see which of the two pages is active in the Page list panel, or by turning on the page margins  in the View panel.

You can choose to show the trim boxes of the two page margins in different colors, or to only show the trim box of the active page.

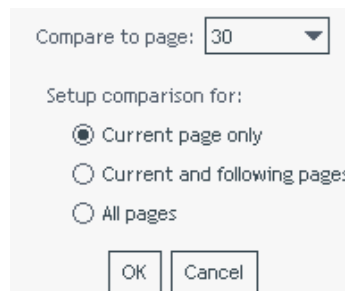
You can also clip the reader spread on its trim box.



Note: The document comparison is disabled while reader spread view is active. If document comparison is turned on, reader spread view is automatically turned off and visa versa.

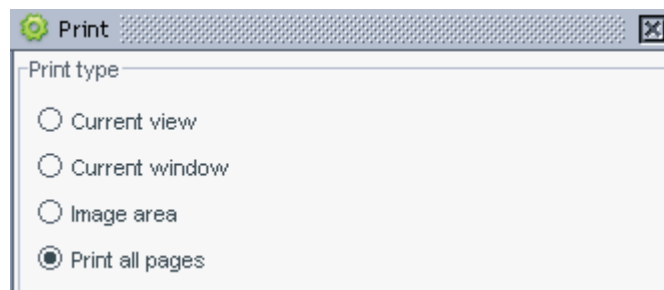
Comparing Mutipage Documents

When comparing two multi-page documents, the view will present an option panel where you decide which pages to compare to which.



Printing Multipage Documents

When viewing a multipage document, you can also choose to print all pages.



The Toolbar




The toolbar contains:

- a progress bar,
- multi page options (when viewing a multi page file). See [Viewing Multipage Files](#) on page 611.
- a Compare tool that allows you to compare two files. See [Comparing Files using the Viewer](#) on page 608.
- an Information button that opens the Info dialog box (containing XMP metadata). See [File Information \(XMP metadata\)](#).
- a Print button that opens the Print options dialog box. See [Print](#).
- a Help button that opens the Viewer online help.
- a Close button that closes the file being viewed (but not the Viewer).

Note: The Viewer documentation included in Automation Engine only highlights the most important tools and options. Please refer to the online Viewer documentation for more elaborate information.

The Top Tools Panel




The top tools panel contains some of the Viewer's most used tools. You can customize which tools you see by right-clicking it and selecting or deselecting some tools.

You can show or hide it by clicking  (see [Customizing the Viewer Window](#) on page 583). Hiding the top tools panel shows the [Tools Panel](#) on the right (containing the **Inspect** and **Measure** tools), and vice-versa.


Note: You can select a tool here even when it is not currently available in the Viewer (for example, **Page list** when viewing a single page file). This tool will then automatically appear in the top tools panel the next time it can be used (for example, when viewing a multi-page file).



The top tools panel can have the following tools:

Page List Tools

-  Reader Spread view (only when viewing a multi-page file, see [Viewing Multipage Files](#) on page 611).
-  Arrows to navigate between pages or spreads (only when viewing a multi-page file).
-  Mark tool to mark a certain page.



Inspect Tools

-  Zoom tool

-  Zoom out tool
-  Pan tool

See [The Tools Panel](#) on page 592 for more information.







Measure Tools

-  Ruler tool
-  Measure color percentages tool (densitometer)

See [The Ruler](#) on page 593 and [The Densitometer](#) on page 594 for more information.



Compare Tools

You can only see the **Compare** tools when you have two print jobs open to compare (see [Comparing Files in the Viewer](#)).

-  View current version
-  View reference version
-  View both - highlight difference
-  View both - blinking
-  View both
-  View both - side by side

See [Comparison Modes](#) for more information.

View Tools


-  Rotate view tool
-  Mirror view tool

See [The View Panel](#) on page 596 for more information.

The Tools Panel





The **Tools** panel holds the most frequently used tools.

The Tools panel contains the following tools:

-  the **Zoom** tool, that guarantees indefinite zoom.

Click the tool or use **Ctrl +** to zoom in.

Use **Ctrl -** to zoom out.

- : The **Pan** tool, that allows you to pan through your file,
- : The **Ruler** tool (see [The Ruler](#) on page 593),
- : The **Densitometer** tool (see [The Densitometer](#) on page 594),
- : The **Screen ruling and angle** tool.

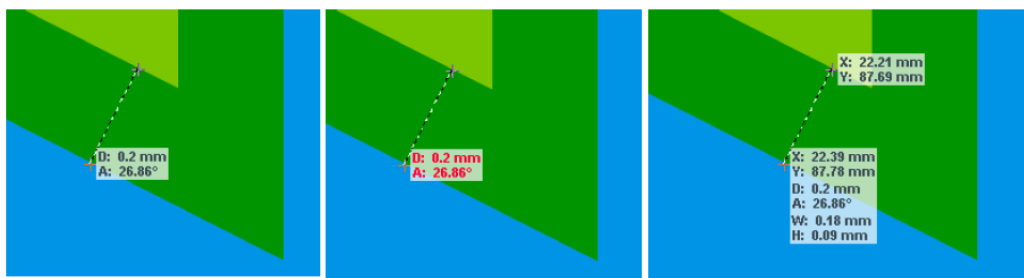
Note: The Viewer documentation included in Automation Engine only highlights the most important tools and options. Please refer to the online Viewer documentation for more elaborate information.

The Ruler

The Ruler tool calculates the distance between any two points in the work area.

When you measure from one point to another, a non-printing line is drawn and you can see the following information (in a pop-up and in the **Info** panel):

- The starting location (X and Y).
- The horizontal (W) and vertical (H) distances traveled from the x- and y-axes.
- The angle measured relative to the axis (A).
- The total distance traveled (D).



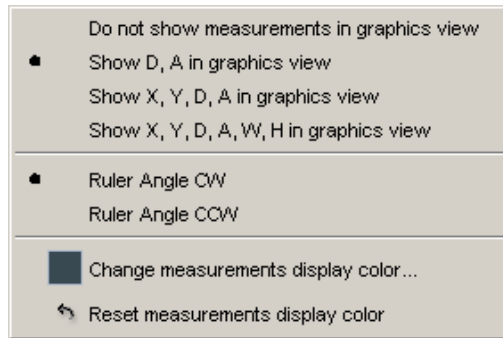
To measure between two points:

1. Select the Ruler tool.
2. Drag from the starting point to the ending point.

Tip: Hold down the Shift key to constrain the tool to multiples of 45°.

You can also:

- Right-click the measure icon to set some extra options, like, for example, showing or hiding units, changing the ruler angle, or the display color...



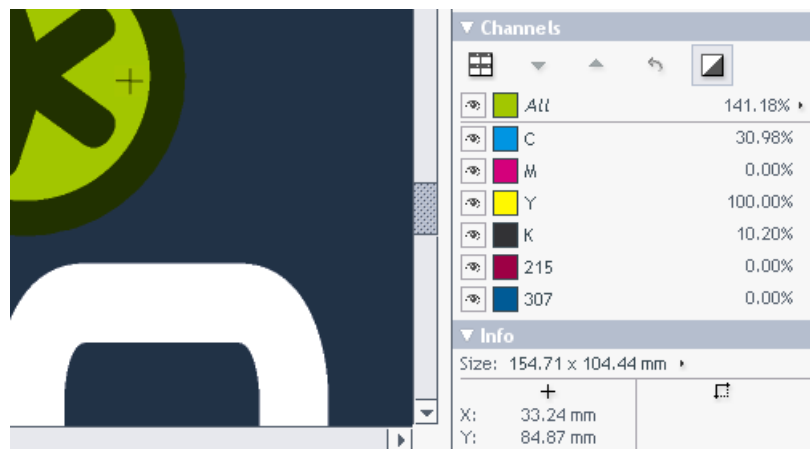
- Click the unit in the **Size** line of the **Info** panel to switch between points, millimeters, centimeters and inches.

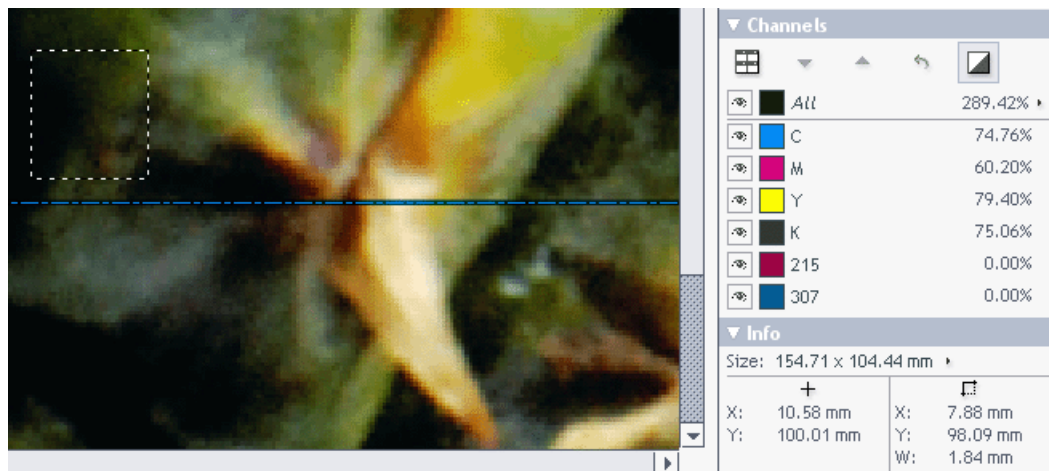
Tip: The Ruler tool can be combined with different view tools. You can for example indicate a certain point, then change the view by zooming out, navigating, or turning back to a previous view, and then indicate the second measuring point.

The Densitometer

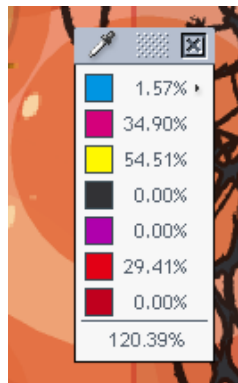
Click in your file with the densitometer tool to measure a certain point, or drag it to measure an area.

The densities are shown in the **Channels** panel, and the location measured in the **Info** panel.





By default, the densities are also shown in a pop-up near the location you measured.



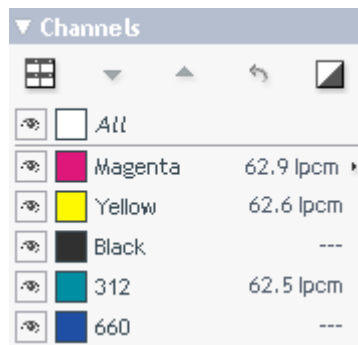
Right-click the densitometer tool to hide or show this pop-up.

Calculate Screen Ruling and Angle

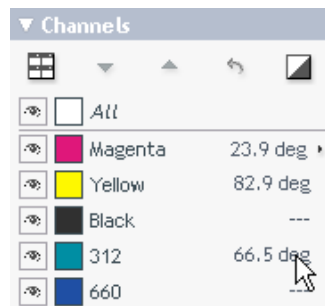
Note: You can only use this tool when working with digital film (RIP^{ed} files).

1. Select the screen ruling tool.
2. To select the area in which to measure the screen:
 - click a point in the image,
 - hold down your mouse and drag across an area (to make a more precise selection),
 - make a selection and double-click the tool,
 - double-click the tool to make a measurement based on the current view.

The **Channels** panel displays the screen rulings:



- To show a channel's screen angle (in the measured area), click its screen ruling value in the **Channel** panel.

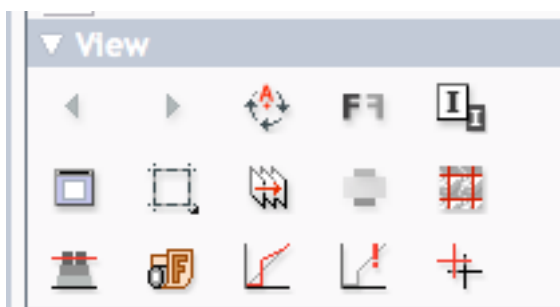


- To change the screen angle orientation, right-click the screen ruling tool button and choose either:
 - **Screen Orientation from XMP** (used by default),
 - **Screen Orientation CW**,
 - **Screen Orientation CCW**.

The Navigator Panel






In the **Navigator** panel you can see the part of your file you zoomed in on, and reposition the zoom if necessary.

The View Panel








The **View** panel contains the following tools, divided over three lines:

First line:






-  **Previous view:** Go to the previous view. Hold the Shift key to go to the first view.
-  **Next view:** Go to the next view. Hold the Shift key to go to the last view.
-  **Rotate view:** Rotates the current view by 90 degrees clockwise.
-  **Mirror view:** Mirrors the current view.
-  **Invert view:** Inverts the current view.

Note: The Viewer documentation included in Automation Engine only highlights the most important tools and options. Please refer to the online Viewer documentation for more elaborate information.

Second line:

-  **Background Color:** Changes the image's or the window's background color (see [Background Color](#) on page 597).
-  **Show margins (trim box):** Displays your file's trim box (see [Show Margins](#) on page 599).
-  **View from back:** Mirrors the current view and the sequence of the channels will be reversed.
-  **Set overprint highlight color:** Highlights the overprints in the color of your choice (see [Overprint Highlight](#) on page 600).
-  **Seamless view:** Allows you to see several one-ups of your file without any space in between (see [Seamless View](#) on page 601).

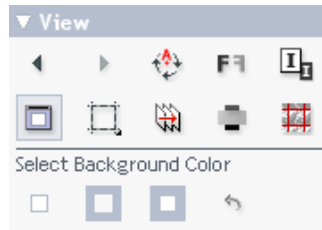
Third line:

-  **Total Area Coverage:** Shows the sum of all separation-densities at a certain point in your document.
-  **Flexo Plate:** Shows a single separation (by default the first in the list) as a simulated flexo plate.
-  **Flexo Print:** Shows the effect of highlight dot gain on the image.
-  **Break Out:** Shows an area where the percentage of a single separation is lower than the First Visible Dot value. This can be used to track areas that won't show up on print because the values are too low.
-  **Registration Error:** Shows a simulation of a design printed with registration errors of a specific press (as defined in the Press Settings).

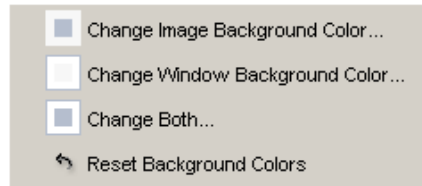
Background Color

To change the window and/or image background color:

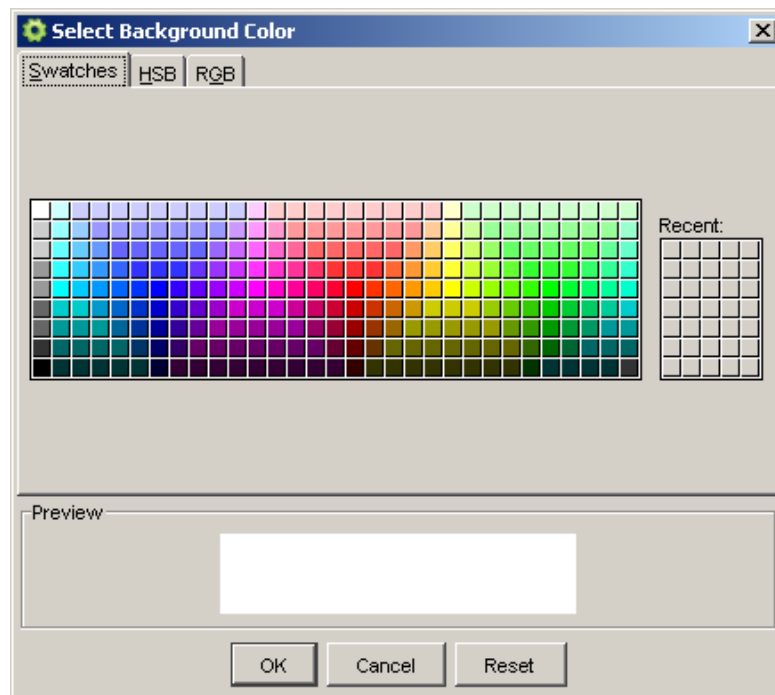
1. Either:
 - Click the Background color tool then the appropriate icon in the **Select Background Color** group that appears.



- Right click the Background color tool then select one of the options from the context menu.

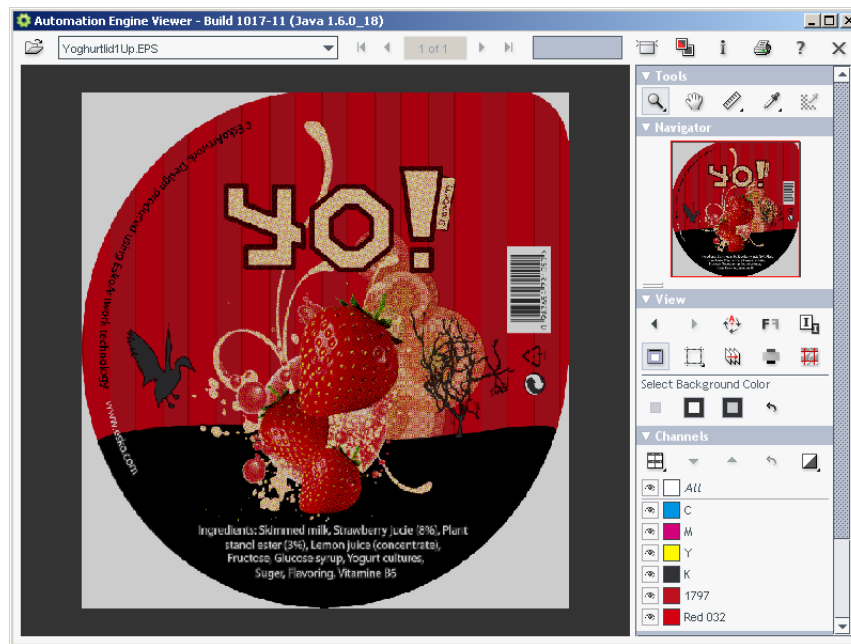


2. In the **Select Background Color** dialog that opens, select either a color swatch, or an HSB or RGB value.



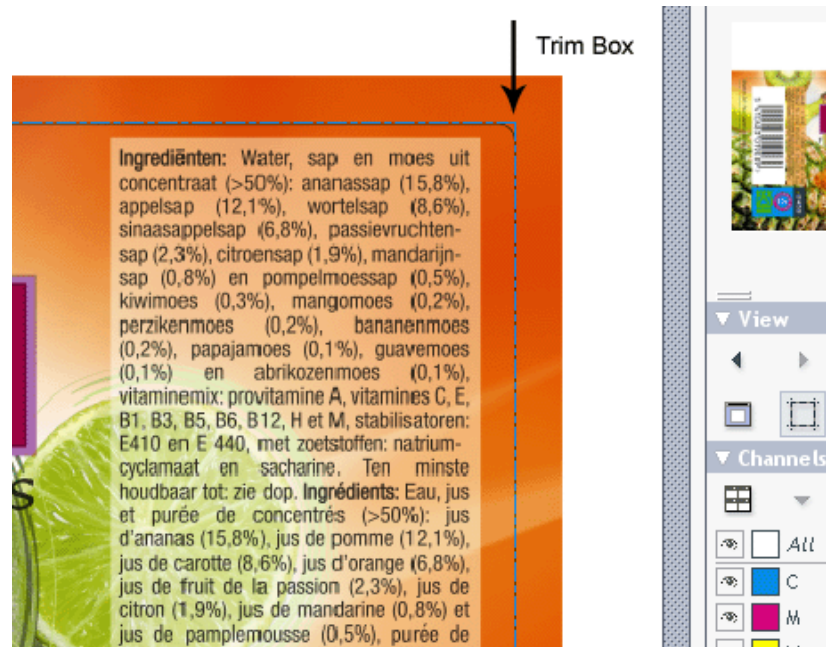
In the example below, the window color was set to black.

The image background was set to light grey, and whereas white is shown as light grey, all separation measurements remain unchanged!



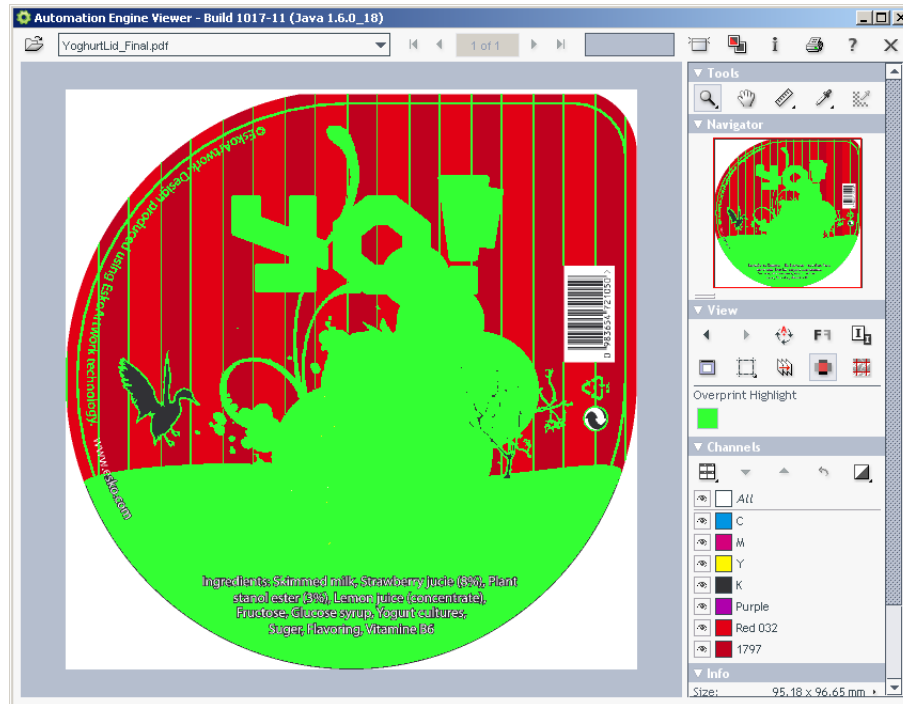
Show Margins

When the trim box is set correctly, you can easily check the correct die settings and possibly predict a problem in the later repeat layout.

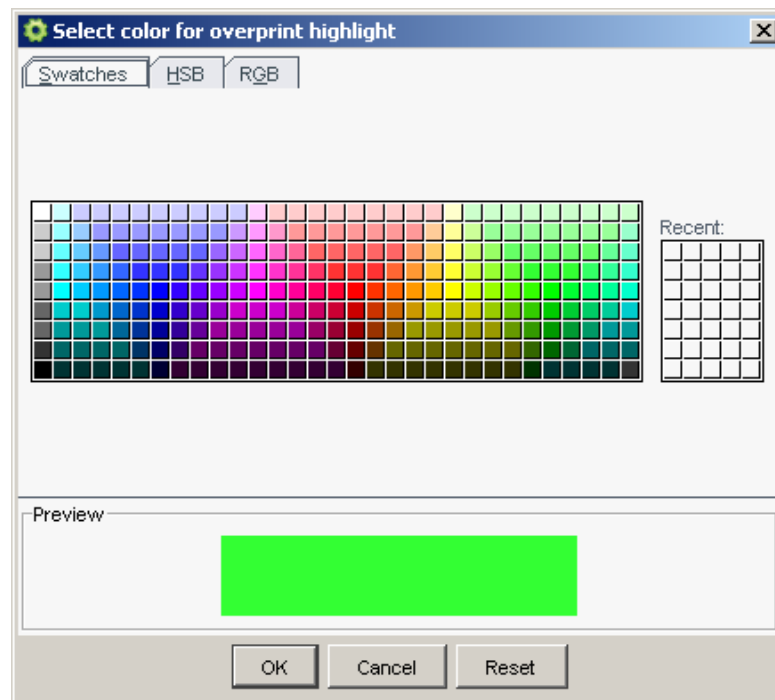


Overprint Highlight

Click the **Set overprint highlight color** button to get a clear indication of all areas with any form of overprint of 2 or more separations.

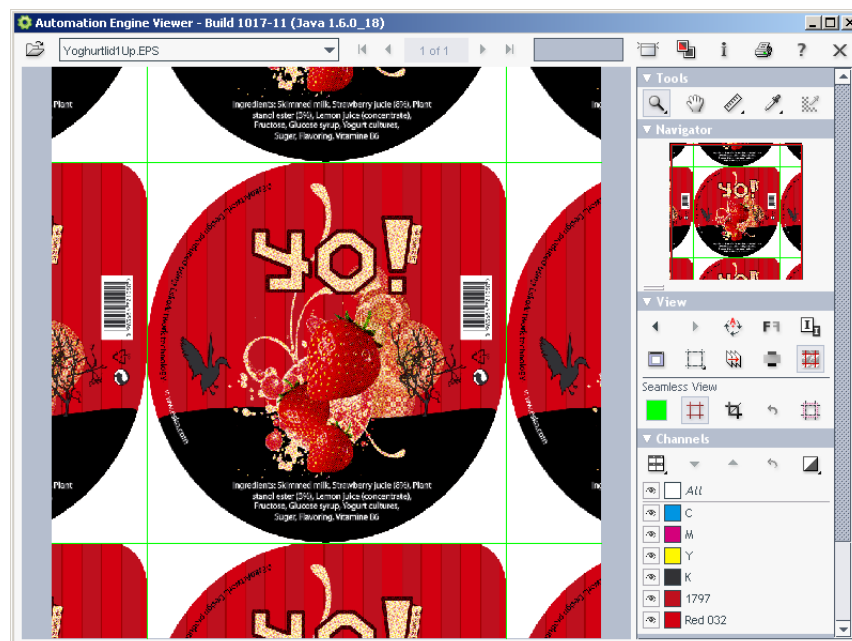



Click the color patch to select the highlight color of your choice.



Seamless View

- Click the **Seamless view** button to see a multi-up layout of the open file.
By default, the seamless repetition will be made around the trim-box.
If the trim-box is not defined (in the XMP metadata), it will be made around the image's boundaries.



- If necessary, you can adjust the seamless guides:
 - a) click the seamless crop tool  to select the one-up's trim box (or the image's boundary if no trim box is defined),
 - b) drag the trim box sides or corners to resize the area that is repeated.

Seamless with Overlap

By default, the seamless repetition will be made by clipping the image at the trim-box, and repeat the clipped part.

Sometimes however, the design to be checked is created to have a non-rectangular clipping.



To check such designs, the Overlap button  can often be helpful.

When turned on, the viewer will overlap the repeated areas rather than tiling them at the trim-box.



Total Area Coverage

The Total Area Coverage (TAC) is the sum of all separation-densities at a certain point in your document.

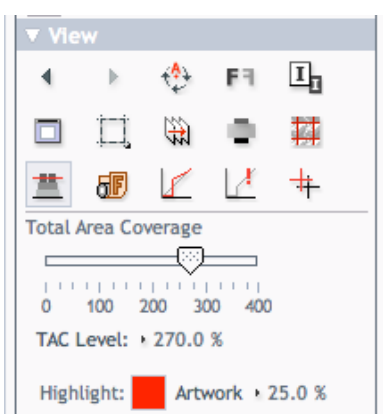
TAC Level Limit

The **TAC Level limit** is the maximal Total Area Coverage that your document may contain. This value depends on the press and the substrate your design will be printed on. Your printer should provide you with the TAC Limit value to use.

Defining the TAC Limit

The TAC limit is defined in the Press settings. By default the TAC Limit is set to 270%.

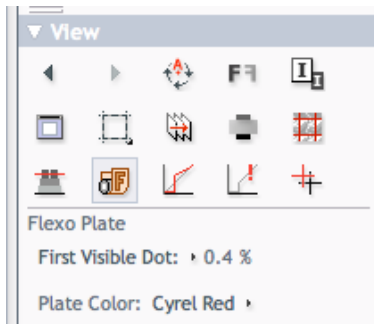
To quickly see the maximum TAC in your document, use the **TAC Limit** slider.



Move the slider to the right to use a TAC Limit value higher than that used by your current Press Setting. The more you move the slider to the right, the less pixels will typically be over the TAC Limit.

Flexo Plate

If you change the **Viewer** Preview mode to **Flexo Plate**, a single separation (by default the first in the list) will be shown as a simulated flexo plate.



In this Preview mode, only one separation can be viewed at a time.

Note:

Viewer uses the same technology as the Esko Flexo Tools for Photoshop.

First Visible Dot

The Flexo Plate preview will use the **First Visible Dot** percentage from the press settings:

- Pixels with a percentage lower than the First Visible Dot percentage will be shown as areas without dots.
- Pixels with a percentage equal or higher than the First Visible Dot percentage will be shown as areas with dots.

Plate Color

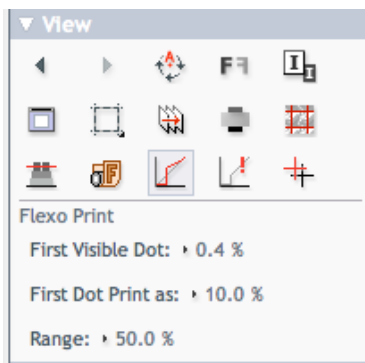
This preview predicts how clean your flexo plates will be. Especially in very light or very dark areas, you could end up with isolated dots or holes on the flexo plate, and those areas are difficult to hold on the plate or the substrate.

You can choose the plate color (Cyrel Red, Blue, Green or Safran Yellow, or a High Contrast Blue&White) to match the type of plate you are using.

Flexo Print

This **Viewer** preview mode shows the effect of highlight dot gain on the image.

This effect is equivalent to a curve adjustment. To determine this curve, the Flexo Print Preview will use the **First dot** and the **Dot gain in highlights** settings from the Press Settings.



Going from light to dark:

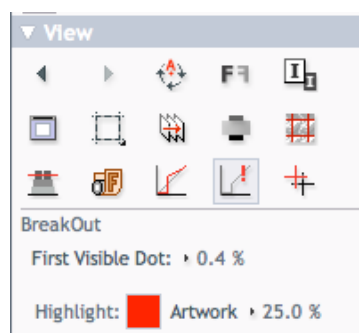
- The Print Preview layer will display percentages below the **First visible dot** percentage as 0%.
- The **First visible dot** percentage will be shown using the **First Dot Prints as** percentage.
- Percentages above the **First visible dot** percentage will be shown darker.
- Towards the midtones this darkening effect will fade out until the **Range** value.
- Percentages above than the **Range** value are displayed unchanged.

Break Out

A Breakout is an area where the percentage of a single separation is lower than the **First Visible Dot** value. This can be used to track areas that won't show up on print because the values are too low.

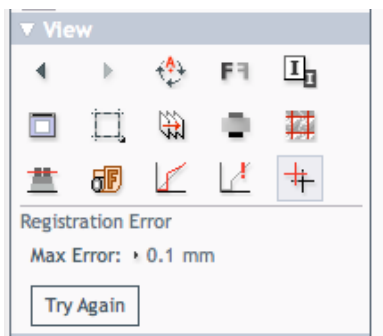
Your printer should provide you with the First visible Dot value. The First visible dot is set in the Press Settings. When switching to Breakout mode, the preview will be dimmed and all areas where a single separation has a value lower than the **First Visible Dot** value (but higher than 0%), will be displayed in highlight color.

You can measure the actual percentages in that specific area using the densitometer.



Registration error

The Registration Error Preview is a simulation of a design printed with registration errors of a specific press (defined in the Press Settings, see). It can be very helpful to see if trapping is applied sufficiently and correctly.



The **Registration Error** value is taken from the Press Setting you selected. The separations are randomly shifted. In fact, all separations are moved exactly the distance of the Registration Error, but all under a random angle.

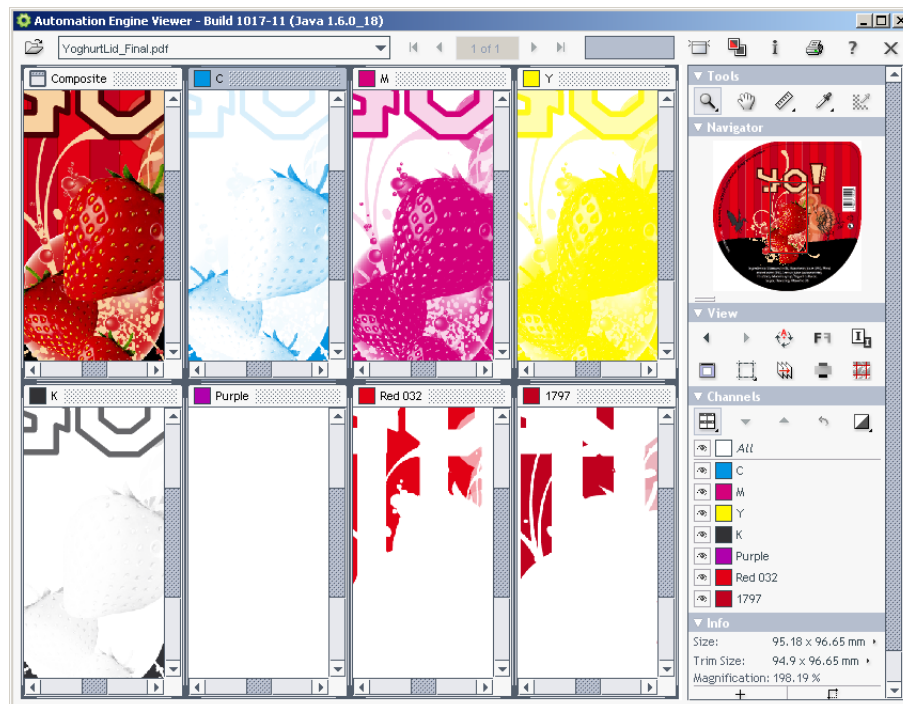
If you want to see another random registration error simulation, click the **Simulate Again** button.

In the example above, you can see a small area of a job in regular preview (left) and with registration errors (right). In this case, you can clearly see no trapping was applied, so white lines appear at the registration shifts.

The Channels Panel

The **Channels** panel shows the file's channels and offers extensive separation viewing possibilities.

Click the **View channels in separate windows** button  to switch between a **Composite view** and a tiled view of the individual separations. Right-click it to choose the kind of tiling to use (**Automatic tiling**, **Horizontal tiling** or **Vertical tiling**).

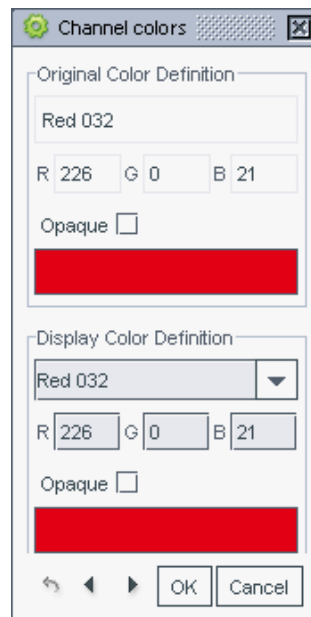


Click the separations' box to add or remove channels from the selection.

Tip: Right-click a separation's box to see only this channel.

You can decide to show separations with the same ink name as one, or to split them. For example, the K and K/1 Esko inks are shown as one by default.

Double-click a channel to show the **Channel Colors** dialog.



In this dialog, you can edit the channel's display color definition: Select a new display color from the list, or set new RGB values.

Tip:

This is useful, for example, if your print job will be printed on clear plastic and contains a white separation. To see that separation better in the Viewer, you might want to display it as pink.

Note that this choice is not remembered the next time you open the Viewer.

Check the **Opaque** checkbox to make the selected separation opaque.

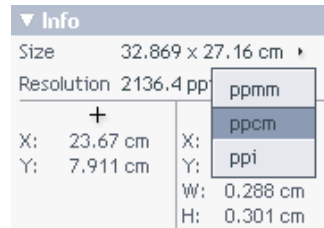
You can use the arrows at the bottom to change the order of separations that are shown in the list.

Note: The Viewer documentation included in Automation Engine only highlights the most important tools and options. Please refer to the online Viewer documentation for more elaborate information.

The Info Panel

The **Info** panel displays file information (dimensions, resolution), and viewing conditions (magnification used, mouse position, readouts from the ruler selection when using the ruler...).


To change the current units, click on one of the small triangles and select the desired unit in the pop-up window that opens. You can select units independently for sizes and resolutions.

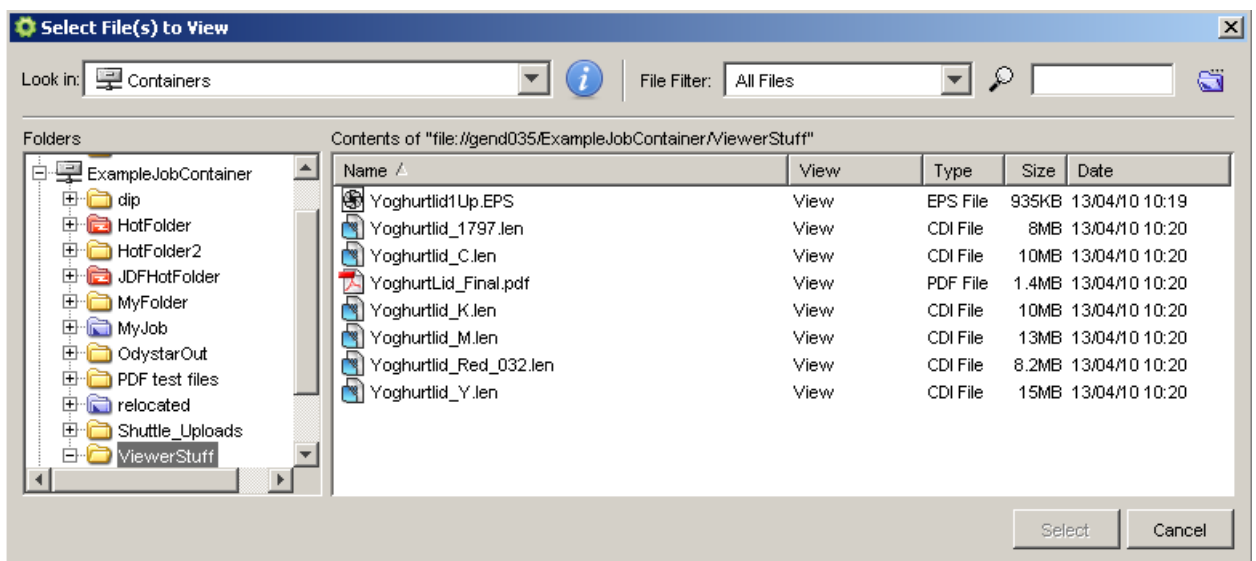


2.8.5 Working with the Viewer

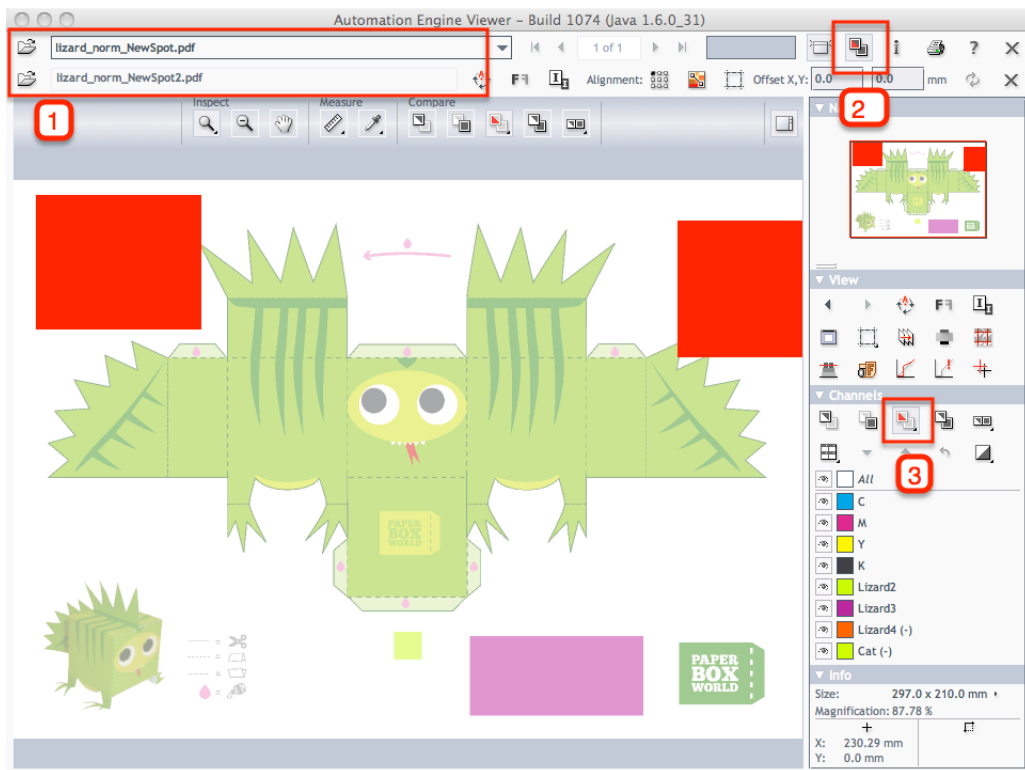
Comparing Files using the Viewer

The Viewer is not only a powerful viewing tool, but also contains a strong comparison tool.

To compare two files, open one file in the Viewer, then click the Compare  button to open the second file.



This enables you to have a close look at the differences between the two files. If needed, you can see the differences per separation, including any changes in the images (internal or referenced).




While comparing you can use other tools like for example [View channels in separate windows](#), [the ruler](#), (de)select, [rotate](#), etc...

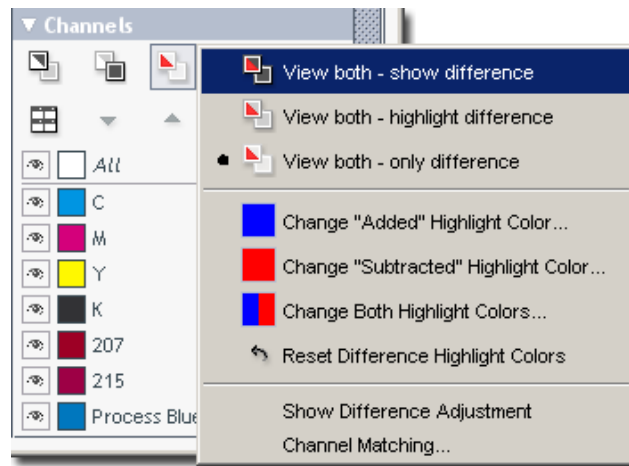
Viewing Differences in the Separations

In case you compare files that have different separations, then all differences will be shown in both common and unique separations.

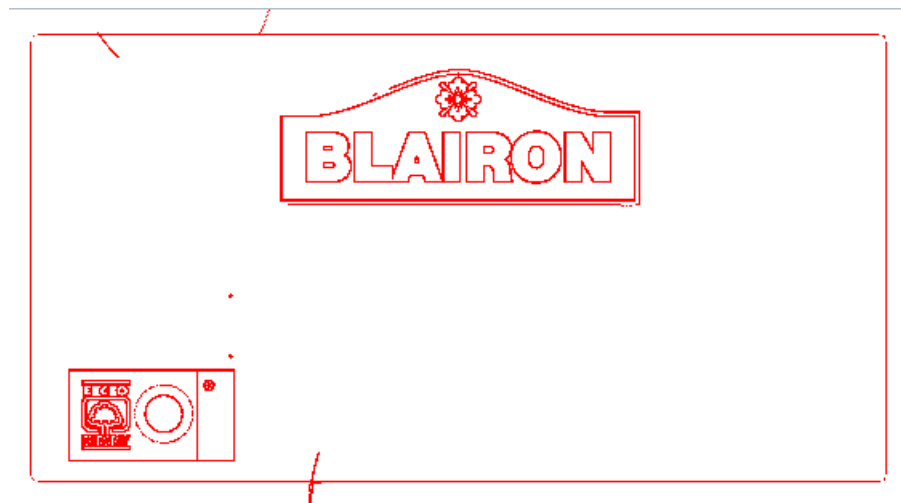
When the compare tool is activated, the **Channels** group will display an additional number of icons.

Move over the icon with the cursor and the tool tip will indicate what it stands for.

Right-click the following icons  to get additional options that allow you to change view modes, highlight colors and difference levels.



In the example below, only the difference is shown.



Comparing Files of Different Sizes

Comparing files with different sizes may seem like a challenge as they are not aligned in the Viewer, but the Viewer has some tools that make it very easy.

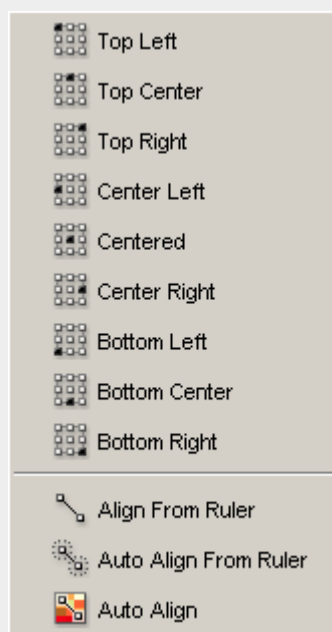


You can align both files by:



- Rotating one file (by 90 degrees)
- Mirroring one file
- Inverting one file
- Aligning both files on one corner
- Aligning with the help of a ruler (see [Align from Ruler](#) on page 586, [Auto Align from Ruler](#) on page 587 and [Auto Align](#) on page 587)

Note: You can find the ruler tools under the alignment button.

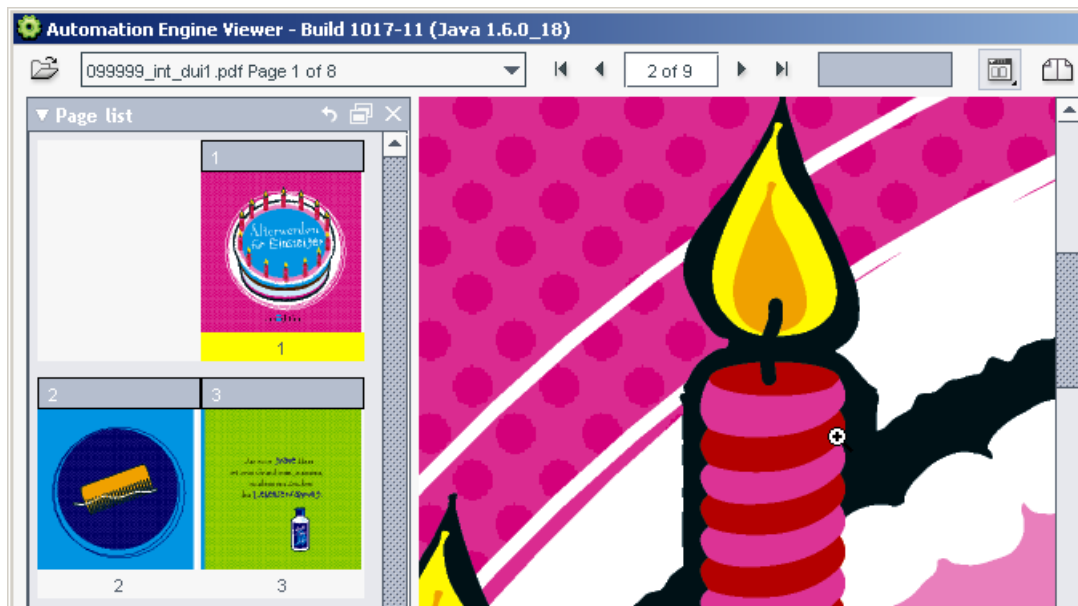



- Indicating the offset manually (typing in the fields)

Viewing Multipage Files

Page Lists

When opening a multi-page document, the Viewer offers a special preview and navigation panel called the Page list panel. The Page list panel occupies the left part of the Viewer window.



The Page list panel can be shown (or hidden) by clicking the  button in the top right part of the file selection bar. This button is not visible if only single page documents are opened in the viewer.

The major part of the Page list panel is occupied by the page list thumbnail view. This view shows thumbnails of the all the pages in the current multi-page document paired two and two according to the file's binding definition.

The size of the thumbnails can be changed by right-clicking the Page list view button in the file selection bar.




You can switch the main view to a specific page by clicking the corresponding page thumbnail.

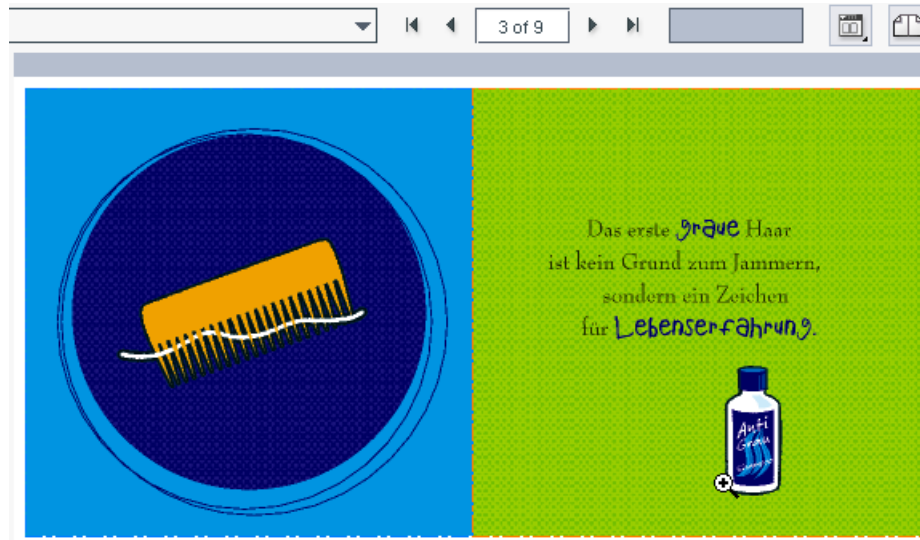
Below the thumbnail view is a navigation bar that applies only to the currently selected multi-page document.






Reader Spreads

You can use the Reader spread button  in the file selection bar, to show your file in reader spread view (pages side-by-side). The reader spread view shows pairs of pages as they will appear in the final document.

The pages are assembled by stitching two pages together on the pages' trim-boxes. If one of the two pages is missing (if the active page is the first or the last), the active page will be shown with an empty (white) area next to it.

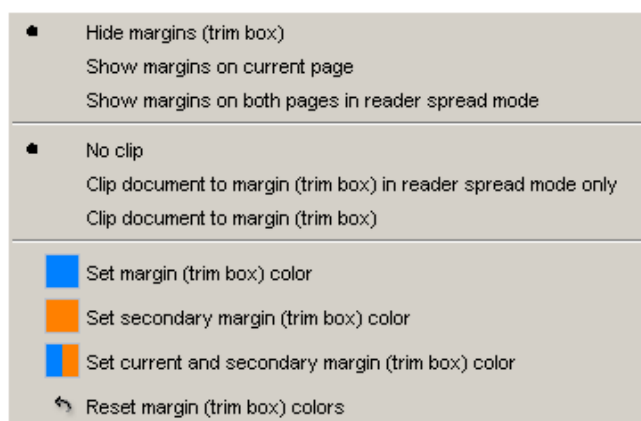


Note: In reader spread view, clicking   displays the previous / next reader spread (set of two pages).

There is a single active page in reader spread view. You can see which of the two pages is active in the Page list panel, or by turning on the page margins  in the View panel.

You can choose to show the trim boxes of the two page margins in different colors, or to only show the trim box of the active page.

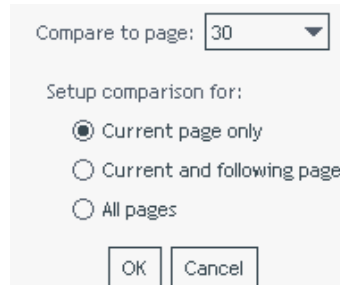
You can also clip the reader spread on its trim box.



Note: The document comparison is disabled while reader spread view is active. If document comparison is turned on, reader spread view is automatically turned off and visa versa.

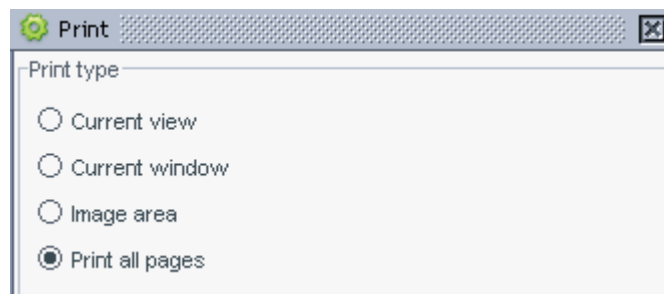
Comparing Mutipage Documents

When comparing two multi-page documents, the view will present an option panel where you decide which pages to compare to which.



Printing Multipage Documents

When viewing a multipage document, you can also choose to print all pages.



Compensating Distortion

What is Distortion?


Distortion is a technique used to pre-compensate graphics that are to be printed with a non-uniform printing technology. Distortion is typically done in the ripping process, the last step before printing.

The Viewer uses the file's XMP metadata to detect if the files are distorted in the vertical and/or horizontal directions, and by how much.

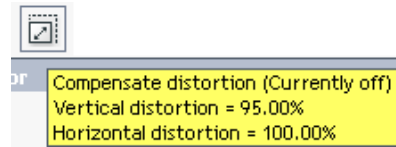
The Compensate Distortion Tool

To make comparison of files with and without distortion, or files with different distortion values easier, the Viewer offers a simple distortion compensation tool.

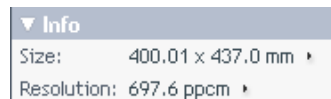
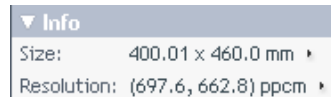
Note: You should compensate the distortion before aligning the files for comparison.

Use the **Compensate Distortion** button  to activate or deactivate the distortion compensation (this button is only shown if one of the open files is distorted).

The tool tip of the **Compensate Distortion** button will show what the distortion of the current file is.



When the distortion is compensated, the resolutions and the size will be adjusted in the Info panel.



Managing View Data

You can create View Data using the **Prepare for Viewer** task. See [Prepare for Viewer](#) on page 415 for more information.

Should you decide to work with prepared View Data, keep in mind that Automation Engine creates this View Data in a (hidden) `.view` folder next to the opened file.

The size of this view data depends on the size of your designs. By default an automatic resolution is chosen. But you need to balance these 2 factors:

- the time that the server spends on creating the View Data
- versus the speed improvement that you notice when initially opening a file that has prepared View Data (instant).

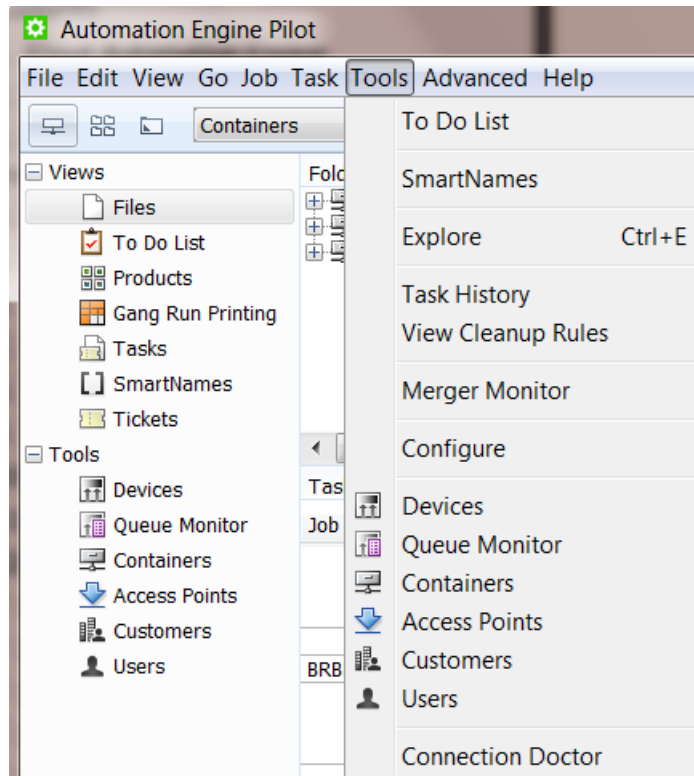
Danger: Manually manipulating this data will surely create Viewer errors and is therefore strongly dis-advised.

A mechanism is provided to delete the View Data:

- You can **manually** delete the (hidden) view data, by:
 - right-clicking a file and choosing 'Delete View Data'. This will remove the View Data for the selected file only.
 - right-clicking a folder and choosing 'Delete View Data'. This will remove the View Data for all files in the folder.
- When you remove or demote a Job (or Export / Archive it), the View Data is **automatically** deleted.

2.9 The Tools

The **Tools** menu contains tools useful to administrate Automation Engine.



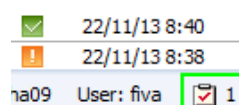
You can also find the most important tools under the **Views**, in the left pane of the Pilot. But this list of tools does not contain all of the tools. For example the **Configure** tool is not shown there.

2.9.1 To Do List

The **To Do List** lists all actions that need user intervention. Among these actions are Messages, Checkpoints, Red Alerts, Milestone Actions.

You can open the **To Do List** using one of the following methods:

- Go to **Tools > To Do List**.
- In the left window, click on the **To Do List** view.
- In the **Status bar**, click on the **To Do Icon**.



Use the **To Do List** to view, modify, forward and delete these actions, and all related files.

Checking To Dos

Actions

You can see a list of To Dos in the **Actions** panel. For each To Do, you can see:

- what the To Do is about (its **Subject**),
- which user it is assigned **To**,
- it's current **State**,
- if it is linked to a **Job** and to which,
- the **Type** of the To Do (Message, Checkpoint, Milestone Action...),
- the To Do's **Due Date**.

Tip:

- Initially, the entries in the To Do List are not sorted. You can sort them by clicking the appropriate column heading.
You can also show / hide or reorganise columns by going to **View > Select Columns in 'To Do List'...**
- You can refresh the list using **Ctrl + R** or **Edit > Refresh** .

Filter

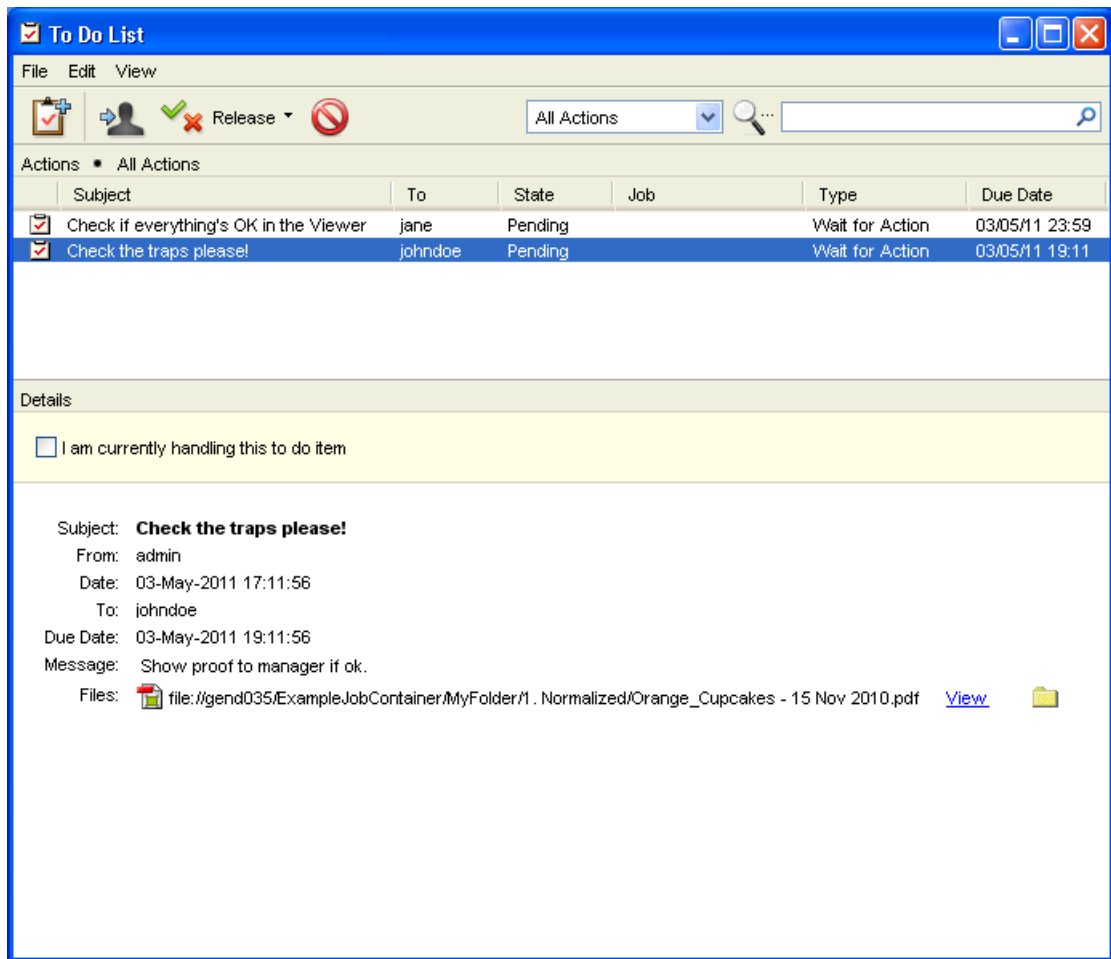
You can filter the list of actions using the drop-down menu. The default filters are **My Actions**, **All Actions**, **Checkpoint Actions**, **Messages** and [Milestone Actions](#).

You can define a **custom** filter. See [Defining a Custom Filter](#) on page 618.

You can also type text in the input field to search on the **Subject** text or the name of the file(s) linked to the To Do.

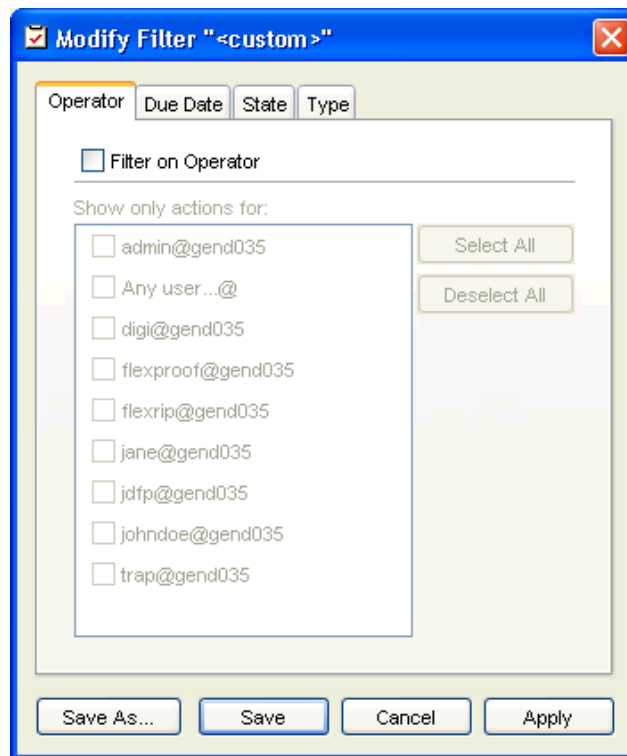
Details

Select a To Do in the **Actions** panel to show its **Details** below. They include extra information, like where the **Files** linked to the To Do are located, and if the To Do includes a **Message**...

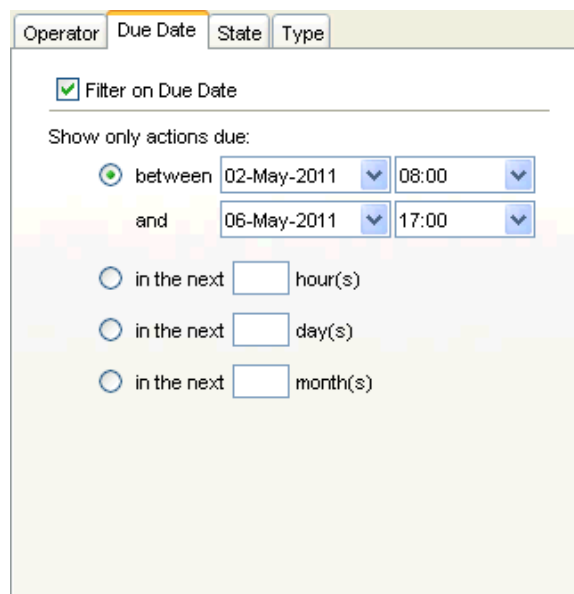


Defining a Custom Filter

1. Select **<custom>** in the filter drop-down menu and click .
This opens the **Modify Filter** dialog.



2. Use the **Operator** tab to choose whose To Dos you want to see.
Select **Filter on Operator** and select the desired operators (you can also **Select All**).
3. Use the **Due Date** tab to filter on the To Dos' due dates.
Select **Filter on Due Date** and select either a range of due dates, or a time span from the current date and time.



4. Use the **State** tab to filter on the To Dos' states.

Select **Filter on State** and select the states (either **Pending**, **In Progress** or both) to show with this filter.

5. Use the **Type** tab to filter on the To Dos' types.

Select **Filter on Type** and choose from the list (**Wait for Action**, **Milestone Action**, **Message**).


6. When you are done, click **Save As...** and give your new filter a name.
You can also **Apply** the filter straightaway.

Handling a To Do

To handle a To Do, select it in the **To Do List** and:

- To mark that you are handling it, select the **I am currently handling this to do item** in the **Details** pane.
- To open a To Do's file in the Viewer, click **View** next to the file path in the **Details** pane.



- To open a To Do's file in the associated application, double-click it in the **Details** pane.
- To open the folder containing the file(s), click the folder icon  next to the file path in the **Details** pane.
- To go to the job in the **Files** View (if the To Do concerns a job), go to **File > Go to Job** or use **Ctrl + Alt + O**.
- To release the files of a **Wait for Action** To Do to one of the output states of the **Wait for Action**

(**Checkpoint**) task / workflow step, click  then the output state you want to release the file to (by default **Completed** or **Aborted**).

See [The Wait for Action \(Checkpoint\) Task](#) on page 1003 for more information about the **Wait for Action (Checkpoint)** task / workflow step.


- To forward a To Do to another user, click  and select the user to forward the To Do to.

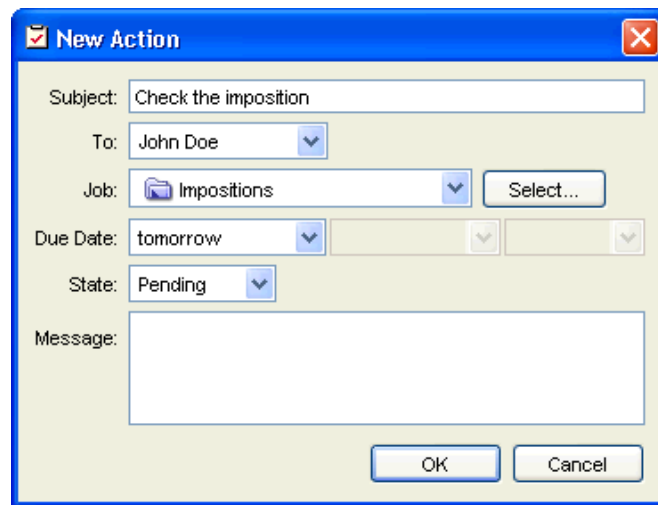
Note: You can also change the **Subject**, **Due Date**, **State** and **Message** at this point if desired.

- To delete a To Do, click .

Creating a To Do

You can only create **Message** To Dos from the **To Do List**.

1. Click  in the **To Do List** tool bar.
2. In the **New Action** dialog that opens:
 - a) Enter a **Subject**.
 - b) Choose who to assign the To Do **To**.
 - c) Link this To Do to a **Job** if needed.
 - d) Choose a **Due Date**.
 - e) Choose the To Do **State**.
 - f) Enter a **Message** if desired.



The screenshot shows a 'New Action' dialog box with the following fields and values:

- Subject: Check the imposition
- To: John Doe
- Job: Impositions (with a 'Select...' button)
- Due Date: tomorrow (with two empty dropdowns)
- State: Pending
- Message: (empty text area)

Buttons: OK, Cancel

3. Click **OK** when you are done.

2.9.2 SmartNames


SmartNames are documented as a separate chapter in the *Complete Manual*. Find it [here](#).

2.9.3 Explore

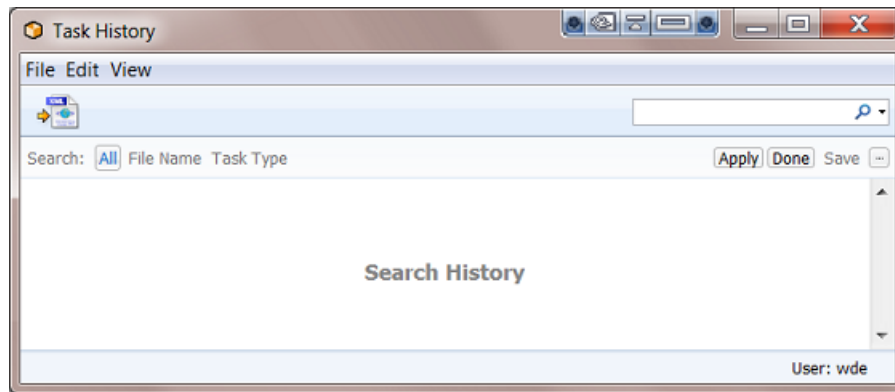
For data that cannot be accessed via the Pilot, you can use this shortcut to see your data directly on the file server, through the client's OS. This means that when you are using a Windows Pilot client, this tool will open a Windows Explorer to show your data. When you are using a Mac OS Client, this 'Explore' tool will show the data through the Mac OS file system browser, provided that the file server has the right share mapping to that Mac OS client.

2.9.4 Task History

To keep your Tasks view manageable, you might have to clean the task lists. If you do not want to keep some tasks, you can delete them. If you want to keep certain tasks or tickets for later use,

you can add them to **Task History**. After choosing a task, you can click on  in the Tasks View to move it to task history.

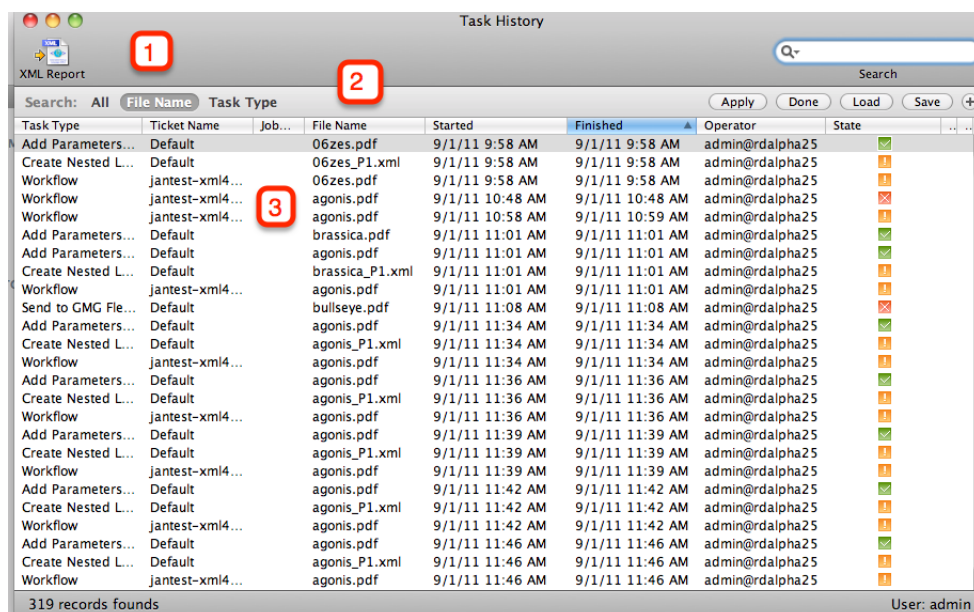
To view the history Choose **Tools > Task History** . You can only view the tasks in the history after clicking **Apply**. Alternatively you can press Enter in your keyboard to view the tasks in the history.



The Task History has a lot in common with the Tasks View. It lists all 'moved' tasks which you can relaunch, view and save.

You can archive entries in the Task History (more on that in the section on [How To Archive?](#) on page 625) which you can restore later. You can make XML reports of the tasks and use them to bill your customers.


The Task History has three parts:



1. Task History XML Report
2. Task History Search options
3. Task History List

Task History Menus


File Menu

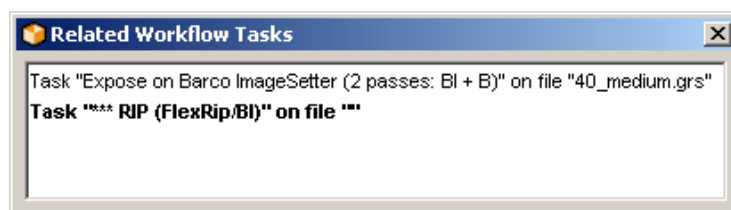
- **Open Task:** opens the ticket used for the selected task.
- **Export XML Report...**  Writes a report of the current selection. This XML report contains task name, ticket name (whether or not the ticket was modified), the Operator, when it started, when the task ended, Status, Priority, job folder...
- **Archive...** It is possible to archive the data in the database to an XML file. This allows you to put the file on CD or store it somewhere else.
- **Import Archive...** Restores the data in the selected XML file in the database. A browse window pops up, browse to the archive.XML and click OK. The archive will be restored.
- **Restart History Server** This will shut down the History Server and reboot it.
- **Close Window** [Ctrl + W] Closes the window. If only one window is left, this is disabled (use Exit, to close the last window).

Edit Menu

- **Select All** [Ctrl+ A] Selects all entries in the table.

View Menu




- **Columns:** define which columns you want to see, for more information, please refer to the section on [Columns](#).
- **Customize Current Filter:** Please see [filtering](#).
-  **Manage Views** [Ctrl + Shift + F] Here you can save the View Settings or select a previously created view.
- **Related Workflow Tasks:** Some tasks are part of a workflow, use this function to get a list of related tasks.



- **Stop** [Escape] This cancels the current database query and lets you set up a different specification.
- **Refresh** [F5] Perform another database query with the current settings.

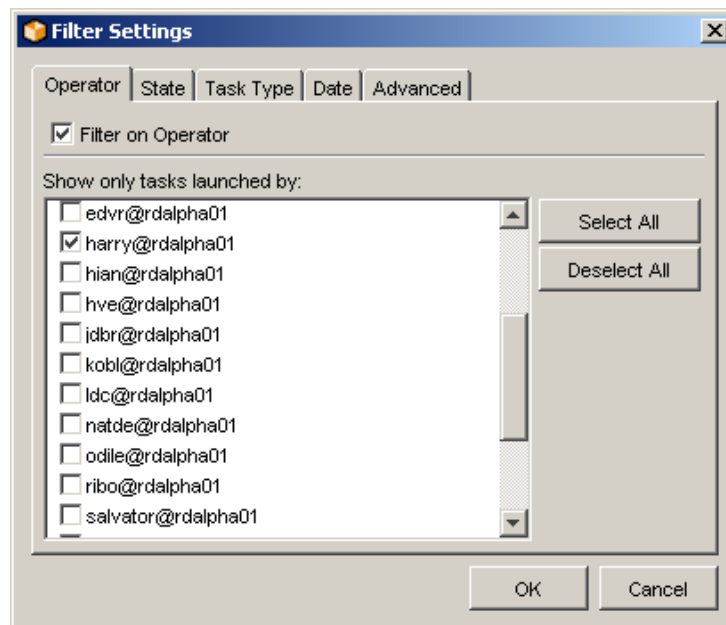
Task History Toolbar

-  **Open Task:** opens the selected task.

-  XML Report... Writes a report of the current selection. This XML report contains task name, ticket name (whether or not the ticket was modified), the Operator, when it started, when the task ended, Status, Priority, Job folder...
-  Opens the Filter Settings dialog. Please see [filtering](#).
-  Views: use the dropdown list to select 'Manage Views...' or an already defined view.

Filter

Filter Settings allow specific modification of the way the Tasks View is filtered. There are five tabs.



- **Operator** Define the filter according to the Operator. Tick the ones you want to monitor.
- **State** Define the filter according to the Status of the task. Just tick the ones you'd like to view.
- **Task Type** By default, the Tasks View will show all tasks launched, but you can modify the filter in such a way that the Tasks View shows nothing but the type of tasks you are interested in.
- **Date** Define the Filter according to date and time.
 - in a given time window (between [date and time] and [date and time])
 - during the previous X hour(s)
 - during the previous X day(s)
 - during the previous X month(s)
- **Advanced**

The Advanced Filter settings allow for precise queries.

Hit 'New' to define a rule. The set up is easy. You define a rule that shows X containing a certain string.

Task History List

The Task History list looks a lot like the [Tasks View task list](#).

The list lists all tasks that have been moved from the Tasks View to Task History. You define what you want to see. You can define the number of columns (see [View Menu > Columns](#)), you can click the head of a column to get the list to sort the way you like it, and you can filter the entries (see [View Menu > Filter](#)).

Task Type	Ticket N...	Job Name	File Name	Started	Finished	Opera...	State	Order ID	Subor...
Export to PDF ...	PDFX-3*	TFW_Out...	Sanity_2...	12/23/05 ...	12/23/05 ...	tfw_out...	✓	TFW_O...	
Export to PDF ...	PDFX-1*	TFW_Out...	Sanity_3...	12/23/05 ...	12/23/05 ...	tfw_out...	✓	TFW_O...	
Create Report...	TFW_rep_...		EG_exter...	12/23/05 ...	12/23/05 ...	TFW_Br...	!		
Export to PDF ...	PDFX-3*	TFW_Out...	Sanity_2...	12/23/05 ...	12/23/05 ...	tfw_out...	✓	TFW_O...	
Normalize Pos...	Default	TFW_Illus...	Symbols ...	12/23/05 ...	12/23/05 ...	kobl@r...	✓	TFW_Illu...	
Create Report...	taskinput_...	TFW_Re...	taskinput...	12/23/05 ...	12/23/05 ...	madl@r...	✓	TFW_R...	
Create Report...	TFW_rep_...		70004_s...	12/23/05 ...	12/23/05 ...	TFW_Br...	✓		
Export to PDF ...	PDFX-3*	TFW_Out...	Sanity_1...	12/23/05 ...	12/23/05 ...	tfw_out...	✓	TFW_O...	
Normalize Pos...	Default	TFW_Illus...	ScopeBo...	12/23/05 ...	12/23/05 ...	kobl@r...	✓	TFW_Illu...	
Create Report...	taskinput_...	TFW_Re...	taskinput...	12/23/05 ...	12/23/05 ...	madl@r...	✓	TFW_R...	
Export to PDF ...	PDFX-3*	TFW_Out...	Sanity_1...	12/23/05 ...	12/23/05 ...	tfw_out...	✓	TFW_O...	
Create Report...	taskinput_...	TFW_Re...	taskinput...	12/23/05 ...	12/23/05 ...	madl@r...	✓	TFW_R...	
Normalize Pos...	Default	TFW_Illus...	rgb_eps.ai	12/23/05 ...	12/23/05 ...	kobl@r...	✓	TFW_Illu...	
Create Report...	TFW_rep_...		70003_tu...	12/23/05 ...	12/23/05 ...	TFW_Br...	✓		
Create Report...	taskinput_...	TFW_Re...	taskinput...	12/23/05 ...	12/23/05 ...	madl@r...	✓	TFW_R...	
Normalize Pos...	Default	TFW_Illus...	OpacityM...	12/23/05 ...	12/23/05 ...	kobl@r...	✓	TFW_Illu...	

View: <all> 22 Records found. User: admin

How To Archive?

Archiving serves two purposes:

- The Task History is used to make the Tasks View quicker, more user-friendly. All tasks stored in the Task History can be relaunched, it is a so called 'living archive'. As time goes by, you will end up with a Task History containing a lot of tasks that you may never want to relaunch. You can delete these tasks, but if you think that's not a good idea, you can use the Archive function. so archiving can be a part of the Task History database maintenance.
- Archiving can be used to make a backup of the Task History data for safety reasons. You archive your data frequently, not because you need the space, but because you want to store your XML archive safely elsewhere, so you can easily restore it when problems should arise.

Archiving will store all information of certain tasks in an XML file. This adds the possibility of restoring those XML archives into the Task History. So your information will never be lost.


Note:

Archiving only works on selected tasks.

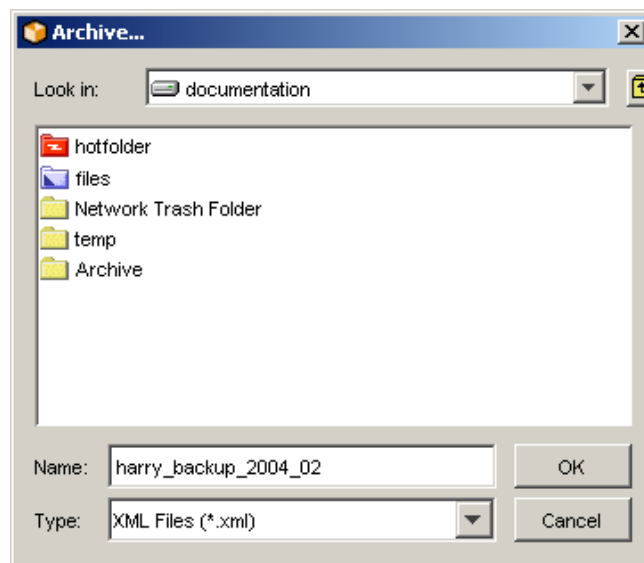
Archiving doesn't delete the archived tasks from the Task History.

1. Select the tasks you want to archive.

If you want to archive all tasks, press CTRL + A. If you want to make a sub selection, the Task History reacts to the known shortcuts such as CTRL + click and SHIFT + click.

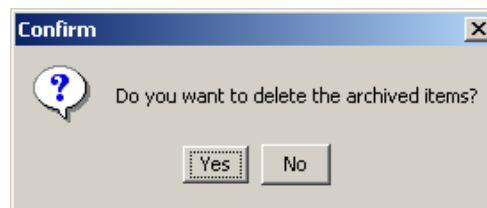
If you want to make an intelligent selection, and you don't want to lose time, you'd better use the [Filter](#) . You define what you want to see in the Task History (for example all tasks originating from one particular job, or handled by one particular operator, ...), click CTRL + A and you're ready to archive.

2. Click Archive (File Menu > Archive).
3. A dialog pops up, allowing you to browse to the place where you want the archive XML to be stored. Make sure to give the archive XML a meaningful name in order to avoid confusion later on. Confusion might arise if you bear in mind that the Task History also uses the XML language to write an XML report.



Click OK.

4. A 'Confirm' dialog pops up and asks you whether or not you want to delete the archived items:



2.9.5 View Cleanup Rules

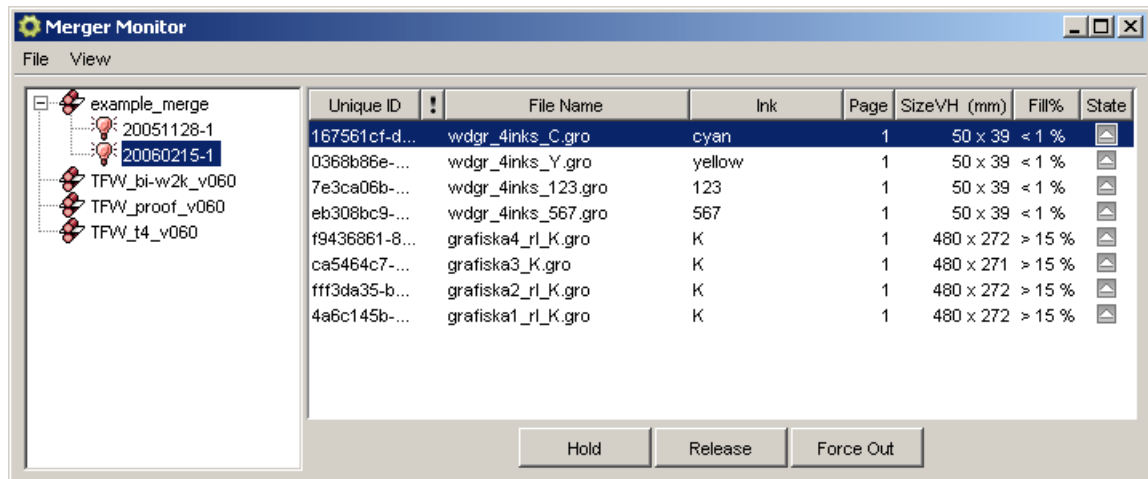
The **Cleanup Rules** dialog offers a view only mode of the task cleanup rules that have already been defined in the **Configure Tool**, in [Task Cleanup Rules](#) on page 682.


Click the '**Details...**' button to inspect the settings (viewing only).

2.9.6 Merger Monitor

If you use an **Expose On ticket** to expose on a device that has been configured to work with merge queues, you can monitor the merging activities.

Start the Merger Monitor by selecting 'Merger Monitor' from the Tools Menu.





The Merger Monitor consists of **two panels**. The **left-hand panel** lists all devices  on which jobs have been merged or are currently waiting to be merged. In the example above there's one device listed **example_merge**. Every device on which something has been merged stays listed until the server is rebooted.

Select a device to see what is in the merge queue. Jobs that are to be merged are assembled in pools.

A **pool** is a group of jobs with the same characteristics (same plate size, negative or positive, ...). These characteristics are defined by the so-called **extract restrictions**. For example: jobs with a different Dot Gain Compensation setting will not be merged together. They will not belong to the same pool.


Some of these restrictions depend on the setting in the **General tab** of the ticket you use.

1. **VDistortion** and **HDistortion**: Jobs must have the same vertical and horizontal distortion before they can be in the same pool.
2. **Reverse** : Jobs have to be reversed in the same way before they can be in the same pool.
3. **Mirror** : Jobs have to be mirrored in the same way before they can be in the same pool.
4. **Dot Gain Compensation**: Only jobs with the same Dot Gain Compensation belong in the same pool.

The other restrictions are defined in the **Output** tab of the ticket you use.

1. **Conf**: Jobs need to have the same plate size before they can be in the same pool.

2. **Output path** and **Output File Name**: Jobs with different Output paths or Output File names will end up in different merge pools.
3. **Resolution**: Jobs need to have the same resolution before they can be in the same pool.
4. **Compression**: Jobs need to have the same compression before they can be in the same pool.

Notice that **example_merge** has two pools : 20051128-1 and 20060215-1. When you select the device, but not a pool, you get an overview of all jobs waiting to be merged. In order to visually indicate that jobs belong to different pools they have a different background color. White and gray are used consecutively.

The **right hand panel** provides detailed information on jobs in the merge queue.

- The taskID
- Priority: this is the priority you defined in the Expose On ticket, not the priority of the task.
- The name of the Job
- The inks you are exposing, you defined them in the Expose On ticket.

Note: There are two ways in which the inks are listed. C&M&Y&K or C+M+Y+K. They can be merged on separate films or on the same film.

- The Page
- The Size HV (mm)




Note: The symbols used to indicate the size tell you whether or not the job can be rotated in the merger. <> (**154<>191**) indicate that the job can be rotated, an **x (154x191)** indicates that it cannot.

Rotation is an option you can set in the Transformations part of General tab of the ticket you use to launch the task.

- Fill %: Jobsize/Platesize

These percentages are relative, the overall fill depends on the margins and gap as they are currently defined in your ticket. You cannot simply add the percentages to get a correct idea of the overall fill. There are all kinds of reasons why jobs cannot be merged together and why the fill percentage is -as a result- not yet reached.

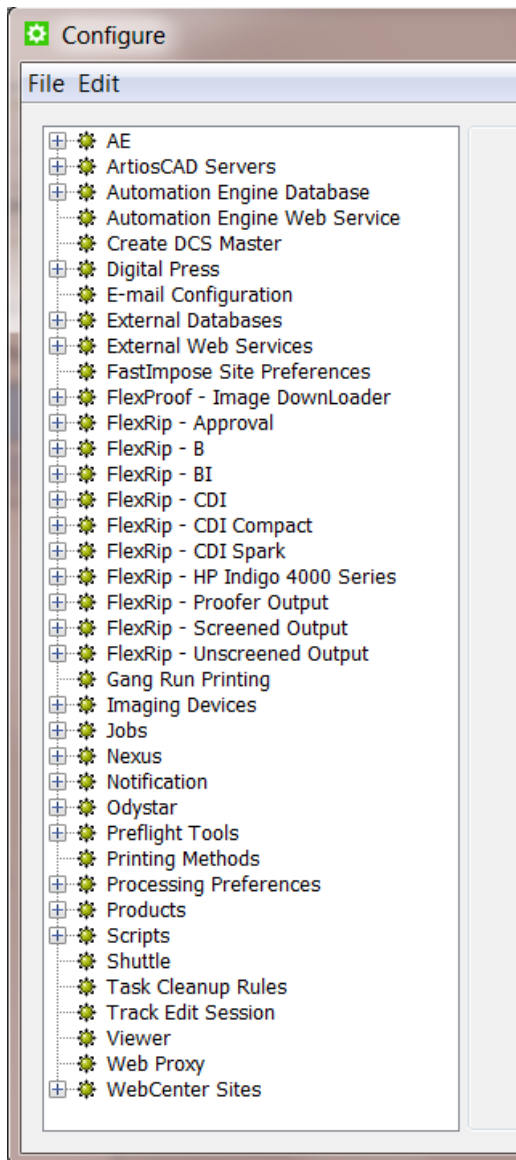
- The State of the job in the Merger queue

A job can be **on hold**  (it's an option you can choose in the 'Expose On ticket'). A job can be **released** , but that does not mean that it will be merged, it still has to wait till the fill percentage is reached. You can **Force** a job **out**  of the Merger queue. The forced out job will take as many jobs possible, if they have a 'release status', with it.

2.9.7 Configure

Use the Configure tool to connect to a variety of devices and applications, and define general Automation Engine settings.

To start Configure, go to **Tools > Configure** in the Pilot.



To Define Settings in Configure

1. Select the settings category, or the device / application you want to connect to, in the left pane.
2. Go to **File > New**.
3. Define your settings.
4. Go to **File > Save**.

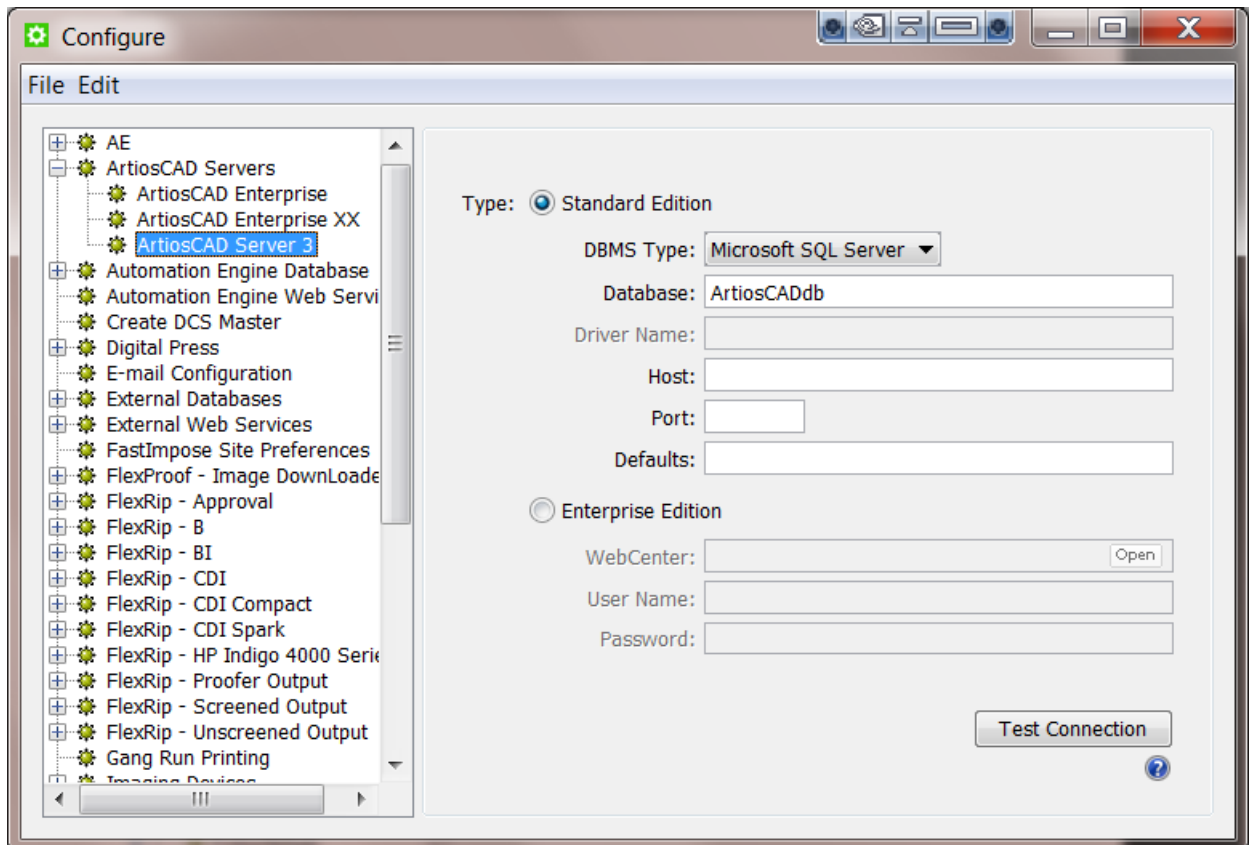
Note:

Windows shortcuts:

- 'Insert': to add a device
- 'Delete': to delete the selected device
- 'F2': to rename the selected device.

ArtiosCAD Servers

You can configure a new ArtiosCAD Server in Automation Engine via the pilot, by choosing **Tools > Configure > Select ArtiosCAD Servers** and using Ctrl / Cmd + N



How to configure a Standard Edition ArtiosCAD Server

1. Set up the ArtiosCAD Database Settings
 - a) When using a **Microsoft SQL Server**:
 1. Enter the **Name** of the ArtiosCAD database. Default is `ArtiosCADdb`
 2. Enter the database **host** name and the name of the SQL Server Instance when ArtiosCAD is not deployed in the default unnamed instance. (typically `<ArtiosCAD Server>\Esko`)
 3. Enter the **port** when not using the default.
 - b) When using **Oracle**:

Important: The **Oracle Client Software** (*ODBC: Additional libraries for enabling ODBC applications*) needs to be installed on the Automation Engine Master and its Assistants. When not installed, the **Create CAD Report from XML** Task will not be able to connect to the Oracle ArtiosCAD database.

1. Enter the name of the Oracle TNS **Service name**, for example Oracle XE

2. Enter the name of the Oracle Database **Driver**

Note: The name of the **Oracle database Driver** can be found in the Drivers tab of the ODBC Control Panel (%systemdrive%\Windows\ SysWoW64\Odbcad32.exe) after installation of the Oracle Client Software

3. Enter the database **host** name
4. Enter the **port** when not using the default.

2. Enter the location of the **ArtiosCAD Defaults**

This should be a **file:** URL, pointing to the ArtiosCAD Defaults., for example file://MyMachine/ArtiosCAD.

The Automation Engine Server should be able to reach this folder, and the service account of the BGMD Service (typically BGSystem) should have read access to the folder.

Note: ArtiosCAD Defaults are **version specific**. The version of ArtiosCAD hosting the Defaults must match with the version of CADX installed on the Automation Engine computer (Master and its Assistants)

If no ArtiosCAD Defaults location is specified, the **Create CAD Report from XML** task will use the ArtiosCAD Defaults installed on the Automation Engine computer (Master and its Assistants).

3. Test the connection by clicking the **Test Connection** button.

You will get a message stating "Connection to the database OK", or get an error message:

- if the Automation Engine Server can't connect to the database
- the Automation Engine Server can't get info about the Defaults location. Check if the location exists, and can be reached from the server.
- the Defaults location is not a folder.

How to configure an Enterprise Edition ArtiosCAD Server

1. Select **Enterprise Edition**
2. Enter the **WebCenter** location, typically `http://<WebCenterServer>/WebCenter_Inst`
3. Click **Open** to open the specified location in your default browser.
4. Enter the **User Name** and **Password**
5. Test the connection by clicking the **Test Connection** button.

You will get a message stating "Connection to the database OK", or get an error message if the Automation Engine can't connect to WebCenter.

Automation Engine Database

Here you can configure the following databases:

- The Task Monitor database,
- The FlexoPlates database,
- The JDF database,
- The Gang Run Printing database,
- The Task History database,
- The Jobs database,

- The Application Server database.

FlexoPlates

This database stores all the information about LEN files sent to the Digital Flexo Suite using the **Make Plate on CDI** task.

You can see this information in the **CDI** view (see [CDI View](#)).

By default, Automation Engine uses the [Jobs](#) on page 633 database as **FlexoPlates** database. To use a custom database, do the following:

1. In **Database**, enter the name to give to your database. By default, this is FlexoPlateHistory.
2. In **DBMS Type**, choose the type of database you are using.
3. In **Host**, enter the name or IP address of your database server.
4. Enter the **Port** to use (or leave it blank to use the default port).
5. Enter the **User** name and **Password** used to connect to the database.
6. Click **Synchronize** to synchronize the information in the database with your Automation Engine server.

Task History

Setup

These settings will be filled out correctly when the system is installed.

The screenshot shows a 'Setup Maintenance' dialog box with the following fields and values:

- Database:** fastlane
- DBMS type:** MSSQLServer
- DB Engine:**
 - Host:** RDALPHA16
 - Port:** (empty)
- Authorization:**
 - User:** admin
 - Password:** *****

A 'Synchronize' button is located at the bottom right of the dialog.

Note: DBMS stands for DataBase Management System.

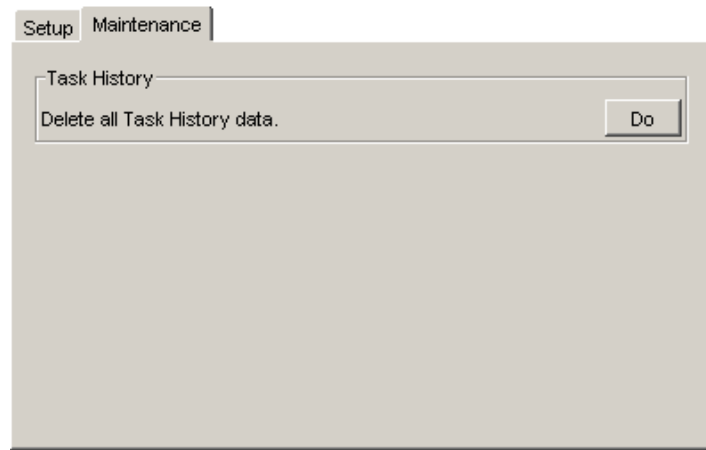
- Database: Enter the name of the database.

- DBMS type: Select the correct driver type from the list. The screenshot shows the driver type for an SGLdriven database.
- Host: Enter the name of the server.
- Port: Indicate which port the system uses to talk to the database.
- Enter the correct user name and password.

Use the **Synchronize** button to test the settings immediately. If you do not click Synchronize, the changes will be effective the next time the system is rebooted.

Maintenance

This tab has a button to **Delete all Task History data**. This is only useful when you have major problems with your database that cannot be fixed without deleting the database, or when you want to alter your database (for example add columns and you don't want to run a conversion script on the existing data, so you delete all Task History data and start with a fresh, altered database). The system will ask for confirmation.



Jobs

Setup

These settings will be filled out correctly when the system is installed. However, you can change these settings.

Note: DBMS stands for DataBase Management System.

- Database: Enter the name of the database.
- DBMS type: Select the correct driver type from the list. The screenshot shows the driver type for an SGLdriven database.
- Host: Enter the name of the server.
- Port: Indicate which port the system uses to talk to the database.
- Enter the correct user name and password.

Use the **Synchronize** button to test the settings immediately. If you do not click Synchronize, the changes will be effective the next time the system is rebooted.

Maintenance

Dangling Jobs are active Automation Engine Jobs that have a link to job folder that is not (or no longer) there, probably because the folder was (re)moved, or renamed.

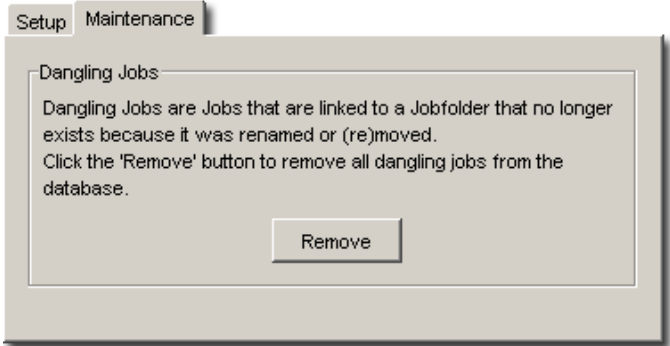
These Jobs can be removed from the Database via the Maintenance tab.

Ideally, each Job corresponds to one folder on disk. But... it may occur that a user or operator moves or deletes a folder of a Job via Windows Explorer or Macintosh Finder. This way, discrepancies can arise between the information of a job in the Automation Engine database and the actual situation on disk.

Removing dangling jobs will bring the folders on disk and the Automation Engine database back in sync.

Click **Remove** to make the database check for each folder whether or not the folder - as contained in the database - still exists.

If the folder does not exist, the job will be removed from the Automation Engine database. This implies that instead of deleting jobs via the Pilot, you are able to delete jobs by deleting the corresponding folder.



Create DCS Master

Inks definition	
Short Name	Full Name
k	Black
y	Yellow
m	Magenta
c	Cyan

When you use the Create Master DCS task you might want to map short names used in plates (that might not mean anything) to full ink names. This is handy for example for plates with international ink definitions. Default, C,M,Y and K are mapped unto Cyan, Magenta, Yellow and Black.

To add an ink definition, double-click an input field and you can start typing. Use TAB to switch fields.

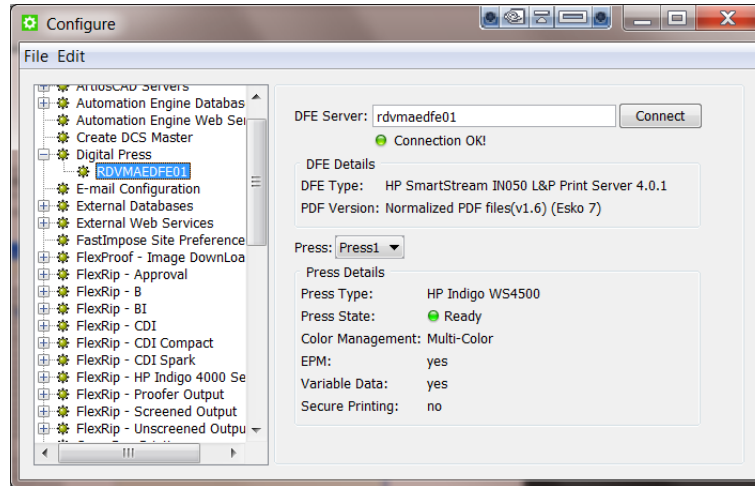
Digital Press

Use this to connect to a digital press through the DFE (Digital Front End) server that drives the press.

1. Select **Digital Press** and use **File > New** to create a new digital press connection.
2. Enter the name of your **DFE Server** and click **Connect**.

Note: Make sure the DFE server is running and reachable over the network.

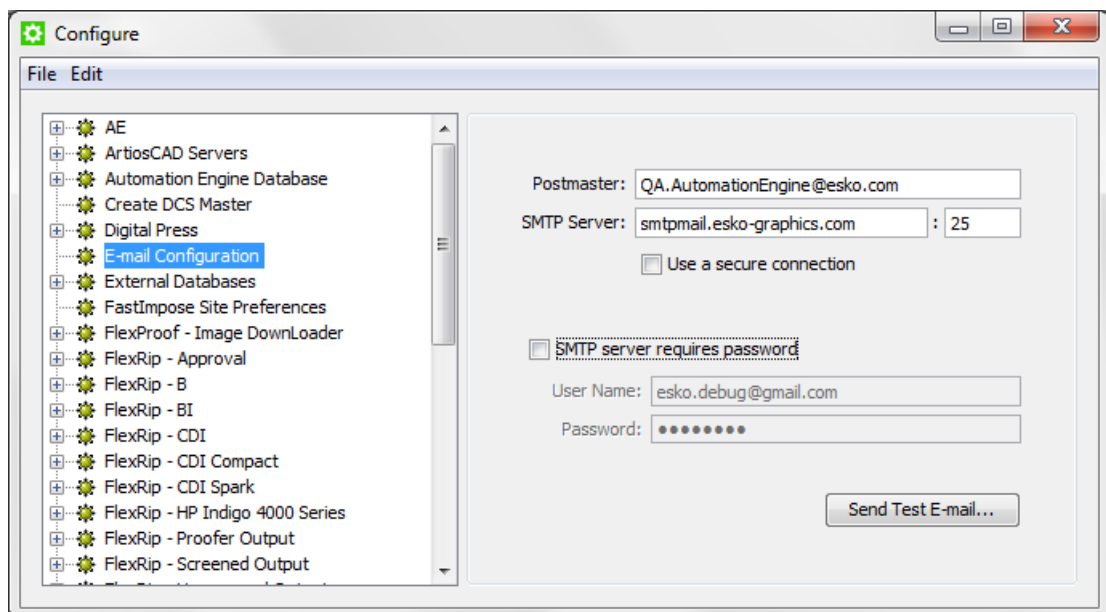
If your connection is successful, you will see **Connection OK!** and details about the DFE server.



3. If your DFE server drives several digital presses, select the **Press** you want to connect to from Automation Engine.
You will see details about the digital press.
4. Use **File > Save** to save your connection.

E-Mail Configuration

Mail Server Tab



- **PostMaster** is the email address that will receive a warning mail when an email could not be delivered.
- Check with your IT department for the correct settings of the **SMTP Server** and **SMTP Port**.
- Select the checkbox if you want to **Use a secure connection** instead of an unrestricted connection.
- If the **SMTP server** requests authentication, fill out the **User Name** and **Password** fields.
- Click on the button to **Send a Test E-mail**.

The following authentication schemes are supported by the Automation Engine mail component: **PLAIN**, **LOGIN** and **DIGEST-MD5**.

Send Test E-mail

- **From:** Indicates the address you're using to send the test email. If no address is filled in, the postmaster address is used. You can choose a different sender to test if the mail server allows you to send emails on behalf of others. The **Send E-Mail** task relies on this behavior as it tries to send emails on behalf of the user that launches the task.
- **To:** Fill in a valid email address that will receive the test mail.
- **Message:** You can enter a personal message or leave the default text.
- When finished, click **Send** to see whether the mail system is working properly.

External Databases

External Databases are databases from external systems (not part of Automation Engine). Automation Engine uses links to such databases in

- a **Database Query SmartName**. Read more in [Database Query SmartNames](#) on page 968.
- a **Database Access Point**. Read more in [Database Access Point](#) on page 1245.
- the **Interact with Database** task. Read more in [Interact with Database task](#) on page 1274.



Attention: When integrating Automation Engine with an external database make sure you are compliant with the license agreements of that database manufacturer.

Note: Configuring these links requires knowledge about database setups. If necessary contact your IT database administrator to figure out correct settings.

Configuring External Databases

1. Go to **Tools > Configure > External Databases** .

2. Click **File > New** to configure a new link to a database.

A new database name is created with the name **db_number** (the number is the chronological number of the database).

3. In the left-hand window, click to select and then click again to rename on the name of that new database link and enter the name you want. Press **Enter** to confirm.

4. Choose the appropriate **DBMS Type**.

These Database Management Systems (DBMS) are supported:

- **Microsoft SQL Server**
- **ODBC Source**

Note: ODBC SmartNames created before version 14 remain unchanged but are deprecated and renamed to the **Type '32 Bit ODBC Query'**.

- **Oracle**
- **Other** (when a JDBC driver is used).



Attention:

Esko provides the drivers for both SQL and Oracle DBMS types. Not all database drivers can be freely distributed by Esko, and not all drivers have been tested. If you need to connect to **Other** types you have 2 options:

- connect using an ODBC System DSN for the database
- connect using a JDBC driver.

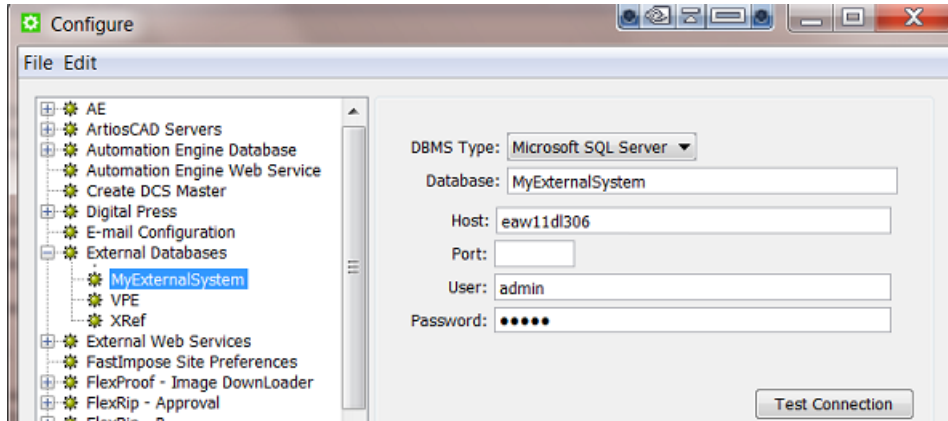
When your database manufacturer allows both ways, we recommend to use JDBC as a connection method. A JDBC connection is directly configured in Automation Engine, whereas for ODBC you need to pass Window's system DSN setup.

5. Enter the connection details. These are different per type of DBMS. See examples in [Specifics per DBMS](#) on page 639.
6. If required for the DBMS, enter the name of the **Host**. This is the database server (computer) name.
7. If required for the DBMS, and if required, enter the **Port** number.
8. If required for the DBMS, enter the **User** name and **Password** that will be used to access the database.
9. Click '**Test Connection**' to test if your setup is valid. If it is not valid, the error message might contain details about the problem.
10. **Disclaimer:** Be aware of the risks of direct database queries and of the fact that Esko takes no liability for them. You need to agree before this link can be saved.
11. Click **File > Save** to save the configuration.

See the next section for some [specifics per DBMS](#).

Specifics per DBMS

Connecting to a Microsoft SQL Server



Connecting to a database server via ODBC

If the database manufacturer does not provide a JDBC driver, you can connect to the database using an ODBC link.

When the name of your system data source is for example 'ProductionDetails', then enter it in the **Database** name field as 'jdbc:odbc:ProductionDetails'.

Note:

32-bit versus 64-bit ODBC:

- On 32-bit hardware (for example Windows Server 2003), Automation Engine tools that use these database links can only work with **32-bit Windows ODBC Data Sources**. You need to add the Data Source using the 32-bit ODBC Administrator tool (%systemdrive%\Windows\System32\Odbcad32.exe).
- On 64-bit hardware (for example Windows Server 2008 R2), Automation Engine tools that use these database links can only work with **64-bit Windows ODBC Data Sources**. You need to add the Data Source using the 64-bit ODBC Administrator tool (%systemdrive%\Windows\System32\Odbcad32.exe).

Note: As of Automation Engine 14.1(2015), Esko will only support 64-bit systems for the Automation Engine server(s).

Connecting to an Oracle Server

DBMS Type: Oracle

Service Name: DBPOLLER

Host: DBServer

Port: 1521

User: SYSTEM

Password: ●●●●●●

Test Connection

Connecting an "Other" DBMS

Other DBMS can be accessed if you can get the necessary drivers and information on how to access the DBMS via standard JDBC technology.

- Download the JDBC driver (a jar file) from the manufacturer's website into a **Container** that is accessible to the Automation Engine (Master) server..
- Enter the required configuration fields. See an example:

Connection Type: Other

Driver(JDBC): com.mysql.jdbc.Driver

Driver File: file:///DBSERVER01/Share/jdbcdrivers/mysql-connectorj.jar

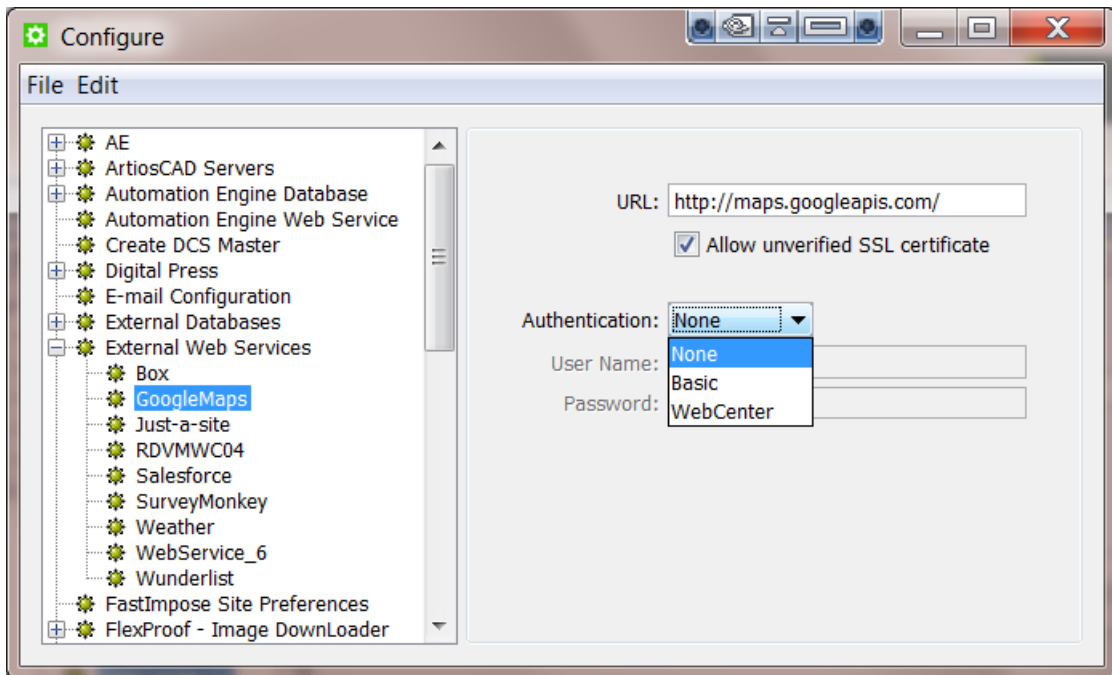
Database URL: jdbc:mysql:///DBSERVER01/ProductionDetails?user=admin&password=admin&autoReconnect=true

Test Connection

- Check your JDBC driver manual for more details.

External Web Services

Using the **Interact with Web Service task**, Automation Engine can communicate with web services from external systems. Read more about this task in [Interact with Web Service task](#) on page 1276.



To configure such a web service,

1. Go to **Tools > Configure** .
2. In the **Configure** window, select **External Web Services**.
3. Press the **Insert** key or go to **File > New** . Specify a name for the new **External Web Service**.
4. Specify **URL**.

You can use a secure connection via SSL (a URL starting with `https://`) with or without a valid SSL certificate signed by a "Certificate Authority" (CA).

When using a secure connection without a valid certificate, import the certificate into the server **java KeyStore**. Use the following command on the Automation Engine server:

```
keytool -import -alias mydomain -file myfile.crt -keystore C:\Esko\bg_prog_fastserver_v40\jre\lib\security\cacerts
```

Check the **Allow unverified SSL Certificate** option to do this.



Warning: This setting is not secure.

If you need to pass via a proxy, configure the proxy settings via **Tools > Configure > Web Proxy** .

5. Select the **Authentication** mode from the drop-down options: **None**, **Basic** or **WebCenter**.



Caution: About the "**WebCenter**" type authentication: if you try to connect to a WebCenter 14 or more recent, you do not need to configure the WebCenter server, you can indicate it in the task. If you try to connect to an older WebCenter server, you have to configure the WebCenter server here and use the **Interact with Web Service** task instead of the **Interact with WebCenter** task.

6. Provide **User Name** and **Password**.

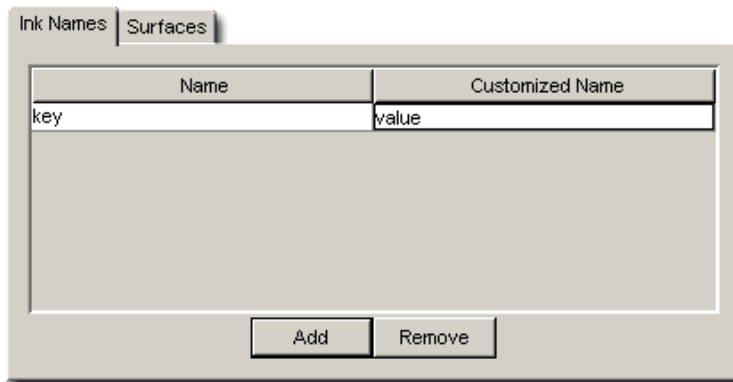
7. Click **File > Save** .

FastImpose Site Preferences

The FastImpose site preferences are useful when you want to map certain names to customized ones. For example if you want to map 'c' to the French 'bleu' on your plates.

Ink Names

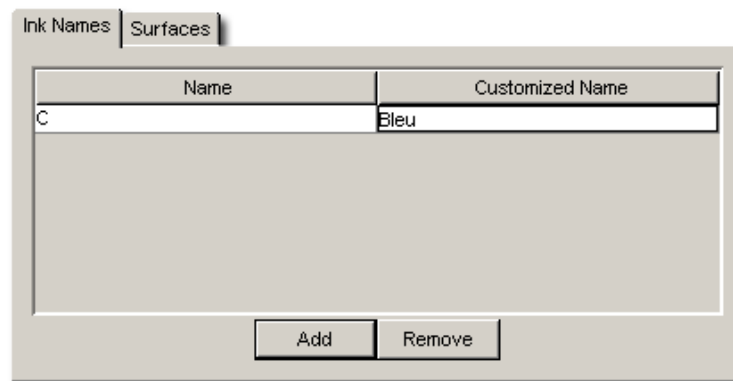
Click Add and two values are added to the list: key and value, double click the words to edit them:



Tip:

A black rectangle around the field indicates that you can edit it.

If you want to 'C' to 'Bleu' you enter C in the Name column, and 'Bleu' in the Customized Name column:



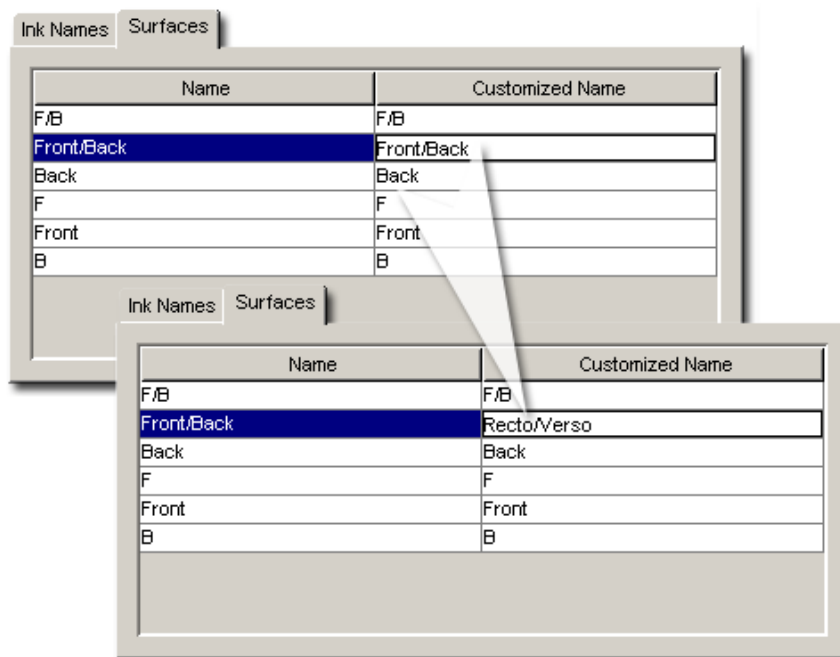
This will change nothing to your job, but if you used marks that would output 'C' on plate, now it will read 'Bleu'.

Surfaces

A number of values are defined (F/B, Front/Back, Back, F, Front, B) but sometimes these definitions do not conform to your language or your way of working, so you can change them. Let's say you want Front/Back to read Recto/Verso, you just edit the Customized Name by double clicking.

Tip:

A black rectangle around the field indicates that you can edit it.



FlexProof - Image Downloader

Select the Image Downloader folder icon, open the **File** Menu and choose **New**. Three tabs appear:

- General
- Queue
- Merge

Note:

When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Proofer Node Enter the FlexProof computer name.

Fill out these fields before you click **Synchronize with ProofNode**. This may take a few seconds, the system will fill out the 'Type' field.

Note:

The name of the Image Downloader ProofNode is case sensitive. This is not the case for the other devices.

Tip:

You can retrieve your host name by typing host name in a command prompt.

Queue

Queue This field will have ImageDownloader1Queue in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge: Yes is the default setting.

Select a **unit**. **Margin** Set an extra margin around the separations.

Gap Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

FlexRip - Approval

Select the FlexRip - Approval folder icon, open the **File** Menu and choose **New**. Three tabs appear:

- General
- Queue
- Merge

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip or FlexProof computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Approval1 Queue` in it initially. It will update automatically when you rename the device.

Queue Job Limit A Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2.

Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the Rip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations. If you decide to set it to **Yes**, other toggles will appear.

Gap Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

FlexRip - B

Select the correct device folder icon, open the **File** Menu and choose **New**.

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

Three tabs appear:

- General
- Queue
- Merge

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will initially read MegaSetter1Queue. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2.

Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Optimize Expose Sequence This toggle ensures 'faster' output (all separations will be ripped in one pass). For some jobs, however, this way of working results in memory problems, when this happens, you can deselect this toggle, all separations will be ripped separately (the rip will be restarted after every separation).

Merge

Merge We advise to set this to **No** as merging will happen after this pass on the way to the Imagesetter, in doing so you achieve high flexibility in merging (positive and negative can be merged on one film).

Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear.

Gap Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

FlexRip - BI (Barco Imagesetter)

Select the BI folder icon, open the **File** Menu and choose **New**. Three tabs appear:

- General
- Queue
- Merge

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have BiModel11 Queue in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2.

Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Optimize Expose Sequence: This toggle ensures 'faster' output (all separations will be ripped in one pass). For some jobs, however, this way of working results in memory problems, when this happens, you can deselect this toggle, all separations will be ripped separately (the rip will be restarted after every separation).

Merge

Merge The dropdown list only offers 'No' as an option. That's because the BI cannot merge jobs. Select a **unit**.

Margin Set an extra margin around the separations.

FlexRip CDI, CDI Spark and CDI Compact

These three can be configured in the same way.

Select the correct device folder icon, open the **File Menu** and choose **New**.

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

Four tabs appear:

- General
- Queue
- Merge
- JDF
- CDI

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its **EskoLink** queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will initially read CDI1 Queue, CDI Compact1 Queue or CDI Spark1 Queue. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher **EskoLink** queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that **EskoLink** being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Optimize Expose Sequence: This toggle ensures 'faster' output (all separations will be ripped in one pass). For some jobs, however, this way of working results in memory problems, when this happens, you can deselect this toggle, all separations will be ripped separately (the rip will be restarted after every separation).

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear.

Gap Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

JDF

If your CDI is equipped with a JDF/JMF enabled Device Controller, you can specify on which URL (http address) this JDF/JMF Device Controller can be reached. To this address an imagesetting JDF file, together with the ripped data will be sent. The accompanying JMF file will indicate that the data can be submitted in the queue.

A JDF/JMF Device Controller installs a tight link between the Automation Engine Server and the Grapholas system. Advantages are:

- automatic transport of the ripped bitmap files towards the Grapholas system for access in the PlatePrep application.
- closer followup of the real plate exposure process from within the Automation Engine Pilot.

CDI

Pixelclock is set to 360000

The **Perimeter** of the drum in mm, inches or mil.

The content of this tab depends on the Grapholas software. If you have recent Grapholas software these settings will be 'automatic'. The CDI Spark for example does not have these settings any more.

FlexRip Indigo 4000 Series

Select the Indigo 4000 Series icon, open the **File** Menu and choose **New**.

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

Two tabs appear:

- General
- Queue

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have **HP Indigo 45000 Queue** in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2.

Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

FlexRip - Proofer Output

Select the Proofer Output icon, open the **File** Menu and choose **New**.

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

Three tabs appear:

- General
- Queue
- Proof

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip or FlexProof computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Proofer Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Proof

A workflow choice must be made: or you choose to let the FlexRip recombine the separations or you choose to add a 'Proof (FlexProof)' task.

Recombine separations:

- **On the FlexRip**, you can add a margin (in mm/mil/inch), no merging is possible.
- **By sequencing a 'Proof (Image Downloader)' Task**: you have to specify the 'FlexProof - Image Downloader' device from the dropdown list. (The list only shows those devices that are configured for the same node (RIP and Proof node must be identical).

FlexRip - Screened Output

DCS

These are the settings of FlexRip devices.

Select Screened Output > select DCS > Select 'New' from the File Menu.

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Screened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **Gap** Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

TIFF

Select **Screened Output** > select **TIFF** > open the **File** Menu and choose **New**.

Note: When you have defined settings, do not forget to **save** them. Click **Save** in the **File** Menu.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Screened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose

commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Optimize Expose Sequence This toggle ensures 'faster' output (all separations will be ripped in one pass). For some jobs, however, this way of working results in memory problems, when this happens, you can deselect this toggle, all separations will be ripped separately (the rip will be restarted after every separation).

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **Gap** Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

Note:

Putting Merge on 'Yes' may result in a very large output file.

PDF

Select Screened Output > select PDF > open the **File** Menu and choose **New**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Screened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **GapExtra** added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

Note:

Putting Merge on 'Yes' may result in a very large output file.

PS

Select Screened Output > select PS >open the **File** Menu and choose **New**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Screened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting

this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **Gap** Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

Note:

Putting Merge on 'Yes' may result in a very large output file.

FlexRip - Unscreened Output

DCS

These are the settings of FlexRip devices.

Select Unscreened Output > Select DCS > open the **File** Menu and choose **new**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Screened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose

commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **Gap** Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

Note:

Putting Merge on 'Yes' may result in a very large output file.

TIFF

Click the Screened Output icon, select TIFF from the list, open the **File** Menu and choose **New**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Unscreened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the

following rules:- having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Optimize Expose Sequence This toggle ensures 'faster' output (all separations will be ripped in one pass). For some jobs, however, this way of working results in memory problems, when this happens, you can deselect this toggle, all separations will be ripped separately (the rip will be restarted after every separation).

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear.

Gap Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

PDF

Click the Screened Output icon, select PDF from the list, open the **File** Menu and choose **New**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Unscreened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules:- having jobs ripped earlier on the FlexRip does not mean your output device

can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **Gap** Extra added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

PS

Click the Screened Output icon, select PS from the list, open the **File** Menu and choose **New**.

Three tabs appear:

- **General**
- **Queue**
- **Merge**

Note: When you have defined settings, do not forget to **save** them. Click Save in the File Menu.

General

Type When you configure a new device, the system will tell you that the Type is Unknown.

Rip Node Enter the FlexRip computer name. Also enter the exact **EskoLink Portnr.** that the specific Dispatcher has for its EskoLink queue.

Fill out these fields before you click **Synchronize with Rip**. This may take a few seconds.

Queue

Queue This field will have `Unscreened Output1 Queue` in it initially. This field will update automatically when you rename the device.

Queue Job Limit A FlexRip's Dispatcher EskoLink queue can rip (= handle the expose-command of) 2 jobs at the same time. If you want the Automation Engine Queue to allow sending 2 Expose commands at the same time, then set the job-limit of the Automation Engine queue on 2. Putting this job-limit on 1 will result in each expose command to that EskoLink being handled one by one: the second job will be ripped after the first one is completely finished. Also remember the following rules: - having jobs ripped earlier on the FlexRip does not mean your output device can output them any earlier. - some people prefer the Automation Engine queue 'job limit' on 1 because that guarantees the 'first in first out' rule. Do consult your Esko workflow trainer or customer service for more advice on these Queue 'job limits'.

Merge

Merge Select a **unit**.

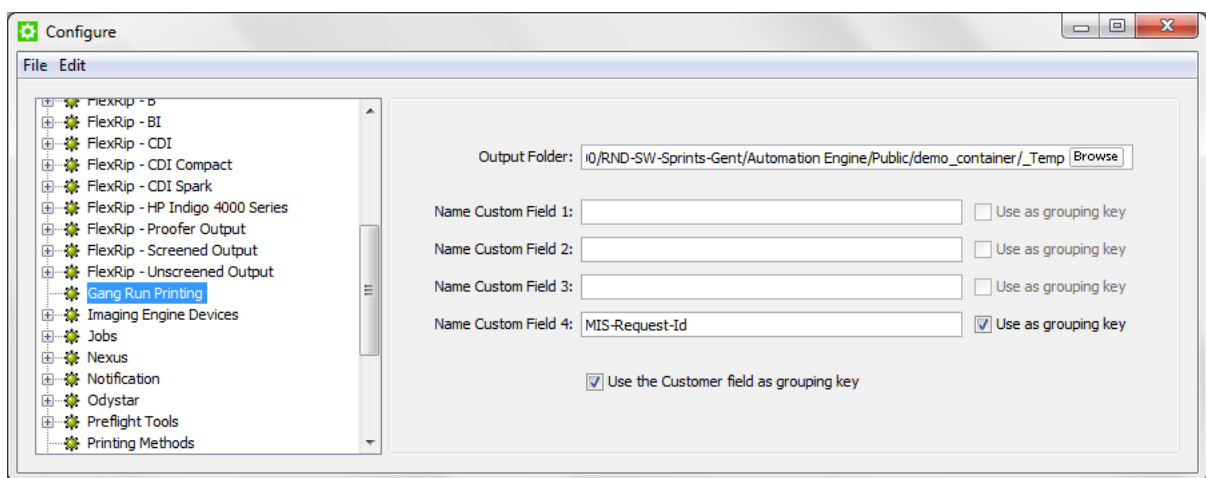
Margin Set an extra margin around the separations.

If you decide to set it to **Yes**, other toggles will appear. **GapExtra** added zone (of empty pixels) around each separation that will result in a 'gap' between the merged separations.

Minimum Fill Enter a percentage. The minimum fill is an economic measure, it ensures that no output is generated before the fill quota is reached.

Gang Run Printing

You can group compatible jobs in the Gang Run view. For this, define one or more custom fields and specify their use as a grouping key. When jobs are submitted to the gang run for a certain substrate, they are grouped automatically when they have the same grouping key values. Typical examples of grouping keys are: Customer, Printing Method, Finishing Method, etc.



1. Go to **Tools > Configure** .
2. In the **Configure** window, select **Gang Run Printing**.
3. Press **Insert** key or go to **File > New** .
4. Specify names for each custom field under **Name Custom Field**.
5. Enable **Use as grouping key** to use Custom Field as grouping key.
6. You may select **Use the Customer field as grouping key**.
7. Click **File > Save** .

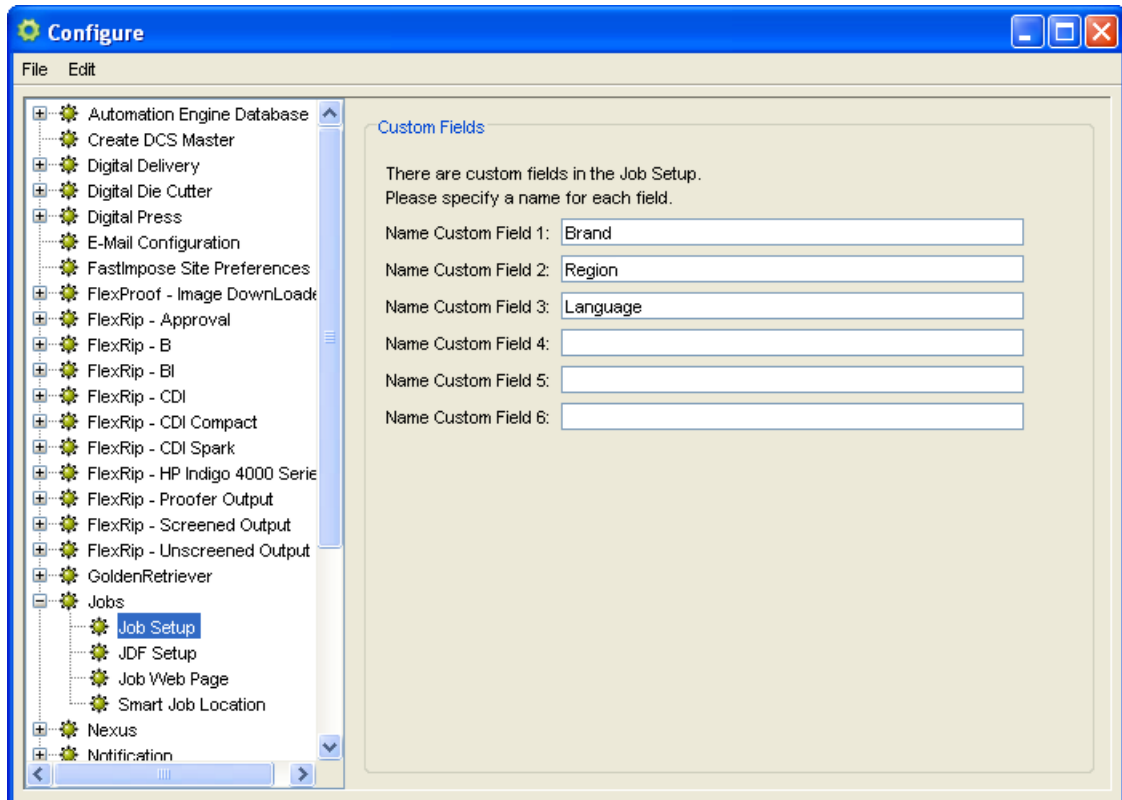
Jobs

Job Setup

Six Custom Fields can be specified. When entering names in custom fields 3 to 6, an extra tab **Categorization** will appear in the **Job Setup** dialog box.

Tip: Examples: region - brand.

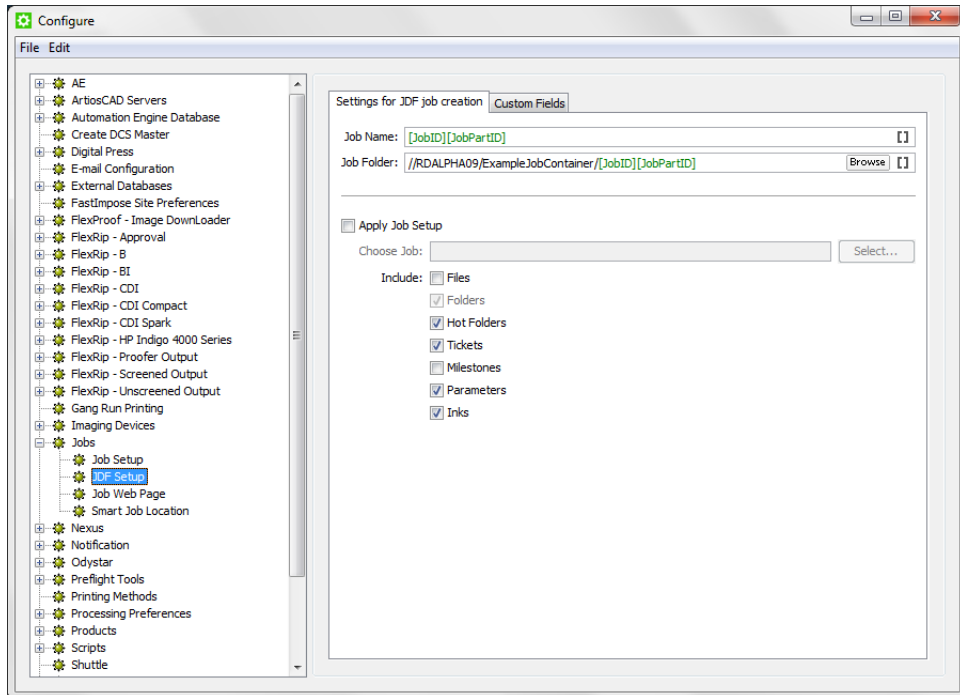
Note: Think about these carefully, changing this later will create confusing lists of values for these fields.



JDF Setup

JDF Setup offers the possibility to define default settings in case you create (new) Jobs via JDF/JMF.

If you want to use the same Job structure each time a new Job is created via JDF, this can be done by taking over the Job structure of a specified Job. Check **Apply Job Setup** and click **Select** if you want to copy the setup from another Job. You can indicate what items should be copied. Traditionally it will be **Tickets**, **Parameters** and sometimes **Inks**.

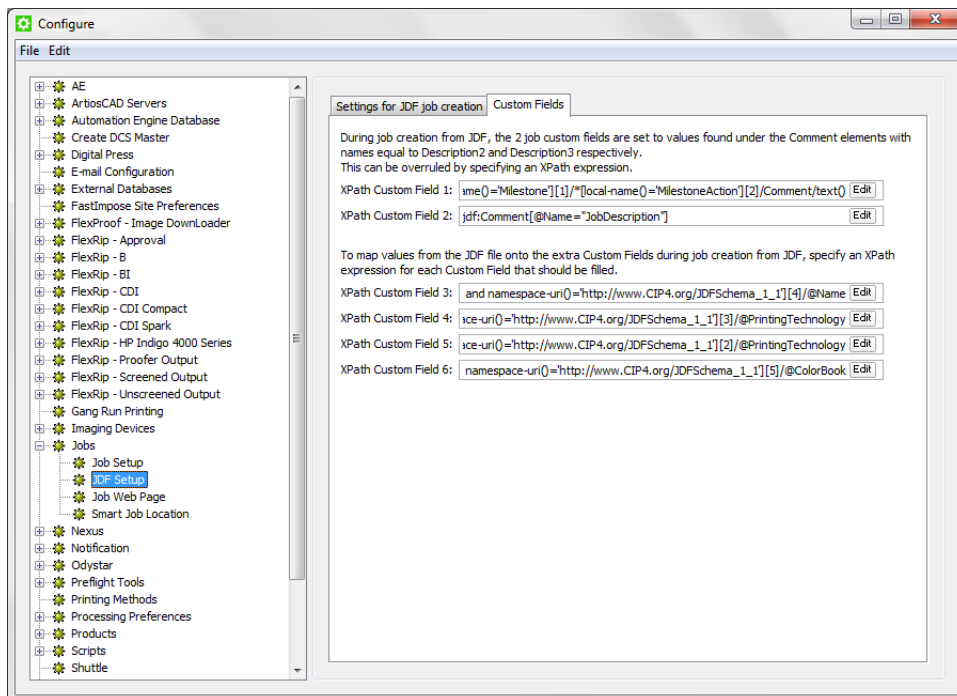


On the **Custom Fields** tab, you can define how the data in the Comment elements from the JDF file is used to populate the custom fields (Categories) that appear in the Job Setup dialog.

To do this, type an XPath expression into the field corresponding to the category that you want to fill in automatically, or click **Edit** to open the XPath builder.

Note:

- The JDF that was loaded in the XPath builder is remembered while the **Configure** dialog is open.
- When editing the next XPath, the same JDF will be automatically used as the example XML in the XPath builder.

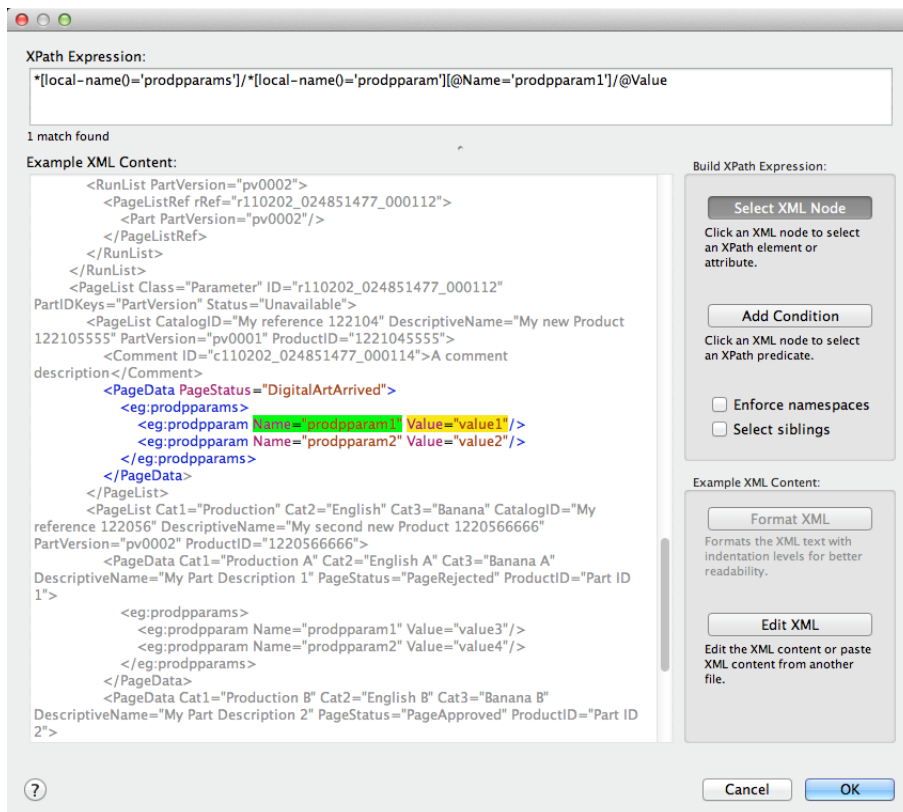


XPath Builder

When you click **Edit** in any field that allows XPath Expressions, the **XPath Builder** opens.

The XPath Builder supports alternative starting points:

- The XPath is not resolved relative to the root node of the XML document, but from another XML element.
- When a proper JDF is used as the example XML, a list of possible starting points is generated.
- The first possible starting point is automatically used to configure the XPath builder.
- In this example, the XPath builder is used to create an XPath for a product part attribute:

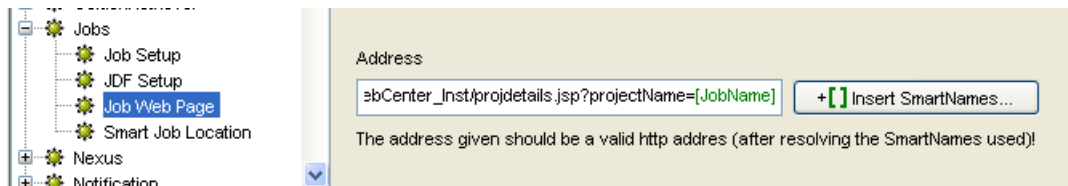


- The XPath builder automatically uses the first **PageData** node as starting point. This is visualised by showing the content outside the **PageData** node in grey.
- It is still possible to select nodes outside the **PageData** node as result node or as predicate node.
- When selecting nodes outside the **PageData** node, the XPath builder will create absolute XPaths.
- When you want, you can modify these absolute paths by making them relative to the **PageData** node using `../`
- While the XPath builder automatically selects the appropriate starting point, it is still possible to manually change the starting point by clicking on another node while holding down Command (Mac) or CTRL (Windows).

Job Web Page

Define the Job Web Page's path here. Obviously, it needs to be a valid URL, so if you use SmartNames, make sure they resolve correctly. This allows you to go to your WebCenter project from within your Automation Engine Job!

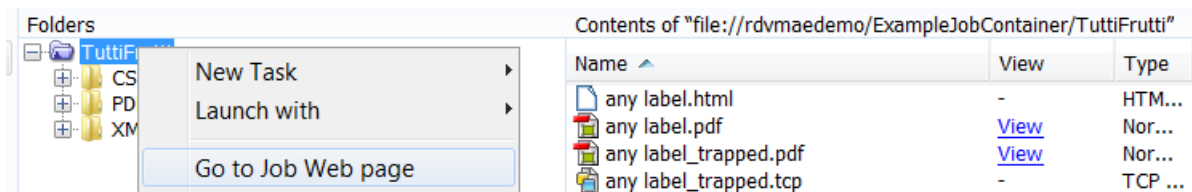
Enter the name of your WebCenter site, and add `"/projdetails.jsp?projectName=[JobName]` at the end.



In the Pilot, you can right-click a job folder and select 'Go to Job Web Page' from the pop up menu. A browser will start up and load the path defined here.



Attention: This only works if you map the job on a project by name, but this will be the most common case.



Smart Job Location

The Job's job folder can be defined by a smart path named **Smart Job Location**. Its configuration calculates the location of the job folder based on a set of rules as defined in Configure panel. This avoids having to enter the path manually in the 'New Job' or 'Job Setup' dialogs.

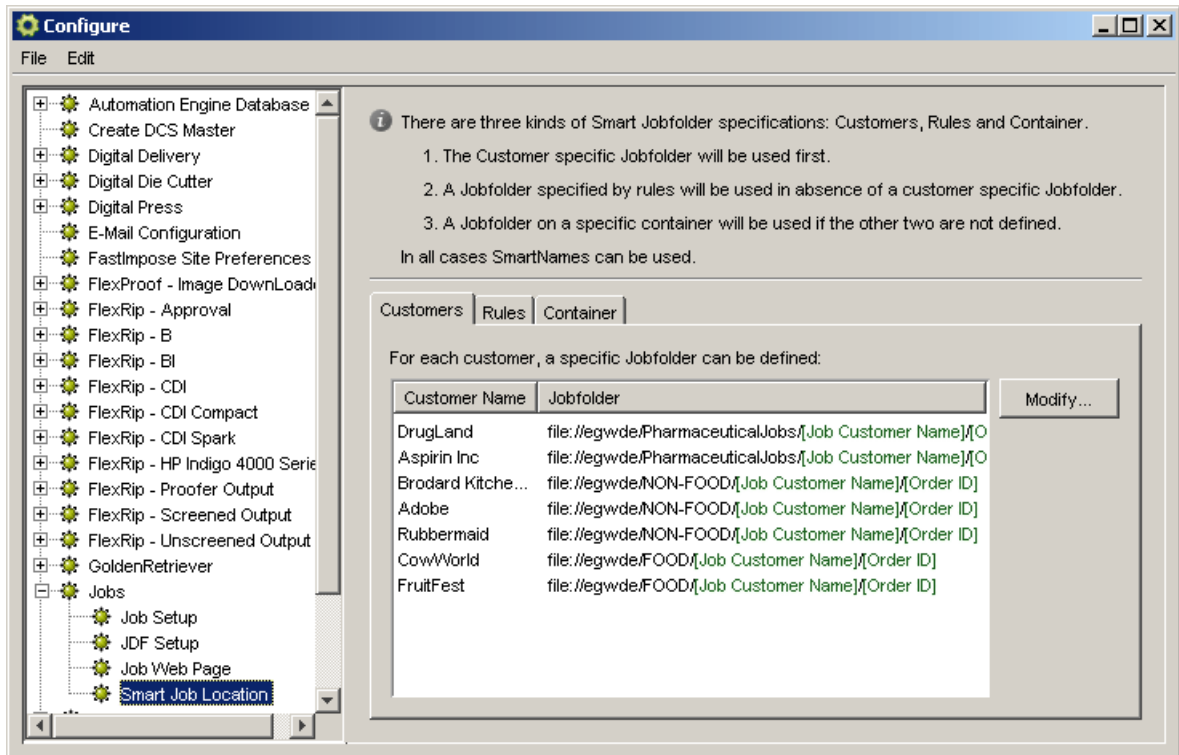
There are three types of specifications that you can use to set up the Smart Job Location. You can base the location on the **Customer** of that job, or on a custom **Rule** or you can specify (only) the **Container**.

These 3 methods will be checked in their specific order: if no rule is found for the Customer, the set of custom rules will be checked. If there is none that matches (that returns a valid path for the job folder), then the Container setting will be checked.

In the 'New Job' or 'Job Setup' dialog boxes you will see an extra toggle **Show Job Location**. Check this toggle to make Automation Engine use the Smart Job Location rules.

We now have a closer look at these types of rules:

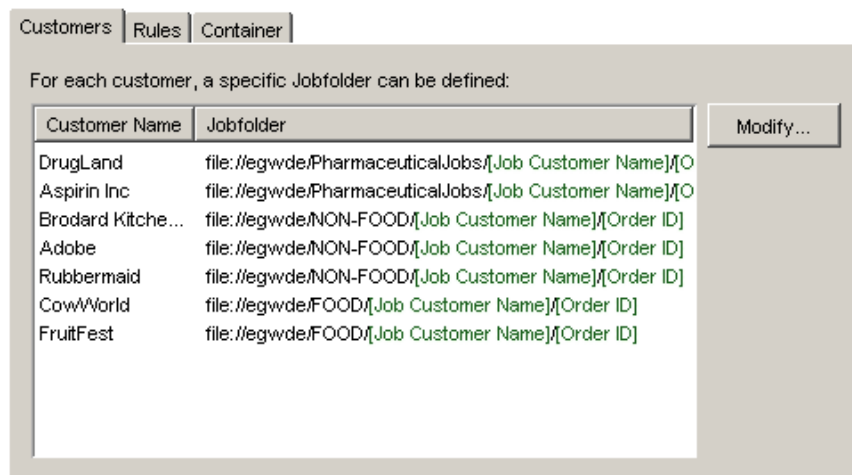
- **Customers:** allows to set simple rule based on the customer name. It allows you to specify 1 smart place for all jobs of that specific customer.
- **Rules:** allows you to set extensive conditional rules.
- **Container:** allows you to set a fixed Container for *all* job folders to be created (from now on).



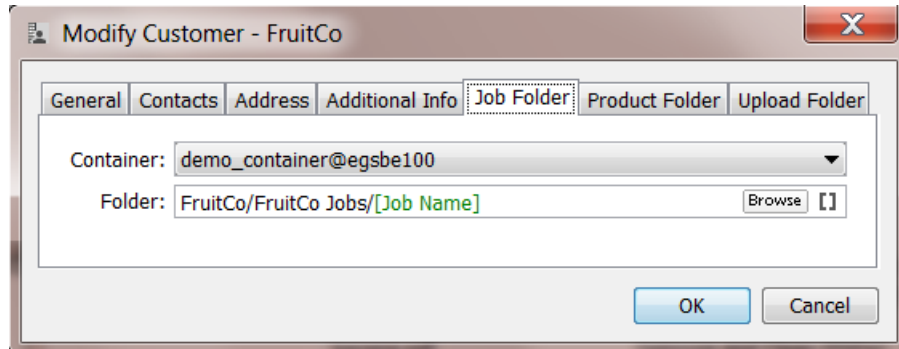
Tip: It is possible to use this Smart Job Location as a SmartName for the indication of the folder for the Job in JDF. Use <SJL\>.

Customers

Set a rule here if you want to control the job location per customer. This so offers 1 smart place for all jobs of that customer.



The rule that you set here will become part of the customer definition in **Tools > Customers**. See this example:

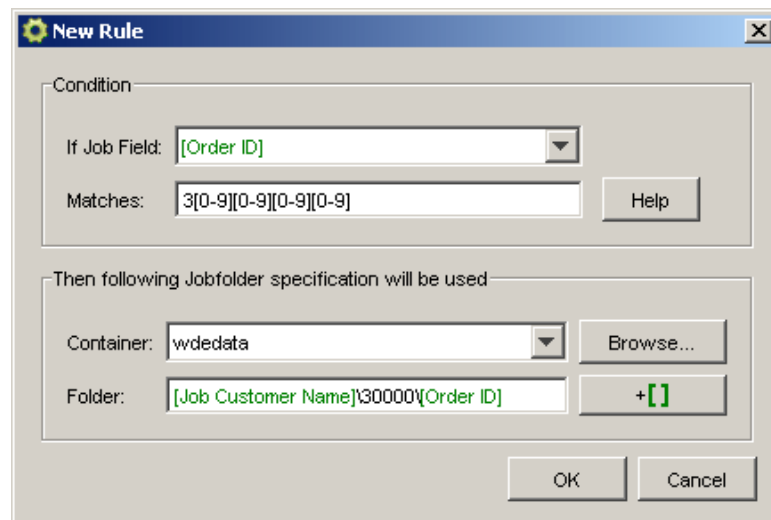


Rules

If no customer is selected for the new job, or no location is defined for the customer, the systems checks your set of custom rules.

Click the **New Rule** button in the **Rules** tab to set up new rules. It enables extra conditions that you can make as extensive as necessary.

In the **Matches** field, you can use [Regular Expressions](#) can be used. Click the **Help** button for an overview of the characters that can be used to indicate your regular expressions.



This example shows how you can distribute many jobs automatically over subfolders, based on the number in the job's Order ID:

Customers Rules Container

The first matching rule will be used:

Condition	Jobfolder
[Order ID] matches [0-1][0-9][0-9][0-9]	wdedata\[Job Customer Name\]00000\[Order ID\]
[Order ID] matches 2[0-9][0-9][0-9]	wdedata\[Job Customer Name\]20000\[Order ID\]
[Order ID] matches 3[0-9][0-9][0-9]	wdedata\[Job Customer Name\]30000\[Order ID\]

New Rule...
Remove Rule
Modify Rule...
Move Up
Move Down

Container

If no rules apply to the new job, the default job location will be used.

In case the Checks on **Customers** and **Rules** did not return a valid path for a job folder, then this rule will be taken: you here define a main choice for the Container.

Customers Rules Container

Container: NON-FOOD

Folder: [Job Customer Name]\[Order ID]

This rule is particularly useful when you have a system where you have all jobs created on a Container (a share on a hard disk) until that share runs full. In that case, you create a next Container and change the general rule here to the new Container, again until that one fills up.

Nexus

1. Choose **Tools > Configure** and select the **Nexus** category.
2. Click **File > New**
3. Enter the name of the Nexus server in the **Host** field.
4. Enter the **Port** on which the Nexus server is accessible.
5. Click **Test Connection** to verify the settings you entered.

Note: You can also test whether the Nexus SoapServer is online by browsing to the following url in your favorite web browser: <http://NexusServerName:5182>

Notification

The configuration of the notification consists of 3 parts:

- Rules
- Via File
- Templates

Notification Rules

Instead of deciding on notifications per ticket, you can define general notification rules. You can set a notification rule on a specific task or on a specific device (queue).

There are two tabs: **Tasks** and **Devices**. Both these tabs list either tasks or devices for which you can define notification rules.

For example:

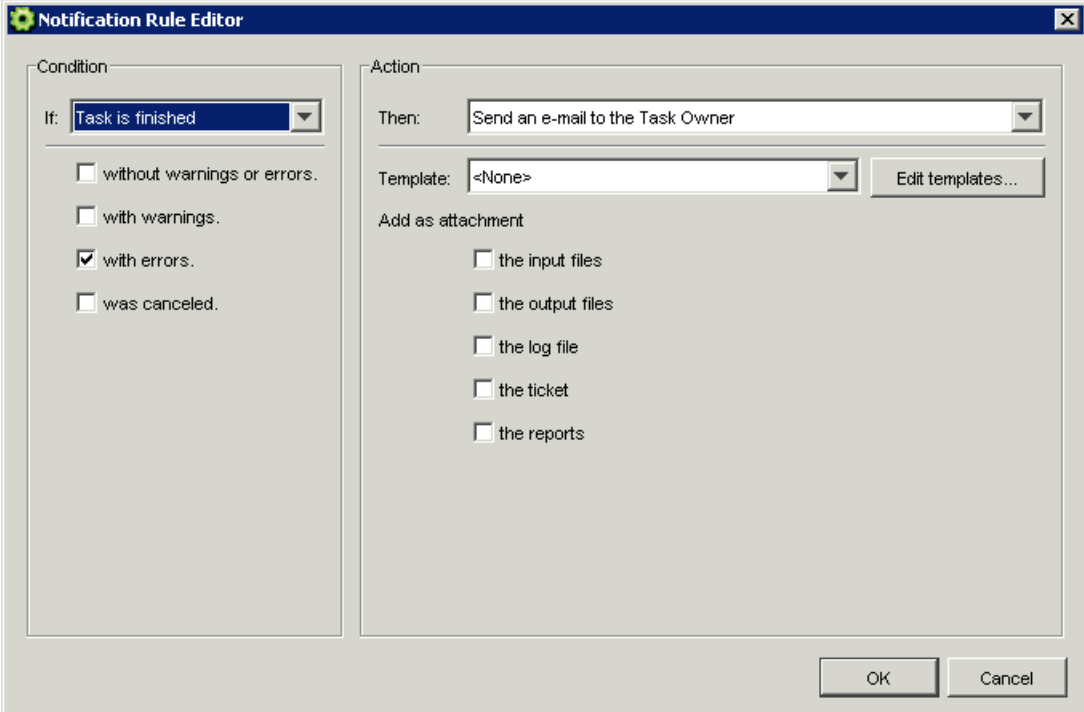
- Milestone task: send out an XML every time a Milestone task ends.
- CDI Compact device: send out XML on all actions on that CDI device. Have each notification added to the main XML representing that CDI.

Note: It is not possible to set up a notification rule for a device if a dedicated rule can be set up for the corresponding task.

Defining a Notification Rule

1. Click 'Add'.

The **Notification Rule Editor** pops up.



The screenshot shows the 'Notification Rule Editor' dialog box. It is divided into two main sections: 'Condition' and 'Action'.

Condition Section:

- 'If:' dropdown menu is set to 'Task is finished'.
- Four checkboxes are listed below:
 - without warnings or errors.
 - with warnings.
 - with errors.
 - was canceled.

Action Section:

- 'Then:' dropdown menu is set to 'Send an e-mail to the Task Owner'.
- 'Template:' dropdown menu is set to '<None>'. There is an 'Edit templates...' button to the right.
- 'Add as attachment' section has five checkboxes:
 - the input files
 - the output files
 - the log file
 - the ticket
 - the reports

At the bottom right, there are 'OK' and 'Cancel' buttons.

2. Define the Notification.

Define the condition for the notification **and** the action.

A notification may consist of an e-mail sent to the task owner (or to any other valid address), it may also be an entry in a XML/text file, or an XML/text file in a certain folder.

The **place** where the Text and XML notifications end up is defined in the [Notification via file](#) part of the Configure Tool.

Note: When selecting “Send an e-mail”, you can **add more than 1 e-mail address** in the 'To' field (including SmartNames) by using ' ; ' as a separator.

You can add the input files, the output files, the task's log file and or the ticket to the notification.

The notification can be based on a template. Hit the **Edit templates ...** button to create and modify templates.

For more information, please see ['Edit Notification Templates'](#).

Note: The Template list and Edit templates button will only be visible if templates have been created before.

Note that **SmartNames** can be used in the **To** field, for example to refer to e-mail addresses of:

- the Job's Customer Service Representative,
- up to 3 customer contacts.

3. Click OK.

The notification rule is added to the 'Notification' dialog.

Via File

There are two tabs: this allows you to determine the default location of the **Text** and **XML** report (entries). You can enter a path for both Text and XML notifications. Use the **Browse** buttons to define a different location.

The image shows two panels of a configuration tool. The top panel is titled 'Text' and has a sub-header 'Default place for storing messages:'. It contains two radio button options. The first option, 'Append the message in a LogFile.', is selected and has a text field with the value 'file:./backstage1/Notification/operation.log' and a 'Browse...' button. The second option, 'Create a new file per message in a folder.', is unselected and has a text field with the value 'file:./backstage1/notification/' and a 'Browse...' button. The bottom panel is titled 'XML' and also has a sub-header 'Default place for storing messages:'. It contains two radio button options. The first option, 'Append the message in a LogFile.', is unselected and has an empty text field and a 'Browse...' button. The second option, 'Create a new file per message in a folder.', is selected and has a text field with the value 'rics/log_data_fastserver_v030/Notification/' and a 'Browse...' button. At the bottom of the XML panel, there are two checkboxes: 'Include task messages' (unchecked) and 'Include task parameters' (checked).

Text file notification: You can append the message in a log file (*.log). You can create a new file for each message in the same folder.

XML notification: You can append the message in a log file (*.XML). You can create a new XML file for each message in the same folder.

To set general rules, you need the [Notification Rules](#) part of the Configure tool.

Include task messages: the task messages and / or task parameters will be added to the notification.

Notification Templates

You can base your notification on a template. This means that you prepare a template with a default structure.

You can also use the default template.

Click 'Edit Templates' in the Notification Rule Editor to add a new one, or to copy, modify, rename or delete an existing template.

Please see [Edit Notification Templates](#) for an explanation of how to create a new template.

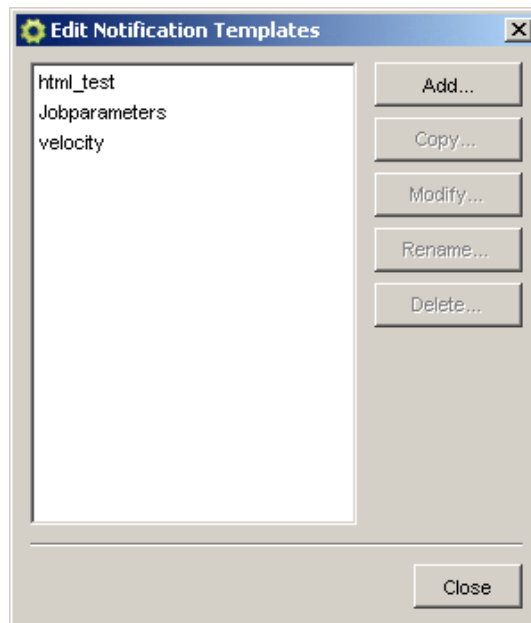
Edit Notification Templates

You can base notifications on a template. This means that the notification will be structured exactly the way you want it to be structured.

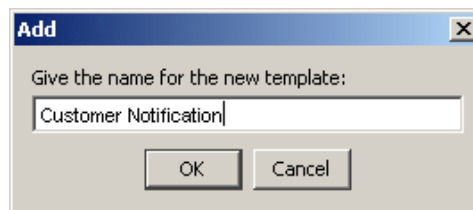
You can either choose an existing template from the list, create a new one, or modify an existing template.

Creating a template

1. Click **Add**.



2. The **Add** dialog pops up. Enter a name for the template. Click **OK**.



3. The **Edit Template** dialog pops up. It has dedicated fields for:
 - the E-mail subject line (SmartName enabled),
 - the text of the E-mail message (SmartName enabled),
 - code-based conditional information (SmartName enabled),
 - attachments.

Edit Template - Smart_Workflow_News

Subject: Automated news from Prepress dep. at Xample - Graphics

Message: Dear [Customer Name Contact 1] at [Customer Name]
Attached is an export PDF from the production file of your design. Please check the document and contact our prepress department asap to discuss or confirm the approval.
Thank you
your CSR:[Job CSR], (Email: [Job CSR E-Mail]) and your prepress operator: [Operator]
at Xample Graphics Ltd.
phone: reception +32.9.216.9.211 - phone: prepress +32.9.216.9.212

Task Information: \$msg.toString()
#end

#end
#end
#end

Attach:

Input Files
 Output Files
 Log File
 Ticket
 Reports

OK Cancel

Tip: You can also **Edit Notification Templates** in the **Configure** tool, provided that you already have templates available: go to **Configure > Notification > Rules > Select a task > Add...**

Odystar

Jobs
Nexus
Notification
Odystar
OdystarServer_1
Pretflight Tools
Processing Preferences

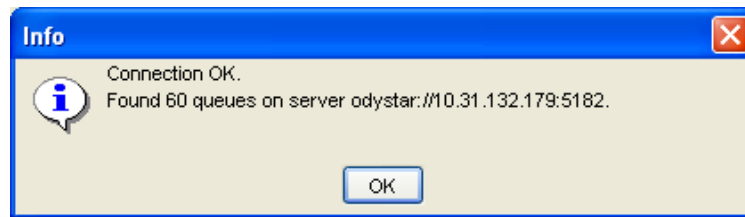
Host: 10.31.132.179
Port: 5182

Test Connection

Enter the name or IP address of the Odystar server in the **Host** field.

Enter the **Port** on which the Odystar server is accessible.

Click **Test Connection** to verify the settings you entered.



Note: You can also test whether the Odystar Hub is online by browsing to the following url in your favorite web browser: `http://OdystarServerNameorAddress:5182`.

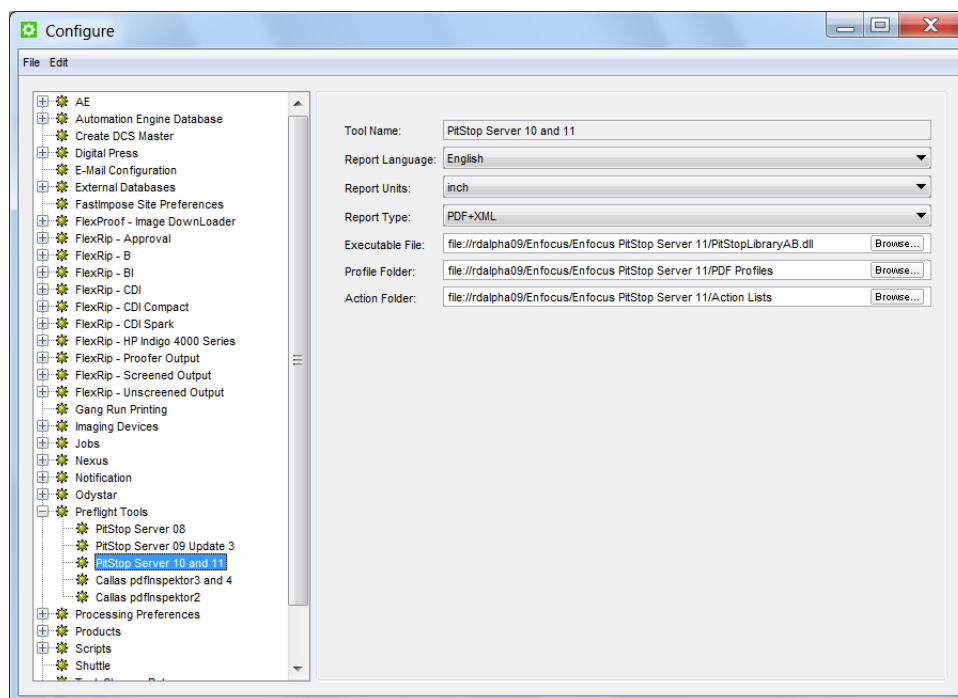
Preflight Tools

Use the **Preflight Tools** category in the Configure tool to define the settings necessary for Automation Engine to work with the supported preflight tools.

This is used in the *Preflight via External Application* on page 405 task.

Important:

- Your PitStop Server application has to run on your Automation Engine server.
- You must share your PitStop folder (located for example in `C:\Program Files\Enfocus\Enfocus PitStop Server 09`) to be able to access it from the Preflight ticket.



Tool Name: select one of the supported preflight tools.

Report Language: select the appropriate language from the list. The report language selection for Enfocus PitStop Server can be set in the PitStop Server preferences.

Report Units: select the unit to be used in the preflight report.

Report Type: select the output type of the preflight report: PDF, XML or both.

Executable File: enter (or browse to) the path to the preflight tool executable file.

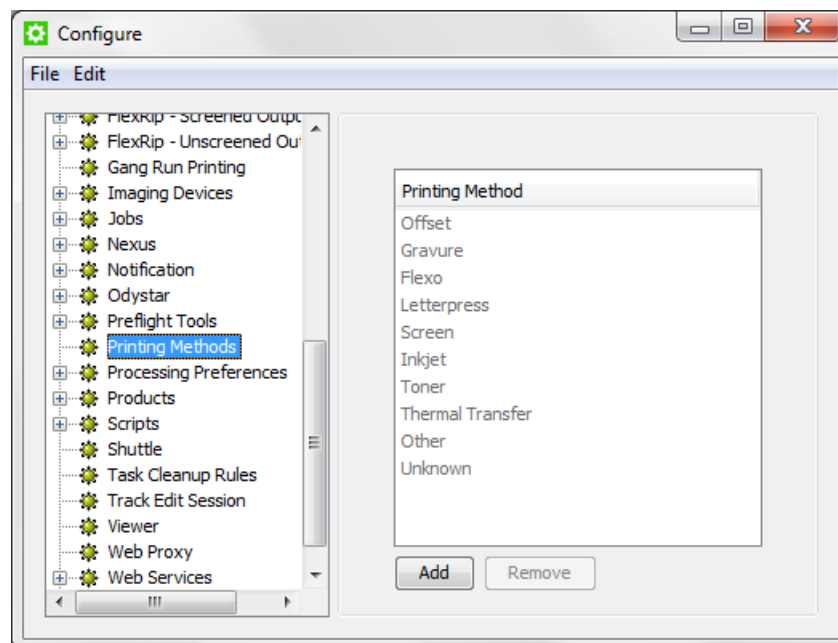
Profile Folder: enter (or browse to) the path to the folder where the default profiles for the selected preflight tool are stored.

Action Folder: enter (or browse to) the path to the folder where the action lists for the selected preflight tool are stored (only from PitStop Server 4). The Preflight task will allow you to select multiple actions from this folder.

Note: Tickets created prior to BackStage 7 Release 5 will be loaded with one single action in the list.

Printing Methods

Printing Methods are defined in the Configure tool. You get a list of standard names. You can add custom ones.

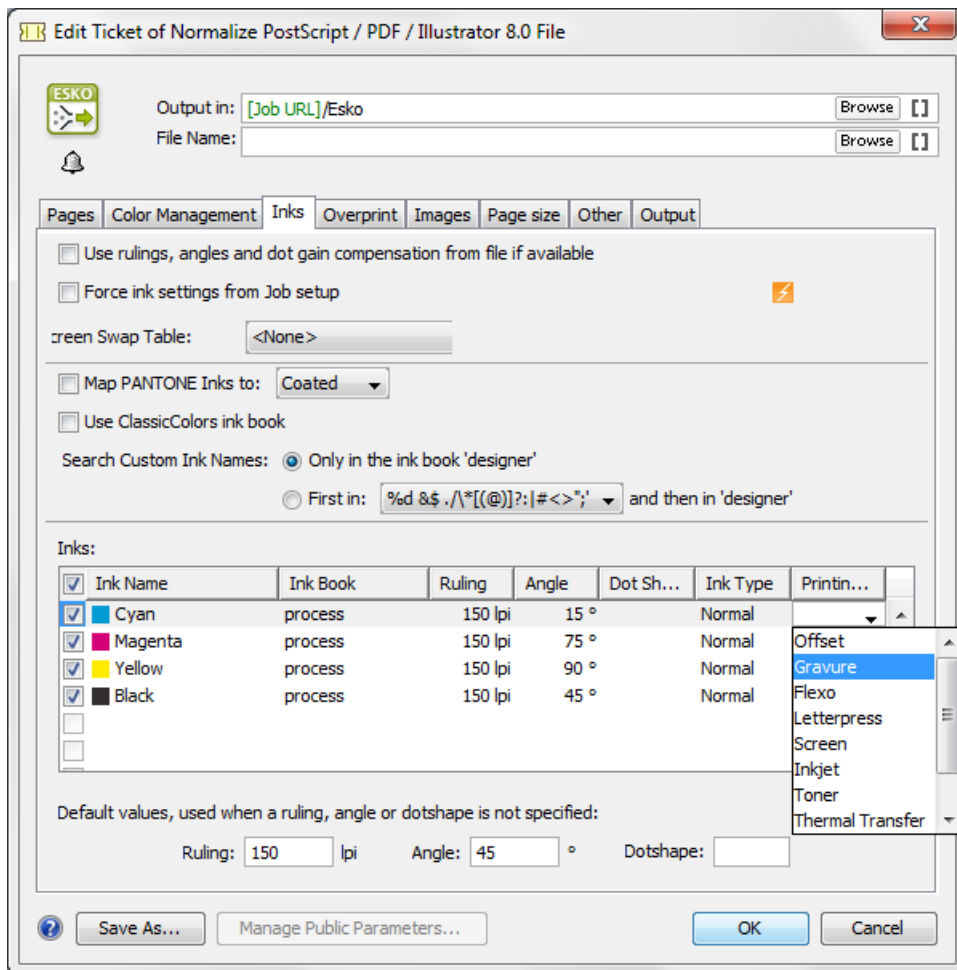


You can set an ink to a specific printing method by using the Job setup's [Inks tab](#) on page 840 or by using a background task.

These tasks can set and/or use **Printing Methods**:

- [Adjust PDF Screening](#) on page 254
- [Extract Inks](#) on page 350
- [Normalize PDF File](#) on page 377
- [Normalize PostScript / PDF / Illustrator 8.0 File](#) on page 379
- [Optimize PDF Separations](#) on page 399
- [Trap with PowerTrapper](#) on page 511
- [RIP'ing with Imaging Engine](#) on page 1123

For example:



These are typical examples of how to make use of a **Printing Method**:

- In your **Imaging Engine** (RIP) task for flexo, you select only the inks of printing method 'flexo'.
- You make a special ReportMaker PDF that focusses on the embossing 'ink'. In your workflow, you use the [Extract Inks](#) on page 350 task to select the wanted ink type before you start the ReportMaker task.

Processing Preferences

Embedded Images

Normalized PDF files have the ability to save all their images internally, but this may result in oversized files. You can define the maximum size for internal images here.

Images exceeding the defined size will be saved externally.

Automatic Normalization

Tasks that [require Normalized PDF files](#) as input will automatically normalize any regular PDF you launch them on using the **Normalize ticket** you select here.

By default, Automation Engine uses the **Normalize PostScript/PDF/Illustrator 8.0 File** ticket called **OnDemand**.

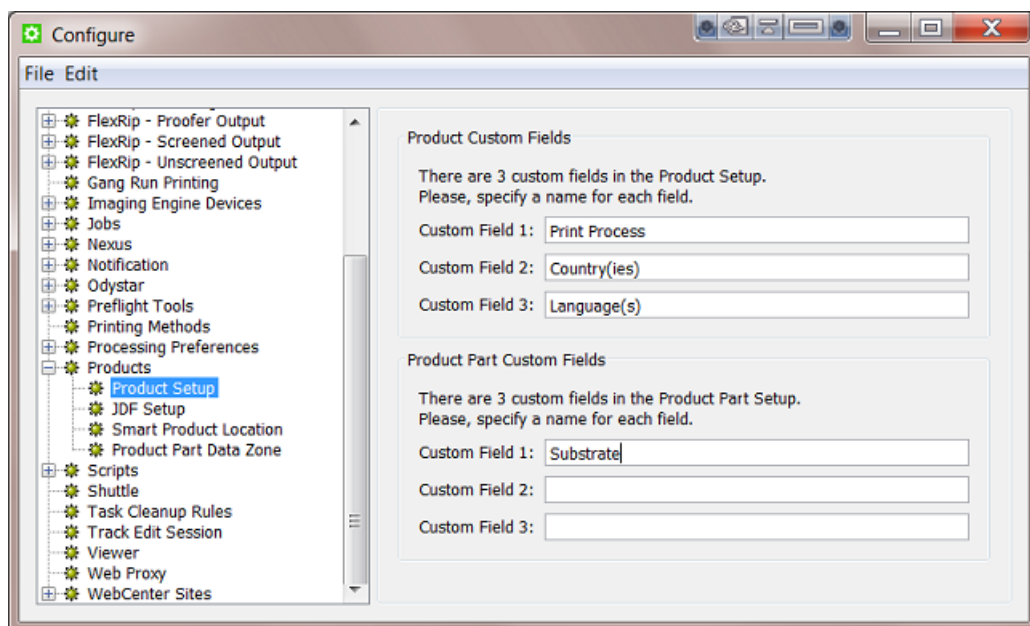
From this ticket only the settings in the **Color Management** tab are taken into account.

All other settings of the automatic normalization are forced to fixed default settings. For example, automatic normalization will always embed all images.

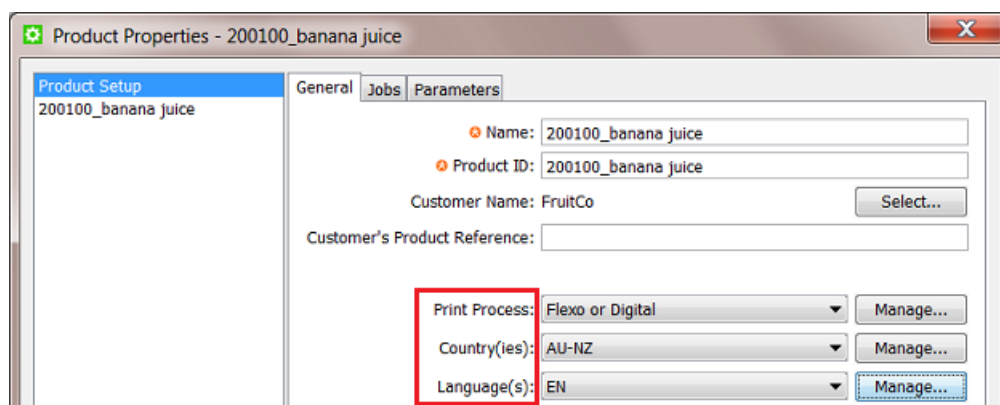
Products

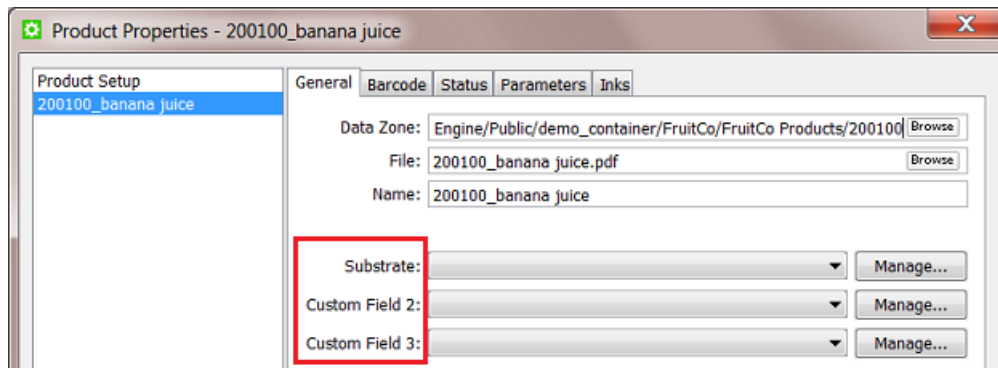
Product Setup

In the item **Product Setup**, you can name the custom fields that appear in the *Product Setup* and in the *Product Part Setup*.



Above example configuration results in this:



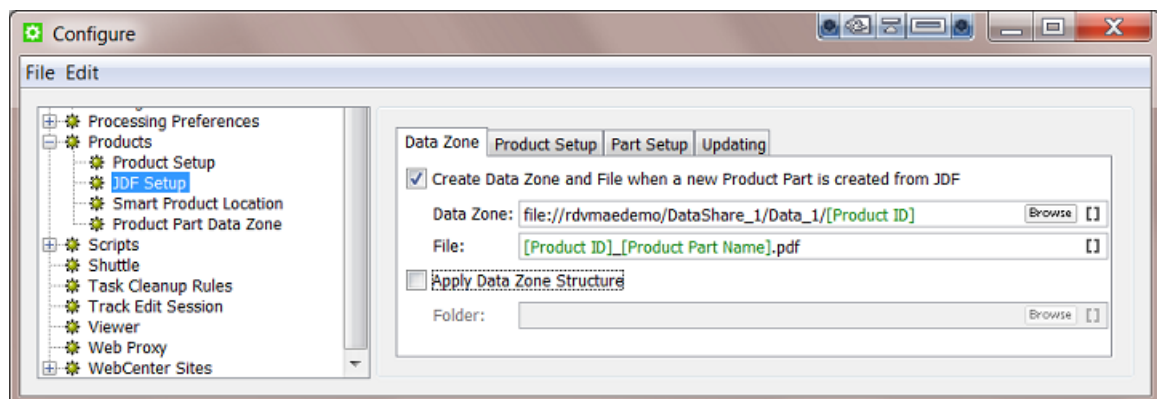


JDF Setup

Note: JDF workflows are not documented in this user manual. Learn more about the basics of JDF in the chapter [Integrating with External Systems](#) on page 1228. For more information, contact Esko Solution Services.

Data Zone

Note: The **Data Zone** concept is documented in [Products Data Structure](#) on page 881.



- **Create Data Zone and File when a new Product Part is created from JDF:**
 - **Data Zone:** When the JDF references a non-existing product part, define here where to create the data zone for that product part (the physical folder containing the part's file).
 - **File:** When the JDF references a non-existing product part, define here where to create the subfolder and/or file in the data zone.
- **Apply Data Zone Structure:** Use this option when you want to add a specific substructure to the data zone. Browse to an example data zone (folder with subfolders).

Product (Part) Setup

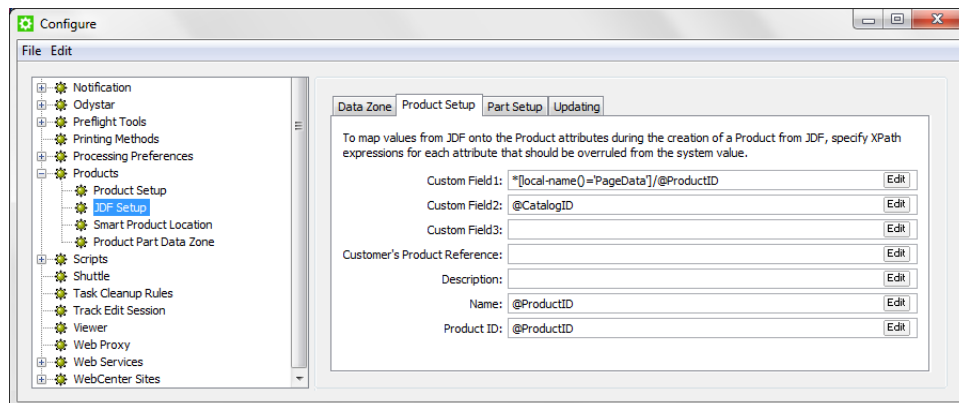
In the **Product Setup** and **Part Setup** tabs, you can add rules to map values from the JDF onto the attributes of the products that the JDF creates.

To do this, type an XPath expression into the field for each attribute that should be overruled by the system value, or click **Edit** to open the XPath builder.

Note:

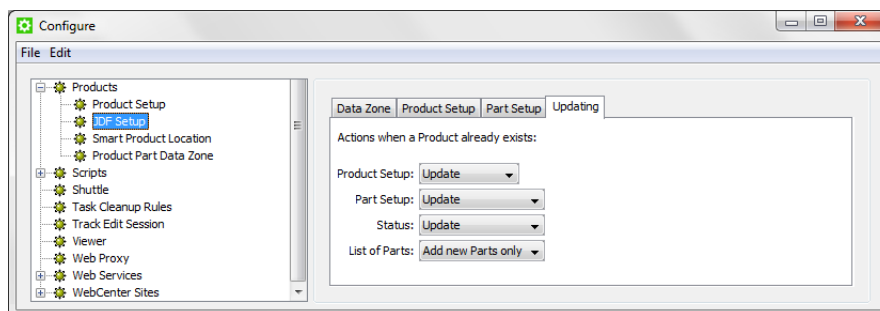
- The JDF that was loaded in the XPath builder is remembered while the Configure dialog is open.
- When editing the next XPath, the same JDF will automatically be used as the example XML in the XPath builder.

Learn more about using XPath in [XPath Builder](#) on page 661.



Updating

In the **Updating** tab, indicate what should happen when an existing product is updated over JDF.



Learn more about updating products in [The Add to Products task](#) on page 908.

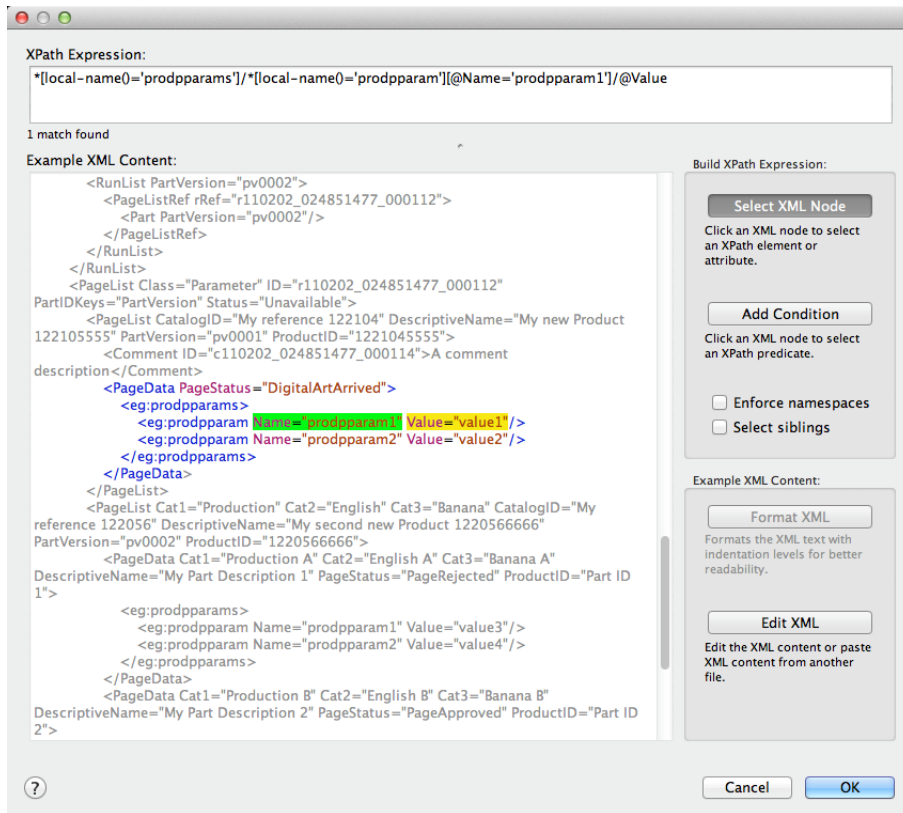
XPath Builder

When you click **Edit** in any field that allows XPath Expressions, the **XPath Builder** opens.

The XPath Builder supports alternative starting points:

- The XPath is not resolved relative to the root node of the XML document, but from another XML element.
- When a proper JDF is used as the example XML, a list of possible starting points is generated.
- The first possible starting point is automatically used to configure the XPath builder.

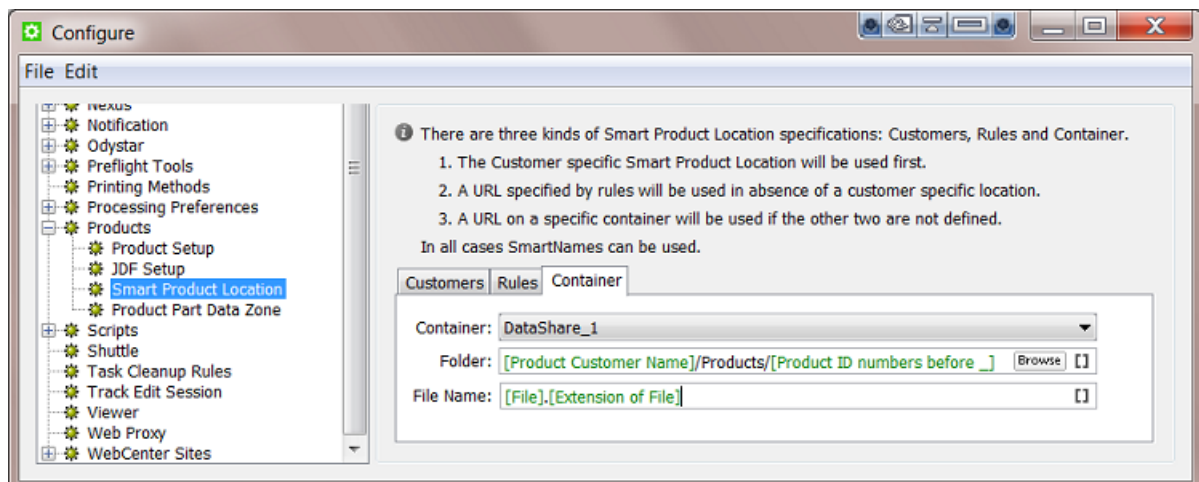
- In this example, the XPath builder is used to create an XPath for a product part attribute:



- The XPath builder automatically uses the first **PageData** node as starting point. This is visualised by showing the content outside the **PageData** node in grey.
- It is still possible to select nodes outside the **PageData** node as result node or as predicate node.
- When selecting nodes outside the **PageData** node, the XPath builder will create absolute XPaths.
- When you want, you can modify these absolute paths by making them relative to the **PageData** node using `.. /`
- While the XPath builder automatically selects the appropriate starting point, it is still possible to manually change the starting point by clicking on another node while holding down Command (Mac) or CTRL (Windows).

Smart Product Location

Smart Product Location allows to automatically decide the location for a new product based on a set of rules defined in **Configure > Products > Smart Product Location** .



Learn more about creating products in [Creating Products](#) and about their file location settings in [Product \(Part\) Properties](#) on page 893.

Important: You do not need this tool when your file is already in the right place when you make it a product part.

Three Ways to Decide the Location

You can use three kinds of specifications to set up the Smart Product Location: based on **Customers**, on custom **Rules** and on a chosen **Container**.

These three ways are checked in a specific order: if the **Customers** check does not return a valid path, then **Rules** will be checked. If this still does not return a valid path, then the **Container** rule will be used.

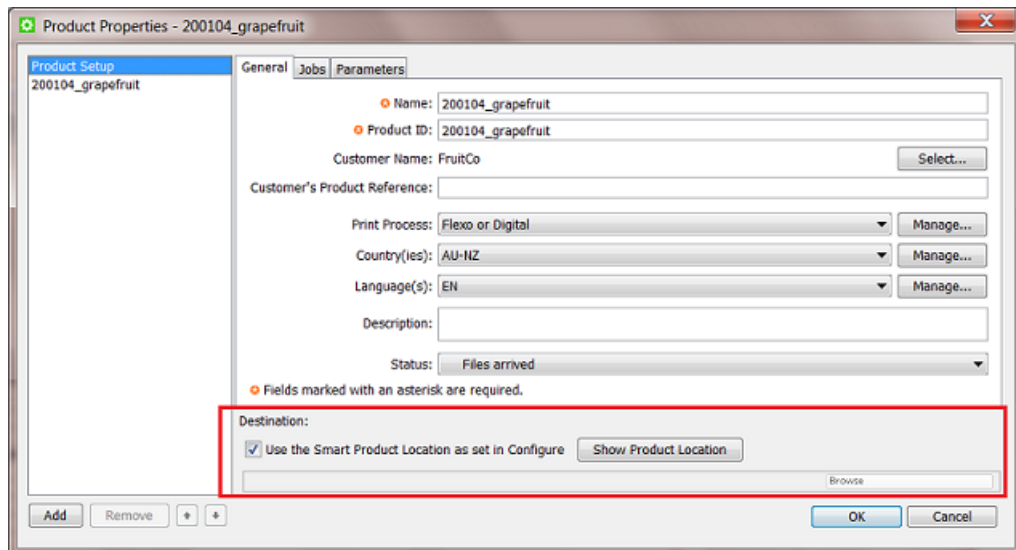
- **Customers:** This rule allows to specify per customer a SmartName based path where all its products will be created.
- **Rules:** This tab allows to set extensive conditional rules. For example you could create all the products in a structure based on their `[custom field 1]`. You can create several rules and they will be checked from top to bottom.
- **Container:** This tab allows to set a smart path in a fixed Container where all products will be created (from now on). For example: All products are created in Container `Data1` and when the disk of that Container is full, you here configure that from now all new products will be created in Container `Data2`.

Using the Smart Product Location

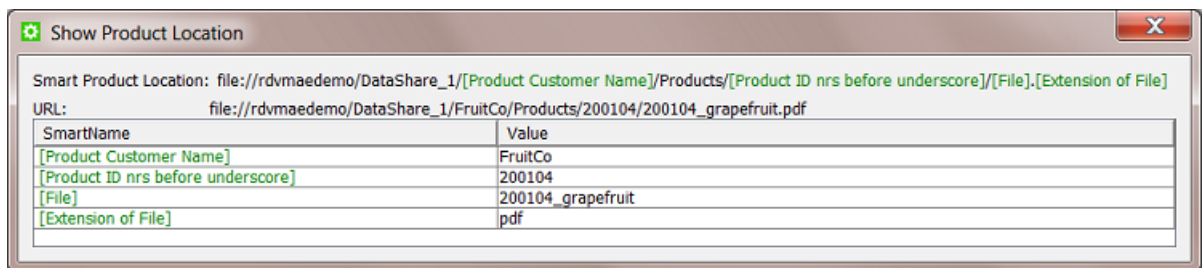
Smart Product Location is used in these cases:

- When choosing [Uploading As a New Product](#) on page 898. This feature is used for example when your product data is still on a CD or on your desktop.
- When creating products via JDF or when placing products via JDF (a product part URL is already decided but 'empty' and the JDF then delivers the file to the URL that expects the file).

When [Uploading a File as a Product](#), the panel shows the extra Smart Product Location tool at the bottom:



Click **Show Product Location** to see a preview of where the uploaded file will be placed. An example:

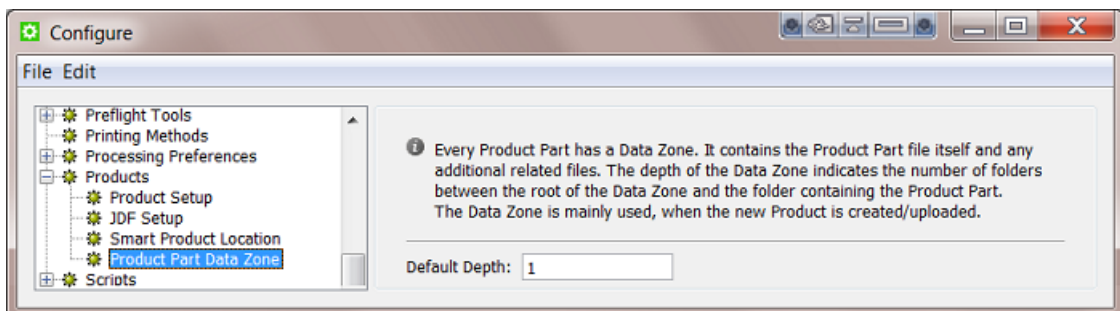


Product Part Data Zone

Note: Learn about the concept of **Data Zone** and see some examples in [Products Data Structure](#) on page 881.

Every product part has a data zone. It contains the product part file itself and any additional related files.

When you look at a product's data zone in the Pilot, the depth of the data zone indicates the number of folders that you will see between the root of the data zone and the folder containing the actual product part file.



The **Default Depth** value that you enter here is used when you create or change the product part properties.

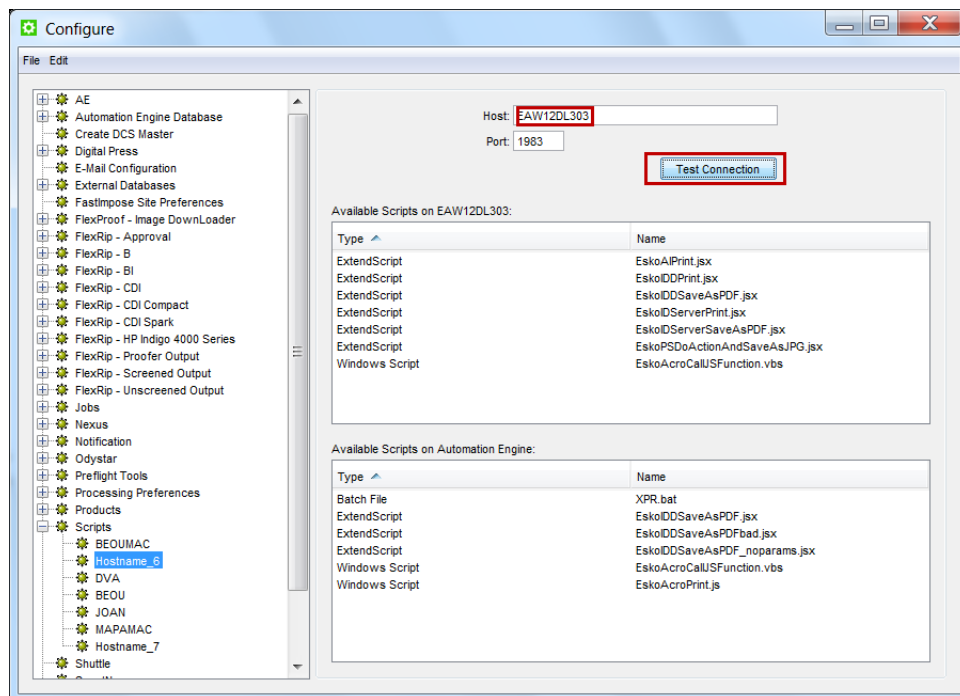
Scripts

You can configure your Script Runner to your Automation Engine server. A Script Runner is a stand alone Automation Engine Server component on which you can create scripts. These scripts can be used in the [Run Script task](#).

This computer must have Script Runner installed and running. For more information on installing a Script Runner, refer to [Getting Started with Scripting](#) on page 123

1. Choose **Scripts** from the categories on the left. Choose **File > New** .

1.



2. Give a suitable name to the Script Runner by choosing **File > Rename**

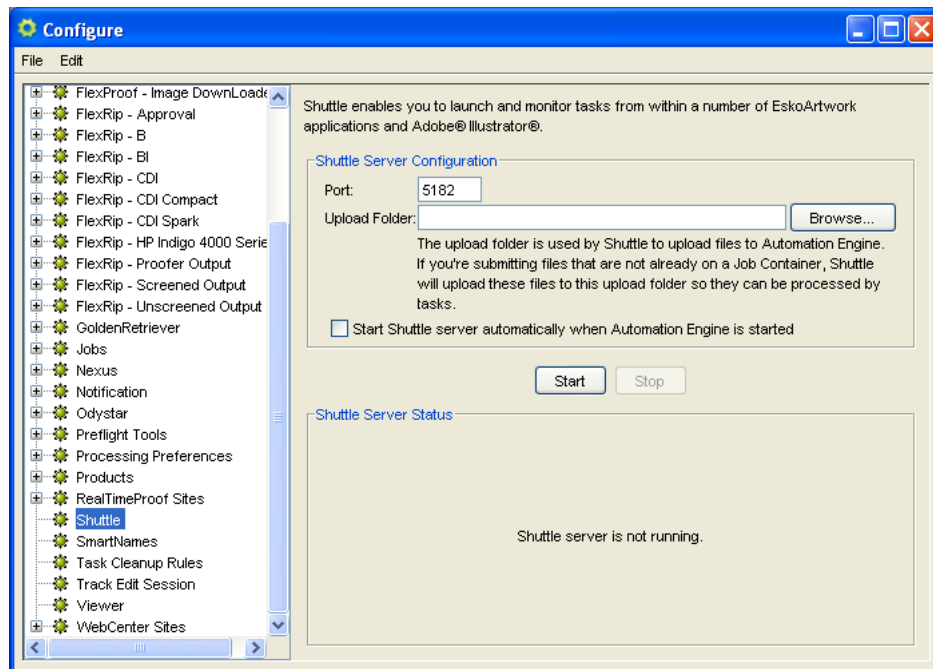
3. Enter the computer name or IP address of the local computer in **Host** field.

4. Enter the **Port** used to connect this computer to your Automation Engine server.

By default, this is 1983.

5. Click the **Test Connection** button. You will see all the scripts available on the Script Runner or a message 'No scripts available'.

Shuttle



Enter the **Port** to use for communication between your Automation Engine server and the Shuttle clients.

Note: By default this is port 5182, but if this port is already used by other processes, you should enter the next available port.

Choose an **Upload Folder** using the **Browse...** button.

This is where Shuttle clients will upload files that are not located in a Container (except the Shuttle DeskPack plug-in, that will always upload files to the **DeskPackContainer**).

Note: The **Upload Folder** must be located on a Container.

Select **Start Shuttle server automatically when Automation Engine is started** if you plan to make frequent use of the Shuttle technology.

Click the **Start** button to start Shuttle server. This is not required to use the Shuttle application. However, starting the server is required to use Shuttle via applications like ArtPro, PackEdge and Illustrator. Shuttle server makes Automation Engine's workflows, tasks and files accessible to the Shuttle clients.

Save the settings by clicking **File > Save** .

Task Cleanup Rules

You can define **Task Cleanup Rules** to automate the cleaning up of the tasks pane.

These rules can be selected and applied in the Monitor.

1. Click **New...**

This opens the **New Rule** dialog box.

2. Enter a **Rule Name** of your choice.

3. Choose if this rule will **Move** entries to **History**, or **Delete** them.

4. Choose what to apply the rule on: **Single Tasks** or **Workflows**.
5. Use the **Operator** tab to choose whose entries you want the rule to clean up.
Select **Filter on Operator** and select the desired operators (you can also **Select All**).
6. Use the **Time** tab to select the entries to clean up based on their date.
Select **Filter on Date** and choose to move / delete the tasks / workflows finished since a certain number of **minute(s)**, **hour(s)**, **day(s)** or **month(s)**.
7. If you are cleaning up **Single Tasks**, you can use the **Task Type** tab to only clean up certain task types.
Select **Filter on Task Type** and select the desired task types.
8. Use the **State** tab to select the entries to clean up based on their state.
Select **Filter on State** and select the desired states (**Successfully done**, **Done with warning**, **Cancelled**, **Failed**).
9. When you are done defining the rule, click **OK**.

New Rule

Rule Name: Move my old tasks to history

Action: Move to History
 Delete

Applies To: Single Tasks
 Chains

Operator Time Task Type State

Filter on Date

Move only tasks finished:

since [] minutes(s)
 since [] hour(s)
 since [] day(s)
 since 1 month(s)

OK Cancel

**Attention:**

The order of the Task Cleanup rules is very important. You have to define them in such a way that a rule does not block the following rules.

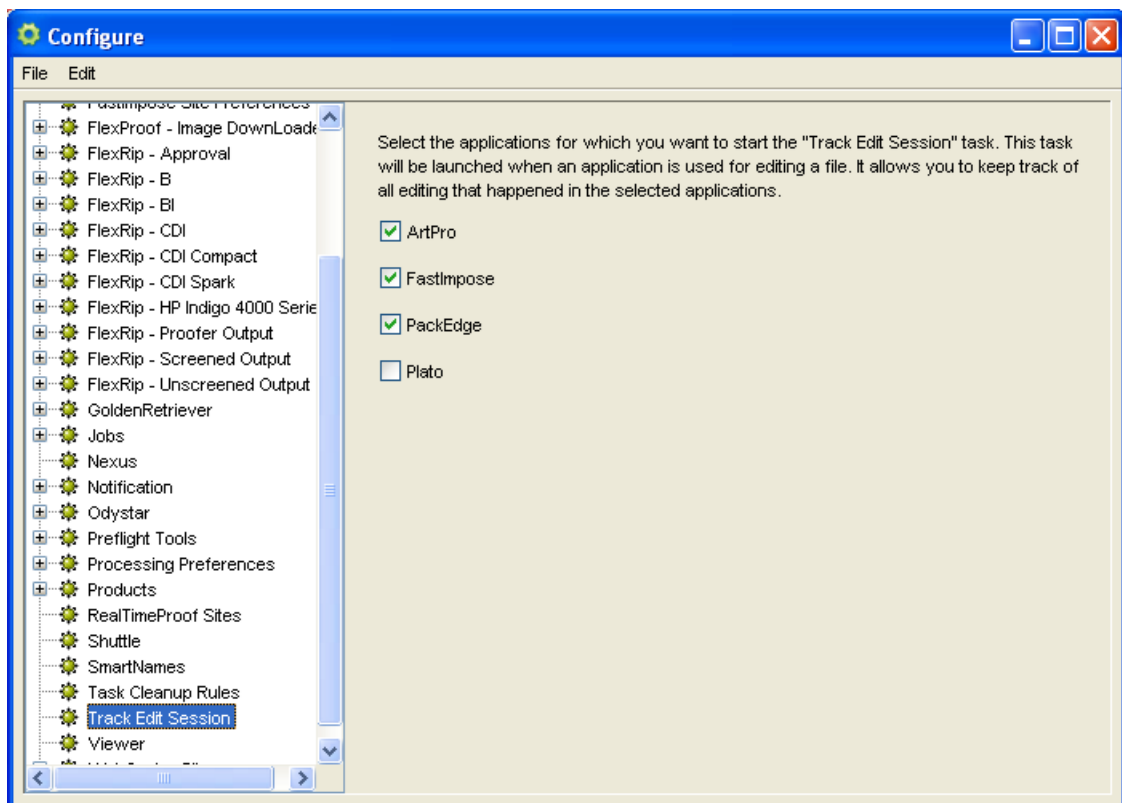
For example, if your first rule moves all successful tasks after two days and the second rule deletes all Proof tasks after 10 hours, the result will be that the first rule dominates and successful Proof tasks will only be moved after 2 days (first rule) instead of being deleted after 10 hours (second rule).

You can overcome this behavior by changing the order of the rules (use **Move Up / Move Down**).

Track Edit Session

Automation Engine can be configured so that all sessions in the Esko editors are logged.

You can view this information in the Task View of the Pilot or as an XML file.



Select the application(s) for which you want to start the “Track Edit Session” task.

When you open a file that resides in an active Job (folder) in an Esko editor, the Track Edit Session task will automatically start logging the interactive editing actions. This tracking will stop when the file is closed.

Note: Keep in mind that the editor needs to be restarted to activate the settings.

Note: For ArtPro, the Track Edit Session only records session times, not editing details.

Viewer

About the Viewer configuration settings

When a folder is marked as a **Speed Up for Viewing** folder, Automation Engine creates View Data for any new files it finds there every time it accesses the folder (for example when a task writes or reads files in this folder).

The 'Speed up for Viewing' settings in Configure allow you to select:

- which **Prepare for Automation Engine Viewer** ticket should be used and which types of files should be taken into account;
- a file streaming session limit.

Note: Files relating to Digital Film will be handled separation by separation.

How to configure the Viewer

Proceed as follows:

1. In the **Preferences for automatic normalization** area, select the **Normalize ticket** of your choice.

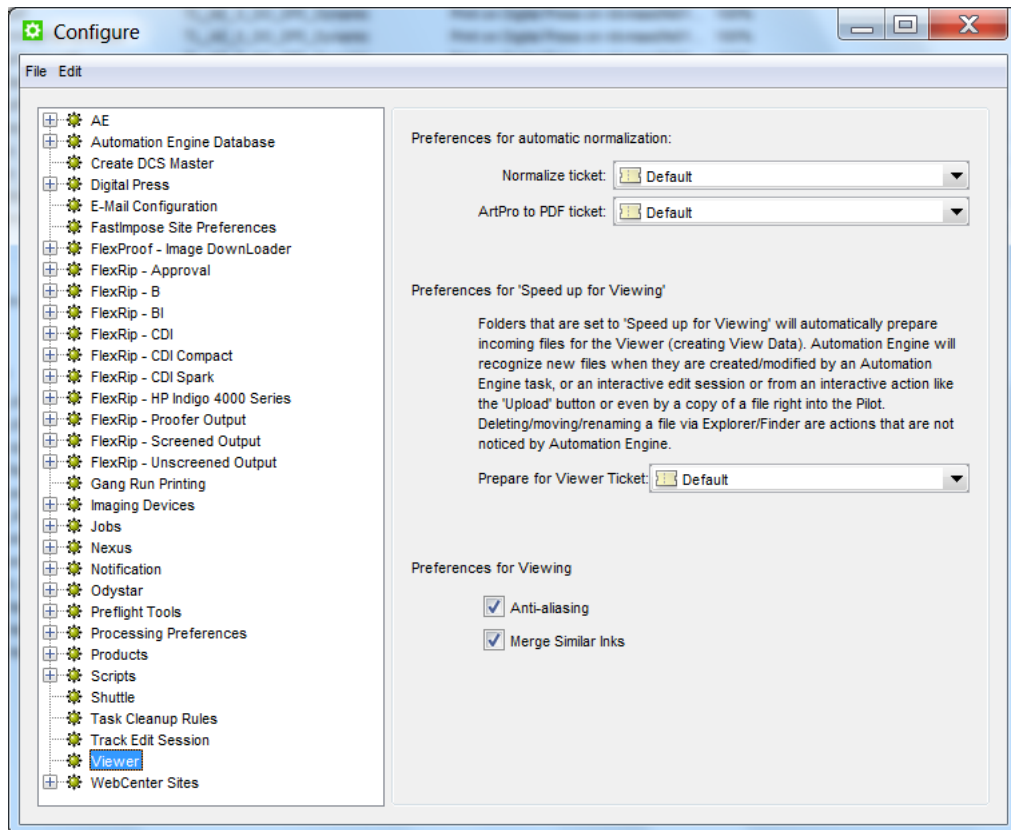
Files that have not yet been normalized require a normalization step before Automation Engine is able to create and show View Data.

2. Select the **ArtPro to PDF ticket** of your choice.

ArtPro files need to be converted to Normalized PDF before you can view them.

3. Select the **Prepare for Viewer** ticket you want to use for view data generation.
4. Configure how many Viewers can be used simultaneously to view files with no view data prepared using the **File Streaming Session Limit** option in the Server Admin Wizard.

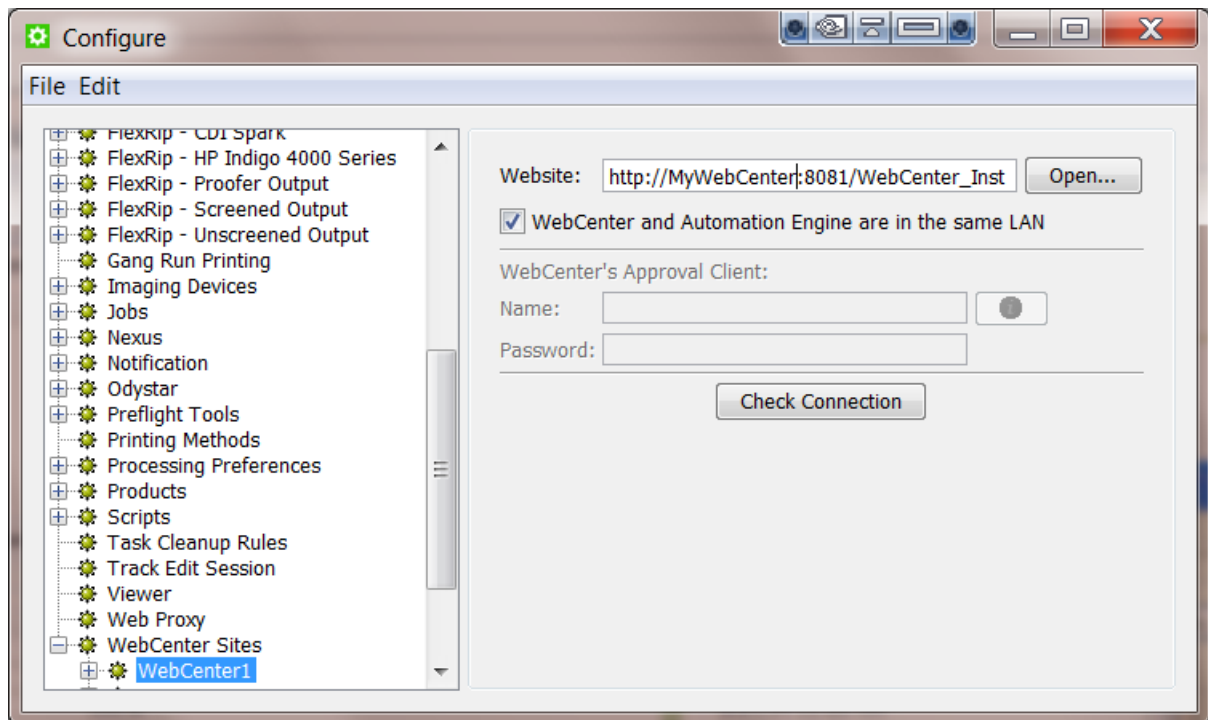
See more in [Processing Capabilities](#) on page 218



You can also set your preferences for viewing by enabling the options :**Anti-aliasing** and /Or **Merge Similiar Inks**.

WebCenter Sites

To configure links from Automation Engine to WebCenter sites, go to the item **WebCenter sites** in the **Configure** tool (**Pilot > Tools > Configure**).



Website:

Specify the web address of the WebCenter site. After you entered the WebCenter address, click **Open...** to check if the WebCenter site is up and running. If you are redirected to the login page, it means that the address is correct and that the site is up and running.

Tip: Be aware that this address is also the address that will be included in the notification e-mails when adding links to documents (approval cycle start e-mail). You should therefore make sure that the address can be accessed from the outside. Make sure to not use an internal address, but one that you want your users to use when *they* use WebCenter.

WebCenter and Automation Engine are in the same LAN

Check this toggle if this is true. This will most likely be the case. Un-check this toggle if WebCenter and Automation Engine are remote.

Note: How can you verify that WebCenter and Automation Engine are in the same LAN? WebCenter and Automation Engine are in the same LAN when you can connect from WebCenter's application server, this is the system where JBOSS is installed, to an Automation Engine Job Folder with the credentials of the user 'BGSystem'.

WebCenter's Approval Client:

If WebCenter and Automation Engine are not in the same LAN, you first need to create an Automation Engine **Approval Client** in WebCenter. This is necessary to give Automation Engine permissions to listen to the approval messages that are stored in WebCenter.

Tip: If the **Approval Client** does not exist yet, follow these steps:

1. In WebCenter, log in as `Administrator`.
2. Go to the **Admin** section. In **Configurations**, select **Workflow Approval Clients**.
3. Click the link "**New Workflow Approval Client**".
4. Enter a **User name** and a **Password** for the Automation Engine client. Mind that this is also the user name and password that have to be entered when configuring the WebCenter site in Automation Engine.

Fill out the client **Name** and **Password** and click the '**Check Connection**' button. This will check if that user can log on successfully.

The **Info** button displays the following information: *"In case Automation Engine and WebCenter are not in the same local area network (LAN), the retrieval of the approval results is secured by a name and password. The name and password specified in Automation Engine must match the name and password specified in WebCenter by the WebCenter administrator. When clicking the 'Check' button, the WebCenter version will be checked and the name and password will be validated by WebCenter."*

Check Connection

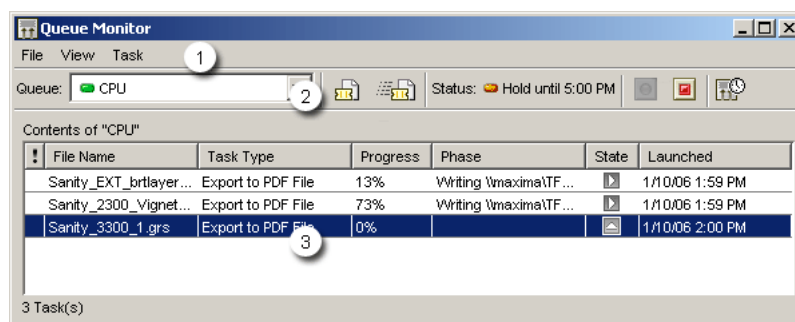
When you click this button, you should, after a few seconds, get a window saying "**Check OK**". The software version of that WebCenter will also be mentioned.

Note: The button will check the validity of the user name or password to connect to that WebCenter but **not** of your WebCenter's **Approval Client**.

Note: When you connect to a secured WebCenter site (for example one that uses HTTPS://), then you have to export its security certificate and import it on the Automation Engine server. If this is not done, publishing from Automation Engine to WebCenter will result in an error. Read more on how to do this in the chapter **Install SSL Certificates on the Workflow Production Server** of the WebCenter documentation.

Tip: After configuring a new WebCenter site, it is recommended to close your Pilot and start it again in order to see the updated list of WebCenter sites in your tickets.

2.9.8 Queue Monitor





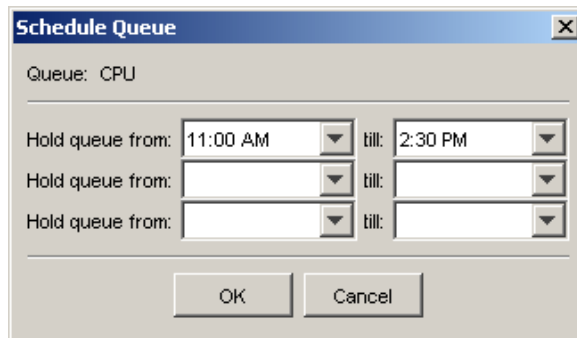
1. Menus
2. Toolbar
3. Contents: lists all tasks in a selected queue. Right-clicking a task activates a context menu which offers the same functionality as the Task menu.

Queue Monitor menus

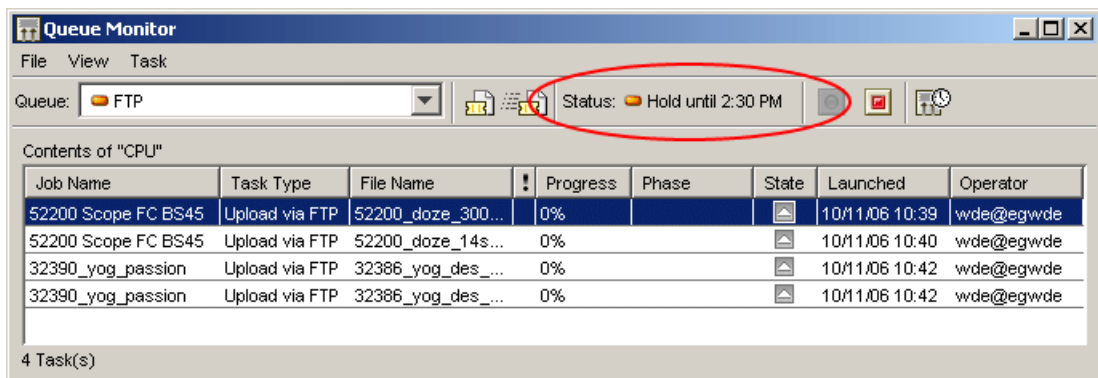
File Menu

File Menu options:

- New Window: opens a new Queue Monitor window.
- Start / Stop queue   : These buttons allow you to start or stop the selected queue.
You can also do this by selecting Start / Stop Queue from the 'Advanced Menu' or by right-clicking a device in the Device List. The devices' status is visible in the Queue State column.
- Schedule queue... allows you to schedule the activity of specific action queues. You can define 3 time intervals during which queues are kept on hold.
Set the FTP queue on hold till 2.30 PM. At that time your waiting FTP tasks will start processing



The scheduled status will be visualized in the [Queue Monitor toolbar](#).



- Close window: closes the Queue Monitor.

View

The View Menu offers 'Columns' from which you can select the columns you want to see or hide. For more info, please refer to the part on '[Columns](#)'.






Task

The Task Menu offers the same functionality as the [Pilot's Task Menu](#). You can also right-click a task in the queue. The context menu will offer you the following options:

- Open Task
- Log Info ...
- Hold
- Hold Until...
- Release
- Cancel
- Annotation...
- Priority...
- Relaunch

Toolbar



1. The Queue drop down list. Select the queue you want to monitor from the list.
2.  Opens the selected Tasks.
3.  Relaunches the selected task(s).
4. Status indicates the status of the selected queue.
5. The **Start / Stop queues** buttons  . These buttons allow you to start or stop the selected queue.
6. The **Schedule Queue** button  opens the Schedule Queue dialog box.

Contents of selected queue

Lists all tasks in the selected queue.

Right-clicking a task opens a pop up menu with the same functionality as the Task Menus.

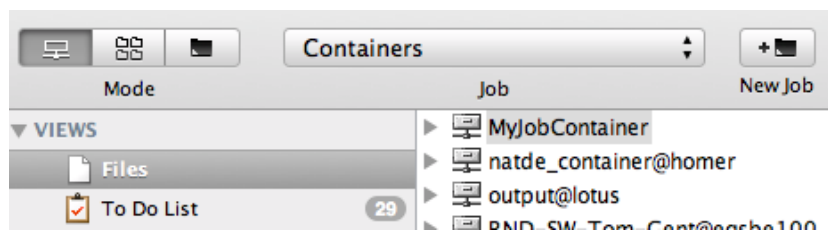
You can modify the look of the list by drag & dropping the heads of the columns, and by selecting 'Columns...' from the [View Menu](#).

2.9.9 Containers

A **Container** is a network data share that contains the data that Automation Engine can work with. You can make (part of) an existing network share a Container or you can have Automation Engine create such network shares. These Containers can be located either on your Automation Engine server or another computer in the same LAN.

By default, you have an empty **ExampleJobContainer**, on the disk drive that Automation Engine was installed on.

You can have many Containers. You can see and browse through them in the Pilots' **Containers** mode.

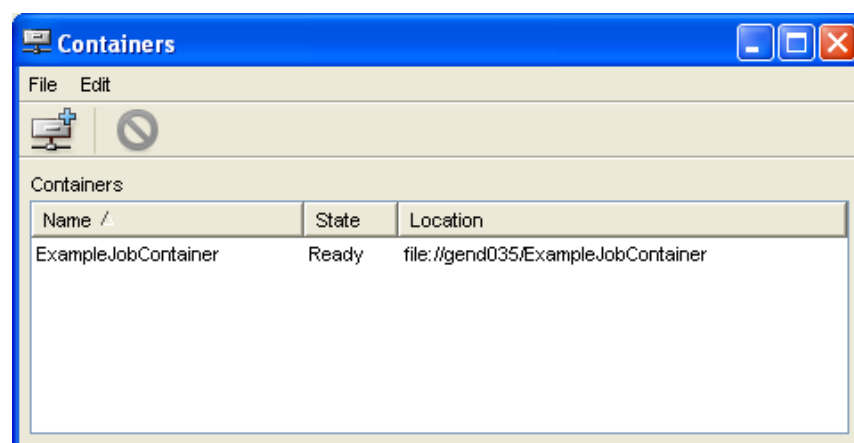



To create a Container, see [Creating a Container](#) on page 691.

Creating a Container

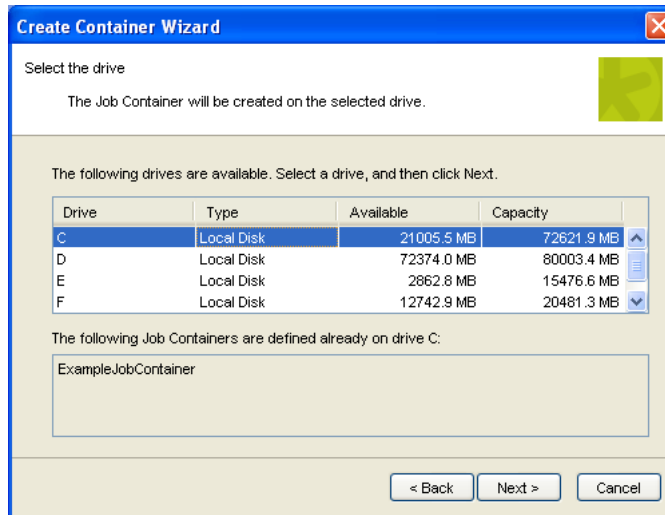
To create a container, do the following:

1. Go to **Tools > Containers** to open the **Containers** window.



2. In that window, go to **File > New** or click . This opens the **Create Container Wizard**.

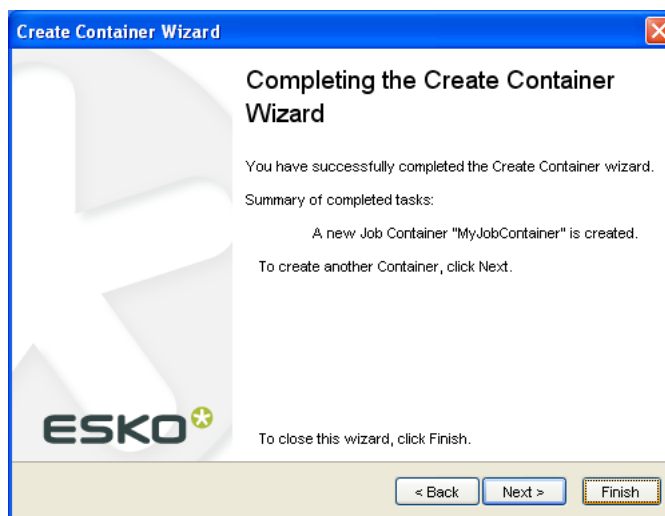
3. Click **Next** in the wizard's first screen.
4. Select either:
 - **Local Folder on the Server** to create a container on your Automation Engine server,
 - **Existing Shared Folder**
5. Click **Next**.
6. If you selected **Local Folder on the Server**:
 - a) Select in which drive you want your container to be.



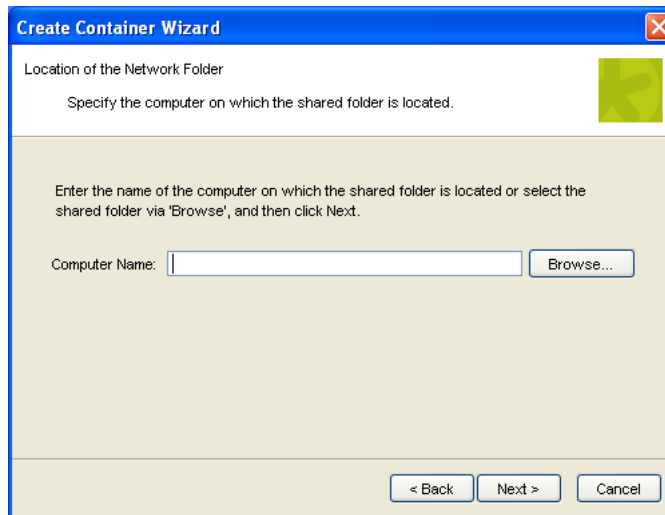
Selecting a drive shows the containers already existing on that drive.

- b) Click **Next**.
- c) Enter the name to give your new container.
- d) Enable **Make this Job Container available for Macintosh** if you want to be able to access it from Pilots running on Macintosh machines.
- e) Click **Next**.

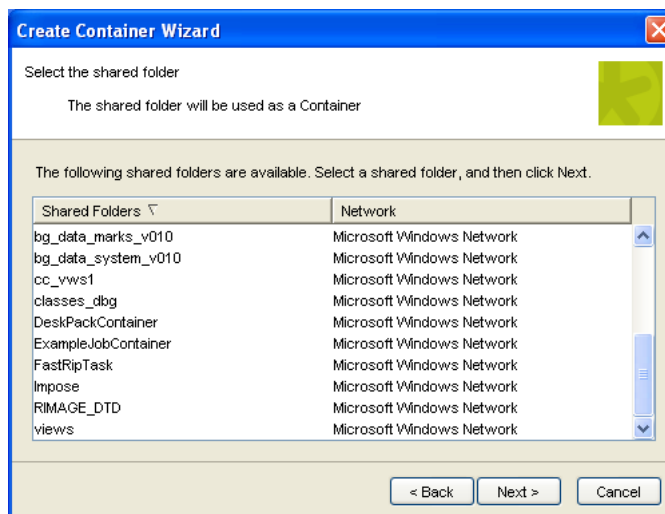
Your container is created.



- f) Click **Next** to create another container, or **Finish** to leave the wizard.
7. If you selected **Existing Shared Folder**:
- a) Enter the **Computer Name** of the machine where the share folder is located, or use the **Browse...** button to select that machine in your network.



- b) Click **Next**.
- c) The wizard shows a list of all available shares on that machine. Select the share you want to use as a container.




Attention: The existing shared folders will not be displayed if the system does not recognize the BGSYSTEM user (ask its default password to Esko).

- d) Click **Next**.
- Your container is created.
- e) Click **Next** to create another container, or **Finish** to leave the wizard.

Deleting a Container

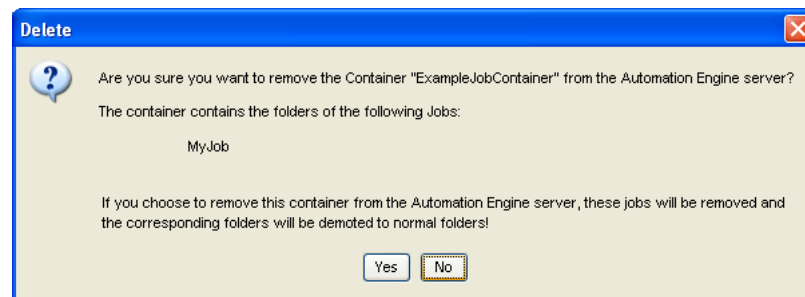
You can delete a container from Automation Engine without losing the container's contents. Deleting a container simply makes its contents invisible to your Automation Engine server.

To delete a container:

1. Select the container to delete in the **Containers** window.
2. Click  or use **File > Remove**.

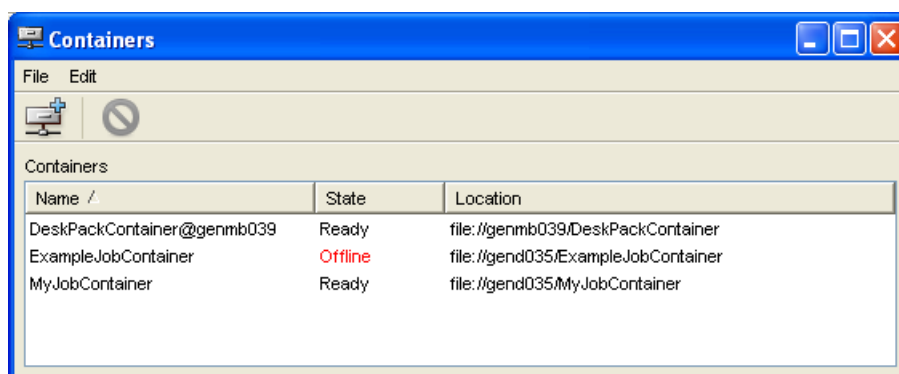
Note:

If there are jobs in the container you want to delete, they will be listed in the confirmation dialog. If you delete the container anyway, the job folders will become normal folders.

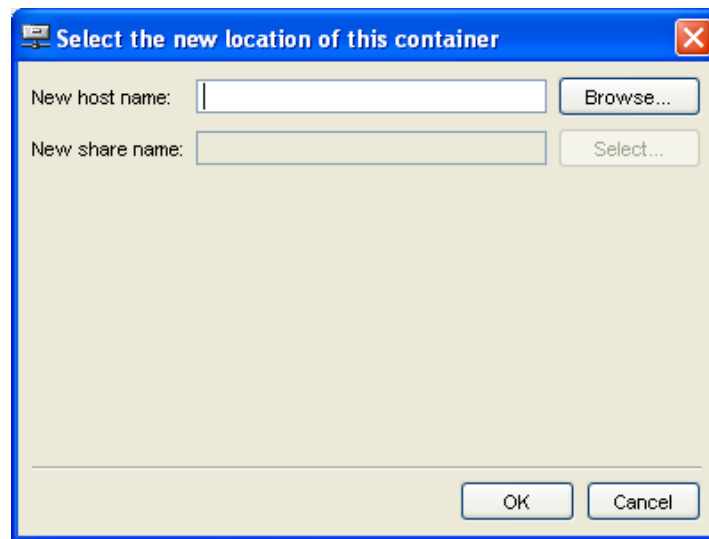


Relocating a Container

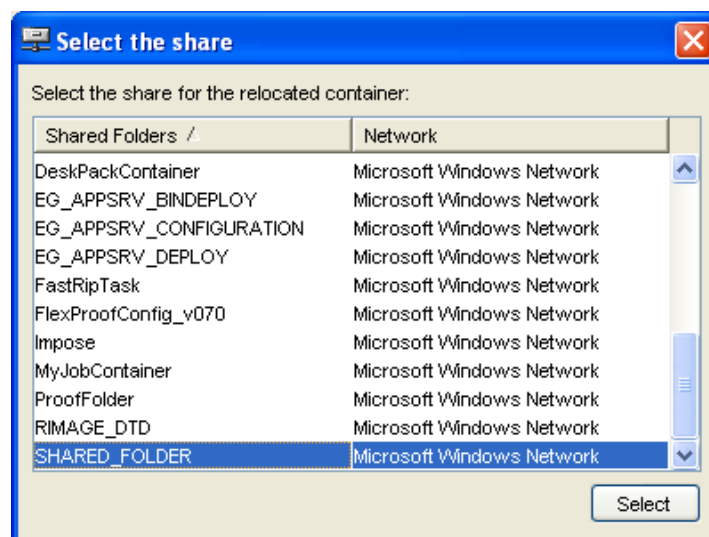
If one of your containers goes offline (for example the disk hosting it crashes) but you have a backup of it, you can restore this container on another disk.



1. Select the offline container and go to **Edit > Relocate**.
2. Enter the name of the computer you want to restore your container on, or browse to it.



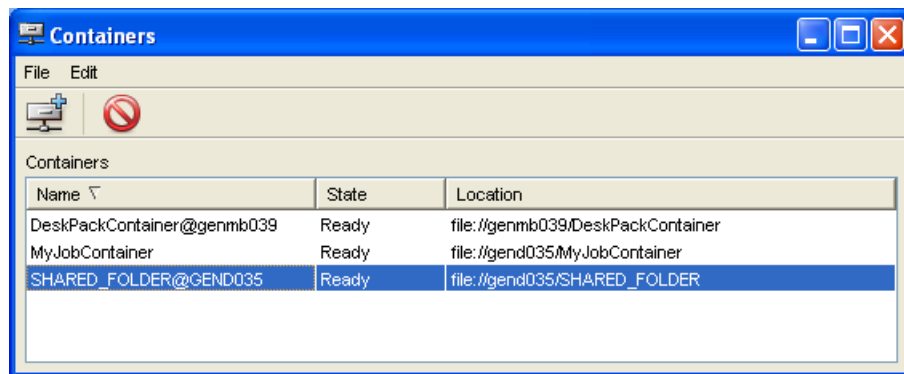
3. Select the share you want to restore your container on.



The **Select the share** dialog shows all shares available on the computer you have chosen.

4. Click **OK** in the **Select the new location of this container** dialog.

Your container is now restored on the share you selected (and has the name of the share).



Refreshing the Containers' List

The containers' availability is automatically checked every 5 minutes. If you need to check it manually/refresh the connection, you can:

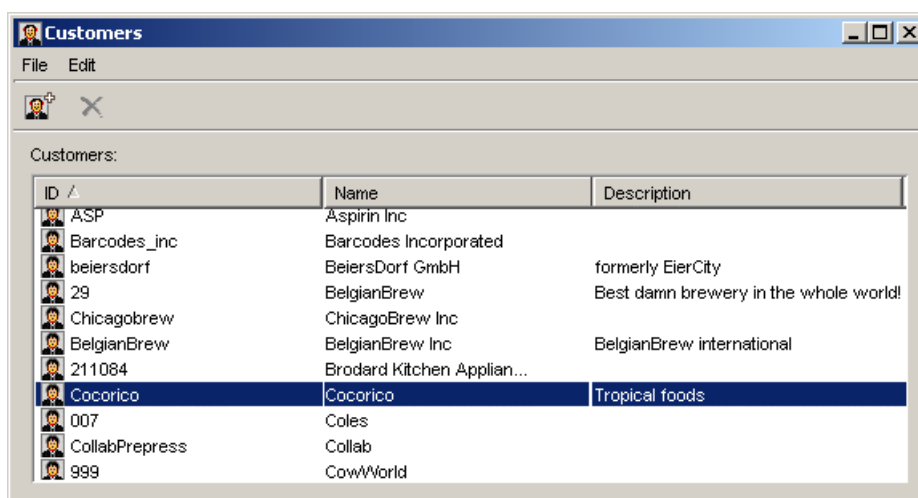
- go to **Edit > Rescan**,
- use **CTRL + R**,
- press **F5** in the Pilot's **Files** view when using the **Container** mode.

2.9.10 Access Points

Access Points are documented in the chapter **Integrating with External Systems**. [Here](#) is a direct link.

2.9.11 Customers


This tool allows you to manage the customers list on your system. A customer has an ID, a name and a description.




Ideally, you do not type in these customer data but get it automatically via the integration with your business system. Read more about such integrations in [Integrating with External Systems](#) on page 1228.

Customers menus

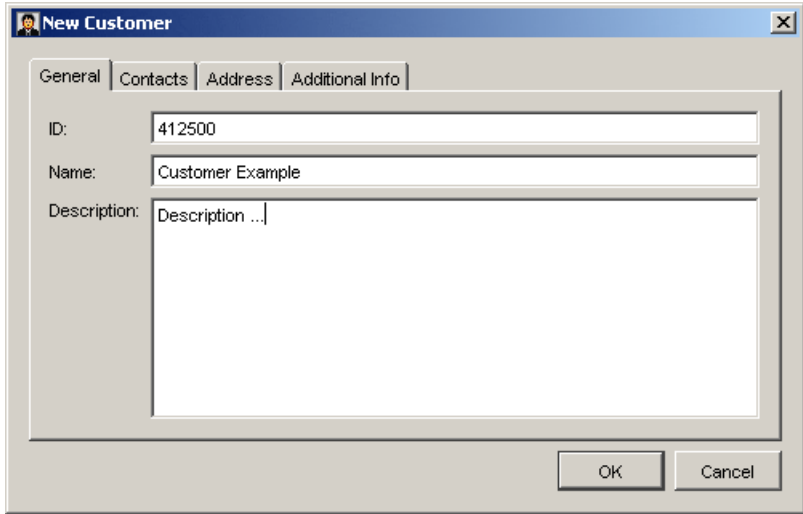
File

- New Customer [CTRL + N]: opens the '[New Customer](#)' dialog. You can also click the new customer icon in the toolbar 
- Modify Customer...: allows you to edit the selected customer data.
- Close Window [CTRL + W]: closes the Customer tool.

Edit

- Delete [Del]: removes the customer from the list. You can also use the delete button 

New Customer



The **General** tab contains three fields:

- ID
- Name
- Description

In the **Contacts** and **Address** tab, 3 contact names with e-mails and detailed addresses can be specified.

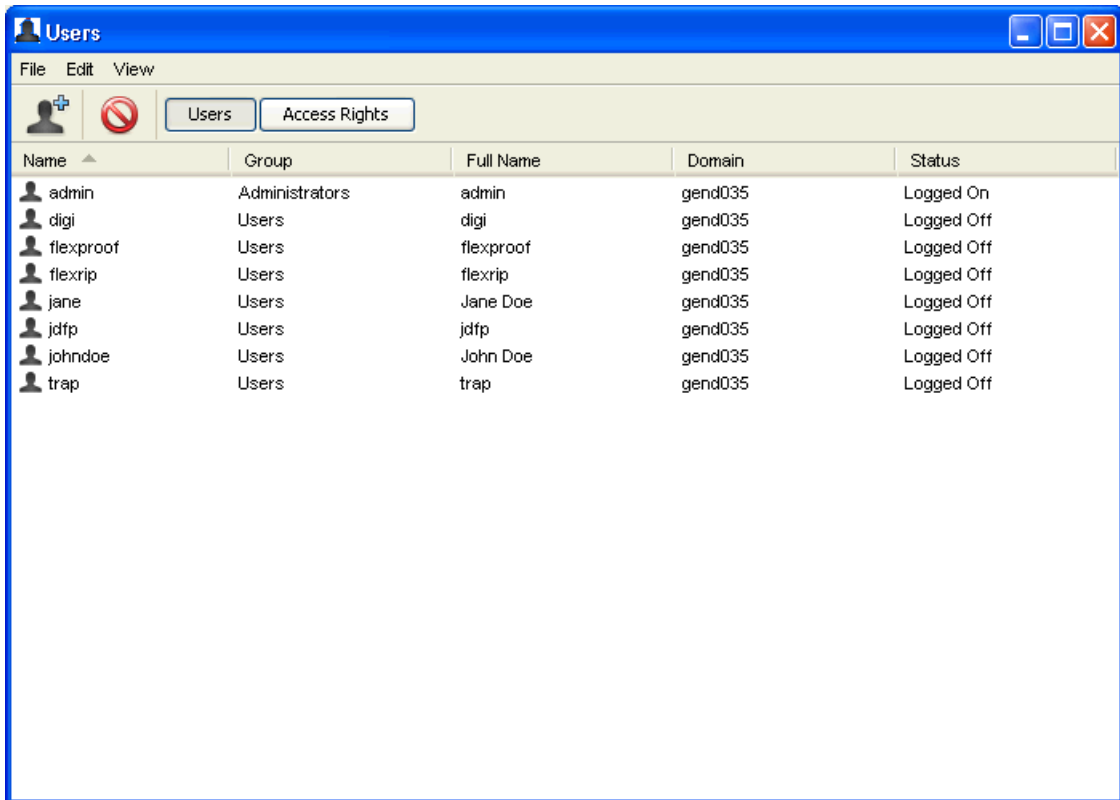
The **Additional Info** tab provides 5 extra custom fields for even more information.

This data will be used in the customer tab of the [Job Setup](#).


To edit the data select **Modify Customer** from the File Menu or double-click a customer in the list.

2.9.12 Users

To manage users, go to **Tools > Users...** in the Pilot to open the **Users** window (or click **Users** under **Tools** in the Pilot left pane).



From this window, you can:

- create a user (see [Creating Users](#) on page 32),
- delete a user (select the user to delete and click the delete icon ) ,
- view which group a user belongs to,
- view which users are logged on,
- change users' properties
- go to **Access Rights** , [Red Alerts](#), and [Connections](#)).

Menus in the Users window

File Menu

The File Menu consists of the following items:

- **New User...** allows you to create a new user. For more information, see [Creating Users](#) on page 32.
- **New Action...** opens the [New Action](#) window, allows you to add a Milestone action to an user.
- **Delete** deletes the selected user(s).
- **Properties...** displays the selected user's properties. For more information, see [Properties](#).
- **Close Window** closes the **Users** window.

When the tab **Groups** is selected, the **File** menu contains:

- **New Group...** allows you to create a new user group. For more information, see [Creating Groups](#) on page 706.
- **New Action..., Delete, Properties** and **Close Window**.

Edit Menu

You can use the **Edit** menu to **Copy** and **Paste** a user's **Favorites, Views** and/or **Red Alerts** to another user.

Note:

- Regular users can only copy **Favorites** from another user and paste them to their own user profile.
- Administrators can copy and paste **Favorites, Views** and/or **Red Alerts** as they wish. They can also copy **Favorites** to multiple users.


View Menu

The Users window has two ways of presenting things:

1. **Users:** shows all defined users and lists their [access rights](#),
2. **Access Rights:** displays all [access rights](#) and lists all known users.

You can also click the **Users** and **Access Rights** buttons.

Creating Users

1. In the **Pilot**, go to **Tools > Users** .
2. In the **Users** window, do one of the following:
 - go to **File > New User** ,
 - use the **CTRL + N** shortcut,
 - click the  icon.

The **New User** dialog opens.

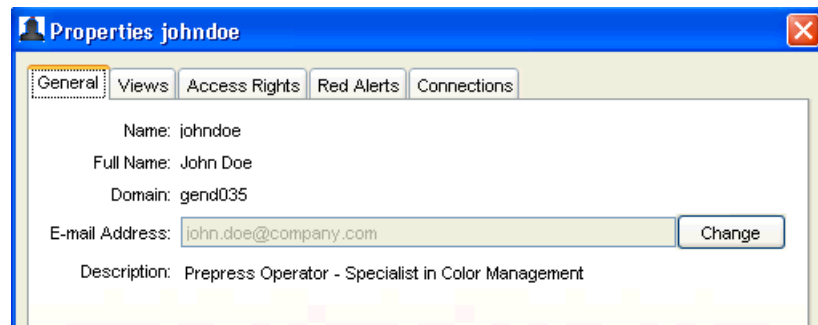
3. Enter the user's **Name, Full Name, Password, Description** and valid **E-mail address**.
4. If you want the user to be able to administer Automation Engine, check the toggle. This will give all user access rights to that user.
5. Click **OK** to add your new user to the **Users** list.

**Attention:**

When creating a user, Automation Engine requests permission from the Windows OS. Therefore, some changes (for example, changing the password) require the use of Windows User Administration tools. These changes must also meet the existing domain policies (for example, the length and complexity of the password).

Properties

Tip: Double click a user to open the **Properties** window.



- The **General** tab lists information entered when creating the user.

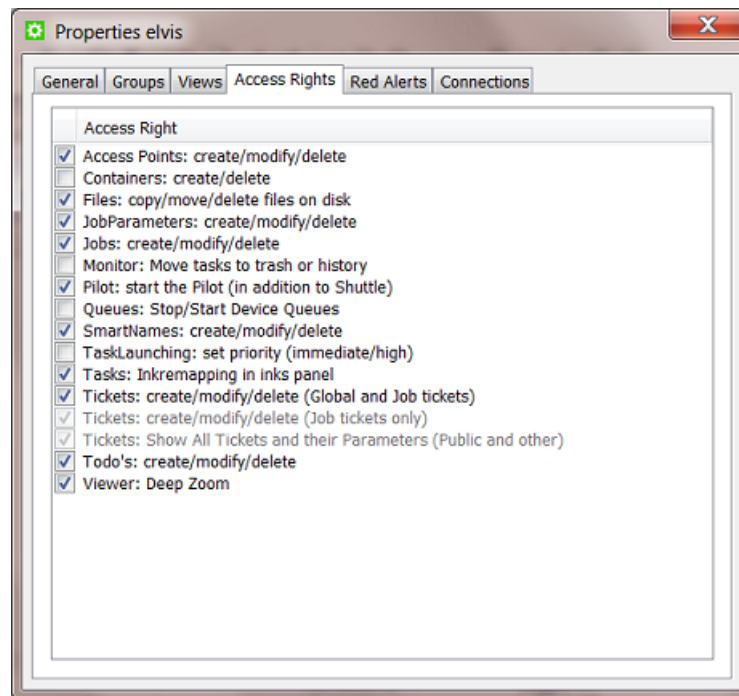
Note: The information cannot be edited in any way, except for the e-mail address (click the **Change** button to make the field editable).

- The **Views** tab lists which Views the user has access to (see [Views](#) on page 703).
- The **Access Rights** tab lists the user's access rights (see [Defining User Access Rights](#) on page 33).
- The **Red Alerts** tab lists which Red Alerts are presented to a specific user (see [Red Alerts](#) on page 705).
- The **Connections** tab lists from where and with what application the user is connected to the server. Naturally, several connections are possible and the **Connections** tab will be empty when the user is logged off.

Defining User Access Rights

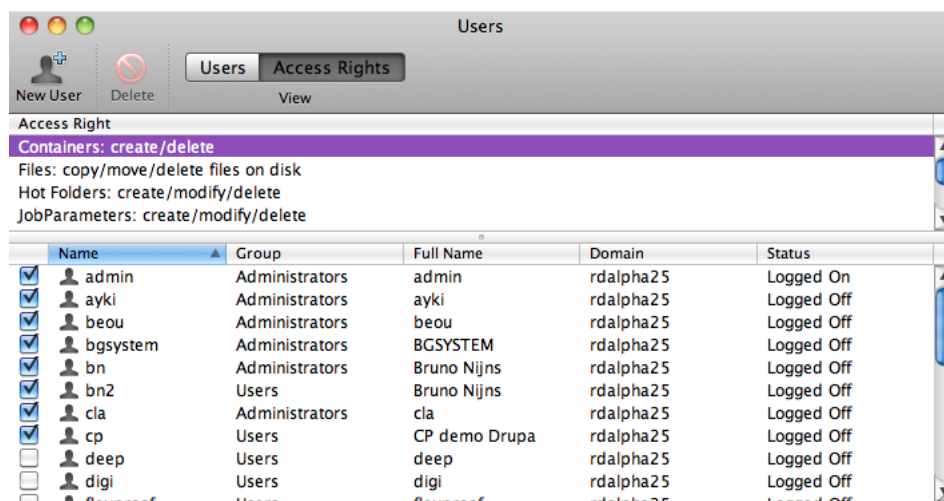
Not all users have the same responsibilities, so you can assign them different access rights in Automation Engine.

- To give access rights to a single user:
 - a) in the **Users** window, double the user you want to give access rights to,
 - b) in the **Access Rights** tab of the **Properties** window, check the access rights you want this user to have.



Note: Automation Engine Administrators automatically have all access rights. You cannot change this.

- To give a particular access right to several users:
 - a) in the **Users** window, click the **Access Rights** button to switch to the **Access Rights** view,
 - b) select the access right you want to give to your users,
 - c) check the users you want to give this access right to.



Access Rights

This access right:	affects...
Containers: create / delete	<i>Creation</i> and <i>deletion</i> of containers.
Files: copy / move / delete files on disk	what you can do with files in the Pilot interface. See the <i>Copy or Move File</i> on page 270 and <i>Delete File</i> on page 308 tasks.
Hot Folders: create / modify / delete	if you can manage Hot Folders or not. See <i>Creating a Task Hot Folder</i> and <i>Modifying Hot Folder Settings</i> in the User Guide.
Job Parameters: create / modify / delete	what you can do with Job parameters. See <i>Job Parameters</i> .
Jobs: create / modify / delete	if you can manage Jobs or not. See <i>Creating a Job</i> on page 835.
Monitor: move tasks to trash or history	what you can do in the Tasks list. See <i>Tasks list</i> and <i>Task History</i> on page 621.
Pilot : Start the Pilot (in addition to Shuttle)	your access to Pilot
Queues: stop / start device queues	what you can do with Queues. See <i>Queue Monitor</i> on page 688.
Task launching: set priority (immediate / high)	if you can set a higher priority to the tasks you launch.
Tasks: ink remapping in inks panel	if you can edit inks when <i>ripping to a TIFF file</i> or <i>proofing to Kodak Approval</i> .
Tickets: create / modify / delete (Global and Job tickets)	if you can manage Global and Job tickets. See <i>Tickets</i> on page 26 and <i>Working with Tickets</i> on page 177.
Tickets: create / modify / delete (Job tickets only)	if you can manage Job tickets. See <i>Working with Tickets</i> on page 177.
Tickets: Show All Tickets and their Parameters (Public and other)	<p>what tickets and parameters you can see in the Pilot (in the Tickets View, when launching a task or workflow on the file in the Files View, or when opening a processed file from the Tasks list).</p> <p>If this is deselected, you will only be able to use:</p> <ul style="list-style-type: none"> • public tickets (see <i>Tickets</i> on page 26), • public parameters in those public tickets (see <i>Using Public Parameters</i> on page 68), • the Launch with option on files (not the New Task option) (see <i>Launching a Task on a File</i> on page 76 and <i>Launching a Workflow on a File</i> on page 1034),

This access right:	affects...
	<p>Note:</p> <ul style="list-style-type: none"> Newly created users have this access right by default. You will not be able to create any ticket if you don't have this access right. Users with this access right can use a Pilot preference to simulate how the Pilot would look without this right (see Tickets Preferences). <p>This is useful for administrators who want to check how users without the full Pilot access right will see the tickets and their parameters.</p>
Access points: Create/Modify/Delete	whether you can create/modify/delete Access points
Viewer : Deep Zoom	the deep zoom rights in unprepared files as well as very deep zoom viewing in prepared files.

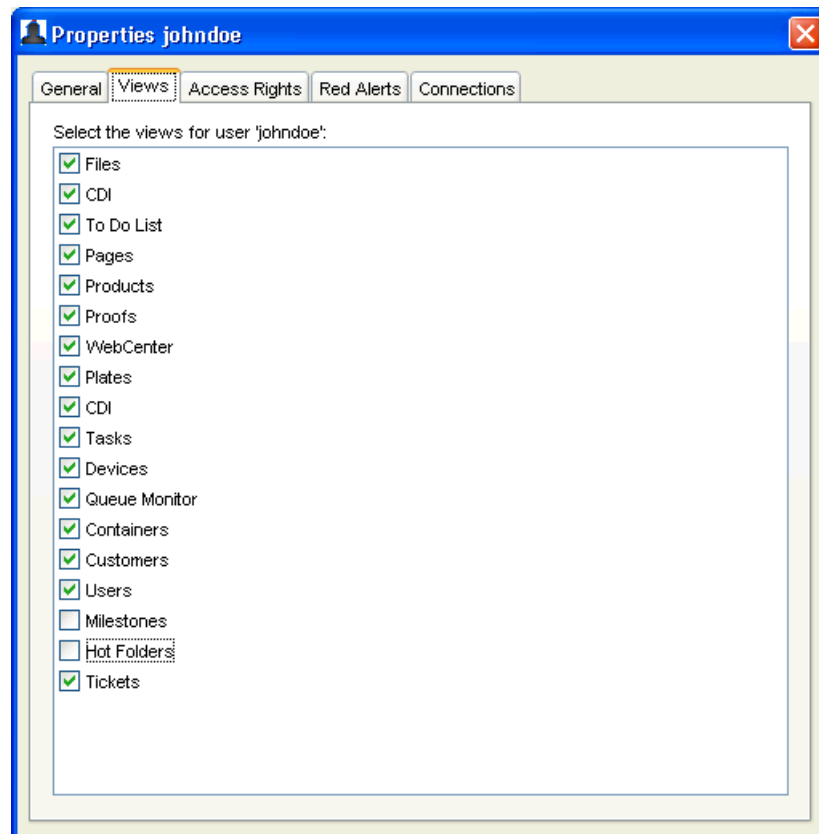
Views

For Administrator Users

As an administrator, you can decide:

- which views you want to see and which not,
- which views other (non-administrator) users are allowed to see.

In the example below, the user called johndoe is not allowed to see the **Milestones** and **Hot Folders** views.

**Note:**

You can also setup the views assigned to users from the Pilot's **Views** pane:

1. Right click in the **Views** pane and select **Select Views...**
2. The **Properties** window opens, showing the **Views** available for the user currently logged on. Select or deselect views as necessary.

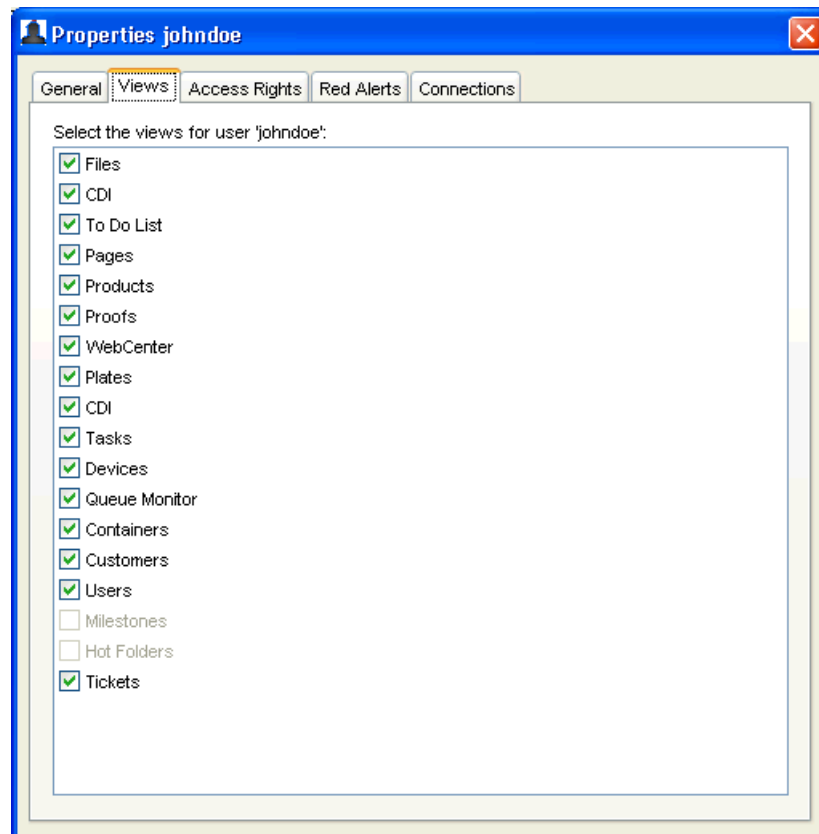
Administrators can also copy the view settings of a user to one or several other users:

1. Select the user you want to copy the settings from, and go to **View > Copy Views** (or click right that user and select **Copy Views**).
2. Select the user you want to give the settings to, and go to **View > Paste Views** (or click right that user and select **Paste Views**).

For Non-Administrator Users

As a non-administrator user, you can only remove some of the available views for yourself. You cannot add views that the administrator disabled, or change the views for another user.

In the example below, the **Milestones** and **Hot Folders** views are not available because the administrator disabled them for this user.

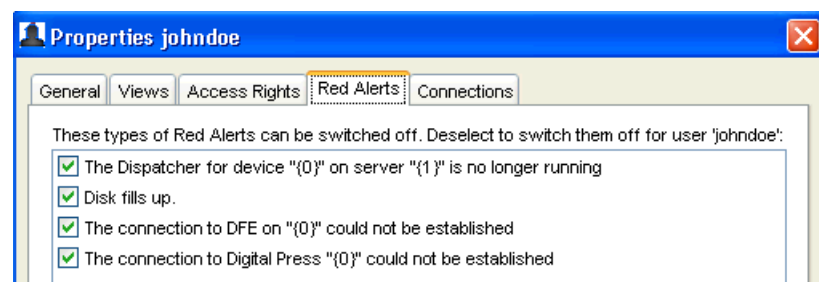


Note: When a user has changed his own view preferences, Automation Engine will automatically refresh after closing the **Users** window.

Red Alerts


When certain events are detected by the Automation Engine server, a Red Alert is broadcast to users.

Use the **Red Alerts** tab in the user **Properties** to configure which Red Alerts are shown to a specific user (deselect the alerts you don't want to show to that user).

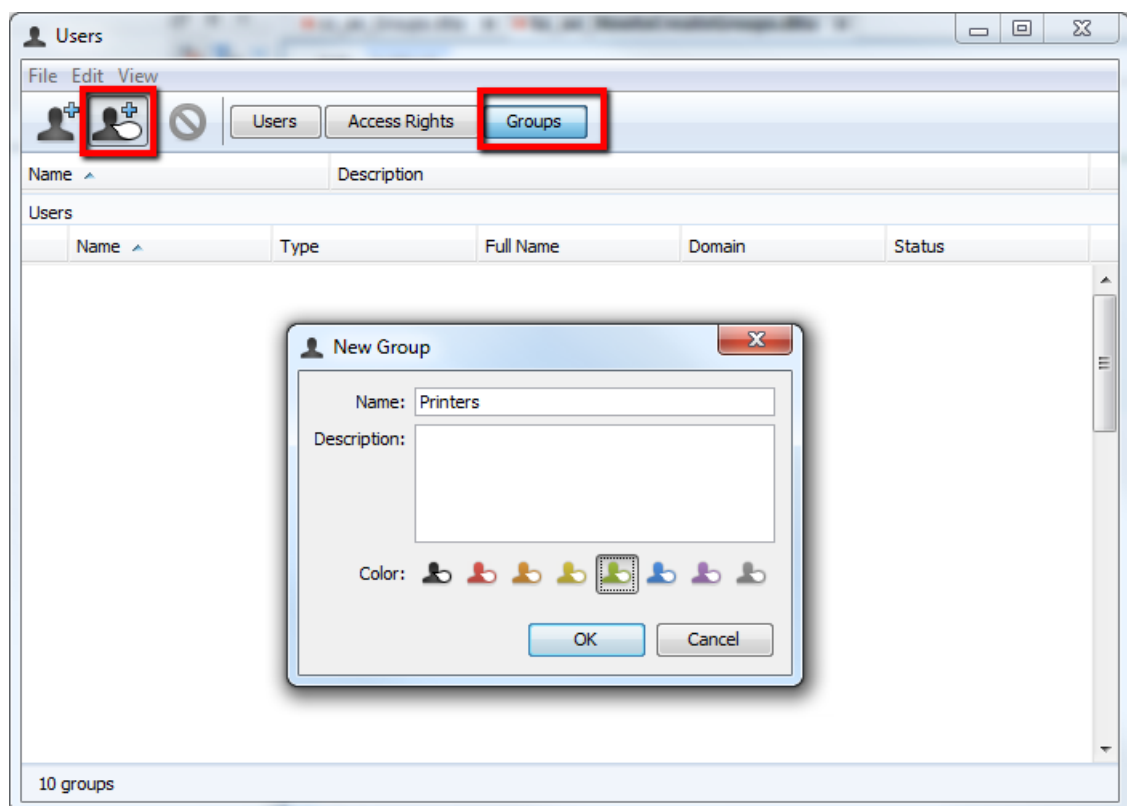


Groups

Creating Groups

1. In the **Pilot**, go to **Tools > Users**.
2. In the **Users** window, go to **Groups** tab.
3. Do any of the following action in the **Groups** tab.
 - Go to **File > New Group..**
 - Use the **CTRL + N** shortcut.
 - Click on the  icon.

The **New Group** dialog opens.



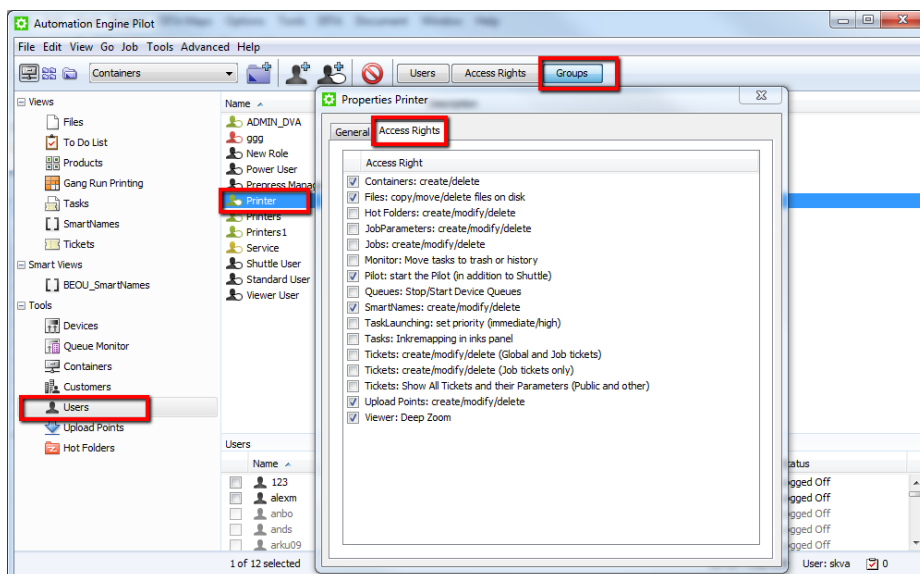
4. Enter the **Name**, **Description** and select a **Color** for the new Group.
5. Click **OK**. The new **Group** will be added to the database.
You can view the existing Groups under the **Groups** tab.

Defining Group Access Rights

You can assign relevant access rights to your **Groups**.

- In the **Users** window under the **Group** tab:
 - a) Double-click the group to which you will assign access rights.
The **Properties** window will pop up.

- b) Check the access rights relevant for the group in the **Access Rights** tab of the **Properties** window.



Note: Automation Engine Administrators automatically have all access rights. You cannot change this.

- You can give a particular access right to several groups/users in one go. Read more in [Defining User Access Rights](#) on page 33

A **Group** member has access rights that are assigned to the **Group**. This overrules the User Access Rights previously defined. If a user is a member in different groups, the relevant access rights will be a sum of the access rights assigned to these groups.

Assigning a Group to a User

- To assign members to a specific **Group**:
 - Go to the **Groups** tab.
 - Click on one of the Groups. All the users will be listed underneath.
 - Select the users to be assigned to this group from the list.
- To assign a **Group** or **Groups** to a **User**:
 - Double-click the user from the **Users** window.
 - Go to the **Groups** tab of the User properties window.
 - Add the member to one or more of the groups from the group overview.

2.9.13 Connection Doctor

As an Automation Engine administrator, you can run diagnostic tests from an operator's workstation to verify that the Pilot's connection to the Automation Engine server works properly. With the provided tools, you can attempt to fix the problems yourself without the need to call Esko Customer Support.

The **Connection Doctor** is documented in [Troubleshooting the Automation Engine Pilot](#) on page 1369.

2.10 SmartNames

SmartNames are documented as a separate chapter in the "Complete Manual". Find it here: [SmartNames](#) on page 916.

2.11 The ArtPro Action List Editor

2.11.1 Automating ArtPro Processing

Automation Engine can automate tasks in ArtPro and convert Nexus workflows into one or more Automation Engine workflows. You can do this using the **Apply ArtPro Action List** task (see [Apply ArtPro Action List](#)), which takes native ArtPro files as input, processes them and can output either ArtPro files or a range of vector formats. The task uses an ArtPro Action List to define the chain of actions to be executed. This Action List file can be created and edited by using the **ArtPro Action List Editor**.

Note: A considerable number of actions are identical or very similar to functionality that can be found in ArtPro. In these cases, a basic explanation is given in this reference manual. For a fully detailed description please refer to the ArtPro manual.

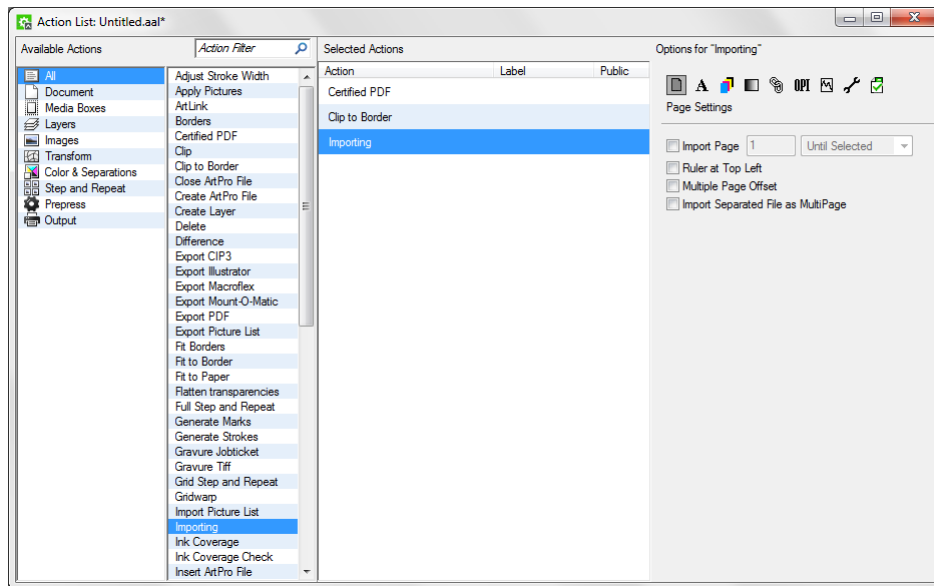
The **ArtPro Action List Editor** actions contain an elaborate number of vector outputs which converts the ArtPro native file format to a range of vector output formats.

2.11.2 Working with ArtPro Action Lists

Download the **ArtPro Action List Editor** from the **Server Administration** web tool, as described in: [Downloading A Client App](#) on page 208.

Install the **ArtPro Action List Editor** on your client machine as follows:

To open the **ArtPro Action List Editor**, select an Action List in the **Apply ArtPro Action List** task panel and click the **Edit** button.



The Action List you selected opens in the **Action List** dialog box. It is divided into three sections:

- **Available Actions**
- **Selected Actions** and
- **Options for "selected action"**

The **Available Actions** section has two columns. The left most is populated with all the available actions. The next column is populated with the filtered actions. Drag and drop desired actions from this column to the **Selected Actions** section. Enter an optional label to each action in this section to help identify these unique actions when using public parameters.

In the **Options for "selected action"** section, the main parameters of the selected action are displayed. These parameters can be edited. When you close the window, you are provided with an option to save the changes. The default file extension for the ActionList is `*.aal` and the format is always UTF8.

Action List Menu Options

The following is a list of menu options available in the **ArtPro Action List Editor**:

1. File menu:

- **New Action List...:** Click this option to open a new Action List.
- **Open Action List...:** Click this option to open an existing Action List.
- **Save Action List...:** Click this option to save an Action List.
- **Save Action List As...:** Click this option to provide a name to the Action List and save it in a desired location.
- **Exit:** Click this option to exit the **ArtPro Action List Editor**.

2. Edit menu:

- **Cut:** Use this option to cut an action from the **Selected Actions** section.

- **Copy:** Use this option to copy an action in the **Selected Actions** section.
 - **Paste:** Use this option to paste a copied action into the **Selected Actions** section.
 - **Delete:** Use this option to delete an action from the **Selected Actions** section.
 - **Select All:**
 - **Edit Preferences...:** Click this option to view the **Preferences** dialog box which contains two tabs: **GUI** and **Server**. Configure the **Distance Unit** and **Font Size Unit** in the **GUI** tab. Configure the Server details in **Server** tab.
3. **View** menu:
- Hide/Show Library:** Use this option to hide or show the Library (**Available Actions** section). **Hide Library** option is initially displayed in this menu which toggles to **Show Library** after user has hidden the Library.
4. **Window** menu:
- Action List: name.aal:** This menu displays names of all the Action Lists currently opened by a user.
5. **Help** menu:
- About ArtPro Action List Editor:** Click this option to view a window which displays information about the **ArtPro Action List Editor**.

Creating an Action List

To create an Action List:

1. Click the **New Action List...** in the **File** menu.
2. The **Action List: Untitled.aal*** dialog box appears. Using this dialog box, add actions to the Action List and define attributes for actions.
3. In the first column of the **Available Actions** section, select **All** to view all the available actions in the second column. Or filter the actions to view only some particular set (selection-related actions, check-related actions...). Also, use the search field to display certain actions (for example, type "OPI" in it to display all OPI-related actions).
4. Double click or drag actions from the second column of the **Available Actions** section to the **Selected Actions** section.
5. Choose an action in the **Selected Actions** section and click **Delete** in the **Edit** menu to remove it from the section.
6. Click an action in the **Selected Actions** section to view the associated parameters in the **Options for "selected action"** section. These parameters can be edited.
7. Click **Save Action List As** in the **File** menu to save this action list in a desired location.

Action lists have to be stored in the `\\server\bg_data_custom_v010\dat\ArtPro Action Lists` folder for the task panel to pick them up. By default, the **ArtPro Action List Editor** also saves them there, but you can choose a different location in the **Save Action List As...** dialog box.

2.11.3 Handling Fonts in ArtPro Files

Important: All fonts used in ArtPro files handled by Automation Engine should either be:

- embedded,
- referenced and located in a folder **accessible to the Automation Engine server**.

If your file has non-accessible fonts, you will get an error when trying to add or edit text, update text objects using ArtLink, export the file to PS or PDF...

ArtPro Files coming from a Nexus on Mac System

Type of ArtPro file in Nexus workflows	Consequence on fonts in Automation Engine
ArtPro files created, edited or opened in ArtPro.	<p>These ArtPro files have all fonts and outlines embedded (subset).</p> <p>They can be opened, repeated, output... in any ArtPro action list in Automation Engine, without any font problems.</p> <p>Exception: ArtPro files containing ArtLinked text containing a character that was not embedded (not yet used in the ArtPro file and the font is not available).</p>
ArtPro files created by a Nexus workflow (files that have not been opened and saved by ArtPro).	<p>These files do not contain any font embedding. Therefore, they cannot be used to generate PDF export, unless:</p> <ul style="list-style-type: none"> • You add a Text to Paths action before outputting the file to PDF. • You add all the necessary font folders (see below) in your first action.
ArtPro files created by a Nexus workflow starting with a PS/PDF import.	<p>Referenced fonts are not available during this workflow!</p> <p>Any action that needs text outlines (Generate Marks, Print to File, Export PDF...) will error.</p> <p>To avoid this, you need to:</p> <ol style="list-style-type: none"> 1. Insert a Save immediately followed by an Open ArtPro file. 2. In the Open ArtPro file action, define the necessary font folder.

Handling the Fonts Correctly in an ArtPro Action List

In general, opening an ArtPro file does not need the glyphs from the characters in the job, since ArtPro always stores the vectorized representation of the complete text block in the file.

When using ArtPro Action Lists and importing PDF files with non-embedded fonts, the fonts need to be available on the Automation Engine server. Do the following to make them available:

1. Create a fonts folder in a container of the Automation Engine server.
2. Move the fonts from Mac to (that folder on) Windows using a third party font conversion tool (for example [CrossFont](#)).
3. Adapt the actions that need font resources (for example the Importing action) in your ArtPro action lists.

Note: We recommend using TrueType fonts.

2.11.4 Available Actions

Adjust Stroke Width

This action allows to change the width of all (or all selected) strokes. For example: This can be used to give hairlines a minimal width. First use the Selector to select all strokes smaller than the minimum width, then use the **Adjust Stroke Width** action to change the width of the strokes.

Apply Pictures

This action applies an ArtPro file or picture to all objects in the file.

This function may be useful to replace the graphics in an existing step and repeat template with another graphics file. The box popup determines which box of the picture (only for ArtPro files and PDF files) is used to center the graphics when applying the pictures to the objects in the file.

Remove Unused Separations removes all empty separations after the **Apply Pictures**.

The **Place Single Separation ArtPro File in Single Separation** option defines how to insert ArtPro files with only one separation. If the option is switched off, objects in this AP file with 100% of the single separation are considered to be registration color and thus get 100% of every ink in the ArtPro file after insertion. If this option is switched on, the file is inserted in one single separation instead.

ArtLink

This action gives full access to ArtPro's ArtLink module from within the ArtPro Action List Editor and allows automatic generation of multiple versions of a template file based on database information. The different versions are in different files or different layers, depending on the setting of the radio button at the bottom.

Since the template file needs to be made in the ArtPro application, please refer to the ArtPro manual for a full description of the ArtLink module. For specific information on the use of Text parameters in ArtLink, please refer the Technote on ArtLink Text Parameters.

The template file which is generated in ArtPro contains all "hot linked" objects which refer to parameters in the database or automatic parameters based on job content. In what follows we describe how an action list needs to be set up in order to automatically generate ArtLink versions with the ArtPro Action List Editor.

Using ArtLink in an Action List

Example:

A basic ArtPro action list could contain a **Select File** action and an **ArtLink** action.

The input folder contains an ArtPro template file "ice cream template.ap", a database file "ice cream.txt" and an images folder which contains the various images which will be used for the versioning: strawberry, lemon, lime, orange, peach and tomato.

The Select File action contains two entries whose name ends with ".txt", referring to the database file and whose name ends with ".ap", referring to the ArtPro template file.

The **Group Files** option should be on, meaning these two files are grouped together for the ArtLink action, which needs the database file and the ArtPro file. Note that the database file always needs to be the FIRST entry in the list.

Setting the ArtLink Action

The ArtLink action has a number of parameters set.

ArtPro Template

The ArtPro file selection does not need to be selected here since it is actually determined by the Select File mechanism. This can be used if the ArtPro File with the links is a fixed file.

Additional Resources

Additional Resources will allow to locate the Color Book and the Ink Book that needs to be used, similar as one would load these in ArtPro when colors or inks need to be defined from a library (example: PANTONE™ library).

File & Fonts Settings...

The **File & Fonts Settings...** button allows working with root folders, folder mapping and font folders for the ArtPro template file, since this is actually an ArtPro file and may contain images (on the same or a different server) and fonts. The fonts are obviously necessary if variable text needs to be applied depending on the database parameters. For a full description of the **File & Fonts Settings...** we refer to the **Open ArtPro File** action.

Save ArtPro

The **Output File Naming Template** button determines how the files, generated by the ArtLink action are named and where they are saved.

As is clear from this resulting folder when the job has been executed, six jobs have been generated, each with a variable extension: .Alnk_1 to .Alnk_6.

It is also possible to define if a Preview needs to be saved and at what resolution.

Save Records

It is possible to define if the results of the different records should be saved in multiple ArtPro files ("Files") or in one single ArtPro file containing different Layers ("Layers"). In that case, the name of the different layers is defined using the File Naming Template.

The Database File

The database needs to be a text file as shown in this example:

```
PANTONE;ice;barcode;pict
"300";"Blue Curacao Ice Cream";"54327719";":~images:strawberry"
"399";"Margharita Ice Cream";"54327627";":~images:lemon"
"370";"Caipirinha Ice Cream";"54327719";":~images:lime"
"orange";"Tequila Sunrise";"54327726";":~images:orange"
"1645";"Peché Mignon Ice Cream";"54327733";":~images:peach"
1797";"Bloody Mary Ice Cream";"54327740";":~images:tomato"
```

The first line describes what variables will be used, while the values for the variables are each described on one line in the file.

- PANTONE™: the color entry in the color book to be used
- ice: a text string
- barcode: the value of a barcode to be used
- pict: an image to be used

Note: The images are described as ~images:xxx where ~ describes the current position of the template file and images is a subdirectory which is located relative to the current position of the template file. Images will also be searched through the Files & Folders settings, so the full exact path is not always necessary.

The Results

After processing, six ArtPro files have been generated, a zoomed view of the first ArtPro file clearly shows the variables which have been used: the color, the text string, the barcode and the image.

The **File Naming Template** in the ArtLink action allows the use of parameter values from the database records. The value of the n-th field can be used as a template key for the **Save ArtPro** file template.

Borders

The **Borders** action allows to modify the job size information of the current file.

Different job sizes can be defined in one file: media box, crop box, bleed box, trim box and art box. Note that if multiple boxes need to be modified, multiple Borders actions need to be set in the action list, as one Borders action changes only one of the Borders.

Each box sizes can be set to a fixed size depending on the size dialog or to a fixed size depending on dimensions which are put in the Vertical and Horizontal fields.

The square buttons in the lower part of the window determines in what direction the new box size is applied, depending on the current content of the file.

It is equally possible to calculate a difference to the current size of any page box (selected by the "start from Box" option), using the DV field (difference vertical) and DH (difference horizontal) field. In this case the Vertical and Horizontal fields needs to be set to 0, while the DV and DH values (positive or negative) are used to make the current box size larger or smaller again with the direction depending on the selected square button in the lower part of the window.

Suppose a file was imported from PostScript with the bleed contained in the job size, it could now have a Trim box which is for example 3mm smaller in all directions when setting DV and DH both to -3 mm. After being handled by the Borders action, it contains the page box information defined by these settings. Hence Step and Repeat applications or imposition applications can now place the files based on the Trim box (for example).

Certified PDF

Certified PDF allows to integrate the Certified PDF Workflow functionality into Automation Engine. The aim is to provide functionality to import Certified PDF, edit and then export as Certified PDF with information about edits and changes to the document. Certified can be divided in three parts: Input, Output and Certified ArtPro sessions.

Certified ArtPro Sessions

Automation Engine can work in Certified Mode, by switching it on in the Certified PDF action.

User name, Short user name, etc. - provides the information about the user who is editing the PDF document. That information is stored in Certified PDF each time when a new session is started.

If Certified Mode is enabled, the field "User name" is mandatory. So, before switching on Certified Mode a User Name should be specified. Otherwise, switching to Certified Mode is not allowed and a message box is displayed stating that User Name must be defined before turning Certified PDF on.

Working Certified Mode in Automation Engine affects the way it works for all following open/save actions. Whatever the Certified options states are, Automation Engine is now recording the history of changes made to the file and saves them with the ArtPro files. From version 8.0, all ArtPro files have the possibility to be exported to Certified PDF and all the history of changes are recorded. Those changes are recorded as sessions.

A session is a track of changes from beginning of the session to the end of the session.

A session starts when:

- A file that does not yet contain a session is created or opened. This can be: opening an ArtPro file from version lower than 8.0, importing a PDF (Certified or not) or importing any other file format.
- An ArtPro 8.0 or higher is opened, changed and saved. The active session is closed and a new session is opened.
- Certified PDF file is exported and Certified mode is on. In that case, the old sessions are recorded to PDF file and a new session is opened.

The session is not begun when:

- Certified PDF file is exported and Certified mode is off. The active session is continued.

Clip

Clip uses the frontmost selected path as a cutting form to adjust the shape of the underlying paths. Any part of the underlying paths that falls outside the frontmost one will be cropped. Thus it masks out the selected paths underneath.

The original paths are modified. For a more detailed description, see the ArtPro manual.

Clip to Border

The **Clip to Border** action clips all the objects to the edge of the selected PageBox.

Objects which are completely out of the pagebox are removed, objects partially inside the selected pagebox are clipped to the pagebox.

Choose **All** radio button to apply the **Clip to Border** action to all objects or choose **Selections Only** radio button apply it to the selected objects only.

Close ArtPro File

This action removes the currently open ArtPro file from memory. Note that every ArtPro action list needs to be terminated by a **Close ArtPro File** action.

When building an ArtPro action list that does not contain a Close ArtPro File action, the system issues a warning when closing the action list. Click the **Cancel** button and make the necessary adjustments.

Clicking the **OK** button merely closes the warning window, it is up to the operator to go back to the action list and add the **Close ArtPro File** action at the appropriate location. Failing to do so results in wrong output. Since data continues to remain in memory, Automation Engine continues to process the different pages and documents on top of each other until it runs out of memory.

Create ArtPro File

This action generates a blank ArtPro file which can be used later for adding pictures, inserting other files, etc.

Create Layer

The **Create Layer** action creates a new layer in the ArtPro file currently open in memory. Further processing takes place in the newly created layer.

A typical example is a double burn workflow, where multiple files are imported (CMYK of first version, K of second version, K or third version, etc...). To make later editing easier, set a **Create Layer** action before each of the imports, hence layering the final output file. The name of the layer is determined by the template.

It is possible to move the selection to the newly created layer, by switching on the appropriate option. Using the **Position** dropdown, create a layer on top or at the bottom of the file.

Delete

This action allows to delete objects, unused separations, unused layers and defined separations from the current file.

Objects

Delete objects

Select **Delete Objects** checkbox, click **All** radio button to delete all objects or click **Selection Only** radio button to delete only selected objects. Although not often used, deleting all objects could be used to delete the contents from a file but retain all pagebox settings for another document (or picture) to be imported into or placed. For more information on selections, see the Selector.

Separations

Separations mode

Select **Leave as is** in the dropdown menu to leave the separation "as is". Select **Remove** in the dropdown menu to define what separations to delete (Remove) or select **Only Include** in the dropdown menu to define what separations to keep. The **Delete Unused Separation** option is not influenced by this setting.

Die/ Transparent/ Varnish/ Opaque

The checkboxes allow to set if separations should be removed or included based on the separation type.

Separations list

If **Separations** mode is set to **Remove**, all separations defined in the list are deleted from the file. If **Separations** mode is set to **Only Include**, only the separations defined in the list are kept and all others are removed.

A separation name can be added by clicking the + button and typing the name in the text field. By selecting a separation from the list and clicking -, a separation name can be removed from the list. The separation names in the list can contain wildcards (*).

Delete Unused Separations

If this option is selected, unused separations are removed, regardless of the Separation mode setting.

Layers

Delete Layers

If **Delete Layers** is selected, all layers defined in the list are deleted from the file. A layer name can be added by clicking the + button and typing in the name in the text field. By selecting a layer name from the list and clicking -, a layer name can be removed from the list. The layer names in the list can contain wildcards (*).

Delete Unused Layers

If this option is selected, unused layers are removed.

Difference

The **Difference** action is used to calculate the differences between two ArtPro files.

This function is useful to verify that the graphics of two different language versions fit, to find out what changes a customer made to the new label, to verify that corrections on files have been executed, etc.

The difference calculation is based on a pixel comparison. Based on the resolution set in the **Resolution** field, the system compares the pixels of the two files at the same location.

When a difference is found between two pixels, one pixel is put in the resulting difference file at the same location. When no difference is found, no pixel (or a gray pixel in separated mode) is put in the resulting file. This means that when two files are compared that are identical, the resulting difference file is either empty or completely gray.

The comparison and the type of resulting file is dependent on the setting of the radio buttons at the bottom of the dialog box:

Separated

Each separation of the two files are compared on a separation basis. For each separation, a grayscale TIFF is written to disk. In the resulting grayscale images, a 50% gray pixel means there is no difference for that specific pixel, if the first file for the pixel is darker than in the other file, a darker pixel is put in the difference files and vice versa.

CMYK

CMYK pixel data of the two files are compared in this case. If no difference is found, the resulting difference pixel is white, when there is a difference, the pixel (CMYK) of the second file at that location is put in the difference file. The resulting file in this case is a CMYK TIFF file.

Note that with this comparison method, **ONLY** the CMYK information is taken into account, differences in PANTONE™ separations are disregarded.

Binary

This results in one single 1-bit TIFF file where differences are shown as black pixels.

Align Horizontal / Vertical

When the ArtPro files to be compared have different job sizes, these popup menus allow to define how the jobs have to be aligned before comparison.

Delete Matching Files

Removes the TIFF files of the separations in which no difference between the two jobs has been found. The Separated mode is the only setting that results in multiple files, hence this setting is the only one affected by this option.

Make Report Job

Generates a new ArtPro document, with the name of the compared files, date and time of comparison, used resolution and all difference files placed underneath each other, preceded by the separation name.

Output

Allows to specify the name and the location of the difference file.

File & Font Settings

Since the **Difference** action actually opens two ArtPro files, in some cases it may be necessary to specify Root Folders, Folder Mapping and Fonts folders. Please refer to the **Open ArtPro File** action for full details.

Export CIP3

The **Export CIP3** action creates a CIP3 file of the active job.

CIP3 is a file format used to transfer data about a specific file through the production process. Besides a preview which can be used to extract ink key settings, this CIP3 file contains information concerning separations, ink coverage, job dimensions, etc....

Two main settings are present: the oversampling of the preview as well as the resolution of the preview that is included in the CIP3 file.

Export Illustrator

This action exports an Illustrator native file (up to Illustrator 7.0) from the current ArtPro document.

Include Traps

Determines if traps are included in the Illustrator file. Multistep gradations are not converted into Illustrator multistep gradations but are exported as CT's with the resolution as determined by this input field.

Compatibility

Determines whether an Illustrator 7.0, 5.0, or 3.0 file is exported. Depending on this setting, a number of restrictions apply:

1. Export to Illustrator 3.0 was developed for line art only:
 - a. text is not supported, use the Text to Paths action before this export
 - b. images are not supported
 - c. gradations are not converted, only the first color of the vignette is exported
 - d. multistep gradations are not exported
2. Export to Illustrator 5.0 does not support TIFF images:

- a. placed TIFFS, mapped TIFFS and internal images are not supported
 - b. non linear multistep gradations are not supported
3. Export to Illustrator 7.0:
- a. mapped pictures are exported as CMYK TIFF images

Select the **Output Fully Transparent Object** checkbox to obtain transparencies as output using Export Illustrator.

Downsample Images

To sample the images in the file 2 times, 4 times, 8 times or not at all.

Export Macroflex

The Cyrel Macroflex uses the same philosophy as the Mount-O-Matic, but is manufactured by DuPont and has some differences.

The minimum distance between the two registration marks is 127 mm. The registration marks also contain characters that refer to the used separations in the group. **A** refers to the first separation, **B** to the second,...

The Macroflex is not using a cylinder to put the polymer on, but is a flat table model. First of all, an adhesive sheet of laminate the size of the cylinder is put on the table. The camera can move horizontally and vertically to show the exact position on the sheet for each piece of polymer. An operator glues the polymer onto the sheet. Once completed, the sheets of all separations are put on the cylinders of the flexo printing press.

Export Mount-O-Matic

With **Export Mount-O-Matic** it is possible to export an ASCII text file that contains all information specifically for the Mount-O-Matic machine.

Mount-O-Matic is a device allowing flexo printers to reduce the use of photopolymer in the plate making process. This machine is useful when only small parts of the file contain information.

Shrink means the same as vertical distortion and serves as information for the Mount-O-Matic. As the Mount-O-Matic file only contains coordinates and not the actual contents of the file, the same vertical distortion has to be used when printing the job to film.

Export PDF

This action converts the ArtPro file in memory to a vector PDF file. By default this PDF file is written to the job directory.

The **Export PDF** dialog box is based on different sets of settings or Flavours: "For Press", "For inPDF", "For Viewing", "Normalized PDF" and "JDF". The settings for each of these flavours can be changed. "inPDF" is used to create inPDF files.

Next to the flavour, an indication to the compliancy is given: this shows that, based on the settings, the file will be PDF 1.3, 1.4, 1.5 or JDF compliant.

Placed Files (S&R)

When outputting a PDF of a file containing placed EPS, this EPS is placed in the PDF file as a PDF XObject. This works on certain Rips. The Rip must accept PDF XObjects containing PostScript.

Placed DCS works only with Separated PS Output and not when outputting PDF.

Load Settings from XML

Select an XML file using a **Select File** action. The settings from that XML file can be loaded in the **Export PDF** action, overwriting the settings. The XML file can be saved (for example) in ArtPro, using the Export PDF Flavor option.

Normalized PDF

With the Normalized PDF flavour, output PDF files that can be opened and edited in PackEdge, Automation Engine etc.

1. Normalized PDF Options:

When using Normalized PDF as flavour, some specific functions are used and some general options are changed.

a. Generate Normalized PDF:

With this option in the **General** tab, the PDF file structure is If no screening is defined on all or some objects, a default screening will be applied to these objects. The defaults are: PDF and XMP information is included in the PDF file.

b. Force Default Screening

This option (in the **Optimize** tab) is used in combination with the **Include Screening** option. Since Normalized PDF does not allow objects without any screening definition, this option applies a default screening on objects that do not have any screening defined yet. This default screening is the default screening angles (15, 45, 75...) and a default ruling of 120 lpi.

c. Images:

Images are included as Esko links and also embedded at low resolution. This means image downsampling is set at 96 dpi, the **Link** option is set to "Esko" and the **Embed Images** option is on.

2. Normalized PDF Restrictions:

a. Screenings:

If no screening is defined on all or some objects, a default screening will be applied to these objects. The defaults are:

- ruling: 120
- dotshape: simpledot (round)
- angles C: 15, M:75, Y:0,B:45 any other : 45

b. PANTONE inks not in Esko color books:

These inks are saved as unregistered inks. Inks found in Esko ink books get the correct ink book reference. Ink type is kept (technical, opaque, varnish, normal).

c. Placed or repeated Files:

Placed or repeated files (ArtPro, Normalized PDF) are expanded in PackEdge.

d. No barcode information.

e. No font file information.

f. Text:

All text is saved and the font is subsetted and embedded. This results in real text in Esko Software. However, 2-byte fonts are always vectorized when exported to any PDF flavour in ArtPro.

g. Images:

Images keep their links if possible (most TIFF, JPEG and PSD files). This is also the case for mapped images. Some image formats such as DCS images, PSD with multiple alpha channels,

TIFFs with extra channels (often generated by ArtPro PS/PDF import) are not supported or incompatible with the Esko Software. These images are embedded or generate a .ct file on Esko software - depending on the settings.

General Settings

Generate Normalized PDF

Please refer **Normalized PDF** above.

Generate Separated PDF

With this option on, the ArtPro document is exported as a separated PDF in which each page represents a separation of the ArtPro document. It can be combined with the **Add Composite on Page 1** option.

Add Composite on Page 1

This option is available when **Generate Separated PDF** is on. If enabled, the first page of a separated PDF will contain a composite proof of the ArtPro document. All subsequent pages represent a separation used in the ArtPro document.

Add Notes

With this option on, all ArtPro notes are included in the PDF file and appear in Acrobat or the Reader as normal, editable PDF notes.

Add Non Printing Layers

If switched on, layers set to non printing are exported into the PDF file. With this option off, these layers are not output, so the PDF file contains only layers with the Print option on.

Include 1.5 Layers

PDF 1.5 allows the use of layers. If the option is switched on, the ArtPro layers are put in the PDF file as 1.5 layers. The compliancy is set to PDF 1.5. This allows user to switch layers on/ off in Acrobat.

Output Fully Transparent Objects

Without this option, fully transparent objects are ignored. With this option on, transparent objects are output.

Page Layout

The Page Layout section allows to set Orientation, Mirror and Distortion.

The Orientation can be set to portrait, turn right, turn left or upside down. It defines the direction in which the file is put on film or on paper. Choosing portrait outputs the file in the upright position, turn right rotates the file 90° and turn left rotates it -90°. By activating Mirror, the job is mirrored before exporting, resulting in a wrong-reading output. Distortion is defined as a percentage of the jobs original dimensions. This distortion is specifically important for flexo jobs. On mounting a flexo plate on the press, it is stretched out in one direction. By making the output slightly smaller than needed, we compensate for this distortion. The original dimensions of the ArtPro file remain unchanged. Positive values make the job bigger and negative values make the job smaller. Example: Vert. distortion = -2.35% makes the output 97.65%. In this way the output can also be scales: example, entering -50% for both Vertical and Horizontal Distortion outputs the job at half of its original size.

Add Thumbnail

With this option on, a thumbnail is created of the file, a small preview, that can be shown in thumbnails in Acrobat. With this option off, the thumbnail in Acrobat is gray.

Add Standard Preview

Preview can be defined by resolution (default 72 ppi) or by max size (in kb). This can be useful when creating a PDF of a large size S&R, of which the preview at 72 ppi would be several Mb.

Add Separated Preview

It is now possible to save a separated 72dpi preview in the PDF file. This non-standard preview can only be used by other Artwork Systems software, such as ArtPro S&R...

Use Anti-Aliasing

Allows to define if Anti-Aliasing needs to be applied when calculating the preview.

Previews & Page Boxes

Thumbnails are created using the Media box. Separated previews are also created using the Media box. The Standard Preview is based on the Art box, in order to be compatible with other applications. This implies that the Standard Preview does not show any bleed. For JDF output, these settings are used for all PDF files created.

Marks

This option allows to add marks into the PDF file. The **Marks** button and option work in exact the same way as in the **Generate Marks** Action.

Optimize Settings

Use Picture Caching

Picture Caching makes the output faster by keeping all images that are put in the PDF file in the RAM, until the complete file is written. This keeps application from having to read an image multiple times from the hard disk, if it is used multiple times in the file. The condition is that enough RAM is required on the machine. This is very useful if images are stored on a remote file server.

Output Gradients as Images

If this option is switched on, all gradients are converted to images instead of including them in the PDF file as PDF Shadings.

Compress Page Description

When this settings is switched on, the PDF file is compressed. This option is switched on by default.

Include Screening

If the option Include Screening is on, Extended Screening and PostScript Screening information is saved in the PDF file. screening is not included.

Generate JDF Output

JDF Output is an optimization system for outputting S&R of PDF files. When outputting a Step & Repeat to JDF, multiple files are generated. First of all, the one up file(s) are exported to PDF file(s). If the One up already was PDF, the file is just copied, unless the option **Copy File** is off. Second, one or more PDF files are generated containing all elements that are on the Repetition: registration marks, test strips, extra information, etc. Items that are grouped, are in the same file. Third, a JDF file is created, containing the positioning of all these PDF files.

Copy PDF

If the Copy PDF option is on, the original PDF files in the S&R will be copied to the output folder of the JDF Output. If the option is off, these single PDF files will not be copied, and the JDF will contain a link to their original location.

Target Size

This option can be used to ensure that the size of the PDF file does not exceed a certain size (example: 5000 kb) This can be used for files that need to be sent, example, by e-mail. In order to keep the file size below the maximum size, images may have to be downsampled and/ or compressed more than set in the dialog box. Do not use this for files that are sent to Press!

Split Long Paths

Just like PS output, paths with many anchor points can be split up as parts. The more parts that need to be exported, the longer it takes. The default settings is 500.000 points.

Text Settings

Vectorise text

Before exporting, ArtPro converts all text to paths to avoid font problems at a later stage. When this option is switched off, the text is in the PDF file as real text. If text is not set to be vectorised, it is always embedded. Japanese 2-byte characters are always vectorised.

Subset Embedded fonts...

Subset Fonts define whether fonts should be embedded as subset if only a part of the characters are used. How much of the font characters should be used before the font is embedded completely, can be set in the When percentage of characters used is less than field. If this value is set to 100%, a subset is created for every font which is not used completely.

Color Settings

Force Opaque Inks to knockout

If the option **Force Opaque Inks to Knockout** is on, overprints is first flattened, so a visually correct result is generated, but the resulting PDF is NOT OK for printing! In order to warn that the result is correct visually, but not for print, a red exclamation mark is shown when switched on.

Fail on Nonseparable Blend modes

If the file to be exported contains Nonseparable Blend modes (Hue, Saturation, Color, Luminosity), the file fails if this option is switched on. These Nonseparable Blend modes operate on a 3 component color space. This means that to calculate the resulting color, CMYK needs to be converted to 3 components, the Blend needs to be calculated, and the result needs to be converted to CMYK again. However, no instructions are given on how these conversion from one color space to another should happen. This results in different behaviors and different color results in different applications/ RIPs.

Exclude Separations

This option can be used to exclude separations from the output. By clicking the Options button, the Output Separations dialog is opened. By Adding a separation (click the + button and define its name. Wildcards can be used), you can exclude this separation for output. By selecting an entry in the list and clicking the - button, you can remove it from the list. The **Search Unused Separation** setting at the top, excludes separations that are not used. It only turns separations off, separations excluded earlier can not be removed from the Exclude list, even if they are in use. The **Search Unused Separation** option does not take registration color into account. The **Exclude separations** can not be used in combination with the **Include separations** option.

Include Separations

This option can be used to include only the selected separations from the output. By clicking the **Options** button, the **Include Separations** dialog box is opened. By Adding a separation (click the + button and define its name. Wildcards can be used), you can include this separation for output. By selecting an entry in the list and clicking the - button, you can remove it from the list. The Include separations can not be used in combination with the Exclude separations option.

Color Management

Color management is a technique used on color in documents to ensure that consistent color is reproduced throughout the print production cycle. Managing color means making sure color is reproduced in the same way for each device the system is outputting to.

If a profile has been defined at the time of importing or opening a file in the action list, it is possible to override the settings for output (example: color profiles for proofers or separated output) in this menu.

DotGain

This allows to apply a dotgain file on the output. For a full explanation on Dotgain, we refer to the ArtPro manual, in the Screening section. The applied dot gain curve is applied to the PDF as a

transfer function in the Graphic state. The dot gain curve for placed files works without problems with placed ArtPro files and PDF files coming from ArtPro or the ArtPro Action List Editor. Placed PDF files from other applications work fine, as long as they do not define their own dot gain curve inside the PDF, as the embedded curve has priority. If a placed PDF file coming from ArtPro or the ArtPro Action List Editor already contains a dot gain curve, a warning is generated on PDF Output.

Always Generate CMYK Separations

This option includes all CMYK separations to deviceN color spaces as soon as one of the CMYK separations is used. This option is only useful for customers that use a GMG color manager on their PDF files.

Flattener

The Flattener is used to translate transparencies and blends. This is necessary when outputting PDF 1.3 compliant.

The Flattener section, contains all settings to flatten Blend modes and overprints. It can also be run as a separate action: Flatten Transparencies. The Flattener replace all blend modes by “normal” objects, maintaining the final result, that is, the blend is calculated to whatever is underneath at that time, and is adapted in a linear object (if possible) or an image.

Image Settings

Use Image Downsampling At

All images with a resolution higher than this value, is resampled. This allows you to create smaller PDF files, example: for a quick soft proof.

For Images Having at Least

If this option is on, only images with a resolution higher than the entered resolution, is downsampled to the down-sample resolution. In the example above, when switching on the option, images with a resolution below 450 ppi remain unmodified, while images with a resolution above 450 ppi are downsampled to 300 ppi.

Use Averaging When Downsampling

This option is the same as in the more print settings. When it is activated, image quality is better, but the output is slower. The difference with PostScript output is that CCITT images are averaged too when outputting to PDF.

Image Compression

No Compression, Lossless compression (Flate), Maximum Quality, High Quality, Medium Quality or High Compression. None means that all compression is taken off. Compressed images are decompressed in the PDF file. A lossless compression means that the file will still be complete after decompression. There is no loss of quality. In the three other cases, the files are compressed with a JPEG compression. A higher compression means smaller files, but worse quality and vice versa.

Include RGB preview of images

This adds a lowres 72ppi preview of the large images inside the PDF. These previews are used by some Acrobat plug-ins, to speed up previewing files.

Image Clipping

The Image Clipping option allows to force clip images to the smallest size inside the clipping path. Allows calculation to include the full picture if clipping does not give any benefit, that is, if the sum of the image parts is close to or bigger than the size of the full image. A full image is only included once in the PDF file, even if it is placed multiple times.

Links

This option allows to include OPI comments into the PDF file. Choose between None (no links), OPI links or Esko links. The option Embed Images allows to define if images with OPI comments should

be embedded as well. OPI comments can not be used in, for example, Illustrator, unless the InPDF plug-in is used. For more information, see the InPDF documentation.

File & Fonts settings

This option allows to open the Root Folder, Folder Mapping, Fonts Folder and Color Management settings. These are the same as when opening an ArtPro file.

Certified PDF

If the “Output profile” file is selected then PDF file is preflighted after being exported. The Output profile can be cleared by shift-clicking the **Browse** button. If **Certified PDF Enabled** is unchecked, then regular non-certified PDF file is saved and no new session is started. The **Incremental save** option works only if the last Certified PDF (that is, the imported Certified PDF or the last Exported PDF) is still available. The option means that the previous Certified PDF file’s contents are merged with the exported PDF, so that “rollback” the PDF document to the previous session is possible. Using this option can result in a large PDF file. To keep both the PDF as the ArtPro file in a certified (“green”) state, it is necessary to save the ArtPro file after the Certified PDF Export.

Export Picture List

The **Export Picture List** action creates a Picture List file from the Step and Repeat file. An option allows to define whether mapped pictures should be included or not. The File Name Template allows to define name and location. A Picture List can be used in the **Import Picture List** action.

Fit Borders

The **Fit Borders** action adjusts the job size and the selected page boxes (All or a selection from the Border popup menu) to all the elements which are on the current document. Note that when **All** is selected, this means that the media box, crop box, bleed box, trim box and artbox are all set to the same size. Users can define if they want to fit the borders only vertically, horizontally or both. They can define whether it should be applied to selections only or on all.

Fit to Border

The **Fit to Border** action fits all the objects to the edge of the selected PageBox. If the option **Scale Proportionally** is on, all objects are scaled proportionally, until the bounding box of the selected object reaches the selected border, either in Vertical or in Horizontal direction. If the **Scale Proportionally** option is off, the selection is scaled to fit the borders both vertically and horizontally. This can sometimes be used when creating sleeves.

The Scale factor can be rounded off using the **Scale round to** value. If set to, for example, 5, the scale is rounded down to the closest multiple of 5. If set to 0, no rounding is applied. The upscaling can be limited using the **Limit scale to** field. The **Fit To Border** action can be applied to all objects or only to the Selection. Apply on **Notes** transforms notes along with the rest of the file, so that the positioning of the notes inside the file remains the same.

Fit to Paper

The **Fit To Paper** action selects the most appropriate paper size from a list of paper sizes. This allows to select the most appropriate page size (for proofers) or plate size (for image setters) for a job, based on the available proofers and/ or plate sizes. These paper sizes can be entered in the Action settings, using the + button. The Media Box defines what box is to be used to determine the optimal paper size. It can be set to **Center on Page**, **Upper Left on Page** or **Fit To Page**. Only this last option

actually scales the job to fit on the optimal paper size. If the **Rotate to fit** option is on, it is allowed to rotate the job when searching for the optimal paper size.

Flatten Transparencies

The **Flattener** section, as found in the Export PDF and Print dialog boxes, contains all settings to flatten blend modes and overprints. It can also be run as a separate action: **Flatten Transparencies**. The Flattener replaces all blend modes by "normal" objects, maintaining the final result, that is, the blend is calculated to whatever is underneath at that time and is adapted in a linear object (if possible) or an image.

Please note that placed PDF or placed ArtPro files can not be flattened! In those cases a warning is generated: `Flattener was unable to process placed art...` This happens only if in the job, some objects with transparency have to be flattened with a placed image.

Resolution

The Resolution allows to set at which resolution an image should be made, when necessary.

Noise for Gradients

If it is necessary to translate a gradation into a ct, the **Noise For Gradients** field sets the amount of noise that has to be applied.

No noise from/ to

This allows to define a range in which no noise should be applied to gradations. This is useful, for example, in autoblend screening.

Vectorise Opacity Channel

This is an optimization tool for the flattener. It reduces the path containing an ArtColor internal image, removing all areas being completely transparent (by vectorizing the opacity channel) before flattening. This way, objects underneath a part of an image where it is 100% transparent, no longer need to be flattened.

Flatten Overprints

If the **Flatten Overprints** is on, objects containing blend modes as well as, objects containing overprint (that is, "trans" in the Paint Style) are flattened. This can be used to get a correct preview in older Acrobat versions and Mac OS X Preview.

Ignore Overprints on Placed Art

This option allows to ignore Overprints on top of placed Art. If this option is not on, Overprints on top of placed Art (example: a S&R) results in a failed job.

Full Step and Repeat

The Full Step and Repeat action is based on ArtPro Interactive Step And Repeat. This functionality allows you to create a repetition of one single file at a time, based on either count or fill size.

General

Count/ Fill size

There are two ways to make a step and repeat in **Full Step and Repeat**: count, where the horizontal and vertical number of labels is defined, or fill size, where the size of the paper is defined and as many labels as possible is put in this area.

When using the fill size, the number of labels is calculated and equally divided in the remaining space around the S&R. When count is used, the fill size is calculated. When a group of labels is selected in ArtPro afterwards, all the used parameters are displayed.

Step

The **Step** is the space used for every file. Unlike the Size shown above, the step size takes rotation and bleed (margin) into account.

Margin

Margin is used to define the bleed between the labels. It is the difference between the Step and the box size. The horizontal and vertical margin can be set separately. If the bleed of the single file exceeds the bleed applied on the repeat, the jobs are cut off in the middle.

Gap/ After

The **Gap** is the distance in between labels. **After** defines after how many labels the gap should be entered. A value of 2, for example, results in two labels without gap, then the gap and then again two labels without, etc.

Distort

The **Distort** value allows to define a distortion for every repeat block separately.

Bleed

The **Bleed** defines how much bleed is used to the outside of the repetition. So there is a different setting for bleed in between jobs and around the repetition. It is ignored if the **Keep Bleeds** checkbox is selected. It is always check if there is enough bleed in the selected file. The trim box size is subtracted from the bleed box and divided by two. This size has to be bigger than the bleed, otherwise an error message is displayed.

Keep Bleeds

When the Step distance is smaller than the bounding box of the ArtPro file, the different labels do not overlap, but they are cut off. **Keep Bleeds** forces bleeds in the original file to be used, even if there is no place to put them. This function can be useful to generate repeats of round labels.

Angle

Using the Angles popup, define rotation for the labels. The options next to the angle, allow to set labels to switch direction every other label, either vertical or horizontally.

Cut Marks

If the **Cut Marks** option is on, cut marks is automatically generated around the repeat block. The cut marks are generated on the used box size. When bleed is applied in between labels, double cut marks automatically appear. The cut marks are filled with registration color.

Pattern

This option is used to make nested S&R. This means that the second row or column is shifted each time and the step distance can be decreased so the labels fit into one another.

Pattern

Using the **Pattern** buttons, define whether every second column or every second row should be shifted. The first button creates a rectangular pattern, with no staggering. The second two create most basic staggering: every second row or column is shifted, and contains one label less than the first row or column. The next two buttons generate rows and columns with the same number of labels. The buttons for round labels shift round labels into each other to save space.

Stagger offset

The offset can either be half of the step size or any user defined value. When using the Round Label pattern, the offset is automatically derived from the size of the job and can no be changed.

Continuous Stagger

Without the **Continuous Stagger** option, all even rows or columns are shifted, the odd rows or columns remain unchanged. When using the **Continuous Stagger** option, every row or column is shifted in regard to the previous.

Include Partial Labels/ Do not clip partial labels

When using the Include Partial Labels option, the gaps created by the shift of a row or a column is filled up with the part of the label as if the repeat was continued outside the job size. When setting the **Do Not Clip Partial Labels** option on, these labels are placed completely instead of just a part.

Advanced

Page Box

Before stepping the selected file, it is possible to decide which box size needs to be used. By default, this value is set to **Trim Box**. When **Page Box** is modified, the values in step distance and size are automatically adapted.

Place Single-Separation ArtPro File in Single Separation

This option defines how to handle repeating ArtPro files with only one separation. If the option is switched off, objects in this AP file with 100% of the single separation are considered to be registration color, and thus get 100% of every ink in the ArtPro file after repetition. If this option is switched on, the file is placed in one single separation.

File & Font Settings

Provides access to the root folders, folder mapping and fonts folders when the file to be stepped is an ArtPro file. For a description of this function, please refer to the **Open ArtPro File** action.

Color Management Settings

This setting determines if color management has to be applied for the Step and Repeat job. When enabling color management, it is possible to select a color management settings file, previously saved by ArtPro. This file contains all ICC profiles and settings to determine how color has to be treated throughout the rest of the workflow (example: proofing of the Step and Repeat document), including ICC profiles for proofing, printing, etc.

Generate Marks

This action physically adds marks to the file, based on the dimensions of the box set in the action dialog box.

Marks

The **Marks** settings define what marks will be used:

User Text

This allows to add user defined text. The text can be defined using the **File Naming Template**, so that file names, user names, user keywords etc. can be included in the marks text.

Job Name

Puts the name of the ArtPro file in the lower left corner in all separations.

Version Number

This is the number that is shown in the **Job Size** dialog box. Every time an ArtPro job is modified after saving, the version number is increased.

Date & Time

Puts the current date and time in the lower left corner in all ink separations.

Separation Names

Puts the names of the separations on the separate films.

Distortion

Puts the distortion from the **Print** dialog box in the file. The value is preceded by V or H, depending on the direction of the distortion.

Dot Gain File name

If a Dot Gain file is selected in the print dialog box and the Dot Gain compensation option is tagged, the Dot Gain file name can be added to the comments.

Color Setting Name

If a set of Color Management settings are loaded, the name of this set can be added too.

Crop Marks

The crop marks are put on the bounding box.

Registration Marks

Appears in the four corners outside the job or centered on the job when the option centered is on. The width of the registration marks is 1/300 inch (about 0.08 mm) and the color is always registration.

Marks File

This option actually allows to select a physical marks file from disk, clicking on the button with the three dots opens the browser for selecting the file. For a description on marks files please refer to the ArtPro reference manual.

Options

The Options define how the marks will be handled.

Vectorize Text

Converts all the text to paths, to avoid font problems on the RIP.

Set Inside Border

Sets the text on the inside of the borders instead of outside the job borders.

Use Text Settings

Define the font to be used for the marks using this settings. This can be interesting for Japanese users. If a default Japanese font (example: Osaka) is defined, all marks text will also contain Japanese characters.

Box

The Box drop down allows to define to what borders the marks should be applied.

Adjust Crop Box/ Adjust Media Box

This modifies the crop box after creation of the marks, so that the job and the marks are completely inside the crop box. In that case, it may be necessary to change the media box as well, to avoid a crop box bigger than the media box. Therefore the option **Adjust Media Box** can be used.

Files & Fonts

This button allows to define root folders, folder mapping and fonts folders, similar as in the **Open ArtPro File** action (please refer to this action for a full description). This functionality has been made available in case ArtPro files have been placed in an open job during the action list, to allow for the retrieval of the linked images or used fonts at the time of printing.

Unit

Allows to define what unit should be used in the Marks. This unit can also be used for the ArtLink parameters added by the **Marks** action.

For PS/ For PDF

As the Dot Gain curve and Distortion can be placed as Marks and these two settings can be made in the **Print** menu and in the **Export PDF** dialog, a pop-up has been added next to the **Generate Marks** button. The pop-up allows to set what Dot Gain and Distortion should be shown in the marks: the settings for PS (from the **Print** dialog) or those from the **Export PDF** dialog. The pop-up only has affect when the **Generate Marks** action is used. If Marks are added, for example in the **Print** action, the appropriate settings (in this example, the settings as defined in the **Print** dialog box) is shown in the marks.

Generate Strokes

The **Generate Strokes** action creates strokes around all objects or around all selected objects (see Select Objects for more information).

Width/ Inside/ Outside

The **Width** defines the line width for the Stroke. Filling in the width results in a symmetrical Stroke. Otherwise Outside/ Inside distances can be defined separately.

Corners/ Caps

The **Corners** and **Caps** settings allow to define the shape of the strokes in the (outside) corners and at the end points. For elaborate information, we refer to the ArtPro manual.

Separation Name/ Ink book

Strokes generated by this action gets 100% of the separation entered here and overprint for all other separations. If the separation does not exist yet, it is created. In that case, the specifications for the ink is searched in the defined Ink book.

Apply on All/ Selections

Allows to define whether strokes should be applied on all objects or only on the selected objects.

Add Stroke to Crop Box/ Trim Box/ Bleed Box

Allows to add a stroke on the selected page box.

Dashed

The **Dashed** option allows to generate dashed strokes. The Dash distances and Gap distances define the length of the individual dashes and gaps.

Gravure Job Ticket

Gravure job ticket offers export for Gravure systems from Hell Gravure.

You can define the **Resolution** and **Oversampling** rate defined in the action, if they can not be retrieved from the job ticket. Each job ticket and thus every separation can have a different value.

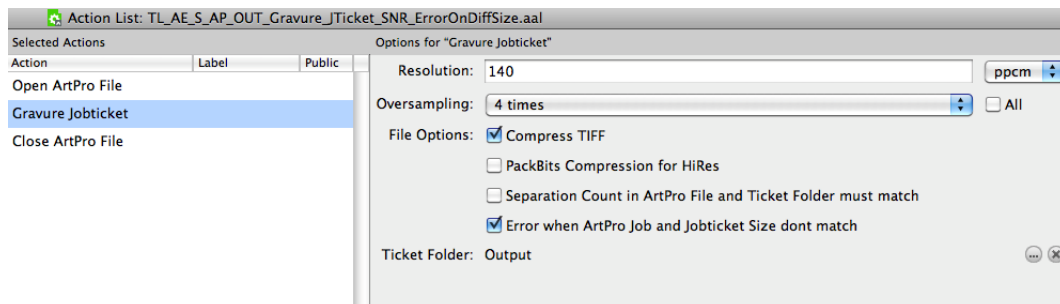
For separations with a resolution above 600 dpi and without oversampling, you can select **PackBits Compression for HiRes**. Otherwise, you can **Compress the TIFF** using LZW, or leave the separation uncompressed (if you did not select **Compress TIFF**). PackBits compression results in lower compression rate, but will be calculated much faster than LZW compression, so it is the best solution for high resolution files.

Selecting **Separation Count in ArtPro File and Ticket Folder must match** will ensure the **Gravure job ticket** will check if the separation count in the ArtPro file matches with the ticket count in the folder.

It is possible let the task **Gravure Jobticket** generate a warning or an error if the ArtPro job and jobticket size don't match:

- Selecting **Error when ArtPro job and jobticket size don't match** will generate an error. Also, no file will be generated.
- Deselecting **Error when ArtPro job and jobticket size don't match** will generate a warning instead and Gravure JobTicket output file will be generated.

For other specific information, contact Hell Gravure.



Gravure Tiff

Gravure Tiff is used to output ArtPro files directly as TIFF data for gravure printing. It creates one TIFF file per separation.

The parameters

1. Output

Choose TIFF in the **Output** menu.

2. Oversampling

Oversampling means that ArtPro makes the calculation of the TIFF on a resolution which is too high, that is, higher than the one needed. Afterwards it is 'averaged', this means that it is converted to low resolution by calculating the average of each group of lines. This gives a kind of "blurred" effect. Oversampling requires a long calculation time and therefore needs optimization.

The oversampling filter can be chosen from none to 8 times:

- none: a lot of jaggies
- 4 times: default value, this is the oversampling value used the most in the packaging and label market
- 6 times: very high, example: for stamps

Example:

If you have an oblique black line on 360 lpi (= 140 l/cm), the edges are very much pixelized. **Oversampling** means that you calculate with a higher resolution than the one you need. Oversampling 4, means that the image is calculated at 4x the resolution (360 lpi) = 1440 lpi. Every line is divided by 4. So there are 16 pixels read (4x4) for one output pixel. The output pixel gets the grayvalue of the 16.

3. Resolution

The resolution of the TIFF file can be defined in l/cm or l/inch. A normal engraving resolution is 140 l/cm or ±360 l/inch.

4. Noise for gradation

The default : 3 %

1 byte = 256 gray values

Gradations normally do not have more than 250 bands (steps of 0.4 %). ArtPro creates 400 bands instead of 250. The added noise makes the steps invisible. The desired amount of noise can be entered in this field.

5. IBM PC Byte order

By default this option is selected, because most engraving machines are IBM-oriented. The system IBM uses 'Little-endian' while Macintosh uses 'Bigendian'.

6. 1 Bit

This option allows you to choose between 1 bit data (checked) and 8 bit 256 graylevel data (default, unchecked). Most systems take 8 bit data.

7. Save Fast Forward Data

This option creates an extra text-file (.SVS) with a description of which parts of the TIFF-output file are blank and hence can be skipped. Some engraving machines have the possibility to take this information into account.

Grid Step and Repeat

Similar to the other step and repeat action, this action gives access to ArtPro's grid step and repeat module from within the ArtPro Action List Editor.

The grid step and repeat allows to create a repetition of different ArtPro (or other supported) files with the same size that are all placed in the same folder. The one-up files can be defined using a **Select File** action before the **Grid Step and Repeat** action.

This repetition can be executed with or without a layout file which is actually a simple spreadsheet containing the positioning of the different files.

For more information we refer to the Grid Step and Repeat section of the ArtPro manual.

Gridwarp

The **Gridwarp** action allows to calculate complex warping of documents based on a source grid and a destination grid.

This action can handle the warping of all element types like gradations, images, etc..... and is typically used for the warping of conic cups, metal containers, etc....

Grid Warp can be done using two files, an ArtPro and a Grid file. In that case, the operation of the **Gridwarp** action is similar to the operation of the ArtLink action where two files are selected by the **Select File** mechanism. Using the **Select Grid** button, a fixed Grid File can be used. If no Grid File is presented, the Grid defined in the ArtPro file itself (if provided) will be used. This allows to set up grid warp in ArtPro, and let Automation Engine do the actual calculation.

Import Picture List

With **Import Picture List** it is possible to import an ASCII text file containing information on name, position and size of the bounding box of placed pictures.

These Picture List files can be created in ArtPro, using the **Export Picture List** function (see the ArtPro Manual) or from any other application, if the ASCII text file has a construction as shown in the example below. If not defined in the file, the placing is done on the box defined next to **Use Box**, centering the image on the bounding box. The other options are the same as for opening an ArtPro file.

```

ArtPro Picture File V1.0

units mm

size 1005.72826.63

Bambolina1 -1.99 -1.99 142.4 126.99 0 16 32 TrimBox
Bambolina1 -1.99 125 142.4 125 0 16 32 TrimBox Bambolina1 -1.99 250 142.4
125 0 16 32 TrimBox
Bambolina1 -1.99 375 142.4 125 0 100 60 TrimBox
Bambolina1 -1.99 500 142.4 125 0 16 32 TrimBox
Bambolina1 -1.99 625 142.4 126.99 0 16 32 TrimBox
Bambolina1 140.41 -1.99 140.41 126.99 0 16 32 TrimBox
Bambolina1 140.41 125 140.41 125 0 16 32 TrimBox
Bambolina1 140.41 250 140.41 125 0 100 60 TrimBox
Bambolina1 140.41 375 140.41 125 0 16 32 TrimBox
Bambolina1 140.41 500 140.41 125 0 16 32 TrimBox
Bambolina1 140.41 625 140.41 126.99 0 16 32 TrimBox
Bambolina1 280.83 -1.99 142.4 126.99 0 16 32 TrimBox
Bambolina1 280.83 125 142.4 125 0 100 60 TrimBox
Bambolina1 280.83 250 142.4 125 0 16 32 TrimBox
Bambolina1 280.83 375 142.4 125 0 16 32 TrimBox
Flatten 421.23 0 292.24 413.32 0 16 32 TrimBox
Flatten 421.23 413.32 292.24 413.32 0 16 32 TrimBox
Flatten 713.47 0 292.24 413.32 0 16 32 TrimBox

```

Construction of a Picture List File

1. ArtPro Picture File V1.0
2. Units
3. Size; the job size, in the unit set above.
4. Filename; when importing a picture list, Nexus will place the pictures described in the Picture List file, if the files are in the same folder as the picture list. If not, ArtPro will prompt for the specific file. If the file is found in the folder or a file has been selected by the user, the picture will be applied on the path, placing its center on the center of the bounding box.
5. Position of the bounding box; the position of the top left corner of the bounding box is defined. The first value is the vertical distance from the top left corner of the job size, the second is the horizontal distance.
6. Size of the bounding box; the next two values define the vertical and horizontal size of the bounding box.
7. Rotation; the rotation can be 0, 90, 180 or -90 degrees.
8. Picture center offset, defining where to center the image, based on the topleft corner of the bounding box.
9. Page box to be used.

Importing

This action is probably the most important action and allows to Import PostScript 3 and PDF files in to the native ArtPro file format (with full preflight and correction capabilities).

When a file is imported, typically all elements are made editable and all paths in the PostScript or PDF document are converted to internal ArtPro paths.

This allows for imported documents to be edited with the Edit or ArtPro application. However, text in the PS or PDF documents, in some cases will be converted to outlines (depending on the settings in the **Font Settings** dialog box and depending on Creator of the PS or PDF file). This means that although text can be added in Edit or ArtPro, text from the original document cannot always be modified using the text editing functionality.

Importing Screening

1. Detecting Esko screening

When importing a PS/ PDF file containing Esko screenings, these screenings are recognized and automatically converted to a PostScript screening with the correct Esko dot shape.

2. Different rulings / dot shape per separation

In ArtPro files, it is not possible to have screenings with different rulings or dot shapes per separation. However, in Esko software these screenings can be defined. When importing such a file, these screenings are split up into different ArtPro screenings. The objects using this screening are split up accordingly and put on top of each other in overprint.

Importing Normalized PDF

Normalized PDF files can be imported just like any other PDF. Keep in mind that in Normalized PDF, image links are included as EskoLinks.

1. Normalized PDF Restrictions

2. No support for Esko .ct file links

If you want to import normalized PDF back in Artpro, make sure the .ct files are embedded before generating the PDF file in the Esko Software.

Page Settings

When opening the **Import PostScript/PDF** dialog box, the **Page** button is selected.

1. Import Page

If a multiple page PS or PDF document has to be imported, this setting defines what pages should be imported. If Import Page is inactive, all pages of the PS or PDF file will be imported. This function would typically be inactive, while the page range to be imported would be determined by the Select Page action. This would actually invoke the importing action multiple times (creating a document for every page).

If the Select Page action was not used for multipage documents, all pages would be imported in one document. The positioning of the individual pages depend on the setting of the Multiple Page offset option (see below).

If Import Page is active, only one page out of the multipage document will be imported. The page that has to be imported can be entered in the Import Page field. The Import Page field can take a string input (a page label) as well as a numeric input (a page number).

For PS files, the PS import has to jump to the right page of the file before starting import. (Selected Only). However, some application define crucial information at the beginning of the PS file, on

page 1 (example: Quark defines all fonts on page 1). Therefore, we have to interpret page 1 as well (without generating output) before importing the selected page (First and Selected). In some poorly constructed PS files, it is possible that the information needed for the selected page is scattered over the pages in front of it. In that case, the option **Until Selected** should be used.

2. Ruler at top left

By default a PostScript or PDF file will be imported placing the PostScript origin (= 0,0) on the center of the ruler. If you switch the option Ruler at top left on, the top left of the page will be placed in the center of the ruler, placing the file in the lower right section. The origin in the PS file is in most cases the bottom-left corner of the page, although this is not a general rule. Example: In EPS files from Adobe Illustrator, the PS origin is the same as the origin of the ruler in the Illustrator file.

3. Multiple Page Offset

When importing a multipage PostScript or PDF file without a **Select Page** action in front, ArtPro puts all pages with the 0,0 point in the center of the ruler. This results in multipage documents being placed on top of each other. If you use the **Multiple Page Offset**, all pages are put next to each other.

4. Import Separated File as Multipage

This option should be switched off in an action list. It relates to importing a separated file, for example files using the %PLATECOLOR command to identify the separation name or older separated PostScript files.

With the Import Separation File as Multipage on, all the elements of a separated file will come in as grayscale, and the different separations will be treated as individual pages, so they will be placed on top of each other, unless the Multiple Page Offset is on.

When importing a separated file with the Import Separated File as Multipage option off, the Separated PostScript file converted to one (composite) ArtPro file with all the separations.

The individual paths coming from the different separations, will be merged together into a path with the combined color information, based on the Merge Equal Paths option (see later). Images will be recombined into one mapped picture.

Font Settings

1. Use Replacement Font instead of included Fonts

If this option is disabled the system uses the fonts in the file (if embedded). If they are not embedded the system uses the fonts in the Fonts Folder. If this option is enabled the system uses the fonts in the Fonts Folder first and uses the embedded fonts when the fonts are not available in the Fonts folder.

2. Fail if Missing Font

When this option is enabled and fonts are not available in the Fonts Folders or embedded in the PS or PDF file, an error appears on the **Importing** action when executing the action list, regardless of the Preflight Settings (Cruise Controls). Use this option when no Cruise Controls are in use.

3. Fail if Courier

Similar to **Fail if Missing Font**, an error appears if there is a Courier font being used in the document, regardless if it is a missing font or no missing font.

4. Convert to real text

When this option is enabled, the system attempts to convert the text in the document to real editable text which can be modified in Edit or ArtPro. To do so, the font must be present on the Edit or ArtPro system. Note that this option works only for files coming from certain applications. Also note that only roman characters can be converted to real text. All other characters (special, accented...) can not be kept, and will be vectorized.

5. Font Folders...

It is possible to select multiple Font folders, in which system looks for fonts that are not installed nor embedded in the PS file.

This allows importing a file with the fonts in a separate folder, without activating the fonts. Keep in mind that these fonts are treated the same way as fonts embedded in the PS file, so all text are converted to outlines. The font folder can be used for Type 1 fonts and Truetype.

To add font folders, go to the **Font Folder...**, click the + button. A dialog box appears to select a folder. Click on the correct folder and click **Choose**. The font folder is added immediately and system searches the folder and all subfolders for printer fonts.

A font folder is shown with its complete path. To stop using a font folder, simply select it from the list and click the - button. The **Reset** button deletes all the font folders from the list.

The PS default fonts toggle makes system use a set of default fonts, like Helvetica, Courier, Times, Symbol... They are stored in a folder called PS Fonts. On Windows, this folder should be on your hard disk alongside the application file. On Mac OS, this folder should be inside the bundle of the ArtPro Action List Editor.

This set cannot be modified. The system tries to use the embedded fonts first and then the font folders.

You can open a **File Naming Template** dialog box allowing to define the name for the font folder based on the job name, user name...

Color Settings

The **Color Settings** contain all separation related settings and determine how the custom colors or CMYK separations are handled in the file. All functionality is available to either keep separations as they are defined in the PostScript or PDF file, to convert them to CMYK or to merge them with another separation. This can be done either in an automated way (example: Convert All Custom Colors to CMYK) or for specific separations only which are known to be in the files that are processed.

This is discussed in the **Custom Colors** section. The list of the custom colors can be seen by clicking the **Custom Colors...** button.

1. Single Separation

This option allows to import a single separation from a PS file or a PDF file. The name of the separation is to be filled in the text field, example: "Black" or the name of any of the other separations in the file to be processed. A typical example of the use of this functionality is double burn or versioning, example: language changes, packaging versions, etc....

To summarize an action list like this would handle folders containing two or more files, preferably with a naming convention which identifies each version. For example, the English version of a document could have the extension "_eng", the Spanish version of a document could have the extension "_esp".

To simulate a plate change of black only on the press and allow for the merging of the black plate of the Spanish version with the color plates of the English version following actions would be defined:

- a. Select file: English version.
- b. Importing: CMYK of English version.
- c. Save ArtPro file: English version.
- d. Export PDF: English version.
- e. Select file: Spanish version.
- f. Importing: Black Plate only of Spanish version, the Remove Separation option will ensure all Black from the English version is made transparent.
- g. Save ArtPro File: English CMY with Spanish K.

- h. Export PDF: English CMY with Spanish K. (for example for proofing)
- i. Close File: remove the file from memory.

This allows to actually proof the merged files exactly the way they would be put on press. Since this merging takes place at the single file level, it is possible to identify potential problems (example: moved graphics or images, text running through images, etc....) early in the process and physically correct them in ArtPro, even if the original documents are not available.

Double burn action lists are supported for single page as well as multipage documents, while more than two versions are equally supported.

2. Remove Separation

When **Single Separation** is active, **Remove Separation** makes the original separation transparent while the new separation from the file is being imported.

3. Force Black Overprint

When **Force Black Overprint** is checked, 100% black in the PS or PDF file is converted to all transparent + 100% black in the ArtPro file. If **Force Black Overprint** is not checked (default), all 100% Black in the PostScript or PDF file is treated as any other color, meaning that the result depends on the **Ignore all Overprints** flag.

4. Ignore All Overprints

When **Ignore All Overprint** is not checked (default), overprint in the PostScript or PDF file is converted to transparent in the ArtPro file. If **Ignore All Overprint** is checked, overprint in the PostScript or PDF file is translated to 0% in the ArtPro file. This option can be used to take out the trapping. Note that this option is not removing the transparent strokes, but only taking off overprints. If both **Force Black Overprint** and **Ignore All Overprints** are switched off, the file is not changed and can still be modified later.

5. Keep Screenings

This option keeps the screening definitions, if they were available, in the PostScript file or PDF file.

6. Fail on Nonseparable Blendmodes

A file to be imported can be set to fail if it contains Nonseparable Blendmodes (Hue, Saturation, Color, Luminosity). These Nonseparable Blendmodes operate on a 3 component color space. This means that to calculate the resulting color, CMYK needs to be converted to 3 components, the Blend needs to be calculated and the result needs to be converted to CMYK again. However, instructions on how these conversion from one color space to another should happen cannot be given. This results in different behaviors and different color results in different applications / RIPs.

7. Fail if Transparencies in Incompatible Colorspace

If the file to be imported contains Transparencies (Blendmodes) on objects in incompatible Colorspaces (example: RGB), the file fails when this option is switched on.

8. Fail if Transparencies imported on Existing Job

If PDF 1.4 transparencies are imported on top of a job containing any objects, this option causes the job to fail.

9. Remove unused separations

This option automatically removes all separations that are defined in the file but which are not used.

10.Convert C/CV/CVC/CVU/CVP to...

All the separations in the custom color list, that have a different extension only are automatically converted to the one extension defined in this menu. So if there is a PANTONE 300 CVU, PANTONE 300 CVC and PANTONE 300 CV, they all can be converted to example: PANTONE 300 CVC, automatically.

11.Convert CMYK to Hexachrome CMYK

This option allows to replace CMYK by Hexachrome CMYK and thus allowing a smoother integration with Hexachrome compatibility.

12. Convert all custom colors / Others

With this option checked, all separations in the file are converted to process colors, example CMYK, based on the CMYK values for the separations as they are defined in the PS or PDF file. The **Others** option is used for separations that are available in the file but not described in the header. This should typically not be the case in a well constructed PS or PDF file, but does sometimes happen. This option equally makes sure that these colors, if they appear, are converted to process colors as well.

13. Custom Colors

The **Custom Colors** button enlarges the **Color Settings** menu with additional fields and a color list which allows to merge PANTONE™ colors or selectively convert PANTONE™ colors to CMYK without having to convert them all.

The additional fields that become available show a color list, a **From** and **To** field and an **Add**, **Delete** and **Reset** button. In the example, some entries has been made in the color list: Blue is merged into Reflex Blue. This is used if there are, for example, two special blue colors in a file which need to merge together in one color. It is obvious that the names of the separations have to be known before this action list is executed. If the **Convert To** color does not exist in the file, it is automatically created.

To make an entry, type the name of the “From” separation in the left field (for example Warm Red), and the name of the “To” separation in right field (example: PANTONE™ Red) and click Add. Note that this first will leave the color (indicated by the color swatch in front of the entry in the list) by default black. To change this, click on the swatch and set the desired colors as CMYK values, these values will be used for display (for example when using ArtPro) and for proofing to a CMYK device.

If a separation needs to be converted to CMYK, simply check the **Convert** button. Set **Exclude** to keep the specified separation, even if **Convert All Custom Colors** is used.

To delete an entry from the list, select the entry and click **Delete** button. A **Color** list can be saved to an external file, and loaded.

14. Color Management

Define different profiles for importing CMYK, RGB or grayscale files through Import PS. Click the **Color Management** button to open the **Import** tab in the **Color Management** dialog box.

15. Color Book

The **Color Book** for import is the same (and uses the same UI) as the ArtPro color book. Custom colors can be defined in import to be converted into the color book inks. This means, for example, convert them into hexachrome values, if a hexachrome color book was selected. The selected Color book is shown next to the **Color Book** button and an option is provided to turn it on or off.

Blends, Shadings and Gradations

Use this to define how gradients, blends and shadings should be handled on PS or PDF import. It is advised to switch **Recognize Gradations** and **Convert Shadings into gradations** on, to recognize gradations from most popular applications.

1. Recognize gradations (PS Only)

In a PostScript file, gradations can be described by a large number of paths or by a TIFF. In most cases, these can be recognized and replaced by ArtPro gradations, if the option is checked (default).

With this option, gradations from Freehand and Illustrator are automatically converted into ArtPro gradations or multistep gradations. Freehand and Illustrator vignettes which are not linear are converted into multistep vignettes with midpoints. Gradations from QuarkXPress are converted into ArtPro linear or multistep gradations.

If the option is not checked, depending on how the gradation was defined in the PostScript file, it is imported as a series of paths, or it is converted to a TIFF image.

2. Convert Shadings into Gradations

PostScript Level 3 and PDF 1.3 and above often use the shading command to define gradations. With this option, most shadings are converted into AP gradations or Multistep gradations. If this option is switched off, an image is generated.

3. Convert blends into CT at

When blends are used in a document, they normally come in as a series of paths, with changing paint styles. These blends are very difficult to select, modify,... Especially trapping becomes very difficult. Therefore, usually blends are flattened into images, with one single path and one single paint style.

With this option on, blends are even converted to internal ArtPro CT's automatically. This means the pictures are embedded in the ArtPro document in a '.ct' file. The resolution for these images can be set and the unit can be changed from pixels per inch to pixels per cm.

4. Resolution for Shadings

This option is for shadings that cannot be converted to ArtPro gradations (example: mesh gradations). All these gradations are automatically converted and extracted as a TIFF image on disk. The resolution for those TIFFs can be set here. Shading conversion to images is a very time consuming calculation. Therefore, the default resolution is set to 50ppi, but a higher resolution is recommended in some cases, depending on the shape of the object. Check the objects carefully.

Images generated during PS/PDF import always generate a mapped image.

5. Interpolate Shadings to 300ppi minimum

When working on certified or using **Cruise Controls**, it is very likely that lowres images generate a warning or error. This would also happen for images generated when converting a shading.

To avoid this, the Resolution for Shadings can be set to a higher resolution (300ppi), but the sampling of shadings at a high resolution is very slow and time consuming.

The option **Interpolate Shadings to 300ppi minimum** samples at a low resolution and creates a higher resolution image from it. The lowres images are interpolated to a resolution higher than 300ppi. The resolution is a multiple of the sample resolution, example: if the sample resolution is 70ppi, the image after interpolation is $70 \times 5 = 350$ ppi. This results in a high resolution image, generated at higher speed.

Links

This determines how links are handled upon import. With links we refer to either OPI links (%ALDImage commands) in PS or PDF files or references to EPS images in PS documents or EskoLinks.

- Placing or importing

The different pictures that could be used in the original document are TIFF (with or without clipping path), DCS, JPEG, PDF, EPS lineart, regular EPS image (with or without clipping path), EPS duotone, tritones..., LoRes or HiRes images from OPI systems...

Most of these images can be brought into the ArtPro file in different ways:

Most of them can either be placed or imported, depending on the kind of image and the desired result. Example: When EPS images are imported, EPS is converted into a TIFF on disk and the TIFF is placed in the ArtPro file. The big advantage is that the TIFF image can be mapped, which gives us benefit for trapping, previewing, ink swapping, ink coverage... Therefore in ArtPro it is recommended to work with mapped images (TIFF, JPG, Photoshop PSD and DCS) all the time.

Because of functionality, it is always recommended to map images upon import except for specific cases like copydot images, PDF images, etc. When images are placed and not mapped, they remain as they are (example: compression, content, etc.) and appear identically in the output as they have been placed. This method is typically recommended for copydot images. Note that only TIFF images, JPG images, PhotoShop PSD and PhotoShop DCS (without vector data) files can be mapped, hence EPS images (any type, also DCS) can ONLY be mapped if they have been imported and thus converted to a TIFF image on disk. Copydot images can equally be imported, (example: DCS copydot – 1 bit data) but this is not recommended since the image would be decompressed while writing to a TIFF file hence produce extremely large files.

Note:

TIFF images are not imported but only placed (and possibly mapped). Importing is only relevant for PostScript images (for example: EPS images) which are then processed through the Import action separately if “importing” has been set.

Summarized:

TIFF: A TIFF image can only be placed, not imported. A placed TIFF file can be mapped and thus trapped automatically.

EPS image: An EPS image can be placed or imported. A placed EPS can not be mapped. When it is imported, the pixels are converted to TIFF data. If the image has a clipping path, it can be converted to an internal path with the image inside.

EPS graphics: If an EPS graphics file, example: a logo or chart, is placed, it can be transformed (rotated, scaled,...) but not edited (it is treated as one object). By importing that same logo, all the vectors are converted into internal paths which can be edited and trapped.

EPS duotones (+ monotonnes, tritonnes and quadtonnes): Can be placed or imported. An imported duotone is correctly converted to a mapped picture. The condition here is that they should be created with Adobe Photoshop 5 or later. Duotones from older versions of Photoshop are extracted as CMYK images when imported, so these should be placed instead.

DCS: A DCS image is in fact a pre-separated EPS. So the same rules apply. Importing makes it a TIFF, which allows us to map, preview, trap,...But a DCS cannot be imported directly (example: dropped in a hot folder to be imported), it has to be used in a PS or PDF file. In other words, if the DCS image is placed in a (Illustrator, XPress,...) document that is saved out as a PS or PDF it can be imported and mapped correctly.

OPI images (Open Prepress Interface): When OPI low resolution EPS sample images are used in a document that needs to be imported, the images always have to be placed, if they are not automatically replaced by their high resolution version upon import (see the OPI section). If they are imported, completely new (low resolution) TIFF images which is not recognized and replaced by the OPI server are created, resulting in a low resolution output. If low resolution TIFF images are used, they can typically (for most OPI servers) mapped and further worked with (example: for trapping).

- Omit/ Embed

There are three ways to put an image in a PS or PDF file.

- ‘OPI’: The image is omitted (TIFF, EPS, DCS,...). This means the original picture is not included in the PS file, only the link is available. The link is a path, indicating the place where the image

can be found on the hard disk or anywhere else in the network (example: on the file server). These PS/PDF files are obviously very small. They are quick to write and copy.

In the **Links** tab of PS/PDF Import, they respond to the settings under Omit. In the picture list, under display links, these images have **OPI:** in front of the name.

- ‘EPS’: The image is embedded. The complete original picture as well as the link is included in the PS file. This means in ArtPro we can choose which to use. These are usually EPS. They respond to the settings under Embed. These images can be recognized in the picture list, by the ‘EPS:’ in front of the name. These PS/PDF files are larger.
- The image can be completely embedded without the link. This is usually with TIFF images. (Freehand is an exception. All TIFFs are always embedded in the PS files, with the link). Because no link or name is available, ArtPro can not display these images in the picture list. No link settings can be defined. These images are extracted to TIFFs on disk during import.
- EskoLink: These are linked images in a Normalized PDF. These act in the same way as OPI links but use a different syntax. EskoLinks allow to define different density per separation and allows the definition of separation remapping.

How to handle links on Import:

1. Use Link and Place All

The original file referred to by the link is searched and this file is placed. The embedded information (if present) is ignored. The file is searched at its original location, unless the link is re-routed in the **Display links...** part of the dialog box or if rerouting is defined in the OPI settings. If the image is not found, the embedded information is used. If it is not available, the result is an empty path.

This setting is mostly used for TIFFs and JPEGs, omitted from the file, lores OPI images, single file DCS or duotones from Photoshop (version 5 or below).

2. Use Link and Import EPS

The file referred to by the link is searched and this file is used. The embedded information (if present) is ignored. If the file referred to is an EPS/DCS, it is imported, converting lineart to ArtPro native lineart and extracting images to disk according to the settings in the Images settings. If the file referred to is not an EPS or DCS (example: TIFF), it is placed.

The file is searched at its original location, unless re-routing is defined in the OPI settings. If the image is not found, the embedded information is used. If it is not available, the result is an empty path.

This setting has to be used for DCS and duotones (if you want them converted to mapped TIFFs). It is also used if there is an OPI swap. In general, most OPI images have the same format (example: EPS). ArtPro is rerouted to the hires. If that hires is an EPS.duotone/ DCS it is imported and converted to a TIFF. In all other cases it is placed. This option is used for EPS, when they are either omitted from the PS file or embedded and the original has been modified since.

3. Ignore All Links

The link is ignored and only the embedded information (if present) is used. This embedded information is imported, converting lineart to ArtPro native lineart and extracting images to disk according to the settings in the Image settings.

This setting is used for embedded (hires) EPS and TIFFs. There is no difference in result whether you apply use link and import or ignore all links. However, if the links are ignored, we need not worry about ArtPro finding the original on disk (is the link correct?, does it need re-routing?,...).

When importing a PS/PDF file, 1, 2 or 3 needs to be chosen for all ‘omitted’ images (the ones in the list with OPI in front of the name) and also for all the embedded images (the ones with EPS).

4. Extensions

It is possible to override the above option settings with the **Extensions** fields. Consider the example where all EPS images are linked and imported. Depending on the types of EPS pictures, it may be considered to make an exception to this setting, example: one would not want to equally import copydot EPS pictures. To avoid this, but still allow for other EPS pictures to be imported and hence trapped, it is possible to work with an extension for the copydot images. If the copydot images would be defined as “.cpd” (as an example), then typing “.cpd” in the **Extensions** field next to **Use Link and Place All** would place ONLY the images with a .cpd extension, while importing the others.

It is equally possible to add multiple entries in the **Extensions** fields separated by a space, example: “.cpd .main .duotone” etc....

5. Into Opacity: B/W, Gray, 5th Channel, Invert

With the Into Opacity options, you can define that linked TIFFs in the PostScript file should be put into the opacity channel, and thus be made transparent.

1. B/W

If the **B/W** option is checked, all black&white bitmaps referred to by link are put into opacity with a solid black image channel. So, all black pixels remain black and all white pixels become transparent.

2. Gray

If the **Gray** option is on, all grayscales, referred to by link, will be put into opacity, with a solid black image channel. The resulting CT will be black for the percentage of the grayscale, and transparent for the remaining percentage.

3. 5th Channel

If this option is checked, a 5 channel TIFF referred to by link, results in a four channel internal CT and the 5th channel is used as opacity channel. The areas of the Photoshop alpha channel that were 100% masked, result in 100% image in ArtPro. The areas that were 0% masked, become completely transparent. Any value in between (example: soft edges on the mask) result in a partial transparency (see pictures). If the option **Invert** is on, the opacity channel is inverted, using the **Invert** button in the Mapped Picture paint style. This way, the alpha channel can be made up in two ways, positive or negative. With **Into Opacity** it is also possible to work with extensions (see above): Example: B/W is off, gray is on, extension = .gr This means all B/W is not affected and all grayscales with an extension .gr is put into opacity. All other grayscales are not affected.

OPI

This is used for rerouting linked images. The rerouting could happen based on the name of the image or based on the path of the images. There are mainly two reasons why rerouting may need to take place:

- Low resolution images need to be replaced by (rerouted to) high resolution images.
- Missing images need to be rerouted to their correct (current) location.

Note: Re-routing can only be done when OPI links are available for the images and the **Use Link and Place All** or **Use Link and Import EPS** options are set.

1. Perform OPI swap

This option enables the rerouting as defined in this menu. If the option has not been set, all settings are ignored and no rerouting takes place when OPI links are used by the system.

2. Sample Extension / Hires Extension

The **Sample Extension** and **Hires Extension** fields are used to swap the extension of the image that is linked. This is typically used for OPI servers which are set to add an extension to or replace an existing extension with another when low resolution images are generated from high resolution images.

Every extension that is entered in the **Sample Extension** field is cut off from the linked file before it is actually searched, while every extension that is entered in the **Hires Extension** field is added to the linked file before it is actually searched.

If the **Sample Extension** field contains “.lay” and the **Hires Extension** field is empty and the image that is linked is “image.lay”, then the system actually searches for “image”, the “.lay” extension is cut off.

If the **Sample Extension** field contains “.lr” and the **Hires Extension** field contains “.hr”, and the image that is linked is “image.lr”, then the system actually searches for “image.hr”, the “.lr” extension is cut off and the “.hr” extension is added.

It is equally possible to enter multiple extensions in the fields, separated by spaces. If the extensions of the file referred to in the link matches an extension entered in the **Sample Extension** field, the extension is cut off and when present, it is replaced by the extension in the **Hires Extension** field at the same position. Extensions that only need to be cut off and not replaced should be at the end of the list.

Example:

- Sample Ext. field: “.lr .low .lay”
- Hires Ext. field: “.hr .high”

The result of this operation is:

- “.lr” is replaced by “.hr”
- “.low” is replace by “.high”
- “.lay” is cut off

3. Sample Folder / Hires Folder

Similar to the **Sample Extension** and **Hires Extension** fields, the **Sample Folder** and **Hires Folder** fields are used to reroute the system to search for an image in a folder different from the folder determined in the path of the OPI link. In OPI swapping environments, this is typically used for OPI systems that are set to generate low resolution images of high resolution images with the same name but in a different folder.

Suppose all low resolution images are stored in a folder “ForLayoutUse” and all high resolution images are stored in the folder “Originals”, both under the same subdirectory, then “ForLayoutUse” would be filled in the **Sample Folder** field, while “Originals” would be filled in the **Hires Folder** field.

When the OPI link in the file is then: E:/CustomerX/Images/ForLayoutUse/Image, the system will actually search for and use: E:/CustomerX/Images/Originals/Image

It is also possible that one path is changed twice, for example: by putting multiple entries in the **Sample Folder** or **Hires Folder** fields separated by spaces. This implies that there should be no spaces in folder names that need to be replaced.

For example, if the **Sample Folder** contains “LR Customer” and the **Hires Folder** contains “HR”, then the OPI path E:/JOBNR/ImagesCustomer/LR/Image is replaced by E:/JOBNR/Images/HR/Image.

In the above examples, it is clear that all OPI paths have been generated by a system on Windows. The above would work for cases where the front-end station creating the PostScript or PDF file with OPI links is based on Windows and has the same mount point and drive letter for the image

server as the Automation Engine. As in the first example, for both Automation Engine and the front-end Windows machine, E:/ is the network drive for the image server entered at the mount point where the CustomerX folder is the first available subfolder.

It is clear that in a day to day production environment, the above situation is mostly hypothetical. Indeed, mostly Macs are being used as the front-end, while if even Windows machines would be used, they would often have a different mount point to the image server than Automation Engine.

In these cases, the Folder Mapping mechanism can be used to remap the drives for the OPI paths.

An example clearly illustrates this. Suppose the Image Server has following structure for the images that are made available to the network: E:/Server/Images/...

Suppose that the Images folder has been made available to the Mac environment as Image Server and this volume gives access to the content of the Images folder.

Suppose that the Server folder has been made available to Automation Engine and this folder is mounted as a network drive under G:/, hence this G:/ gives access to the contents of the Server directory.

If now an OPI link on the Mac is defined as: ImageServer:image then Automation Engine actually needs to find: G:/Images/image To achieve this, we would enter "Image Server:" in the **Client** field, while entering "G:/Images/" in the **Server** field. Clicking **Add** appends this definition to the list. It is possible to have multiple OPI folder mappings active at the same time. The system equally automatically converts between Macintosh notation of subfolders and Windows notation of subfolders (: and /).

4. Root Folder

If a linked file cannot be found at its original location or at the location it is supposed to be found after re-routing, the file will be searched for in the folder or volume that is defined as Root folder. Nexus will not only search in the Root Folder but equally in all subfolders. When a Root Folder has been defined, it is displayed underneath the Root Folder button. To remove the Root Folder setting, click on the Root Folder button while holding the SHIFT key.

Note that Nexus will use the first instance of the file that it finds, which may be potentially dangerous in production environments. Suppose the image to be found is called "logo.TIFF" and the logo is not found at its location, Nexus checks the Root Folder and its subfolders.

Suppose in the production environment, two customer directories have been set underneath the root folder directory: RootFolder/CustomerA/Images/logo.TIFF and RootFolder/CustomerB/Images/logo.TIFF.

Even if the CustomerB logo needs to be used, the system always picks the first instance of the name that it finds, in this case the logo of CustomerA! It is also not advisable (for speed reasons) to select a complete disk or a very big image tree as root folder.

5. Exclude Folders from Search

When searching through the Root Folder, folders names entered in this field are not searched. This can be used when all images can be found on the same volume, with the possibility to exclude all folders containing low resolution files, example: all low resolution files may always be put in folders called ForLayoutUse.

Image Settings

1. Vectorise Bitmaps: Always/ Most/ Never

When importing a PostScript or PDF file containing 1-bit images or a 1-bit image is detected via OPI, this setting defines how to handle those.

When set to **Never**, the 1-bit images are placed as images. When set to **Always**, 1-bit images are always vectorised. When set to **Most**, the 1-bit images containing more than 5mio pixels are

placed, images smaller than 5mio pixels are vectorised. For example: This option can be used to vectorise small logos but leave large scans such as copydot files, unmodified.

When vectorizing a Bitmap, the **Precision** can be set to **Exact** (no reduction factor and straighten margin is applied), **Fine**, **Normal** and **Coarse**. For more information, refer the ArtPro Manual.

If a bitmap is placed as an image, the bitmap is converted into a Mapped TIFF with image channels and an opacity channel. The image pixels constitute the image and the white ones are transparent to the background.

Both a vectorised TIFF and a bitmap mapped TIFF are colored exactly like in the original document. When importing a PDF file, the result is always a grayscale TIFF. In most cases a vectorised bitmap is smaller and quicker to handle than a mapped TIFF.

2. Make Images Internal

After importing, all TIFF and JPG pictures (placed in the PostScript or PDF file or extracted by Automation Engine) can be automatically converted into Internal Images in ArtPro, by activating the option **Make Images Internal**. It is only used in exceptional cases. For more information on internal images, see Chapter 5: Contone of the ArtPro Manual.

3. Extract included images

If an image has to be extracted on disk during PS/PDF import (see also Links), a file is saved to disk. During Import, identical images are detected and deleted. This means that, if Automation Engine extracts images from the PS or PDF file and writes them to disk, these images are compared and if the same image is used several times, only one of them is kept. The comparison is done after writing the TIFFs and before making them internal (if necessary). This comparing can take some time but results in faster editing and smaller file sizes.

The **File Name** field in this dialog box is set to define how the extracted images of the PS or PDF file is named and to what location they are saved. The path name in this case determines where they are saved. If empty they are saved to the current job folder. It is advisable to use the document page number in the Template, as otherwise some files can be overwritten with multipage PS import and Select Page Action.

4. Resolution / Detect Image Resolutions

When images are extracted during the import of a PDF or PS file and the **Detect Image Resolutions** in PostScript / PDF Import option is on, Automation Engine checks the internal resolution of the image.

If this resolution is an integer (example: 360 ppi), the image saved uses this resolution and is placed in the ArtPro file at 100% scaling. If the resolution is not an integer (example: 243,6 ppi), or if the **Detect Image Resolutions** in PostScript / PDF Import option is off, the resolution as defined in the Import PS/PDF dialog is used and the scaling is adjusted (in this case, the 300 ppi image is placed at a scaling of 123.56%).

5. Use Original Name

When the original name of an image is known, the extracted file is saved with the original name if this option is checked. If this option is off or the original name is not known, the user is prompted for a name but by default the name of the PostScript file followed by .# and a number is used.

6. Use Photoshop clipping path

If a PS file is imported, in which Photoshop TIFF files are placed, containing a clipping path, this clipping path can be converted to ArtPro line art, with the TIFF file placed inside it's path. This option only affects TIFF files coming in with OPI. Clipping paths on EPS images are always used. Another difference is that during Import PS, all paths are imported as straight line vectors, while when using the **Use external clipping path** option when placing a picture, the curves are kept as curves.

7. Recognize TIFF Channels

When OPI linked pictures have to be placed, Automation Engine tries to recognize any spot colors when placing a TIFF file and if the **Recognize TIFF Channels** is switched on.

A TIFF file can contain channels indicating masked areas (type1), channels indicating selected areas (type2) and channels indicating spot colors (type3).

Channels of type 1 and 2 are treated as alpha channels, thus put into the opacity channel. If the TIFF contains more than one alpha channel, only the first is used and all others are ignored, though the separation are generated based on the channel names present in the TIFF. These separations have CMYK value 0, 0, 0, 100 and only reflect the number of channels in the TIFF.

If spot color channels are found, a separation is created with the same name as the TIFF file channel.

The color definition (CMYK representation) for this spot color is first searched in the ink book. If not found there, the color information from the TIFF file is used.

If ArtPro encounters an unknown color space, a new separation is created with the channel name and with CMYK representation 0, 0, 0, 100.

General Settings

1. Clip Text

Text that was clipped in another application is clipped in ArtPro as well after Import PostScript/PDF. In order to make this possible, though, clipped text is vectorised in ArtPro and thus no longer available as real text. Normal text (not clipped) is not affected by this option, which is checked by default.

2. Merge Equal Paths

If **Merge Equal Paths** is unchecked, all paths are converted to ArtPro in the normal way. If checked, two or more identical paths on top of each other are merged into one single path with the combined color information. This option is automatically enabled for separated files.

3. Recognize Illustrator Layers / Merge Equal Layers

When checked, all the original layers from imported Illustrator (<9) EPS files are recreated in Automation Engine and all the elements are put back on the correct layers. All PDF files containing PDF 1.5 layer information, also generate the correct layers in Automation Engine. Make sure to generate PDF 1.5 output from certain applications (example: Illustrator CS) to include all layer information.

If unchecked, all vectors are put on one single layer called PostScript Import. This option can be used with or without **Merge Equal Layers**. If Automation Engine finds multiple layers with the same name, those layers are merged into one (example: An EPS with a 'Dielines' layer is imported on an ArtPro template with 'Dielines' in it). It is possible in Automation Engine to have several layers with the same name.

4. Ignore Page Commands

In a normal PostScript file, the document size is defined at two places: the bounding box, in the header of the file and page commands, later in the PostScript. In most cases, the Page Commands are more exact than the bounding box. For files coming from certain versions of Barco systems, the Page Commands are rounded off, while the bounding box is exact. In those cases, you can choose to **Ignore Page Commands** and use the PS bounding box instead.

5. Keep Strokes

When checked, all strokes found in the PostScript or PDF file are imported as ArtPro Strokes. If the option is unchecked, the strokes are converted to regular ArtPro paths.

6. Memory

This parameter indicates how much Virtual Memory is allocated to the Import function, in order to store its variables, key words, dictionaries, etc. Generally, 20000 kB (default) is sufficient to perform the function. However, when the error message VMError appears, the value should be increased.

Certified PDF

When importing any file type other than PDF, the Sessions log indicates that no input profile was applied and the session begins AFTER the import.

- Importing non-Certified PDF

When importing a non-Certified PDF file and Certified Import is enabled, Automation Engine Certifies the PDF file and import the Certified PDF instead of the original. Unless **Overwrite input file on re-preflight** is checked, a new certified PDF is saved (according to the Template settings) and this certified PDF is imported. This is saved as a separate session.

- Importing Certified PDF

1. Re-Preflight if profile does not match: Selected

The job is re-preflighted if the input profile and the embedded profile do not match. If they match, the session log indicates that both profiles are identical. When comparing two profiles, their content is checked and not just the name.

2. Re-Preflight if profile does not match: Unselected

The Certified PDF is imported without re-preflighting it. The Session window indicates the profile it was originally certified by.

- Re-preflight

When Re-preflighting, Automation Engine creates a new session, preflight the file and close the session. The file is saved according to the File Naming Template settings.

1. Overwrite input file on re-preflight

When checked, there is no new PDF file created by Automation Engine. Instead, it replaces the original PDF file.

2. Get all bounding boxes for preflight report

When checked, this option forces Automation Engine to obtain bounding boxes for the last preflight report. If activated, it gets full information for every report item along with bounding box of the area where the error/ warning occurred. This causes an additional preflight of the input document and consume additional time.

Cruise Controls (Preflight Settings)

This dialog box is used to set a preflight profile.

Several controls are available which are split up in four categories: Lineart Color Controls, Image Controls, Object Controls and Text Controls. These controls are found in the left pane. Clicking the triangle in front of each of the controls will show the available controls for that category.

In the right window the selected controls, for example preflight settings, are shown, each with an either green, orange or red circle in front. The color determines the severity of the control: red refers to error level, orange to warning level and green to note level.

To change the severity level for a preflight item, click on the preflight item and select the appropriate severity level from the dropdown menu. To add a preflight item to the list, double-click on the control in the left window.

To remove a preflight item from the list, select “Delete Control” from the contextual menu, or select it and press Cmd - Backspace.

The **Save Controls**, **Clear Controls** and **Load Controls** buttons allow to respectively save all the selected controls with their severity level to a file, clear the list or load controls and their severity level from a previously saved file.

Note that a job is not held based on preflight, in the case of a warning level or error level there is only an exclamation mark in front of the job in the Job list indicating that the error button should be clicked to verify the error log information for that specific job. However, even if there was a preflight error, the job still continues to run until completed!

When preflight has been performed for a file and the file has been saved it contains a complete Cruise Report (preflight report). Using ArtPro it is possible to view this report and actually use it to navigate to the appropriate object in the file that triggered that specific entry in the Cruise Report.

By switching on the **Save Report** option, a preflight report can be saved in PDF format to the location specified in the Template. Using an E-mail action, this report can be emailed as an attachment, using a Select File action to select the PDF file only.

Following are the different **Cruise Controls** or preflight settings which are available:

1. Info

With these settings, information about the imported file can be found:

Filename: The original name of the imported file

Format: What file format the imported file has (as found in the header of the file)

Title: Title of the document of which the imported file was created

Creator: Application in which the imported file was created

Date: Date on which the imported file was created

Type: What type of file was imported (Comp. / Separated)

2. Lineart Color

* Overprint: Overprint defined in the imported file is detected. Overprints in a document could indicate that the file may already contain some trapping.

* Custom Colors: Using this control, get notified when and where a custom color is used in the imported file.

* Ink Coverage: Ink coverage can be checked not to exceed a given percentage, example: 300%. This is used to find a common error, like the use of registration color, where simple black was needed.

3. Image

This category contains all controls to avoid most common errors in images.

* Resolution: if the resolution of all images used in the imported file is bigger or smaller than 72, 300, 1000 ppi or a freely entered value, a warning is generated. This can point out low resolution images or images with an excessive resolution. Very important to know here is that ArtPro takes the scale factor of an image into account. The real resolution is the scan resolution divided by the scale factor in % and multiplied by 100. The real resolution of a 300 ppi image scaled to 200% is 150ppi. The real resolution of a 72 ppi image scaled to 20% is 360ppi.

* Color: Images in the specified color space (RGB, CMYK, LAB, Separated or grayscales) can be found. This could also be very useful because ArtPro does not support any RGB images.

* Links: With the Link settings, OPI or EPS comments can be checked:

OPI Comments: indicates if OPI comments were encountered in the imported file.

OPI Recognized: indicates if the found OPI comments were used.

EPS Linked: indicates if linked EPS files were encountered in the imported file.

EPS Imported: indicates if these found EPS files were imported (instead of being placed).

4. Object Kind

Object controls offer the possibility to point out objects smaller or bigger than a given value. For example, to avoid hair lines or small text printed in flexography.

* Text: text smaller or bigger than 12pt, 6pt or any entered value can be looked up and reported.

* Strokes: strokes can be checked for size to be bigger or smaller than 2 mm, 1 mm, 0.5 mm or a user defined value.

5. Text

Text controls contains three criteria for font control:

* Font Missing: A criteria can be set to search for missing fonts during import. ArtPro reports if no font information was found at all, meaning the printer as well as screen fonts were missing.

* Font Name: If a font, used in the imported PS or PDF file matches the name entered here, a warning is displayed. This way, the system can notify the operator when for example, a courier is used in a file, for this is often a replacement font.

In the definition of the font name, a wild card can be used. If you enter `Cour*`, all font names starting with "cour" are found, including courier bold, courier italic, etc. The name is not case sensitive.

To add controls to the list, double click on the control or select it and click on the button with the arrow to the right. To remove controls from the list, select it and click on the button with the arrow to the left.

* TrueType: checks if TrueType fonts were used in the imported file.

6. Transparency

Transparency controls contains three criteria:

* Blend Modes: A criteria can be set to search for used blend modes.

* Opacity: If an opacity higher, lower or equal to the entered value is encountered. If set to "Lower than 100%" is set, all objects with an opacity other than 100% trigger the control.

* Opacity Masks: checks if Opacity Masks were used in the imported file.

7. Preflight Report

The **Preflight Report** allow to get information from the PDF Preflight Reports. This of course only applies on importing PDF files that were preflighted. The Cruise Controls that can be used are the three levels of Preflight report: fixes, warnings and errors.

Please note that Bounding Boxes for preflight entries are only available if the file was preflighted or re-preflighted during import, with the **Get all bounding boxes for preflight report** option selected (see above).

Ink Coverage

The **Ink Coverage** action calculates the surface ink coverage of the document and saves the information as a text file on disk. The **Template Editor** button again determines the naming and location of the file.

Per separation in the job, a value is given as a percentage and as a surface coverage (example: square mm). Equally a total surface value is given.

Note:

In the case of pictures, ink coverage only works when there are mapped pictures or when placed ArtPro pictures contain a separated preview. In other cases an average value is used. When using ArtPro, this average value can be adapted in the paint style. The only relevant setting in this action is the samples per inch that is taken which determines the accuracy of the ink coverage. After running the job, the results can be seen in the Job Editor.

Ink Coverage Check

Ink Coverage Check verifies if the amount of ink does not exceed the **Ink Limit**.

Special separations such as opaque separations, die or transparent separations can be excluded. The page box to be used can be set, as well as the resolution at which the job needs to be rasterized for checking.

Join

The Join Paths function joins the two nearest open points of the selected paths with a straight line. This is repeated until no open points are left.

Insert ArtPro file

This action inserts an existing ArtPro file or picture into the file which is currently being processed.

The file that is inserted can either be a fixed file (name is selected in the **Insert File** window) or a file which has been determined by a previous **Select File** action (in this case the file to be placed would be selected from a folder which is dropped in the hot folder) or a file which has been generated elsewhere in the action list. An example of the last case is the result of a **Difference** action that needs to be inserted (place as picture) in a new ArtPro file.

The insertion is done on the bottom right quadrant of the current position of the ruler, determined by the position of the ruler in the **Importing** action, the position of the ruler in an opened ArtPro file or the position of the ruler after the **Move Ruler** action.

Browse

Allows to physically select a fixed file from the system. Click **Browse** to open the **Browse** menu. After selection the name of the file with its full path is displayed underneath the **Browse** button.

When no file is selected, the file to be inserted needs to be determined by a previous **Select File** action in the action list.

Overwrite separation angles and frequency

When inserting an existing ArtPro file, the system loads colors correctly. If separations were different, Automation Engine adds the appropriate separations in the separation table. When there are identical separations, this option overwrites the angles and frequency of the current separations with the values of the file that is being inserted.

Merge Layers with Same Name

The ArtPro file format supports layering. When this option is set, content of layers in the inserted ArtPro file is merged with content of layers in the current file, if they have the same name.

Ignore Screening Errors

Since Extended Screenings are stored differently in newer version of the application, opening a file from older version may result in a conversion of the screening. If this conversion fails, the job fails, unless the **Ignore Screening Errors** option is selected. As a conversion error means the screening settings are changed, it is advised to use this option only for action lists that do not have any screened output or if screening settings are defined in the action list itself.

Place as picture using

This option is used for placing image files or treating an ArtPro file as a picture (which would mean the content of the ArtPro file is not available, for example, for trapping or internal modifications).

Supported picture types include EPS, DCS, TIFF, PSD, ArtPro, PS, PDF and TIFF/IT.

When the **Place as picture using** option is selected, the drop-down menus present beside this are enabled which can be used when inserting a file with it to place the selected file as a picture inside the open file.

At the time of inserting, a rectangle is created of the same size as the border defined in this popup menu, in which the file is placed as a picture. The **Align picture on** option can be used to define on what border the placed file should be positioned. The top left corner of the selected border is placed in the center of the ruler.

For example: combination of both options allow you to place the inserted file positioned on its trim box (top left corner of the trim box in the center of the ruler), but place it inside its bleed box.

Place Single-Separation ArtPro File in Single Separation

This option defines how to handle inserting ArtPro files with only one separation. If the option is switched off, objects in this AP file with 100% of the single separation are considered to be registration color. Thus they get 100% of every ink in the ArtPro file after insertion. If this option is selected, the file is inserted in one single separation instead.

Recognize TIFF Channel

The **Recognize TIFF Channel** Option allows to recognize TIFF Channels when inserting a TIFF File.

A TIFF file can contain channels indicating masked areas (type1), channels indicating selected areas (type2) and channels indicating spot colors (type3).

Channels of type 1 and 2 are treated as alpha channels and put into the opacity channel. If the TIFF contains more than one alpha channel, only the first is used and all others are ignored, though the separations are generated based on the channel names present in the TIFF. These separations have CMYK value 0, 0, 0, 100, and only reflect the number of channels in the TIFF.

If spot color channels are found, a separation is created with the same name as the TIFF file channel. The color definition (CMYK representation) for this spot color is first searched in the ink book. If not found there, the color information from the TIFF file is used.

If Automation Engine encounters an unknown color space, a new separation is created with the channel name and with CMYK representation 0, 0, 0, 100.

Layer Versioning

The **Layer versioning** action allows to create multiple ArtPro files from a single file, based on its layers. The **Base Layers** are used for every ArtPro file, while every **Version Layers** results in a separate ArtPro file. This can be used, for example, to split up a file with layers “Base graphic”, “Base Text” “English” “French” “German”, into 3 files: everyone having the Base layers and one with the French, one with the German and one with the English layer.

Base Layers

The **Base Layers** are the layers that are used for every ArtPro file. These can be defined by name or by range. When defining a base layer by name, it can be added to the list by clicking the + button. Layers can be removed from the **Base Layers** list by selecting it and clicking the - button. The name of a layer can be defined by doubleclicking it in the list. When defining the name of the **Base Layers**, wildcards (*) can be used.

When defining the range, you can, for example, define that layer 1 - 3 are base layers.

Version Layers

The **Version Layers** can be defined by the user or it can be set to use all non base layers as version layers.

When defining the **Version Layers**, you can add and remove versions. The Template key for a version can be used in the **File Naming Template** in order to define the names of the resulting ArtPro File. For every version, you can define what layers need to be used. If the option is selected and the entered Layer is not found in the original ArtPro File, the action fails.

The option **Ignore Case In Layer Names** defines whether the definition of layer names should be case sensitive or not. If other layers than the ones defined in the versions and base layers are found, the action fails if the option **Fail if extra Layers found** is selected. This option is disabled when the **All Non Base Layers** option is selected for the **Version Layers**.

When saving the various ArtPro files, you can define if the file needs to be saved with preview (and its resolution) and if the Certified PDF information should be included.

Make Compound

With **Make Compound** all the selected paths will become one single object.

Where paths are overlapping, a look-through window to the background is created.

For a more detailed description, please refer to the ArtPro manual.

Make Thumbnail

Make Thumbnail is an action to create a TIFF or JPEG preview of an ArtPro file.

Resolution can be set up to 120 dpi. Noise for gradation can be set. There is no oversampling.

Compression can be set to TIFF, High Quality JPEG, Medium Quality JPEG or High Compression JPEG. The Template can be used to define name and location of the thumbnail.

Modify Layers

The **Modify Layers** action allows to change the settings of layers. In the Layer fields you can enter any number of layers, using full name or a wildcard (*).

If the **Matching** option is selected, all layers matching the entered string are affected. If it is not selected, all layers NOT matching the string are affected. This allows, for example, to select all layers except “varnish”

With the checkboxes you can set the layer(s) to (in)visible, (un)locked or (non-)printing.

If a layer name is not found, this line is ignored. However, if the “Abandon” option at the end is on, the rest of the action list is abandoned.

Layers with hidden, locked or non-printing checks, stay like that until other **Modify Layers** actions remove check or until the action list closes the ArtPro file.

Modify Paint

The Modify Paint action allows to modify the paint (in percentage of a given separation) of objects or selected objects, change the overprint, or modify the start and end color of gradations.

Modify Screen

This action will apply user defined screening parameters to all selected objects.

The screening parameters in this case are PostScript based. The **Dot Code** field allows you to enter a PostScript string to execute your own personalized screening.

Use a Screening set allows to select a screening set as saved in ArtPro based on the name in the **Screen** field. The file can be selected using the **Browse** button. See the ArtPro documentation for a full description.

Modify Separations

The Modify Separations allows to change settings for separations.

Rename C/U/CV/ CVC/CVU/CVP to

This allows to convert different CVC flavours to the same extension.

Delete CMYK Flags

This allows to remove CMYK flags. These flags indicate what separations should be handled as CMYK (for placing pictures, importing PS files, etc.) even if they are replaced by spot colors.

Separation list

The separations are selected by name. Wildcards can be used, example: “Die*” selects all separations whose name starts with Die. Separations are added by clicking the + button.

Rename

A separation can be renamed by entering the original separation name and the new separation name and check “Rename”. Note that renaming is NOT the same as remapping !

Create

A separation can be created by entering the separation name and check “Create”. In the example, a separation PANTONE Red will be created.

Set to

1. Remap

A separation can be remapped by entering the original separation name and the “target” separation. If the **Target** separation does not exist, the job will fail.

2. Convert to CMYK

A separation can be converted to CMYK by entering the name of the separation and entering “CMYK” as target.

3. 0 / Trans

A separation can be set to knockout by entering “0”. All objects containing “trans” for that separation are changed to “0”. By entering “Trans”, the separation is set to overprint, changing all “0” values to “trans”. The **Apply on Selection** can be used to set the desired separation to 0% or Trans only for the selected objects.

4. Opaque / Die / Trans

Allows to set the attribute of a separation to opaque, die or transparent (varnish).

The Opaque/ Die/ Trans information is used in the JDF export and in the Vector trapper action.

Move Ruler

Every document in Import has a ruler position, depending on how the ruler position was set when importing a file or when opening a file.

The **Move Ruler** action moves the ruler within the current file to a new position based on an offset from the current ruler position or an offset from the position of any of the corners or centers of the pageboxes. This offset is set by the Vertical and Horizontal fields. This is very useful when pictures or other files need to be added to the current file at an offset from the ruler position or any of the corner or center points of one of the pageboxes.

Nested Step and Repeat

The **Nested Step and Repeat** action is based on the PowerStepper module in ArtPro.

It is used to make step and repeat based on the MFG (ArtiosCAD), CFF2 or DDES information. MFG, CFF2 and DDES files are generated on CAD/CAM systems to create die cuts (example: Dimensional Impressions’ “Score”). Through this action these files can be used to automate the step and repeat process without having to calculate positions, rotation and nesting. For a full explanation of the Nested S&R functionality, we refer to the ArtPro Manual.

The action typically needs two files as input:

- The CAD file
- The ArtPro file

The CAD file can also be specified in the action settings.

Nested Layout File Format

Instead of selecting an image, the user can also select a ‘layout’ file, that contains the names of several images, together with information on where they should be placed.

Layout files are tab-separated text files with a small header. Here is an example:

```
ArtPro Nested S&R Layout V1.0
JokerBoxBerries.aw 1-35
JokerGreen3.aw 46-7
```

The first line identifies the file as a 'layout' file that can be used for Nested S&R. It is followed by an empty line. Each following line contains the name of an image and the station numbers at which these images should be placed. In the example, the file 'JokerBoxBerries.aw' is placed at station numbers 1 through 3 and 5 and the file 'JokerGreen3.aw' is placed at station numbers 4 and 6 through 7.

When using a layout file, you can deselect the option **Filename in Layout File must match** and provide the files to be placed using a **Select File** before the action.

Open ArtPro File

This action opens an existing ArtPro file, either input through the **Apply ArtPro Action List** task, generated earlier in the action list or defined earlier in the action list (example: through the Select File action).

When an ArtPro file is dropped in the hot folder, the line art file AS WELL as the CT file (when available) need to be dropped. The system automatically selects the line art file for opening while recognizing the appropriate CT file. In order for this to work correctly, the corresponding action list has to be created with the option **Preserve Filenames** active. This makes sure that the appropriate line art and CT file names are not modified by the system which would result in the system not finding the CT portion of the file.

The **Open ArtPro File** action contains functionality related to used images, used fonts and color management. Although an ArtPro file can contain a line art portion and a CT portion (for internal images), an ArtPro file can equally contain LINKED images, these are images which are used in the ArtPro file but which are not embedded in the CT portion of the file.

The **Open ArtPro File** action contains four main portions: Root Folders, Folder Mapping, Fonts Folders and Color Management.

Root Folders

When images were used residing on the front-end Macintosh or on a server which can not be mounted on the Nexus system, the Root Folders functionality can be used to define where Nexus has to locate linked images which are used in the ArtPro file for opening.

In this example one Root Folder has been defined: E:/AE/ AP_files. When an ArtPro file that contains linked images which were previously stored on a local Mac needs to be opened, they need to be copied to this folder or a subfolder of this folder first. A Root Folder can be added by typing in the appropriate path and clicking the **Add** button or by selecting it through the **Browse** mechanism. Equally, a Root Folder can be removed by clicking the **Remove** button.

Multiple Root Folders can be set, when opening an ArtPro file, Automation Engine goes through the respective root folders (and subfolders) until the image has been located.

Note: You can use a **File Naming Template** to locate the root of the images.

Folder Mapping

The **Folder Mapping** mechanism can be used when images which are residing on a server and which can be mounted on the Automation Engine are used.

As an example, in the ArtPro file, the path of an externally linked image can be defined as: Intel_Mac:images:image1.

Obviously, on the Automation Engine, the file server would be mounted at a different mount point with a different name. Typically the image server would be mounted to a drive letter, for example, G:.

To allow Automation Engine to automatically locate images on the Image Server when opening an ArtPro File, the **Folder Mapping** mechanism allows to substitute a portion of the path name used in

the ArtPro file with another name. In above example, Intel_Mac would be substituted with G:/, which would tell the Automation Engine to find above image as G:/images/image1.

To define a **Folder Mapping** substitution, enter the respective fields **On Client** (how the local Macintosh sees the file server) and **On Server** (how the Automation Engine sees the file server) and click **Add**. Similarly, **Folder Mapping** substitutions can be removed by clicking the **Remove** button.

Fonts Folders...

When live text has been used in the ArtPro document, **Fonts Folders** can be used to allow Automation Engine to extract the outline information from the Mac fonts for trapping, printing, exporting, ArtLink functionality, etc.... Please note this is only valid for Type 1 fonts and Truetype fonts.

To define a fonts folder click the **Add** button and browse to the appropriate folder. Similarly, fonts folders can be removed by selecting them and clicking the **Delete** button.

Although the ArtPro file format (version 6 and onwards) contains all the outline information for live text in the document, the font is still important for special functionality like ArtLink, Regmarks, etc. ... By clicking the folder icon next to a font folder, you can select **Edit**, which opens a **File Naming Template** dialog box allowing to define the name for the font folder based on the job name, user name,...

Color Management

This setting determines if color management has to be applied during the action list. When enabling color management, it is possible to select a color management settings file, previously saved by ArtPro. This file contains all ICC profiles and settings to determine how color has to be treated throughout the rest of the action list, including ICC profiles for proofing, printing, etc.

Tip:

You can use **Make SMB Compatible** to rename all the linked images with a crossed edge sign (#). This character will be replaced by an underscore. Use this option to make these images compatible with Windows fileserver. In this case, the ArtPro file should be saved to disk so that the image links point to the renamed images.

If the ArtPro file itself contains a working space different from the one defined in the Color settings file, the action list stops if the option **Stop when working space profiles are different** is checked.

Map Pictures

If the **Map Pictures** option is selected, images are converted to mapped pictures if possible. For more information on Mapped Pictures, we refer to the ArtPro documentation.

Ignore Screening Errors

Since Extended Screenings are stored differently in files from older version, opening a file from them results in a conversion of the screening. If this conversion fails, the job also fails, unless the **Ignore Screening Errors** option is selected. As a conversion error means the screening settings are changed, it is advised to use the **Ignore Screening Errors** option only for action lists that do not have any screened output or if screening settings are defined in the action list itself.

OPI Generation

The **OPI Generation** action allows to generate low resolution images from high resolution images, example: for use in a page layout environment.

In this action the resolution and format (TIFF or EPS) can be selected. When a document is printed with **Omit Images**, the **Importing** action can then recognize the OPI comments in the PostScript or PDF file and reroute back to the high resolution images.

Optimizing

The **Optimizing** action deletes (parts of) the contours that are not needed to produce correct print output. The function makes it possible to “clean” or flatten a document, removing all the superfluous and unnecessary paths. For each path the program checks all the contours on top of it and decides which objects should be cut, united or left untouched to maintain the same structure in the job.

Optimize is especially useful to eliminate unnecessary contours from an imported PostScript file, to perform faster trapping, faster printing,... Keep in mind though, that the objects are typically less editable after optimization than before. We hence only recommend to use the **Optimizing** action for complicated graphics documents.

The effect of optimizing are, the graphics look a lot cleaner and are faster to print and trap. However, the editability is considerably reduced.

PowerTrapper

The **PowerTrapper** dialog box is used to make trapping settings.

1. Choose your trapping settings in the **Distance & Direction**, **Color & Shape** and **Processing** tabs.
2. If needed, define trapping rules using **Add Rule**.
3. **Save** your trapping settings as a **Trap Preset** to be able to reuse them on another document.

Distance & Direction

1. Trapping

You can choose between two trapping modes:

- **Normal Trapping:** This selects adjoining color pairs, which are likely to cause registration problems like ugly light gaps, and unwanted halo effects. PowerTrapper Standalone automatically chooses the most appropriate trap direction based on the relative luminance of the adjoining colors. Lighter colors are generally trapped into darker colors to minimize the visual effect of the trap.
- **Reverse Trapping:** This prevents overprint of adjoining color areas by trapping these color pairs with a white knockout (erasing) trap. The white knockout trap is put on top of the lighter of the two colors.

This is useful to prepare a job for Dry Offset printing (example: metal beverage cans) where overprinting of inks is not allowed as the inks would contaminate each other on the blanket.

2. Trapping Distance

Enter the width you want to give your traps. The unit used is the General unit defined in Illustrator's Unit & Display Performance Preferences. The default trap width is 0.2 mm (or equivalent in your chosen unit).

3. Minimum Ink Difference

Trapping is applied to two adjoining colors when:

- the two colors each consist of at least one ink but they have no common inks.
- the two colors have common inks but there is a difference of at least the **Minimum Ink Difference** percentage for two or more inks making up these colors.

These differences in ink percentages must be in opposite directions.

For example, with the default minimum ink difference percentage of 10%, the following two colors are trapped:

1st color: C 5% M 10% Y 50% K 15% (+10% K)

2nd color: C 5% M 10% Y 85% K 5% (+35% Y)

Increase the **Minimum Ink Difference** to trap colors that are more different, lower it to trap colors that are more similar.

4. Image Trapping

- **Automatic** (default): this determines the most suitable trap direction automatically, by comparing the average lightness of the CT image with the lightness of the adjoining area, then traps the lighter color into the darker one.
- **Always Towards Images**: this always traps line art under adjoining CT images.
- **Always Towards Line Art**: this always traps CT images under adjoining line art.

5. Pull Back

An ink pull back keeps away all but the darkest ink of a color area composed of multiple inks that borders to a very light background.

It prevents individual inks from the composed color to become visible on the background due to registration errors during the printing process.

- **Automatic** (default): this creates pull backs on rich black or rich spot colors, when the color left in the pull back area is not very different from the composed color.
- **Only Pull Back Rich Black**: this only creates pull backs on rich black.
- **Do Not Pull Back**: this never creates any pull back.

6. Advanced Settings

Expand the collapsible button to view the **Advanced Settings**.

1. If you want to use a different distance when trapping into black, spot colors, images or when creating pull backs, enable the corresponding option and enter the trapping distance to use.

The unit used is the **General** unit defined in Illustrator's **Unit & Display Performance Preferences**.

2. Image Trapping

If you want images to be trapped with each other, enable **Trap images to images**.

3. Pull Back

If you chose to use **Automatic** pull back (see Pull Back), the **Also pull back light inks** option is available. Enable it if you want to create pull backs even when the pullback ink (ink remaining in the pull back area) is visibly different from the composed color. Enable **Do not pull back images and gradients** if you do not want to create pull backs on images and gradients.

4. Overshoot Mode

You can choose between the following three overshoot modes:

- **Automatic** (default): The centerline behavior is according the rules determined in the color pair list.
- **Never on Dark Areas**: The centerline is forced not to get beyond the center of dark areas.
- **Never**: The centerline is forced not to get beyond the center of any area.

Color & Shapes

1. Trap Color Intensity

By default, **PowerTrapper** uses the full color of the object to spread in the trap (100% trap color intensity).

However, you can create a trap of a lighter color by reducing the trap color intensity percentage.

2. Image Trap Color

Choose how to trap your images:

- **Use Original Image Data** (default): When spreading an image, **PowerTrapper** uses the underlying image data (when the image is clipped) to create the trap.

Note: When there is no underlying image data (the image is not clipped), it creates an empty trap.

- **Extend Image Data:** **PowerTrapper** automatically extrapolates the image along its top, right, bottom and left edges to ensure that the trap looks as expected even if there is no clipped image data outside the visible image.
- **Use an Approximate Flat Color:** Instead of using existing or extrapolated image data in the trap, **PowerTrapper** uses a solid color. A suitable color is determined by averaging the image's colors along its boundary with the object to trap.

3. Truncate Traps

PowerTrapper can truncate the trap so that it does not stick out on another color. There are two truncation modes:

- **On Center** (default): when a portion of a trap comes too close to the contour of another object, **PowerTrapper** limits it to half the distance between the trapped object and the other object.
- **On Edge:** when a portion of a trap comes too close to the contour of another object, it is truncated on the other object's edge.

4. End Caps

This option specifies how to shape the ends of an open trap.

- **Square** (default): this option ends the trap at right angle to the adjoining object.
- **Round:** this rounds the ends of the trap. This option is typically only used in combination with white knockout (reverse trapping).

Note: We recommend you not to use **Round End Caps** together with **Truncate Traps On Center**, as this can generate some artefacts (the round caps are truncated).

- **Object Dependent:** with this option, the trap is a logical continuation of the contour of the spread object.

5. Trap Corners

This option allows you to change how the traps' sharp corners are handled.

- **Round** (default): a round cap will be placed at all corners.
- **Beveled:** this cuts sharp corners off.
- **Mitered:** this option works with a miter ratio.

The miter ratio serves to limit the length of the sharp corner (the distance from the base of the trap to the corner point).

The default miter ratio value is 4. This means that if the length of the sharp corner is more than 4 times the **Trapping Distance**, then the corner are cut off (beveled). If it is less than 4 times the **Trapping Distance**, the corner are left as it is.

6. Advanced Settings

Click the collapsible button to show the advanced settings.

Truncate Traps

If you want to truncate traps into black differently, select **Into black** and choose the truncation mode (**On Center** or **On Edge**).

Processing

1. Color pairs

PowerTrapper can trap all hits (occurrences) of a color pair the same way or recalculate the trap direction for different hits.

- Each hit can have its own decision (default): this recalculates the trap direction for each hit of a color pair.
- Use the same decision for all hits: this traps all hits of the same color pair in the same direction.
- Use the same decision for objects smaller than: this uses the same trap direction for hits of a color pair belonging to small objects, and recalculates the trap direction for hits belonging to bigger objects.

Use the field next to the option to determine the maximum size of small objects. The unit used is the **General** unit defined in Illustrator's **Unit & Display Performance Preferences**.

The default size is 1 mm (or equivalent in your chosen unit).

Restrictions

In some cases **PowerTrapper** can safely ignore certain object types to speed up the trapping.

- Ignore existing trap layer: if your document has already been trapped, select this option to keep the existing trap layer out of the trapping process.
- Ignore bitmaps: this keeps bitmap pixel data from being trapped. Bitmap data is contourized before trapping which may slow down trapping depending on the bitmap's contents. Bitmaps with lots of small isolated islands typically take a longer time to trap.

Note: When you indicate an object type to ignore, it is ignored completely as if it did not exist in the job. Also, the ignored objects does not truncate trapping areas resulting from other objects below them.

2. Small Gaps

Some input jobs contain very small unintentional gaps between neighboring objects, preventing the correct trapping of these objects. Although it is better to clean up such jobs before trapping, **PowerTrapper** can ignore these gaps automatically.

To use this option, select **Close when smaller than** and enter the maximum size small gaps can have.

The unit used is the **General** unit defined in Illustrator's **Unit & Display Performance Preferences**.

The default size is 0.01 mm (or equivalent in your chosen unit).

Note: If you set a big gap size, trapping may become slower and small parts of the job may be ignored during trapping.

Add Rule

Add Rule allows to define custom trapping settings for specific trapping pairs.

1. When to use rules:

In most cases the general trapping settings entered in the **Trap** dialog box gives you excellent results, but in some cases you may need to refine them using rules.

Add Rule allows to specify exceptions to the general trapping settings for certain color pairs. They are an advanced feature and should be used only by experienced users.

For example, you can use them in the cases below:

- To set a different trap distance for a specific spot ink.
Sometimes you need a larger trap distance for spot inks if there is no other ink to help cover leaks on the press.
- To prevent trapping or ink pull back in specific cases.
Depending on the type of the adjoining colors, trapping and / or pull back may be undesirable.
- To force a specific trap direction.
PowerTrapper Standalone determines the trap direction based on the relative lightness of the adjoining colors. If the colors have a similar lightness level, it selects an arbitrary direction. You can use a rule to make it always use a particular direction for a color pair.
- To specify a different shape and/ or truncation mode for certain color pairs.
For example, you can choose to use “On Center” truncation when trapping under a 100% black line and “On Edge” truncation on all other color pairs.

2. Creating a rule

- Click the **Add Rule** button at the bottom of the **Trap** dialog.
- In the pop-up that opens, choose the trapping pair to apply the rule to using the **From** and **To** lists. You can create a rule to trap from/to: any color, a particular ink (process, spot or opaque), a particular object (image, gradient or empty background) or the registration color.

Note: When trapping a particular ink, you can choose the ink name from which **Minimum Density** the rule is applied and if the color must be pure (no other separation in the color object) to apply the rule.

- Choose the contents of the rule.

Options	Description
Trapping Direction	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Standalone calculate the trapping direction. • Force Trapping from ‘From’ to ‘To’ always spreads the “From” color/object on the “To” color/object.

Options	Description
Trapping Distance	<ul style="list-style-type: none"> • Do Not Trap 'From' to 'To' never traps the "From" and "To" colors/objects together. <p>Use this to define a special trapping distance for the trapping pair.</p>
Trap Color Intensity	<p>Use this to define a special trap color intensity for the trapping pair.</p>
Pull Back	<ul style="list-style-type: none"> • Automatic lets PowerTrapper decide if there should be a pull back on the trapping pair. • Always Pull Back always creates a pull back on the trapping pair. The darkest ink of the pair is the pullback ink (remaining ink in the pull back area). • Do Not Pull Back never creates a pull back on the trapping pair.
Pull Back Distance	<p>Use this to define a special pull back distance for the trapping pair.</p>
Reverse	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Standalone decide if the trapping pair should have a reversed trap. • Always Trap Reversed always generates a reversed trap for the trapping pair. • Do Not Trap Reversed never generates a reversed trap for the trapping pair.
Truncate Traps	<p>Choose if traps generated for the trapping pair should be truncated on center or on edge.</p>
End Caps	<p>Choose how to shape the ends of open traps for the trapping pair (square, round or object dependent).</p>
Trap Corners	<p>Choose how to shape the trapping pair's trap corners (round, beveled or mitered).</p>

- If necessary, use the + button to add contents to your rule. You can use the - button to remove contents.
- After defining the rules, click **OK** button. You can now see your rule at the bottom of the **Trap** dialog box.

When trapping your document, PowerTrapper traps the trapping pairs that match rules according to those rules and the rest of your document according to the main trapping settings. If necessary,

you can add another rule or remove a rule previously created. You can also edit a rule by double-clicking it in the **Trap** dialog box.

Print to File

The **Print to File** action converts the ArtPro native file format to a composite or separated EPS file, depending on the **Device Type** that has been chosen.

Depending on the action list, the **Print to File** action can be simply followed by a **Close File** action, an FTP action, a **Copy File** action or a **Convert EPS to PS** and a **Send over PAP** action. This depends on what device the file needs to be printed to and how the device is physically connected to Automation Engine. This is described later in this section.

Load Settings from XML

After using a **Select File** action to select an XML file, the settings from that XML file can be loaded in the **Print to File** action, overwriting the existing settings.

General Concepts

1. Flattening

When creating a PS file, the ArtPro file is flattened if needed, as PS does not support blend modes, opacity, etc. The settings of the **Flattening** Options are used.

2. Composite PostScript®

Composite PostScript means all information is calculated into a single page, unseparated PostScript file.

In a Composite file, placed EPS files are copied unmodified. This means that if you work with DCS for example, the placed low resolution is sent to the printer. Both TIFF files and ArtPro files are converted into PostScript at full resolution (or any resolution defined by other print settings).

There are mainly 4 ways of generating PostScript for Printers/ Proofers.

1. Using DeviceN (Composite L3 Printer)

When outputting composite PostScript using the Level 3 DeviceN comments, a composite PostScript is generated, containing all transparencies and spot colors. If this composite output is handled by a RIP, capable of handling these level 3 commands, a correct output can be achieved. This option is preferred if the Proofer/Printer can handle it.

Please note that composite output, even when using the DeviceN command does not include dotgain or screening information.

2. DeviceN Off, Use Setcustomcolor OFF

The complete job, if necessary, is flattened to remove all overprints and transparencies, and converts all PANTONE™ colors into CMYK.

The result is a flattened CMYK composite PostScript file which can be sent to any color proofer.

3. DeviceN Off, Use Setcustomcolor ON

This creates a flattened file, flattening all CMYK overprints and blend modes.

The resulting PostScript file contains CMYK objects in combination with separation color objects. This PostScript file still contains overprinted objects. Not every color Proofer can handle this.

4. Bitmap Proofer

This option forces to Rip the complete job into a CMYK image at a given resolution and generate a PS file containing only the Ripped CMYK image.

3. Separated PostScript®

If you make a file with All Separations, a multipage PostScript file is created. Each page contains black and white information describing the separation of one ink. So the file contains as many pages as there are inks in the ArtPro file.

If you make a file per separation, a single page PostScript file is created containing the black and white information of the film (ink) you asked for.

Placed art in separated output:

- EPS files containing CMYK, grays and custom colors are separated by ArtPro using **separation code**. Some EPS files however, are not suited for this separation code. Therefore an alternative separation code is provided (by switching on the **Simple separation code** option) for these situations. If you work with DCS files (or DCS-2), ArtPro replaces the low resolution file by the high resolution separations.
- TIFF CMYK files are separated and converted into PS.
- ArtPro files are converted as normal, this means any picture in the file is expanded with high resolution.
- OPI: PostScript output of TIFF files always contain OPI comments (as specified in Aldus Open Prepress Interface Specification 1.2 and 1.3). This makes it possible to work with low resolution images on the Mac and in proofs (= unseparated output) and high resolution images on film (= separated output). Even if the TIFF file type is not supported (not CMYK), OPI comments are generated.

Examples of TIFF files which are supported by OPI are RGB and palette TIFF files. These files can be placed in ArtPro, can be seen on screen and make a color proofer output. However, separation cannot be made for these TIFFs, unless you use OPI replacement.

4. Placed Files (S&R)

Placed PDF 1.3 files are converted to PS during output. Placed PDF 1.4 files cannot be converted to PostScript Level 3 and generate an error `Cannot convert PDF 1.4 into PostScript`. Note that the distinction between PDF 1.3 and 1.4 is not done on the header, but on the presence of PDF 1.4 functionality.

Device Type

1. Color Proofer

Color Proofer is the standard Composite PostScript output to send to PostScript proofing devices.

2. Color L3 Proofer

Color L3 Proofer generates composite output, using Level 3 commands such as DeviceN and shadings. This results in composite output, containing all spot colors, overprints, etc. Screening and Dotgain Compensation is not included.

3. Separated Proofer

Separated Proofer is a device to print separations on a color proofer, for example: a LaserWriter. Although it is sent to a Proofer, the output is separated.

4. Bitmap Proofer

This output, originally developed to be able to output overprints and transparencies, has become obsolete because of the L3 commands. With the Bitmap Proofer output, the ArtPro file is RIP'ed into image data which is sent to the proofer, preceded by a PostScript header and followed by

a PostScript footer. The image data consists of CMYK information. This output can get quite substantial, depending completely on the output resolution and the file size.

In the **RIP** tab of **More Settings**, the desired resolution can be defined. The ideal resolution is equal to the resolution of the proofer device being used.

5. Custom proofer

Custom proofer is the output device for proofers with all **More Settings** available. Any device type actually has a number of preset **More Settings**, of which some can be modified and others are grayed out. **Custom Proofer** has the preset **More Settings** from **Color Proofer**, but all settings are available for modification.

6. Generic ImageSetter

This is the default device for the output of **Separated PostScript**, for a general type of imagesetter or platesetter.

7. Agfa

To generate output for an Agfa imagesetter, choose **Agfa** in the **Device Type** list. The type of PostScript generated for Agfa is separated PostScript.

8. Barco RipScript

This device type generates separated PostScript, to be exposed through **Barco RipScript**.

9. Barco TransScript

This output generates EPS files which can be sent to an old Barco system, through **Barco TransScript**. Note that these files are not standard EPS files and are not to be used in applications other than Barco TransScript. For more information on how to output through Barco TransScript and the limitations of this method of output please refer to the ArtPro manual.

10. Creo (Layout / Single Job)

These two devices make it possible to use OPI server functionality for Step & Repeat on Creo Servers. Whereby, high resolution image data is stored on the server and a preview version is stored with the original file. At output the server automatically replaces the preview version with the high resolution data, according to the OPI comments contained in the Step & Repeat Print file. Printing will therefore be considerably faster, as network data transfer between Automation Engine and the Creo server system is reduced to a minimum.

1. Procedure

- a. Print the single job in the **Print** menu with the option **Creo** (Single Job) and send it through to the Creo server.
- b. Create the S&R file of the single original ArtPro job and save it.
- c. Print the repetition file to Creo (Layout)

Besides the OPI possibilities, the Creo output contains DIG comments, allowing Print files from Automation Engine to be used in the VPS, the Creo Virtual Proofing System. Therefore, even if the OPI Step&Repeat is not used, Creo Layout should still be used to enable the VPS.

11. Crosfield

Crosfield output is based on Generic Imagesetter but fine-tuned to make the Crosfield OPI work. The setting **Include Gradations** comments allows Crosfield to put gradations (converted to CTs) in the **Lineart** level (at high resolution) and CTs in the CT level.

12. DCS-2 Single File / DCS-2 Multifile

The DCS-2 output creates a DCS file, a separated file format including all separations and a preview. With **Multifile**, the result is as many files as there are separations and an additional preview file. For Single file, separations as well as preview are contained in one single file.

The DCS files can be placed in other applications (example: page layout or imposition applications,...) or can be used with OPI servers when printing Step & Repeat files or from imposition applications. Hereby, the high resolution image data (DCS 2 file) is stored on an OPI server and a preview version, generated by the OPI system is stored with the original file. At output the OPI server automatically replaces the preview version with the high resolution data, according to the OPI comments contained in the Step & Repeat file or the imposition file.

To print Step and Repeat files from one-up ArtPro files follow this procedure:

Save the single job in the **Print** menu with the option DCS-2 and send it through to the OPI server.

Create the Step and Repeat file of the single original ArtPro job, either in a step and repeat action list or manually on an ArtPro front end station and save it.

Print the repetition file to Generic Imagesetter in the **Print** menu, with the option **Replace Placed ArtPro by OPI** in the **More Settings**.

13. Delta Technology

This output device is only to be used for repetitions. For every group, a Delta list is created. In the case of S&R, a delta list will be created for a single label. The next time the same Delta list is used, the RIP only has to place it, resulting in a very high S&R optimization. To use this S&R optimization, the clipping box has to be a rectangle. For S&R Tabular, this is no problem, but nested CFF2 repetition can cause troubles.

14. LinoType-Hell

LinoType-Hell output is based on Generic Imagesetter. This output supports the **Automatic Page Positioning** feature from Linotype-Hell.

15. Optronics

The Optronics is the same as Generic Imagesetter output.

16. Nexus

This **Device Type** should be used when printing to NexusRIP workflows as it results in separated PostScript optimized for NexusRIP. This device is used to print single documents (to a low resolution link file) as well as S&R or imposition data with links.

17. Purup

Purup is based on the Generic Imagesetter output, but fine-tuned for use with the Purup OPI system, Imagemark, which is based on low resolution DCS files.

18. Quark Separable EPS

To generate an EPS file that can be separated by Quark XPress® you can choose Quark Separable EPS as **Device Type**. As this is a composite output, it has some serious limitations concerning transparencies. Therefore, **DCS-2**, **L3 Proofer** or **PDF Export** is more advisable. For more information please refer to the ArtPro manual.

19. Scitex Separated / VIP / PCF

The best way to send to Scitex is using separated output, as composite output has too many limitations. The only reason to use composite (VIP) output is for APR, Scitex OPI system on the VIP rips or if the files are to be handled further on the Scitex systems.

To generate composite output for the Visionary™ Interpreter for PostScript® from Scitex, choose Scitex VIP in the **Device Type** list.

Scitex VIP is a computer application which converts PostScript files into the Scitex formats. The Scitex VIP output always makes PostScript output to a file, as it is not a real device but a computer application.

In some cases the VIP can be selected in the chooser, in other cases VIP has to be opened and the PostScript file loaded into the VIP. This depends on the customers' configuration. The type

of PostScript generated for Scitex VIP is composite PostScript. The composite output is then to be separated by the Scitex RIP. Please bare in mind that composite PostScript is much more limited when it comes down to supporting transparencies, or not standard ArtPro functionality like mapped pictures, etc.

Output to Scitex PCF is specifically meant to output step and repeat data to Scitex's native format. For more information on these device types, please refer to the ArtPro manual.

20. Screen

The Screen is the same as Generic Imagesetter output.

21. Think Laboratories

A Generic Imagesetter output, fine-tuned for Think systems.

22. Custom ImageSetter

Custom ImageSetter is the output device for imagesetters with all **More Settings** available. Any device type will actually have a number of preset **More Settings**, of which some can be modified and others are grayed out. Custom ImageSetter has the preset **More Settings** from Generic ImageSetter, but all settings are available for modification.

Other Settings

1. Paper Size

The **Paper Size** drop-down menu includes the different sizes. If a PPD needs to be assigned, it should be done through the **Convert EPS to PS** action. **from media box** always uses the Media Box. **from page box** uses the active (that is, the last selected) page box. If any objects are outside the selected page box, a warning is generated. When choosing **from Job** (default), the active page box is used, enlarged if necessary to fit all objects.

Following sizes are included in the Paper Size menu:

- A4: 210 x 297 mm
- A3: 297 x 420 mm
- US Letter: 8.5 x 11 inch
- US Legal: 8.5 x 14 inch
- Tabloid: 11 x 17 inch
- Tabloid extra: 12 x 18 inch

2. Layout

For proofer output, choose between **Center**, **Fit always**, **Fit Large Paper** or **Tile** output. **Center** will position the job centered on the paper size, taking into account the indicated **Distortion** for size. A distortion of 0% results in a true size job output. If the job is bigger than the indicated paper size, it is clipped on the edges. **Fit Always** enlarges or reduces, so that the ArtPro file matches the paper size. **Fit Large Paper** fits large format ArtPro jobs to smaller paper sizes, without enlarging the job when the paper size is larger than the job size. **Tile** is used when a job is larger than the paper size. It divides the job over several pages, with the indicated **Tile Margin** and **Tile Overlap**. Define a **Tile Margin** if you do not want to use the complete paper size. The **Tile Overlap** indicates how much of the tile edges are printed on the next page, as to create an overlap area.

3. Orientation

The **Orientation** can be set to **Portrait**, **Turn right**, **Turn left** or **Upside down**. It defines the direction in which the file is put on film or on paper. Choosing portrait outputs the file in the upright position, turn right rotates the file 90° and turn left rotates it -90°.

4. Mirror / Negative

When **Mirror** check box is selected, the job is mirrored before exposing, resulting in a wrong-reading output. **Negative** allows outputting separations as negatives. Note that these settings are additional to the settings on the RIP actions.

Example: a negative setting on **Print to File** and a negative setting on the **Assemble a Screen Page Sub** action results in a positive output. The same is true for **Mirror**. Equally note that the imagesetter or platesetter device can also have a negative and mirror setting.

5. Output

This allows to define the name of the output file, using the **File Naming Template**.

6. Title

The **Title** allows to define the title comment in the PostScript file. This comment can be extracted, example, in a NexusRip workflow, to extract the job name instead of the PostScript file name.

7. Distortion

A document can be output with a vertical and/or horizontal distortion. It is defined as a percentage of the jobs original dimensions and is applied to the PostScript file.

This distortion is specifically important for flexo jobs. On mounting a flexo plate on the press, it is stretched out in one direction. By making the output slightly smaller than needed, we compensate for this distortion. The original dimensions of the file remain unchanged.

Positive values make the job bigger and negative values make the job smaller. Example: When vertical distortion is -2.35% it makes the output 97.65%. In this way you can also scale the proofer output: example, entering - 50% for both vertical and horizontal distortion outputs the job at half of its original size.

8. Clip

If the job is bigger than the paper or the film, you may output in parts. If **Clip** is not active, the complete job is printed or exposed and is thus clipped on the output when it is bigger than the paper size. When Clip is checked, you can define the tile dimensions by clicking on the **Rectangle...** button. The tile indicated in the Rectangle dialog will output.

9. Rectangle...

Clip Rectangle allows to define what part of the job should be printed.

1. page box Clipping

When **page box Clipping** is selected, the job size chosen under **Use Box** is used to clip the output. The different boxes can be defined in the **Job Size** dialog box.

2. Manual Clipping

If **Manual Clipping** is selected, the size and position of the part to be printed can be set manually. In the **Vertical** and **Horizontal** field, the size of the rectangle to be printed can be entered. If the **Lock to Ruler** option is selected, the rectangle to be printed moves along when moving the ruler (see **Move Ruler** action).

3. Automatic Clipping

If **Automatic Clipping** is selected, the size and position of the part to be printed is defined by the job size and the settings of the **Divide** and **Print Part** fields.

Divide: in the **Divide** field, enter how many parts the job should be split up, for both vertical and horizontal.

Print Part: in the **Print Part** field, define which part of the job should be printed.

10. Marks

Besides the **Cut** marks, other objects like registration marks as well as job and separation identification can be added to the file through Generate mark.

11. Separations

The **Separations** menu has a dual function, to define which separations have to be output and to define how separations need to be mapped to other separations. By default the window is blank and the **Search Unused Separations** option is selected. This means that the system checks which separations are in the document and output separations which are not empty. When there are non-mapped pictures in the file though, all separations are output. The **Separations** window contains two areas, the **Separation Mappings** area and the **Exclude** area.

1. Separation Mappings

To add an entry to the **Separation Mappings** list, click the + button and enter the separation name in the **From** field. By entering the **To** separation, you can remap the separation to another plate. Note that it is equally possible to enter “CMYK”, which allows to convert a separation to CMYK on output without actually modifying the document itself.

The **Print** icon can be toggled on or off by clicking the **Print** button.

An example on the use of the **Separations** dialog box, is in a double burn action list, where there are three different black separations in the file: English Black, Spanish Black, and Graphics Black. In this case, the Graphics Black should be mapped to the English Black as well as to the Spanish Black, hence they are merged. Seven separations are output into six plates.

2. Notes on separation mapping

Mapping of output separations is fully supported for all paint styles. Even placed ArtPro jobs can be merged. Example: for a layout, containing two ArtPro jobs, each with a different spot color, both separations can be merged during output. This is even possible if the jobs contain TIFFs. This is because no merging is done within the placed ArtPro jobs. Note however that remapping is only available for Separated outputs!

Separating placed EPS- and PDF-files takes into account overprints, PS3 and PDF1.3 vignettes (Illustrator 8 and 9) and Photoshop duotones.

3. Screening for mapped separation

When you have a step & repeat file containing multiple jobs and you want to map separation X from job A into separation Y from job B, separation X will still be output with its own screening settings as long as this job (A) does not contain separation Y. When job A contains separation X as well as separation Y, the screening which will be used to output the mapped separation X into Y will be separation Y.

4. Exclude Separations

This option can be used to exclude separations from the output. By clicking the **Options** button, the **Output Separations** dialog box is opened. By Adding a separation (click the + button and define its name. Wildcards can be used), you can exclude this separation for this. By selecting an entry in the list and clicking the - button, you can remove it from the list.

5. Search Unused Separations

The **Search Unused Separations** setting excludes separations that are not used. It only turns separations off, separations excluded earlier are not removed from the **Exclude** list, even if they are in use. The **Search Unused Separations** option does not take registration color into account.

12. Dot Gain Compensation

The **Dot Gain Compensation** button defines if the dot gain curve(s) selected in the **Dot Gain File...** menu (see below), has to be calculated on the PostScript output. With this option off, all curves are ignored for output. **Dot Gain Compensation** is only available on separated output.

13. Dot Gain File ...

The **Dot Gain File ...** button allows you to select a Dot Gain File. The file name of the selected Dot Gain file is shown on the right side of the button. If a Dot Gain File is selected in the **Print** menu, this Dot Gain File overrides all Dot Gain settings in the **Screening** dialog (example: an opened ArtPro file). The exception is a placed EPS that contains a dot gain compensation, in which case both curves are used. A selected file can be removed by clicking the Dot Gain File button while holding down the SHIFT key. Dotgain files are generated in and saved from the screening dialog. (please refer to ArtPro manual for more details).

14. Color Management

If a color management profile has been defined at the time of importing or opening a file in the action list, it is possible to override the settings for output (example: color profiles for proofers or for separated output) in this menu.

15. Flattening Options

When creating a PS file, the ArtPro file is flattened, as PS does not support blend modes, opacity, etc. The settings of the **Flattening Options** are used.

16. File & Fonts Settings

The **File & Font Settings** allows to define root folders, folder mapping and fonts folders, similar to the **Open ArtPro File** action (please refer to this action for a full description). This functionality has been made available in case ArtPro files have been placed in an open job during the action list, to allow for the retrieval of the linked images or used fonts at the time of printing.

More Settings

In this dialog a whole list of **Print Settings** are available. Depending on the active **Device Type** the settings are defined automatically, whereby some are checked by default. For some devices, some settings are not accessible (gray) and others can be (de)activated. For **Custom Proofer** and **Custom Imagesetter** all the options are available so as to allow specific customized output. The **More Settings** are divided up in seven categories: **Page, Separations, Paints, Text, Optimize, RIP** and **Preview**. For more information, please refer to the ArtPro reference manual.

1. Page

These settings are related to the type of file of the EPS (PS) file that is generated by the **Print to File** action (composite versus separated,...), and the comments in the header. Changing these settings may in some specific cases be necessary to have a receiving application or printer recognize the file or to have the correct type of file generated.

2. Separations

These settings determine how separations and page commands are defined in the file. Depending on the **Device Type** that has been selected in the main **Print to File** dialog box, these settings are correctly defined by default. Only in specific cases should changes be made.

3. Paints

These settings are related to how colors and overprints are defined in the file and how gradations and images are handled. The system supports applying image averaging at the time of down sampling. This results in better image quality when downsampling has been applied, but slows down the generation of the PS output.

4. Text

These settings determine how text is handled in the file. Settings can be made to vectorise text or to embed fonts.

5. Optimize

These settings determine how the PostScript needs to be optimized and how OPI comments need to be handled in the file, including the use of OPI comments for placed ArtPro files, allowing for a productive Step and Repeat solution (example: with the Creo Layout device).

The PS Optimize settings include the ability to split long paths (important for output to PS Level 1 RIPs or for RIPS with limits to the amount of points on a path due to insufficient memory) and the use of picture caching which improves output speed when images need to be put into the output multiple times (example: when image trapping has been applied).

6. RIP

These settings allow for specifying special comments needed to output PostScript to a RIP system, including the NexusRIP. Note that the ArtPro Action List Editor has the ability to add an XML job ticket (Write Job Ticket) to the PS file, that contains specific information about the job, including the job size, separations, screening used, etc. This job ticket is then read by NexusRIP and put into the link file when a link file is generated. This allow ArtPro to display all the information about a specific job (the job ticket) when this link file is placed (example: a step and repeat file). For more information and an example please refer to the ArtPro manual.

7. Preview

These settings determine the type of preview that is generated (if any) and if a mask needs to be generated in the preview (example: to support transparent views for clipping paths). Some receiving systems may not support this preview mask. In these cases the mask can then be switched off, but the file is still output correctly.

PrintCheck

The **Printcheck** option allows to create an image of the complete job. Besides the resolution, the noise for gradation can be set. The compression allows to choose between tiff and jpeg, with 3 possibilities on compression : High compression, Medium compression or High Quality. The file can be exported as a Tiff/JPEG or an EPS. The **Box** drop-down menu allows to define what border has to define the boundaries of the image to be created.

Regmarks

The **Regmarks** action allows to use personalized marks.

The idea is to create one ArtPro file with the marks you need based on a blank document. The job size is not relevant for applying the registration marks.

When using this selected ArtPro file with this action, the marks will automatically adapt to the job size of the current document while the inks will equally be adapted.

There are two ways of selecting a Regmarks file: either select a fixed file by using the **Select File** button in the action, or select a variable Regmarks file by using the **Select File** action.

For a description of how to build a registration mark file in ArtPro we refer to "Import marks" in the ArtPro manual.

With the **Box** menu, define on what page box the regmarks need to be applied or to use the bounding box of the selection.

With the **Merge Layers** function, you can define that layers with the same name in the Regmark file and in the open file, should be merged together.

Reorder Layers

The **Reorder Layers** action allows to change the order of layers.

Layers with a name that match one of the entries are placed in the specified order. Other layers are placed underneath in their original order.

Entries in the list can be added and removed using the **Add** and **Remove** buttons. An entry can contain a wildcard (*).

If two different layers match the same entry, their relative order depends on the original order.

If a layer name matches different entries, the best matching is chosen. Example: a layer name **Trap layer** matches more with “Trap*” than with “**”.

Reorder Separations

Reorder Separations allows to change the order of the separations in the **Separations** list.

Separations with a name that match one of the entries are placed in the specified order. Other separations are placed underneath in their original order.

Entries in the list can be added and removed using the + and - buttons.

An entry can contain wildcards (*). If a separation matches two different entries, it is placed at the topmost position. If two different separations match the same entry, their relative order depends on the original order.

There is an option to reverse the separation order. This option can be used for some special printing processes like dry offset. When reverse is enabled, separation reorder cannot be done.

Save ArtPro File

This action saves the currently processed file to an ArtPro file on disk.

Output

The Template allows to set the name of the file to be saved.

Compatibility

The **Compatibility** allows to choose to save the document as ArtPro 9.x/8.x or as ArtPro 7.5 / 7.0 / 6.5 file or as ArtPro 6.0 file. An ArtPro document saved in an earlier file format does not contain version specific information. Example: an 8.0 document saved as ArtPro 7.5 document will not contain blend modes. The file is flattened before saving.

Include Preview

When **With Preview** is checked, a (separated) preview is included in the ArtPro file. The **Resolution** field allows to define the resolution (in ppi) for this preview. Default is 72 ppi.

When the preview is added to the file, it is shown in the **Job Editor**, if the **Thumbnail** option is enabled. in the **Preferences**.

Save Hidden Layers

This option defines whether layers with a “Hidden” state should be included in the saved file or not. See also the **Layers** related actions.

Profiles

Embed entire profile in ArtPro files allows to include the used **Color Management** profile in the ArtPro file. This can be useful when moving files to another system. The option **Don't embed profiles for imported pictures with embedded profiles** disables embedding profiles already embedded in the pictures.

Flattener Options

When saving as ArtPro 7.5 file, the ArtPro file is flattened. The settings of the **Flattening Options** are used.

Certified PDF sessions information

The option **Include Certified PDF information** allows to define if Certified PDF information should be included in the ArtPro file. If the option is checked, the information is included in the file, even when saving as ArtPro 7.5 file. If that file is open in ArtPro 7.5, that information is ignored. If the option **Save Certified PDF Information** is selected, the Certified PDF information is saved in a separate XML document. The **File Naming Template** is used to define its name and location.

Flattener Template

If the Flattener needs to save images, the name of these images are defined by the **File Naming Template** next to the **Flattener Options**.

Select File

Automation Engine is capable of working with hot folders that can accept files, as well as hot folders which can accept folders. When a folder is submitted to a hot folder, this action determines which file(s) from the folder is selected for processing. It can also be used to filter on a list of files which should be used as input to the following action. The **Select File** contains several items for selecting a file for further processing.

Select Options

The **Select Options** show the criteria used. A line can be added using the + button or removed using the - button. When a criteria is selected, its settings are shown in the **Select Item Options**. The available settings depend on the **Select** mode (By Name, By File Type,... see below).

Select By Name

1. Whose Name

The system selects a file based on the name of the file. The parameters are: **is anything, starts with**, “ends with”, “contains”, “does not contain”, “does not start with” and “does not end with”.

The **Edit Template** button allows to use the **File Naming Template**. It is also possible to use **Regular Expressions**.

2. Position From 0 To 500

The **Position From... To...** fields work in combination with the **Whose name contains...** option. Using the position fields, you can point out what character positions of the file name is checked. When setting **Whose name contains** as “APR” from position 2 to 4, files which have APR on character position 2 to 4 are searched. As the first character is position 0, the 3rd, 4th and 5th characters are searched.

Select By Documenttype

1. File Type

You can use this to select an ArtPro, PDF or PS file which matches the other criteria.

2. Number of Separations

The **Number of Separations** can be set to Equals, More than or Less than.

3. Number of Pages

The **Number of Pages** can be set to Equals, More than or Less than.

4. Page Box

The selection can be set for the selected page box to be within the entered values.

5. Orientation

The orientation criteria can be set to All, Portrait or Landscape.

Select Objects

Select Objects allows to select/ deselect objects.

First

The radio buttons define what to do prior to the selection.

Do nothing means leaving the selection as is. This can be useful to combine multiple **Select Objects** actions.

Invert Selections inverts the selection prior to applying the select action.

Select All or **Deselect All** allows to start from scratch with either everything selected or everything deselected.

Paint Style

1. Flat Paints

Allows to Select/ Deselect **Flat Paints**. This can be used in combination with the **Matching** option (see below).

2. Gradations

Allows to Select/ Deselect **Gradations**, either Linear, Circular or Multistep gradations. This can be used in combination with the **Matching** option (see below).

3. Matching

Using the **Matching** option in the **Paint Style** tab, it is possible to select objects with a **Flat Paints** or a **Gradation** within the defined percentage range. The separation names can contain wildcards. The **Overprint** column allows to select objects that are transparent for that specific separation.

4. Pictures

Allows to Select/ Deselect Pictures, either placed, mapped or Patterns.

5. Images Type

Selects all placed files of the file type defined by the menu: TIFF, ArtPro files, EPS, TIFF, TIFF/IT or PDF.

6. Images Color

Selects all images in the file based on the setting of the dropdown menu. Choices are: CMYK, RGB, grayscale, B/W (Black and white one bit images) or All.

Transparency

1. Blend Mode

Selects all objects with a certain **Blend Mode**. By setting the dropdown menu to **All**, you can select all objects containing any **Blend Mode**.

2. Opacity

Allows to select all objects with an opacity higher (At Least), lower (At Most) or Equal to the value entered.

3. Opacity Masks

Selects all objects containing a **Soft Mask**.

4. Objects/ Groups with Transparency

Selects all objects or groups containing any transparency.

5. Overprint

Selects all objects containing overprint.

Objects

1. Text

To select all text, or text smaller or bigger than the entered pointsize.

2. Vectorized Text

To select vectorized text. Text that was converted to outlines using the **Text to paths** action or during import is remembered.

3. Font Name

Selects all text items or text blocks containing the font entered in the input field. A text item containing more than one font, is selected completely if the entered font is used.

4. Screen Name

Selects all items using the screen settings entered in the input field.

5. Object Name

This helps to select objects with that name (eg. a layer from an imported MFG file which imported as a sub-layer in ArtPro.)

6. Strokes

To select all strokes or those smaller or bigger than the entered value.

7. Traps

Selects all traps created by **Automatic Trapping**. A trap that has been edited is longer be recognized as a trap and thus not be selected.

8. Barcodes

Selects all Barcodes containing bar code information. With some transforms (example: unproportional scaling), an invalid barcode is created. A warning is given when transforming and the bar code is longer selected with this function.

9. Thin Part Fixes

Select the thin part fixes done in ArtPro.

10.Slices

Selects all slices (used for Grid Warping).

11.Open Paths

Selects all paths that are open.

12.Blends

Selects all blends.

13.Link

To select all objects containing a link for ArtLink.

14.Objects Inside / Outside bounding box

In the **Objects** tab you can define to select or deselect objects inside or outside a defined pagebox.

15.Layers

This allows to select or deselect objects that are in a **Layer** that matches (or does not match) the entered name. The layer name can contain wildcards.

Selections

The radio buttons at the bottom define whether the defined objects should be selected or deselected.

Using **Within Selections** in combination with **Select** searches for objects that match the defined criteria inside the current selection. Objects that do not match these criteria are deselected.

Using **Within Selections** in combination with **Deselect**, deselects the objects inside the selection that match the defined criteria. Objects that do not match these criteria remain selected.

The **Fail if Number of Selections...** option allows to have the action fail if the number of selections is higher than, lower than or equal to the entered value. This can be used to have the action fail if any text is smaller than 7 pt, for example.

Separate

This action separates the file or repetitions into its separations. The separations are placed under and next to each other in a single ArtPro file. You can define the **Vertical Gap** and **Horizontal Gap** to be left in between the separations and the number of columns to be used (that is, the number of separations to be placed next to each other). In order to print a “separated” file like this, the option **All Separated Pages on Same Film** should be selected, in the **More Settings** of the **Print to File** action.

Set Station Numbers

The **Set Station Numbers** action is based on the **Station Number** part of the **Step And Repeat Nested** action. It allows to set station numbers on an existing step and repeat file. For more information on Station Numbers, we refer to the ArtPro manual.

It is possible to define the origin for the **Offset**, using the square buttons. By default, this is set to place the Station Numbers centered on the bounding box of a station.

The **File Naming Template** can be used to define the text of the Station Numbers to be added.

Spread

This action allows to apply a spread on all objects or on the selection. The action is based on the **Interactive Trapping** function in ArtPro. For a more elaborate description, we refer to the ArtPro documentation.

When a **Spread** is applied on an object:

1. The paint style of the original object is changed to 0% for every separation.
2. A slightly larger **Spread** object is generated.
3. The **Spread** object gets the paint style of the original object, BUT changed to “trans” for every separation that had a 0% value.
4. If the original object had a blend mode, the original and spread object are grouped and the blend mode is applied on the group.

Tabular Step and Repeat

The **Tabular Step and Repeat** action gives access to ArtPro’s tabular step and repeat module from within the ArtPro Action List Editor. For a full description we therefore refer to the ArtPro manual.

This action can be used to Step and Repeat not only ArtPro files but also TIFF files, EPS files, DCS files and PDF files. Hence any of these file formats could be handled by an action list that starts with a **Tabular Step and Repeat** action.

Equally, the **Tabular Step and Repeat** action could be preceded by a **Select File** action which would select a file from a folder.

This would typically be the case, when an ArtPro file and a **Regmarks** file are bundled in a folder for **Tabular Step and Repeat**. The **Select File** action would then select the ArtPro file (or any other of the supported formats) that needs to be stepped, and be followed by a **Select File** action to select the **Regmarks** (ArtPro) file that needs to be used to generate the register marks.

The first drop-down menu, **Use Box**, allows to determine which page box (example: media box, bleed box, trim box, etc.) needs to be used as reference for the Step distance. For example: if the Trim box is selected, the step distance horizontally and vertically is based on the size of the Trim box (unless **Freeze Step** is selected).

Step and Repeat Settings

1. Count

Amount of instances of the file, horizontally and vertically.

2. Begin Pos.

Horizontal and vertical offset to the ruler position for the first instance of the document to be stepped.

3. Step

Only when in combination with the **Freeze Step** option, this fields determine the offset between the steps. When left (0,0), the step distance is actually based on the selected pagebox (in the example: the Trim box) of the file that is stepped.

4. Angle Setting

The **Angle setting** determines the rotation of the file to be stepped.

5. Cut Marks

The **Cut Marks** option adds cut marks for any of the stepped instances.

6. Keep Bleeds

Includes the bleed of the stepped file in the step and repeat document. Bleeds are always kept outside of the step and repeat job size. Note that bleeds overlap when the step distance is set to the Trim box.

7. Place Single-Separation ArtPro File in Single Sep

This option defines how to handle repeating ArtPro files with only one separation. If the option is not selected, objects in this AP file with 100% of the single separation are considered to be registration color and thus get 100% of every ink in the repetition. If this option is selected, the file is inserted in one single separation instead.

8. File & Font Settings

This gives access to the root folders, folder mapping and fonts folders when the file to be stepped is an ArtPro file. For a description of this function, please refer to the **Open ArtPro File** action.

a. Color Management Settings

This setting determines if color management has to be applied for the step and repeat job. When enabling color management, it is possible to select a color management settings file, previously saved by ArtPro. This file contains all ICC profiles and settings to determine how color has to be treated throughout the rest of the action list (example: proofing of the step and repeat document), including ICC profiles for proofing, printing, etc.

Text to Paths

This action automatically converts all text or the selected text to outlines. Note that when importing PS or PDF files, text could already have been converted to outlines if the **Convert to Real Text** option has not been set during the **Importing** action. This is especially useful when ArtPro files are used and files need to be exported to environments where the fonts are not available.

Transform

This action applies a transformation to objects in the document. The transformation is done either to all objects or to the selected objects only (see **Selector**).

Following transformations can be executed:

- Move horizontally and vertically
- Scale horizontally and vertically
- Rotate
- Shear

Options

1. Repeat

The repetition field allows to execute the transformation multiple times.

2. Copy

The **Copy** option copies the contents of the file before executing the transformation. If at the same time repetition is set there will equally be multiple copies.

3. Apply on

a. Path, Paint and Soft Mask

The **Path**, **Paint** and **Soft Mask** options determine if the objects and their contents are transformed or only the contents without the objects or the objects without the contents. Suppose a clipping path with an image would be transformed and Path is not checked, then only the image itself is transformed and not the clip path.

b. Notes

Transforms the notes along with the rest of the file, so that the positioning of the notes inside the file remains the same.

c. Center Paint

The **Center Paint** option is only available if the **Apply on: Paint** and **Soft Mask** is off. It transforms the path only, but then reposition the paint based on the center point of the path: the point in the paint that was on the center point of the original path is placed on the center point of the transformed path. For example: this can be used when scaling or distorting Step and Repeat files, without scaling the contents of the paths, but still retaining their position inside the paths.

4. Scale Strokes

If the **Scale Strokes** option is unchecked, the stroke width does not change when scaling. If the option is checked, the stroke remains a stroke and its width is changed for the same percentage as the parent path was scaled. Thus, scaling an object with a 3 mm stroke to 150%, results in an object with a 4.5 mm stroke. When scaling is unproportional, the stroke width is scaled by the geometric mean of vertical and horizontal scaling. This means the stroke is scaled proportionally. The geometric mean is the square root of the product of vertical and horizontal scaling.

5. Use Ruler

For scaling and rotating, the **Use Ruler** option defines to use the center of the ruler as the center for rotation and scaling. If the option is unchecked, the center of the selected objects is used.

Trim & Bleed

This function calculate trim and bleed for all objects in the file.

The action is typically when performing in Step and Repeat and has been described in the **Nested Step and Repeat** section.

Unite

Unite creates the union of the selected paths.

Select the necessary paths with the **Select Objects** action and then apply the **Unite** action.

For a more detailed description, please refer to the ArtPro manual.

Update ArtLink

The **Update ArtLink** action is comparable to the **ArtLink** action. All of its settings can be found in the **ArtLink** dialog box.

While the **ArtLink** action creates as many versions as there are records in the database, the **Update Artlink** action does not create any files or versions. It modifies the open ArtLink file using the information from one specific database record (defined in the **Record Number** field) or from information included in the XML file.

Additional Key Values

The **Additional Key Values** allow to define keys to be used in the **File Naming Template** when updating the file. These keys are defined from the action list rather than coming from the database. For example: this can be useful to set the name of the operator.

Vector Trapping

This action gives access to ArtPro's **PowerTrapper** module from within the ArtPro Action List Editor and performs the trapping on all elements in the file.

For a full description of all the different settings we refer to the **Automatic Trapping** section in the ArtPro manual.

Since this action actually generates additional objects containing the trapping information it is possible to view and edit the traps in ArtPro or to export them to PDF or composite PS.

1. Remove traps first

If the **Remove traps first** checkbox is selected, all existing traps are removed before trapping is calculated. The selection of the original objects is kept, allowing to immediately trap the same selection again with different parameters.

2. Trapper Version

The Version defines what trapping routine has to be used. Version 1 (Packaging) starts to trap in the background and works its way to the front, while version 2 (Commercial) starts in the front and works towards the back. This second routine should be faster in general, though version 1 might

give more consistent results, as it has a longer development history. Therefore we recommend to use version 1 for packaging and labels and version 2 for commercial work.

4stage Trapper uses a totally different kind of procedure. Unlike Version 1 and 2, it does not work object based and only calculates traps for where different colors actually meet.

The 4stage Trapper is the only good way to trap objects containing transparencies. The traps made by the 4stage trapper are defined using transparencies as well.

Distance

1. Into Color

Defines the spread distance for trapping colors under colors. When you enter a value for **Into Color**, it is automatically copied for **Into Black** as well.

2. Into Black

Defines the spread distance for trapping colors under black. If **Into Black** needs a different value than **Into Color**, enter **Into Color** first and then **Into Black** because by default the value from the first field is copied into the second. Sometimes you may want a bigger spread under the black color, example, for a centerspread. To create a centerspread (center trapping in black, example, for comic strip drawings), the spread value should be set to slightly more than half the width of the thickest black line. When the spread value is higher than half of the black line width, in the middle, where the spreads meet from both sides, the spreads is limited to the center and does not overlap.

3. Into CT

Defines the spread distance for trapping colors to pictures. For Version 1 and Version 2, the color type can be flat color or linear gradation. Multistep gradations and other pictures are not trapped automatically. The 4stage trapper can trap anything.

4. Center Error

These fields specify the accuracy for the centerspread (see example on the right). When **Off** option is selected for **Center Error** means no centerspread is calculated. The percentages, together with the distance, define the precision for center trapping. The lower the percentage, the more precise. Example: with a trapping distance of 0.5 mm and 8% accuracy, the trap centerline is at a maximum distance of 0.04 mm off the real centerline. Keep in mind that increasing the accuracy increases the trapping time! Also note that a smaller trap distance results in more accurate centerspreads, so do not set the distance higher than necessary. The accuracy cannot be different for normal colors and for black for the 4Stage trapper or if the trapping distances are equal. This means that if the black trap distance is changed to equal the normal trap distance, the accuracy is automatically set to the one for the normal colors. General settings are 8% for normal colors and black. If you have a job where centerspread under black is important, put the accuracy to 2%, 1% or even 0.5%.

5. Corners/ Miter

This field defines the trap corners, which can be mitered, rounded, bevel or square. The miter limit can be set to a maximum of 4.

Direction

1. Separation Threshold

It indicates the minimum difference that must exist between touching colors before they are trapped. There must be a difference more than the **Separation Threshold** in at least one separation in each direction. The usage for the different versions is described below.

2. Black Margin

The **Black Margin** defines what colors are treated as black. From what percentage should an object be trapped with the distance **Into Black** and from what percentage **Into Color**. The default value is 85%, meaning a box of 100% yellow is spread into a box of 90% black with the distance

of **Into Black**. If the black in this example was only 80%, it would trap at the distance for **Into Color**. It actually does not just apply for black but in general for all dark colors. If the black value in color on proofer multiplied with the percent value in paint style is higher than the black margin, the distance into black is used.

3. Trap Intensity

The Intensity, by default 100%, defines the "intensity" of the trap and can be adjusted according to the needs. When a job is trapped, the traps should be as inconspicuous as possible. With a lower intensity, the traps become lighter and so less visible. But keep in mind that if the intensity is too low, the trapping will not be correct.

4. Keep Away Rich Black

A rich black is a black element, to which another color was added, to make it look darker. Usually, a tint of cyan is added to the black, but this could be done with any separation, cmy or spot. The idea is to make 100% black look darker on print, as black, like other inks, is a transparent ink, which sometimes looks more like a dark gray than black.

If the Keep Away Rich Black option is on, a rich black area is trapped so that the supporting color is kept away from the edge of the black, on white as well as on color. This means that on a white element, a trap line is created with only black and no supporting color.

5. Split Gradations

Vignettes are by default spread or choked as one object, that is either completely spread or completely choked. In theory, though, this trapping is not always correct. A gradient could be darker than the background at one end (choke) and lighter at the other (spread).

This option makes it possible to partially spread and choke one vignette. When it is active, the calculation of the automatic traps divide the vignette path into several parts depending on the colors. On the basis of the light-dark principle the different parts are then spread or choked. Hence, some parts of the same vignette can spread to the outside whereas other parts can choke to the inside.

6. Gradation Mode

When set to **Step into Gradations**, the program decides on the trap direction according to the light-dark principle. When set to **Always towards Flat**, gradient is always forced to spread into flat colored backgrounds, even if this background is lighter.

This option is not applied for multistep gradations. Forcing a multistep gradient into a flat color has to be done manually. The **4stage Trapper** always uses **Step into Gradations**.

7. Trap Images / All Separations (Packaging and Commercial only)

In order to trap mapped pictures automatically, enter a distance for **Into CT** and activate the **Trap Images** option. By default it is switched off so that selected pictures are ignored during autotrapping.

Using Version 1 & 2, mapped pictures trap with flat colors or linear gradations, not with multistep gradations or other pictures. The autotrap result is a mapped picture with a minimum value added for the underlying colors or a gradient transparent on top of the picture.

When all separations are active (default), it traps all the separations of the color, whether they are in the picture or not. When the option is not active, the image is only trapped into separations that are not in it.

For example, a cmyk image is put on top of a cyan background and a pantone 032. With the option unchecked, it only traps into the pantone 032. With the option checked, it traps in both. By default, all colors are spread in the image and black is choked. This can be adjusted using the Black Margin (see earlier). The 4stage Trapper always traps **All Separations**.

8. Image to Image (4stage Trapper only)

Using the 4stage Trapper, images are always trapped towards linear. If the **Image to Image** option is checked, images are also trapped towards images.

9. Image Mode (4stage Trapper only)

When set to **Step into Image**, the program decides on the trap direction according to the light-dark principle. When set to **Always towards images**, the background is always forced to spread into the image, even if this image is lighter.

10. Trapping Pairs

The **Trapping Pairs** button opens the dialog that holds the actual trapping pairs. A trapping pair is represented by a **From** color, a **To** color and a **Mix** color. The **From** and **To** are the original colors, whereas the **From** is spread into the **To** color. The **Mix** is the color of the actual trap.

2.12 The PitStop Profile Editor

You can easily create and manage PitStop PDF **Profiles** and **Action Lists** using **PitStop Profile Editor** from Automation Engine.

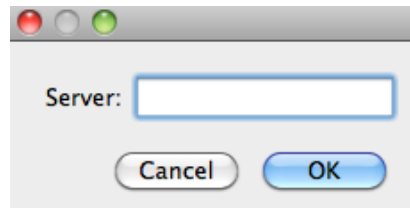
Read more about PitStop in <http://www.enfocus.com/en/support/knowledge-base>

2.12.1 Setting Up the PitStop Profile Editor

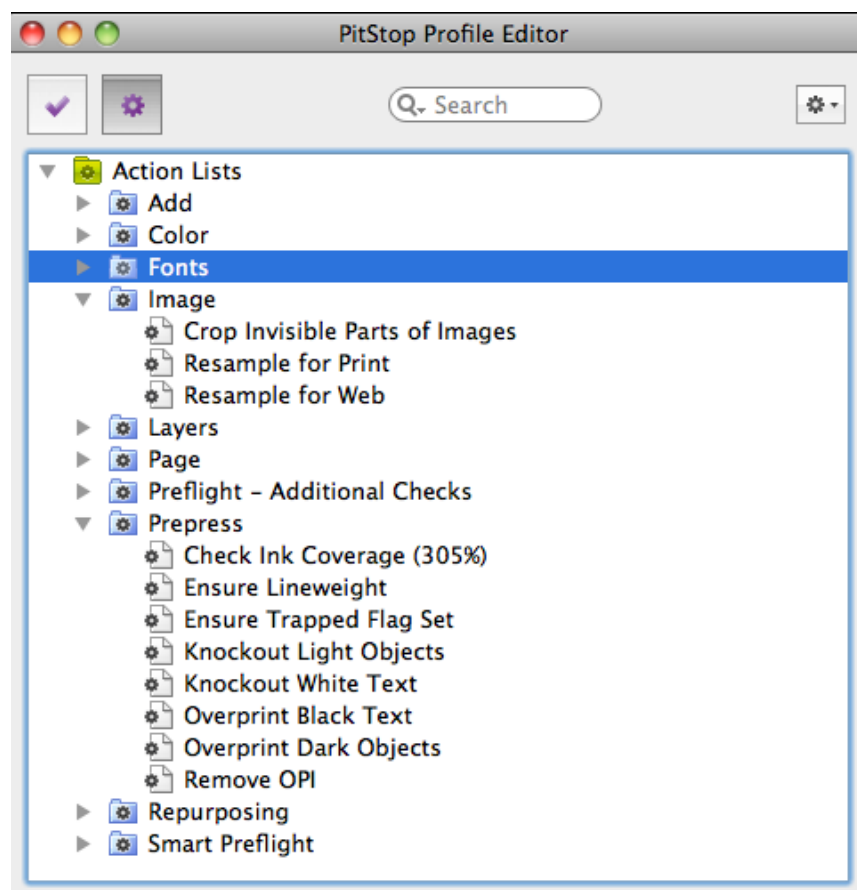
You can download **PitStop Profile Editor** the Client Apps section of of Automation Engine server web page. You can access it by browsing to `http://<name of your server>` or `http://<name of your server>:9999`. Read more in [Main](#) on page 208

1. Download and install the **PitStop Profile Editor** on your client computer where the Pilot is installed.
 - On Windows:
 1. Double-click on the Automation Engine PitStop Profile Editor.exe.
 2. Click **Next** on **Esko Installshield Wizard** on your client machine.
 3. Confirm on Installshield Wizard that you want to install PitStop Profile Editor by clicking **Yes**
 4. **Browse** to the location where you want the Wizard to create subdirectories. The default location will be `C:\Esko`.
 5. Click **Next**.
 6. **Finish** the installation.
 - On Mac:
 1. Double-click on the downloaded Automation Engine PitStop Profile Editor.dmg.
 2. Double-click on the Automation Engine PitStop Profile Editor.pkg. The Assistant will guide you through the installation.
 3. Click **Continue**.
 4. Click on **Change Install Location** to specify the location where you want the subdirectories.
 5. Click **Install**. Fill in your credentials to confirm the installation.
 6. A message **The installation was successful** will appear. **Close** the installation.

2. Start the application from:
 - **All Programmes > Esko > Automation Engine PitStop Profile Editor > PitStop Profile Editor .**
 - **Applications > Automation Engine PitStop Profile Editor > PitStopProfile Editor .**
3. Fill in the host name of the Automation Engine server in the **Edit > Preferences** window. This will integrate the application with the PDF Profiles and Action Lists from this server.




You can now access the the PDF Profiles and Action Lists from the Automation Engine Server.

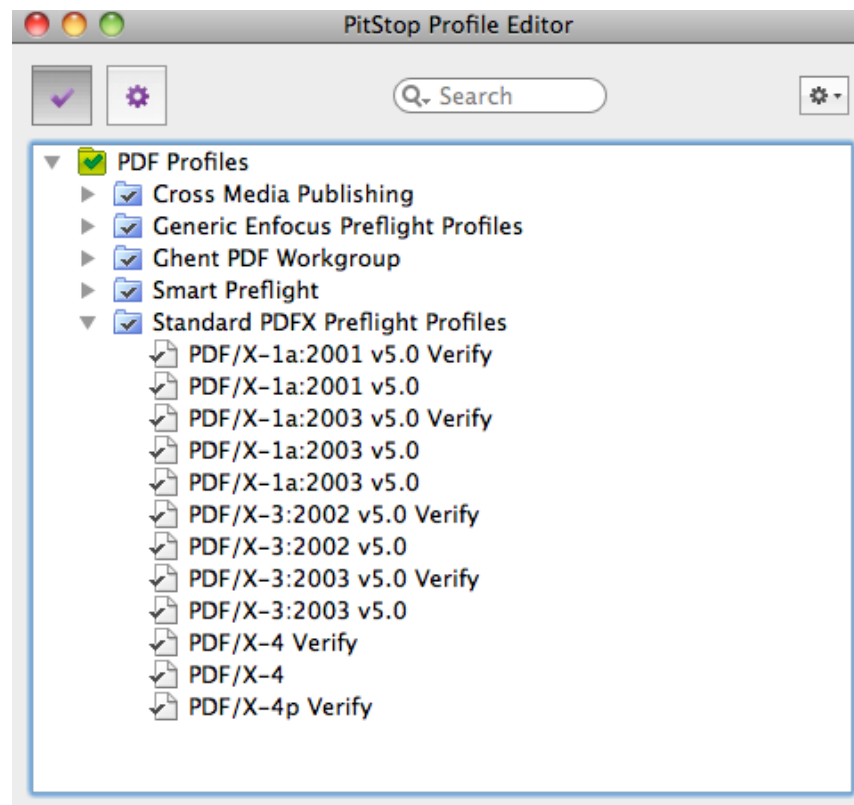



2.12.2 Using SmartNames in PitStop Profile Editor


You can use SmartNames in PitStop Profiles and Action Lists in Automation Engine Profile Editor. You can make use of PitStop's "Smart Preflight" and reduce the the number of Profiles and Action Lists. The PitStop Profile Editor loads the global SmartNames from the Automation Engine server that has been entered in the preferences.

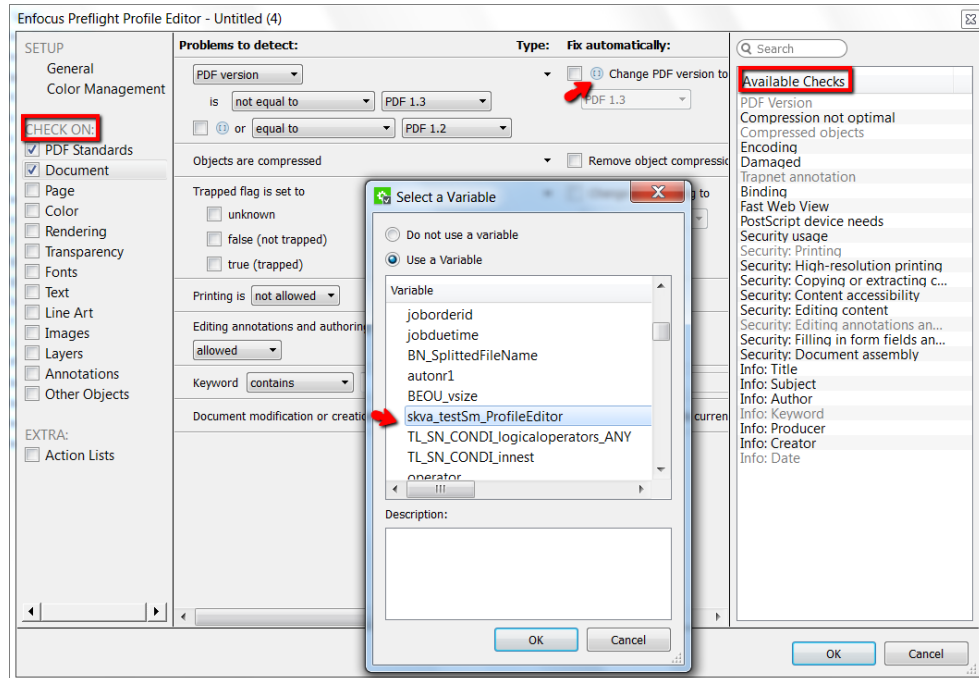
1. Create SmartNames in SmartNames View by clicking **New SmartName** icon  in the toolbar.
Read more in [Creating a SmartName](#)
2. Start the application from :
 - **All Programmes > Esko > Automation Engine PitStop Profile Editor > PitStop Profile Editor .**
 - **Applications > Automation Engine PitStop Profile Editor > PitStopProfile Editor .**


You can access the PDF Profiles and Action lists in the PitStop Profile Editor.



3. Select one of the **PDF Profiles** and click on the menu button  to expand menu options. From the following drop-down, choose **New**.
4. Enable one of the **CHECK ONs**.

- a) You can now choose from **Available Checks** pertinent to your **CHECK ON**.
- b) Click on **Actions**  and **Enable Variable Names**.



5. Click on the blue SmartName (Variable)  icon. Created SmartNames (Global) and System SmartNames will be available.
6. Choose the desired SmartName and Click **OK**. You can also give a brief **Description**.

You can use a SmartName in fields requiring:

- text
- number
- measurements (size in "mm", "cm", "in", "p" (picas) or "pt" (points)).



Attention: Specify units in the measurement SmartNames (i.e. "mm", "cm", "in", "p" etc). . If the unit is not specified, SmartName value assumes points as its units. For example, SmartNames representing A4 dimensions, should resolve to "210 mm" and "297 mm". If the SmartNames resolves to "210" and "297", the dimensions will be taken as 74mm x 105mm while preflighting.

- toggling on or off.

Note: The SmartName value "0", "FALSE" or empty, corresponds to check box which is off. "1", "TRUE" or anything else corresponds to on.

2.13 The RunList Editor

The **RunList Editor** enables you to create and modify RunLists.

A **RunList** is a file used to collect several files into one Job Folder, or to make one final PDF out of several smaller PDFs (for example for imposition). It contains the names of the files to be collected or joined, their collecting / joining order and their original location.

You can use this for example with the [Wait for Files \(via JDF\) task](#). If you want to collect a number of files with different file names, you can create a RunList referencing those files, and feed that RunList to the **Wait for Files (via JDF)** task.

2.13.1 Setting Up the RunList Editor

1. Install the **RunList Editor** on your client machine (where the Pilot is installed):

- On Windows:

1. Open an Explorer window and type the following in the address bar: `\\Automation Engine Server Name\Automation Engine 10.1 Windows Client` (where `Automation Engine Server Name` is the name or IP address of your Automation Engine server).
2. Copy `Automation Engine RunList Editor 10.1.0.exe` on your client machine.
3. Double-click the `exe` file to install the application.

- On Mac:

1. In Finder, go to **Go > Connect to Server...** (or use **Command + K**).
2. In the **Server Address** field of the **Connect to Server** dialog, type `smb://Automation Engine Server Name/Automation Engine 10.1 Mac Client` (where `Automation Engine Server Name` is the name or IP address of your Automation Engine server).
3. Copy `Automation Engine RunList Editor 10.1.0.dmg` on your client machine.
4. Double-click the `dmg` file to install the application.

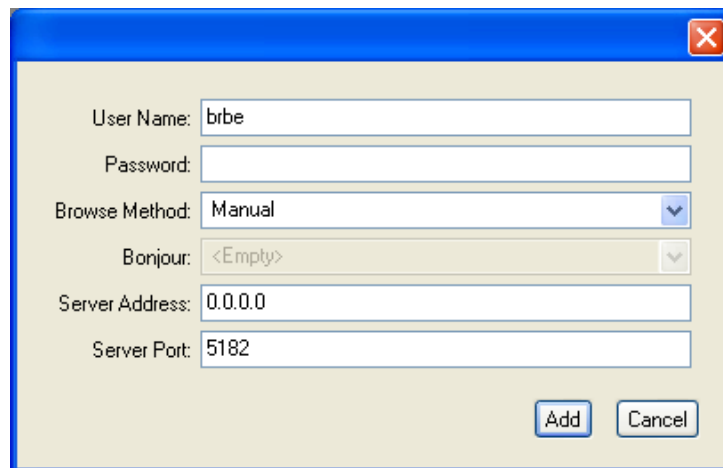
2.

Double-click the RunList Editor shortcut  on your desktop to open the RunList Editor application.

3. Go to **Edit > Preferences** (on Windows) or **RunList Editor > Preferences** (on Mac) to open the **Preferences** window.

4. Connect the RunList Editor to your Automation Engine server:

- a) In the **Servers** tab, click the  button at the bottom of the **Servers** list.
In the pop-up that opens:



The screenshot shows a dialog box with a blue title bar and a close button in the top right corner. The dialog contains the following fields and controls:

- User Name:
- Password:
- Browse Method: (dropdown menu)
- Bonjour: (dropdown menu)
- Server Address:
- Server Port:
- Buttons: and

- b) enter the **User Name** and **Password** you use to login to the Pilot,
- c) select **Manual** in the **Browse Method** list,
- d) enter your server's name or IP address in **Server Address**,
- e) fill in the **Server Port** used by your server.

This is the port you defined in Configure for Shuttle communication (see [Automation Engine Setup for Shuttle](#) on page 48).

- f) Click the **Add** button.

This adds a connection to your server in the **Servers** list. You should see a green dot if the connection was successful.

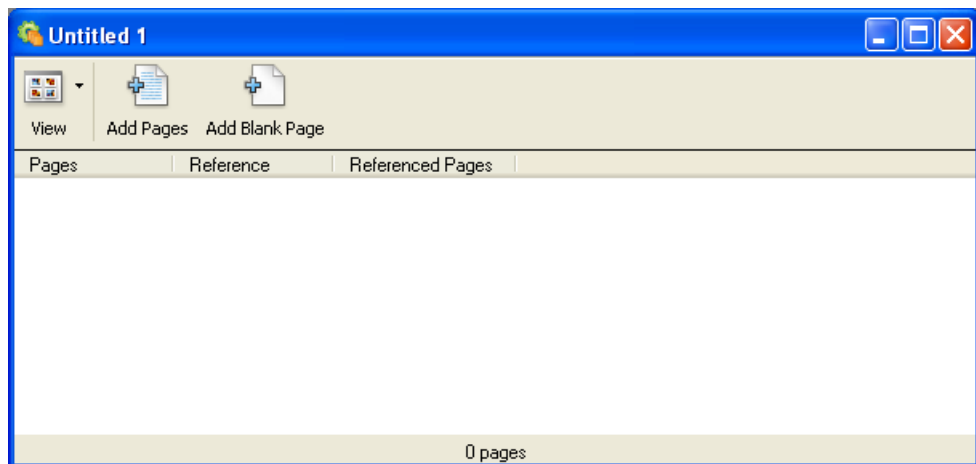
5. In the **Units** tab, choose the units to use when submitting to a workflow that has public parameters:
 - a) Choose to define the **Distance** in **Millimeter**, **Point** or **Inch**.
 - b) Choose to define the **Resolution** in **Pixels Per Millimeter** or **Pixels Per Inch**.
 - c) Choose to define the **Screen Ruling** in **Lines Per Millimeter** or **Lines Per Inch**.

2.13.2 Creating a RunList

1. Go to **File > New** to create a RunList.

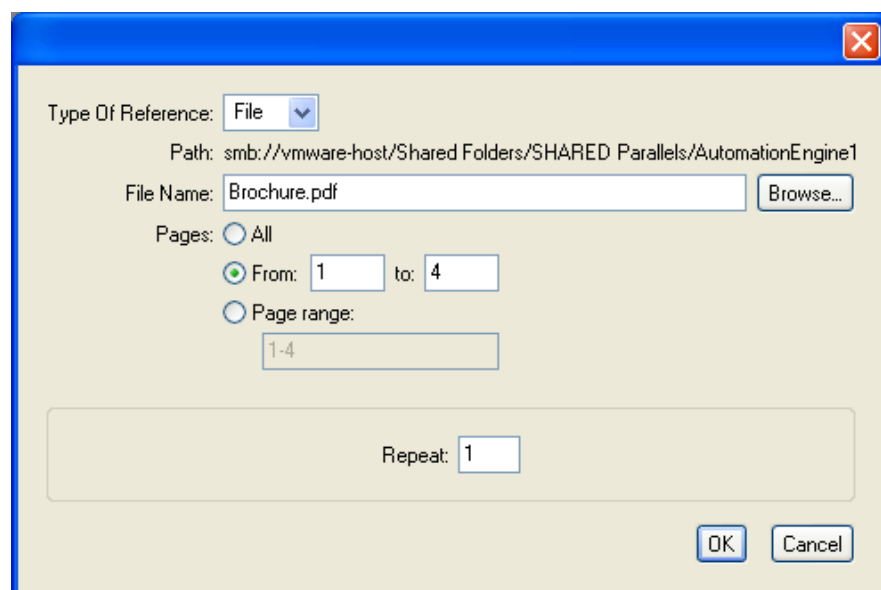
You can also use **Ctrl + N** (Windows) or **Command + N** (Mac).

This opens the following dialog:



2. Add files to the RunList:

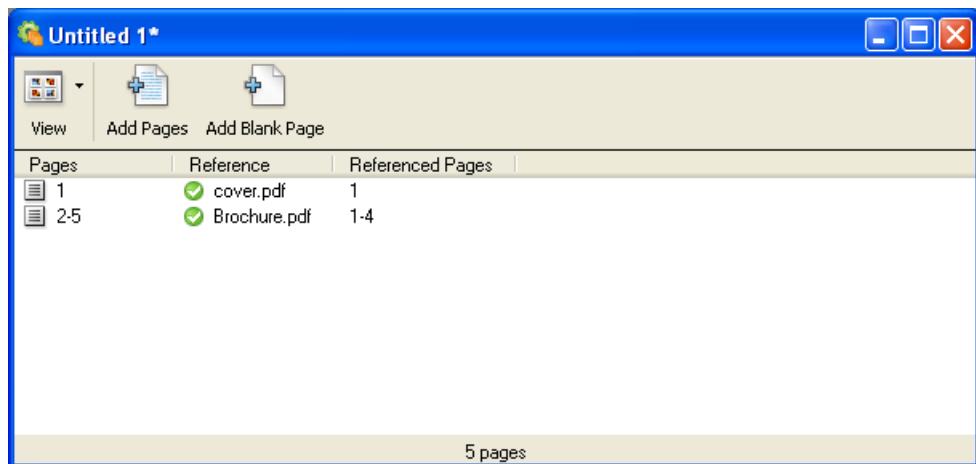
- Click the **Add Pages** button in the toolbar.
- In the pop-up that opens, use the **Browse...** button to browse to your file.
- Define which of the file's pages you want to add to the RunList (for example, if you want to add the pages 1, 2, 3 and 7, enter 1-3, 7 in **Page range**).



Note: By default, the **From 1 to ...** option is selected, selecting all pages.

- Click OK.

This creates an entry for every file you add.



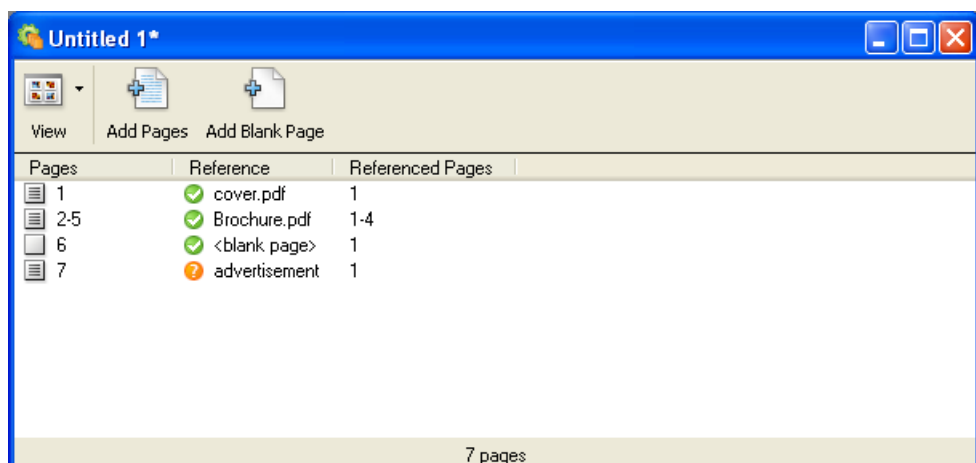
You can see the RunList's total number of pages at the bottom of the dialog.

Note:

You can also drag and drop one or several file(s) into the dialog to add them to the RunList. By default, this adds all of the files' pages, but you can double-click an entry to edit it.

3. If necessary, click the **Add Blank Page** button in the toolbar to add blank pages to the RunList.
4. You can also add a placeholder for a file that is not available yet:
 - a) Click the **Add Pages** button.
 - b) Type the file name in the pop-up's **File Name** field.
 - c) Click OK.

The RunList dialog will show an orange "?" icon, to indicate that you still have to provide a link to the file.



Tip:

- You can also create placeholders for several files that are not yet available. See [Adding a Number of Placeholders](#) on page 790.
- You can use **Undo** (**Ctrl + Z** (Windows) or **Command + Z** (Mac)) or **Redo** (**Ctrl + Shift + Z** (Windows) or **Shift + Command + Z** (Mac)) when creating or editing a RunList.
- To delete a RunList entry, select it and press **Delete** (Windows) or **Command + BackSpace** / **Delete** (Mac).
You can use **Ctrl + A** / **Command + A** first to select all entries for deletion.
- To duplicate a RunList entry, select it and press **Ctrl + D** (Windows) or **Command + D** (Mac).

5. When you are done, go to **File > Save** (or use **Ctrl + S** (Windows) or **Command + S** (Mac)) and give your RunList a name.

RunLists are saved in the JDF format.

Adding a Number of Placeholders

To create placeholders for several files that are not yet available:

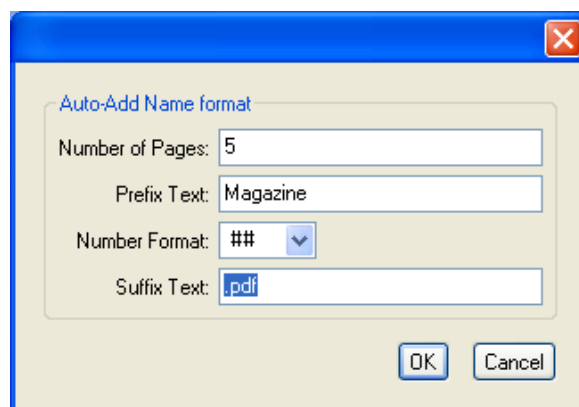
1. Go to **RunList > Auto Add...**

This opens the **Auto Add Name Format** dialog.

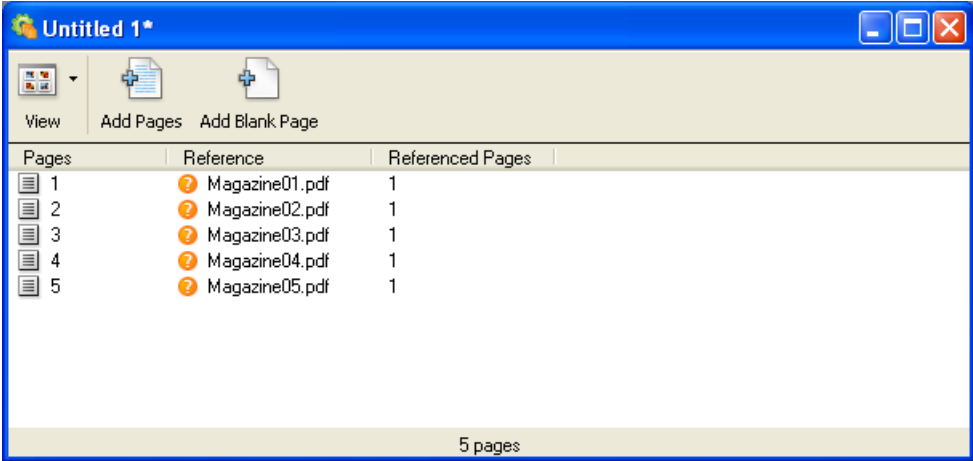
2. In **Number of Pages**, enter the number of placeholder entries to create.
3. Use the **Prefix Text**, **Number Format** and **Suffix Text** fields to define the name to give to the placeholders.

The **Number Format** specifies the number of digits to use in the file name.

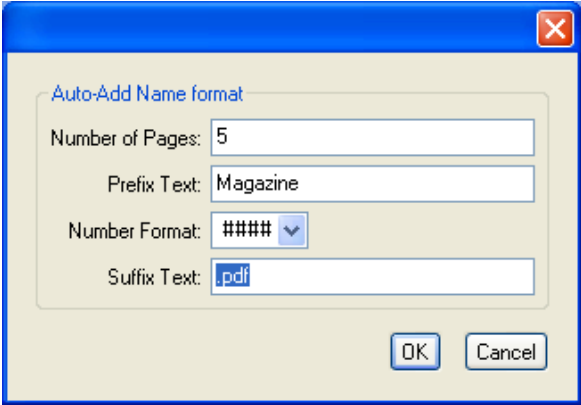
For example, this format:



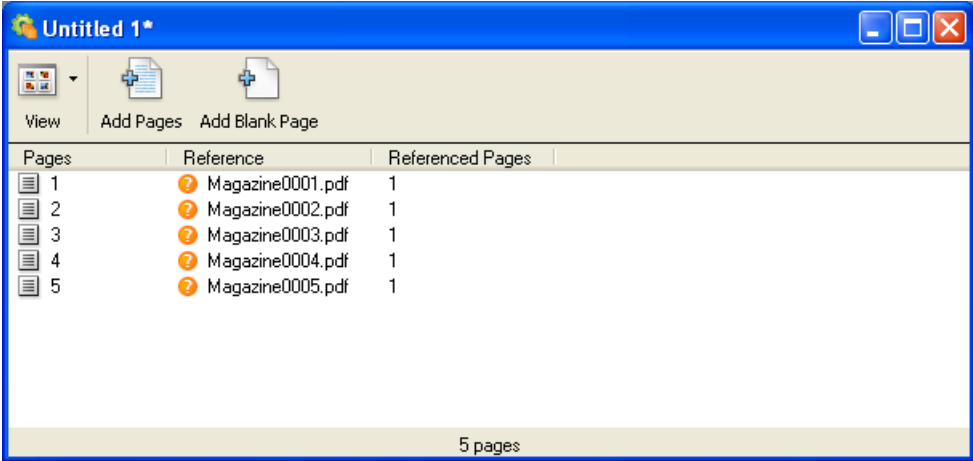
generates the entries below:



While this format:



generates those entries:

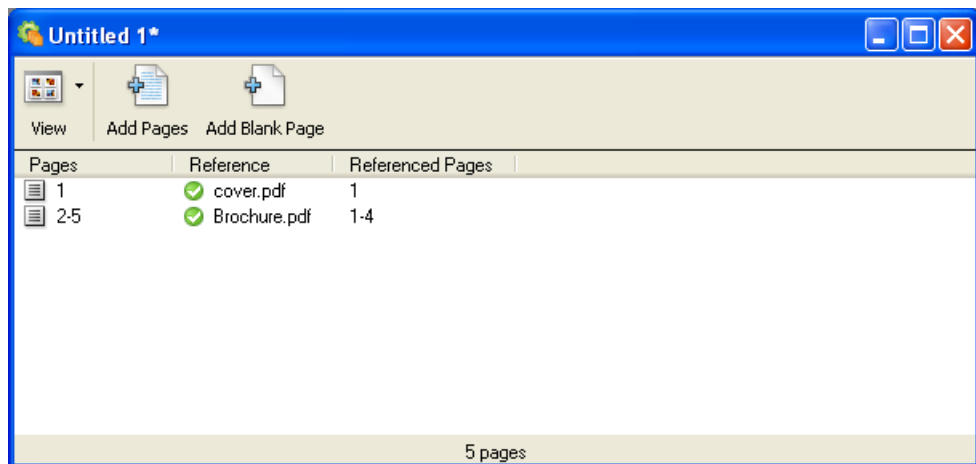


2.13.3 Viewing the RunList Files



Context for the current task

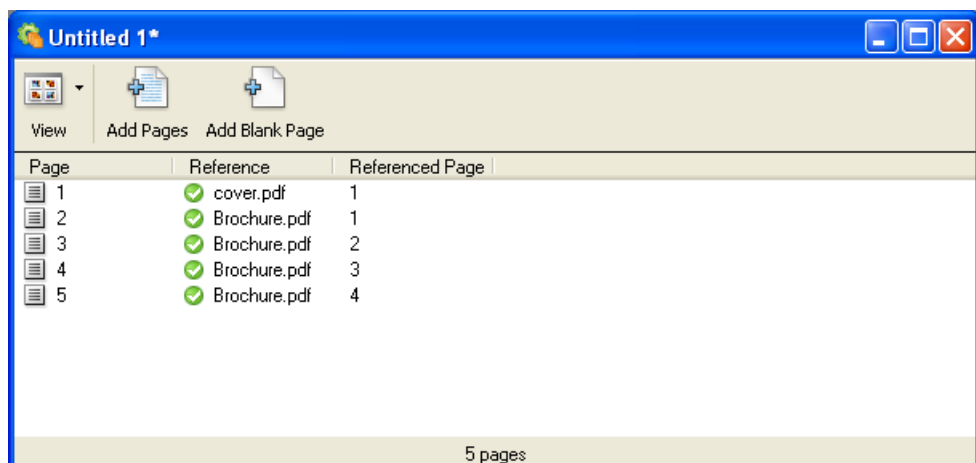
-

To view the pages grouped by file, click  and select **RunList** (Windows), or click  (Mac).



-

To view each page individually, click  and select **Page List** (Windows), or click  (Mac).



2.13.4 Changing the Files' Extension

- Go to **RunList > Change Extension...** to change the extension of all your individual files at once.

For example, you can use this to create the RunList from original InDesign files, then change the extension to PDF to use individual PDF files in your workflow.

Note: If you don't have the files with the new extension at the location defined in the RunList, they will have an orange "?" icon.

2.13.5 Sorting the RunList's Files

- Select entries in the main dialog and go to **RunList > Sort Selection By Name** to sort them alphabetically.
- You can also drag the entries to reorder them.

This updates the order of the RunList pages automatically.

Pages	Reference	Referenced Pages
☰ 1-5	✔ 4463_Neurologen.pdf	1-5
☰ 6-9	✔ Brochure.pdf	1-4

Pages	Reference	Referenced Pages
☰ 1-4	✔ Brochure.pdf	1-4
☰ 5-9	✔ 4463_Neurologen.pdf	1-5

2.13.6 Submitting the RunList to a Workflow

Once your RunList is ready, you can submit it to an Automation Engine workflow of your choice.

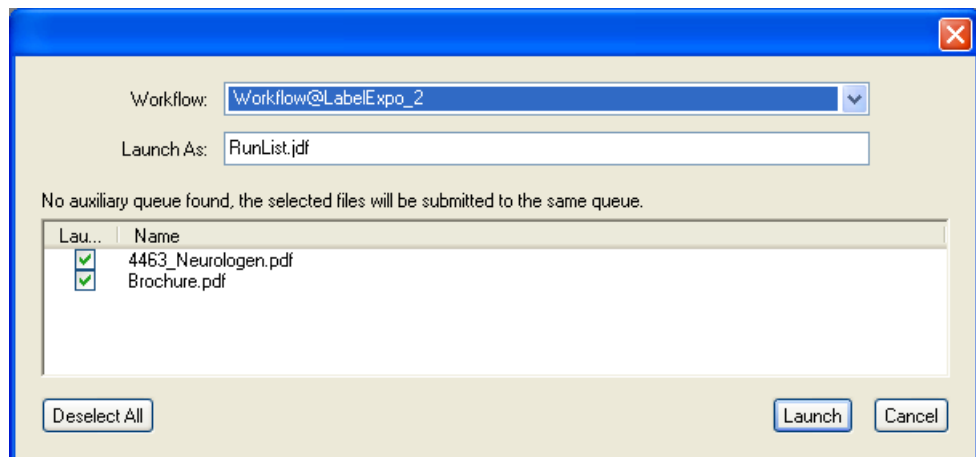


Attention: Make sure that:

- The RunList Editor is connected to your Automation Engine server (see [Setting Up the RunList Editor](#) on page 786).
- The workflow you want to submit your RunList to is public (see [Making Your Ticket Public](#) on page 49 in the User Guide).
- The RunList and the individual files it references are accessible to the Automation Engine server (saved on an Automation Engine container).
- You have replaced any placeholders in the RunList by the actual files.

1. Go to **File > Submit...**

This opens the following dialog:



2. Select the workflow you want to launch on your file in the **Workflow** list.
3. In **Launch As**, you can change the name of your RunList before launching it into the workflow if desired.
4. If you also want to submit the individual files referenced in the RunList, select their check boxes. For example, when submitting to a workflow starting with a **Wait for Files (via JDF)** workflow step:
 - submit the referenced files along with the RunList from the RunList Editor,
 - in the **Wait for Files (via JDF)** parameters, use **[URL of Folder of Input]** for **Look in Folder**.

The referenced files will be copied to a temporary folder alongside the RunList, and picked up by the workflow step.
5. Click **Launch**.

2.14 The Script Runner

2.14.1 Scripting for Script Runner on Mac OS

A Script Runner on Mac OS supports AppleScript and Shell Script to automate operations.

Note:

Sample scripts are provided as-is with no warranty of fitness for a particular purpose. These scripts are solely intended to demonstrate techniques for accomplishing common tasks. Additional script logic and error-handling may need to be added to achieve the desired results in your specific environment.

It is up to the user to verify that his intended use of the offered automation functionality is compliant with any third party license agreement and/or other restrictions applicable to any non-Esko products.

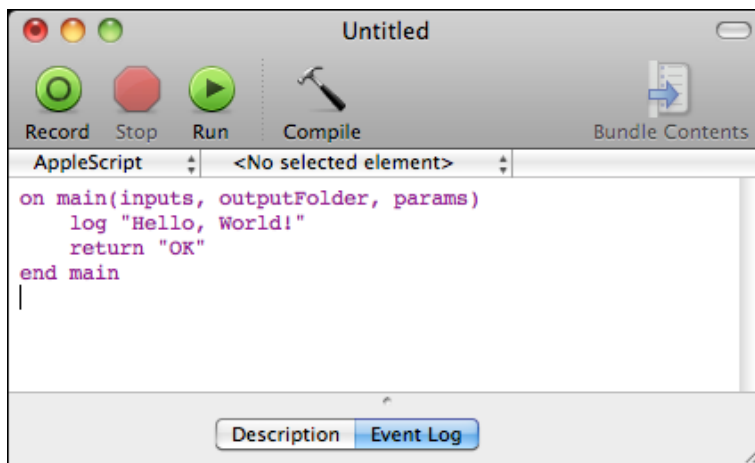
Using AppleScript on Script Runner

AppleScript is a scripting language that makes direct control of scriptable applications and of many parts of the Mac OS possible. An AppleScriptable application is one that makes its operations and data available in response to AppleScript messages, called Apple events.

We recommend using AppleScript because:

- it is highly integrated into the Mac OS
- it is supported by a lot of third party applications
- it has a high level of accessibility for scripting beginners

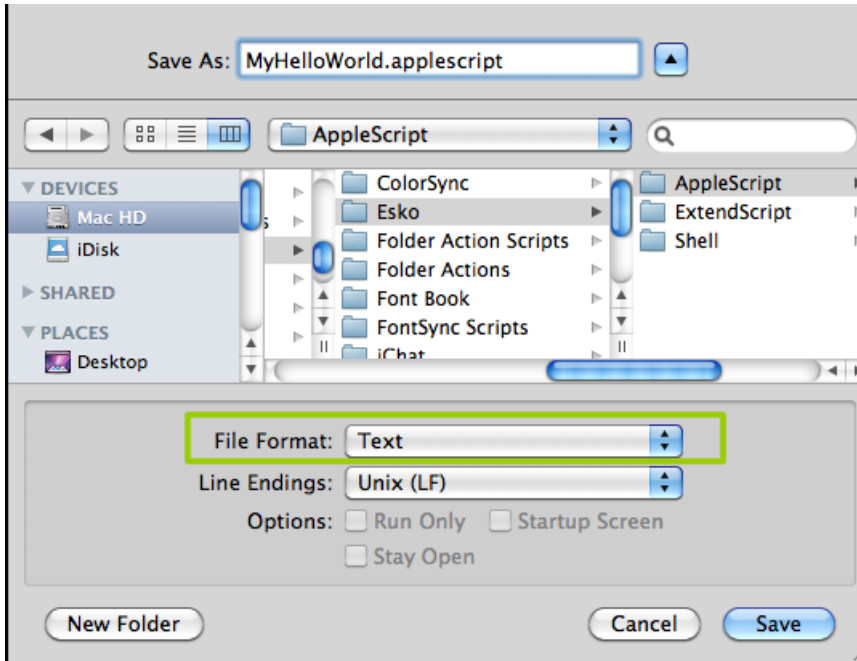
1. Open the AppleScript Editor and add following code.



Options	Description
main	This function will be called by the Script Runner. Only the code in this main function gets executed.
inputs	The first argument of the main function: a list of input file paths (type: list of strings).
outputFolder	Second argument of the main function: the folder where AE expects the script's result files. AE will continue the flow with the files you write in this folder. If you leave this folder empty, AE will continue the flow with the inputs of the Run Script task (type: string).
params	Third argument of the main function: additional script parameters injected into the script via the Run Script ticket (type: list of strings).
log	Extra log information in the Run Script task details
return "OK"	This will communicate with the Run Script task that everything went fine. Other

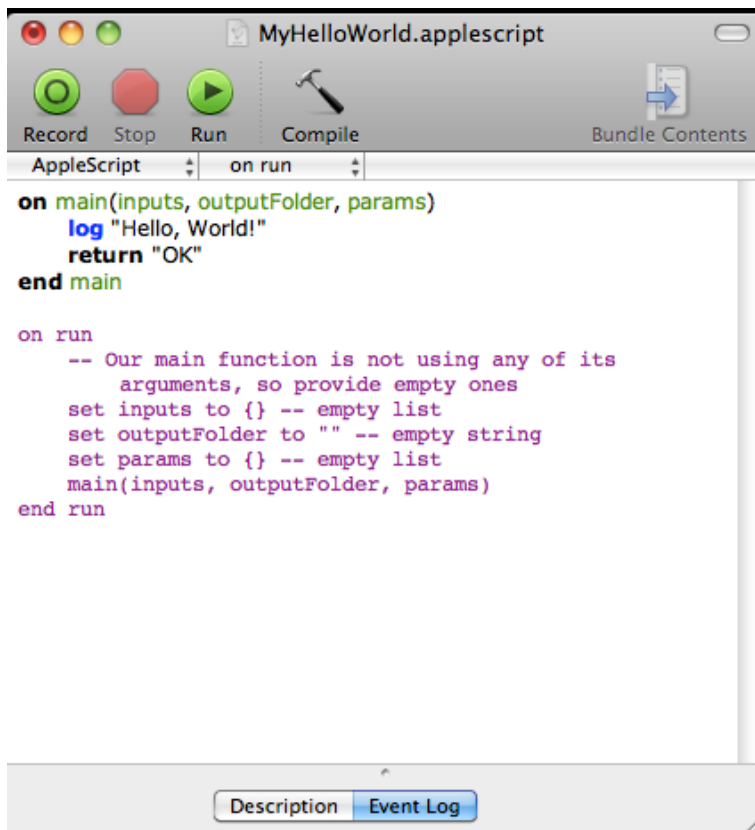
Options	Description
	possibilities are <code>return "Warning"</code> and <code>return "Error"</code> .

2. Save this code as an AppleScript text file in the Script Runner's AppleScript folder (**default: / Library/Scripts/Esko/AppleScript**) or in the Automation Engine AppleScript folder.



Note: Script Runner supports 'Text' format. Therefore it is essential to change the file format to 'Text'.

3. You can add following code to test this script locally in the AppleScript Editor. Save the file and click Run to execute the script.

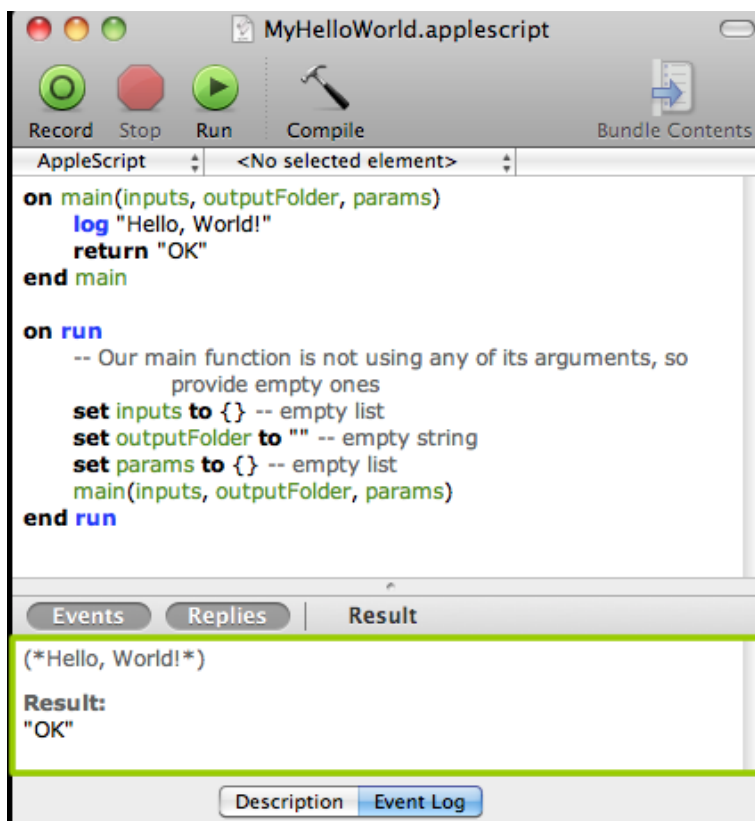


The screenshot shows the 'MyHelloWorld.applescript' editor window. The toolbar includes 'Record', 'Stop', 'Run', 'Compile', and 'Bundle Contents'. The script code is as follows:

```
AppleScript
on run
on main(inputs, outputFolder, params)
    log "Hello, World!"
    return "OK"
end main

on run
    -- Our main function is not using any of its
    arguments, so provide empty ones
    set inputs to {} -- empty list
    set outputFolder to "" -- empty string
    set params to {} -- empty list
    main(inputs, outputFolder, params)
end run
```

The 'Event Log' tab is selected at the bottom.



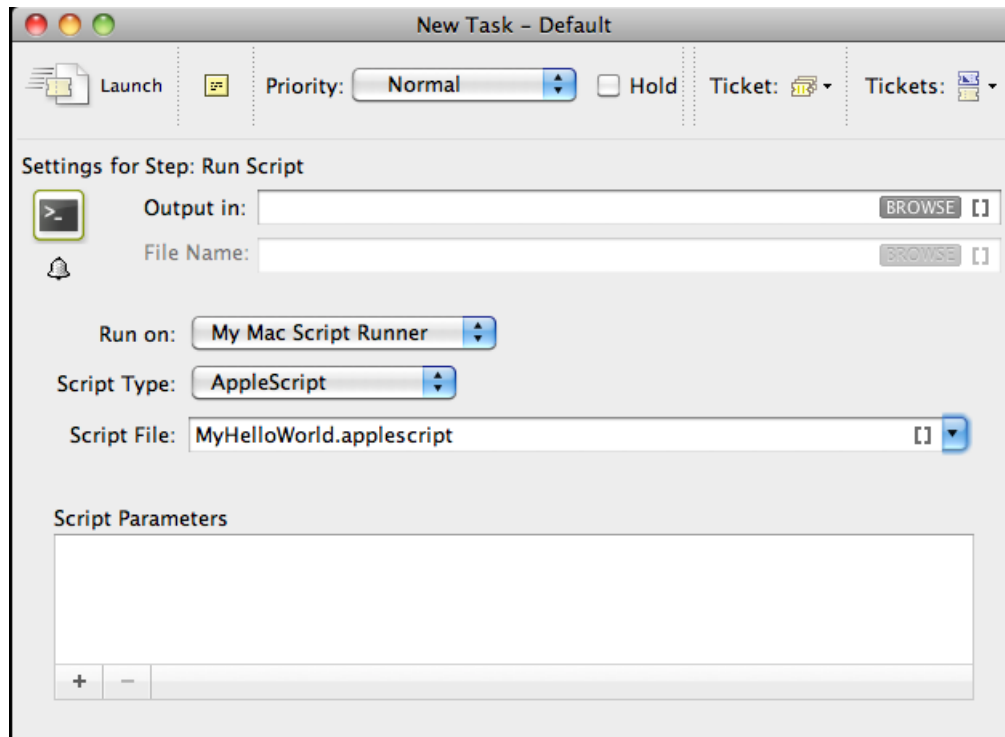
The screenshot shows the 'MyHelloWorld.applescript' editor window after execution. The toolbar includes 'Record', 'Stop', 'Run', 'Compile', and 'Bundle Contents'. The script code is the same as in the previous screenshot. The 'Result' tab is selected at the bottom, showing the output:

```
Events | Replies | Result
(*Hello, World!*)
Result:
"OK"
```

The 'Event Log' tab is selected at the bottom.

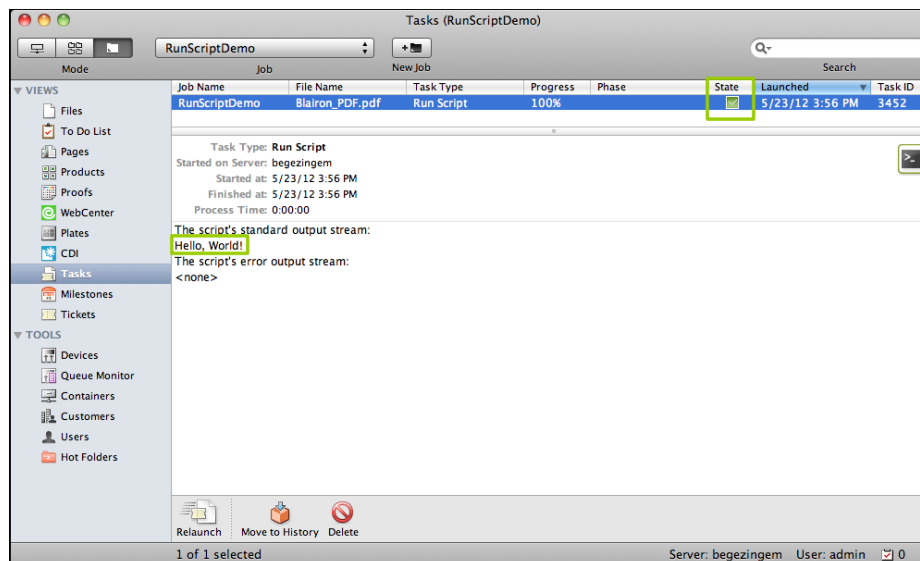
Notice the 'Hello, World!' and 'OK' result in the event log. The Script Runner does not attend to the test code in your script. It will execute the contents of the main function and ignore the rest. Therefore, you can keep your test code for future local testing.

4. Open the Automation Engine Pilot. Go to **Files** view where you can select a file and open a **New Task**. Choose the **Run Script** task, modify its settings and launch the task.



Read more in [Run Script](#) on page 469

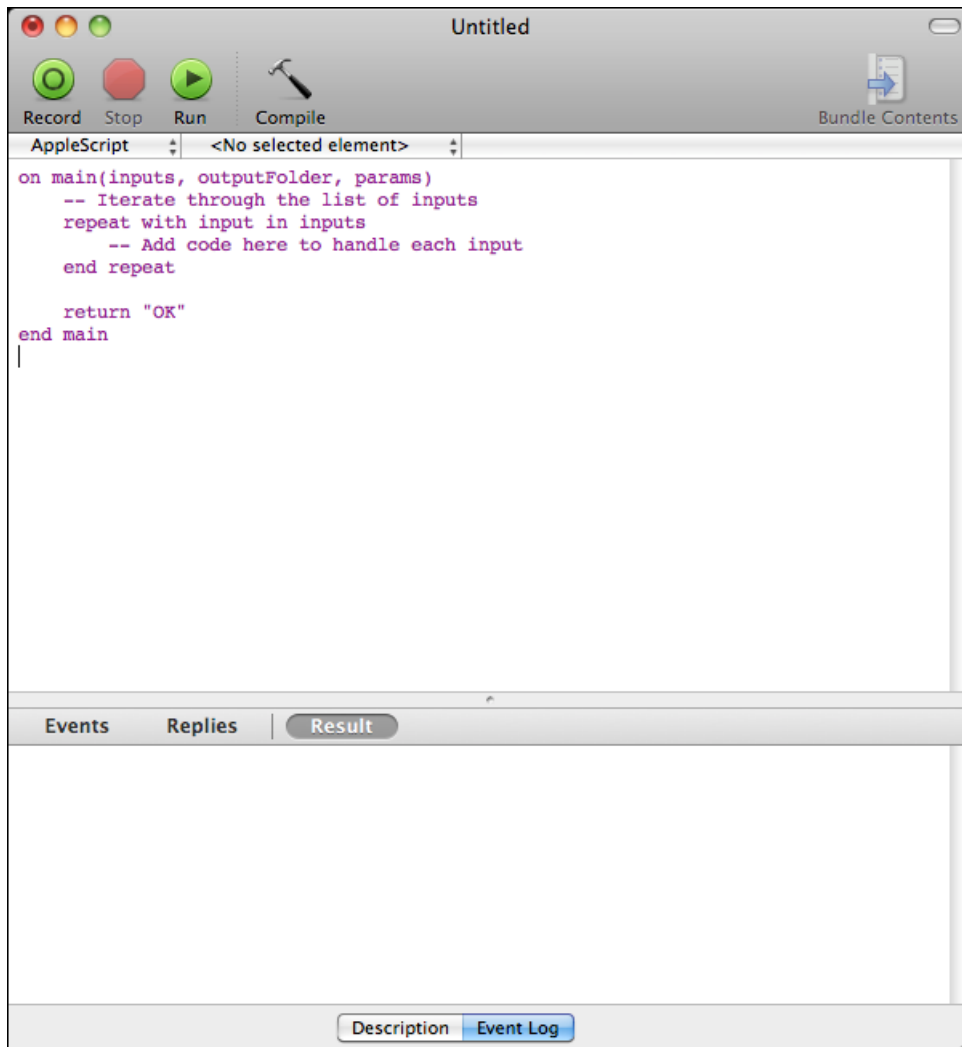
Note that the 'Hello, World!' in the task details and 'OK' state are corresponding with `log "Hello, World!"` and `return "OK"` in the script.



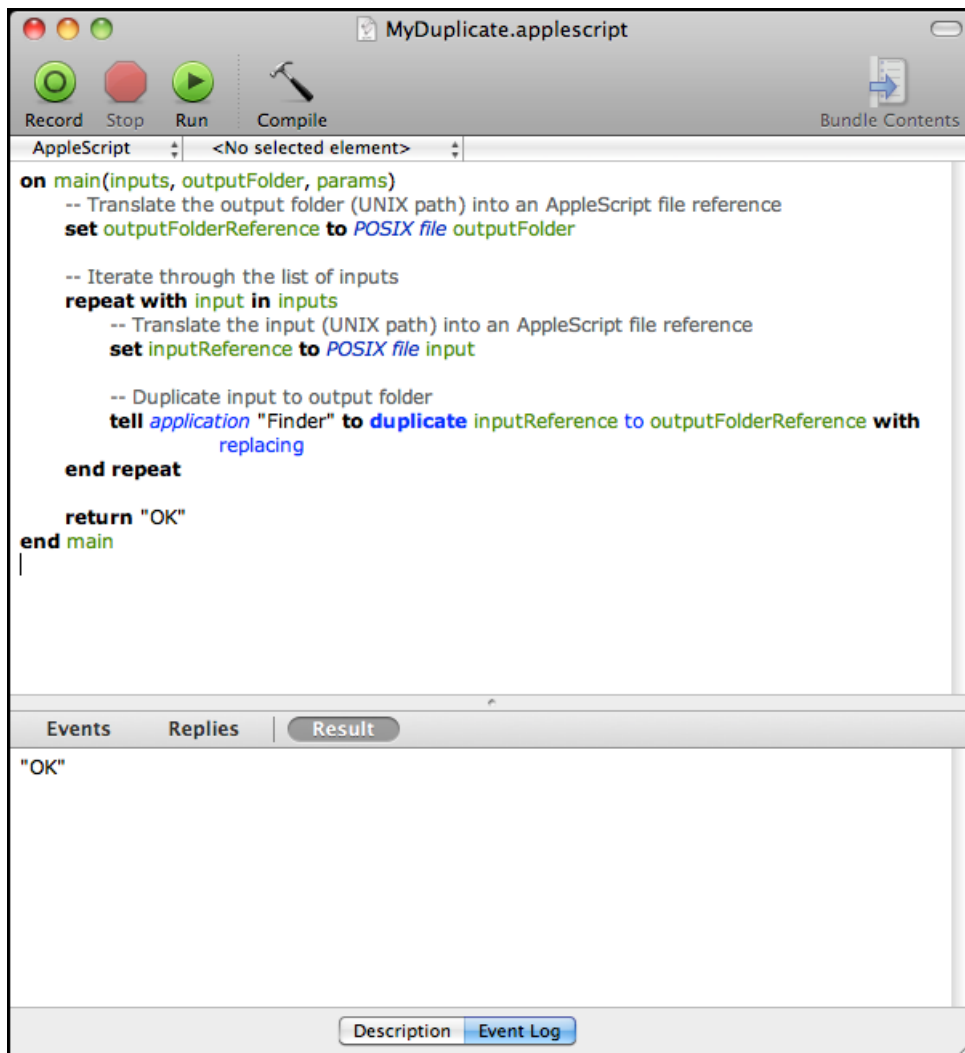
Using AppleScript On Script Runner: A Sample Case

We can use `inputs`, `outputFolder` and `params` in the AppleScript to achieve our objective. First, we demonstrate how to duplicate files without the size restriction and then proceed with the actual case.

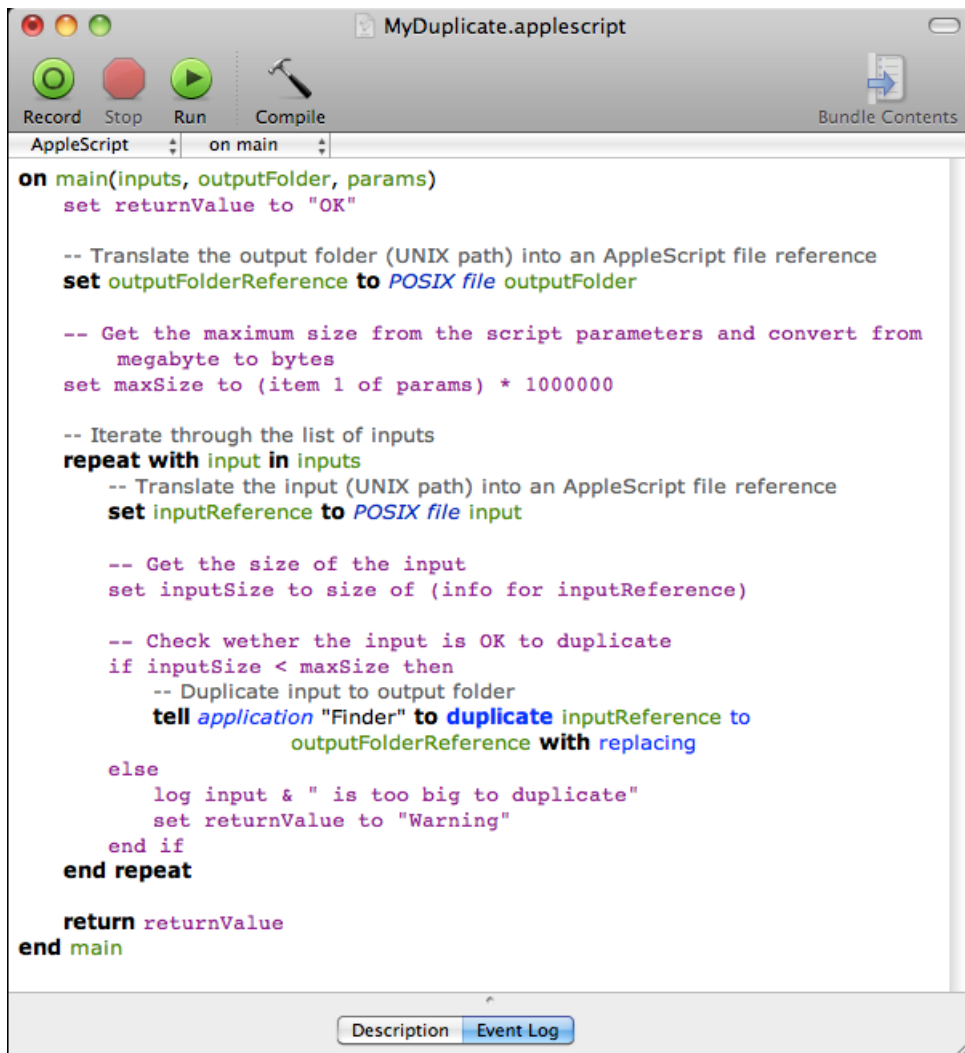
1. Open the AppleScript Editor and add the code given below. This code is aimed to iterate through the list of inputs. It enables you to handle the inputs one by one, via the 'input' variable.



2. You can modify the Script as given below to duplicate the files to a specified output folder without size restrictions. Save this code as an AppleScript text file in the default AppleScript folder of Script Runner(default: **/Library/Scripts/Esko/AppleScript**) or in the Automation Engine AppleScript folder.



3. Add the file size check in the code as given below. This will duplicate the file when the input file size is smaller than the maximum size from the script parameters. If this condition is not met it will add an entry in the log and there will be "Warning". Save the file.



```
on main(inputs, outputFolder, params)
    set returnValue to "OK"

    -- Translate the output folder (UNIX path) into an AppleScript file reference
    set outputFolderReference to POSIX file outputFolder

    -- Get the maximum size from the script parameters and convert from
    megabyte to bytes
    set maxSize to (item 1 of params) * 1000000

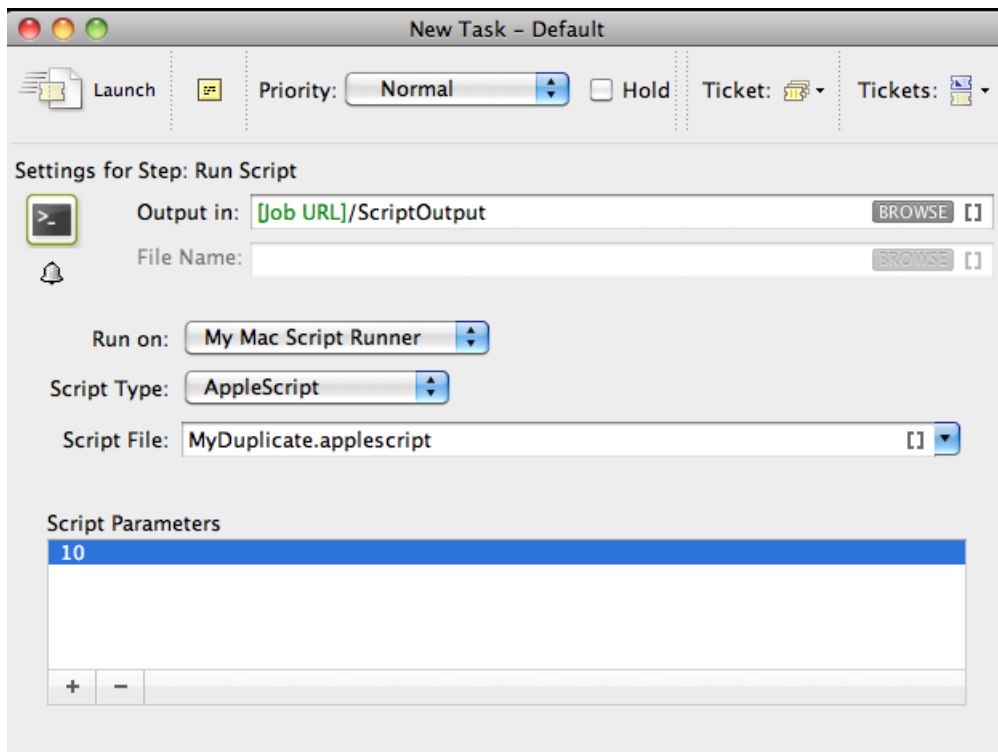
    -- Iterate through the list of inputs
    repeat with input in inputs
        -- Translate the input (UNIX path) into an AppleScript file reference
        set inputReference to POSIX file input

        -- Get the size of the input
        set inputSize to size of (info for inputReference)

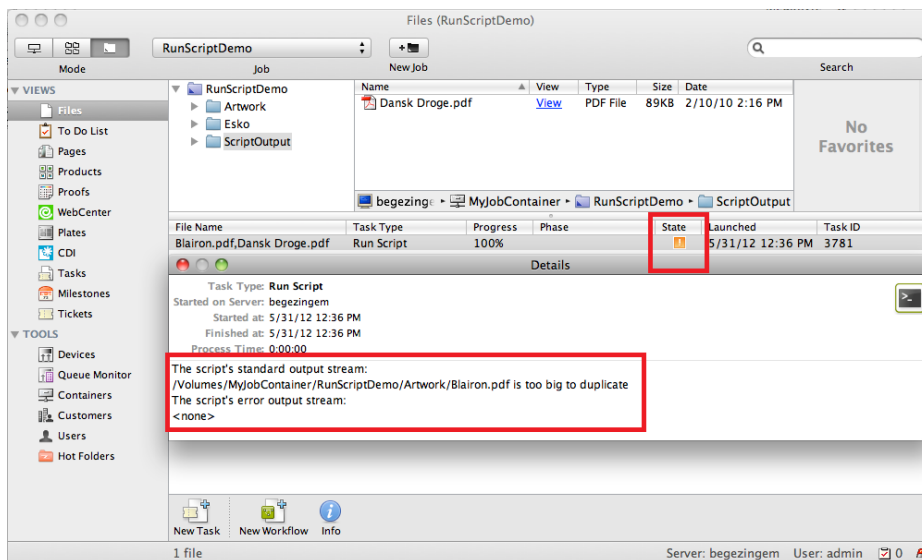
        -- Check whether the input is OK to duplicate
        if inputSize < maxSize then
            -- Duplicate input to output folder
            tell application "Finder" to duplicate inputReference to
                outputFolderReference with replacing
        else
            log input & " is too big to duplicate"
            set returnValue to "Warning"
        end if
    end repeat

    return returnValue
end main
```

4. Open the Automation Engine Pilot. Go to **Files** view where you can select the files to be copied and open a **New Task**. Choose the **Run Script** task, modify its settings and launch. This modified ticket will duplicate every selected file which is smaller than 10MB to the current job's Script Output folder. In this example, we executed this task for two files (Blairon.pdf: 22MB and Dansk Droge.pdf: <1MB).



'Dansk Droge.pdf' is duplicated into the job's Script Output folder. 'Blairon.pdf' was too big to duplicate (> 10MB). Therefore, the task ended in 'Warning' state and added an entry in the task details.

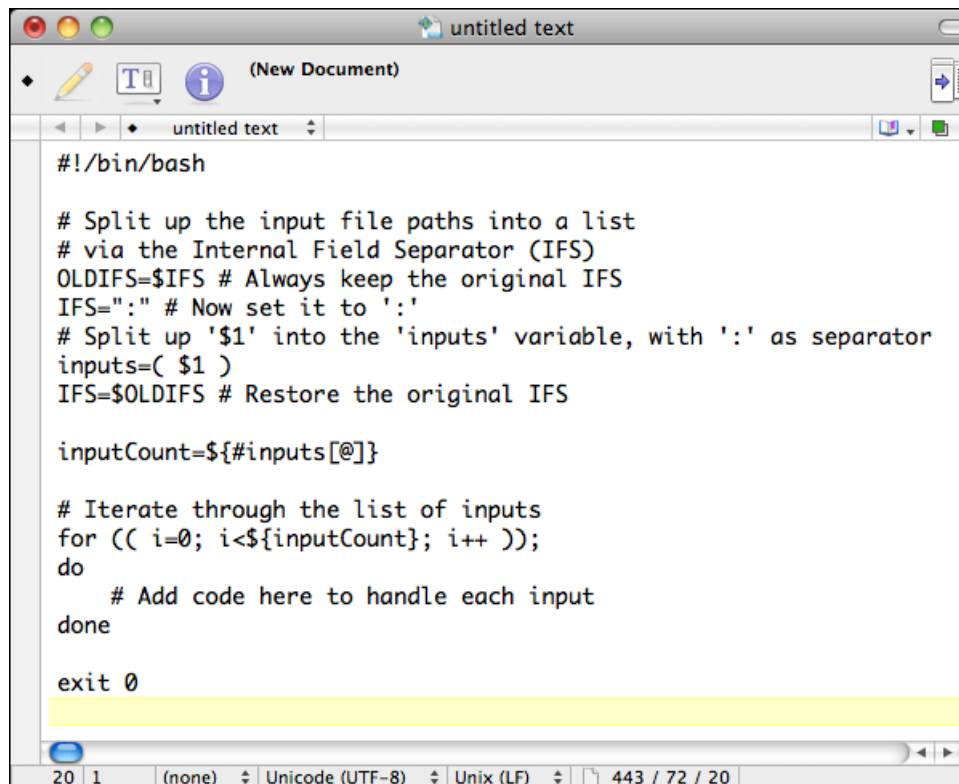


Using Shell Script on Script Runner

In this example, we use a Shell Script to copy every input file with a size smaller than the specified size in the script parameters to the output folder.

1. Open a text editor and add following code. When the Script Runner executes this code, \$1 (the script's first argument) will contain a string of input file paths separated by :. The code splits up

the concatenated file paths into a real list. This helps to iterate through the list and handle the **Run Script** task's inputs one by one.



```
#!/bin/bash

# Split up the input file paths into a list
# via the Internal Field Separator (IFS)
OLDIFS=$IFS # Always keep the original IFS
IFS=":" # Now set it to ':'
# Split up '$1' into the 'inputs' variable, with ':' as separator
inputs=( $1 )
IFS=$OLDIFS # Restore the original IFS

inputCount=${#inputs[@]}

# Iterate through the list of inputs
for (( i=0; i<${inputCount}; i++ ));
do
    # Add code here to handle each input
done

exit 0
```

2. Write and save the code as below. This script copies the input to the output folder if the input's file size is smaller than the maximum size from the script parameters. If the size of the file is bigger, it adds an entry in the log and makes sure the task ends in 'Warning' state (via exit value '1'). Save this code as a text file to the Script Runner's Shell folder (default: **/Library/Scripts/Esko/Shell**) or to the Automation Engine Shell folder.

```

#!/bin/bash

# Split up the input file paths into a list
# via the Internal Field Separator (IFS)
OLDIFS=$IFS # Always keep the original IFS
IFS=":" # Now set it to ':'
# Split up '$1' into the 'inputs' variable, with ':' as separator
inputs=( $1 )
IFS=$OLDIFS # Restore the original IFS

inputCount=${#inputs[@]}

# Get the output folder
outputFolder=$2

# Get the maximum size from the script parameters
# and convert from megabyte to bytes
maxSize=`expr $3 \* 1000000`

exitValue=0

# Iterate through the list of inputs
for (( i=0; i<${inputCount}; i++ ));
do
    input=${inputs[$i]}
    # Get the size of the input
    inputSize=`ls -l "$input" | awk '{print $5}'`

    # Check whether the input is OK to copy
    if [ $inputSize -lt $maxSize ]; then
        # Copy input to output folder
        cp -f "$input" "$outputFolder"
    else
        echo "$input is too big to duplicate"
        exitValue=1
    fi
done

exit $exitValue

```

\$1

First shell script argument: the Run Script task's inputs. A string of input file paths, separated by ':'.

\$2

Second shell script argument or output folder: This is the folder where Automation Engine expects the script's result files. AE will continue the flow with the files you write in this folder. If you leave this folder empty,

\$3, \$4, \$5, ...

AE will continue the flow with the inputs of the Run Script task.

Remaining shell script arguments: additional script parameters which you can inject into the script via the Run Script ticket.

exitValue

0

1

2

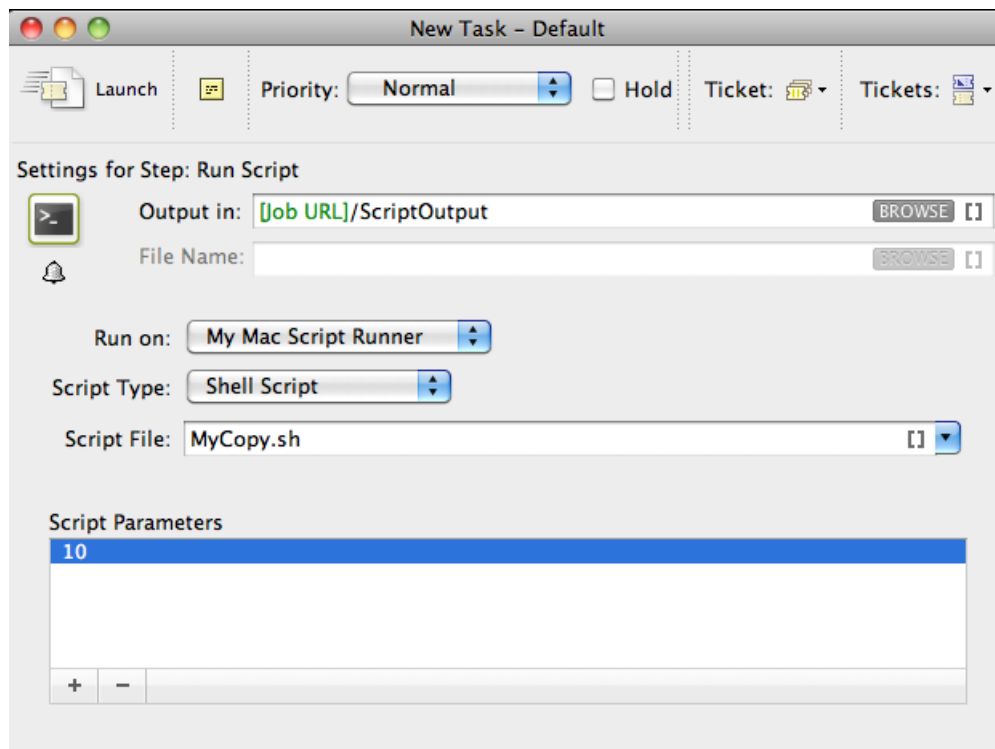
Ending Status of the task

OK

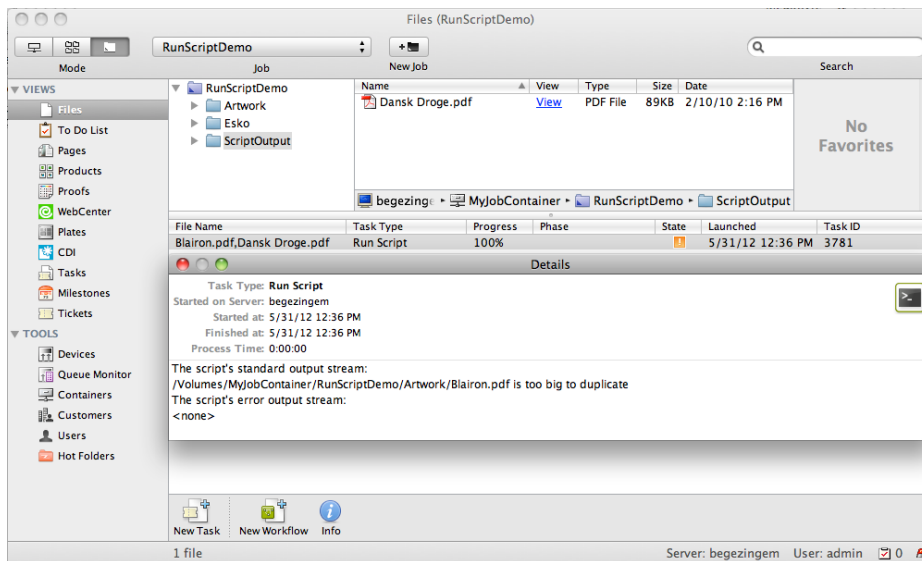
Warning

Error

- Open the Automation Engine Pilot. Go to **Files** view where you can select a file and open a **New Task**. Choose the **Run Script** task, modify its settings and launch. This modified ticket will duplicate every selected file which is smaller than 10MB to the current job's Script Output folder. In this example, we executed this task for two files (Blairon.pdf: 22MB and Dansk Droge.pdf: <1MB).



'Dansk Droge.pdf' is duplicated into the job's Script Output folder. 'Blairon.pdf' was too big to duplicate (> 10MB). Therefore, the task ended in 'Warning' state (due to `exitValue=1` in the code) and added an entry in the task details.



2.14.2 Scripting for Script Runner on Windows

A Script Runner on Windows supports Windows Script and Batch File. We recommend Windows Script for its scripting abilities comparable to batch files, its wider range of supported features and the simpler syntax. Windows Script is plain-text VBScript or JScript which is interpreted and run by the Windows Script Host.

Using Windows Script on Script Runner

1. Open a text editor and add the following code:

```

1
2
3 Function Main(inputs, outputFolder, params)
4     WScript.Echo "Hello, World!"
5
6     ' Set the function's return value.
7     Main = "OK"
8 End Function
9
length:148 Ln:6 Col:39 Sel:0 Dos\Windows ANSI INS

```

Function Main

The function that will be called by the Script Runner. Script Runner executes only the code in this main function.

```
inputs
```

```
outputFolder
```

```
params
```

```
WScript.Echo
```

```
Main = "OK"
```

First argument of main function: a list of input file paths(type: list of strings).

Second argument of the main function: the folder where AE expects the script's result files. AE will continue the flow with the files you write in this folder. If you leave this folder empty, AE will continue the flow with the inputs of the Run Script task (type: string).

Third argument of the main function: additional script parameters injected into the script via the Run Script ticket (type: list of strings).

This puts some extra log info in the Run Script task details and log. This call prints text to the Console and adds a newline character without Script Runner context.

This will communicate with the **Run Script** task that everything went fine. Other possibilities are `Main = "Warning"` and `Main = "Error"`.

2. You can test this script locally by adding the following code: Save this file. Open command prompt. Change the directory to the script's parent directory. Run command 'cscript MyHelloWorld.vbs'.

```

1
2
3 Function Main(inputs, outputFolder, params)
4     WScript.Echo "Hello, World!"
5
6     ' Set the function's return value.
7     Main = "OK"
8 End Function
9
10 ' Our main function is not using any of its arguments,
11 ' so provide empty ones.
12 Dim inputs()
13 Dim outputFolder
14 Dim params()
15 Main inputs, outputFolder, params
16
length: 314 Ln: 16 Col: 1 Sel: 164 Dos\Windows ANSI INS

```

This will produce the output 'Hello, World!' to the console. The Script Runner does not attend to the test code in your script. It will execute the contents of the main function and ignore the rest. Therefore, you can keep your test code for future local testing.

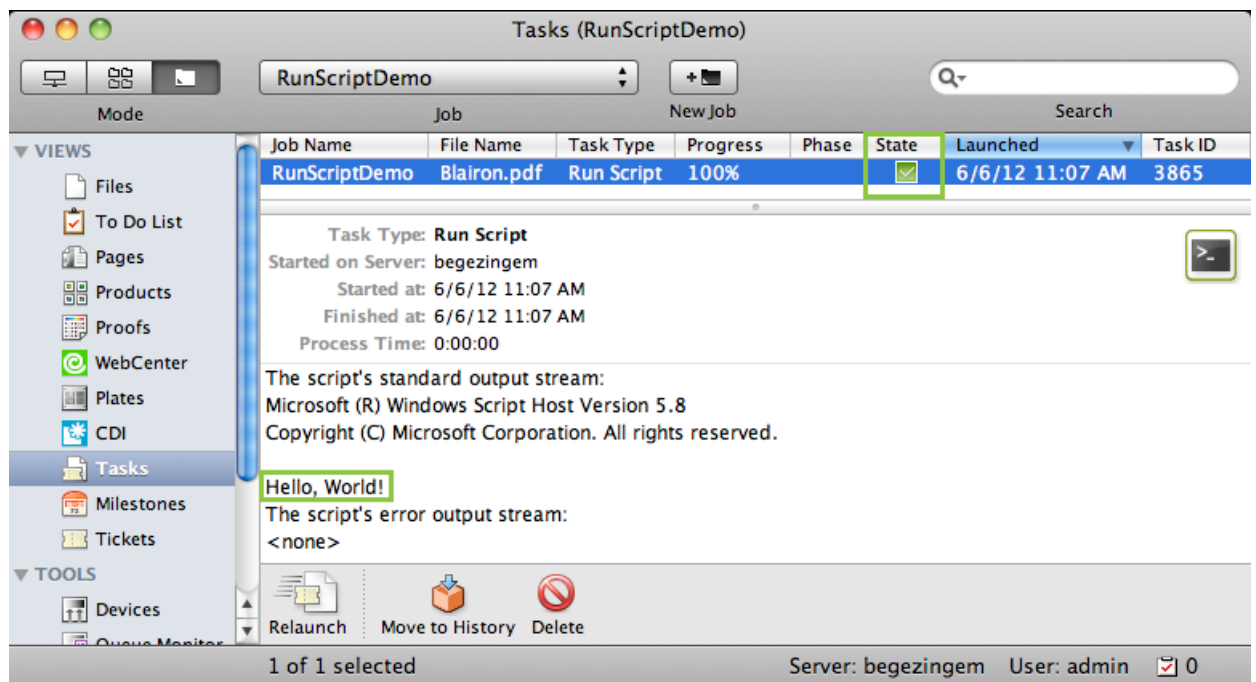
3. Open the Automation Engine Pilot. Go to **Files** view where you can select a file and open a **New Task**. Choose the **Run Script** task, modify its settings and launch the task.

The screenshot shows a window titled "New Task - Default" with a toolbar containing "Launch", "Priority: Normal", "Hold", "Ticket:", and "Tickets:". Below the toolbar, the "Settings for Step: Run Script" section contains the following fields:

- Output in:** [Text field] [BROWSE] [Icon]
- File Name:** [Text field] [BROWSE] [Icon]
- Run on:** [Dropdown menu] (My Win Script Runner)
- Script Type:** [Dropdown menu] (Windows Script)
- Script File:** [Text field] (MyHelloWorld.vbs) [Icon]

Below these fields is a "Script Parameters" section with a large empty text area and "+" and "-" buttons at the bottom left.

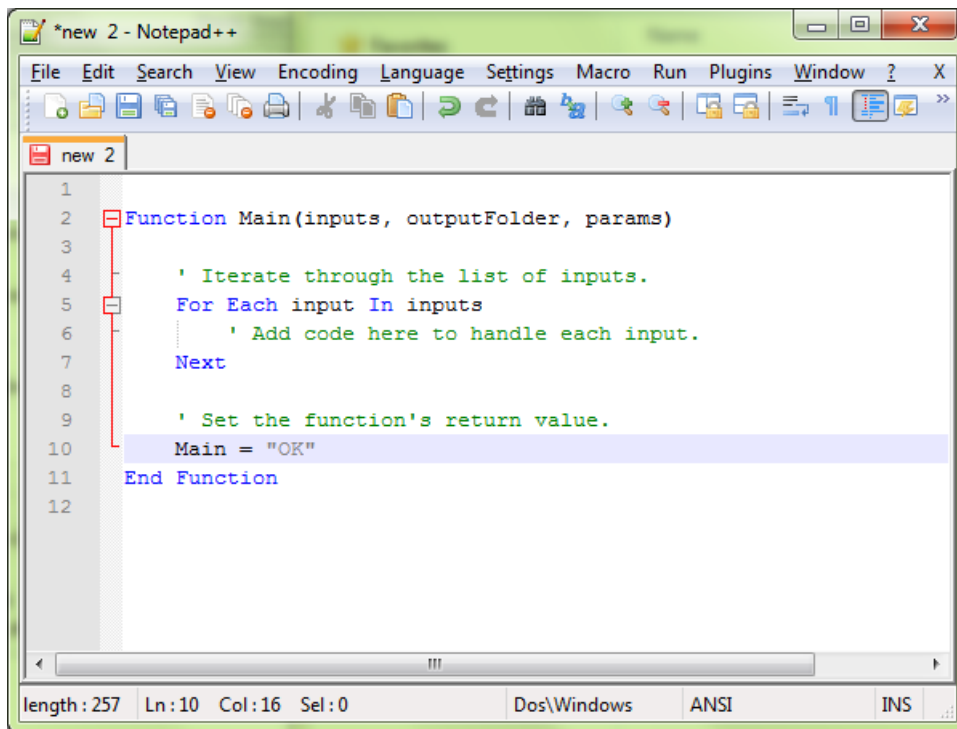
Note that the 'Hello, World!' in the task details and 'OK' state are corresponding with `WScript.Echo "Hello, World!"` and `Main = "OK"` in the script.



Using Windows Script On Script Runner: A Sample Case

We can use `inputs`, `outputFolder` and `params` in the AppleScript to achieve our objective. First, we demonstrate how to duplicate files without the size restriction and then proceed with the actual use case.

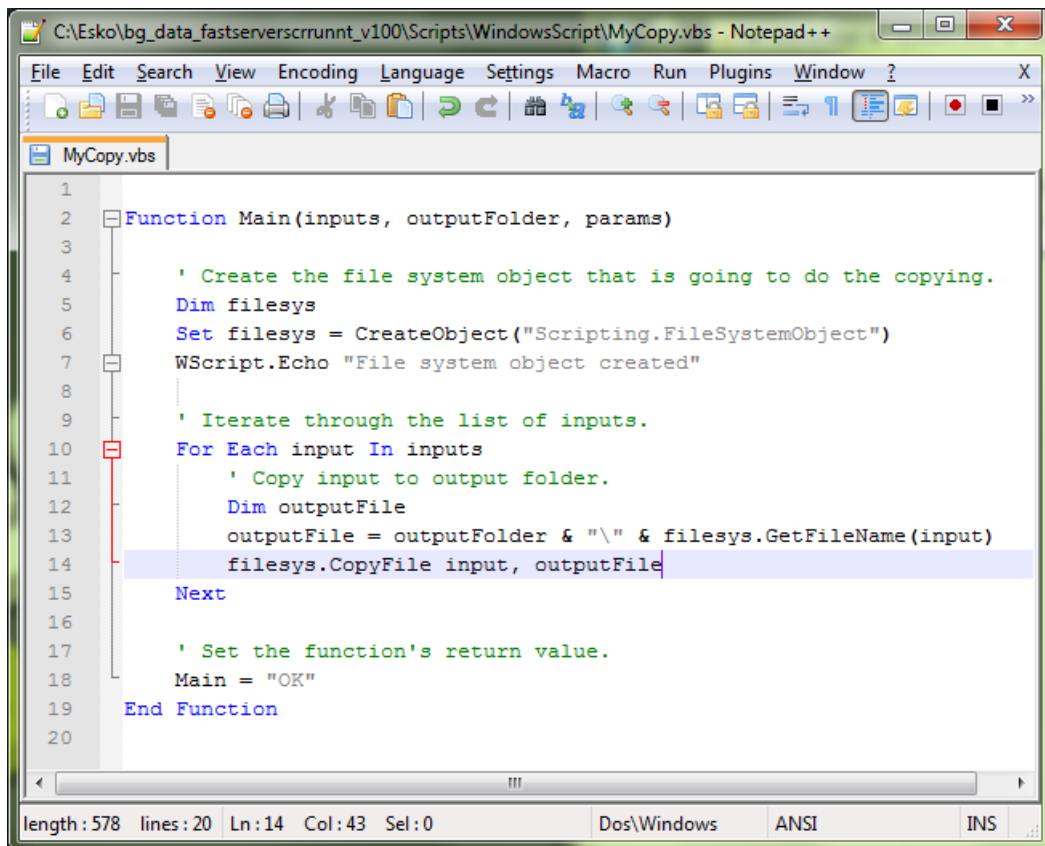
1. Open a text editor and add the code given below. This code is aimed to iterate through the list of inputs. It enables you to handle the inputs one by one, via the 'input' variable.



```
1
2 Function Main(inputs, outputFolder, params)
3
4     ' Iterate through the list of inputs.
5     For Each input In inputs
6         ' Add code here to handle each input.
7     Next
8
9     ' Set the function's return value.
10    Main = "OK"
11 End Function
12
```

length: 257 Ln: 10 Col: 16 Sel: 0 Dos\Windows ANSI INS

2. You can modify the Script as given below to duplicate the files to a specified output folder without size restrictions. Save this code as a text file with '.vbs' extension (VBScript) in the Windows Script folder of Script Runner (default: **C:\Esko\bg_data_fastserverscrunnt_v100\Scripts\WindowsScript**) or in the Automation Engine WindowsScript folder.



```
1
2 Function Main(inputs, outputFolder, params)
3
4     ' Create the file system object that is going to do the copying.
5     Dim filesystems
6     Set filesystems = CreateObject("Scripting.FileSystemObject")
7     WScript.Echo "File system object created"
8
9     ' Iterate through the list of inputs.
10    For Each input In inputs
11        ' Copy input to output folder.
12        Dim outputFile
13        outputFile = outputFolder & "\" & filesystems.GetFileName(input)
14        filesystems.CopyFile input, outputFile
15    Next
16
17    ' Set the function's return value.
18    Main = "OK"
19 End Function
20
```

length: 578 lines: 20 Ln: 14 Col: 43 Sel: 0 Dos\Windows ANSI INS

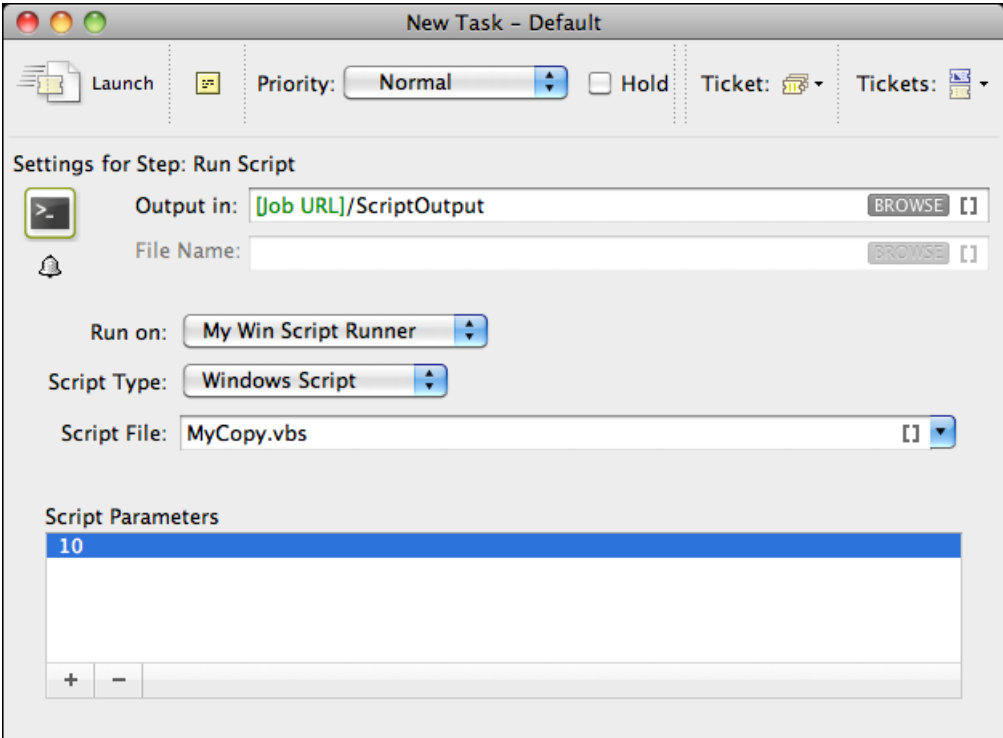
3. Add the file size check in the code as given below. This will duplicate the file when the input file size is smaller than the maximum size from the script parameters. If this condition is not met it will add an entry in the log and there will be "Warning". Save the file.

```

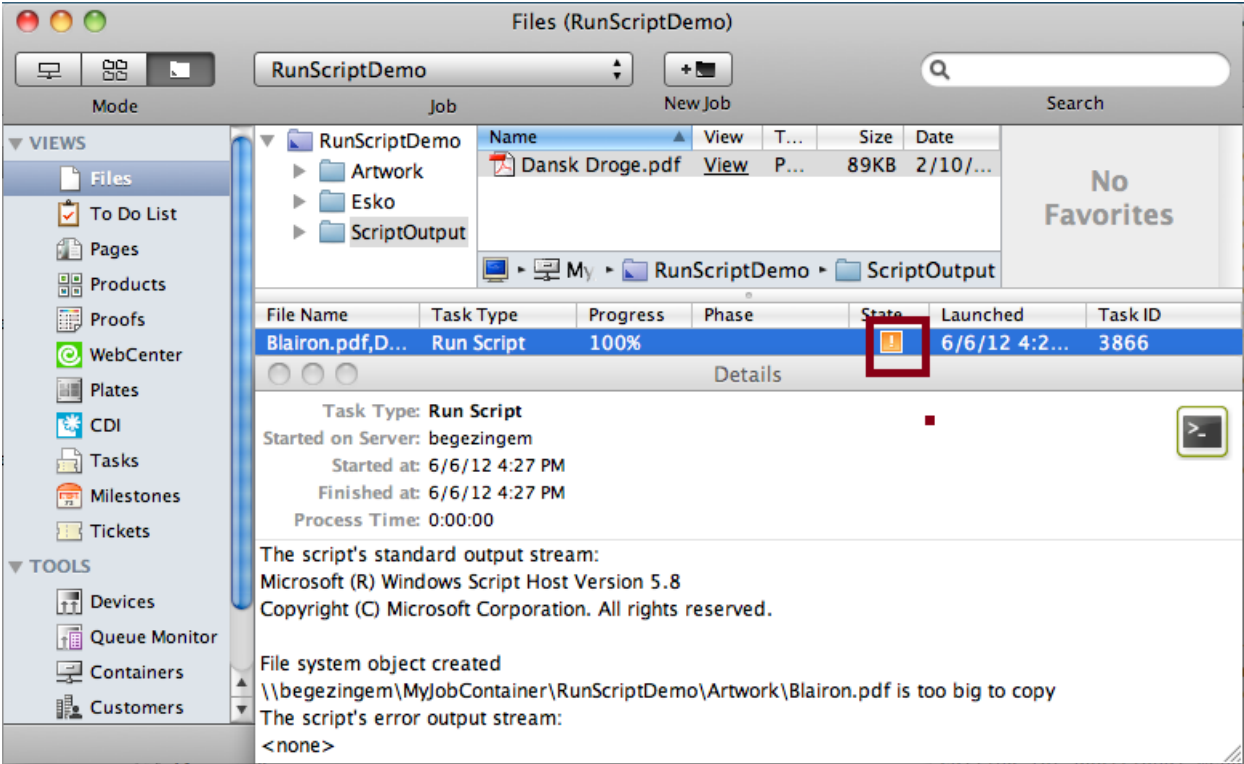
1
2  Function Main(inputs, outputFolder, params)
3      Dim returnValue
4      returnValue = "OK"
5
6      ' Get the maximum size from the script parameters
7      ' and convert from megabyte to bytes.
8      Dim maxSize
9      maxSize = params(0) * 1000000
10
11     ' Create the file system object that is going to do the copying.
12     Dim fileSys
13     Set fileSys = CreateObject("Scripting.FileSystemObject")
14     WScript.Echo "File system object created"
15
16     ' Iterate through the list of inputs.
17     For Each input In inputs
18         ' Turn the input path into a file object.
19         Dim inputReference
20         Set inputReference = fileSys.GetFile(input)
21
22         ' Check whether the input is OK to copy.
23         If inputReference.Size < maxSize Then
24             ' Copy input to output folder.
25             Dim outputFile
26             outputFile = outputFolder & "\" & fileSys.GetFileName(input)
27             fileSys.CopyFile input, outputFile
28         Else
29             WScript.Echo input & " is too big to copy"
30             returnValue = "Warning"
31         End If
32     Next
33
34     ' Set the function's return value.
35     Main = returnValue
36 End Function
37
length:1046 lines:37 Ln:31 Col:15 Sel:0 Dos\Windows ANSI INS

```

4. Open the Automation Engine Pilot. Go to **Files** view where you can select the files to be copied and open a **New Task**. Choose the **Run Script** task, modify its settings and launch. This modified ticket will duplicate every selected file which is smaller than 10MB to the current job's Script Output folder. In this example, we executed this task for two files (Blairon.pdf: 22MB and Dansk Droge.pdf: <1MB).

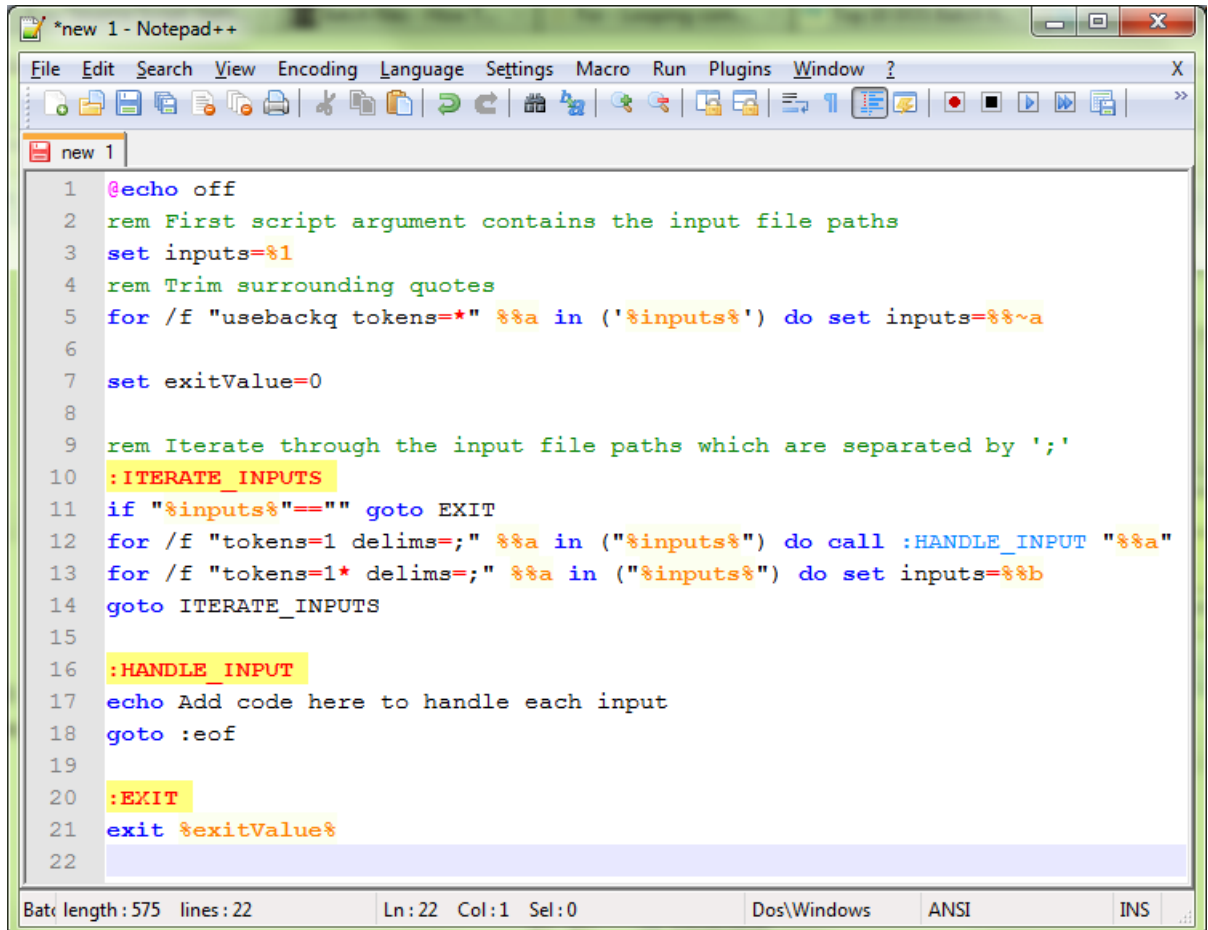


'Dansk Droge.pdf' is duplicated into the job's Script Output folder. 'Blairon.pdf' was too big to duplicate (> 10MB). Therefore, the task ended in 'Warning' state and added an entry in the task details.



Using Batch File for Scripting on Windows

1. Open a text editor and add the code given below. This code is aimed to iterate through the list of inputs. It enables you to handle the inputs one by one, via the command %1 (the script's first argument) will contain a string of input file paths, separated by ';'.



```

1 @echo off
2 rem First script argument contains the input file paths
3 set inputs=%1
4 rem Trim surrounding quotes
5 for /f "usebackq tokens=*" %%a in ('%inputs%') do set inputs=%%~a
6
7 set exitValue=0
8
9 rem Iterate through the input file paths which are separated by ';'
10 :ITERATE_INPUTS
11 if "%inputs%"==" " goto EXIT
12 for /f "tokens=1 delims=;" %%a in ("%inputs%") do call :HANDLE_INPUT "%a"
13 for /f "tokens=1* delims=;" %%a in ("%inputs%") do set inputs=%%b
14 goto ITERATE_INPUTS
15
16 :HANDLE_INPUT
17 echo Add code here to handle each input
18 goto :eof
19
20 :EXIT
21 exit %exitValue%
22

```

2. You can modify the Script as given below to duplicate the files to a specified output folder with a size check. Save this code as a text file with '. bat ' extension in the Script Runner Batch File Folder (default: **C:\Esko\bg_data_fastserverscrunnt_v100\Scripts\BatchFile**) or in the Automation Engine BatchFile folder.

```

C:\Esko\bg_data_fastserverscrunnt_v100\Scripts\Batch\myCopy.bat - Notepad++
File Edit Search View Encoding Language Settings Macro Run Plugins Window ?
myCopy.bat
1 @echo off
2 rem First script argument contains the input file paths
3 set inputs=%1
4 rem Trim surrounding quotes
5 for /f "usebackq tokens=*" %%a in ('%inputs%') do set inputs=%%~a
6
7 rem Get the output folder
8 set outputFolder=%2
9
10 rem Get the maximum size from the script parameters
11 rem and convert from megabyte to bytes
12 set /a maxSize=%3*1000000
13
14 set exitValue=0
15
16 rem Iterate through the input file paths which are separated by ';'
17 :ITERATE_INPUTS
18 if "%inputs%"==" " goto EXIT
19 for /f "tokens=1 delims=;" %%a in ("%inputs%") do call :HANDLE_INPUT "%%a"
20 for /f "tokens=1* delims=;" %%a in ("%inputs%") do set inputs=%%b
21 goto ITERATE_INPUTS
22
23 :HANDLE_INPUT
24 rem Get the size of the input
25 for %%? in (%1) do set inputSize=%%~z?
26 rem Check whether the input is OK to copy
27 if %inputSize% lss %maxSize% (
28     rem Copy input to output folder
29     copy /y %1 %outputFolder%
30 ) else (
31     echo %1 is too big to copy
32     set exitValue=1
33 )
34 goto :eof
35
36 :EXIT
37 exit %exitValue%
38
Bat: length: 973 lines: 38 Ln: 38 Col: 1 Sel: 0 Dos\Windows ANSI INS

```

%1

First batch file argument: the Run Script task's inputs. A string of input file paths, separated by ';'.

%2

Second batch file argument or output folder: the folder where AE expects the script's result files. AE will continue the flow

%3, %4,%5...

with the files you write in this folder. If you leave this folder empty, AE will continue the flow with the inputs of the Run Script task. Remaining batch file arguments: additional script parameters, injected into the script via the **Run Script** ticket.

Exit value

0

1

2

Ending Status of the task

OK

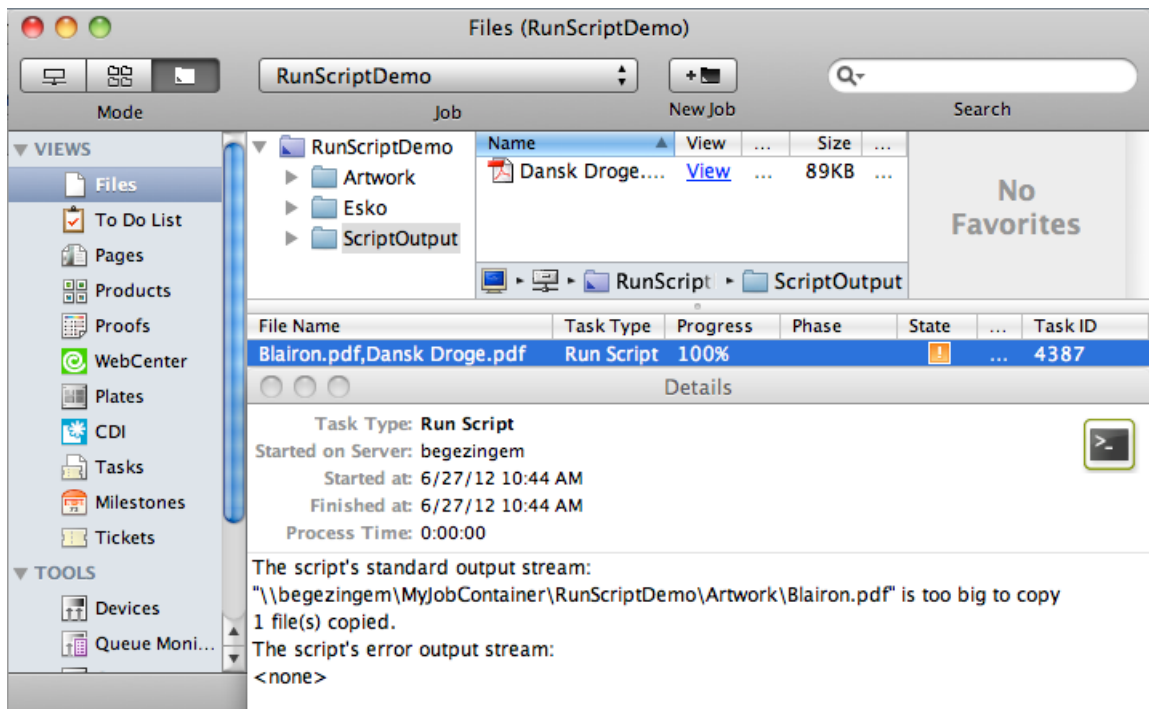
Warning

Error

- Open the Automation Engine Pilot. Go to **Files** view where you can select the files to be copied and open a **New Task**. Choose the **Run Script** task, modify its settings and launch. This modified ticket will duplicate every selected file which is smaller than 10MB to the current job's Script Output folder. In this example, we executed this task for two files (Blairon.pdf: 22MB and Dansk Droge.pdf: <1MB).

The screenshot shows the 'New Task - Default' configuration window. At the top, there are controls for 'Launch', 'Priority' (set to Normal), 'Hold' (checkbox), 'Ticket', and 'Tickets'. Below this is the 'Settings for Step: Run Script' section. It includes a terminal icon, 'Output in:' (set to [job URL]/ScriptOutput), 'File Name:' (empty), 'Run on:' (set to My Win Script Runner), 'Script Type:' (set to Batch File), and 'Script File:' (set to myCopy.bat). At the bottom, there is a 'Script Parameters' table with one row containing the value '10'.

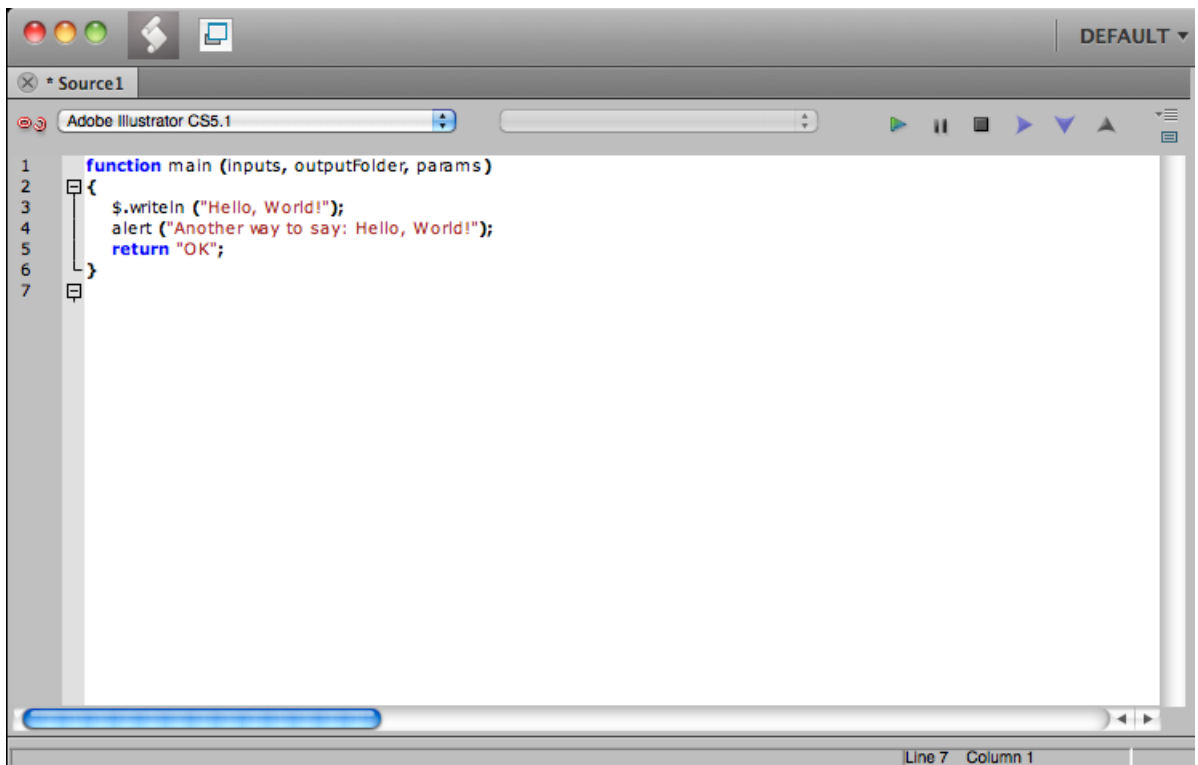
'Dansk Droge.pdf' is duplicated into the job's Script Output folder. 'Blairon.pdf' was too big to duplicate (> 10MB). Therefore, the task ended in 'Warning' state and added an entry in the task details.



2.14.3 Using ExtendScript on Script Runner (On MacOS/Windows)

ExtendScript is JavaScript extended for Adobe CS applications. Adobe provides a complete integrated development environment (IDE) for programming ExtendScript which is the ExtendScript Toolkit (ESTK). Latest versions of the ESTK are available with the Creative Suite. For more info and Adobe scripting resources visit the Adobe Scripting Center.

1. Open the ExtendScript Toolkit and add following code. Save this code in the Script Runner's ExtendScript folder. The default location is : **/Library/Scripts/Esko/ExtendScript** for a Script Runner on Mac OS or **C:\Esko\bg_data_fastserverscrunnt_v100\Scripts\ExtendScript** on Windows. Alternatively, you can save them in the ExtendScript folder of Automation Engine.



Main

This function will be called by the Script Runner. Only the code in this main function gets executed.

Inputs

First argument of main function: a list of input file paths (type: list of strings).

outputFolder

Second argument of the main function: the folder where AE expects the script's result files. AE will continue the flow with the files you write in this folder. If you leave this folder empty, AE will continue the flow with the inputs of the Run Script task (type: string).

params

Third argument of main function: additional script parameters, injected into the script via the **Run Script** ticket. (type: list of strings)

\$.writeln

This writes extra log information in the Run Script task details and log. Without Script Runner context this call prints text to the Console, and adds a newline character.

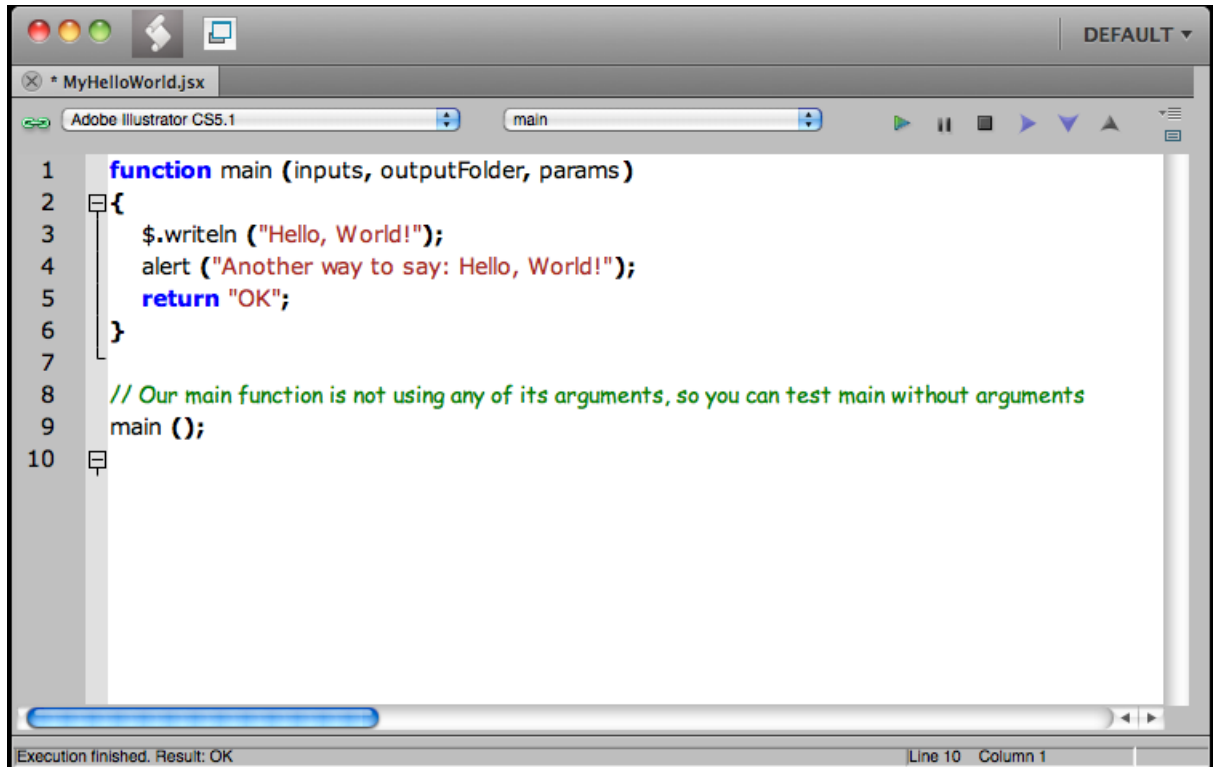
alert

This registers some extra log info in the **Run Script** task details and log. Without Script Runner context this call displays an alert box.

```
return "OK";
```

This will communicate with the **Run Script** task that everything went fine. Other possibilities are `Return = "Warning"` and `Return = "Error"`.

- To test the script locally in the ExtendScript Toolkit, add following code, save and run.

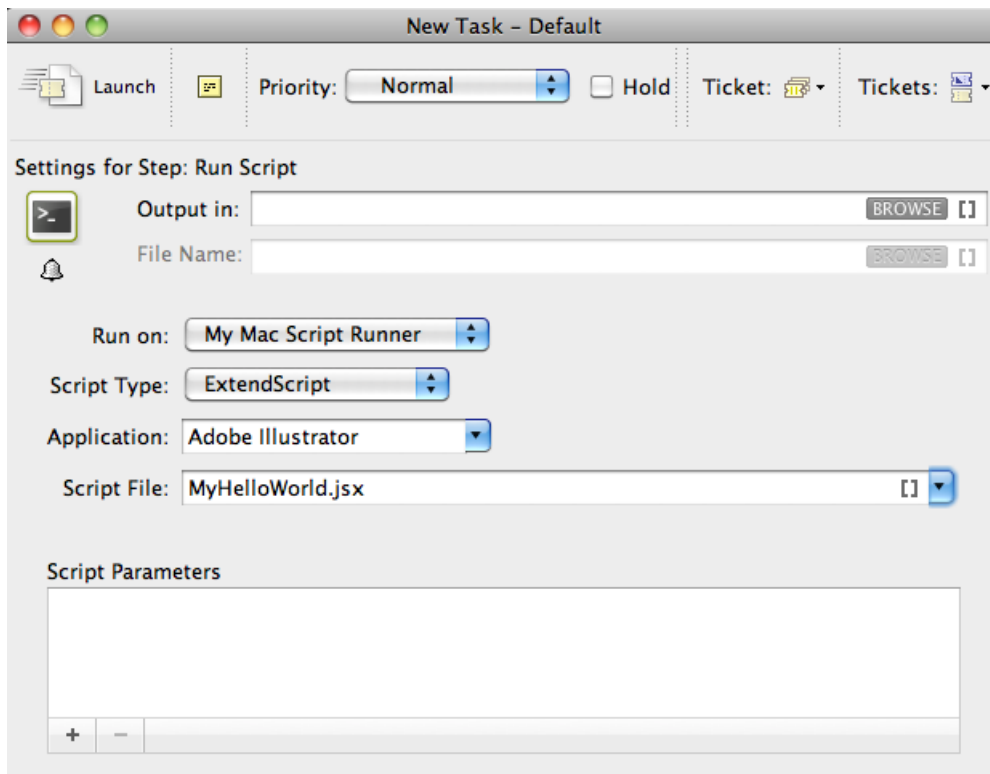


Note that the alert box pops up and the 'Hello, World!' and 'OK' result in the Console.

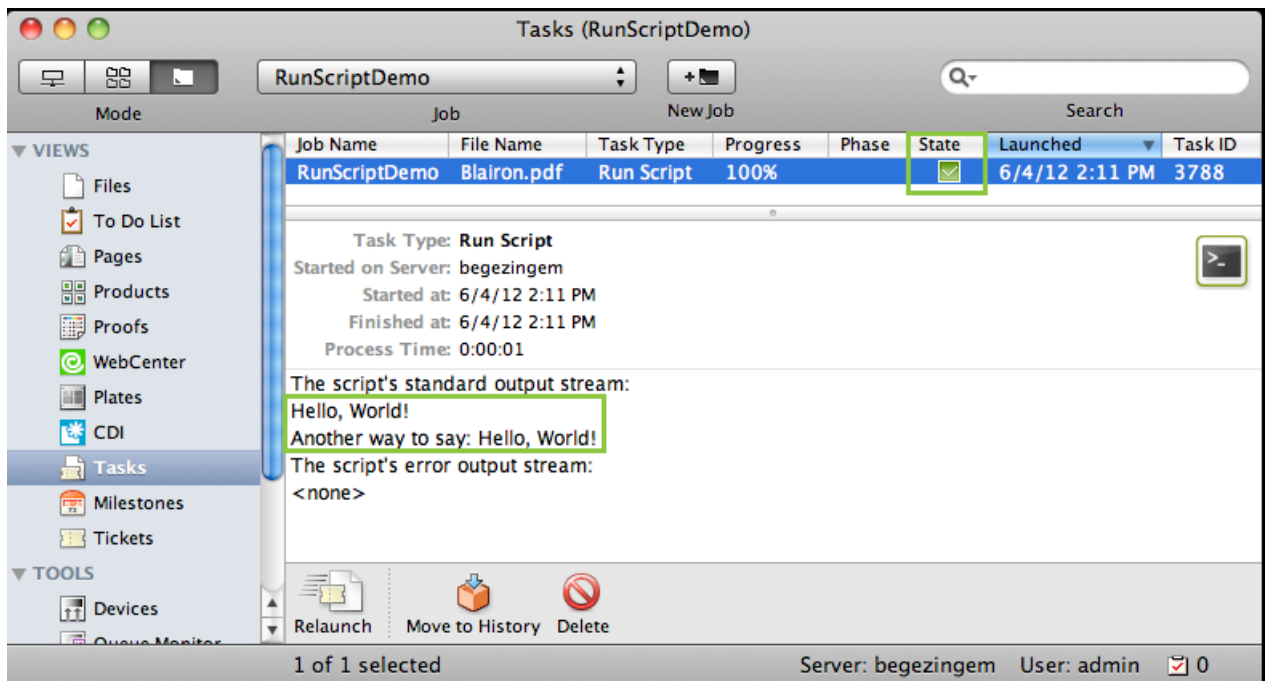


The Script Runner does not attend to the test code in your script. It will execute only the contents of the main function and ignore the rest. Therefore, you can keep your test code for future local testing.

- Open the Automation Engine Pilot. Go to **Files** view where you can select a file and open a **New Task**. Choose the **Run Script** task, modify its settings and launch the task.



Note that the 'Hello, World!' in the task details and 'OK' state are corresponding with `$.writeln("Hello World!")` and `Return = "OK"` in the script.



Using ExtendScript on Script Runner: A Sample Case.

Note: When you use ExtendScript on Windows, you can avoid troubles while accessing your user specific settings such as Adobe applications' **Presets** , **Actions** etc by stopping the Script Runner service and running it as an application for the logged in user (who also defined Adobe settings).

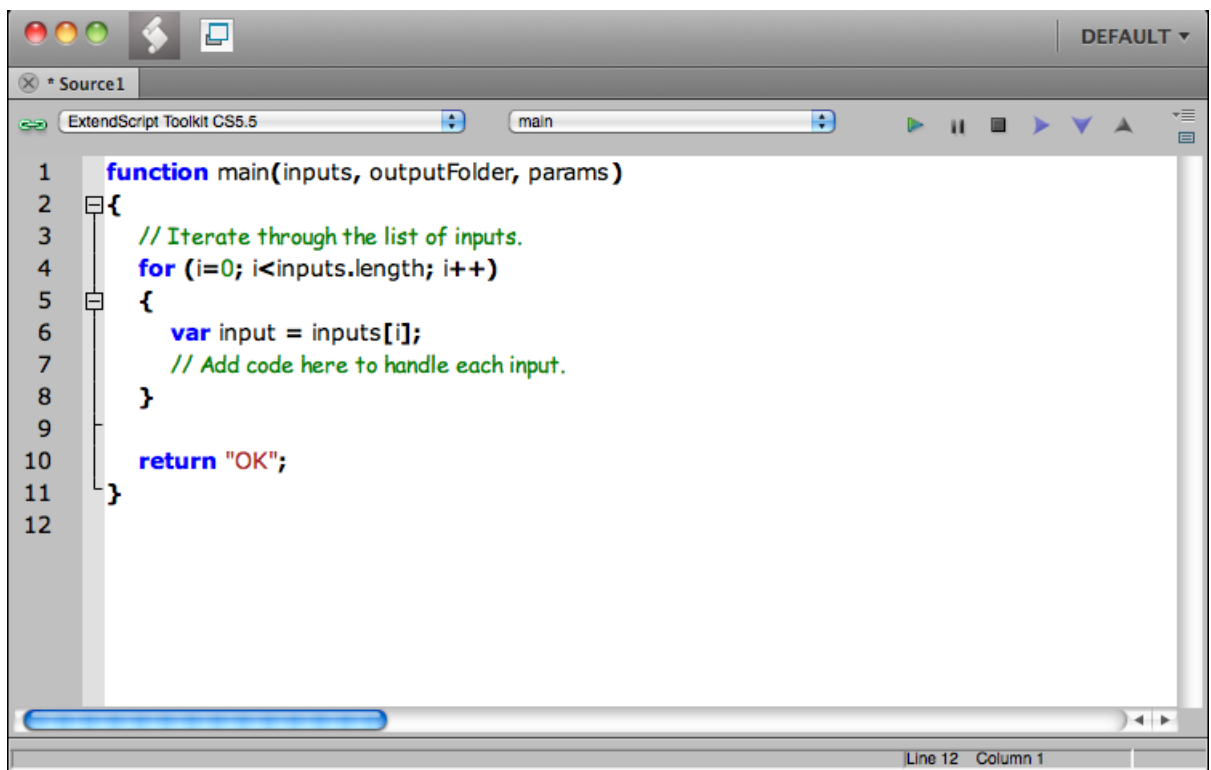
Open **Start > All Programs > Esko > Automation Engine Script Runner > Preferences** .

Stop the Script Runner and uncheck **Start at login** (which actually means 'Start as service') and Close **Preferences**.

Start as a console application by double clicking its executable: Script Runner's program folder>\bin_ix86\egscrnun.exe (for example C:\Esko\bg_prog_fastserverscrnunnt_v120\bin_ix86\egscrnun.exe)

Note: To prevent having to start up the Script Runner every time you log in, add its executable to your user's/system's Startup Items.

1. Define a **My Print Preset** in your Adobe Illustrator application.
2. Open the ExtendScript Toolkit and add following code. This code is aimed to iterate through the list of inputs. It enables you to handle the inputs one by one, via the `input` variable.

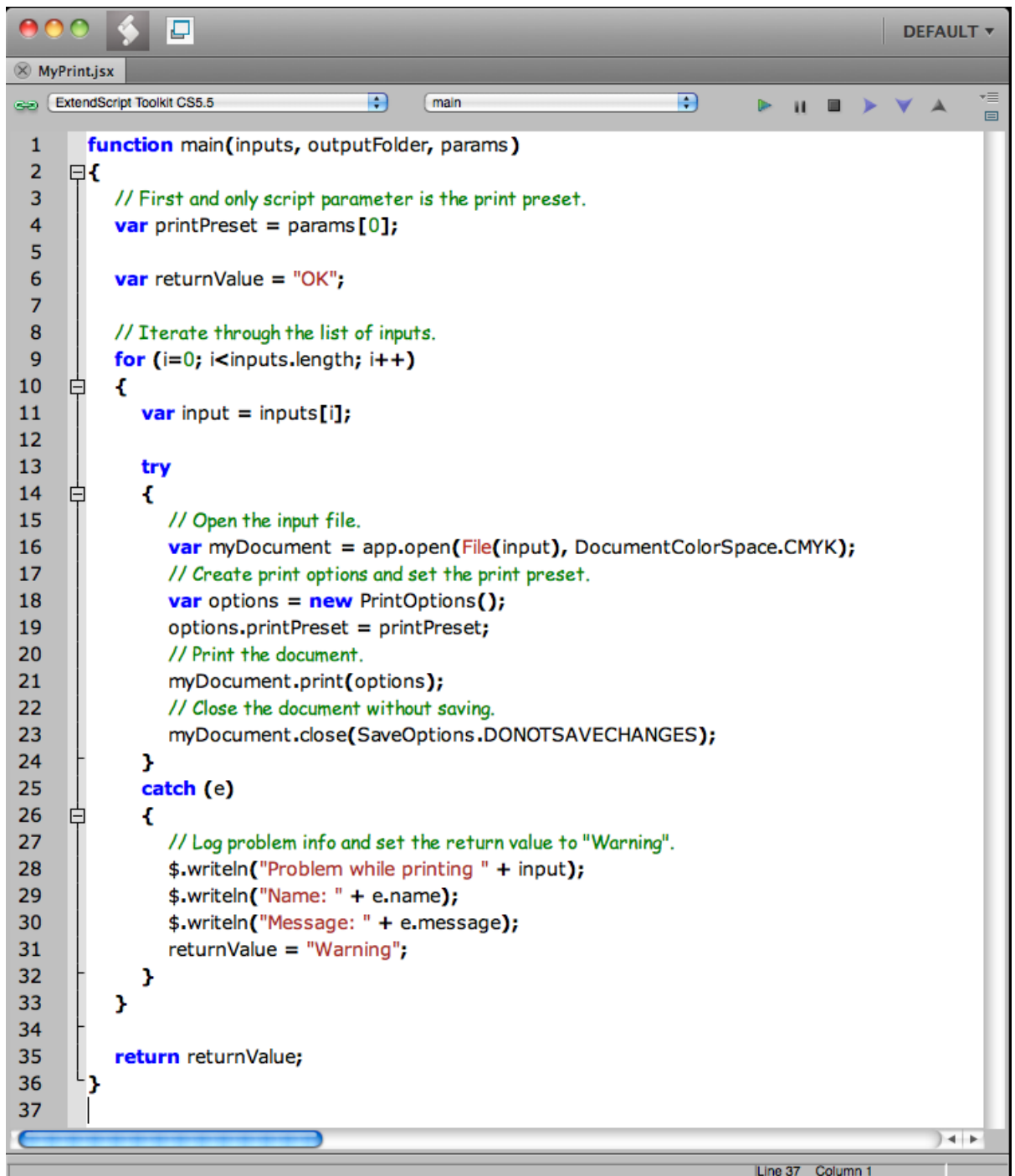


```

1  function main(inputs, outputFolder, params)
2  {
3      // Iterate through the list of inputs.
4      for (i=0; i<inputs.length; i++)
5      {
6          var input = inputs[i];
7          // Add code here to handle each input.
8      }
9
10     return "OK";
11 }
12

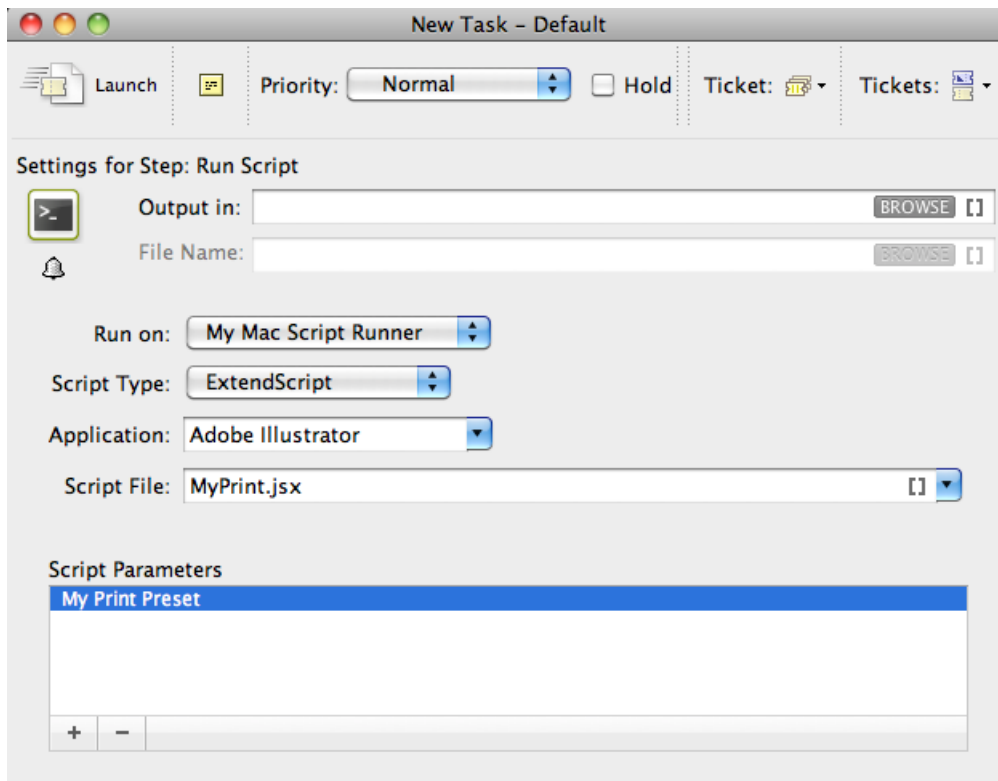
```

3. To print every input file using a **Print Preset** from the script parameters of the **Run Script** task, write the code as below. Save this code in the default ExtendScript folder(of Script Runner or Automation Engine).



```
1  function main(inputs, outputFolder, params)
2  {
3      // First and only script parameter is the print preset.
4      var printPreset = params[0];
5
6      var returnValue = "OK";
7
8      // Iterate through the list of inputs.
9      for (i=0; i<inputs.length; i++)
10     {
11         var input = inputs[i];
12
13         try
14         {
15             // Open the input file.
16             var myDocument = app.open(File(input), DocumentColorSpace.CMYK);
17             // Create print options and set the print preset.
18             var options = new PrintOptions();
19             options.printPreset = printPreset;
20             // Print the document.
21             myDocument.print(options);
22             // Close the document without saving.
23             myDocument.close(SaveOptions.DONOTSAVECHANGES);
24         }
25         catch (e)
26         {
27             // Log problem info and set the return value to "Warning".
28             $.writeln("Problem while printing " + input);
29             $.writeln("Name: " + e.name);
30             $.writeln("Message: " + e.message);
31             returnValue = "Warning";
32         }
33     }
34
35     return returnValue;
36 }
37
```

4. Open the Automation Engine Pilot. Go to **Files** view where you can select a file and open a **New Task**. Choose the **Run Script** task, modify its settings and launch the task.



Launching this ticket will result in the selected Illustrator files being printed using the **Print Preset** specified as script parameters (`My Print Preset`) in the task.

2.14.4 Appendix : Script Samples

Note:

Sample scripts are provided as-is with no warranty of fitness for a particular purpose. These scripts are solely intended to demonstrate techniques for accomplishing common tasks. Additional script logic and error-handling may need to be added to achieve the desired results in your specific environment.

It is up to the user to verify that his intended use of the offered automation functionality is compliant with any third party license agreement and/or other restrictions applicable to any non-Esko products.

AppleScript Code Samples

MyHelloWorld.applescript

```
on main(inputs, outputFolder, params)
    log "Hello, World!"
    return "OK"
end main
on run
```

```
-- Our main function is not using any of its arguments, so provide empty ones
set inputs to {} -- empty list
set outputFolder to "" -- empty string
set params to {} -- empty list
main(inputs, outputFolder, params)
end run
```

MyDuplicate.applescript

```
on main(inputs, outputFolder, params)
    set returnValue to "OK"

    -- Translate the output folder (UNIX path) into an AppleScript file
    reference
    set outputFolderReference to POSIX file outputFolder

    -- Get the maximum size from the script parameters and convert from megabyte
    to bytes
    set maxSize to (item 1 of params) * 1000000

    -- Iterate through the list of inputs
    repeat with input in inputs
        -- Translate the input (UNIX path) into an AppleScript file reference
        set inputReference to POSIX file input

        -- Get the size of the input
        set inputSize to size of (info for inputReference)

        -- Check whether the input is OK to duplicate
        if inputSize < maxSize then
            -- Duplicate input to output folder
            tell application "Finder" to duplicate inputReference to
outputFolderReference with replacing
        else
            log input & " is too big to duplicate"
            set returnValue to "Warning"
        end if
    end repeat

    return returnValue
end main
```

Shell Script Code Sample

MyCopy.sh

```
#!/bin/bash

# Split up the input file paths into a list
# via the Internal Field Separator (IFS)
OLDIFS=$IFS # Always keep the original IFS
IFS=":" # Now set it to ':'
# Split up '$1' into the 'inputs' variable, with ':' as separator
inputs=( $1 )
IFS=$OLDIFS # Restore the original IFS

inputCount=${#inputs[@]}

# Get the output folder
```



```

outputFolder=$2

# Get the maximum size from the script parameters
# and convert from megabyte to bytes
maxSize=`expr $3 \* 1000000`

exitValue=0

# Iterate through the list of inputs
for (( i=0; i<${inputCount}; i++ ));
do
  input=${inputs[$i]}
  # Get the size of the input
  inputSize=`ls -l "$input" | awk '{print $5}'`

  # Check whether the input is OK to copy
  if [ $inputSize -lt $maxSize ]; then
    # Copy input to output folder
    cp -f "$input" "$outputFolder"
  else
    echo "$input is too big to duplicate"
    exitValue=1
  fi
done

exit $exitValue

```

Windows Script Code Samples

MyHelloWorld.vbs

```

Function Main(inputs, outputFolder, params)
  WScript.Echo "Hello, World!"
  ' Set the function's return value.
  Main = "OK"
End Function

' Our main function is not using any of its arguments,
' so provide empty ones.
Dim inputs()
Dim outputFolder
Dim params()
Main inputs, outputFolder, params

```

MyCopy.vbs

```

Function Main(inputs, outputFolder, params)
  Dim returnValue
  returnValue = "OK"

  ' Get the maximum size from the script parameters
  ' and convert from megabyte to bytes.
  Dim maxSize
  maxSize = params(0) * 1000000

  ' Create the file system object that is going to do the copying.
  Dim filesystem
  Set filesystem = CreateObject("Scripting.FileSystemObject")

```

```

WScript.Echo "File system object created"

' Iterate through the list of inputs.
For Each input In inputs
' Turn the input path into a file object.
Dim inputReference
Set inputReference = filesys.GetFile(input)

' Check whether the input is OK to copy.
If inputReference.Size < maxSize Then
' Copy input to output folder.
Dim outputFile
outputFile = outputFolder & "\" & filesys.GetFileName(input)
filesys.CopyFile input, outputFile
Else
WScript.Echo input & " is too big to copy"
returnValue = "Warning"
End If
Next

' Set the function's return value.
Main = returnValue
End Function

```

Batch File Code Sample

MyCopy.bat

```

@echo off
rem First script argument contains the input file paths
set inputs=%1
rem Trim surrounding quotes
for /f "usebackq tokens=*" %%a in ('%inputs%') do set inputs=%%~a

rem Get the output folder
set outputFolder=%2

rem Get the maximum size from the script parameters
rem and convert from megabyte to bytes
set /a maxSize=%3*1000000

set exitValue=0

rem Iterate through the input file paths which are separated by ';'
:ITERATE_INPUTS
if "%inputs%"==" " goto EXIT
for /f "tokens=1 delims=;" %%a in ("%inputs%") do call :HANDLE_INPUT "%%a"
for /f "tokens=1* delims=;" %%a in ("%inputs%") do set inputs=%%~b
goto ITERATE_INPUTS

:HANDLE_INPUT
rem Get the size of the input
for %%? in (%1) do set inputSize=%%~z?
rem Check whether the input is OK to copy
if %inputSize% lss %maxSize% (
rem Copy input to output folder
copy /y %1 %outputFolder%
) else (
echo %1 is too big to copy
set exitValue=1

```

```
)  
goto :eof  
  
:EXIT  
exit %exitValue%
```

ExtendScript Code Samples

MyHelloWorld.jsx

```
function main (inputs, outputFolder, params) {  
    $.writeln ("Hello, World!");  
    alert ("Another way to say: Hello, World!");  
    return "OK";  
}  
// Our main function is not using any of its arguments, so you can test  
main without arguments  
main ();
```

MyPrint.jsx

```
function main(inputs, outputFolder, params) {  
    // First and only script parameter is the print preset.  
    var printPreset = params[0];  
  
    var returnValue = "OK";  
  
    // Iterate through the list of inputs.  
    for (i=0; i<inputs.length; i++)  
    {  
        var input = inputs[i];  
  
        try  
        {  
            // Open the input file.  
            var myDocument = app.open(File(input), DocumentColorSpace.CMYK);  
            // Create print options and set the print preset.  
            var options = new PrintOptions();  
            options.printPreset = printPreset;  
            // Print the document.  
            myDocument.print(options);  
            // Close the document without saving.  
            myDocument.close(SaveOptions.DONOTSAVECHANGES);  
        }  
        catch (e)  
        {  
            // Log problem info and set the return value to "Warning".  
            $.writeln("Problem while printing " + input);  
            $.writeln("Name: " + e.name);  
            $.writeln("Message: " + e.message);  
            returnValue = "Warning";  
        }  
    }  
  
    return returnValue;  
}
```


3. Jobs

3.1 Job Basics

3.1.1 The Job Concept

What is an Automation Engine Job?

An Automation Engine Job represents an amount of work to be done for a certain order in your business system. This can be a classic order management system but it could also come from a web application that needs Automation Engine to execute some prepress tasks.

A Job is an entry in the Automation Engine Jobs database and has a unique name. In the Pilot, Jobs are represented by this blue folder icon .

In Automation Engine, a Job consists of 2 things:

- **data:** files and folders, stored under the Job's **Job Folder**, in a Container.
- **metadata:** the parameters of that job order, usually how the order management system describes this job. They can be general parameters like the job's name, order ID, customer and due date but can also be parameters describing what to do with the input files to get to the targeted result.

Jobs can be created manually by an Automation Engine user or automatically via some integration with an external system.

A Job represents Work In Progress (WIP). This means that you should remove Jobs that are 'done'. Removing a Job from the Automation Engine Jobs database does not mean that you have to delete the related data files.

Most Automation is based on Jobs

A lot of the automation that Automation Engine brings comes from the fact that tasks can automatically use parameters from their Job context. Job parameters automatically become SmartNames and the (workflow) task tickets can use those SmartNames to pick up their values.

Note: Customers that do not use Jobs at all find a similar level of automation by using **Workflow Parameters**. Learn more in [Using Workflow Parameters](#) on page 1065.

Launching tasks on Jobs

Jobs can be managed manually in the Pilot, but several actions are also possible by launching tasks.

- The [Create Job task](#) on page 1280 task allows to create and modify Jobs.
- The [The Check Job Parameters Task](#) on page 874 task allows to check and correct barcode and ink parameters used in designs.

- The [Archive Job](#) task and the [Restore Job](#) task allow to remove Jobs, store their data on a different network place and, if needed later, restore them back, including their metadata.
- The [Relocate Job](#) task allows to move a Job's Job Folder to a different place. For example to another Container that has more disk space.

Learn more about these tasks in [Tasks you can Launch on Jobs](#) on page 869.

Like all tasks, these tasks can also be triggered by an external system. Learn more in [Integrating with External Systems](#) on page 1228.

3.1.2 Types of Jobs

Conceptually, we can distinguish several types of Automation Engine Jobs. This categorisation usually depends on these factors:

- What type of site are you? Does your company only do prepress or do you also produce? And if you produce, do you also re-produce items that you produced before?
- Is your Automation Engine closely integrated with your order management system? If it is, then Automation Engine best follows a similar process logic as in that system. Terms like jobs, orders, products, items, milestones etc. will need to have the same meaning.

We distinguish 3 types of Jobs:



Attention: The categorization of these job types does not appear in the user interface of Automation Engine. Their names are only used here to illustrate different ways of working with the tools Jobs and Products. Learn more about Products in [Products](#) on page 876.

Type 1: A Prepress Job

The goal of a Prepress Job is to make a single one-up design print-ready. Typical steps are preflight, making print-ready and approval. The result is a file that is ready to be produced.

Some variations:

- The production can then be done in-house (in another Job, under another order number) or the print-ready file can be sent out to a company that produces.
- The resulting file can be registered as an Automation Engine Product.
- A Prepress Job can result in multiple files, but only when they belong to the same product item, for example front and back graphics. They probably also share the same approval cycle.
- At (flexo) prepress houses ('trade-shops'), the result could also be a set of plates.

The Job can be removed when the work is done.

Type 2: A Production Job

A Production Job is a job that uses one or multiple product items for a specific production run. The product items were already approved and print-ready. The goal of this type of job is output.

A Production Job does not contain the one-up's data in its job folder, but links to one or multiple **Products**. The job folder of a Production Job typically only stores the layout files or other data that are unique to this specific production order. The product data stay in their **Product Data Zone**.

The Job can be removed when the work is done. The Products are not removed, they stay 'live' until a next Job links them in for another production run.

Type 3: An All-In-One Job

Some customers have Jobs in which they handle both the prepress work and the output production. The one-up data is stored inside the job folder, not registered as a Product and the workflow continues within this Job with the productions steps (layout, plates, send to press...). Production related data is also kept inside this job folder.

In some cases, when the output is combined with other (already existing) items, this can mean that the one-up data is copied from other (older) jobs into in this job folder.

The Job can be removed when the complete work is done.



Attention: In this chapter, we will describe the use of Jobs *without* the use of the Products tool.

3.1.3 The Job Views

General Job Views

The Pilot and its Views were introduced in [The Pilot's Main Window](#) on page 20. We here illustrate some job-related specifics:




The screenshot shows the 'Automation Engine Pilot' window. The 'Job Overview' view is active, displaying a table of jobs. The table has columns for Name, Description, Due Date, Order ID, and Sub Or... The job '1404-L86203_003' is selected. Below the table, the 'Job Details' panel shows information for the selected job, including Job Name, Order ID, SubOrder ID, Last Actor, Due Date, Category, Customer ID, Customer Name, Job Folder, and Description. The status bar at the bottom indicates '1 of 58 selected'.

Name	Description	Due Date	Order ID	Sub Or...
000100001			000100001	
002_barcode			002_barcode	
1009_BusinessC...			1009_BusinessC...	
1235		Dec 27, 2014	1235	
1236		Oct 11, 2014	1236	
1237		Dec 2, 2014	1237	
1404-L74536_002	New flavour Cocos and lemon	Jun 1, 2015	1404-L74536	002
1404-L86203_003	Promo strawberry 100ml south Europe cluster	Jun 1, 2015	1404-L86203	003
1404-W74536_002	New flavour Cocos and lemon _1	Jun 1, 2015	1404-W74536	002
1404-W74599_009	New flavour Banana	Jun 1, 2015	1404-W74599	009
1404-W86203_003	Promo strawberry 100ml South Europe_1	Jun 1, 2015	1404-W86203	003
5411028040230			5411028040230	
666-97634			666-97634	

Job Details for 1404-L86203_003:

- Job Name: 1404-L86203_003
- Order ID: 1404-L86203
- SubOrder ID: 003
- Last Actor: dva (rdvmaedemo)
- Due Date: Jun 1, 2015 10:00:00 AM
- Category: flexo
- Customer ID: 123456
- Customer Name: YoghurtXL Company
- Job Folder: file:///egsbe100/RND-SW-Sprints-Gent/Automation Engine/Public/demo_container/DVA/1404-L86203
- Description: Promo strawberry 100ml south Europe cluster

1. Type of View.

- The  **Containers** view shows the folder structure of your Containers but will show a folder in blue when it is an active Job Folder. Double click any blue folder to select and view that job.
- The  **Job Overview** shows all jobs (or a filtered list). You can combine this with *Smart Views* on page 163.
- The  **Last used Job** view opens the last used job in the *job specific view*.

2. **Info.** This button opens the **Job Setup** of the selected job. Learn more in *Job Setup Parameters* on page 837.

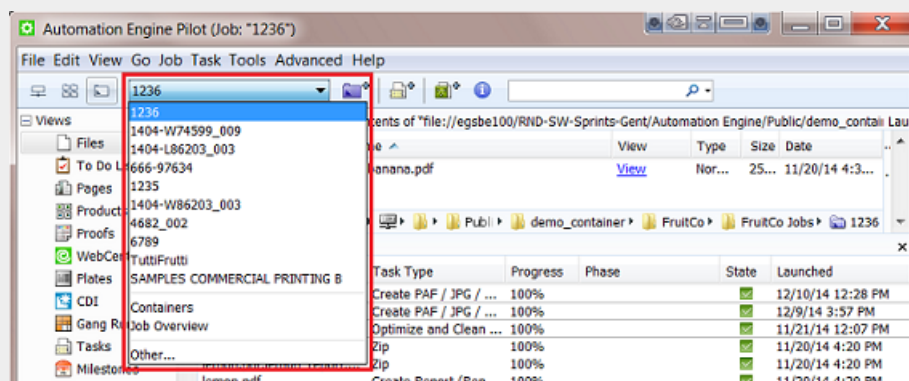
3. **Search** input field. Learn more in *Job View Filters* on page 832.

4. **Job Overview.** Define the columns and their order in **View > Select Columns in 'Jobs'...**

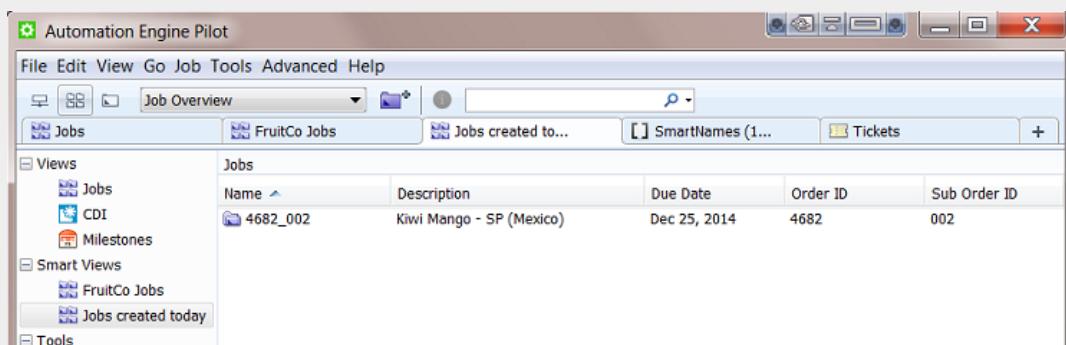
5. **Job Details.** See some of the job's details. Learn more in *Job Setup Parameters* on page 837.

6. **Amount of Jobs (selected).**

Note: When you open the dropdown list, you can select another job that you recently used. You can set the amount of remembered jobs in **Edit > Preferences > Pilot**.



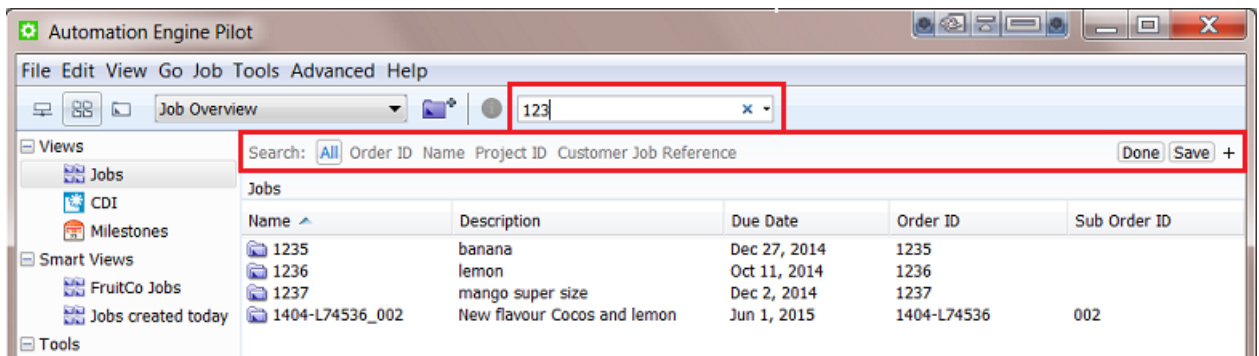
Tip: If you regularly switch between same views, consider using multiple view tabs (**File > New Tab**). For example:



Job View Filters

Basic Searching

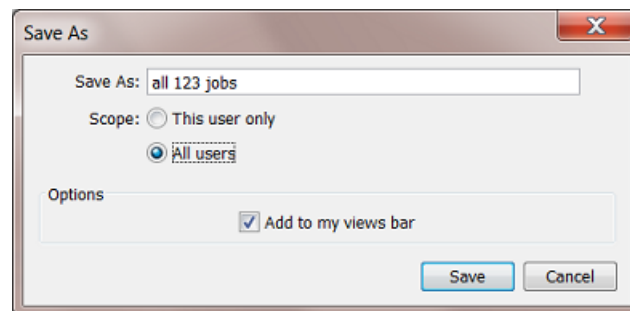
When you type in a filter in the **Search** input field, the filter will be looked for in **All** of the default attributes Order ID, Name, Project ID, Customer Job Reference.



Tip: You can use double quotes to restrict the search. For example

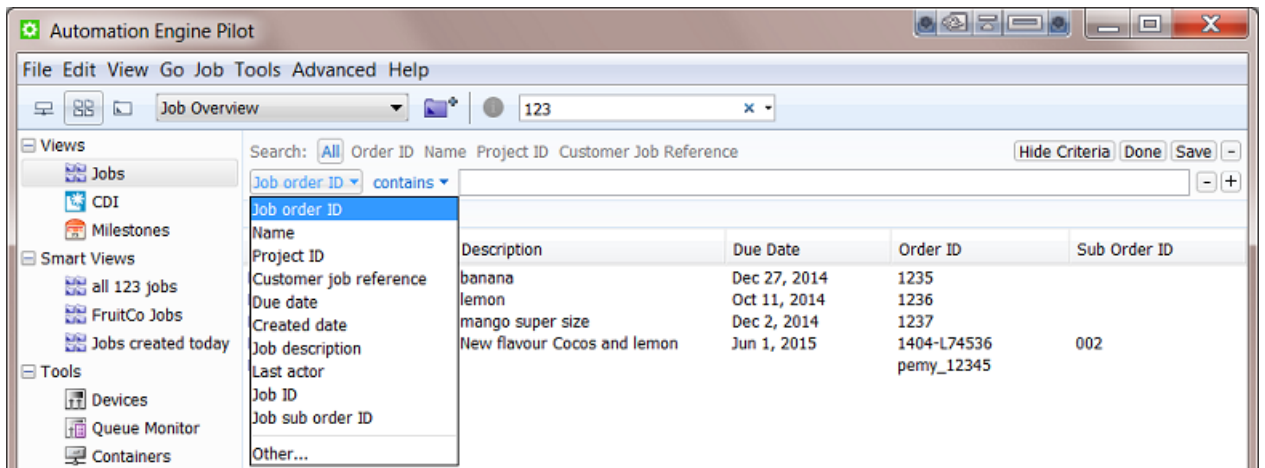
- FL 123 will list jobs that have both FL and 123 in at least one attribute.
- "FL 123" will only list jobs that have this exact string in at least one attribute.

Click **Save** to save this search. Decide if it is for all users or only yourself and if you want to add it to your views bar.

**Advanced Searching**

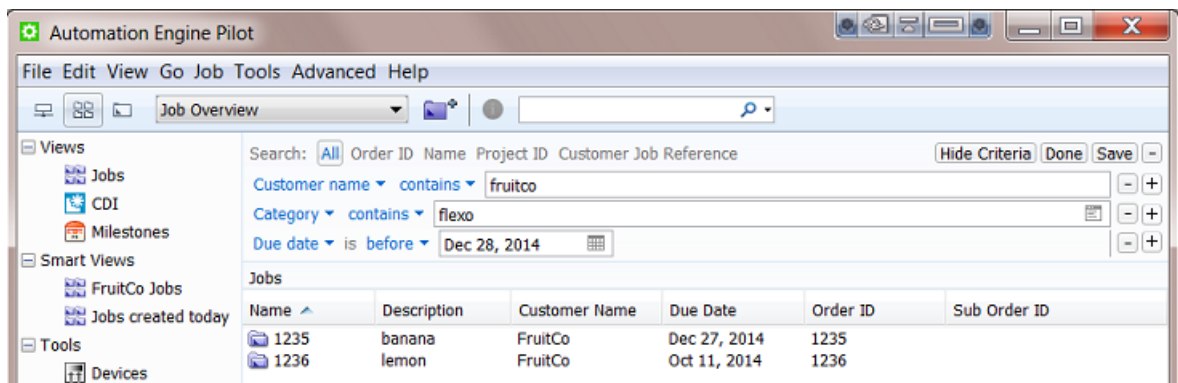
If you want to search in other attributes, or if you want to create advanced search criteria, click on the + button. A new search criterion appears.

See how the attributes dropdown list offers many more attributes:

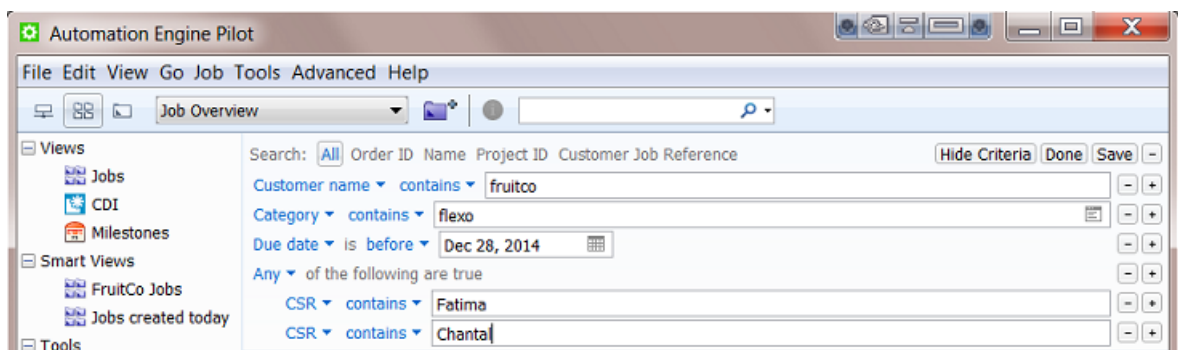


In the dropdown list, select **Other...** to add even more attributes to this list. For example **Customer name**.

Use the + button to continue adding search criteria. For example:

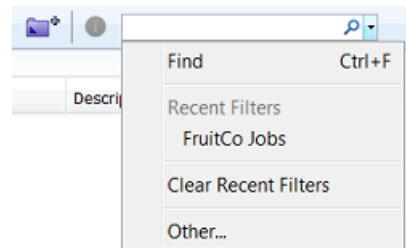


You can even add 'logical operators' to your criterion. Hold **Alt** when clicking on the + to start your criterion with **All**, **Any** or **None**. See this example where you are not sure which CSR handles this job order:



Click **Hide Criteria** to hide the lines with the criteria but still have them active.

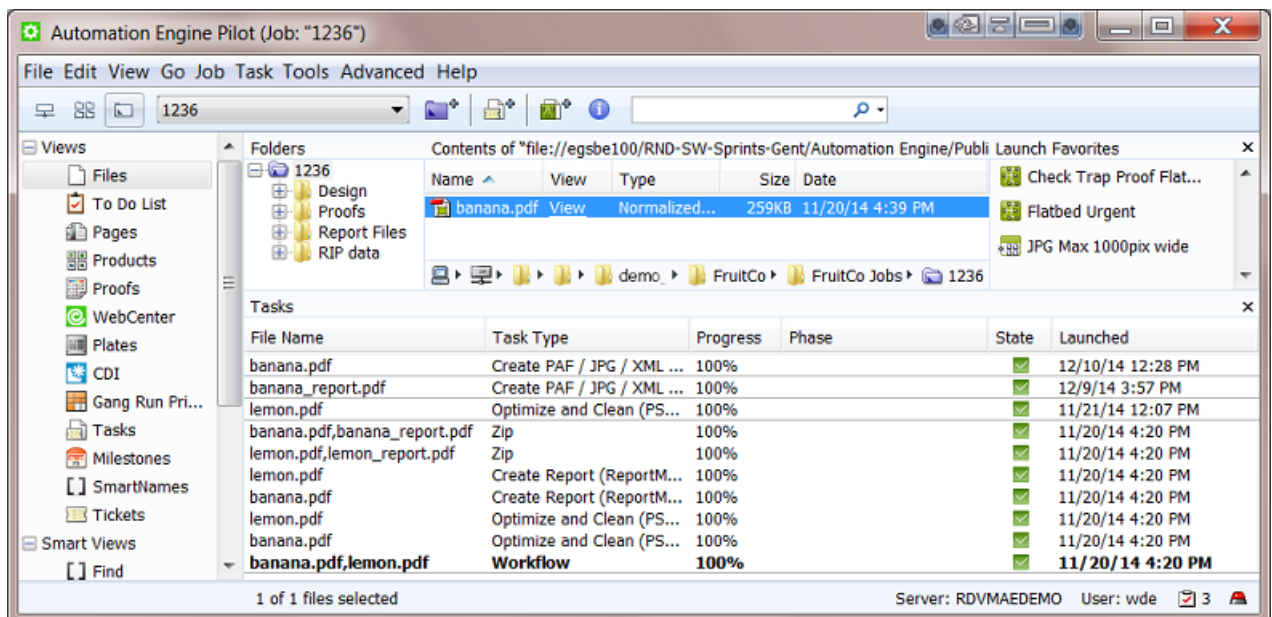
As shown above, you can save the (set of) search criteria. You can re-use it later as a Smart View or pick it from the dropdown list in the search field:



Job Specific View

When you select a job from the dropdown list of last used jobs, or when you double click a blue job folder, the view adapts and focuses on that job.

- The Pilot top bar mentions the name of the selected job.
- The **Last Used Job** drop down list mentions the name of the job folder.
- The Folders pane shows the (blue) job folder, its subfolders and their files.
- The Tasks pane shows the tasks that were launched in this job context.
- The list of favorite tickets also shows (blue) job tickets (saved for this job only).



All tasks that are launched in this view will run "in this job". Job-based SmartNames will now get their values from this job.

3.1.4 The Job Menu

Note: When no job is selected, some of the menu options are unavailable.


- **New Job....** Learn more in [Creating a Job](#) on page 835.
- **Go to Job.** When a job is selected in a list, use this to go to that job ('open' the job).
- **Job Setup....** This opens the Job Setup dialog of the selected job.
- **Find Job....** This opens a separate dialog to search for jobs. Learn more in [Job View Filters](#) on page 832.
- **Relocate Job.** This moves the job folder of a (live) job. Select the ticket of the **Relocate Job** task. Learn more in [Tasks you can Launch on Jobs](#) on page 869.
- **Remove Job....** This removes the open or selected job(s) from the jobs database. You will be asked to confirm and you can choose to keep the files and folders. Learn more about removing multiple jobs and re-creating them in [About Removing Jobs](#) on page 867.
- **Delete View Data....** This deletes all View Data present in the job folder. View data are stored in hidden subfolders of files that were prepared for viewing in [the Automation Engine Viewer](#). You will be asked to confirm. You can choose to **Also unmark subfolders that are marked for Speed for Viewing**. Learn more about this feature and View Data in [Prepare for Viewer](#) on page 415.
- **Manage Job Filters....** This opens a dedicated dialog to manage job filters. Learn more in [Job View Filters](#) on page 832.

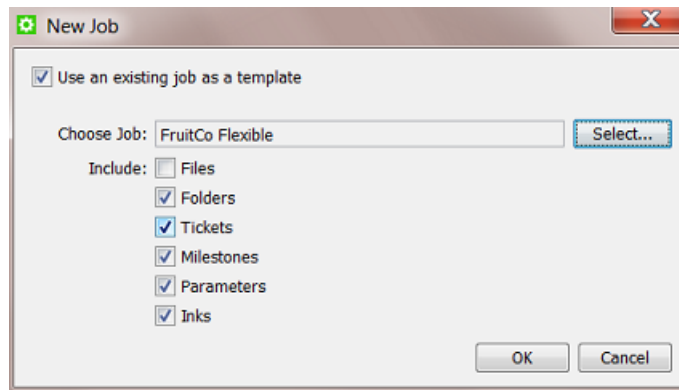
3.2 Creating a Job

You can create jobs manually or they can be created automatically.

Creating a Job Manually

To create a job, you can first select an existing folder that will become the job folder, or you can have a new job folder created while you create the job. Follow these steps:

1. In the Pilot, choose one of these methods to create a new job:
 - go to **Job > New Job...**
 - right-click (on an existing folder) and select **New Job...**
 - click the **Create Job** icon 
 - (select a folder and) use **Ctrl + Alt + N** (Windows) or **Cmd + Alt + N** (Mac).
2. In the **New Job** dialog, decide if you want to **use an existing job as a template**.
 - If you don't use a template, click **OK** and fill in all the parameters in the next dialog.
 - If you want to an existing job as a template, select it and decide which type of parameters you want to include from that job in your new job. Then click **OK**.



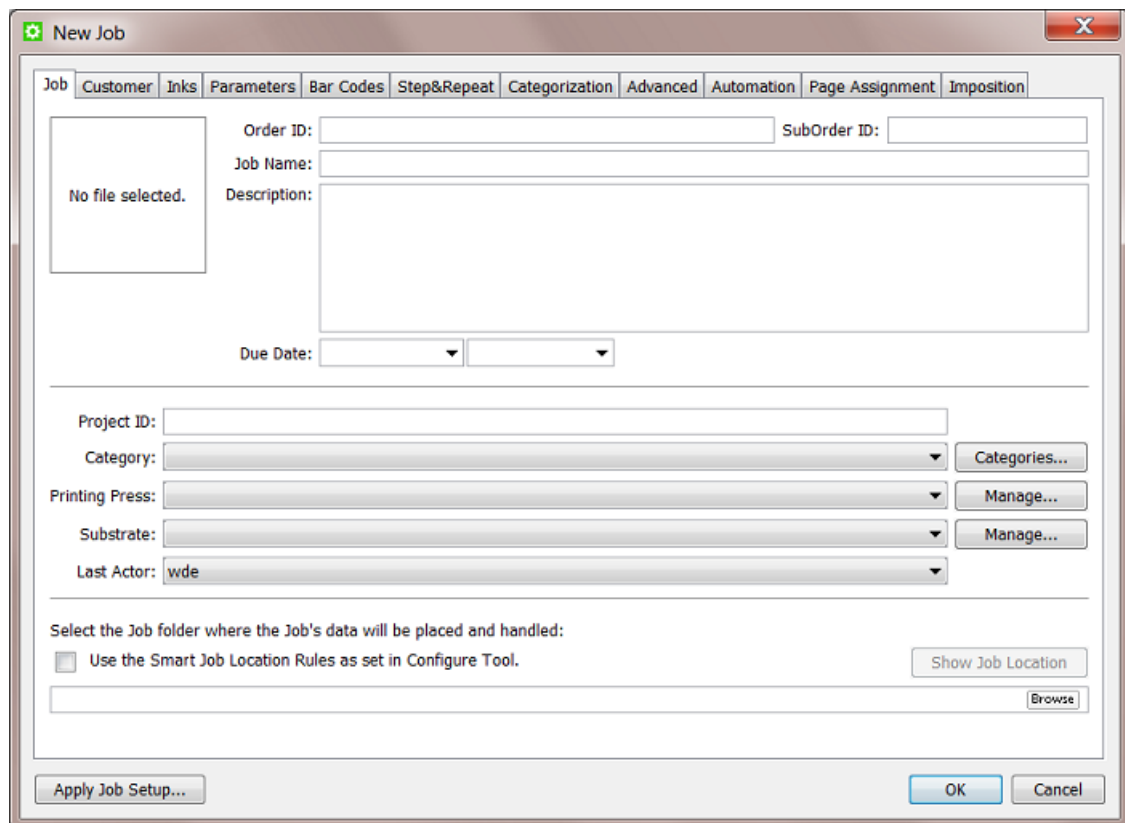
The job you use as a template could be a job that is similar to the new one (for example the 123-lemon job has many same attributes as the 234-banana job). It could also be a job that you specifically created to be used as a template (the typical 'flexibles' job for example).

The milestones that you copy from the template job will be added in the state that they were: confirmed or not, with related milestone actions or not etc. Their time stamp will be relative to the creation date and time of your new job. For example: you want your new job to also get the milestone 'proof ready' 2 days after it was created.

Tip: Copying milestones from another job will be useful when they are not confirmed yet.

3. A dialog opens with the Job Setup. Now you can fill in all the parameters for the job.

If you started from an existing folder or used a template, several parameters will already be filled in.



Learn more in [Job Setup Parameters](#) on page 837.

Creating Jobs Automatically

In most cases, the Automation Engine Job originates in another system. Recreating that job manually in the Pilot is not an efficient way of working. It is strongly advised to integrate Automation Engine with that other system and so have the jobs come in automatically. This will both win you time and reduce errors.

This integration can be based on an incoming file that describes the job or could be a bi-directional link with that external system. In both cases, the [Create Job task](#) on page 1280 task will create the Job in background (probably as part of a workflow task). Such an automated workflow can also update Jobs with new or changed parameters.

Learn more in [Integrating with External Systems](#) on page 1228.

3.3 Job Setup Parameters

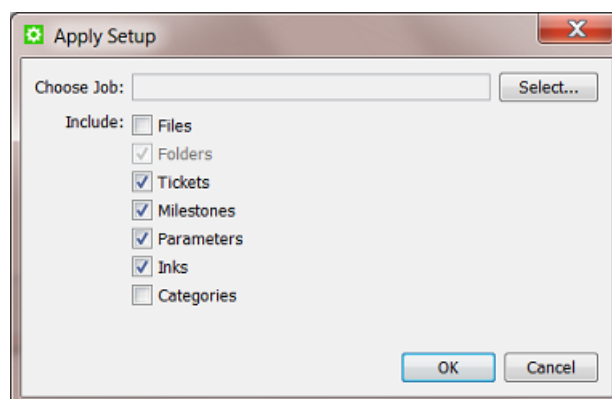
You can fill in the job setup parameters while [creating a job](#), or, when you are in a job context, by going to **Job > Job Setup...** .

In an existing job, you can add or modify parameters at any time, except the **Order ID** and **SubOrder ID**. Learn more in [Job tab](#) on page 838.

The parameters are spread over multiple tab sheets. The tab sheets **Automation**, **Page Assignment** and **Imposition** only appear when you are using the module for page workflows.

Most job parameters are automatically offered as Job SmartNames. Learn more in [SmartNames](#) on page 916.

The button **Apply Job Setup...** allows to copy parameters from other jobs. Select the job and define which type of parameters you want to include from that job.



This is similar to how you can use a template job at the moment of job creation. Learn more in [Creating a Job](#) on page 835.

3.3.1 Job tab

The **Job** tab contains the main job order information:

The screenshot shows the 'New Job' dialog box with the following details:

- Order ID:** 1245
- SubOrder ID:** 001
- Job Name:** 1245_001
- Description:** lemon flavor, Japanese market
- Due Date:** Jan 21, 2015, 12:30 PM
- Project ID:** (empty)
- Category:** flexo
- Printing Press:** Gallus ECS 340
- Substrate:** (empty)
- Last Actor:** Thomas
- Job Location:** file://aeserver01/demo_container/FruitCo/FruitCo Jobs/1245_001

Order ID: order management systems typically define a job order by an identification number. This is a mandatory field.



Caution: You can not change it once the job is created.

SubOrder ID: often, a job order also has a sub number (or text). You can leave this field blank.



Caution: You can not change it once the job is created.

Important: An Automation Engine job needs to be unique. More specifically, the combination of Order ID and Suborder ID needs to be unique.

Job Name: this is the name that you see in the job overview lists. This is a mandatory field. If you do not enter a name here, the name will be "OrderID_SubOrderID".

Important: Also the name of the job needs to be unique.

Note: You can change the name after the job was created.

Note: The job name will also be proposed as name for the job folder. Learn more below.

Description: a short text describing the job order. By using its SmartName version, this text can for example also be picked up in a report PDF.

Due Date: the due date and time for this job order. This date is used in job filtering and also in several tasks that allow choosing the **Job Due Date** as one of the choices for the task due date. For example the [Submit to Gang Run](#) on page 1109 task and the [Publish on WebCenter task](#) on page 1353 task.

Project ID: the identification number of the project that this job order belongs to. Project numbers are typically defined by the order management system or any other external system. This parameter can be useful to find jobs that belong to a same project. This parameter can match to a project on WebCenter.


Category: choose a category from the list. Categories are often used to help you find jobs or to define the route of a workflow. Click the button **Categories...** to add new categories manually or by loading a list from an XML file.

You can add more custom categories. Two are offered in this tab sheet: **Custom Field 1** and **Custom Field 2**. You can set four more in the tab [Categorization](#). You can configure their custom names in [Configure - Jobs - Job Setup](#). See an example in the above screen shot.

Tip: In the page about the tab [Categorization](#), you can find a table with matching names of categories, custom fields and their SmartName versions.

Last Actor: the last Automation Engine user that worked with this Job (launched a task, opened a file). This is filled in automatically but you can set it manually too.

Job Folder: define the path of the job folder for this job. You can choose one of these methods:

- browse to an existing folder (somewhere in one of your containers).
- browse to the root of where you want to create the new job folder and click . By default, the **Job Name** will be proposed as name for the job folder. You can change that if you want to.
- **Use the Smart Job Location** defined in the Configure Tool: when this toggle is selected, the location and name of the job folder will be calculated based on a job parameter or on the default location for that customer. A preview of the folder's location and name can be asked by clicking the **Show Job Location** button. Learn more in [Smart Job Location](#) on page 663.

Tip: The **browse** button can also be used to relocate an existing job folder. When you here change the path of an existing job folder, you will be asked if you want to move or copy the data to the new job folder.

3.3.2 Customer tab

Customer ID: click **Select...** to open the **Select Customers** dialog. Choose one or add a new customer. Learn more about creating new customers in [New Customer](#) on page 697.

Customer Name and **Customer Description:** these fields will be filled out automatically after selecting a customer.

Customer's Job Reference: this is how your customer refers to that job, how they named it (initially). When communicating with your customer, he will appreciate if you mention this name (as well). You

can for example add this name as a SmartName to a report PDF or add it to a name of a file that you send him.

Customer Service Representative (CSR): select or type the name of the person in your sales or service department who manages this customer account.

The **E-mail** address of the CSR that you selected is automatically displayed if it was used before. You can also enter the E-mail address manually.

You can click the **Contact...** button to open an E-mail message with that E-mail address in the **To** field (if an E-mail client is configured).

Tip: All these parameters

- can be used to filter or search for a job.
- can be entered via XML/JDF/JMF. It is strongly advised not to re-enter these admin data manually. Learn more on how this can be done in [Integrating with External Systems](#) on page 1228.
- exist as **System SmartNames**.

3.3.3 Inks tab

This tab sheet allows to store ink specifications for this job.

The screenshot shows the 'Job Setup' dialog box with the 'Inks' tab selected. The 'Preferred Ink Book' is set to 'PANTONE+ Solid Coated'. The table below lists the ink specifications:

Ink Name	Ink Book	Ruling	Angle	Dot Shape	Ink Type	Printing Method
Cyan	process	95 lpi	7 °		Normal	Flexo
Magenta	process	95 lpi	37 °		Normal	Flexo
Yellow	process	95 lpi	82 °		Normal	Flexo
Black	process	95 lpi	67 °		Normal	Flexo
PANTONE 123 C	PANTONE+ Solid Coated	95 lpi	7 °		Normal	Flexo
die	designer	95 lpi			Technical	Other
gold	designer	95 lpi			Opaque	Screen

Default values, used when a ruling, angle or dotshape is not specified:
 Ruling: 120 lpi Angle: 37 ° Dotshape:

If there are inks specified in the list, but no default values are set, factory settings will be used for ruling (120 lpi) and angle (45°).

What you enter here serves as a *minimal* requirement for this job. This means that you can for example only enter the angles for a specific set of ink names and leave all other fields or columns blank.

As with most tab sheets, you can enter this data manually, copy these parameters from another job (see the button **Apply Job Setup...**) or get them from an XML when using the [Create Job task](#) on page 1280.

Use cases

Job ink parameters can be used in these cases:

- When you import design files, the task [Normalize PostScript / PDF / Illustrator 8.0 File](#) on page 379 offers to enforce these job ink parameters in the output file.
- In PackEdge or ArtPro, when you open a file that is located in a job folder, the editor will check if the file's inks match the job ink parameters and show a warning when they don't. You can also use the **Load Job Parameters** feature in the editor's **Inks** dialog to enforce the job ink parameters.
- When you use the [Check Job Parameter task](#), you can check and or modify the file's inks.
- In page workflows, when you use the **Variant Setup** in the [Imposition tab](#), the job's inks will be used.

Note: A job can only have one set of ink parameters.

Preferred Ink Book: when you then type an ink name in the table, Automation Engine searches for the ink in your preferred ink book first. For example, if you type 154 as ink name, and your preferred ink book is `Pantone Colors Uncoated`, Automation Engine will automatically suggest the ink `PANTONE 154 Uncoated`.

Printing Method: you can set a printing method per ink. The list shows those defined in the **Configure** tool.

These are typical examples of how to make use of a **Printing Method**:

- In your **Imaging Engine** (RIP) task for flexo, you select only the inks of printing method 'flexo'.
- You make a special ReportMaker PDF that focusses on the embossing 'ink'. In your workflow, you use the [Extract Inks](#) on page 350 task to select the wanted ink type before you start the ReportMaker task.

Learn more in [Printing Methods](#) on page 673.

Some tips on editing the ink list

- To start, click in the empty canvas under the column **Ink Name** and type in an ink name. For example 'C'.
- You can define specify ruling, angle and dot shape per ink name or use the buttons at the bottom of the tab sheet to specify them for *all* inks.
- **Ruling** uses the unit as defined in **Edit > Preferences... > Units** .
- To change all fields in a column with a same value, double click the column header (valid for **Ruling**, **Angle** or **Dot Shape**).

3.3.4 Parameters tab

This tab allows to store two types of job parameters:

- custom job parameters, with a name and value that you decide. They will also appear as a SmartName.
- FlexRip/Proof job parameters. This is an older technology that allows enforcing job parameters into the FlexRip or FlexProof task tickets. Learn more in [FlexRip/Proof Job Parameters](#) on page 843.

Note: With Imaging Engine, this concept is replaced by the use of SmartNames.

These job parameters can be created

- manually. Learn more in [Creating and Modifying a Custom Job Parameter](#) on page 842.
- by loading values from an XML file. Learn more in [Loading Job Parameters from an XML File](#) on page 842.
- by using the **Create Job** task, where you use the (XML) input from an external system. Learn more in [Create Job task](#) on page 1280.

Creating and Modifying a Custom Job Parameter

Use the buttons **Add...**, **Modify...** and **Remove** to manage your custom parameters manually.

Follow these steps to add a custom job parameter:

1. Click **Add...** to open the **Add Job Parameter** dialog.
2. Choose the **Scope** 'General'. Learn more about the other Scopes in [FlexRip/Proof Job Parameters](#) on page 843.
3. Type in the **Name** and **Value** and click **OK**.

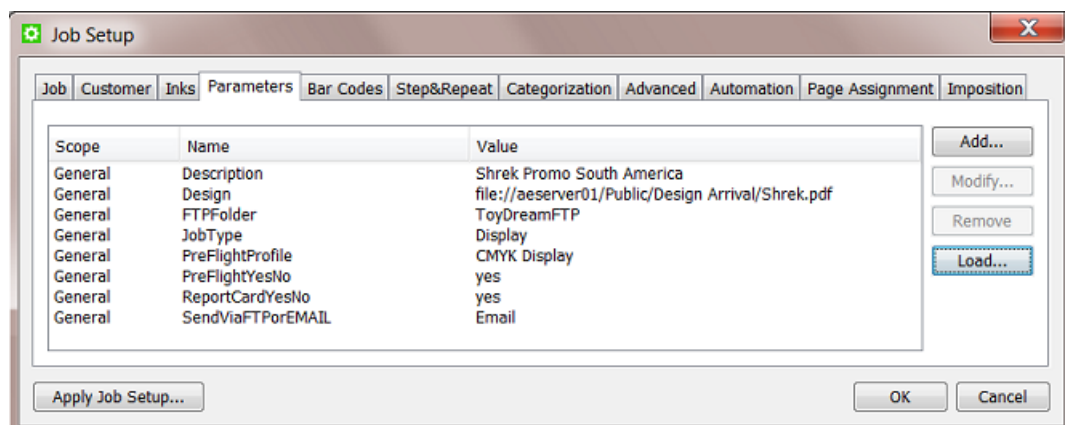
The SmartName version will be created automatically. To easily recognize which SmartNames are job specific, their name starts with [jp.] .

Loading Job Parameters from an XML File

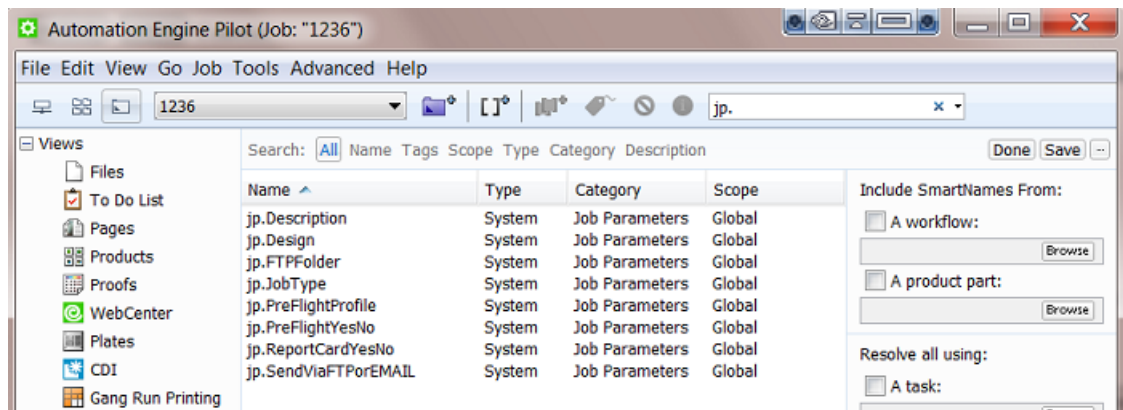
If you have an XML file that contains the custom job parameters, you can create them automatically by loading them into your job setup:

1. In the job setup tab **Parameters**, click **Load....**
2. In the **Load Job Parameters from XML** dialog, click **Browse** and browse to your XML file.
3. Click **OK**. You now have a custom parameter for all value pairs the XML contains from the root node.

For example: the result after loading the XML file:



For this example, see the automatically created SmartNames of these custom job parameters (for this job '1236'):

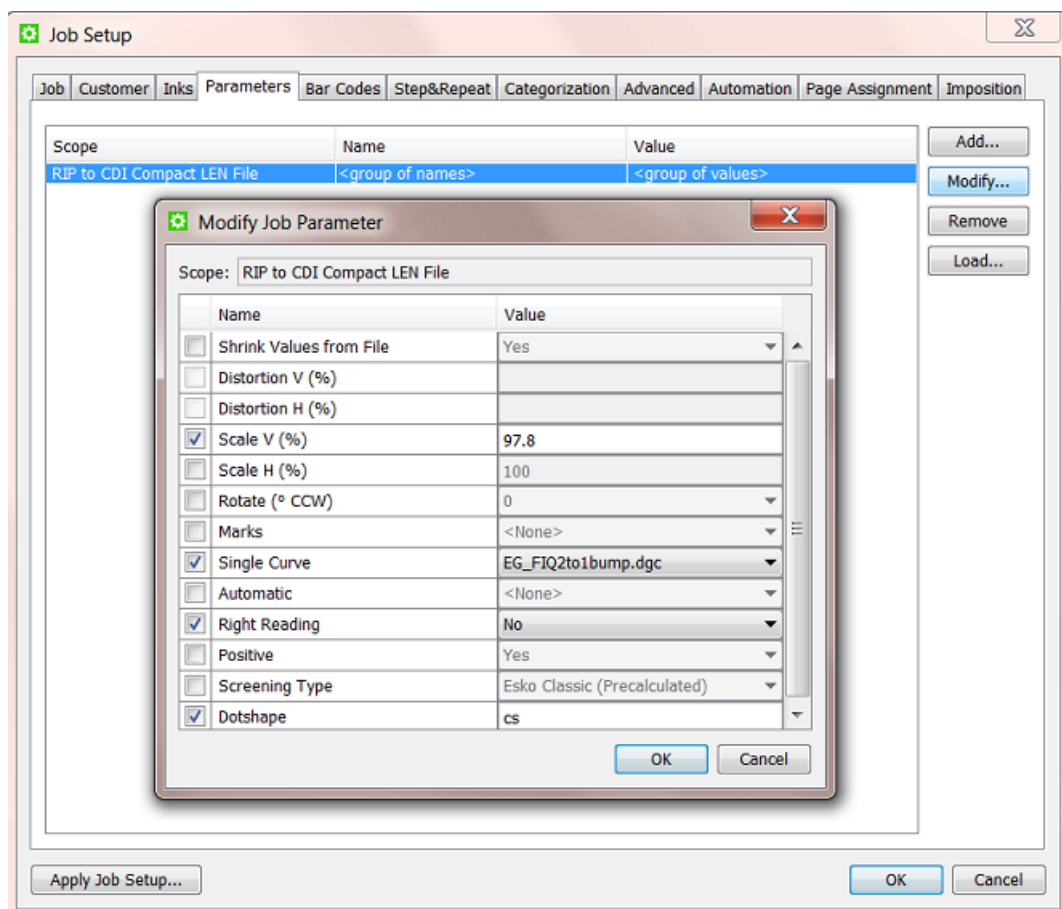


Learn more about using SmartNames in [SmartNames](#) on page 916.

FlexRip/Proof Job Parameters


Creating Job Parameters for FlexRip or FlexProof

1. Click **Add...** to open the **Add Job Parameter** dialog.
2. Choose the **Scope** of a specific type of FlexRip or FlexProof.
3. Check the parameters that you want to enforce during RIP'ing and define their value. For example:



Click **OK**. The job parameters for this type of FlexRip/Proof will be added as one line.

Using Job Parameters for FlexRip or FlexProof

When you now select a file in that job and open a task ticket for that particular FlexRip/Proof, you will see fields that are marked by an orange icon . These are fields that are overruled by the job parameters.

For example:

Note: With Imaging Engine, this concept is replaced by the use of SmartNames.

3.3.5 Bar Codes tab


In the **Bar Codes** tab you can create parameters for multiple bar codes of different types.

Tip: One file can have for example a retail bar code and also a general purpose bar code that will be used for a filling station.

As with most tab sheets, you can enter this data manually or get them from an XML file when using the [Create Job task](#) on page 1280.

Tip: When selecting a bar code, the specified parameters are shown in the **Details** area.

Similar to the [Inks tab](#) on page 840, these parameters can be used to enforce their parameters into the bar codes of the job files that you opened in PackEdge or ArtPro. Also here, the [Check Job Parameter task](#) can be used to check or correct the bar codes in the files versus the parameters you specify here.

See this example where the job parameters are enforced  into the bar code in PackEdge:

In AE Job Setup



In PackEdge

We now briefly describe the available parameters. We refer to the [documentation of your editor](#) for more details.

Type

Selecting a Type is a mandatory choice.

Note: When you define more than one bar code, the checking mechanism with the editors and the [Check Job Parameter task](#) will recognize the type of bar code and check accordingly.

Sub Type

Some bar code types offer sub types. For example a **Datamatrix** code.

Code

The validity of the code is not checked here. This is done in the editors.

Composite Code

A **Composite Code** is a 2D symbol placed on top of a GS1 barcode, that encodes extra information (batch number, expiration date...).

Output Resolution

Enter the resolution (in ppi) that you will use to RIP your bar code. This parameter helps preventing unwanted levels of rounding in the bar width.)

Bar Width Reduction (BWR)

Use this parameter to adjust the width of the bar code bars, in order to compensate for ink bleeding when printing on your substrate.

Your printer or customer will usually provide you with the value to use. For example if your printer tells you that the line width will increase by 0.01 mm, you should use a **Bar Width Reduction** value of 0.01 mm.

Important: This field expects a value in millimeters (mm).

Important: If you use a comma as a decimal symbol, it will be replaced by a dot.

Tip: You can enter a negative value if you want to print thicker and not thinner bars.

Device Compensation

Use this if your printer gave you a separate device bleed value. This parameter adjusts the bar's width to compensate for bleed caused by the plate or film making process.

Important: This field expects a value in millimeters (mm).

Note: If you use both a **Bar Width Reduction** and a **Device Compensation** value, the software will add them together to adjust the bar's width.

Tip: You can enter a negative value if you want to print thicker and not thinner bars.

Narrow Bar

This parameter determines the width of the narrowest bar of your bar code (and affects the width of your bar code proportionally). For GS1 bar codes, this affects the size (height and width) of your bar code proportionally.

Magnification

Use this parameter to increase or decrease the bar code size proportionally. For example: enter 1.25 for a magnification of 125%.

Important: The value needs to be between 80% and 200%.

Ratio

The **Ratio** is the factor between the width of the narrowest bar and the width of the widest bar in your barcode. For example, enter 3 if you want your widest bar to be three times as wide as your narrowest bar.

3.3.6 Step&Repeat tab

Specific JDF

This tab with Step and Repeat parameters is only useful in a workflow where you receive a JDF from an external system that describes the Step and Repeat via “JDF for Stripping for Packaging”. Such a JDF will not only create the job but also store those specific Step and Repeat parameters in this tab.

These parameters also become job parameter SmartNames and can so be picked up by Step and Repeat tasks. They will appear as `[jp.steprepeat.]`.

Note: The task *Import JDF Step & Repeat* on page 355 uses the JDF itself as input. In that case the values shown here are just informational and the task does not use SmartNames.

Also via XML?

Step & Repeat specifications can also come from external systems via an XML file or database queries or HTTP calls. In such cases, these parameters do **not** end up in this tab sheet and they also become other types of SmartNames. Learn more about this in *Integrating with External Systems* on page 1228.

Manual entry

You can also enter values manually here but that usage is not different than when you would add them in the custom **Parameters** tab sheet.

3.3.7 Categorization tab



Attention: This tab only appears if you created custom categories in the Configure tool (see further).

The **Categorization** tab allows you to manage up to four extra custom Job Categories (you already have three category fields in the **Job** tab).

You can use these custom categories as extra ways to search jobs. And you can also use their SmartNames in task tickets.

Creating and modifying a Custom Category

Go to **Tools > Configure > Jobs > Job Setup** and type the name of custom category.

Managing their Values

To manage the list of values you can choose for each category, click the corresponding **Manage...** button, and add or remove items to the list. In this dialog, you can also load lists of items from an XML file by clicking **Load...**

Overview of Custom Categories and their SmartNames

Because there is already 1 *standard Job Category*, the number of your custom categories will be 1 higher. See this overview:

Table: Names of custom Categories

name of Job setup tab	name of category in Job setup	custom field number	name of SmartName
Job	"Category"	/	Job Category
Job	name you gave	custom field 1	Job Category 2
Job	name you gave	custom field 2	Job Category 3
Categorization	name you gave	custom field 3	Job Category 4
Categorization	name you gave	custom field 4	Job Category 5
Categorization	name you gave	custom field 5	Job Category 6
Categorization	name you gave	custom field 6	Job Category 7

3.3.8 Advanced tab

This tab sheet offers a feature to automatically start a task or workflow when you create or update a job that uses this job as a template.



Caution: This is an old feature. Please check below for the alternative ways to reach the same goal.

If you wish to use this, check the toggle and select the **Task Type** and **Ticket Name**.

When is a job updated?

A job is updated in these cases:

- when the JDF that created the job is sent again to Automation Engine. Learn more about JDF in [Integrating with External Systems](#) on page 1228.
- when a **Create Job** task is launched again on the same job.

When is this feature useful?

This feature can be useful when you have jobs that are created automatically via JDF without the automatic launching of tasks at that moment. Your JDF can then specify that a job needs to be used as a template. And that template job can have this parameter active to launch a task.

Alternative methods

There are other ways to launch a workflow in a job that was just created or updated.

The [Create Job task](#) on page 1280 can

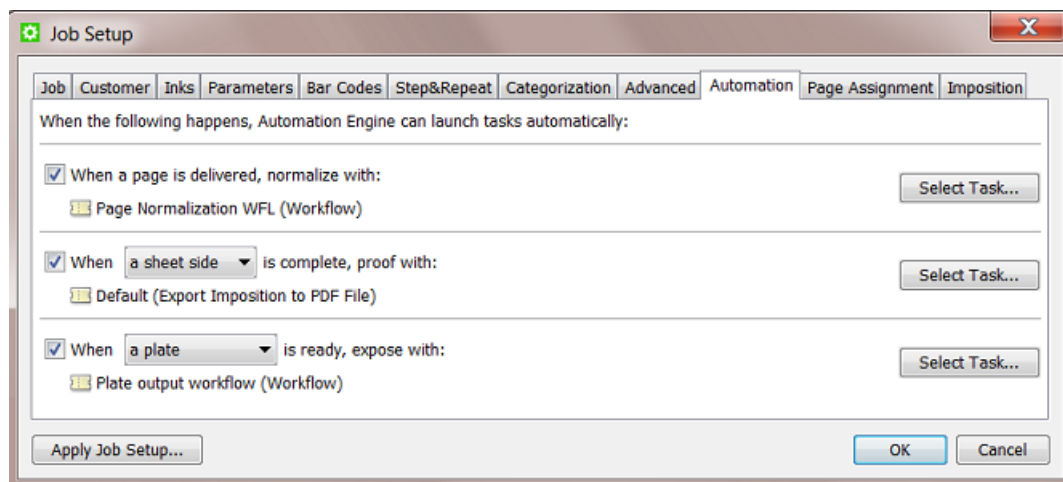
- be part of a workflow, so right after this step where the job is created your workflow continues with more steps.
- can be started automatically by an Access Point. Learn more in [Access Points](#) on page 1236.
- also has a tab sheet with this same feature. Learn more in [Create Job task](#) on page 1280.

3.3.9 Automation tab

Note: This tab is only offered in page workflows.

The **Automation** tab offers tools to automate your page workflow. When a certain job related event happens, a specific task can be launched automatically, which then can lead to a next status in the workflow.

You can do this for **Normalization, Proofing** and **Exposing**.



- Choose how you want to normalize the file **when a page is delivered**. Select a single task or a workflow.
- Choose when you want to launch a **proofing** output task: when **a sheet side, a variant or an imposition** is complete. Select a task.
- Choose when to **expose** your job: when **a plate, a separation set or an imposition** is ready (complete and approved if required). Select a single task or a workflow.

3.3.10 Page Assignment tab

Note: This tab is only offered in page workflows.

This tab offers settings to automate the assigning of incoming PDF files to Page slots.

Learn more in [Page Assignment via Naming Convention](#).

Learn more about the Pages tool in [Working with Pages](#) on page 182.

3.3.11 Imposition tab

Note: This tab is only offered in page workflows.

- **Variant Setup**

When working with variants, this allows you to select the correct use case and variant separation from the list. Learn more in [Variant Setup](#), a section of [Working with Plates](#) on page 192.

- **Output prerequisites**

These settings allow you to decide when a separation can be considered as being ready.

- **When no page is rejected:** This is called "implicit approval". The plate status in the **Plates View** will indicate 'ready for ripping' if all the pages of the plate are assigned and if no page is rejected. When a page for that plate is unassigned or rejected, the output task on a plate will fail.
- **When all pages are approved:** This is called "explicit approval", it is more strict than "implicit approval". The plate status in the **Plates View** will only indicate 'ready for ripping' if all the pages of the plate are approved. When not all the pages on that plate are approved, the output task on a plate will fail.

Learn more in [Task status information](#).

Learn more about the Plates tool in [Working with Plates](#) on page 192.

3.4 About Job Folder Location

3.4.1 General Note on Data Structure

Automation Engine allows you to choose your own data structure

As mentioned in [Data Management in Automation Engine](#) on page 23, Automation Engine will work with the data structure you have.

This freedom also brings requests for advice on what data structure could serve your specific needs the best. The more you learn about the available tools (Jobs, Products, SmartNames, Workflow Parameters etc.) the more you will learn what is (no longer) important in that data structure.

Historical View on Data Structure

Before there were workflow automation servers like Automation Engine, prepress managers built up their own logic in the prepress data structure. Prepress operators browsed many folders in the network to open a file in their workstation editor and then saved it again, often with a changed name.

Very typical was the habit to write information about the files into both their name and their location. For example: a final version was named "name_final.pdf" or written in a folder "Final". Or a file was named "lemon_c_m_y_k_pms123.pdf". Or when the job was ready to be archived they moved the whole job folder to a share "For Archive".

Then came workflow servers like Automation Engine where the information is (automatically) stored in database fields. And the user can see information via a dedicated interface: the Pilot. You can for example list all jobs of the category "NON-FOOD" or you can use the "i" button to see the PDF's inks (XMP metadata).

Does this mean that the data structure and the file naming is less important than before? No, actually the opposite is true.

Why is it still Advised to have a Logical Data Structure?

There are two main reasons why a logical data structure is more important than ever:

- **for the people that do not use the Pilot.**

Are there non-prepress users looking directly at your prepress data structure? If those people are not using the Pilot interface, they will still want to get information from the location or from the name (like in the 'old days').

Note: We advise to check if those colleagues will not prefer to install a local Pilot and benefit from that information-rich view. Remember you can use User Access Rights to limit their access. And they could even benefit from using the **Viewer**.

- **for external systems** that look at, use or deliver the data.

What other systems are looking at your data structure or at files that come from there? What systems send you data into that data structure?

The integration of Automation Engine with many other systems is the main reason that your data more than ever needs to be well structured and named. Learn more in [Integrating with External Systems](#) on page 1228.

For these reasons we advise to:

- investigate thoroughly what data structure will serve your needs and those of the system(s) you (will) integrate with. This includes avoiding copied data.
- use all the tools that can automate the creation of those folders and file names (templates, SmartNames, Smart Job Location etc.).

3.4.2 FAQs on Job Folder Location

Can I Move a Job Folder?

As explained in [The Job Concept](#) on page 24, a job is a temporary object. But even during its short life time, it can happen that you want to move the physical data to another place in the network.

In this case, it is very important that you use one of the tools in Automation Engine to do this. If you use the OS tools (Windows Explorer, Mac Finder etc.), Automation Engine will not be aware of this move and its database field for the job folder will still refer to the old path.

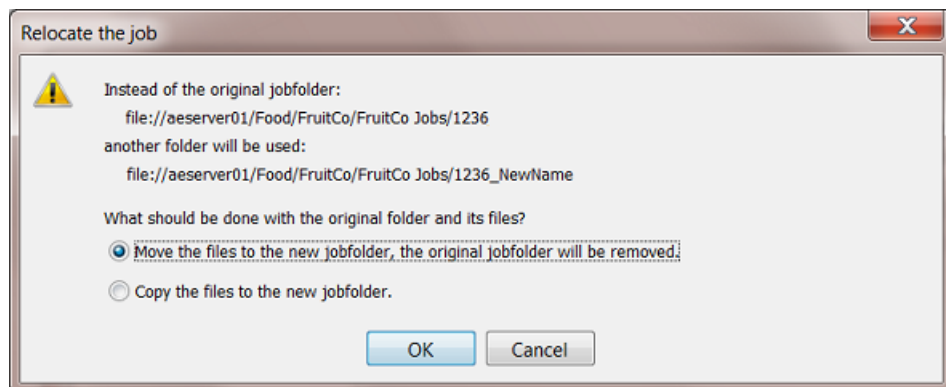
You can relocate a job folder

- manually, one by one. By using the **browse** button in the [Job tab](#) on page 838 of the job setup. You will have the choice to move or copy the folder.
- by using the [Relocate Job](#) task. You can launch this task on multiple jobs (for example on a smart filtered selection).

Can I Rename a Job Folder?

To rename a job folder you can use the same manual relocation tool as mentioned above.

1. In the **Job** tab of the job setup, in the field with the job folder path, change the name of the folder.
2. Click **OK**. The resulting dialog offers a choice on what to do with the original folder and files.



3. Make your choice and click **OK**.

What happens when I Remove a Job Folder from a live Job ?

When Automation Engine can not access the job folder of a job that you are trying to open or use, a warning dialog will mention "**Could not retrieve the list of files**". The Pilot will show the job folder in yellow.

This is the same warning that you will get when there is network problem and Automation Engine can for example not access a complete **Container**.

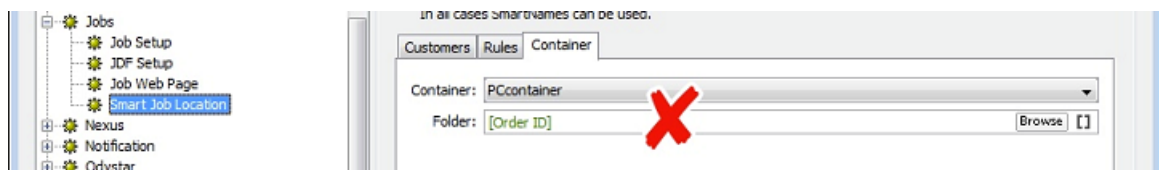
When Automation Engine can again access that folder again, with the place and name it was expecting, the job folder will again be shown in blue and ready for use.

Note: In the Configure tool, there is a feature that allows to remove all "dangling jobs" (jobs that have no more link to their job folder). Learn more [here](#).

3.4.3 Good Practices regarding Job Folder Location

Creating all your job folders under 1 same root folder can result in a slow Automation Engine Pilot and even slow tasks. We advise not to do this.

Here is an example setup that certainly causes too many subfolders in one same folder:



To keep your system healthy, it is good practice to use subfolders in your data structure. Here is some advise on what logic you could use to decide that data structure:

About using Subfolders per Customer Name

We do not advise this, mainly because customers tend to change their names. It looks like a good idea at first, but Esko has seen many customers migrating away from this structure.

About using Subfolders per Customer ID

We do not advise this. It is true that using a customer ID is more stable, but it also has some disadvantages:

- That customer ID number is often meaningless to the Esko users.
- Because legal regulations may require your company to keep track of order history, it is possible that old customers may not be deleted in the business system. That is why, when your customer changes his name, some business systems then change the ID anyway.
- Another reason to not base your folders on customer ID is that customer companies also merge.

These reasons show that we again face the same issues as when we use the customer's name.

About Using Subfolders per Type of Job Order

We do advise to base a data structure on types of jobs.

If you have prepress orders (creating and getting a one-up print ready) and production orders (outputting existing approved data), then structure them in separate folders or even different **Containers**.

Or you can use the product or production type: labels, carton, digital, flexo, offset etc. A typical way to set this up is by using the SmartName of a Job (custom) Category.

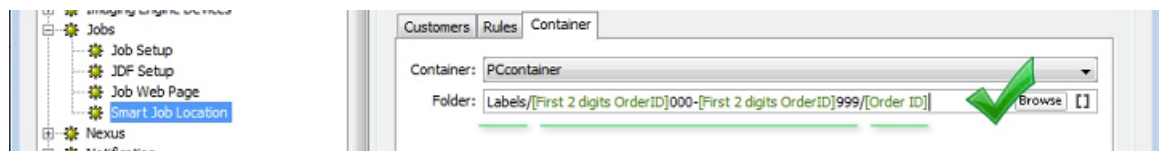
About Using Subfolders per Order Range

We do advise to create a data structure where you split folders based on the job's Order ID.

For example: all jobs in the 1000 range in one folder, all jobs in the 2000 range in the next folder, etc. This is a straight forward structure that will not allow more than 1000 subfolders in one folder. Use [String Extract SmartNames](#) on page 948 to set this up.

This is considered to be the best practice.

For example:



Note: All these principles also apply to folders where you store Products (Product Data Zone). Learn more in the dedicated chapter on [Products](#) on page 876.

3.5 Uploading Files to Jobs

There are many ways to upload files onto a job folder.

- Manually:
 - using the Pilot

- drag and drop files or folders from outside the Pilot onto a job (sub)folder in the Pilot. Or use copy/paste shortcuts.
- right-click a job (sub)folder and choose **Upload Files...** Learn more [here](#).
- right-click a job (sub)folder and choose **Upload Files from FTP Site...** Learn more [here](#).
- using Shuttle, as start of a workflow in this job. Learn more [here](#).
- Automatically:
 - from [Access Points](#) on page 1236. Any of the 5 types of Access Points can upload data to a job (sub)folder (and launch a workflow). The Folder Access Point can also be a sub-folder of the job folder. Learn more in [Integrating with External Systems](#) on page 1228.
 - using the [Download WebCenter Document task](#) on page 1361. When your Automation Engine is connected to a WebCenter, this task logs on to WebCenter and downloads documents from a specified WebCenter project. Learn more in [Integrating with WebCenter](#) on page 1346.
 - from WebCenter,
 - by synchronizing the WebCenter project with an Automation Engine job. When a WebCenter project has a **workflow connection** with an Automation Engine job, documents can be copied to that job when synchronizing. Set this up in the **Configure** page of the WebCenter project.
 - by using WebCenter's **Sync Workflow** task in a WebCenter workflow. Learn more in the [WebCenter documentation](#).

3.6 Milestones: Tracking Job Status

3.6.1 Concept

Milestones represent important moments in the life of an Automation Engine job. Each job has at least one Milestone: the moment it was created. You can add Milestones, predefined ones or custom ones. Each Milestone has a due date and several other attributes (see below). The goal of a Milestone is that it gets confirmed.

Note: The **Job Due Date** that you set in the job setup is not automatically created as a Milestone.

A Milestone has these attributes:

- a **Name**. For example "Approved".
- a **Due Date**. This is the time that this Milestone is expected to be confirmed. It could represent when deliverables are expected (plates, print production, shipping, invoicing). Or it could stand for the time an action is expected to be finished, for example "Approved" in 3 days.
- a status. Users can decide on the status **Confirmed** or not (still "pending").
- a set of **Actions**. You can optionally attach Actions that need to be done to get that Milestone confirmed.

- a **Responsible** user.

Milestones can be created and managed

- manually, by a user in the Pilot.
- by copying them from a template job (manually or via the [Create Job task](#) on page 1280).
- by the [Manage Milestone task](#).
- by an external system, using JDF/JMF. Learn more in [Integrating with External Systems](#) on page 1228.

How can you Use Job Milestones?

Milestones can be used

- as the basis for **status feedback to external systems**. This is the main reason to invest in setting up and using Milestones.









Automation Engine focuses on reporting what happens with a job but not on job organization or job distribution: that is the task of the order management system. What is important is that Automation Engine reports back to such systems and that this ideally happens without manual intervention of prepress users (who have other things to focus on). Learn more in [Informing Other Systems of the Milestones Statuses](#) on page 866.

- in the Pilot,
 - when **filtering** jobs. For example you want to remove all jobs that have a milestone "Finished" confirmed more than 2 weeks ago.
 - as **visual help when tracking the status of jobs**.

3.6.2 The Milestones View

This view is different when you are using it in the **Job Overview** mode or in the **Last Used Job** mode.

Both these views use these icons for different milestone statuses:


-  indicates the time the job was created.
-  indicates the current time on the time line.
-  indicates a pending milestone (not confirmed).
-  indicates a selected milestone.
-  indicates an overdue milestone: the current time is later than the due date of the (pending) milestone.
-  indicates a milestone that contains one or more **Actions** that are not all **completed**. Learn more in [Working with Milestone Actions](#) on page 861.
-  indicates a pending milestone of which all **Actions** are **completed**.
-  indicates a confirmed milestone. A milestone must be completed before it can be confirmed.

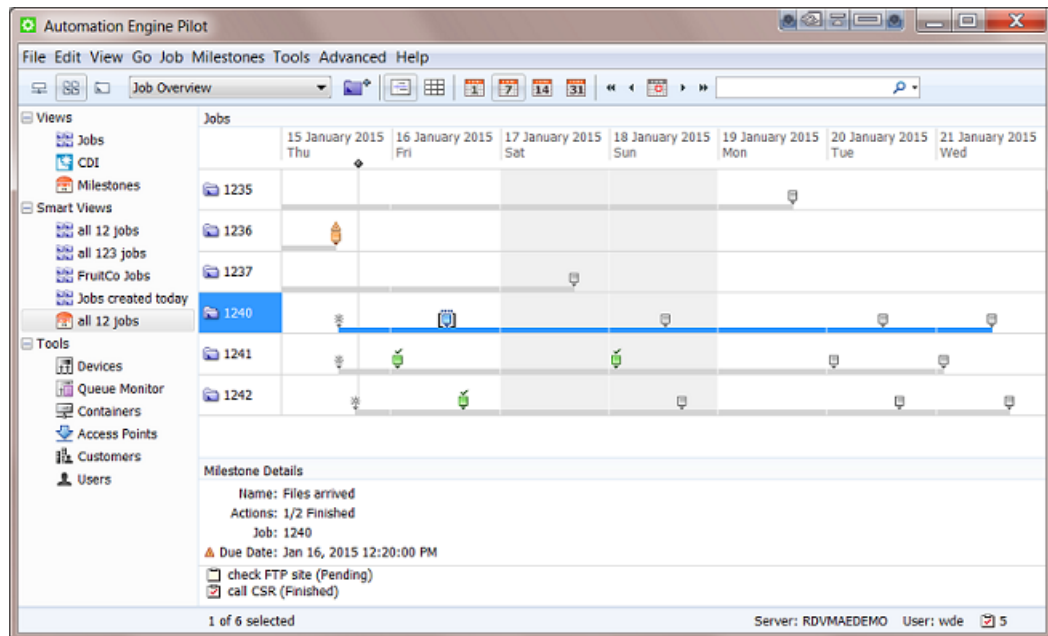
In Job Overview Mode

When you are in **Job Overview** mode, you can see all milestones of a (filtered) list of jobs.










The **Milestones** view offers two types of overview.

The Timeline View

The **Timeline View**  shows you the milestones for a defined period.



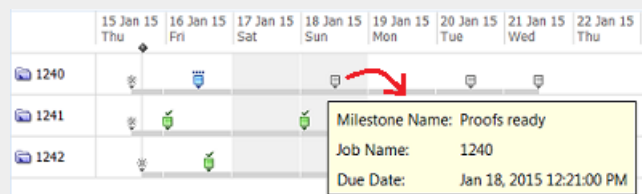
Use these buttons to define the time period you are viewing:

-     to show 1 day, 7 days, 14 days or 31 days.
-      to show the previous week, previous day, the current day, the next day or the next week.

From the menu **Milestones**, use **Go to Date...** to specify a specific date in the center of your view.

Select a milestone to see **Milestone Details** in the bottom pane.

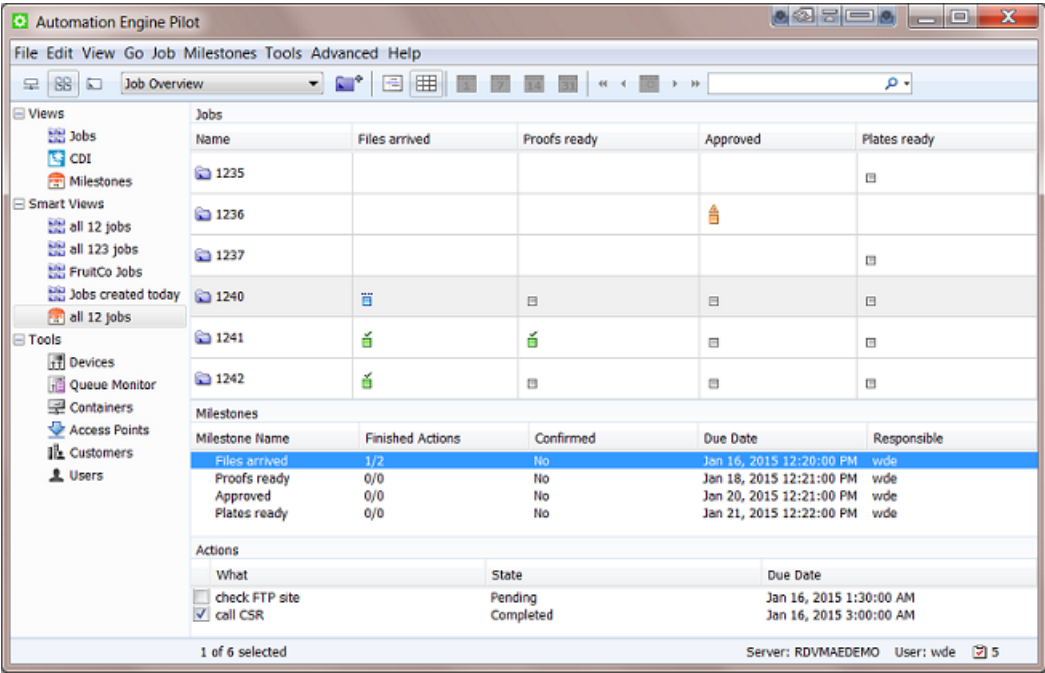
Tip: When hovering over a milestone icon, you will see its name and more details:



Tip: Double click a milestone icon to open the milestone view for that specific job.

The Status View

The **Status View**  shows the milestones and their status in columns.

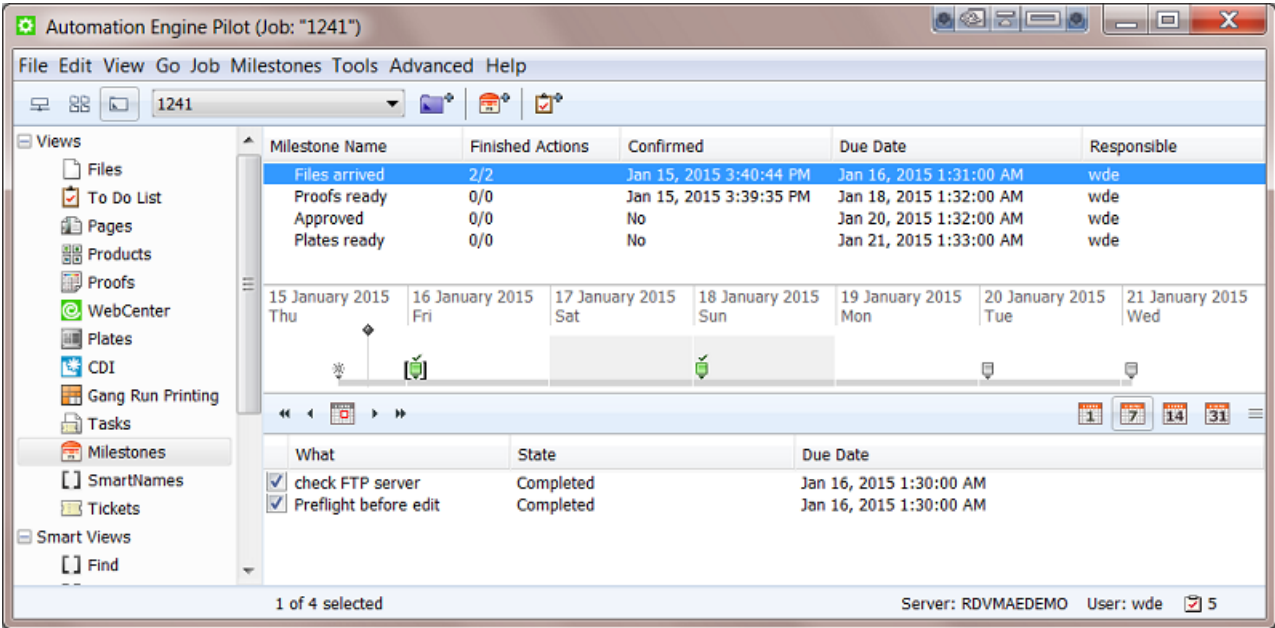


Go to **View > Select Columns...** to choose which milestones will be shown here and to define the order of these columns.

If no icon appears in a milestone column, this means that this milestone is not defined for that job.

In Job Mode

In **Last Used Job** mode, the **Milestones** view shows all the milestones for a specific job.



Select a milestone to see it selected in the time line and to see details below about any actions for this milestone.


Printing out the Job's Milestones

You can use **Milestones > Print Milestones...** to print (or save to PDF) a view of a job's milestones. You can define the printer's page format but the content of the page is fixed. The page shows some major job info and the milestones' timeline and overview.

An example:

Milestones 42100 Scope label

Job Info



Job Name: 42100 Scope label
 Due Date: Apr 30, 2004 4:30:00 PM
 Order ID: 42100
 SubOrder ID: 00002_0000001
 Category: Scope jobs
 Customer: Visionline
 Customer Reference: B 406/2004
 Last Actor: wde@egwde
 Folder: file://egwde/wde\data/Scope_jobs/42100_Scope_Label
 Description: Scope Binoculars label (European release end 2004)

TimeLine

apr 22 23 24 25 26 27 28 29 30 apr


* [check] [check] [check] [check] [warning]

Milestone	Responsible	Actions	Δ	Due Date
* Start				Apr 22, 2004 3:18:04 PM
[check] Files Available	justine (egwde)	0 of 0 - Confirmed		Apr 23, 2004 4:30:00 PM
[check] editing in BSE or AI	wde (egwde)	1 of 1 - Confirmed		Apr 26, 2004 7:00:00 PM
		<input checked="" type="checkbox"/> editing todo		Apr 24, 2004 8:30:00 AM
[check] Approved digitally	admin (egwde)	0 of 0 - Confirmed		Apr 28, 2004 2:30:00 PM
[check] Plates Ready	george (egwde)	0 of 0 - Confirmed		Apr 29, 2004 10:00:00 AM
[warning] End Job	no owner	0 of 0 - Not Confirmed		Apr 30, 2004 4:30:00 PM

Note: The thumbnail of the job shown in this printout is the one you can (manually) add in the **Job** tab sheet of the job setup.

3.6.3 Adding Milestones to a Job

Adding a Milestone Manually

1. Select your job and open the **Milestones** view.
2. Click , or go to **File > New Milestone** .

The 'New Milestone' dialog box contains the following fields and controls:

- Milestone Name:** A dropdown menu showing 'Approved' and a 'Manage...' button.
- Description:** A large empty text area.
- Due Date:** Two dropdown menus showing 'Jan 23, 2015' and '9:00 AM'.
- Reminder:** A checked checkbox, a dropdown menu showing '2 hours', and a clock icon.
- Responsible:** A dropdown menu showing 'Elvis Aaron Presley'.
- Buttons:** 'OK' and 'Cancel' buttons at the bottom right.

3. From the **Milestone Name** list, select the milestone you want to add to this job. If the one you want is not yet in the list, click **Manage...** and create it there. Learn more in [Manage \(Custom\) Milestones](#) on page 860.
4. If you want, add a **Description** for your milestone.
5. Define a **Due Date** and time. This is a mandatory field.
6. If you want, add a **Reminder**. Choose an absolute time or a time relative to the due date. At that time, a reminder will be E-mailed to the **Responsible** user.
7. Select the user that is **Responsible** for this milestone. This is a mandatory field when you set a reminder.
8. Click **OK**.

Copying the Milestones from another Job

When you create a new job (manually or via the [Create Job task](#) on page 1280 or via JDF), you can copy the properties of another job (template) to that new job. This can include the milestones of that other job.

The 'New Job' dialog box contains the following fields and controls:

- Use an existing job as a template:** A checked checkbox.
- Choose Job:** A dropdown menu showing 'FruitCo Label template' and a 'Select...' button.
- Include:** A list of checkboxes:
 - Files
 - Folders
 - Tickets
 - Milestones
 - Parameters
 - Inks
- Buttons:** 'OK' and 'Cancel' buttons at the bottom right.

The copied milestones will be set relative to the creation date and time of the new job.

For example, a typical job for your customer FruitCo has a job due date of 9 days after job creation and during that period it includes 3 custom milestones. The due date of the new job that copies

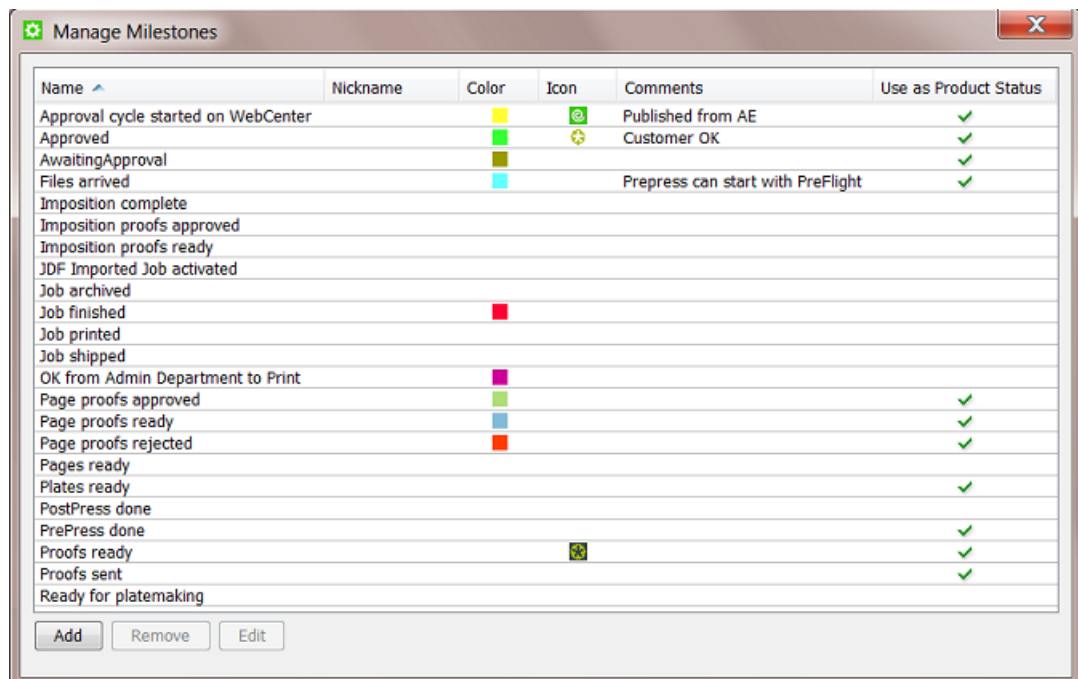
those milestones will then be set to 9 days after its creation date. And its milestones will be created relative to the new creation date.

3.6.4 Manage (Custom) Milestones

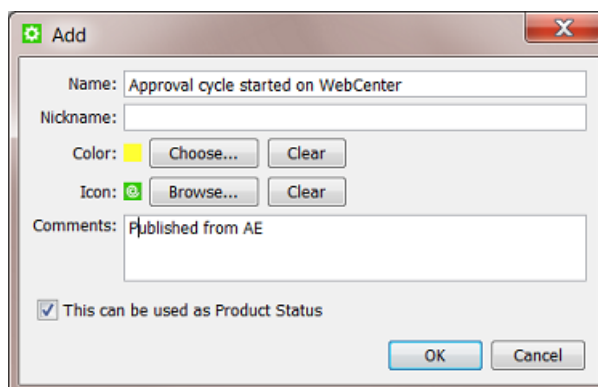
There are many predefined milestones, but you can also create custom milestones.

In the **Milestones** view (in **Last Used Job** mode):

1. Go to **Milestones > Manage Milestones** . This dialog shows you all your milestones and their attributes:



2. Click the **Add** button.



In this dialog:

- enter a **Name** for your custom milestone.

- you can also enter a **Nickname**, this is the name that will be shown in the Pilot interface. This is also useful when your milestones are managed by JDF/JMF. Learn more in [Integrating with External Systems](#) on page 1228.
- if you want, choose a **Color** and / or browse to an **Icon** to help recognize this milestone. Icons can be based on JPEG/PNG/GIF files and will be automatically scaled to a fitting resolution.
- you can also enter **Comments**. For example to explain all users how this milestone should be used or if an external system is using this milestone information.
- if you use the [Products](#) on page 876 tool, define if this milestone can also be used as Products status.
- click **OK**. Your custom milestone now appears in the list and is ready to be added to jobs..

Note: You can safely remove milestones that you already used in jobs. Those jobs will not lose their (local copy) of the milestone.

3.6.5 Working with Milestone Actions


Concept

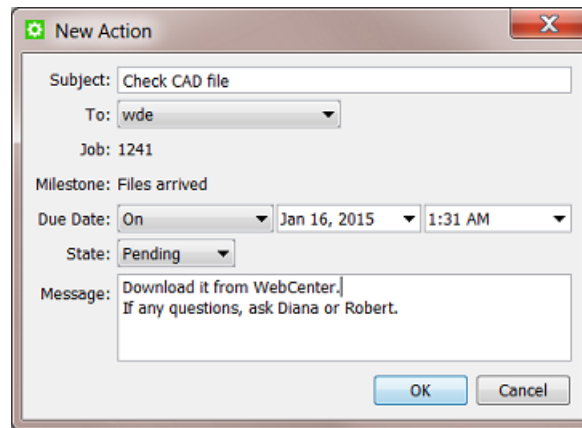
A milestone represents a status that you want to reach. A milestone can also list what **Actions** need to be done to reach that state. A milestone can only be **confirmed** when all its actions were set to **completed**.

Some customers use these tools extensively and ask their users to base their daily work on the milestone actions appearing in their **To Do List**. These users then use custom filters when over-viewing (their) actions. They typically filter their actions based on user, state and urgency.

Creating Milestone Actions

To add an action to a milestone:

1. Select the milestone in the **Milestones** view.
2. Click  or go to **Milestones > New Action...** or right-click on the canvas in the bottom part of the Pilot.
3. In the **New Action** dialog, enter what needs to be done, who needs to do it and by when.

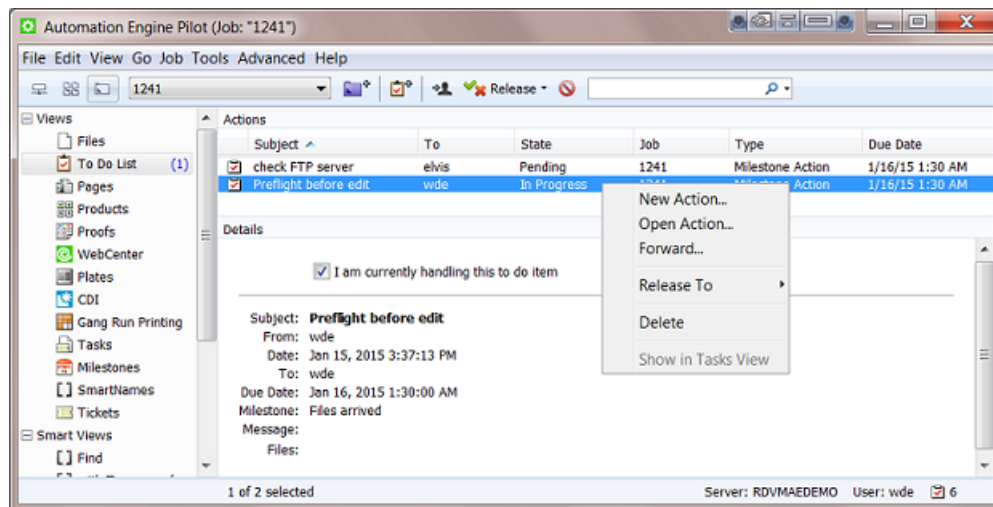



The user that needs to do the action can later change the **State** from **Pending** to **In Progress**, **Completed** or **Aborted**.

Working with Milestone Actions

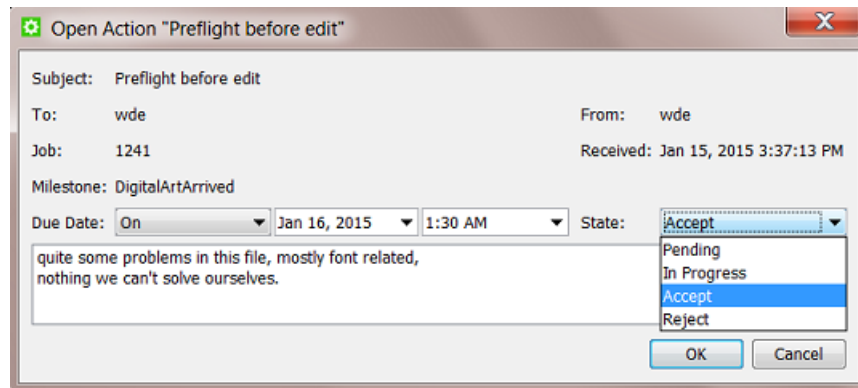
Milestone actions appear

- in the **Milestone view**, when in job mode. See a screen shot [here](#).
- in the **To Do List**, as **Milestone Actions**. The To Do list offers some more functionality on these actions:



- Right-click to open, delete or create a new action.
- Right-click or click  to **Forward** the action to another user.
- Check **'I am currently handling this item'** to change the state to **In progress**.

Open (double-click) the action to use these extra tools:



- change the **State**.

Note: Because the names of these states are shared with **CheckPoint Actions**, choosing **Accept** or **Reject** will for a milestone action result in **Done**.

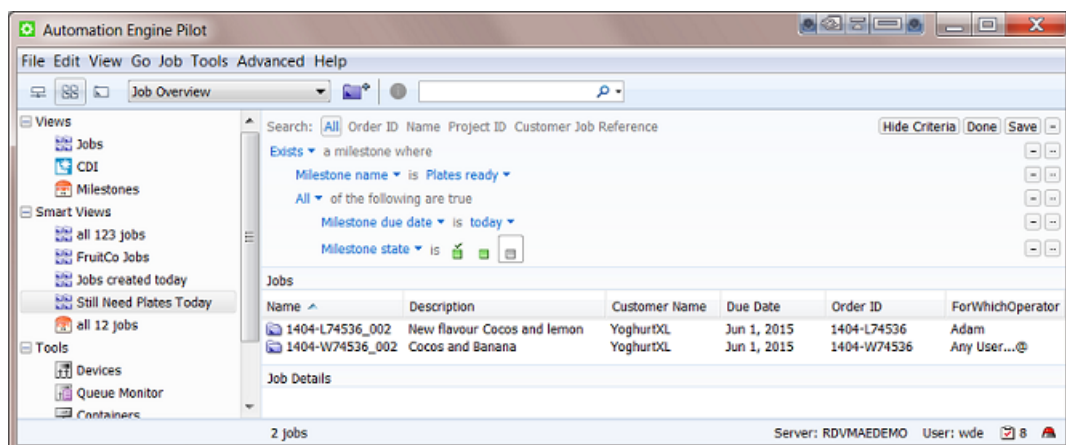
- change the **Due Date**.
- add a **Message**.

For example: a user opens his to do list in the morning and uses a filter showing all the actions on his name that have a due date before noon, in the state **Pending** or **In Progress**. He can also use the column **due date** to sort by urgency.

3.6.6 Using Milestone Statuses to Filter Jobs

The filters that you make in Job Overview mode can also be based on milestones.

See this example of a filter that was saved as a **Smart View** named "Still Need Plates Today". It only shows jobs with an unconfirmed milestone "Plates Ready" that has a due date today:



Learn more on creating such filters in [Job View Filters](#) on page 832.

3.6.7 Using Milestones in Workflows: the Manage Milestone Task

Concept

Using milestones in workflows enables these principles:

- Milestones can help control a workflow. A workflow can check the status of a milestone before continuing (in a specific direction). Use the workflow control **Router** to do this. See an example [here](#). Learn more in [Router](#) on page 1024.

Tip: When you also work with the Products tool, then check if the dedicated **Wait for Product Status** task is a better alternative.

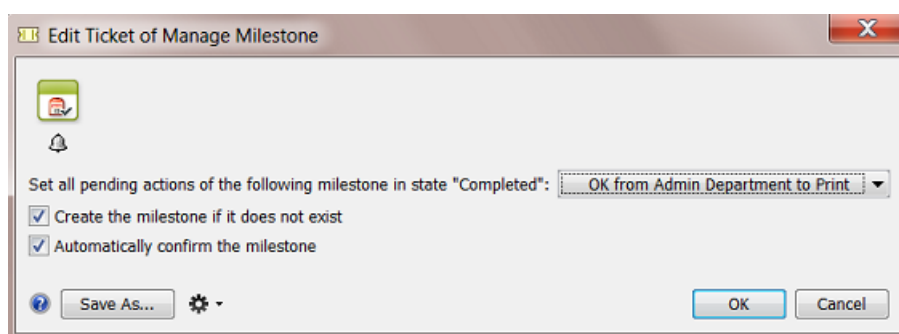
- The **Manage Milestone** task helps to automate working with milestones. This task serves to create and confirm milestones as a workflow step. This enables a lot of automation when working with milestones. And you can also use it to hold workflows. Learn more below.

See an example of a workflow that automatically confirms four milestones:



The Manage Milestone Task

You can use this task to automatically decide one or multiple of these milestone attributes:



- **Set all pending actions of the following milestone in state "Completed"**: select the milestone you want to set as "Completed". Learn more about **Milestone Actions** in [Working with Milestone Actions](#). This setting is mandatory.
- **Create the milestone if it does not exist**: if the above selected milestone you selected was not yet added to the job, check this option to add it (otherwise the task will fail). The milestone will be created with the status "Completed".
- **Automatically confirm the milestone**:

- Check this option to set the above selected milestone as "Confirmed" and have the workflow continue with its next step(s) or with any task that was waiting for this milestone to be confirmed.
- Do not check this option if you are still waiting for a confirmation. The task will then serve as a hold-point: it will pause the workflow until that milestone is confirmed.
- In the **Workflow Editor**, this step then will jump up and down, as a signal that a user intervention is needed:



Right-click the file-icon and choose to **Open Milestone...** or to **Show in Tasks View**.

- In the **Tasks** view, you will see the phase **User intervention needed**.

File Name	Task Type	Progress	Phase	State
banana.pdf	Manage Milestone	0%	User intervention needed	
banana.pdf	Workflow	100%		
banana.pdf	Workflow	0%	Running	

Click on the red text to open the milestone dialog where you can change any setting.

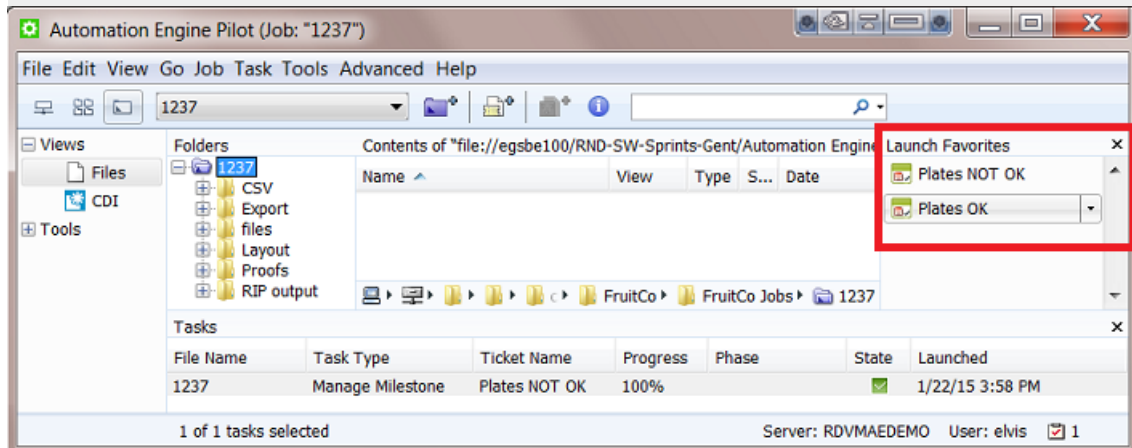
If you check **Confirm Milestone** (and click **OK**), the workflow will continue. If you can not confirm this milestone yourself but for example need the help of someone else, then type in a description and change the responsible user.

- In the **Milestones** view, right-click to open the milestone or directly click in the column **Confirmed** to change it to **Yes**.

Milestone Name	Finished Actions	Confirmed	Due Date	Responsible
PrePress done	0/0	No	Jan 22, 2015 2:43:00 PM	toto

Tip: Some customers make the manual confirmation of a milestone easy by offering the users a favorite ticket that only (creates and) confirms a milestone. This is useful when that user is aware of a status that Automation Engine has no information on.

For example: A plate room operator checks plates that come out of the developer. He reads their job number on the plate and types it in as a job filter in his simplified Pilot. He then launches the appropriate favorite ticket: **Plates OK** or **Plates NOT OK**.



This information can now automatically change a workflow and/or be sent out to the system that does the press planning. Learn more about such feedback in the next section.

3.6.8 Informing Other Systems of the Milestones Statuses

The milestone status of an Automation Engine job is often more important to external people than to users of the Pilot. Many parties want to be kept up to date about the latest status in prepress, both colleagues and customers.

External systems are not always interested in all prepress details. You can choose to only inform them on some specific milestones.

These are tools that help inform other systems and their users:

- Add a **notification** to your **Manage Milestone** task. This can be an E-mail notification or the same message can also be written in an XML file for an external system to read. Learn more about the type and content of such messages in [Notifications](#) on page 175.
- Or you can use one of many tools dedicated to help integrate Automation Engine with external systems. Typical methods are
 - write out XML files
 - write directly in the database of the external system
 - when the external system is capable of subscribing to JMF signals, you can set up the automatic sending of JMF signals when a milestone is confirmed.

For more information on these feedback tools, see the dedicated chapter [Integrating with External Systems](#) on page 1228.

3.7 About Removing Jobs

Concept

An Automation Engine job represents a Work In Progress. It is expected that you remove the job when the work is done.

Removing a job means that it is removed from the Automation Engine jobs database. You have the option to keep the job folder data.

When you remove a job,

- all finished tasks in that job will be deleted. If you wish to keep those tasks, use the [Task Cleanup Rules](#) on page 682 to move them (earlier) to the [Task History](#) on page 621 tool.
- if that job still has unfinished tasks, you will be asked before removing the job to confirm that they can be cancelled and deleted.
- all **Actions** for that job in the **To Do List** will be deleted.
- any setup of Hot Folders in that job folder will be removed.
- when you choose to not delete the job folder, all the job's setup parameters will be stored in a subfolder 'config' (this is the information that was inside the database).
- the job will disappear from the Pilot's Job Overview.
- in the Containers view, the blue colored folder icon will turn back into a standard yellow folder icon. You will be able to restore it later.

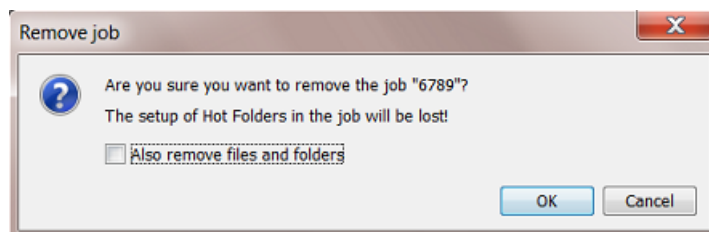
Note: The **Archive Job** task allows to remove the job (from the database) and move the job folder to a different place in the network. Learn more [here](#).

You also have tools to reactivate the job, to recreate it back into the jobs database. See below.

Removing a Single Job

To remove a single job, select the job and press **Delete**. You can also right-click (or go to the menu **Job**) and choose **Remove Job...**

You will be asked to confirm. You can choose to also remove the job folder and its contents.



Caution: There is no way to **Undo** the removal of the files and folders.

Removing Multiple Jobs

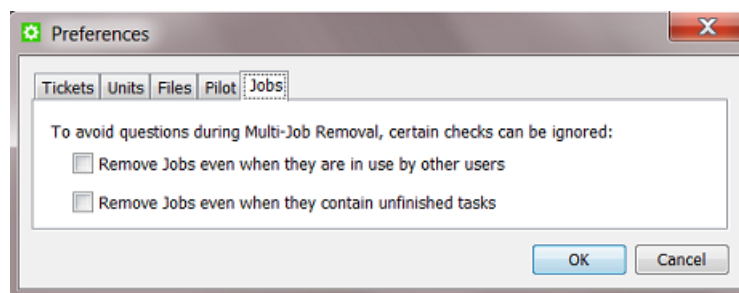
In the Job Overview mode, you can select multiple jobs and remove them in the same way as described above. You will also get the option to keep or remove (all) their job folder data.

Note: It is good practice to use a smart filter when you select multiple jobs to be deleted. For example: all jobs that have their milestone 'finished' confirmed more than a month ago.

Note: There is no "Remove Job" task, so you can not remove jobs as a workflow step.

In **Edit > Preferences**, you can decide to avoid getting (many) questions when you remove multiple jobs:

- questions about jobs being in use by other users. A job is 'in use' when a user has this job open in his Pilot.
- questions about jobs containing unfinished tasks.



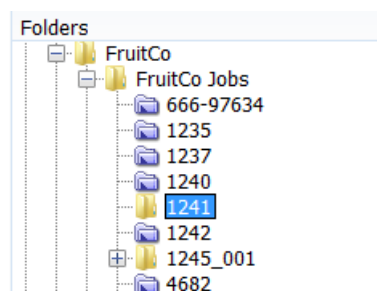
Learn more in [Pilot Preferences on Multi Job Removal](#).

Recreating a Job that was Removed

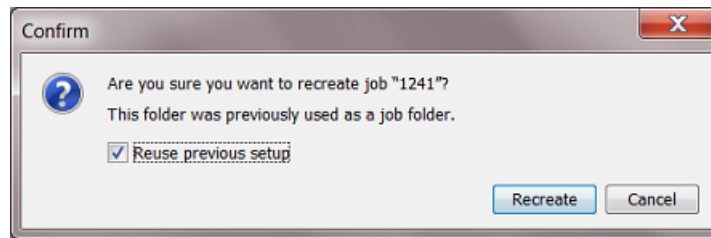
When a job was removed but the job folder data is still available, you can recreate that job back into the jobs database.

There are two ways to recreate a job:

- Recreating a job manually. Follow these steps:
 1. In Containers view, select the yellow folder that used to be the job folder of a removed job.



2. Right-click (or go to the menu **Job**) and choose **New Job....** When a previous job setup is detected in that job folder (in the subfolder 'config'), you will be able to recreate that job. Choose to **Reuse the previous setup**.



3. Click **Recreate**. The renewed job will be opened and will contain the previous job setup parameters, subfolders and local (blue) job tickets.

- Recreating one or multiple jobs with the **Restore Job** task.

Learn more in [The Restore Job Task](#) on page 873.

3.8 Tasks you can Launch on Jobs

These are the tasks that you can launch **only** on Automation Engine jobs:

- The [Relocate Job task](#) moves the job folder of a job to another place.
- The [Archive Job task](#) removes a job and exports all its data and metadata into a ZIP file.
- The [Restore Job task](#) re-creates a job from a ZIP file that was created by the **Archive Job** task.

Like all tasks, these tasks can also be triggered by an external system. Learn more in [Integrating with External Systems](#) on page 1228.

The **Create Job** task is documented in the chapter [Integrating with External Systems](#) on page 1228. Find a direct link [here](#).

Note: The **Remove Job** feature removes one or multiple jobs. This feature is not available as a task, it is something that a user chooses in the Pilot. Learn more in [About Removing Jobs](#) on page 867.

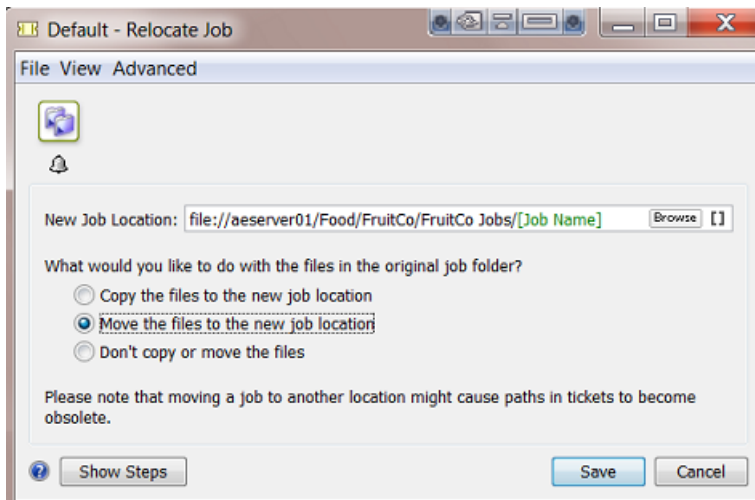
3.8.1 The Relocate Job Task

Concept

This task allows to change the location of an Automation Engine job. The job itself does not change, only the path of its job folder.

Restriction: You can not launch this task as part of a workflow.

Task Options



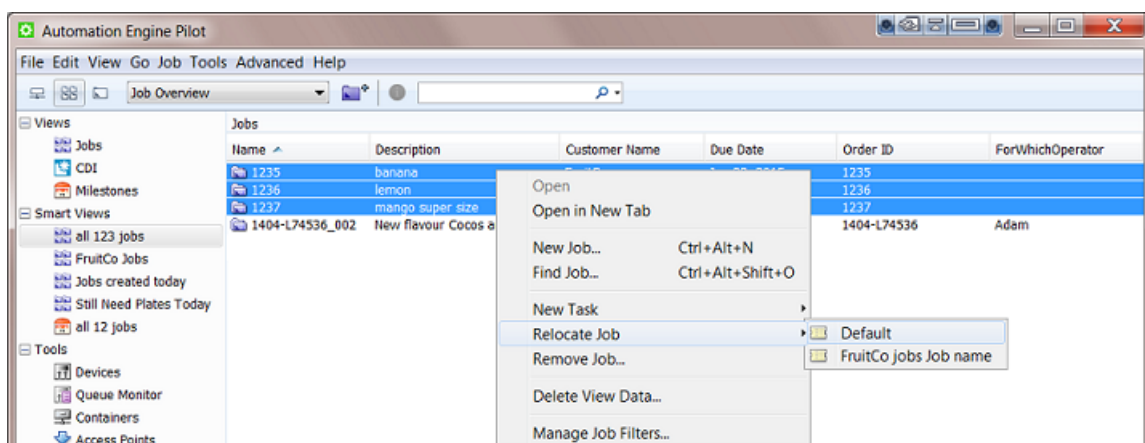
- Enter the **New Job Location**. Keep in mind to end the location with the name of the job folder.
- You can choose to:
 - Copy the files to the new location.
 - Move the files from the original job folder to the new one.
 - Not copy or move the files. The relocated job will then have an empty job folder.



Warning: If you move a job folder without using a tool from Automation Engine, the job setup field for the job folder will not be updated and the Pilot will show warnings. Automation Engine will keep trying to retrieve your missing files and this can slow down your server.

Relocating Multiple Jobs

In Job Overview mode, (filter and) select the jobs you want to relocate. Right-click and choose a **Relocate Job** ticket:



3.8.2 The Archive Job Task

Concept

The **Archive Job** task removes a job from the jobs database and exports the data in a form that enables to restore the job later.

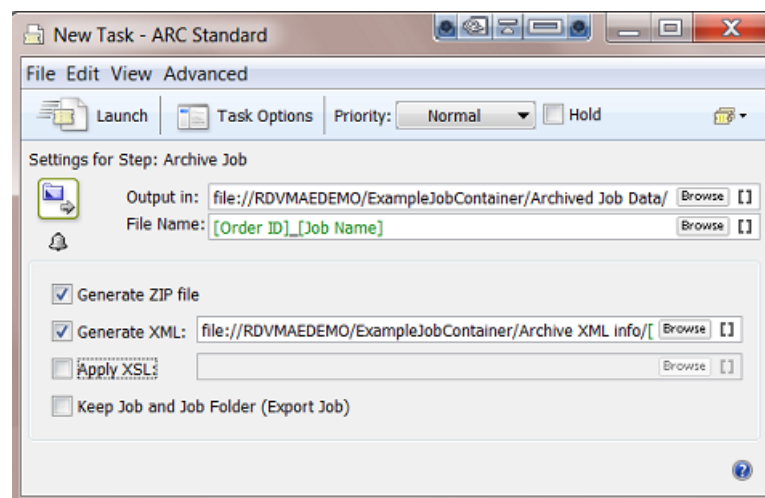
Typical for this task is that it exports the job metadata as well, these are the job setup parameters. This metadata makes it possible to restore the job (back) later.

Note: The job's metadata is exported in a subfolder named 'config'.

Note: When you launch the **Zip task** on a job, the task only ZIP's the data inside the job folder, not the job's metadata.

Note: When archiving (removing) a job, the tasks that were launched in that job will be deleted. If you want to keep these tasks, use the [Task History](#) on page 621 tool. This tool allows setting up rules that automatically move those tasks to a separate database (before archiving the job).

Task Options



- **Output in:** define where you want to write this archive data. For example some customers use a 3rd party backup/archive solution that scans specific network folders for new incoming data. This system then archives that data online or writes it on a backup medium.
- **File Name:** define the name of the file in which all data will be stored. When you do not choose to generate a ZIP file, this will be a folder with subfolders.
- **Generate ZIP file:** deselect this option if you want to move the job folder to the above location without ZIP'ing all data.
- **Generate XML:** select this option to generate an XML file that describes this job. Such an XML file can be useful for a 3rd party archive solution that for example uses it to build up its search database.

- **Apply XSL:** select this option if you want to format the XML file using an XSL style sheet from a 3rd party archive solution.
- **Keep Job and Job Folder (Export Job):** deselect this option if you want to delete the job folder once it is successfully archived.

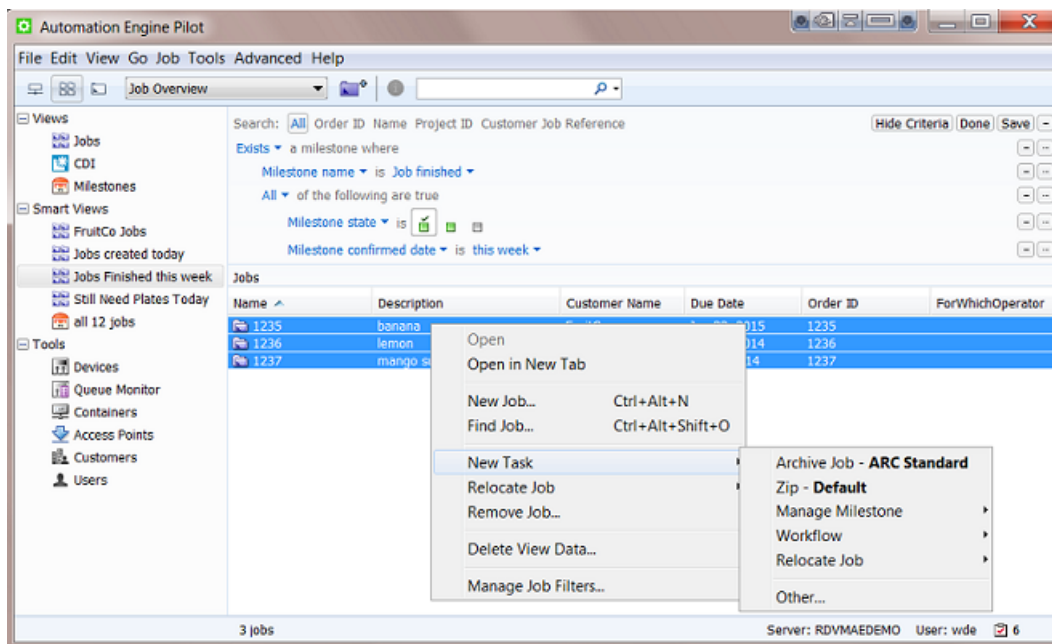
Select this option if you still want to keep the job in the job database and also keep the job folder in its original location. Here are some examples when to choose this:

- To use this task as a job backup tool.
- To exchange that job with another Automation Engine. For example to send to a job to another plant of your company (that also has Automation Engine) or to share the job with Esko Customer Service.

Archiving Multiple Jobs

In **Job Overview** mode, right-click the selected jobs and choose your **Archive Job** ticket.

Here is an example of a smart way to archive multiple jobs. The filter lists all jobs that have their milestone "Job Finished" confirmed this week:



Using the Archive Job Task in a Workflow

You can end a workflow by archiving the job that the workflow was launched in.

Here is an example:



Notice these steps:

- After the plate making stage, the workflow waits for 2 weeks.
- It then sets the milestone "Ready for Archive" but without the option to confirm it. This triggers a request for a user intervention (learn more about [this option the Manage Milestone task](#)).
- When a user confirms this milestone, the workflow continues and archives the job.

3.8.3 The Restore Job Task

Concept

This task allows to restore jobs back to the Automation Engine job database that were previously archived using the [Archive Job](#) task.

Restriction: You can not launch this task as part of a workflow.

Possible Input Types

The input file for this task can be

- a ZIP file created by the **Archive Job** task.
- a job folder created by the **Archive Job** task.
- an XML file that refers to the location of a ZIP or folder created by the **Archive Job** task.

The root node of the XML needs to have an XML attribute named "Location". The value of this attribute is the URL to the ZIP or archived job folder. Here is an example XML (it looks like the XML generated by the Archive Task):

```
<?xml version="1.0" encoding="UTF-8"?>
<Archive Location="file://aeserver01/Food/Archived Job Data/1237.zip"/>
```

Task Options

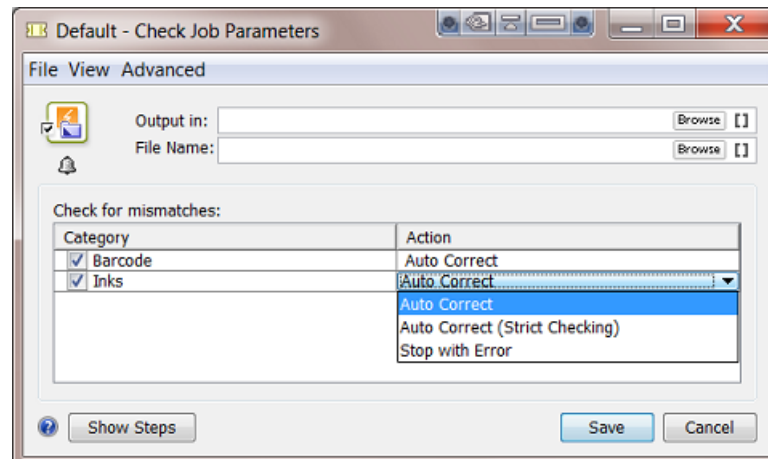
- **Output in:** define where you want to place the restored job folder. Use [Job URL] when you want to restore the job to its original location and with its original job folder name.
- **Job Name:** define the name of job once it is restored. Use [Job Name] when you want the job's original name.

- **Order ID** and **SubOrder ID**: similar to the job name, define a new ID or use SmartNames to use the ID's of the original job.
- **Due Date**: set the job's new due date.
- **Delete archived data**: select this option if you want to delete the archive data after restoring the job.

3.8.4 The Check Job Parameters Task

This task serves to check if your file matches specific parameters as defined in your job setup. You can also use it to automatically correct the file according those parameters.

Note: For the task to function properly, the file should be part of an Automation Engine Job.



For **Barcode** settings, you can select:

- **Auto Correct** when you want to correct the file with what is specified in the job's [Bar Codes tab](#) on page 844 .

For example, your Normalized PDF includes an Esko barcode of type EAN-13 and code 123456789012, and your job parameters specify an EAN-13 with code 5449000050205, then this task will automatically correct the barcode number in the PDF to 5449000050205.

- **Stop with Error** when you want the task to stop with an error if the barcode in the job's [Bar Codes tab](#) on page 844 does not match the barcode in the file.

Note: This task only checks barcodes created in Esko PackEdge or by using Esko's barcode plugin for Adobe Illustrator (DeskPack). If the task does not find such a barcode, the task will end in a warning.

Note: You cannot use this task to check and auto correct barcodes in normalized PDFs generated by **ArtPro**.

For **Inks**, you can select:

- **Auto Correct** when you want the task to correct the file's ink settings according what is specified in the job's [Inks tab](#) on page 840.

- **Auto Correct (Strict Checking):** when you want to correct the file's ink settings and order according to the job's ink parameters.

In this strict setting, the task will finish with an error if it finds unexpected inks (inks that are in the file, but not defined in the job parameters) or if it finds missing inks (inks that are not in the file but are defined in the job parameters).

- **Stop with Error** when you want the task to stop with an error if the job's ink parameters are different than those in the file.

3.9 Automated Creation of Jobs

Most of the job setup parameters originate in another system, often the order entry part of the business system. It could also be a web-portal where job specifications were first entered.

When someone already entered this data manually, we advise to not re-enter that data. We therefore strongly advise to get your jobs created automatically.

The chapter [Integrating with External Systems](#) on page 1228 explains how this can be done. It describes the many available techniques and also offers exercises.

Here is a summary of how jobs can be created automatically:

- by the [Create Job task](#) on page 1280 that, as part of a workflow, picks up the job setup parameters via SmartNames or from an external (XML) file(s).
- by an [Access Point](#) that interacts with the external system and launches a workflow that uses the found information in the **Create Job** task.
- via JDF/JMF commands from an external system, sent directly to Automation Engine's JDF channel.

4. Products

4.1 Products Basics

4.1.1 The Products Concept

What is an Automation Engine Product?

An Automation Engine Product is a one-up graphics production item stored in the Automation Engine Products database. A Product represents a Stock Keeping Unit (SKU). A Product is what you Step&Repeat on a plate or press, possibly combined with other Products.

As explained in [The Job Concept](#) on page 24, a Job is a to-do for Automation Engine. A Job is not a thing, it is an activity with a specific goal. A Job is temporary, you delete it when the Job is done. A Product does represent a thing and is not temporary, you expect it to come back in other production (output) orders. For example, you could have 200 Jobs, 20.000 Products and 20.000.000 files.

In [Types of Jobs](#) on page 829, we also describe that a Product is the result of a "Prepress Job-Order", in which the goal is to get a design ready for production and customer approved. From that moment on, the Product can be used in many "Production Job-Orders", where the only task is to produce (output) it.

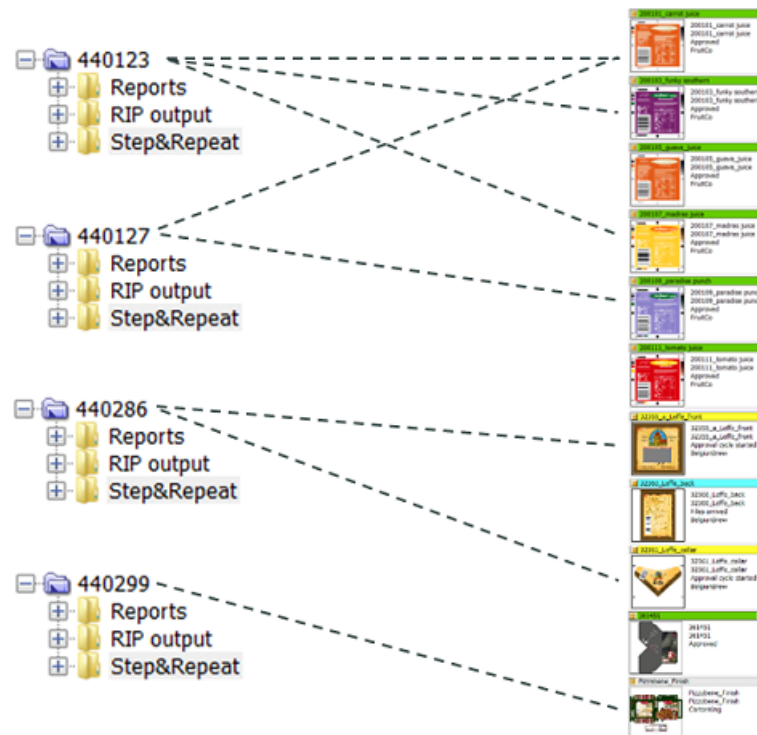
It is easier to integrate Automation Engine with a business system when both use the same concepts. It will be easier to translate the incoming orders into a prepress (data) context. It will also be easier to give feedback to that system about statuses in prepress.

The Main Principles when Working with Products

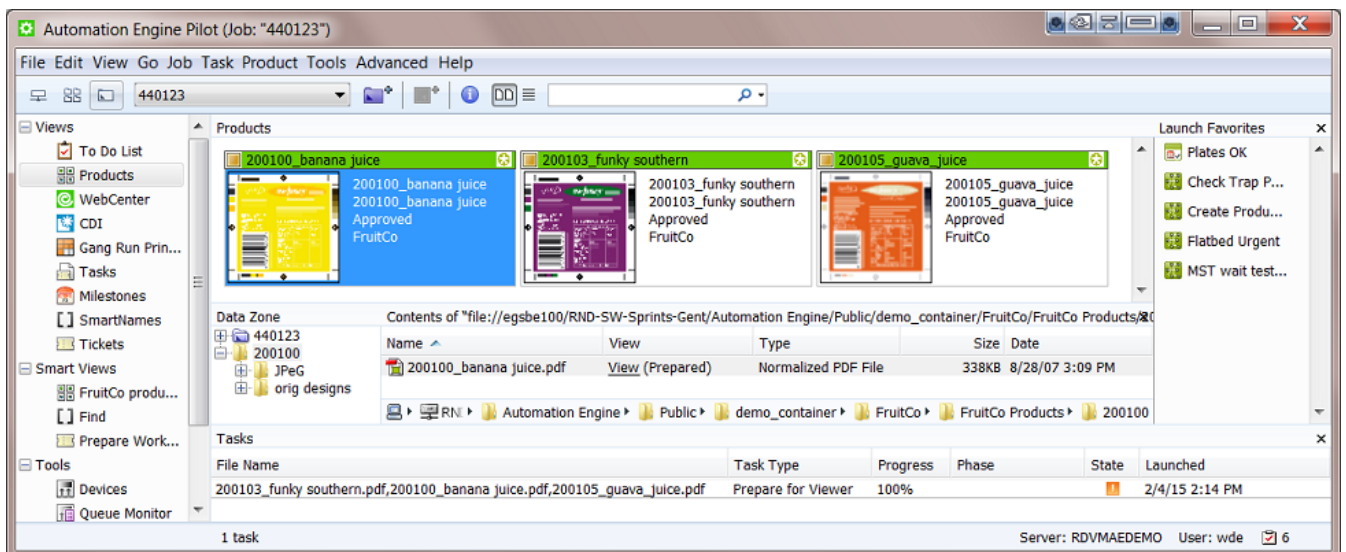
Working with Products typically means following this process:

- You first check your data structure. Working with Products requires that your Job data are split from your Product data. Learn more about in [Products Data Structure](#) on page 881.
- Then you build up your Products database. Learn more about how to automate this in [Creating Products](#) on page 897.
- Then you start linking Product(s) into your Jobs. This way, the Product data files are not inside the Job Folder. Product data are independent: today they can be linked to Job 123 and next time to Job 456. They can even be linked to several Jobs at the same time.

In Automation Engine, this temporary relation to a Job is a temporary link between items in the Product database and items in the Jobs database:



Here is an example how this looks in the Pilot: the Job 440123 links to 3 Products:



Note: You can work with Products without working with Jobs. However, in all classic print production environments Esko advises to use the Jobs concept.

Product File Formats

These file formats are supported as Automation Engine Products:

- PDF

Note: Non-Normalized PDFs cannot be checked for barcode parameters. Learn more in [The Check Product Parameters task](#) on page 912.

- ArtPro
- Adobe Illustrator (Esko DeskPack workflow).

Note: You will only see a product part thumbnail when the AI file was saved as PDF compatible.

It is possible to add other file types to the Products database, but it will become clear that Automation Engine does not offer (m)any valuable features for such files (no thumbnail, few tasks possible, provided parameters are irrelevant etc.).

Who typically uses the Products tool?

The Products tool benefits all customers that have product items coming back into production, all environments that do some form of re-runs. These sites usually are printers/converters.

If your business is to produce an item only once, then you may conclude it is not worth to add these files to the Products database. If you produce an item only once, you then probably end by removing the Job and storing its data somewhere as a backup, just in case you need it again for some unexpected reason.

Benefits of working with Products

- Data efficiency: no more needless copying or searching of data.
- A transparent data structure that supports the process logic of re-runs.
- A more user-friendly interface: thumbnails and colors indicating status.
- Automation Engine now speaks the same language as your business system, this enables a higher form of integration, leading to even more automation.

What is the Products tool not?

The goal of the Products tool is *not* to make Automation Engine a Digital Asset Management system (DAM).

These are some differences with a DAM:

- The Products database only stores one version. There is no automatic versioning tool.
- Searching for references is limited. You can see which Products are used in a Job and you can see in which Jobs a Product is used.
- File support: the Products tool is designed for one-up graphics production files, not for images or CAD or 3D or admin data.

Note: You can use WebCenter as a DAM for your graphics production data. WebCenter offers automatic versioning and supports all file formats. There are also standard available tasks to up- and download data from WebCenter to Automation Engine in case you need it (back) for production. Learn more in [Synchronizing Products with WebCenter](#), a section of the dedicated chapter [Integrating with WebCenter](#) on page 1346.

Do I have to Install an Extra Database?

No. An installation of Automation Engine already provides the required database.

Note: Product attributes are stored in the same database as the one that is configured for Jobs (see **Tools > Configure > Automation Engine Database > Jobs**).

Automated management of Products (and Jobs) via Integration

The goal of using the Products tool is to work more efficient. It is therefore important that you automate the management of your Products. Creating them manually or changing their status manually would again reduce efficiency.

You can automate managing (Jobs and) Products by integrating Automation Engine with your order management system.

These should be your goals for such automation:



- All new Jobs and Products appear automatically.
- Jobs automatically link in their Products.
- Products statuses change automatically.
- Ideally, the business system makes a difference between Prepress and Production Jobs.
- Production Jobs should be fully automatic, the Product(s) they link to are already approved so the only limitation should lie in how far you can fully automate the Step&Repeat.

See [Tasks on Products](#) to learn about tasks you can use in a Products workflow. As usual, these tasks can also be triggered from an external system. Learn more in [Integrating with External Systems](#) on page 1228.

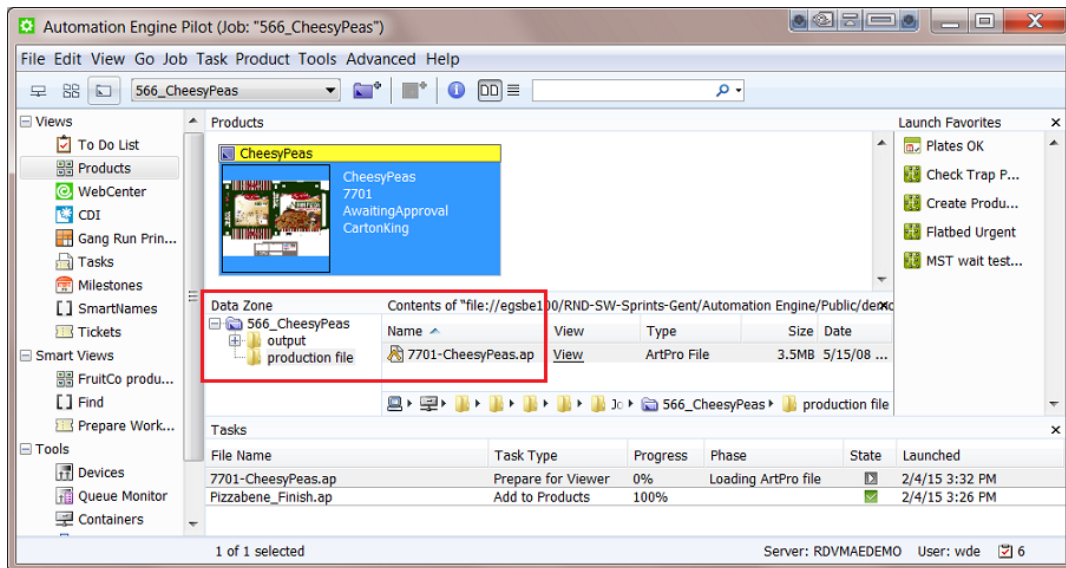
4.1.2 Global Products and Job Products

A **Global Product** is the one we described [above](#): its data is not inside a job folder, it is independent. Global Products can be linked into several jobs (even at the same time).

It is however possible to store the data of a product *inside* a job folder. We then call it a **Job Product**, it can only be used in that job. Some Prepress Service Providers (trade shops) or commercial printers, who don not normally produce re-runs, use this concept.

Like Tickets, the global version is visualized with a yellow icon  and the job (local) version with a blue icon . See some examples in [Products Views and Menu](#) on page 886.

See this example of a Job Product, notice how the product file is located inside the job folder:

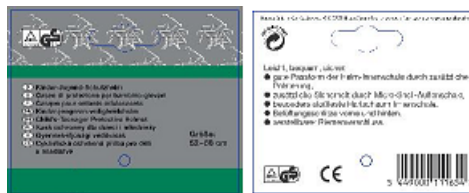


4.1.3 Products can have Multiple Parts

A Product can be a combination of multiple Product Parts. These parts then come together on press or later in finishing and so end up as 1 SKU in your warehouse.

Some examples:

- A bicycle helmet is sold in a plastic wrap that also contains a paper sheet with product information. The sheet was printed in 2 print runs: first the front in 2 colors (front.pdf) and then the back in 1 color (back.pdf). This front-back printing made these 2 Product Parts become 1 Product (after cutting). 1 Product is kept in the warehouse (1 SKU), 1 Product is ordered by your customer.



Note: There could be other Products that use the same front Part but a different back Part as their (language) variant.

- A booklet is glued to a label. These 2 Product Parts come together in finishing and so become 1 SKU, 1 Product to be ordered.



These are Not Multiple Parts of one Product:

At first glance, these 3 labels look like Parts of a same Product. This may look that way to the brewery, for whom the product is the beer bottle.



But, for the label printer, these are **3 different Products**.

- They do not come together in printing or in finishing.
- In the warehouse, they are 3 separate SKUs.
- They can be ordered separately: your customer can for example order 6.000 front labels, 3.000 French back labels and 3.000 Spanish back labels.

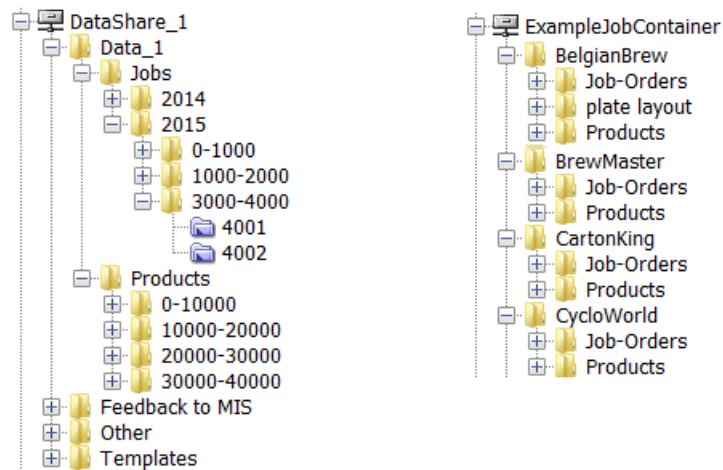
4.2 Products Data Structure

Important: This page illustrates data structures to work with *Global Products* (where the product data is not inside a job folder).

Splitting Product data from Job data

When jobs link to product (data) it is important that they do not share any folders. This data separation can be achieved in many ways.

Here are 2 examples:



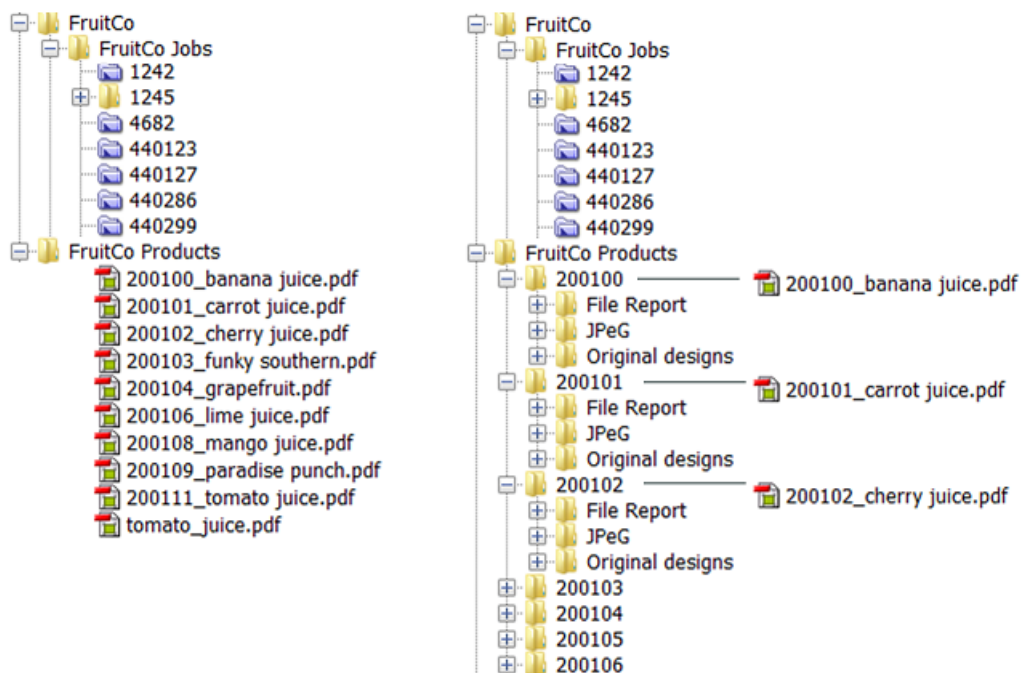
- **On the left**, an example where the data split between jobs and products happens quite high in the container.
- **On the right**, an example where this data split is done per customer.

Note: Using customer names for folder has its disadvantages too. Customer tend to changes names for example. Learn more about your options in [Good Practices regarding Job Folder Location](#) on page 852.

Organization of Product data

Once your product data is split from the job data, you can freely decide how to structure that product data.

See these 2 examples:



- **On the left** all the product data is in 1 same folder `FruitCo Products`. This is usually not advised because it quickly leads to having too many files in a same folder (slow browsing, general system slowness etc.). And if you want to select a file related to a product file, you will again have to browse, even when they are in subfolders like `Originals` or `Proofs`.

If you create further subfolders that avoid having too many files in one folder, then this is a valid way to organize your data.

- **On the right** each product has its own folder. This is advised. When each product has its own folder, you can
 - use subfolders to store related files.
 - choose whether you store the actual product file in its root folder (as in above example) or in its own dedicated subfolder, for example named `Production file`.



Attention: The more structure and consistency you have, the easier it will be for other systems to also benefit from this data structure. They can then rely on finding files in specific places and will also be able to provide you files in specific (sub)folders.

Note: When creating Product (Parts) via JDF, see [Products JDF setup](#) to learn about extra possibilities.

The Product Data Zone (PDZ)

A product has one or more **Product Parts** and each part refers to a specific product part file. So when we talk about the product data we more precisely mean the product part data. Learn more in [Product \(Part\) Properties](#) on page 893.

The place where the product part file and its related files are located is also referred to as the product part **Data Zone**. In the Pilot, when working with products, you sometimes want to launch tasks on files related to that product. For example: selecting its report file and launching the E-mail task. That is why it is advised to see more data in the Pilot than just the product part file.

The Depth of a Data Zone

The Data Zone is not just the folder where the product part file is located, it can be a larger zone. How large this zone is is decided by its **Depth**.

The reason to set this depth is to control how much data the Pilot users will see and have access to when selecting a product in the Pilot.

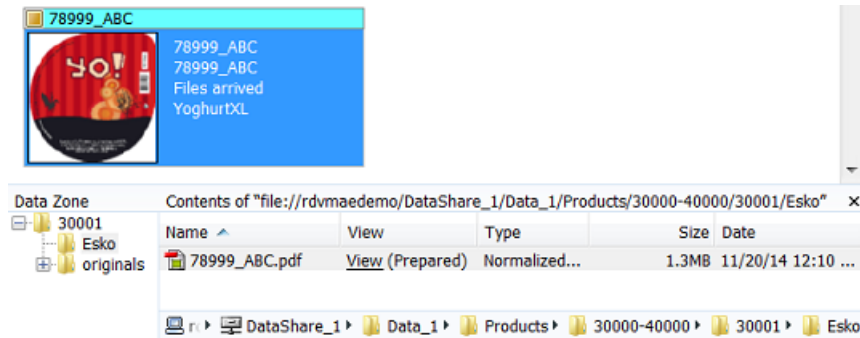
The depth of a product part data zone is decided when the product is created. It then takes the value as defined in **Tools > Configure > Products > Product Part Data Zone** . Learn more in [Configuring the Data Zone Depth](#).

Here are some examples:

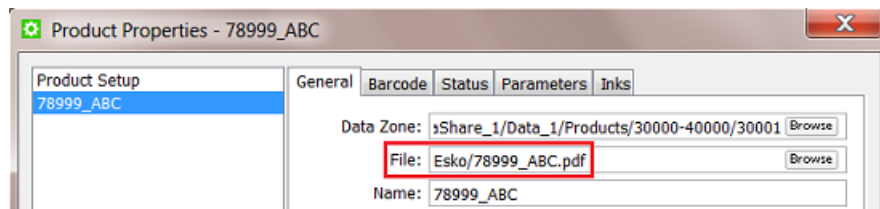
4.2.1 Examples of Data Zone Depth

Example 1: a Product Part with a Data Zone Depth of 1

The product shown below was created with a data zone depth of 1. See how, when you select the product thumbnail, the **Data Zone** pane not only shows the folder `Esko` where the product part file is located in, but also `30001`, which is 1 folder higher. If the depth was 0, you would only see the folder `Esko` and so have no direct access to other product's related files.



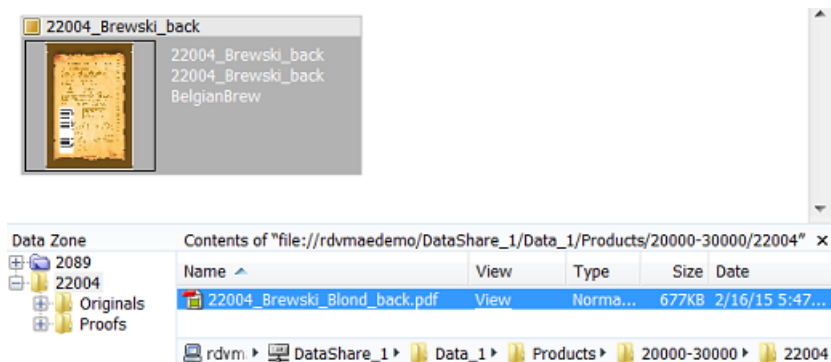
Another place where this depth is visible is in the properties of the product part. Notice how the definition of **File** in this case starts with (1) folder name:



Note: In this properties dialog, it is possible to manually change this depth. This might be useful when a wrong depth was used at product creation time. In above case you could for example remove Esko/ from the **File** definition and it to the **Data Zone** path.

Example 2: a Product Part with a Data Zone Depth of 0

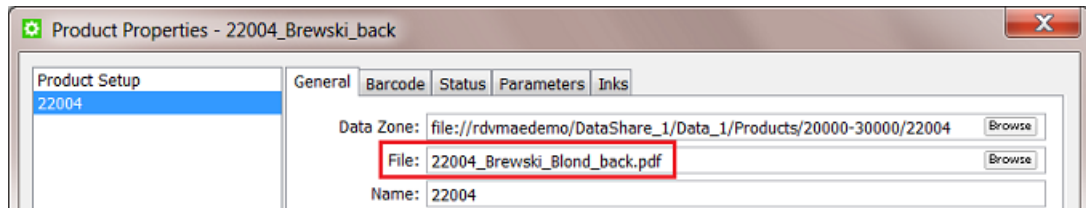
As in the above example, the product has its own folder and subfolders. But here, the actual part file is not stored in a subfolder, but it is located in the root of that product folder. See how the PDF is in the folder 22004 :



This indicates a Data Zone depth of 0. There is no need to, when you select the product, to see a folder higher than 22004.

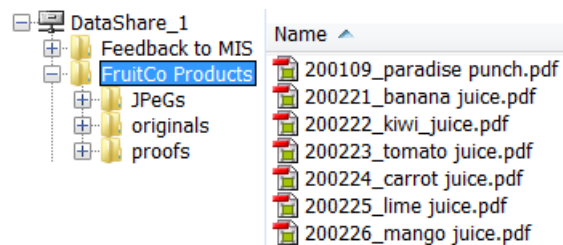
Note: In the **Data Zone** pane, folders below the selected part are always visible.

In the product part setup, notice how in this case the definition of **File** only contains the name:

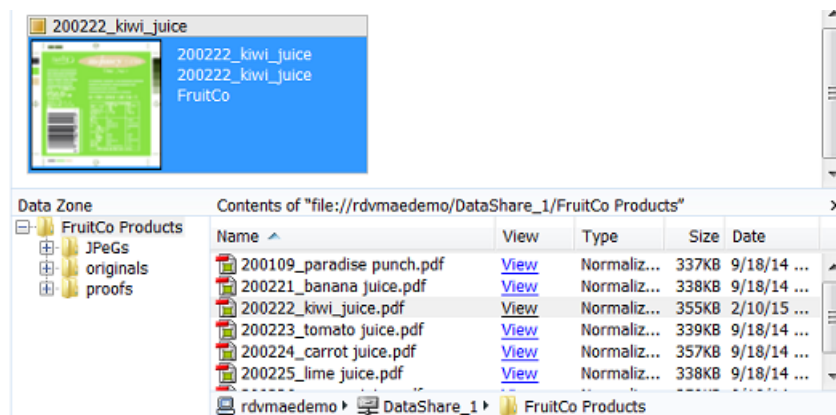


Example 3: another Product Part with a Data Zone Depth of 0

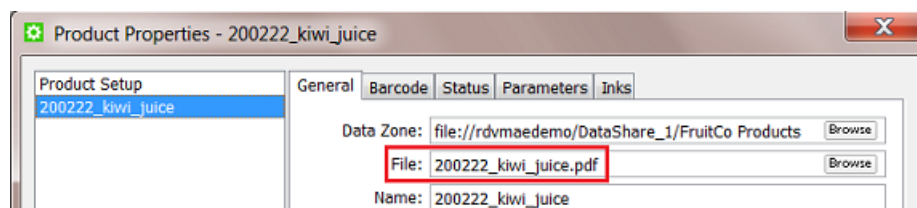
In this example many product part files share a same folder, they do not each have their own folder and subfolders. Here's how it looks in Container mode:



See below how, when the product thumbnail is selected, the **Data Zone** pane only shows the file's parent folder `FruitCo Products`. Also here, there is no need to see 'higher', the related files are in the folders below:



In the product part setup, the definition of **File** does not contain a folder name:

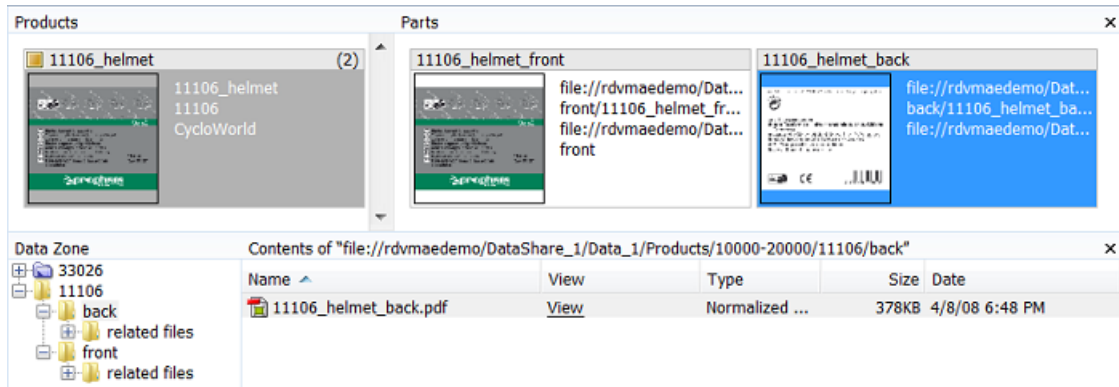


Example 4: a Product with 2 Parts with a Data Zone Depth of 1

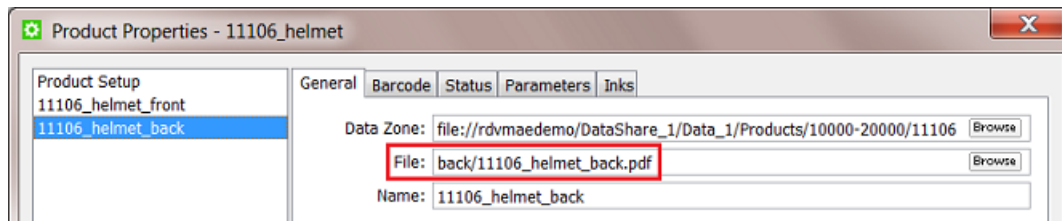
Note: Multi-part products were introduced [here](#).

This example shows a product that has 2 parts. Each product part file is stored in its own folder but they share a common root folder.

See below how, when you select a part, the **Data Zone** pane also shows 11106, which is 1 level higher than the folder the part file is located in:



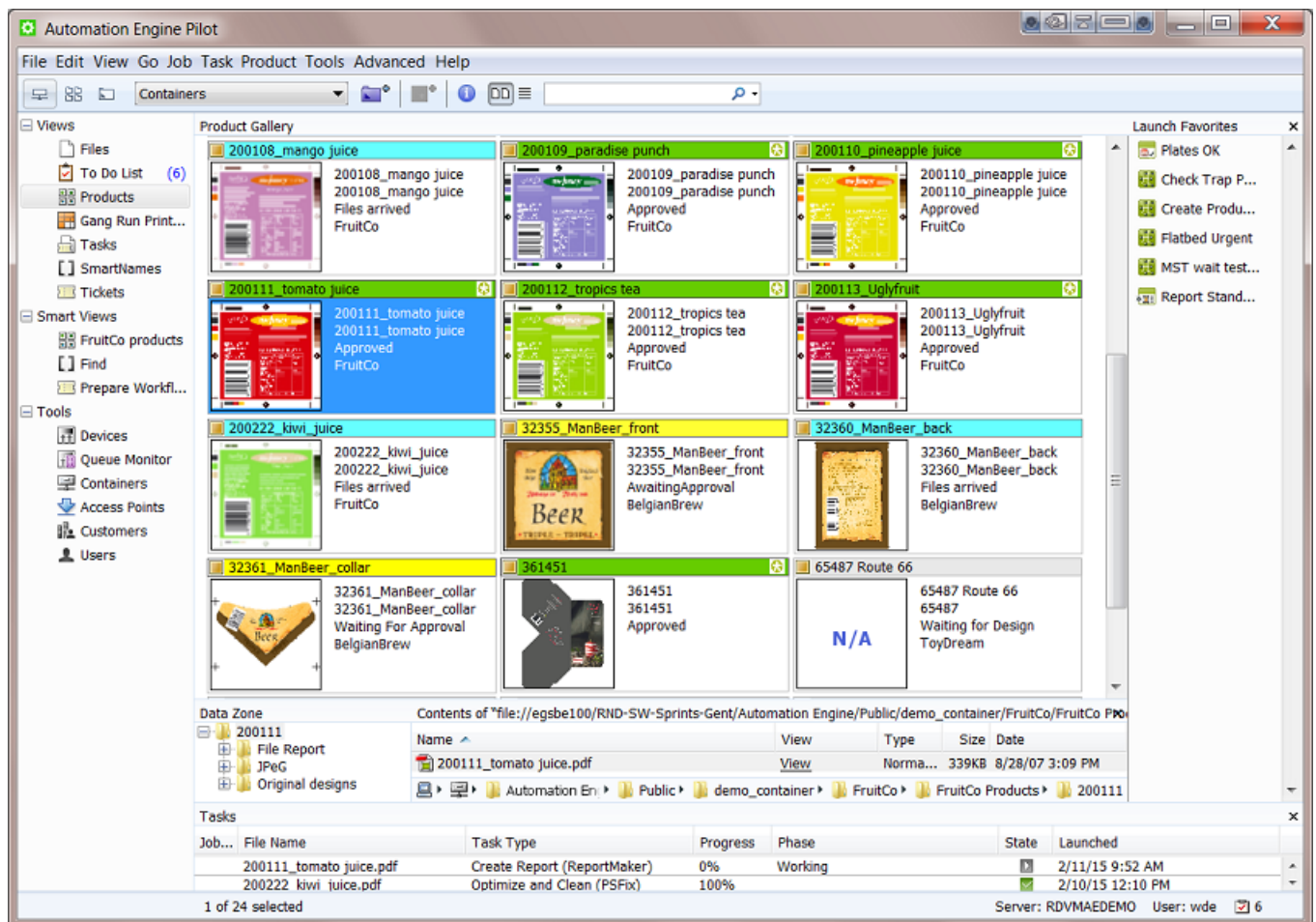
In the product part setup, the definition of **File** contains 1 folder name:



4.3 Products Views and Menu

Note: In the **Files** view you can not see if a file is a **Product Part**. Use the here described view modes to work with products:

4.3.1 The Products Gallery View



In Container mode, select the **Products** view to see

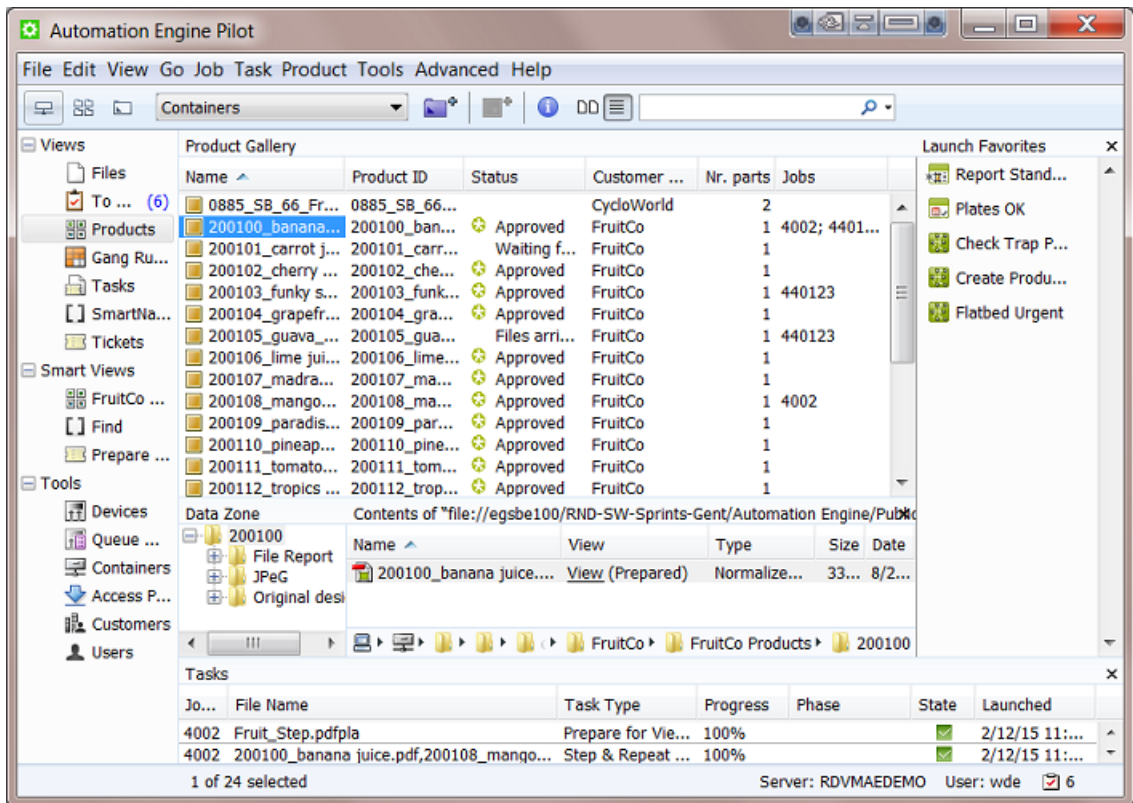
- the **Product Gallery**: a view on all or a filtered list of your Products. Choose the thumbnail view or the list view .
- the **Data Zone** pane (switch it on or off in the menu **Product**). This shows the data zone of the selected product. Learn more in [Products Data Structure](#) on page 881.
- the **Tasks** pane (switch it on or off in the menu **View**). This shows all tasks launched on any of the viewed products.
- the **Launch Favorites** pane (switch it on or off in the menu **View**). This shows your favorite tasks available for the file type of the selected product(s).

Note: A thumbnail mentions **N/A** when the thumbnail is not available. The reason for this can be

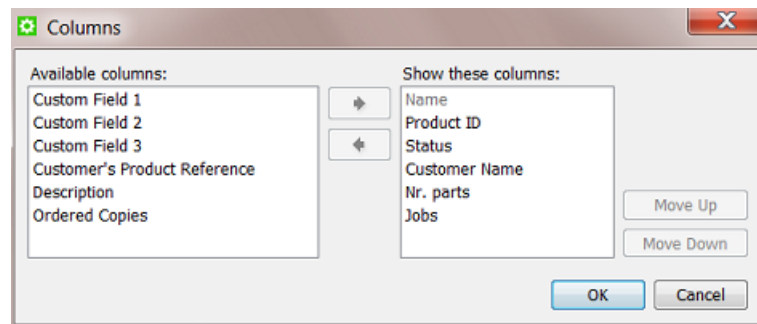
- because the product is an "empty" product: in JDF workflows, it is possible to create a product part that has no link to a file yet.
- because the product part file is not a supported graphics file type (as we listed [here](#)).

The Products List View

The list view offers the same functionality but shows the **Product Gallery** in columns:



Go to **View > Select Columns in > Product Gallery...** to customize which columns you see.



Tip: You can also right-click on a column title and deselect that column there.




Note: You can choose different columns for product parts (when also viewing the **Parts** pane).

Working in the Gallery view

Tip: You can select multiple products by using Command-click (Mac) or CTRL-click (Windows) or by using SHIFT-click to select a series.

You can use the Products Gallery view to

- create a new product (📄+). Learn more in [Creating Products](#) on page 897.

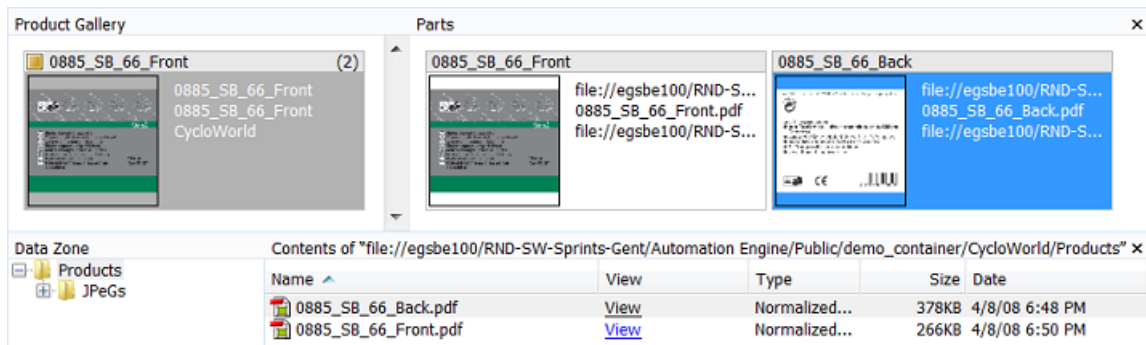
- get information about products:
 -  indicates a **Global Product**,  a **Job Product**. Learn more [here](#).
 - To see if a product is linked to one or more jobs, you need to open the product's **Properties**. Learn more in [Product \(Part\) Properties](#) on page 893.
 - In thumbnail view, the color of the top bar indicates the status. Learn more about statuses and their colors in [Product Properties](#). When a product has multiple parts that do not have the same status, this bar will be shown in grey. Learn more about multi-part products below.
 - In thumbnail view, a number in the top right corner indicates that the product has more than one product part. Learn more below.
 - In thumbnail view, the text fields describe the main product attributes. Use **View > Select Thumbnail info in > Product Gallery...** to customize the information that is shown there.
- manually change product properties. Press  or right-click the product(s) and choose **Product Properties....** Learn more in [Product \(Part\) Properties](#) on page 893.
- launch tasks on one or multiple products. Select the product(s), choose a favorite ticket or right-click and choose **New Task** or **Launch with**.

Note: A task that you launch on a product part that is linked to a job will run in the context of that job. This is important when you use job related SmartNames in the task's ticket(s).

Note: If a product is linked to more than one job, the task will run in the context of the job that this product was most recently linked to. Learn more in [Product Setup](#) on page 894.

Viewing Multiple Parts

When you select a product that has more than 1 part, an extra pane opens that shows those parts. This happens automatically when you activated the function '**Show Parts**' from the menu **Products**.



Tip: You can re-size this **Parts** pane, the Pilot will remember its size.

Note: In the **Data Zone** pane, the file(s) representing the selected product part(s) will be highlighted in grey (and vice versa).

Note: The name of the main product thumbnail is the name of the first product part (as defined in its properties). Learn more [here](#).

Note: The thumbnail info of a product part can be different than for the main product thumbnail. Set this info in **View > Select Thumbnail info in > Parts...** .

By seeing the separate parts, you have all the above mentioned functionality also available on product parts.

If however you want the functionality (a task or managing properties) to happen on all parts, then there is no need to view or select any parts. For example launching a task on the main product thumbnail will automatically launch this task on all parts.

Filtering and Searching

The filter and search tools enable to quickly make smart product selections. You can also save these filters as a [Smart View](#).

You can search and filter products in the same way as you do with Jobs. Learn more in [Job View Filters](#) on page 832.

More Functionality in this View

A lot of functionality is offered when you right-click one or multiple products:

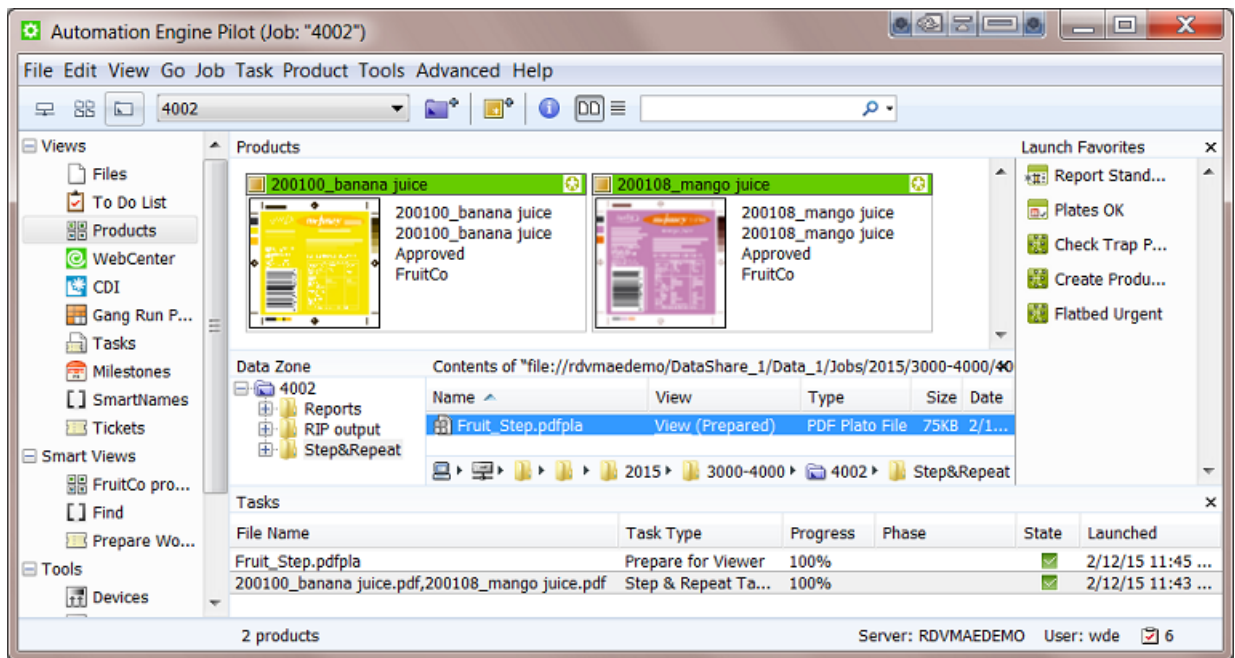
- Use **New Task** and **Launch with...** to launch a task or workflow on your selected product(s). Learn more in [Launching a Workflow on a File](#) on page 1034.
- Use **Open...**, **Open With** and **View...** to open the product part file in a chosen application or in the Automation Engine Viewer.
- Use **File Info...** to see the metadata of the product part file.
- All other features are described in [The Product Menu](#) on page 892.

4.3.2 The Product(s) in a Job View

When in **Last Used Job** mode, the **Products** view shows which products are linked into that job.

When this job is a "prepress job", your goal will be to get that product approved and ready for output. When this job is a "production job", you will probably work on step & repeat (in an editor or via tasks) and/or output tasks. Learn more about these job types in [Types of Jobs](#) on page 829.

See this an example of a production job that needs to output 2 labels:



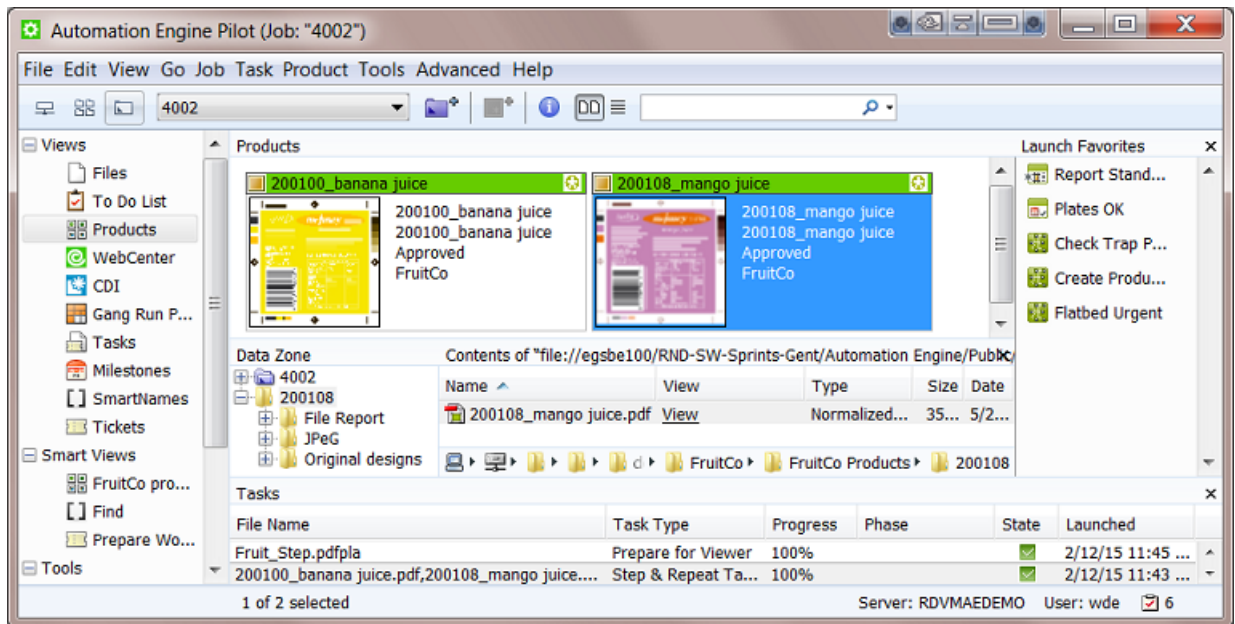
Notice how

- the 2 products have a status 'approved', which is typical when you start outputting them.
- a Step & Repeat task was launched on the 2 products.
- the resulting pdfpla file was output in the job's subfolder "Step&Repeat".
- that pdfpla file was selected to launch the **Prepare for Viewer** task.

Using the Data Zone pane to see All Related Data

In job mode, the **Data Zone** pane always shows the job folder. When a product is selected, you will also see the product's data zone.

In this example, selecting the 'mango' product made the **Data Zone** pane also show folder '200108' and its subfolders:



Note: The data zone of a product is decided at its creation time. Learn more in [Products Data Structure](#) on page 881.

In job mode, the **Data Zone** pane offers access to all the data relevant for that job:

- When you want to access a file in the job folder, browse to it in the (blue) job folder.
- When you want to access a product part file, click on the product part thumbnail and choose your action from there.
- When you want to access another file in the product's data zone, select the product part thumbnail to visualize it's data zone and then browse in those (yellow) product folder(s).

Hiding the Files View

The fact that you can access all data using the **Data Zone** pane, means that there is no more need to see the **Files** view (when you work with products).

Also, when you use the **Products** view and then switch to another job, the **Files** view will again be shown. To avoid that you need to click on the **Products** view again, you can choose to hide the **Files** View.

To do this, right-click on the **Files** view and choose **Remove from Views Bar**.

Tip: To see the **Job Setup** while staying in the **Products** view, select the blue job folder icon and use the shortcut CTRL-ALT-J (Windows) or Command-Alt-J (Mac).

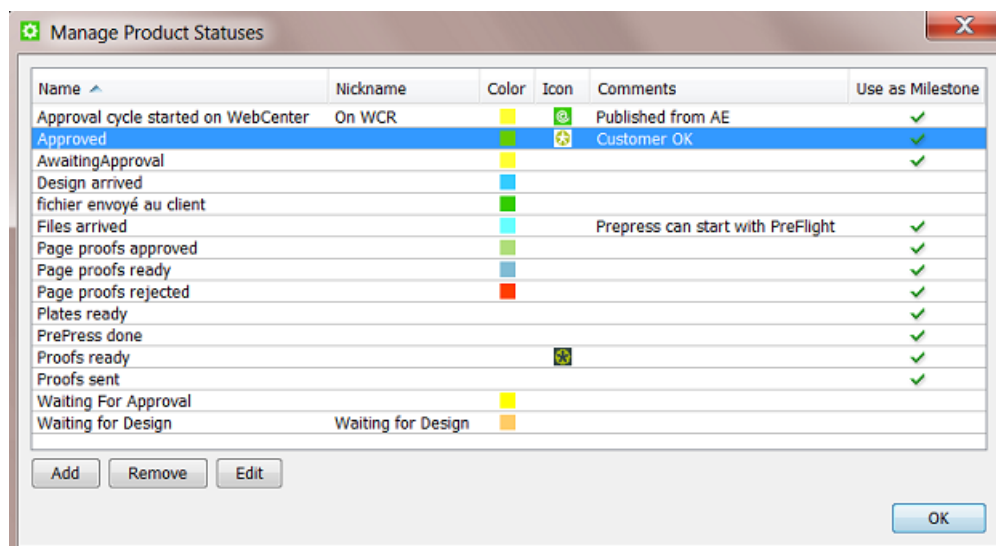
4.3.3 The Product Menu

Tip: Many of the features offered in the **Product** menu are also offered when you right-click on a selection that you made.

- **New Product...** allows you to browse to a file, create it as a product and link it to the job you are in. Learn more in [Creating Products Manually](#) on page 897.
- **Upload as Product...** does the same as above but also uploads the chosen file to its new product data zone. You can use this for example when the file is still on your local computer, on a CD/DVD drive or already in a Container but not yet in its proper product place. Learn more in [Uploading As a New Product](#) on page 898.

Note: These features are also available when you right-click on an empty piece of canvas in the Product Gallery.

- **Link Products....** Use this feature to manually link product(s) to the job you are in.
- **Product Properties...** opens the setup of the product and its parts. Learn more in [Product \(Part\) Properties](#) on page 893.
- **Go To File** shows you the product part file in the **Files** view of the **Container** mode.
- **Explore Folder...** opens a disk explorer view of the folder of the selected product part file.
- **Link to Job...** enables a quick way you to link a product(s) to a job from a list of most recent used jobs (or choose **Select Job...** to select any other). You will get a warning when the product is already linked to that job.
- **Unlink Product.** In **Job** mode, use this to remove the selected product(s) from that job. This does not delete any files.
- **Hide / Show Parts.** Learn more in [the Products Gallery View](#) about working with Products that have **Parts**.
- **Manage Product Filters....** If you saved filters in your **Products** view, manage them here and decide if you want to add them in the **Views** bar (as a **Smart View**).
- **Manage Statuses....** This is where you can add or modify custom product statuses. This list also contains **Milestone** names that were allowed to be used as product status. The way to add or modify them is the same as described in [Manage \(Custom\) Milestones](#) on page 860.



4.4 Product (Part) Properties

Product properties are the attributes that are stored in the Automation Engine Products database.

The concept is very similar to the **Job Setup** fields for Jobs. Most of the properties also become available as SmartNames, in a dedicated category **Product Information**. Learn more on how to use SmartNames [here](#).

Note: The **SmartNames** view allows to specifically test product part SmartNames. Learn more in [Also Show Product \(Part\) Parameter SmartNames](#) on page 932.

The properties are divided in general product properties ([Product Setup](#)) and properties specific to each product part ([Product Part Setup](#)).

Note: The concept of multi-part products was introduced [here](#).

You can also use the **Product Properties** dialog to manually **Add** or **Remove Parts**. You can use the arrow buttons to sort the parts. A product will get its name and thumbnail from the product part that is highest in this list.

4.4.1 Product Setup

General tab

Product Properties - 200100_banana juice

Product Setup
200100_banana juice

General Jobs Parameters

Name: 200100_banana juice

Product ID: 200100_banana juice

Customer Name: FruitCo

Customer's Product Reference: 2014-04-88 banana

Custom Field 1: Flexo or Digital

Custom Field 2: English

Custom Field 3: Australia

Description: new design summer 2014

Status:

* Fields marked with an asterisk are required.

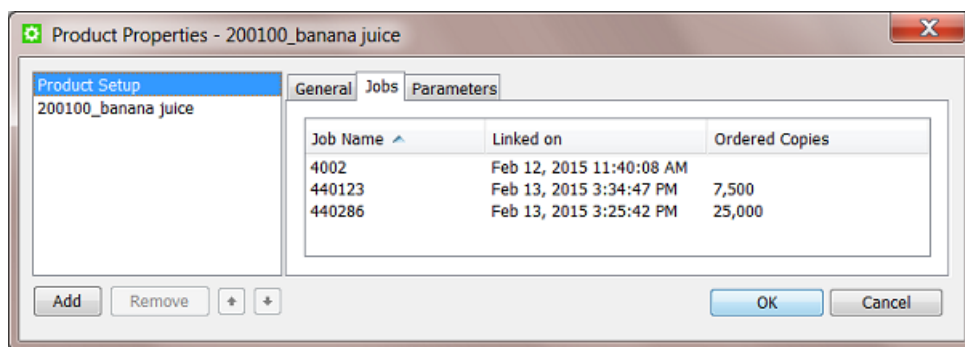
- **Name:** For global products, the product name has to be unique. For job products, the name has to be unique within a job. For example there can be multiple job products with the same product name as long as they exist in different jobs. Learn more about global and job products [here](#).
- **Product ID:** This is the product identification code, often (only) a number. The same remark as mentioned above about uniqueness is also valid here. The maximum length of a product ID is 32 characters.
- **Customer Name:** Select the customer that this product belongs to.
- **Customer's Product Reference:** This is the name your customer uses for this product. It can be useful as a search filter or for example to mention on a report file for this customer.

- **Custom Fields:** You can define up to three custom fields to help you search the products database or to use them as SmartNames.
- **Description:** You can add a description (maximum 128 characters).
- **Status:** Choose the products status from the list. Learn about standard and custom statuses in [The Product Menu](#) on page 892.

Tip: You can use the [Wait for Product Status](#) on page 525 task to hold and continue a workflow based on this status.

Jobs tab

This is where you see which job(s) that this product is currently linked to, and when that link was made.



Attention: If the product is linked to more than one job, a task that is launched on that product will run in the context of the job that this product part was most recently linked to.

Ordered Copies is an attribute that can be set when you use the [The Link Product to Job task](#) on page 909. This number is picked up in the **SmartLayout** function of Esko's Step & Repeat editor **Plato** (learn more in [Working with Products in Editors](#) on page 905). This number also becomes a SmartName that for example can add its value to a report file.

Parameters tab

You can here add custom parameters that also become available as SmartNames.



Attention: Mind that these parameters need to reflect what is specific for this product (range). Parameters that relate to how you *output* this product (Step&Repeat, press...) are probably more job related parameters and so belong in the job setup.

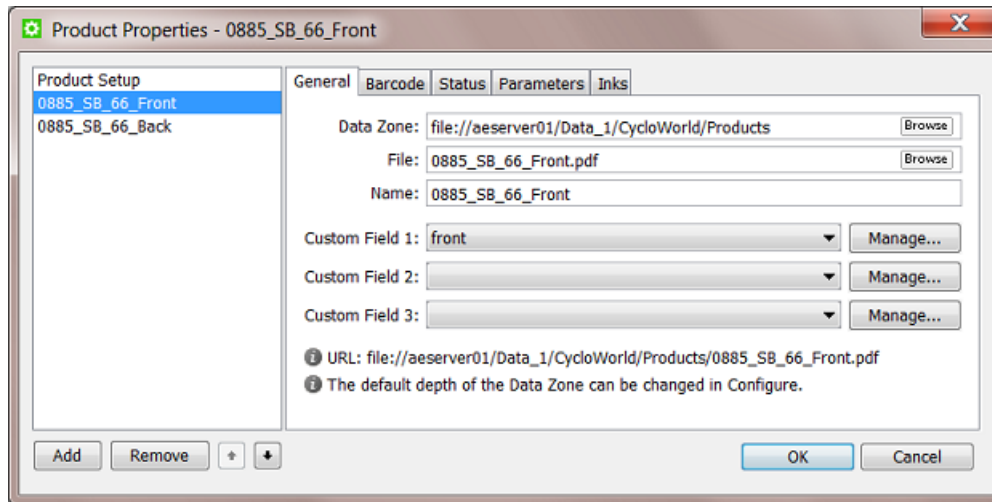
4.4.2 Product Part Setup

You can specify more and different properties for a product part because

- a part refers to a file. Some part properties relate to the file's location and its graphics characteristics.
- a product can have several parts with different properties (for example the front has the barcode and the back only has one ink).

Most properties also become available as SmartNames (in the category **Product Information**).

See this example of a product with 2 parts:



General tab

- **Data Zone:** This is the path to the folder containing the product part. This could also be a folder one or more levels higher. Learn more about the important concept **Data Zone** in [Products Data Structure](#) on page 881.
- **File:** This is the name of the file for that product part.

Note: If the parts file is in a subfolder within the parts data zone, you can include a folder name in this field: `subfoldername/file`. Learn more in [Products Data Structure](#) on page 881.

- **Name:** This is the name of the product part. This can be different than the file name.
- **Custom Fields:** You can add up to 3 custom fields.

Barcode tab

If the product part file has an (Esko) barcode, you can specify its properties here. The barcode can be checked and corrected using the [interaction with Esko editors](#) or by using the [Check Product Parameters](#) task. The details and usage is identical as with jobs: learn more in [the Job Setup Barcodes tab](#).

Status tab

Each product part can have its own status. This is relevant when a product has multiple parts.

Note: A product will automatically get the status of its parts when there is only one part or when the status of all its parts is the same.

To set a status, pick it from the list or use the [The Manage Product Status task](#) on page 910 task. If the status you choose has a **Nickname**, then that is the name you will see. Learn more about creating statuses in [The Product Menu](#) on page 892.

You see when and by whom the status of this product part was last changed. An extra note can also help clarify why the status was changed.

Tip: When the change was done by the [The Manage Product Status task](#) on page 910 task, the user name includes the name of the server.

Parameters tab

You can set custom parameters per product part. Their SmartName version shows up in the category **Product Part Parameters** and their name starts with [ppp.].

Inks tab

The inks in the product part file can be checked and corrected by the [interaction with Esko editors](#) or by using the [Check Product Parameters](#) task. Details and usage is identical as with jobs, learn more in [the Job Setup Inks tab](#).

Note: JDF workflows that create product (parts) can also specify such ink properties.

4.5 Creating Products

Products can be created

- manually, using the Pilot
 - by making a file a product part file
 - by uploading a file to a new place and then making that file a product part file.
- automatically
 - by using the [Add To Products](#) on page 254. The task then defines the product properties based on the file's URL or metadata or out of some XML file(s) etc.
 - by direct JDF/JMF commands. This technique communicates directly to the server software. For more information on JDF contact Esko Solution Services or find some basic answers in the chapter [Integrating with External Systems](#) on page 1228.

Note: You can also use above tools to modify products (except the upload method).


Note: Removing products is described [in this separate page](#).

4.5.1 Creating Products Manually

Note: Learn about the difference between global and job products [here](#).

Creating a Global Product

Follow these steps to manually create a new product:

1. In the Pilot, go to **Container** mode and **Products** view and click on . Alternatively, go to **Product > New Product...** or right-click on an empty part of the canvas in the **Product Gallery** pane and choose **New Product...** .
2. The dialog first asks you to select the file that you want to add as product part file. Make your selection and click **OK** to continue.

Important: When you select multiple files, the next steps will create them as parts of a same product.

Important: If you select a file that is located in an (active) job folder, you will automatically create a job product.

3. Now define the product's properties, both of the product and of the product part(s). Learn more in [Product \(Part\) Properties](#) on page 893.

Tip: At this point, you can use the **Add** button to add extra product parts.

4. Click **OK** to confirm and create the product. The new product appears in the product gallery.

Note: You will get a warning when a product part with the same name or ID already exists.

Creating a Job Product

To create a job product, you can use the same steps as above but you start in **Last Used Job** mode (you are in a job and you select a file located somewhere in that job folder).

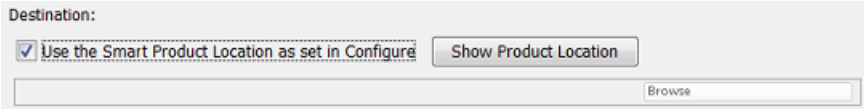
4.5.2 Uploading As a New Product

Upload as Product... does the same as creating a new product but also uploads the selected file to its new product data zone.

You can use this for example when the file is still on your local computer, on a CD/DVD drive or already in a Container but not yet in its correct product place.

Much like [Creating Products Manually](#), follow these steps to upload one or multiple file(s) as a new product (parts):

1. In **Container** or **Last used Job** mode, go to the menu **Product > Upload as Product...** or right-click on an empty part of the canvas in the **Product Gallery** pane and choose **Upload as Product...** .
2. Select the file(s) and fill in their properties (same steps as when [Creating Products Manually](#) on page 897).
3. In the product part setup, there is now one extra field where you have to define the **Destination** of the file. Choose the **Use the Smart Product Location as set in Configure** or browse to where you want to store the product part.



The **Smart Product Location** is a tool that can define the location and name of the product part based on a product parameter or on the default location for that customer. Learn more in [Smart Product Location](#) on page 678.

Note: The **Smart Product Location** can be checked by clicking on **Show Product Location**.

4. Confirm with **OK**.

4.5.3 Creating Products Automatically

You can create products automatically by using the [The Add to Products task](#) on page 908.

You can launch this task

- as a user in the Pilot, after selecting an input file. This is advised when you use a task ticket in which SmartNames already fill in many properties automatically.
- or this task can be part of a workflow that selects the input file(s) and also provides the values for the required product properties.

To automate the creation of products, you typically create workflows that combine the [The Add to Products task](#) on page 908 with some of these tools:

- the [Select File](#) on page 470 and the [Select Referenced File](#) on page 471 workflow control (when the file path is mentioned in an XML or CSV).
- [Using Regular Expressions](#) on page 978.
- the [Data Splitter](#) on page 308 workflow control.
- **System Value SmartNames** Learn more in [Creating System Value SmartNames](#) on page 944 and see an [Example of Looking for the Product Part URL](#) on page 945.

Some of these are used in these examples of the two main types of product creating workflows:

Example Workflow 1: Building your Products Database from your Existing Data

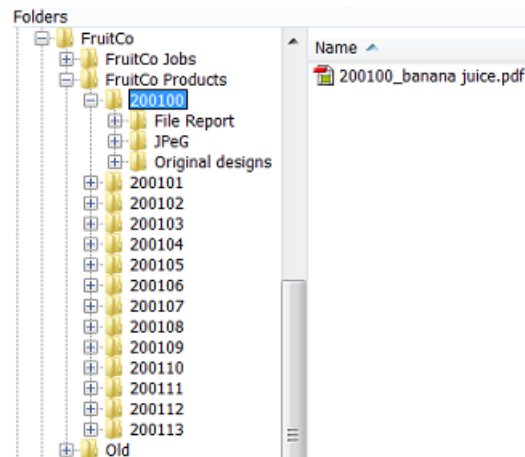
When you start to work with the products tool, you may already have a lot of data that you want to add to your products database. With this existing data, your goal is to launch workflows that create many dozens or hundreds of products in one go.



Attention: Based on what you learned in [Products Data Structure](#), you can decide to first re-arrange your data structure or to keep the data in their existing folders before you start creating your products.

Tip: It is good practise to first test this import on a part of your data, for example limited to one of your customers. We advise to also test your output workflows with some of that first set of products. You might learn details that you want to change before importing all your other products.

See this example where all the 200* folders contain one product part file in their root folder:



This is the workflow we will use to automate creating these products:



We first use a smart way to select only the part file we need. We then add it as a product. We then set the status.

These are important steps and settings in this workflow:

1. First manually select all the 200* folders and then launch or open your workflow.
2. The **Data splitter** (default settings) will split that group of folders and will so launch the **Select File** step on every separate folder.
3. In **Select File**, use these settings:

- **File Type**: in this example the type we want is a normalized PDF.

- In **Select other Files**, we select all those that are in the URL of what this step received as input: that (single) 200... - folder.
 - Do not **Look in subfolders** to avoid selecting the other PDFs or any report files.
4. In the **Add to Products** task, make sure to indicate the customer. We know the customer because we make a manual selection under the product folders of a specific customer.

Tip: When you do not set the customer via a SmartName, this workflow ticket becomes customer specific. In such cases we advise to add that customer name to the name of your workflow ticket.

5. We propose to already set a status. When these data were produced earlier, you can for example set their status to 'Approved'.

Tip: When you are testing these workflows, make sure to check the **Updating** options in the **Add to Products** task.

Example Workflow 2: Adding New Incoming Designs as Products

When Should a New Design become a Product?

The concept of products allows to add them to the database very early. As soon as you have a representation, a new design, you can add this file as a product part.

You can use the product status to indicate what stage this product is in, for example `New` or `Preflight OK` or `Waiting for Approval` etc.

Changing the Product Part File During Prepress Stages

You can also change the product part file during a workflow. For example you first use the design file as the product part file but once you import or normalize or start editing you then move that original design file to a subfolder `original` and update the product part with the more recent (normalized) PDF file.

Note: To update a product you use the same **Add to Products** task but use specific settings in its **Updating** tab.

Creating a Product in a Job Context or Not

It is your choice to add the product while you are in a job context (in Last Used Job mode) or not (in Container mode).

As mentioned in [Types of Jobs](#) on page 829, when you do not work in a job context, it will be difficult to give feedback to your business system. However, many such systems do not give you a job-order number to do prepress work on new designs. Many of them only send you job numbers for 'production jobs' in which you output already approved products.

In that case, you still have a choice to create your own 'prepress jobs'. A reason to do that is consistency, because then everything that your users do in prepress (with Automation Engine) happens in a job context.

Tip: But how do you name those self-created prepress jobs? You could base their name on the name of the product they are supposed to create. For example if you know you are preparing a new product that should be named 7890, then you could do this work in a job named `Preparing_7890`.

Avoid Changing a Product Name or ID: Get it Right from the Start

It is very important to get the product name and ID as soon as possible. You do not want to create a product and later change its name or ID. If this name is not sync with what the business system expects then there will be no or little integration possible. For example if you send the business system an early JPEG of the new design, it will need to have the correct name: the name of the item that this system expects.

In an ideal situation, your business system sends you a *prepress job* and in there mentions the name and ID of the product that it expects you to create.

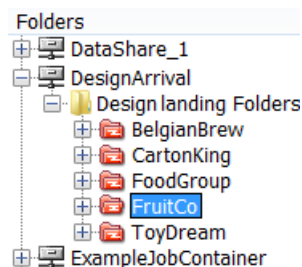
In all other cases, to get this correct product name or ID, you will probably need to get this information from people working with the business system. It is in that admin department that this number was created. In the business process, those colleagues made the decision that you should prepare and output a new design or variant, so they will have a record of that item number (and name).

Example Workflow

Your new designs can come in via many different ways. An important tool to help automate creating them as a product are *Access Points* on page 1236. An Access Point can detect a new incoming design and immediately launch a workflow that moves the file to the right folder for the product part and then adds it to the product database.

See this example:

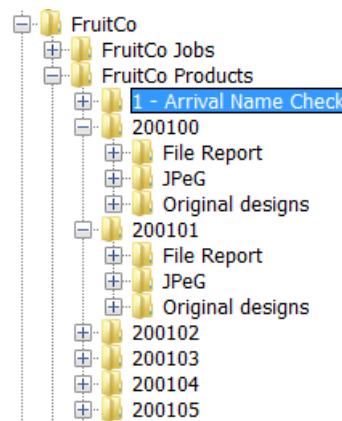
- A **Folder Access Point** detects new designs coming in from the customer `FruitCo`.



- This access point launches this workflow:



1. The found data is moved away from the access point folder to an 'arrival hall' for new designs for this customer:



Important: We use this intermediate place because we first want to check what the correct name and ID for this product will be.

2. The **CheckPoint** task asks a user to check with the admin department what the correct name and ID needs to be. When needed, the user manually changes the name of that design file.
3. The next **Move** task is now able to move the design to a new own product folder. The folder is decided by using SmartNames, based on the customer name and on the first 6 characters of the designs name.
4. Now that we have the file in the right place and with the right name, we create the product.
5. A preflight is executed on the file.
6. The status we set depends on the preflight result.

Note: You can of course extend this workflow with steps like editing and proofing. The goal of this 'prepress job' is to have an approved product item. An other job should then serve to output this product, possibly together with others.

Note: There are countless variations of how new designs can be processed. To create smart ways of automating their import to the products database, we advise to study all the possible tools mentioned in [Creating Products Automatically](#) on page 899.

4.6 Linking Products to a Job

A global product can be linked to one or multiple jobs. A job product can only be linked to one job.

You can link a product to a job

- manually, using the Pilot, in **Products** view,
 - in **Container** mode, by right-clicking one or multiple products, choosing **Link to Job** and selecting the job from the list of recently used jobs (or by using the dedicated menu item).
 - in **Last used Job** mode, by right-clicking in the **Products** panel, choosing **Link Products...** and selecting the one(s) you want (or by using the dedicated menu item).

- in **Last used Job** mode, in the **Data Zone** panel, by selecting a file in the job (sub)folder and choosing the menu item **Product > New Product...** . This will create a job product and automatically link it to this job.
- automatically,
 - by using the [The Link Product to Job task](#) on page 909.
 - by using JDF/JMF commands. This technique communicates directly to the server software. For more information on JDF contact Esko Solution Services or find some basic answers in the chapter [Integrating with External Systems](#) on page 1228.

Which Job(s) is this Product Linked To?

You can see the job(s) that a product is linked to

- in the product's properties. Learn more in [Product Setup](#).
- in ☰ the list view. Learn more in [Products Views and Menu](#) on page 886.

Which Product(s) does this Job Link to?

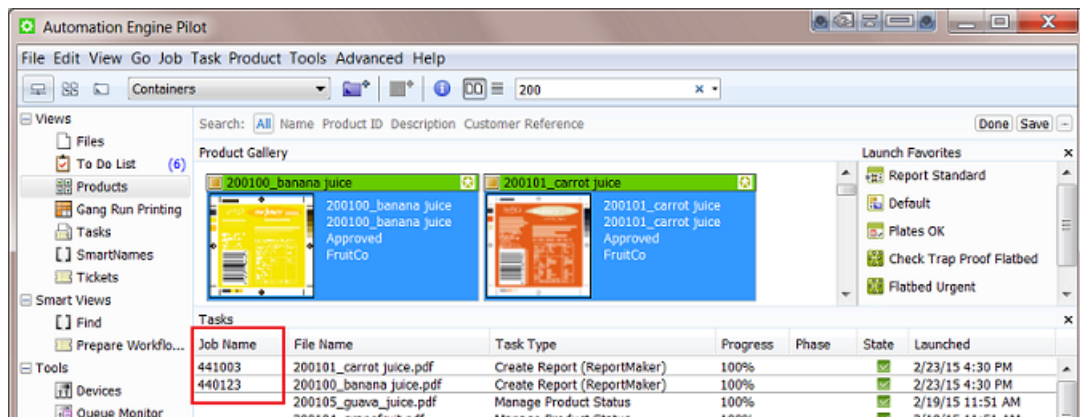
You can only see which products a job links to by opening the job in **Last Used Job** mode.

4.7 Launching Tasks on Products

Launching tasks on products is very similar to launching tasks on files.

Here is some information specific to launching tasks on products:

- As with files, you can right-click on products and launch tasks from there. You can also launch **Favorites**.
- Launching a task on a product will launch it on all its product parts. If you want to launch a task on a specific part, use [Show Parts](#) to see the **Parts** panel and make your specific selection there.
- In **Last used Job** mode, in **Products** view, when you select one or more products and launch a task, it will run in the context of that job. The SmartNames will relate to that job.
- In **Container** mode,
 - tasks launched on products check if those products are linked to a job and will run in the context of that job. See an example where two products were selected and the ReportMaker task was launched:



- when a product is linked to multiple jobs, the task will run in the context of the job that the product was last linked to. Learn more in the [Product Setup](#) on page 894.
- In a workflow, you can use the [Select File](#) on page 470 task to select all the product parts of the job (context) that that workflow is running in. The next workflow step is then launched on those product parts.

4.8 Working with Products in Editors

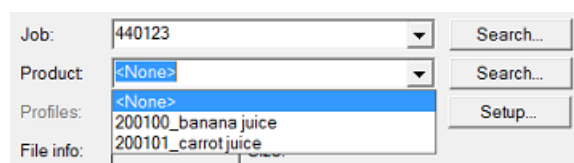
Editors that have been configured to be a client of your Automation Engine server, will show extra tools that help you work with products.

Easy Access to Open Products in Editors

Esko editors like ArtPro, PackEdge and Plato that communicate with an Automation Engine server, offer an easy way to open product part files.

In their **Open** dialog, extra **Job** and **Product** lists help avoid a lot of browsing:

- You can select a job first and then the products linked in that job appear in the list of products:



- You can press **Search...** to open a dedicated **Select Job** or **Select Product** dialog.
- You can click on the drop-down arrow to pick one from the memory of last used jobs or products.
- When you open a product that has multiple parts, an extra dialog will open to let you choose which part you want to open.

Tip: When you are editing a product and you want to update it with your changes, then you best use **Save** and not **Save as**. Saving as another name and/or in another folder will not change the product part's URL.

Editors Checking a Product's Barcode and Ink Parameters

As mentioned in the chapter on [Jobs](#) on page 828, Esko editors check if the file's barcode and ink settings match those that were defined in the file's job setup (in [Job Ink parameters](#) and [Job Barcode parameters](#)).

The same principle and tools are valid when your editor opens a product part file that has properties for barcode and/or inks.

Note: When both the file's product part and the file's job have such parameters defined, then the product part parameters are checked.

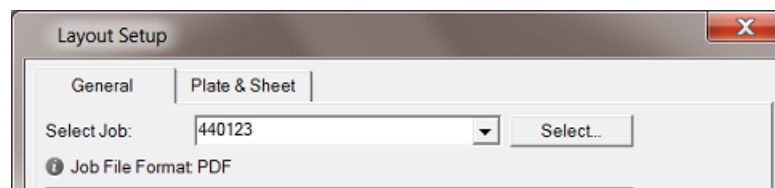
Note: When an editor is checking product parameters, the dialogs use the same wording: they also mention "Check *Job* Parameters" when they check *product* parameters.

And, as with job parameters, there is also a dedicated task to check or correct such product parameters. Learn more in [the Check Product Parameters task](#).

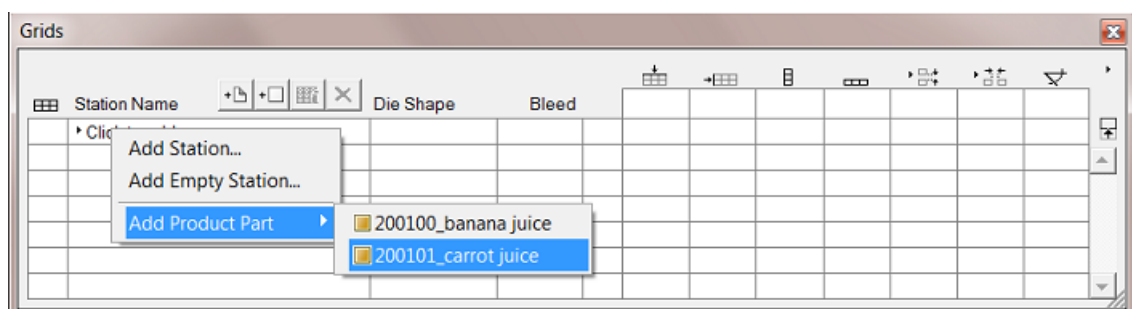
In Esko Plato: Fast Ways to Load Products onto a Layout

Esko's Step & Repeat editor Plato offers extra tools that also avoid searching and browsing when you put **stations** on the layout.

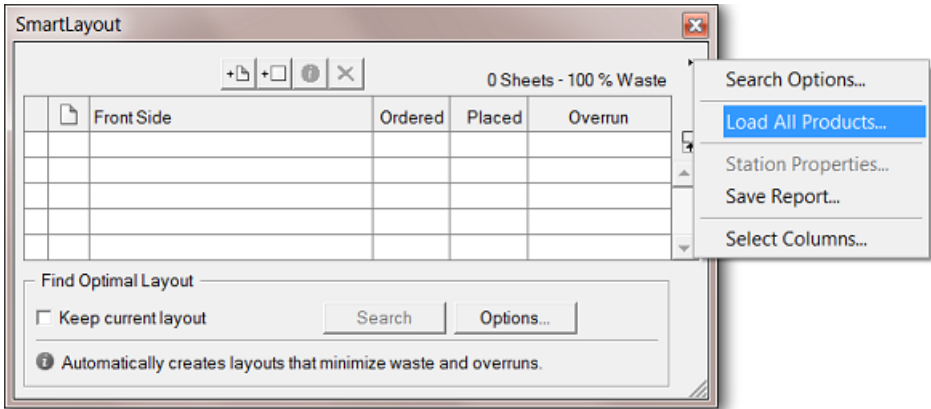
For this to work, Plato first needs to be aware in which job you are working. This is done in the **Layout Setup** dialog:



- When you create a **Manual Layout**, the dialog **Place Station** offers the same job and products lists as in the **Open** dialog, as shown above.
- When you create a new grid based layout, your right-click to add a station now also offers to add the product parts (that are linked in that job):



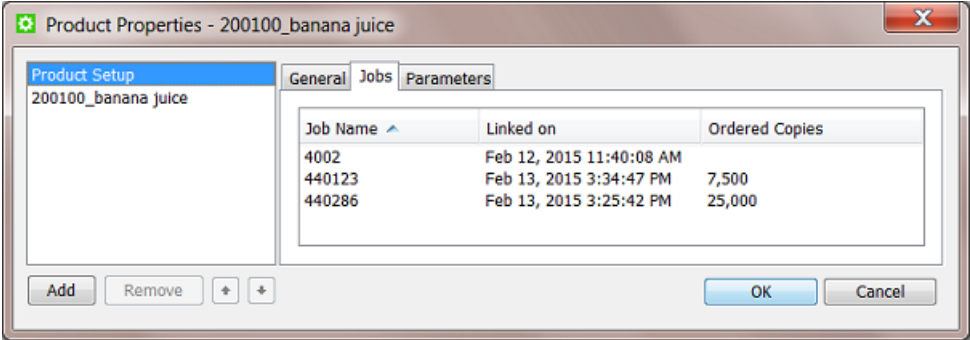
- When you create a **SmartLayout**, you can choose to **Load All Products...** that are linked into that job. Customers who often make layouts with many different files will quickly see the benefit of this function:



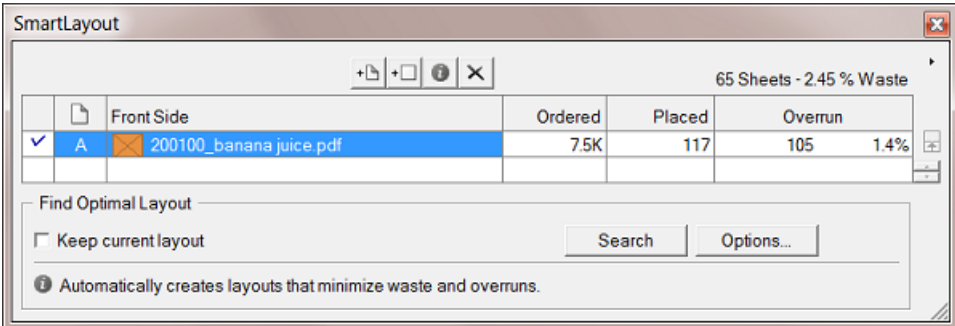
In Esko Plato: Using the Product Parameter "Ordered Copies"

When you load a product part in **SmartLayout**, Plato also checks the *Product Setup* parameter **Ordered Copies** and automatically fills it in in the **SmartLayout** dialog.

An example: This product is linked into 3 jobs but for job 440123 there are 7.500 **Ordered Copies**:



When Plato works in the context of job 440123, and when you use that product in your SmartLayout, that number of ordered copies will be filled in automatically:



Learn more about Plato [here](#).

4.9 Tasks specific for Products

4.9.1 The Add to Products task



Attention: This task requires you to be familiar with the manual setup of [Products](#) on page 876.

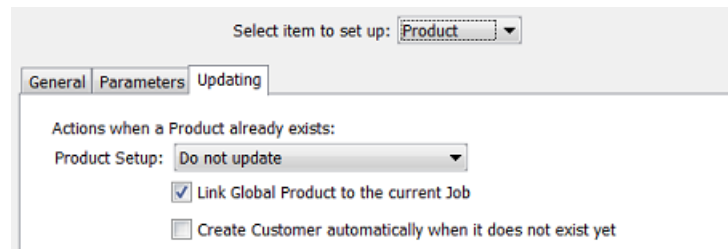
You can use this task to create a product (part) or to add or modify parameters of a product (part). You can not delete product (part) parameters using this task. Learn more below on how to update fields. The setup fields can be filled with SmartNames, typed in values or values chosen from a list.

Select item to set up: There are different tab sheets for product and for product part setup. Every product has at least one product part, so both are important.

Setting Up the Product

- **General:** You here define the main product properties. This tab is similar to the **General** tab when you manually define a **Product Setup**. Learn more about each field in [Product Setup](#) on page 894.

- **Parameters:** You can here define custom product parameters. This tab is identical to the **Parameters** tab in the **Product Setup**. Learn more about each field in [Product Setup](#) on page 894.
- **Updating:** When a product with the same **Product ID** or **Product Name** already exists, this task will update that product. Define the rules here.



- **Update:** If you want the existing product to be updated with the info from the task.
- **Do not update:** If you do not want the existing parameters to be updated.
- **Update and provide a warning.**
- **Do not update and provide an error.**

Link Global Product to the current Job: If you enable this the product in the task will be linked to the current job context.

- **Create Customer automatically when it does not exist yet.** Indicate if you want this.

Note: Mind that a customer exists when the **Customer ID** already exists.

Setting Up the Product Part

The settings for the product part are very similar to the ones for the product, that we explained [above](#).

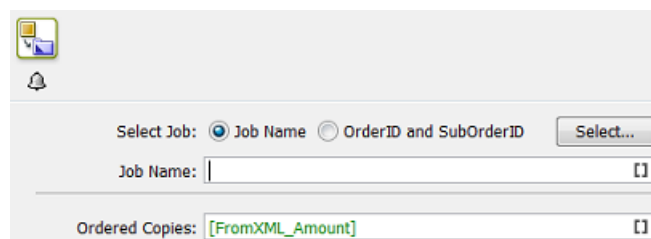
For the part, there are extra settings for the part's **File**:

- **Update:** If you want to update the product part with your task input file.
- **Do not update:** When the product part should not be updated with your task input file.
- **Update and provide a warning.**

4.9.2 The Link Product to Job task

The **Link Product to Job** task allows to link one or multiple products to a job.

It also allows to specify the **Ordered Copies**: this is the *amount* of product items that need to be produced in that job order.



Select Job

The job to link to can be indicated via its **Name** or via the combination of **OrderID** and **SubOrderID**. You can select a specific job from the list but typically you will enter a SmartName here.

Note: When the job does not exist, the task generates an error.

Note: When the input file belongs to a product that is already linked to the specified job, the task does nothing. This also means that the **Ordered Copies** specified in the task is then not set in the product properties.

Ordered Copies

Enter the amount manually or use a SmartName. Learn more in [Product Setup](#) and in [using the ordered copies parameter in Esko Plato](#).

About the Task Input File

- When the input file is a product part of an existing **Global Product**, the product is linked to the specified job and the ordered copies is set for that job (if the product was not yet linked to the specified job).
- When the input file is a product part of an existing **Job Product**, the task generates an error.
- When the input file is not yet a product part, the task generates an error.

4.9.3 The Manage Product Status task

The **Manage Product Status** task allows to set the status of a product.

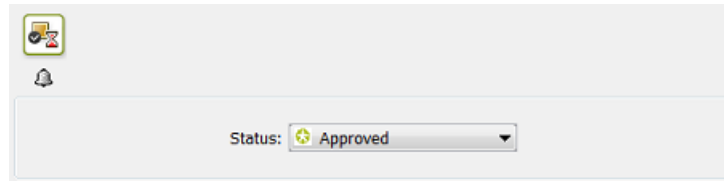
You can add an extra comment.

Tip: Adding a comment is most useful when the status change was a user's decision. For example a comment that mentions that the customer couldn't approve via E-mail or web due to network problems and so exceptionally called you to confirm that he approves the file.

This task is often used to automatically set the product status during a workflow. See some examples in [Creating Products Automatically](#) on page 899.

4.9.4 The Wait for Product Status task

The **Wait for Product Status** task allows to hold a workflow until (all) the input product part(s) reach the status specified in the ticket.



This task will wait until **all** Product Parts reach that status.

Learn more about product (part) status in [The Product Menu](#) on page 892 and [Product \(Part\) Properties](#) on page 893.

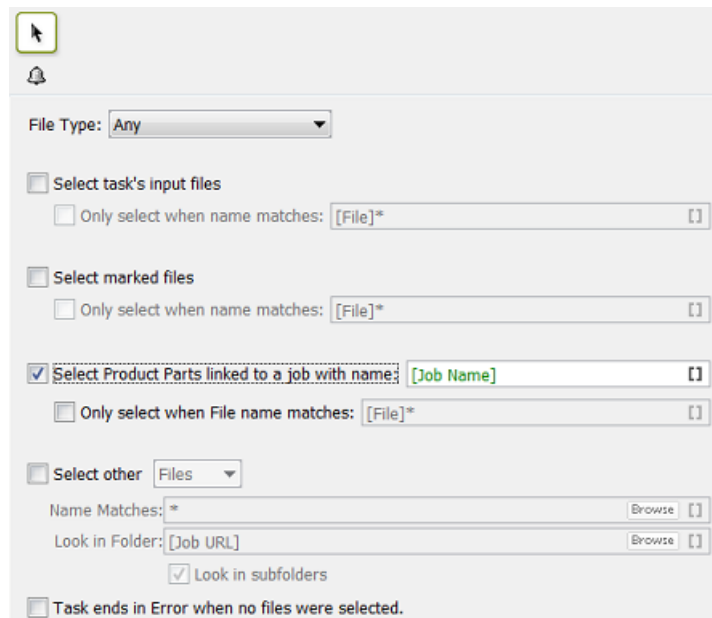
Note: This task ends in error

- when none of the input files are product parts.
- when one of the input product parts is deleted from the products database while the task is waiting.

This task is typically used to prevent output before the product part(s) are approved. See this example workflow:



Notice how the input of the **Wait for Product Status** task is defined by the [Select File](#) on page 470 task that has a specific option to select all the product parts in the job:



4.9.5 The Check Product Parameters task

Note: The concept and tools explained here are very similar to those in the [The Check Job Parameters Task](#) on page 874.

The **Check Product Parameters** task allows to make sure that the barcode and/or inks of the task's input file match the properties specified in its [Product Part Setup](#). You can use the task just to check the file or you can use it to automatically correct the file.

Category	Action
<input checked="" type="checkbox"/> Barcode	Auto Correct
<input checked="" type="checkbox"/> Inks	Auto Correct
	Auto Correct (Strict Checking)
	Stop with Error

Note: The input file must be a **Product Part** file.

Important: When no output file name or folder were specified in the ticket, the **Auto Correct** actions will overwrite the input file.

For **Barcode** settings, you can select:

- **Auto Correct** when you want to correct the file according to what is specified in its [Product Part Setup](#).

For example, your normalized PDF includes an Esko barcode of type EAN-13 and code 123456789012, and your product part specifies EAN-13 and code 5449000050205, then this task will automatically correct the barcode number in the PDF to 5449000050205.

- **Stop with Error** when you want the task to stop with an error if the barcode in the [Product Part Setup](#) does not fully match the barcode in the file.

Note: This task only checks barcodes created in Esko PackEdge or by using Esko's barcode plug-in for Adobe Illustrator (DeskPack). If the task does not find such a barcode, the task will end in a warning.

Note: You cannot use this task to check and auto correct barcodes in normalized PDFs generated by **ArtPro**.

For **Inks**, you can select:

- **Auto Correct:** For each ink in the input file, the task checks if the characteristics of the ink match the settings in product part parameters. If they don't match the file is corrected.
- **Auto Correct (Strict Checking):** Same as above but also generates an error if the file's list of inks and the order of inks is not exactly as defined in the product part parameters.

- **Stop with Error:** This is a strict check on the list of inks, their order and their characteristics.

4.10 Publishing Products to WebCenter

You can use the [Publish On WebCenter](#) on page 449 task to publish files from within an Automation Engine container to a WebCenter project.

When you launch that task on an Automation Engine product,

- the product part file will be stored as a project document
- the product part file will also be stored as a document in the **Bill Of Material** of that WebCenter project. The product (part) properties can be stored as WebCenter document attributes.

Note: The reverse is not possible: it is not possible to push WebCenter document attributes to Automation Engine product parameters.

Learn more about this in [Publishing Products to WebCenter](#), a page in the dedicated chapter on [Integrating with WebCenter](#) on page 1346.

4.11 Backing Up and Archiving Products

Backing Up Product Data

As explained in [The Products Concept](#) on page 876, data belonging to products are important data that you want to keep online for a long time. This is why we advise to schedule a regular backup of your product data. This can be done with many specialized (non-Esko) backup applications.

Additionally, we advise to use the [database backup tools of Automation Engine](#). The products database contains the properties of all your products. Remember that you can not restore selections of this database, restoring is done on the complete database. And remember that the products database is not a separate database but a database table in the jobs database.

Note: Some customers also use the Pilot to back up the product part files. You can select all or groups of products in your products gallery and launch tasks that can copy that data to a near-line disk or off-line to some device (CD/DVD). You can launch the [Create PAF / JPG / XML \(LinkEdge\)](#) on page 285 task (on normalized PDFs) or the [Copy](#) task or you can create workflows that copy the whole product part data zone. If you accidentally lost product data, it will then be easy to recuperate a selection of that copied data.

Archiving Product Data

As explained in [The Products Concept](#) on page 876, a product should stay in your database as long as a job-order can come in that needs (to output) it. Ideally, the business system informs you when its users have decided that a product item is 'end-of-life'.

Some customers keep some outdated products in the database to use them as basis for new designs. When you do this, mind that exceptionally large databases may cause your system to slow down.

Tip: When you do keep outdated products in the database, make sure to give them a status that is easy to recognize, for example `end-of-life` or `do not output`.

Another reason to remove old products is to avoid the risk of using them again in production.

Tip: You can reduce that risk by using a workflow that checks the status before outputting. See an example in [The Wait for Product Status task](#) on page 910.

Important: The [Archive Job](#) on page 260 task archives and restores job folders and job products. Global product files are not stored inside job folders and will not be archived by this task.

In summary, there is no specific task or procedure to archive products. We advise to stick to these general rules:

- Make regular backups of all your product data and database.
- Remove products from the database when you can.
- Use the product status to help indicate if a product can still be used for output.

4.12 Removing Products

What Happens when you Remove a Product?

When you remove a product, the product and all its parts are removed from the product database. The product part files on disk are not removed.

Removing a **Global Product** does not leave a memory of the product (part) parameters. When you make the file a product (part) again, it will not propose you any previous settings.

This is different when the product was a **Job Product** that was removed together with the job. When you remove a job it always stores a memory of all its settings in a subfolder and this is also done for settings of any job products. When you then use that ex-jobfolder to (re-)create a job, the old product(s) are re-created in this job as job products and will have all their product (part) settings again.

When do you Remove a Product?

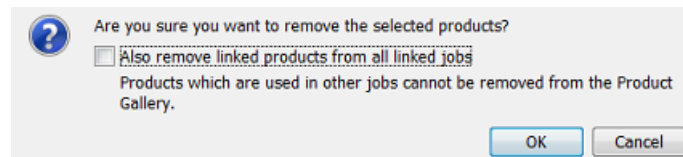
As mentioned in [The Products Concept](#) and in [Backing Up and Archiving Products](#), (global) products are created to be used for a long term.

It makes sense to remove a product when you are sure it will never need to be produced (output) again. In other words, when there are no more benefits of having it in a database: when you don't link it into jobs anymore or you don't use its SmartNames any more.

Removing a Product when in Container mode

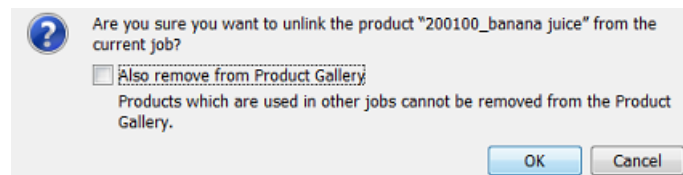
In the Product Gallery, select the product(s) and press `Delete` or use the right-click or menu function **Product > Remove Product**. A dialog will ask you to confirm.

By default, you can not remove products when they are still linked to a job. However, you can force this removal by using the option to also remove those job-links at the moment you remove the product(s). This is the option **Also remove from linked jobs** or **Also remove linked products from all linked jobs**.

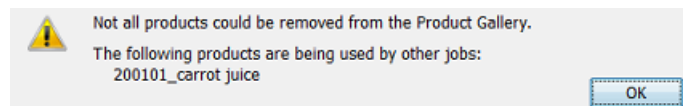


Removing a Product when in Job mode

When you are in a job's **Products** view and you select a linked product, pressing `Delete` serves to **Unlink** the product from that job. However, the resulting dialog offers an option to **Also remove** the product from the products database:



When you are asking to remove a product that is still linked to other jobs, you will see this warning:



If you want to remove products that are linked to multiple jobs, you need to do this in Container mode (in the Product Gallery).

5. SmartNames

5.1 Concept

What is a SmartName?

A SmartName is a variable that refers to a value. You use them by inserting them in input fields all across Automation Engine. For example the SmartName `[File]` refers to the name of the task's input file, and the SmartName `[Date]` refers to the date the task is launched.

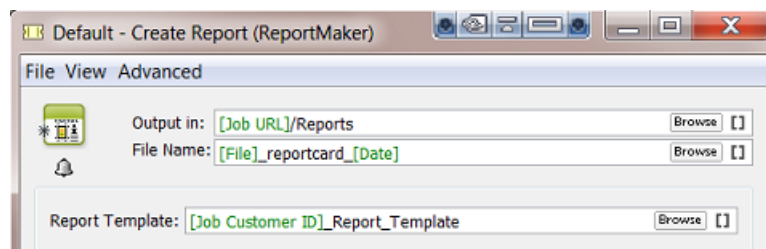
SmartNames are shown in green and between square brackets. For example `[File]`.

Where can SmartNames be used?

Note: Not all SmartNames can be used everywhere. To learn about limiting the places where they can be used see [SmartName Scopes](#) on page 926.

- **In Task Tickets, Workflow Steps and Access Points**

See an example of a **ReportMaker** ticket:



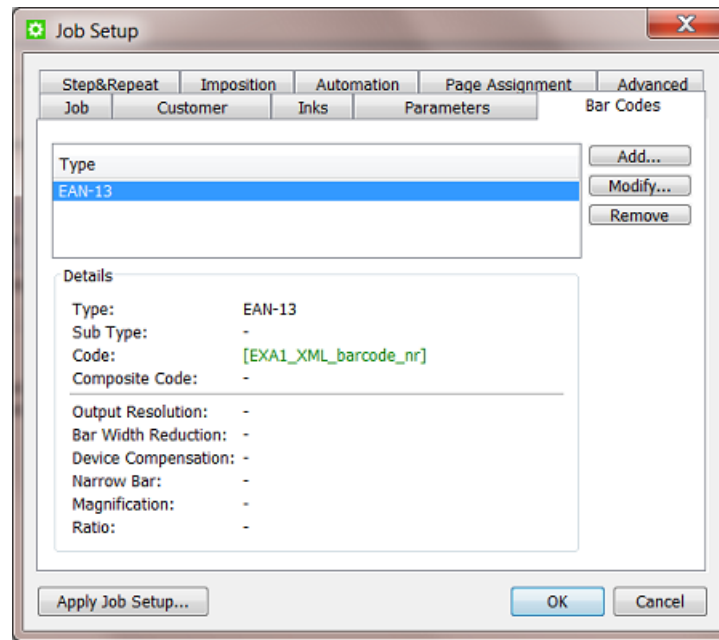
The output file will be written in the Job Folder, in a sub folder 'Reports'. The output file name will start with the name of the input file name plus the suffix '_reportcard_' and ending with the date when this task is launched.

Tip: You here also see how you can make combinations of SmartNames and regular (static) text.

The report template that will be used starts with the customer's ID number and ends with '_Report_Template'. This makes the ticket smart enough to select the template that this customer prefers.

- **In some fields of the Job Setup**

In this example a SmartName defines the code of the bar code in the Job Setup (the code is a variable that will be coming in from an XML file):

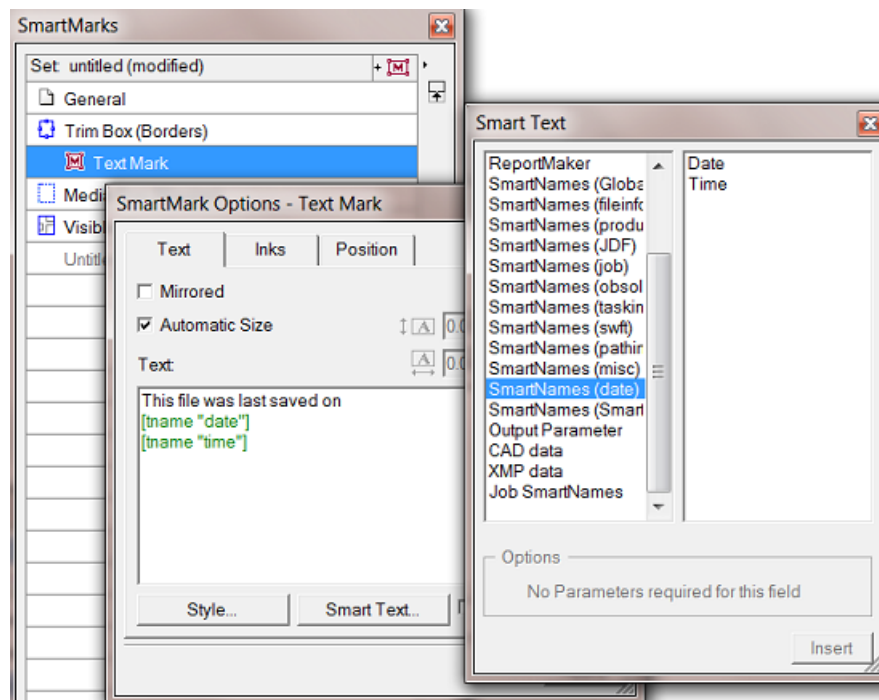


- **As Workflow Parameters in Workflow Steps**

When your workflow contains **Workflow Parameters**, you can pick them up as **Workflow Parameters SmartNames** in task tickets of tasks used in that workflow. Learn more about **Workflow Parameters** in [Workflow Parameters](#).

- **Inside SmartMarks**

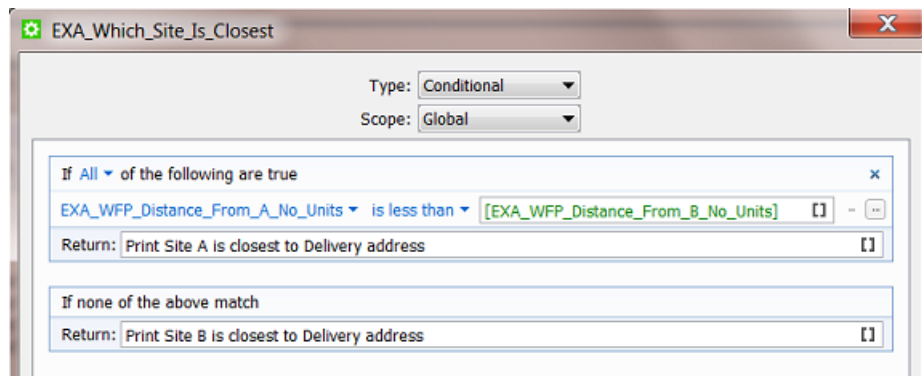
See this example of how a **SmartMark** (here in **PackEdge**) uses a **Smart Text** that also picks up 2 SmartNames:



- **Inside the setup of other SmartNames**

Some examples:

- You can create a custom SmartName that extracts the first 5 characters from the [Job Name]
- In below example, a conditional SmartName 'EXA_Which_Site_Is_Closest' compares the value of two other SmartNames



- **Inside some fields in the Customer setup**
- **In Configure, in some setup fields of Jobs and Products**

Advantages

SmartNames are a crucial part of Automation Engine's power. They bring speed and error reduction to your workflow through these effects:

- **Consistency.** Folders and file names will be consistent, no typing errors.
- **Automation.** You will rarely need operator intervention to edit workflows.
- **Integration.** SmartNames can pick up their value from other systems without human intervention.
- **Organisation.** Making workflows smart means you need to prepare less workflows. Smart workflows adapt to the input files and the variable job instructions.

When you first install or update Automation Engine, you already get many **Default SmartNames**. You can also make your own.

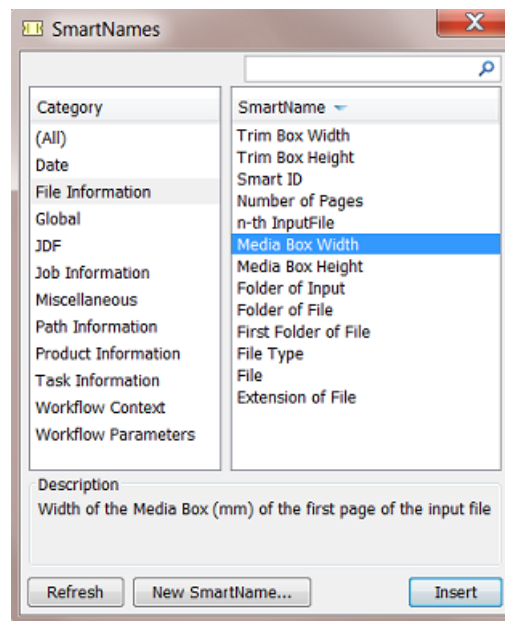
In the next chapters, we first give an overview of the general principles. Then we describe the different types of SmartNames in more detail. We both explain the default ones and then show you how to create your own.

5.2 General Principles

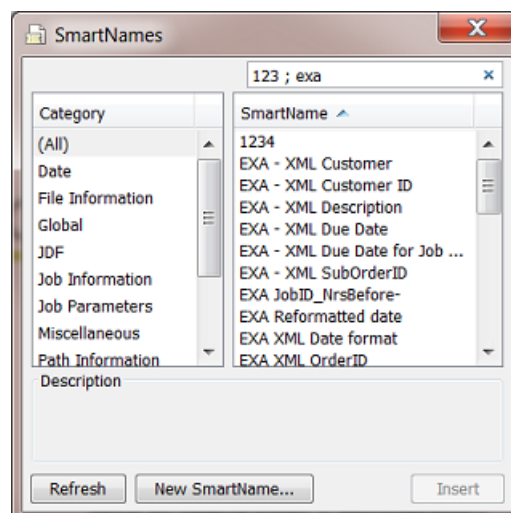
5.2.1 Using a SmartName

There are 2 ways to use an existing SmartName:

- click on **[]** to open the **SmartName Browser**. Choose a category, select the SmartName and click **Insert**.

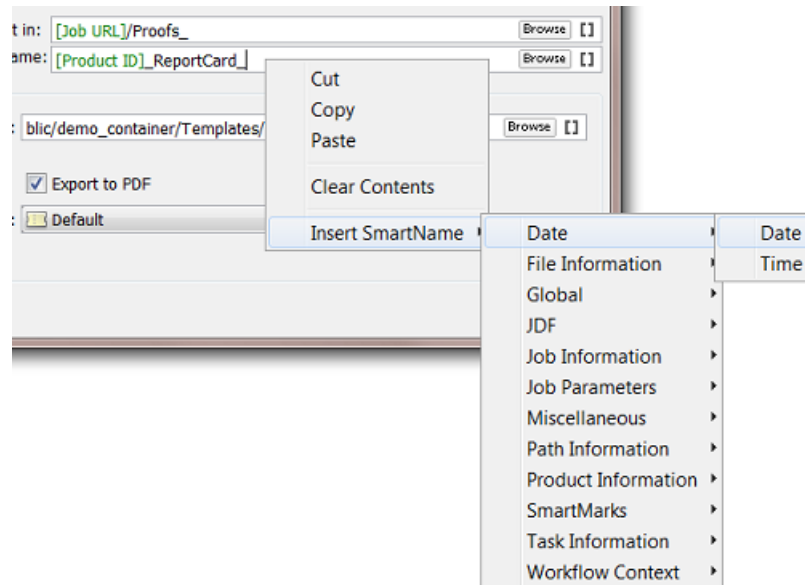


- When you want to insert multiple SmartNames, press ALT while clicking **Insert**. Then, the **SmartName Browser** will not close.
- You can filter the shown SmartNames by entering text in the search filter top-right. Use double quotes (") when an exact match is required.
- You can even combine search filters. For example when you type '123 exa', you ask to see SmartNames that contain both 123 **and** exa in their name or **Tag** (learn about tags [here](#)). Add a semicolon '123 ; exa' to see SmartNames that contain 123 **or** exa.



- Click **Refresh** to make sure that the most recent created SmartNames are also shown (or press CTRL-R or CMD-R).
- Click **New SmartName** if you want to create a new one from here, while keeping this dialog open.

- right-click in the field, choose **Insert SmartName...** and choose a SmartName from the offered categories.



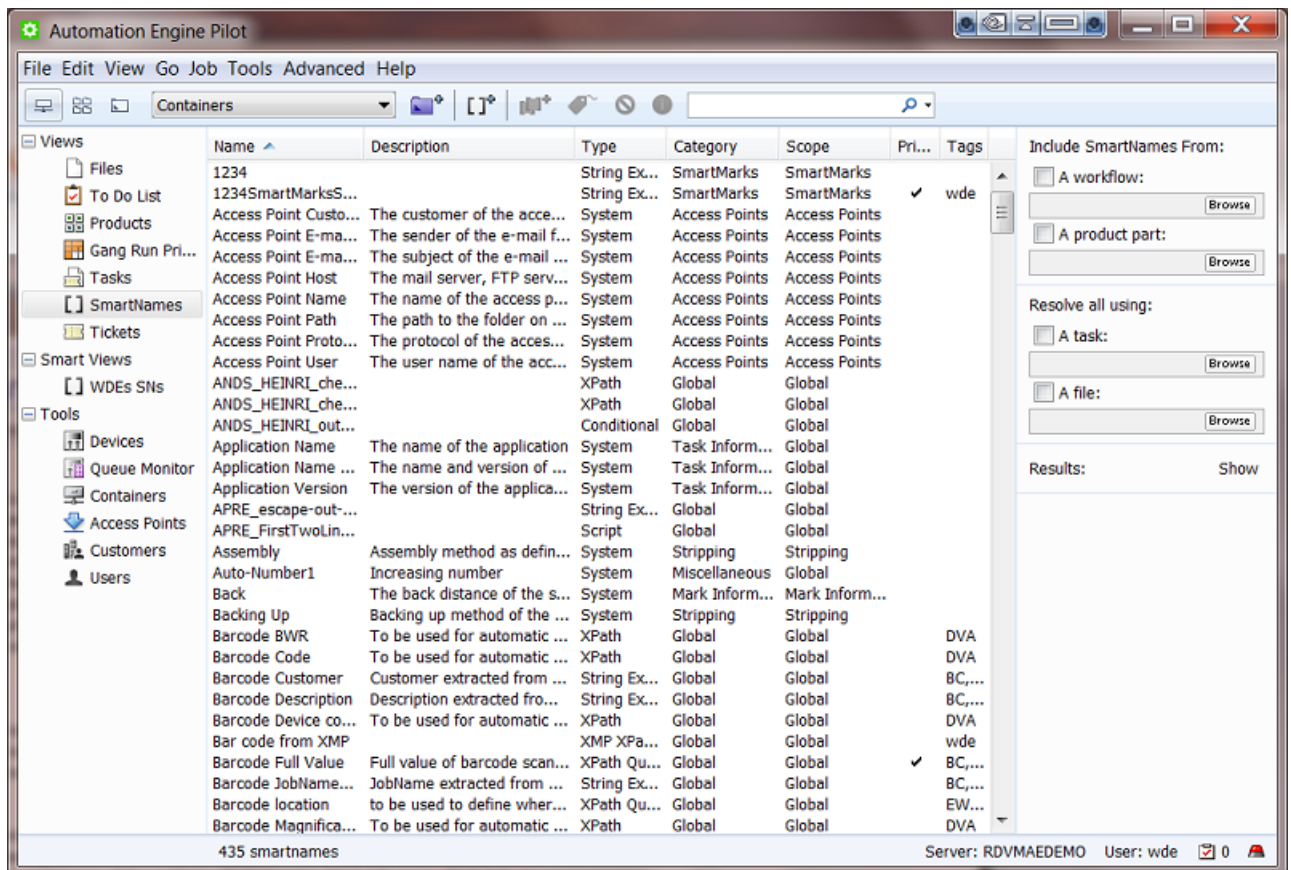
Important: In both ways, make sure you first position your cursor in the exact place where you want to insert the SmartName.

Tip: It is possible to select and cut, copy and paste SmartNames.

5.2.2 The SmartNames View

The **SmartNames View** in your **Pilot** is where you

- see all your SmartNames, both the default ones that came with Automation Engine and the ones you created extra
- add filters to help find and organize them
- create, modify or delete SmartNames (if you have the **User Access Right**. Read more in [Defining User Access Rights](#) on page 33)
- test your SmartNames ("resolve").



As usual, you can control the columns in this view. To do this, go to the menu **View > Select Columns in 'SmartNames'!** (by default they are all shown).

You can add a **Description** to your custom SmartNames. This quickly reminds you what they are for, without the need to open and analyse them.

Tip: The **SmartNames View** is not available when your Pilot is in **Jobs Overview** mode.

Tip: When you choose **Tools > SmartNames**, the **SmartName View** opens as a separate window.

Type - Category - Scope - Private - Tags (Overview)

We here briefly describe what these columns mean. Find more detailed descriptions on some of them in the next pages.

- **Type.** The type is a very important distinction. It defines what kind of technique the SmartName will use to get (resolve) its value. Will it get the value from a database inside Automation Engine? Or by asking the database of an external system? Or from an XML file or from another SmartName? All types are documented in [SmartName Types \(Overview\)](#) on page 923 and later in even more detail.
- **Category.** Categories are an old way of helping you filter and group SmartNames. We recommend using **Tags** instead (see further).
- **Scope.** By default the **Scope** of a SmartName is **global**. This means you can use it everywhere. Defining a different (smaller) **Scope** for a SmartName means that you want to limit the zone where

it can be used. For example: if you define the **Scope** 'Step and Repeat' then you indicate that this SmartName makes no sense to be used outside that concept of Step and Repeat. More on this in [SmartName Scopes](#) on page 926 .

- **Private.** You can save a SmartName as **Private**. Private SmartNames are not visible when applying SmartNames in a ticket. They do not appear in the list or the **SmartName Browser**. Two example use cases:
 - the SmartName is only used as a sub-component of one or more other SmartNames (that users *will* see and use)
 - you are still testing a new SmartName in the **SmartName View** and you do not want other users to already use it.
- **Tags.** Adding custom tags to a SmartName helps you filter and group them. Read more on this in [SmartName Tags](#) on page 927.

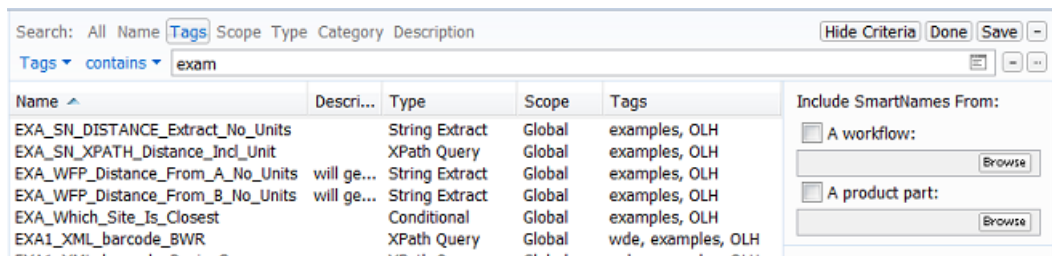
The right panel

On the right side of the **SmartNames View**, you can add more SmartNames to the list and you can test them extensively. Read more in [Testing SmartNames](#) on page 931.

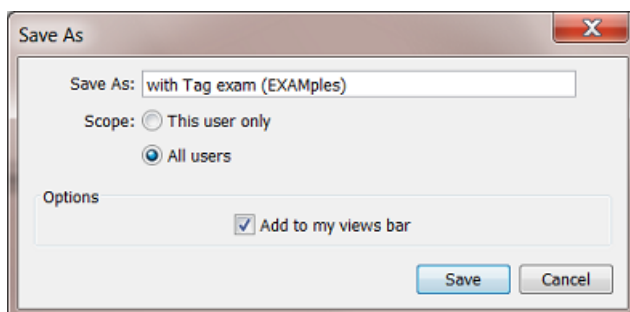
Advanced Search Criteria for SmartNames

As in many other **Views** in the **Pilot**, you can use **Advanced Search Criteria**. To activate this, type in the **Find** field or press CMD-F (Mac) or CTRL-F (Windows). You see the criteria of the current filter. Click the + or - to add or remove extra lines of criteria.

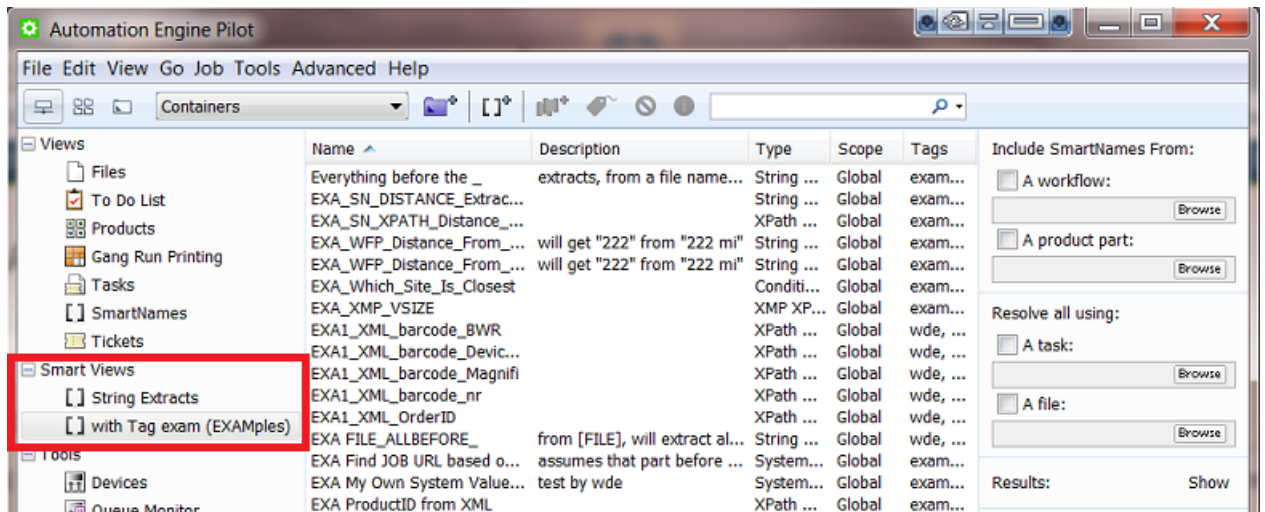
This screen shot is an example of a filter that shows all SmartNames with `exam` in their **Tag**.



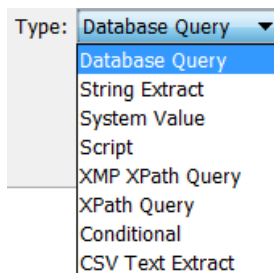
Click **Save** to save this filter. Define if this filter is for yourself or for all users. Decide if you want to **Add to my views bar**.



See an example of custom **Smart Views**:



5.2.3 SmartName Types (Overview)



Later in this document we describe each type in detail. Here is an introduction:

Database Query SmartNames

These retrieve a value from an external database. This is useful when there is no other way that the external system can send the information to Automation Engine (via XML or CSV or E-mail ...)

Read more on this type of SmartNames in [Database Query SmartNames](#) on page 968.

String Extract SmartNames

These SmartNames extract a part of a string of characters (text or numbers). For example: when the input is '123-abc', then extracting all characters before the '-' will result in '123'.

Note: The string from which the value of the SmartName is extracted can contain other SmartNames. For example: extracting the first 5 characters of the [JOB ID].

Read more on this type of SmartNames in [String Extract SmartNames](#) on page 948.

System Value SmartNames

These retrieve a value from one of Automation Engine's own databases. You get these when you install or update your Automation Engine server software. For example [Access Point E-mail Sender] (only valid in the context of an **Access Point**)

Read more on this type of SmartNames in [System \(Value\) SmartNames](#) on page 939.

Script SmartNames

These use **JavaScript** to define a value.

Read more on this type of SmartNames in [Script SmartNames](#) on page 970.

XMP Path Query SmartNames

These retrieve a value from the XMP (metadata) section of a PDF. For example when it was last modified or the PDF version number.

Read more on this type of SmartNames in [XMP path Query SmartNames](#) on page 958.

Xpath Query SmartNames

These use XPath Queries to retrieve a value from an XML. The **Xpath Builder** is a great help to construct these **Xpath expressions**.

Read more on this type of SmartNames in [Xpath Query SmartNames](#) on page 953 .

Conditional SmartNames

These return a value based on a condition. For example a SmartName 'Trapping size' has these conditions: if the workflow parameter 'Printprocess' is 'flexo', then return '0,2' and if the workflow parameter is 'digital' then return '0'.

These SmartNames are also often used to route workflows in one or the other direction.

Read more on this type of SmartNames in [Conditional SmartNames](#) on page 961.

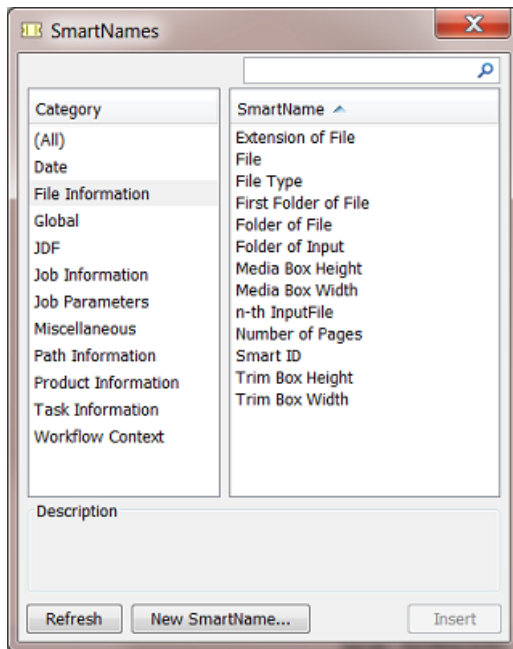
CSV Text Extract SmartNames

These extract a value from a CSV file. For example take the value from field 5 on line 2.

Read more on this type of SmartNames in [CSV Text Extract SmartNames](#) on page 951.

5.2.4 SmartName Categories

SmartNames are sorted into categories, so you can find them more easily. The **SmartName View** and the **SmartName Browser** show you these categories and their members:



Date

Use SmartNames of this category to insert the [Date] or [Time] of their execution.

File Information

These SmartNames return file-related information from within the file itself (for example [Media Box Width]) or from within the Automation Engine database ([File Type], [Folder of File], [Smart ID] ...). The most popular one is [File], representing the name of the ticket's input file.

Global

This is a full list of all the SmartNames that can be used anywhere throughout Automation Engine. The ones with a limited **Scope** will not be listed here.

JDF

These SmartNames are only used in tasks where JDF constructs are used. Read more about JDF in [What about JDF?](#) on page 1230.

Job Information

These SmartNames refer to most of the fields in a **Job** setup, including the details of **Customer** and **CSR** for that **Job**. Information on the location of the **Job** is part of the category **Path Information**. Any custom **Job Parameters** are also a separate category.

Job Parameters

When you are in a Job context of a Job that has custom **Job Parameters**, you will see them here. Their name automatically starts with 'jp. '

Miscellaneous

- Some SmartNames return the name of the Automation Engine server(s). They are useful when you use **Assistant** servers.
- [Auto-Number1] will add a sort-number to a group of output files. This helps sorting files. Find more information [here](#).

Path Information

These SmartNames correspond to the Job's location. Find more details in [System \(Value\) SmartNames](#) on page 939.

Tip: If these SmartNames are greyed out (unavailable), it means that the field you want to insert them into is not a path field.

Product Information

These SmartNames are only useful when you use the [Products](#) on page 876 tool in Automation Engine. Much like the category **Job Information**, this category relates to the many fields in the **Products** database of Automation Engine.

Read more about setting them up in [Product \(Part\) Properties](#) on page 893.

Task Information

This category of SmartNames allows to insert task related information (Task Name, Operator, Application Version...). Find more details in [System \(Value\) SmartNames](#) on page 939.

Workflow Context

These SmartNames pick up specific names or paths of folder or files in a workflow context. They help avoiding that you have to make complex workflows or create complex custom SmartNames to achieve the same result.

Find more details in [System \(Value\) SmartNames](#) on page 939.

5.2.5 SmartName Scopes

Depending on where you are in Automation Engine, different SmartNames may be available for use.

Global SmartNames

These SmartNames are available everywhere where SmartNames are offered.

SmartMarks SmartNames

These SmartNames are only available when using SmartMarks. In [Concept](#) on page 916, find an example where a SmartName was used to decide the Smart Text in a SmartMark.

Step and Repeat SmartNames

These SmartNames are **only** available in **Step and Repeat** tasks.

Mark Information

These default SmartNames are only available in the task **Add Marks to Imposition** (page workflows only).

Page Information

These default SmartNames offer information on the (PDF) page attributes, typically height and width. You can use these in the (page) imposition related tasks like the **Export Imposition to PDF File**.

Plate Information

These default SmartNames offer information on the plate attribute of a (page) imposition. You can use these in the task **Export Imposition to PDF File**.

Stripping SmartNames

These default SmartNames are only available when retrieving values from JDF files (with the **Import JDF Stripping** task).

Imposition SmartNames

These default SmartNames are only available when working with impositions (for example, in the **Export Imposition to PDF File** ticket).

Unit Text Field SmartNames

SmartName enabled unit text fields are available in tasks where units have to be entered. You can do unit conversions and simple arithmetic using the operators +, -, * and / . You can also use parenthesis () if necessary while calculating. **PowerTrapper**, **Change Imposition Layout** and **Create CAD sheet** and a few examples of tasks that offer such unit text fields.

Some examples:

[Width] inch

9 inch + 3mm

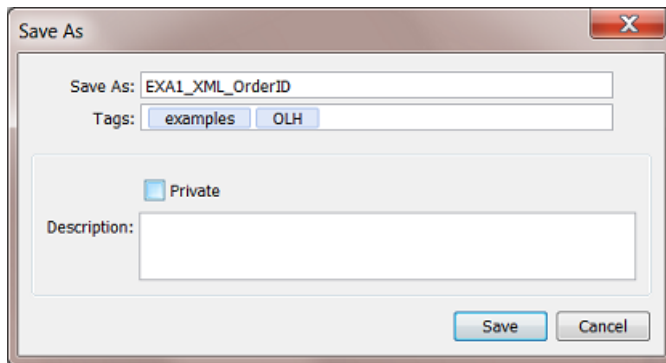
5.2.6 SmartName Tags

Creating a Tag

The **Scope** of a SmartName defines *where it can* be used. The **Category** of a SmartName helps *filtering*. Adding **Tags** to a SmartName is an extra tool that enables you to *organize them via custom filters*.

Note: Tags are also shown in a separate column in the **SmartNames View**.

You can add one or more Tags when you save a new SmartName. When the Tag already exists, it will appear in light blue:



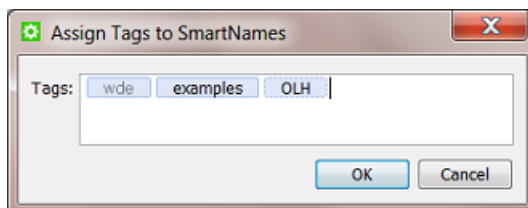
Tip: In the Tags field, press the space bar to get a drop down list of all the Tags that were already created on your Automation Engine.

To add a Tag to an existing SmartName, select it, right-click and choose **Info** to edit and add one or more Tags.

To remove a Tag from a SmartName, select it and press the **Delete** button.

Tagging multiple SmartNames

1. In the **SmartNames View**, (filter and) select the SmartNames you want to tag.
2. Right-click and choose **Tags....** A dialog will show what Tags they already have in common: Tags that are not assigned to *all* selected SmartNames are shown in grey.



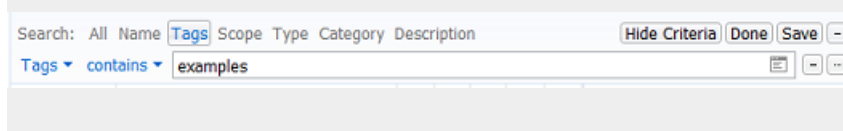
3. To add a Tag to all selected SmartNames, add the Tag and click **OK**.
4. To remove a Tag from all selected SmartNames, delete the Tag and click **OK**.

Filtering SmartNames on Tags

To filter SmartNames on Tags,

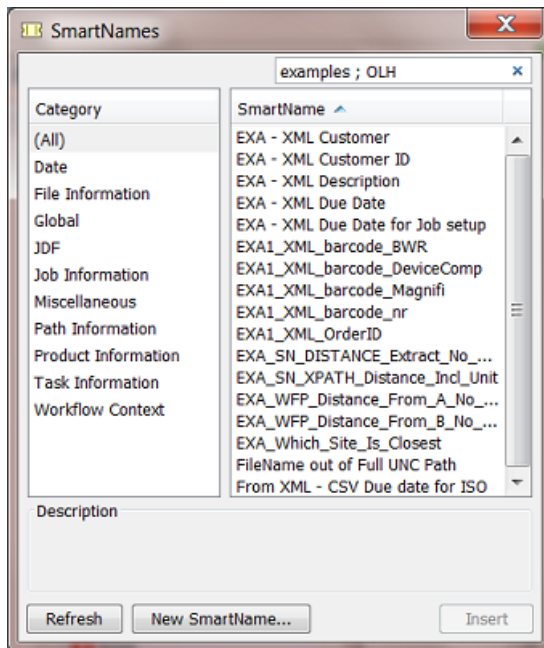
- in the **SmartNames View**, select the search field and type the name of a Tag.

Tip: It is also possible to add a search row by pressing the **Add "+"** button in the **Scope** bar, and choosing **Tags** from the attributes list (that pops up on the left).



- when selecting a SmartName from the SmartNames browser, enter the Tag in the search field.

To filter on *multiple* Tags, separate the names by a space. For example to filter on SmartNames that have *both* examples and OLH as Tag, type "examples OLH" into the search field.



To filter on SmartNames that have *either* examples *or* OLH as a Tag, then separate the Tags by a semicolon ";" (see screen shot).

Renaming a Tag

To rename a Tag, follow these steps:

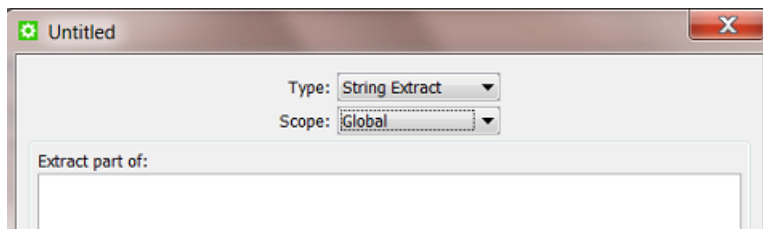
1. Go to the **SmartNames View**, filter on the **Tag**.
2. Select all SmartNames that contain that Tag.
3. Right-click and choose **Tags...**
4. Double click the Tag to edit it and type the new name.
5. Click **OK**.


5.2.7 Creating a SmartName (General)

These are the general steps to create a new SmartName. Find more detailed instructions per SmartName type further in this document.

1. In your Pilot, select **Views > SmartNames** .
2. Click on the **Create a new SmartName** icon  in the toolbar. Alternately, choose **File > New SmartName** .

This opens the SmartName setup dialog:

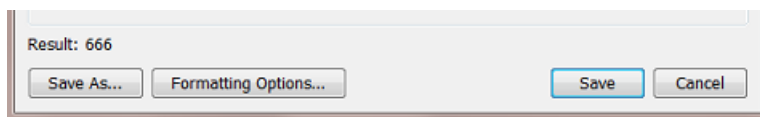


Note: Click  to select the attribute from a drop down list.

3. Select the **Type**. We introduced the available types in [SmartName Types \(Overview\)](#) on page 923. See further in this document for a detailed setup instruction per type. Depending on which **Type** you choose, the dialog adapts to offer different options.
4. Select the **Scope**. We introduced the available scopes in [SmartName Scopes](#) on page 926.

Note: You can easily change the **Scope** later ; this does not affect the setup options.

5. Set all the required attributes. This is a general introduction ; see the specific settings per type further in this document.
6. Check the **Result** value (bottom left in the dialog).

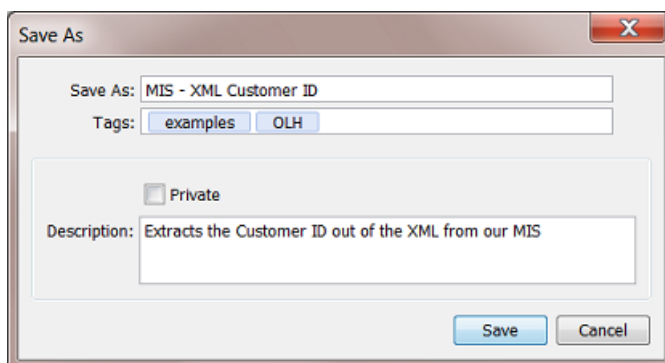


Depending on the example input data (file), you can already here see the example result with the already chosen settings.

7. **Formatting Options...** . You can still change the formatting of that result (text, number, data and time). This is explained in detail in [Formatting SmartNames](#) on page 972.

For example your SmartName already extracts a date out of an XML, but you also want to change the format of that date from 25-Aug-14 to 25-08-14.

8. Click **Save** and enter a logical **Name** and a suitable **Description**.



When you use **Save As...**, the dialog stays open. This is very useful when creating multiple variations of a SmartName one after the other.

[SmartName Tags](#) on page 927 explains all about adding **Tags**.

Decide if your SmartName should stay **Private**. Read more about Private SmartNames this in [The SmartNames View](#) on page 920.

9. Click **OK**.

You can use **Save As...** in the **Add SmartNames** dialog to enter a new name for the SmartName. This is useful when creating multiple variations of a SmartName one after the other from the same dialog.

10. Check the result of your new SmartName in more situations or tasks. Read more about this in [SmartName Relations and Dependencies](#) on page 935.

5.2.8 Testing SmartNames

While creating new SmartNames you will want to test them. Instead of having to test them in a often complex workflow, you can use tools available in the **SmartNames View**.

This part of the View contains 2 main sections:

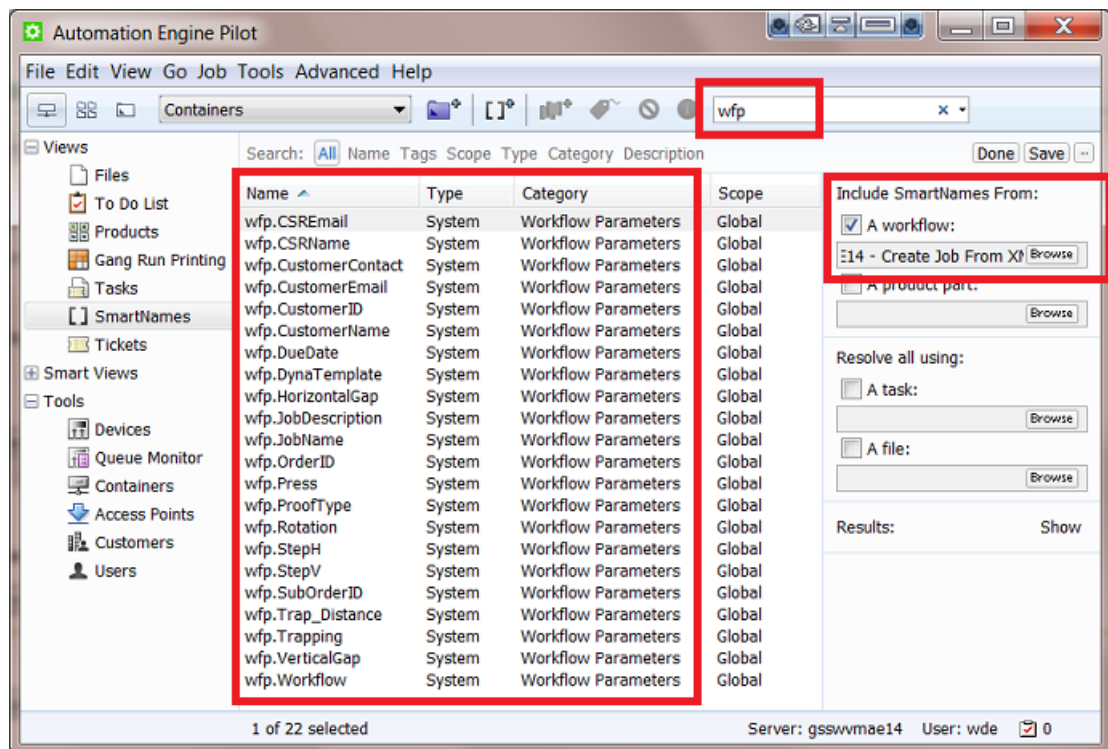
- **Include SmartNames From:** This part serves to show extra SmartNames that were not appearing in your list yet. Choose if you also want to show **Workflow Parameters** and/or **Product (Part) Parameters**. Read more in [Also Show Workflow Parameter SmartNames](#) on page 931 and [Also Show Product \(Part\) Parameter SmartNames](#) on page 932 The reason that you also want to see them here is to test them as well. The next feature enables that:
- **Resolve all using:** This part allows you to test your SmartNames. You can test them on a task or on a file. Read more in [Test your SmartNames with 'Resolve All Using'](#) on page 933.

Also Show Workflow Parameter SmartNames

Follow these steps to show extra **Workflow Parameters** in your list of SmartNames:

1. In **Include SmartNames From**, check **A workflow** and browse to the workflow that has the **Workflow Parameters** you want to see here. They will immediately appear in your list of SmartNames (unless you have a filter setting that prevents that).
2. Tip: When your list shows many SmartNames already, it is possible you do not easily see them appear. Because they are here all shown starting with the prefix **wfp**, you can easily check their

presence by typing a name filter 'wfp'. Alternatively, you could also use the **Category** filter **Workflow Parameters**.

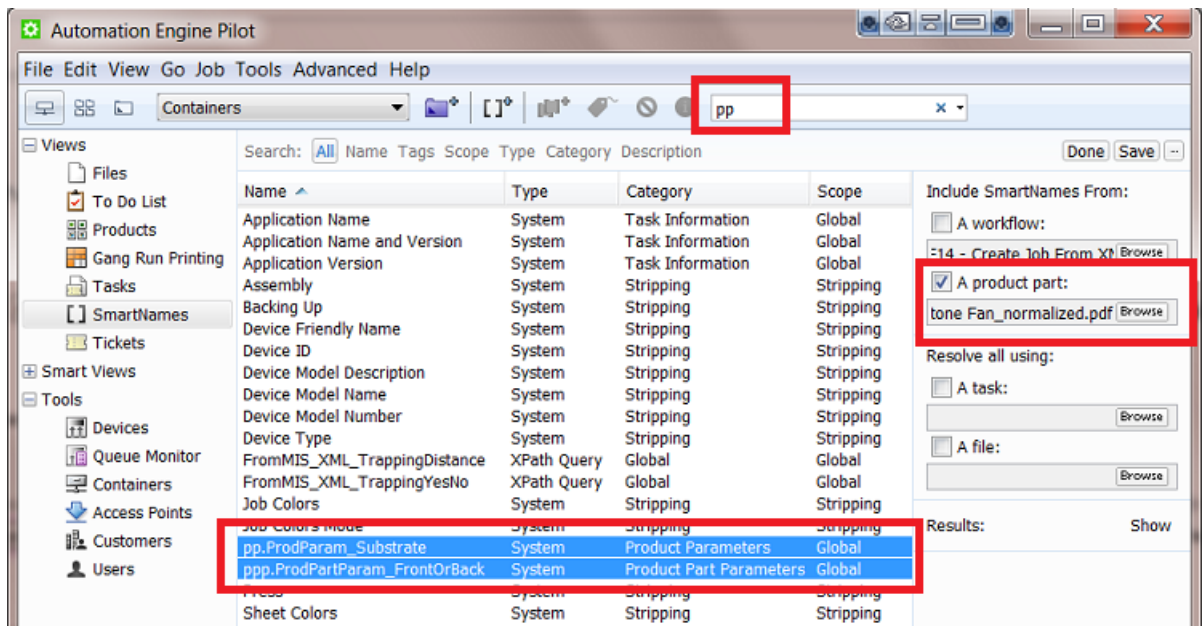


Also Show Product (Part) Parameter SmartNames

Note: This feature is only useful when you are using Automation Engine's [Products](#) on page 876 tool.

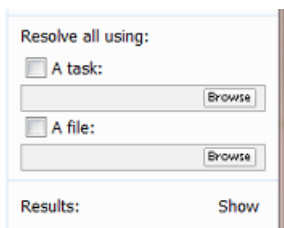
Follow these steps to show extra **Product (Part) Parameters** in your list of SmartNames:

1. In **Include SmartNames From**, check **A product part:** and browse to the product file that has the Product (Part) Parameters that you want to see here. They will immediately appear in your list of SmartNames (unless you have a filter setting that prevents that).
2. Tip: When your list shows many SmartNames already, it can be hard to find them back in the list. Because they are here all shown starting with the prefix **pp**, you can easily see them appear by typing a Name filter 'pp' or 'ppp'. Alternatively, you could also use the Category filter **Product (Part) Parameters**.



Test your SmartNames with 'Resolve All Using'

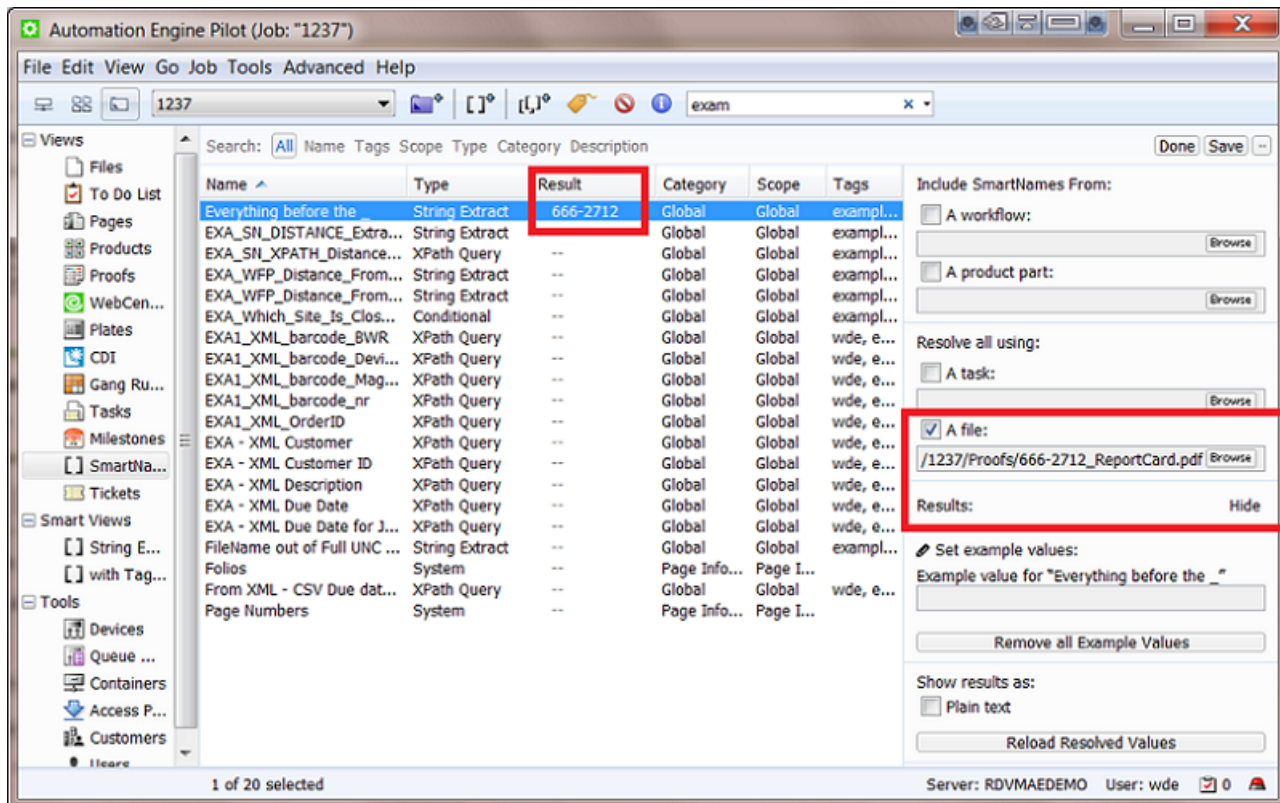
The **Resolve All Using** tool in the **SmartNames View** enables testing your SmartNames. The basic principle is that you give an example input which is then tested on *all* SmartNames.



Tip: If the SmartName you want to test is a **Workflow** or **Product (Part) Parameter**, then first make it appear in the list by using the feature **Include SmartNames from**. Read more on how to this in [Also Show Workflow Parameter SmartNames](#) on page 931 and [Also Show Product \(Part\) Parameter SmartNames](#) on page 932.

"Resolve all using:" - An example:

1. Make sure the SmartName you want to test is visible in the list.
2. If you want to test the SmartName by using it in a task, then first make sure you have a finished task given a certain input. Then check **A task** and browse to the task in your task monitor dialog. If you want to test the SmartName by using it on a file, check **A file** and browse to that file.
3. Click **Show** to start resolving all your SmartNames with the selected file or task. This also adds an extra column to your **SmartNames View**. The example in this screen shot shows how the selected file `666-2712_ReportCard.pdf` made the SmartName `[Everything before the _]'` resolved (resulted) in the `666-2712`.




Set example values

This is useful for SmartNames that have dependencies. For the selected SmartName, you can type in a manual value instead of needing to select a file or a task as input example.

Tip: Not all SmartNames allow you to enter an example value. This will for example work for most **System** type SmartNames and for the **CSV Text Extract** type SmartNames. This is because the value of these SmartNames is a "given" and can be overruled by your new example value. The value of SmartNames that are "calculated" can not be overruled ; the resolving mechanism will simply calculate them when needed.

See an example in the below screen shot.

- You made a custom SmartName `[File - only part before _]`, which is a string extract with the `_` (underscore) as separating character.
- This SmartName of course depends on the system SmartName `[File]`. If you then select the main SmartName `[File]`, you will be able to set an example value.
- In our example, we enter the value `"coco_765869"`.
 - See how this value is shown as a manually entered example value: the  icon indicates this.
 - See how the SmartNames (that depend on the SmartName `[File]`), also react and resolve a value: the SmartName `[File - only part before _]` results in `'coco'`.

Name ^	Result	Type
Assembly	?	System
Backing Up	?	System
Company Code from FSP	[Company Code from FSP]	CSV Text Extract
Design Name from FSP	[Design Name from FSP]	CSV Text Extract
Design Name from FSP (witho...	<<Design%20Name%20f...	String Extract
Design Revision from FSP	>	String Extract
Everything before the _	coco	String Extract
Extension of File		System
File	coco_765869	System
FileName out of Full UNC Path		String Extract
FileName Part Of Full Path	file://file1/file2/file.pdf	String Extract
File - only part before _	coco	String Extract
File Type	[File Type]	System
File URL	[File URL]	System
First Folder of File		System
Folder of File	>>	System
Folder of Input		System
Job File Format	PDF	System
Joint Code from FSP	[Joint Code from FSP]	CSV Text Extract
Media Box Height	[Media Box Height]	System

Include SmartNames From:

A workflow:

A product part:

Resolve all using:

A task:

A file:

/1237/Proofs/666-2712_ReportCi

Results:

Set example values:

Example value for "File"

coco_765869

Remove all Examples Values

Click this to remove all the example values from your SmartNames View.

Show results as Plain text

In some exceptional cases it can be necessary to insert a SmartName although the user interface does not offer to insert this SmartName in this place. Use this function to see the SmartName as plain text so you can then select, copy and paste it anyway in the dialog or ticket where you really want it. Test carefully, because these cases are really exceptional.

A **'plain text'** version of a SmartName looks a bit like XML code. For example [File URL] is in plain text written as <<fileurl/>> .

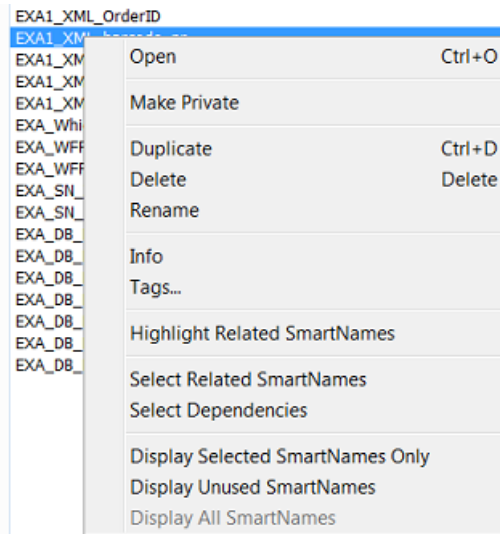
Reload Resolved Values

This 'refresh' function is useful when the input for you test has changed in the meantime.

For example you are setting up SmartNames that are reading parts of an XML file and you already did a test here. You typically want to test again after you know that the content of the XML file changed (your test will show new values).

5.2.9 SmartName Relations and Dependencies

In **SmartNames View**, when you right-click on one or on a selection of multiple SmartNames, you get these extra functions:



Tip: Most of those are also available from the **View** menu.



Attention: How SmartNames relate to each other is very important when you need to manage many of them. Such checks are very important especially when you plan to rename or delete a SmartName!

Highlight Related SmartNames

Choose this function to **highlight** all SmartNames **that need one or more** of the currently selected SmartNames: 'Need' can mean that the one you selected is based on the other(s) or it can mean that the other(s) are based on the one you selected. This highlighting is a feature that stays active until you switch it off again.

For example: see this screen shot of the ones related to the SmartName [File]:

Name ^	Result	Type
Design Name from FSP	[Design Name from FSP]	CSV Text Extract
Design Name from FSP (without Revision)	<<Design%20Name%20from%20...	String Extract
Design Revision from FSP	>	String Extract
Everything before the _	[File]	String Extract
Extension of File		System
File	[File]	System
FileName out of Full UNC Path		String Extract
FileName Part Of Full Path	file://file1/file2/file.pdf	String Extract
File - only part before _	[File]	String Extract
File Type	[File Type]	System
File URL	[File URL]	System
First Folder of File		System
Folder of File	>>	System
Folder of Input		System
Job File Format	PDF	System
Joint Code from FSP	[Joint Code from FSP]	CSV Text Extract
Media Box Height	[Media Box Height]	System
Media Box Width	[Media Box Width]	System
n-th InputFile	0	System
Number of Pages	[Number of Pages]	System
Page Range	?	System
partOfFileNameBefore-	[File]	String Extract
pemy_BOX_DFile-id	[pemy_BOX_DFile-id]	Script
pemy_BOX_DFile-name	[pemy_BOX_DFile-name]	Script
pemy_BOX_File-etag	[pemy_BOX_File-etag]	Script
pemy_BOX_File-id	[pemy_BOX_File-id]	Script
pemy_BOX_FolderInfo-json	[pemy_BOX_FolderInfo-json]	CSV Text Extract

Select Related SmartNames

Choose this function to select all SmartNames **that need one or more** of the currently selected SmartName. In this example [Fuel Cost] also gets selected because it [Fuel Cost no km] needs it (to extract the unit 'km').

Fuel Cost	--	Script
Fuel Cost no km	--	String Extract

Select Dependencies

Choose this function to select all SmartNames **that are used in** the currently selected SmartNames. If you select the SmartName [Fuel Cost no km] in the example below and then activate **Select Dependencies**, the SmartName [Distance] will also get selected. This is because [Distance] is used in the definition of [Fuel Cost no km].

Name ^	Result	Type
Back	--	System
Distance	--	XPath
EXA_SN_DISTANCE_Extract_No_Units	--	String Extract
EXA_SN_XPATH_Distance_Incl_Unit	--	XPath Query
EXA_WFP_Distance_From_A_No_Units	--	String Extract
EXA_WFP_Distance_From_B_No_Units	--	String Extract
Fuel Cost	--	Script
Fuel Cost no km	--	String Extract
IWW distance to france	--	XPath

Display Selected SmartNames Only

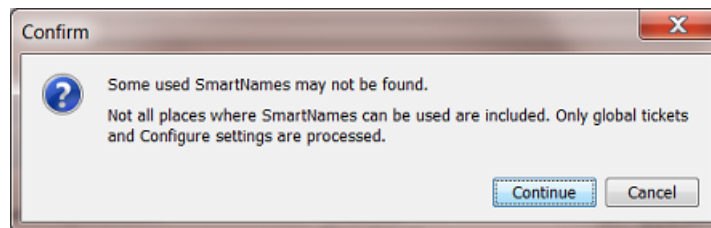
This function is no longer about selecting but about limiting the SmartNames that are displayed. This function helps when the selected ones are spread far out over your long list of SmartNames. See how the limited focus is also shown at the top.

Focused on 7 SmartNames Search: <input type="text" value="All"/>			
Name	Result	Type	Description
EXA_DB_Myorders_CustomerID	--	XPath Query	
EXA_DB_Myorders_CustomerName	--	XPath Query	
EXA_DB_Myorders_DueDate	--	XPath Query	
EXA_DB_Myorders_OrderNr	--	XPath Query	
EXA_DB_Myorders_Process	--	XPath Query	
EXA_DB_Myorders_ProofType	--	XPath Query	
EXA_DB_Myorders_Status	--	XPath Query	

When you want to see all your SmartNames again, then simply choose the function `Display All SmartNames`.

Display Unused SmartNames

When it's time to cleanup your SmartNames you will want to know which ones are not being used (any more). Choose this function to display only the unused SmartNames. You will first get this warning dialog:



Press continue to start the search. When done, decide what to do with them (delete? tag as 'unused?').

5.2.10 Exporting and Importing SmartNames

SmartNames can be exchanged by exporting and importing (workflow) tickets that use those SmartNames.

Learn more about exporting and importing (workflow) tickets in [Exchanging Workflows](#) on page 1009.



Caution: SmartNames of *custom Job Parameters* ([j.p.]) will not be imported. When you import a workflow that uses them, they will not be added into the job setup tab 'Parameters'.

Why Exchange SmartNames?

Exchanging SmartNames is typically done

- with other people (during an Esko training or to send to Esko Customer Service).
- with other servers (colleagues that use another Automation Engine server in another plant of your company).

How to Recover a SmartName that was Deleted?

Your SmartNames are a standard part of the Automation Engine backup data. When you want to restore SmartNames that were (accidentally) deleted, you can get them back by restoring a backup of that configuration. The deleted SmartName of course has to be part of that backup.

Read more about backups and how to restore them in [Server Administration](#) on page 38.

5.3 System (Value) SmartNames

We will first list the default SmartNames of this type (**System SmartNames**), then show you how to create your own (**System Value SmartNames**) and end with some examples.

5.3.1 Default System SmartNames

We here describe the **System** type SmartNames that come with an install or update of your Automation Engine server.

Remember: In **SmartNames View**, the column **Description** already describes the SmartName briefly.

Note: Default System type SmartNames can not be changed. You can not double click and open them. You can not copy them.

- **Access Point Customer, Access Point E-mail Sender, Access Point E-mail Subject, Access Point Host, Access Point Name:** these can be used within **Access Points**.
- **Access Point Path:** the path to the folder on the FTP server that will be scanned.
- **Access Point Protocol:** describes the used **File Transfer** protocol in an **FTP Access Point**.
- **Application Name** and **Application Version:** mentions the name and or the version of your Automation Engine software.
- **Assembly:** Assembly method as defined in the JDF file (`Assembly/@order`), only used in page imposition tasks.
- **Auto-Number1:** the main reason to use this SmartName is to add control over sorting the files in the workflow. This SmartName is therefore only valid when the task has multiple input files. If you choose as output file name `[File]_[Auto-Number1]` then each output file will get an (increasing) number. For example: your task input files are `banana`, `lemon` and `mango.pdf`. Then the output file names will be `banana_1.pdf`, `lemon_2.pdf` and `mango_3.pdf`.

Note: Notice that this SmartName is not related to version numbering!

- **Back:** the distance between the center of the back of the section and the trim box of the page (typically used to add collating marks to a page imposition).

- **Backing up:** the "backing up" method of the press (turn or tumble) as defined in the JDF file.
- **Book Name** and **Book Number:** name and number of the book (**ImposeProof** only).
- **CSR for the Job:** name of the customer service representative of the Job.
- **Customer's Product Reference:** name that your customer gives to the Product item.
- **Customer's Job Reference:** name that your customer gives to "his" job order.
- **Date:** the current date, formatted YYYYMMDD .
- **Device ID, Device Model Description, Device Model name, Device Model Number, Device Type:** as defined in the JDF file, used for page imposition "Stripping" tasks.
- **E-mail Address of the CSR for the Job:** the e-mail address of the customer service representative for the Job.
- **Extension of File:** the extension of the input file.
- **File:** the name of the input file without extension.
- **File Type:** the file type of the input file, for example PDF or PDFSC (= Esko Normalized PDF).
- **File URL:** the URL of the folder of the input file, the full path.
- **First Folder of File:** the name of the first folder of the input file. For example: the File URL is `file://aeserver01/ShareThatYouMadeContainer/FirstFolder/SecondFolder/banana.pdf` here, the **First Folder of File** is 'FirstFolder'.
- **Folder of File:** the name of the folder of the input file. In above example this is 'SecondFolder'.
- **Folder of First Input of Master Workflow:** name of the folder of the input file of the most outer workflow.
- **Folder of Input:** the name of the folder of the input file (same as **Folder of File**).
- **Folio:** the page name of the current page (relates to **Mark Information**).
- **Folios:** a range of page names (folios) containing prefixes and suffixes (for example ~A~,~C~--~E~,~G~).
- **Imposition Name:** the name of the imposition (page workflows).
- **Ink Name:** the name of the ink(s) (relates to **Plate Information**).
- **JDF Device ID** and **JDF Device Name:** the ID and name of the machine or software (RIP or workflow) that interprets JDF.
- **JDF Sheet Name** and **JDF Signature name:** the sheet and signature name in the JDF file.
- **Job Category ...:** the category of the Job as defined in the Job setup, maximum 7 categories.
- **Job Colors:** the numbers of colors in the job (file) as found in the JDF file.
- **Job Colors Mode:** the Job color mode ('Black&White' or 'Full') as found in the JDF file.
- **Job Container Name:** the name of the **Container** where the Job has its **Job Folder**.
- **Job Container URL:** the URL of the **Container** where the Job has its **Job Folder**.
- **Job CT URL:** the URL of the "ct" folder of the Job (= where image files can be created during normalization).

- **Job Customer Contact 1, 2, 3** : the name of the 1st, 2nd and 3rd contact of the **Customer** of the Job.
- **Job Customer Description**: the description of the **Customer** of the Job.
- **Job Customer E-mail 1, 2, 3**: the e-mail address of the 1st, 2nd and 3rd contact of the **Customer** of the Job.
- **Job Customer ID** and **Job Customer Name**: the ID and name of the **Customer** of the Job.
- **Job Customer Info 1, 2, 3, 4, 5**: the 1st etc.. info of the **Customer** of the Job.
- **Job Description**: the description of the Job.
- **Job Due Date**: the due date of the Job (YYYYMMDD).
- **Job Due Time**: the due time of the Job (HHMMSS).
- **Job File Format**: the file format of the Job (standard format is Esko normalized PDF).
- **Job ID** and **Job Name**: the ID and name of the Job.
- **Job Temp URL**: the URL of the "temp" sub folder of the Job Folder.
- **Job URL**: the URL of the Job's **Job Folder**.
- **Layer Name**: the name(s) of the layer(s).
- **Layer Names**: the names of the layers used in this sheet (page imposition workflows).
- **Layer Number**: the number(s) of the layer(s).
- **Local Server**: the name of the server where the task runs.
- **Mark Height** and **Mark Width**: height and width of the current mark.
- **Master Server**: name of the Automation Engine master server. This is the one that runs the "task manager".
- **Media Box Height** and **Media Box Width**: the height and width (mm) of the Media Box of the first page of the input file.
- **Name (without extension) of originating Input of (enclosing) Workflow**: the name of the first input of the (enclosing) workflow.
- **Name of first Input of Master Workflow**: the name of the first input of the (most outer) workflow.
- **Name of Folder of Job**: the name of the **Job Folder**.
- **Name of Folder of originating Input of (enclosing) Workflow**: the name of the *folder* of the first input of the (enclosing) Workflow.
- **Name of originating Input of (enclosing) Workflow**: the name of the first input of the (enclosing) Workflow.
- **n-th Input File**: the index number of the input file (the how many'th it is).
- **Number of Pages**: the number of pages of the input file.
- **Number of Pages (RunList)**: the number of pages in the **RunList**.
- **Operator**: the Automation Engine user that launched the task.

- **Order ID:** the Order ID of the Job.
- **Padded Page Label:** the page label padded with zeros (extra zeros in front).
- **Padded Page Number:** the page number padded with zeros (extra zeros in front). Only used in the **Split Pages** task.
- **Page Label:** the label of the page. Only used in the **Split Pages** task.
- **Page Height:** and **Page Width:** the vertical and horizontal trim size of the page.
- **Page Number:** the page number (starting from 1) or page range if the 'One Ups' contain more than one page.
- **Page Numbers:** a range of page numbers. For example: 1-5, 7, 9-11 .
- **Page Range:** the range of imposed pages available in the output file (**ImposeProof** only).
- **Paper Height** and **Paper Width:** the vertical and horizontal size of the paper (page workflows).
- **Plate Height** and **Plate Width:** the vertical and horizontal size of the plate (page workflows).
- **Plate Template Name:** the name of the plate template (page workflows).
- **Press:** the name of the press as defined in the JDF file (Device/@FriendlyName or Device/@DeviceID).
- **Print Group Name:** the name of the print group (collection of sheets) or sheet. This is used in (Black & White) page workflows.
- **Product Customer Description, Product Customer ID** and **Product Customer Name:** the description, ID and name of the customer for that **Product**.
- **Product Custom Field 1, 2, 3:** the custom field 1, 2 and 3 of the **Product**.
- **Product Description, Product ID** and **Product Name:** the description, ID and name of the **Product**.
- **Product Ordered Copies:** the amount of ordered copies that were ordered of this **Product Part** in this Job.
- **Product Part Custom Field 1, 2, 3:** the custom field 1, 2 and 3 for the **Product Part**.
- **Product Part Data Zone:** the URL of the **Data Zone** of the **Product Part**.
- **Product Part Name:** the name of the **Product Part**.
- **Product Part Status:** the status of the **Product Part** (the Esko system name).
- **Project ID:** the Project ID of the Job.
- **RunList Name** and **RunList Number:** the name of the RunList and its number in the imposition.
- **Section Number:** the number of the section, starting from 1 (this is related to **Marks**).
- **Server:** the name of Automation Engine's (**Master**) Server.
- **Sheet Colors** and **Sheet Colors Mode:** the number of colors in the sheet and their color mode ('Black&White' or 'Full'), as found in the JDF file (page workflows).
- **Sheet Descriptive Name:** a short description of the sheet as defined in the JDF file (StrippingParams/@DescriptiveName), used in page workflows.

- **Sheet Index:** the index of the sheet (starting from 0), used in page workflows.
- **Sheet Name** (scope 'Stripping'): the name of the sheet as defined in the JDF file (StrippingParams/@SheetName), used in page workflows.
- **Sheet Name** (scope 'Plate Information'): the name of the sheet, used in page workflows.
- **Sheet Number:** the number of the sheet (starting from 1), used in page workflows.
- **Sheet Side:** the side of the sheet: Front or Back (used in page workflows).
- **Sheet Side Smart ID:** the **Smart ID** of the sheet side (unique identification number).
- **Short Task ID** and **Short Workflow Task ID:** the short version of the task ID number and the (most outer) workflow task ID number. For example: '5439'.
- **Signature Name:** the name of the signature as defined in the JDF file (StrippingParams/@SignatureName), used in page workflows.
- **Smart ID:** the **Smart ID** of the file (unique identification number based on last modification dates of the file and all its references).
- **Stripping Part Number:** Stripping part number as defined in the JDF file, used in page workflows.
- **Sub Order ID:** the sub order ID of the Job.
- **System Defined:** this is used when outputting .imp files (page impositions). It is a combination of the imp name, sheet number, sheet side and layer name.
- **Task ID:** the ID of the task in Automation Engine, the long version. For example: 'afbdf6ac-8d82-4eee-af4d-b2408041640f'.
- **Task Name:** the Esko system name of the task (workflow) ticket. For example the **Add SmartMarks** task is called "batchbrix.smartmarks".
- **Task Owner E-mail:** the e-mail address of the owner of the task.
- **Ticket Name:** the name of the ticket of the task (the name you gave the ticket).
- **Time:** the current time (when the SmartName is calculated), formatted HHMMSS.
- **Trim Box Height** and **Trim Box Width:** Height and width of the Trim Box (mm) of the first page of the input file.
- **URL of first Input of Master Workflow:** the URL of the first input file of the (most outer) workflow.
- **URL of Folder of Input:** the URL of the *folder* of the input file.
- **URL of Folder of originating Input of (enclosing) workflow:** the URL of the folder of the first input of the (enclosing) workflow.
- **URL of Input:** the URL of the input file.
- **URL of originating input of (enclosing) workflow:** the URL of the first input of the (enclosing) workflow.
- **Variant Name** and **Variant Number:** the name(s) and number(s) of the variant(s) (used in page imposition workflows).
- **Web** (scope 'Imposition'): this is the index of the web in the print group (used in page workflows).

- **Web** (scope 'Plate Information'): this is the number of the web (for multi-web printing in page workflows).
- **Workflow Task ID**: the ID of the (most outer) workflow task.
- **Work Method**: the working method for 'backing up' Back to Front (page workflows).
- **Work Style**: the name of the work style defined in the JDF file (StrippingParams/@WorkStyle).

5.3.2 Creating System Value SmartNames

As we introduced in [SmartName Types \(Overview\)](#) on page 923, a **System SmartName** is a query to one of Automation Engine's own databases. Think of it as a question to Automation Engine.

As you can see in [Default System SmartNames](#) on page 939, there are already a lot **Default System SmartNames**. Still, there are still some cases where it makes sense to create your own extra **System Value SmartNames**.

Concept

When creating your own **System Value SmartName**, your question can be one of these 3:

- "Give me the Job URL, the full network path to the Job's Job Folder"
- "Give me the Product Part URL, the full network path to the Product Part file"
- "Give me the Product Part Data Zone URL, the full network path to the Product Part Data Zone"

These questions usually come up in these 2 cases:


- when you receive a file and you want to store it in a Job or Product related folder
- when you want to get a file from (under) those places.

In both cases, you need to add 1 or more parameters in your question that you do know. For example: "Where is the Job Folder if I tell you that the Order ID is 12345 and the Customer ID is 67888 ?"

Note: In a perfect world, you have a perfect data structure that is very logic and it never changes. Then you would maybe not have these questions. However, it can for example happen that your data **Containers** fill up and you may not be sure on which one the Job Folder was created. Or the files that you receive need more analysis to interpret where they belong to.

Creating a System Value SmartName - Generic Steps

We here only show the basic generic steps to create such a SmartName. See further for 3 specific examples.

1. In the **SmartNames View**, click on 
2. Choose the **Type 'System Value'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
4. **Which System Value:** choose the type you are you are looking for.
A list of attributes will appear. These will help to find what you want.
5. Check the attribute(s) **that you do know**, one could be enough.
6. Enter the value of each checked attribute. This will probably also be a SmartName.

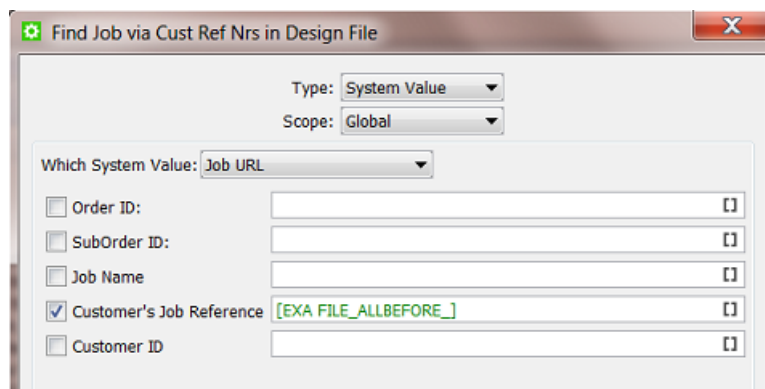
You obviously need to prepare these SmartName(s) first.

7. Check the **Result** value already shown below left. This will only be possible when you are in **SmartNames View** and are using the **Resolve all using** tool. Read more about these in [Test your SmartNames with 'Resolve All Using'](#) on page 933.
8. Click **Save as...** to save your SmartName.

Example of Looking for the Job URL

Example: Finding the Job URL by using the name of an incoming design file

- You have created a Job '1237' and you are waiting for the customers design file.
- You receive a design file via E-mail or FTP. Your customer does not know what Job name you already created. The name of the design file is '78999_Banana.pdf'. This name starts with how the *customer* names this job/design: the **Customer 's Job Reference**, followed by a free name reflecting the design variant (here the *banana* flavor).
- The setup of your Job '1237' also includes this **Customer 's Job reference** '78999' (this information came in earlier via your business system).
- You created a SmartName 'EXA FILE_ALLBEFORE_', a **String Extract SmartName** that extracts from a file name all characters before the '_'. Read more about such SmartNames in [String Extract SmartNames](#) on page 948.
- Your **System Value SmartName** SmartName now needs the settings as in this screen shot:

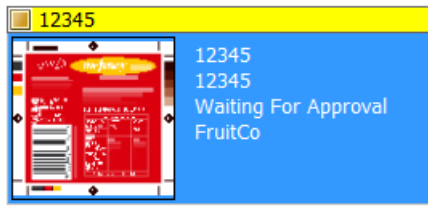


- Save your SmartName with a logical name. For example 'Find Job via Cust Ref Nrs in Design File'.

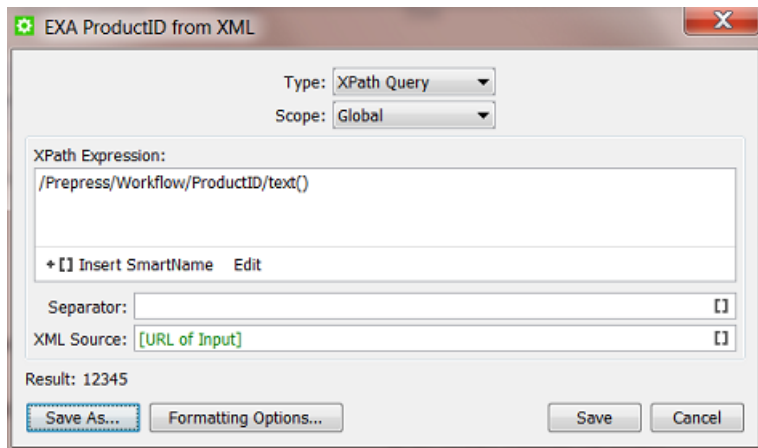
Example of Looking for the Product Part URL

Example: Finding the full Product Part URL by asking the Product ID

- Your business system sends you an XML with a list of product items on which you need to launch a workflow.
- Business systems often do not know the details of your prepress data structure. The XML mentions the **Product ID** but does not know the full file URL to that **Product Part**. So you need to find out that URL.
- For example, we look for the full URL of the Product ID '12345'. What the business system does not know is the folder and the name of the PDF. The PDF name is 'tomato_juice.pdf', it has no reference to the **Product ID**.

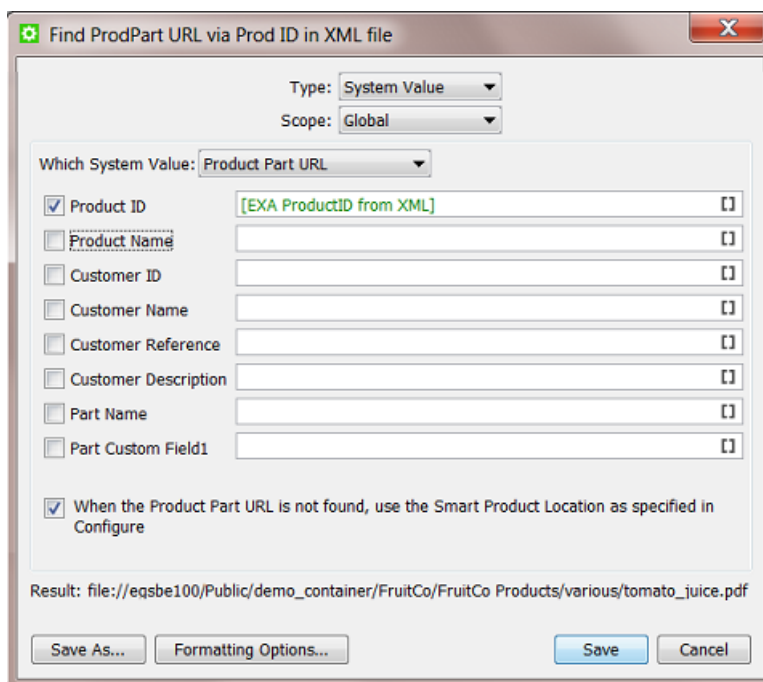


- You first need to create a SmartName that extracts the **Product ID** from the XML. See this example:



Learn more about extracting values from an XML file in [XPath Query SmartNames](#) on page 953.

- Your **System Value SmartName** then uses that **Xpath Query SmartName**:



See (below left) how the **Result** shows the complete URL to the PDF. This is what we were looking for.

- Save your SmartName with a logical name. For example 'Find ProdPart URL via Prod ID in XML file'.

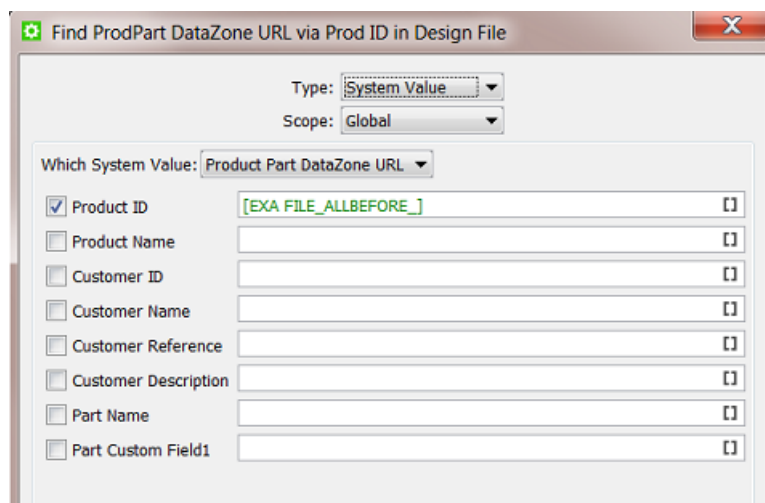
Example of Looking for the Product Part Data Zone URL

Example: Finding the Product Part Data Zone URL by using the name of an incoming design file

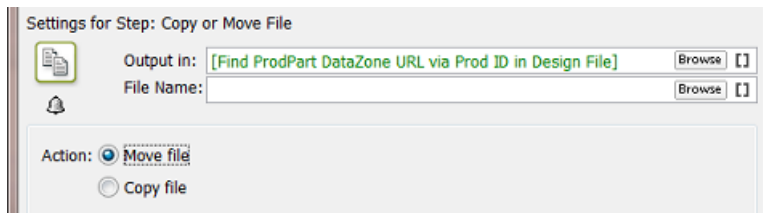
- You have a workflow where your business system creates new **Products** on your Automation Engine, even *before* the design file has arrived.
- One of those is the **Product** '6662712_Yolo_Juice' that is waiting for its design (and has no thumbnail display yet). The number before the underscore is the product ID.



- You have a **Folder Access Point** (or E-mail or FTP or WebCenter) that you scan for incoming designs.
- When a design comes in, you first want to copy it to the right destination: to its **Product Part Data Zone**.
- You need a **System Value SmartName** as in this screen shot:



- Save your SmartName with a logical name. For example 'Find ProdPart DataZone URL via ProdID in Design File'.
- Your **Copy or Move File** ticket now uses this SmartName:



- You probably want to continue your workflow with the **Manage Product Status** task to change the status of this **Product Part** to 'Design File Arrived'.

5.4 String Extract SmartNames

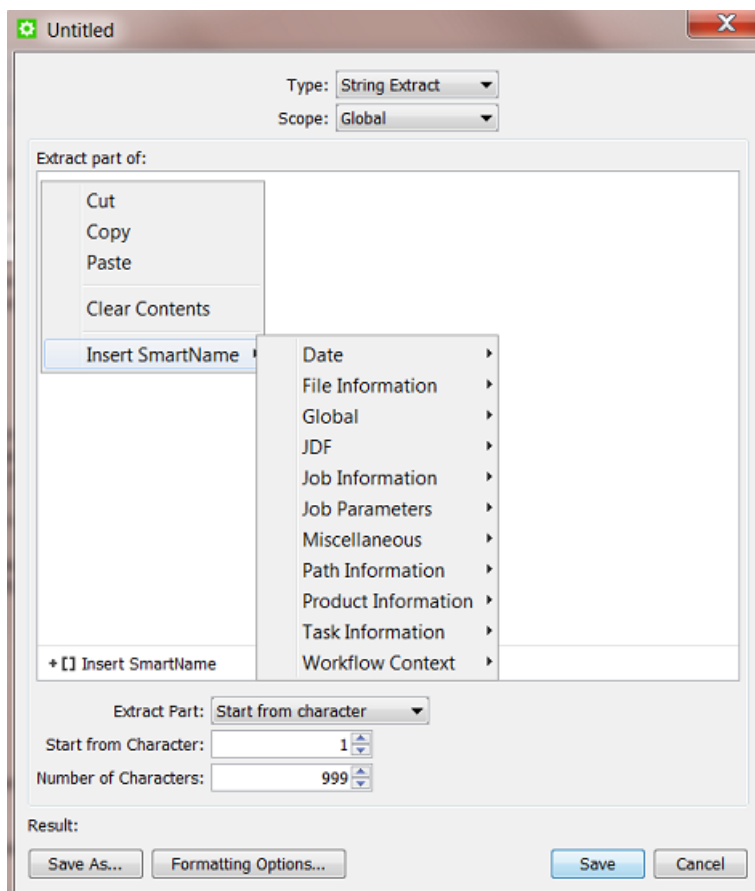
When automating your workflow, you will quickly feel the need to extract a part of a file name or a network path. **String Extract SmartNames** offer to do just this.

Note: A **String** is a series of characters. These can be numbers, text or even symbols.

5.4.1 Creating a String Extract SmartName

To create a **String Extract SmartName**, follow these steps:

1. In **SmartNames View**, click on **[]⁺**.
2. Choose the **Type 'String Extract'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
4. In **Extract part of...**, decide *from what* you will extract something. In the canvas, you can
 - type. This is mostly done to test your SmartName while creating it (you see the result below left)
 - insert **SmartNames**. This is the most typical use, for example to extract parts of a `[File]` or a `[File URL]`.
 - right-click and choose an option: **Cut**, **Copy**, **Paste**, **Clear Contents** or **Insert SmartName**.



5. Define *what* you want to extract. Choose from these 3 methods:

- **Start from character.** Learn more in [Start From Character](#) on page 949.
- **Use separation character.** Learn more in [Use Separation Character](#) on page 950.
- **Use regular expression.** A "Regular Expression" is a standard language in computer science. It enables very specific ways to extract parts of a string. Learn more in [Using Regular Expressions](#) on page 978.



Caution: Results can vary if your input string is considered as a text or as a number! This is decided in the **Formatting Options**. The default is that your string is assumed to be text. Learn more about this in [Formatting SmartNames](#) on page 972.

Note: The **Result** will update itself only after you changed a setting or clicked somewhere in a different field. Tip: press the 'Tab' button to make sure that the **Result** is re-calculated.

6. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

Start From Character

This method allows to decide the start and end character that you extract. Some examples:

Example 1: "First 5"

Your input string is '874095620banana' and you want to extract the first 5 characters.

Choose **Start from character**. For **Start from Character** choose '1' and for **Number of Characters** choose '5'. This will result in '87409'.

Note: This method is mostly used to extract the *first* characters of a file name. To extract the *last* part of a file name, you need to use **Regular Expressions**. Learn more in [Using Regular Expressions](#) on page 978.

Example 2: "All except the first 3"

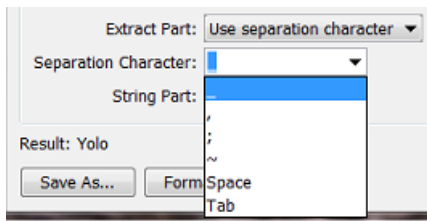
Your input string is '8740956202537435'. You want to extract all of the characters except the first 3.

Choose **Start from character**. For **Start from Character** choose '4' and for **Number of Characters** leave the default setting to '999'. This will result in '0956202537435'.

Use Separation Character

This method allows to separate the string into parts every time a certain **Separation Character** occurs, and extract one of the parts.

You can choose a **Separation Character** from the list but you **can also type in a different one**.



The **String Part** is where you choose which part of the string you want to extract.

Example 1: "Everything before the (first) underscore"

Your input string is '12345_678901' and you want to extract everything before the underscore '_'.

Choose **Separation Character** '_' and set the **String Part** to '1'.

This will result in '12345'.

Example 2: "The 3rd part"

Your input string is '87409_5620_27_923' and you want to extract the number '27' which is the 3rd part of the string (if you separate it with the '_').

Choose **Separation Character** '_' and set the **String Part** to '3'.

This will result in '27'.

Note: If you want the part after the **last** '_', then this is not possible using the **Use separation character**. To do that, you need to use the method '**Use regular expression**'. Learn more in [Using Regular Expressions](#) on page 978.

Example 3: "Everything after the '@' "

Your input string is 'Franky.Fruit@FruitCo.com' and you want to extract everything after the '@'.

In **Separation Character**, type in '@' and set the **String Part** to '2'.

This will result in 'FruitCo.com'.

Use Regular Expression ([link](#))

Regular expressions are not only available in **String Extract SmartNames**. There are documented in [Using Regular Expressions](#) on page 978.


5.5 CSV Text Extract SmartNames

A **CSV Text Extract SmartName** allows you to create a SmartName that finds a value in a specific field of a CSV file. We will first illustrate the generic steps to create such a SmartName and then show a concrete example.

Note: Although the name of this SmartName type contains the word 'text', the value that is extracted can also be a number or any other character type.

5.5.1 Creating a CSV Text Extract SmartName

To create a **CSV Text Extract SmartName**, follow these steps:

1. In **SmartNames View**, click on .
2. Choose the **Type 'CSV Text Extract'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
4. Define where your CSV can be found and set some options:
 - **CSV Source:** Define the path and name of the CSV file. You typically use SmartNames here.
 - **Text Encoding:** Indicate the **Text Encoding** of the CSV file. The default encoding is Unicode UTF-8.
 - **Field Separator:** Indicate which character is used to separate the fields in the CSV. If you're not sure, open your CSV in a simple text editor.
 - **Quote Character:** Indicate if the value in the CSV contains quotes that you do not want to keep. For example: in the CSV a value is written as "25000" but you want to extract only the 25000.
 - **Ignore line breaks:** Check this option if the CSV only contains one record (column).
 - **Ignore first line:** Check this option if the first line of the CSV contains the names of the columns. Use **Select Field...** to preview the effect of this.

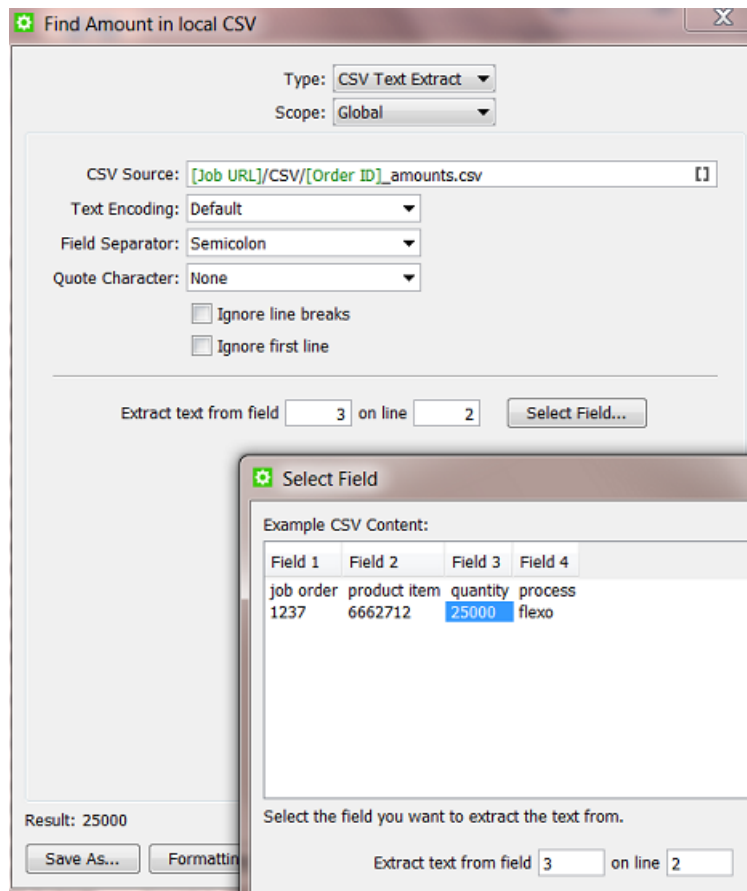
5. Define which specific field in the CSV you want to extract.
 - Type the number of the field and line in **Extract text from field ... on line ...**
 - or click on **Select Field...** to get a preview of your CSV file and there click the field you want to extract. The field and line number will then be entered automatically.

Note: A preview of your CSV file will only appear if the field **CSV Source** can already find your CSV input file.

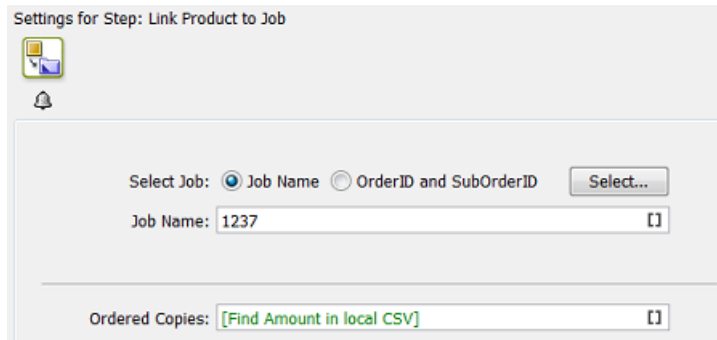
6. Check the result in **Result**.
7. If needed, set any **Formatting Options**. Learn more in [Formatting SmartNames](#) on page 972.
8. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

5.5.2 Example

- You are working with the **Products** tool and have set up some integration with your business system. You have set up a workflow where your **Jobs** are created automatically.
- You also get (from another system) a CSV file with the ordered copies (amount) that a **Product** is to be produced in that Job context. The CSV is written in the **Job Folder**, in a sub folder **CSV** and with a specific name. The screen shot shows how the **CSV Text Extract SmartName** can be set up:



- This SmartName [Find Amount in local CSV] can now be used in your automated workflow where you use the **Link Product to Job** task.



5.6 Xpath Query SmartNames

As mentioned in [SmartName Types \(Overview\)](#) on page 923 **Xpath Query SmartNames** use "XPath queries" to extract a value from an XML file. XPath, the XML Path Language, is a standard query language for selecting nodes from an XML.

In Automation Engine, there are many tasks that can read parts of an XML, often without the need to create a SmartName first.

Tip: Many of these tasks and tools are described in the chapter [Integrating with External Systems](#) on page 1228.

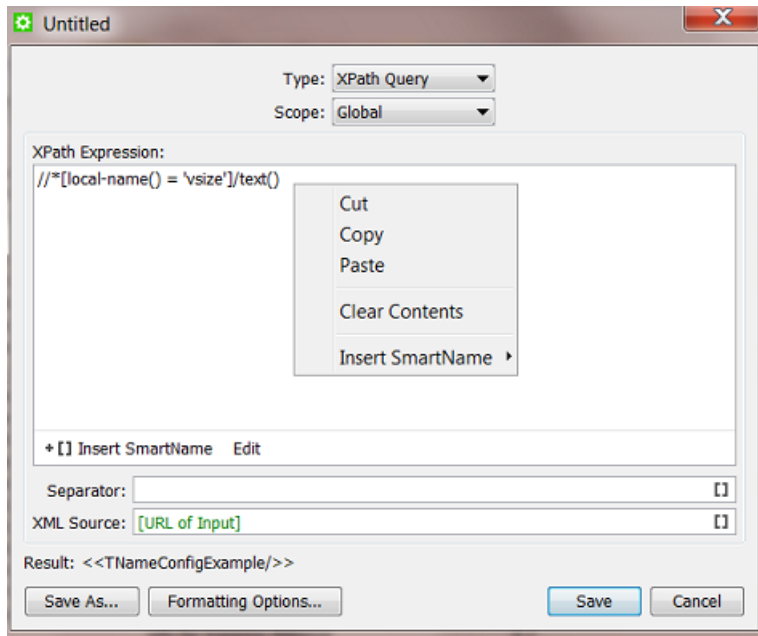
To help you build such XPath queries, **Esko** offers the [The XPath Builder](#) on page 954. This tool is available throughout Automation Engine, wherever you can create XPath queries.

5.6.1 Creating an Xpath Query SmartName

To create an **Xpath Query SmartName**, follow these steps:

1. In **SmartNames View**, click on **[]⁺**.
2. Choose the **Type 'Xpath Query'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
4. The large canvas is where you define your **Xpath Expression**. Initially, there is an example expression about 'vsize'. It will be removed automatically when you have used the **Edit** button and created your own. In this canvas, you can
 - type an Xpath Expression (if you know how to do this).
 - click on **Edit** to open the **XPath Builder** and create your expression there. Learn more in [The XPath Builder](#) on page 954.

- click on **Insert SmartName** to insert SmartName(s) into your **Xpath Expression**. This is in most cases not needed.
- right-click and choose an option: **Cut**, **Copy**, **Paste**, **Clear Contents** or **Insert SmartName**.



5. **Separator:** If the **XPath Expression** returns multiple elements, decide what character you want to use as separator.
6. **XML Source:** Define the path and name of the XML. Or keep the default SmartName [URL of Input].
7. **Result:** See the result of your **XPath Expression**. This is possible when the **XML Source** can be found. The **Result** is also visible when you are in **SmartNames View** and are using the **Resolve all using** tool and have manually selected an example XML. Read more about this in [Test your SmartNames with 'Resolve All Using'](#) on page 933.
8. **Formatting Options:** Learn more in [Formatting SmartNames](#) on page 972.
9. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

5.6.2 The XPath Builder

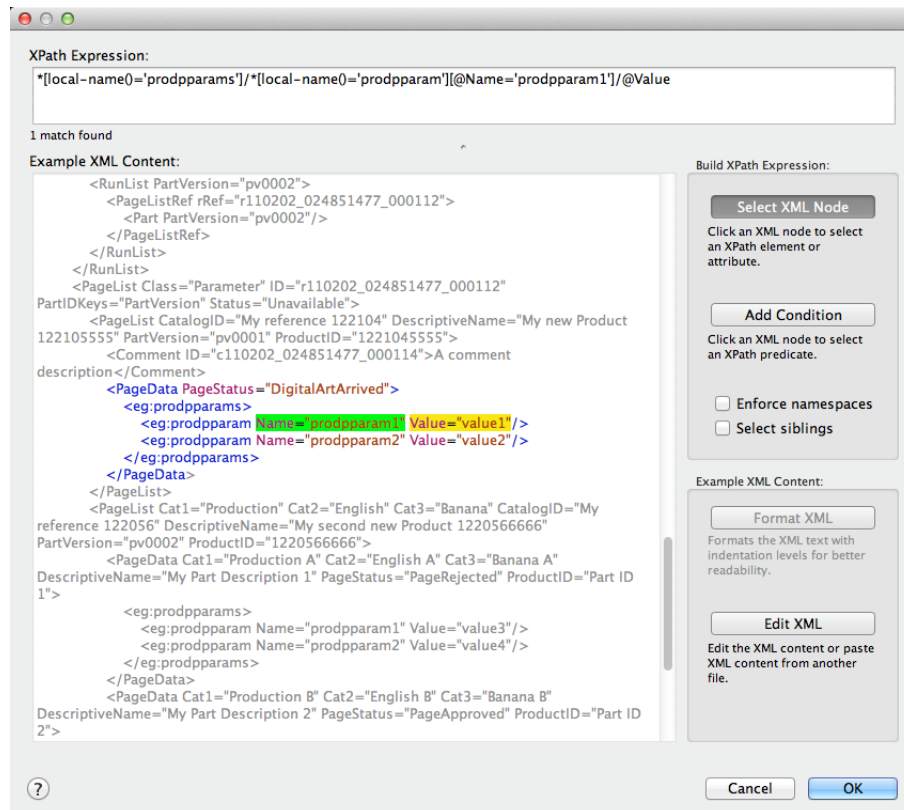
Introduction

XPath is a language that describes how to locate elements and attributes in XML (Extensible Markup Language) files. You can find a full specification on <http://www.w3.org/TR/xpath/> and a tutorial on <http://www.w3schools.com/xpath/>.

Think of an **Xpath Query SmartName** as a **String Extract SmartName** but then to get a string from an XML file.

The **XPath Builder** is an XML editor available in the **Automation Engine Pilot**. Anywhere in the Automation Engine where there is an **XPath Expression**, you can access the **XPath Builder** by clicking **Edit**.

The XPath Builder panel



- On top of the panel, you see the resulting **XPath Expression** that is probably a result of what you have clicked below.
- The large canvas shows the **Example XML Content**.
 - When possible, the example XML is automatically taken from the context. For example when you open the **XPath builder** from the **XPath Query SmartName** dialog, then the input file is used as an example XML.
 - When no example XML is available automatically, right-click in the canvas to see these extra ways to get and manage XML content: **Cut**, **Copy**, **Paste**, **Clear** and **Paste from File....**
 - Use CTRL-F (Windows) or CMD-F (Mac) to open a **Find** tool. This is very handy when your XML is a large file and you quickly want to find the string you want to select.

Note: This area has XML syntax highlighting: specific colors indicate a specific kind of node.

- On the right you have tools that help you create the **Xpath Expression** by offering smart selections and even help you edit the XML content.

Building XPath Expressions

Select XML node

To build an **XPath Expression**, make sure that the **Select XML Node** button is clicked.

Simply click on an XML node to automatically create an XPath to that node. The resulting XPath is shown in the **XPath Expression** field at the top.

Note:

- The created XPath will be an "absolute XPath" from the root node to the selected node.
- The selected node will be highlighted in yellow.
- When the selected node is used in a list, an index number will be used in the XPath. For example: `/PurchaseOrders/PurchaseOrder[1]/Address[2]/Street/text()`.
- The generated XPath takes **namespaces** into account by using the **local-name** function. For example: `/*[local-name()='PurchaseOrder'][@*[local-name()='PurchaseOrderNumber']='99503']/*[local-name()='Address'][@*[local-name()='Type']='Billing']/*[local-name()='Street']/text()`

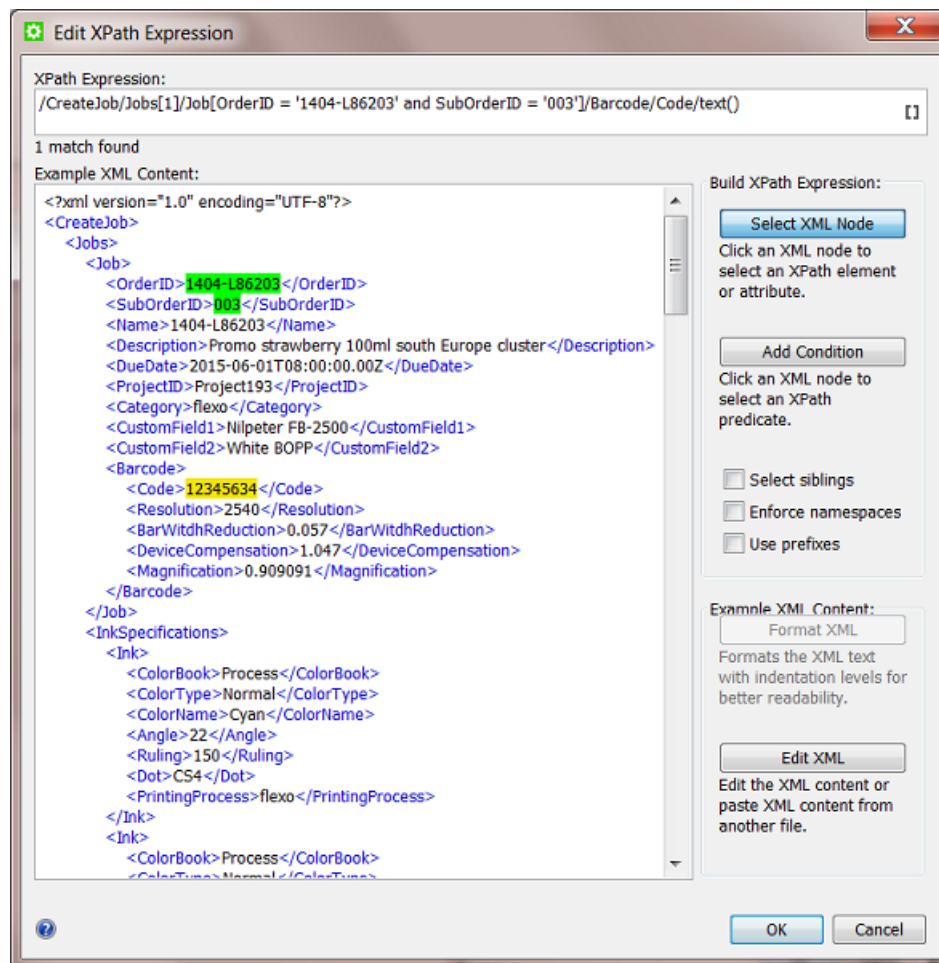
Note: By selecting **Enforce namespaces**, the XPath Builder will verify any used **namespace**.

Add Condition

To add certain conditions to the XPath, click **Add Condition**. Then click on the desired XML node to add it as a 'predicate' to the **XPath Expression**.

Note:

- You can also add a condition by clicking on the desired XML node while holding the Alt key.
- The predicate nodes are highlighted in green.
- When using predicate nodes, the indices in the XPath are replaced with predicates. For example: `/PurchaseOrders/PurchaseOrder[@PurchaseOrderNumber='99503']/Address[@Type='Billing']/Street/text()`.
- When using predicates, it is possible that the **XPath Expression** selects multiple XML nodes. The number of matches is displayed beneath the **XPath Expression**.



Options

- **Select siblings.** Select this option to select all occurrences of the selected element at this level.

Tip: This option is especially relevant when working with 'repeating content' in the [Map data task](#). You there need to **Select siblings** to avoid that only the first element would be repeated (you would so get a list of only 1).

- **Enforce namespaces.** Select this option to enforce **XML namespaces** ('xml:ns').

Tip: XML **namespaces** provide a method to avoid element name conflicts. In XML, element names are defined by the developer. This often results in a conflict when trying to mix XML files from different XML applications.

- **Use prefixes.** Check this option if you want to use **XML namespaces** in your **Xpath Expression** in a shorter way.

Note: This option overrules the option **Enforce namespaces**.

Tip: This option is especially useful when querying XMP.

Editing Example XML Content

Format XML

Under **Example XML Content**, click on **Format XML** to format the example XML. This can be used to make the XML more readable. It will typically remove unnecessary spaces and lines.

Edit XML

To edit the example XML, follow these steps:

1. Under **Example XML Content**, click on **Edit XML**.

Note: This will make the XML content editable. Also, the buttons **Revert** and **Apply** will become visible.

2. Edit the XML in any way you want.
3. When finished making changes, click **Apply**.

Note: The changes will be automatically validated. If the XML is not valid, an error message will appear and the location where the parse error occurred will be visible and highlighted in a red color.

4. Before you click **Apply**, you can click on **Revert** to undo *all* changes made to the XML content.

Note: You can undo/redo the last edits using (on Windows) Ctrl-Z and Ctrl-shift-Z or (on Mac) Command-Z and Command-Shift-Z.

Note: If you clicked **Apply** but did this in error, you can still click **Cancel**.

Note: It is also possible to edit the **XPath Expression itself** manually. When you do this, take the following into account:

- After editing the XPath, press **Enter** to validate the updated **XPath Expression**.
- If the updated **XPath Expression** is not valid, an error message will be displayed beneath the **XPath Expression**.
- The corresponding XML nodes will be highlighted in grey.

5.7 XMP path Query SmartNames

XMP stands for 'eXtensible Metadata Platform'. It allows embedding data *about* a file, known as metadata, *into* the file itself. It is a standard defined by Adobe.

Esko also uses XMP as format for metadata in its processed files (Normalized PDF, View data, LEN and other RIP output)

Note: A full specification of how Esko uses XMP is available as part of the [Automation Engine Online Help](#), Find the white paper in its section "Related Documentation".

Because XMP is actually XML embedded in a file, you can also use SmartNames to query that metadata. **XMP path Query** SmartNames are very similar to **Xpath Query** SmartNames. The main difference is that the source file is not a XML but a file with embedded XMP. The way to that metadata is also an **Xpath Expression**.

5.7.1 Creating an XMP path Query SmartName

When creating an **XMP path Query SmartName**, it is typical is that we first need to indicate our example input file, the one that contains the example XMP.

To create an **XMP path Query SmartName**, follow these steps:

1. In **SmartNames View**, in **Resolve all using**, check **A file** and browse to your (example) file that has the XMP metadata.
2. In **SmartNames View**, click on **[]⁺**.
3. Choose the **Type 'XMP path Query'**.
4. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
5. The large canvas is where you define your **Xpath Expression**. Initially, there is an example expression about 'vsize'. If 'vsize' is mentioned in the XMP of the input file you selected, it will already return a **Result** value (below left). This example expression will be removed automatically when you have used the **Edit** button and created your own. In this canvas, you can
 - type an Xpath Expression (if you know how to do this).
 - click on **Edit** to open the **XPath Builder** and create your expression there. Learn more in [The XPath Builder](#) on page 954.
 - click on **Insert SmartName** to insert SmartName(s) into your **Xpath Expression**. This is in most cases not needed.
 - right-click and choose an option: **Cut, Copy, Paste, Clear Contents** or **Insert SmartName**.
6. **Separator**: If the **XPath Expression** returns multiple elements, decide what character you want to use as separator.
7. **Result**: See the result of your **XPath Expression**.
8. **Formatting Options**: Learn more in [Formatting SmartNames](#) on page 972.
9. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

5.7.2 Examples

Remember the many System SmartNames on 'Size'

There are many SmartNames that you no longer need to create yourself via XMP queries. Remember these default (system type) SmartNames with **Scope Page Information** that may already deliver exactly what you were looking for:

Name	Description	Type	Scope ^
Folios	A range of page names (folios) containing prefixes and suff...	System	Page Information
Layer Name	The name(s) of the layer(s)	System	Page Information
Layer Number	The number(s) of the layer(s)	System	Page Information
Number of Pages (RunList)	Number of pages in the RunList	System	Page Information
Page Height	Vertical trim size of the page	System	Page Information
Page Numbers	A range of page numbers (numbers are going from 1, for e...	System	Page Information
Page Width	Horizontal trim size of the page	System	Page Information
RunList Name	Name of the RunList	System	Page Information
RunList Number	Number of the RunList in the imposition	System	Page Information
Variant Name	The name(s) of the variant(s)	System	Page Information
Variant Number	The number(s) of the variant(s)	System	Page Information

Learn more about them in [Default System SmartNames](#) on page 939.

What if my file does not have the XMP information I'm looking for?

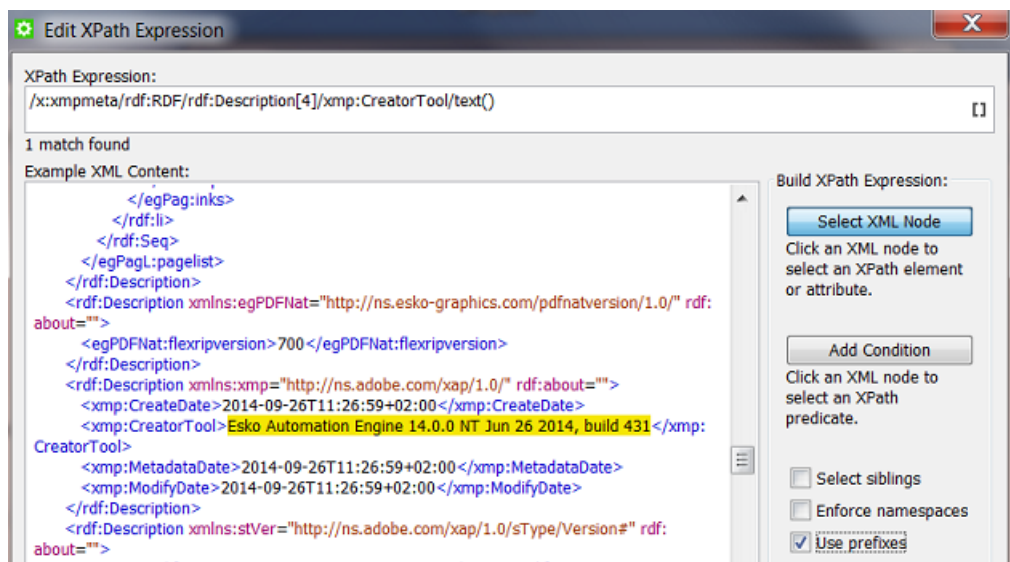
For example: you want to get the 'vsize' of a native PDF. Your workflow does not create a normalized PDF version of it (where you would automatically get that extra XMP). In that case, you can use the **LinkEdge** task to create an XML containing the same XMP as you would find inside a normalized file.

The next step is then to use an **Xpath Query SmartName** to get that specific XMP information from inside that XML file. Learn more on how to do this in [Xpath Query SmartNames](#) on page 953.

Which application created this file?

Most files with metadata will have a note about which application created the file. Follow these steps:

1. Follow the generic steps as explained in [Creating an XMP path Query SmartName](#) on page 959.
2. In the example XML, use CTRL-F or CMD-F to find for the string 'CreatorTool'.
3. Select the node. For example: see how the 'CreatorTool' is selected in the example below:

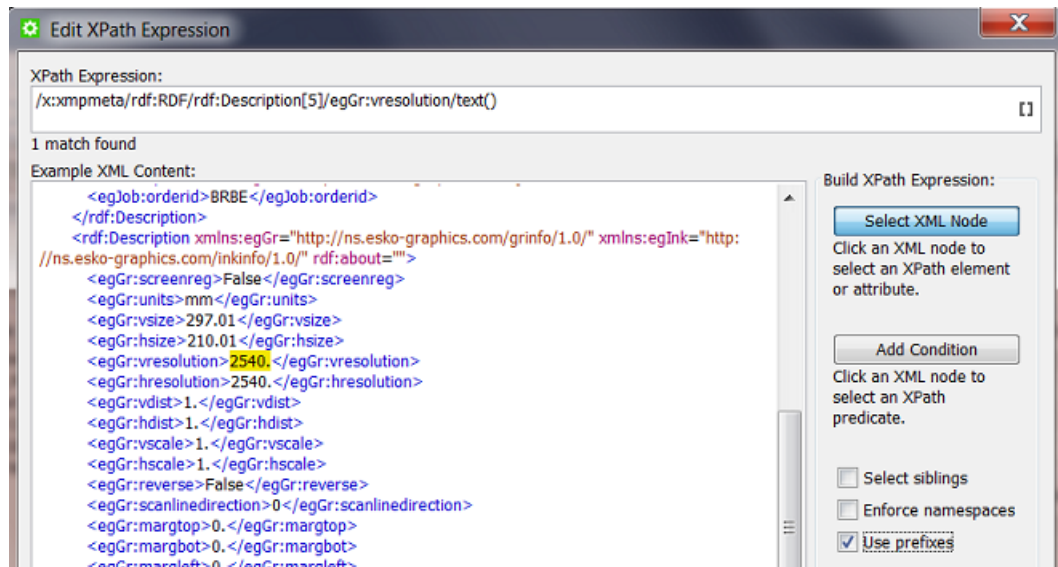


4. Click **OK**.
5. Check the **Result** and **Save** your SmartName.
6. Tip: to shorten this name, create another [String Extract SmartName](#) or a [Regular Expression](#).

Which options were used when RIP'ing this file?

It can be useful to see with what resolution or distortion your RIP file was made. There are tools to quickly detect that in Automation Engine, but your remote print site may not have those tools. It can therefore be useful to extract that value and use it in a file name or (FTP) folder name.

Follow the similar steps as in above example. See the screen shot to detect even more RIP options in the XMP of a LEN file (made with **FlexRip** or **Imaging Engine**).




5.8 Conditional SmartNames

Conditional SmartNames base their value on one or more conditions. The condition can be very basic or can be built up of several criteria and sub criteria. You can also define the **order** in which the conditions need to be met.

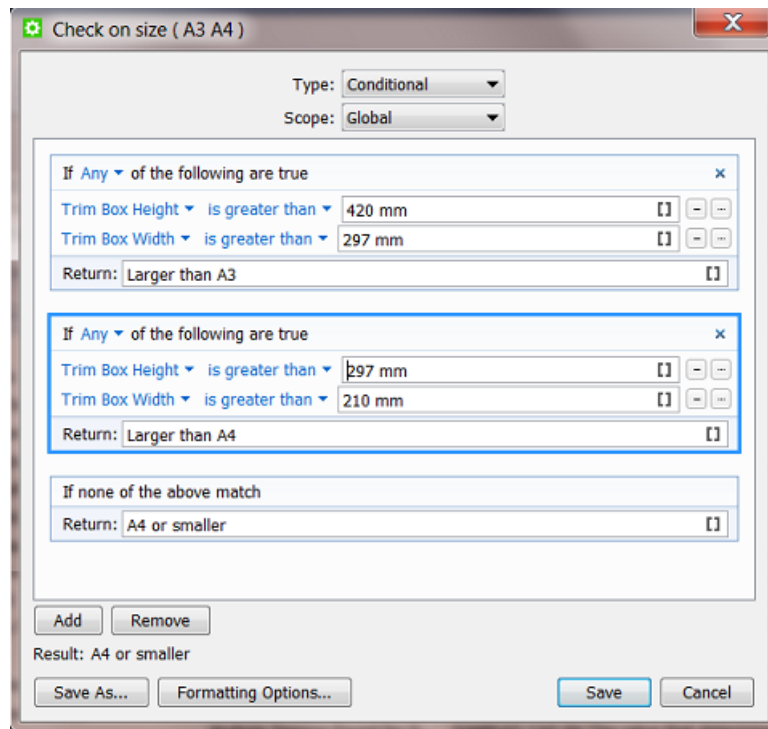
After documenting how to create a **Conditional SmartName**, we illustrate their power with some examples.

5.8.1 Creating a Conditional SmartName (generic)

To create a **Conditional SmartName**, follow these steps:

1. In **SmartNames View**, click on .
2. Choose the **Type 'Conditional'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more about [SmartName Scopes](#) on page 926.
4. The dialog initially contains 2 blocks.
 - A "**conditional block**". Each conditional block has one or more criteria and ends with the **Return** value for that block.

- You can create extra conditional blocks using the buttons **Add / Remove** (Windows) or **+ -** (Mac).
- You can also right-click in the line **If xxx of the following are true:** and **Duplicate** that block. Use the functions **Move Up / Down** or **drag and drop** the blocks to define their order.
- The order of the blocks is the order that they will be checked, **from top to bottom**.



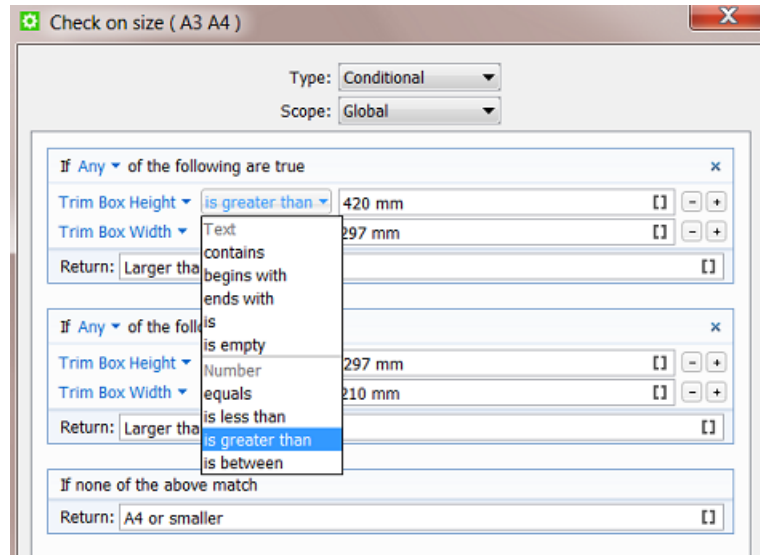
- Read more detail in [Setting up a Conditional Block](#) on page 962.
 - An "**alternative block**". When none of the above conditions match, this is what the SmartName's return value will be. Type in the value (or use another SmartName).
5. Check the result in **Result**.
 6. If needed, set any **Formatting Options**. Learn more in [Formatting SmartNames](#) on page 972.
 7. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

5.8.2 Setting up a Conditional Block

To set up a conditional block, follow these steps:

- **If All - Any - None of the following are true.** Choose the relation between the condition and your (one or more) criteria.
- Per criterion, you need to

- select the attribute from the drop down. Choose between the SmartName **File** or any **Other** SmartName.
- select the text or number expression from the drop down list. You can choose any of the following expressions from the drop down, depending on whether it is a text or numeric condition:



- For a **text** criterion, choose from the following expressions:
 - **Contains, begins with, ends with, is** and **is empty**.
Use ' ' (space) for 'AND' and ';' for 'OR'. For example, a text criterion specifying:
 - "'Inkname' **contains** 'Pantone C' ": this will resolve all ink names that contain 'Pantone' *and* all that contain 'C'.
 - "'Inkname' **is** 'C; M; Y; K; ' ": this will resolve the ink names that are specifically *any of these*.
 - Use double quotes to specify the exact text corresponding to the attribute. For example: " Inkname **contains** "reflex blue" ": this will *only* resolve for "reflex blue".
- For a **number** criterion, choose from the following expressions: **equals**, **is greater than**, **is less than**, and **is between**. Some important notes:
 - If the attribute does not correspond to a number, it will be ignored.
 - You can use standard unit symbols and unit conversions are done in background. For example: if you specify "**equals** 5 mm" then this is not a problem when the file's XMP is defined in in (inches).

Note: Unit conversions only work with units of the same kind. For example if the specification is "**equals** 5 mm" then this will fail to resolve when the XMP mentions "6 ppi" or just "6".

- **Note:** Number expressions will not resolve to text strings. For example: "**Job Name equals** MyJobName " will be ignored because **equals** is used for numbers, not text.

Note: Quotes (") and semicolons (;) are not accepted as values in the text and number conditions.

- Adding criteria: You can add a criterion to a condition by clicking the +.
- Adding **sub criteria**. When you hold `Alt` while clicking the +, your new criterion will be a sub-criteria of the one above. See [Examples](#) on page 964 where we use it to exclude some attributes that were selected in the one above.

5.8.3 Examples

Plate distortion depending on cylinder size

A **Job Category** is used to define size of the cylinder on which the product will be printed. This circumference has an effect on the distortion that is required for the plate. A **Conditional SmartName** defines the distortion setting for the RIP.

Distortion based on Cylinder Size

Type: Conditional
Scope: Global

If All of the following are true
Job Category is 480
Return: 97,995

If All of the following are true
Job Category is 500
Return: 98,075

If All of the following are true
Job Category is 520
Return: 98,149

If All of the following are true
Job Category is 540
Return: 98,217

If none of the above match
Return: Error - Cylinder type (Job Category) Not Found

Add Remove

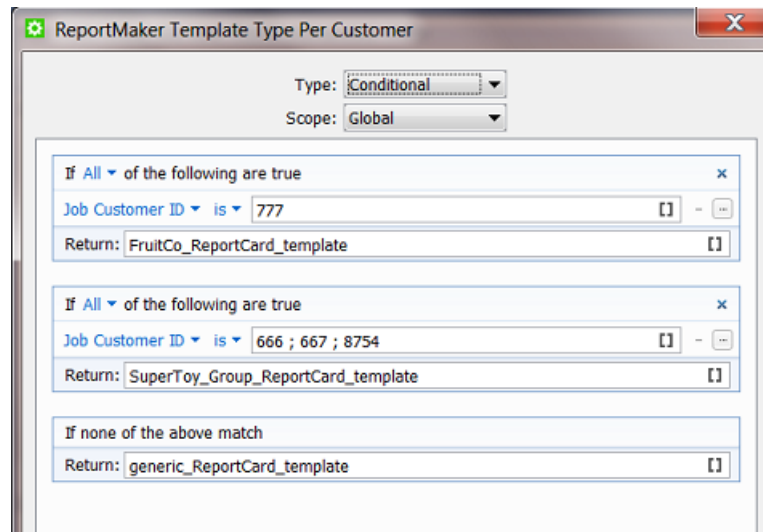
Result: Error - Cylinder type (Job Category) Not Found

Save As... Formatting Options... Save Cancel

ReportMaker Template depends on the Customer

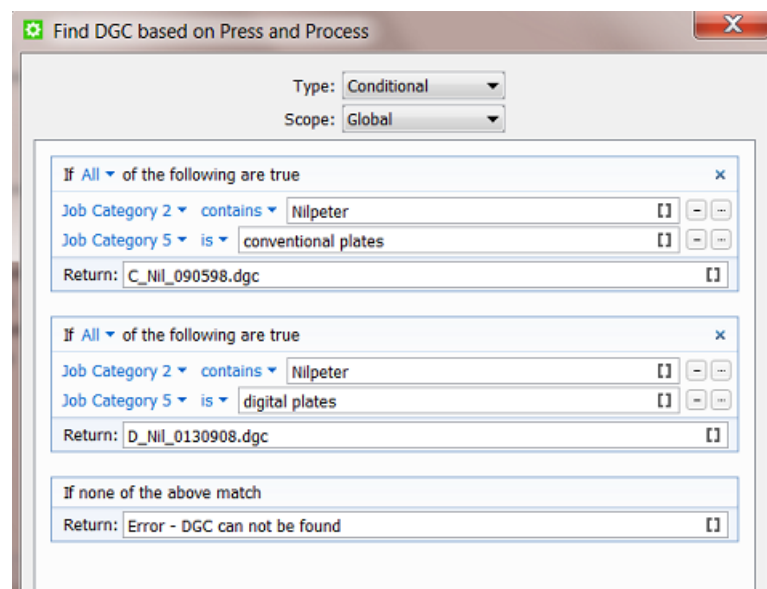
The ReportMaker ticket uses a SmartName to select its template. A conditional SmartName checks who the customer is.

- First we check if it is our biggest customer 'FruitCo' (ID 777).
- Then we check if it is any customer of the group 'SuperToy' (any of those 3 IDs).
- If it is none of the above, we use a generic template for all other customers:



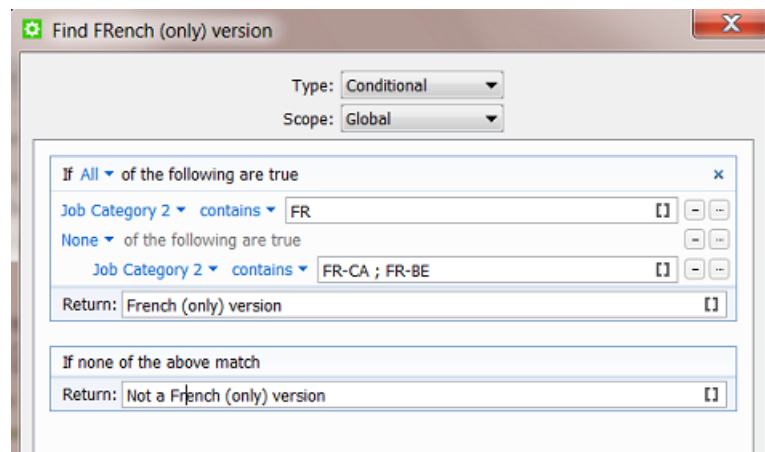
Dot Gain Compensation depending on the Press *and* type of flexo plates

If the press chosen for the Job is the flexo press Nilpeter, then the DGC file depends on the category where the type of plates is defined: conventional flexo plates or digital flexo plates (CDI):



Find the French only version (using a sub criterion to exclude some attributes)

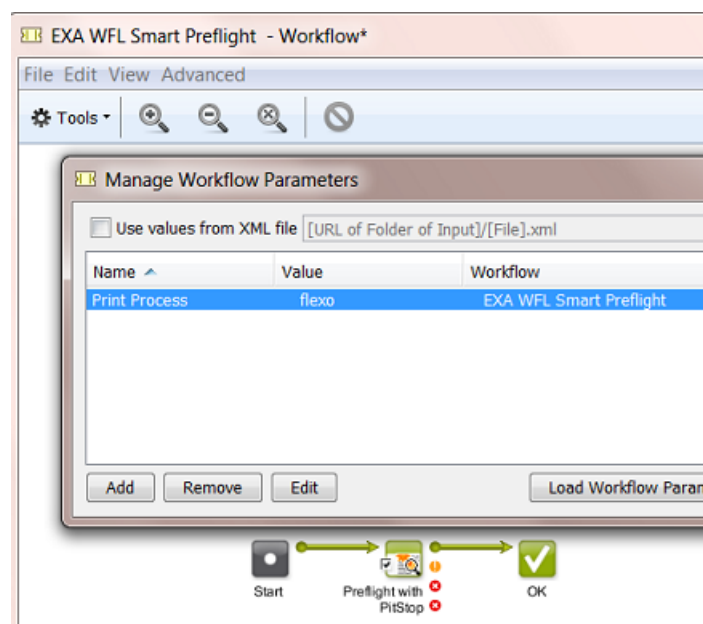
Our SmartName is looking for Jobs that are in the French language, but not in combined language versions. See how the extra sub criterion (ALT-'+') excludes the categories that contain the Canadian and Belgian version Jobs that combine French with another language:



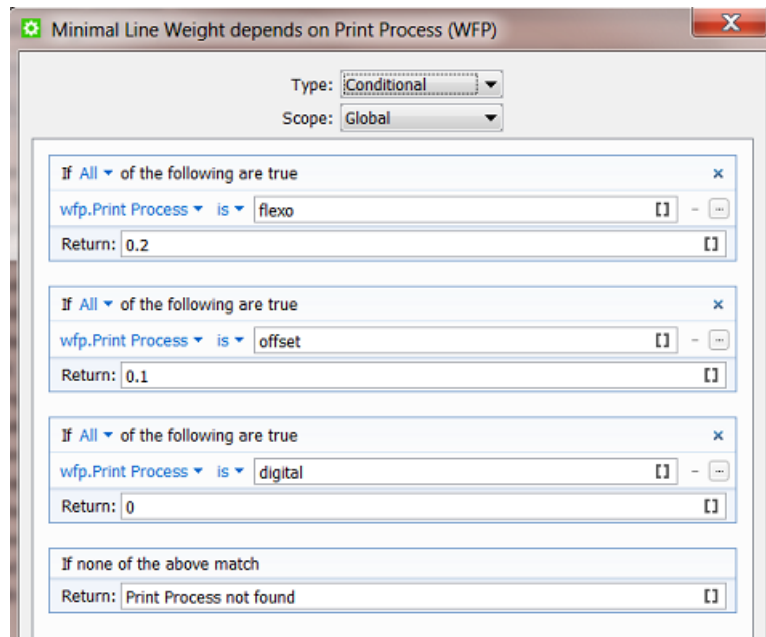
Smart preflight checking a workflow parameter 'Print Process' to check allowed line width

Our workflow includes **PitStop** preflight. It is **Smart Preflight** because it uses SmartNames. Our example shows how the **preflight profile** will check line width but have it depend on the print process that the file will be printed in. The print process is defined as a workflow parameter.

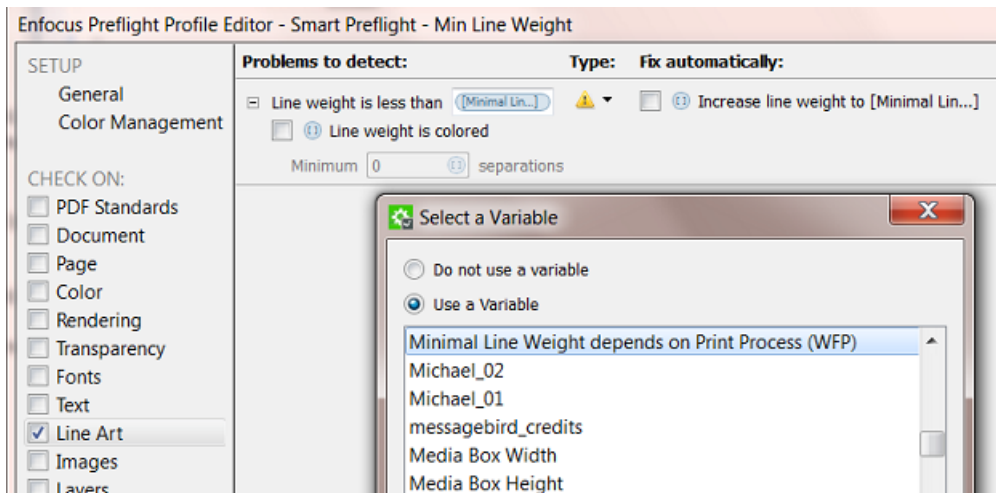
- This is our workflow parameter: [wfp.Print Process]



- This is how we set up our conditional SmartName (3 print processes each get a condition on line width (mm)):



- This is how our SmartName is used inside the PitStop Profile (there named 'variable'):



5.9 Database Query SmartNames

5.9.1 Concept

Database Query SmartNames ask a question to an external database. They pick up information from other systems, typically business systems (planning, production...). They only *ask* information, they do not write information into that external system.

Note: Database Query SmartNames may not be used to pick up information from a database from Automation Engine itself. For such use cases, you use the **System (Value) SmartNames**. Learn more in [System \(Value\) SmartNames](#) on page 939.

Some typical examples when you use Database Query SmartNames:

- In the **Tabular Step & Repeat** task. You can fill its many fields with **Database Query SmartNames** and so pick up all the required values. Narrow-web label printers often use this and base their queries on press type and die-shape.
- As a RIP option that you pick up from an external system. For example your business system that created your Job may have informed you about main production decisions like the press, the amount of plates and their size. But the distortion factor may need to be picked up from another external system closer to the press room. In that case, you use a **Database Query SmartName** in your RIP's setting for distortion.

How do Database Query SmartNames compare to the other tools that access databases?

- A **Database Access Point** also does a database query but its other main feature is that it triggers a workflow, and maybe you do not want that to happen at the moment you pick the information.
- The **Interact with Database** task can read *and* write into external databases and can be used as any step of a workflow. But it does not return you the values immediately, you get them in a separate XML file from where you need to pick them up again (then via **Xpath Query SmartNames**).

Note: In cases where that XML is *already* available, it is advised to use the information from the XML. For example: the XML that was used to create the Job, is stored in the Job Folder and it also contains all the Step & Repeat information. In that case it is advised to pick up that Step & Repeat info locally instead of asking it again to an external system (which always means network traffic and small delays).

Basically, **Database Query SmartNames** is the preferred tool when you *only* want to pick up a value and when an extra return-XML has no added value.


5.9.2 Creating a Database Query SmartName

First set up the link to that external database

Setting up a link to an external database is done in **Tools > Configure > External Databases** . Learn more on this in [Configuring links to External Databases](#).

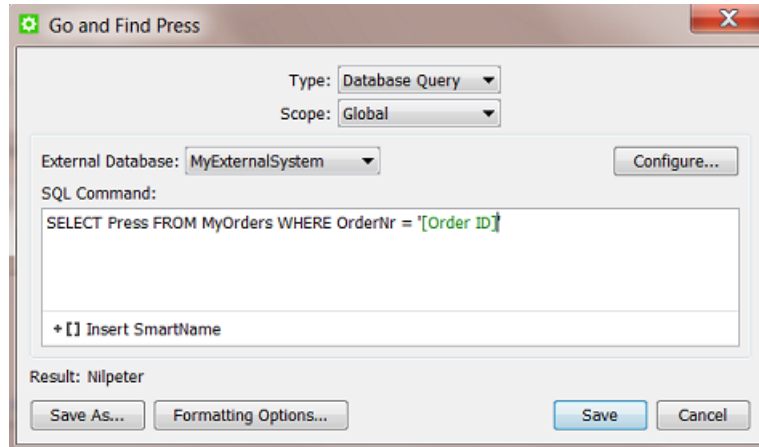
Now create the Database Query SmartName

To create a **Database Query SmartName**, follow these steps:

1. In **SmartNames View**, click on .
2. Choose the **Type 'Database Query'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more in [SmartName Scopes](#) on page 926.

4. Choose the **External Database** from the drop down list. If you need to create, change or test the link to an external database, click **Configure**.
5. Write your **SQL Command** in the canvas. You can **Insert SmartNames** into your command.

Note: Esko does not provide training in SQL commands. Contact your own IT database specialist to help set these up.



6. Check the **Result**. The result will be (re-)calculated when you click any option outside the canvas.
7. If needed, set any **Formatting Options**. Learn more in [Formatting SmartNames](#) on page 972.
8. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

Learn more about integrating with external databases

Learn more about using SQL commands in the chapter [Integrating with External Systems](#) on page 1228, in the sections

- [Database Access Point](#) on page 1245
- [Interact with Database task](#) on page 1274
- [An elaborate example](#).

5.10 Script SmartNames

Script SmartNames use **JavaScript** to get a value. Typically, other SmartNames are inserted into the script.

JavaScript is a standard scripting language. Learn more on http://en.wikipedia.org/wiki/JavaScript_syntax or on http://www.w3schools.com/js/js_syntax.asp.




Attention: Before you create a **Script SmartName**, check if it can be built using standard tools as

- a **Conditional SmartName**
- a **Regular Expression** (in a **String Extract SmartName**).

5.10.1 Creating a Script SmartName

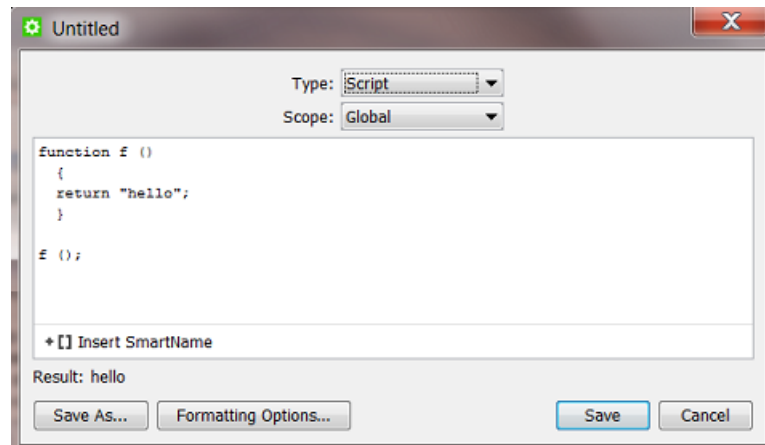
To create a **Script SmartName**, follow these steps:

1. In **SmartNames View**, click on .
2. Choose the **Type 'Script'**.
3. Choose a **Scope**. If you are not sure, choose **Global**. Learn more in [SmartName Scopes](#) on page 926.
4. The canvas initially shows a JavaScript template that returns the value "hello".



Attention: This template is important. The script needs to be a function returning a value that is called from the body of the script. Learn more in this [Esko KB article](#).

Replace it with your own JavaScript. You will probably **Insert SmartNames** into your script. See some examples in [Examples](#) on page 971.



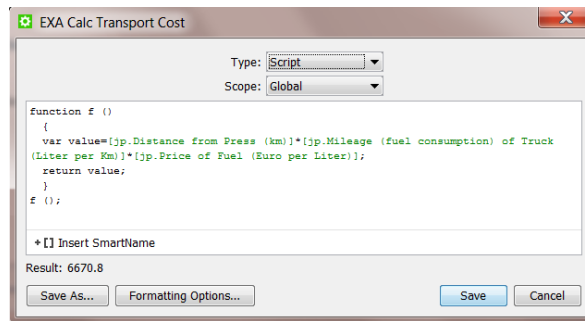
5. Check the result in **Result**. The result will be (re-)calculated when you click any option outside the canvas.
6. If needed, set any **Formatting Options**. Learn more in [Formatting SmartNames](#) on page 972.
7. Click **Save** to save your SmartName and close the dialog. Click **Save As...** to save your SmartName but keep the dialog open (to create more SmartNames).

5.10.2 Examples

The following examples are just a few basic samples of what a JavaScript can do. If you need help, Esko has specialists available through [Esko Solution Services](#) (Not Esko Customer Service).

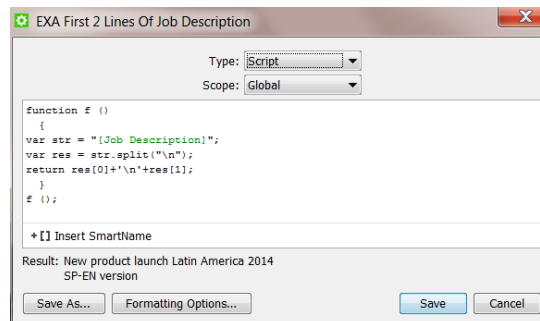
Calculating: Multiply

This JavaScript multiplies the values of 3 **Job Parameters** (number SmartNames):



Extracting a few lines from a text:

This JavaScript returns the first 2 lines of a SmartName, in this case the **Job Description**:



5.11 Formatting SmartNames

A lot of data that end up in SmartNames can come from external systems. And a lot of information that you send back out to external systems may also be decided by SmartNames. These systems can store their information in a different language, in a different regional format or even a specific custom format. That is why it may be needed to change the format of a SmartName.

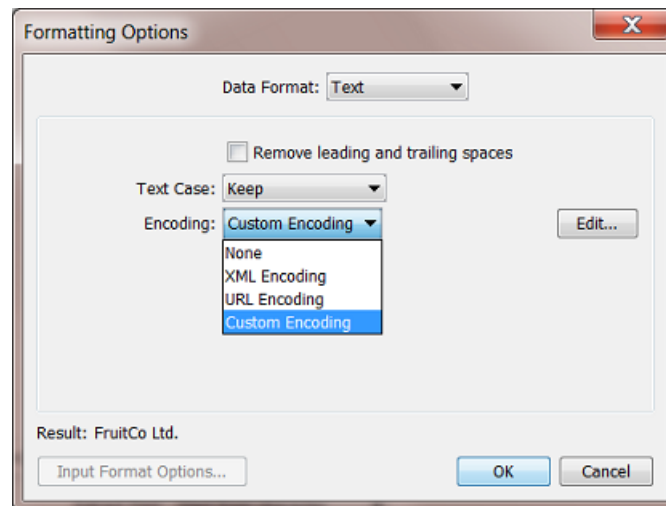
The formatting options are different whether your data is a **Text**, a **Number** or a **Date or Time**.

Note: A data string that contains both numbers *and* text characters is seen as **Text**. For example: 789_87_banana is **Text**.

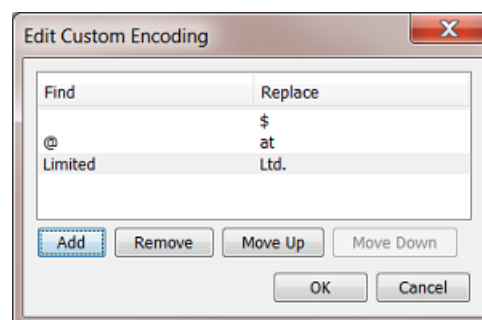
The option **Formatting Options...** is available when you create any SmartName.

5.11.1 Text Formatting

When, in **Formatting Options...**, you select the **Data Format 'Text'**, you get the following options:



- **Remove leading and trailing spaces:** If your SmartName value includes space characters at the beginning or end, use this option to remove those spaces.
- **Text Case:**
 - **Keep** the original case of the SmartName value.
 - change the text case using **Make Upper Case** or **Make Lower Case**.
- Specify any special **Encoding** of the SmartName value. Choose
 - **None** to leave it like it is.
 - **XML Encoding.** If your SmartName output will be used inside an XML file, your value can contain a character that has a special meaning in XML (a 'reserved character'), then this encoding will replace that character by a by an XML entity reference. For example '**10<20**' will be formatted to '**10<20**'.
 - **URL Encoding.** Similar to the above, this will replace the reserved characters for URLs with the typical 'percent encoding'. For example '**Digital Flexo**' will be formatted to '**Digital%20Flexo**'.
 - **Custom Encoding:** Use it to define custom replacements of characters or strings. Click **Edit** and add one or more lines of rules. The rules will be executed from top to bottom. For example:



- First all ' ' (spaces) will be replaced by a Dollar sign
- then all '@' will be replaced by 'at'

- then all words 'Limited' will be replaced by 'Ltd.'

The **Result** will update after a new setting as soon as you click in a different field.

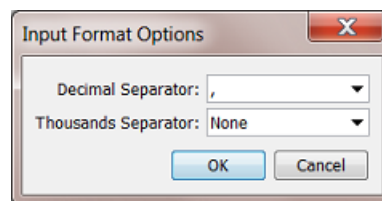
5.11.2 Number Formatting

When, in **Formatting Options...**, you select the **Data Format 'Number'**, you get the following options:

First check or define the Input Format Options.

Why first? Because this might already have an effect on the **Result** that you will be checking when you start the actual (output) formatting options.

The **Input Format Options** allow you to inform Automation Engine how your input is constructed, how it should be interpreted.

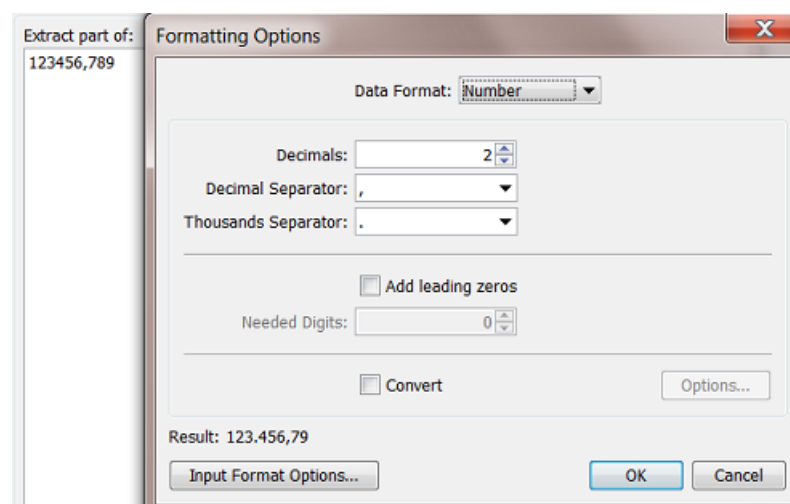


For example: Your input is '123456,789' and you want to indicate now that your **Decimal Separator** is the ',' (comma) and that there is no **Thousands Separator (None)**. You will already see a change in the **Result**.

(Output) Formatting Options

Now define how you want this **Result** formatted.

Note: If the **Result** is *already* what you want, this means that the default settings in the **Formatting Options** panel may already be the ones you want.



- **Decimals:** Indicate how many decimals you want. Select **Auto** to keep the number of decimals like it was in the original value of the SmartName. See in above screen shot how the setting 'decimals: 2' takes one decimal away and rounds the number to '79'. Another example is to round values with too detailed numbers like '67,5000000000' to shorter ones like '67,5' (one decimal left).
- **Decimal Separator:** Select the type you want. In above screen shot we confirm that the comma is the one we want (to keep).
- **Thousands Separator:** Select the type you want. See in above screen shot how it adds the point in between '3' and '4'.

Note: The decimal and thousands separators cannot be identical.

- You can choose to **Add leading zeros** to the value of the SmartName to get a certain number of **Needed Digits**. If the value of the SmartName has less digits in the integer part than the specified number of needed digits, leading zeros will be added until the number of needed digits is reached. For example, if the number of needed digits is 5, "123" becomes "00123".
- You can choose to **Convert** the value of the SmartName to another unit.
 - **Length.** Choose, in both directions, from **Millimeters, Centimeters, Decimeters, Meters, Inches, Mils** and **Points**
 - **Angle.** Choose, in both directions, from **Radians** and **Degrees**.
 - **File Size.** Choose, in both directions, from **Bytes, Kilobytes, Megabytes, Gigabytes** and **Terabytes**.
 - **Resolution.** Choose, in both directions, from **Pixels per Inch, Pixels per Millimeter, Pixels per Centimeter, Lines per Inch, Lines per Millimeter** and **Lines per Centimeter**.
 - **Time.** Choose, in both directions, from **Milliseconds, Seconds, Minutes, Hours, Days** and **Weeks**.

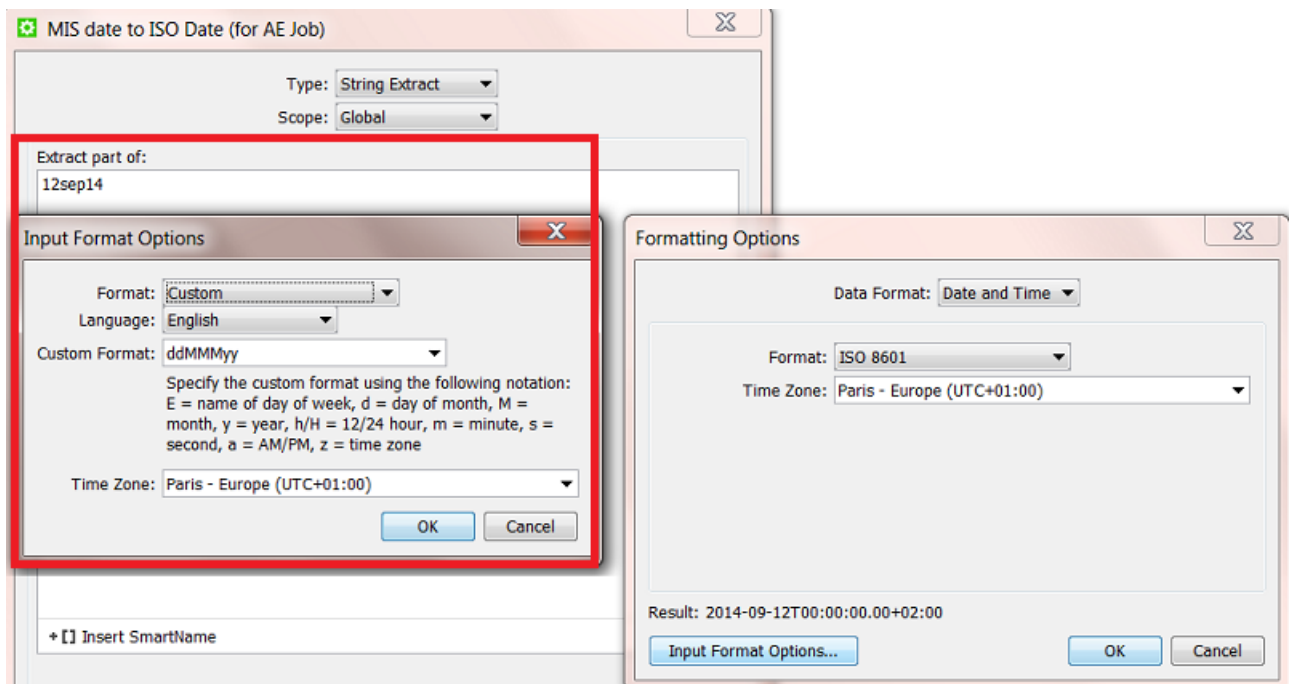
5.11.3 Date and Time Formatting

When, in **Formatting Options...**, you select the **Data Format 'Date and Time'**.

It is very important that you **first** check or define the **Input Format Options...**

Input Format Options

Date and Time descriptions appear in many different formats. Before a SmartName value can be re-formatted, it will be required to first explain Automation Engine how your input value is constructed. This is what you do in **Input Format Options**.



Format: ISO 8601

Select this when your input date and time is in this standard format. This will be the case when the date was constructed by Automation Engine. For example: your input data is '2014-09-27T16:30:00.00+02:00' (September 27 2014, 4:30 PM in the time zone UTC+2).

Format: Unix Time (in milliseconds)

Select this when your input date and time is in this standard format. Unix time is used widely between computers, it is the amount of seconds that have passed since 1 January 1970. For example: '1410532200000' (milliseconds).

Format: Custom

Select this when your input date and time is in any other format than the 2 above mentioned standards.

- **Language:** the language that you select here affects the list of default custom date and time formats in the **Custom Format** drop-down list.
- **Custom Format:** indicate here how your custom input is constructed. You can specify the custom format using the following notation: E = name of day of week, d = day of month, M = month, y = year, h/H = 12/24 hour, m = minute, s = second, a = AM/PM, z = time zone. Learn more in this [Example](#) on page 977.
- **Time Zone:** If you want, you can select the **Time Zone** of the original value of the SmartName.

Output Formatting Options

Once your input format is well understood by Automation Engine, you define the output **Formatting Options**.

You here have the same options as in [Input Format Options](#) on page 975.

Note: If your **Date and Time** serves a field in any Automation Engine database, it needs to be formatted according **ISO 8601**.

Example

You have a workflow where your MIS sends you an XML every time a new Job has to be created on Automation Engine (see an example of such workflows in [Examples of integrating with an external system](#)).

The XML from the MIS mentions the Job's **Due Date** but has its own particular way of mentioning the date and time. We will need to reformat it to the standard **ISO 8601** (that Automation Engine requires in this example).

Follow these steps to get the result as shown in the screen shot:

The screenshot displays the Automation Engine interface for configuring a date format. The main window is titled "MIS Due Date to format for AE Job Due Date - sample". It shows the following settings:

- Type: String Extract
- Scope: Global
- Extract part of: Wednesday, October 1 2014, 3:35 AM
- Data Format: Date and Time
- Result: 2014-10-01T09:35:00.00+02:00

Two dialog boxes are open, both highlighted with red boxes:

- Input Format Options:** Shows the format set to "Custom" with the custom format "EEEE, MMMM d y, h:mm a" and the time zone set to "New York - America (UTC-05:00)".
- Formatting Options:** Shows the format set to "ISO 8601" and the time zone set to "New York - America (UTC-05:00)".

Buttons for "OK", "Cancel", "Input Format Options...", and "Save" are visible.

- In the setup of the SmartName, you see our input value 'Wednesday, October 1 2014, 3:35 AM'. Normally this date would in this canvas not be written out like this but it would show up as another SmartName, one that picks up that date from the XML that we mentioned earlier.
- Click **Formatting Options...**, do not check or change any settings there yet.
- Click **Input Format Options...**, Choose **Custom** and **English**.
- In **Custom Format**, type the string that you see in the screen shot. This is what those codes mean:
 - 'EEEE': 'E' means 'day of the week'. 4 times 'E' means the day is written out in full text.
 - then there is a comma and a space
 - 'MMM': 'M' means 'month'. 4 times 'M' means the month written out in full (text).
 - then a space followed by 'd', meaning the day of the month (number)
 - then a space followed by 'y', meaning the year, in (full) number
 - then again a comma and a space
 - then 'h:mm', meaning the hour (in 12 hour system), followed by a colon, followed by the minutes
 - then a space and 'a', meaning 'AM'.
- Indicate that this time is one from the *New York* time zone.
- Click **OK** to confirm and close the **Input Format Options** dialog.
- Now set the (output) **Formatting Options**: Choose the **ISO 8601 format** and choose the same *New York* time zone
- Check the **Result**: you should see a ISO 8601 format of the custom description that we received.
- **Save** the SmartName with a logical name.
- This SmartName (date) is one that we can now use in the **Create Job** task, for the **Job Due date**.

5.12 Using Regular Expressions

Regular Expressions are text patterns that are used to match strings. They can contain a mix of plain text and special characters to indicate what to match. Find more information about this standard tool on [Wikipedia](#).

In Automation Engine, they are used for special cases where SmartNames do not offer a solution. You can use **Regular Expressions** in

- **String Extract SmartNames**
- Some tasks: the **Select File, Wait for Files, ...**
- Some **Workflow Controls: Data Splitter, Sort and Router**.

5.12.1 A Selection of Useful Characters in Regular Expressions

Character	Description	Example
^	Beginning of the string. ^abc will only lead to a match if the string begins with abc.	^C matches "Cyan" but doesn't match "PANTONE 120 C".

Character	Description	Example
	OR	cat dog matches "cat" or "dog"
\$	End of the string. xyz\$ will only lead to a match if the string ends with xyz.	120\$ matches "PANTONE 120" but doesn't match "PANTONE 120 C".
.	The . matches any single character.	ab.d matches "abcd" as well as "ab9d" or "ab-d".
[abc]	Brackets enclosing a set of characters indicates that any of the enclosed characters may match the target character.	[abc] matches "a", "b" or "c" but doesn't match "x" or "y".
[^abc]	The ^ inside the brackets means that <i>none</i> of the enclosed characters may match the target character.	[^abc] matches "x" or "y" but doesn't match "a", "b" or "c".
[a-zA-Z]	The - inside the brackets is used to show a range of characters.	[a-zA-Z] matches all upper and lowercase characters, [0-9] matches all digits.
\d	matches a numeric character (digit) (exactly the same as [0-9]).	\d\d matches "96" but doesn't match "5a".
\D	matches any non-numeric character (exactly the same as [^0-9]).	\D\D matches "A+" but doesn't match "A4".
\s	matches a whitespace.	\d\s matches "5 " but doesn't match "5a".
\S	matches any non-whitespace character.	\s\S matches " 6" but doesn't match " ".
\w	matches an alphabetic (word) character (exactly the same as [a-zA-Z0-9]).	\w\w matches "9d" but doesn't match "5+".
\W	matches any non-alphabetic character (exactly the same as [^a-zA-Z0-9] or [^\w]).	\W\W matches "+ " but doesn't match "5+".
?	The ? matches the character to its left 0 or 1 times.	esk\w? matches "esk" as well as "esko" but doesn't match "eskonv".
*	The * matches the character to its left 0 or more times.	c* matches "", "c", "cccc" etcetera. esk\w* matches "esk" as well as "Esko".
+	The + matches the character to its left 1 or more times.	c+ matches "c", "cccc" but not "". esk\w+ matches "esko" as well as "Esko" but doesn't match "esk".
{n}	The {n} matches the character to its left exactly n times.	a{3} matches "aaa" but doesn't match "aa" or "abc". ab{3}c matches only "abbbc".
{n,}	The {n,} matches the character to its left at least n times.	a{3,} matches "aaa" or "aaaaaa" but doesn't match "aa" or "abc".
{n,m}	The {n,m} matches the character to its left at least n times, but no more than m times.	a{3,5} matches "aaa", "aaaa" or "aaaaa" but doesn't match "aa" or "aaaaaa".
\	Escapes characters with a special meaning. The \ can precede a special character (like * or [) if you want to search for it literally.	esko\\artwork*nv matches "esko\artwork*nv". \?\$ matches "?" at the end of a line.

Some examples based on the above list

Regular Expression	Matches	Doesn't Match
<code>\d{6}-\w{5}-\d{1,3}\.pdf</code>	312454-ESKO4-1.pdf 235682-USA12-23.pdf	31245A-ESKO4-1.pdf 312456-USA-23.pdf
<code>PANTONE \d+ C\$</code>	PANTONE 120 C PANTONE 15 C	PANTONE 120 U PANTONE 15 CU
<code>[Yy].*</code>	Yes y	N no
<code>(Cyan) (Magenta) (Yellow)</code>	Cyan Yellow	Black Cyan SpotColor

5.12.2 Using Parentheses to Extract

Use () in the regular expression to define what to extract

Parentheses are used to define what you want to extract (to "capture a group").

Example:

Example input string: `lemon`

Regular expression: `le(mo)n`

Output: `mo`

What if I want to extract multiple groups?

Automation Engine does not support this. Only the first group in the string between () will be extracted.

Example:

Example input string: `lemonandmango`

Regular expression: `le(mo)na(nd)man(go)`

Output: `mo`

What if the input string already contains () ?

When the input string already contains a set of parentheses, you have to "escape" those characters, by adding a `\` in front of them.

Example:

Example input string: `ab(cd)e` We want to extract the group between the () .

Regular expression: `.*\((.*)\).*`

In human language: *something, then an open parentheses, then the group we want to extract, until there's a closing parentheses, followed by something.*

Output: cd

5.12.3 Examples from Users

These examples came from actual users. You may find more on [Esko's Knowledge Base](#).

Extracting the last character of a string

The SmartNames function [Start From Character](#) on page 949 makes it easy to extract the *first* x characters of a string. With **Regular Expressions**, you can extract the *last* character(s). See this example (you can change the number to extract more (last) characters):

Example input string: banana

Regular expression: .* (. {1})

Output: a

Extracting the server name from the SmartName 'Operator'

Example input string: Chantal@AESERVER01

Regular expression: (.+)@.+

Output: Chantal

Extracting the domain from an E-mail address

Example input string: michael.jackson@esko.com

Regular expression: .+@(.+)\.[a-z]+

Output: esko

Extracting the part of a file name after the *last* underscore

Example input string: Banana_v03_271265_p003.pdf

- with the file extension
 - Regular expression: .*_(.*)
 - Output: p003.pdf
- without the file extension
 - Regular expression: .*_(.*)\..*
 - Output: p003

Extracting the part of a file name between the 2nd and 3rd underscore

Example input string: Banana_v03_271265_p003.pdf

Regular expression: [^_]*_[^_]*_(.*)_.*

Output: 271265

Extracting a number from a file name

Example input string: Cosmo453_Advertisement_4.pdf

Convention in human language: *something_something_number.pdf*

Regular expression: `._.+_([0-9]+).pdf`

Output: 4

Defining a specific file selection - containing an 'exclude'

In the **Select File** task, you can use regular expressions in the input fields that end with **Name Matches**: by ending your input with `@regex`. This creates possibilities to define very specific selections. Some examples:

- To select all files except files where the name contains `_report` :

```
(?!.*_report).*@regex
```

- To select all files but exclude MAC files (starting with `.`) :

```
!\.).*@regex
```

- To select all ArtPro files, but exclude the automatically generated backup file:

```
(?!.* BU).*\.ap@regex
```

Defining a specific file selection

- To select the file and the report of the file:

```
<file/>.pdf|<file/>_report.pdf@regex
```

- To select only files with specific extensions. For example only the ARD and MFG files:

```
.*\.ard|.*\.mfg@regex
```

5.13 SmartNames of Parameter Values

The chapter "SmartNames" describes the many ways how to create a SmartName that will resolve in a string like "flexo" or "123". But how do you create a SmartName that resolves in **Parameter Values** that are not a free text string? For example you want a SmartName to decide a button in the interface that indicates to rotate 90 degrees. Or you want a SmartName to decide an internal value that is shown in a dropdown list, for example "Any User"?

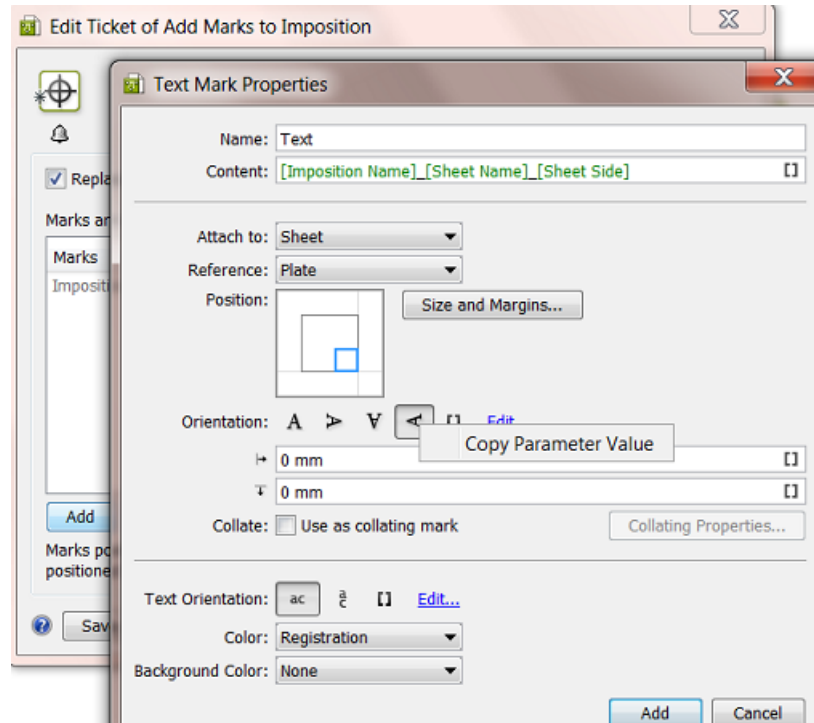
We illustrate both cases with an example:

Creating a SmartName of a User Interface Button

Some buttons in Automation Engine Tickets can be transformed into SmartNames. Follow these steps to create such a SmartName:

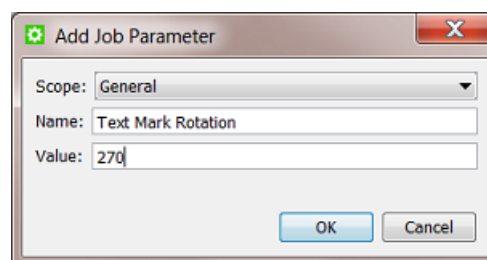
1. Right-click the button and see if the option **Copy Parameter Value** appears. If so, click the option.

For example this option is available for the **Orientation** buttons of the Marks that you can set in the **Add Marks to Imposition** task. Notice how you can copy their Parameter Value and how the list of buttons also offers to be decided by a SmartName.

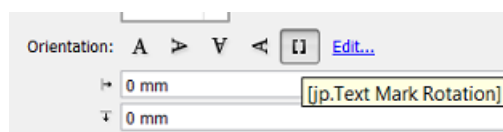


2. Go to the dialog where you create the SmartName and right-click and **Paste** the Parameter Value.

In this example we create a Job Parameter (that will become available as a SmartName). We here show the manual creation but it would normally be created automatically based on input from an external system. In the Job Setup, we create the Job Parameter **Text Mark Rotation** and for its **Value** we paste (**Ctrl-V**) the Parameter Value of the button. The screen shot shows that the resulting value is "270".



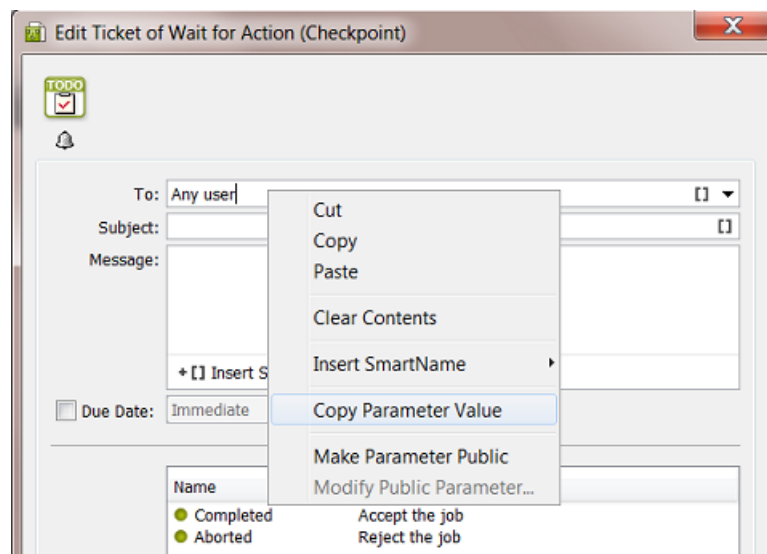
3. Now use that SmartName in the Ticket instead of the button. Click on the SmartName icon and select it.



Creating a SmartName of an Automation Engine internal value

This example shows how to create a SmartName that will return the value of "Any user" (a choice from a dropdown list):

1. Right-click the interface value and see if the option **Copy Parameter Value** appears. If so, click the option.

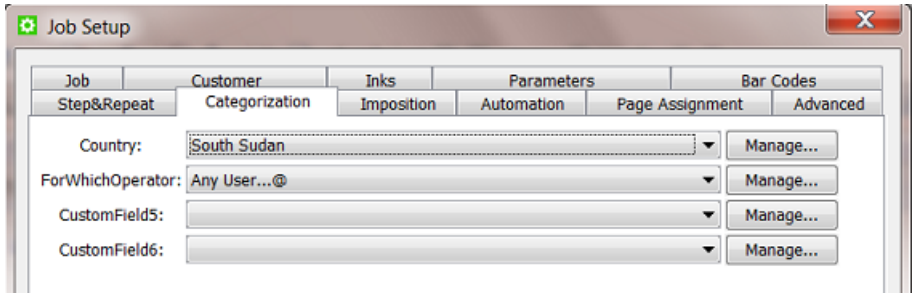


2. Go to the dialog where you are creating the SmartName and **Paste** the value there. You will see it appear as "Any User...@".

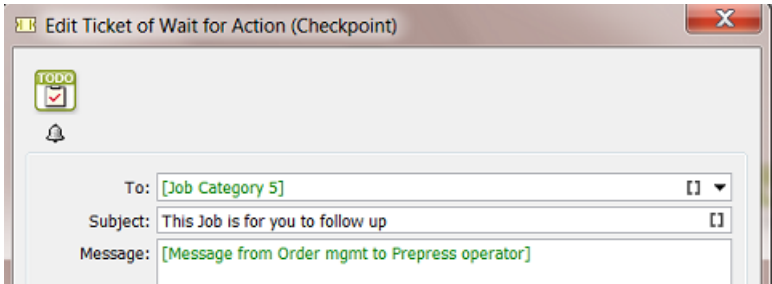
Here is an example workflow: your Jobs are created via XML files that you receive from your business system. The XML contains the name of the Esko user that should handle this Job and therefore should receive an item in his **To-Do list** (from the **Wait for Action (Checkpoint)** task that is automatically started when the Job is created). The **Create Job** task maps that user to the custom **Job Category 5**. The **Wait for Action (Checkpoint)** task then picks it up as the SmartName **Job Category 5**.

For this to work, the business system needs to know exactly how to write the Automation Engine User names. In our example, the business system also needs to learn how to tell Automation Engine that the To-Do of a Job is for **Any User**. In that case, the business system needs to mention in the XML that the Job is for "Any User...@", because only that value will match the dropdown choice **Any user**.

This screen shot shows how the **Create Job** task created the correct string in the custom Job Category named **ForWhichOperator**:



And the SmartName of this Job Category is then used in the **To:** field of the Checkpoint task:



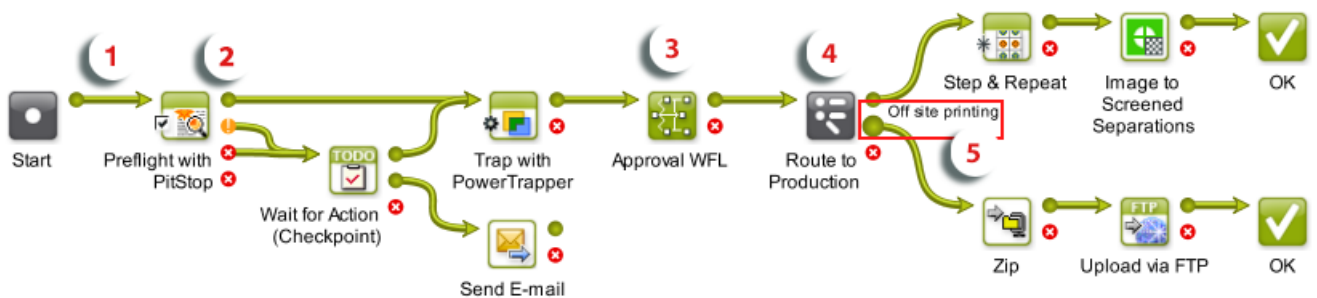
6. Workflows

6.1 Workflow Basics

Automation Engine enables you to create very powerful and automated workflows. You can customize them to do exactly what makes sense in your environment. You can make them extra smart by using SmartNames, subworkflows, routing, integration points with external systems and much more.

A step in a workflow is an Automation Engine **Task** performing an action on an input file to produce an output file. It can be also be a **Workflow Control**. Workflow controls can be about deciding which route the workflow will take or about file management (selecting, sorting etc.) or about other administrative decisions in the workflow.

Here is an example introducing some main tools:



1. A **Transition** is the green line that connects each step in the workflow. It is a graphical representation of an output file of a step being used as the input for the next step.
2. By default each step has an OK and Error **Output Pin**. When a step ends in the OK state, the output file will be sent to the step connected to the OK output pin. When a step errors out the workflow will continue with the step connected with the Error output pin. In the example workflow above the **Preflight with PitStop** task has 2 extra pins, matching the possible Preflight statuses.
3. This **subworkflow** handles the approval process. This sub-workflow opens when you double click it.
4. Here we used the workflow control **Router**.
5. See how this Router shows 3 output pins. When hovering over them with your mouse you see their name, in this case the route to 'Off site printing'

Apart from creating your own workflows, you can also download sample workflows per market segment. These are advanced workflows that can serve as a head-start when creating your own. Learn more about these in [Sample Workflows](#) on page 1076.



Workflows can be started

- manually, by an Esko user,

- using the Pilot
- using the Shuttle client tool or the Shuttle plug-in of Esko editors and Adobe Illustrator.
- automatically
 - by an Access Point
 - by an external system that was integrated with Automation Engine. This could be WebCenter or any non-Esko system.

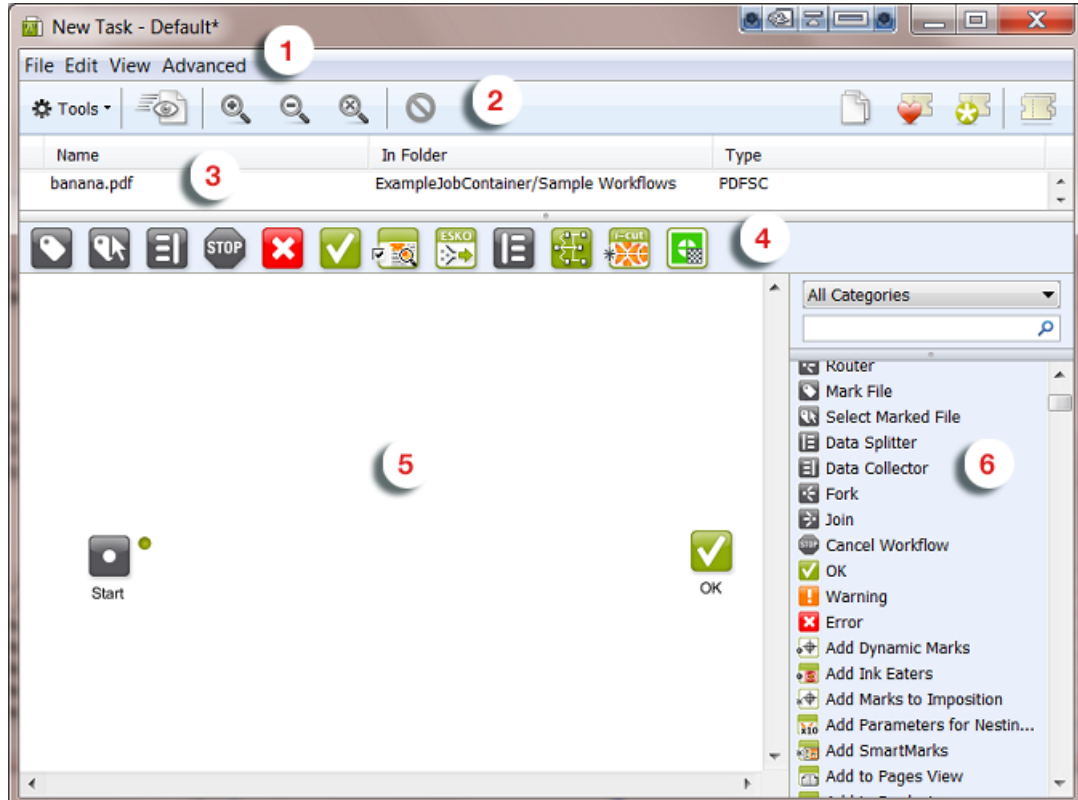
Note: Learn more about Access Points and integrations in [Integrating with External Systems](#) on page 1228.

6.2 Building Workflows using the Workflow Editor

You can open the workflow editor from the Pilot using  (from the **Tickets** view) or  (from the **Files** view).

6.2.1 The Workflow Editor Window

The workflow editor window consists of these main areas:



1. The workflow editor menus. Learn more in [The Workflow Editor Menus](#) on page 988.
2. The workflow editor tool bar. Learn more in [The Workflow Editor Tool Bar](#) on page 990.
3. The **files** pane shows the files that you selected before opening this workflow (only when opened from the **Files** view) or it shows the files that you launched this workflow on.
4. The **favorites** bar shows your favorite workflow steps that you often use when building workflows. Add new ones by drag and dropping new steps or custom tickets into the bar. To remove any, right-click and click 'remove'.
5. The **canvas** shows your workflow. You can monitor your workflow while it is processing files and you can diagnose a finished workflow.
6. The **tasks pane** shows all the tasks and workflow controls that you can use in a workflow. You simply drag and drop them onto the canvas. You can easily find tasks by using the filter and the search field.

6.2.2 The Workflow Editor Menus

Some menu options are only available while creating a workflow, others are only available for a saved workflow ticket.







Use this menu...	to ...
File	<ul style="list-style-type: none"> • Launch a file into the workflow (when you selected a file before building the workflow). • Launch a file into the workflow and keep the workflow editor window open (Launch And Monitor). • Select one or multiple files to launch in the workflow (Select File...). • Save your workflow and all its parameters in a Ticket (Save Ticket). • Save your workflow under a different name (Save Ticket as...). • Save your workflow on your client computer (Export Ticket...). Learn more about exporting workflows in Exchanging Workflows on page 1009. • Show information about the workflow ticket (Show Info): the Name of the ticket, the Task Type, the Tags, whether or not the ticket is Public, a Description of the ticket. • Open the selected step. • Show Parameter Inspector. Use this tool to easily manage the ticket parameters of all the used tasks. Learn more in The Parameter Inspector on page 1033. • Find: puts your cursor in the search field of the tasks pane and clears any previously entered text string. • Search: puts your cursor in the search field of the tasks pane or highlights any previously entered text string. • Close this workflow editor window.

Use this menu...	to ...
	<ul style="list-style-type: none"> • Print your workflow canvas on a network printer (only available for a saved workflow ticket)
Edit	<ul style="list-style-type: none"> • Undo or Redo your latest changes. The menu option also mentions which type of change it can undo. For example: 'Undo Change Parameters' or 'Undo move' etc. • Cut, Copy or Paste the selected step(s). You can also copy-paste between multiple workflow editor windows. • Select All workflow steps. • Delete the selected step.
View	<ul style="list-style-type: none"> • Change the zoom of the workflow to its Actual Size. Alternatively, you can also Zoom In, Zoom Out, or Zoom to Fit in the window. Check the available shortcuts. • Show / Hide Favorites, Files, Tasks and Sticky Notes. Learn more about sticky notes here. • Expand or Collapse the Queue. When a workflow is running, a Queue is the vertical list of icons of workflow items that are being processed by a same step. These items represent (grouped) files or concepts like Job, Plate etc. If the amount of these icons is disturbing, choose to collapse that queue.
Advanced	<ul style="list-style-type: none"> • Add / Remove this workflow ticket to / from your Favorites bar. • See a processing Log (when the workflow is running). You can also send the Log contents as an E-mail. • Set up Task Options: Priority, Hold / Release, Launching a separate task per input file and Annotations. • Set up Notification Rules. Learn more in Defining a Notification Rule on page 667. Tip: check if using the separate Send E-mail task is a better alternative. • Manage Workflow Parameters... and Workflow Parameter Values.... Learn more about these in Using Workflow Parameters on page 1065. • Manage Public Parameters for the selected step or check and Modify Public Parameter Values for the whole workflow. Learn more in Using Public Parameters in Workflows on page 1051.

Use this menu...	to ...
	<ul style="list-style-type: none"> • Validate the current workflow (including all subworkflows). Any problems will be shown and explained in a separate dialog.






6.2.3 The Workflow Editor Tool Bar

The tool bar offers buttons to:

- perform specific actions on the workflow ticket:  **Tools**. All available actions are explained in [The Workflow Editor Menus](#) on page 988.
- launch the workflow on the selected files  (only when you opened the workflow editor from the **Files** view). Press **Alt** to see the button change to  and **Launch and Monitor** the workflow (without closing the workflow editor).
- change the zoom in the workflow canvas (zoom in , zoom out , zoom to fit ),

Tip:

You can also use these shortcuts:

- **Ctrl + Space** (on Windows) or **Space + Cmd** (on Mac) to activate the zoom in tool, then click or drag in the window.
- **Ctrl + Alt + Space** (on Windows) or **Cmd + Option + Space** (on Mac) to activate the zoom out tool, then click in the window.
- Scroll your mouse up or down to move the whole workflow up / down in the window. Add **Shift** to move left-right.
- delete the selected step(s) .
- show / hide certain areas of the window: Files  (when opened from the **Files** or **Tasks** view), Favorites , Tasks , Tickets .

Tip: Use **Ctrl + click** or click and drag to select multiple steps.

6.2.4 Building a Workflow from Scratch

1. In the **Tickets** view, click  to open the workflow editor.

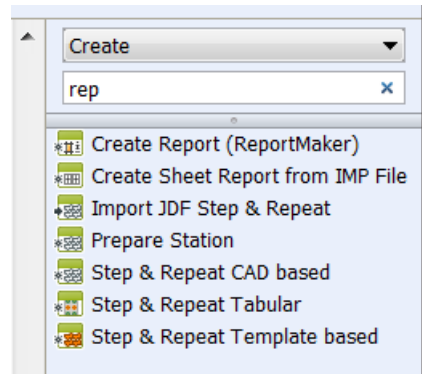


The canvas shows these two workflow controls by default: **Start** and **OK**.

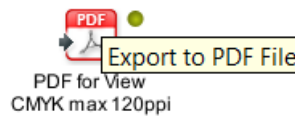
2. Drag and drop the desired steps from the tasks pane onto the canvas.

Note: If the tasks pane is not visible, click **View > Show Tasks** .

Note: To find a task quickly, use the filter and the search field.



3. If you want to give one or more steps a more descriptive name to make the workflow self documenting, double-click the name underneath the icon and change it. You can still see the original name of the step by hovering over the step.



4. Connect your steps with one another. To do this, click on a step's green output **Pin** and drag it onto the next step.

This will link the two steps with a green arrow, called a **Transition**.

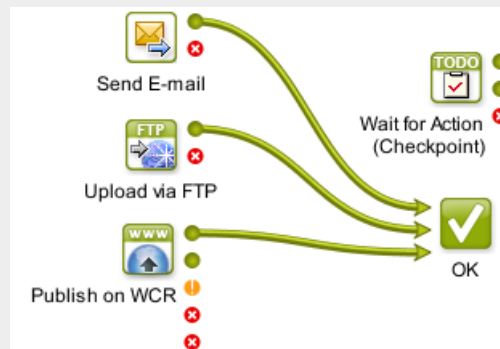


Note: Remove the transition by selecting the end part and dragging it away from the step that it was connected to.

Note: You can connect a step's output to multiple other steps. These steps will receive the same input file and will execute in parallel. See an example below ;



Note: When you move your mouse to a point where multiple transitions come together, they will split up to make it easier for you to pick one of them (see below). When you want to move them together, hold the **Alt** button to select and move them together.



Tip: Learn more about keeping your workflow clean in [Visual Optimizations of a Workflow](#) on page 995.

5. Double-click each step and fill in its parameters in its ticket. Click **OK** to close the ticket. There is no need to save these tickets. Their settings will be saved inside the workflow ticket.
6. Save your workflow using either:
 - **File > Save** or **Ctrl + S**,
 - **File > Save As...** or **Ctrl + Shift + S**.

Note: Until your workflow is saved, you can undo your changes. In **Edit > Undo** you can read which change can be undone.

Note: Until you save your changes, the name of the workflow will show an * behind it's name.

Tip: Before saving your workflow, it is good practice to check its validity. Use **Advanced > Validate Workflow** or **Ctrl + Alt + V**.

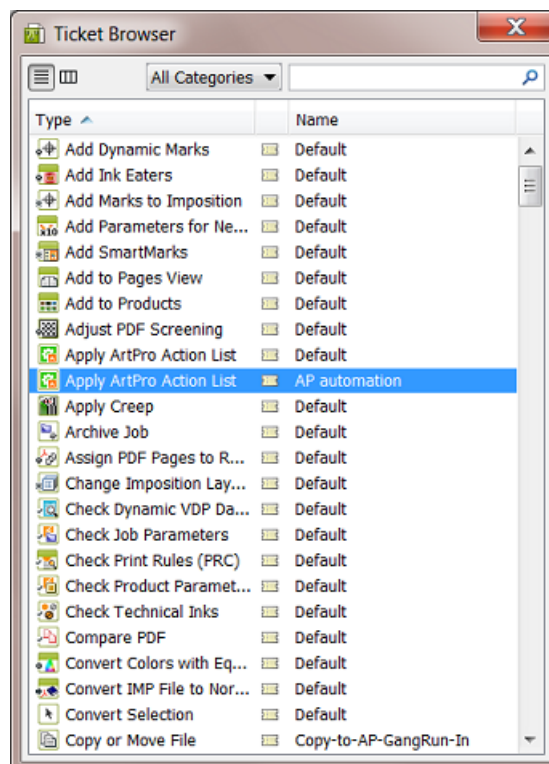
6.2.5 Building a Workflow using Custom Tickets

Compared to *building a workflow from zero*, using already made custom tickets as workflow steps adds these extra possibilities:

Using the Ticket Browser

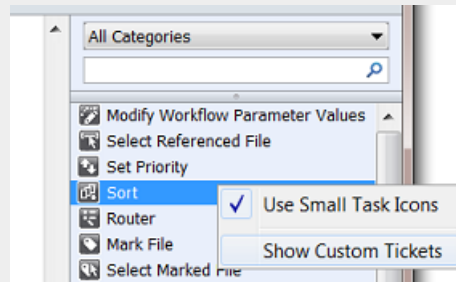
In the tool bar of the workflow editor, click  to open the **Ticket Browser** pop-up.

 shows the complete list, sorted per task type.  shows only the tickets of the selected task type.



1. Use the **Categories** filter and /or the search field to quickly find your custom ticket.
2. Select and drag the ticket onto your workflow canvas.

Note: Alternatively, you can choose to not use this **Ticket Browser**, but use the **Tasks pane** and activate **Show Custom Tickets** (right-click in the list).



Custom Tickets in your Workflow can be Copies or References

When you add a custom ticket onto your workflow canvas, it is by default a *copy* of that ticket. This means that any changes you make to that ticket will *not* be reflected in the original ticket and vice versa..

When you hold **Ctrl + Shift** or **Cmd + Alt** (Mac only) while you drag a custom ticket onto your workflow canvas, a *reference* to that ticket is added to your workflow. This means that any changes you make to the ticket will also be done in the original ticket and vice versa.

Note: This is equally valid when your ticket is a workflow ticket.

Referenced tickets also show their original name, with a small arrow in front. In this example you see 3 workflow steps: a copied ticket on the left, a referenced one in the middle and on the right a referenced one that was renamed in this workflow:



Note: If the ticket that you refer to in your workflow uses any SmartName containing [... originating Input ...], be aware that this will reflect to the input of the ticket that you refer to, not of the workflow that refers to it.

Adding a Part of another Workflow to your Workflow

If you want to add parts of a workflow to another workflow, follow these steps:

1. Open both workflows in a workflow editor.
2. Select a part of one workflow and choose **Copy**.
3. Click on the canvas of the other workflow and choose **Paste**.
4. Add the desired transitions and save your new workflow.

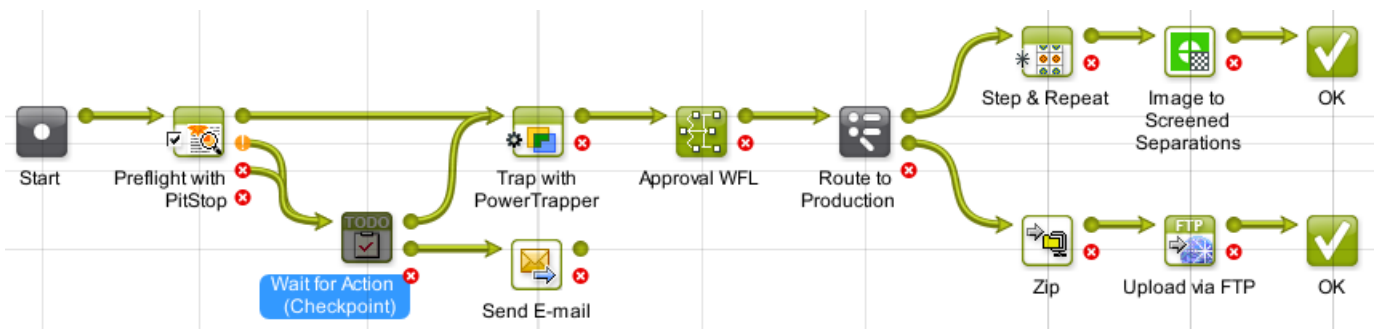
Note: If you plan to reuse workflows or parts of a workflow several times, we advise to create subworkflows. Learn more in [Subworkflows](#) on page 998.

6.2.6 Visual Optimizations of a Workflow

Workflows can become very complex. The following tools help to visually optimize them, which will make it easier to understand them.

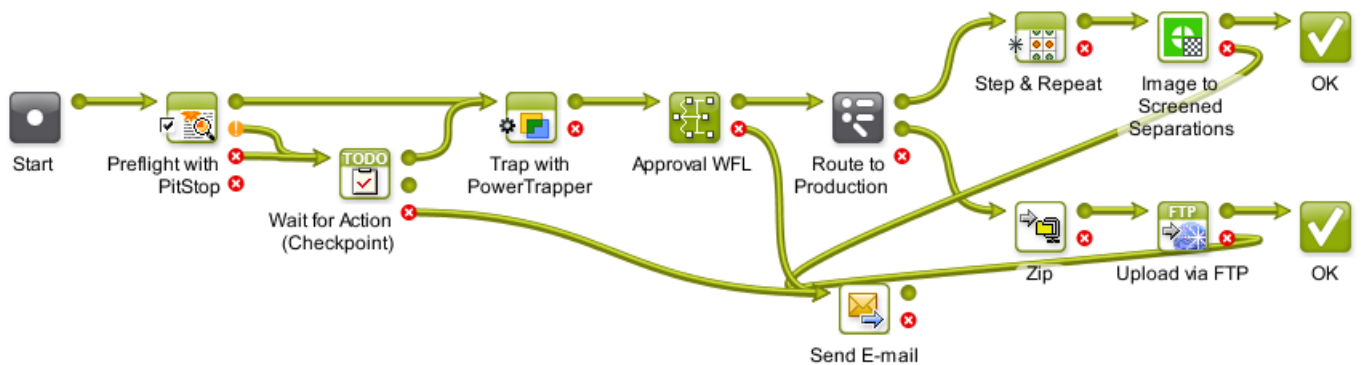
Aligning Workflow Steps on a Grid

You can align your workflow steps on a grid. To do this, hold the **Shift** key while dragging a step to the desired position. It will snap to a place on the grid.



Hiding and Showing Transitions

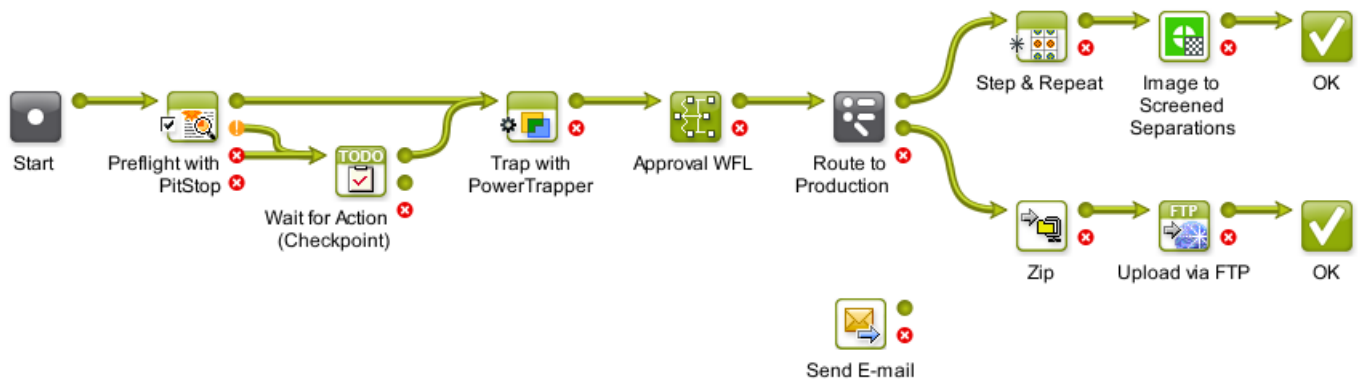
To optimize the overall look and readability of a workflow, you can hide specific transitions. See this example where the transitions to one **Send E-mail** step make the workflow look more complicated than it is:



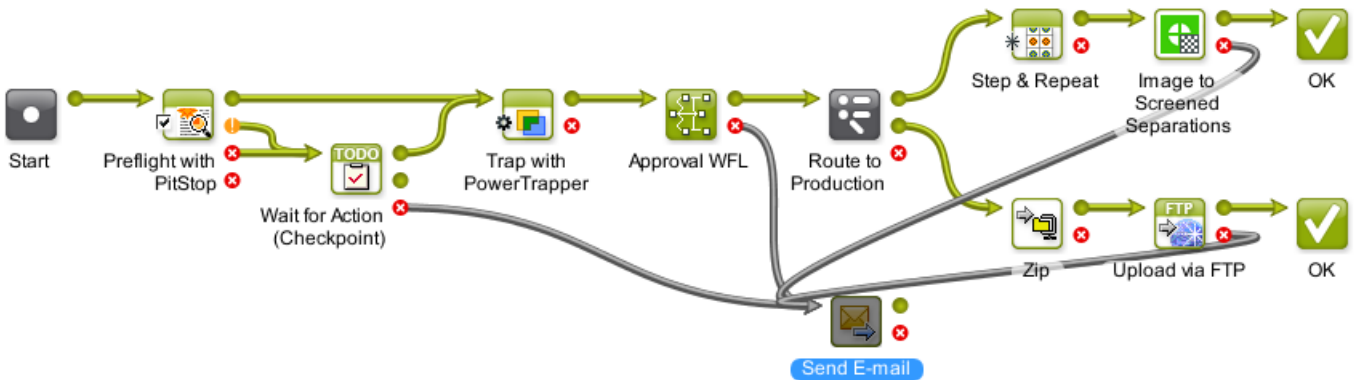
To hide the transitions to a (less important) step:

1. Right-click on the desired workflow step.
2. Click **Hide Transitions**. All transitions going to this workflow step will now be hidden. All other new transitions made to this step will automatically be hidden as well.

See this example where all transitions to the **Send E-mail** step are hidden:



If you want to see which transitions were hidden, click on the step (start or end). The hidden transitions will be revealed in a grey color:



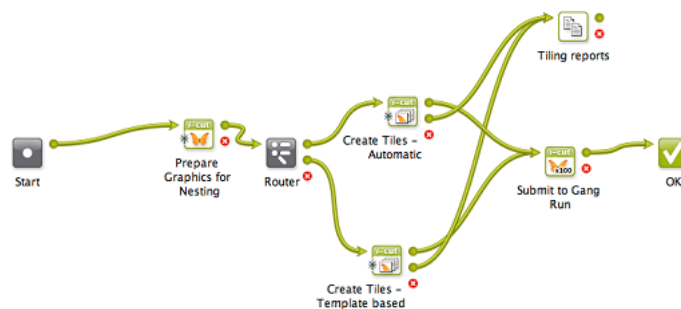
To reveal the transitions again, right-click the step and choose **Show Transitions**.

Distributing the Space between Workflow Steps

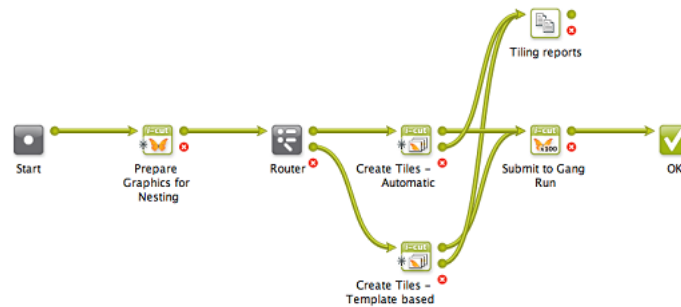
To distribute the space between all (or selected) steps evenly and also to align them on a grid, proceed as follows:

1. Right-click anywhere in the workflow window.
2. To align the tasks horizontally, click **Horizontal Distribute Space** or use **Ctrl + Shift + H**.
3. To align the tasks vertically, click **Vertical Distribute Space** or use **Ctrl + Shift + V**.

For example, this workflow needs some realignment:

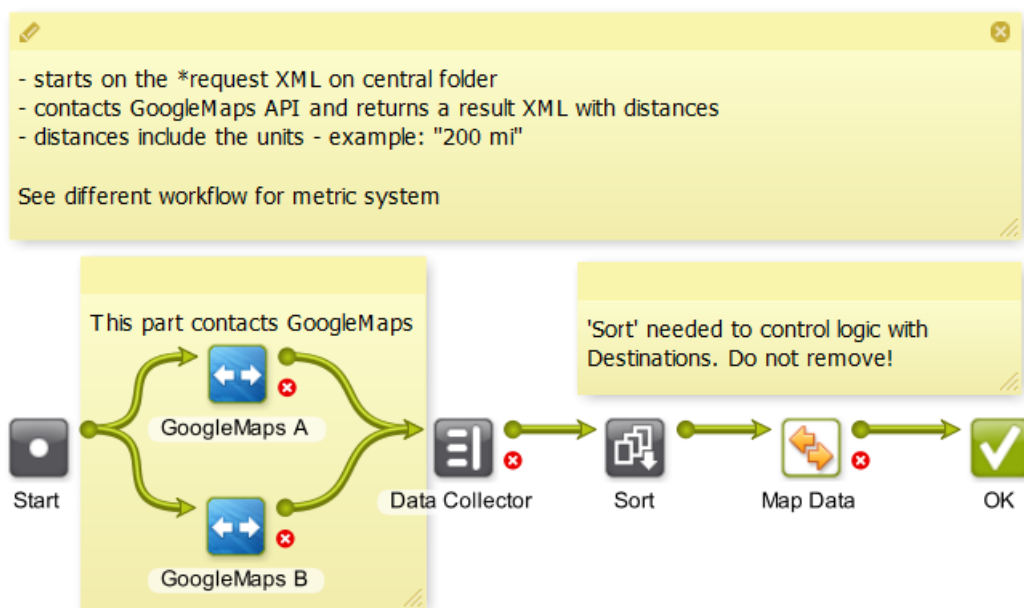


This is the result after distributing the space:



Adding Sticky Notes

Right-click anywhere on your canvas to add a **Sticky Note** to your workflow. Use these notes to add background information and so help users understand the workflow. They can serve as warnings or to remind users why some parts are constructed that way.



Customizing the Canvas Background

You can choose a custom canvas background for all your workflows. Right-click on the canvas, choose **Change Canvas Background...** and browse to an image on your local computer.

You can remove it again with **Clear Canvas Background**.

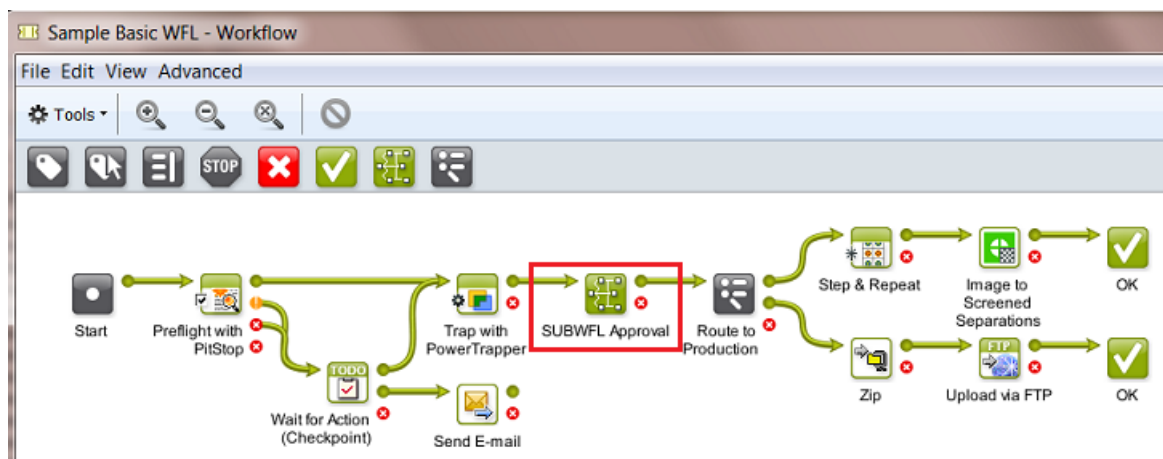
6.2.7 Subworkflows

A subworkflow is a workflow that is part of its master workflow. It is included in the workflow ticket of the master workflow. A subworkflow can also include another subworkflow. You can create workflows several levels deep.

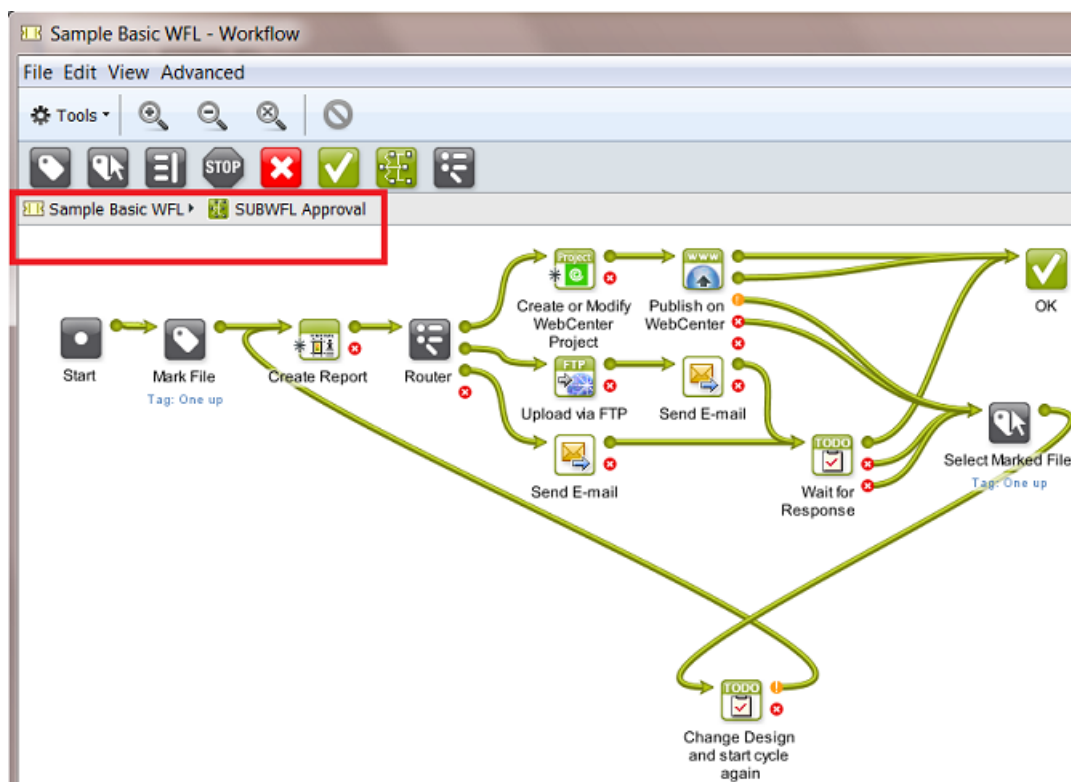
These are two main reasons to create a subworkflow:

- The master workflow will *look less complex* when a part of its flow is represented by only 1 workflow step.
- To stop creating the same (parts of) workflows again. If you notice that you keep creating the same or similar (parts of) workflows, then consider building that recurring part once and then copying it as a subworkflow into the other workflows.

In this example we see that the complete approval workflow is represented by a subworkflow named "SUBWFL Approval":



Double clicking the subworkflow step opens it in the same workflow window. Notice the indication of the path:



The Difference between a Subworkflow and a Referenced Workflow

A referenced workflow is not included in the master workflow, it is not copied on board. It is a link to a separate (workflow) ticket.

Note: Learn about adding referenced tickets or workflows in [Building a Workflow using Custom Tickets](#) on page 993.

The advantage of a referenced workflow is that by updating your referenced workflow you so automatically update all the master workflows that refer to it. See some examples in [Good Practices for Building Workflows](#) on page 1010.

See this example where the approval part of the workflow is added as a reference. Notice the small arrow and extra name of the original workflow ticket:



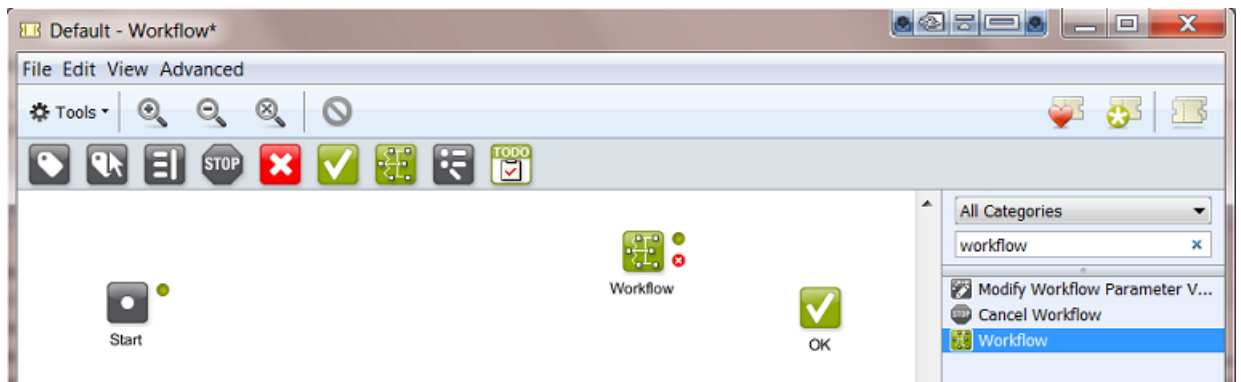
When you double-click a referenced workflow step, it will open in a separate window.

Building a Subworkflow

To add a subworkflow into your workflow:

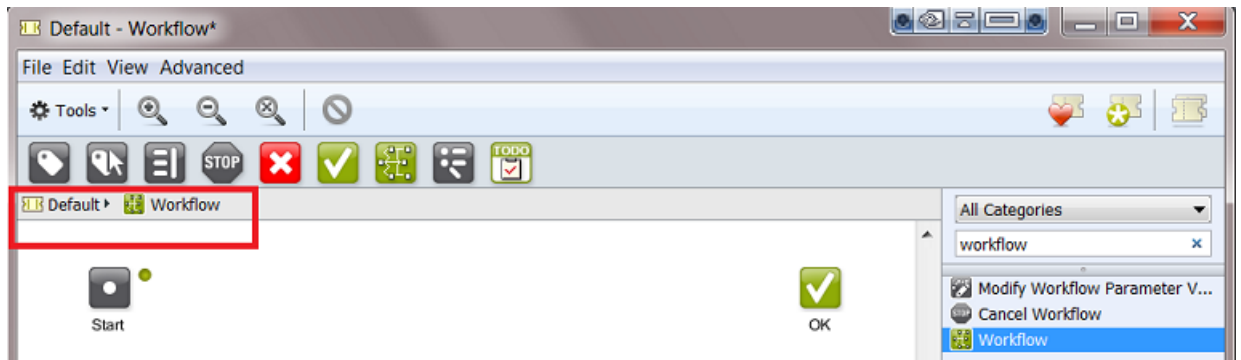
1. In the workflow editor, drag a **Workflow** task into your workflow. Or use an existing workflow ticket ; in that case skip step 3.

Tip: To find it quickly, select the category **Workflow** in the tasks pane or type 'workflow' in the search field.



2. Double-click the workflow step.

The workflow step opens in a new canvas, where you can build your subworkflow. The extra level in the path indicates that this is a subworkflow:



Note: This is an example of a path indication of a workflow with 2 levels of subworkflows: Flat Display WFL > One Up Approval > WebCenter approval

3. Build your subworkflow: drag and drop steps into its canvas, add transitions, open each step to change its parameters. There is no need to save at this point.
4. In the workflow path, click on your master workflow to go back to it (in above example: click on Default >).

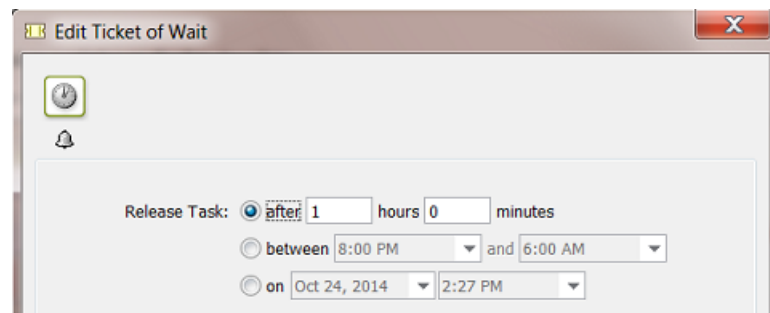
5. Save your master workflow. Your subworkflow(s) will be saved as a part of it.

6.2.8 Inserting Wait Points

Sometimes you want the workflow to wait a while before continuing with the next task. There are several ways to do this:

The Wait Task

Use the **Wait** task in a workflow to postpone the next task to a relative or absolute time.



A few examples of workflows where this task is useful:

- Your workflow sends a proof to a proofing device. You want to wait 30 minutes and then send the user a *To Do* to go and pick up the proof.



- A **Folder Access Point** launches a workflow on incoming design files. These design files have been backed up already somewhere else. That is why you can delete them from the Automation Engine Container when you are sure that the workflow no longer needs them (for example after 4 hours).



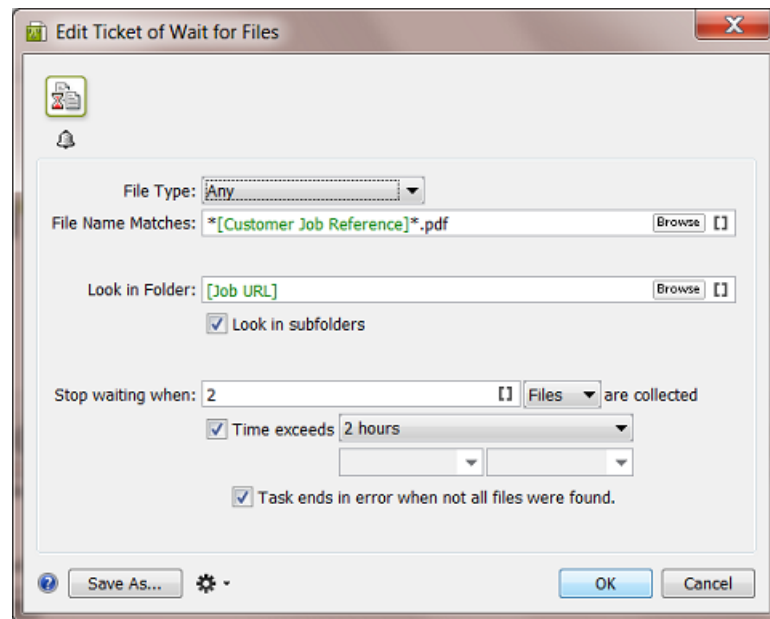
- Every night you want to clean up a specific type of files that you consider temporary. You have built this into your workflows, where a **Wait** task is set to release between 3 and 5 AM, and is followed by a **Delete** task.

The Wait for Files Task

Use this task when you want to process files that are not present yet.

The task will look for the files at a regular interval, and when it finds them, it will pass them on to the next workflow step.

You can also define how long the task should wait, and what to do if the files aren't found after that time period.




1. Select the File Type of the file(s) you want to wait for.

You can either:

- choose to wait for **Any** file type,
- select a specific file type from the list,
- select several file types (see below).

To select several file types:

- a) Choose **Select File Types...** in the **File Type** list.
This opens the **Select File Types** dialog.
- b) In this dialog, select the files types you want from the **Available File Types** column.
- c) Click  to send them to the **Selected File Types** column.
- d) Click **OK** when you are done.

You will see your selected file types listed in parentheses:



2. Use the File Name Matches parameter to search on file names.

You can use wildcards and SmartNames. You can also browse to a file with a name close to what you need, and add wildcards and/or SmartNames to that example name.

3. Define where to look for the files in the Look in Folder parameter.

Choose to **Look in subfolders** or not.

4. Define when to Stop waiting for the files:

- when a certain number of **Files** or **Pages** are collected,
- after a certain time if the files haven't been found (when **Time exceeds ...**).

To set a custom time, select **other...** and select a date and time from the dropdown list.

Depending on what you choose, the task will finish either after it has found all the files/pages, or after a certain time if it can not detect all the files/pages.

Note: If you do not give the task a time limit, and it can not detect all the files, it will run indefinitely.

5. Select **Task ends in error when not all files were found** if you want the task to finish with an **Error** state when:
 - the time that you set in **Stop waiting when time exceeds ...** has passed,
 - the task has not detected all files at that set time.

The Wait for Product Status task

This task allows to hold a workflow until one or more **Products** have reached a certain status.

This task is documented in the chapter on [Products](#) on page 876. Find a direct link [here](#).

The Wait for Action (Checkpoint) Task

Use this task to hold your automated workflow for any form of user intervention. This task adds a Checkpoint in the user's [To Do List](#). When the user finished the required action, he releases the To Do item. He can choose to let the workflow continue, to abort it or to have it take another route.

Task Settings:

- **To:** From the dropdown list you can choose a specific user or **Task Owner** (the user who launched the task) or **Any user** (all users will get the To Do).

If you use a SmartName to specify the user, make sure it resolves to the method "name@domain" (for example "franky@aeserver01" or "franky@fruitco-domain").

Note:

The users **Any user** and **Task owner** can also be addressed through a SmartName. This SmartName needs to resolve to their internal **Parameter Value** "Task Owner...@" and "Any User...@". Learn more about this in [SmartNames of Parameter Values](#) on page 982.

- **Subject:** This subject line is shown in the To Do List overview.
- **Message:** Optionally, add extra information.
- **Due date:** Setting a due date becomes useful in the To Do List overview for sorting and creating filters (learn more in [To Do List](#) on page 616). Choose from the dropdown list to set a relative or an absolute time ('On').
- **Output States:** You can here define custom output states. In a workflow, they are shown as output Pins.
 - Click **Add** to add a new output state. Enter a **Name** and **Description** (optionally) for the output state. Choose the **Type** from the dropdown symbols. An example:

Name	Description (optional)
Completed	Accept the job, continue workflow
Not sure. I will ask my boss	Route to workflow with extra Checkpoint
Aborted	Reject the job

Output States:

Add Remove Move Up Move Down

The receiver of this task needs to select one of the output states.
The file(s) will be released to the selected output state.

- **Release automatically:** Select this if you want the task to end after a specific time to a specific state.

Example of a workflow combining a To Do for "Any user" with one for "Task Owner"

It is possible to use the To Do List to distribute tasks. The first user who picks up a To Do Action can become the owner of the rest of the workflow.

See this example of a workflow that is launched by a **Folder Access Point**: the first Checkpoint task sends the To Do to **Any User**. The second Checkpoint task sends it to the **Task Owner**:



The Folder Access Point launches this workflow with a non-specified user:

Options

Operator: []

Priority: Normal

OK Cancel

Learn more about Access Points in [Access Points](#) on page 1236.

This is how the result looks like in the Tasks view:

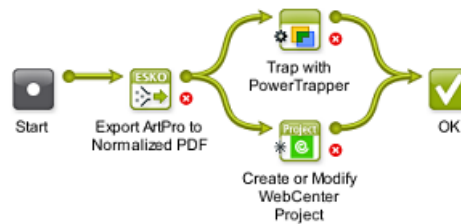
File Name	Task Type	Progress	Phase	State	Launched	Operator
banana.pdf	Wait for Action (Checkpoint)	0%	User intervention needed	[X]	10/30/14 4:24 PM	wde@rdvmaedemo
banana.pdf	Wait	100%		[X]	10/30/14 4:23 PM	wde@rdvmaedemo
banana.pdf	Wait for Action (Checkpoint)	100%		[X]	10/30/14 4:20 PM	UPLC@RDVMAEDEMO
banana.pdf	Workflow	0%	Running	[X]	10/30/14 4:20 PM	UPLC@RDVMAEDEMO

- First, the user was "UPLC" (the system user when no user is specified in an Access Point).
- The first To Do became visible in the list of All users.
- This first To Do was handled by the user "wde", which made him/her the Task Owner from that point on in the workflow.
- The second To Do goes to the Task Owner, which is still "wde".

6.2.9 Routing in Workflows

Simple Processing Routing

As mentioned in [Building a Workflow from Scratch](#), Transitions can route your files to different workflow steps. Automation Engine will automatically check the format of the files going through this step, and send them to the appropriate step(s) for that file format. See an example below where the PDF will go to will go to the trapping step:



However, to keep an easier visual overview of your workflow, we advise to use a Router workflow control instead (learn more in [Automatic Routing](#) on page 1008).

Routing in Subworkflows

When a subworkflow has multiple end steps, an extra type of routing becomes possible. The end steps of your subworkflow correspond with the outputs of the their workflow step in your master workflow.

For example, this subworkflow ends with two 'OK' end steps that were renamed to "Small Press" and "Big Press".



In the master workflow, you can see that they correspond to the two green output pins of the subworkflow step "Layout".



This means that:


- files sent to the "Small Press" end of your subworkflow will go through the "Small Press" output pin in your master workflow,
- files sent to the "Big Press" end of your subworkflow will go through the "Big Press" output pin in your master workflow.





Routing via To Do List Action Items

Users can decide the route of the workflow (files) through **Action** items in their **To Do List**. They can also choose to forward the To Do Action item to another Automation Engine user.

Learn more about Workflows creating Actions in a user's To Do List in [The Wait for Action \(Checkpoint\) Task](#) on page 1003. Learn more about using the To Do List in [To Do List](#) on page 616.

In the Pilot Tasks View

In the Tasks view, the **Wait for Action (Checkpoint)** task shows the Phase "**User intervention needed**". Below right, the icon  shows that one (extra) item is added to this user's To Do List.

Job Name	File Name	Task Type	Progress	Phase	State	Launched
ta_shu_Plugin_Set...		Wait for Action (Checkp...	0%	User intervention needed		11/02/10 10:43
ta_shu_Plugin_Set...		Trap - Prepare and Cre...	100%			11/02/10 10:43
ta_shu_Plugin_Set...		Normalize PostScript / P...	100%			11/02/10 10:42
ta_shu_Plugin_S...		Workflow	0%	Running		11/02/10 10:42

You can click either of these elements to **Release** the item to one of the outputs that were defined in the **Wait for Action (Checkpoint)** task. This will close the To Do item.

I am currently handling this to do item

Release to:

<please select>

<please select>

Send to proofer - Traps ok - to be proofed

Re-run - Change parameters to improve traps

Not good - Wrong trapping

In the Workflow Editor

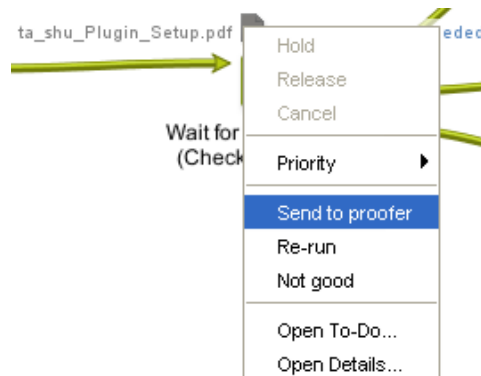
In the workflow editor, the user will see **User intervention needed** as the file status and the **Checkpoint** task step will jump up and down:




The workflow editor will also show one To Do  (for this workflow).

To route (and close) the To Do item, you can

- right-click the file icon and select one of the outputs defined in the **Checkpoint** task.



- right-click the file icon and select **Open To Do...** and there select one of the outputs offered in the **Release** button
- click the To Do icon  to open the item in the To Do List and there select one of the outputs.

Note: In the To Do List, make sure the **Actions Filter** shows the **Wait for Action** items.

In Shuttle

In the **Automation Engine Shuttle** window, Checkpoint tasks and To Dos can be handled in the same way as you do [in the Pilot Tasks View](#).



Attention: The user name you used in the **Checkpoint** task and the user name used in Shuttle to connect to Automation Engine must match.

Note: Learn more about Shuttle in [Using Shuttle to Upload and Process Files](#) on page 47 (User Guide) and in [The Automation Engine Shuttle](#) on page 47 (Reference Guide).

Automatic Routing



The **Router** workflow control enables a workflow to automatically decide the route that your workflow (files) should follow.

The criteria that you define to route to a specific output Pin can be based on many different types: the file type, the number of separations, the value of a Job Parameter, etc.

For example: routing files based on their **File Type**:

File Type		Route to
is	ArtPro File	ArtPro workflow
is	PDF File	PDF workflow
is	Normalized PDF File	PDF workflow
no match		Error

Buttons: Add, Remove, Move Up, Move Down

... and so send each file type to an appropriate next task in the workflow:



In the above example, ArtPro files are routed to their own normalization task, while Native or Normalized PDF files are sent directly to the Trap task.

Learn more about the Router in [Router](#) on page 1024.

6.2.10 Exchanging Workflows

The two main reasons to export and import workflows are

- to exchange them with other plants of your company that also use Automation Engine.
- to exchange them with Esko Customer Service.

Exchanging workflow tickets is done *in the same way as for single tickets*:

1. **Export** the workflow to a ZIP file (in Tickets view or in the Workflow Editor).
2. Send the ZIP file to the other Automation Engine (onto a client computer of that server).
3. At the receiving site, in Pilot Tickets View, select **Import**, browse to the ZIP file, and click **OK**.

Note: The Automation Engine software version on the server that imports the ZIP may not be on an older version than the Automation Engine server that created the ZIP.

What about references?

As mentioned in [Working with Tickets](#) on page 177, references will also be copied into the ZIP file. These references are not just files like templates but also SmartNames, profiles, scripts, referenced workflow tickets etc. See a full list of included references in [Exporting Tickets](#) on page 178.



Attention: Workflows that include referenced (blue) Job Tickets can not be exported. An error message will appear.

Note: Exchanging workflows is often done just to exchange the SmartNames used in those workflows. Learn more in [Exporting and Importing SmartNames](#) on page 938.

6.2.11 Good Practices for Building Workflows


Below tips help you build a workflow that is sustainable and easy to maintain. They also avoid extra work and cost when you upgrade or migrate server computers.

Note: Some of these tips are also present as an Esko Knowledge Base article. Find these and more on <http://help.esko.com/knowledgebase>.

Avoid Absolute Paths and Server Names

Do you know the name of your file server in five years from now? Probably not. That is why we recommend to avoid using that computer name in your Tickets and Workflows.

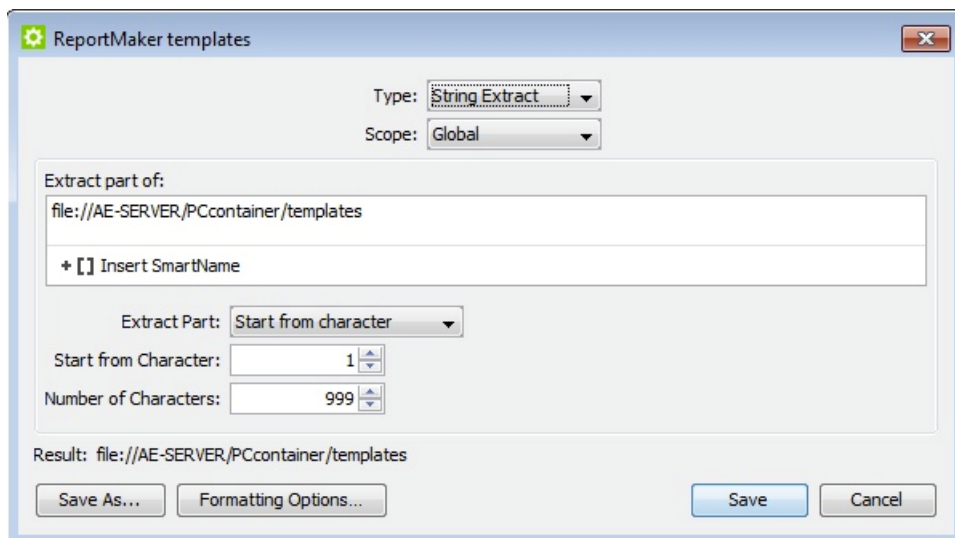
For example when you define a template for the ReportMaker task. See below example of a Ticket with an absolute path to one specific template. If you have many such tickets, and the location of the templates changes, you then have to adjust all your ReportMaker tickets.

Report Template: 

This can be avoided by using a SmartName in the path. When the path changes, you then only have to update the SmartName.

Report Template: 

The SmartName can be a **String Extract SmartName** that contains the path or even just the server name:



ReportMaker templates

Type:

Scope:

Extract part of:

+ Insert SmartName

Extract Part:

Start from Character:

Number of Characters:

Result: file://AE-SERVER/PCcontainer/templates

The same principle applies to other tasks with references to files or folders like **Compare PDF**, **Create CAD Sheet** and **Select File**. If you copy a PDF file to an external system, a CIP3 file to a press, a TIFF file to an imaging device, a JPEG preview to an MIS, a ZIP file to an archiving system etc., always avoid using server names and fixed locations.

Note: You can keep it even more generic. For example if your templates are saved on the Automation Engine Server, you can use the system SmartName `[Server]` to make your path smart. No need to create an extra SmartName in this case, but you will have to keep the templates on the Automation Engine Server when you migrate to new hardware.

Some typical situations where you better use SmartNames in paths:

- ReportMaker templates
- CAD files repository
- Image Mark files in SmartMark sets
- Files loaded in ArtPro Action Lists
- PDF, XML, JPEG communication with MIS/ERP
- TIFF, LEN, PS data sent to output devices
- Defining the file location of a Product.

For some situations, there is a default solution:

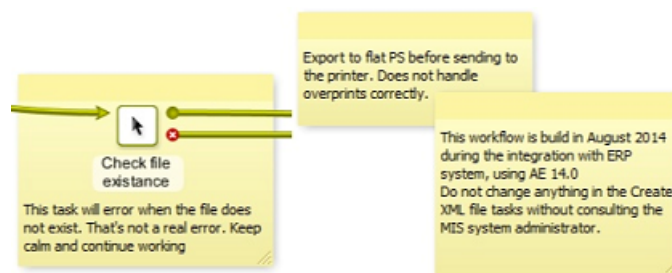
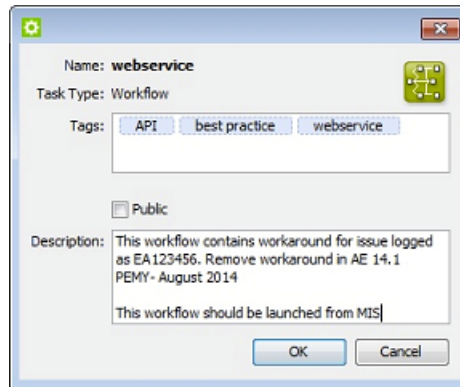
- In the Create Job ticket, use the Smart Job Location instead.
- Save PitStop Profiles in the default "PitStop Root" container (only visible in the PitStop ticket).
- FastImpose templates have a default location on the Automation Engine Server.

Annotate

If you are implementing a temporary workaround or even misusing a task to achieve your goals, make sure that you later remember why you did this. There might be a better solution or less strict requirements in a next version of Automation Engine. An exotic part of the workflow for which you don't remember why or how you did it, is a part that nobody later dares to change or remove. It will cause problems sooner or later.

The best way to manage this is to document on the spot. Automation Engine provides plenty of possibilities to do so:

- Description field for tickets and workflows
- Tags for tickets and workflows (also helps filtering)
- Renaming the workflow steps
- Description field for SmartNames
- Tags for SmartNames
- Sticky Notes in workflows

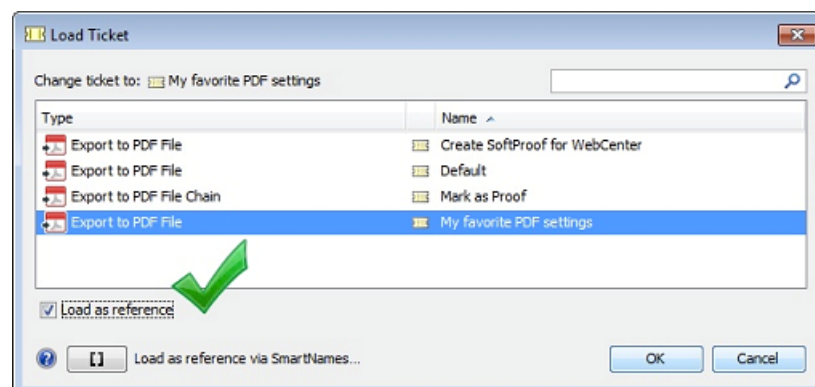


Note: Documenting your experiences while building a workflow is not only a great help for yourself and your colleagues but also for anybody at Esko Customer Service.

Use Referenced Tickets and Workflows

Referenced Tickets

You probably use several Export to PDF tickets. We advise to not copy-paste such a ticket in your workflows but to add a reference to this ticket. Then, when you decide to change a setting (for example "outline fonts"), you only have to change this single ticket and all the workflows that use it will so adapt and use these latest settings.



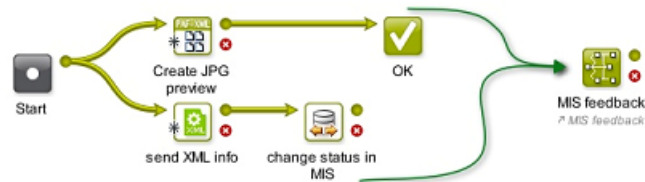


Learn more about referenced tickets in [Building a Workflow using Custom Tickets](#) on page 993.

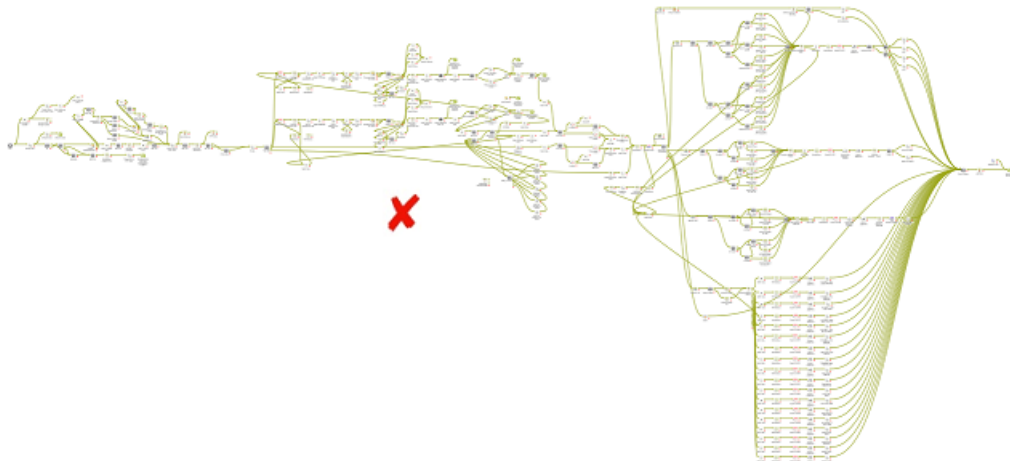
Reoccurring Workflows Parts

When part of a workflow is identical in other workflows, we advise to create it as a separate workflow and then refer to it from the master workflow(s).

For example, a part of your workflow sends feedback about a file to the business system. You probably use this part in several workflows. We advise to create it once and use it many times as a workflow step in various master workflows. We again advise to not copy it but to add it to the master workflow as a reference. This way, when settings change, you will only have to adapt it once.

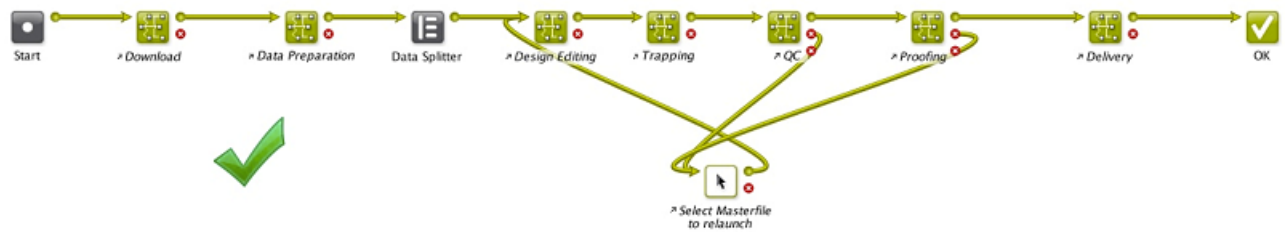


Workflows can get gigantic. When launching such a workflow, the Pilot will read all the settings of all the tasks in this workflow, even in the subworkflows. This is why launching bigger workflows seems slow. See an example of such huge workflows that you should try to avoid creating:



However, at the moment of launching, the Pilot will not immediately read the settings of a *referenced* workflow. This adds an extra reason to divide such huge workflows into several referenced workflows. The master workflow will not only react quicker, but will be much more structured and easier to understand.











Here is a good example of a master workflow that quickly visualises the logic workflow process steps:









Learn more about copied and referenced subworkflows in [Subworkflows](#) on page 998.

6.3 Workflow Controls

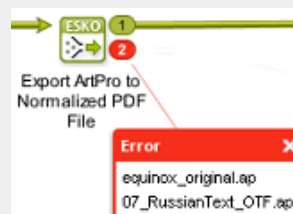
Workflow Controls are steps in a workflow. Instead of executing a task on an input file, they make workflow decisions on routing, file selection, file order etc.

- 
Start: Every workflow needs to start with this step. By default it is present on every new workflow canvas.
- 
Modify Workflow Parameter Values: Use this to modify workflow parameter values at some point during the workflow. Learn more in [Modify Workflow Parameter Values](#) on page 1015.
- 
Select Referenced File: Use this to easily select files that are referred to in an XML file. Learn more in [Select Referenced File](#) on page 1018.
- 
Set Priority: Use this to assign a different priority to parts of your workflow. Learn more in [Set Priority](#) on page 1021.
- 
Sort: When the order of the input files is important in the next workflow step, use this one to manage that order. Learn more in [Sort](#) on page 1021.
- 
Router: This step enables automatic routing of files based on many possible criteria. Learn more in [Router](#) on page 1024.
- 
Mark File: Use this step to mark files that you will select later in the workflow using the **Select Marked File** workflow control. Learn more in [Mark File and Select Marked File](#) on page 1029.
- 
Select Marked File: Use this step to select the files that you marked earlier in the workflow using the **Mark File** workflow control. Learn more in [Mark File and Select Marked File](#) on page 1029.
- 
Data Splitter: Use this step when you want to send the files one by one to the next task instead of all together. Learn more in [Data Splitter](#) on page 1030.
- 
Data Collector: Use this to collect all incoming files before sending them to the next task. Learn more in [Data Collector](#) on page 1029.

- 
Fork: Use this to tag incoming and related files with a group tag. Learn more in [Fork and Join](#) on page 1032.
- 
Join: Use this to select the files tagged as a group by the **Fork** workflow control. Learn more in [Fork and Join](#) on page 1032.
- 
Cancel Workflow: When a file is sent to it, this step cancels the processing of the entire workflow. This is useful when you also want to stop not just the branch of the workflow where this step is used, but all branches. For example: when the previous step **Manage Product Status** results in 'not approved' then the next step is **Stop**.
- 
OK: By default, there is already one **OK** step in the canvas. You can add more and route files processed successfully to the appropriate **OK** step. When a subworkflow ends with multiple **OK**'s, they will show up in the master workflow as multiple green output pins.
- 
Warning: Add this step at the end of your workflow to route files that generated a warning during processing. For example preflight tasks or approval processes typically also offer a warning as end type.
- 
Error: Add this step at the end of your workflow to route files that generated an error during processing.

Note: A task that has one or more files ending with an error automatically triggers an error status for both that task and the whole workflow. That is why it is not necessary to add this **Error** workflow control and connect to it from the error output pin of a task.

To see which files caused this error, you can also click on the error output pin and open the output widget:



Learn more in [Diagnosing a Workflow in the Workflow Editor](#) on page 1047.

6.3.1 Modify Workflow Parameter Values

This workflow control allows to modify values of workflow parameters during the workflow. Learn more about workflow parameters in [Using Workflow Parameters](#) on page 1065.

Concept

The typical reason to modify a workflow parameter during a workflow is that there is new or changed information that was not there earlier or at the start of that workflow. Some example cases:


- The new or changed information is based on a user's decision. For example the decision how to release a To Do Action item. See an example below.
- The new or changed information is coming from an external system (and was not available earlier).

Note: Mind these exceptions:

- New information coming from WebCenter does not need this separate workflow control. The tasks that communicate with WebCenter offer their own built-in tool to map their output parameters to workflow parameters. Learn more in [Modifying Workflow Parameters During a Workflow](#) on page 1075.
- When, during a workflow, an external system sends an updated XML with new or modified parameters, check if you can use the feature [Using Workflow Parameter Values from XML](#) on page 1072. This feature loads that XML at every step of the workflow, not just at the start. This means that the incoming updated XML can trigger an updated workflow parameter without the use of this separate workflow control.

Using the workflow control **Modify Workflow Parameter Values**

To use this workflow control, proceed as follows:

1. Select and drag the **Modify Workflow Parameter Values** workflow control onto your workflow canvas. Double-click it to edit.
2. Click **Add** to add workflow parameters or double-click existing ones to modify.
3. Use  to select an existing parameter or type the desired workflow parameter name and provide a new value. This can be a fixed value, a SmartName or an XPath that retrieves the new value from the workflow control's XML input. Learn more on Xpath in [The XPath Builder](#) on page 954.
4. Click **Add** to confirm.

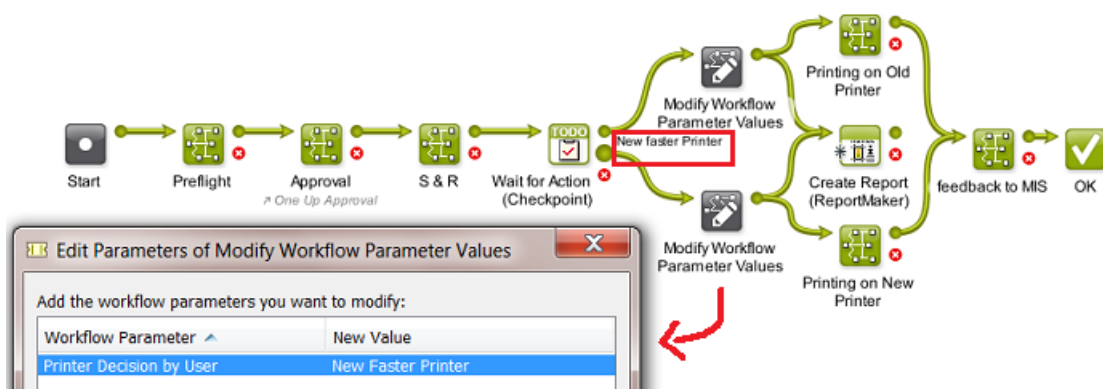
Note: When the parameters list contains a new one, you will be asked to **Confirm**.

5. Click **OK** to close. Save your workflow.

Example: Capturing the To Do decision as a Workflow Parameter

Our workflow asks a user to decide if the output will be done on the old or on the new printer. This decision should also be mentioned on the Report PDF.

- See how the To Do output Pin 'New faster Printer' routes to the workflow control that sets the value of the workflow parameter 'Printer Decision by User' to 'New Faster Printer'.




- The ReportMaker template needs to pick up that workflow parameter. Those templates are created in PackEdge. Because PackEdge SmartMarks can not pick up workflow parameters, you first need to create a SmartName of that workflow parameter and then use that SmartName in the template. Follow these steps to do this:
 1. In Pilot, in the SmartNames View, in **Include SmartNames From**, check the option **A workflow** and select the workflow that contains the desired workflow parameter.
 2. Click **[]⁺** to create a new SmartName.
 3. Set the Scope to **Global** and the SmartName Type to **String Extract**.
 4. Click **+ [] Insert SmartName** and select the category **Workflow Parameters**.
 5. Select the desired workflow parameter [wfp.] and click **Insert**.
 6. Save the SmartName.
 7. Open the ReportMaker template in PackEdge and open the **SmartMark** palette.
 8. Select **Text Mark** from the **Add SmartName** palette.
 9. Click the **Smart Text...** button, select the category **SmartNames (Global)** and select the SmartName you just created.
 10. Save the ReportMaker template
 11. Use this template in the ReportMaker task. The resulting PDF can now mention what the user decided in his To Do. Here is an example of how this can look like:

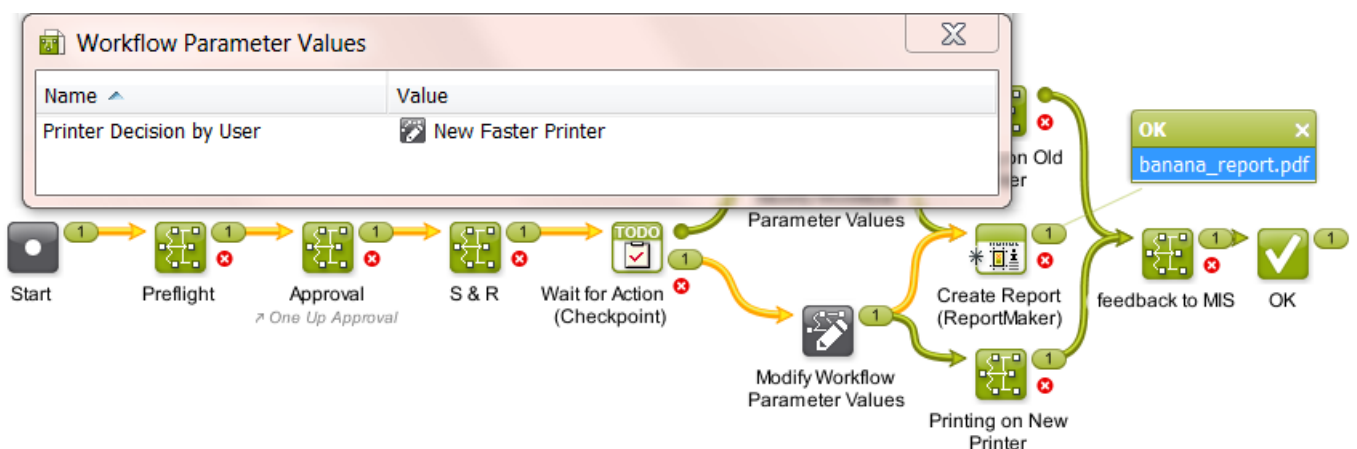


Visualizing the Workflow Parameter Values of a Step Output

To visualize the Workflow Parameter Values of a selected file, you can:

- choose the menu item **Advanced > Workflow Parameter Values**. This dialog stays opens while you click on output files.
- right-click on the desired step output and select **Workflow Parameter Values**.

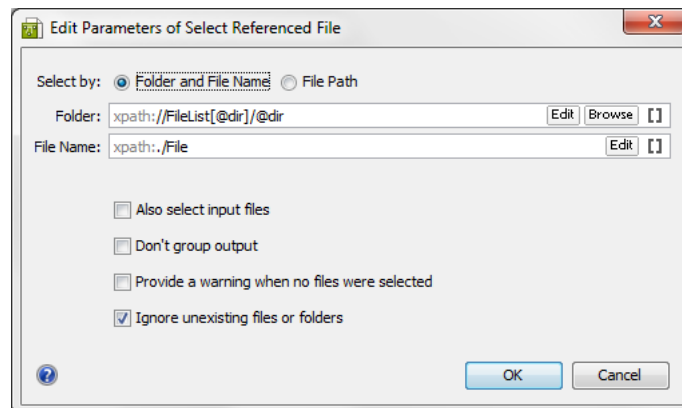
Using above mentioned example, see the workflow parameter values of the Report PDF. The small icon  indicates that the value was modified during the workflow:



Note: The set of workflow parameters can be different for each step output. For more info, see [Using Workflow Parameters](#) on page 1065.

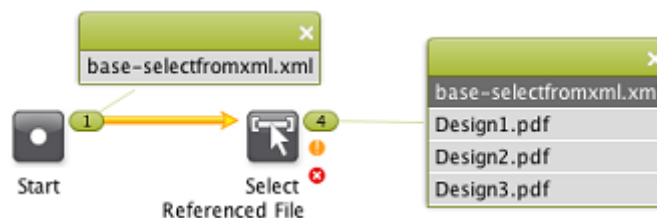
6.3.2 Select Referenced File

Use the workflow control **Select Referenced File** to easily select files that are referred to in an XML file.



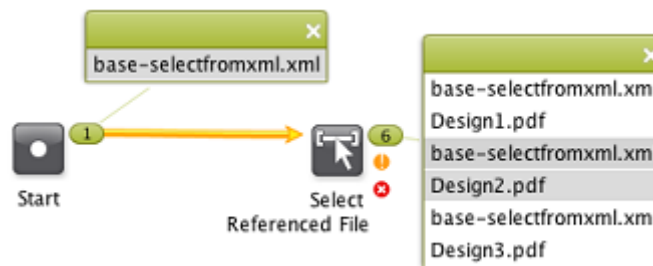
Note: The **Edit** button offers to use the **Xpath Builder**. Learn more about creating Xpath Expressions [here](#).

- **Select by:** You can indicate (via SmartNames or XPath) which XML content to take to build up the corresponding file path(s) and name(s). You can choose between:
 - **Folder and File Name:** Choose this option if the source XML contains entries that define the file path and file name separately. If this option is chosen, specify both the **Folder** and the **File Name**.
 - **File Path:** Use this when each file in the source XML has a fully specified absolute file path.
- **Also select input files:** Select this option to also add the step's input file(s) to the group of output files.



- **Don't group output:** When this option is selected, the selected files will be offered separately to the next step in the workflow instead of as a group.

When **Don't group output** is not selected (so output is grouped) and **Also Select Input Files** is selected, the task will generate a group for each output:



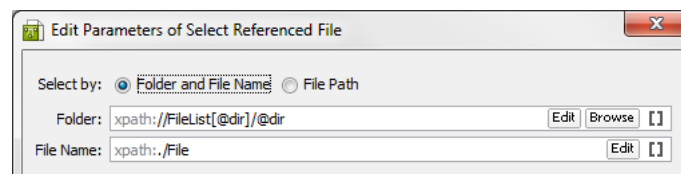
- **Provide a warning when no files were selected:** Select this option to get a warning when no files are selected.
- **Ignore unexisting file or folder:** Select this option to prevent workflow errors when one or more file paths refer to non-existing files. When selected, the non-existing files will be ignored.

Example 1: Folder and File Name

Our example XML refers to one file in a first Job and then to 3 files in another Job:

```
<?xml version="1.0" encoding="UTF-8"?>
- <doc>
  - <FileList dir="file://loppem/ExampleJobContainer/Jobs/Order-2014-100024">
    <File>PDF/Design1.pdf</File>
  </FileList>
  - <FileList dir="file://loppem/ExampleJobContainer/Jobs/Order-2014-100025">
    <File>PDF/Design1.pdf</File>
    <File>PDF/Design2.pdf</File>
    <File>PDF/Design3.pdf</File>
  </FileList>
</doc>
```

We used the **Edit** button to get to these settings:



After the **Select Referenced File** step ran, you will notice these 4 files on the OK output pin of the workflow control (assuming the files exist).

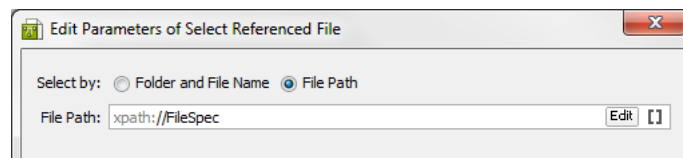
- file://loppem/ExampleJobContainer/Jobs/Order-2014-100024/PDF/Design1.pdf
- file://loppem/ExampleJobContainer/Jobs/Order-2014-100025/PDF/Design1.pdf
- file://loppem/ExampleJobContainer/Jobs/Order-2014-100025/PDF/Design2.pdf
- file://loppem/ExampleJobContainer/Jobs/Order-2014-100025/PDF/Design3.pdf

Example 2: Absolute File Path

This example XML contains several absolute file paths:

```
<?xml version="1.0" encoding="UTF-8"?>
...
<FileSpec>file://AEServer01/Food/Jobs/1234/PDF/design1.pdf</FileSpec>
<FileSpec>file://AEServer01/Food/Jobs/1234/PDF/design2.pdf</FileSpec>
<FileSpec>file://AEServer01/Food/Jobs/1234/PDF/design3.pdf</FileSpec>
...
```

We chose the option **File Path** and used the **Edit** button to get to this setting:



The output pin of the **Select Referenced File** step will show these 3 file paths:

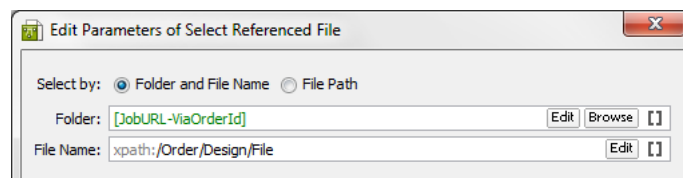
- file://AEServer01/Food/Jobs/1234/PDF/design1.pdf
- file://AEServer01/Food/Jobs/1234/PDF/design2.pdf
- file://AEServer01/Food/Jobs/1234/PDF/design3.pdf

Example 3: Using SmartNames

This example XML has a different structure:

```
<?xml version="1.0" encoding="UTF-8"?>
<Order id="2014-102457">
  <Customer id="JuiceCo"/>
  <Design>
    <File>PDF/design1.pdf</File>
    <File>PDF/design2.pdf</File>
  </Design>
</Order>
```

In this case, we find the Folder via a SmartName, based on the Order ID, and the File Name via an Xpath:



This is the resulting selection ; the part of the path including the Job number was found via the SmartName, followed by what the XML defined in <File> :

- file://AEServer01/Food/Jobs/102457/PDF/design1.pdf
- file://AEServer01/Food/Jobs/102457/PDF/design2.pdf

6.3.3 Set Priority

Use the **Set Priority** workflow control to assign a different priority to certain branches of your workflow.

You can set the priority to:

- **Workflow** (to use the general priority assigned to the workflow)

Note: By default, the general workflow priority is set to **Normal**. You can change it by going to **Advanced > Task Options...** in the workflow editor window.

- **Low**
- **Normal**
- **High**
- **Immediate.**

You can see the chosen priority in blue under the workflow control:



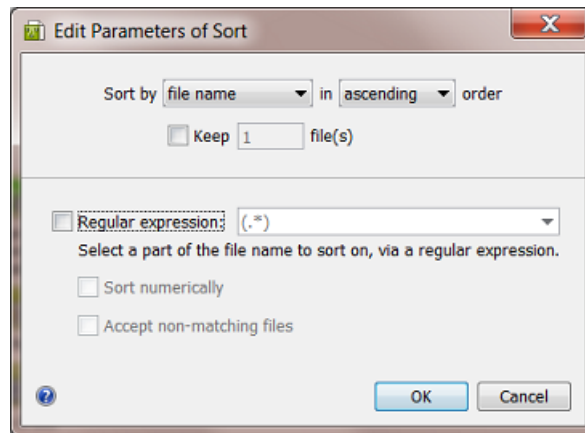
Example

In this example, the most urgent operation is RIP'ing the files and sending them to the device. Creating a JPEG and a Report PDF is less urgent:



6.3.4 Sort

Use the **Sort** workflow control to define the order of the files in a file set. You do this because the order is important in the next workflow step.



You can sort the files by:

- **file name.**
- **workflow inputs.** Files that entered the workflow first will be at the top of the list.
- **SmartName.** The value of a SmartName is calculated per file and that value determines the order.

You can sort in **ascending** or **descending** order.

Keep ... file(s): Use this if you only want to send a specific number of sorted files to the next workflow step.

Sort by file name

Use a **Regular expression** to define the part of the file name that you want to sort on. The dropdown list shows two example regular expressions:

- `(.*)` matches the entire file name
- `.*?([0-9]+)` matches the number in the file name (for example to sort files named `cover1.pdf`, `page_2.pdf`, etc.).

Learn more about regular expressions in [Using Regular Expressions](#) on page 978.

- **Sort numerically**

Select this option if you are using a regular expression to extract a number from file names. The results will then be sorted numerically and not as text. For example, a value of 10 will be sorted after 9 when this option is selected. When you don't sort numerically, the values are considered as text: then 10 will come before 9 (as text, the first character 1 comes before 9).

- **Accept non-matching files**

Select this option if you also want to sort files that don't match the criteria. For example your regular expression is based on numbers but the name of one file name doesn't contain numbers.

Any non-matching file(s) will be put at the top of the sorted list.

If you don't use this option and a file does not match, all your files will go to the error output pin.

Sort by workflow inputs

Files that entered the workflow first will be at the top of the list. You can sort in **ascending** or **descending** order and optionally define how many files you want to keep.

Sort by SmartName

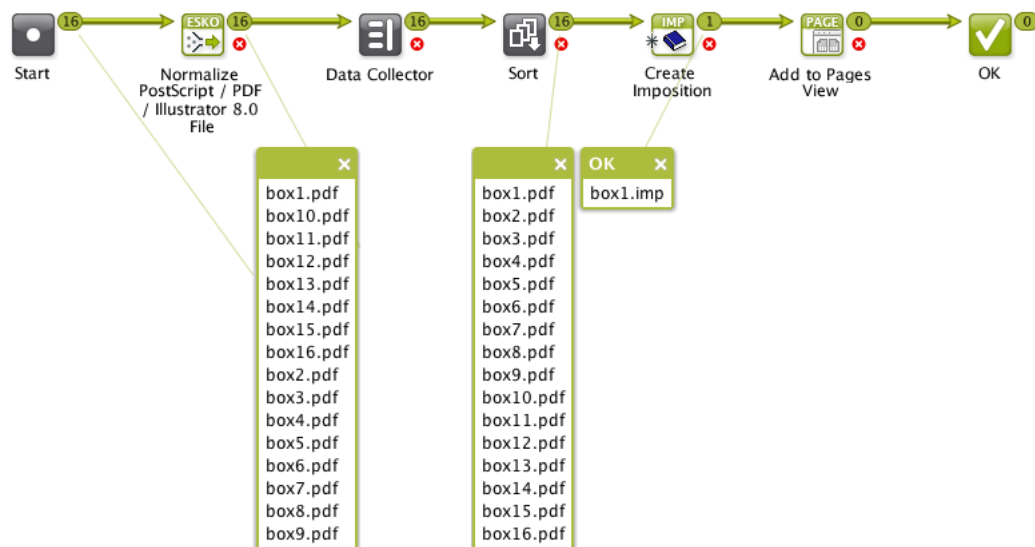
Select the SmartName you want to sort on. This list only offers custom SmartNames.

Numerical versus text based sorting works as described above for **Sort by file name**. The same is true for the handling of non matching files.

Example: Sorting by Page Number

In this workflow, the **Data Collector** collects the pages of a page list after Normalizing, and sends them to the **Sort** workflow control. Learn more about **Data Collector** [here](#).

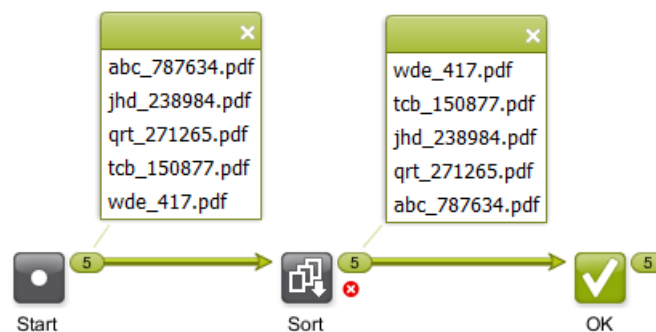
The **Sort** workflow control then sorts them in the right order, before sending them to the **Create Imposition** task. The files are sorted on **file name**, in **ascending order**, using a **Regular expression** to extract the number in the file name (and with the **Sort numerically** option):



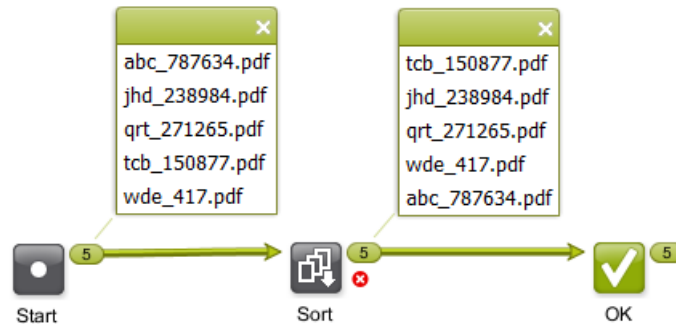
Example: Sorting by SmartName, Numerically and As Text

In this example, you want to sort your files by the number that they contain. The number is always mentioned after an underscore. You created a SmartName that extracts, from a file name, everything after the underscore.


This is the result when you **Sort numerically**:



This is the result when you don't **Sort numerically** ; the part of the name after the underscore is now considered as text:



6.3.5 Router

The **Router** workflow control  serves to automatically decide the route that your workflow (files) should follow.

Let's take the example where a Router decides the workflow track based on the size of the incoming file. When the file is smaller than 5 MB, the file will be sent out via E-mail, otherwise via FTP.

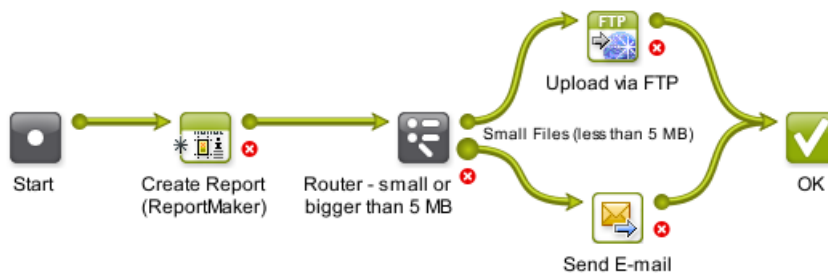
- These are the settings in the Router:

Two rules were defined and the **Route to** names were customized.

Note: The rules are executed from top to bottom. Their order can be changed with **Move Up / Move Down**.

- This is the workflow:

Notice how (when the mouse hovers over the Router step) the name of the middle output pin appears and how it matches the name that was defined in the **Route to** column of the Router settings.



Routing can be based on many different criteria (see the list below). Several of them offer these generic options:

- **Treat as numeric value:** When you select this, the options in the **value** column offer numeric possibilities (is less than, is greater than, etc.). This option is not there when it is obvious that the value is numeric.
- **Case sensitive:** Select this if you want to match the exact case of the value.

Note: If a value has to **match**, wildcards and [Regular Expressions](#) can be used in the value field.

Routing Based on Job Parameter Value

Choose this to route your file based on a Job Parameter. The **Parameter Name** dropdown list offers the parameters of the Job setup tab **Parameters** and also those of the tab **Step&Repeat**.

Note: You can also type in the parameter. This is not advised to avoid typing errors. However this can be useful when you build the workflow and router settings outside of a Job context in which you will be using it. In that case the parameters will not show up in the dropdown list yet.

An example:

Route based on: Job Parameter Value		
Parameter Name: Trapping needed		
<input type="checkbox"/> Treat as numeric value	<input type="checkbox"/> Case sensitive	
Job Parameter Value	Route to	Add
contains yes	Trapping needed	Remove
contains no	No Trapping needed	Move Up
no match	No Trapping needed	Move Down

Notice how you do not always need an Error route and how same routes can be chosen for more than 1 value.

Routing Based on Job Field Value

Choose this to route your file based on one of the Job setup fields listed in the **Field Name** dropdown list. The list also includes the custom **Job Categories** you may have defined.

The field **Job Operation Context** can only have these 2 values: `update` or `creation`. This value was useful in cases where we now advise to use the **Create Job** task as a workflow step. Learn more in the [Advanced](#) on page 1286 tab of that task.

Routing Based on Number of Files

Choose this to route collections of input files based on the number of files in that collection.

Some examples:

- When you get not 1 but 2 files, you want to route to the recto/verso (front/back) workflow.
- You created various **Tabular Step & Repeat** tickets, one for each amount of **Grids** that they offer. You use the Router to route to the Step & Repeat ticket with the same number of grids as the amount of input files.

Routing Based on Milestone

Choose this to route based on the fact if a certain Milestone is **Set** or **Not Set**.

For example: you have a workflow where someone from the administration department uses the Pilot to select a Job and set the Milestone 'OK from Admin department to Print'. The Router checks if this OK was already given before continuing the workflow to the press.

The screenshot shows the Router configuration interface for routing based on a milestone. The 'Route based on:' dropdown is set to 'Milestone'. The 'Milestone:' dropdown is set to 'OK from Admin Department to Print'. Below this, there are two columns: 'Milestone' and 'Route to'. The 'Milestone' column has three rows: 'is Set', 'is Not Set', and 'no match'. The 'Route to' column has three rows: 'Print workflow', 'Wait for 1 hour and check again', and 'Error'. To the right of these rows are buttons: 'Add', 'Remove', 'Move Up', and 'Move Down'.

Routing Based on File Name

Choose this to route files based on their name. Choose to **Include the file extension** or not.

For example: You agreed with the person sending you design files that no trapping will be done if he ends the file name with 'notrap'. You created an Access Point that launches a workflow that starts with this Router:

The screenshot shows the Router configuration interface for routing based on file name. The 'Route based on:' dropdown is set to 'File Name'. There are two checkboxes: 'Case sensitive' and 'Include the file extension', both of which are unchecked. Below this, there are two columns: 'File Name' and 'Route to'. The 'File Name' column has two rows: 'ends with notrap' and 'no match'. The 'Route to' column has two rows: 'WFL without Trapping' and 'WFL with Trapping'. To the right of these rows are buttons: 'Add', 'Remove', and 'Move Up'.

Routing Based on File Type

Choose this to route files based on their type.

For example: this Router categorizes many file types into specific routes:

Route based on: File Type

File Type	Route to
is PDF File	Graphics file
is PDF Plato File	Graphics file
is Normalized PDF File	Graphics file
is PDF Station File	Graphics file
is ARD File	CAD file
is MFG File	CAD file
is A3D File	3D file
is VRML File	3D file
is U3D File	3D file
no match	Strange file type - Check

Routing Based on File Size

Choose this to route files based on their size.

Select **Consider the combined size of all files in the collection** if you want to match the size of the whole file collection and not of its individual files.

See an example in the above introduction.

Routing Based on Number of Pages

Choose this to route files based on their number of pages.

Select **Consider the combined page count of all files in the collection** if you want to match the page count of the whole file collection and not of its individual files.

For example: if the file has 2 pages, then you know it is a recto/verso (front/back) file.

Routing Based on Page Orientation

Choose this to route files based on their orientation (**Portrait, Landscape** or **Square**).

Some examples:

- your RIP does not have an auto-rotate function. Your workflow will then need to make sure that you send the data in the wanted orientation.
- you want to know if a file fits on an output sheet. Before you compare the sizes, you will want to know which side is the longest. If your output sheet is 100 horizontal by 60 vertical, you best only compare sizes when your input file is also in landscape mode.

Routing Based on Page Size

Choose this to route files based on their page size.

This is typically used to route files to the output device with the right size. For example you want to send small files to the small proofer, and bigger ones to the big proofer.

In **Pagebox to use**, you can choose either **Trimbox** or **Mediabox**. For images both settings Trimbox and Mediabox will have the same result. The image size will be used for the comparison.

Note: This does not take the page orientation into account. Both page orientations will always be checked. For example 'Page size' 'is equal to' '210 x 297 mm' will match both portrait and landscape A4 files.

Routing Based on Number of Separations

Choose this to route files based on their number of separations.

For example: when a flexo job has more than 6 inks, you use a different SmartMarks set. It then uses one with smaller rectangles for the color strips. You use a Router that checks the number of separations. If it detects more than 6 inks, the next step is the Step & Repeat ticket that uses the set of SmartMarks with the smaller rectangles.

Select **Consider the combined separation count of all files in the collection** if you want to match the number of separation of the whole file collection and not of its individual files.

Note: This counts the number of **unique** separations. For example, if you have a CMYK+Red and a CMYK+Green file, the collection's combined separation count will be 6.

Routing Based on Separation Name

Choose this to route files based on the name of their separation(s).

You can check if **any** (one or more), **all** or **none** of the separation names match or contain parts of a specified string.

For example:

- when 'any' separation names 'contain' the text string 'pantone', then route to the part of the workflow where you process the files that are not CMYK-only.
- when 'all' separation names 'does not contain' the text string 'Die', then route to the **Checkpoint** task that will add an Action to the To Do List. The Action will ask the user to add that separation.

Routing Based on Layer Name

Choose this to route files based on the name of their layer(s).

For example: you sometimes receive PDFs with several layers of black text in different languages. The Router checks if the file has any layers with these names: FR, SP, EN, DE, IT, or PT. If so, it routes them to the workflow where you first use **Version PDF** to create PDFs with only 1 language in black (k): name_cmy_FR_k.pdf, name_cmy_SP_k.pdf, etc.

You can check if **any**, **all** or **none** of the layer names match. When using **any**, only one layer name needs to match for the file to be considered matching.

Routing Based on SmartName Value

Choose this to route files based on a SmartName value.

For example: the Router checks the workflow parameters 'Print Process' or 'Preflight Yes or No'. The values of these parameters decide what route the file needs to follow.

Find a good example in [Launching a workflow with parameters from an XML](#) on page 1319. Step 5 of that example illustrates how the Routers use SmartNames.

Note:

Routing on SmartNames requires more resources from Automation Engine. If you have an alternative, we advise you to use another routing method instead.

If your SmartName only contains **Job** related parameters, and you have many input files, we advise to add a *Data Collector* in front of your **Router**. This will group the files into a collection and allow the Router to perform the SmartNames check only once.

6.3.6 Mark File and Select Marked File

These 2 workflow controls are used as a pair. First you use **Mark File** to tag files, then you use **Select Marked File** to select those files later in your workflow.

You can have several pairs of **Mark File** and **Select Marked File** in your workflow, as long as each pair has its own tag.

Mark File Settings

In **Mark all incoming items with tag**, enter the tag to set on all files passing through this workflow control.

Select Marked File Settings

In **Select all items with tag**, use the dropdown list to select a tag that you entered earlier in the corresponding **Mark File** workflow control.

You can filter the selection of tagged files. Select **Only select when name matches** and enter what name the file should match. You can use SmartName(s) and the asterisk * as a wildcard.

If you are working with subworkflows, you can choose to **Also search workflows containing this workflow** (to search 'up'). Learn more about subworkflows [here](#).

Example

A typical example is launching tasks on the one-up PDF after your workflow was already working on the Step & Repeat version. In this case, you can tag the one-up file with **Mark File**, later select it with **Select Marked File** and then for example archive it:



Note: You need to use the exact same tag in the **Mark File** and the **Select Marked File** step. It is case-sensitive, so we advise to always use the dropdown list.

6.3.7 Data Collector

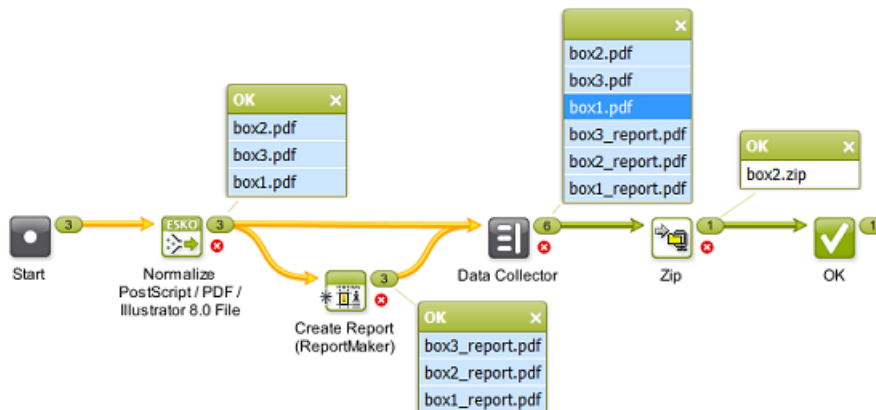
Use the **Data Collector** workflow control

- to collect all files that travel separately in the workflow and then send them all together to the next step.
- to wait for all files before continuing the workflow (without grouping them)

Example

In this example, the Data Collector waits for all of the files to arrive (one-ups and their reports), and then sends them as one **group** to the **Zip** task.

The result is one ZIP file containing both the one-ups and their reports.



Settings

- **Error Policy.** Decide what needs to happen when errors appeared earlier in the workflow, before this Data Collector: **Fail when errors occur** or **Ignore errors**.
- **Grouping**

Group all collected data is active by default. In the above screenshot, notice how a mouse-click on the `box1.pdf` in the output widget of the Data Collector, also highlights all the other members of the group in light-blue (and as usual also the related files in previous steps).

Note: Such a relation also means that they will share the same Workflow Parameters. Learn more [here](#).

Switch this function off when you want to use Data Collector as a wait point in your workflow. It then only serves to hold the workflow until all previous steps are finished and then continues the workflow with the same input (collections) as they entered the Data Collector.

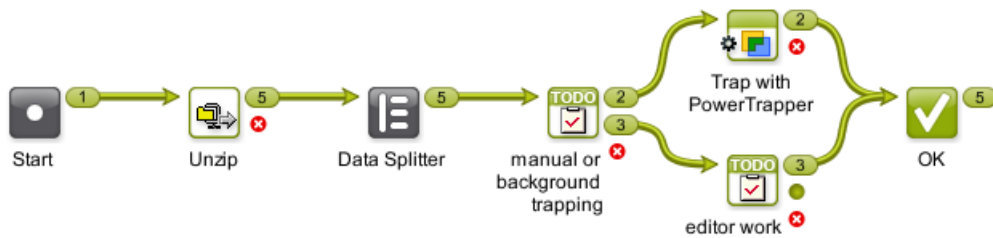
Note: When the Data Collector does not **group**, the Workflow Parameters will not be grouped either. They remain attached to their specific file set. Learn more [here](#).

6.3.8 Data Splitter

Use the **Data Splitter** if you want the next step in the workflow to process each file individually.

Example

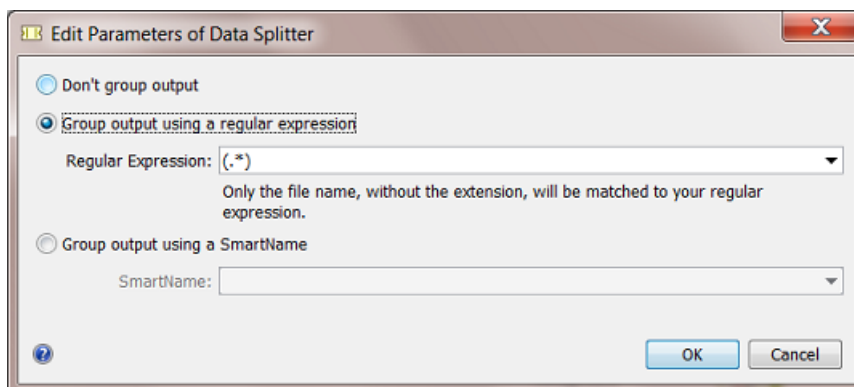
In this example, the Data Splitter (with the default setting **Don't group output**) sends the unzipped files to the **Checkpoint** task one by one. This way, 5 To-Do Actions are created. The user needs to decide for each file if it needs manual or background trapping:



Without the Data Splitter in this workflow, there would only be one To-Do Action. The Action item would provide a link to the 5 files, but they would need to share the same decision.

Split and Create a (new) Group

You can also use the Data Splitter to split the collected files and **group** them according to a criterion you define.



You can group the output by using a Regular Expression or by using a SmartName.

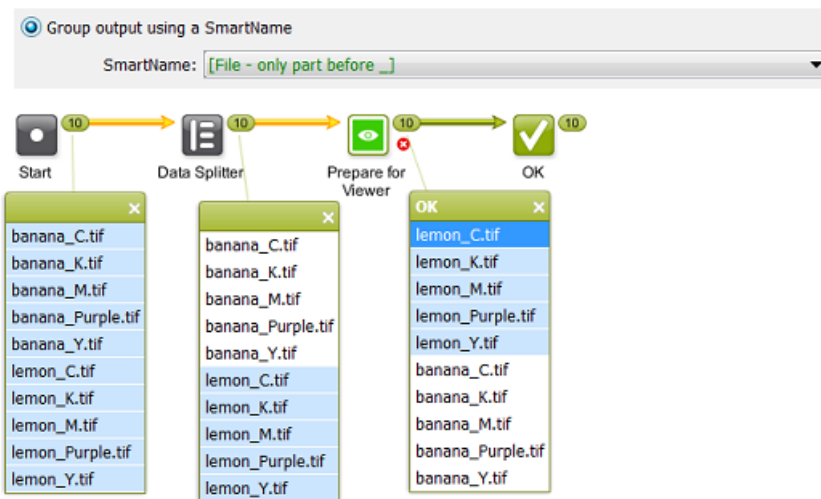
Two default regular expressions are available from the dropdown list:

- `(.*)` matches the entire file name without the extension
- `.*_pag([0-9]+).*?` matches the number directly after `_pag` in the file name (use this for example for files named `page_1.pdf`, `pagina_2.pdf`, etc.).

Only the file name will be matched, without the extension. Learn more about regular expressions in [Using Regular Expressions](#) on page 978.

Example: Group RIP output to Prepare for Viewing

The Automation Engine **Viewer** enables viewing of RIP'ed data when they were **prepared for viewing** together, as a group. When you have a folder containing RIP output of several jobs, it is important that you send the right groups of separations to the **Prepare for Viewing** task. You can use the Data Splitter and group the separations by using a SmartName.



Note: you can use the same principle in a workflow where you use the **Create Wrapper file** task.

Example: Group based on the Name of the Substrate

To automate the nesting process in an i-cut workflow, you can produce a grouped output using **Xpath Query SmartNames** mapping to the name of the Substrate. That way, all data under the same Substrate will be produced as one output for the nesting.

6.3.9 Fork and Join

The **Fork** workflow control automatically tags

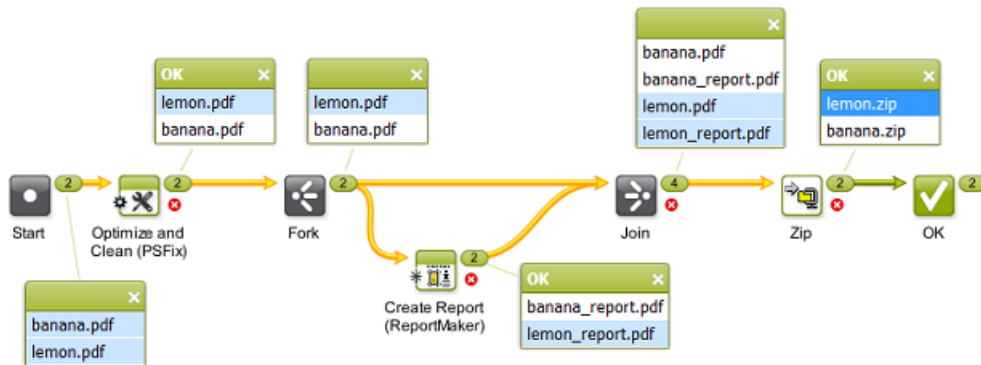
- every file running through it
- and all the files that those files generate in the next steps (for example Report files)

This creates a **Group Tag**.

The **Join** workflow control then groups all the files with the same tag.

Example of Group Tagging

Our example workflow creates a ZIP containing the one-up PDF and its report PDF. This is easy to set up when the workflow starts with only one file. This is different when your workflow starts with multiple files:



In this example,

- we have 2 input files that will travel through this workflow together
- the Fork tags both the input files and the 2 report files created from those input files
- the Join creates 2 groups, each with the input file and its report file.
- these groups are the inputs for the ZIP task. You now have a ZIP per group.

Without the Fork and Join steps, you get 4 ZIP files: one for every file. If you use a Data Collector to group all of them, you get 1 ZIP of all 4 files.

6.4 Launching and Diagnosing Workflows

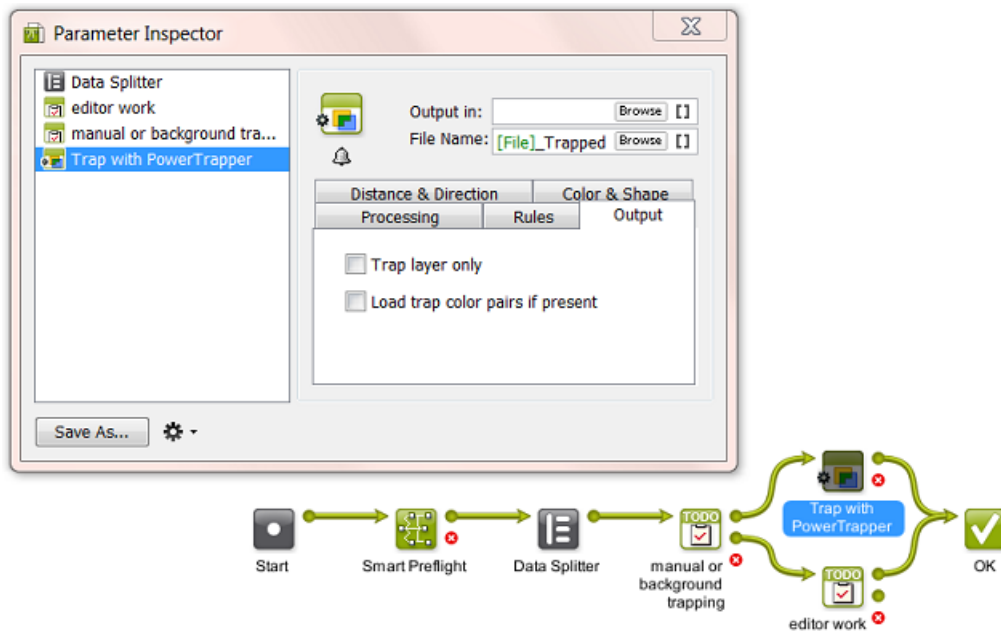
6.4.1 The Parameter Inspector

The **Parameter Inspector** enables you to quickly view or change the ticket parameters of your workflow steps, without having to open and close each workflow step separately.

Open it by choosing **Show Parameter Inspector...** from the workflow editor **File** menu or by choosing the same option after a right-click on the canvas.

Note: The parameter inspector is not available for workflows that were already launched (running or finished).


The window stays open while you navigate through the steps. The steps are shown in alphabetical order. You can navigate through the steps by selecting them in the parameter inspector or by clicking on steps in the workflow editor canvas.



The parameter inspector does not show

- steps that have no parameters (Start - Stop - Warning - Error - Cancel)
- subworkflow steps. Double click the subworkflow step on the canvas to see its own steps appear in the same parameter inspector window.
- referenced steps.

When available, you can

- choose **Save As...** to save the selected ticket as a separate task ticket.
- choose  to set extra options that you can also set when you open the ticket separately: [Notification Rules](#), [Manage Public parameters](#) and [Map Output Parameters](#).

6.4.2 Launching a Workflow on a File

There are several ways to launch a workflow on a file: from the Pilot, from the Workflow Editor, from [Shuttle](#), via Access Points or via JDF/JMF.

Note: Learn more about launching workflows from external systems in [Integrating with External Systems](#) on page 1228.

Launching a workflow from the Pilot's **Files** View is very similar to launching a single Ticket. Follow these steps:

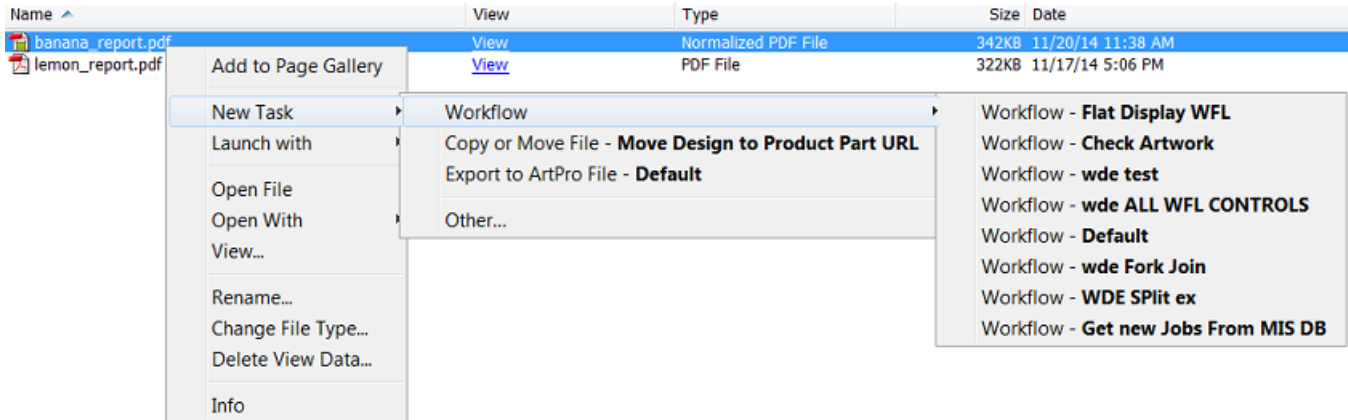
1. In the **Files** view, right-click the file(s) you want to process and select:
 - **New Task**. You can then select the workflow and change its settings before launching it.
 - **Launch with**. You can then select the workflow and launch it without changing its settings.

Note:

If you don't have the access right **Tickets: Show All Tickets and their Parameters (Public and other)**, you will only see **Launch with**. You will also only be able to select public workflow tickets. Learn more in [Access Rights](#) on page 702.

2. Select the workflow ticket to use.

- If you have launched workflows previously, the most recent workflow tickets you used are listed under **Workflow**.

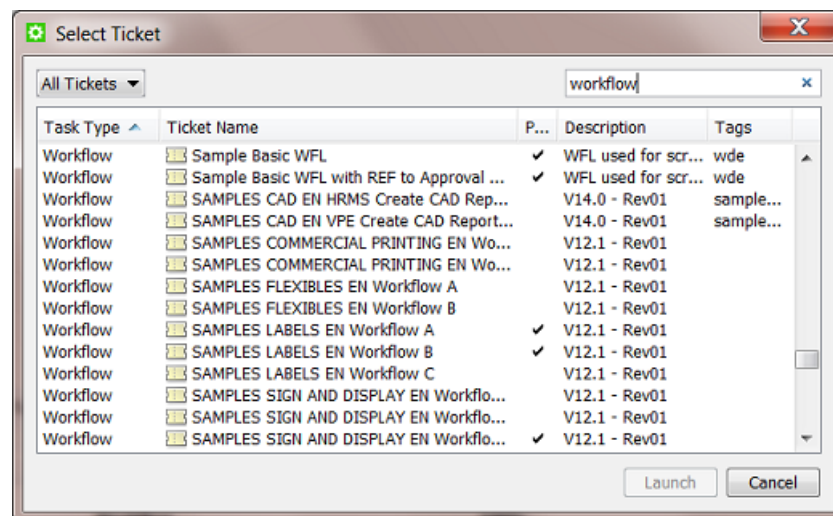


- If you don't need a recently used workflow, select **Other...**



In the **Select Ticket** dialog that opens, select the workflow ticket you want to use and click **OK**.

Tip: Type `Workflow` in the search field to quickly filter to the task type **Workflow**.

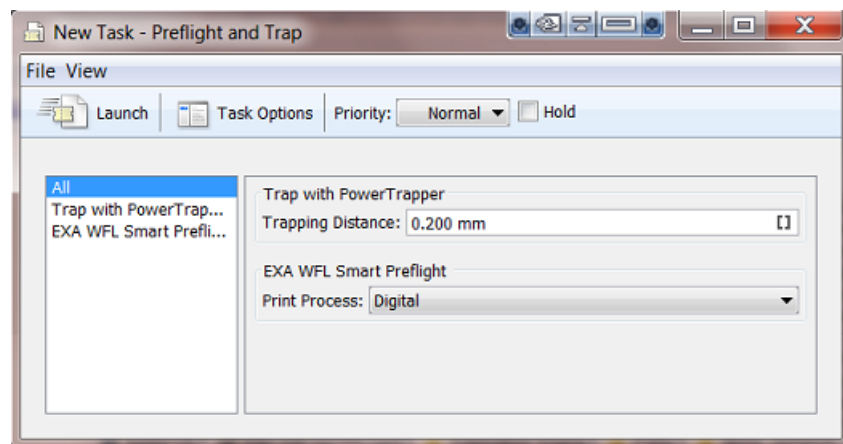
Tip: If you hold the **Alt**, the **OK** button changes into **Launch**.



3. Some variations are possible:

- To keep the workflow editor window open and monitor the workflow while it is executing, hold **Alt** while clicking Launch to **Launch and Monitor** (the Launch button changes to ).
- If you choose **New Task**, the workflow editor opens. Fill in the settings you want to use in each workflow step and click the Launch  button.
- If you choose **Launch with** and you don't have the access right **Tickets: Show All Tickets and their Parameters (Public and other)**, then
 - if the ticket does not contain public parameters, the workflow is launched on the file as soon as you select the workflow ticket.
 - if the ticket contains public parameters, it will open for you to fill them in. You will only see the public parameters. Click the **Launch** button when you are done.

In this example the user is asked to decide 2 public parameters:

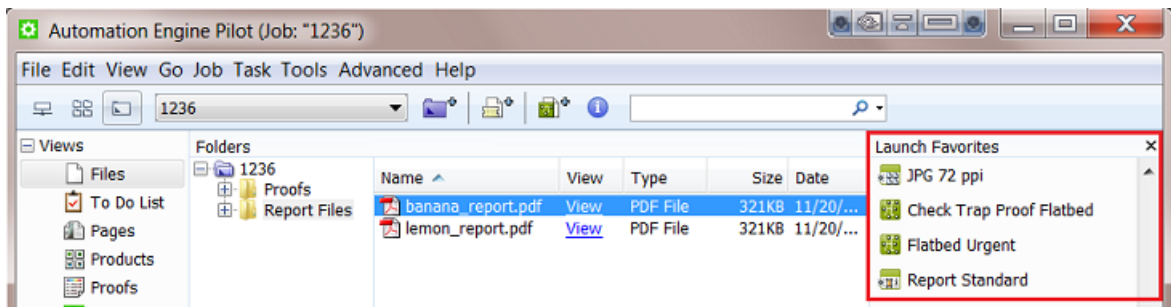


Note: If a workflow is launched on files that are linked to several Jobs, a separate workflow will be launched for each Job context. This can happen for example when you use the **Files** view or Shuttle or a Folder Access Point.

6.4.3 Using Favorite Workflow Tickets

Note: In [Favorite Tickets](#) on page 179, you can read how to build and launch **Favorite** tickets for single task tickets. You can do the same for workflows tickets. We here repeat some information and add specifics about working with favorite workflows.

Favorite tickets are links to tickets that you use a lot. They show up in the **Favorites** pane, both in the Files and in the Tickets View. You can use **Pilot > View > Favorites** to show or hide the **Favorites** pane.



Favorites tickets show up depending on

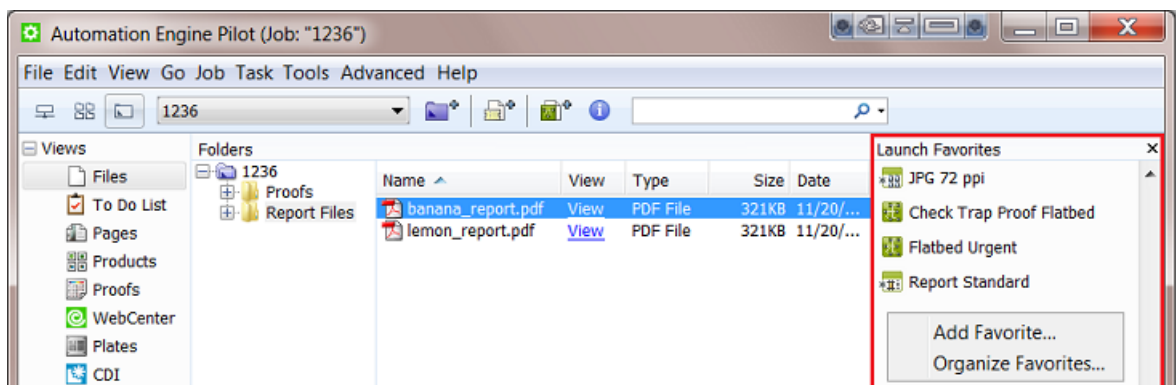
- the file type you selected first.
- the user that is logged in. They are your personal favorites.

Tip: You can copy and paste favorite tickets from one user to another. In **Pilot > Tools > Users**, right-click a user name and choose **Copy** or **Paste Favorites**.

Adding Favorite Workflow Tickets

There are 3 ways to create a favorite (workflow) ticket:

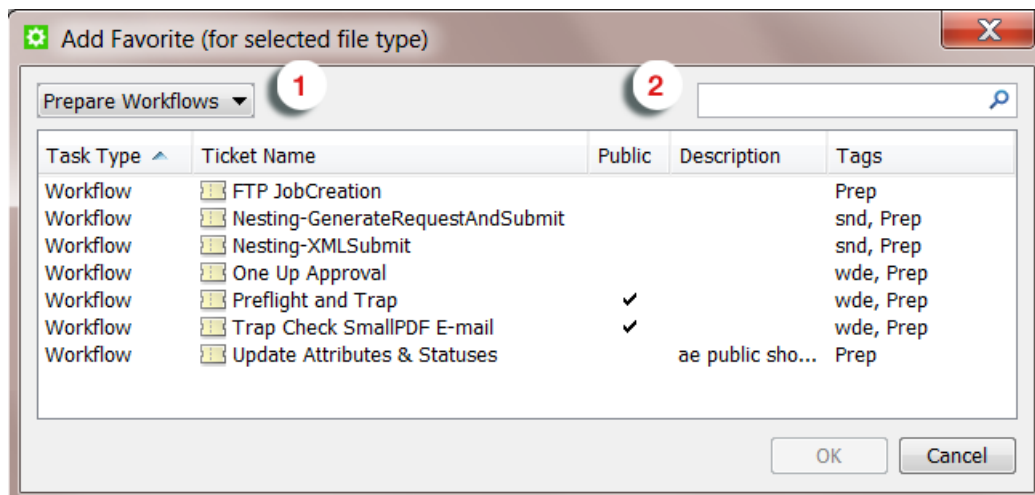
- when you first save a workflow or use 'save as', then check **Add to Favorites**.
- in the Tickets View, right-click a ticket and select **Add to Favorites**. Use **Shift** to select multiple tickets.
- both in the Files and in the Tickets View, in the **Favorites** pane, right-click and choose **Add Favorite....**



The Add Favorite Dialog

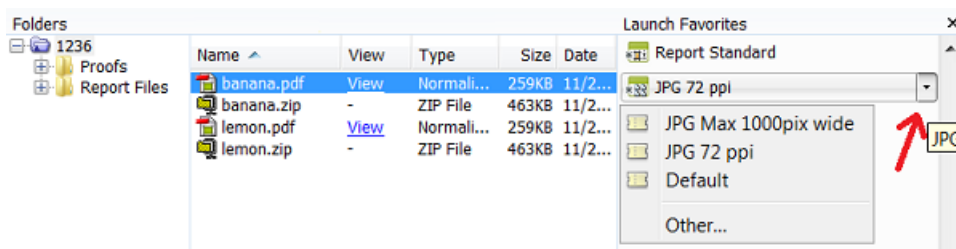
When selecting a (workflow) ticket to add as a favorite, these 2 tools help you filter your list of tickets:

1. Instead of seeing **All Tickets**, use the dropdown list and select one of the tickets filters that you created earlier and saved into your **Smart Views**. Learn more about Smart Views [here](#).
2. Type in a filter in the search field. What you type will be used to filter in all the columns.



There is No Need to add Many Favorites of Same Task Type

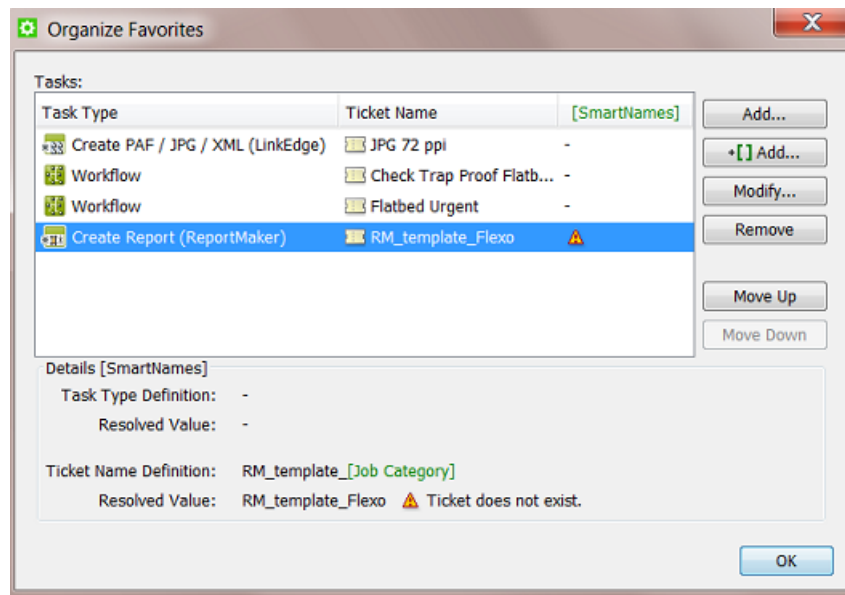
When you click on the right side of the favorite, a dropdown list will show you all tickets of the same task type (with a maximum of 10). This easy access to similar tickets means that you do not have to create a favorite of those other similar ones (when you also use them often).



When the ticket that you look for is not in that list (remember that maximum 10 are shown), you can choose **Other...** and select it there.

Organizing Favorite Workflow Tickets

In the **Favorites** pane, right-click and choose **Organize Favorites...** to organize favorites and to create favorites based on SmartNames.



- Click **Add...** to add a favorite by selecting it from the list of tickets.
- Click **+ [] Add...** to add a favorite based on a SmartName. Find instructions below.
- Click **Modify...** or **Remove** to modify or remove a favorite.
- Click **Move Up** or **Move Down** to change the order in which the favorite will appear in the favorites pane.

Adding a Favorite based on a SmartName

Click **+ [] Add...** to add a favorite based on a *SmartName*.

- Define the **Task Type**. Select it from the dropdown list or specify the type by using SmartNames.
- Define the **Ticket Name**. Choose the **Scope** and define the name by typing in the name and optionally using SmartNames in that name.

See this example where your favorite ReportMaker ticket is partially defined by the **Job Category**: In the screenshot above you can see the feedback when the Resolved Value does not match. It is good practice to use the **Check** button to check that value before clicking **OK**.

Launching Favorite Workflow Tickets

There are various ways to launch a favorite (workflow) ticket:

Launch Immediately

1. Select file(s).
2. Hold the **Alt** button.
3. Click on your favorite. The task is launched.

Choose a Ticket of the Same Task Type and Launch Immediately

1. Select file(s).
2. Hold the **Alt** button.
3. Click on the right-side of a favorite to open the dropdown list of tickets of the same task type.
4. Move your mouse down to the chosen ticket.
5. Click on the chosen ticket. The task is launched.
6. Release the **Alt** button.

Open the Favorite Ticket in the Workflow Editor

1. Select file(s).
2. Click on your favorite.
3. In the workflow editor, check or change the settings and then launch or 'launch and monitor'.

Choose a Ticket of the Same Task Type and Open it in the Workflow Editor

1. Select file(s).
2. Click on the right-side of a favorite to open the dropdown list of tickets of the same task type.
3. Move your mouse down to chosen ticket.

4. Click on the chosen ticket.
5. In the workflow editor, check or change the settings and then launch or 'launch and monitor'.


6.4.4 Building and Launching a Workflow On The Fly

If you build a workflow from the **Tickets** view, you cannot launch it until you finished building it. Your Workflow Editor will not have a **Files** pane to select files and then launch it. Working this way is typically something done by Automation Engine administrators while building the workflows. Once the workflow is ready for testing, users can then go to the **Files** view, make a file selection, select the workflow ticket and launch it.

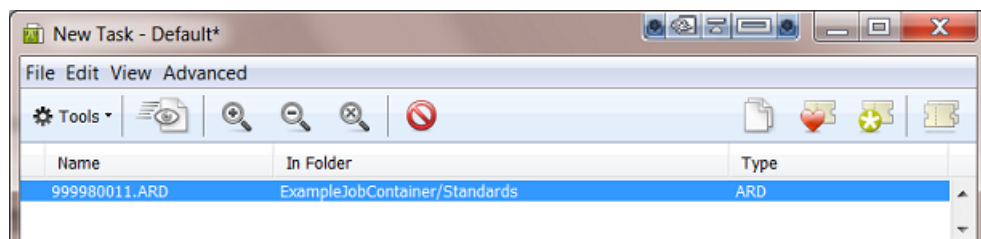
However, you do not always have to prepare a workflow ticket before you can launch it. It is possible to build the workflow and launch it on a file without first saving the workflow ticket. Follow these steps to do this:

1. In the **Files** view, select the file(s) you want to launch your workflow on.

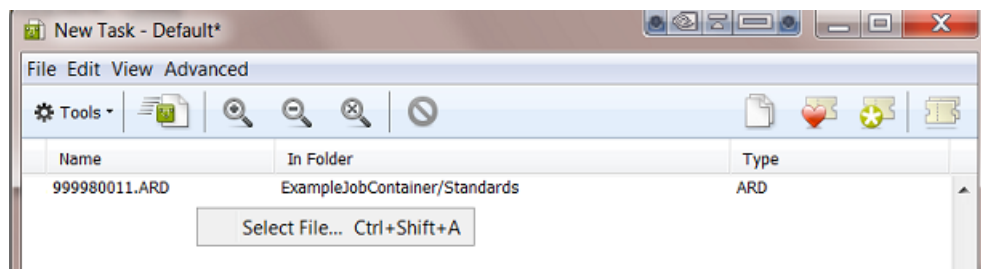
Note: If you need to select files from different (Job) folders, then first select those in one same folder, then choose **New Task** or **New Workflow** to open a Workflow Editor and there use the **Select File** tool to select the files from any other folders (see step 4).

2. Click  to open the workflow editor.


You can see the file(s) you selected in the **Files** pane of the Workflow Editor.



3. Build your workflow and define the settings of every step.
4. If you want, select more files. Right-click in the **Files** pane and choose **Select File...**



In the dialog that appears, select files from folders in **Jobs** or **Containers**.

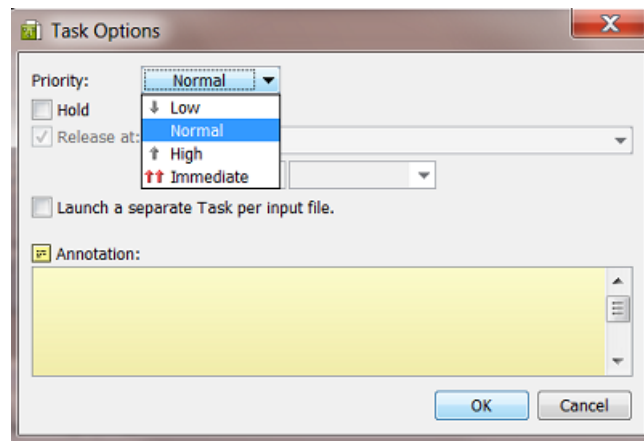
5. Click the Launch  button. Hold **Alt** to  **Launch and Monitor** and keep the workflow editor open.

- If you kept the workflow editor open, continue by changing the workflow settings, [relaunching](#) or launching on different files. When ready, save your workflow ticket.

6.4.5 General Workflow Task Options

Task Options are options that will be used when you push the **Launch** button in the workflow editor. These settings are not saved into the workflow ticket.


Open **Task Options** from the menu **Advanced** in the workflow editor:



- Priority:** Choose the general priority with which you want to launch this workflow. The default is **Normal**.

Note: To choose **High** or **Immediate**, you need a special [User Access Right](#).

Note: You can also choose to set the priority of only a part of the workflow. Learn more in [Set Priority](#) on page 1021.

- Hold:** Check this when you want to hold the workflow when you launch it. The workflow will start but immediately go into 'pause' mode. You can then later decide when to use the  button to start it for real.
- Release:** When you decided to use **Hold**, you can set a time when the workflow should be released automatically. Choose a relative or absolute time.
- Launch a separate task per input file:** Choose this option when you selected multiple input files but when you still want to launch a separate workflow task per input file.
- Annotation:** When testing many workflow tasks or many versions of a workflow, it can be useful afterwards to read some notes that you added before launching it. Once the task was launched, you can read these annotations from this same menu or you can read them in the Tasks view when you right-click the main workflow task line.

For example: "This task was only launched to create the temporary files that I need to test the next one"

6.4.6 Relaunching Your Workflow with Different Settings

You can relaunch a workflow from the **Tasks** view and from the **Workflow Editor** (skip step 1):

1. In the Tasks view, double-click the main **Workflow** line to open your workflow in the workflow editor.

1236	banana.pdf,banana_report.pdf	Zip	100%	✓	11/20/14 4:20 PM
1236	lemon.pdf,lemon_report.pdf	Zip	100%	✓	11/20/14 4:20 PM
1236	lemon.pdf	Create Report (ReportMaker)	100%	✓	11/20/14 4:20 PM
1236	banana.pdf	Create Report (ReportMaker)	100%	✓	11/20/14 4:20 PM
1236	lemon.pdf	Optimize and Clean (PSFix)	100%	✓	11/20/14 4:20 PM
1236	banana.pdf	Optimize and Clean (PSFix)	100%	✓	11/20/14 4:20 PM
1236	banana.pdf,lemon.pdf	Workflow	100%	✓	11/20/14 4:20 PM

Tip: The workflow can be opened and relaunched even when it is still running.

Tip: If you double-click any sub-step of the workflow, only that ticket will open. You can also relaunch just that separate single step (if the input files are still available).

2. In the workflow editor window, click the **Edit Workflow** button  if you want to make any changes.

When you are finished making changes, and if you want to keep the new settings before you try them out, save your workflow now.

Note: As soon as you make one change, the name of the workflow ticket in the top bar will show an asterisk * behind the name to indicate that it has been changed.

3. Click Launch  or press **Alt** while clicking Launch to **Launch and Monitor** .

6.4.7 Diagnosing a Workflow in the Tasks Pane

You can check the processing status of a workflow in the Pilot's **Tasks** pane. The **Tasks** pane appears in the **Files**, **Products** and **Tasks** View.

Job Name	File Name	Task Type	Progress	Phase	State	Launched
1236	lemon.pdf	Create Report (ReportMaker)	0%	Reading \\...	▶	11/20/14 4:30 PM
1236	banana.pdf	Create Report (ReportMaker)	0%	Reading \\...	▶	11/20/14 4:30 PM
1236	lemon.pdf	Optimize and Clean (PSFix)	100%		✓	11/20/14 4:29 PM
1236	banana.pdf	Optimize and Clean (PSFix)	100%		✓	11/20/14 4:29 PM
1236	banana.pdf,lemon.pdf	Workflow	0%	Running	▶	11/20/14 4:29 PM



Attention: The **Tasks** pane does not show the **Workflow Controls** that may be part of your workflow. To diagnose them, use the Workflow Editor.

Workflow Status

The workflow status can be

- ▶: running, when one or more steps are waiting to be handled.
- ✓: success, when all steps finished successfully.
- ⚠: warning, when one or more steps finished with a warning.
- ✗: failed, when one or more steps finished with an error.
- ⏹: cancelled, when a user cancelled the processing.

Right-Click and Diagnose Further

Right-click any task line to see many functions available for that task or its related files.

The availability of the functions depends on

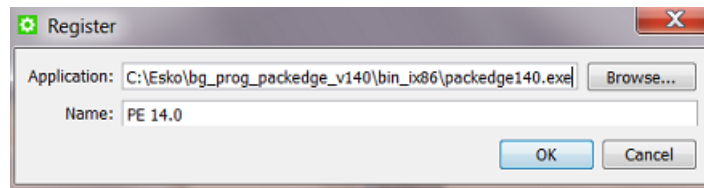
- if the task is already finished or not
- if the task is the initial 'workflow' task type or any sub-step of that workflow.

Job Name	File Name	Task Type	Progress	Phase	State	Launched
1236	lemon.pdf,lemon_report.pdf	Zip	100%		✗	11/20/14 4:30 PM
1236	banana.pdf,banana_report.pdf	Zip	100%		✗	11/20/14 4:30 PM
1236	lemon.pdf	ReportMaker	100%		✓	11/20/14 4:30 PM
1236	banana.pdf	ReportMaker	100%		✓	11/20/14 4:30 PM
1236	lemon.pdf	an (PSFix)	100%		✓	11/20/14 4:29 PM
1236	banana.pdf	an (PSFix)	100%		✓	11/20/14 4:29 PM
1236	banana.pdf,lemon.pdf		100%		✗	11/20/14 4:29 PM
1236	banana.pdf,banana_report.pdf		100%		✓	11/20/14 4:20 PM
1236	lemon.pdf,lemon_report.pdf		100%		✓	11/20/14 4:20 PM
1236	lemon.pdf	ReportMaker	100%		✓	11/20/14 4:20 PM
1236	banana.pdf	ReportMaker	100%		✓	11/20/14 4:20 PM
1236	lemon.pdf	an (PSFix)	100%		✓	11/20/14 4:20 PM
1236	banana.pdf	an (PSFix)	100%		✓	11/20/14 4:20 PM
1236	banana.pdf,lemon.pdf		100%		✓	11/20/14 4:20 PM
WDETESTS	78999_ABC_report.pdf		100%		✓	11/20/14 4:08 PM
WDETESTS	99998011.ARD		100%		⚠	11/20/14 3:13 PM
WDETESTS	78999_ABC_report.pdf		100%		⏹	11/20/14 2:25 PM
1237	South.pdf	Checkpoint)	100%		✓	11/20/14 1:48 PM
1237	South.pdf		100%		✓	11/20/14 1:48 PM
1237	South.pdf		100%		✓	11/20/14 1:48 PM
WDETESTS	78999_ABC.pdf	Checkpoint)	100%		✓	11/20/14 1:48 PM
WDETESTS	78999_ABC.pdf		100%		✓	11/20/14 1:48 PM
WDETESTS	78999_ABC.pdf		100%		✓	11/20/14 1:48 PM
WDETESTS	banana_report.pdf	Checkpoint)	100%		✓	11/20/14 1:28 PM
WDETESTS	banana_report.pdf		100%		✓	11/20/14 1:28 PM
WDETESTS	banana_report.pdf		100%		✓	11/20/14 1:28 PM
WDETESTS	banana_report.pdf	Checkpoint)	100%		✓	11/20/14 1:27 PM
WDETESTS	banana_report.pdf		100%		✓	11/20/14 1:27 PM
WDETESTS	banana_report.pdf		100%		✓	11/20/14 1:27 PM
WDETESTS	banana_report.pdf	base	100%		✗	11/20/14 12:14 PM

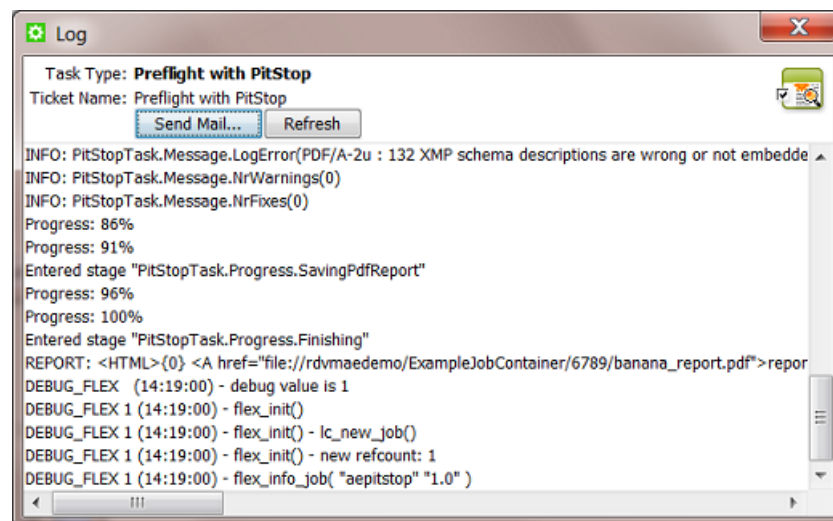
Many of these functions are documented in [Task Menu](#) on page 168.

Some specific notes:

- **Open Output File.** This opens the output file in the default application for that file type on your computer. For example: all PDF files are opened in Adobe Acrobat Reader.
- **Open Output File With.** This allows to chose from applications that you registered. For example, this is how you register a locally installed PackEdge 14.0 :



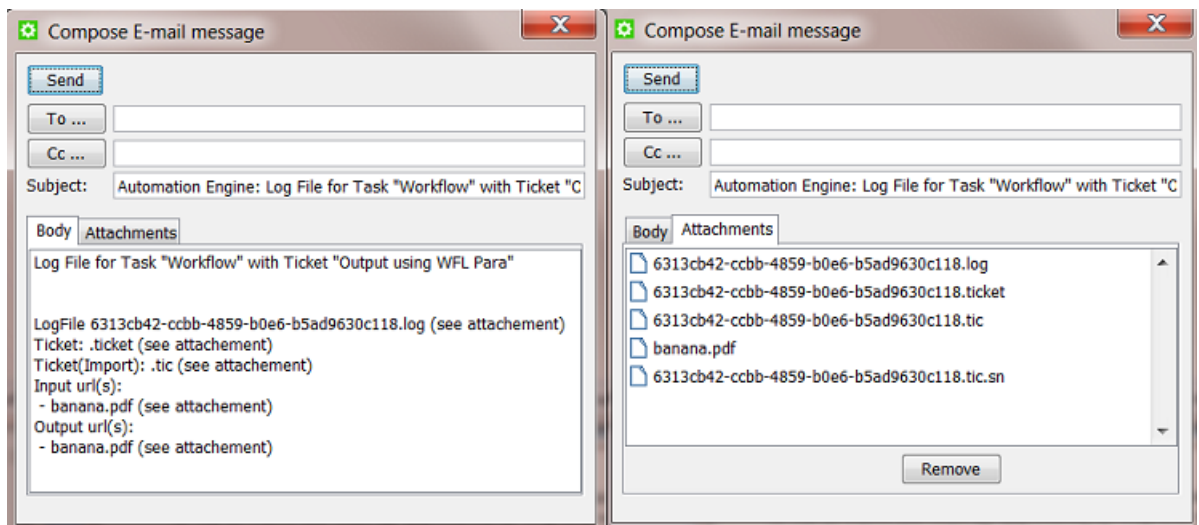
- **Delete.** If you decide to manually delete tasks, then you best delete the main workflow task step so that all related task steps are also cleaned up. In general we advise to use the automated version based on the standard **Task Cleanup Rules** that you set up in the Configure tool.
- **Move to History.** Same comments as above for the **Delete** function. Learn more about storing task details in [Task History](#) on page 621.
- **Log Info.** The Log file contains the most detailed information about the processed task. You will not only find the standard task **Details** (see also below) but also progress status and complex technical debugging information. This level of detail can be interesting when searching for the cause of errors. It is mainly meant for Esko service people.
- **Open....** This shows the log information of the selected step in the **Log** dialog. You can send an E-mail containing this log file and attachments (see below) or, when the task is still running, **Refresh** to see the latest content.



- **Open with** allows to open the log file in a text editor installed on your computer. **Register** that application in the same way as we explained for the option **Open Output File With** (see above).

Note: This requires the Automation Engine client to have access to the share 'AutomationEngineLogging' on the Automation Engine Server.

- **E-mail.** The **Compose E-mail message** dialog allows to send this log info in an E-mail, using the E-mail configuration as set up in **Tools -> Configure**. Fill in the E-mail address(es) and change the body message as you wish. By default all relevant files are already attached (remove any if you want to). The files with extension `.ticket`, `.tic` and `.sn` are file types that Esko customer service can read.



- **Details** shows you a summary of the log file with information meant for the user. It is also translated. It can also contain links to related files. Here is an example of the details of the preflight task that offers a direct link to the preflight report file:

File Name	Task Type	Progress	Phase	State	Launched
banana.pdf	Trap with PowerTrapper	100%		✓	12/2/14 2:19 PM
banana.pdf	Preflight with PitStop	100%		✗	12/2/14 2:19 PM
banana.pdf	Workflow	100%		✓	12/2/14 2:18 PM


Details	
Task Type:	Preflight with PitStop
Started on Server:	rdvmaedemo
Started at:	12/2/14 2:19 PM
Finished at:	12/2/14 2:19 PM
Process Time:	0:00:04
Input file:	file:///rdvmaedemo/ExampleJobContainer/6789/banana.pdf
Number of errors:	6
Error:	PDF/A-2u : PDF/A Output Intent is missing
Error:	PDF/A-2u : Non- PDF/A-2 compliant compression filters have been found
Error:	PDF/A-2u : PDF/A version is not PDF/A-2u
Error:	PDF/A-2u : CMYK is used while Output Intent is not defined (106x on page 1)
Error:	PDF/A-2u : Halftone is not PDF/X compliant (498x on page 1)
Error:	PDF/A-2u : 132 XMP schema descriptions are wrong or not embedded in the metadata streams
Number of warnings:	0
Number of fixed issues:	0
Report of the PitStop engine check (PDF)	report


Note: In the Tasks View, you default get the **Details** pane.

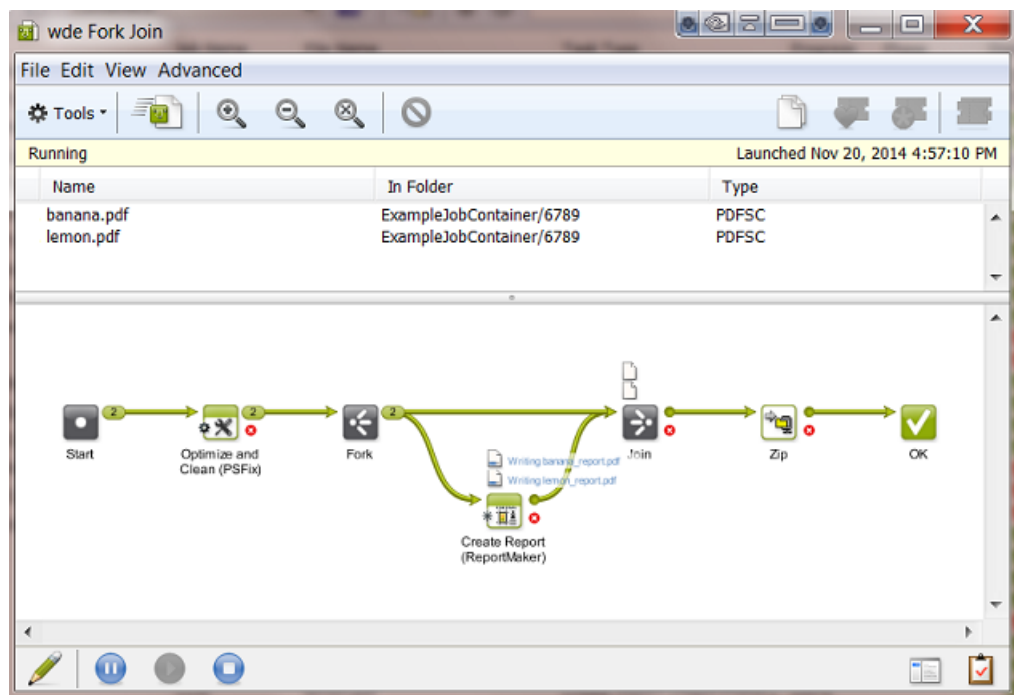
- **Group Workflows.** If you do not filter the view by the column '**Launched**', then tasks of a same workflow are not shown together. Use this function to show them in a group anyway.

6.4.8 Diagnosing a Workflow in the Workflow Editor

General View


In the workflow editor, you can see your file(s)  going from step to step, with their processing **Phase** displayed in blue text.

Note: This icon  can represent a single file or a collection of related files.



- Click  to see the **Details** of the main workflow task (names of output files, stranded files, reading of workflow parameters...).

Tip: If you want the details of a specific workflow step, then first select one of the output files of that step and then right-click and choose **Details**.

- Click  to see the **To Do** Action items associated with this workflow.

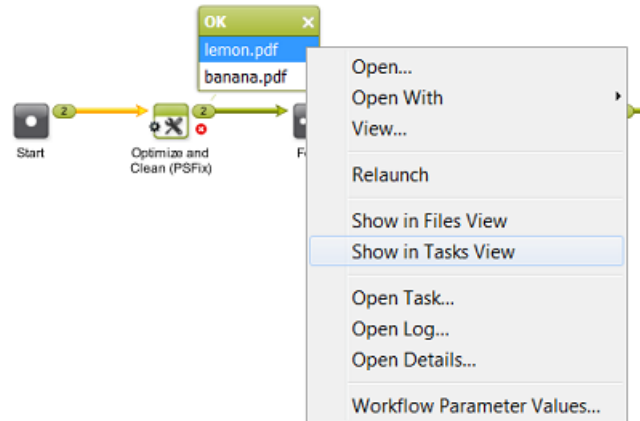
Output Pins and Widgets


A number in the output pin of a step indicates the number of files that ran through it.



Click on the number to open a widget showing the file names.

Inside the widget, right-click the name of a file to see more options. Many of these options were explained in [Diagnosing a Workflow in the Tasks Pane](#) on page 1043.

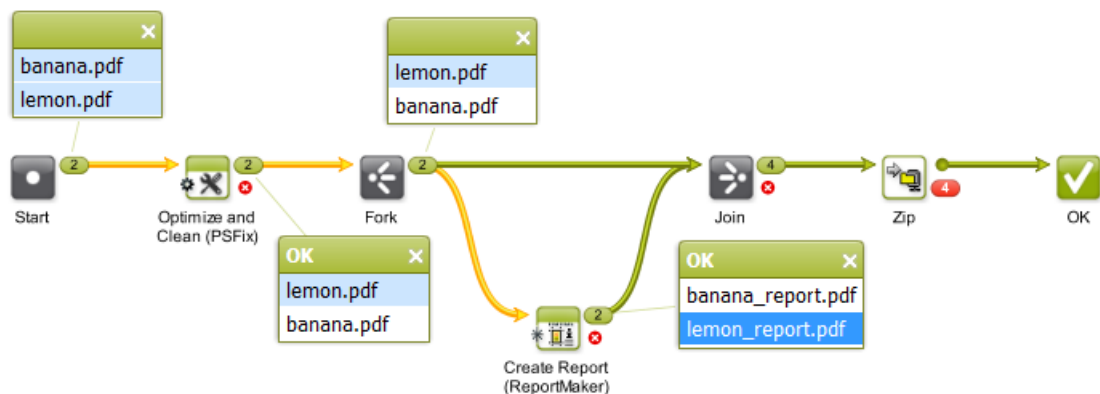


Note: You can also right-click the icons  of files that are still being processed and get more options there.

Note: To check the **Workflow Parameter Values** of a (collection of) file(s): learn more in [Using Workflow Parameters](#) on page 1065.

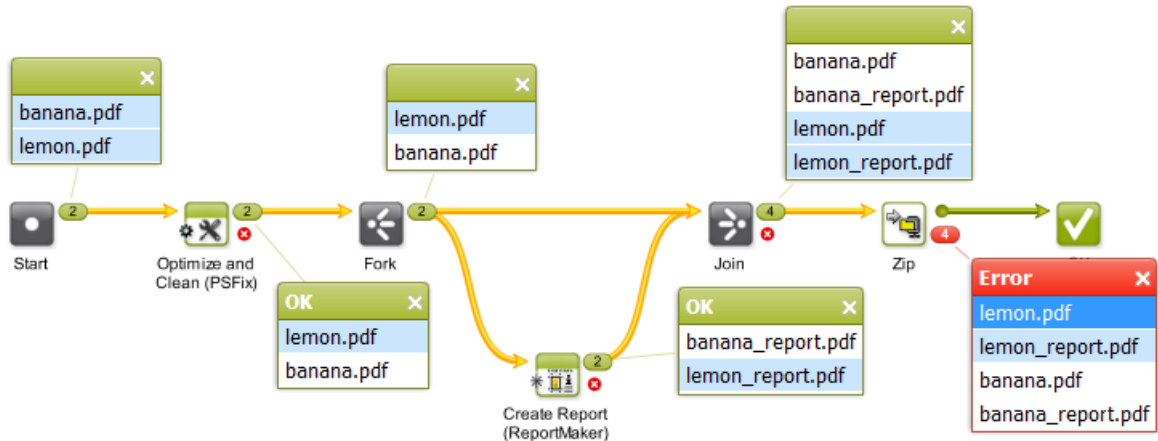
Seeing the Followed Path

In an output widget, click on a file name to see which path it followed. In this example we clicked on `lemon_report.pdf`. The name is highlighted in blue and the path that this file followed so far is shown in yellow.



Seeing the Related Files

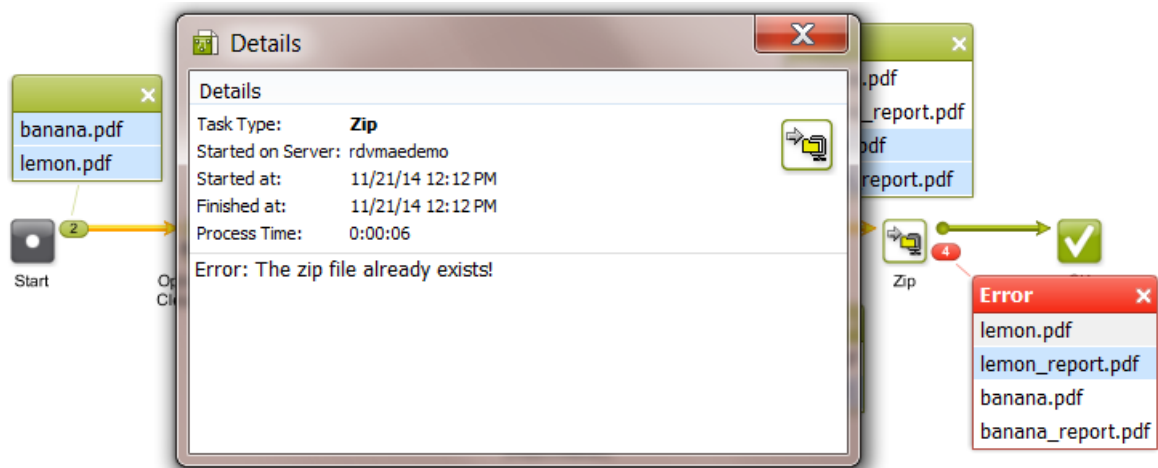
Clicking on a file name also helps diagnosing the relations between files. In below example we clicked on `lemon.pdf` in the 'Error' widget. This highlights all related files in a light blue color. These relations were created during their processing.



Note: Files that are related to each other in a workflow will also share the same workflow parameters at that specific step of the workflow. Learn more in [Using Workflow Parameters](#) on page 1065.

Diagnosing an Error



In below example, we right-clicked on a file in the 'Error' widget and asked the **Open Details** This explains the error: the zip was not created because it already exists (and the ticket had the setting to fail in that case). Sometimes, you will need to **Open Log ...** to find what you are looking for.



Details is meant for the user and is translated. **Log** info is mainly for Esko service people, it is more complex and not translated.

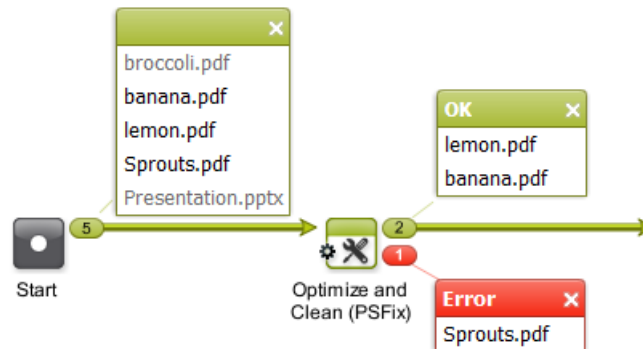
In this example, you could now continue with these steps:

1. Click on  to edit the workflow.

2. Double click the **Zip** task, change the setting that generated the error and click **OK** to confirm and close the ticket.
3. Launch this changed workflow: click  or press **Alt** and click .
4. If you prefer this new setting, save your workflow.

Error Files or Invalid Files

See the open widgets of this example workflow:






- In the widget of the **Start** step, `broccoli.pdf` and `Presentation.pptx` are shown in grey because they are not valid for the next step in the workflow.
 - `broccoli.pdf` is invalid because it has a size of 0 Kb.
 - `Presentation.pptx` is a PowerPoint file and that is not a valid input file type for the next step.

Note: Both file names would be shown in black when the next step would for example be a **Copy or Move File** task.

- `banana.pdf` and `lemon.pdf` are valid PDFs.
- `Sprouts.pdf`: based on its extension '.pdf', the file was valid for the next step but it still resulted in an error (the task detected that the file was not really a PDF file).

6.4.9 Pausing or Cancelling Your Workflow

While monitoring your file's progress in the workflow editor, you can:

- click  to pause your workflow (at the end of the task that is currently running),
- click  to make your paused workflow run again,
- click  to cancel your workflow's processing.

6.5 Using Public Parameters in Workflows

Note: Public Parameters are not only available for Workflows but also for single Task Tickets. This is documented in [Using Public Parameters](#) on page 68. We here only add the specifics about using public parameters in workflows.

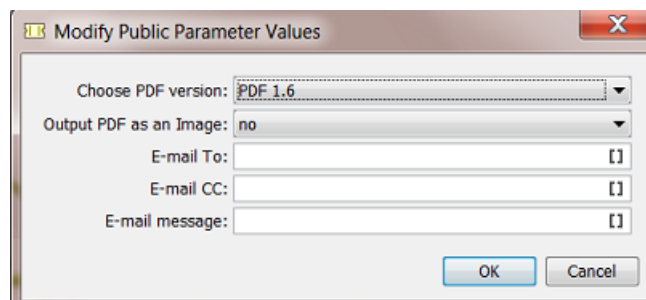
6.5.1 Concept

Most Task Tickets offer many settings ('parameters'). And Workflows can be quite complex combinations of many such Tickets. That is why, when an administrator creates a workflow, it can be useful to not show this whole complexity to every day users of those workflows.

This can be done by making only some parameters public. Thus, the daily user can only see and modify a limited set of parameters that he understands and is allowed to change.

For example:

A user wants to launch a workflow and needs to check or change only a few settings. Instead of having him open the workflow and open several steps to change only 5 parameters, you offer him this simple dialog, where he is asked to define the 5 parameters that you made public to him:



When the user presses **OK**, the workflow starts and uses the chosen settings.

Available from many Client Applications

This functionality is available for users launching workflows from these applications that have a connection with the Automation Engine server:

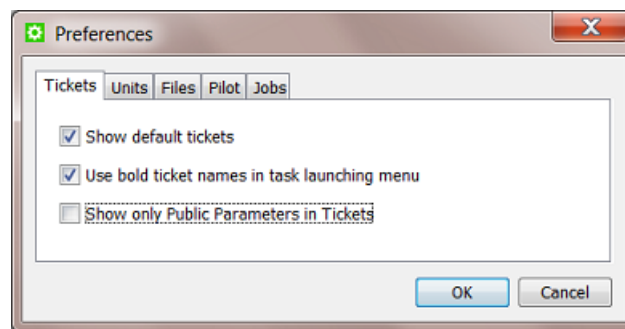
- Automation Engine Shuttle
- Automation Engine Pilot
- ArtPro, Neo
- PackEdge, Plato, FastImpose
- DeskPack (Adobe Illustrator)

Important:

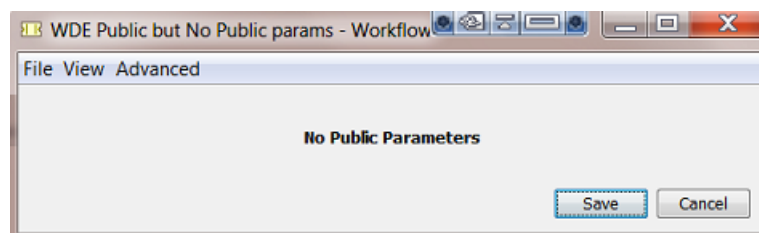
- Client applications only show **Public** tickets and workflows, no matter what user is logged in. In other words, they only show the same list of tickets that a user sees who does *not* have the User Access Right "**Tickets: Show all Tickets and their Parameters (Public and other)**".
- If those public tickets and workflows do not have any public parameters, the user will not be able to open them to change any parameters. They will be launched right after he selected the (workflow) ticket.

Preference to have the Pilot show Only Public Parameters

In the Pilot, in **Edit > Preferences** the option **Show only Public Parameters in Tickets** can help you prevent users from opening workflows in the workflow editor and so having access to all settings.



- When this option is not checked, users will be able to open workflow tickets and see all steps and settings. If they can make changes and save the ticket depends on their **User Access Rights**.
- When this option is checked, a users that opens a workflow ticket will only be shown a dialog with the public parameters of that workflow. When that workflow has no public parameters, the user will see this dialog:

**About Public Parameters in a Workflow Ticket that is Not Public**

It is possible to set public parameters in a workflow ticket that is not saved as public. This can be useful when the non public workflow ticket is referenced inside a master workflow that is public. The public parameters of the referenced subworkflow will show up in the list of public parameters of the master workflow.

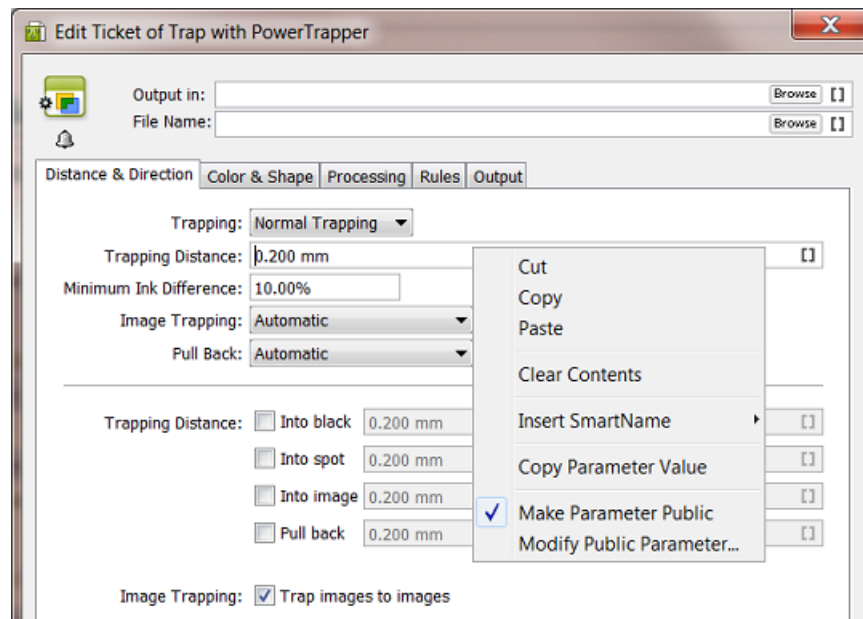
6.5.2 Creating Public Parameters in a Workflow

Creating public parameters in a workflow is very similar to creating them for a single task ticket, as explained in [Making a Parameter Public](#) on page 69.

A summary:

1. In the workflow editor, select a step and open the ticket.
2. Right-click the parameter you want to make public and choose **Make Parameter Public**.

We here right-clicked in the field for **Trapping Distance**:



Note: Not all parameters of all tickets can be made public.

3. Right-click the same parameter again and choose **Modify Public Parameter...**. Decide here if you want the parameter to be a free or predefined value.

Note: You can also skip this step and later manage this and all other public parameters in the [Manage Public Parameters](#) dialog. You can also use both dialogs.

4. Save your ticket to also save these changes to your public parameters.

Note: Public parameters belonging to steps that you later deleted from the workflow are not removed. You can remove them using the **Manage Public Parameters** dialog.

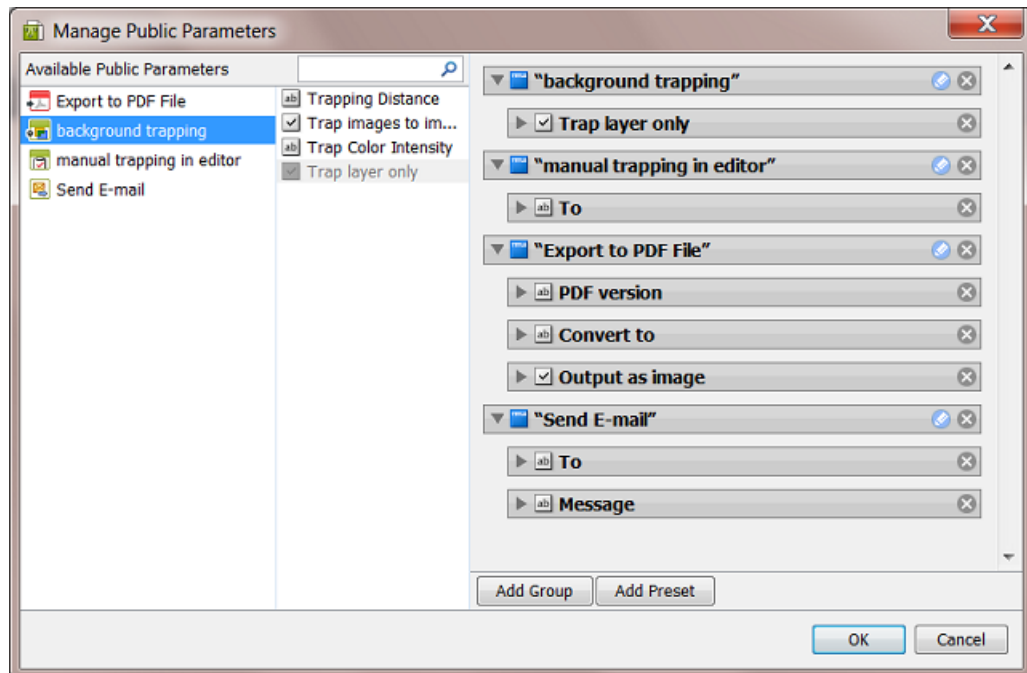
Next step is to manage the workflow parameters in a special dialog where you will define more precisely how they will present themselves to the user.

6.5.3 Managing Public Parameters in a Workflow

Once you have made parameters public in your workflow steps, you can open the **Manage Public Parameters** dialog to get an overview and make changes.

1. In the workflow editor, go to **Advanced > Manage Public Parameters...**



The dialog shows all public parameters of this workflow, sorted by workflow step:

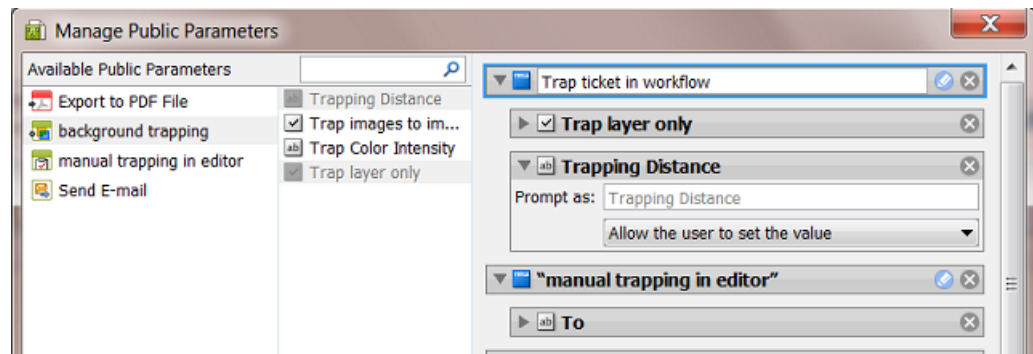


- **The left pane** shows the workflow steps for which you have made parameters public.
- **The middle pane** shows the public parameters of the workflow step that was selected on the left. They are greyed out when they are already used in the right pane. When parameters are not used in the right pane, they will not be public parameters.
- **The right pane** shows all the public parameters in detail (name, type of value, etc.). They are default organized in **Groups** matching the (custom) name of the step where they came from. This is where you build the dialog that the users will see and use to decide the values before launching the workflow.

To help you trace their origin, selecting items in the panels also shows selections of related items in the other panels.

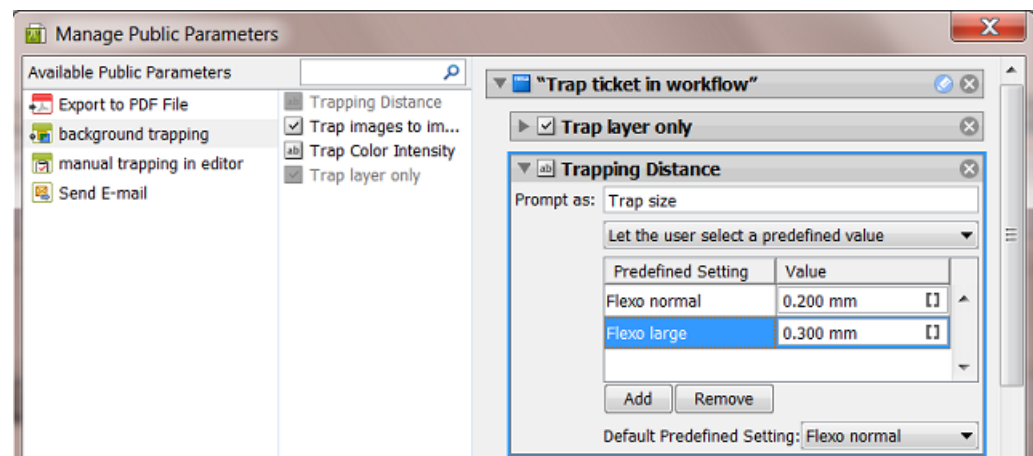
2. You can

- drag parameters from the middle to the right pane to make them public. For example dragging the parameter **Trapping Distance** into the group **"background trapping"**.
- click on  to remove the parameter or group from the right pane. It will then not be public anymore.
- re-arrange the order of your public parameters or groups (drag them up or down).
- click on  to change the default name of the group.



- change the settings of the public parameters. Define in **Prompt as** is what the user will see. Instead of **Allowing the user to set a value**, you can offer the user a choice of predefined values.

In below example the user will get a dropdown to choose between **Flexo normal** and **Flexo large**. The user making this choice when launching the workflow will not see the actual values of 0,2 and 0,3 mm.



Learn more about editing and re-designing this pane in [Making Presets to Simplify the Users' Choices](#) on page 72 and [Example of Further Customizing the Public Parameters Dialog](#) on page 1058.

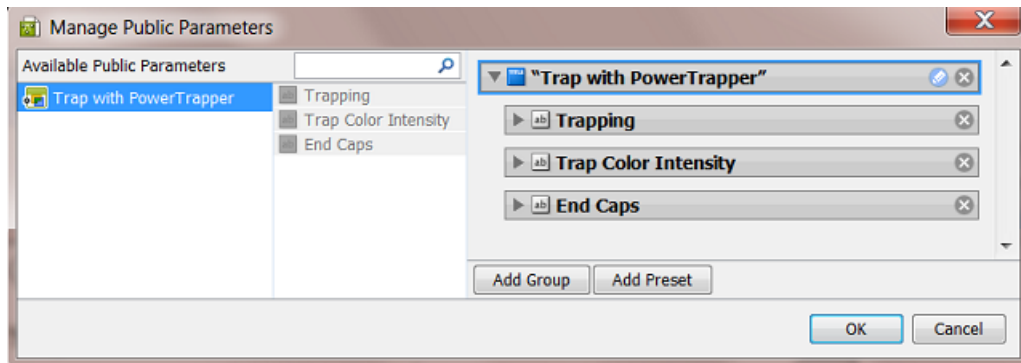
6.5.4 Making Presets to Simplify the Users' Choices




Presets are custom combinations of parameters. They simplify the choice users have to make. They limit the amount of decisions that users have to make.

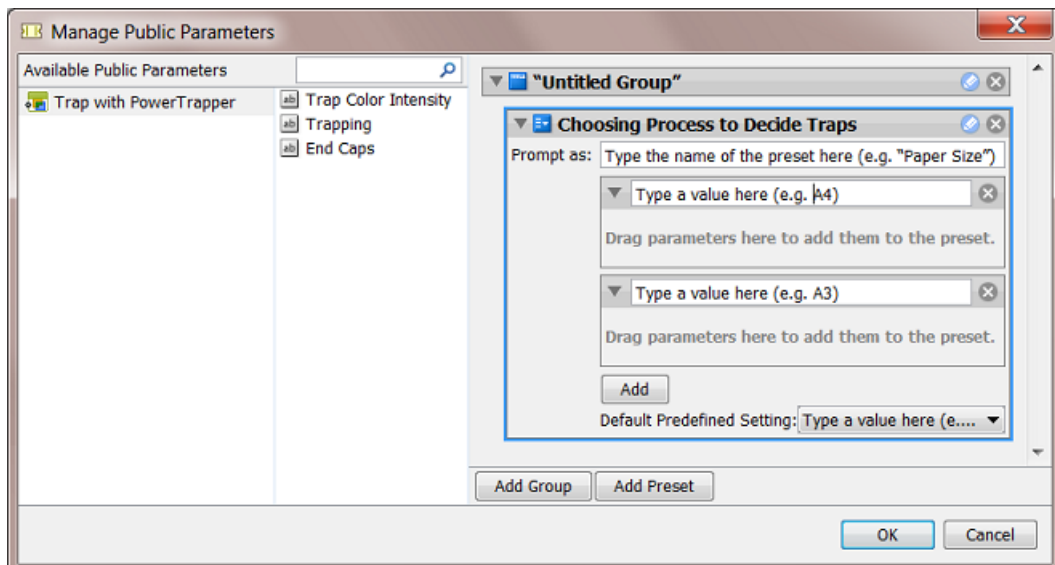
Example

For example, you want users to use different trapping settings when printing offset versus dry offset. You can define those trapping settings in a preset, so that users will only have to choose between 'offset' and 'dry offset'.

We start from a **Manage Public Parameters** dialog where the trapping step has 3 public parameters:

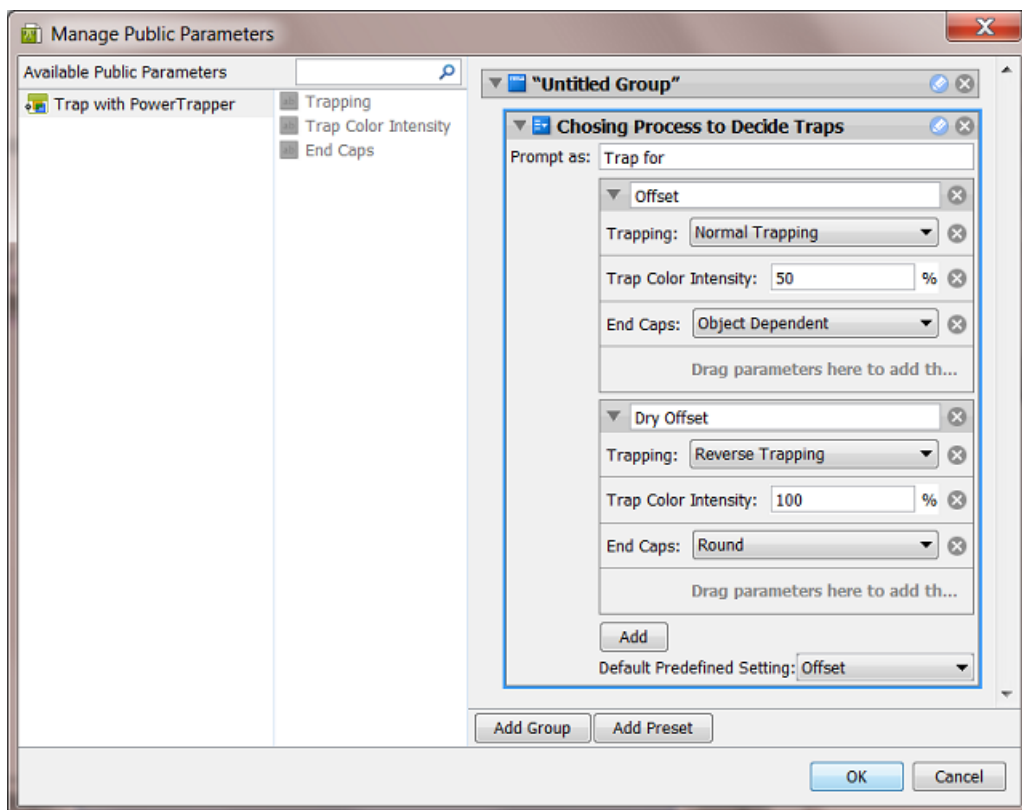
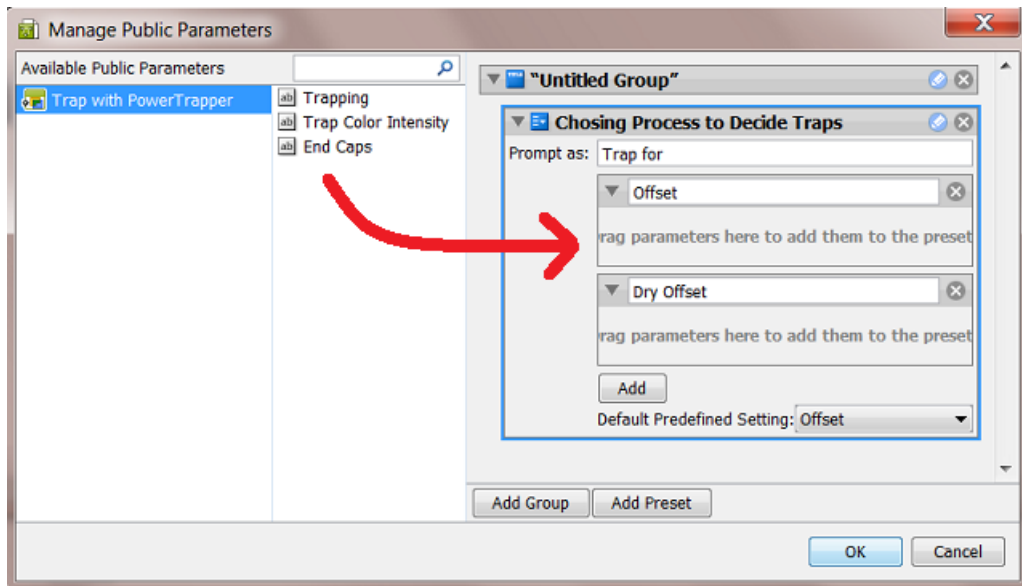


1. Click on  to remove the (default) **Group** named "Trap with PowerTrapper"
2. Press the **Add Group** button.
A new "Untitled Group" is created. Click on  to change the name.
3. Press the **Add Preset** button.
A new "Preset" is created. Click on  to change the name.
By default, two example values will appear. the preset was renamed to 'Choosing Process to Decide Traps'.



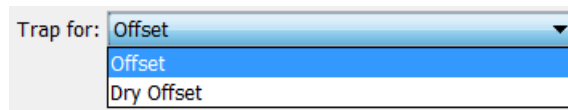
4. Define what the users will see:
 - a) in **Prompt as**, enter the text that users will see in the client application (replacing 'Type the name of the preset here (e.g. "Paper Size")').
 - b) enter the values that the users will choose from (replacing 'Type a value here (e.g. A4)'). We here added 'Offset' and 'Dry Offset'.

Click **Add** if you want to add extra values (not the case in our example).
5. Drag the parameters from the dialog's middle pane onto one preset item. This will make them automatically appear under each preset item (here both under *Offset* and *Dry Offset*).



6. Define the parameters values for each preset item.
See the above screenshot for a good example.
7. Click **OK** to confirm and close the dialog.
8. Save your ticket.

Result: this is how the resulting dialog will look like to the users of the client applications:



6.5.5 Example of Further Customizing the Public Parameters Dialog

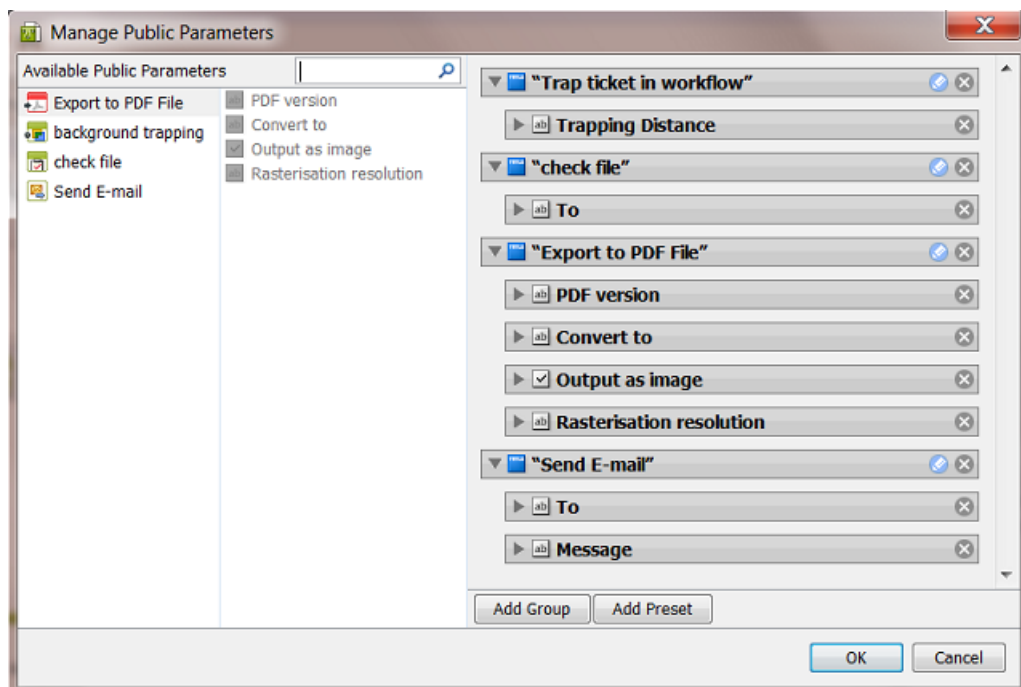
In this example, we further customize the public parameters dialog by reorganizing the **Groups** and adding a **Preset** to combine parameters.

Step 1 - Example Workflow and its Initial Public Parameters Dialog

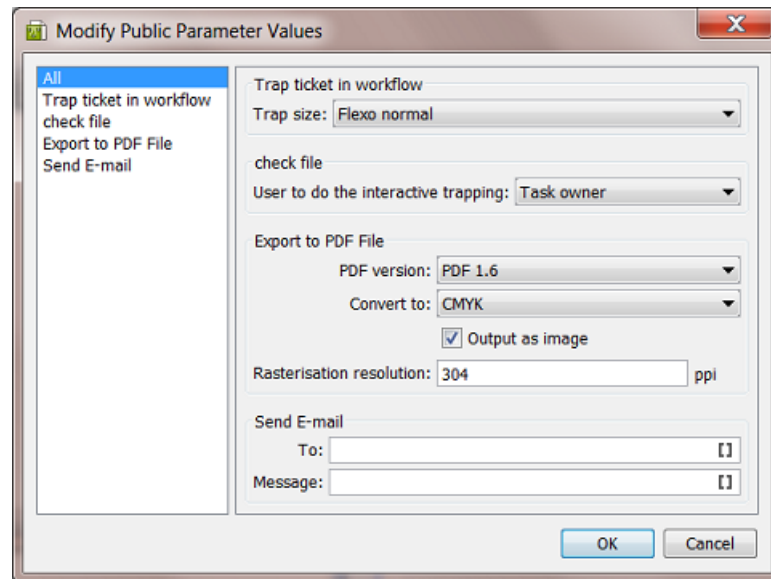
This is our example workflow:



We created public parameters in 4 steps. This is the initial **Manage Public Parameters** dialog, where the groups match the workflow steps:



If you don't customize this, the resulting dialog, that will be shown to the users launching a task, will look like this:



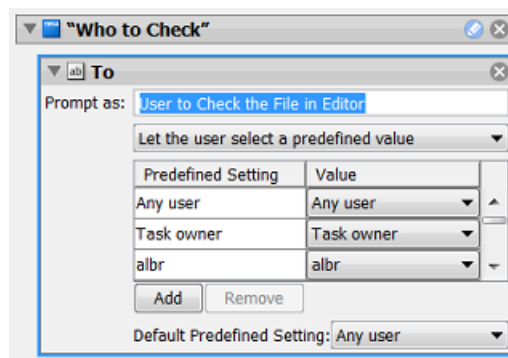
In this dialog, you can see

- that the items on the left pane of this dialog match the groups in the **Manage Public Parameters** dialog.
- that when there are multiple groups, an extra one named 'All' is automatically created. This group offers all parameters together.
- that the parameter 'Trap size' is offered as a choice of presets. This was illustrated [earlier](#).

In the next steps we will further optimize this dialog for the end user.

Step 2 - Customizing the Groups and the Text that the User will See

- In the group "check file",
 - rename the group to "Who to Check",
 - change **Prompt as** to 'User to Check the File in Editor',
 - set the **Default Predefined Setting** to 'Any User'.

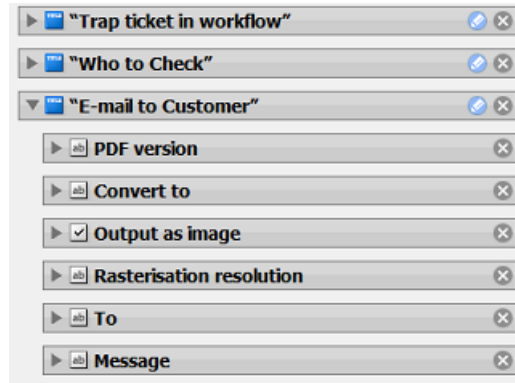


- Press **Add Group** to create a new group and name it "E-mail to Customer".

- In the right pane, drag the parameters from the groups "Export to PDF File" and "Send E-mail" into this new group.

Tip: You can press **Shift** to select and drag multiple parameters.

- Remove these two groups that are now empty: "Export to PDF File" and "Send E-mail". This is the result:

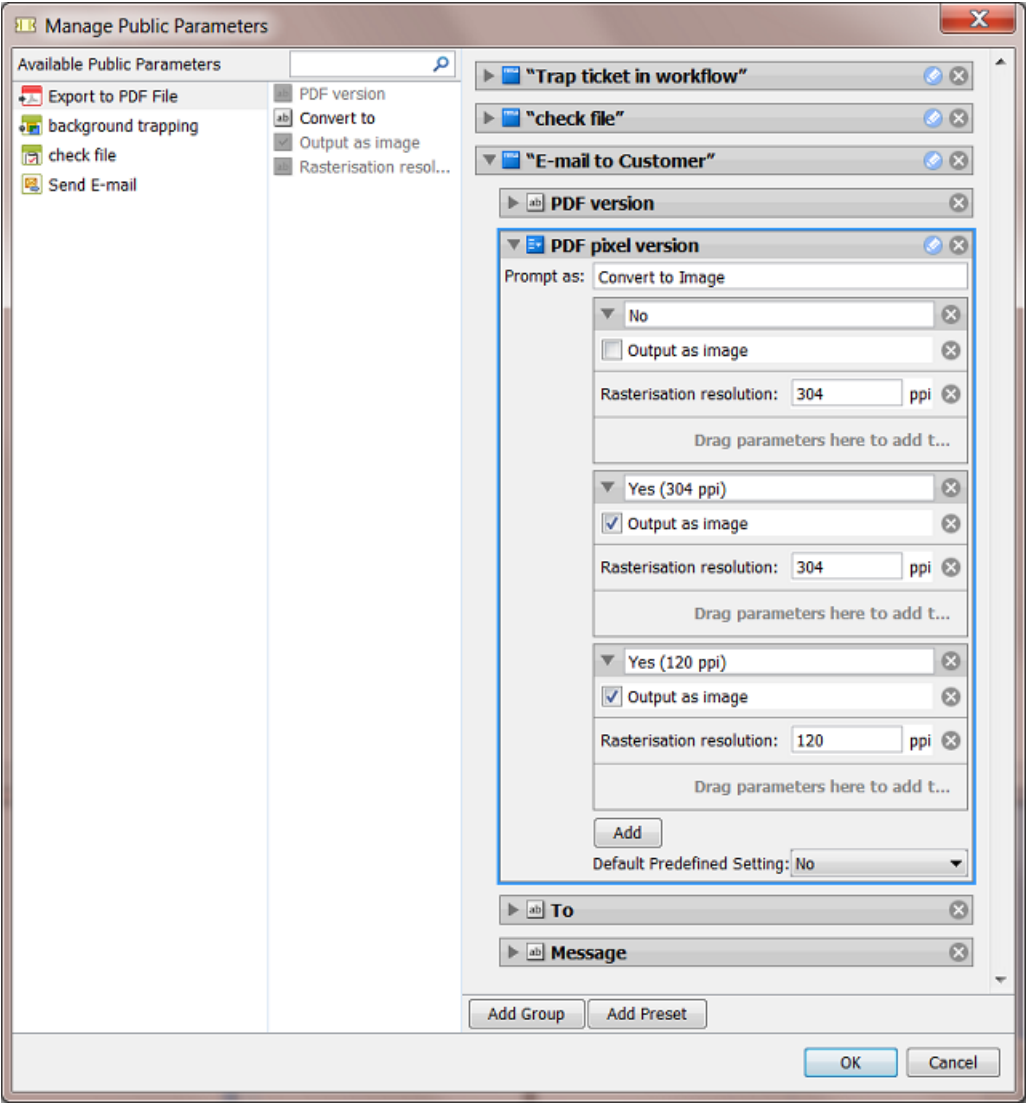


Note: An alternative way to do this, is to first delete the 2 groups, then create the new one and then drag the 5 parameters from the middle pane into the new group.

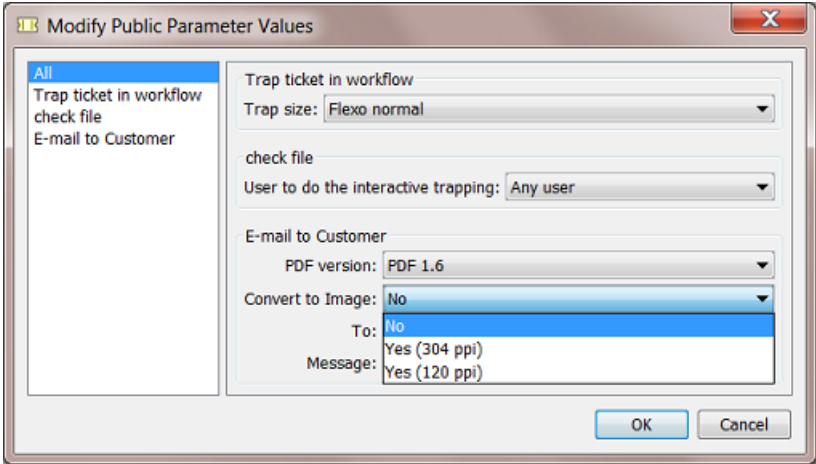
Step 3 - Adding a Preset and Combining Parameters

In the new **Group** "E-mail to Customer", we now combine 2 parameters into a new **Preset**:

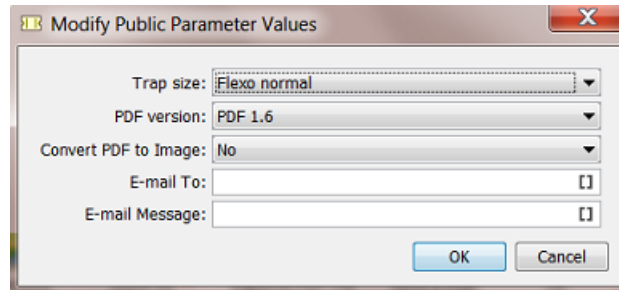
- Press **Add Preset**. A new preset appears at the bottom of the bottom group. Rename it to 'PDF pixel version'.
- Set the **Prompt As** to 'Convert to Image'
- Press **Add** to add a third parameter and rename the 3 parameters to 'No', 'Yes (304 ppi)' and 'Yes (120 ppi)'.
- Drag the parameters 'Output as Image' and 'Rasterization Resolution' onto the first value of this new preset.
- Adapt the parameter values as shown in this screenshot:



For the user launching the workflow, the resulting dialog already looks a bit simpler:



In this example, the list of parameters is not that long. That is why you could also decide to *only* offer the **All** view in this dialog. You can do this easily by moving all parameters into one group. The result can look like this:



Note: Many tasks have Tickets with a conditional user interface: where settings depend on another setting. For example settings that only become available after you first make another choice. Example: 'Apply color conversion?' If 'Yes', then there is the choice 'Convert to CMYK' or 'Convert to RGB'.

These kind of conditions can not be created in the **Manage Public Parameters** dialog. If you wish to recreate such a user decision, you have to create a preset that offers a choice for each combination. In our example this would mean a dropdown list with these 3 choices: 'No color conversion', 'Convert to CMYK' and 'Convert to RGB'.

6.5.6 Launching a Workflow with Public Parameters

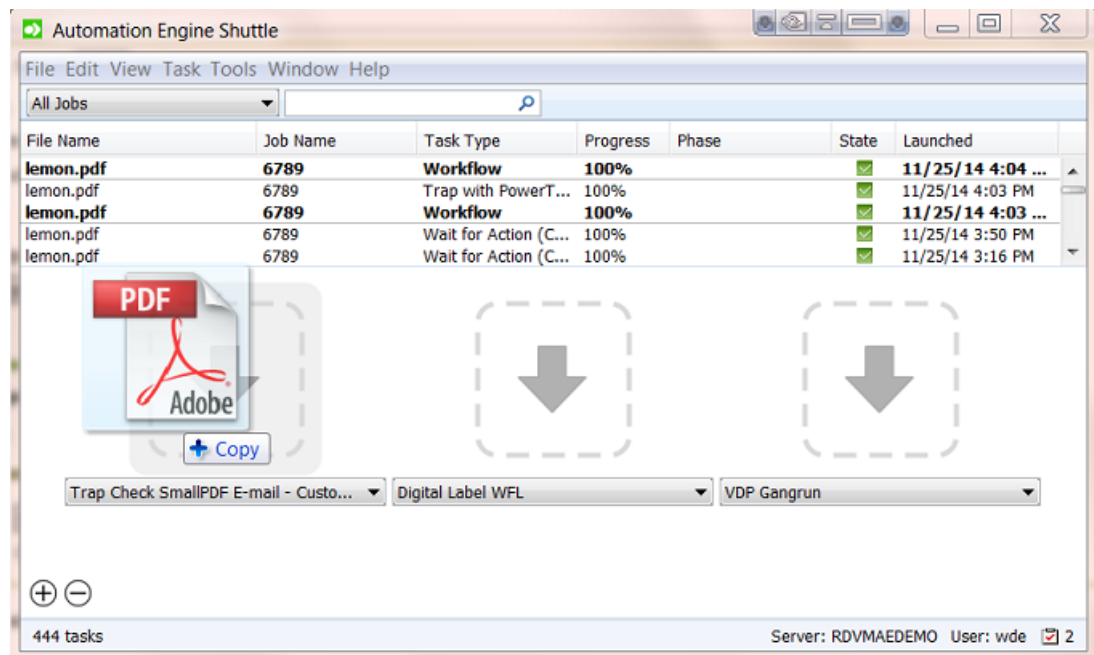
Launching a workflow with public parameters is very similar to how it is done with single task tickets.

Learn more about all variants of launching workflows from Shuttle client applications in [Using Shuttle to Upload and Process Files](#) on page 47.

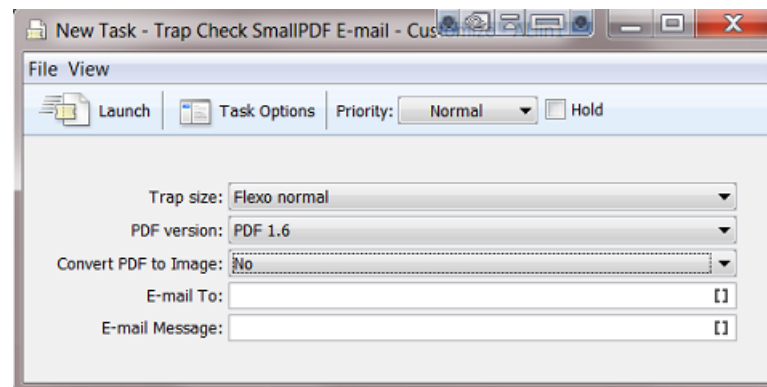
Here are some examples:

Example when launching from Shuttle:

Drop the input file on the workflow in the **Launch panel**.



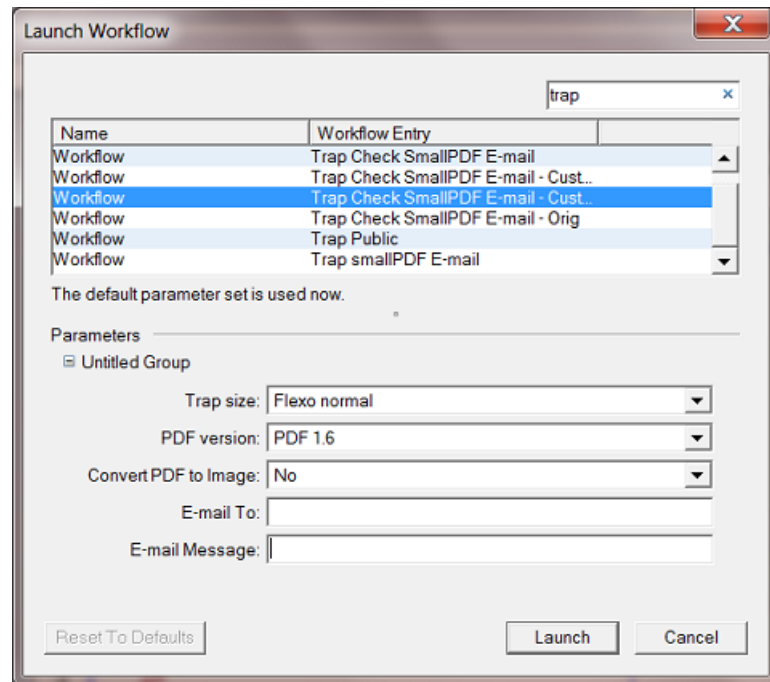
If you drop a file on the launch field of a workflow that has public parameters, a dialog will open asking you to decide these public parameters:



Click **Launch** when ready.

Example when launching from PackEdge:

Choose **File > Launch Workflow** from the menu, select the workflow, decide the public parameters and press **Launch**.



Example when launching from ArtPro

In the **Launch Workflow** dialog, ArtPro offers extra functionality to convert the ArtPro file before it enters the workflow and to set a Job or Product context (which is important when using SmartNames).

Launch Workflow

Launch as: PDF for Press

Job: <None> Search...

Product: <None> Search...

Job ID: Job Part ID:

Workflow

Q all

Name	Workflow Entry	Server
Workflow	Call Webservice Demo	rdalpha25
Workflow	Trap Check SmallPDF E-mail	rdvmaedemo
Workflow	Trap Check SmallPDF E-mail - Customize	rdvmaedemo
Workflow	Trap Check SmallPDF E-mail - Customize - ALLin1	rdvmaedemo
Workflow	Trap Check SmallPDF E-mail - Orig	rdvmaedemo
Workflow	Trap smallPDF E-mail	rdvmaedemo

The default parameter set is used now.

Parameters

▼ Untitled Group

Trap size: Flexo normal

PDF version: PDF 1.6

Convert PDF to Image: No

E-mail To:

E-mail Message:

Reset To Defaults Cancel Launch

6.6 Using Workflow Parameters

6.6.1 Concept

Workflow Parameters are parameters that are specific for a workflow. Their values are picked up by steps in the workflow.

For example: you print displays and you mostly use the same output workflow. Your workflow ticket has 3 workflow parameters: `substrate`, `sheet size` and `quantity`. The values of these parameters can be typed in by a user or can be picked up from an external source.

Workflow parameters show up as SmartNames starting with `[wfp.]`. Much like Job and Product parameters, you can use them by inserting them in the SmartName enabled field in your workflow steps.

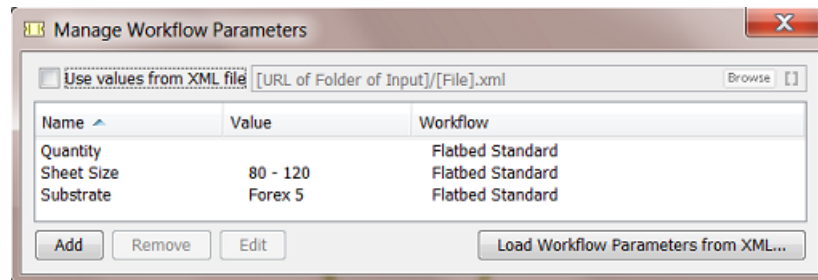
In many cases, you could accomplish the same by using Job or Product parameters, but some Automation Engine customers choose to not use the Jobs or Products concept.

Workflow Parameters are Loaded at Workflow Start-Up, but can Change Later

Workflow parameters and their values are loaded when the workflow starts up. However they can change during the workflow. This can happen via a user intervention or more automated steps. Learn more in [reading their value from an XML file](#) and in [modifying workflow parameters during a workflow](#).

Managing Workflow Parameters

In the workflow editor, go to **Advanced / Tools > Manage Workflow Parameters...**



- The **Manage Workflow Parameters** dialog is where you **Add**, **Remove** and **Edit** workflow parameters.
- This dialog lists all the workflow parameters defined in the current workflow and all of its subworkflows.

Note: If you are in a subworkflow, the workflow parameters defined in the master workflow will not be shown in this dialog.

- You can keep this dialog open while making changes in the current or subworkflow(s) and while navigating to and from subworkflows. The list of workflow parameters is instantly updated to reflect changes.

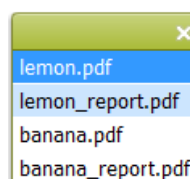
Workflow Parameters can be Different per Group of Files

Workflow parameters do not have to be valid for *all* files in that workflow, they can be different per file or per group of files.

For example: the banana and lemon displays share the same `substrate` and `sheet_size`, but have a different value for the parameter `quantity`.

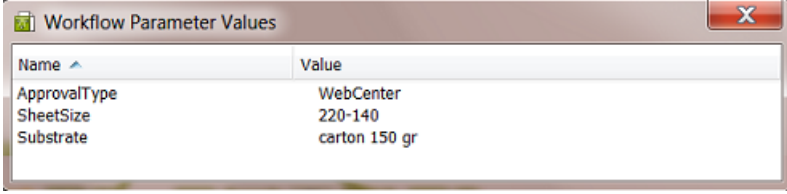
Workflow parameters are valid for one file or for a group of files. You can see such grouping of files when you open a Widget and select an item in its list. This will not only highlight the selected item (in blue) but also the other items belonging to that group (in light blue). Such a group is sometimes referred to as a "Token", it can represent 1 or more items.

For example: when you select `lemon.pdf`, `lemon_report.pdf` also gets highlighted. This means that they belong together in a group. This means that they share the same workflow parameters (at that point in the workflow).



These groups are created automatically by the file management logic in the workflow steps. However, sometimes a setting lets you decide how these groups need to be created. A typical example is the [Data Collector](#) on page 1029.

When you right-click on an item in a widget and select **Workflow Parameter Values**, you will see the values for that group of files at that point in the workflow. An example:



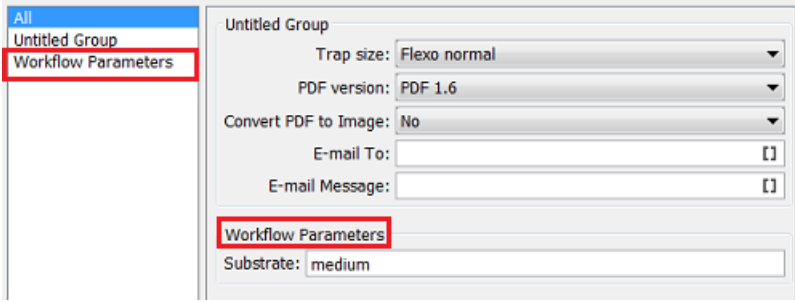
Name	Value
ApprovalType	WebCenter
SheetSize	220-140
Substrate	carton 150 gr

Workflow Parameters versus Public Parameters

Public Parameters serve to simplify the decisions users have to make when they launch workflows from client applications. **Workflow Parameters** serve to make your workflow more automated.

Workflow parameters are also SmartNames, public parameters are not.

In the **Manage Public parameters** dialog that is shown to the user launching the workflow, workflow parameters show up as a separate parameter group named **Workflow Parameters**:



You can easily remove workflow parameters from the list of public parameters using the **Manage Public Parameters dialog**. Learn more about public parameters in [Using Public Parameters in Workflows](#) on page 1051.

Note: There is an exception: if you [use workflow parameters values from an XML file](#), the workflow parameters will not be public.

6.6.2 Most Typical Use Cases for Using Workflow Parameters

1. Automating workflows by Loading Workflow Parameters from an XML file

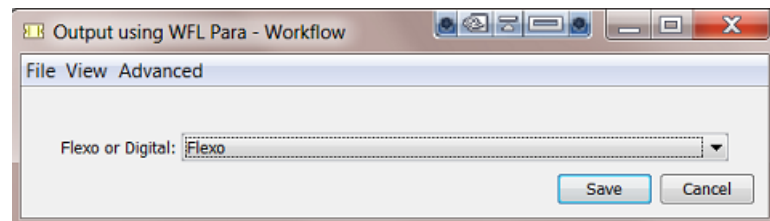
This is the case where the order management or production planning system sends Automation Engine an XML with the details of the output workflow. These production specifics are automatically picked up and serve as parameters for the workflow. Learn more in [Using Workflow Parameter Values from XML](#) on page 1072.

2. Using Workflow Parameters as a Smart Public Parameter

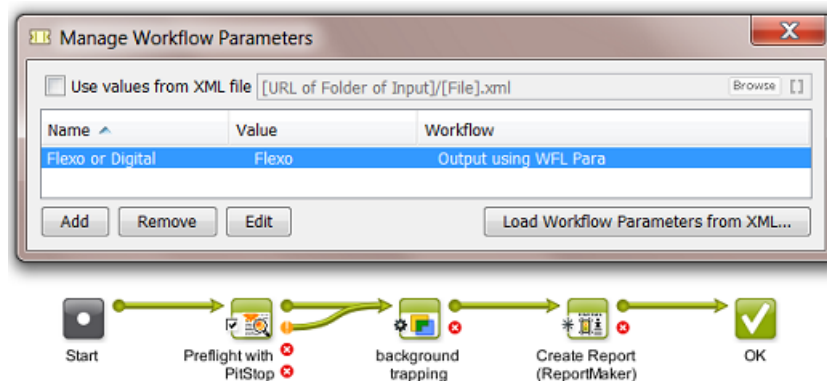
When users enter the value of workflow parameters manually, it usually resembles the case of a using public parameters, but then also taking advantage of the referencing power of a SmartName.

For example:

Your users need to decide if the workflow is `flexo` or `digital`. You want them to answer this one question as a public parameter:



This is only 1 question but we will use the answer for 3 different decisions in our workflow, once in each of below workflow steps:



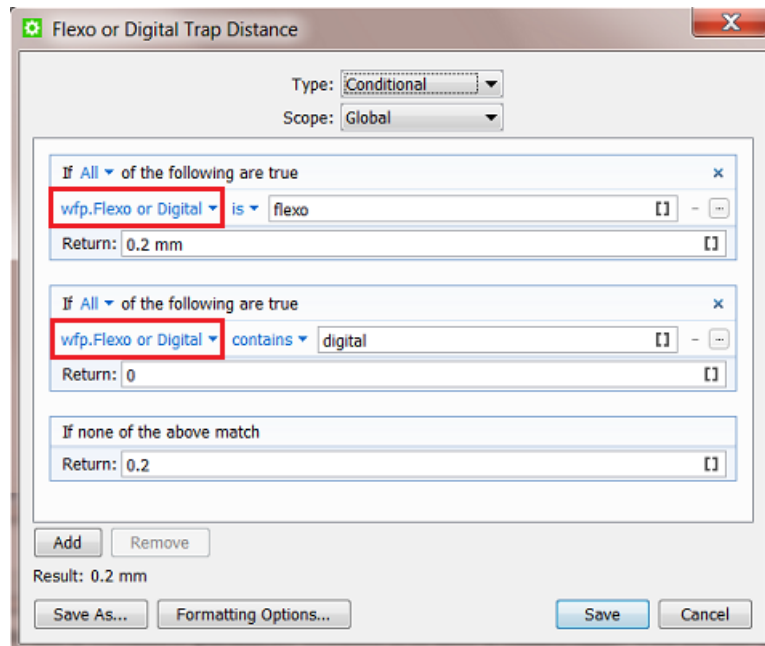
Here is an example how this can be done. We also illustrate two different levels of complexity:

- You **use the value of the workflow parameter as text**. For example the Preflight task, depending on the user's answer, uses the profile `flexo.ppp` or `digital.ppp`. In this case you pick up the workflow parameter as a SmartName:

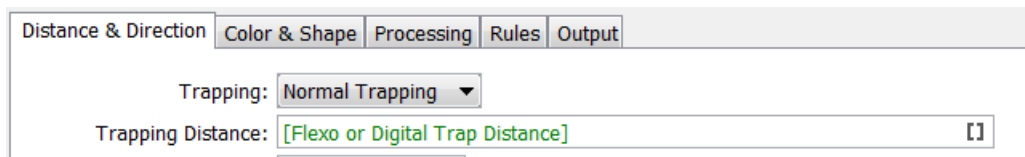
Profile:

In the same way, the ReportMaker task uses `flexo.pdf` or `digital.pdf` as template. One decision from the user so already decided values in 2 tickets.

- You can also **use value of the workflow parameter inside another SmartName**. For example this conditional SmartName to define the trapping distance:



You then use that conditional SmartName in the trapping task:



In this example, when you do not use a workflow parameter, the user launching the task would have to decide 3 different public parameters. Now, the user is only asked 1 decision.

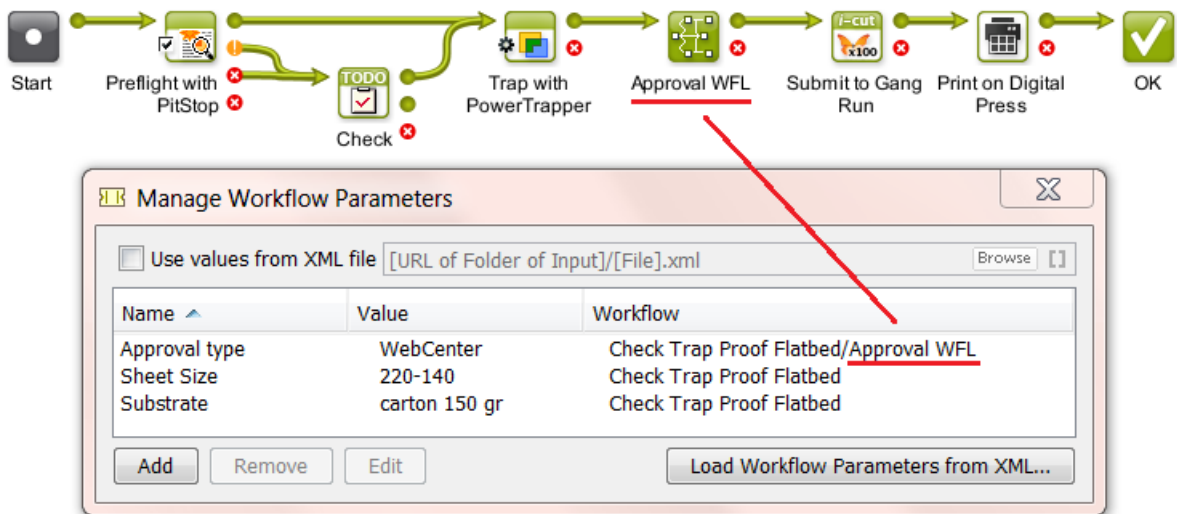
6.6.3 Creating Workflow Parameters Manually

Follow these steps to create workflow parameters manually:

1. Open the (sub)workflow in the workflow editor.
2. Open **Advanced / Tools > Manage Workflow Parameters...**
3. In the **Manage Workflow Parameters**, click on **Add** and fill in the **Name** and **Value**.

Note: You can leave the value empty at this time. The value has to be defined at the latest by the time the workflow is launched (for example when a user who launches the workflow fills in the value in the dialog of public parameters).

In the dialog, the column **Workflow** shows if the workflow parameter is present in the current workflow or was created in a subworkflow.



Note: If you need to add many workflow parameters and if their names are already present in an XML file, you can load that list of names by using the button **Load Workflow Parameters from XML....** Learn more [here](#).

4. Save your workflow to also save your workflow parameters.

6.6.4 Loading Workflow Parameters from XML

Instead of typing them in manually, you can load **Workflow Parameters** from a XML file. You can select all parameters from the XML file or only those of a specific XML Element. After loading them, you can keep the values that came from the XML file or change them.

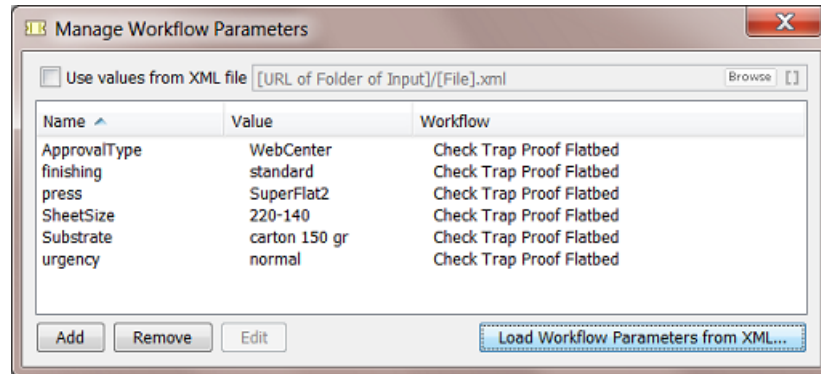
This is an example of a valid XML file:

```
<?xml version="1.0"?>
- <job>
  - <production>
    <ApprovalType>WebCenter</ApprovalType>
    <Substrate>carton 150 gr</Substrate>
    <SheetSize>220-140</SheetSize>
    <press>SuperFlat2</press>
    <finishing>standard</finishing>
    <urgency>normal</urgency>
  </production>
  - <customer>
    <name>FruitCo</name>
    <number>2712</number>
    <currency>USD</currency>
  </customer>
</job>
```

Follow these steps to load only the items of the XML Element 'production' :

1. Open the (sub)workflow in the workflow editor.
2. Open **Advanced / Tools > Manage Workflow Parameters...**
3. Click **Load Workflow Parameters from XML....**


4. Browse to the XML file, select it and click **Open**.
A dialog pops up asking you to specify an XML **Element** or to leave the field blank to select all items in the Root Element of the XML file.
5. In that dialog, type in the element name `production` and click **OK** to confirm.
This will load these workflow parameters:

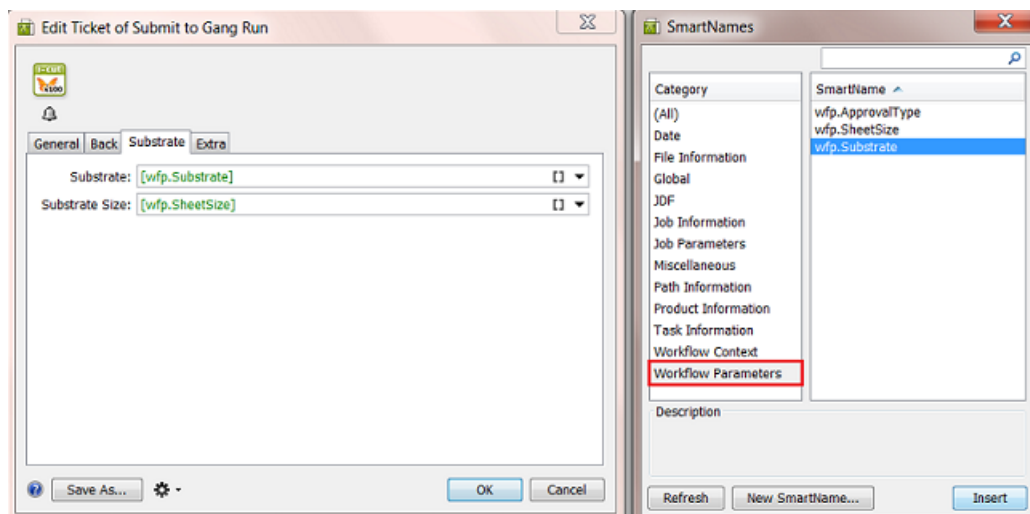


6. If you want you can make any modifications like changing a value or even removing a parameter.
In our example, we don't need the parameter 'finishing' so we can remove it.
7. Save your workflow to also save your workflow parameters.

6.6.5 Using Workflow Parameters

Once you have created them, you use the workflow parameters by picking up their SmartName version in the workflow step ticket.

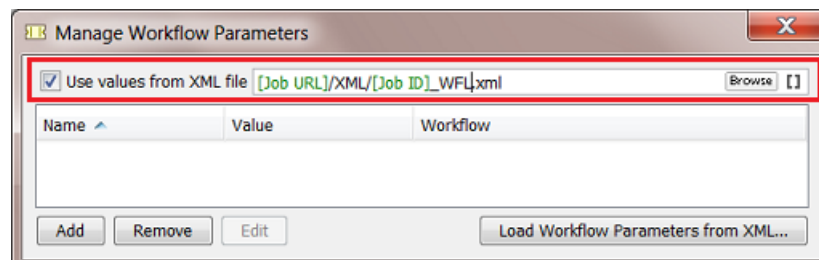
1. Open the (sub)workflow in the workflow editor.
2. Double-click the ticket in which you want to use a workflow parameter.
3. Click on  in the SmartName-enabled field where you want to add the workflow parameter.
In the SmartNames Browser, select the **Category 'Workflow Parameters'**



4. Select the workflow parameter [wfp.] that you want and click **Insert** (or double-click) to add it to the ticket.
5. Click **OK** to confirm and close the ticket.
6. Save your workflow.

6.6.6 Using Workflow Parameter Values from XML

Workflow parameters and their values can be defined by an XML file.



This is an example of a valid XML file:

```
<?xml version="1.0"?>
- <job>
  - <production>
    <ApprovalType>WebCenter</ApprovalType>
    <Substrate>carton 150 gr</Substrate>
    <SheetSize>220-140</SheetSize>
    <press>SuperFlat2</press>
    <finishing>standard</finishing>
    <urgency>normal</urgency>
  </production>
  - <customer>
    <name>FruitCo</name>
    <number>2712</number>
    <currency>USD</currency>
  </customer>
</job>
```

In this concept, there are 2 differences versus *creating workflow parameters manually* or *loading them once from an XML*:

- the workflow will load and re-read this XML file at every step of the workflow. This also means
 - that you will probably not use the step *Modify Workflow Parameter Values* in this workflow because its modifications would be overruled.
 - that, during the workflow, the XML file can be updated with new or changed values.
- the resulting workflow parameters will not be public parameters. The concept here is that it is not a user who decides the values, but an XML file.

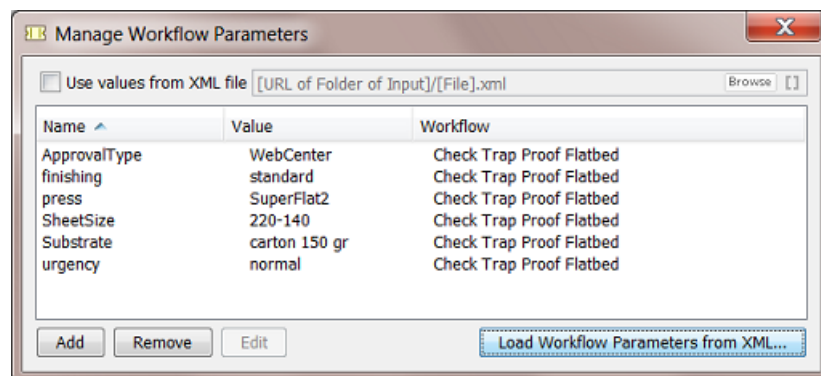
Example

Follow these steps to set up a workflow where workflow parameters values are used from an XML file:



Attention: Before we can ask a workflow to use the XML file, we first have to insert the workflow parameters as [wfp.] SmartNames into the workflow step tickets. To be able to do that, we will first load a sample of that XML and so create these required SmartNames first.

1. Open the (sub)workflow in the workflow editor.
2. Open **Advanced / Tools > Manage Workflow Parameters...**
3. Click **Load Workflow Parameters from XML....** Browse to your sample XML file, select it and click **Open**.
A dialog pops up asking you to specify an XML **Element** or to leave the field blank to select all items in the Root Element of the XML file.
4. Using the example XML mentioned in [Loading Workflow Parameters from XML](#), type in the element `production` in that dialog and click **OK** to confirm.
This will load these workflow parameters and create a [wfp.] SmartName of each of them:



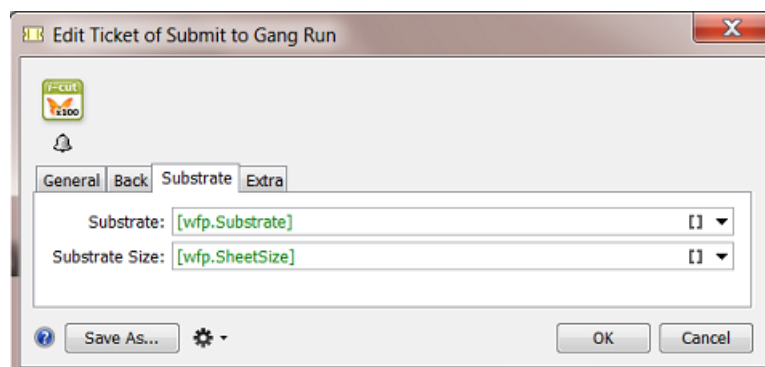
5. From that list, remove any parameters that you will not be using.

Note:

- The **Values** in this list will not be used anyway, they will come from the XML file (see step 7).
- The **Names** in this list are the basis for the [wfp.] SmartNames. So do not delete them if you need the SmartName version.


6. Use the resulting SmartNames in your workflow step tickets (as shown in [Using Workflow Parameters](#) on page 1071).

An example:




7. In the **Manage Workflow Parameters** dialog, check **Use values from XML file**. You will be asked to confirm this step. Click **OK**.
8. Select the XML file using **Browse** or use SmartNames. This is the place and name that the workflow will look for at the start of every step of the workflow.
9. Save your workflow to also save this setup of your workflow parameters.

6.6.7 Changing the Value of a Workflow Parameter


You can easily change the value of a workflow parameter by using the **Edit** or  (Mac) button in the **Manage Workflow Parameters** dialog.

When would you change the value of a workflow parameter?

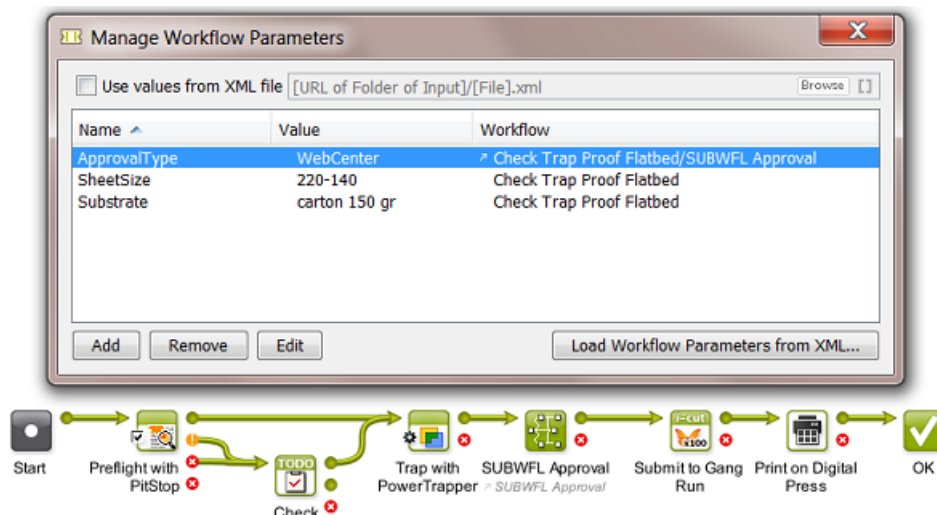
When you are testing a workflow while staying in the workflow editor. You edit the parameter value, relaunch the workflow () and monitor the result.


Tip: When *using values from an XML file*, some administrators use this "edit and test" method to learn how the XML file should look like.

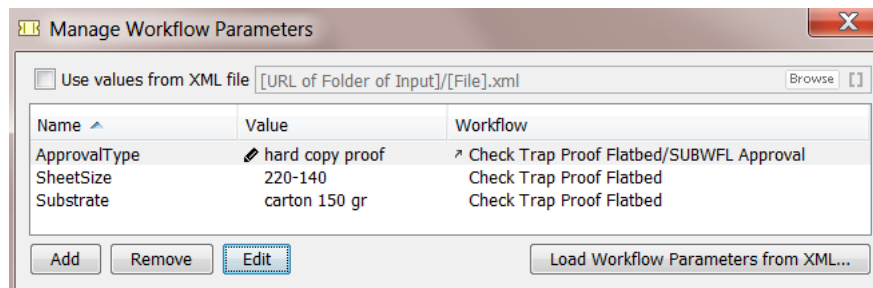
Changing the value of a workflow parameter from a referenced subworkflow

If you edit the value of a workflow parameter that is defined in a referenced workflow, its value will be overruled with the new value without changing that value in the referenced workflow. In the **Manage Workflow Parameters** dialog, a pencil icon  will appear next to the workflow parameter that was overruled.

For example, our workflow named `Check Trap Proof Flatbed` has a referenced subworkflow named `SUBWFL Approval` in which a workflow parameter `ApprovalType` has the value `WebCenter`:



When you now edit this value, for example into `hard copy proof`, this icon  will appear next to the value to indicate that this parameter was overruled:



6.6.8 Modifying Workflow Parameters During a Workflow

The **Value** of a workflow parameter can be modified while the workflow is running. This can be done because

- the value changed or was empty before and has now 'arrived'. This change can be decided by a user or by an XML file from an external system.
- the new value did not exist yet because it is based on an result of a previous workflow step.

These workflow steps can modify workflow parameters:

- The workflow control **Modify Workflow Parameter Values**. Learn more in [Modify Workflow Parameter Values](#) on page 1015.
- The **Integrate with WebCenter** task.
- The **Create or Modify WebCenter Project** task.
- The **Publish on WebCenter** task.


These WebCenter related tasks are then part of workflows that use the result of their interaction with WebCenter as a workflow parameter value in one of the next steps of that workflow.

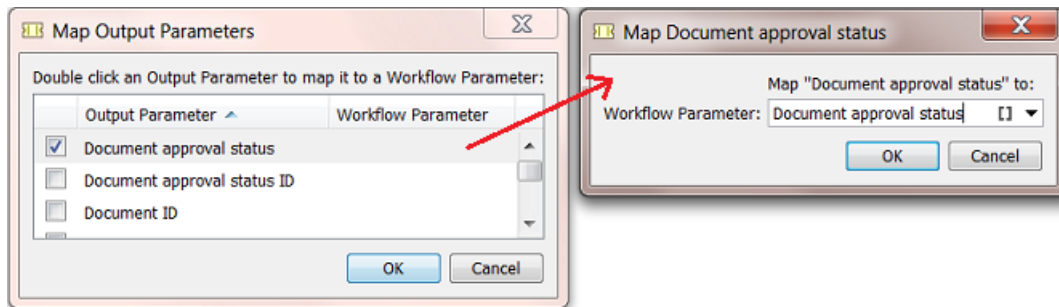
Using the Integrate with WebCenter task

Some **Actions** in the **Integrate with WebCenter** task allow you to use the output of the Action to modify the value of a workflow parameter.

To do this, you need to **Map** one of the task's **Output Parameters** to a workflow parameter.

For example, a workflow asks WebCenter what the approval status is of a document and then uses that status information in the next part of the workflow:

1. In the workflow editor, in the **Integrate with WebCenter** ticket, select an **Action**. For example 'Get Document History'.
2. Click on the Tools icon  and choose **Map Output Parameters...**. A dialog will open.
3. Double-click an **Output Parameter**. For example **Document approval status**.
4. Define to which workflow parameter this **Output Parameter** needs to be mapped to. Select it from the dropdown list or from the **SmartNames Browser** or type in a (new) name. If the workflow parameter does not exist yet, you will be asked to confirm its creation.



Learn more about the **Integrate with WebCenter** task [here](#).

Using the Create or Modify WebCenter Project Task

The **Create or Modify WebCenter Project** task allows you to modify the value of a workflow parameter with the value of the **Project ID** of that WebCenter Project.

To map the **Project ID** to a workflow parameter, use the same **Map Output Parameters** dialog as explained in the [previous section](#).

For example, you did not create the WebCenter project but you want to know the ID of this project because you will publish data to that project later in your workflow.

Learn more about the **Create or Modify WebCenter Project** task [here](#).

Using the Publish on WebCenter Task

The **Publish on WebCenter** task allows you to modify the value of a specific workflow parameter with the value of the **Document Version ID** of the published document in WebCenter.

To map the **Document Version ID** to a specific workflow parameter, use the **Map Output Parameters** dialog, as explained [above](#).

For example:

You use WebCenter in your approval workflow. You have already published your production PDF twice to WebCenter and the customer still wants changes. You make more changes and as usual overwrite the PDF before publishing again to WebCenter.

Now the customer checks this 3rd version and changes his mind: he says that he liked version 2 better. This is a problem because nobody has version 2 any more. Remember that the document versions that WebCenter keeps are versions of only the View Data.

This can be solved when, after publishing your PDF, you ask WebCenter the document version ID and then make a copy of your production file and add that version number in its name. In this way, you also keep versions of your production PDFs.

Learn more about the **Publish on WebCenter** task [here](#).

6.7 Sample Workflows

You can download sample workflows from your Automation Engine server. You can use these

- to learn good practices in building workflows

- as starting points when building your own workflows, rather than starting from scratch.

The sample workflows are categorized per market segment.

The installation will also install the required resources like input files, SmartNames, templates etc.

Installing Sample Workflows

1. Browse to the **Server Admin** page via `http://servername` or `http://servername:9999/`.
2. Click on **Samples**.
3. Click on the **Install** button next to the desired market segment.

The screenshot shows the 'Automation Engine Server' interface. The top navigation bar includes 'CLIENT APPS', 'SAMPLES', 'SERVER ADMIN', and 'DIAGNOSTICS'. The breadcrumb trail is 'Home > Samples > Workflows'. The main content area is titled 'Server RDVMAEDEMO' and shows a list of 'SAMPLE WORKFLOWS PER MARKET SEGMENT'. The list includes five categories: 'Labels', 'Folding Carton', 'Flexibles', 'Sign & Display', and 'Commercial Print'. Each category has a description and an 'INSTALL' button. At the bottom, there is a lock icon and the text 'Click the lock to prevent further changes.'

The **Install** button will not be available when you haven't logged in and 'unlocked' this feature.

Click, at the bottom of the page, on  and confirm your credentials. This will unlock this feature and make the **Install** button available.

Important: These sample workflows are hosted on <https://mysoftware.esko.com/>. If no Internet connection is available while trying to install the sample workflows, an error message appears.



Caution: The installation of sample workflows will overwrite any workflows that were previously installed (for that same market segment). Any changes you made to those sample workflows will then be lost. Therefore, we advise to make a copy of the sample workflow in case you really wish to use it.

Tip: Click the link in the market segment description to read more about the workflow and to watch a video tutorial (if available).

4. Your Tickets View will show the resulting sample workflows:

The screenshot shows a search interface with a search bar containing the word 'sample'. Below the search bar is a table with three columns: 'Task Type', 'Ticket Name', and 'Description'. The table lists 15 workflow entries, all of which are 'Workflow' type. The ticket names are various combinations of 'SAMPLES SIGN AND DISPLAY EN Workflow', 'SAMPLES LABELS EN Workflow', 'SAMPLES FLEXIBLES EN Workflow', and 'SAMPLES COMMERCIAL PRINTING EN Workflow'. The descriptions are version numbers like 'V12.1 - Rev01' and 'V14.0 - Rev01'.

Task Type	Ticket Name	Description
Workflow	SAMPLES SIGN AND DISPLAY EN Workflow S	V12.1 - Rev01
Workflow	SAMPLES SIGN AND DISPLAY EN Workflow D	V12.1 - Rev01
Workflow	SAMPLES SIGN AND DISPLAY EN Workflow C	V12.1 - Rev01
Workflow	SAMPLES SIGN AND DISPLAY EN Workflow B	V12.1 - Rev01
Workflow	SAMPLES SIGN AND DISPLAY EN Workflow A	V12.1 - Rev01
Workflow	SAMPLES LABELS EN Workflow C	V12.1 - Rev01
Workflow	SAMPLES LABELS EN Workflow B	V12.1 - Rev01
Workflow	SAMPLES LABELS EN Workflow A	V12.1 - Rev01
Workflow	SAMPLES FLEXIBLES EN Workflow B	V12.1 - Rev01
Workflow	SAMPLES FLEXIBLES EN Workflow A	V12.1 - Rev01
Workflow	SAMPLES COMMERCIAL PRINTING EN Workflow B	V12.1 - Rev01
Workflow	SAMPLES COMMERCIAL PRINTING EN Workflow A	V12.1 - Rev01
Workflow	SAMPLES CAD EN VPE Create CAD Report Workflow	V14.0 - Rev01
Workflow	SAMPLES CAD EN HRMS Create CAD Report Workflow	V14.0 - Rev01

Sample input files and Job folders will be created in `/ExampleJobContainer/Sample Workflows`.

5. Rename the sample workflow, to avoid losing any changes.

6.8 Automated Launching of Workflows

These tools allow to launch workflows automatically:

- Access Points launch a workflow after they detected a signal from 'outside'. There are 5 types of Access Points. The Folder Access Point is the most classic example and is also often referred to as a 'hot folder'. Access Points have their own View in the Pilot.

Learn more in [Access Points](#) on page 1236.

- WebCenter can also launch workflows on Automation Engine (when they are in the same LAN). WebCenter can do this as a specific step in its own workflow, there called 'Submit Workflow'.

Learn more in the [WebCenter documentation](#). Here is a [direct link](#) to this specific topic (in v14.0).

- Some external systems can launch a workflow on Automation Engine via JDF or JMF commands. Although most of this functionality is also available through use of XML, some MIS vendors prefer to use JDF/JMF because of specific additional features.

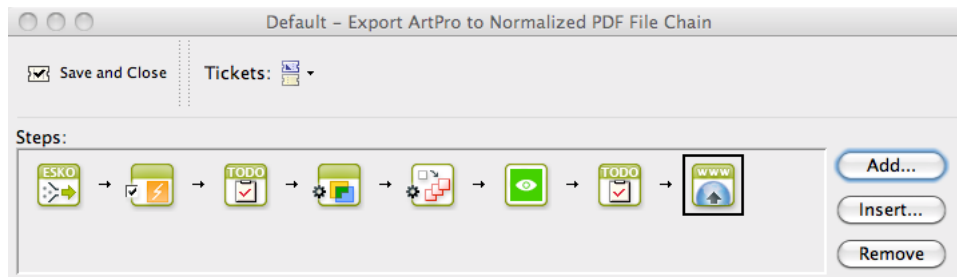
Learn more in [What about JDF?](#) on page 1230.

6.9 Migrating Old Task Chains

Old Workflow Technology

The Esko product BackStage, a predecessor of Automation Engine, had a limited workflow technology where workflows were only linear and were named **Task Chains**.

For example:



If you still have such task chains, they will still work but Esko strongly advises to migrate them to Automation Engine **Workflows**. See below how to do this.

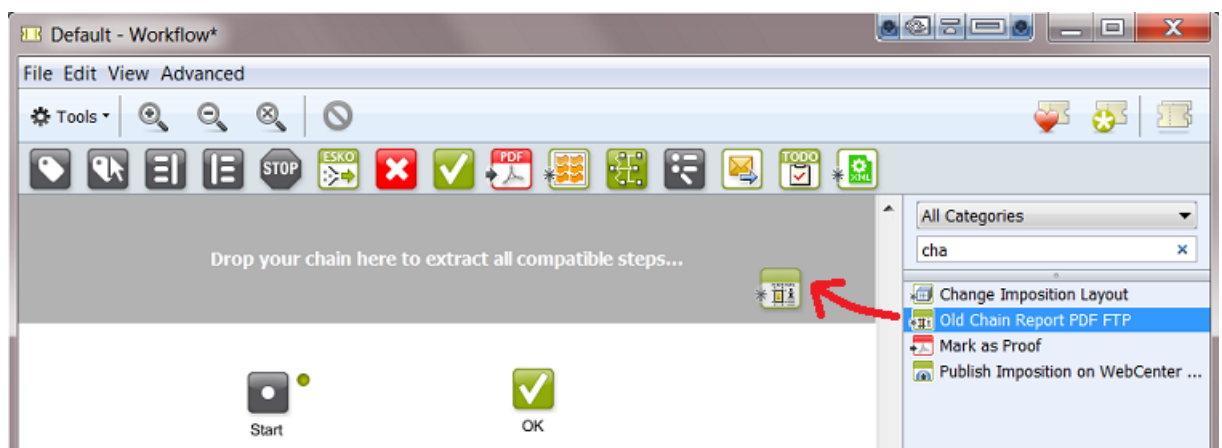
Note: Depending on your version and configuration, it is still possible that your Automation Engine offers some task chains by default. Page printing workflows still offer **Export to PDF File Chain (Mark as Proof)** and **Publish Imposition on WebCenter Chain** as chains of multiple steps. Also, a 'FlexRip plus InkPlanner Chain' ticket is still available (until you can add it to a workflow with an **Imaging Engine** step).

Migrating old Task Chains to new Workflows

You can not automatically convert task chain tickets into workflow tickets, but the workflow editor offers a tool that helps you migrating a task chain to a workflow that will keep all your settings.

Follow these steps to migrate a task chain to a workflow:

1. In the Pilot, click on **New Workflow**.
2. In the workflow editor, in the Tasks pane, select your task chain ticket.
3. Drag and drop your task chain ticket into the top part of the workflow editor window. A grey area will appear, showing you a message where to drop the ticket.



4. This starts a conversion tool that extracts all compatible steps:



- Each extracted step keeps all of its ticket settings.
- The step is named after the name of the task chain.
- The steps are not connected. You are invited to optimize your workflow construction. There may be better ways to build your workflow in the current version.

Note: When the chain contains deprecated steps, a warning appears:





Some chain steps will be excluded.
Only chain steps that are compatible with this workflow system are included.

Excluded chain steps:

#	Task Type
3	Mark File

OK

When you click **OK**, the chain is migrated but the deprecated step will disappear after a few seconds. This means that you should replace that step with a corresponding step that is available in your current version. For example you should replace the old **Mark File**  by **Mark** .

5. Based on these extracted steps, build your new workflow.

In below example we removed the old **Export to PDF** step because this functionality is now available on board the **ReportMaker** ticket:



6. Save your new workflow.
7. Test it and, when ready, delete the old task chain from your list of Tickets.

7. Working with Gang Run Printing

7.1 What is Gang Run Printing?

Gang Run Printing enables you to efficiently match graphic designs to be printed with the available substrate (to have as little substrate waste as possible and faster turnaround), with:

- *Nesting*,
- *Ganging*,
- *Tiling*.

This is especially useful when printing Sign and Display material and for shorter production runs.

It takes into account the designs' sizes, shapes and ordered quantities to produce layouts on substrate sheets.

7.1.1 Nesting

Nesting is laying out several graphic files in the most efficient way so that waste is minimized.

When working with non-rectangular graphics, their bounding boxes may overlap in the layout.



For more information about nesting in Gang Run Printing, see [Creating Nested Layouts on the Nesting Server](#) on page 1087.

7.1.2 Ganging

Ganging is combining several different graphic files, in one print job, even if those files belong to several orders (for example if one order doesn't fill a whole substrate sheet).



This helps drive production costs down and helps you offer a competitive price to your customers.

7.1.3 Tiling

Tiling is splitting oversize graphics, used for example for billboards, into rectangular pieces (tiles) for printing. These tiles can then be glued together.





When printing the tiles, Gang Run Printing can *nest* them to get the most efficient layout.

7.1.4 Manual or Automated Gang Run Printing

- In **i-cut Layout**, you can perform manual Gang Run Printing: selecting and preparing each graphic individually, then finding layouts (the most efficient layouts are calculated automatically).

When working manually in i-cut Layout, you can tweak the results if necessary. See the i-cut Layout manual for more information.

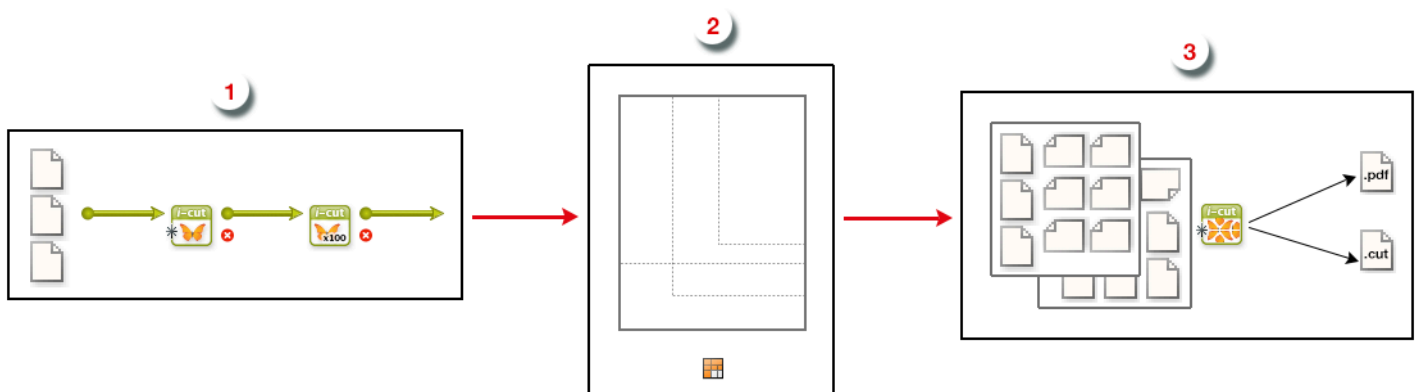
- In **Automation Engine**, you can automate the graphics selection and preparation too.

You can also submit your graphics for nesting on your chosen substrate, and manage your substrate queues.

When working automatically in Automation Engine, you can process more graphics in parallel, and you don't need an operator to keep checking when to generate layouts (this is done automatically).

7.2 Working with Gang Run Printing

Here is an overview of how you can work with the Gang Run Printing module.



1. You first **prepare** your graphics and **submit** them to be nested on a particular substrate, in your chosen quantity.

You do this through an **input** workflow, that should contain the **Prepare Graphics for Nesting** and the **Submit to Gang Run** tasks.

See [Preparing and Submitting your Graphics \(Input Workflow\)](#) on page 1085.

2. Once the graphics are submitted, they go in a **substrate queue**, and the **nesting server** calculates the best possible **nested layouts**, based on the **sheet sizes** and **nesting parameters** you defined.

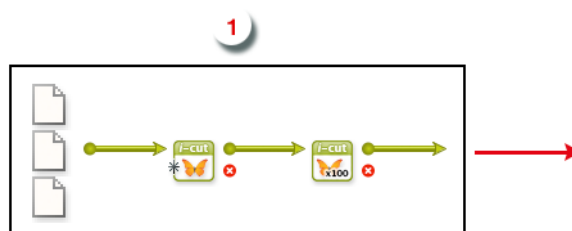
See [Creating Nested Layouts on the Nesting Server](#) on page 1087.

3. Once the nesting server has found one or more **optimal nested layouts**, it delivers them to your **output workflow**, that creates **production files** (layout files and cut files).

The output workflow must contain the **Export Nested Layouts** task. You can set this workflow to copy the production files to a specific folder and/or send the files to a press.

See [Getting Production Files out of your Nested Layouts \(Output Workflow\)](#) on page 1090.

7.2.1 Preparing and Submitting your Graphics (Input Workflow)



Preparing your Graphics

Before sending your graphics to be nested on the substrate of your choice, you need to make sure they have at least a **cut path**. You may also want to add **bleed** and **SmartMarks**.

You can use the **Prepare Graphics for Nesting** task to identify or extract a cut path from your graphic, and if necessary to add bleed and SmartMarks to it.

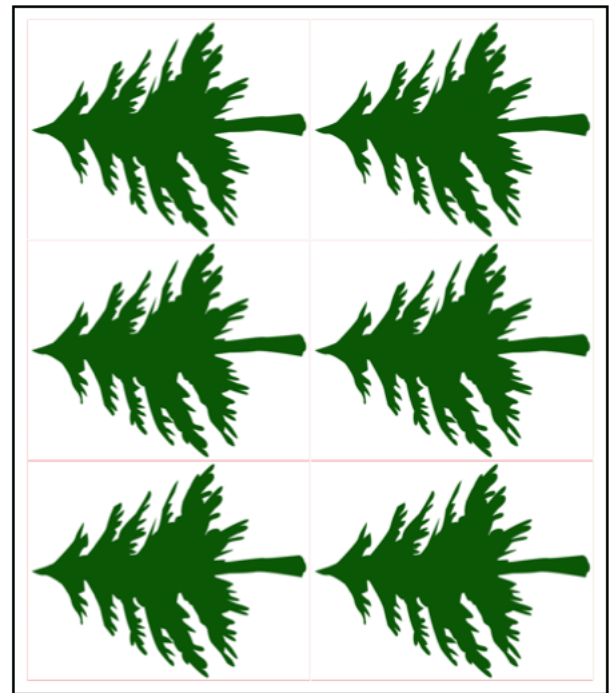
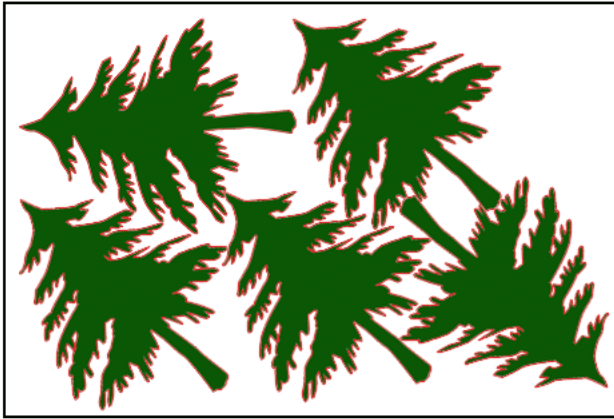
For more information about specific task parameters, please see [Prepare Graphics for Nesting](#) on page 1100.

Identifying or Extracting a Cut Path

Your graphics must have a cut path as they will be nested based on their cut path.

For example, cardboard cut-outs of a tree (as below left) will be nested and cut differently than postcards of a tree (as below right).

Depending on the case, your tree graphic will need a different cut path (shown in red).



The cut path is typically provided in the file as a contour line in a spot color.

For graphics that don't already contain a cut path, you can generate one from the white background, the trim box, or a separate file.

Adding Bleed

If your file doesn't already have bleed outside the cut path or trim box, you can generate some in the Prepare Graphics for Nesting task.

Adding SmartMarks

SmartMarks are marks that automatically update based on changes to the sheet size, the number of inks on the sheet, etc. You can use them as regular marks or as production controls.

There are many types of SmartMarks (registration marks, barcodes, text, placeholders for eyelets...).

You typically add SmartMarks to your nested layout in your [output workflow](#) (mostly for cutting the sheet), but in some cases you need to add some SmartMarks to your graphic in your input workflow.

For example:

- You can add eyelet marks to a graphic that will be used for a large eyeleted banner. You need to add the eyelet marks before [tiling the graphic](#) to fit the substrate.
- When defining the cut path in the input workflow, you can add text on this cut path for easy processing after cutting (for example to indicate the job and customer reference, or the order the tiles must be glued in...).

Tiling Oversize Graphics

In case the substrate size is smaller than the final printed object (for example when printing billboards), you can split the graphic into rectangular tiles (fitting the substrate size) that can later be glued together.

You can do this with the [Create Tiles](#) task. This task allows adding overlaps to be able to glue the tiles on top of each other.

Submitting your Graphics

Once your graphics *are ready*, you can send them for nesting using the [Submit to Gang Run](#) task.

The most important things to define in that task are:

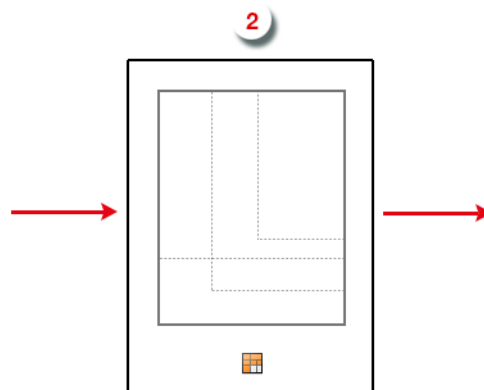
- how many copies of that graphic you want,
- on which substrate (and optionally on which substrate size) you want it printed,
- if you have a specific due date to get that graphic printed.

Note:

Substrates are organized into substrate queues, which not only define the substrate to use but also how the graphics will be nested.

See [Working with Substrate Queues](#) on page 1088 for more information.

7.2.2 Creating Nested Layouts on the Nesting Server



Automatic Calculation of Nested Layouts

Once your graphic(s) arrive in the [substrate queue](#), the nesting server calculates possible layouts to nest those graphics, together with any graphic(s) already in the queue.

The nesting server tries to find the most optimal layout (or layouts, depending on your graphics) possible, taking into account these substrate queue settings:

- the [nesting settings](#) you set up,
- the different possible substrate sizes (if applicable),
- the minimum fill percentage for that substrate queue.

If the minimum fill is reached, the nesting server sends the layout(s) to the output workflow to generate production files. All graphics that have been nested in these layouts are then removed from the substrate queue.

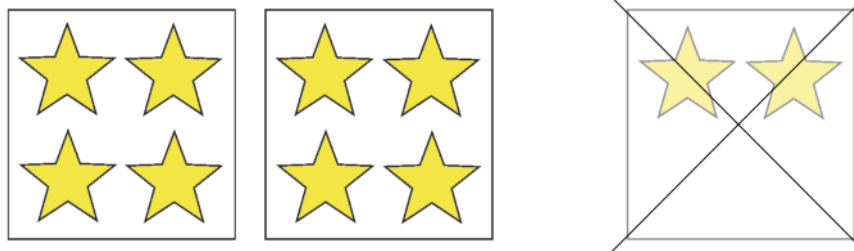
If it isn't reached, the graphics stay in the substrate queue until new graphics are submitted to the queue, at which point the nesting server calculates possible layouts again.

Outputting a Partial Quantity of a Graphic's Order

Sometimes the nesting server cannot fit all of the ordered copies of a graphic on layout(s) that satisfy the substrate queue settings.

In this case, rather than generating an extra layout that does not satisfy the minimum fill, the nesting server only outputs the quantity that does fit on the optimal layouts.

For example, if 10 copies of a graphic are ordered but only 8 fit on a layout satisfying the minimum fill, the nesting server does not generate a layout with only the remaining 2 copies. They will stay in the substrate queue.



Once new graphics are submitted to the queue, the nesting server will try to generate a nested layout containing the remaining 2 copies and the new graphics.

Generating Nested Layouts in Time for the Due Date

As your customers expect their printed goods to be delivered on a certain date, sometimes you cannot wait for the minimum fill to be reached. In this case, you can either:

- manually force a layout to be generated from the urgent graphics,
- enter a due date when submitting these graphics. The graphics will then be force-nested in time for that due date.

In both cases the nesting server generates a layout ignoring the minimum fill. However, it may also use other graphics from the queue in the layout to still raise the substrate's fill percentage.

Working with Substrate Queues

A substrate queue acts like a printer queue to which you can send jobs you want to nest and print on a particular substrate.

Substrate Queues Based on Material Types

For each type of substrate material you have in your stock, you should have a substrate queue.

For example:

In your stock you have 2 piles of PVC substrates:

- Thickness 3 mm
- Thickness 5 mm

For this stock you should set up 2 substrate queues:

- PVC 3mm
- PVC 5mm

Substrate Queues Based on Nesting Settings

It can also be useful to define substrate queues based on how the graphics will be nested and printed.

- If you have an expensive substrate, you can set the nesting server to always nest the graphics so that the substrate waste is minimal.
- If your graphics need to be nested on a material that will be printed in offset, you could aim for less plates (so less layouts) at the cost of extra waste.

So if some graphics needs to be nested differently (even if it is on the same material), you might need to create a different substrate queue for each type of printing/processing.

For example:

In your stock you have a pile of 250 g/m cardboard both for business cards and door hanger orders.

Business cards and door hangers will be cut differently:

- business card will be cut by a paper guillotine,
- door hangers will be cut on a cutting table.

The way graphics are cut influences the way they need to be nested. Since nesting settings are attached to a substrate queue, you need different substrate queues.

In this case you should set up 2 substrate queues:

- Cardboard for Business Cards
- Cardboard for Door Hangers

Grouping Certain Graphics Together

In certain cases, you don't want to group certain graphics on the same nested layout, even when they are to be printed on the same substrate or using the same printer.

For example you do not want to mix graphics that need varnishing with normal graphics, or you do not want to mix graphics from different customers.

In this case you do not necessarily need to set up separate substrate queues for graphics to be nested separately, but you can use grouping keys when submitting the graphics.

Even when submitting them to the same substrate queue, the nesting server will only nest graphics that have the same grouping key together.

For example:

In your stock you have 1 pile of 220 gsm photo paper. Depending on the order, some graphics printed on this paper need to be laminated, some don't.

In this case you only need to set up 1 substrate queue, for both regular and laminated graphics.

You can use a grouping key called "Lamination", with the values "Yes" or "No", to prevent laminated graphics being mixed with normal graphics.

Substrate Sheet Sizes

If you have a substrate material in several sheet sizes, you can add these sizes to the substrate queue for that material.

For example:

In your stock you have 3 piles of substrates:

- 1 pile of cardboard 2000x1000m
- 1 pile of cardboard 750x1200mm
- 1 pile of polystyrene 750x1200mm

For this stock you should set up 2 substrate queues:

- Cardboard (with 2 substrate sizes)
- Polystyrene (with 1 substrate size)

Substrate Rolls

If you are working with substrate rolls, you can still use Gang Run Printing: you can define your sheets sizes based on your roll's dimensions.

Keeping Track of your Stock

If desired, you can keep track of your stock of each different sheet size.

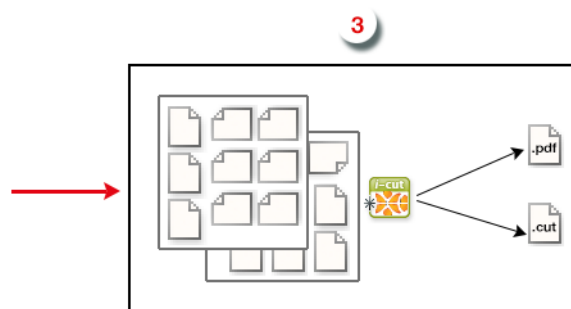
You indicate the quantity of sheets you have in that size, and every time the nesting server uses that sheet size for a nested layout, it will be subtracted from the available quantity.

Substrate Offcuts

The nesting server always tries to produce sheets with as little waste as possible. However, an approaching due date might force you to produce a sheet that has a significant portion of waste that could actually be re-used.

You can bring those rectangular leftovers (also called offcuts) back into circulation, by adding them to the substrate queue as a smaller sheet size, together with their quantity.

7.2.3 Getting Production Files out of your Nested Layouts (Output Workflow)



Every *substrate queue* is linked to an output workflow. You can build this workflow to your own needs, but it should at least contain the *Export Nested Layouts* task.

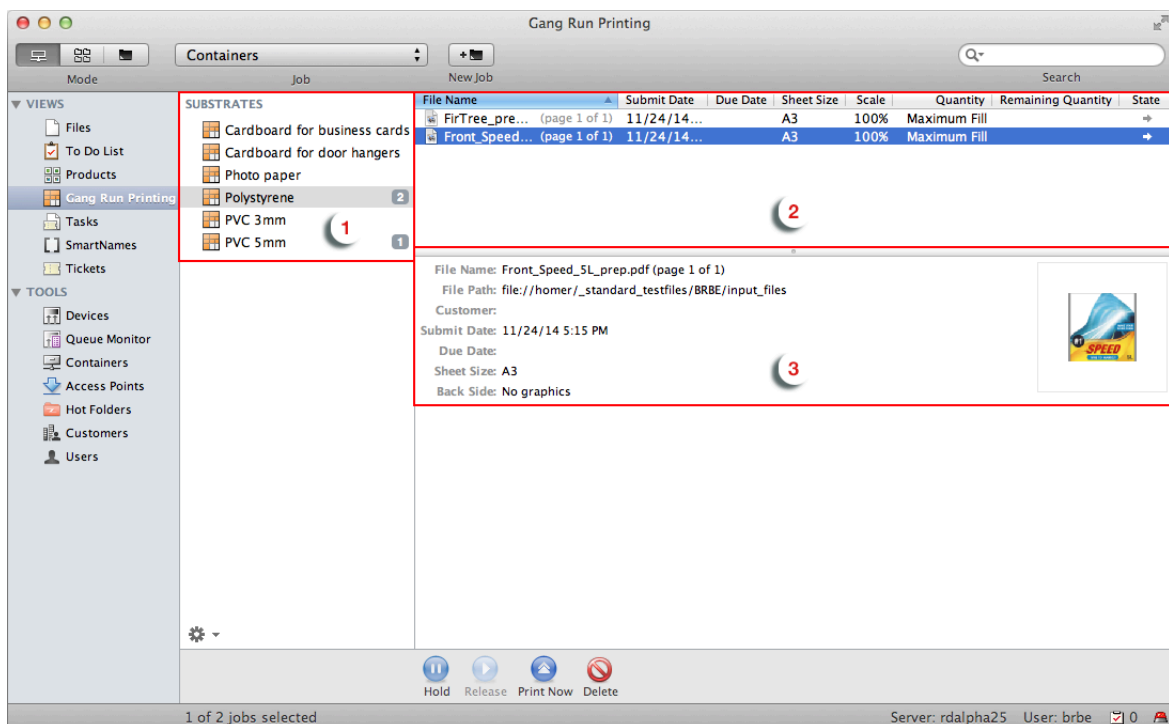
As soon as the nesting server has found one or more optimal nested layouts for your substrate queue, it delivers them to its output workflow, which is started automatically.

The output workflow then generates production files:

- A PDF file for the press, containing the nested layout, and marks for printing and cutting, but no technical inks.
- A CUT file for the cutting table, containing the combined cut path of the nested graphics.
- (Optionally) a nesting report, in PDF format (which can be used as guide by the cutting table operator) and/or in XML format (which can be used by the MIS for invoicing).

7.3 The Gang Run Printing View

The **Gang Run Printing** view in the Pilot shows an overview of all graphics that have been submitted for nesting on each of the available substrates.





1. Substrates available
2. Graphics submitted for nesting on the selected substrate
3. Details of the selected graphic

From this view, you can:

- *define substrates* and their characteristics,
- have an overview of all graphics *submitted* to each substrate queue, and their *state*,
- *control the printing flow* (for example you can put some graphics on hold, or force-print some of them).


7.3.1 Working with Substrates

When *submitting graphics* to be nested on a substrate, the graphics arrive in the corresponding *substrate queue* in the **Gang Run Printing** view.

In the **Substrates** list, you can see at a glance how many graphics have been submitted for nesting on a particular substrate.  Polystyrene 

The nesting server then calculates the best possible nested layout(s) for these graphics, based on:

- the graphics quantity ordered,
- the sheet sizes and nesting parameters you defined for your substrate queue.

The substrate queue icon  becomes animated during this calculation.

We strongly recommend you [define your substrates queues](#) here before submitting graphics to them.


Note:

If you submit graphics to a substrate queue that is not defined in the **Substrates** list (by entering an unknown substrate in the [Submit to Gang Run](#) task), this substrate queue will be created automatically in the **Substrates** list, but will miss crucial parameters (such as sheet sizes and a link to an output workflow), so the nesting server won't be able to calculate layouts for it.

You will see a warning sign on the substrate icon  to indicate that the substrate queue isn't properly configured. In this case, make sure to [add the missing parameters](#).

You can also manage your substrates (edit, duplicate or delete them) from the **Gang Run Printing** view.

Adding a Substrate

1. Click the action button  at the bottom of the **Substrates** list and select **New Substrate...**

You can also go to **File > New Substrate...** or press **Ctrl + N** on Windows / **Cmd + N** on Mac.

2. In the **New Substrate** dialog, enter the substrate **Name**.
3. In the **Sheet Sizes** tab, [specify the sheet size\(s\)](#) your substrate comes in.



Attention: You need to define at least one sheet size for the substrate, otherwise graphics sent to this substrate's queue will not be nested and printed.

4. In the **Nesting** tab, set the [nesting settings](#) for your substrate.

Graphics submitted for Gang Run Printing on this substrate will be nested according to these settings.


5. In the **Workflow** tab, [select the output workflow](#) to launch on the graphics submitted to that substrate's queue.




Attention: You must link an output workflow to your substrate, otherwise you won't get production files for the graphics you submitted to that substrate queue!

6. Click **OK** to save your substrate.

Note:

If you save your substrate without having defined all the necessary settings (for example if you haven't linked a workflow to your substrate), you will get a warning message, and there will be a warning sign on the substrate icon .

If this is the case, you need to [edit your substrate](#) to define the missing settings. When you are done, you should have a regular substrate icon .

Defining Sheet Sizes

For every sheet size you want to add:

1. Click **Add** (on Windows) or **+** (on Mac).
2. In the **Add Sheet Size** dialog, enter a **Name** for the sheet size you are adding.
3. In the **Size** parameters, enter the sheet's width and height.

The units used are those you defined in the Pilot Preferences (in **Edit > Preferences > Units** on Windows or **Automation Engine > Preferences > Units** on Mac).

4. Enter the **Margins** to use.

The margins won't contain nested graphics, but can contain marks.

5. If desired, you can keep track of the **Stock** you have of that sheet size.

To do this, select **Use available quantity of** and enter the number of sheets you have in that size.

Every time the nesting server uses that sheet size for a nested layout, it will be subtracted from the available quantity.

**Attention:**

If you are keeping track of the stock here and the stock reaches 0, the nesting server will not nest layouts on that sheet size anymore.

This can cause graphics to be stuck in the substrate queue (if this was the only sheet size defined for that substrate, or you chose to nest the graphics *on that sheet size specifically*).

When you get more sheets in that size, you should [edit](#) the available quantity.

6. Click **OK**.

Defining Nesting Settings

1. Select the **Nesting Type** to use for the layout:

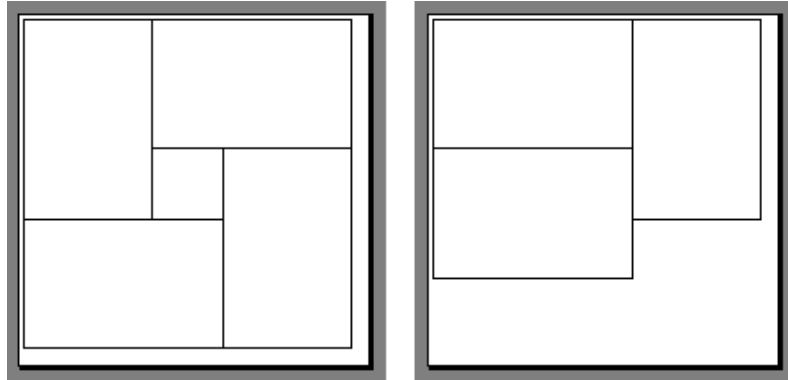
- **Minimal Waste** tries to reduce the waste in material to a minimum in the layouts generated.
- **Minimum Layouts** aims to generate as few layouts as possible, potentially at the cost of having some more waste.

Having fewer layouts can reduce setup time, making this option interesting for offset printing.

- **Guillotine Cut** generates layouts that can be cut on a Guillotine cutting table (which can only cut in straight edge-to-edge movements).

This usually reduces the number of cuts per layout but may increase the number of layouts and the waste.

In the example below, the layout at left cannot be cut on a Guillotine table. A Guillotine nesting can generate for example the layout at right, which can be guillotine-cut. The graphics that don't fit on that layout will be placed on another layout.



2. Select the **Search Method** to use for calculating the layout.

Depending on if you are running a test or processing graphics for production, you can set the layout search from **Very Fast** to **Slow (Very Accurate)** (the faster the search, the less accurate the result can be).

3. If you want to nest the graphics in the order in which they are coming in, select **Respect the graphics order**.
4. Depending on the **Nesting Type** you selected, you can set extra options:

If you have selected...	you can set:
Minimal Waste or Minimum Layouts	<ul style="list-style-type: none"> the Fill Starting Point on Sheet (where the first graphic will be placed): at the Top Left, Top Right, Bottom Left or Bottom Right, the Primary Fill Direction (in which direction the sheet will be filled): Horizontal or Vertical.
Guillotine Cut	the direction of the First Cut : Horizontal , Vertical or Don't care .

5. If desired, you can add a **Gutter** in between cut paths (specify the gutter width).
6. If the graphics in your files have a bleed outside the cut path, you can clip it away by entering the bleed distance into **Bleed Clipping Mask**.
7. When working with **Guillotine Cut**, you can choose to allow the nesting server to print a few extra copies of some of the graphics, so that you have more printed sheets with the same layout.

This reduces the amount of plates needed for offset printing, keeping the costs down.

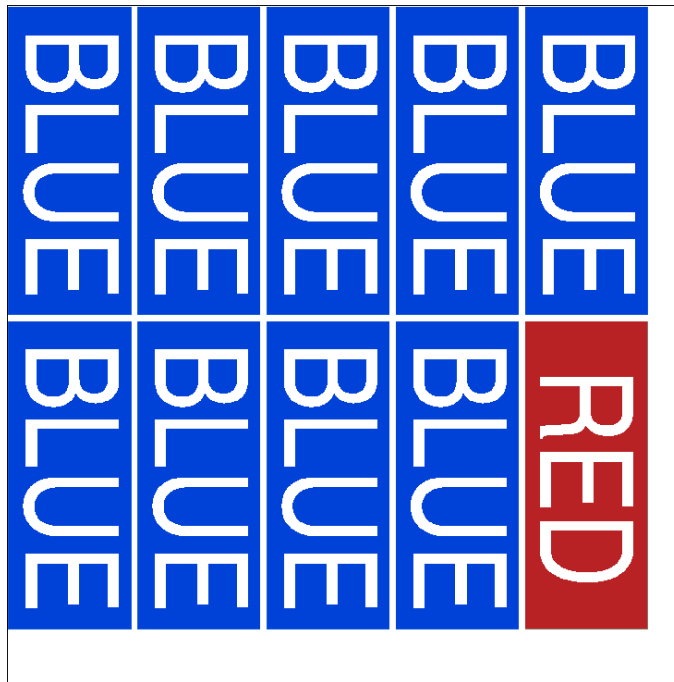
- If you want to do this, set a **Maximum Overrun**.

For example set a **Maximum Overrun** of 10.0% if you don't mind printing up to 10% more of some of the graphics to have less layouts.

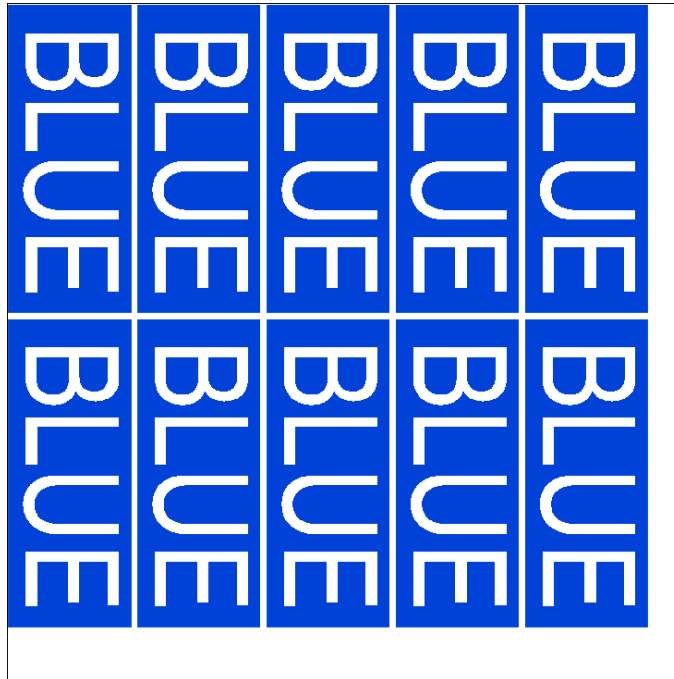
- If you want to only print the exact ordered quantity of each graphic, and don't mind the number of layouts, keep the **Maximum Overrun** at 0.0%.

For example, you need to print 100 "BLUE" graphics and 10 "RED" graphics, on a sheet size that can fit 10 graphics.

- If you don't allow overrun, you will have 2 layouts:
 - A layout with 9 "BLUE" and 1 "RED" graphics, to be printed 10 times (giving 90 "BLUE" and 10 "RED" graphics).

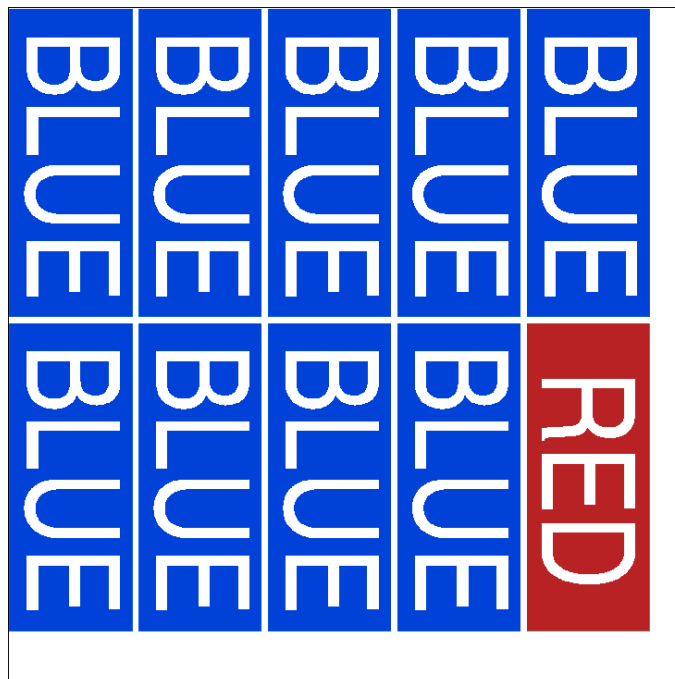


- A layout with 10 "BLUE" graphics, to be printed 1 time (for the remaining "BLUE" graphics).



- If you allow a **Maximum Overrun** of 20.0%, you will have only 1 layout, with 9 "BLUE" and 1 "RED" graphics, to be printed 12 times.

This gives 108 "BLUE" graphics (8% more than ordered) and 12 "RED" graphics (20% more than ordered).



8. In **Rotation**, specify if the task can rotate the graphics in the layout to get a better fit. You can choose:
 - **Allow any angle**,
 - **Allow 90 degree**,
 - **Allow 180 degree**,
 - **No rotation**.
9. Enter a **Minimum Fill** if you only want to output the sheet when it's filled to a certain level.

**Attention:**

Do not put 100% here as only the graphics inside the cut path count as fill. The margins, gutters etc. all count as waste.

By default, the minimum fill is set to 75%.

Linking a Workflow to Your Substrate

1. Select the output **Workflow** to launch on the graphics submitted to that substrate's queue.

This *output workflow* will be launched either:

- when the *minimum fill percentage* for that substrate is reached,
- in time for the submitted graphics's *due date* (if they have one),
- when you *force print* one or more graphics from the substrate queue.




Attention: You must link an output workflow to your substrate, otherwise you won't get production files for the graphics you submitted to that substrate queue!

2. If you have *set a due date* for the graphics to be printed, you can choose to **Launch the workflow automatically** a certain number of **hours** and/or **minutes before the due date**.

If the workflow hasn't been launched by then (because the minimum fill percentage for that substrate hasn't been reached), it will be automatically launched at that time (even if the fill percentage isn't reached).

Editing a Substrate

1. Select the substrate that you want to edit, click the action button  and select **Open**.


Tip: You can also go to **File > Open** or press **Ctrl + O** on Windows / **Cmd + O** on Mac, or just double-click the substrate to edit.

2. In the **Edit Substrate** dialog, you cannot change the substrate **Name**, but you can edit all the other settings.

Note: If you want to change the substrate name, we recommend you *duplicate the substrate* under your chosen name, then delete the original substrate.


3. In the **Sheet Sizes** tab, you can add, modify or delete sheet sizes for the selected substrate.

- to add a sheet size, click **Add** (on Windows) or **+** (on Mac) and define the *sheet size settings*.

- to modify a sheet size, select it, click **Edit** (on Windows) or  (on Mac), and change the settings as desired.
 - to delete a sheet size, select it and click **Remove** (on Windows) or - (on Mac).
4. In the **Nesting** tab, you can edit the *nesting settings* for the substrate.
 5. In the **Workflow** tab, you can *link a different workflow to the substrate* or change when to launch it automatically.
 6. Click **OK** to save your changes or **Cancel** to revert.

Note: After changing the substrate settings, the nesting server will automatically re-calculate nesting layouts for the graphics in that substrate queue, even if no new graphic was submitted.

Duplicating a Substrate

1. Select the substrate that you want to duplicate, click the action button  and select **Duplicate...**

Tip: You can also go to **File > Duplicate...** or press **Ctrl + D** on Windows / **Cmd + D** on Mac.


2. In the **Duplicate Substrate** dialog, enter a **Name** for the new substrate.
3. The substrate settings are the same as the old substrate's, but you can modify them if necessary.

See:

- [Defining Sheet Sizes](#) on page 1093
- [Defining Nesting Settings](#) on page 1093,
- [Linking a Workflow to Your Substrate](#) on page 1097.

4. Click **OK** to save your new substrate.

Deleting a Substrate

1. Select the substrate that you want to delete, click the action button  and select **Delete**.



Tip: You can also go to **File > Delete** or press the **delete** key.

2. In the dialog that pops up, click **Delete** to confirm that you want to delete that substrate.

7.3.2 Working with Graphics Submitted for Printing

Graphics Overview

List of Graphics per Substrate

If there are graphics submitted for nesting on a substrate, you will see the number of graphics to be nested next to that substrate queue.  Polystyrene  2

Click the substrate to see the list of graphics and their printing information at right.

Printing Information

For each graphic, you will see:

- the **File Name** and the page to be printed in that file,
- the **Sheet Size** it should be printed on (if this was defined when submitting the graphic),
- the **Quantity** ordered,
- the **Remaining Quantity** to be printed (if some copies of the graphic have already been included in a nested layout this is what remains in the queue),
- the **State** of the graphic in the substrate queue,
- the date it was submitted (**Submit Date**),
- the **Due Date**.

This information comes from the settings defined when *submitting* the graphic to the queue.

If you want to find a specific graphic in the list, you can filter the graphics' list using the search field

, or sort the graphics by any of the columns by clicking the column header.

The sorting order has no influence on the output.

Tip:

You can also show / hide or reorganise columns by going to **View > Select columns in 'Gang Run Printing'...**

For example, you can show columns with your *labels and grouping keys* and use that to sort graphics.

Your column settings are saved when you quit the Pilot and restored the next time you log in.

Graphics States



➔	<p>Graphics ready to be nested and output.</p> <p>When the nesting server calculates possible layouts, it will try to include all of the graphics with this state in the most efficient layout(s) possible.</p>
	<p>Graphics on hold.</p> <p>Graphics with this state will not be included in the layout calculations.</p>
✘	<p>Graphics that couldn't be placed on a layout because they are too big. It could be that:</p> <ul style="list-style-type: none"> • The graphic is too big for any available sheet size of that substrate. In this case, consider either <i>defining a bigger sheet size</i> (if you have some available), or choosing a different substrate when <i>submitting</i> the graphic to the queue. • The graphic is too big for the sheet size you chose to nest it on (when submitting it to the queue). In this case, choose a bigger sheet size, or set the graphic to be nested on Any sheet size of your substrate when submitting it to the queue. • It's too big to fit within the rotation possibilities defined for that substrate queue. In this case, change the <i>rotation parameter</i> to Allow any angle.

If none of these solutions help, you need to [tile](#) your graphic before submitting it to the substrate queue.




Controlling the Printing Flow

By default, graphics will be sent to the output workflow automatically either when the [desired fill](#) is reached or at the [time you set](#) (a certain amount of time before the due date).

However, you can:

- Put graphics on hold: select the graphic(s) and click the **Hold** button . Graphics on hold have a hold icon  next to them in the table.


Note: You can put one or more graphics (or even all graphics in the queue) on hold, but you cannot put the queue itself on hold: even if all of the queue's graphics are on hold, new graphics submitted to the queue will still trigger a layout calculation.

- Release graphics that are on hold: select the graphic(s) and click the **Release** button . Graphics that are not on hold have a ready icon .
- Force-print graphics: select the graphic(s) and click the **Print Now** button .

Note:

To get a better substrate usage, some other graphics may be printed along the graphic(s) you force-print. In this case, you will see that either these graphics disappear from the list, or their **Remaining Quantity** is reduced.

Graphics that are on hold will not be included. If you want to force-print a graphic on hold, you need to release it first.

- Remove a graphic from the queue: select the graphic and click the **Delete** button .

Note: This will not delete the graphic file itself, only remove it from the substrate queue.

7.4 The Gang Run Printing Tasks

7.4.1 Prepare Graphics for Nesting

Use this task to [prepare](#) individual graphics for Gang Run Printing (you can add a cut path, bleed, and SmartMarks in the [General](#) tab, and optimize your cut path in the [Optimize](#) tab).


You can send the following graphic file formats to this task:

- TIFF (CMYK or grayscale),
- JPEG (CMYK or grayscale, progressive JPEG compression is not supported),
- PSD (CMYK or grayscale),
- PDF (single page or multipage),
- Normalized PDF (single page or multipage).

Typically, you use this task before a [Submit to Gang Run](#) task in an [input workflow](#). The input workflow prepares the graphics and submits them to a [substrate queue](#) on the nesting server (see [Working with Gang Run Printing](#) on page 1084 for an overview of the whole Gang Run Printing process).

General Tab

1. In **Create Cut Path**, define what to use as a cut path, or how to generate one.

If you want to...	do the following...
use an ink as cut path	<ol style="list-style-type: none"> 1. Select From Ink or Trim Box. 2. Enter the name of the ink to use in the field below. <ul style="list-style-type: none"> Tip: <ul style="list-style-type: none"> • If the ink used for the cut path has different names in different input files (for example if you have customers from different countries using names like <code>cut</code>, <code>découpe</code>, <code>taglio...</code>), you can add several ink names, separated by a comma (,). <p>The order you use is important as the task will use the first ink it finds in the list as cut path!</p> <ul style="list-style-type: none"> • You can use wildcards and/or use SmartNames to define ink names. For example use <code>Pantone*</code> if you want to use any Pantone ink used in the file as a cut path. <ul style="list-style-type: none"> Tip: <p>When building a workflow using an example file, click  to select one or more of the inks from the example file.</p> <p>If you always receive CMYK graphics with one spot color used for the cut path, select Any SpotColor.</p> <ol style="list-style-type: none"> 3. Define what to do if the ink(s) you defined are not present in the file: <ul style="list-style-type: none"> • If you want to use the trim box as cut path instead, select Create cut path from trim box if no ink is found. • If you don't want to use the trim box but want the task to end in error, leave Create cut path from trim box if no ink is found deselected.
use the trim box as cut path	<p>Select From Ink or Trim Box (without specifying an ink to search for).</p> <p>If you always want to use the trim box as cut path (usually for rectangular graphics like business cards), you don't need to select Create cut path from trim box if no ink is found (as this</p>

If you want to...	do the following...
	option is only applicable when you want an ink as cut path as your first choice).
use the cut path defined in an external ARD, PDF or Normalized PDF file	Select From File and Browse to that file (or use SmartNames to define the path to the file).
create a cut path from the contour of the graphic on its white background	<p>Select From White Background.</p> <p>Note:</p> <p>When creating a cut path this way, we recommend you select the Flatten Objects option.</p> <p>This will include both the contour of the design on the white background and the contours of any "white holes" in the design in the cut path. You can then remove the "white holes" contours using the Remove holes option.</p>

2. In **Cut Path's Ink Name**, enter the ink name to use for the cut path you just defined.

- If you defined the cut path based on an ink in your file, this will rename that ink to the name you enter here.

Tip: Use this to standardize cut path ink names across files (especially when your files come from different customers).

- If you defined the cut path from the file's trim box, white background, or an external file, this will add the cut path in that ink in your graphic file.

The cut path ink name you enter here will be used to identify the cut path in the [Submit to Gang Run](#) task later in the [input workflow](#).

3. If the graphic contains no bleed, you can add some here.

Select **Create Bleed** and enter the bleed distance (the distance outside the cut path). You can also use a SmartName.

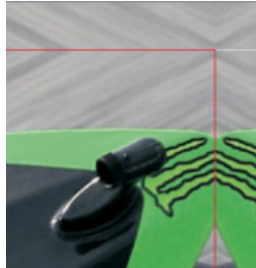
You can choose between two **Bleed Modes**:

- Choose **Clone** to take the part of the graphic just inside the cut path and replicate it to the outside of the cut path.



This is typically the best option for non rectangular cut path shapes.

- Choose **Mirror** to take a part of the graphic just inside the cut path (as much as the bleed distance) and mirror it to the outside of the cut path.



This is typically the best option for rectangular cut path shapes.

4. If desired, you can add **SmartMarks** (a SmartMarks set that you created in i-cut Layout or PackEdge) to the resulting graphics.

For example, you can add marks around the cut path to indicate the customer and the job name. These will be visible in the bleed on the cutting table and can help organize your printed graphics after cutting.

You can use SmartNames to enter your SmartMarks set's name.

Note:

Before you can do this, you need to:

1. Make sure your icut Layout or PackEdge application is connected to your Automation Engine server.
2. Create and save a SmartMarks set in icut Layout or PackEdge (this automatically saves it to the correct location on your Automation Engine server: \\AEservername\bg_data_marks_v010\dat).

See the icut Layout or PackEdge manual for more information about connecting to an Automation Engine server and creating a SmartMarks set.

Optimize Tab

If you created the cut path with the **From White Background** method, you typically need to optimize the cut path contours that were automatically generated, to facilitate and speed up the cutting process.

You can:

- **Remove holes** in your cut path.

Select this if the cut path was extracted from a white background and your graphic contains white. This option makes sure that the white parts in your graphic are not considered part of the background (which would create holes in your graphic).

- Smooth small jagged edges in your cut path.

Select **Smooth cut path with minimum distance between path points** and enter the desired distance. Any point that deviates from the cut path by less than that distance will be smoothed.

- Remove small objects from your cut path.

Select **Remove contours smaller than** and enter the minimum size for your graphic's small objects. Any objects smaller than this will be removed from the cut path.

- Move the cut path inwards or outwards.

Select **Offset cut path** and enter a positive offset value (to move it outwards) or a negative one (to move it inwards).

You can use SmartNames to specify any of these values.

7.4.2 Create Tiles

Use this task to tile oversize graphics, used for example for billboards, into smaller graphics to be printed. You can select the size/number of the tiles, and allow for an overlap if necessary.



You can send the following graphic file formats to this task:

- TIFF (CMYK or grayscale),
- JPEG (CMYK or grayscale, progressive JPEG compression is not supported),
- PSD (CMYK or grayscale),
- PDF (single page or multipage),
- Normalized PDF (single page or multipage).

Note:





- For each image file or single page PDF, the task outputs a multipage PDF in which each page contains a tile.
- For multipage PDFs, the task outputs a multipage PDF (in which each page contains a tile) for each page of the input multipage PDF.

1. In **Method**, choose how to determine the number and size of the tiles:

- If the tile size is the most important (if for example you can only print a certain size), choose **Use Fixed Tile Size** and enter the horizontal  and vertical  **Tile Size**.

Note: The last tiles might be smaller (depending on the tile size and the size of the original file).

Select **Allow Rotation** if you don't mind having the graphic rotated by 90 degrees in case this fits better and generates less tiles.

- If the number of tiles is more important to you than their size, choose **Use Grid** and enter the **Number of Tiles** you want horizontally  and vertically .
- If you don't know the exact number or size of the tiles you want, but you can only print up to a certain size, choose **Automatic Tiling** and enter the **Maximum Tile Size** horizontally  and vertically .

The number and size of the tiles are calculated to have the least waste and the least tiles possible, within that maximum size.

All tiles will have the same size, except when [adding an overlap](#).

Select **Allow Rotation** if you don't mind having the graphic rotated by 90 degrees in case this fits better and generates less tiles.

- If you have a tiling template that you have created in icut Layout and want to use here, choose **Use Template** and select your template in the **Template** list.

Note:

To do this, you need to:

1. Make sure your icut Layout application is connected to your Automation Engine server.
2. Create and save the template in icut Layout (this automatically saves it to the correct location on your Automation Engine server: \\AEservername\bg_data_custom_v010\dat\tiling).

See the icut Layout manual for more information about connecting to an Automation Engine server and creating a tiling template.

With a tiling template:

- you can see exactly how your graphic will be tiled when creating the template in icut Layout (if you create the template from the same/a similar graphic that you will use in Automation Engine),
- you make sure you have consistent tiling in icut Layout and Automation Engine (for example if you use icut Layout for some graphics orders and Automation Engine for others).




Attention: If you want to add overlaps to your tiles when using a template, you need to already add them in the template in icut Layout. You cannot add overlaps to tiles created from a template in the **Create Tiles** task.

If you enable **Fit graphics to grid**, your graphic will be scaled to match the grid size, both vertically and horizontally. This can result in non-proportional scaling!

2. Define what to use as **Cut Path**.

Note: Whereas the [Prepare Graphics for Nesting](#) task allows you to generate a cut path if the graphic doesn't contain one, in the **Create Tiles** task the cut path needs to be already available, either as an ink or by using the trim box as cut path.

If you want to...	do the following...
use an ink as cut path	<ol style="list-style-type: none"> 1. Enter it in the Cut Path field.

If you want to...	do the following...
	<p>Tip:</p> <ul style="list-style-type: none"> If the ink used for the cut path has different names in different input files (for example if you have customers from different countries using names like <code>cut</code>, <code>découpe</code>, <code>taglio...</code>), you can add several ink names, separated by a comma (,). <p>The order you use is important as the task will use the first ink it finds in the list as cut path!</p> <ul style="list-style-type: none"> You can use wildcards and/or use SmartNames to define ink names. For example use <code>Pantone*</code> if you want to use any Pantone ink used in the file as a cut path. <hr/> <p>Tip:</p> <p>When building a workflow using an example file, click  to select one or more of the inks from the example file.</p> <p>If you always receive CMYK graphics with one spot color used for the cut path, select Any SpotColor.</p> <p>2. Define what to do if the ink(s) you defined are not present in the file:</p> <ul style="list-style-type: none"> If you want to use the trim box as cut path instead, select Create cut path from trim box if no ink is found. If you don't want to use the trim box but want the task to end in error, leave Create cut path from trim box if no ink is found deselected.
use the trim box as cut path	<p>Leave the Cut Path field empty.</p> <p>If you always want to use the trim box as cut path (usually for rectangular graphics like business cards), you don't need to select Create cut path from trim box if no ink is found (as this option is only applicable when you want an ink as cut path as your first choice).</p>

The task will tile the graphic that is inside the cut path. The tiles created will have a cut path in the same ink as the input graphic's cut path (if the cut path is created from the trim box, this will be the `Cut` ink).

- Typically, graphics for large banners or billboards are not supplied in the size at which they will be printed out. In this case, you can **Scale** them up here.

For example, enter `1000%` if your graphic is only `10%` of the size you will need it to be.

- If the graphics in your files have a bleed outside the cut path, you can clip it away by entering the bleed distance into **Bleed Clipping Mask**.

5. If desired, you can add **SmartMarks** (a SmartMarks set that you created in i-cut Layout or PackEdge) to the tiles.

For example, you can add SmartMarks containing glueing instructions in the overlap zones. You can use SmartNames to enter your SmartMarks set's name.

Note:

Before you can do this, you need to:





1. Make sure your icut Layout or PackEdge application is connected to your Automation Engine server.
2. Create and save a SmartMarks set in icut Layout or PackEdge (this automatically saves it to the correct location on your Automation Engine server: \\AEservername\bg_data_marks_v010\dat).

See the icut Layout or PackEdge manual for more information about connecting to an Automation Engine server and creating a SmartMarks set.


Tip: We recommend you define your SmartMarks set in i-cut Layout, as it contains tiling-specific SmartMarks fields (for adding tile numbers).

6. If you want to include overlaps (for example if the tiles' edges are to be glued on top of each other), select **Add Overlap** and click the **Options...** button to set how exactly where and how big the overlaps should be.

In the **Overlap Options** dialog:

- a) Enter the **Overlap** distance.
- b) In **Tile Start Position**, select where you want the first overlap to be: on the top left , top right , bottom left  or bottom right  tile.

Note:

You can use SmartNames for most options of this dialog. For button options like **Tile Start Position**, **Direction** and **Pattern**, you can set a SmartName by clicking , and edit it by clicking **Edit...**

Tip:



To know what the SmartName should resolve to to get a certain value for a parameter, do the following:





1. Right-click the parameter value and select **Copy Parameter Value**.

- For a parameter with a field to edit (like **Overlap**), enter the desired value in the field then right-click in that field.
- For a parameter with buttons (like **Tile Start Position**), right-click the button of your choice.

2. Paste this in a text editor.


For example, the SmartName to get a top left  **Tile Start Position** should resolve to t1.

- c) In **Direction**, select whether the overlaps should run horizontally  or vertically .

Depending on the **Tile Start Position** you chose, horizontal overlaps can run from left to right  or from right to left , and vertical overlaps can run from top to bottom  or from bottom to top .

- d) In **Pattern**, choose whether you want the overlaps to run snake-wise  or in zig-zag .

The exact snake-wise or zig-zag pattern depends on what **Tile Start Position** and **Direction** you selected.

For example, when starting on the top left tile  and running horizontally  with a snake-wise pattern , you will get something like this:



We recommend you run a test to determine exactly what pattern works best for your graphic.

- e) You can choose to add an **Overlap on both sides** of each tile (using the overlap distance you set on both sides).

In this case, you cannot add a glue area to your overlaps.

- f) If desired, add a **Glue Area** to your overlaps.

This area will not be printed and you can add glue to it after cutting the tiles.

- g) Click **OK**.



Attention: Adding overlaps increases the tiles size (of all tiles except the last ones). If you selected the **Use Fixed Tile Size** or **Automatic Tiling Method**, you should take the overlap distance into account.

7. If you want a report detailing how the graphic was tiled, select **Create tiling report** and click the **Options...** button to choose how the report will look like.
- In **Report Size**, select if you want your report on an **A4** or a **Letter** page.
 - In **Units**, choose if you want the report to use **mm** or **inch**.
 - Click **OK**.

7.4.3 Submit to Gang Run

Use this task to *submit* a graphic file to a *substrate queue*, where it will be nested with other graphics to optimize substrate usage.

In this task, you can specify parameters (quantity, scale factor...) for nesting graphics based on their cut path.

You can send the following graphic file formats to this task:

- TIFF (CMYK or grayscale),
- JPEG (CMYK or grayscale, progressive JPEG compression is not supported),
- PSD (CMYK or grayscale),
- PDF (single page or multipage),
- Normalized PDF (single page or multipage).

Read more in:

[General Tab](#) on page 1109

[Back Tab](#) on page 1112

[Substrate Tab](#) on page 1112

[Extra Tab](#) on page 1113

General Tab

1. Define how many of each input graphic you want to print.


You can either:

- Enter the **Quantity** to print.
 - Select **Maximum Fill** if you want to print as many copies as will fit in one sheet.
2. Typically, graphics for large banners or billboards are not supplied in the size at which they will be printed out. In this case, you can **Scale** them up here.

For example, enter 1000% if your graphic is only 10% of the size you will need it to be.

3. Define what to use as **Cut Path**.

Note: Whereas the *Prepare Graphics for Nesting* task allows you to generate a cut path if the graphic doesn't contain one, in the **Submit to Gang Run** task the cut path needs to be already available, either as an ink or by using the trim box as cut path.

If you want to...	do the following...
use an ink as cut path	<p>1. Enter it in the Cut Path field.</p> <p>Tip:</p> <ul style="list-style-type: none"> If the ink used for the cut path has different names in different input files (for example if you have customers from different countries using names like <i>cut</i>, <i>découpe</i>, <i>taglio...</i>), you can add several ink names, separated by a comma (,). <p>The order you use is important as the task will use the first ink it finds in the list as cut path!</p> <ul style="list-style-type: none"> You can use wildcards and/or use SmartNames to define ink names. For example use <i>Pantone*</i> if you want to use any Pantone ink used in the file as a cut path. <p>Tip:</p> <p>When building a workflow using an example file, click  to select one or more of the inks from the example file.</p> <p>If you always receive CMYK graphics with one spot color used for the cut path, select Any SpotColor.</p> <p>2. Define what to do if the ink(s) you defined are not present in the file:</p> <ul style="list-style-type: none"> If you want to use the trim box as cut path instead, select Create cut path from trim box if no ink is found. If you don't want to use the trim box but want the task to end in error, leave Create cut path from trim box if no ink is found deselected.
use the trim box as cut path	<p>Leave the Cut Path field empty.</p> <p>If you always want to use the trim box as cut path (usually for rectangular graphics like business cards), you don't need to select Create cut path from trim box if no ink is found (as this option is only applicable when you want an ink as cut path as your first choice).</p>

4. If you want to set a due date (when the sheet with the nested graphics needs to be printed), select **Due Date** and choose when you want the due date (and the due time) to be.

- If you set a due date, the nesting server will always *generate a nested layout* that includes this graphic before your due date, even when the minimum fill has not been reached.

Tip: In your substrate queue settings, you can choose to generate the nested layout and send it to the output workflow *a specific time before the due date*. This allows time for printing, production...

- If you don't set a due date, the nesting server will only generate a nested layout when the minimum fill is reached.

Select...	to...
Immediate	generate a nested layout including your graphic immediately (this is the same as when you use <i>Print Now</i> in the Gang Run Printing view).
Today	set the due time to a certain time today (this is the default option). The nesting server will generate a nested layout including your graphic <i>the specified amount of time</i> before the due time.
Tomorrow	set the due date to a certain time tomorrow. The nesting server will generate a nested layout including your graphic the specified amount of time before the due time.
In ... hours	set the due time a certain number of hours after the task is launched (you can set this with a SmartName).
In ... days	set the due date a certain number of days after the task is launched (you can set this with a SmartName). You also need to set the due time.
On	set the due date to a specified date (and time) in the future.
Job due date	set the due date to the due date specified in the job setup.
Select SmartName...	set the due date using a SmartName. This opens the SmartNames dialog where you can select your SmartName.

Select...	to...
	<p>Note:</p> <ul style="list-style-type: none"> • See SmartNames in the general Automation Engine manual for information about SmartNames and how to create custom ones. • Your SmartNames must resolve to a value in the ISO 8601 standard format (for example 2014-09-27T11:08:00+02:00). • If your due dates are supplied in a different format (for example 27/09/14), use the formatting options for date and time SmartNames to convert this format to the ISO standard.

Back Tab

- On the **Back** tab, you can specify what to print on the back of each graphic:
 - **No graphics:** choose this if you don't want any back side for the graphic.
 - **Even pages of multipage file:** choose this to print the odd pages of the file on the front, and the even pages on the back (this only works with multipage PDFs).
 - **Graphics from front side:** choose this to print the same graphic on the front and on the back.

If your graphic has an irregular shape, you can select **Flip horizontally** so both graphics face the same direction and have the exact same contour for cutting.

Note: If your graphic contains text, you shouldn't flip it as this will make the text unreadable.

- **Other graphics:** choose this to print different graphics on the back, and **Browse** to these graphics.

Tip: Give the graphics to print on the back a consistent name, and use SmartNames to retrieve them (for example, [File]_back.[Extension of File]).

Substrate Tab

1. Choose the **Substrate** to print your graphics on.



Attention:

You should choose (one of) the substrate(s) you have defined as a [substrate queue](#) in the [Gang Run Printing view](#).

If you submit your graphics to a non-existent substrate, a substrate queue will be created automatically for this substrate, but it won't be ready to process your graphics (so your graphics will be stuck in the queue until you [configure it](#) properly).

2. If this substrate has several sizes defined, you can choose which **Substrate Size** to use.

Or leave this on **Any** to let Automation Engine find the best fit.

Extra Tab

In this tab you can enter:

1. A **Customer ID** identifying the customer who ordered the graphics.
2. Your desired values for the custom fields you defined in the **Configure** tool.

You can use both customer ID and custom fields either as:

- labels in the [reports](#) about the nested layouts or the graphics,
- grouping keys to define [which graphics can be nested together](#).

Example 1: Using Custom Fields as Labels

You get graphics orders from national and international customers. You are using different shipping companies for national and international orders.

When generating layouts, you want to indicate which graphics are for national and for international shipping, so the cutting operator can store the finished graphics in the right places.

1. In the [Configure tool](#), define a **Custom Field** as `Shipping`.
The corresponding **Custom Field** parameter in this task tab will now be called **Shipping** instead.
2. When launching the task on graphics from international orders, enter `International` in that parameter, otherwise enter `National`.
3. In your [output workflow](#), choose to generate a **layout PDF report** in the [Export Nested Layouts task](#).
4. In the layout report, you will see for each graphic name:
 - the label **National** if the copies of this graphic need to be shipped nationally,
 - the label **International** if the copies of this graphic need to be shipped internationally.

Layers

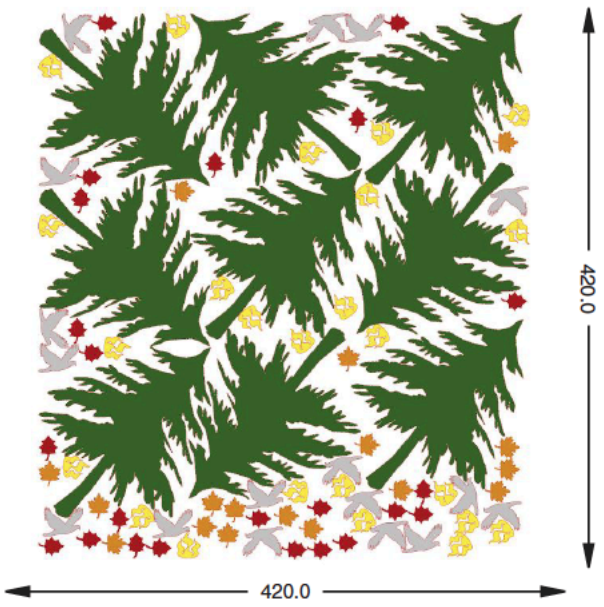
Preview

Dummy Colors

Shipping

Base











nesting11700_254677088



420.0

420.0

BRBE - A3 # = 1 48.95% 51.05%
 \\homer_standard_testfiles\BRBE\nesting11700_254677088.pdf
 \\homer_standard_testfiles\BRBE\nesting11700_254677088.cut

	AutumnLeaf_prep.pdf	18.8 x 16.51	17	 National
	AutumnLeaf2_prep.pdf	19.15 x 14.67	23	 National
	Dove_prep.pdf	21.93 x 29.97	17	 National
	FirTree_prep.pdf	170.09 x 132.17	9	 International
	Fire_prep.pdf	20.65 x 17.02	24	 International

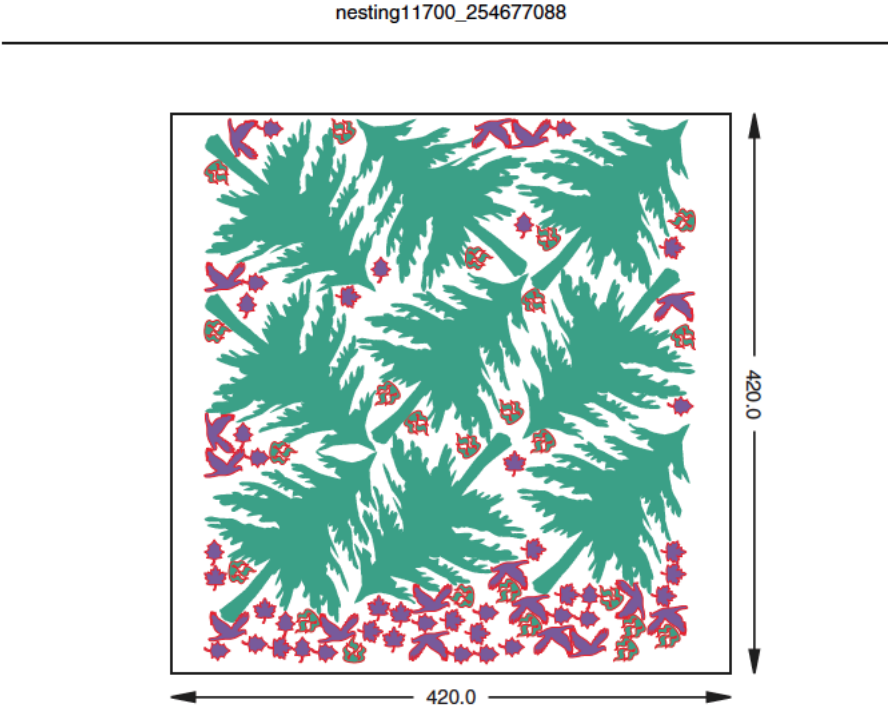
5. If you open the report in an application that can show PDF layers (for example Adobe Reader), you will see that the report has a layer called **Shipping**.

If you hide the layers above this layer (so it only shows the **Base** and **Shipping** layers), you will see that:

- Graphics that need to be shipped nationally are shown in the color of the **National** label (purple in the example below).
- Graphics that need to be shipped internationally are shown in the color of the **International** label (green in the example below, cut paths are shown in red).

Layers

- Preview
- Dummy Colors
- Shipping
- Base



BRBE - A3 # = 1 48.95% 51.05%
 \\homer_standard_testfiles\BRBE\nesting11700_254677088.pdf
 \\homer_standard_testfiles\BRBE\nesting11700_254677088.cut

AutumnLeaf_prep.pdf	18.8 x 16.51	17	National
AutumnLeaf2_prep.pdf	19.15 x 14.67	23	National
Dove_prep.pdf	21.93 x 29.97	17	National
FirTree_prep.pdf	170.09 x 132.17	9	International
Fire_prep.pdf	20.65 x 17.02	24	International

Example 2: Using Custom Fields as Grouping Keys

You need to print different graphics on the same substrate, but some will need to be laminated and some not.

1. In the *Configure tool*, define a **Custom Field** as `Lamination`, and select **Use as grouping key**.
 The corresponding **Custom Field** parameter in this task tab will now be called **Lamination** instead.
2. When launching the task on graphics to be laminated, enter `Yes` in that parameter, otherwise enter `No`.
3. The nesting server will generate layouts with either:
 - only graphics to be laminated,
 - only graphics that shouldn't be laminated.

Note: If you generate a layout report, it will have the label **Yes** if the graphics on that layout need to be laminated, **No** if they don't.

7.4.4 Export Nested Layouts

Use this task to create production files (a PDF print file and a cutting file) from your nested Gang Run Printing layouts.

Typically, the **Export Nested Layouts** task is the main part of your Gang Run Printing **output workflow**.

When working with Gang Run Printing:

1. You submit the graphics to be nested and printed through the *input workflow*.
2. The *nesting server* automatically nests the graphics on the substrate, and sends the resulting nested layouts to the output workflow *linked to the substrate queue*.
3. The *output workflow* generates production files from the nested layouts.

Note:

In a workflow, this task has five output pins (**Print Files**, **Cutting Files**, **Done Graphics**, **Input** and **Error**), so you can send each type of file to the right place more easily.

If you select **Create report**, the task will have an extra output pin for the report.

Read more in:

- [Sheet Tab](#) on page 1116
- [Production Files Tab](#) on page 1120
- [Reports Tab](#) on page 1122

Sheet Tab

1. If desired, you can add **SmartMarks** (a SmartMarks set that you created in i-cut Layout or PackEdge) to your sheet.

For example, you can use a SmartMarks set containing registration and/or cut marks. You can use SmartNames to enter your SmartMarks set's name.

Note:

Before you can do this, you need to:

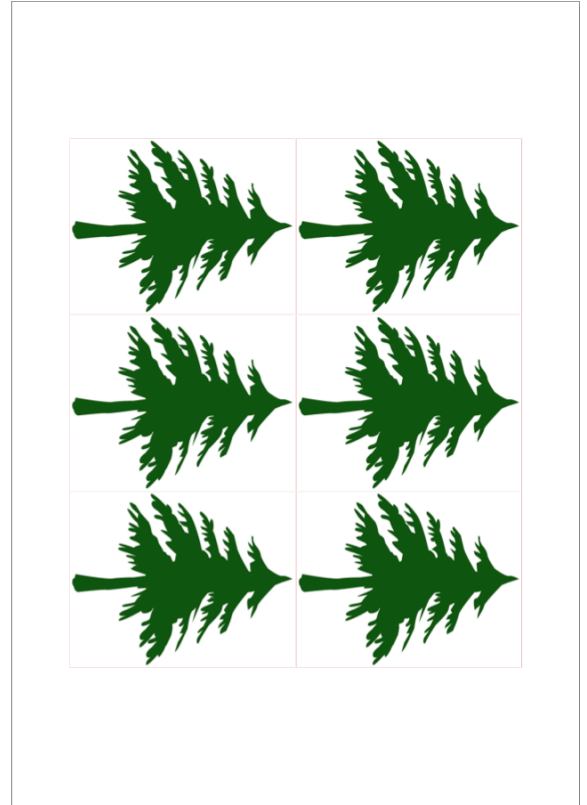
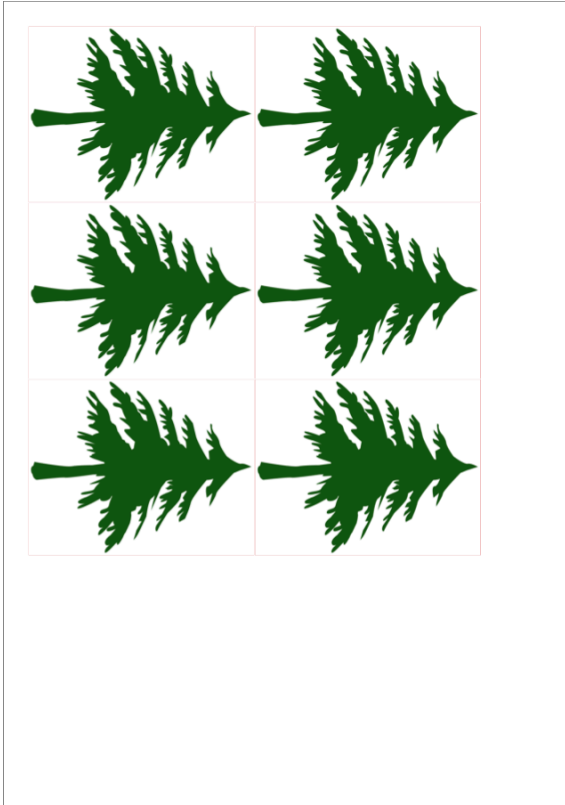
1. Make sure your i-cut Layout or PackEdge application is connected to your Automation Engine server.
2. Create and save a SmartMarks set in i-cut Layout or PackEdge (this automatically saves it to the correct location on your Automation Engine server: \\AEServername\bg_data_marks_v010\dat).

See the i-cut Layout or PackEdge manual for more information about connecting to an Automation Engine server and creating a SmartMarks set.

2. When the *nesting server* generates your nested layout, it places the layout on the sheet based on the settings of the substrate queue you chose (*sheet size and margins* and *nesting settings*).

However, you may want to move the nested layout on the sheet for printing and finishing purposes. In **Center Layout on Sheet** you can choose to center the layout either **Vertically**, **Horizontally**, or **Vertically and Horizontally**.

In the example below, the nested layout has been centered both vertically and horizontally.

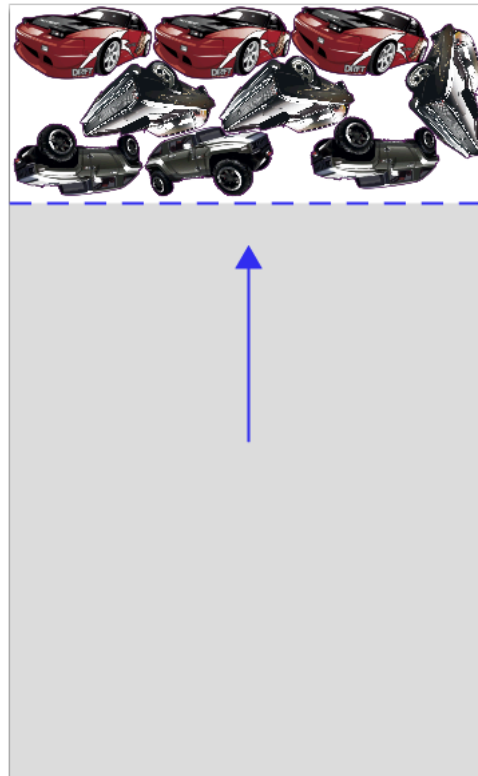


3. If you are working with substrate rolls or large sheets, you can choose to fit the final PDF that will be printed (the "sheet" file containing your nested layout) to one or both of the layout's dimensions.

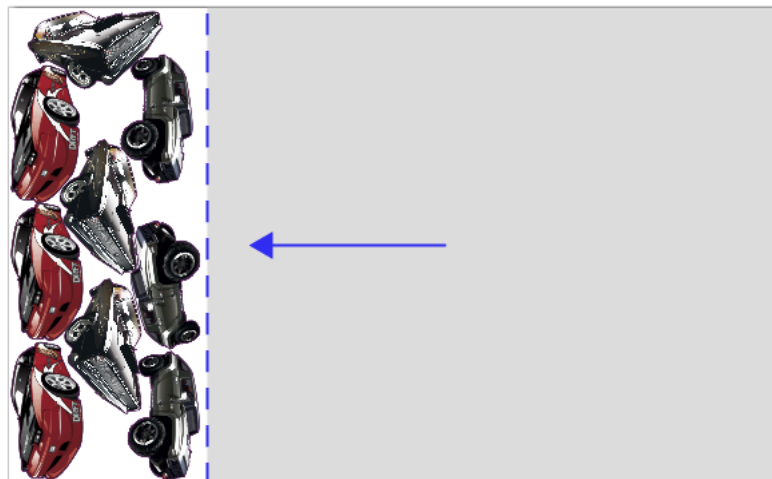
This way, you can cut closer to the layout and have less substrate waste (for rolls) or save the remaining blank area as an offcut so you can reuse it (for large sheets).

For **Fit Sheet**, choose either:

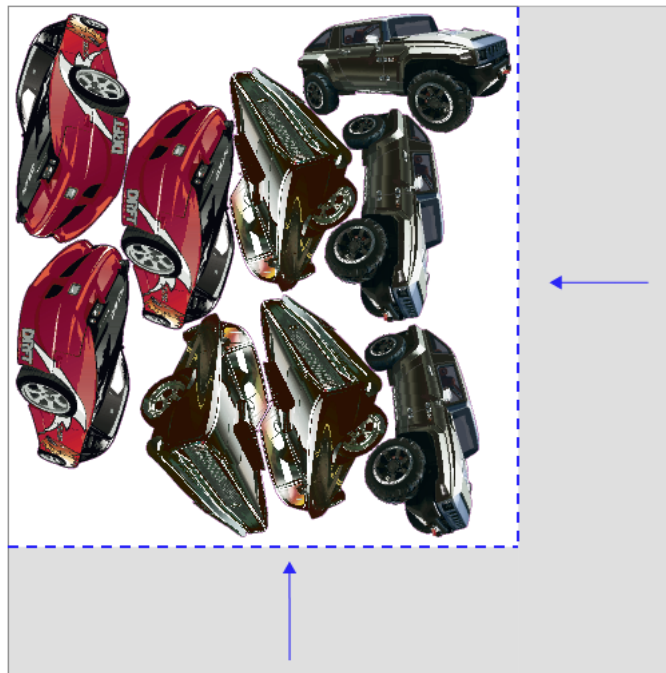
- **To Layout Height** to fit the sheet height to the height of your nested layout.



- **To Layout Width** to fit the sheet width to the width of your nested layout.



- **To Layout Width & Height** to fit the sheet height and width to the height and width of your nested layout.



4. If you are printing Front and Back, define how the sheet will be turned for back printing in **Backing Up**: either **Work-and-Turn** or **Work-and-Tumble**.

This influences how the back graphics will be placed in your final PDF.

Front (at left) and back (at right) with **Work-and-Turn**:



Front and back with **Work-and-Tumble**:



Production Files Tab

The task will output:

- PDF print files,
- cutting files (in your desired format).

Note: You will always have a single page PDF print file per layout, unless you are printing front and back (in this case, the back graphic(s) will be on a second page).

1. In **Save Cutting Files**, choose the cutting file format you want to generate (**i-cut**, **HPGL**, **PDF**, **MFG**, **ACM** or **JDF**).

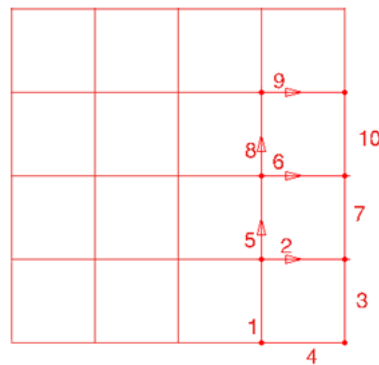
Tip:

- If you have several cutting tables and you don't know yet which one you will use to cut the sheet, you can generate several file formats: choose **Select File Types...**, select all the file types you want to generate and click **OK**.
- If you want to check your cutting files for quality control, you can also output a PDF file and inspect it in Acrobat.

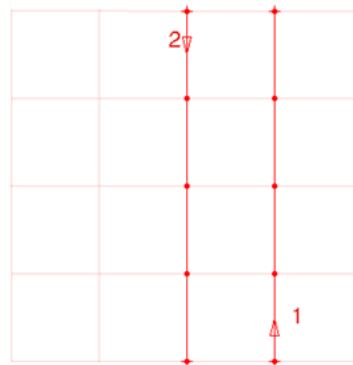
When generating an **ACM** file (for a Kongsberg table), you can select **Print Side Up** to cut with the print side visible.

For all cutting file types except **JDF**, you can influence the knife or cutting head's cutting sequence:

- Leave **Sequence lines** deselected if you want to use the original cut paths for the cutting lines (this way, common lines between graphics are only cut once, but every graphic is still cut individually).



- Select **Sequence lines** if you want the vertical and horizontal cutting lines to be concatenated so that the knife can cut longer lines at once (so the knife doesn't need to be lifted and brought down, and accelerate and decelerate, as often, which speeds up the cutting process).



If you have selected **Sequence lines**, you can choose to **Cut Across Gaps** to can cut longer lines at once even if there is a *gutter* between your graphics.



Attention:

If you are generating an **i-cut** file, make sure you have **i-cut marks** in your file as they are needed to define where to start cutting.

Other SmartMarks in your file will not be used for that purpose when working with an i-cut file.

For more information about i-cut marks, please see the i-cut Layout manual.

2. By default, all inks with the `technical` ink type (including the ink used for the cut path) are considered structural inks, and are included in the cutting file but not in the print file.

If you want to include other inks from your graphic in the cutting file but not in the print file, enter their names in **Add Extra Structural Inks**.

When adding several inks, you should separate the ink names with a comma (for example `Crease,Kiss-cut`). You can also use wildcards (*) and SmartNames.

3. By default, the output files have an automatically generated name containing unique identifiers (for example `nestingXXXX_YYYYYYY.cut` and `nestingXXXX_YYYYYYY.pdf`).

Note: If multiple copies of the same layout need to be printed, the number of copies will be part of the PDF file name.

If you want to give your output files a specific name, enter it in **Output Name**.



Attention:

Make sure you include a unique identifier in the name so that your output files do not overwrite each other! This can happen for example if several layouts are generated.

To prevent this, we recommend you include a SmartName such as **Task ID**.

4. Define where to save the production files and the report(s):
 - If you want to save all output files in the same folder, select **Save all files in the same folder**, and browse to that **Output Folder**.
 - Otherwise, make sure **Save all files in the same folder** isn't selected, and browse to your chosen folder for the **Print Files**, **Cutting Files** and **Report Files**.

Reports Tab

You can create one or more nesting reports if desired.

- If you want XML reports (typically used as feedback for an MIS system):
 - Select **Create report per graphic** if you want a report for each graphic nested (detailing on which layout and substrate it is nested).
 - Select **Create layout XML report** if you want a report detailing the different graphics used in each layout.
- If you want a PDF report detailing the different graphics used in each layout, select **Create layout PDF report**.

This is typically used as instructions for the cutting table operator.

You can select:

- the **Report Size (A4 or Letter)**,
- the **Resolution** used for graphics in the report (in the unit defined in the Pilot Preferences),
- the **Units** used in the report (**mm** or **inch**).

8. RIP'ing with Imaging Engine

8.1 Copyright Notice

© Copyright 2015 Esko Software BVBA, Gent, Belgium

All rights reserved. This material, information and instructions for use contained herein are the property of Esko Software BVBA. The material, information and instructions are provided on an AS IS basis without warranty of any kind. There are no warranties granted or extended by this document. Furthermore Esko Software BVBA does not warrant, guarantee or make any representations regarding the use, or the results of the use of the software or the information contained herein. Esko Software BVBA shall not be liable for any direct, indirect, consequential or incidental damages arising out of the use or inability to use the software or the information contained herein.

The information contained herein is subject to change without notice. Revisions may be issued from time to time to advise of such changes and/or additions.

No part of this document may be reproduced, stored in a data base or retrieval system, or published, in any form or in any way, electronically, mechanically, by print, photoprint, microfilm or any other means without prior written permission from Esko Software BVBA.

This document supersedes all previous dated versions.

PANTONE[®], PantoneLIVE and other Pantone trademarks are the property of Pantone LLC. All other trademarks or registered trademarks are the property of their respective owners. Pantone is a wholly owned subsidiary of X-Rite, Incorporated. © Pantone LLC, 2014. All rights reserved.

This software is based in part on the work of the Independent JPEG Group.

Portions of this software are copyright © 1996-2002 The FreeType Project (www.freetype.org). All rights reserved.

Portions of this software are copyright 2006 Feeling Software, copyright 2005-2006 Autodesk Media Entertainment.

Portions of this software are copyright ©1998-2003 Daniel Veillard. All rights reserved.

Portions of this software are copyright ©1999-2006 The Botan Project. All rights reserved.

Part of the software embedded in this product is gSOAP software. Portions created by gSOAP are Copyright ©2001-2004 Robert A. van Engelen, Genivia inc. All rights reserved.

Portions of this software are copyright ©1998-2008 The OpenSSL Project and ©1995-1998 Eric Young (ey@cryptsoft.com). All rights reserved.

This product includes software developed by the Apache Software Foundation (<http://www.apache.org/>).

Adobe, the Adobe logo, Acrobat, the Acrobat logo, Adobe Creative Suite, Illustrator, InDesign, PDF, Photoshop, PostScript, XMP and the Powered by XMP logo are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Microsoft and the Microsoft logo are registered trademarks of Microsoft Corporation in the United States and other countries.

SolidWorks is a registered trademark of SolidWorks Corporation.

Portions of this software are owned by Spatial Corp. 1986 2003. All Rights Reserved.

JDF and the JDF logo are trademarks of the CIP4 Organisation. Copyright 2001 The International Cooperation for the Integration of Processes in Prepress, Press and Postpress (CIP4). All rights reserved.

The Esko software contains the RSA Data Security, Inc. MD5 Message-Digest Algorithm.

Java and all Java-based trademarks and logos are trademarks or registered trademarks of Sun Microsystems in the U.S. and other countries.

Part of this software uses technology by Best™ Color Technology (EFI). EFI and Bestcolor are registered trademarks of Electronics For Imaging GmbH in the U.S. Patent and Trademark Office.

Contains PowerNest library Copyrighted and Licensed by Alma, 2005 – 2007.

All other product names are trademarks or registered trademarks of their respective owners.

Correspondence regarding this publication should be forwarded to:

Esko Software BVBA

Kortrijksesteenweg 1095

B – 9051 Gent

info.eur@esko.com

8.2 What's New in Imaging Engine 14.1 Prerelease Version X?

New Proof Task

A new **Proof** task was added, for proofing to a physical device. See [Proof](#) on page 1193 for more information.

The **Proof - Unscreened Proofing** task was renamed to **Proof to File - Unscreened**.

8.3 What is Imaging Engine?

An Adobe PDF Print Engine Based RIP

Imaging Engine is Esko's **Raster Image Processor (RIP)**, based on the industry standard Adobe PDF Print Engine.

On top of Adobe PDF Print Engine, Imaging Engine includes Esko's renowned [color management](#), [screening](#), [dot gain compensation](#) and [output formats](#) (PDF, TIFF and LEN).

A RIP Integrated Seamlessly into Automation Engine

Imaging Engine is completely integrated into Automation Engine. This means that the Automation Engine Pilot is your single point of contact for using Imaging Engine (you do not need to launch or use any separate software).

You can run the [Imaging Engine tasks](#) on any PDF or Normalized PDF.

A Powerful Multi-Files Processing RIP

Taking advantage of Adobe PDF Print Engine's capabilities (64-bit, multi-threaded, multi-process architecture), Imaging Engine is powerful and highly scalable.

You can add more RAM and cores to your server hardware to process more files in parallel, and faster.

Note: The amount of parallel processing you can do also depends on your [license](#).

8.4 Using Imaging Engine for Proofing and RIP'ing

Files coming from a graphic designer often need a lot of prepress work before being printed (for example, they contain objects in the wrong color space, vector graphics, etc.). You can do this prepress work in Automation Engine for example.

After that, you can use Imaging Engine to both **proof** and **RIP** your files.

- When using the same solution for both proofing and RIP'ing, your proof will look much more like your final print than if you had used two different solutions. This makes Imaging Engine ideally suited for **contract proofing**.
- After the proof is approved, the RIP is the last essential step to make sure your files are ready to send to your **output device**. Imaging Engine can generate **screened** or **unscreened** files, depending on the device you want to use.

8.5 What Does Imaging Engine Do?

When **proofing** with Imaging Engine, you can:

- perform general file transformations to make sure the output is as expected (select the right page box, rotate/invert/mirror the file as needed, apply a distortion to pre-compensate for substrate distortion...),
- [color manage](#) objects in your file, or the whole file, so they look good on press,
- [rasterize](#) the vector graphics to continuous tone bitmaps,
- output your file to the [file format](#) needed for your proofing workflow or proofing device (TIFF or PDF).

When **RIP'ing** with Imaging Engine, you can:

- perform general file transformations to make sure the output is as expected,
- [rasterize](#) the vector graphics to continuous tone bitmaps,
- [screen](#) those bitmaps at the correct resolution for your output device and which the screening settings most applicable to your printing method,
- add [dot gain compensation](#) to compensate for plate and/or press dot gain,
- output your file to the [file format](#) needed for your imaging device (TIFF, LEN, PDF).

8.5.1 Color Management

Color management is making sure that colors are reproduced accurately at different stages of the printing process.

A computer where graphics files are created in a pre-press editor, a proofer and a press all have a different a color gamut, which means that the extent of the colors they can reproduce is somewhat different.

With color management, your application performs a controlled conversion of the colors between these different devices, so that the colors are as consistent as possible. This consistency is especially important for specific "brand" colors, that have to look the same on different substrates (cardboard, paper, plastic...).

Colors in graphic files such as photographs are generally defined in the **RGB** color space (Red, Green, Blue).

Proofers and presses use **CMYK** (Cyan, Magenta, Yellow and Black -or "Key") for printing, and sometimes additional inks (for example Orange, Green and Blue, to be able to reproduce more colors, or one or two spot inks -generally Pantone colors- used in a particular file).

Files coming from a graphic designer may contain CMYK, spot colors and sometimes images in RGB.

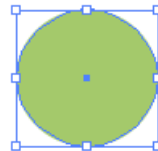
You can use color management to convert your input file's colors into your proofer's color space in Imagine Engine's proofing tasks ([Proof](#) and [Proof to File - Unscreened](#)).

Imagine Engine uses Color Strategies created in the Color Engine Pilot for this purpose (for more information about Color Strategies, see the [Color Engine Pilot documentation](#)).

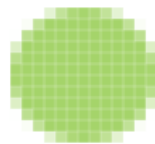
8.5.2 Rasterizing Vector Graphics to Bitmaps

Files coming from a graphic designer may contain **vector graphics** or "linework" (especially for fonts or logos).

Vector graphics are highly scalable without any loss of quality (as they are not based on pixels but on mathematical formulas), but they are not printable.



To be able to print those graphics, you need to **rasterize** them with a RIP (Raster Image Processor). This means converting them to **bitmap images**, or "contones" (continuous tone images), which are based on pixels.



Low-resolution bitmaps can look pixelated, but Imaging Engine generates high-resolution bitmaps.

In Imaging Engine, you can choose to:

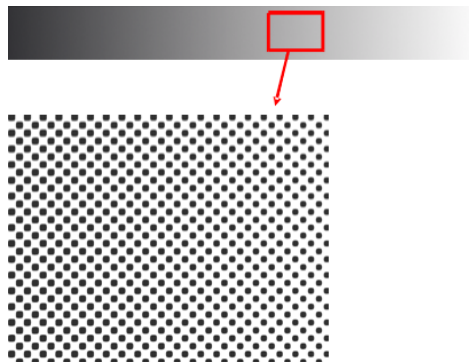
- output your files as contones if you need *unscreened output* (you can do this with the *Image to Unscreened Separations* and the *Proof to File - Unscreened* tasks).
- screen those contones to get "halftones" if you need *screened output* (you can do this with the *Image to Screened Separations* task).

8.5.3 Screening

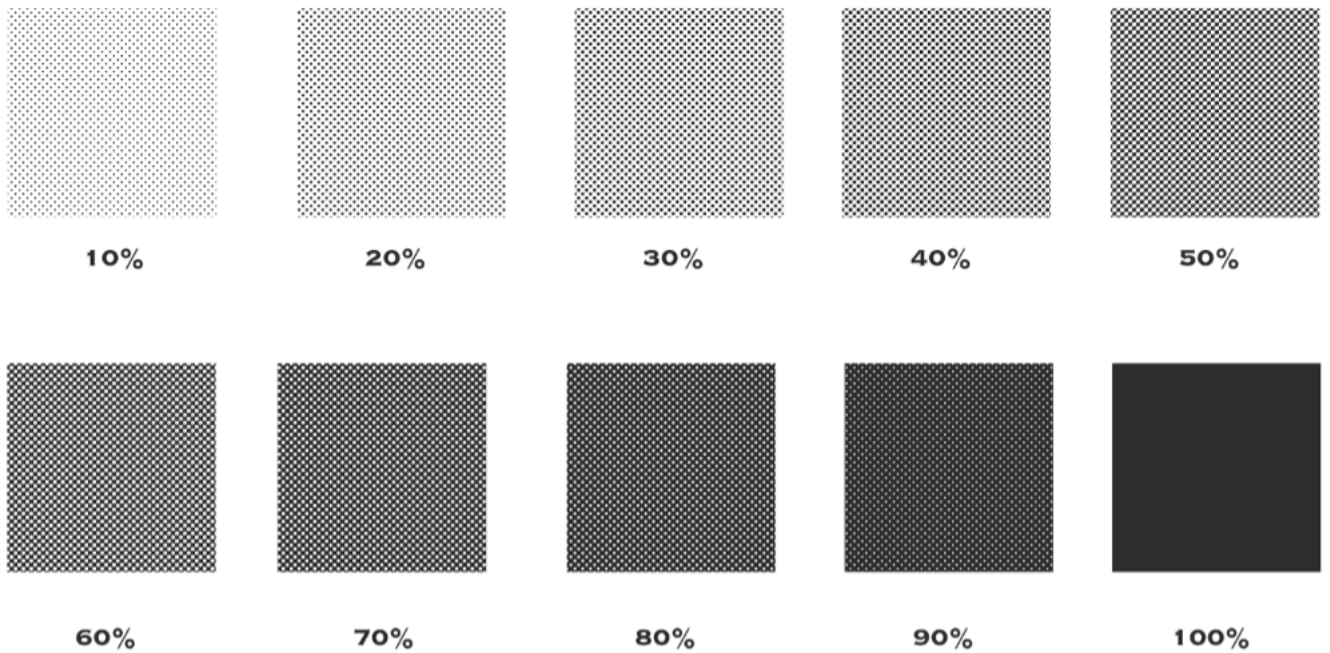
What is Screening?

When printing an image on press, each ink (Cyan, Magenta, Yellow, Black and any additional ink) is laid out separately on the substrate, and the super-imposition gives the final colors.

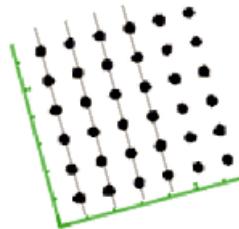
At a high detail level, the press can either print ink or not print ink, so to create differences of color intensity within one ink, you use a small scale pattern of dots of varying size, called a screen.



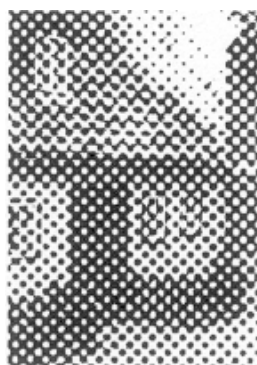
When viewed from a regular distance, this pattern looks like a lighter or darker shade of that color, depending on how big the dots are (what percentage of the area they cover).



The screen can also be coarser or finer, so that you have to be more or less far away to see it as shades of a color. This depends on how many lines of dots can fit in a certain measurement. This is expressed in lines per inch (**lpi**), lines per centimeter (**lpcm**) or lines per millimeter (**lpmm**), and is called the **screen ruling**.



A low screen ruling as below left looks very coarse, and the quality improves as the screen ruling gets higher.



25 Lpi



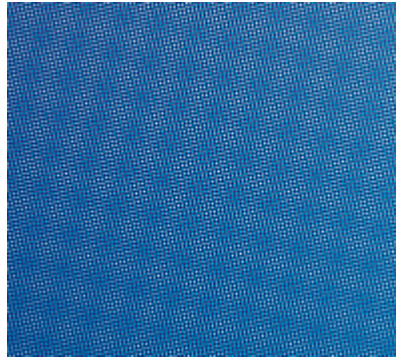
45 Lpi



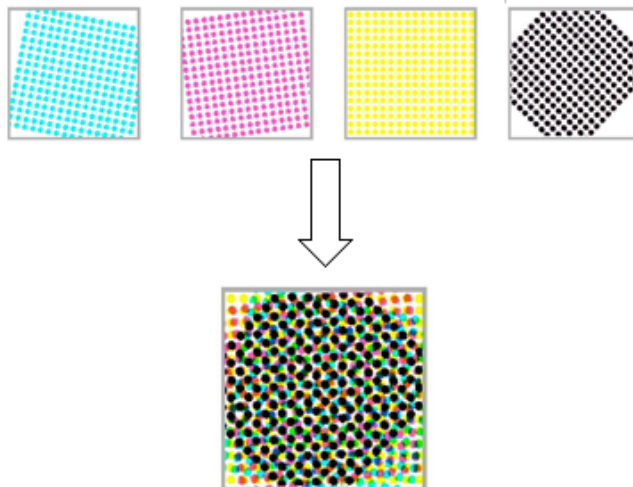
85 Lpi

Screen Angles and Moiré

When placing the screens of different separations on top of each other, a visually disturbing effect called moiré can occur.



To avoid this effect, the screen of each separation is rotated at a specific angle, with a difference of 30° for the most "visible" inks (Cyan, Magenta and Black), and 15° for Yellow.



Note:

The exact screen angles used depend on the printing process:

- Basic offset angles are: Y at 0 degrees, C, M, and K at 15, 45 or 75 degrees depending on the application.
- Basic flexo angles are: Y at 7.5 degrees, C, M and K at 22.5, 52.5 or 82.5 degrees depending on the application (or Y at 82.5 degrees, C, M and K at 7.5, 37.5 or 67.5 degrees depending on the application).

Flexo angles are different from offset angles due to the anilox roll used in flexo presses (a cylinder which is engraved with millions of small cells that carry a thin film of ink which is deposited on the plate).

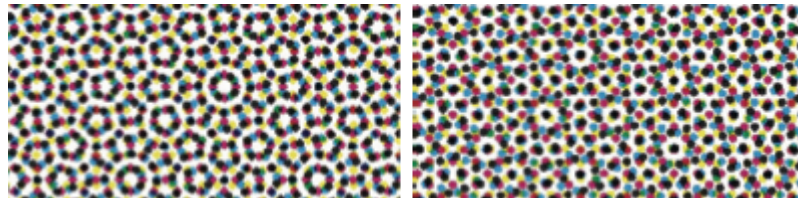
Putting the flexo screens on angles based on a 7.5° offset is needed to prevent moiré between the screen and the pattern of the inking cells of the anilox roll.

Rosettes

When overlaying the screens of the different separations at *their respective screen angles*, you get a rosette pattern. At a normal viewing distance, this is much less visible and distracting than moiré.

A rosette pattern can be clear centered or dot centered.

- Clear centered rosettes (also called hole centered rosettes) (below left) are built up around a hole, and generally considered to be more stable.
- Dot centered rosettes (below right) are built up around a dot, and can give a more saturated result.



Types of Screens Supported

Imaging Engine supports different types of screens:

- *screens with standard dot shapes*,
- *screens with advanced dot shapes*,
- *stochastic (FM) screens*,
- *legacy screens* (screens that you have from an old version of FlexRip or Nexus RIP),
- *custom screens* (screens you have created in Screen Manager or HD Flexo).

Depending on your particular license, some of these screens will be included in your Imaging Engine, and others can be purchased as an option.

Please check the sales material of your product to know what is included.

Screens have a short name and a long name. For example **R** -> **Round Fogra** (**R** being the short name and **Round Fogra** the long name). When selecting a screen, they are listed alphabetically by short name.

Note:

Imaging Engine's halftone phase origin (the point where it starts screening the input file) is the top left corner.

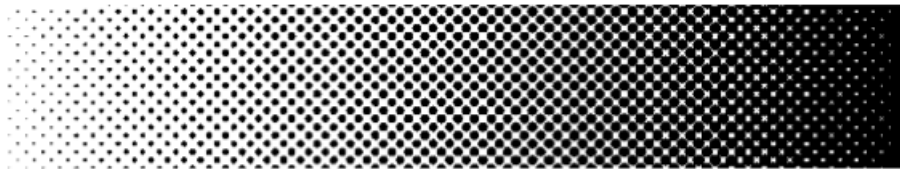
This is the same origin point as FlexRip and Nexus RIP, so you should be able to re-run old jobs and get the same alignment (you may notice 1 pixel difference with old Nexus jobs).

Screens with Standard Dot Shapes

Depending on your license, your Imaging Engine product may include the following screens:

Circular (Euclidean) (short name: C)

This screen uses circular dots, that grow continuously circular until 100%.

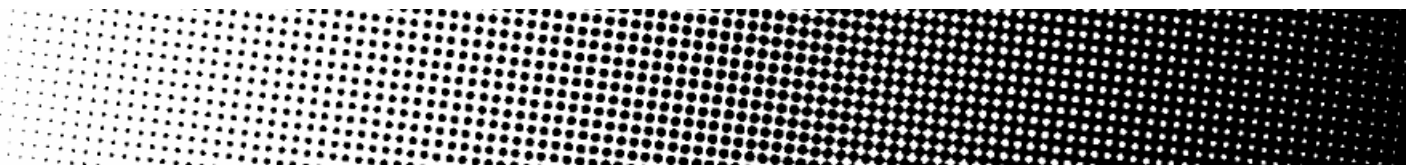


The circular dot is typically used for **flexography** and **letterpress**.

This dot gives good results in many situations. However in some printing conditions the holes in the shadows can fill in, which can result in unstable or high dot gain, and cause a loss in tonal range. In this case consider using the [Round Fogra](#) dot instead.

Double Circular (short name: F)

This screen has circular dots in the highlights and mid-tones, and circular holes in the shadows.

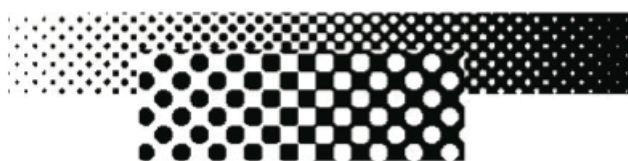


Note that this screen can cause irregular ink bridging ("bridges" between dots) around the mid-tones, which can give unstable dot gain in that tonal range. In this case use the [Round Fogra](#) dot instead.

Paragon Euclidean (short name: PAREUC)

Paragon screens are designed for **offset** and **flexo** environments, to eliminate rosette drift and suppress red and green patterns.

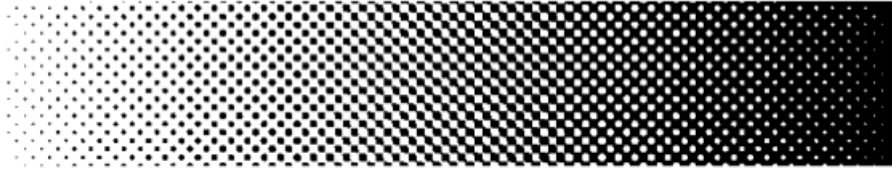
This is a Paragon screen with a round dot in the highlights, a checkerboard pattern in the 50% region, and a negative round dot in the shadows.



Round Fogra (short name: R)

This screen uses the Round Fogra dot, that closely resembles the *elliptical* and *circular* dots, and can be used the same way. It is also a better alternative for square dots.

It goes from a round dot to a round hole, with a square/diamond shape in the mid-tones.



As with elliptical dots, the touching of the dots at the four corners at 50% (leading to a 50% intensity jump) is avoided by using a more elongated dot shape so that the dots first touch around 45% forming a chain and touching for the second time around 55%. This causes less artefacts and less dot gain when printing.

The Round Fogra dot can be used for virtually all printing processes. It is very popular for **offset** printing, but it can also be used for **flexography**, **letterpress**, **silk screen printing** and for making films for **offset-helio conversion**. Only chemical etching gravure cannot be done with this dot shape.

Screens with Advanced Dot Shapes

Depending on your license, your Imaging Engine product may include screens with advanced dot shapes.

Rugby (short name: D)

This screen has a special dot shape and is used for **screen printing**. Rugby dots are shaped like a rugby ball, or a diamond with round edges, which dramatically reduces the interference of the dots with the mesh structure of the silk screen.



The dots touch around 35% to form chains. At 65%, a negative dot with rugby shape appears. This dot shape is fully symmetrical.

Elliptical (short name: E)

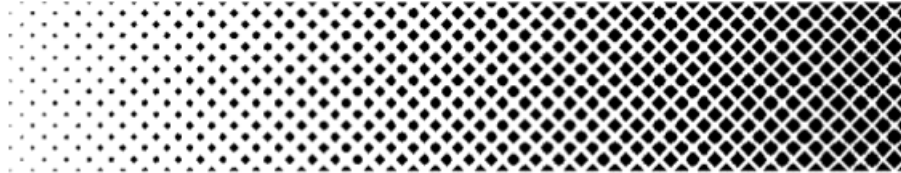
This screen uses a more elliptical version of the *Round Fogra* dots.



For these dots, the first touching point is around 35%. Between 35% and 65%, a chain is formed with the same orientations as for the Round Fogra dots.

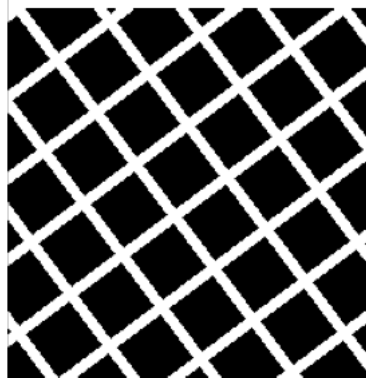
Helio (short name: H)

Helio dots continuously grow square, even above 50%, forming "helio walls".





This dot is needed for the **classic gravure etching** processes only. The dot shape is highly optimized for both contone reproduction and linework areas.

Convex helio dots are the standard helio dots. The dot is as square as possible.



However in some circumstances the etching process smooths out the square shape of the dots, which can cause the helio walls to break during printing (especially at the highest dot percentages), resulting in bad print quality.

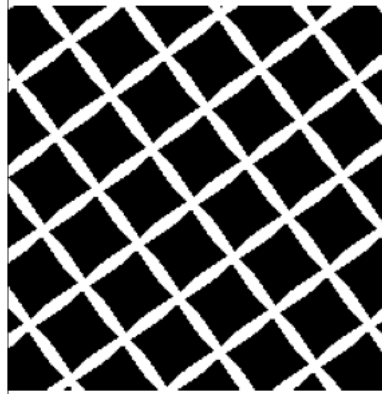
On film	On plate
	

In this case, you should use [concave helio dots](#) instead.



Helio Concave (short name: G)

Concave helio dots are designed to avoid the helio walls breaking that sometimes happen with *standard helio dots*.

In the concave helio screens, the square-shaped screen dots are replaced by more pillow-shaped dots.



When etching the cylinder, the sharp ends are rounded off and the final result is closer to a square than the result of the standard helio dot.

On film	On plate
	

This dot shape will be useable up to slightly higher screen levels than the standard helio dot.

Line (short name: L)

This screen use lines instead of dots.



The advantages of using line screens are:

- there is no tonal jump because the lines never touch,
- the rosette patterns disappear completely, which gives the illusion of a higher ruling.

Paragon Diamond35 (short name: PARDIA35)

Paragon screens are designed for **offset** and **flexo** environments, to eliminate rosette drift and suppress red and green patterns.

Paragon Diamond35 is a screen with diamond shaped dots.



Paragon Elliptical35 (short name: PARELL35)

This is a Paragon screen with *elliptical* dots.



Square (short name: S)

This screen uses square dots in the highlights, forms a checkerboard at 50% and has square holes in the shadows.



At 50%, the tonal jump can be very noticeable, especially when the printing process causes the holes to fill (for example in Flexo), which results in a large dot gain. If the tonal jump is too obvious, use *Round Fogra* or *elliptical* dots instead.

Eccentric 0 (short name: X0)

Eccentric dots are a series of more or less elongated dots, X0 being the most elongated (the furthest from a round shape). A screen with X0 dots is close to forming *lines*.



Because of the more or less elongated shape of the dots, different eccentric dots have different touching points. The dots form a chain between the two touching points.

Eccentric 0 has a first touching point at 5% and a second touching point at 95%.

Note:

Eccentric dots have specific advantages for the **flexo** printing process:

- The flexo process is characterized by a high dot gain. Eccentric dots tend to lower this dot gain, as well as the tonal jumps caused by dot gain.
- The more eccentric the dot is (X0 and X1 dots), the less noticeable rosette patterns it generates. The difference in rosette patterns between eccentric dots and conventional round or circular dots is more striking in flexo than in offset or gravure, due to the lower rulings used in flexo.

Eccentric 1 (short name: X1)

Eccentric 1 dots are slightly less elongated than X0 dots.



Eccentric 1 has a first touching point at 10% and a second touching point at 90%.

Eccentric 2 (short name: X2)

Eccentric 2 dots are less elongated again.



Eccentric 2 has a first touching point at 20% and a second touching point at 80%.

Eccentric 3 (short name: X3)

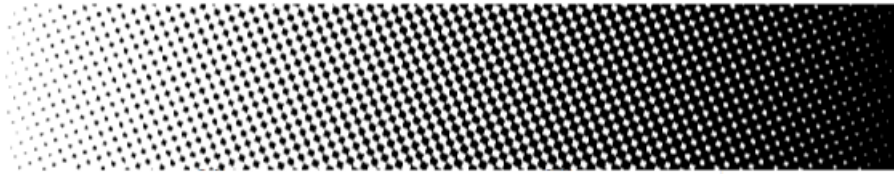
Eccentric 3 dots are only slightly elongated.



Eccentric 3 has a first touching point at 30% and a second touching point at 70%.

Eccentric 4 (short name: X4)

Eccentric 4 dots are the least elongated of the eccentric dots. They are close to *Round Fogra* dots.



Eccentric 4 has a first touching point at 40% and a second touching point at 60%. At 50% the dots form a checkerboard.

Stochastic (FM) Screens

In "stochastic" (or Frequency Modulated) screens, dots are not placed on a regular grid pattern like in conventional (Amplitude Modulated) screens.

In a conventional screen, the dot size grows with the intensity (you have bigger dots at 50% than at 25%). In a stochastic screen, the dot size stays the same, but you get more dots per area as the intensity increases.

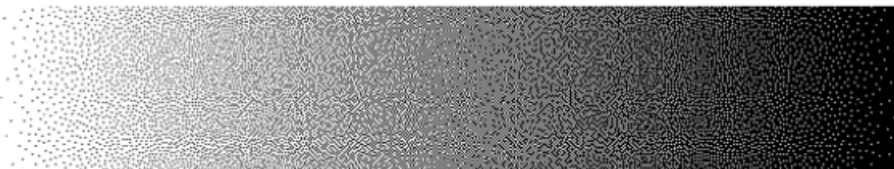
The advantages of stochastic screens are:

- sharper images,
- the full elimination of disturbing rosette patterns,
- no color shift caused by small registration errors (those errors are less visible than the same errors with conventional screens, which can reduce press preparation time),
- the possibility to combine more than 4 separations without *moiré*.

Monet (short name: M)

The Monet screen has dot sizes exactly equal all over the density range, and is optimized for **offset** printing.

In many cases, it can also be used in combination with a *Rugby dot* for **silk screen printing**.

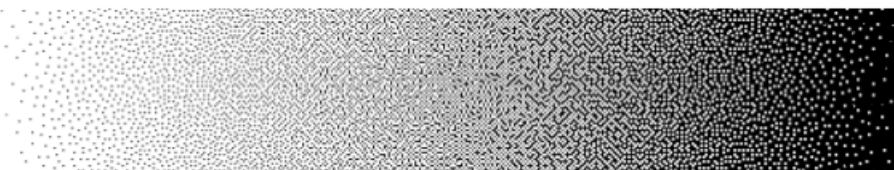


If you experience high dot gain when using a Monet screen, we recommend you switch to an *Organic screen*.

Monet for Flexo (short name: MF)

This screen is based on the Monet screen.

The Monet for Flexo screen has holes in the shadows that are bigger than the dots in the highlights, so they don't fill up as much with dot gain (not increasing the shadows' darkness too much).



This is especially beneficial for printing processes with high dot gain such as **flexo**.

Organic (short name: O)

Unlike first generation stochastic screens like *Monet* or *Monet for Flexo*, where the single dots are left on their own, the organic dots are combined in clusters, forming organic-like random structures. This reduces the dot gain around the mid-tones.

However, Organic screens can generate noise in images, giving a grainy appearance. If you experience this problem, we recommend you switch to a Monet screen.

The Organic screen is optimized for **offset** printing.



Legacy Screens

If you have some screens that you got with an old version of FlexRip or Nexus RIP, they will still be available in Imaging Engine.

Note:

Those are pre-calculated screens that were created for specific printing environments or to solve specific problems. They can only perfectly match certain sets of conditions.

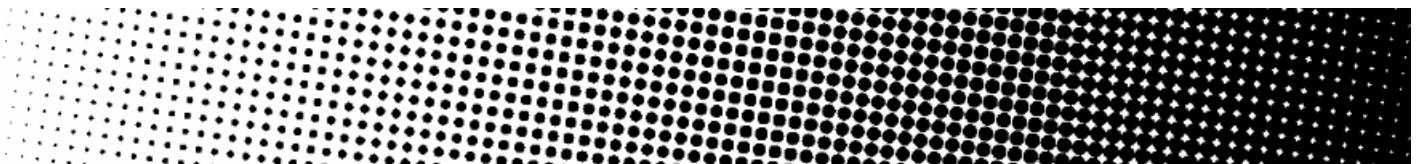
Today, you can get the same effects, fine-tuned for your particular printing conditions, by creating [custom screens](#) in HD Flexo and/or Screen Manager.

See the HD Flexo and/or Screen Manager documentation for more information.

Conventional (AM) Screens

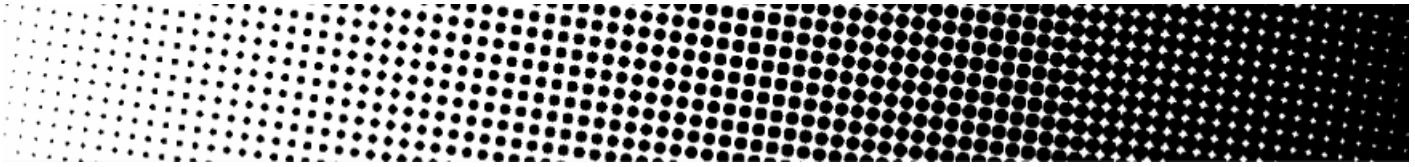
TwinBeam (short name: CD0)

The TwinBeam screen was developed for the CDI. It uses round dots that are exposed by two laser beams.



CDI MultiBeam Optimized (short name: CD1)

The CDI MultiBeam Optimized screen was also developed for the CDI, as an improvement to the TwinBeam screen. It uses round dots that are exposed by several laser beams.

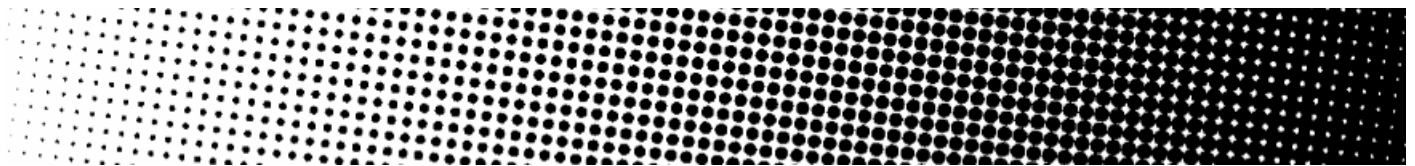


Endless (short name: CO)

This screen was used in combination with cylindrical sleeves on the CDI, to make several copies of a job print continuously on the substrate, without broken dots at the edges of each job.

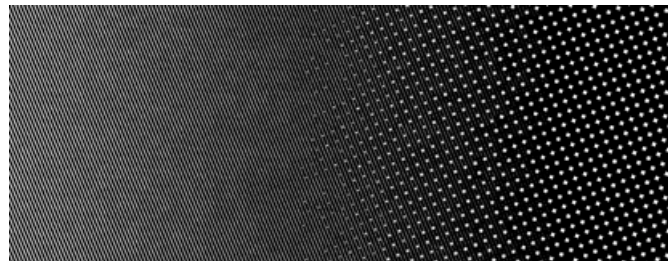
For best results, we recommend you use Screen Manager to create a custom seamless screen instead of using the Endless dot here.

This screen uses the circular dot.



Groovy (short names: GY1 to GY5)

Groovy screens put a line pattern in the dark areas of the screen only. The lighter areas have a [Double Circular](#) dot pattern.



The transition between the normal screen pattern and the line pattern is smooth, leading to a smooth gradation on print between the lower density (non-groovy) and higher density areas (groovy).

They improve solid rendering in **flexo** printing (the lines help even out the ink distribution) and have their best performance on foils. Performance on other substrates (paper, metal) is heavily dependent on the printing circumstances.

You can choose between different groovy screens:

Long name	Short name	Pattern angle	Pattern ruling	Transition start	Maximum pattern percentage	Pattern
Groovy1	GY1	Close to 45 degrees	3 times the original ruling	95%	45%	Lines
Groovy2	GY2	Close to 45 degrees	3 times the original ruling	90%	45%	Lines

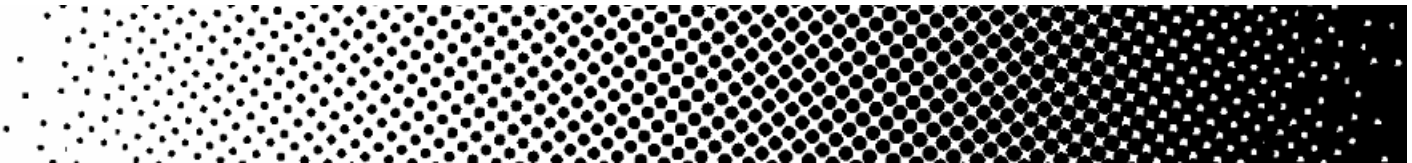
Long name	Short name	Pattern angle	Pattern ruling	Transition start	Maximum pattern percentage	Pattern
Groovy3	GVY3	Close to 45 degrees	4 times the original ruling	95%	45%	Lines
Groovy4	GVY4	Close to 45 degrees	4 times the original ruling	90%	45%	Lines
Groovy5	GVY5	Close to 45 degrees	3 times the original ruling	93%	45%	Eccentric1

Transitional (AM/FM) Screens

Circular Samba (short name: CS)

Samba screens are transitional screens, that were designed to solve problems in the highlights in **flexo**.

They have a *stochastic* part in highlights and shadows, a conventional part in the mid-tones and the transition between both contains a pattern that is gradually going from stochastic to conventional over a large number of intermediate patterns.



SambaFlex (short names: CS4 to CS45)

Like Samba screens (which they are based on), SambaFlex screens are transitional screens, going from stochastic to conventional and back.

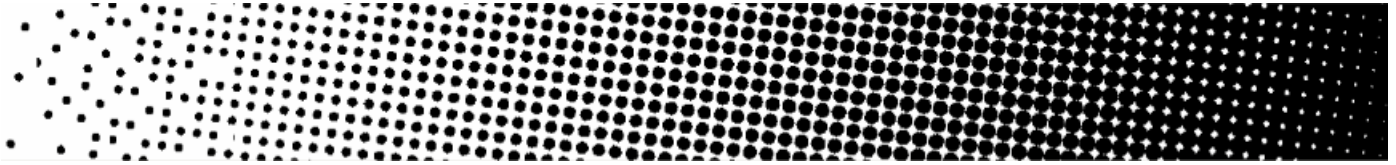


SambaFlex screens support seven different minimum dot sizes, for every imagesetter resolution. These sizes cover a very broad range, from hardly visible dots until dots so big that they can only be used for artistic effects.

- Screens with a small minimum dot size (like **CS4**) have almost unnoticeable stochastic effects.
- Screens with a bigger minimum dot size (like **CS45**) have a coarser stochastic pattern, and less dots in the highlights. The transition point in the highlights shifts up to higher dot percentages.

SambaFlex Highlight (short names: CS4H to CS45H)

These screens are based on the SambaFlex screens, and only have a stochastic part in the highlights. In the shadows, they stop at the darkest point, whether it is 100% or not (for these screens it is usually 95%).



Custom Screens

In Imaging Engine, you can also use any custom screen you created in the **HD Flexo** and **Screen Manager** applications. You will recognize those custom screens because:

- Screens created in **HD Flexo** have a short name beginning with **HD** (**HD01**, **HD02**...).
- Screens created in **Screen Manager** have a short name beginning with **SCR** (**SCR01**, **SCR02**...).

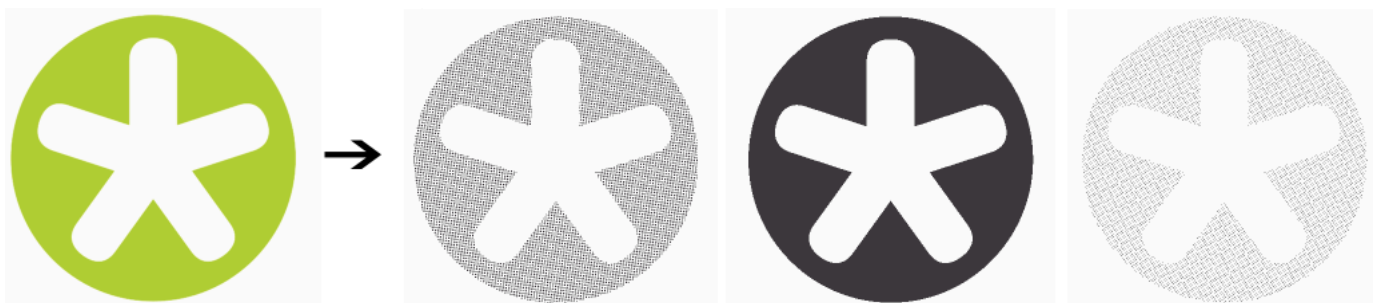
Please see the HD Flexo and/or Screen Manager documentation for more information.

Screened or Unscreened Output?

When RIP'ing your files with Imaging Engine, you can produce either screened or unscreened output.

- You typically need **screened output** to send to a platesetter or filmsetter (for offset or flexo for example).

Run the [Image to Screened Separations task](#) on your input file to produce screened output (a screened file per separation).



- You typically need **unscreened output** to send to a device that will take care of the screening, or a device that doesn't need screened input (for gravure for example).

Run the [Image to Unscreened Separations task](#) on your input file to produce unscreened output (an unscreened file per separation).



8.5.4 Dot Gain Compensation

What is Dot Gain?

When printing a job, the dots tend to print larger on the press than on the plate, which makes the output darker than the original file.

Original job	
Digital film	
Print	





This is due to different factors:

- the type of ink: the more fluid the ink is, the more it will spread under pressure of the press.
- the type of substrate: paper with a rougher and more porous surface (like uncoated paper) reflects less light, making the printed area look darker.
- the type of press:

- offset presses typically print at a 67% density for a 50% dot,
- flexo presses have an even higher dot gain as they put more pressure on the substrate,
- gravure presses have a dot gain similar to offset presses for small dot sizes (up to about 30%) and a sharply increasing dot gain for higher percentages (70% prints as solid).

What is Dot Gain Compensation?

To compensate for dot gain and have the printed output look like the original file, you use **dot gain compensation (DGC)**. This consists in making the dots on the plate smaller, so that with dot gain they print to the correct density.

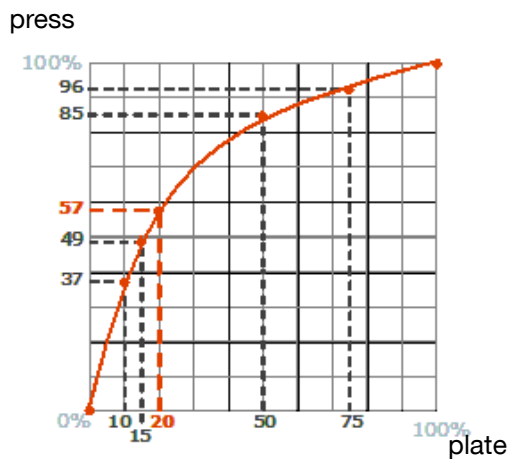
	Dot on plate	Dot on press
Without DGC		
With DGC		

Dot Gain Compensation Curves

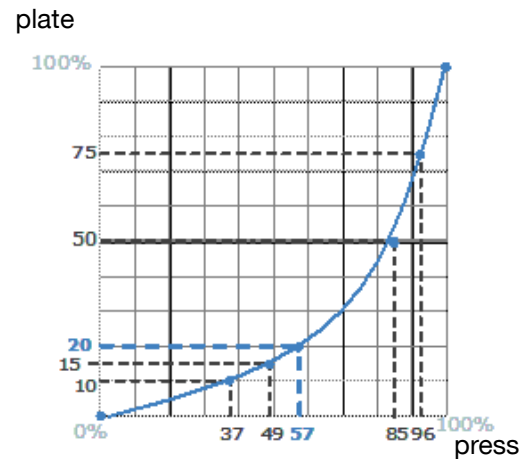
You apply dot gain compensation by using a dot gain compensation curve, which defines which percentage to use on plate for each desired percentage on press.

In the example below, you see how a press prints without any dot gain compensation in the **dot gain curve** at left: a 20% density on the plate prints to 57% on press. 57% on the plate would print to almost 90% on press.

At right, you see the **dot gain compensation curve** used to make the press print to the densities expected in the job: to print a 57% density on press, a 20% density will be used on the plate. To print a 20% density on press, you would need a density of about 5% on the plate.



Dot gain curve



Dot gain compensation curve

Creating a Dot Gain Compensation Curve

You can create dot gain compensation curves in Curve Pilot (see the Curve Pilot documentation for more information).

Dot gain compensation curve files have the `.dgc` extension.

Using a Dot Gain Compensation Curve

You can apply a dot gain compensation curve on all or some separations of your job in the [Image to Screened Separations](#) and [Image to Unscreened Separations](#) tasks in Imaging Engine.

DGC Curve Strategies

Different Dot Gains within a Job

When using a [single dot gain compensation curve](#), the same dot gain compensation is applied to the complete job, without making any compensation difference in the screening parameters, the inks or the type of content (continuous tone images or line work).

However, dot gain is influenced by those factors.

Screening parameters

The following parameters influence dot gain:

- The **dot shape**: for example square, round and elliptic dots have slightly different dot gain characteristics.

Stochastic screens produce more dot gain in the mid-tones than conventional screens.

See [Types of Screens Supported](#) on page 1130 for more information about dot shapes and screens.

- The **screen ruling**: dot gain only happens on the perimeter of the printing dots, which means that higher rulings always result in higher dot gain.

For example, a double ruling means double dot gain.

Inks

Dot gain compensation on a Yellow separation is often less important and less needed than dot gain compensation on a Cyan separation.

The viscosity of the ink also has a big influence.

Continuous tone images or line work

Sometimes, images have been pre-compensated for dot gain in an image editing program, so they need less dot gain compensation than linework.

Dot Gain Compensation Curve Strategies

A **dot gain compensation curve strategy** contains several dot gain compensation curves and defines when to apply which curve.

Creating a Dot Gain Compensation Curve Strategy

You can create dot gain compensation curve strategies in Curve Pilot (see the [Curve Pilot documentation](#) for more information).

Dot gain compensation curve files have the `.icpro` extension.

Using a Dot Gain Compensation Curve Strategy

You can apply a dot gain compensation curve strategy in the [Image to Screened Separations](#) and [Image to Unscreened Separations](#) tasks in Imaging Engine.

Each job separation or object is automatically compensated with the appropriate dot gain compensation curve.

Note: You can also use legacy screen-based DGC files (`.scrdgc`) as dot gain compensation curve strategies in Imaging Engine.

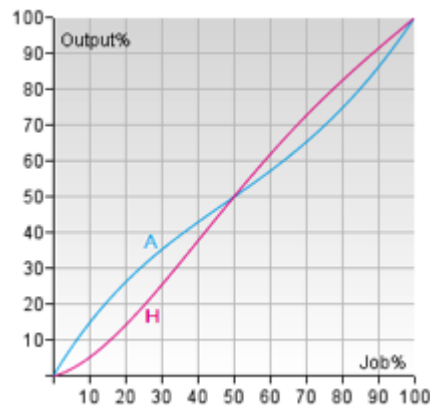
PressSync Curves

PressSync curves are predefined compensation curves that can cover the dot gain compensation needs of most presses, while significantly reducing the complexity of dot gain compensation in workflows.

Slope and Mid-tone Compensation

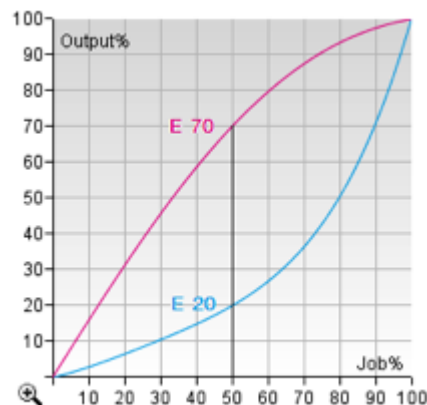
Curves are defined by a letter and a number.

- The letter (A to H) describes how the press behaves in the highlights and shadows (the slope): an A curve makes the press print darker in the highlights and lighter in the shadows (for presses who have the opposite problem), while an H curve, on the contrary, makes the press print lighter in the highlights and darker in the shadows.



An E curve is halfway in between and has a straight slope: it corrects the output the same way throughout the range. Use it for presses whose output is consistent in the highlights, mid-tones and shadows.

- The number indicates how much the 50% dot (mid-tone) prints to: an E20 curve compensates the 50% dot to 20%, while an E70 curve compensates 50% to 70%.



Creating a PressSync Curve

You cannot create PressSync curves, they are predefined. There are 400 different PressSync curves, to match most dot gain compensation needs.

If you wish, you can see each PressSync curve on a graph in PressSync Pilot (see the PressSync Pilot documentation for more information).

Using a PressSync Curve

You can apply a PressSync curve as press curve on all or some separations of your job in the *Image to Screened Separations* and *Image to Unscreened Separations* tasks in Imaging Engine (you just need to enter the letter and number of the PressSync curve).

PressSync Curve Sets

A PressSync curve set contains several PressSync curves, each attached to an ink (a process ink or any spot color defined in Esko's CMS database).

Creating a PressSync Curve Set

You can create PressSync curve sets in PressSync Pilot (see the PressSync Pilot documentation for more information).

PressSync curve sets files have the `.prsync` extension.

Using a PressSync Curve Set

You can apply a PressSync curve set as press curve in the *Image to Screened Separations* and *Image to Unscreened Separations* tasks in Imaging Engine.

Each job separation is automatically compensated with the appropriate PressSync curve.

8.5.5 RIP'ed File Formats

Imaging Engine can output to the following formats:

- **TIFF**, a bitmap file format widely supported by image-processing, publishing, DTP and other applications. You can output to *screened* or *unscreened* TIFF.
- **PDF**, a multifunctional file format, supported by most graphics-related applications. You can output to screened or unscreened PDF.
- **LEN**, a bitmap file format developed for Esko's CDI (*Cyrel Digital Imager*). You can send LEN files to the Grapholas software, which will use them to expose flexo plates on the CDI drum.
LEN files have high resolutions of 2100 ppi and more. They are always screened.

Composite and Separated Files

When **proofing** with Imagine Engine, you can generate either **composite** or **separated** files.

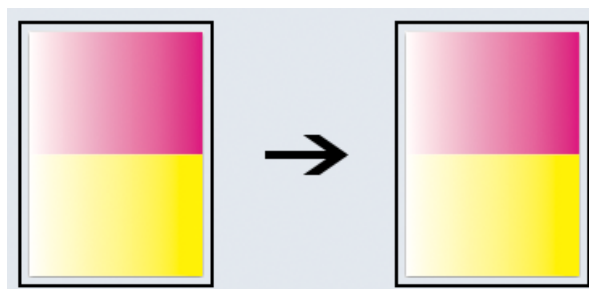
RIP'ing with Imagine Engine always generates **separated** files.

Composite Files

Composite files are files that contain several separations. Imaging Engine can generate composite **PDF** and composite **TIFF** files.

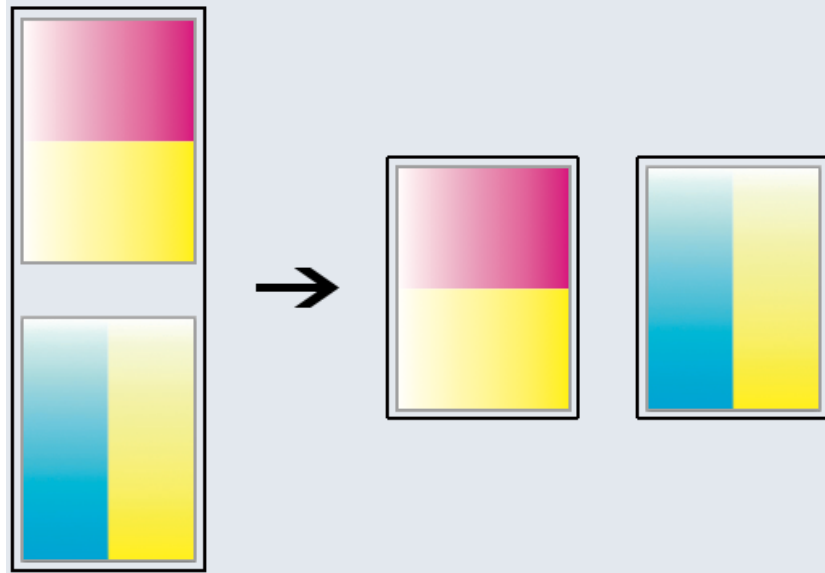
When generating composite output for:

- a single page input file, you get a single page output file containing all of the separations you choose to output (after excluding technical inks for example).



- a multi-page input file, you get one output file per page of the input file, containing all of the separations you choose to output from that input file page.

In the example below, the input file has 2 pages, containing respectively Magenta and Yellow, and Yellow and Cyan. When outputting to composite files, you get 2 one-page files, each containing the separations of the corresponding page in the input file.



Separated Files

Separated files are files that only contain one separation each. Imaging Engine can generate separated **PDF**, **TIFF** and **LEN** files.

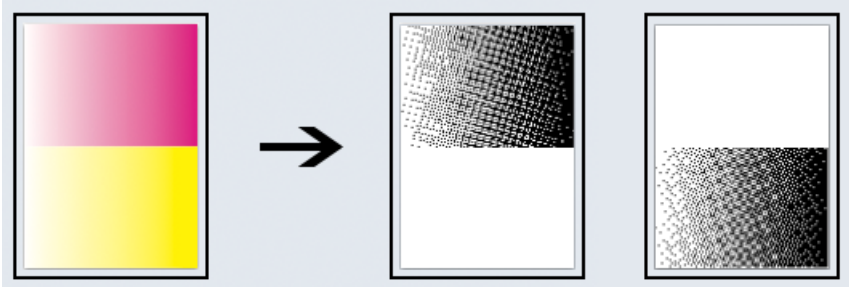
When generating separated output for:

- a single page input file, you get one output file per separation of the input file that you choose to output (after excluding technical inks for example).

In the example below, the input file has one page with a Magenta and a Yellow vignette. After RIP'ing, you get 2 files (one for each separation), each containing a single channel bitmap.



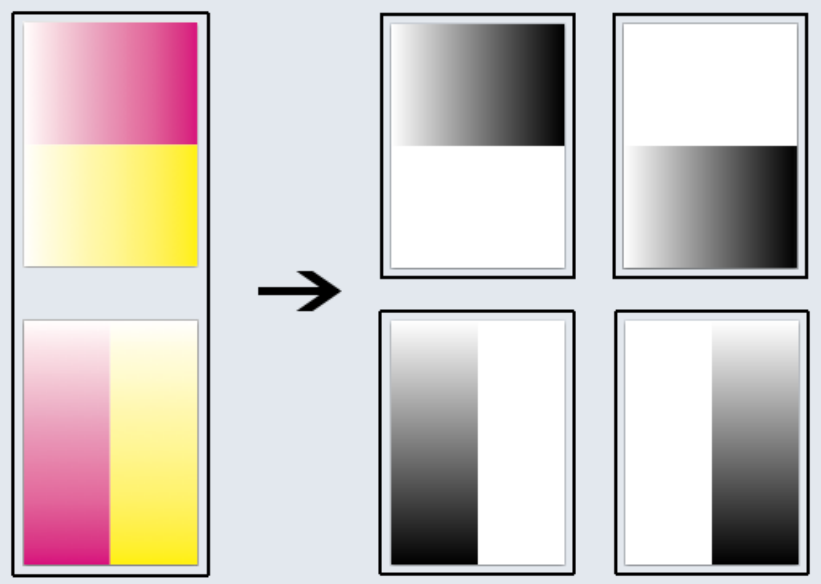
Unscreened separated output



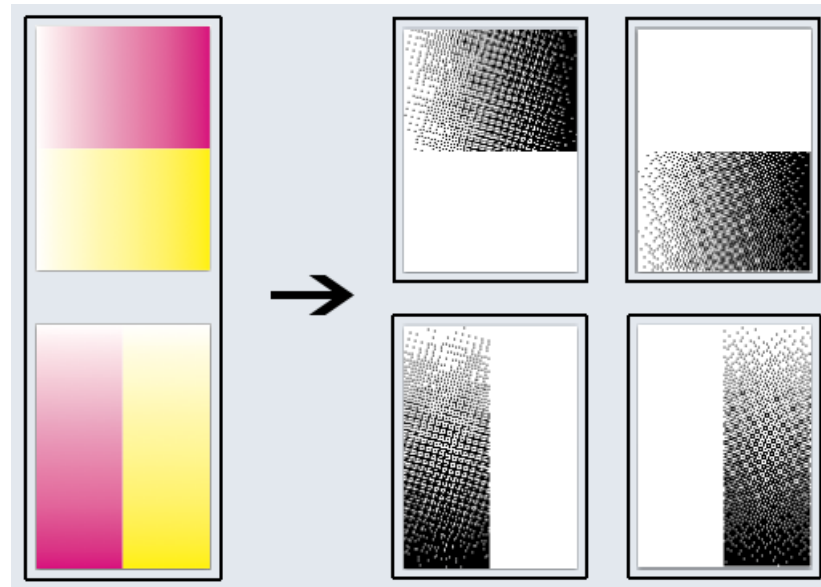
Screened separated output

- a multi-page input file, you get one output file per page of the input file and per separation that you choose to output.

In the example below, the input file has 2 pages, each containing a Magenta and a Yellow vignette. After RIP'ing, you get 4 files (one for the 1st separation of the 1st page, one for the 2nd separation of the 1st page, etc.), each containing a single channel bitmap.



Unscrapped separated output



Screened separated output

8.6 Imaging Engine Installation and Configuration

8.6.1 Licensing

To use the Imaging Engine functionality in Automation Engine, you need an Imaging Engine license (in the form of a product key file).

If you don't have this license, you will not be able to [configure Imaging Engine in the Automation Engine Pilot](#), or see [the Imaging Engine tasks](#).

Depending on the license you have, you will be able to output to some or all of the supported [output formats](#) (TIFF, PDF and LEN), and you may be limited to a certain output file size or not (for LEN files).

For more information about the available licenses and which would fit your production best, please talk to your Esko sales representative.

8.6.2 Installation

Before Installing Imaging Engine 14

Before you install Imaging Engine 14 on your system, check the following:

- You have the Suite 14 Imaging Engine Software Installation DVD.

Note: If you downloaded the software from MySoftware.esko.com, you will need to mount the corresponding .iso files as virtual DVDs, or unpack the .iso files first.

- You need to have Administrator privileges.

- You need Internet access.
- Your computer must meet the minimum requirements for the installation, as described on www.esko.com/en/SystemRequirements/.
- **Data Execution Prevention (DEP)** must not block the installation.
- The **Password Policy** must not block the installation.
- You must make sure your anti-virus software is not blocking the installation or configuration.
- You need a product key file for Imaging Engine 14. If you do not have a product key file yet, contact [Esko Customer Support](#).

How to Change the Data Execution Prevention (DEP) Settings

Data Execution Prevention (DEP) is a security feature that helps prevent damage from viruses and other security threats by monitoring your programs to make sure they use the computer's memory safely. Before installing Imaging Engine 14, make sure that DEP does not block the installation.

1. In **Control Panel**, click **System > Advanced System Settings**.
2. Click the **Advanced** tab.
3. Under **Performance**, click **Settings**.
4. Click the **Data Execution Prevention** tab.
5. Select **Turn on DEP for essential Windows programs and services only**.
6. Click **OK**.
7. Restart the computer.

How to Change the Password Policy

A password policy is a set of rules designed to enhance computer security by encouraging users to employ strong passwords and use them properly. Before installing Imaging Engine 14, make sure that the password policy on your system does not block the installation.

1. In **Control Panel**, click **Administrative Tools**.
2. In the **Administrative Tools** window, double-click **Local Security Policy**.
3. In the **Local Security Policy** window, double-click **Account Policies > Password Policy**.
4. In the right pane, double-click **Password must meet complexity requirements**.
5. Select **Disabled**.
6. Click **OK**.

Licensing and Activation Requirements

Imaging Engine 14 is protected with licenses. These licenses are distributed as a product key file (in .html). To import the licenses into the License Server, the product keys in the product key file must be activated using the **Activate Server Licenses** utility.

You need:

- A product key file for Imaging Engine 14. If you do not have a product key file yet, contact [Esko Customer Support](#).
- Internet access to activate the licenses.
- An **Esko ID** in order to activate the licenses. Get one at <http://www.esko.com/licensing/CreateAccount>.

Installing Imaging Engine 14

You need the Suite 14 Imaging Engine Software Installation DVD for the installation. Use the following procedure to install Imaging Engine 14:

1. Open the Suite 14 Imaging Engine Software Installation window:
 - If you insert the Suite 14 Imaging Engine Software Installation DVD into the DVD drive, or mount the corresponding .iso file, the window appears automatically.
 - If you unpacked a downloaded .iso file, or if the window doesn't open automatically, double-click `setup.exe` in the root folder.
2. Select the installation language.
3. Click **Proceed to the installation**.
4. Under **Imaging Engine 14.0 installation, including related tools (Color Engine Pilot, Curve Pilot, HD Flexo Screens and ScreenManager)**, click **Install**.
5. On the **Imaging Engine 14.0 Installer** page:
 - a) Install the License Server Components and activate the licenses. Read more in [How to Install the License Server Components](#) on page 1152.
 - b) Check and install the prerequisites. Read more in [How to Install the Imaging Engine 14 Prerequisites](#) on page 1153.
 - c) Proceed with the software installation. Read more in [How to Install Imaging Engine 14](#) on page 1154.
 - d) Install the Bitmap Viewer. Read more in [How to Install the Bitmap Viewer](#) on page 1154.

How to Install the License Server Components

1. On the **Imaging Engine 14.0 Installer** page, click on **Install the System Controller 1.0**.
2. Follow the instructions of the installation wizard.
3. When the installation is finished, click **Activate the licenses** on the **Imaging Engine 14.0 Installer** page.
4. In the **Esko Network License Manager**, do one of the following:
 - click in the **Click to activate your Product Key File** area,
 - drag your product key file in the area marked **or drag your Product Key File here**,
 - click the **Activate** button.
5. Follow the instructions of the Activate License Wizard.
6. When asked about the product keys, choose **I have received an HTML file containing my Product Keys**.
7. Click **Browse** to select the product key file.
8. When asked, fill in your **Esko ID** and **Password**.
9. Click **Finish**.

The licensed products you can use are now visible in the License Manager.
10. Close the License Manager to go back to the **Imaging Engine 14.0 Installer** page.

How to Install the Imaging Engine 14 Prerequisites

Before you install Imaging Engine 14, you need to install the Imaging Engine 14 prerequisites.

1. On the **Imaging Engine 14.0 Installer** page, click on **Check and install the Imaging Engine System prerequisites**.

This checks whether the operating system is suited for installing and running Imaging Engine 14, and starts the Imaging Engine 14 prerequisites installation.

2. If:

- you see an error message about the Windows version, your operating system is not suited for running Imaging Engine 14. Read the [System Requirements](#) again and install Imaging Engine 14 on a suitable operating system.
- you see the message `Ready to install the prerequisite components for the Imaging Engine software. Do you want to start the installation?`, click **Yes**.

3. The installation wizard for the prerequisite components will check the licenses, list the prerequisite components to install on the system, request confirmation for installation and then install the required components.

Follow the instructions of the installation wizard.

The installation wizard will install the following components:

- Microsoft Visual Studio Redistributables
- Message Passing Interface service

If you are prompted to reboot your computer, do so and restart this installation procedure.

4. If the installer doesn't find a running Automation Engine 14 server on the local machine, you will see the **Enter Server name** window, prompting you to enter the name of the server running your Automation Engine.

Note:

- You should be able to access this server from your Imaging Engine 14 machine.
- You can still use the local machine as Automation Engine server (if Automation Engine 14 is installed on it): start up Automation Engine, and leave the field in the **Enter Server name** window blank.

The installer then checks if the server you entered has a running Automation Engine 14 server.

If you get the following error:	then:
<pre>ERROR - Imaging Engine requires Automation Engine 14.0.0 The provided master server does not have Automation Engine 14.0.0 running Checked server: [yourservername]</pre>	<p>Automation Engine 14 is not running on the server you entered.</p> <ol style="list-style-type: none"> 1. Check that you entered the server name correctly. 2. Make sure Automation Engine 14 is running on it.

If you get the following error:	then:
	<ol style="list-style-type: none"> Restart the Imaging Engine 14 prerequisites installation.
<pre>ERROR - Imaging Engine requires Automation Engine 14.0.0 The provided master server does not have Automation Engine 14.0.0 installed Checked server: [yourservername] (version [yourserverversion])</pre>	<p>Automation Engine 14 is not installed on the server you entered.</p> <ol style="list-style-type: none"> Check that you entered the server name correctly. Install Automation Engine 14 on it. Restart the Imaging Engine 14 prerequisites installation.

- Click **Exit**.

How to Install Imaging Engine 14

- On the **Imaging Engine 14.0 Installer** page, click **Install the Imaging Engine 14.0 products**.
- Follow the instructions of the installation wizard.
- Read the **End user license agreement** carefully and select **Yes, I accept the license agreement**.
- When asked to select the products to install on this system, select Imaging Engine and the desired related products (Screen Manager, Curve Pilot, Color Engine Pilot...).

The **System Components** group cannot be deselected and will be installed on all systems.

- Select a **Destination Folder** for the chosen products or accept the defaults.
- At the end of the installation, click **Finish**.

How to Install the Bitmap Viewer

- On the installation page, click **Install the Bitmap Viewer 14.0 software**.
- Follow the instructions of the installation wizard.
- Select the **Destination Directory** or accept the default.
- Select the file types which you want to associate with the **Bitmap Viewer** or accept the defaults.
- Select the language to use or leave **Automatic** selected to use the computer's language.
- Click **Install**.
- When all components have been installed, close the installation window and eject the DVD.

8.6.3 Configuration in the Automation Engine Pilot


Note:




- You need to perform this configuration before you can use Imaging Engine.
- If you have already configured Imaging Engine, you can only edit the configuration while it is not processing files (if it is processing files while you try to change the configuration, you will see a warning, and will have to wait until the processing is done).


To connect your Automation Engine server to your Imaging Device:

1. In the Automation Engine Pilot, go to **Tools > Configure** .
2. Select **Imaging Engine Devices** in the left pane.
3. Go to **File > New** or press the **Insert** key (on PC).

This creates an **Imaging1** entry under **Imaging Engine Devices** and shows settings in the right pane.

4. In the right pane's **General** tab:
 - a) In **Host Name**, enter the name of the server running your Imaging Engine and press **Enter** or **Tab** (or click the **Connect** button next to **Imaging Engine**).
 - b) Check that you see a green dot  with the message **Imaging Engine is up and running**.

If you see...	it means that...
a grey dot  with the message Imaging Engine is starting up on host	you need to wait a few minutes before you can use Imaging Engine.
a red dot  with the message No license for Imaging Engine on host	your Imaging Engine <i>license</i> is missing. Please check your licenses.
a red dot  with the message No Imaging Engine available on host	the server you have entered doesn't have Imaging Engine or even Automation Engine installed. Please check if you have entered the server name wrong.

You may see other error messages in some cases, but you will always get some advice as to what you need to do (for example with the  **Missing resource for Imaging Engine on host** message, you will get advised to `Please reinstall the Imaging Engine`).

Note: You can **Stop** and **Start** Imaging Engine from here for troubleshooting purposes, but this is generally not necessary, as Imaging Engine starts automatically, and will restart automatically in case of a crash or after being force quitted.

- c) Set the **Number of Workers**. This is the number of parallel processes used to work on multiple pages of a file, or multiple files at the same time.



By default, the number of workers is set to 1.

Note:

The maximum number of workers you can set depends on your license and on the number of cores on your server.

If your license allows it, we recommend you set the number of workers to the number of cores of your server minus 1, for optimal processing.

5. In the **Settings** tab, select some settings to use in the Imaging Engine tasks when working with the device you are configuring here:

a) Select the **Rotation Direction** to use for all angle values (for example for transformation and screening angles): either **Clockwise**  or **Counterclockwise** .

b) Select what kind of **Rosettes** you want to use in the *Image to Screened Separations task*:

- **Clear Centered** rosettes are generally considered to be more stable. This is the default.
- **Dot Centered** rosettes can give a more saturated result.

You can see which imaging device and which kind of rosettes have been used for each file in that file's log.

6. In the **Queue** tab, enter a name for your Imaging Engine **Queue**.

7. In the left pane, select the **Imaging1** entry, go to **File > Rename** and give it a meaningful name.

You can also select it and press **F2** (on PC).


8. Go to **File > Save** or use **Ctrl+S** (on PC) to save your configuration.

8.7 The Imaging Engine Tasks

Imagine Engine consists of:

- two proofing tasks: *Proof* and *Proof to File - Unscreened*,
- two RIP'ing tasks: *Image to Screened Separations* and *Image to Unscreened Separations*.

Use the Imaging Engine tasks as you would use other Automation Engine tasks. For most of the settings, you can also use public parameters (so an operator can define them through Shuttle) and/or SmartNames.

You can use SmartNames everywhere you see the SmartName icon , and you can use SmartNames in public parameters even for settings without that icon (for example if the setting is a button).

Important: SmartNames values must resolve to a value that is valid for that setting at the moment the task runs. Any invalid SmartName value will make the task end in error.

Tip:

- SmartNames are case insensitive (for example it doesn't matter whether your SmartName resolves to `AngleAndRuling` or `angleandruling`).
- To know what value a SmartName must resolve to for a specific setting:
 1. select the setting value you want (for example **Scale : Fit to page size**),
 2. right-click that value (**Fit to page size**) and select **Copy Parameter Value**,
 3. paste the value in a text editor (in this example it is `FitSize`).

For more information about SmartNames and public parameters, see the *SmartNames* and *Using Public Parameters* sections of the Automation Engine manual.

8.7.1 Image to Screened Separations

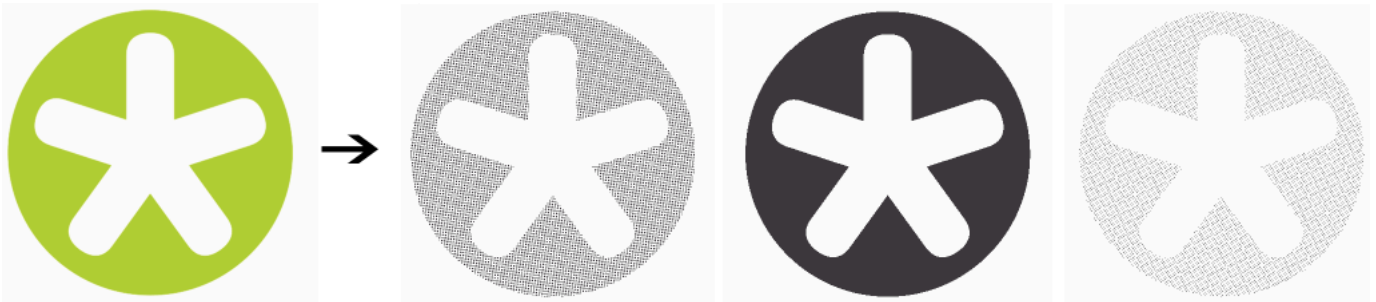
Use this task if you need *screened* output to send to a platesetter or filmsetter (for offset or flexo for example).

Task Output

The **Image to Screened Separations** task generates one file per separation, in either *PDF, TIFF or LEN format*.

For example, you can send screened LEN separations to the CDI.

In the example below, processing a file containing the lime green Esko star generates screened files for the Cyan, Yellow and Black separations.



Task Parameters

In the **Device** field, select the imaging device *that you configured*.

Then use the selector pane at left to show more settings to fill in. See:

- [General](#) on page 1157,
- [Transformations](#) on page 1159,
- [PDF Objects](#) on page 1160,
- [Document Inks](#) on page 1161,
- [Output](#) on page 1161,
- [Separations](#) on page 1168,
- [Summary](#) on page 1175.

General

1. Define what **Page Box** of the input PDF you want to RIP.

You can either select **Media Box**, **Trim Box**, or insert a SmartName `[]` that must resolve to either `media` or `trim`.

After selecting your file's page box you will see its dimensions under the **Page Box** option.

Note: This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If your input file is a multi-page PDF, select which **Pages** you want to RIP:
 - a) First select **All pages**, **Odd pages only** or **Even pages only** in the **Pages** option.

You can also insert a SmartName resolving to either `All`, `Even` or `Odd`.

- b) Then you can refine your selection by entering page numbers or a **Page Range**.

You can separate page numbers by a comma, or use a dash to indicate a page range (for example `3, 6-9`). You can also use a SmartName resolving to page numbers and/or page ranges using this same format.

Note:

- Page numbers are always physical pages. If your document starts with page 5, you should not use 5 but 1 to get the first page.
- If you enter page numbers that are not part of your document, they will be excluded from the output (all the other valid pages will be RIP'ed).
- If you enter the same page(s) more than once (for example 1, 1-2), it/they will only be output once.
- Pages are RIP'ed in the order of the document (entering 5-1 will be automatically changed to 1-5).

3. If you want to invert your output so it is compatible with the output device or software system that will process it, click **A**.

You can also click **I** or **Edit...** to insert a SmartName that must resolve to 1 or `true` (to invert the output), or to 0 or `false` (to leave the output as it is).

4. If you want to mirror your output so it is compatible with the output device or software system that will process it, click **M**.

You can also click **I** or **Edit...** to insert a SmartName that must resolve to 1 or `true` (to mirror the output), or to 0 or `false` (to leave the output as it is).

5. If you want to add a **Smart Mark** to your output file(s), select it in the **Smart Mark** list, or use a SmartName that will resolve to the Smart Mark name.

A **Smart Mark** is a mark whose content is variable, so you can make it display information about that specific job. For example, you can include some of the parameters used during RIP'ing in your output file(s) by using a Smart Mark.

You need to:

- a) First create the Smart Mark you want to use in **PackEdge** (see the **SmartMarks** section in the PackEdge manual for details).

You can create either:

- a Smart Mark with RIP-specific parameters,

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
Resolution	the <i>Resolution</i> of your output file(s)
Screen based DGC	the <i>Curve Strategy</i> you applied to your separation(s)
DGC	the <i>Single Curve</i> you applied to your separation(s)
Dot shape	the <i>screen or dot shape</i> you used on your separation(s)

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
V Distortion	the <i>vertical distortion</i> you applied to your output file(s)
H Distortion	the <i>horizontal distortion</i> you applied to your output file(s)
Device	the <i>imaging device</i> you used
Negative Printing	whether you <i>inverted</i> your output
Mirror Printing	whether you <i>mirrored</i> your output



- another type of Smart Mark (for example a corner mark).
- b) Then select that Smart Mark in your **Imaging Engine** task.

Transformations

Use the **Transformations** settings to rotate or distort your output relative to your input file.

1. If you want to **Rotate** your file's contents, select the rotation you want to apply here. This can be 0, 90, 180 or 270 degrees. Any other value will give an error.

Note:



- The direction of the rotation is the one you selected *when configuring your Imaging Engine* (in the **Settings** tab). You will see  if you selected a clockwise or  if you selected a counterclockwise rotation.
- You can use a SmartName that resolves to 0, 90, 180 or 270 (if you use a SmartName, the task's default rotation direction, *counterclockwise*, will be used, regardless of what you have configured for your device).
- You need to use a numerical value (*ninety* will not work).
- This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If you want to apply **Distortion** to your file's contents:

- a) Select the type of distortion to apply: **No distortion**, **Flexo distortion** or **Values from file**.

You can also use a SmartName that resolves to `Not`, `Flexo` or `File`.

Note: If you selected **Values from file** and the input file contains distortion values, you will see these in the fields below (greyed out).

- b) If you want to apply **Flexo distortion**, enter the vertical  and horizontal  distortion percentages to use.

You can enter numerical values, or insert SmartNames that resolve to numerical values (without the % sign).

Note:

- The values should be within the 90%-110% range. If you enter a percentage outside of this range it will be automatically adjusted to the nearest value in the range (for example 85 will be adjusted to 90%).

However, if you use a SmartName that resolves to a percentage outside the range, the task will end in error.

- You can enter a simple formula and it will be calculated automatically (for example you can enter `100+2` and this will be changed to `102%`). This only works with numerical values and not with SmartNames.

PDF Objects

Use the **PDF Objects** settings to define:

- how to color manage RGB objects in your input file,
- what dot gain compensation to apply to your input file's images.

All parameters in this tab are public parameters, that you can fill in when submitting files through Shuttle.

1. In **Default RGB Profile**, specify the profile to use if no profile is specified for a certain RGB object in the input file. The default is `srgb.icc`.

You can use a SmartName that resolves to the full profile name.

2. Select what output profile you want to use when converting RGB objects to CMYK:
 - If you want to use the output profile from the input file (if there is one available), leave **Use output intent from file** selected (it is selected by default).
 - If you don't want to use the output profile from the input file, deselect **Use output intent from file** and specify the output profile to use in **Output Color Profile**.

This output profile will also be used if there is none available in the input file.

The default is `ISOcoated_v2_300_eci.icc`.

For the **Use output intent from file** option, you can use a SmartName that resolves to `true` (to use the file's profile) or `false` (to not use it).

For **Output Color Profile**, you can use a SmartName that resolves to the full profile name.

3. Select **Use PDF transfer curves** if you want to use the *dot gain compensation curves* attached to the images in your input PDF.

These curves compensate for the unwanted characteristics of the device that created the images (scanner, digital camera...).

Aside from these curves applied on individual images, you can still apply:

- a *plate curve* (to compensate for the dot growth or reduction -for example in Digital Flexo-happening when making the plate)
- and/or a *press curve* (to compensate for the dot gain on press)

... to the full file(s) before output.

You can use a SmartName that resolves to `true` (to use the input file's curves) or `false` (to not use them).

Document Inks

Use the **Document Inks** settings to select which separations of the input file(s) you want to RIP.

For example, you can use these settings to exclude technical and/or varnish inks from your output.

See [Selecting the Separations to Keep in the Output](#) on page 1219 for more information.

Note:

If your file contains PantoneLIVE inks and you don't have a PantoneLIVE license, the **Ink Book** column will show `unregistered`, followed by an ID number (for example `<unregistered "b0c3126e-4f74-4402-8e47-e5e9fe50195a">`) instead of the PantoneLIVE ink book name for those inks.

Output

Use the **Output** settings to define what type of output you want and where to output it.

You can set these through public parameters or using SmartNames.

1. Enter your output **File Name** and output **Folder**.

Note:

- You can use Imaging-specific SmartNames for the **File Name**.

By default, it is set to:

- `[File]_[ShortInk]` for single page input files (which gives output file names like `Myfile_C.tif`),
- `[File]_p[Page]_[ShortInk]` for multi-page input files (which gives output file names like `Myfile_p1_C.tif`).
- The **File Name** and **Folder** can contain special characters (except `\ / : * ? < > " |`).
- The whole path to the output file (including the container, **Folder** and **File Name**) can be up to 255 characters.
- You can set a local or remote URL for the output **Folder**, but it must be an absolute path.
- If your output folder doesn't exist yet, the task will create it if the location you request is valid (you should place this folder on an Automation Engine container accessible by your Automation Engine user).


If the output folder creation fails, please check the task log for the full list of conditions and adapt your output folder path accordingly.

2. Define what to do if a file of the same name already exists in your output folder:

- a) Click **Set Overwrite Policy...**
- b) Choose what to do **If files exist already**. You can either:
 - **Extend the file name** if you want to keep the existing output file and give the new output file a different name. In this case, your new output file name consists of:
 - the **Output File Name** (the **File Name** you set earlier) - which is what the file would be called if no file with the same name existed,
 - a custom suffix that you can define using plain text and/or SmartNames (by default, this is `_c`, for "copy"),
 - an automatic suffix made of the three last digits of the task **ID**,

Tip:

To see the task ID of your tasks:

1. Go to the Pilot's **Tasks** pane (in the **Files** or **Tasks** view).
2. Go to **View > Select Columns in 'Tasks'...**
3. In the **Columns** dialog, select **Task ID** in the **Available columns** list and click  to move it to the **Show these columns:** list.
4. Click **OK**.
5. You now have a **Task ID** column in the **Tasks** pane, listing the task ID of all the tasks.

- the file extension.

For example, you are working with a CMYK input file called `file.pdf` and you have left the tasks' output **File Name** to its default value (the SmartName **[File]**).

The first TIFF files you generate will be called:

- `file_C.tif`,
- `file_M.tif`,
- `file_Y.tif`,
- `file_Y.tif`,

If you generate output from this input file again, with the task 3669, and you have set the overwrite policy to **Extend the file name** with the default **Output File Name + `_c` + ID** name, the new output files will be called:

- `file_C_c669.tif`,
- `file_M_c669.tif`,
- `file_Y_c669.tif`,
- `file_Y_c669.tif`,

Tip:

- If you are working with the Bitmap Viewer, we recommend you keep the default overwrite policy (**Output File Name + _c + ID**), which is compatible with the Bitmap Viewer.
- If you are not, you can define the custom suffix as you wish. For example, you could use date and time SmartNames.

- **Overwrite** the existing file(s).
- **End the task in error.**

Note: If you use a SmartName to set this, it must resolve to `extend` or 0 (for **Extend the file name**), `overwrite` or 1 (for **Overwrite**), or `error` or 2 (for **End the task in error**).

c) Click **Set**.

3. Select the output **File Type** you want. This can be a **TIFF File**, a **PDF File** or a **LEN File** (see [RIP'ed File Formats](#) on page 1147 for more information).


The task generates *Separated* output for all file types. This means you get one output file per page of the input file, and per output separation.

The **File Assembly** is automatically set to **1 File Per Separation**, and the **Output Type** set to **Separate** (for both file types).

Note:

- Every output file that Imaging Engine generates contains XMP information (metadata about file creation, separations, calibration curves, etc.).

You can view this XMP information by selecting your output file in the Automation Engine

Pilot's **Files** view and clicking .

For more information about XMP metadata, see [the XMP Metadata white paper on help.esko.com](http://help.esko.com).

- TIFF files produced by Imaging Engine are fully compatible with Esko's Bitmap Viewer and DotSpy.
- LEN files produced by Imaging Engine are fully compatible with Esko's CDI (Cyrel Digital Imager), just as LEN files produced by FlexRip.

4. Choose the **Compression** to use for your output files.

Note:

For screened output, all available compression methods are lossless (so you don't lose any of the high-resolution detail).

You should choose a compression method based on the compression speed and the compressed file size. Typically, faster compression methods compress the file less, so the output file is bigger.

Note that TIFF files can be a maximum of 4GB, PDF files 12GB, and LEN files are unlimited. Files using stochastic screening are generally larger.

- For **TIFF** output, choose either:

- **PackBits**, a fast lossless compression method for graphic files.

Tip: Use this if you want fast output and file size is not too much of an issue (if you have a fast network for example).

- **LZW**, a lossless compression method for graphic files and other files.

Tip: This compression method works best for files with large images.

- **CCITT Group 4**, a lossless compression method only used for files containing black and white.

Tip:

- This compression method works best for files with a lot of text.
- Use this if having a small output file is more important than processing speed (for example, if you want to store your screened files on a server to rerun jobs later, or if you are working with really large input files that may give screened TIFFs of over 4GB).

- For **PDF** output, choose either:

- **PackBits** (see above).

- **Flate**, a lossless compression method similar to zipping, based on the LZW compression.

Tip: This compression method works best for images with large single color areas or repeating patterns.

- **CCITT Group 4** (see above).

- For **LEN** output, choose either:

- **Flate** (see above).

- no compression (**None**).

Tip: The **Compression** method offered by default for the output **File Type** you select (**LZW** for **TIFF**, **PackBits** for **PDF** and **Flate** for **LEN**) is generally the best for this file type.

**Attention:**

- When using a SmartName to set the output file's **Compression**, it must resolve to one of the following (case insensitive):
 - **PackBits, LZW** or **CCITT-G4** for **TIFF** output,
 - **PackBits, Flate** or **CCITT-G4** for **PDF** output,
 - **Flate** or **None** for **LEN** output.

Setting any other value, or an unsupported compression for the file type you chose, will make the task error (with the message `Unsupported combination of output type and compression` in the log).

- If you have already set the **Compression** but decide to change the output **File Type**, by default the task will either:
 - keep the same **Compression** (if it is supported for the new output **File Type**),
 - or (if it isn't), set it to the first **Compression** available for this **File Type**.

You can still change it as desired.

5. Set the **Resolution** of your output files.

Note:

- The **Resolution** unit is taken from the general Automation Engine Preferences.
- You can enter values from 1 to 25400 ppi (or equivalent in your unit). SmartNames must also resolve to a value in that range.
- Values in another unit than the one from the general Preferences will be recalculated, and values like `300ppi + 500ppm` will be calculated correctly.
- Depending on the **Resolution** you set here, you will have different screen **Rulings** available in the [Separations tab](#).

6. If your input file contains images, the images' pixels may not map exactly to output pixels because:

- the images' resolution is different from the output resolution you chose,
- scaling and rotation effects change the size and placement of the images' pixels.

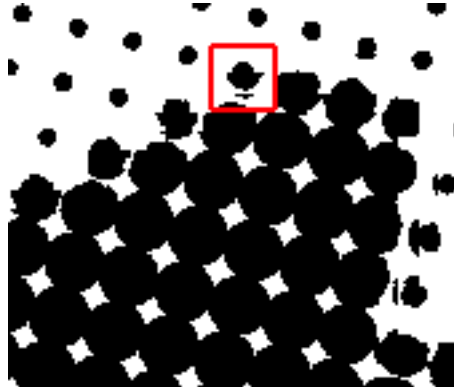
You then need to resample these images to be able to output them at the same resolution as the rest of the file.

In the **Resample Images** option, choose the resampling method you want to use (you will have to make a trade-off between processing time and image quality).

- Choose **From File** to use the resampling method defined in the input PDF (this is the default option). This method can be either:
 - the "nearest neighbor" algorithm, which keeps the image very sharp but can make it look jagged (pixelated but sharp). This is mostly recommended if you have small images with

sharp lines/sudden differences in intensity, and/or you do not want your images to be fuzzy at all.

This resampling method is very fast but may cause broken dots if the original image has sharp jumps in intensity, small and sharp detail, hairlines...



Broken dots can be more susceptible to dot gain than regular dots, they can be washed away while making the plate or can become unstable during printing (for example on flexo plates they can flap around and smear ink on the substrate).

They can have an especially noticeable effect when using advanced screening technologies such as Concentric.

- the "PDF interpolation" algorithm, that smoothes the image while resampling (see below).

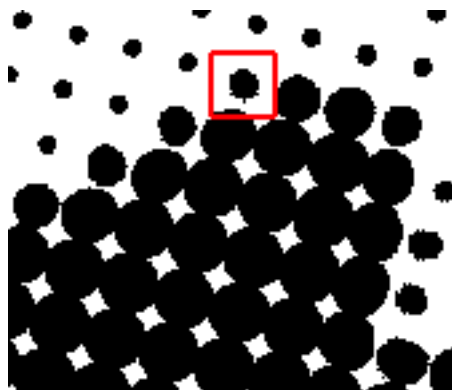
Note: Most PDF files don't specify a preferred resampling method, so the default ("nearest neighbor" algorithm) is used. If you want to make sure that the "PDF interpolation" algorithm is used, choose **PDF Interpolation** instead of **From File** in the **Resample Images** option.

You can also use a SmartName resolving to `FromPDF`.

- Choose **PDF Interpolation** to smooth the image while resampling.

With this method, you get images that are pixellated and fuzzy, so it is only recommended if you have small images with gradual differences in intensity, like photographs of nature.

This is slightly less fast but reduces the chance of having broken dots.

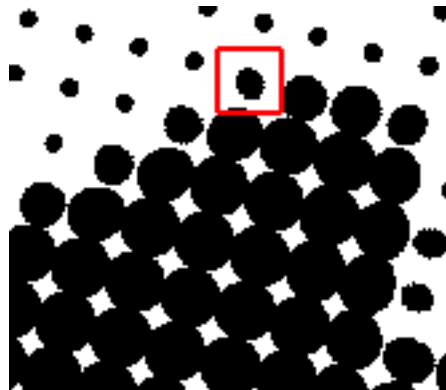


You can also use a SmartName resolving to `PDFInterpolate`.

- Choose **Auto Resampling** to upsample the images to an automatically calculated resolution (generally half or a third of the output resolution) before smoothing it.

When using this, images are a lot less pixellated and fuzzy.

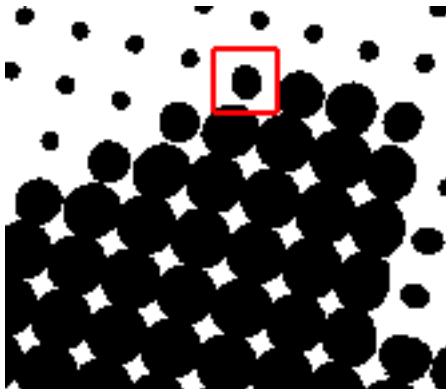
This method is quite slow but gives images of a good quality and reduces the risk of broken dots a lot, so it is recommended in most cases (unless you want an extra sharp image and have no broken dots problem, in which case you should choose **From File**).



You can also use a SmartName resolving to `ResampleAuto`.

- Choose **Full Resampling** to upsample the images to the full output resolution.

This is the slowest method but gives the highest quality images (not pixellated and only very slightly fuzzy), with no risk of broken dots.



Use this if **Auto Resampling** doesn't give you the results you want.



Attention:

Resampling to full resolution can require significant amounts of memory when the output resolution is high (especially from 4000 dpi).

If you want to do this, we recommend that you have at least 8GB of RAM available for each worker (see [Installation](#)).

You can also use a SmartName resolving to `ResampleFull`.

Note:

If your image is not...

- an 8 bit image,
- an image with 1, 3 or 4 components (for example grey, RGB or CMYK),

then the **From File** option is always used.

Separations

Use the **Separations** settings to define the *screening* and *dot gain compensation* to apply to the separations you selected in the *Document Inks settings*.

You can define *default* screening and dot gain compensation settings for most or all of these separations, and *exceptions* to these settings if you want to RIP some of these separations differently.

Defining Default Settings

Define the default screening and dot gain compensation (DGC) settings you want to use on the separations to RIP. You will still be able to define *exceptions* to these settings for certain inks.

Under **Default Separation Settings**:

1. If your input file contains screening settings, you can have the task use (some of) those if desired.

In **Use from file**, select which setting(s) to use:

- select **None** if you want to ignore any screening settings present in the input file and only use settings you define here,
- select **All** if you want to use all the screening settings from the file,
- select **Angle only** if you want to use the screen angles defined in the file, but use the dot shape and ruling defined here,
- select **Ruling only** if you want to use the ruling defined in the file, but use the dot shape defined here (the task then calculates the appropriate angles automatically),
- select **Angle and ruling** if you want to use the angles and ruling defined in the file, but use the dot shape defined here.

Note:

- If you choose to use a certain screening setting from the file but that setting is not present in the file, the task will use the settings defined here as a back-up.
- You can use SmartNames resolving to `none`, `all`, `angle`, `ruling` or `angleandruling`.

2. In **Dot**, select the screen/dot shape to use (or to use as a back-up if you are using the screening settings from the input file).

You can choose from a list of screens installed on your Automation Engine server (on the screens server).

Screens have a short name and a long name. For example **R** -> **Round Fogra** (**R** being the short name and **Round Fogra** the long name).

- Screens with a short name beginning with **HD** are custom screens created in **HD Flexo**. Please see the HD Flexo documentation for more information.

Note: You can only use an HD Flexo screen when generating *LEN files* at a *resolution* of 4000 ppi.

- Screens with a short name beginning with **SCR** are custom screens created in **Screen Manager**. Please see the Screen Manager documentation for more information.
- Screens with a short name beginning with another prefix are predefined screens:
 - *screens with standard dot shapes*,
 - *screens with advanced dot shapes*,
 - *stochastic (FM) screens*,
 - *legacy screens* you have from an old version of FlexRip or Nexus RIP (any legacy screen for which you have a license will have been copied to your Automation Engine server during installation).

Tip:

- When selecting a screen, they are listed alphabetically by short name.
- When using a SmartName to set the screen, it should resolve to either the screen's short name or its long name (not both).

3. Select the **Ruling** to use (or to use as a back-up if you are using the ruling from the input file).

Note:

- The ruling values offered depend on the **Dot** you selected, and on the **Resolution** you entered in the *Output settings*.
- If you have used SmartNames for the **Dot** and/or the **Resolution**, you will not see **Ruling** values to choose from, and will need to enter a ruling manually (which must be a valid ruling for the dot shape and output resolution your SmartNames resolve to).
- When using a SmartName for the **Ruling**, make sure it resolves to the ruling value in lpi but without the unit (for example it should resolve to 150 for 150 lpi).
- This ruling is applied to all separations, but you can change the yellow ruling to minimize yellow moiré (see below).

The task will automatically calculate the correct angles to use based on the **Resolution**, **Dot** shape and **Ruling** you set.

If you are using SmartNames, make sure they resolve to a correct **Resolution**, **Dot** shape and **Ruling** combination. If your combination is incorrect and no valid set of angles can be found, the task will end in error.

4. Click **Advanced Screening Settings** to set options to minimize yellow moiré.
 - a) In **Change Y Ruling**, select **Yes (conventional)** if you want to minimize yellow moiré by changing the ruling (when working with AM Screening).

When using a SmartName to set this, it must resolve to `no` if you don't want to use yellow moiré reduction, or `conventional` if you do.

- b) If you are changing the ruling, choose **by** how much: **-15**, **-7**, **+7** or **+15**.

When using a SmartName to set this, it must resolve to `-15`, `-7`, `+7` or `+15`.

- c) Click **Set**.

Note:

- If your file doesn't contain a yellow separation, yellow moiré reduction won't be applied. The task will run as normal but you will get a warning in the log.
- You cannot set yellow moiré reduction options when using a Paragon dot, as Paragon dots already have built-in yellow moiré reduction.

After the screening settings, you now need to define dot gain compensation settings.

5. Define the **Plate Curve** you want to use on the separations to RIP.

This curve is used to compensate for the dot growth or dot reduction (for example in Digital Flexo) happening when making the plate. Choose either:

- **None** if you don't want to apply any plate curve.
- **Single Curve** if you want to apply one *curve* for all separations (you can still define exceptions for some of the separations later).

In this case, select the `.dgc` curve to apply at right.

You can choose from a list of curves available in the resource server defined for your Automation Engine server.

- **Curve Strategy** if you want to apply a *curve strategy* (a file created in Curve Pilot that contains several curves to apply in different cases: it can have different curves per separation, per dot shape, per ruling...).

In this case, select the `.icpro` curve strategy or legacy `.scrdgc` screen-based DGC file to apply at right.

You can choose from a list of curve strategies available in the resource server defined for your Automation Engine server.

Note:

- You can use SmartNames resolving to the exact name of the **Single Curve** or **Curve Strategy** (including the extension).
- This will set the **Plate Curve** type to `Unknown` in the ticket, but will resolve to the correct curve type if the SmartNames are correct.

6. Define the **Press Curve** you want to use on the separations to RIP.

This curve is used to compensate for the dot gain on press. Choose either:

- **None** if you don't want to apply any press curve,
- **Single Curve** if you want to apply one [curve](#) for all separations (you can still define exceptions for some of the separations later).

In this case, select the `.dgc` curve to apply at right.

You can choose from a list of curves available in the resource server defined for your Automation Engine server.

- **Curve Strategy** if you want to apply a [curve strategy](#) (a file created in Curve Pilot that contains several curves to apply in different cases: it can have different curves per separation, per dot shape, per ruling...).

In this case, select the curve strategy to apply at right (you can select a `.icpro` curve strategy, a `.prsync` [PressSync curve set](#) or a legacy `.scrDGC` screen-based DGC file).

You can choose from a list of curve strategies available in the resource server defined for your Automation Engine server.

- **PressSync Curve** if you want to apply a [PressSync curve](#) (a predefined curve applicable to your printing environment).

In this case, select the letter and number of the PressSync curve.

Note:

- You can use SmartNames resolving to the exact name of the **Single Curve**, **Curve Strategy** (including the extension) or **PressSync Curve**.
- This will set the **Press Curve** type to `Unknown` in the ticket, but will resolve to the correct curve type if the SmartNames are correct.

Defining Exception Settings

If you don't want to use the [default settings](#) you defined for certain inks, you can define specific screening and dot gain compensation settings for those here.

1. Click **Define Exceptions...**

2. In the **Define Exceptions** dialog, select to **Define Exceptions** either:

- **by using exception criteria** (see [Defining Exceptions Using Exception Criteria](#) on page 1171),
- **Manually** (see [Defining Exceptions Manually](#) on page 1173).

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

Defining Exceptions Using Exception Criteria

If you want to define exceptions to the default screening and/or dot gain compensation settings for certain inks, you can do it using exception criteria.

This works as follows:

- You create a rule per separation or set of separations that need specific screening and/or dot gain compensation settings.
- Within each rule:
 - You define criteria to select the separation(s) you want to apply specific settings to.


Each criterion will check something about the inks in your file.

This check can be about the **Ink name**, **Ink book**, **Ink Index**, **Ink type** or **Printing method**. For example, you can check if the **Ink book** is *designer*.

Tip:

The **Ink index** is the order in which the ink will be printed (the ink with ink index 1 is at the bottom of the document and will be printed first).

To find the index of a file's inks:

1. select the file in the Automation Engine Pilot's **Files** view,
2. click ,
3. in the **Info** dialog that opens, select **Inks** in the left panel.

The order of the inks at right is the ink index (the topmost ink has an index of 1, the second of 2, etc.).

Note that if you reorder the inks by one of the columns (for example by **Ink name**), they won't be ordered by ink index anymore. You will need to close that dialog and reopen it to see the ink index order again.

- You select these settings.
- If you have several rules, you can reorder them in order of priority (the top one is applied first).


Note: You can do this either when running the task on one or more files, or when creating a task ticket to use later.

If you *have chosen* to **Define Exceptions by using exception criteria**, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, enter a **Rule Name**.
3. Under **If separation matches**, define one or more criteria to select your desired separation(s).
 - a) Define your criteria the following way:
 - **Ink name:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink book:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink index:** you can define if it **equals/is less than/is greater than** the number you enter, or **is between** the two numbers you enter (in this case the ink indexes you enter as boundaries are also selected).
 - **Ink type:** you can define if it **is** a piece of text (alphanumerical characters).
 - **Printing method:** you can define if it **is** a piece of text (alphanumerical characters).

See [the Printing methods section](#) of the Automation Engine documentation for more information about printing methods.

Tip:

- You can enter several values to check for: just use a semicolon (;) between them.
For example, to check for both inks called `Cyan` and inks called `Magenta`, choose **Ink name is** and enter `Cyan;Magenta`.
- When launching the task on a file, you can also click  to select one or more of the **Ink names/Ink books/Ink types/Printing methods** used in your input file(s).
A criterion will automatically be created based on your selection.

- b) Click **+** to define more criteria.

Note: You can create as many criteria as you want, they will be applied from top to bottom (the top one will be applied first).

4. Define the settings (**Dot**, **Ruling**, **Angle** and/or **Press Curve**) you want to use for the selected separation(s).

5. Click **Add**.

You can see your rule and the settings you selected for the exception separation(s) in the **Define Exceptions** dialog.

Tip: You can double-click your rule if you still want to edit it.

6. Repeat steps 1 to 5 to define more exception rules if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog so you'll be able to see your exception separation(s) with its/their settings in the main task panel.

7. Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Defining Exceptions Manually

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

If you *have chosen* to define exceptions manually, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, select the **Separation** you want to make an exception for.
3. Define the settings you want to use for this separation (**Dot**, **Ruling**, **Angle** and/or **Press Curve**).
4. Click **Add**.

You can see your exception separation with its settings in the **Define Exceptions** dialog.

Tip: You can double-click your exception if you still want to edit it.

5. Repeat steps 1 to 4 to define more exceptions if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog selected so you'll be able to see the settings for that separation in the main task panel.

- Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Switching How You Define Exceptions

You can switch between the two methods of defining exceptions, and this will give the same results.

Switching from Defining Exceptions Manually to Defining Exceptions Using Exception Criteria

If you defined exceptions manually but then choose to **Define Exceptions by using exception criteria**, the task will translate your manual exceptions into rules, so that the same inks are selected, and they have the same exception settings.

For example, if you manually selected **Cyan**, **Magenta** and **Black** and assigned to each of these separations the **Press Curve** `CMK.dgc`, you will get the following rules:

Rule Name: Cyan (process)

Ink name is Cyan

Press Curve: `CMK.dgc`

Rule Name: Magenta (process)

Ink name is Magenta

Press Curve: `CMK.dgc`

Rule Name: Black (process)

Ink name is Black

Press Curve: `CMK.dgc`

Tip: The rule names are created automatically from the contents of the exceptions, but you can still double-click each rule to edit them.

Note:

If you...

- first defined rules with exception criteria,
- then switched to defining manual exceptions but didn't make any changes,
- then switched back to exception criteria,

... then your rules will be kept.

If you did change the exceptions in manual mode, the task will automatically generate new rules.

Switching from Defining Exceptions Using Exception Criteria to Defining Exceptions Manually

If you defined rules with exception criteria but then choose to **Define Exceptions Manually**, each separation selected by a rule will be selected individually, and have the settings you defined in the rule.

For example if you had as criteria:

Ink name is Cyan;Magenta;Black

Ink book is process

and you selected as **Press Curve** CMK.dgc, you will get the following manual exceptions:

Ink Name	Ink Book	Press Curve
Cyan	process	CMK.dgc
Magenta	process	CMK.dgc
Black	process	CMK.dgc

Note: You can only switch to selecting inks manually when you are launching the task on a (single) file.

Summary

This shows a summary of all the settings you defined in the ticket.

- To show the settings from a tab, click + beside the tab name in the right pane.
Click **Expand All** to show the settings from all tabs.
- To hide the settings from a tab, click - beside the tab name in the right pane.
Click **Collapse All** to hide the settings from all tabs.

8.7.2 Image to Unscreened Separations

Use this task if you need unscreened output to send to a device that will take care of the screening, or a device that doesn't need screened input (for gravure for example).

Task Output

The **Image to Unscreened Separations** task generates one file per separation, in either *PDF or TIFF format*.

In the example below, processing a file containing the lime green Esko star generates unscreened files for the Cyan, Yellow and Black separations.



Task Parameters

In the **Device** field, select the imaging device *that you configured*.

Then use the selector pane at left to show more settings to fill in. See:

- [General](#) on page 1176,
- [Transformations](#) on page 1177,
- [PDF Objects](#) on page 1178,
- [Document Inks](#) on page 1179,
- [Output](#) on page 1179,
- [Separations](#) on page 1187,
- [Summary](#) on page 1193.

General

1. Define what **Page Box** of the input PDF you want to RIP.

You can either select **Media Box**, **Trim Box**, or insert a SmartName `[]` that must resolve to either `media` or `trim`.

After selecting your file's page box you will see its dimensions under the **Page Box** option.

Note: This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If your input file is a multi-page PDF, select which **Pages** you want to RIP:
 - a) First select **All pages**, **Odd pages only** or **Even pages only** in the **Pages** option.

You can also insert a SmartName resolving to either `All`, `Even` or `Odd`.

- b) Then you can refine your selection by entering page numbers or a **Page Range**.

You can separate page numbers by a comma, or use a dash to indicate a page range (for example `3, 6-9`). You can also use a SmartName resolving to page numbers and/or page ranges using this same format.

Note:

- Page numbers are always physical pages. If your document starts with page 5, you should not use 5 but 1 to get the first page.
- If you enter page numbers that are not part of your document, they will be excluded from the output (all the other valid pages will be RIP'ed).
- If you enter the same page(s) more than once (for example `1, 1-2`), it/they will only be output once.
- Pages are RIP'ed in the order of the document (entering `5-1` will be automatically changed to `1-5`).

3. If you want to invert your output so it is compatible with the output device or software system that will process it, click **A**.

You can also click `[]` or **Edit...** to insert a SmartName that must resolve to `1` or `true` (to invert the output), or to `0` or `false` (to leave the output as it is).

4. If you want to mirror your output so it is compatible with the output device or software system that will process it, click **M**.

You can also click **[]** or **Edit...** to insert a SmartName that must resolve to 1 or `true` (to mirror the output), or to 0 or `false` (to leave the output as it is).

5. If you want to add a **Smart Mark** to your output file(s), select it in the **Smart Mark** list, or use a SmartName that will resolve to the Smart Mark name.

A **Smart Mark** is a mark whose content is variable, so you can make it display information about that specific job. For example, you can include some of the parameters used during RIP'ing in your output file(s) by using a Smart Mark.

You need to:

- a) First create the Smart Mark you want to use in **PackEdge** (see the **SmartMarks** section in the PackEdge manual for details).

You can create either:

- a Smart Mark with RIP-specific parameters,

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
Resolution	the <i>Resolution</i> of your output file(s)
Screen based DGC	the <i>Curve Strategy</i> you applied to your separation(s)
DGC	the <i>Single Curve</i> you applied to your separation(s)
V Distortion	the <i>vertical distortion</i> you applied to your output file(s)
H Distortion	the <i>horizontal distortion</i> you applied to your output file(s)
Device	the <i>imaging device</i> you used
Negative Printing	whether you <i>inverted</i> your output
Mirror Printing	whether you <i>mirrored</i> your output



- another type of Smart Mark (for example a corner mark).
- b) Then select that Smart Mark in your **Imaging Engine task**.

Transformations

Use the **Transformations** settings to rotate or distort your output relative to your input file.

1. If you want to **Rotate** your file's contents, select the rotation you want to apply here. This can be 0, 90, 180 or 270 degrees. Any other value will give an error.

Note:



- The direction of the rotation is the one you selected *when configuring your Imaging Engine* (in the **Settings** tab). You will see  if you selected a clockwise or  if you selected a counterclockwise rotation.
- You can use a SmartName that resolves to 0, 90, 180 or 270 (if you use a SmartName, the task's default rotation direction, *counterclockwise*, will be used, regardless of what you have configured for your device).
- You need to use a numerical value (*ninety* will not work).
- This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If you want to apply **Distortion** to your file's contents:

- a) Select the type of distortion to apply: **No distortion**, **Flexo distortion** or **Values from file**.

You can also use a SmartName that resolves to *Not*, *Flexo* or *File*.

Note: If you selected **Values from file** and the input file contains distortion values, you will see these in the fields below (greyed out).

- b) If you want to apply **Flexo distortion**, enter the vertical  and horizontal  distortion percentages to use.

You can enter numerical values, or insert SmartNames that resolve to numerical values (without the % sign).

Note:

- The values should be within the 90%-110% range. If you enter a percentage outside of this range it will be automatically adjusted to the nearest value in the range (for example 85 will be adjusted to 90%).
However, if you use a SmartName that resolves to a percentage outside the range, the task will end in error.
- You can enter a simple formula and it will be calculated automatically (for example you can enter $100+2$ and this will be changed to 102%). This only works with numerical values and not with SmartNames.

PDF Objects

Use the **PDF Objects** settings to define:

- how to color manage RGB objects in your input file,
- what dot gain compensation to apply to your input file's images.

All parameters in this tab are public parameters, that you can fill in when submitting files through Shuttle.

1. In **Default RGB Profile**, specify the profile to use if no profile is specified for a certain RGB object in the input file. The default is `srgb.icc`.

You can use a SmartName that resolves to the full profile name.

2. Select what output profile you want to use when converting RGB objects to CMYK:
 - If you want to use the output profile from the input file (if there is one available), leave **Use output intent from file** selected (it is selected by default).
 - If you don't want to use the output profile from the input file, deselect **Use output intent from file** and specify the output profile to use in **Output Color Profile**.

This output profile will also be used if there is none available in the input file.

The default is `ISOcoated_v2_300_eci.icc`.

For the **Use output intent from file** option, you can use a SmartName that resolves to `true` (to use the file's profile) or `false` (to not use it).

For **Output Color Profile**, you can use a SmartName that resolves to the full profile name.

3. Select **Use PDF transfer curves** if you want to use the [dot gain compensation curves](#) attached to the images in your input PDF.

These curves compensate for the unwanted characteristics of the device that created the images (scanner, digital camera...).

You can use a SmartName that resolves to `true` (to use the input file's curves) or `false` (to not use them).

Document Inks

Use the **Document Inks** settings to select which separations of the input file(s) you want to RIP.

For example, you can use these settings to exclude technical and/or varnish inks from your output.

See [Selecting the Separations to Keep in the Output](#) on page 1219 for more information.

Note:

If your file contains PantoneLIVE inks and you don't have a PantoneLIVE license, the **Ink Book** column will show `unregistered`, followed by an ID number (for example `<unregistered "b0c3126e-4f74-4402-8e47-e5e9fe50195a">`) instead of the PantoneLIVE ink book name for those inks.

Output

Use the **Output** settings to define what type of output you want and where to output it.

You can set these through public parameters or using SmartNames.

1. Enter your output **File Name** and output **Folder**.

Note:

- You can use Imaging-specific SmartNames for the **File Name**.

By default, it is set to:

- [File]_[ShortInk] for single page input files (which gives output file names like Myfile_C.tif),
- [File]_p[Page]_[ShortInk] for multi-page input files (which gives output file names like Myfile_p1_C.tif).
- The **File Name** and **Folder** can contain special characters (except \ / : * ? < > " |).
- The whole path to the output file (including the container, **Folder** and **File Name**) can be up to 255 characters.
- You can set a local or remote URL for the output **Folder**, but it must be an absolute path.
- If your output folder doesn't exist yet, the task will create it if the location you request is valid (you should place this folder on an Automation Engine container accessible by your Automation Engine user).

If the output folder creation fails, please check the task log for the full list of conditions and adapt your output folder path accordingly.

2. Define what to do if a file of the same name already exists in your output folder:


a) Click **Set Overwrite Policy...**

b) Choose what to do **If files exist already**. You can either:

- Extend the file name** if you want to keep the existing output file and give the new output file a different name. In this case, your new output file name consists of:
 - the **Output File Name** (the **File Name** you set earlier) - which is what the file would be called if no file with the same name existed,
 - a custom suffix that you can define using plain text and/or SmartNames (by default, this is `_c`, for "copy"),
 - an automatic suffix made of the three last digits of the task **ID**,

Tip:

To see the task ID of your tasks:

- Go to the Pilot's **Tasks** pane (in the **Files** or **Tasks** view).
- Go to **View > Select Columns in 'Tasks'...**
- In the **Columns** dialog, select **Task ID** in the **Available columns** list and click  to move it to the **Show these columns:** list.
- Click **OK**.
- You now have a **Task ID** column in the **Tasks** pane, listing the task ID of all the tasks.

- the file extension.

For example, you are working with a CMYK input file called `file.pdf` and you have left the tasks' output **File Name** to its default value (the SmartName **[File]**).

The first TIFF files you generate will be called:

- `file_C.tif`,
- `file_M.tif`,
- `file_Y.tif`,
- `file_Y.tif`,

If you generate output from this input file again, with the task 3669, and you have set the overwrite policy to **Extend the file name** with the default **Output File Name + _c + ID** name, the new output files will be called:

- `file_C_c669.tif`,
- `file_M_c669.tif`,
- `file_Y_c669.tif`,
- `file_Y_c669.tif`,

Tip:

- If you are working with the Bitmap Viewer, we recommend you keep the default overwrite policy (**Output File Name + _c + ID**), which is compatible with the Bitmap Viewer.
- If you are not, you can define the custom suffix as you wish. For example, you could use date and time SmartNames.

- **Overwrite** the existing file(s).
- **End the task in error.**

Note: If you use a SmartName to set this, it must resolve to `extend` or 0 (for **Extend the file name**), `overwrite` or 1 (for **Overwrite**), or `error` or 2 (for **End the task in error**).

c) Click **Set**.


3. Select the output **File Type** you want. This can be a **TIFF File** or a **PDF File** (see [RIP'ed File Formats](#) on page 1147 for more information).

The task generates *Separated* output for both file types. This means you get one output file per page of the input file, and per output separation.

The **File Assembly** is automatically set to **1 File Per Separation**, and the **Output Type** set to **Separate** (for both file types).

Note:

- Every output file that Imaging Engine generates contains XMP information (metadata about file creation, separations, calibration curves, etc.).

You can view this XMP information by selecting your output file in the Automation Engine Pilot's **Files** view and clicking .

For more information about XMP metadata, see [the XMP Metadata white paper on help.esko.com](http://help.esko.com).

- TIFF files produced by Imaging Engine are fully compatible with Esko's Bitmap Viewer and DotSpy.

4. Choose the **Compression** to use for your output files.

- For **TIFF** output, you can choose between lossless and lossy compression. Lossless compression preserves all the detail in the file, while lossy compression removes some detail.

We recommend you use lossless compression for the final output (for files you will send to a third party RIP for example), and lossy compression only for previews and thumbnails.

For both compression types (lossless and lossy), you should then choose a compression method, based on the compression speed and the compressed file size. Typically, faster compression methods compress the file less, so the output file is bigger (note that TIFF files can be a maximum of 4GB).

If you want lossless compression...	<p>choose either:</p> <ul style="list-style-type: none"> • PackBits, a fast lossless compression method for graphic files. <p>Tip: Use this if you want fast output and file size is not too much of an issue.</p> <ul style="list-style-type: none"> • LZW, a lossless compression method for graphic files and other files. <p>Tip: This compression method works best for files with large images.</p>
If you want lossy compression...	<p>choose one of the JPEG compression methods (those are more suitable for images like photographs and natural artwork than sharp edged or flat color images like line art or lettering, as JPEG compression tends to introduce noise into solid color areas).</p> <p>They are sorted from very low compression (JPEG Limited), which doesn't lose much detail but doesn't reduce the file size much, to very high compression (JPEG Extreme), which reduces the file size a lot but loses a lot of detail.</p>

- For **PDF** output, all available compression methods are lossless.

You should choose a compression method based on the compression speed and the compressed file size. Typically, faster compression methods compress the file less, so the output file is bigger (note that PDF files can be a maximum of 12GB).

Choose either:

- **PackBits** (see above).
- **Flate**, a lossless compression method similar to zipping, based on the LZW compression.

Tip: This compression method works best for images with large single color areas or repeating patterns.

- **JPEG 2000**, a compression method based on **JPEG** and suitable for large images (the version of **JPEG 2000** used by Imaging Engine is lossless).

Tip: The **Compression** method offered by default for the output **File Type** you select (**LZW** for **TIFF** and **PackBits** for **PDF**) is generally the best for this file type.



Attention:

- When using a SmartName to set the output file's **Compression**, it must resolve to one of the following (case insensitive):
 - **PackBits, LZW, JPEGLimited, JPEGLow, JPEGStandardLow, JPEGStandard, JPEGStandardHigh, JPEGHigh, or JPEGExtreme** for **TIFF** output,
 - **PackBits, Flate or JPEG2000** for **PDF** output.

Setting any other value, or an unsupported compression for the file type you chose, will make the task error (with the message `Unsupported combination of output type and compression` in the log).

- If you have already set the **Compression** but decide to change the output **File Type**, by default the task will either:
 - keep the same **Compression** (if it is supported for the new output **File Type**),
 - or (if it isn't), set it to the first **Compression** available for this **File Type**.

You can still change it as desired.

5. Set the **Resolution** of your output files.

Note:

- The **Resolution** unit is taken from the general Automation Engine Preferences.
- You can enter values from 1 to 25400 ppi (or equivalent in your unit). SmartNames must also resolve to a value in that range.
- Values in another unit than the one from the general Preferences will be recalculated, and values like 300ppi + 500ppm will be calculated correctly.

6. If your input file contains images, the images' pixels may not map exactly to output pixels because:

- the images' resolution is different from the output resolution you chose,
- scaling and rotation effects change the size and placement of the images' pixels.

You then need to resample these images to be able to output them at the same resolution as the rest of the file.

In the **Resample Images** option, choose the resampling method you want to use (you will have to make a trade-off between processing time and image quality).

- Choose **From File** to use the resampling method defined in the input PDF (this is the default option). This method can be either:
 - the "nearest neighbor" algorithm, which keeps the image very sharp but can make it look jagged (pixellated but sharp). This is mostly recommended if you have small images with sharp lines/sudden differences in intensity, and/or you do not want your images to be fuzzy at all.



This resampling method is very fast.

- the "PDF interpolation" algorithm, that smoothes the image while resampling (see below).

Note: Most PDF files don't specify a preferred resampling method, so the default ("nearest neighbor" algorithm) is used. If you want to make sure that the "PDF interpolation" algorithm is used, choose **PDF Interpolation** instead of **From File** in the **Resample Images** option.

You can also use a SmartName resolving to `FromPDF`.

- Choose **PDF Interpolation** to smooth the image while resampling.

With this method, you get images that are pixellated and fuzzy, so it is only recommended if you have small images with gradual differences in intensity, like photographs of nature.



This is slightly less fast.

You can also use a SmartName resolving to `PDFInterpolate`.

- Choose **Auto Resampling** to upsample the images to an automatically calculated resolution (generally half or a third of the output resolution) before smoothing it.

When using this, images are a lot less pixellated and fuzzy.



This method is quite slow but gives images of a good quality, so it is recommended in most cases (unless you want an extra sharp image, in which case you should choose **From File**).

You can also use a SmartName resolving to `ResampleAuto`.

- Choose **Full Resampling** to upsample the images to the full output resolution.

This is the slowest method but gives the highest quality images (not pixellated and only slightly fuzzy).



Use this if **Auto Resampling** doesn't give you the results you want.



Attention:

Resampling to full resolution can require significant amounts of memory when the output resolution is high (especially from 4000 dpi).

If you want to do this, we recommend that you have at least 8GB of RAM available for each worker (see [Installation](#)).

You can also use a SmartName resolving to `ResampleFull`.

Note:

If your image is not...

- an 8 bit image,
- an image with 1, 3 or 4 components (for example grey, RGB or CMYK),

then the **From File** option is always used.

Separations

Use the **Separations** settings to select the [dot gain compensation](#) to apply to the separations you selected in the [Document Inks settings](#).

You can define [default](#) dot gain compensation settings for most or all of these separations, and [exceptions](#) to these settings if you want to RIP some of these separations differently.

Defining Default Settings

Define the default dot gain compensation (DGC) settings you want to use on the separations to RIP. You will still be able to define [exceptions](#) to these settings for certain inks.

Under **Default Separation Settings**:

1. Define the **Plate Curve** you want to use on the separations to RIP.

This curve is used to compensate for the dot growth or dot reduction (for example in Digital Flexo) happening when making the plate. Choose either:

- **None** if you don't want to apply any plate curve.
- **Single Curve** if you want to apply one [curve](#) for all separations (you can still define exceptions for some of the separations later).

In this case, select the `.dgc` curve to apply at right.

You can choose from a list of curves available in the resource server defined for your Automation Engine server.

- **Curve Strategy** if you want to apply a [curve strategy](#) (a file created in Curve Pilot that contains several curves to apply in different cases: it can have different curves per separation, per dot shape, per ruling...).

In this case, select the `.icpro` curve strategy or legacy `.scredgc` screen-based DGC file to apply at right.

You can choose from a list of curve strategies available in the resource server defined for your Automation Engine server.

Note:

- You can use SmartNames resolving to the exact name of the **Single Curve** or **Curve Strategy** (including the extension).
- This will set the **Plate Curve** type to `Unknown` in the ticket, but will resolve to the correct curve type if the SmartNames are correct.

2. Define the **Press Curve** you want to use on the separations to RIP.

This curve is used to compensate for the dot gain on press. Choose either:

- **None** if you don't want to apply any press curve,
- **Single Curve** if you want to apply one [curve](#) for all separations (you can still define exceptions for some of the separations later).

In this case, select the `.dgc` curve to apply at right.

You can choose from a list of curves available in the resource server defined for your Automation Engine server.

- **Curve Strategy** if you want to apply a [curve strategy](#) (a file created in Curve Pilot that contains several curves to apply in different cases: it can have different curves per separation, per dot shape, per ruling...).

In this case, select the curve strategy to apply at right (you can select a `.icpro` curve strategy, a `.prsync` [PressSync curve set](#) or a legacy `.scredgc` screen-based DGC file).

You can choose from a list of curve strategies available in the resource server defined for your Automation Engine server.

- **PressSync Curve** if you want to apply a [PressSync curve](#) (a predefined curve applicable to your printing environment).

In this case, select the letter and number of the PressSync curve.

Note:

- You can use SmartNames resolving to the exact name of the **Single Curve, Curve Strategy** (including the extension) or **PressSync Curve**.
- This will set the **Press Curve** type to `Unknown` in the ticket, but will resolve to the correct curve type if the SmartNames are correct.

Defining Exception Settings

If you don't want to use the [default settings](#) you defined for certain inks, you can define specific dot gain compensation settings for those here.

1. Click **Define Exceptions...**
2. In the **Define Exceptions** dialog, select to **Define Exceptions** either:
 - **by using exception criteria** (see [Defining Exceptions Using Exception Criteria](#) on page 1189),
 - **Manually** (see [Defining Exceptions Manually](#) on page 1191).

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

Defining Exceptions Using Exception Criteria

If you want to define exceptions to the default dot gain compensation settings for certain inks, you can do it using exception criteria.

This works as follows:

- You create a rule per separation or set of separations that need a specific dot gain curve.
- Within each rule:
 - You define criteria to select the separation(s) you want to apply a specific curve to.


Each criterion will check something about the inks in your file.

This check can be about the **Ink name, Ink book, Ink Index, Ink type** or **Printing method**. For example, you can check if the **Ink book** is `designer`.

Tip:

The **Ink index** is the order in which the ink will be printed (the ink with ink index 1 is at the bottom of the document and will be printed first).

To find the index of a file's inks:

1. select the file in the Automation Engine Pilot's **Files** view,
2. click ,
3. in the **Info** dialog that opens, select **Inks** in the left panel.

The order of the inks at right is the ink index (the topmost ink has an index of 1, the second of 2, etc.).

Note that if you reorder the inks by one of the columns (for example by **Ink name**), they won't be ordered by ink index anymore. You will need to close that dialog and reopen it to see the ink index order again.

- You select that curve.
- If you have several rules, you can reorder them in order of priority (the top one is applied first).


Note: You can do this either when running the task on one or more files, or when creating a task ticket to use later.

If you *have chosen* to **Define Exceptions by using exception criteria**, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, enter a **Rule Name**.
3. Under **If separation matches**, define one or more criteria to select your desired separation(s).
 - a) Define your criteria the following way:
 - **Ink name:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink book:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink index:** you can define if it **equals/is less than/is greater than** the number you enter, or is **is between** the two numbers you enter (in this case the ink indexes you enter as boundaries are also selected).
 - **Ink type:** you can define if it **is** a piece of text (alphanumerical characters).
 - **Printing method:** you can define if it **is** a piece of text (alphanumerical characters).

See [the Printing methods section](#) of the Automation Engine documentation for more information about printing methods.

Tip:

- You can enter several values to check for: just use a semicolon (;) between them.
For example, to check for both inks called `Cyan` and inks called `Magenta`, choose **Ink name is** and enter `Cyan;Magenta`.
- When launching the task on a file, you can also click  to select one or more of the **Ink names/Ink books/Ink types/Printing methods** used in your input file(s).
A criterion will automatically be created based on your selection.

b) Click **+** to define more criteria.

Note: You can create as many criteria as you want, they will be applied from top to bottom (the top one will be applied first).

4. Select **Press Curve** and choose the press curve to use for the selected separation(s).
5. Click **Add**.

You can see your rule and the **Press Curve** you selected for the exception separation(s) in the **Define Exceptions** dialog.

Tip: You can double-click your rule if you still want to edit it.

6. Repeat steps 1 to 5 to define more exception rules if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog so you'll be able to see your exception separation(s) with its/their **Press Curve** in the main task panel.

7. If desired, you can reorder your exception rules (they are applied from top to bottom). To do this, select a rule and click **Move Up** or **Move Down**.
8. Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Defining Exceptions Manually

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

If you *have chosen* to **Define Exceptions Manually**, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, select the **Separation** you want to make an exception for.
3. Select **Press Curve** and choose the press curve you want to use for this separation.
4. Click **Add**.

You can see your exception separation with its **Press Curve** in the **Define Exceptions** dialog.

Tip: You can double-click your exception if you still want to edit it.

5. Repeat steps 1 to 4 to define more exceptions if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog selected so you'll be able to see the settings for that separation in the main task panel.

- Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Switching How You Define Exceptions

You can switch between the two methods of defining exceptions, and this will give the same results.

Switching from Defining Exceptions Manually to Defining Exceptions Using Exception Criteria

If you defined exceptions manually but then choose to **Define Exceptions by using exception criteria**, the task will translate your manual exceptions into rules, so that the same inks are selected, and they have the same exception settings.

For example, if you manually selected **Cyan**, **Magenta** and **Black** and assigned to each of these separations the **Press Curve** `CMK.dgc`, you will get the following rules:

Rule Name: Cyan (process)

Ink name is Cyan

Press Curve: `CMK.dgc`

Rule Name: Magenta (process)

Ink name is Magenta

Press Curve: `CMK.dgc`

Rule Name: Black (process)

Ink name is Black

Press Curve: `CMK.dgc`

Tip: The rule names are created automatically from the contents of the exceptions, but you can still double-click each rule to edit them.

Note:

If you...

- first defined rules with exception criteria,
- then switched to defining manual exceptions but didn't make any changes,
- then switched back to exception criteria,

... then your rules will be kept.

If you did change the exceptions in manual mode, the task will automatically generate new rules.

Switching from Defining Exceptions Using Exception Criteria to Defining Exceptions Manually

If you defined rules with exception criteria but then choose to **Define Exceptions Manually**, each separation selected by a rule will be selected individually, and have the settings you defined in the rule.

For example if you had as criteria:

Ink name is Cyan;Magenta;Black

Ink book is process

and you selected as **Press Curve** CMK.dgc, you will get the following manual exceptions:

Ink Name	Ink Book	Press Curve
Cyan	process	CMK.dgc
Magenta	process	CMK.dgc
Black	process	CMK.dgc

Note: You can only switch to selecting inks manually when you are launching the task on a (single) file.

Summary

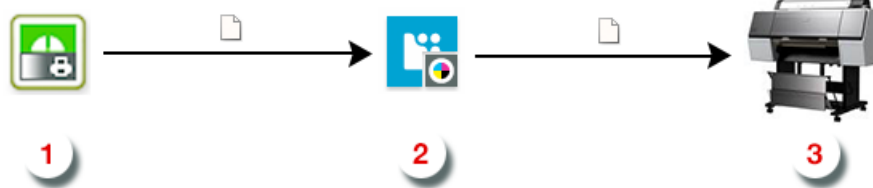
This shows a summary of all the settings you defined in the ticket.

- To show the settings from a tab, click + beside the tab name in the right pane.
Click **Expand All** to show the settings from all tabs.
- To hide the settings from a tab, click - beside the tab name in the right pane.
Click **Collapse All** to hide the settings from all tabs.

8.7.3 Proof

If you have an **Esko Proof Server**, you can use the Imagine Engine **Proof task** to send your RIP'ed output to a **physical proofer**, through the Proof Server.

This works as follows:



1. The **Proof task** *color manages* and *RIPs* your input file, and generates a TIFF file for proofing.
2. It sends this file to a workflow of your choice on the **Esko Proof Server**.
3. The Proof Server proofs it on the **proofing device** connected to that workflow.

Note:

To use this task, you need to have created a Proof Server **workflow** in the Esko Proof Client, and that workflow must:

- be connected to a physical proofer,
- have no color management or verification options set (you can set color management options [in the task itself](#)).

For details on how to create a Proof Server workflow, please see the [Esko Proof Client documentation](#).

Task Parameters

In the **Device** field, select the imaging device [that you configured](#).

Then use the selector pane at left to show more settings to fill in. See:

- [General](#) on page 1194,
- [Proofer](#) on page 1196,
- [Transformations](#) on page 1197,
- [PDF Objects](#) on page 1198,
- [Document Inks](#) on page 1199,
- [Color Management](#) on page 1199,
- [Output](#) on page 1199,
- [Summary](#) on page 1203.

General

1. Define what **Page Box** of the input PDF you want to RIP.

You can either select **Media Box**, **Trim Box**, or insert a SmartName `[]` that must resolve to either `media` or `trim`.

After selecting your file's page box you will see its dimensions under the **Page Box** option.

Note: This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If your input file is a multi-page PDF, select which **Pages** you want to RIP:
 - a) First select **All pages**, **Odd pages only** or **Even pages only** in the **Pages** option.

You can also insert a SmartName resolving to either `All`, `Even` or `Odd`.

- b) Then you can refine your selection by entering page numbers or a **Page Range**.

You can separate page numbers by a comma, or use a dash to indicate a page range (for example `3, 6-9`). You can also use a SmartName resolving to page numbers and/or page ranges using this same format.

Note:

- Page numbers are always physical pages. If your document starts with page 5, you should not use 5 but 1 to get the first page.
- If you enter page numbers that are not part of your document, they will be excluded from the output (all the other valid pages will be RIP'ed).
- If you enter the same page(s) more than once (for example 1, 1-2), it/they will only be output once.
- Pages are RIP'ed in the order of the document (entering 5-1 will be automatically changed to 1-5).

3. If you want to invert your output so it is compatible with the output device or software system that will process it, click **A**.

You can also click **I** or **Edit...** to insert a SmartName that must resolve to 1 or `true` (to invert the output), or to 0 or `false` (to leave the output as it is).

4. If you want to mirror your output so it is compatible with the output device or software system that will process it, click **M**.

You can also click **I** or **Edit...** to insert a SmartName that must resolve to 1 or `true` (to mirror the output), or to 0 or `false` (to leave the output as it is).

5. If you want to add a **Smart Mark** to your output file(s), select it in the **Smart Mark** list, or use a SmartName that will resolve to the Smart Mark name.

A **Smart Mark** is a mark whose content is variable, so you can make it display information about that specific job. For example, you can include some of the parameters used during RIP'ing in your output file(s) by using a Smart Mark.

You need to:

- a) First create the Smart Mark you want to use in **PackEdge** (see the **SmartMarks** section in the PackEdge manual for details).

You can create either:

- a Smart Mark with RIP-specific parameters,

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
Resolution	the <i>Resolution</i> of your output file(s)
V Scale	the <i>vertical scaling</i> you applied to your output file(s)
H Scale	the <i>horizontal scaling</i> you applied to your output file(s)
Color Strategy	the <i>Color Strategy</i> you applied to your output file(s)

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
Source Profile	the Source Profile contained in the Color Strategy you applied to your output file(s)
Destination Profile	the Destination Profile contained in the Color Strategy you applied to your output file(s)
Rendering Intent	the Rendering Intent contained in the Color Strategy you applied to your output file(s)
Device	the <i>imaging device</i> you used
Negative Printing	whether you <i>inverted</i> your output
Mirror Printing	whether you <i>mirrored</i> your output

- another type of Smart Mark (for example a corner mark).
- b) Then select that Smart Mark in your **Imaging Engine task**.

Proofer

This is where you select the [Esko Proof Server workflow](#) to which the task sends your RIP'ed output.

- The **Proof Server Workflow** list contains all of the valid workflows found on your Proof Server.

You can:

- select a workflow from the list,
- use SmartNames (that must resolve to the name of a valid Proof Server workflow),
- set the workflow using public parameters.

Note:

To be valid, your workflow must:

- be connected to a physical proofer,
- have no color management or verification options set (you can set color management options [in the task itself](#)).

Imaging Engine checks whether the Proof Server and the selected workflow are available when defining ticket settings and when (re-)launching the task.

The task will end in error if it can't access the Proof Server workflow or if the workflow is invalid.

Tip:

For details on how to create a Proof Server workflow, please see the [Esko Proof Client documentation](#).

Transformations


Use the **Transformations** settings to scale or rotate your output relative to your input file.

1. If you want to **Scale** your input file's contents, select either:

- **Scale by percent** and enter the vertical  and horizontal  scaling percentages to use.

Note:

By default, you enter the vertical scaling percentage and the horizontal one is calculated automatically to keep the proportions.

If you want to enter the horizontal scaling percentage manually, click  to unlink it from the vertical one first.

However, if you do this then link the percentages again, the last percentage you entered will be kept, and the other one will be recalculated automatically to keep the proportions.

- **Fit to page size** to fit your output to the substrate defined in your [Proof Server workflow](#), while keeping the proportions (this will fit it to either the width or the height of the substrate, while making sure everything is on the substrate).
- **Fit to page width** to fit it to the width of the substrate defined in your [Proof Server workflow](#), while keeping the proportions.

If you don't want to scale your file's contents, leave **Do not scale** selected.

You can use SmartNames resolving to `not` (**Do not scale**), `ByPercent` (**Scale by percent**), `FitSize` (**Fit to page size**) or `FitWidth` (**Fit to page width**).

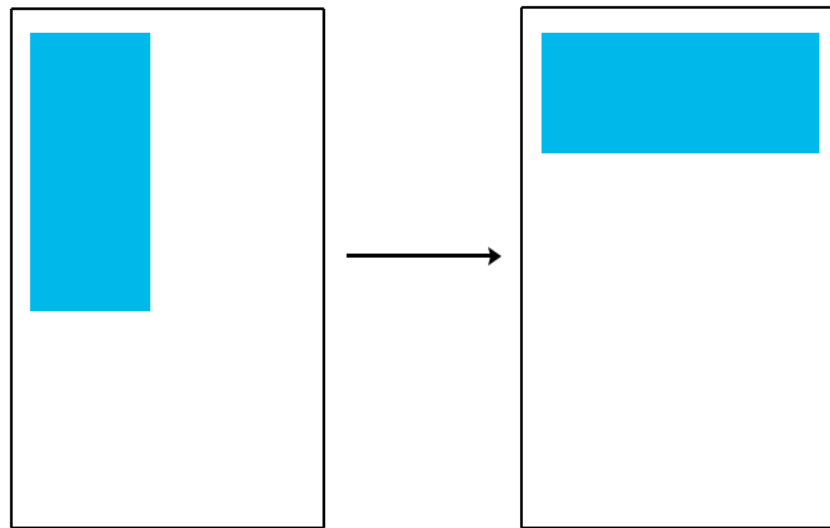
When entering SmartNames for vertical and horizontal scaling percentages, they must resolve to the numerical value (without the % sign).



2. If you want to **Rotate** your file, select the rotation you want to apply here.

You can choose to either:

- use a fixed rotation (this can be 0, 90, 180 or 270 degrees; any other value will give an error),
- rotate your file if necessary to fit, or fit better, on the substrate (select `Minimal Media Consumption`).

If it did fit on the substrate already, this option makes sure that the largest dimension of the file will be placed in the substrate width, to use less substrate material when cutting.

**Note:**

- The direction of the rotation is the one you selected *when configuring your Imaging Engine* (in the **Settings** tab). You will see  if you selected a clockwise or  if you selected a counterclockwise rotation.
- You can use a SmartName that resolves to 0, 90, 180, 270 or minimal (if you use a SmartName, the task's default rotation direction, *counterclockwise*, will be used, regardless of what you have configured for your device).
- When using a fixed rotation, you need to use a numerical value (*ninety* will not work).
- This is a public parameter, that you can fill in when submitting files through Shuttle.

PDF Objects

Use the **PDF Objects** settings to define:

- how to color manage RGB objects in your input file,
- what dot gain compensation to apply to your input file's images.

All parameters in this tab are public parameters, that you can fill in when submitting files through Shuttle.

1. In **Default RGB Profile**, specify the profile to use if no profile is specified for a certain RGB object in the input file. The default is `srgb.icc`.

You can use a SmartName that resolves to the full profile name.

2. Select **Use PDF transfer curves** if you want to use the *dot gain compensation curves* attached to the images in your input PDF.

These curves compensate for the unwanted characteristics of the device that created the images (scanner, digital camera...).

You can use a SmartName that resolves to `true` (to use the input file's curves) or `false` (to not use them).

Document Inks

Use the **Document Inks** settings to select which separations of the input file(s) you want to proof. For example, you can use these settings to exclude technical and/or varnish inks from your output. See [Selecting the Separations to Keep in the Output](#) on page 1219 for more information.

Note:

If your file contains PantoneLIVE inks and you don't have a PantoneLIVE license, the **Ink Book** column will show `unregistered`, followed by an ID number (for example `<unregistered "b0c3126e-4f74-4402-8e47-e5e9fe50195a">`) instead of the PantoneLIVE ink book name for those inks.

In this case, you must exclude these inks from the output, or the task will end in error (with the following message in the log file: `This job contains PantoneLIVE inks, and the necessary license for proofing these is not available.`).

Color Management

- Select a **Color Strategy** to apply *color management* to the output file(s).

You can choose from a list of color strategies available in the resource server defined for your Automation Engine server.

Note:

- The default color strategy is **linear**.
- You can only select a CMYK color strategy.
- You can select it through a public parameter, or using a SmartName. The SmartName for the color strategy must resolve to one of the available color strategies.

Once you have selected the desired color strategy, you can see the **Source** and **Destination** profiles used in the strategy.

Output

Use the **Output** settings to define what type of output you want and where to output it.

You can set these through public parameters or using SmartNames.

1. If your input file contains images, the images' pixels may not map exactly to output pixels because:
 - the images' resolution is different from the output resolution you chose,
 - scaling and rotation effects change the size and placement of the images' pixels.

You then need to resample these images to be able to output them at the same resolution as the rest of the file.

In the **Resample Images** option, choose the resampling method you want to use (you will have to make a trade-off between processing time and image quality).

- Choose **From File** to use the resampling method defined in the input PDF (this is the default option). This method can be either:
 - the "nearest neighbor" algorithm, which keeps the image very sharp but can make it look jagged (pixellated but sharp). This is mostly recommended if you have small images with sharp lines/sudden differences in intensity, and/or you do not want your images to be fuzzy at all.



This resampling method is very fast.

- the "PDF interpolation" algorithm, that smooths the image while resampling (see below).

Note: Most PDF files don't specify a preferred resampling method, so the default ("nearest neighbor" algorithm) is used. If you want to make sure that the "PDF interpolation" algorithm is used, choose **PDF Interpolation** instead of **From File** in the **Resample Images** option.

You can also use a SmartName resolving to `FromPDF`.

- Choose **PDF Interpolation** to smooth the image while resampling.

With this method, you get images that are pixelated and fuzzy, so it is only recommended if you have small images with gradual differences in intensity, like photographs of nature.



This is slightly less fast.

You can also use a SmartName resolving to `PDFInterpolate`.

- Choose **Auto Resampling** to upsample the images to an automatically calculated resolution (generally half or a third of the output resolution) before smoothing it.

When using this, images are a lot less pixelated and fuzzy.



This method is quite slow but gives images of a good quality, so it is recommended in most cases (unless you want an extra sharp image, in which case you should choose **From File**).

You can also use a SmartName resolving to `ResampleAuto`.

- Choose **Full Resampling** to upsample the images to the full output resolution.

This is the slowest method but gives the highest quality images (not pixellated and only slightly fuzzy).



Use this if **Auto Resampling** doesn't give you the results you want.



Attention:

Resampling to full resolution can require significant amounts of memory when the output resolution is high (especially from 4000 dpi).

If you want to do this, we recommend that you have at least 8GB of RAM available for each worker (see [Installation](#)).

You can also use a SmartName resolving to `ResampleFull`.

Note:

If your image is not...

- an 8 bit image,
- an image with 1, 3 or 4 components (for example grey, RGB or CMYK),

then the **From File** option is always used.

2. Enter the number of **Copies** you want to print (at least 1).

You can use SmartNames and/or public parameters.

Summary

This shows a summary of all the settings you defined in the ticket.

- To show the settings from a tab, click + beside the tab name in the right pane.
Click **Expand All** to show the settings from all tabs.

- To hide the settings from a tab, click - beside the tab name in the right pane.
Click **Collapse All** to hide the settings from all tabs.

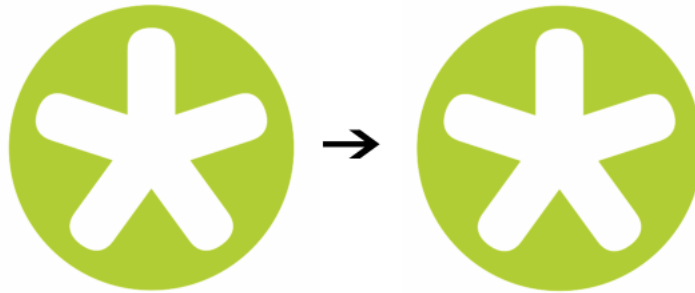
8.7.4 Proof to File - Unscreened

Use this task if you need unscreened but color managed output, in a file that you can send to a proofer.

Task Output

The **Proof to File - Unscreened** task can generate composite *PDF or TIFF files* (one file with all the separations).

In the example below, processing a file containing the lime green Esko star generates one multi-channel unscreened file containing the Cyan, Yellow, Magenta and Black channels, where the Magenta channel is empty.



Task Parameters

In the **Device** field, select the imaging device *that you configured*.

Then use the selector pane at left to show more settings to fill in. See:

- *General* on page 1204,
- *Proofer* on page 1206,
- *Transformations* on page 1207,
- *PDF Objects* on page 1208,
- *Document Inks* on page 1208,
- *Color Management* on page 1209,
- *Output* on page 1209,
- *Summary* on page 1218.

General

1. Define what **Page Box** of the input PDF you want to RIP.

You can either select **Media Box**, **Trim Box**, or insert a SmartName `[]` that must resolve to either `media` or `trim`.

After selecting your file's page box you will see its dimensions under the **Page Box** option.

Note: This is a public parameter, that you can fill in when submitting files through Shuttle.

2. If your input file is a multi-page PDF, select which **Pages** you want to RIP:

- a) First select **All pages, Odd pages only** or **Even pages only** in the **Pages** option.

You can also insert a SmartName resolving to either `All`, `Even` or `Odd`.

- b) Then you can refine your selection by entering page numbers or a **Page Range**.

You can separate page numbers by a comma, or use a dash to indicate a page range (for example `3, 6-9`). You can also use a SmartName resolving to page numbers and/or page ranges using this same format.

Note:

- Page numbers are always physical pages. If your document starts with page 5, you should not use 5 but 1 to get the first page.
- If you enter page numbers that are not part of your document, they will be excluded from the output (all the other valid pages will be RIP'ed).
- If you enter the same page(s) more than once (for example `1, 1-2`), it/they will only be output once.
- Pages are RIP'ed in the order of the document (entering `5-1` will be automatically changed to `1-5`).

3. If you want to invert your output so it is compatible with the output device or software system that will process it, click **A**.

You can also click **I** or **Edit...** to insert a SmartName that must resolve to `1` or `true` (to invert the output), or to `0` or `false` (to leave the output as it is).

4. If you want to mirror your output so it is compatible with the output device or software system that will process it, click **M**.

You can also click **M** or **Edit...** to insert a SmartName that must resolve to `1` or `true` (to mirror the output), or to `0` or `false` (to leave the output as it is).

5. If you want to add a **Smart Mark** to your output file(s), select it in the **Smart Mark** list, or use a SmartName that will resolve to the Smart Mark name.

A **Smart Mark** is a mark whose content is variable, so you can make it display information about that specific job. For example, you can include some of the parameters used during RIP'ing in your output file(s) by using a Smart Mark.

You need to:

- a) First create the Smart Mark you want to use in **PackEdge** (see the **SmartMarks** section in the PackEdge manual for details).

You can create either:

- a Smart Mark with RIP-specific parameters,

Use this Smart Mark parameter in PackEdge:	to display this value from the Imaging Engine parameters:
Resolution	the <i>Resolution</i> of your output file(s)
V Scale	the <i>vertical scaling</i> you applied to your output file(s)
H Scale	the <i>horizontal scaling</i> you applied to your output file(s)
Color Strategy	the <i>Color Strategy</i> you applied to your output file(s)
Source Profile	the Source Profile contained in the Color Strategy you applied to your output file(s)
Destination Profile	the Destination Profile contained in the Color Strategy you applied to your output file(s)
Rendering Intent	the Rendering Intent contained in the Color Strategy you applied to your output file(s)
Device	the <i>imaging device</i> you used
Negative Printing	whether you <i>inverted</i> your output
Mirror Printing	whether you <i>mirrored</i> your output

- another type of Smart Mark (for example a corner mark).
- b) Then select that Smart Mark in your **Imaging Engine task**.

Proofer

When outputting your RIP'ed files to a proofer:

- Select the **Substrate Size** from a list of predefined sizes, or enter the substrate **Width** and **Height** yourself to define a **Custom** substrate.

Note:

- By default, the **Substrate Size** is set to **Custom**, with the **Width** and **Height** values of an A3 page.
- The **Width** and **Height** unit is taken from the general Automation Engine Preferences.

Note:

You can set those values using public parameters and/or SmartNames. If you use SmartNames, make sure that:

- the SmartName for **Substrate Size** resolves to one of the substrate sizes offered in the list (when using a SmartName here, the **Width** and **Height** options will show **0**, but the task will use the correct dimensions when it runs).
- the SmartNames for **Width** and **Height** resolve to a numerical value, without any unit (the default task unit, mm, will then be used, regardless of the unit set in the general Automation Engine Preferences). When using a SmartName in one or both of these options, the **Substrate Size** will show **Custom**.

If one or more of the SmartNames resolves to an invalid value, the task will fail.

Tip:

The **Substrate Size** you select here can be used to scale the input file's contents to the substrate automatically (to do this, select **Fit to page size** or **Fit to page width** in the [Transformations tab](#)).

Transformations


Use the **Transformations** settings to scale or rotate your output relative to your input file.

1. If you want to **Scale** your input file's contents, select either:

- **Scale by percent** and enter the vertical  and horizontal  scaling percentages to use.

Note:

By default, you enter the vertical scaling percentage and the horizontal one is calculated automatically to keep the proportions.

If you want to enter the horizontal scaling percentage manually, click  to unlink it from the vertical one first.

However, if you do this then link the percentages again, the last percentage you entered will be kept, and the other one will be recalculated automatically to keep the proportions.

- **Fit to page size** to fit your output to the substrate you defined in the [Proofer settings](#), while keeping the proportions (this will fit it to either the width or the height of the substrate, while making sure everything is on the substrate).
- **Fit to page width** to fit it to the width of the substrate you defined in the [Proofer settings](#), while keeping the proportions.



If you don't want to scale your file's contents, leave **Do not scale** selected.

You can use SmartNames resolving to `not` (**Do not scale**), `ByPercent` (**Scale by percent**), `FitSize` (**Fit to page size**) or `FitWidth` (**Fit to page width**).

When entering SmartNames for vertical and horizontal scaling percentages, they must resolve to the numerical value (without the % sign).

2. If you want to **Rotate** your file's contents, select the rotation you want to apply here. This can be 0, 90, 180 or 270 degrees. Any other value will give an error.

Note:

- The direction of the rotation is the one you selected *when configuring your Imaging Engine* (in the **Settings** tab). You will see  if you selected a clockwise or  if you selected a counterclockwise rotation.
- You can use a SmartName that resolves to 0, 90, 180 or 270 (if you use a SmartName, the task's default rotation direction, `counterclockwise`, will be used, regardless of what you have configured for your device).
- You need to use a numerical value (`ninety` will not work).
- This is a public parameter, that you can fill in when submitting files through Shuttle.

PDF Objects

Use the **PDF Objects** settings to define:

- how to color manage RGB objects in your input file,
- what dot gain compensation to apply to your input file's images.

All parameters in this tab are public parameters, that you can fill in when submitting files through Shuttle.

1. In **Default RGB Profile**, specify the profile to use if no profile is specified for a certain RGB object in the input file. The default is `srgb.icc`.

You can use a SmartName that resolves to the full profile name.

2. Select **Use PDF transfer curves** if you want to use the *dot gain compensation curves* attached to the images in your input PDF.

These curves compensate for the unwanted characteristics of the device that created the images (scanner, digital camera...).

You can use a SmartName that resolves to `true` (to use the input file's curves) or `false` (to not use them).

Document Inks

Use the **Document Inks** settings to select which separations of the input file(s) you want to proof.

For example, you can use these settings to exclude technical and/or varnish inks from your output.

See *Selecting the Separations to Keep in the Output* on page 1219 for more information.

Note:

If your file contains PantoneLIVE inks and you don't have a PantoneLIVE license, the **Ink Book** column will show `unregistered`, followed by an ID number (for example `<unregistered "b0c3126e-4f74-4402-8e47-e5e9fe50195a">`) instead of the PantoneLIVE ink book name for those inks.

In this case, you must exclude these inks from the output, or the task will end in error (with the following message in the log file: `This job contains PantoneLIVE inks, and the necessary license for proofing these is not available.`).

Color Management

- Select a **Color Strategy** to apply *color management* to the output file(s).

You can choose from a list of color strategies available in the resource server defined for your Automation Engine server.

Note:

- The default color strategy is **linear**.
- You can only select a CMYK color strategy.
- You can select it through a public parameter, or using a SmartName. The SmartName for the color strategy must resolve to one of the available color strategies.

Once you have selected the desired color strategy, you can see the **Source** and **Destination** profiles used in the strategy.

Output

Use the **Output** settings to define what type of output you want and where to output it.

You can set these through public parameters or using SmartNames.

1. Enter your output **File Name** and output **Folder**.

Note:

- You can use Imaging-specific SmartNames for the **File Name**.

By default, it is set to:

- `[File]` for single page input files (which gives output file names like `Myfile.tif`),
- `[File]_p[Page]` for multi-page input files (which gives output file names like `Myfile_p1.tif`).

This is a good default for composite output.

For separated output, we recommend you use:

- `[File]_[ShortInk]` for single page input files (which gives output file names like `Myfile_C.tif`),
- `[File]_p[Page]_[ShortInk]` for multi-page input files (which gives output file names like `Myfile_p1_C.tif`).
- The **File Name** and **Folder** can contain special characters (except `\ / : * ? < > " |`).
- The whole path to the output file (including the container, **Folder** and **File Name**) can be up to 255 characters.
- You can set a local or remote URL for the output **Folder**, but it must be an absolute path.
- If your output folder doesn't exist yet, the task will create it if the location you request is valid (you should place this folder on an Automation Engine container accessible by your Automation Engine user).

If the output folder creation fails, please check the task log for the full list of conditions and adapt your output folder path accordingly.

2. Define what to do if a file of the same name already exists in your output folder:


a) Click **Set Overwrite Policy...**

b) Choose what to do **If files exist already**. You can either:

- **Extend the file name** if you want to keep the existing output file and give the new output file a different name. In this case, your new output file name consists of:
 - the **Output File Name** (the **File Name** you set earlier) - which is what the file would be called if no file with the same name existed,
 - a custom suffix that you can define using plain text and/or SmartNames (by default, this is `_c`, for "copy"),
 - an automatic suffix made of the three last digits of the task **ID**,

Tip:

To see the task ID of your tasks:

1. Go to the Pilot's **Tasks** pane (in the **Files** or **Tasks** view).
2. Go to **View > Select Columns in 'Tasks'...**
3. In the **Columns** dialog, select **Task ID** in the **Available columns** list and click  to move it to the **Show these columns:** list.
4. Click **OK**.
5. You now have a **Task ID** column in the **Tasks** pane, listing the task ID of all the tasks.

- the file extension.

For example, your input file is called `file.pdf` and you have left the tasks' output **File Name** to its default value (the SmartName **[File]**).

The first TIFF you generate will be called `file.tif`.

If you generate output from this input file again, with the task 3668, and you have set the overwrite policy to **Extend the file name** with the default **Output File Name + _c + ID** name, the new output file will be called `file_c668.tif`

Tip:

- If you are working with the Bitmap Viewer, we recommend you keep the default overwrite policy (**Output File Name + _c + ID**), which is compatible with the Bitmap Viewer.
- If you are not, you can define the custom suffix as you wish. For example, you could use date and time SmartNames.

- **Overwrite** the existing file(s).
- **End the task in error.**

Note: If you use a SmartName to set this, it must resolve to `extend` or `0` (for **Extend the file name**), `overwrite` or `1` (for **Overwrite**), or `error` or `2` (for **End the task in error**).

- c) Click **Set**.
3. Select the output **File Type** you want. This can be a **TIFF File** or a **PDF File** (see [RIP'ed File Formats](#) on page 1147 for more information).
 - For TIFF files, you can choose to output either *composite* or *separated* files.

To output **Composite** files, select **1 File Per Page** in **File Assembly** (or use a SmartName that resolves to `FilePerPage`). The **Output Type** is then automatically set to **Composite**.

To output **Separated** files, select **1 File Per Separation** in **File Assembly** (or use a SmartName that resolves to `FilePerSeparation`). The **Output Type** is then automatically set to **Separate**.

Note:

Those separated files are shown in the correct separation in the Bitmap Viewer (as it can read the separation information in the XMP data) and in grayscale in other viewers.

When proofing them, they are also in the correct separation (for example the proofer proofs the Magenta file using the Magenta cartridge).

- For PDF files, the task always generates **Composite** files. The **File Assembly** is automatically set to **1 File Per Page**, and the **Output Type** set to **Composite**.

Note:

With separated output, you can easily see how *color management* affects your output separations relative to the input separations.

- If you are using a **linear Color Strategy** (that doesn't color manage process colors), the process separated output files will contain the same objects as the corresponding separations in the input file.

In the CMYK example below, each solid object in an input separation is the same in the output separation (for example the Cyan bar is still only 100% Cyan).




- If you selected something other than **linear** in the **Color Strategy** (you are color managing all separations of your file), the process output separations will be different from the corresponding input separations.

In the CMYK example below, each solid object in an input separation is now made up of several output separations (for example the Cyan bar is now 60% Cyan, 36% Magenta and 22% Black, and appears in these three separated output files).



Note:

- Every output file that Imaging Engine generates contains XMP information (metadata about file creation, separations, calibration curves, etc.).

You can view this XMP information by selecting your output file in the Automation Engine Pilot's **Files** view and clicking .

For more information about XMP metadata, see [the XMP Metadata white paper on help.esko.com](http://help.esko.com).

- TIFF files produced by Imaging Engine are fully compatible with Esko's Bitmap Viewer and DotSpy.

4. Choose the **Compression** to use for your output file(s).

- For **TIFF** output, you can choose between lossless and lossy compression. Lossless compression preserves all the detail in the file, while lossy compression removes some detail.

We recommend you use lossless compression for the final output (for files you will send to a third party RIP for example), and lossy compression only for previews and thumbnails.

For both compression types (lossless and lossy), you should then choose a compression method, based on the compression speed and the compressed file size. Typically, faster compression methods compress the file less, so the output file is bigger (note that TIFF files can be a maximum of 4GB).

If you want lossless compression...	<p>choose either:</p> <ul style="list-style-type: none"> • PackBits, a fast lossless compression method for graphic files. <p>Tip: Use this if you want fast output and file size is not too much of an issue.</p> <ul style="list-style-type: none"> • LZW, a lossless compression method for graphic files and other files. <p>Tip: This compression method works best for files with large images.</p>
If you want lossy compression...	<p>choose one of the JPEG compression methods (those are more suitable for images like photographs and natural artwork than sharp edged or flat color images like line art or lettering, as JPEG compression tends to introduce noise into solid color areas).</p> <p>They are sorted from very low compression (JPEG Limited), which doesn't lose much detail but doesn't reduce the file size much, to very high compression (JPEG Extreme), which reduces the file size a lot but loses a lot of detail.</p>

- For **PDF** output, all available compression methods are lossless.

You should choose a compression method based on the compression speed and the compressed file size. Typically, faster compression methods compress the file less, so the output file is bigger (note that PDF files can be a maximum of 12GB).

Choose either:

- **PackBits** (see above).
- **Flate**, a lossless compression method similar to zipping, based on the LZW compression.

Tip: This compression method works best for images with large single color areas or repeating patterns.

- **JPEG 2000**, a compression method based on **JPEG** and suitable for large images (the version of **JPEG 2000** used by Imaging Engine is lossless).

Tip: The **Compression** method offered by default for the output **File Type** you select (**LZW** for **TIFF** and **PackBits** for **PDF**) is generally the best for this file type.



Attention:

- When using a SmartName to set the output file's **Compression**, it must resolve to one of the following (case insensitive):
 - **PackBits, LZW, JPEGLimited, JPEGLow, JPEGStandardLow, JPEGStandard, JPEGStandardHigh, JPEGHigh, or JPEGExtreme** for **TIFF** output,
 - **PackBits, Flate or JPEG2000** for **PDF** output.

Setting any other value, or an unsupported compression for the file type you chose, will make the task error (with the message `Unsupported combination of output type and compression` in the log).

- If you have already set the **Compression** but decide to change the output **File Type**, by default the task will either:
 - keep the same **Compression** (if it is supported for the new output **File Type**),
 - or (if it isn't), set it to the first **Compression** available for this **File Type**.

You can still change it as desired.

5. Set the **Resolution** of your output files.

Note:

- The **Resolution** unit is taken from the general Automation Engine Preferences.
- You can enter values from 1 to 25400 ppi (or equivalent in your unit). SmartNames must also resolve to a value in that range.
- Values in another unit than the one from the general Preferences will be recalculated, and values like 300ppi + 500ppm will be calculated correctly.

6. If your input file contains images, the images' pixels may not map exactly to output pixels because:
- the images' resolution is different from the output resolution you chose,
 - scaling and rotation effects change the size and placement of the images' pixels.

You then need to resample these images to be able to output them at the same resolution as the rest of the file.

In the **Resample Images** option, choose the resampling method you want to use (you will have to make a trade-off between processing time and image quality).

- Choose **From File** to use the resampling method defined in the input PDF (this is the default option). This method can be either:
 - the "nearest neighbor" algorithm, which keeps the image very sharp but can make it look jagged (pixellated but sharp). This is mostly recommended if you have small images with sharp lines/sudden differences in intensity, and/or you do not want your images to be fuzzy at all.



This resampling method is very fast.

- the "PDF interpolation" algorithm, that smoothes the image while resampling (see below).

Note: Most PDF files don't specify a preferred resampling method, so the default ("nearest neighbor" algorithm) is used. If you want to make sure that the "PDF interpolation" algorithm is used, choose **PDF Interpolation** instead of **From File** in the **Resample Images** option.

You can also use a SmartName resolving to `FromPDF`.

- Choose **PDF Interpolation** to smooth the image while resampling.

With this method, you get images that are pixellated and fuzzy, so it is only recommended if you have small images with gradual differences in intensity, like photographs of nature.



This is slightly less fast.

You can also use a SmartName resolving to `PDFInterpolate`.

- Choose **Auto Resampling** to upsample the images to an automatically calculated resolution (generally half or a third of the output resolution) before smoothing it.

When using this, images are a lot less pixellated and fuzzy.



This method is quite slow but gives images of a good quality, so it is recommended in most cases (unless you want an extra sharp image, in which case you should choose **From File**).

You can also use a SmartName resolving to `ResampleAuto`.

- Choose **Full Resampling** to upsample the images to the full output resolution.

This is the slowest method but gives the highest quality images (not pixellated and only slightly fuzzy).



Use this if **Auto Resampling** doesn't give you the results you want.



Attention:

Resampling to full resolution can require significant amounts of memory when the output resolution is high (especially from 4000 dpi).

If you want to do this, we recommend that you have at least 8GB of RAM available for each worker (see [Installation](#)).

You can also use a SmartName resolving to `ResampleFull`.

Note:

If your image is not...

- an 8 bit image,
- an image with 1, 3 or 4 components (for example grey, RGB or CMYK),

then the **From File** option is always used.

Summary

This shows a summary of all the settings you defined in the ticket.

- To show the settings from a tab, click + beside the tab name in the right pane.
Click **Expand All** to show the settings from all tabs.
- To hide the settings from a tab, click - beside the tab name in the right pane.

Click **Collapse All** to hide the settings from all tabs.

8.7.5 Working with Separations

When working with the *Imaging Engine tasks*, you will often need to:

- Select which separations you want to include or exclude when proofing or RIP'ing.
See [Selecting the Separations to Keep in the Output](#) on page 1219.
- Apply different (screening and) dot gain compensation settings to the separations you are including in the output (when RIP'ing).
See [Applying Different RIP'ing Settings to your Output Separations](#) on page 1223.

Selecting the Separations to Keep in the Output

When proofing or RIP'ing with one of the Imaging Engine tasks, you can select which separations of the input file(s) you want to actually proof or RIP in your task's **Document Inks** settings.

For example, you may want to exclude technical and/or varnish inks from your output.

You can choose to proof or RIP either:

- **All** separations.
- A number of separations you select **Manually** (see [Selecting Inks Manually](#) on page 1219).

Note:

You can only select separations manually when running the task on a single input file.

- A number of separations you select **by using selection criteria** (see [Selecting Inks Using Selection Criteria](#) on page 1220).

Selection criteria are rules that you define to select separations automatically for you when the task is launched on a file.

This is more advanced but also more powerful than selecting separations manually.

We recommend that you select separations manually when you first start working with Imaging Engine, then switch to using selection criteria when you want to automate your workflow a bit more.

Tip:

To get used to working with selection criteria, try the following on a few of your files:

1. select the separations you want to output manually,
2. then change **Select Inks** to **by using selection criteria** and see which rules have been created based on your manual selection.

Selecting Inks Manually

Note: You can only select inks manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

When choosing to **Select Inks Manually**, you will see a table containing all of your document's inks, showing each ink's:

- **Ink Name**,
- **Ink Book**,
- **Ink Index** (the order of that ink in the document),
- **Ink Type**,
- **Printing Method**.

Select the ink(s) you want to RIP, or select the toggle in the table header to select/deselect all inks.

You will see a table listing the **Selected document inks**, and another table with the **Document inks that are not selected**.



Attention: If no ink is selected, you cannot launch the task.

Switching from Selecting Inks Using Selection Criteria to Selecting Inks Manually

You can switch between the two methods of selecting inks, and this will keep the same inks selected.

When switching from *selecting inks using selection criteria* to selecting inks manually, all the inks selected by the rules you defined (the inks in the **Selected document inks** table) become selected manually.

Note: You can only switch to selecting inks manually when you are launching the task on a (single) file.

Selecting Inks Using Selection Criteria

If you don't want to *select inks manually* every time, you can define some rules to automatically select certain inks in your input file(s).

You can do this:

- either when running the task on one or more files, or when creating a task ticket to use later,
- by choosing to **Select Inks by using selection criteria** in the **Document Inks** settings.

This is how it works:


- Each rule you create will check something about the inks in your file.

This check can be about the **Ink name**, **Ink book**, **Ink Index**, **Ink type** or **Printing method**. For example, you can check if the **Ink book** is *designer*.

Tip:

The **Ink index** is the order in which the ink will be printed (the ink with ink index 1 is at the bottom of the document and will be printed first).

To find the index of a file's inks:

1. select the file in the Automation Engine Pilot's **Files** view,
2. click ,
3. in the **Info** dialog that opens, select **Inks** in the left panel.

The order of the inks at right is the ink index (the topmost ink has an index of 1, the second of 2, etc.).

Note that if you reorder the inks by one of the columns (for example by **Ink name**), they won't be ordered by ink index anymore. You will need to close that dialog and reopen it to see the ink index order again.

- You can create as many rules as you want.
- You can decide if you want to select the inks that match **All** of your rules, **Any** of your rules (at least one) or **None** of your rules (if you want to exclude the inks that match the rules).

Build your rules the following way:

1. Choose how to apply your rules:

- Select **All of the following are true** if you want to only select the inks that match **all** of the rules you define.
- Select **Any of the following are true** if you want to select all the inks that match **at least one** of the rules you define.
- Select **None of the following are true** if you want to only select the inks that match **none** of the rules you define.

2. Define your first selection rule.

You can select an ink based on its:

- **Ink name:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
- **Ink book:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
- **Ink index:** you can define if it **equals/is less than/is greater than** the number you enter, or is **is between** the two numbers you enter (in this case the ink indexes you enter as boundaries are also selected).
- **Ink type:** you can define if it **is** a piece of text (alphanumerical characters).
- **Printing method:** you can define if it **is** a piece of text (alphanumerical characters).


See [the Printing methods section](#) of the Automation Engine documentation for more information about printing methods.

For example, you could define the following rules to exclude a technical ink called "crease":

None of the following are true

Ink name is crease
Ink book is Visualizer Standard Finishes
Ink type is Technical
Printing method is Unknown

Tip:

- You can enter several values to check for: just use a semicolon (;) between them.
 For example, to check for both inks called Cyan and inks called Magenta, choose **Ink name is** and enter Cyan;Magenta.
- When launching the task on a file, you can also click  to select one or more of the **Ink names/Ink books/Ink types/Printing methods** used in your input file(s).
 A rule will automatically be created based on your selection.

3. Click + to add more rules and define them.

To remove a rule, click - next to it.

Tip: If you are launching the task on a single input file, you will see a table listing the **Selected document inks**, and another table with the **Document inks that are not selected** under the rules you defined.



Attention: If no ink is selected, you cannot launch the task.

Switching from Selecting Inks Manually to Selecting Inks Using Selection Criteria

You can switch between the two methods of selecting inks, and this will keep the same inks selected.

When switching from *selecting inks manually* to selecting inks using selection criteria, the task will translate your manual ink selection into rules, so that the same inks are selected (you can see them in the **Selected document inks** table).

The rules you get are based on the characteristics of the inks you selected.

For example, if you selected all the process colors but no spot color in your file (for an offset job), you will get the following rules:

All of the following are true

Ink name is Cyan;Magenta;Yellow;Black
Ink book is process
Ink type is Normal
Printing method is Offset

Note:

If you...

- first selected inks using selection criteria,
- then switched to manual selection without changing the selection,
- then switched back to selection criteria,

... then your rules will be kept.

If you did change the selection in manual mode, the task will automatically generate new rules.

Applying Different RIP'ing Settings to your Output Separations

Once you have *selected* which separations you want to include in your RIP'ed output, you can choose to apply different RIP'ing settings to these separations.

- When generating unscreened output with the *Image to Unscreened Separations* task, you can choose which *dot gain compensation* settings to apply.
- When generating screened output with the *Image to Screened Separations* task, you can choose which *screening* and dot gain compensation settings to apply.

In both cases, you can define **default** settings for most or all of the selected separations, and **exceptions** to these settings if you want to RIP some of these separations differently.

You can define those exceptions in two ways:

- **Manually** (see *Defining Exceptions Manually* on page 1173).

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

- **by using exception criteria** (see *Defining Exceptions Using Exception Criteria* on page 1171).

Exception criteria are rules that you define to apply certain RIP'ing settings to certain separations automatically when the task is launched on a file.

This is more advanced but also more powerful than applying RIP'ing settings to separations manually.

We recommend that you apply RIP'ing settings manually when you first start working with Imaging Engine, then switch to using exception criteria when you want to automate your workflow a bit more.

Tip:

To get used to working with exception criteria, try the following on a few of your files:

1. to apply your desired RIP'ing settings to certain separations manually,
2. then change **Define Exceptions** to **by using exception criteria** and see which rules have been created based on your manual settings.

Defining Exceptions Using Exception Criteria

If you want to define exceptions to the default screening and/or dot gain compensation settings for certain inks, you can do it using exception criteria.

This works as follows:

- You create a rule per separation or set of separations that need specific screening and/or dot gain compensation settings.
- Within each rule:
 - You define criteria to select the separation(s) you want to apply specific settings to.


Each criterion will check something about the inks in your file.

This check can be about the **Ink name**, **Ink book**, **Ink Index**, **Ink type** or **Printing method**. For example, you can check if the **Ink book** is `designer`.

Tip:

The **Ink index** is the order in which the ink will be printed (the ink with ink index 1 is at the bottom of the document and will be printed first).

To find the index of a file's inks:

1. select the file in the Automation Engine Pilot's **Files** view,
2. click ,
3. in the **Info** dialog that opens, select **Inks** in the left panel.

The order of the inks at right is the ink index (the topmost ink has an index of 1, the second of 2, etc.).

Note that if you reorder the inks by one of the columns (for example by **Ink name**), they won't be ordered by ink index anymore. You will need to close that dialog and reopen it to see the ink index order again.

- You select these settings.
- If you have several rules, you can reorder them in order of priority (the top one is applied first).

Note: You can do this either when running the task on one or more files, or when creating a task ticket to use later.


If you *have chosen* to **Define Exceptions by using exception criteria**, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, enter a **Rule Name**.
3. Under **If separation matches**, define one or more criteria to select your desired separation(s).
 - a) Define your criteria the following way:
 - **Ink name:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink book:** you can define if it **contains/begins with/ends with/is** a piece of text (alphanumerical characters), or if it **is empty**.
 - **Ink index:** you can define if it **equals/is less than/is greater than** the number you enter, or **is between** the two numbers you enter (in this case the ink indexes you enter as boundaries are also selected).
 - **Ink type:** you can define if it **is** a piece of text (alphanumerical characters).

- **Printing method:** you can define if it is a piece of text (alphanumerical characters).

See [the Printing methods section](#) of the Automation Engine documentation for more information about printing methods.

Tip:

- You can enter several values to check for: just use a semicolon (;) between them.
For example, to check for both inks called `Cyan` and inks called `Magenta`, choose **Ink name is** and enter `Cyan;Magenta`.
- When launching the task on a file, you can also click  to select one or more of the **Ink names/Ink books/Ink types/Printing methods** used in your input file(s).
A criterion will automatically be created based on your selection.

- b) Click **+** to define more criteria.

Note: You can create as many criteria as you want, they will be applied from top to bottom (the top one will be applied first).

4. Define the settings (**Dot, Ruling, Angle** and/or **Press Curve**) you want to use for the selected separation(s).

5. Click **Add**.

You can see your rule and the settings you selected for the exception separation(s) in the **Define Exceptions** dialog.

Tip: You can double-click your rule if you still want to edit it.

6. Repeat steps 1 to 5 to define more exception rules if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog so you'll be able to see your exception separation(s) with its/their settings in the main task panel.

7. Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Defining Exceptions Manually

Note: You can only define exceptions manually when you are launching the task on a (single) file, not when you are defining a generic ticket without an input file.

If you [have chosen](#) to define exceptions manually, do the following:

1. Click **Add**.
2. In the **Exception Properties** dialog, select the **Separation** you want to make an exception for.
3. Define the settings you want to use for this separation (**Dot, Ruling, Angle** and/or **Press Curve**).
4. Click **Add**.

You can see your exception separation with its settings in the **Define Exceptions** dialog.

Tip: You can double-click your exception if you still want to edit it.

- Repeat steps 1 to 4 to define more exceptions if desired.

If you want to remove an exception, select it and click **Remove**.

Tip: Leave the **Preview** option at the bottom of the dialog selected so you'll be able to see the settings for that separation in the main task panel.

- Click **OK** to save your exceptions and close the **Define Exceptions** dialog.

Switching How You Define Exceptions

You can switch between the two methods of defining exceptions, and this will give the same results.

Switching from Defining Exceptions Manually to Defining Exceptions Using Exception Criteria

If you defined exceptions manually but then choose to **Define Exceptions by using exception criteria**, the task will translate your manual exceptions into rules, so that the same inks are selected, and they have the same exception settings.

For example, if you manually selected **Cyan**, **Magenta** and **Black** and assigned to each of these separations the **Press Curve** `CMK.dgc`, you will get the following rules:

Rule Name: Cyan (process)

Ink name is Cyan

Press Curve: `CMK.dgc`

Rule Name: Magenta (process)

Ink name is Magenta

Press Curve: `CMK.dgc`

Rule Name: Black (process)

Ink name is Black

Press Curve: `CMK.dgc`

Tip: The rule names are created automatically from the contents of the exceptions, but you can still double-click each rule to edit them.

Note:

If you...

- first defined rules with exception criteria,
- then switched to defining manual exceptions but didn't make any changes,
- then switched back to exception criteria,

... then your rules will be kept.

If you did change the exceptions in manual mode, the task will automatically generate new rules.

Switching from Defining Exceptions Using Exception Criteria to Defining Exceptions Manually

If you defined rules with exception criteria but then choose to **Define Exceptions Manually**, each separation selected by a rule will be selected individually, and have the settings you defined in the rule.

For example if you had as criteria:

Ink name is Cyan;Magenta;Black

Ink book is process

and you selected as **Press Curve** CMK.dgc, you will get the following manual exceptions:

Ink Name	Ink Book	Press Curve
Cyan	process	CMK.dgc
Magenta	process	CMK.dgc
Black	process	CMK.dgc

Note: You can only switch to selecting inks manually when you are launching the task on a (single) file.

9. Integrating with External Systems

9.1 Introduction

This documentation is a guide for those who want to connect their Automation Engine server with other systems. Typically these systems are

- Business systems like MIS (Management Information System) or ERP (Enterprise Resource Planning). These systems usually have components like order entry, price calculation, customer database, production planning, stock etc.. These systems can be purchased by your company from an MIS vendor and then adapted to your specific business process or your company could have created her own unique system.
- Asset Management systems. They focus on managing and sharing your prepress data and are usually web-enabled.

A connection between your Automation Engine server and (one of) these systems is of course digital and can be one-directional, bi-directional or, in case more than 2 systems are connected, even triangular.

Note: Throughout this document, for the sake of simplification, we will name such systems 'the external system'.

Note: Many features that we describe in this chapter require the optional module named **Automation Engine Connect**.

Note: A separate document is available on integrating Automation Engine with Esko WebCenter.

9.1.1 Why Integrate?

Faster.

No time is wasted by waiting, usually for a person.

Better.

Error reduction: no errors re-typing or misinterpreting. A small error can have huge costs. A wrong 'status' interpretation can have major production (cost) consequences.

More consistent, more process control.

Having a consistent way of working will benefit all. It is easier to train new people. It will be required when your company is trying to receive quality certifications (ISO etc..).

Operators in both departments can focus on more added value tasks.

The freed up time can now be used to increase the *quality* of what they do or to *think* their actions over. Hiring new (young) people will be easier when they see you have modern ways of working.

Now let's have a look at what information typically is exchanged in each direction.

9.1.2 External System sending commands To Automation Engine

Create an Automation Engine Job and set Job parameters.

An Automation Engine Job represents a job order that you get from the business system. So it is logical not to create those Automation Engine Jobs manually but to get your instructions from that system in an automated digital way. All the parameters should then be picked up in a highly automated workflow.

Create an Automation Engine Product and set Product parameters.

New design data that arrive can be automatically added to the Automation Engine Products database, including the detailed product specifications like barcode and inks. Often, these design data arrive before there is a job order to produce them. The workflow can link this newly created product to a Job.

Start an Automation Engine workflow with parameters defined by the external system.

Not all parameters need to be stored first in a Job or Product: Automation Engine can also retrieve parameters from an external system during a workflow and use these parameters in subsequent steps of the workflow.

9.1.3 Automation Engine sending commands To an External System

Setting Job or Product status in that external system.

The business system is very interested in what is happening with the job in the prepress department. Being automatically informed on the exact status can have major benefits: Your sales/service people will have up to date info for your customer. Your production will be more efficient when its planning is based on reality. Automation Engine can send such feedback automatically. And based on that new status, the external system may also decide to start an own automatic workflow.

Initiating actions on that external system.

Automation Engine could even do more than that. It could really talk to that external application, ask questions and come back with a set of information.

9.1.4 Some general advice

Have a project manager

Appoint someone who owns the mission to do this. This integration can affect many people so someone needs to be in charge. It is actually possible that it is the first time that these 2 different departments (sales-service and prepress) are part of a same project. Each department will defend its way of working. Each department will not know all complexities of 'the other side'.

Change Management

When you start eliminating some human intervention, it is possible that new questions will come to the surface. Any changes in the way you work in such a busy production environment will need to be defended. The integration is indeed technical but it is actually *not* 'just an IT project': the reason

you do this will be of interest to higher management. Often higher management is where such ideas to integrate start. Making some changes will be necessary to get that higher efficiency that they are looking for.

Step by Step

Start with a step that is 'obvious' and build from there. Before 'going live', carefully test and dare to consider all sides of the changes that this integration will cause. Using *both* the new *and* the old workflow can be confusing, so be prepared to switch over fast.

9.1.5 Need any help?

If you want help with your integration project, Esko offers 2 levels of service.

- Standard training by an Esko Academy trainer.

Esko trainers will explain the Esko part of the tools.

Tip: After learning about the concepts and features in this documentation, we advise to also check the chapter [Examples](#) on page 1287.

- Help from an Esko Solution Architect.

Esko Solution Architects are experienced in such system integrations and have a broader knowledge of 'workflow', beyond the world of Esko products. Contact your Esko sales rep if you want to know more.

9.1.6 What about JDF?

Note: Job Definition Format is a standard language based on XML. It was created by the CIP4 group to help standardize system integration in the printing industry. Read more on <http://www.cip4.org>

Integrations that use JDF/JMF require assistance from an Esko Solution Architect. That is why this documentation does not cover JDF/JMF.

Note: With some specific MIS, Esko optionally offers a 'standard' integration package based on a White Paper. Some of these can be seen on <http://www.esko.com/movies>. Contact your sales rep if you want to know more.

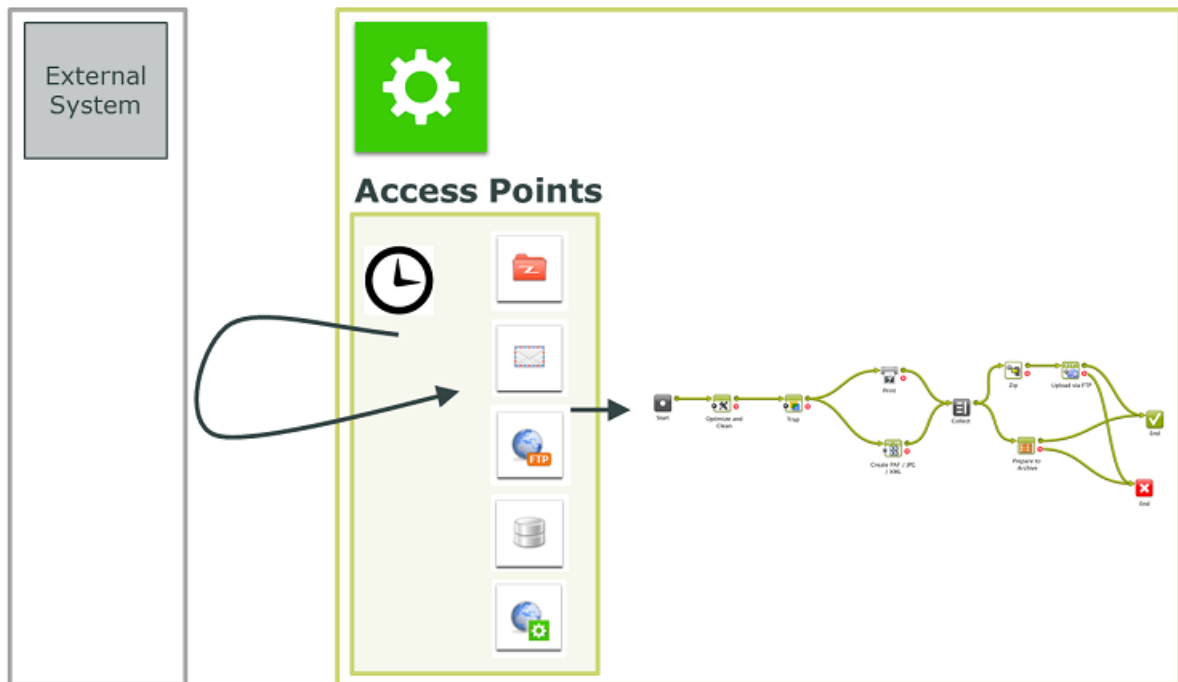
9.2 Integration Concepts

Let's distinguish the main *concepts* in integrating with Automation Engine. In many integrations, several of these concepts are combined. There is no pre-defined order in which these concepts would be used in your particular case.

The tools that we here introduce are documented in detail in the next chapters.

9.2.1 Access Points Triggering a Workflow on Automation Engine

Access Points trigger a workflow after they detected a signal from 'outside'. They use a 'polling' mechanism: they check at a regular time interval if they have to trigger that workflow. There are 5 types of Access Points. You can create as many as you need.



A typical example is a classic hot folder (a **Folder Access Point**), where the incoming data is then processed and the result is output on another folder. For example: design PDFs arrive in a hot folder, from there they will be automatically trapped and then written to another folder.

However, Access points are not only used to start work on that incoming data. They can also receive *parameters* for the workflow that they will start. These parameters define *what* needs to be done in the workflow with some other (set of) file(s). For example: a Folder Access Point receives an XML from the order entry system. The XML describes a list of re-run print Jobs. And for each Job it describes which Products need to be linked into that Job and what quantity needs to be printed.

Or, instead of *receiving* those parameters, Access Points can *go and search* for them. For example: a **Database Access Point** looks up the desired trapping distance in an external system. This results in a SmartName the trapping task then uses.

Other types of **Access Points** are **FTP**, **Mail** and **Web Service**. Find their detailed documentation in the chapter [Access Points](#) on page 1236.

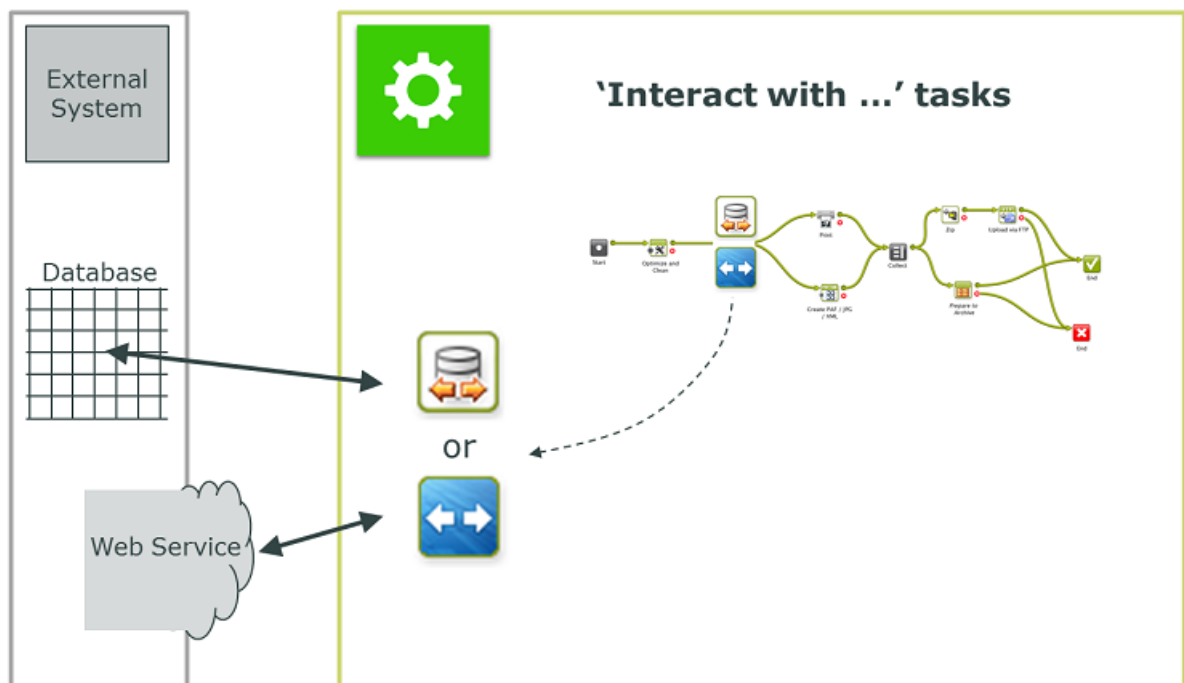
It's all about SmartNames!

In most cases, these parameters that come in from the external system are turned into SmartNames. And then these SmartNames will be picked up in the workflow tasks. These SmartNames can be stored in Job or Product parameters, when they describe job or product characteristics that are

relevant for the lifetime of the job or product. When the parameters are relevant only for the workflow started by the access point, they can be used as workflow parameters.

9.2.2 Automation Engine tasks Interacting directly with the External System

Not only Access Points can contact an external system. There are also 2 Automation Engine tasks that are dedicated to interact with external systems. Contrary to Access Points, their interaction can be **bi-directional**. Besides retrieving data from external systems, these tasks can also be used to provide feedback to external systems. This will be described in more detail later. However, these 2 tasks are typically used because the **workflow needs this interaction before it continues**. The result of these tasks will be used in the next steps of the workflow. As usual, this can be creating Jobs or Products, or get values for SmartNames to be used in that workflow.



- The **Interact with Database** task 

Automation Engine can use this task to **read** a value from an external database. For example: an operator starts a workflow to Step & Repeat and output. The workflow asks the database of the production planning system on which press this job will be printed. This is needed to know the layout size and also any dot gain curve. This press choice can be decided very late, so it might not have been part of the Job parameters that were created earlier.

Note: Reading from an external database is also possible by using a simple SmartName query or by using a Database Access Point.

Automation Engine can also use this task to **write** directly into the database of the external system. For example to inform it on the new status of a Job or Product. When that system can not read

XML messages with such information, then you can consider to have Automation Engine write this information directly into the database of that system.



Caution: This is a very direct way to have applications connect. This must be well designed and carefully tested.

Important: Esko takes no liability for queries that write or change fields in that external database.

- The **Interact with Web Service** task

The concept is very similar to the one above, but here the task uses HTTP calls to a web service. Also here, both **read** ('GET') and **write** ('POST') commands are possible. When the external system is running a web service, you will be able to access it via HTTP (Hypertext Transfer Protocol). However, you also need to learn **how** to communicate to that system. Some applications will indeed publish their API (Application Programming Interface). For example Automation Engine can use this task to ask a question to Google Maps. This is only possible because Google Maps did publish its API (and runs a web service). This example is illustrated in the chapter [Examples](#) on page 1287.

Typically, both these tasks execute such interactions as part of a larger workflow in Automation Engine. Find their detailed documentation in the chapter [Tasks Interacting with other Systems](#) on page 1274.

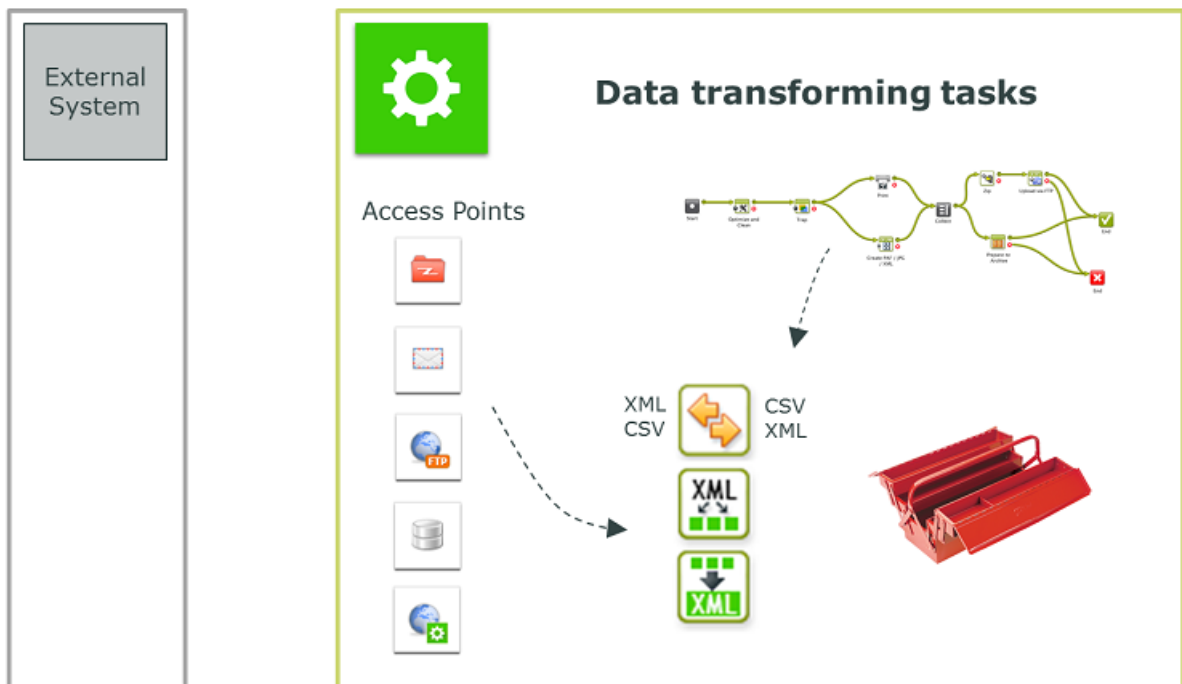
Note: The **Integrate with WebCenter** task is described in [Advanced WebCenter Integration](#) on page 1362, a chapter of the document on integrating Automation Engine with WebCenter.

9.2.3 Automation Engine Mapper tasks Transforming Data

Automation Engine offers many ways to use XML to control the workflow. Automation Engine also offers many features to create XML files. But sometimes you need to create an XML with a different content or formatting.

This **Create XML File** task is typically used to create XML files that are to be read by external systems. In such XML files, you sometimes want more or less information. And sometimes you need to re-format data in a way that the external system prefers.

Then there are the specific data formatting tasks like **Split XML**, **Join XML** or **Map Data** task. These are powerful tools that offer many more ways to control XML files. Sometimes the data (information) that comes from an external system first needs to be transformed before it can be used in Automation Engine. Sometimes, these transformations are classic file conversions like CSV to XML. The **Map Data** task was created to do this. Sometimes the XML will first need to be split in several XMLs (**Split XML** task). Sometimes it will be useful to join several data files into one XML (**Join XML** task).



Data transforming tasks can be used at any step in a workflow. They are used

- at the start of a workflow, to help transform what an Access Point delivered.
- at any other point in a workflow, also often at the end, to translate information into a format that the external system prefers.

Find a detailed description of these tasks in the chapter [Data Transforming Tasks](#) on page 1255).

Transforming data from an Access Point

Here are some examples where such data transforming tasks are used right at the beginning of a workflow that is started by an **Access Point**.

For example:

- The external system provides a list of jobs to be processed under the form of a large XML that it writes on the **Folder Access Point**. The **Split XML file** task then creates 1 XML file per job entry. These separate XML files are then used for the **Create Job** task.
- The external system sends an CSV file on a **Folder Access Point**. The **Map Data** task first transforms it into an XML. Later in the workflow, SmartNames will get their values by reading from that XML file (XPath queries).
- Some Access Points do not even result in data *files* yet. In case of a **Database Access Point** or a **Web Service Access Point**, the accessed data can arrive as a several XML files. This is where you can use the **Join XML** task to create a single and maybe even simpler XML file. Again, that XML file is then typically the base to provide SmartNames their workflow specific value.

Transforming data from a workflow

There are many moments where it can be necessary to transform Automation Engine information. Typically when sending information to external systems. Some of these external systems have their

own mapping tools to read these incoming XMLs anyway. But sometimes you will be asked to have Automation Engine transform them first.

Some examples:

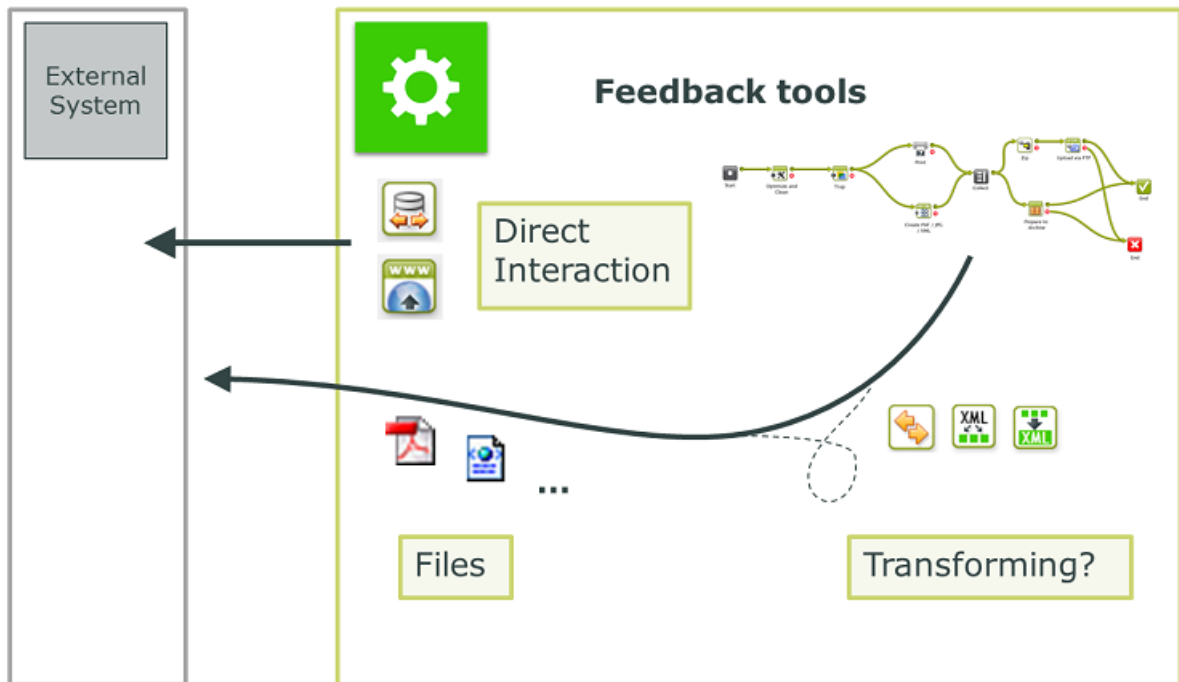
- The MIS can not read an XML file but can read CSV. This is a classic use case for the **Map data** task.
- The XMP (metadata) of a PDF needs to be sent to the external system as an XML. The **LinkEdge** task does that. But maybe the MIS wants to have the ink coverage information formatted in a different way. And maybe it is also necessary to convert the millimeters into inches.
- When using the **Interact with Web Service** task to ask the web service of Google Maps the distance to a customer, the answer needs to end up in an XML or a SmartName.
- A **Step & Repeat** task requires a specific XML as input file, with a different structure than the one from the external system.
- The **Create Job** task requires information that is present in 3 XMLs, from 3 different sources.
- The XML with feedback that you want to send to the external system needs to contain both a Job status and some meta data from the PDF.

9.2.4 Automation Engine sending data Back to the External System

We briefly described this topic in the introduction ([Automation Engine sending commands To an External System](#) on page 1229). Here, we add some detail and mention some tools.

A workflow in Automation Engine always generates output data, and often even many kinds. A lot of that data is not just for other prepress components, but for external systems. We distinguish **2 main types of data**:

- **Sending Prepress data (back)**
 - Data for output devices like proofers and RIPs (CTP, DFE, Press)
 - Data for people, often customer service or sales people or end-customers (PDFs attached in e-mail or sent via FTP)
 - Data for external systems, some of which web-enabled, often to view, approve or exchange 'assets'. In many cases it is not the original production files that are sent but smaller versions (less Mb) or more secure versions like password protected PDFs (JPEGs, RGB-PDFs, viewer streaming data, 3D files...)
- **Sending Communication data (back).** Here, the goal is to inform. Some communication will be about a status, for example 'plates made'. Some will be about describing the (new) prepress data themselves, for example the ink coverage of a production PDF. This communication data can also be files or it could be 'live':
 - Communication *files*: XML, XMP-XML, CSV, TXT
 - written on a network share, typically on a hot folder of the external system
 - sent via e-mail or FTP
 - Communicating *directly* with the external system: (see also [Automation Engine tasks Interacting directly with the External System](#) on page 1232.)
 - directly to its database, writing or updating fields
 - directly to the web service of the API of that external system



Transforming data before sending

Sometimes the external system does not support one of the above file formats or interaction protocols. In those cases, the **Map Data** task can help to convert. Find an introduction to this concept in [Automation Engine Mapper tasks Transforming Data](#) on page 1233.

Tip: You do not always need to send a lot of Mb to send a lot of information. For example: an XML describing the bar code parameters can be more useful than a high-res PDF. You could zoom-in on the PDF but the XML will tell you even the non-visible parameters of that bar code (type, bar width reduction, distortion...)

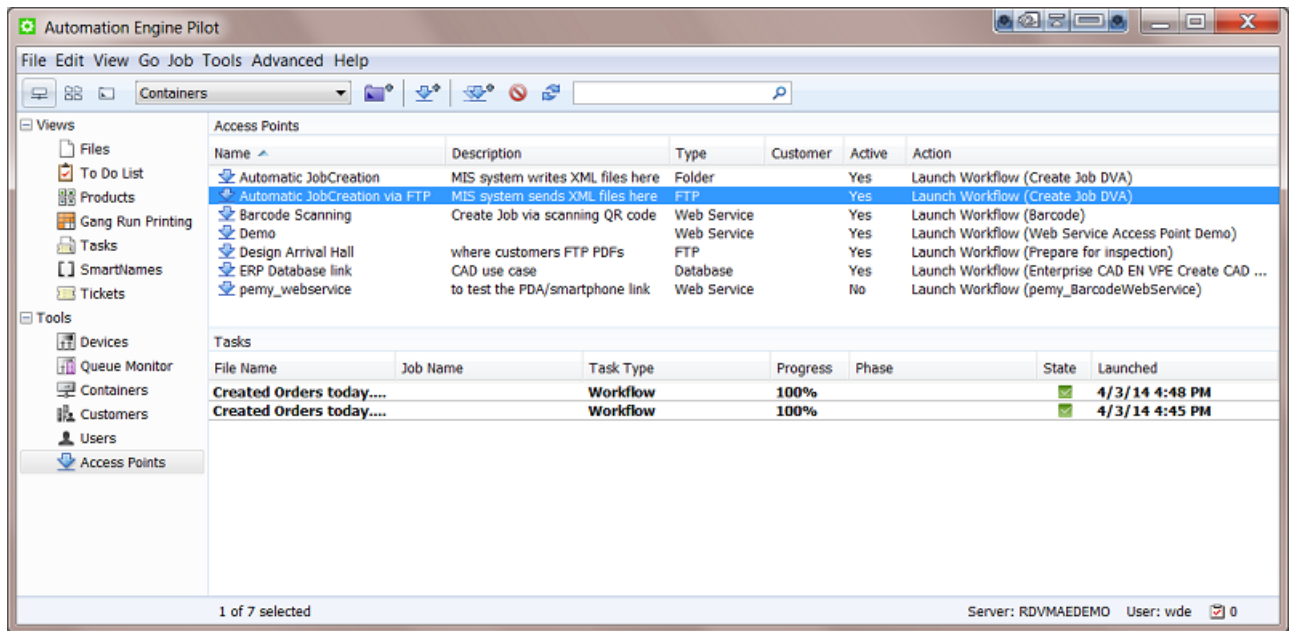
9.3 Access Points

An Access Point launches a workflow after it detected a signal from 'outside'. Automation Engine offers 5 types of Access Points. The **Folder Access Point** is the most classic example and is also often referred to as a 'hot folder'.

Tip: We advise you to first read the introduction about the concept of Access Points in [Access Points Triggering a Workflow on Automation Engine](#) on page 1231.

The Access Points Tool and View

In your **Pilot**, the **Access Points** Tool is where you create, modify and have an overview of your Access Points.







The top part of this view shows a list of all your **Access Points**.

The bottom part is a task monitor that shows the tasks that were executed by the selected **Access Point**.

Working with Access Points (the basics)

We here list the main functionality that is available in tool bar and also via the **File** menu. The next chapters will add detail per *type* of **Access Point**.

-  to **create** a new Access Point.
-  to **duplicate** an Access Point. The selected Access Point will open with empty fields for Name and Description.
-  to **delete** a selected Access Point. You will be asked to confirm.
-  to '**scan now**'. To decide manually when the selected Access Point should scan for new files. This can be useful when you do not want to wait for the time interval (see further).

Note: This function is disabled for the **Web Service Access Point**.

To **modify** an existing **Access Point**, select it from the list and double click it to open and modify.

To **deactivate** an **Access Point**, right-click it and choose **Deactivate Access Point**.

Differences in how they work

This table illustrates how some **Access Points** work in a slightly different way:

Table: Access Points - technical differences

Access Point Type	Checks for data at regular time interval ('Polling')	Uploads that data	Starts a workflow
Folder	V	V	typically
FTP	V	V	typically

Access Point Type	Checks for data at regular time interval ('Polling')	Uploads that data	Starts a workflow
Mail	V	V	typically
Database	V	- (**)	typically
Web Service	- (*)	- (***)	always

(*) The external system decides when to contact the Access Point

(**) *goes to and gets info from* an external database.

(***) *waits until it receives* an HTTP command from the external system

Access Point SmartNames

There is a specific category of SmartNames for Access Points. They enable smart re-using of many attributes of an Access Point. All Access Point SmartNames are also automatically stored as workflow parameters.

For more information on SmartNames, see the dedicated chapter [SmartNames](#) on page 916.

9.3.1 Folder Access Point

Concept

What is a Folder Access Point?

A **Folder Access Point** monitors a folder located somewhere in an Automation Engine **Container**. Every time one or more files are uploaded into that folder, the **Folder Access Point** will move the files to another folder where they will be processed. **Folder Access Points** are typically created to launch a workflow using those incoming files as input files for that workflow.

However, you can also choose *not* launch a workflow. In that case the **Access Point** only serves to upload files onto a specific folder in an Automation Engine **Container**.

Tip: The chapter [Examples](#) on page 1287 contains an example case of a **Folder Access Point**.

Note: **Folder Access Points** can also be used to process incoming JDF files. For more info on using JDF, read more in [What about JDF?](#) on page 1230.

What about Hot Folders?

Before version 14 of Automation Engine, the user could create 'Hot Folders'. When you upgrade to version 14, they will **not** be automatically migrated to **Folder Access Points**. They will stay Hot Folders and will still function as they did before.


If you did create such Hot Folders in a version older than version 14, you will also keep your **Hot Folders View** in the Pilot of your version 14. If you haven't created Hot Folders in Automation Engine before upgrading to version 14, you will not see the Hot Folders view in the Pilot.

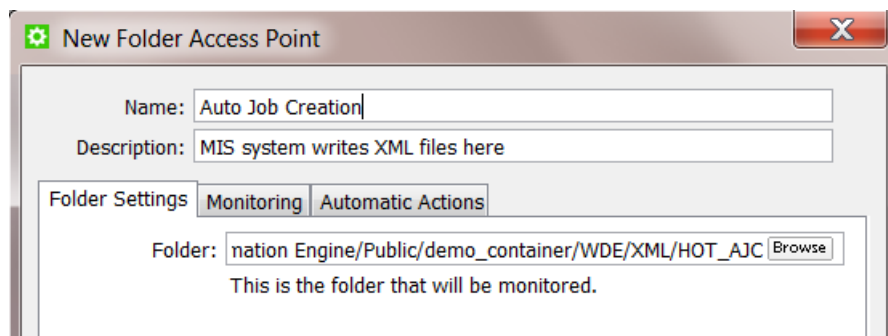
The following options were previously available in Hot Folders but are not available in **Folder Access Points**. This is because there are now other and better ways to do these actions:

- the annotation for the workflow task (you now write a 'description')
- grouped launching (you now can still create the effect by using a ZIP, but the most typical use cases are covered when you use the dedicated 'gang run' related features)
- delete input files when the workflow is finished (this can be done with the **Delete File** task at the end of your workflow)

Creating or modifying a Folder Access Point

Follow these steps to create or modify **Folder Access Points**:

1. In the **Pilot** click on **Access Points** view. Alternatively, choose **Tools > Access Points** .
2. In the dedicated **Access Points** view, click on  . Alternatively, choose **File > New** .
3. In the resulting window, select **Folder** and click **OK**.
The **New Folder Access Point** window will open. Here is one with some typical settings already filled in:



4. Enter a **Name** for the **Access Point**.
5. If you want, enter a **Description**.
This will help identifying your **Access Point** without having to read all its settings.
6. In the tab **Folder Settings**, define which folder will be the actual Access Point that will be monitored by Automation Engine. Use the browse button to indicate it.
This folder needs to be *inside* a **Container**. This folder may not be inside another **Folder Access Point**.
7. In the tab **Monitoring**, define the time interval.
 - **Check Every**. Enter the time interval with which Automation Engine should check if any new files came in. Think of the time interval as the *maximum* time it takes for the new files to be picked up.
 - **Schedule**. If you want this Access Point to be inactive during a part of the day, then activate this option and indicate the start and end time of that period.

Note: Such a temporary inactive state will not be shown in the general Access Points View. In that View, the column 'Active' shows its *general* status.
8. In the tab **Automatic Actions**, define what needs to happen.

- **Move Files to.** This is the folder where the files will be processed. If you want, use **SmartNames**. These files have to be moved to avoid that the **Access Point** would process the same files again and again.
- **Rename Files to.** If you want you can choose to rename these files too. To do this, enter the new file name.

Tip: Typically, SmartNames are used here to help indicate in their name *when* they were processed or *where* they came from.

- **Action.** Choose:
 - **None.** Choose this option when you only want to use this **Access Point** to *move* the files.
 - **Workflow.** Choose this option when you want a workflow to start. Select the workflow ticket from the offered list. The new accessed files will be used as input files for this workflow.



Attention: Make sure your workflow **can** start with the type of files that will arrive.

Options:

- **Operator:** choose the operator that will be shown as operator for the workflow task.

Tip: If you do not want to assign one specific person, you could create an operator named "Access Points" or "Automatic" and assign that one here. Also remember that the Access Points Task Monitor shows you the tasks launched by the selected Access Point.

Note: If you leave this option blank, a system user "UPLC" will be used.

- **Priority:** choose the priority for this workflow: Low, Normal or High. The default priority is 'Normal'.
- **Process JDF.** Choose this option when this Access Point will be used to process incoming JDF files. For more info on JDF, read this section: [What about JDF?](#) on page 1230.

9.3.2 FTP Access Point

Concept

An **FTP Access Point** monitors a folder on an FTP site. At a regular time interval, it will check if new files have arrived in that folder and it will move them to a designated folder in a **Container** of Automation Engine.

Note: There is no FTP server embedded in Automation Engine, so this functionality relies on a FTP server that has been setup separately from Automation Engine.

This also works for new files in any subfolders of the scanned FTP site. Those files will be moved to an identical structure of subfolders that will be automatically created.


The **FTP Access Point** can also start a workflow on those files.

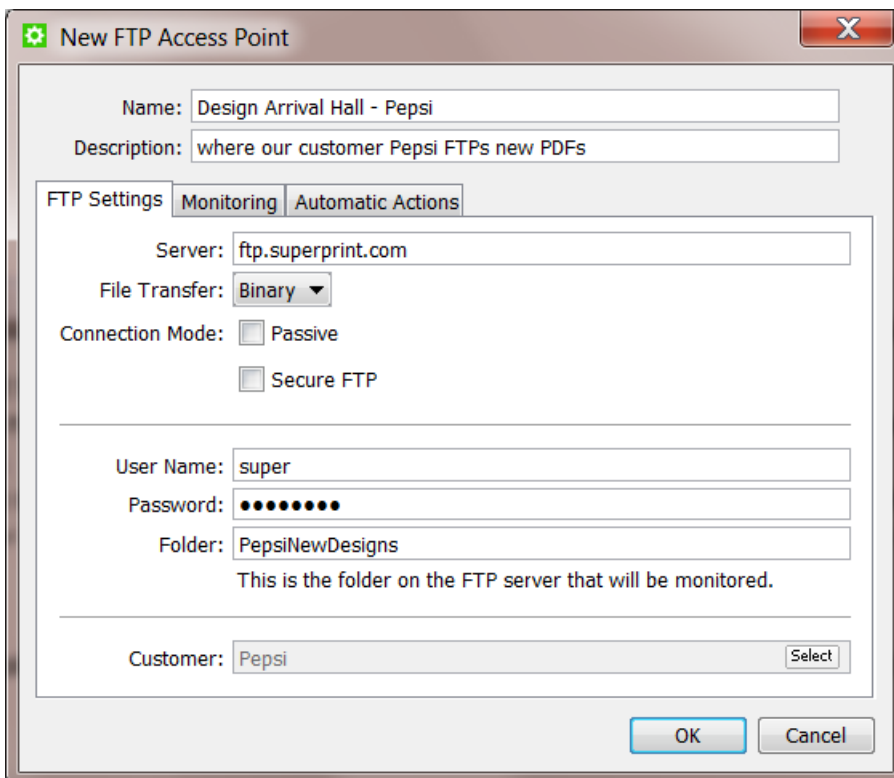
After uploading them, the Access Point deletes these files from the FTP site. However, the folders (and subfolders) are not deleted. This way, the folder structure of FTP site stays intact.

Note: The **Pilot** also offers a manual way to upload files from an FTP site. When you right-click on a folder or Job Folder, the function **Upload from FTP site** is offered. This **FTP Access Point** is an automated version of that feature.

Creating or modifying an FTP Access Point

Follow these steps to create or modify **FTP Access Points**:

1. In the **Automation Engine Pilot** go to the **Access Points** View. Alternatively, choose **Tools > Access Points** .
2. In the dedicated **Access Points** View, click on  . Alternatively, choose **File > New** .
3. In the resulting window, select **FTP** and click **OK**.
The **New FTP Access Point** window will open. Here is one with some typical settings already filled in:



New FTP Access Point

Name: Design Arrival Hall - Pepsi

Description: where our customer Pepsi FTPs new PDFs

FTP Settings | Monitoring | Automatic Actions

Server: ftp.superprint.com

File Transfer: Binary

Connection Mode: Passive

Secure FTP

User Name: super

Password: ●●●●●●●●

Folder: PepsiNewDesigns
This is the folder on the FTP server that will be monitored.

Customer: Pepsi

4. Enter a **Name** for the **Access Point**.
5. If you want, enter a **Description**.
This will help identifying your **Access Point** without having to read all its settings.
6. In the tab **FTP Settings**, define where and how you wish to connect.

- **Server:** Enter the name of the FTP server.
- **File Transfer:** Choose either **Binary** or **ASCII**.

Tip: **Binary** will always work, but if you have an ASCII file, the transfer speed will be higher when you choose **ASCII**.

- **Connection Mode:**
 - Select **Passive** if there is a firewall between the FTP server and Automation Engine that is blocking the FTP server from setting up connections to Automation Engine.

Tip: By default, the FTP transfer uses Active Connection Mode (where the FTP server opens the data connection). When **Passive** is selected, the system will use Passive Connection Mode instead (where the client initiates the connection).

- Select **Secure FTP** if you are connecting to a secure FTP server (over FTPS, not SSL or SSH). If you are experiencing problems using secure FTP, try connecting over regular FTP and ask your IT administrator to check the security settings of your FTP server.
 - **User Name** and **Password:** Enter a valid user name and password for this FTP server.
 - **Folder:** This is the folder on the FTP server that will be monitored. As mentioned in [Concept](#) on page 1240, also its sub folders will be monitored.
 - **Customer:** If you want to link a specific customer to this Access Point, use **Select** to pick it from the list. Then you can use it as a SmartName.
7. In the tab **Monitoring**, define the time interval.

- **Check Every.** Enter the time interval with which Automation Engine should check if any new files came in. Think of the time interval as the *maximum* time it takes for the new files to be picked up.
- **Schedule.** If you want this Access Point to be inactive during a part of the day, then activate this option and indicate the start and end time of that period.

Note: Such a temporary inactive state will not be shown in the general Access Points View. In that View, the column 'Active' shows its *general* status.

8. In the tab **Automatic Actions**, define what needs to happen.

- **Move Files to.**
 - **Folder.** Indicate where you want the files to be moved to. Browse to a folder in a **Container**. If you want, use **SmartNames**.
 - **Customer's Upload Folder.** Alternatively, you can choose to move these files to the **Customer's Upload Folder**. This is a folder you can define in the definition of the **Customer**.
- **Rename Files to.** If you want you can choose to rename these files too. To do this, enter the new file name.

Tip: Typically, SmartNames are used here to help indicate in their name *when* they were processed or *where* they came from.

- **Action.** Choose:
 - **None.** Choose this option when you *only* want to use this **Access Point** to *move* the files.
 - **Workflow.** Choose this option when you also want a workflow to start. Select the workflow ticket from the offered list. The new accessed files will be used as input files for this workflow.



Attention: Make sure your workflow **can** start with the type of files that will arrive.

Options:

- **Operator:** choose the operator that will be shown as operator for the workflow task.

Tip: if you do not want to assign one specific person, you could create an operator named "Access Points" or "Automatic" and assign that one here. Also remember that the Access Points Task Monitor shows you the tasks launched by the selected Access Point.

- **Priority:** choose the priority for this workflow: Low, Normal or High. The default priority is 'Normal'.
- **Process JDF.** Choose this option when this Access Point will be used to process incoming JDF files. For more info on JDF, read this section: [What about JDF?](#) on page 1230.

9.3.3 Mail Access Point

Concept

A **Mail Access Point** scans an e-mail account and extracts the e-mail attachments of new incoming e-mail. These files are uploaded to a folder in a **Container**. A **Mail Access Point** can also start a workflow on those files.


Here are 2 typical examples of how **Mail Access Point** SmartNames can help you do even more:

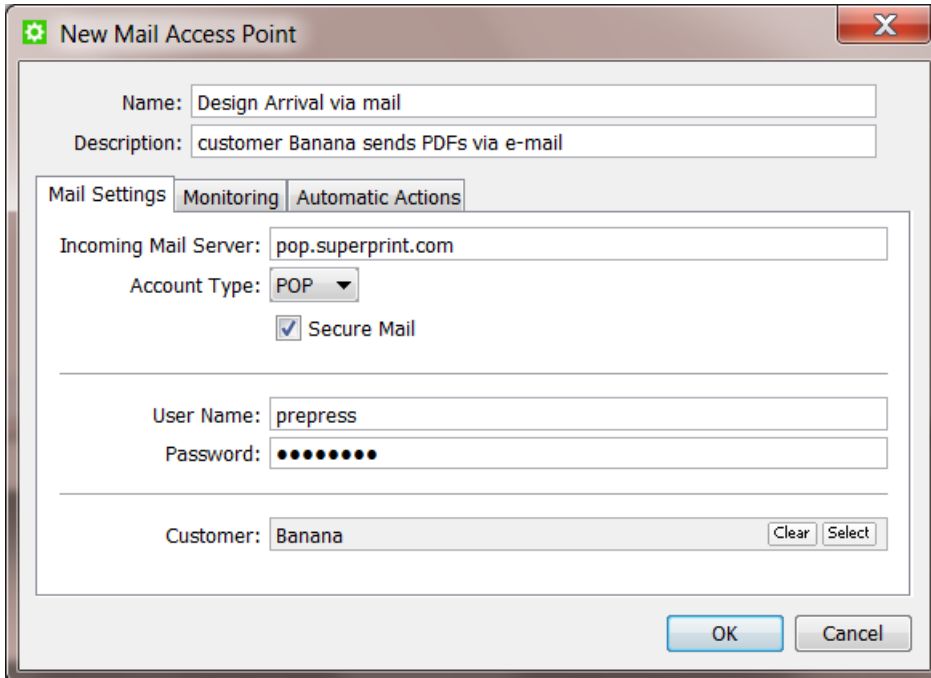
- You can agree with the e-mail sender to start the subject line with the Job number. This helps you direct these files to the right folder.
- The address of the e-mail sender can be used to later send him an e-mail back about the status of the files he sent.

Create or modifying a Mail Access Point

Follow these steps to create or modify **Mail Access Points**:

1. In the **Pilot** click on **Access Points** view. Alternatively, choose **Tools > Access Points** .

2. Click on  to create a new **Access Point**. Alternatively, choose **File > New** .
3. In the resulting window, select **Mail** and click **OK**.
The **New Mail Access Point** window will open up. Here is one with some typical settings already filled in:



4. Enter a **Name** for the **Access Point**.
5. If you want, enter a **Description**.
This will help identifying your **Access Point** without having to read all its settings.
6. Define the **Mail Settings**:
 - **Incoming Mail Server**: Enter the name of the mail server.
 - **Account Type**: Depending on the mailbox configuration, choose either **POP** or **IMAP**.
 - **Secure Mail**: Select this if the mailbox is a secure type.
 - **User Name** and **Password**: Enter the user name and password of the mail account that will receive the e-mails that you want to access.
 - **Customer**: If you want to link a specific customer to this Access Point, use **Select** to pick it from the list. You can then use it as a SmartName.
7. In the tab **Monitoring**, define the time interval.
 - **Check Every**. Enter the time interval with which Automation Engine should check if any new files came in. Think of the time interval as the *maximum* time it takes for the new files to be picked up.

- **Schedule.** If you want this Access Point to be inactive during a part of the day, then activate this option and indicate the start and end time of that period.

Note: Such a temporary inactive state will not be shown in the general Access Points View. In that View, the column 'Active' shows its *general* status.

8. In the tab **Automatic Actions**, define what needs to happen.

- **Move Files to.**
 - **Folder.** Indicate where you want the files to be moved to. Browse to a folder in a **Container**. If you want, use **SmartNames**.
 - **Customer's Upload Folder.** Alternatively, you can choose to move these files to the **Customer's Upload Folder**. This is a folder you can define in the definition of the **Customer**.
- **Rename Files to.** If you want you can choose to rename these files too. To do this, enter the new file name.

Tip: Typically, SmartNames are used here to help indicate in their name *when* they were processed or *where* they came from.

- **Action.** Choose:
 - **None.** Choose this option when you only want to use this **Access Point** to *move* the files.
 - **Workflow.** Choose this option when you want a workflow to start. Select the workflow ticket from the offered list. The new accessed files will be used as input files for this workflow.



Attention: Make sure your workflow **can** start with the type of files that will arrive.

Options:

- **Operator:** choose the operator that will be shown as operator for the workflow task.

Tip: if you do not want to assign one specific person, you could create an operator named "Access Points" or "Automatic" and assign that one here. Also remember that the Access Points Task Monitor shows you the tasks launched by the selected Access Point.
- **Priority:** choose the priority for this workflow: Low, Normal or High. The default priority is 'Normal'.
- **Process JDF.** Choose this option when this Access Point will be used to process incoming JDF files. For more info on JDF, read this section: [What about JDF?](#) on page 1230.

9.3.4 Database Access Point

Concept

A **Database Access Point** reads, at a regular time interval, information from an external database. That information is written into one or more XML files. The **Database Access Point** can also start a workflow.

A typical use case is checking an external system for a new status. That new status will be a reason for Automation Engine to start an action. For example: in that external database, when the status of a Job or Product is found on "preflight required", Automation Engine starts a preflight workflow. Or the output tasks will be started for all jobs that are found in a status "Jobs to expose".

Tip: The chapter [Examples](#) on page 1287 contains an example case of a **Database Access Point**.


To prevent that these statuses keep triggering these actions over and over again, the **Database Access Point** offers the choice to update or delete those (status) fields. For example: the status "Proof required" would be updated to "Proof started on Esko".

These use cases are often identical to those where an external system sends an XML file to Automation Engine. Here are some typical reasons to use a **Database Access Point** instead of such an XML transfer:

- The external system can not export the wanted data to XML. Or it would require an expensive intervention or an upgrade of the software version.
- Exporting an XML would be a manual action by a user of that external system. And all manual actions can be forgotten.

Create or modifying a Database Access Point

Follow these steps to create or modify a **Database Access Point**:

1. In the **Pilot** click on **Access Points** view. Alternatively, choose **Tools > Access Points** .
2. Click on  to create a new **Access Point**. Alternatively, choose **File > New** .
3. In the resulting window, select **Database** and click **OK**.
The **New Database Access Point** window will open up. Here is one with some typical settings already filled in:

New Database Access Point

Name:

Description:

Database Settings | Monitoring | Automatic Actions

External Database:

SQL Select Query:

After Processing: Delete the records
 Update the records

Column Name	New Value
Order_Status	Started in Esko

4. Enter a **Name** for the **Access Point**.

5. If you want, enter a **Description**.

This will help identifying your **Access Point** without having to read all its settings.

6. Define the **Database Settings**:

- **External Database:** Select the database from the list. The list shows the **External Databases** that you configured via **Tools > Configure** .
- Enter the desired **SQL Select Query**: The query will search for records or it will search specific values in one or more columns. An XML file will be generated for each record that matches the query.

Here is an example of a such an XML file. The query was looking for more info on Jobs with the status 'New'. This XML shows the resulting details of the Job 1213. See how the 'Select *' returned all the columns of the row of this Job 1213: name, operation etc...

```
?xml version="1.0" encoding="UTF-8?>
<design select="SELECT * FROM design WHERE status = "NEW""
producer="DBPoller-MIS-1" created="2013-04-24T11:26:14.210+02:00">
  <id>1213</id>
  <name>MyDesign</name>
  <operation>CreateJobCard</operation>
  <width>12</width>
  <height>500</height>
  <depth>50</depth>
  <status>NEW</status>
</design>
```

Note: As soon as you select a **Database Access Point**, the query will be checked if it is valid. If it is not valid, an error message will be shown.

- **After Processing:** Now choose what should happen after the results from the query were processed.
 - **Delete the records:** This will delete the records that the query selected in the external database. It will make them empty.
 - **Update the records:** This will update the records that the query selected in the external database with a new value. Click on **Add** to update the desired parameters. Fill in the **Column Name** and the **New Value** in that database column and click OK.

Note: If you want to update a record that contains a date and/or time, use the format **yyyyMMddHHmmss** for the new value of the column or use **now** to insert the date and time of the moment that the record is updated.

7. In the tab **Monitoring**, define the time interval.

- **Check Every.** Enter the time interval with which Automation Engine should execute this database query.
- **Schedule.** If you want this Access Point to be inactive during a part of the day, then activate this option and indicate the start and end time of that period.

Note: Such a temporary inactive state will not be shown in the general Access Points View. In that View, the column 'Active' shows its *general* status.

8. In the tab **Automatic Actions**, define what needs to happen.

- **Move Files to.** Indicate where you want the resulting XML file(s) to be written. Browse to a folder in a **Container**. If you want, use **SmartNames**.
- **Rename Files to.** If you want you can choose to rename these files too. To do this, enter the new file name.

Tip: Typically, SmartNames are used here to help indicate in their name *when* they were processed or *where* they came from.

- **Action.** Choose:
 - **None.** Choose this option when you want no further action.
 - **Workflow.** Choose this option when you also want a workflow to start. Select the workflow ticket from the list.



Attention: Make sure your workflow finds its input files. These can be the XML files from the query but that is not required.

Options:

- **Operator:** choose the operator that will be shown as operator for the workflow task.

Tip: if you do not want to assign one specific person, you could create an operator named "Access Points" or "Automatic" and assign that one here. Also remember that the Access Points Task Monitor shows you the tasks launched by the selected Access Point.

- **Priority:** choose the priority for this workflow: Low, Normal or High. The default priority is 'Normal'.

9.3.5 Web Service Access Point

Concept

When a system has a web service, other software applications can send commands or queries to that system using HTTP requests to the URL of the web service.

Note: Some users prefer this way of direct communication rather than exchanging (XML) files on hot folders.

Automation Engine also has a web server. A **Web Service Access Point** is a way for external systems within the intranet to start workflows on Automation Engine by sending a command to the web service of Automation Engine.

Tip: In [this section on differences between Access Points](#), you can read how a **Web Service Access Point** works somewhat different than the other types of Access Points. One important difference is that the **Web Service Access Point** does not use the concept of time intervals. It is the external system that fully decides **when** to send a command to Automation Engine.

The HTTP requests can be both HTTP POST requests and HTTP GET requests. If the HTTP POST request also 'posts' data files, then they will be used as input files for the workflow that will be started.

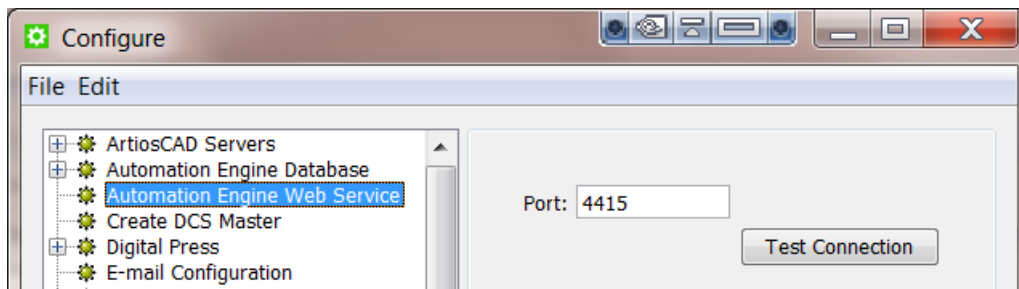
Note: When using a HTTP GET request, parameters of the web service call are encoded in the URL using HTTP query strings. When using HTTP POST requests, the parameters are encoded in the content part of the HTTP request.

Checking the Automation Engine Web Service

The web service of Automation Engine will be running automatically on the Automation Engine server.

Note: You actually also use this web service when you access the Automation Engine **Client Apps** in a browser.

You can check the status of the Automation Engine Web Service by entering this in a browser: `http://<name of your server>:<port>/ws/`. The port number is the one as defined in the **Configure** tool > **Automation Engine Web Service**. By default this port number is 4415.





For example: if your Automation Engine server is named `AESERVER1`, then the URL to enter in your browser is `http://aeserver1:4415/ws/`

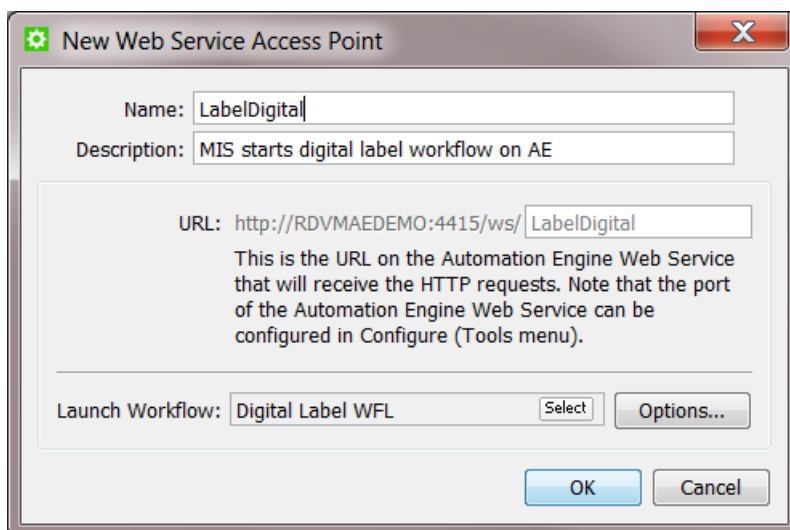
The web page that then appears offers more guidelines on formatting more detailed HTTP commands.

Creating a Web Service Access Point

Note: Setting up a **Web Service Access Point** is simple. The complexity lies in the content of the HTTP requests that are sent to it (see next chapters).

Follow these steps to create or modify a **Web Service Access Point**:

1. In the **Pilot** click on **Access Points** view. Alternatively, choose **Tools > Access Points**.
2.  Click on  to create a new **Access Point**. Alternatively, choose **File > New**.
3. In the resulting window, select **Web Service** and click **OK**.
The **New Web Service Access Point** window will open up. Here is one with some typical settings already filled in:



4. Enter a **Name** for the **Access Point**.
This name will automatically be used to complete the URL for this Access Point.
5. If you want, enter a **Description**.

This will help identifying your **Access Point** without having to read all its settings.

6. See the resulting URL. This is where the external system needs to send its command to launch the workflow defined below.
7. Select the **Workflow** that this Access Point will start.

Options:

- **Operator:** choose the operator that will be shown as operator for the workflow task.

Tip: If you do not want to assign one specific person, you could create an operator named "Access Points" or "Automatic" and assign that one here. Also remember that the Access Points Task Monitor shows you the tasks launched by the selected Access Point.

- **Priority:** choose the priority for this workflow: Low, Normal or High. The default priority is 'Normal'.

HTTP POST Requests

Launching a workflow using an HTTP POST request

The HTTP POST requests must meet the following requirements:

- The value of the **Content-Type** header of the HTTP POST request should be, in most cases, **multipart/form-data**.
- HTTP request parts can be added to the HTTP POST requests to attach data files and/or to add parameters (also see further).
- For every HTTP request part that is added to an HTTP POST request, a **Content-Disposition** header should be present.
 - For each data file, the (UTF-8 encoded) file name of the data file should be specified in the corresponding **Content-Disposition** header.
 - For each parameter, the name of the parameter should be specified in the corresponding **Content-Disposition** header.
- The encoding of the HTTP POST requests should be **base64** or no encoding (for example **binary**, **8bit**,...).

For example:

```
POST /ws/myFavoriteWorkflow HTTP/1.1
Host: aeserver01:4415
Content-Type:                                multipart/form-data;
  boundary=-----146677a63ef
Content-Length: <length of request body in octets (8-bit bytes)>
...
-----146677a63ef
Content-Disposition: form-data; name="wfparam1"

2mm
-----146677a63ef
Content-Disposition: form-data; name="wfparam2"

4mm
-----146677a63ef
```

```
Content-Disposition:          form-data;          name="inputfile1";
  filename="inputfile1.pdf"
Content-Transfer-Encoding: binary

<binary contents of inputfile1.pdf>
-----146677a63ef
Content-Disposition:          form-data;          name="inputfile2";
  filename="inputfile2.pdf"
Content-Transfer-Encoding: binary

<binary contents of inputfile2.pdf>
```

Attaching data files to be used as input files for the workflow that is launched

You can attach data files to be used as input files for the workflow that is launched by adding them as HTTP request parts to the HTTP POST request. For each data file, the (UTF-8 encoded) file name of the data file should be specified in the corresponding **Content-Disposition** header.

For example:

```
POST /ws/myFavoriteWorkflow HTTP/1.1
Host: aeserver01:4415
Content-Type:                               multipart/form-data;
  boundary=-----146677a63ef
Content-Length: <length of request body in octets (8-bit bytes)>
...

-----146677a63ef
Content-Disposition:          form-data;          name="inputfile1";
  filename="inputfile1.pdf"
Content-Transfer-Encoding: binary

<binary contents of inputfile1.pdf>
...
```

When the workflow is launched, the data files are handled in one of the following ways:

- When the workflow is launched in a specific Job, the data files will be moved to the Job Folder.
- When the workflow is not launched in a specific Job and the workflow is successfully finished, the data files will be deleted.
- When the workflow is not launched in a specific Job and the workflow did not finish successfully, the data files will be kept in this shared folder:

```
\\<name of your server>\AutomationEngineTmpFolder
```

When no data files are attached to the HTTP POST request, the **Web Service Access Point** will generate an XML file containing some information about the HTTP GET request. This XML file will then be used as input file for the workflow to be launched and will be handled in the same way as attached data files would be handled.

Launching a workflow in a specific Job

You can specify the Job in which the workflow should be launched by adding parameters as HTTP request parts to the HTTP POST request. The name of each parameter should be specified in the corresponding **Content-Disposition** header.

The Job in which the workflow should be launched can be specified using the **Job ID**, the **Job Name**, the **Order ID** or using both the **Order ID** and the **Sub Order ID**. The parameters you need to add as

HTTP request parts to the HTTP POST request are, respectively, **jobid**, **jobname**, **orderid** or **orderid** and **suborderid** . The values for these parameters should be UTF-8 encoded.

For example:

```
POST /ws/myFavoriteWorkflow HTTP/1.1
Host: aeserver01:4415
Content-Type:                               multipart/form-data;
  boundary=-----146677a63ef
Content-Length: <length of request body in octets (8-bit bytes)>
...

-----146677a63ef
Content-Disposition: form-data; name="orderid"

2014-100254
...
```

Specifying the values of the workflow parameters of the workflow that is launched

You can also set the values of the workflow parameters of the workflow by adding parameters as HTTP request parts to the HTTP POST request. These parameters should have the same names as the names of the workflow parameter(s) which value you want to set. The values for these parameters should be UTF-8 encoded . For each parameter, the name of the parameter should also be specified in the corresponding **Content-Disposition** header.

For example:

```
POST /ws/myFavoriteWorkflow HTTP/1.1
Host: aeserver01:4415
Content-Type:                               multipart/form-data;
  boundary=-----146677a63ef
Content-Length: <length of request body in octets (8-bit bytes)>
...

-----146677a63ef
Content-Disposition: form-data; name="wfparam1"

2mm
...
```

HTTP GET Requests

Launching a workflow using an HTTP GET request

When using HTTP GET requests to launch workflows on a **Web Service Access Point**, no data files can be attached.

Every time an HTTP GET request is received on the URL of the **Web Service Access Point**, it will generate an XML file containing some information about the HTTP GET request. This XML file can then be used as input file for the workflow to be launched.

- When the workflow is launched in a specific Job, this XML file will be moved to the Job Folder.
- When the workflow is not launched in a specific Job and the workflow is successfully finished, this XML file will be deleted.

- When the workflow is not launched in a specific Job and the workflow did not finish successfully, this XML file will be kept in this shared folder:

```
\\<name of your server>\AutomationEngineTmpFolder
```

Launching a workflow in a specific Job

You can specify the Job in which the workflow should be launched by adding parameters to the query string of the HTTP URL. The Job in which the workflow should be launched can be specified using the **Job ID**, the **Job Name**, the **Order ID** or using both the **Order ID** and the **Sub Order ID**. The parameters you need to add to the query string of the HTTP URL are, respectively, **jobid**, **jobname**, **orderid** or **orderid** and **suborderid**. The values for these parameters should be UTF-8 encoded.

For example:

```
http://aeserver01:4415/ws/myFavoriteWorkflow?jobid=ee6b08ca-eeef-4cb7-b2e1-254c20d82546
http://aeserver01:4415/ws/myFavoriteWorkflow?jobname=Libelle-2014-week12
http://aeserver01:4415/ws/myFavoriteWorkflow?orderid=2014-100254
http://aeserver01:4415/ws/myFavoriteWorkflow?orderid=2014-100255&suborderid=001
```

Specifying the values of the workflow parameters of the workflow that is launched

You can also set values for workflow parameters by adding parameters to the query string of the HTTP URL. These parameters should have the same names as the names of the workflow parameter(s) which value you want to set. The values for these parameters should be UTF-8 encoded .

For example:

```
http://aeserver01:4415/ws/myFavoriteWorkflow?wfparam1=2mm
http://aeserver01:4415/ws/myFavoriteWorkflow?wfparam1=2mm&wfparam2=4mm
http://aeserver01:4415/ws/myFavoriteWorkflow?jobname=Libelle-2014-week12&wfparam1=2mm
```

Error Handling

When using this **Web Service Access Point**, the corresponding HTTP responses should always be checked because they can contain error messages.

In case the workflow was successfully launched, the HTTP response will contain the task ID of the workflow task that was launched.

For example:

```
<?xml version="1.0" encoding="UTF-8"?>
<message>
  <response>taskid=dd145a15-49bb-4625-a0da-6d3aee663ae3</response>
</message>
```

In case the workflow could not be launched successfully and/or an error occurred, the HTTP response will contain one or more error messages.

For example:

```
<?xml version="1.0" encoding="UTF-8"?>
<message>
  <error>
```

```
<code>-110</code>
  <msg>Problem loading ticket: swft/Web Service Access Point Test
WF</msg>
  </error>
</message>
```

9.4 Data Transforming Tasks

We introduced the concept of these tasks in the chapter [Automation Engine Mapper tasks Transforming Data](#) on page 1233.



Attention: These types of tasks require good knowledge of SmartNames.



Attention: The **XPath Builder** makes it a lot easier to extract parts from XML files. This tool is documented in a [separate chapter on SmartNames](#).

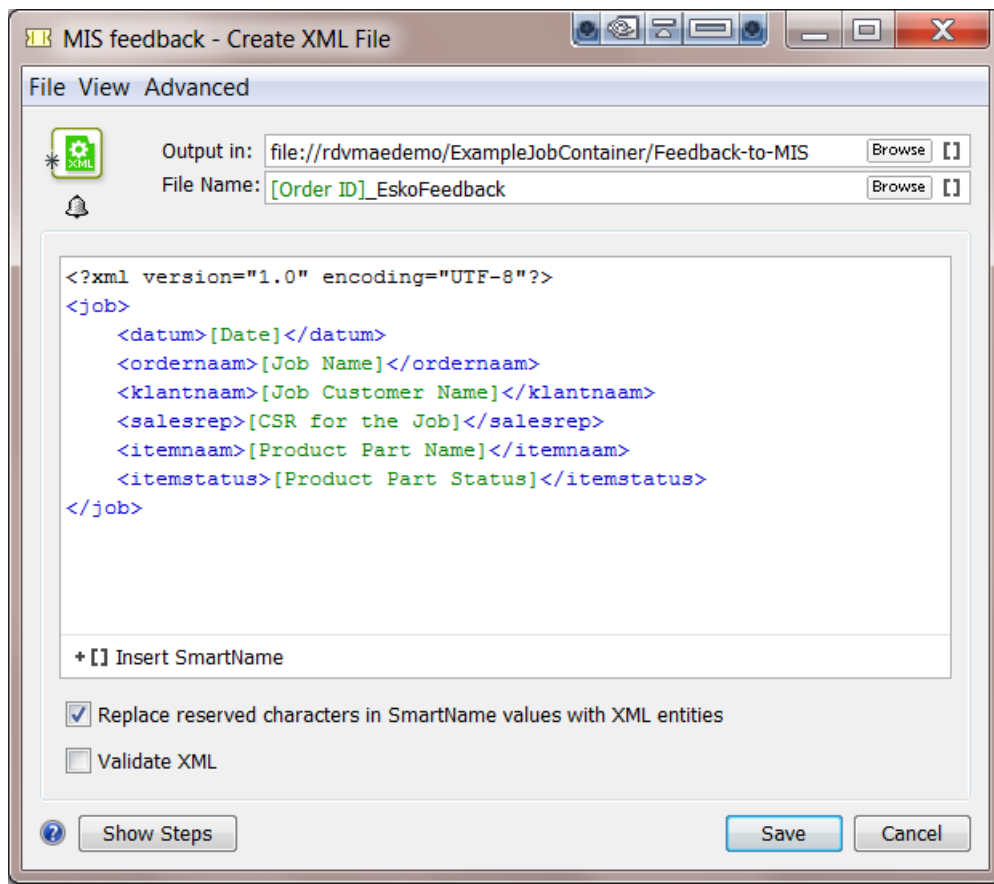
9.4.1 Create XML File task

The **Create XML File** task is typically used to export value from SmartNames to an external system.

Important: This task does not use its task input file to start building the XML from. The content of the XML that you will create is completely defined inside the task window.

Defining the content of the XML

The **Create XML File** task offers a blank canvas where you need to define all the content of the XML that you want to create. In below example you see a combination of self written XML code (blue) and some SmartNames (green). Click on + [] **insert SmartName** to pick a SmartName from the list. Alternatively, right-click and choose a SmartName from the offered categories.



Note: The XML file has to start with an XML declaration. This line indicates the file encoding to the reading application. To add such a line, you can simply right-click and choose **Insert XML Declaration** to insert this industry default one: `<?xml version="1.0" encoding="UTF-8"?>`

Note: Any links to externally defined XML schema (for example a reference to a DTD or XSD) may influence performance. Preferably remove such links from the XML file.

- **Replace reserved characters in SmartName values with XML entities:** It is possible that the SmartName returns characters that are not allowed in XML. With this option, you choose to map these special characters mapped into these 'XML entities'. Below list shows these mappings:

Reserved characters in result value	XML Entities
"	&quot;
'	&apos;
&	&amp;
>	&gt;
<	&lt;

- **Task fails when output XML is not a valid XML file :** Enable this option to have Automation Engine check the validity of the XML. If this option is on, the task will fail if the XML is not valid. The validation checks the XML formatting and the XML tag endings.

Note: This task does not check XML schemas such as XSD, DTD, etc.

9.4.2 Split XML Task

Concept

Sometimes you want to split XML files to make them more easy to use Automation Engine workflows. These XML files typically come from an external system. The **Split XML task** needs 1 XML file as input and will write 1 or more XML files as output. Some typical examples:

- Every morning your MIS system sends you a list with new Jobs and their details. To use that XML for the **Create Job task**, you need to split it into 1 XML per Job.
- Your MIS system sends an XML per customer order, but your workflows are set up per production item of an order. In this case, the **Split XML task** helps you to generate an XML file per production item. This makes it easier to automate your workflow. This example is elaborated below.

Tip: You can also use this task to split an XML that was created by Automation Engine itself. For example to make that XML more manageable for an external system.

Supported input files

The input of this task is XML. Files with a different extension than **.xml** will not be recognised or processed.

Note: Exception: JDF files are also allowed as input. The resulting output will be XML files. These XML files will no longer be valid as JDF file.

How to do you decide what to split?

How you split the task's input file is defined by XPath expressions. The built-in **XPath Builder** will help you to do write those expressions. See further.

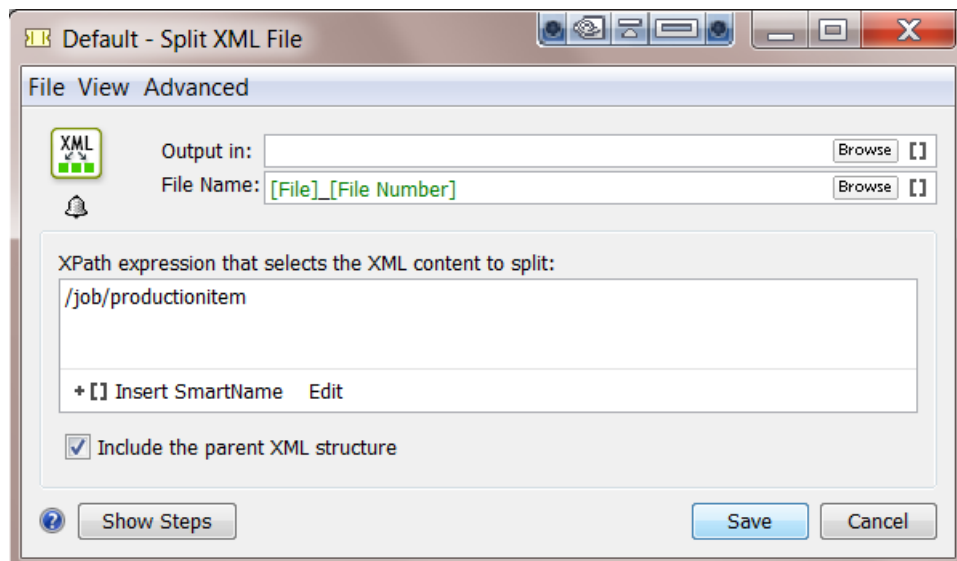
Example and Options

Example: The following code is grouping 2 production items in 1 XML file:

```
<?xml version="1.0"?>
<job>
  <commoncomponent>Z</commoncomponent>
  <commoncomponent content="Y" />
  <productionitem>
    <subelement content="A" />
    <subelement>1</subelement>
  </productionitem>
  <productionitem>
    <subelement content="B" />
    <subelement>2</subelement>
  </productionitem>
</job>
```

To split each production item into a separate XML file, set this expression in the XPath canvas:

/job/productionitem.



This will generate these 2 files:

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<job>
  <commoncomponent>Z</commoncomponent>
  <commoncomponent content="Y" />
  <productionitem>
    <subelement content="A" />
    <subelement>1</subelement>
  </productionitem>
</job>
```

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<job>
  <commoncomponent>Z</commoncomponent>
  <commoncomponent content="Y" />
  <productionitem>
    <subelement content="B" />
    <subelement>2</subelement>
  </productionitem>
</job>
```

Note: The SmartName [File Number] is available for this task only. It automatically generates a unique file name.

Include the parent XML structure

This option will preserve all XML content that the XPath expression does not select.

Note: In case of a conditional expression this will also include siblings that do not match the condition.

If you only want to keep the parts that you selected with the XPath expression, then deselect this option. Using the previous example, this will result in following XML files:

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<productionitem>
  <subelement content="A" />
```

```
<subelement>1</subelement>
</productionitem>
```

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
<productionitem>
  <subelement content="B" />
  <subelement>2</subelement>
</productionitem>
```

When an input file contains a **namespace declaration** in the parent XML structure ('xmlns'), then that namespace declaration will be added as an attribute of the top level element. For example:

```
<?xml version="1.0" encoding="UTF-8"?>
<job xmlns:esko="http://www.esko.com/myowns/">
  <commoncomponent>Z</commoncomponent>
  <commoncomponent content="Y" />
  <esko:productionitem>
    <esko:subelement content="A" />
    <esko:subelement>1</esko:subelement>
  </esko:productionitem>
  <esko:productionitem>
    <esko:subelement content="B" />
    <esko:subelement>2</esko:subelement>
  </esko:productionitem>
</job>
```

Then the split result with namespace declaration on the top level element will be:

```
<?xml version="1.0" encoding="UTF-8" standalone="no"?>
<esko:productionitem xmlns:esko="http://www.esko.com/myowns/">
  <esko:subelement content="A" />
  <esko:subelement>1</esko:subelement>
</esko:productionitem>
```

```
<?xml version="1.0" encoding="UTF-8" standalone="no"?>
<esko:productionitem xmlns:esko="http://www.esko.com/myowns/">
  <esko:subelement content="B" />
  <esko:subelement>2</esko:subelement>
</esko:productionitem>
```

Encoding

If the input XML file contains a header that specifies an encoding (for example encoding="ISO-8859-3"), the task will write the output files in that encoding.

If no header or encoding is specified, the resulting files will have **UTF-8** encoding by default.

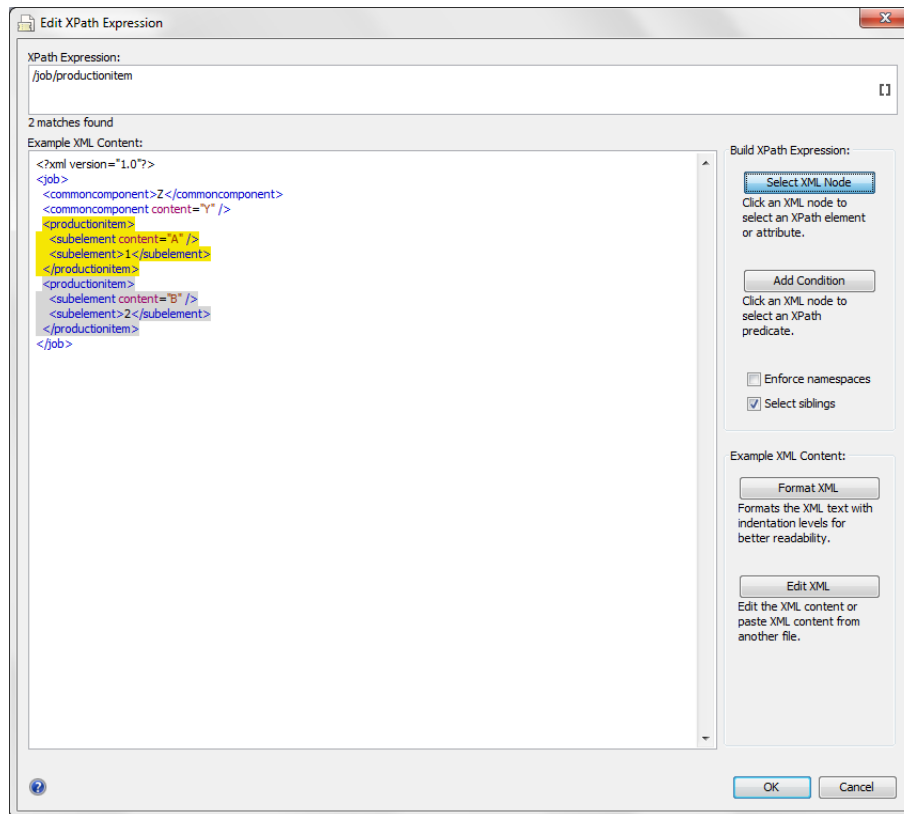
XML Validation

The task will generate well formed XML, but there is no support for schema or DTD validation. References to DTD's for validation will be removed.

Creating XPath Expressions

Usually you will not define the XPath expression by typing it in the task window yourself. Use the **Edit** button to open the **XPath Builder**. This tool will help you create your own correct XPath expression by clicking the variable elements. By selecting parts in the window with **Example XML Content**, you will see the resulting XPath expression automatically appear in the top part of this tool.

Important: When the **Example XML content** window is empty, this means that you forgot to select an XML input file before opening the **Split XML task**.



Selecting the option **Select siblings** allows you to automatically select the right level.

In above example, the user selected the level 'productionitem'. The selection is shown in yellow. The siblings are also automatically selected and are shown in grey. The correct **XPath expression** that is needed to split that level of elements will automatically appear in the top part of this tool.

You can easily build complex expressions with this tool, such as:

- selecting elements conditionally.
- selecting an element from a specific 'namespace'.

Find more information on the **XPath Builder** in *The XPath Builder* on page 954 (chapter SmartNames).

9.4.3 Join XML Files Task

Concept

The **Join XML** task joins the content of its input XML files. In case you do not want the full content of all these input files, there are options to determine what parts you want or from what root level down that you want their content.

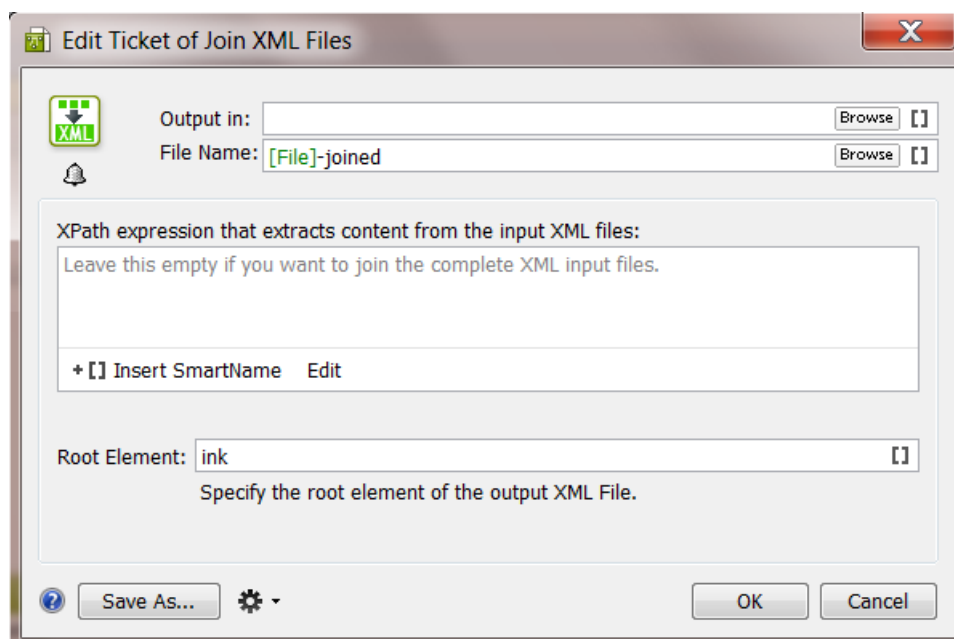
Some typical examples:

- The information you need is spread over several source XML files, maybe even coming from different systems. You need (parts of) them combined in 1 XML file.
- You want to change the content of an XML. After splitting it you can re-join specific parts.

Note: The **Map Data** task can also be used to *join* XML files, but it offers much more features and options than this **Join XML** task.

Example and Options

The **Join XML task** joins multiple input XML files into 1 output XML file. The default option of this task is to take the full content of the input files and join everything together into 1 XML file.



Name of the output XML file

The default name of the output XML file is set to **[File]-joined**. **[File]** is here the name of the first input file. You can of course change this setting.

Xpath expression that extracts content from the input XML files

Tip: In the previous topic about the **Split XML** task, we introduced **Xpath expressions** and also introduced the **XPath Builder** tool.

This canvas is where you can define the content of the output XML. If you do not want the full content of all your input files, use XPath expressions to define what you want to extract from them. Per input file, all nodes that comply with the given XPath expression are imported into the output XML file.

Click on **+ []** to insert **SmartNames** to help build your XPath expression.

Click on to open the **XPath Builder**. For more info on **XPath Builder** please see [The XPath Builder](#) on page 954 (chapter 'SmartNames').

If this canvas is left empty, and if you didn't specify a **Root Element**, then all the nodes under the root node are considered (XPath == /*/*).

Root Element in your output file

Each input XML file has its own **root element**. You can define here what you want as **root element** in the output XML file. See an example where this root element was set to 'ink':

```
<?xml version="1.0" encoding="UTF-8"?>
- <ink>
  <name>BananaYellow</name>
  <type>Designer</type>
  <book>FruitColors</book>
</ink>
```

If this field is left empty, then the tag name of the root of the first input file will be used.

Example

You have these 2 XML files:

- One XML file lists ink attributes of a (first) ink of a PDF:

```
<?xml version="1.0" encoding="UTF-8"?>
- <job>
  <name>banana.pdf</name>
  - <ink>
    <name>803</name>
    <type>pantone</type>
    <book>pms1000c</book>
    <angle>7</angle>
    <ruling>92</ruling>
    <dotshape>round</dotshape>
  </ink>
</job>
```

- Another XML file contains information on the ink coverage of that file:

```
<?xml version="1.0" encoding="UTF-8"?>
- <job>
  <name>banana.pdf</name>
  <customer>FruitCo</customer>
  - <ink>
    - <inkcov>
      <pct>27.12</pct>
      <mm2>6512</mm2>
    </inkcov>
  </ink>
</job>
```

You now want to join these 2 files into 1 XML file with **all** the ink attributes together.

- This is the result if you use **no options**:

```

<?xml version="1.0" encoding="UTF-8"?>
- <job>
  <name>banana.pdf</name>
  - <ink>
    <name>803</name>
    <type>pantone</type>
    <book>pms1000c</book>
    <angle>7</angle>
    <ruling>92</ruling>
    <dotshape>round</dotshape>
  </ink>
  <name>banana.pdf</name>
  <customer>FruitCo</customer>
  - <ink>
    - <inkcov>
      <pct>27.12</pct>
      <mm2>6512</mm2>
    </inkcov>
  </ink>
</job>

```

- This is the result when you set the option **Root Element** to **PrintItem**:

```

<?xml version="1.0" encoding="UTF-8"?>
- <PrintItem>
  <name>banana.pdf</name>
  - <ink>
    <name>803</name>
    <type>pantone</type>
    <book>pms1000c</book>
    <angle>7</angle>
    <ruling>92</ruling>
    <dotshape>round</dotshape>
  </ink>
  <name>banana.pdf</name>
  <customer>FruitCo</customer>
  - <ink>
    - <inkcov>
      <pct>27.12</pct>
      <mm2>6512</mm2>
    </inkcov>
  </ink>
</PrintItem>

```

- This is the result when you **also** used the **Edit** button to add the **XPath Expression** **/Job/ink**

```

<?xml version="1.0" encoding="UTF-8"?>
- <PrintItem>
  - <ink>
    <name>803</name>
    <type>pantone</type>
    <book>pms1000c</book>
    <angle>7</angle>
    <ruling>92</ruling>
    <dotshape>round</dotshape>
  </ink>
  - <ink>
    - <inkcov>
      <pct>27.12</pct>
      <mm2>6512</mm2>
    </inkcov>
  </ink>
</PrintItem>

```

Because the Xpath extracted only the content of <ink>, the pdf file name and the customer name are not kept in this joined file.

Note: To further change the structure, order or content, you can use the **Map Data** task.

9.4.4 Map Data task

Concept

The **Map Data** task is a powerful tool that can be used to

- convert CSV files into XML and vice versa.
- transform the content of a CSV or XML file: extract, join, rename or re-format parts.

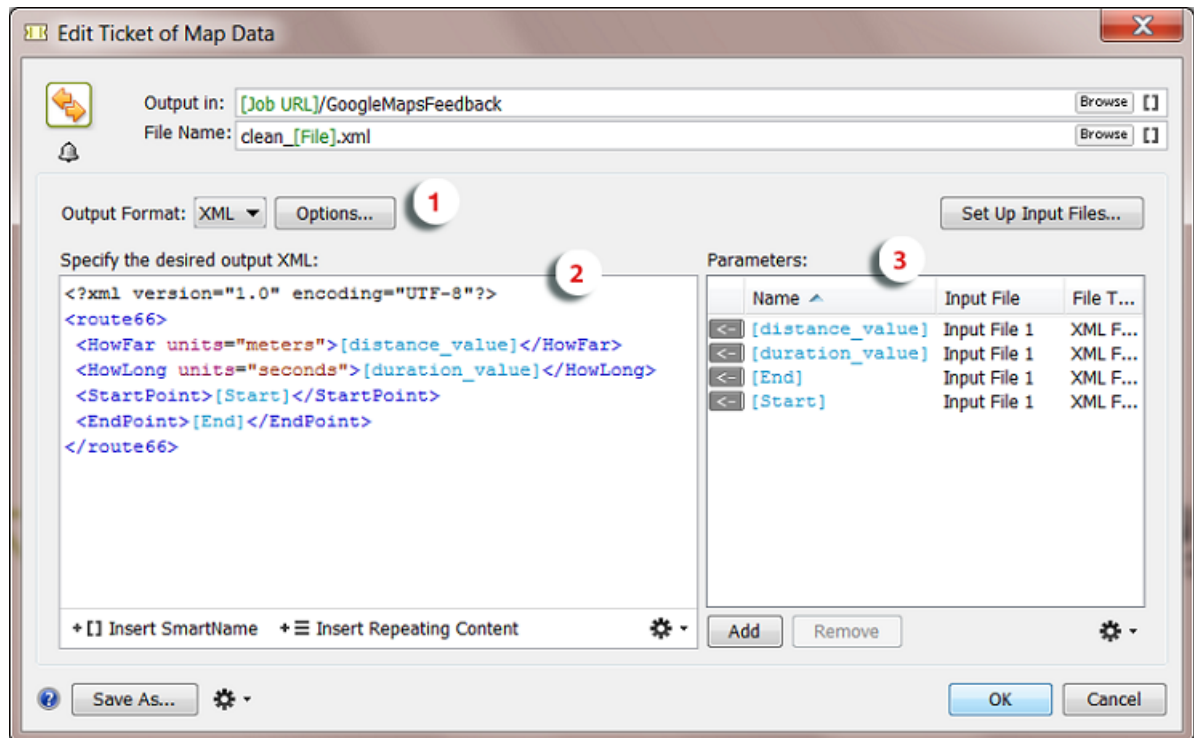
Note: A CSV or comma separated file is a structured text file. In such files commas or some other character are used as field separators. CSV can be generated by exporting an XLS file in Microsoft Excel.

Some example use cases:

- After interacting with an external system, you need to simplify or re-write the XML files that you received.
- You need to combine information from different sources into 1 file, but also reformat it (this is more than what the **Join XML** task can do). For example getting job specification from an incoming XML file and getting ink information from the metadata of an uploaded PDF (XMP). You want to combine those into one XML that can be used for the **Job Creation** task. To do this, the XMP ink information should also be transformed to the XML format that is needed for **Job Creation** task.
- You need to transform an XML that you received before it can be used by the **Create CAD Sheet** task.
- You receive an XML file with all possible sheet layouts. Now you need to create a list of these layouts using a custom text format (for example plain text with line breaks, CSV or XML).

Tip: The chapter [Examples](#) on page 1287 contains many examples that use this task.

Main Tools in the Task Panel



The **Map Data** task panel consists of three main parts:

1. Option to choose the Output Format.
2. Canvas to specify the content of the output file of this task.
3. List of Parameters that you want to read from the task's input files.

Choose the Output Format

- **XML:** Click on **Options** to define these output format options:
 - **Replace reserved characters in parameter values with XML entities:** It is possible that the parameters from the input files return characters that are not allowed in XML. With this option, you choose to map these special characters to 'XML entities'. Below list shows these mappings:

Reserved characters in result value	XML Entities
"	"
'	'
&	&
>	>
<	<

- **Task fails when output XML is not a valid XML file:** Enable this option to have Automation Engine check the validity of the XML. If this option is on, the task will fail if the XML is not valid. The validation checks the XML formatting and the XML tag endings.

Note: This task does not check XML schemas such as XSD, DTD, etc.

- **CSV:** Click on **Options** to define this Output Format option:
 - **Text Encoding:** Choose what type of text encoding that the system that will read this CSV file prefers. The default is UTF-8.

Canvas to Specify the Content of the Output file

The canvas on the left there is where you can

- insert XML or CSV data from another file. This is the typical way to insert your example output file, after which you can start replacing its absolute values with SmartNames or Parameters.
- insert Parameters from your input files (from the list on the right side of the panel)
- insert SmartNames
- insert Xpath expressions (the standard language to extract specific parts of an XML)
- ...

See more in [Specifying the Output File](#) on page 1271.

Tip: Right-click in the canvas to get a list of content to insert.

A List of Parameters from your Input Files

In this field, you build up a list of parameters from your input files. You can then insert these parameters easily in the canvas on the left, similar to how you insert SmartNames. These parameters can come from one or more input files.

At the moment you set up the Map Data task, it is good practice to use *example* input files that contain the same parameters as the files that will be the *real* input files of the Map Data task once it is started in a workflow. It is also a good idea to start from a sample output file (see further).

See more in [Setting Up \(Example\) Input Files](#) on page 1266 and [Adding Parameters](#) on page 1269.

Main Workflow in Using the Map Data Task

Before we go in more detail explaining all the possibilities of this powerful task, it is good practice to stick to following main workflow:

- Step 1: Get an example of the output file you need to create. Analyse its structure.
- Step 2: Get (an) example input file(s). Analyse its structure.
- Step 3: Create the parameters from the input file(s).
- Step 4: Specify the output file. Use your example output file and the parameters that you made from the input file(s).
- Step 5: Test the resulting ticket in a workflow, using real input files.

Setting Up (Example) Input Files



Attention: An example input file is a file that you use while you set up the **Map Data** ticket. Be aware that this example file is often not your real input file once the task is running!

Important: The **Map Data** task allows to create parameters *without* the help of example input files but this will be much more complex. We advise to create the parameters based on example input files because it will help you with correct spelling, see the larger context and give you immediate feedback.

2 ways to select an Input File

- In the **Pilot**, select one or more input files **before** you open the **Map Data** ticket. These files will show up as input files.
- Open the **Map Data** ticket and click on **Set Up Input Files....**

Setting up an **Input File** is explained in more detail in the next sections of this document.

Working with Multiple Task Input files

By default a **Map Data** task will start as soon as it gets one input file. The task will not wait for a list of input files to be complete. But you can easily solve that.

To make the **Map Data** task work on multiple input files, you need to make a workflow where you have a **Data Collector** task before your **Map Data** task.



The **Data Collector** task will wait for outputs of previous tasks and hand them over as one group to the next task.

How to Control the Order of Multiple Input Files

If your **Map Data** task has more than one input file, it is very important to control the order of those input files. What you want to avoid is

- parameters that do not find their value because they look in the wrong input file.
- parameters that find a value but one that comes from the wrong input file.

Some examples that illustrate the importance of parameters looking in the correct input file:

- Your **Map Data** ticket is part of a workflow that creates a new **Job**. Does the parameter [date] or [time] come from the input file from the MIS or from the metadata-XML from the PDF that the customer sent you? The difference can result in a conflict about your due date of that **Job**!
- Your parameter [print technique] selects the field in row 2, column 5 of a CSV file. If this parameter searches this field in the wrong input file, then your search can return "Rio de Janeiro" instead of "Digital Flexo"!

To control the order of your input files, we advise to first use the **Data Collector** task, followed by a **Sort** task and only then followed by the **Map Data** task.



The **Data Collector** task will wait for outputs of previous tasks and hand them over as one group to the next task.

The **Sort** task enables you to sort that group. You can base the order on file name, time or SmartNames.

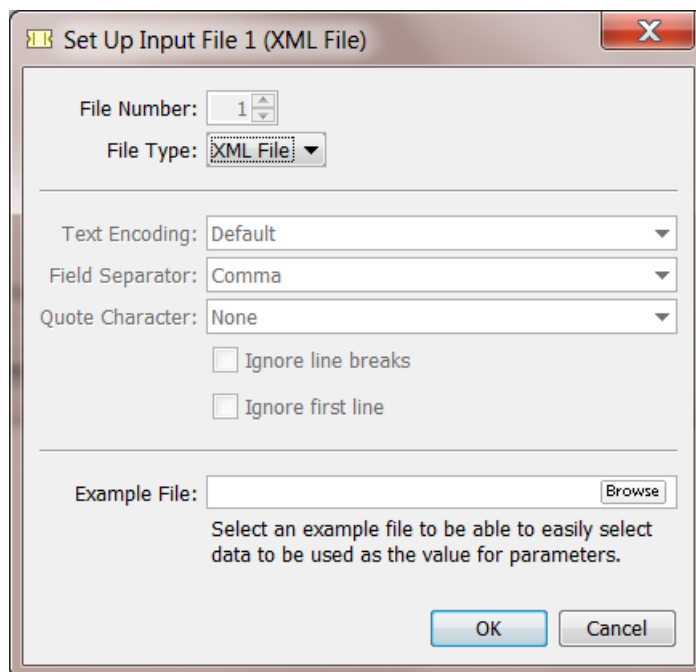
Setting up (Example) Input Files - Details

1. Click on the **Set up Input Files....**

A panel opens that shows your current list of Input Files, their name and order in the list, their type (XML or CSV) and if they link to an example file or not.

2. To add or modify one, click on **Add** or double click an existing Input file definition.

The **Set Up** panel opens.



3. Choose the **File Type** of the desired Input File: **XML File** or **CSV File**.

4. In case you chose **CSV File**, you can change the following CSV options:

- **Text Encoding:** Indicate the **Text Encoding** of the CSV file. The default encoding is Unicode UTF-8.
- **Field Separator:** Indicate which character is used to separate the fields in the CSV. Check your CSV in a simple text editor to detect it.
- **Quote Character:** Indicate if the value in the CSV contains quotes that you do not want to keep. For example: In the CSV the value is written as "25000" but you want to extract only the 25000.
- **Ignore line breaks:** Check this option if the CSV only contains one record (column).
- **Ignore first line:** Check this option if the first line of the CSV contains the names of the columns.

Note: These CSV options have a great impact on the way the CSV input files are handled. A selected value field (using the field number and the line number of the field) can be the wrong one if the CSV options are not set correctly.

5. If you want, select an example file to be able to easily select data to be used as the value for parameters.

Removing an Input file

In the **Set Up Input files** panel, the **Remove** button will remove the selected input file.



When there are no parameters created yet that use that input file, the input file item will be removed. You will not be asked to confirm.

But when the input file is in use, you will see a warning mentioning this use. You will be asked to confirm before deleting the input file item together with all the parameters or 'repeating content' that require that input file.

Note: A similar warning and effect happens when you change the *File Type* of the Input file (from XML to CSV or vice versa).

Adding Parameters

There are basically 3 ways to create new parameters:

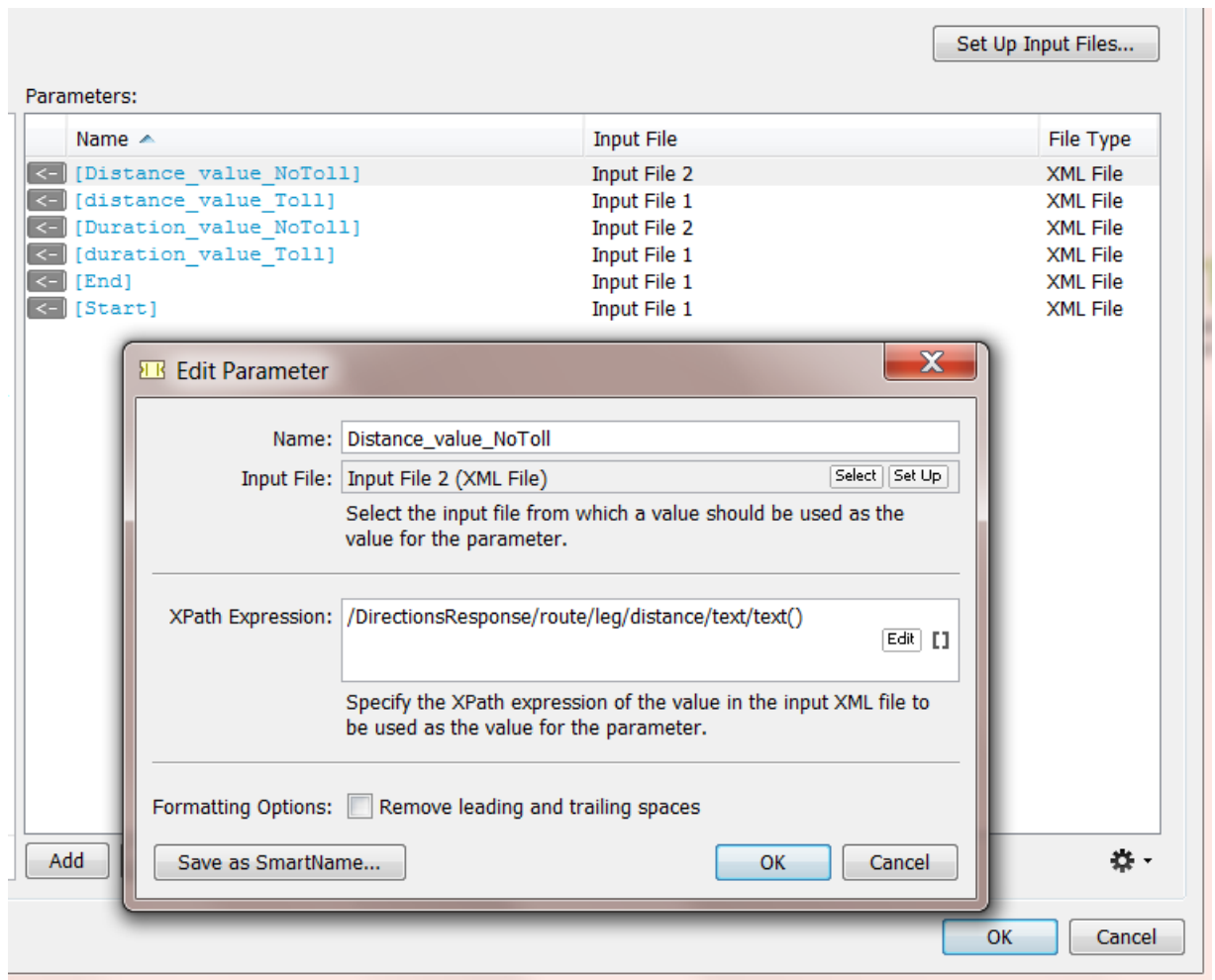
- creating them one by one, using the **Add** button.
- loading **multiple** parameters from a file, using the  button.
- duplicating an existing parameter, using the  button.

Adding Parameters

Once you specified the output format and set up the (example) input files, you can start adding parameters that represent the values that you wish to insert from the task's input files into the output file.

To add parameters, proceed as follows:


1. Click **Add** underneath the **Parameters** list.
The **Add Parameter** dialog prompts.
2. Enter a **Name** for the new parameter.
3. Select the **Input File** that contains the value for the parameter.
4. Specify where the value for the parameter is located in the selected **Input File**:
 - In case you selected an XML **Input File**, specify the **XPath Expression** of the value in the input file to be used as the value for the parameter. If you selected an example XML file for the XML input file, you can easily create the **XPath Expression** using the **Edit** button. This will open the **XPath Builder**. For more information on **XPath Builder**, see [The XPath Builder](#) on page 954 (chapter 'SmartNames').



- In case you selected a CSV **Input File**, specify the **Field** number and the **Line** number of the field in the input CSV file to be used as the value for the parameter. If you selected an example CSV file for the CSV input file, you can select the desired field using the **Select Field** button.
- 5. Separator.** If the XPath expression returns multiple elements, decide what character you want to be used as separator.
 - 6. Formatting Options.** If you suspect that the chosen value might have leading and/or trailing spaces, select **Remove leading and trailing spaces**.
 - 7.** If you want to save the created parameter as a SmartName, then click **Save as SmartName...** In the prompted dialog, enter a comprehensive **Name** and **Description** and choose whether or not you want this SmartName to be **Private**.
 - 8.** Click **OK** to add the parameter to the **Parameters** list.


The created Parameter is now in the **Parameters** list. You can now insert the created parameter in the output XML or CSV (see [Specifying the Output File](#) on page 1271).

Note:

- To edit an existing parameter, select it in the **Parameter** list and double-click it.
- To delete an existing parameter, select the parameter and click **Remove**. You will get a warning when you try to delete a parameter that is in use (in the canvas on the left).
- To delete all unused parameters, click on the action button  underneath the **Parameters** list and choose **Delete Unused Parameters**. You will be asked to confirm.
- To add Parameters from an example input file, see the next section.

Loading Multiple Parameters from a File

Instead of creating parameters one by one, you can choose to load multiple parameters from a file. Those that you would not need are easy to remove from the list anyway. To do so, proceed as follows:


1. Underneath the **Parameters** list, click on the **Action**  button.
2. Choose **Load Parameters from File** in the prompted menu.
The **Load Parameters from File** dialog will prompt.
3. **Select** the desired **Input File** from which you want to load the parameters.
4. If the selected **Input File** is an XML file, proceed as follows:
 - a) Specify the **XPath Expression** to load the XML parameters from the **Input File**. You can click **Edit** to open the Xpath Builder to help you build the desired **XPath Expression**.
 - b) Click **OK**.
All parameters from the selected XML Element will be loaded, including child elements and all attributes
5. If the selected **Input File** is a CSV file, proceed as follows:
 - a) If the **First line contains parameter names**, select the similarly named option and the values from the first line will also become parameters.
 - b) Click **OK**
CSV Parameters will now be added for every field of every record in the **Input File**.

Note: About names of parameters:

- When one or more parameters that are loaded from the example input file have the same name (because the XML element name, the XML attribute name or the CSV field name in the example XML/CSV file is the same), their names will be made unique by auto-numbering them.
- When one or more parameters with the same name already exist, a warning dialog will pop up asking you if you want to replace them or not. If you choose to replace them, this will overwrite their current contents.

Specifying the Output File

The canvas on the left of a **Map Data** ticket is where you define what your resulting XML or CSV output file will contain.

There are many types of smart content that you can add to this canvas. And you can even type in some content yourself. Some of these types of content are listed via the  button. You will also see a complete list of possibilities when you right-click inside the canvas.

Note: Most of these are sensitive to the position of your cursor at the moment you right-clicked. They will not be offered when they are not valid at that specific insertion point.



Caution: There is no 'Undo' function while editing in this canvas.

Tip: Find a good [example](#) in the chapter [Examples](#) on page 1287 . It shows how 2 XMLs from the application Google Maps are mapped into one smaller output XML. That output XML so contains only specific content to be picked up as workflow parameters.

Here are all the methods and data types that you can enter:

Cut-Copy-Paste

These standard functions work on all types of data you have entered here.

Clear Contents

This will clear the whole canvas.

Insert SmartName

Your output XML can also contain SmartNames. This way you can add information to the XML that was maybe not present in the input files but important extra data for the system that will read the output file.

Note: You can also use the dedicated **+[]** button.

Paste from File...

The complete content of the selected file will be added here. This will be possible with all content that is in a format that can be output to XML or CSV.

Important: Use this function to paste your example output file.

Insert XML Declaration

It is a good practice to have an 'XML declaration' at the start of your XML file. This line indicates the file encoding to the reading application. This feature will here insert the industry default one:

```
<?xml version="1.0" encoding="UTF-8"?>
```

Insert XML Element

Type in the name of the **element** that you want to insert. An opening and closing statement of that **element** will be inserted. For example:

```
<Distance></Distance>
```

Often, you will continue by inserting a **Parameter** in between those two.

Insert XML Attribute

Type in the name and value of the **attribute** that you want to insert. For example:



```
units="miles"
```


For example: see how this **attribute** was inserted in the above **element**:

```
<Distance units="miles"></Distance>
```

Insert Parameter

Choose the parameter from the list. Alternatively, select the parameter in the parameter list on the right side and

- Click the  to insert that parameter at the current cursor position in the canvas.
- Drag & drop the  to the right position in the canvas.

For example: the parameter [GoogleMapsDist] was inserted in above example:


```
<Distance units="miles">[GoogleMapsDist]</Distance>
```

Insert New Parameter

This is a shortcut that combines the adding of a new parameter with its immediate placement in the canvas. After you inserted this new one, it will also appear in the parameter list on the right.

Insert Repeating Content (XML statement 'For Each')

For example: for each job that is described in the input file, you want to pick up all its ink specifications (name, type, angle, ruling). This content will be repeated in the output file for each found job.

After making sure your insertion pointer is at the correct place in the canvas, right-click or use the dedicated  button. A dialog appears and in the canvas a visual "repeating block" appears in pink.

Now start typing or add XML elements in the repeating block.

Mind that you insert the parameter at a valid insertion point.




Note: When you add a parameter while your insertion point is *inside* a pink block of repeating content, the dialog will mention that that parameter will only serve to be inserted in repeating content.

Choose to insert a new parameter and click **OK**. The parameter is inserted and also listed in pink in the parameter table.

Specify the desired output XML:

```
<?xml version="1.0" encoding="UTF-8"?>
<general>
  <File>
    <InkName>[Plate]</InkName>
    <InkType>[InkBook]</InkType>
    <PrintAngle>[Angle]</PrintAngle>
  </File>
  REPEATING
</general>
```

Parameters:

	Name ^	Input File	File Type
	[Angle]	Input File 1 - Created Ord...	XML File
	[InkBook]	Input File 1 - Created Ord...	XML File
	[Plate]	Input File 1 - Created Ord...	XML File

Note: When the insertion pointer is out of the repeating block, parameters that were used inside the repeating block cannot be inserted outside the repeating block. Their insert buttons are then disabled (shown in grey).

It is also possible to insert Repeating Content inside another Repeating Content block. This is called **Nested Repeating Content**. See an example:

Specify the desired output XML:

```
<?xml version="1.0" encoding="UTF-8"?>
<general>
  <File>
    <Ink>[Plate]</Ink>
    <InkAngle>[Angle]</InkAngle>
    <InkRuling>[LPI]</InkRuling>
    REPEATING
    <CustomerName>[Cust]</CustomerName>
  </File>
  REPEATING
</general>
```

Parameters:

Name ^	Input File	File Type
<- [Angle]	Input File 1 - Created Ord...	XML File
<- [Cust]	Input File 1 - Created Ord...	XML File
<- [LPI]	Input File 1 - Created Ord...	XML File
<- [Plate]	Input File 1 - Created Ord...	XML File

9.5 Tasks Interacting with other Systems

9.5.1 Interact with Database task

Concept

Tip: The tasks that interact directly with external systems were introduced in [Automation Engine tasks Interacting directly with the External System](#) on page 1232. You there also read how this task compares to the **Database Access Point**.

The **Interact with Database** task allows you to communicate directly with an external database. It allows both getting information from that database (read) and sending information to that external database (write).

As mentioned in the chapter [Integration Concepts](#) on page 1230, a typical use for this task is to feed information back to external systems by using this task in a workflow. For example, you can set up this task to update a record in a table of your MIS when your workflow has reached a certain stage. For example setting the status record to 'Proof Ready'.



Warning: This task may not be used to interact with Automation Engine databases. This is not supported by Esko. This can damage your server configuration.



Warning: See the important remark in the task panel. As mentioned earlier in this document, using SQL statements to change records of an external database can potentially harm that database. If you are using this technique then this means that you have a database specialist who takes all liabilities for these SQL queries.

Tip: The chapter [Examples](#) on page 1287 contains 2 examples that contact an external database.

Set Up and Options

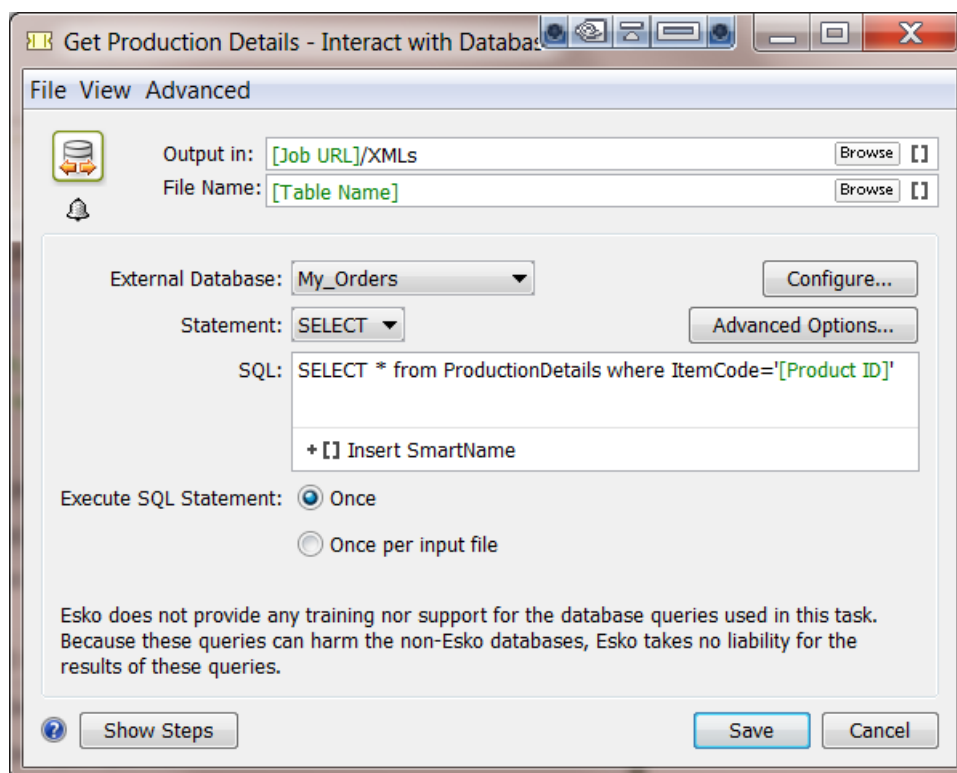
File Name

When the SQL query is a 'select' statement (see further), the task will dump the found records in an XML file. The default ticket offers the SmartName **[Table Name]** as name for that output XML file. In below screenshot, this name would then be 'ProductionDetails'.

Note: The SmartName **[Table Name]** is only offered in a context where it makes sense.



Caution: Not all SQL queries will resolve a name of a database table. Do not use the SmartName **[Table Name]** when it is possible that no table name is found.



External Database

Select a database from the list of configured databases. Click **Configure...** to access the configure tool directly. Read about configuring **External Databases** in the chapter [External Databases](#) on page 637 in the **Reference Guide**.

Important: Before linking Automation Engine with an external database, please check your license agreements with the database manufacturer.

Statement

Choose the type of SQL statement that you want to use: **SELECT** or **OTHER**.

- **SELECT**

When using the **Select** statement, the output of the task will be an XML file.

See an example of such a resulting XML. Notice that the SQL statement is also mentioned inside.

```
<?xml version="1.0" encoding="UTF-8" ?>
- <rows created="2013-05-23T10:41:32.100+02:00" producer="sqlexec" select="select * from ProductionDetails where Factory='FRANCE' startrec='1'">
- <row>
  <c n="Factory">FRANCE</c>
  <c n="Machine">S802</c>
  <c n="ItemCode">FR-04742</c>
  <c n="Run">March13</c>
  <c n="QC">Passed</c>
  <c isNull="1" n="Comments" />
</row>
</rows>
```

Advanced Options...: You can avoid creating an XML file that is too big by limiting the number of records with the option **Maximum number of records per file**. The task will then generate as many files as needed.

- **Other**

In this mode, you can execute any other SQL statement such as INSERT, UPDATE etc. However, the task will in this case not generate an output file.

Note: When the **Statement** type is set to **Other**, you cannot execute a **Select** statement.

In this example the status of the Job is changed in the orders database:

The screenshot shows a configuration window for an external database task. The 'External Database' is set to 'My_Orders'. The 'Statement' type is set to 'Other'. The SQL statement is: UPDATE Orders set Status='Proofready' where OrderNR=[Order ID]. There is a button for 'Advanced Options...' and a checkbox for 'Insert SmartName'.

Execute SQL Statement

Decide if you want the SQL statement to be executed **Once** or **Once per input file**.

For example: You have multiple input files for this task: banana.pdf, lemon.pdf and mango.pdf. When your SQL statement uses the SmartName [File] (= the task's input file), then the option **Once per input file** will make the task run **for each input file**. With the option **Once**, the task would only run for the banana.pdf (the first input file).

Tip: Starting a task once and having it run on multiple input files is much lighter for the software than starting the task multiple times with each time only one input file.

9.5.2 Interact with Web Service task

Concept

Tip: The tasks that interact directly with external systems were introduced in the chapter [Integration Concepts](#) on page 1230.

The **Interact with Web Service** task uses HTTP requests to communicate with a web service of an external system. Both **read** ('GET') and **write** ('POST') commands are possible. The result of this task is typically an XML file.

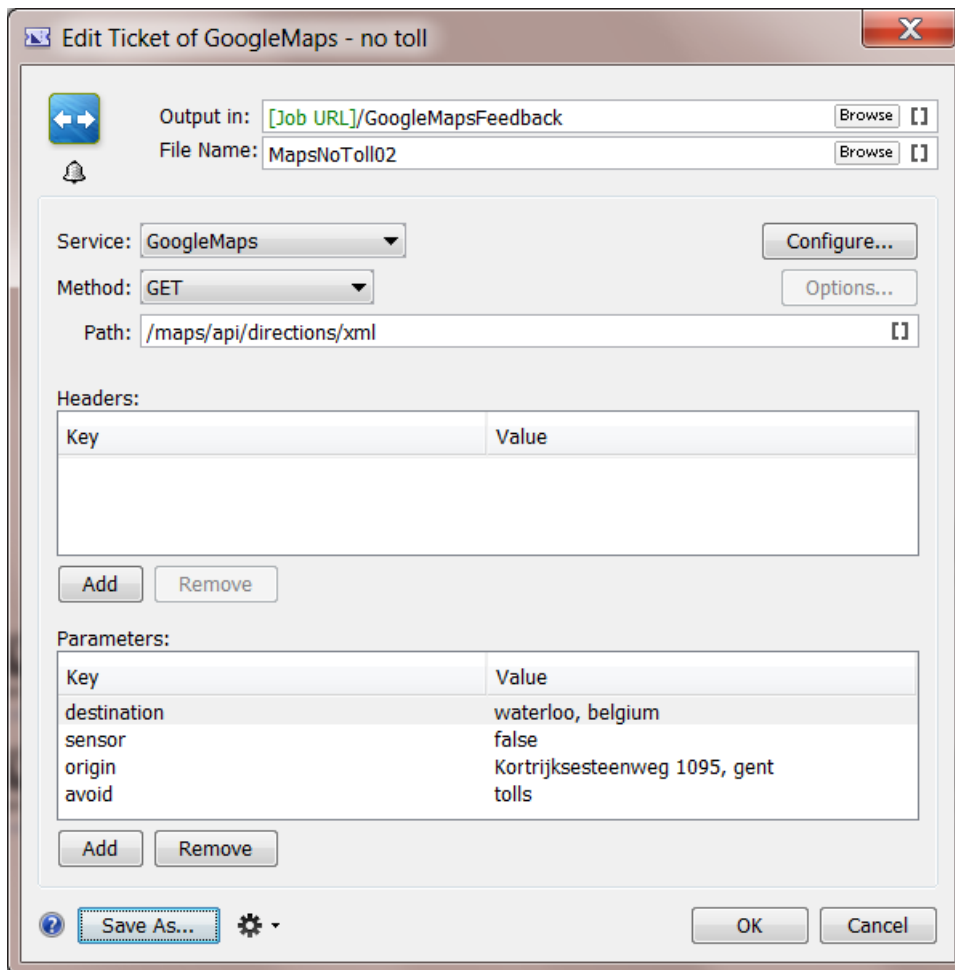
When an external system is running a web service, you will be able to access it via HTTP (HyperText Transfer Protocol). However, to do this, you also need to learn **what** you can communicate to that system, what language you need to speak to it. Some applications will indeed publish their API (Application Programming Interface). And it is that API documentation that will tell you what you can enter in the **Interact with Web Service** ticket.

Tip: The chapter [Examples](#) on page 1287 contains an example where a web service is contacted.

Difference with the Web Service Access Point.

- A **Web Service Access Point** is a way for an external system to speak to the web service of Automation Engine. This **Interact with Web Service** task is **the reverse**: it is Automation Engine that interacts with the web service of an external system.
- An **Access Point** is used to **start** a workflow. This **Interact with Web Service** task can be inserted at any step of the workflow.

Set Up and Options



Output folder and File Name

Define here the path and name of your output file. The type of output file is defined by the API. We advise to choose XML.

When the **Method** is **GET**, this file will contain what the external system returns you. When the **Method** is any other type, the output file will only contain a status of the interaction (for example 'OK').

Service

Choose the web service from the list. Click **Configure...** to access the **Configure** tool where you define such external Web Services. For more information see the chapter [Configure](#) on page 628 in the **Reference Guide**.

Method

There are various types of HTTP methods that you can send to a web service. Choose the method from the list:

- **GET**. This most frequently used method will retrieve data from that external system. Consider the URL as the question and the output file as your answer.

For example: ask Google Maps the distance from your company to your customer.

- **POST**. This method creates a new object on the external system.

- This method is commonly used to post actual data, to upload a file to the external system. Therefore, click on **Options...** and choose to **Attach input files**. All input files are added to a single HTTP request and SmartNames are resolved in the context of the first input file. In case each input file must be sent separately, then add a **Data Splitter** task before the **Interact with Web Service** task in the workflow. In that case SmartNames will be resolved for each file.

Note: The parameters are sent as "application/x-www-form-urlencoded" data. When XML files are attached, the data is sent as "multipart/form-data".

- The POST method also enables to send a SOAP request. The exact text of the SOAP request needs to be put in an XML file and you need to add a single parameter "SOAPAction" with the needed value.

Note: the **Create XML** task can be used to embed SmartNames inside the SOAP request.

Note: Find an example on [Esko Knowledge Base](#).

- **PUT.** This method requests the external system to store data or information under the given (existing) URL.
- **DELETE.** This method requests the external system to delete the information corresponding to the given URL.
- **Select SmartName....** You can even use a SmartName to define the method.

Note: **PUT** and **DELETE** are only used for 'RESTful' web services. 'REST' is the main architecture style of HTTP. An alternative is 'SOAP'.

Path

This path will be documented in the API of the external system. APIs often present several paths that you can address.

In the above example of a ticket to access Google Maps, the information will be gotten from the 'directions' part of the API. The path ends with XML which here means that we choose to get an XML file back (and not a JSON file for example).

Headers

The API documentation will instruct you if you need to use HTTP headers. To do this, add their key and value in the **Headers** list.

Parameters

The API documentation will instruct you what parameters you can use. To do this, add their key and value in the **Parameters** list. This is where the actual 'question' will become more clear.

In the above example, you can see how the ticket asks Google Maps a distance between the origin and destination address. Other parameters could be more optional, like in this case the option to only calculate distances for routes that avoid toll roads.

Tip: Just Google "Google maps API" or click [here](#) to learn how Google Maps is a nice example of how an external system publishes its API.

Tip: The chapter [Examples](#) on page 1287 contains an example that uses the Google Maps API.

9.6 Tasks often used in Integration Workflows

9.6.1 Create Job task

Concept, Workflow and FAQs

The **Create Job** task creates an Automation Engine **Job** based on an XML input file containing the job parameters. Typically this input XML file comes from your business system. It is inside this system that the job-orders are decided and where many of its prepress production parameters are entered first. Typically, a **Folder Access Point** receives this XML file and then starts a workflow containing this **Create Job** task.

Tip: Read more about the concept of integrating with external systems in [Introduction](#) on page 1228.

Tip: The chapter [Examples](#) on page 1287 contains many examples where this task is used.

It is possible that the XML file that feeds this **Create Job** task does not come directly from the business system. Maybe you first have to combine it with another input file that contains other job parameters. Such other parameters could for example arrive as metadata inside a PDF. The data formatting tasks **Split XML**, **Join XML** and **Map Data** offer all tools to combine these parameters into 1 XML file.

Should the XML be in a specific format?

No, but it should be a valid XML of course. You will set up **SmartNames** or **XPath expressions** that map the vocabulary of that business system to the terminology required in Automation Engine. For example your business system uses the term 'Order Number' to describe the name of a job. You can then make a SmartName that maps 'Order Number' to 'Job Name'.

Can one XML file create multiple Jobs?

If your XML contains the description of multiple jobs, then you first need to split that XML into multiple XML files that each describe one job. To do that you will set up a workflow with the **Split XML** task in front of this **Create Job** task. Also use the **Data Splitter** task to make sure that the **Create Job** task is started for each incoming XML file.



Can the input file be a CSV file?

If you want to create a Job from a CSV file, then use the **Map Data** task to convert it to an XML file.

What if my business system can not export an XML file?

When your business system is not capable to export a job description in XML format, you can consider the alternative where Automation Engine asks your business system if there is a new job to be created. Read about the various possible techniques in the chapter [Access Points](#) on page 1236.

Can the task find its parameters *outside* the task's input file?

Yes. The input file of this task does not have to be the XML file that delivers all the parameters. It can even be a PDF. The parameters could also be looked for in other files than the task's input file. Some examples of such special setups:

- A **Create Job** ticket with a PDF as input file and where the SmartNames do SQL queries to an external database.
- A **Create Job** ticket with SmartNames that find their values in **workflow parameters**.
- A **Create Job** ticket with (SmartNames that are) Xpath expressions to some centrally stored XML that the business system writes there every night at 3 AM.

The **Create Job** task can get the values for its parameters from different sources. It can for example

- find the main job order specifications from the business system's XML file (name, ID, customer, due date)
- AND get the some ink information from the metadata of a PDF
- AND get a custom parameter 'Press' from a workflow parameter `[wfp.PressName]`.

What if the Job already exists? Can I use this task to *update* Jobs?

Yes you can also use this task to update existing Jobs. If Automation Engine detects that the Job already exists, it will update the Job with any new or changed values.

This task will function as an *Update Job* task

- when it sees that the [Job Name] already exists
- OR when it sees that the *combination* of [Job Order ID] and [Job Sub Order ID] already exists.

At the end of this chapter, you can find a sample XML that can be used to create jobs from. Remember that the format of the XML is free. This sample is not a template, it is just one example.

We will now document the settings in the different tab sheets.

Important: To set up a **Create Job** ticket, you need advanced knowledge of SmartNames.

Job

File View Advanced

JOB

Job Customer Inks Parameters Bar Codes Template Advanced

Order ID: [FromMIS_XML_OrderID] []

Sub Order ID: [FromMIS_XML_SubOrderID] []

Name: [FromMIS_XML_JobName]_[FromMIS_XML_OrderID] []

Description: Xpath:/Orders/Job/LongName []

DueDate: []

Project ID: [FromMIS_XML_WebProject] []

Category: Digital [] ▾

Quantity: [] ▾

Country: [] ▾

Status: [FromMIS_XML_Status] [] ▾

Job Folder: [Smart Job Location] Browse []

Show Steps Save Cancel

For most of the fields in this task, you can choose to set a SmartName or a combination of SmartNames.

You can also directly write XPath expressions that will extract values from an XML file. The syntax for doing this is "XPath:" followed by a valid XPath expression for the input file.



Caution: If you are not an expert in building XPath expressions, we advise to not type them here but to create them via XPath SmartNames.

- **Order ID.** This field must have a value. You can not leave it empty.
- **Sub Order ID**
- **Name.** In the screen shot example, we combined 2 SmartNames with some absolute text (_).
- **Description.** In the screen shot example, we wrote an XPath expression.
- **Due Date.** The format needs to be compliant with ISO 8601 (YYYY-MM-DDThh:mm:ssTZD) . For example: 2014-07-28T12:00:00+01:00
- **Project ID.**

Tip: Some users use this field to define the name of the project on Esko WebCenter that corresponds with this Job..

- **Category.** For this field, you can also choose a category from the list.

Tip: The categories in the list are all those that were already used in a Job.

- **Custom Fields.** The custom fields that appear here are the ones that you defined in **Configure > Jobs > Job Setup** (maximum 6). Alternatively to using a SmartName or XPath, choose a value from the list.
- **Job Folder.** The system needs to know where to create the actual folder for this Job.

Note: Beware that you do need to define a *complete* network path here, not just a folder name.

You have these choices:

- Use the SmartName [Smart Job Location] to use this setting as defined in **Configure > Jobs > Smart Job Location** .
- Click **Browse** to browse to the desired job location
- Click **Browse** to browse to a folder in a container, and then use SmartNames to define a sub folder.

Note: If you want to make the path smarter by adding the name of the customer, then you best set that up as a **Smart Job Location**.

Customer

Using the same techniques as described for the tab 'Job', you here define the customer details for the Job you are creating.

- **Customer ID.** You can alternatively select the ID from the customer list panel.
- **Customer Name.**
- **Customer Description.**
- **Customer's Job Reference.** This is the name that your customer uses for this Job.
- **Customer Service Representative.** You can alternatively select the CSR from the list.
- **E-mail.**

Inks

Setting up the inks for the new Job can only be done if the input file is an XML file and if the ink properties are stored in attributes of XML elements.

In the example below the XML input file contains an element "Farben" (German for "inks"). This is an array of "Farbe" elements, each describing an ink.

```
- <Farben>
<Farbe Farbbuch="Process" Farbtyp="Normal" Farbnamen="Cyan" Winkel="15" Lineatur="150" Punktform="CS4" Druckmethode="Offset" />
<Farbe Farbbuch="Process" Farbtyp="Normal" Farbnamen="Magenta" Winkel="75" Lineatur="150" Punktform="CS4" Druckmethode="Offset" />
<Farbe Farbbuch="Process" Farbtyp="Normal" Farbnamen="Yellow" Winkel="0" Lineatur="150" Punktform="CS4" Druckmethode="Offset" />
<Farbe Farbbuch="Process" Farbtyp="Normal" Farbnamen="Black" Winkel="45" Lineatur="150" Punktform="CS4" Druckmethode="Offset" />
<Farbe Farbbuch="PANTONE+ Solid Coated" Farbtyp="Normal" Farbnamen="PANTONE Warm Red C" Winkel="45" Lineatur="150" Punktform="R" Dr
<Farbe Farbbuch="ClassicColors" Farbtyp="Opaque" Farbnamen="White" Winkel="45" Lineatur="120" Punktform="R" Druckmethode="Offset" /
<Farbe Farbbuch="PANTONE+ Pastels & Neons Coated" Farbtyp="Transparent" Farbnamen="PANTONE 9284 C" Winkel="45" Lineatur="150" Punkt
<Farbe Farbbuch="Designer" Farbtyp="Technical" Farbnamen="Die" />
</Farben>
```

The **Create Job** task will create an ink in the new Job for each of these elements in the XML. The table in the dialog defines how to map XML attributes of each element to an ink property.

Job Customer **Inks** Parameters Bar Codes Template

Load ink information from an XML element:

XML element name

XPath expression

XML Element Name:

Specify the name of the XML element that contains all ink information.

For each job ink parameter, specify the corresponding XML element that is used in the XML file:

Ink Parameter	XML Element Name
Ink name	Farbnamen
Ink type	Farbtyp
Ink book	Farbbuch
Angle	Winkel
Ruling	Lineatur
Dot shape	Punktform
Printing method	Druckmethode

Edit

- **Load Ink Information from an XML element.** Enable this and then choose how you will load this specification: You have these 2 options:
- **XML element Name:** Indicate the XML element that contains the ink specification. In above example the ink info is described in the element *Farben*.
- **XPath expression :** Alternatively, use an XPath Expression to the XML element containing the ink information. For example: `/CreateJob/Farben`

Note:

- To open the **XPath builder**, select **XPath expression** and click on the **Edit** button in the text field.
- The XML that the ticket sees as input file will be loaded in the **XPath builder**. Alternatively, you can right-click in the canvas and choose **Paste from File...** to select the example XML input file.
- When you edit the XPath expression again, that same XML (example) input file will be automatically used.

For each job ink parameter, specify the corresponding XML element that is used in the XML file. To do this, select the parameter and click **Edit**.

- **Ink name**

- **Ink type**
- **Ink book**
- **Angle**
- **Ruling**
- **Dot Shape**
- **Printing Method**

Parameters

To add custom parameters to the job you are creating, click **Add** and enter its **Name** and **Value**.

Both fields are SmartName enabled. Typically, the **Name** will be an absolute value and the **Value** is a SmartName that extracts the custom parameters from an XML file.

For example:

Name	Value
Trapping	[FromMIS_XML_TrappingYesNo]
Trap Distance	[FromMIS_XML_TrappingDistance]

Bar Codes

The **Create Job** task also allows to add parameters for one or more bar codes.

Important: The **Create Job** task does not check the consistency of these bar code parameters. This means that the creator of the input file is responsible to create bar code parameters that are compliant.

You can **Add** or **Remove** parameters for multiple bar codes.

For each bar code, you can set the following parameters by adding a SmartName or an XPath expression. Some also offer to select a value from a list.

Modify Bar Code

Type: [FromMIS_XML_Barcode_type] [v]

Sub Type: [] [v]

Code: XPath://NewOrders/Job/Barcode/Code []

Composite Code: []

Output Resolution: []

Bar Width Reduction: [FromMIS_XML_Barcode_BWR] []

Device Compensation: []

Narrow Bar: []

Magnification: [FromMIS_XML_Barcode_Magnif] []

Ratio: []

OK Cancel



Attention: Some parameters will not be relevant for some bar code types.

- **Type.**
- **Sub Type.** A list will be offered if you selected a **Type** that offers **Sub Types**..
- **Code.** As mentioned above, the code is not checked for validation here.
- **Composite Code.** Only some types offer to add an extra composite code.
- **Output Resolution**
- **Bar Width Reduction**
- **Device Compensation**
- **Narrow Bar.** Only some types work with a specification of Narrow Bar and Ratio
- **Magnification**
- **Ratio.** Only some types work with a specification of Narrow Bar and Ratio

Template

You can use an existing job as a template job for the new job that you here create. If you decide so, you can select which attributes and parameters that you want to get from that template job.

Note: For more information about using template jobs and their options, see the dedicated chapter [Jobs](#) on page 828.

Select **Use an existing job as a template** if you want to create your job based on another job.

In **Choose Job**, **Select** the job from the list or use a **SmartName** to have the job selected. An example of a SmartName could be `[customer name]_template`

Then choose what to **Include** from the template job into the new job:

- **Files**
- **Folders**
- **Hot Folders.** This item will only appear if your server still has hot folders from before v14.
- **Tickets.** These are the (blue) Job tickets.
- **Milestones.** Times will be set relative to the creation date of the new job.
- **Parameters.** Custom parameters.
- **Inks.**
- **Categories**



Attention: If there any parameters that you pick up **both** from the input file and from a template job, then those from the input file will be kept.

Advanced

The **Create Job** task can automatically launch a workflow when the new job is created and even when the task updates the job.

To do this, select when you want have a workflow launched and select the workflow from the list. You can even choose both moments.

Note: If the **Create Job** task itself is part of a workflow, the workflow that is chosen in this task will start immediately, as a separate workflow. It will not wait for the workflow that contains the **Create Job** task to finish.

The advantage of this function is that the chosen workflow will be launched *in the context of that created/updated job*. This means that all the used SmartNames will get their value from *that* job.



Attention: In case you create a workflow that contains the **Create Job** task as a workflow step and then continues with other steps, then the tasks *following* that **Create Job** step will not run in the context of the new/updated job: they will run in the context where the (main) workflow initially started!

Sample Input XML

Click [this link](#) to find a ZIP with the samples used in the chapter [Examples](#) on page 1287. There is a sample XML in the folder "Create a Job using XML" that you can use to test setting up SmartNames, Xpath expressions and so build and test an example **Create Job** task.

Download the ZIP file and unzip it to a local folder or directly into a **Automation Engine Container**.

9.6.2 Add To Products task

The **Add To Products** task allows to create Automation Engine **Products**. The task can get their properties out of file names, paths, SmartNames or values out of XML data.

This task is often used after workflow steps first made a smart selection of input files.

This task is documented in the chapter [Products](#) on page 876. Find a direct link [here](#).

9.6.3 Link Product To Job task

The **Link Product To Job** task allows to link or multiple **Products** to a **Job**.

This task is documented in the chapter [Products](#) on page 876. Find a direct link [here](#).

9.7 Examples

These examples will help you build and understand the workflows of typical integration use cases.

At the beginning of each example, we also mention which tools and tasks are used. We also add a link where you can find the sample (XML) files used in that example.

Note: An example often requires you to understand the features explained in the previous examples.

9.7.1 Creating a Job using XML

Summary

We will create a **Job** based on the sample XML. Most of the work is creating the XPath Query SmartNames.

Tools used in this example:

- A sample XML describing 1 job order

- **SmartNames** including **Xpath Builder**
- The **Create Job** task

Step 1 - Analyze your Input XML file

Click [this link](#) to find a ZIP with the sample XML file used in this example.

Save the file in a logical place in an **Automation Engine Container**.

Open it (double-click) and have a look at its structure:

```
<?xml version="1.0" encoding="UTF-8" ?>
- <CreateJob>
  - <Job>
    <OrderID>002</OrderID>
    <SubOrderID>002123</SubOrderID>
    <Name>XMLsample_to_CreateJob</Name>
    <Description>A sample XML to test the Create Job Task.</Description>
    <DueDate>2015-06-01T09:00:00.00+01:00</DueDate>
    <ProjectID>Project123</ProjectID>
    <Category>Offset</Category>
    <CustomField1>CustomField1</CustomField1>
    <CustomField2>CustomField2</CustomField2>
    <JobFolder>file://AESERVER01/MainContainer/FruitCo/XMLsample_to_CreateJob</JobFolder>
  + <Barcode>
  </Job>
  <InkSpecifications>
  - <Ink>
    <ColorBook>Process</ColorBook>
    <ColorType>Normal</ColorType>
    <ColorName>Cyan</ColorName>
    <Angle>15</Angle>
    <Ruling>150</Ruling>
    <Dot>CS4</Dot>
    <PrintingProcess>Offset</PrintingProcess>
  </Ink>
  - <Ink>
    <ColorBook>Process</ColorBook>
    <ColorType>Normal</ColorType>
    <ColorName>Magenta</ColorName>
    <Angle>75</Angle>
```

See how a lot of job info is inside the parent element '*CreateJob*' and how the ink information is inside a different parent element '*InkSpecifications*'.

Note: In this sample, even the full path for the *JobFolder* is filled in. This is usually not the case. A business system usually leaves it up to you to decide where to store your prepress data. Some MIS do want to control this as well. Advantage of already knowing exactly where your prepress files are is that they can then also show some of them to their MIS users (final PDF, JPEG..). However, most MIS will not decide that path for you but will ask you to let them know where you decided to create that *JobFolder*. In that workflow, the MIS usually wants you to return them a XML containing that *JobFolder* path.

Step 2 - Create your SmartNames

SmartNames will read specific pieces of that XML and so populate the various properties of the Job you will create. To read parts of that XML, we will use SmartNames of the type **XPath Query**.

Note: You can also create SmartNames directly from within the **Create Job** task. However, because they are Xpath queries that involve a sample input file, we advise to create them using the main SmartNames tool . It will be somewhat easier to do it there.

1. Go to the SmartNames tool and indicate your sample input XML file

Because you will probably make a dozen new XPath SmartNames, it will be easier when you only have to indicate your sample input file once. Do this in the **Resolve all using:** section.

The screenshot shows the 'SmartNames' tool interface. Under the 'Include SmartNames From:' section, there are two options: 'A workflow:' and 'A product part:', both with 'Browse' buttons. Under the 'Resolve all using:' section, there are two options: 'A task:' and 'A file:'. The 'A file:' option is checked. Below it, a text field contains the file path ':create_job_task_sample_XML.xml' and a 'Browse' button. At the bottom, there is a 'Results:' label and a 'Show' button.

Note: You do not have to click on 'Show' at this moment.

2. Make a first XPath Query SmartName

1.

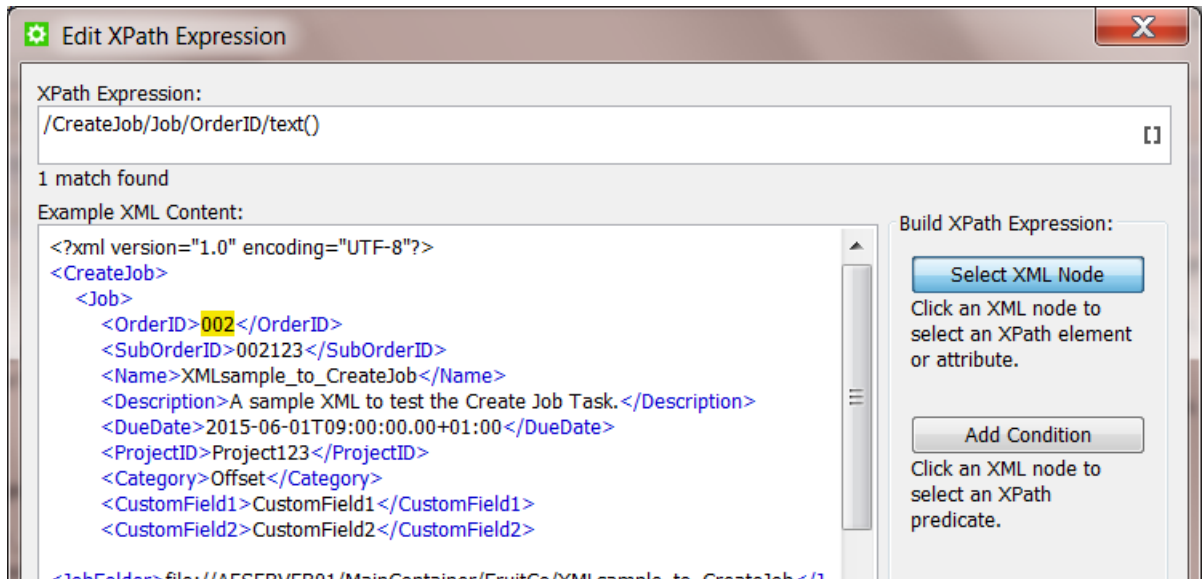
In the SmartNames tool, in the top bar, click on  to create a new SmartName.. Choose Type **XPath Query**.

The screenshot shows a dialog box titled 'Untitled' with a gear icon on the left and a close button on the right. Inside the dialog, there are two dropdown menus: 'Type:' set to 'XPath Query' and 'Scope:' set to 'Global'. Below these is a text area labeled 'XPath Expression:' containing the query: `//*[local-name() = 'vsize']/text()`.

An example query about 'vsize' appears. It will be removed automatically later. See the part `/text()` at the end. This means that the query will not select the full element but only the value of that element.

At the bottom,

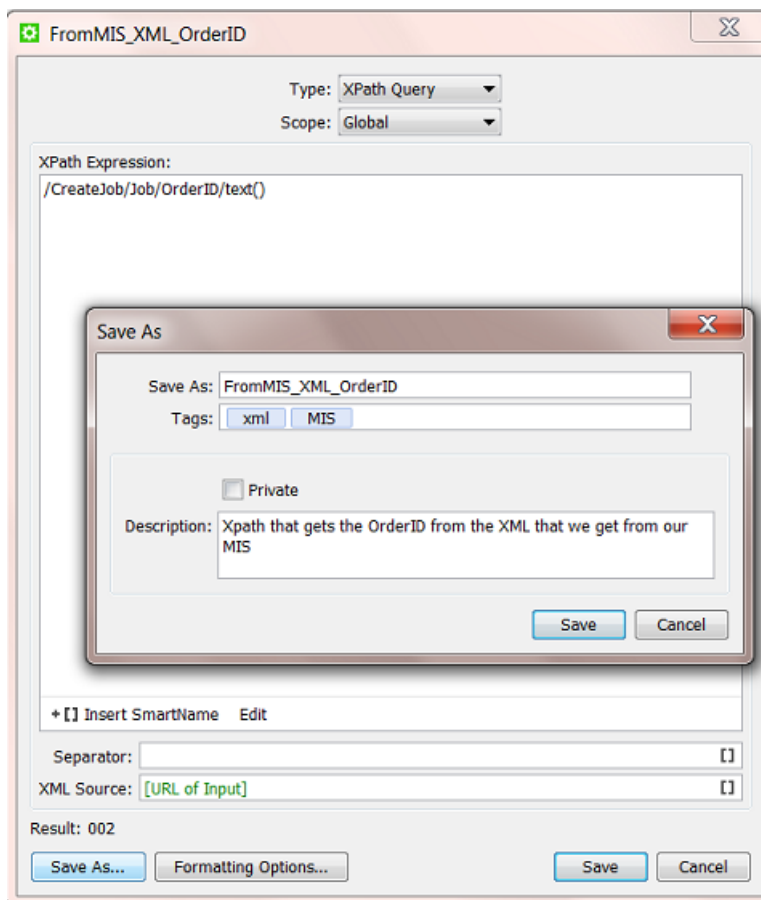
2. Click on 'Edit' to open the **XPath Builder**.



As a first SmartName, we will create the Xpath Query representing the Job's OrderID.

All you need to do here is click on the value that the SmartName will be looking for. It will be highlighted in yellow and the resulting XPath Expression will automatically appear in the top field.

3. Click on 'OK' to close the 'Edit Xpath Expression' dialog. The expression will have replaced the example one. The resulting value will also be shown bottom left in 'Result:' (in below screenshot: "002").
4. Now save your new SmartName. Give it a logical name and maybe add some tags and a description.



3. Now make the other SmartNames (quickly)

The next SmartNames will be made very similarly. Double click the one you just made and while keeping the SmartName dialog open, follow these steps:

- click **Edit**
- in the **Xpath Builder**,
 - click on the value you want the SmartName to represent
 - click **OK** to close the Xpath Builder
- click **Save As...** and enter an appropriate name, tag and descriptions

Repeat this for all the SmartNames you wish to use in the **Create Job** task.

Step 3 - Insert the SmartNames in the Create Job ticket

Open a **Create Job** ticket and insert the created SmartNames in their corresponding fields.

File View Advanced

JOB

Job Customer Inks Parameters Bar Codes Template Advanced

Order ID: [FromMIS_XML_OrderID] []

Sub Order ID: [FromMIS_XML_SubOrderID] []

Name: [FromMIS_XML_JobName]_[FromMIS_XML_OrderID] []

Description: Xpath:/Orders/Job/LongName []

DueDate: [] []

Project ID: [FromMIS_XML_WebProject] []

Category: Digital [] ▾

Quantity: [] ▾

Country: [] ▾

Status: [FromMIS_XML_Status] [] ▾

Job Folder: [Smart Job Location] Browse []

Show Steps Save Cancel

Do the same for all fields in the other tabs.

Don't forget to save your ticket.

Picking up the Ink parameters from the XML

As explained in [Inks](#) on page 1283, the **Inks** tab is constructed differently.

In our sample XML, see how the ink information is described in a parent element **InkSpecifications**. And in there, *per ink* in an element named '**Ink**'. And there are 8 inks described.

```

<?xml version="1.0" encoding="UTF-8" ?>
- <CreateJob>
+ <Job>
- <InkSpecifications>
  - <Ink>
    <ColorBook>Process</ColorBook>
    <ColorType>Normal</ColorType>
    <ColorName>Cyan</ColorName>
    <Angle>15</Angle>
    <Ruling>150</Ruling>
    <Dot>CS4</Dot>
    <PrintingProcess>Offset</PrintingProcess>
  </Ink>
+ <Ink>
+ <Ink>
+ <Ink>
+ <Ink>
+ <Ink>
+ <Ink>
+ <Ink>
  </InkSpecifications>
+ <Customer>
+ <Parameters>
</CreateJob>

```

You also see that the names of the ink parameters are not all identical to those used in the Job setup in Automation Engine. In many cases these words in the XML may not be in English. This terminology in the XML (in the business system) needs to be mapped to the official one in Automation Engine. This is explained in [the documentation of the Create Job task](#).

See the correct settings for our example:

- The element name under which *all* the ink info is found is `InkSpecifications`.
- The parameter names in your XML (right) have been mapped to the standard names in Automation Engine (left):

Job Customer **Inks** Parameters Bar Codes Template Advanced

Load ink information from an XML element:

XML element name
 XPath expression

XML Element Name:

Specify the name of the XML element that contains all ink information.

For each job ink parameter, specify the corresponding XML element that is used in the XML file:

Ink Parameter	XML Element Name
Ink name	ColorName
Ink type	ColorType
Ink book	ColorBook
Angle	Angle
Ruling	Ruling
Dot shape	Dot
Printing method	PrintingProcess

Note: These names are not case-sensitive.

Note: If one XML Element name is wrongly mapped, the Create Job task will not fail but only this wrong parameter will stay empty in the Job Inks setup.

- Alternatively, in our example, You can use this **Xpath Expression** to select that element describing all the inks:

Job Customer **Inks** Parameters Bar Codes Template Advanced

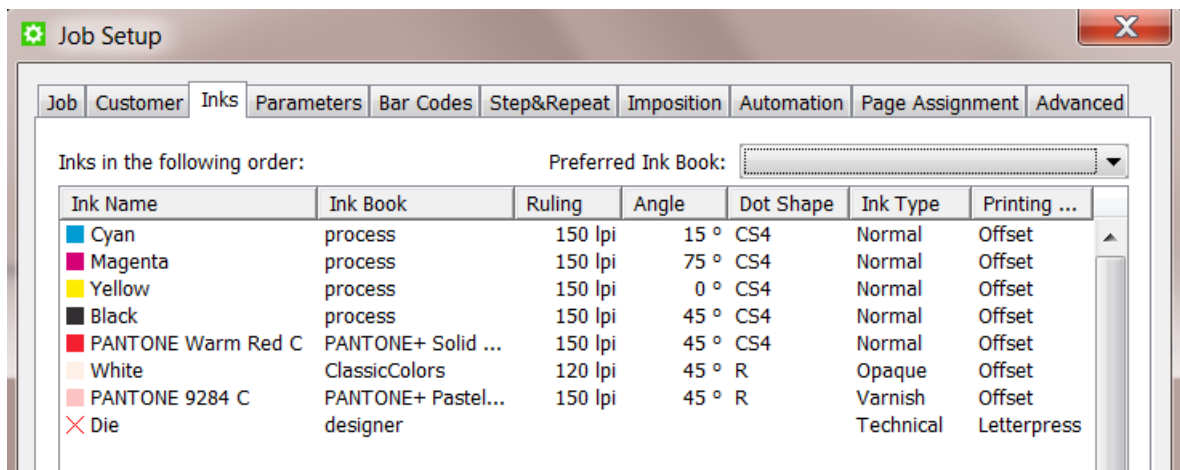
Load ink information from an XML element:

XML element name
 XPath expression

XPath Expression:

Specify the XPath expression to the XML element that contains all ink information.

This is the result in the created Job:



Step 4 - Test your Workflow

Test your workflow by launching it from within the Pilot.

In the Pilot, select the input XML file and launch the workflow. It should create the Job and the Job should contain all the parameters you had set up.

When the Job was created but one or more parameters were not found, the status of that 'Create Job' workflow step will be 'warning' (orange).

- The task's details info will already tell you that "*There were problems resolving some job fields*".
- The task's log file will tell you what specific SmartName(s) has a problem. For example "*Could not resolve(2) job.customer : <NameOfTheSmartNameThatFails>/*"

Step 5 - Add automation by starting this workflow from an Access Point

When an external system sends you a XML files describing Jobs to create, we advise to automate this workflow.

A typical way to do this is to create a **Folder Access Point**. The folder of that **Folder Access Point** is then the place where the XMLs will arrive. The workflow that the **Folder Access Point** will automatically start at a regular time interval is then the one you created in this example.

Note: Read more on setting up such automation in [Folder Access Point](#) on page 1238.



Attention: It is also common for external systems to send the prepress department you an XML file containing the description of **multiple job orders** . And often those systems only send such a list once or twice per day. This use case is explained in a next example [Creating Multiple Jobs from One XML](#) on page 1295.

9.7.2 Creating Multiple Jobs from One XML

Summary

Note: This example requires you to master the features explained in the previous example [Creating a Job using XML](#) on page 1287.

You receive an XML from the business system. It is a list of job-orders and their production parameters. Some are already started in prepress, some are new, some are on hold. Your goal is to create Jobs of all those in the status 'ReadyForPrepress'. The XML needs to be split into multiple XML files each describing one Job. These XML files are then input files for the **Create Job** task. The job-order info in the XML files is too elaborate. You will only pick up some selected parameters from those XML files.

Tools used in this example:

- **SmartNames** including **Xpath Builder**
- **Split XML** task
- **Data Splitter** workflow control
- **Create Job** task

Step 1 - Analyze the XML describing Multiple Jobs

Tip: Click [this link](#) to find a ZIP with the sample XML file used in this example.

The XML in this example describes the 3 job orders. See how 2 of them have the status 'ReadyForPrepress'.


```

<?xml version="1.0" encoding="UTF-8"?>
- <Jobs>
  - <Job>
    <OrderID>1404-W86203</OrderID>
    <SubOrderID>003</SubOrderID>
    <Name>1404-W86203</Name>
    <Description>Promo strawberry 100ml South Europe</Description>
    <DueDate>2015-06-01T08:00:00.00Z</DueDate>
    <Status>ReadyForPrepress</Status>
    <ProjectID>Project193</ProjectID>
    <CustomerID>2712</CustomerID>
    <Category>flexo</Category>
    <CustomField1>Nilpeter FB-2500</CustomField1>
    <CustomField2>White BOPP</CustomField2>
  </Job>
  - <Job>
    <OrderID>1404-W74536</OrderID>
    <SubOrderID>002</SubOrderID>
    <Name>1404-W74536</Name>
    <Description>New flavour Cocos and lemon </Description>
    <DueDate>2015-06-01T08:00:00.00Z</DueDate>
    <Status>OnHold</Status>
    <ProjectID>Project123</ProjectID>
    <CustomerID>2712</CustomerID>
    <Category>flexo</Category>
    <CustomField1>Gallus ECS 340</CustomField1>
    <CustomField2>OPP</CustomField2>
  </Job>
  - <Job>
    <OrderID>1404-W74599</OrderID>
    <SubOrderID>009</SubOrderID>
    <Name>1404-W74599</Name>
    <Description>New flavour Banana</Description>
    <DueDate>2015-06-01T08:00:00.00Z</DueDate>
    <Status>ReadyForPrepress</Status>
    <ProjectID>Project123</ProjectID>
    <CustomerID>2712</CustomerID>
    <Category>flexo</Category>
    <CustomField1>Gallus ECS 340</CustomField1>
    <CustomField2>OPP</CustomField2>
  </Job>
</Jobs>

```

Note: In this example XML, we removed typical extra parameters about the customer, bar code and inks. The principle of those types was already explained in [Creating a Job using XML](#) on page 1287.



Attention: If you send this XML to the **Create Job** task, it would only create the first Job it finds in the XML.

Step 2 - Create the Workflow Ticket

This is the workflow we need:

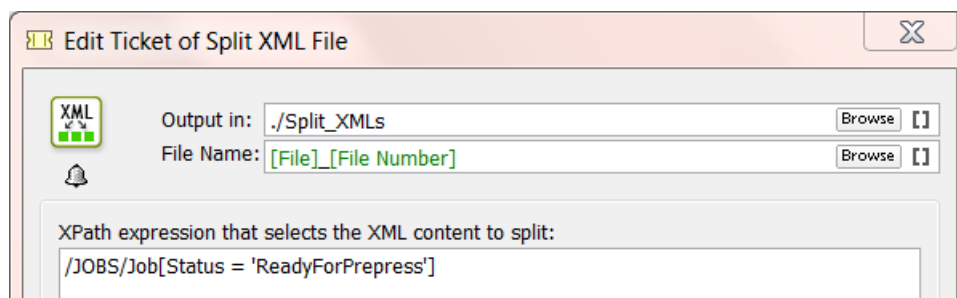


- The **Split XML** task splits the XML into multiple XML files. If you don't do this, the **Create Job** task would only create the first Job it finds in the XML.
- The **Data Splitter** makes sure the next task sees the many input files. It prevents that the task is only launched on the first input file.
- The **Create Job** task creates Jobs, as determined in the **Data Splitter**, for *each* incoming XML file.

Note: The **Create Job** has no output files. If you want to continue the workflow with its input files (the split XML files), then configure a specific **output pin** for that. Read more on these possibilities in [The Workflow Editor](#) and also [Building Workflows](#) in the User Guide.

Step 3 - Splitting the XML but also using a Filter on Status

In the **Split XML** task, we need this XPath Expression:



- The expression `/Jobs/Job` will result in splitting the XML on that level of elements: one per Job.
- The extra filter `[Status = 'ReadyForPrepress']` will limit that to jobs with that status.

This XPath expression can easily be built by using the **Xpath Builder**. Open it via the **Edit** button.

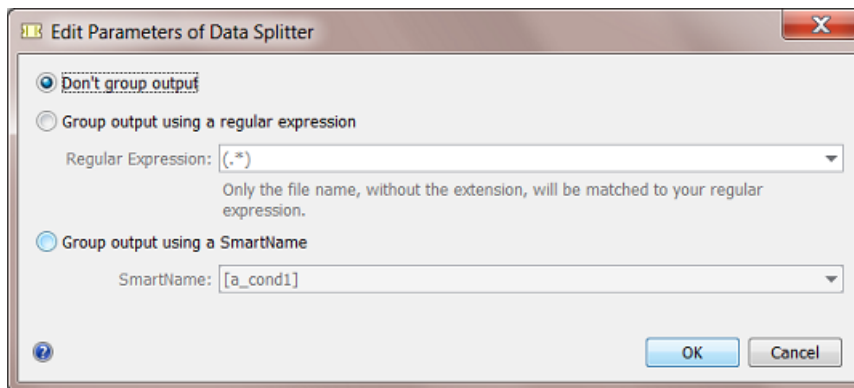
Note: When the filter does not find any matching 'node' (job), the task will error and mention this as the reason.

- **Output in:** Define where you want the split XMLs to be written.
- **File name:** Define the name of the split XML(s). The default setting will add a number.

Step 4 - Importance of the Data Splitter

When you do not use the **Data Splitter**, the **Create Job** task will receive all the XMLs generated by the **Split** task as 1 group on its input pin. In this case only the first incoming XML will be processed and the rest will be ignored. The **Data Splitter** step in this workflow will make sure the **Create Job** task receives the XMLs as separate files so each of them gets processed.

Because we (possibly) have multiple input XML files, we need these (default) settings:



We do not want the split XML files to be grouped. The **Create Job** task needs to be launched on each one of them.

Find more details on this task in [Data Splitter](#) on page 1030.

Step 5 - Create the Xpath SmartNames

Identical as explained in the previous example [Creating a Job using XML](#) on page 1287, you need to

- create SmartNames that you will use in the **Create Job** ticket. As explained in [Step 2 - Create your SmartNames](#) on page 1289.
- insert those SmartNames in the **Create Job** ticket. As explained earlier in [Step 3 - Insert the SmartNames in the Create Job ticket](#) on page 1291.

Step 6 - Workflow Result

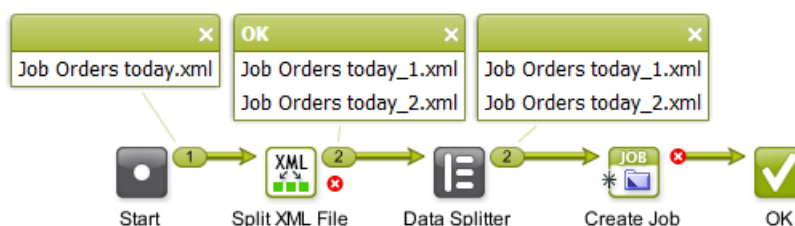
Select the XML describing multiple Jobs and launch your workflow.

The **Create Job** task will be started per incoming XML. In this case 2 times, because only 2 had the status 'ReadyForPrepress'.

File Name	Task Type	Progress	Phase	State	Launched
Job Orders today_2.xml	Create Job	100%		✓	7/11/14 2:49 PM
Job Orders today_1.xml	Create Job	100%		✓	7/11/14 2:49 PM
Job Orders today.xml	Split XML File	100%		✓	7/11/14 2:49 PM
Job Orders today.xml	Workflow	100%		!	7/11/14 2:49 PM

Note: Workflow controls like the **Data Splitter** do not appear in this task overview.

See the files the workflow created:



The 2 Jobs that had the status 'ReadyForPrepress' will be created:

Name	Description	Due Date	Order ID	Sub Order ID	Customer Name
1404-W86203_003	Promo strawberry 100ml South Europe	Jun 1, 2015	1404-W86203	003	FruitCo
1404-W74599_009	New flavour Banana	Jun 1, 2015	1404-W74599	009	FruitCo

9.7.3 Create Multiple Jobs from a CSV file

Summary

Every morning, you receive a CSV file from the planning department with a list of job orders for the prepress department. Each row in the CSV represents a job order. This data needs to be converted into separate XML files to send to the **Create Job** task.

Note: a CSV is a Comma Separated Value file. This is usually an exported file from a database or an XLS file.

Tools used in this example:

- **SmartNames** including **Xpath Builder**
- **Map Data** task
- **Split XML** task
- **Data splitter** workflow control
- **Create Job** task

Step 1 - Analyse the CSV file

Tip: Click [this link](#) to find a ZIP with the sample CSV file used in this example.

Our example CSV contains a description of 2 job orders.

	A1	Order Nr								
	A	B	C	D	E	F	G	H	I	J
1	Order Nr	Customer	Customer Nr	DueDate	CSR email	Country	Process	Shipping	Preflight	Proof
2	EXA_CSV_Job01	FruitCo	2712	30-09-14	Franky.fruit@fruitco.com	USA	Flexo	Fedex		Web
3	EXA_CSV_Job02	Toy Dream	1965	25-10-14	Tony.Toy@Toydream.com	China	Offset	DHL	Yes	PDF

Because the **Create Job** task does not work with CSV files, we first need to create an XML. And as you saw in the previous example [Creating Multiple Jobs from One XML](#) on page 1295, we will need 1 XML *per Job* to create.

Note: At this point it is good to detect any data fields that will need reformatting. In this example workflow, the format of the due date will need to be mapped to the format that is required for that field in the Job Setup.

Step 2 - Create the Workflow Ticket

This is the workflow that we need to create Jobs from a CSV with descriptions of multiple job orders:



- The **Map Data** task defines what parts of the CSV will be mapped into the XML file.
- The **Split XML** task splits that XML into multiple XML files, like we did in the [previous example](#).
- The **Data Splitter** makes sure that the next task sees the many input files. It prevents that the task is only launched on the first input file.
- The **Create Job** task creates Jobs, one for *each* incoming XML file.

Step 3 - Mapping the CSV to XML

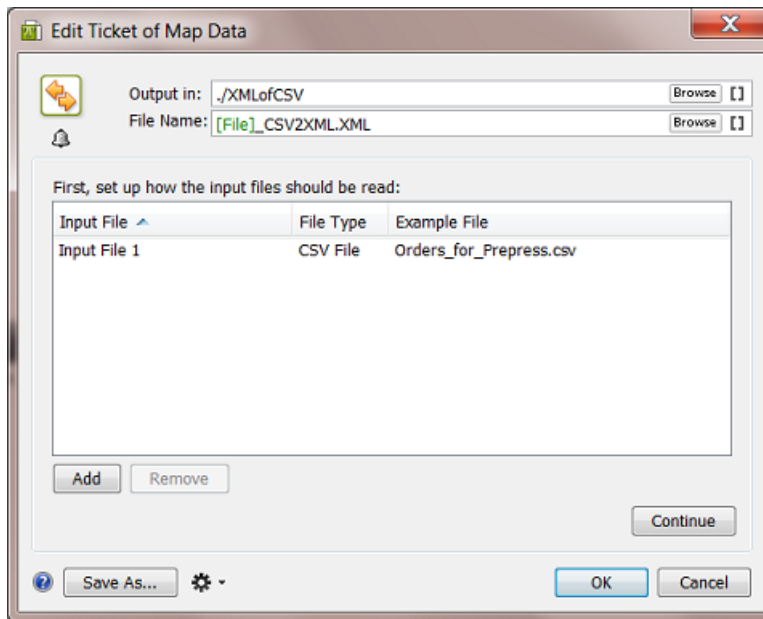
The **Map Data** ticket will

- convert the CSV to XML
- define which parts of the CSV are needed in the XML
- define how the parameters should be named in the XML. You can keep their names if you want to.
- define what part in the XML will be **Repeating Content**. This is needed because you are not sure how many jobs there will be in that CSV / XML. The CSV that you receive tomorrow can have a different amount of rows. This is the '*for each*' principle as explained earlier in [Specifying the Output File](#) on page 1271.

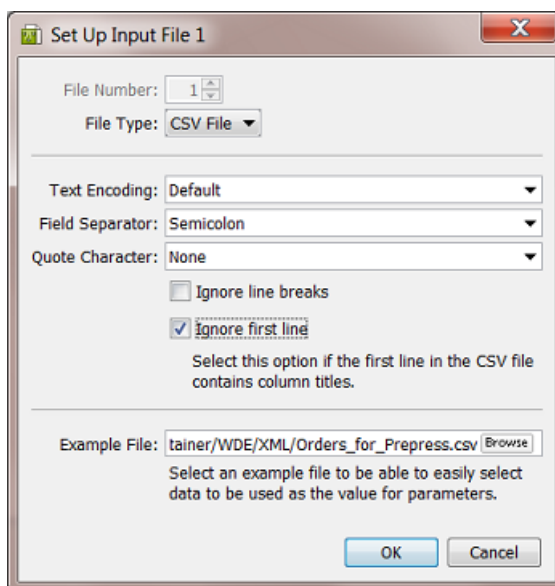
Note: The *order* in which we define the settings in this ticket is important.

1. Define the CSV as input file

- Select your CSV file and open the workflow ticket that you made in the previous step.
- Double click the **Map Data** step. This automatically sets your CSV as input file:



- Now set 2 specific options that define how our example CSV should be read. Double click the line 'Input File 1' to open its setup. Set the **Field Separator** to **Semicolon**. Because the first line of the CSV contains the field headers (not values we want to read), check the option **Ignore first line**. Remember that your specific CSV might be different, so do check all options.



- Click **OK** to close that dialog.
- Maybe now is a good time to already define the folder and name of your output XML. As usual, you can use SmartNames in **Output in:** and **File Name:**.
- Click **Continue**.
- Make sure the **Output Format:** is on 'XML'. In the **Options...** make sure to activate the **Replace reserved characters in parameter values with XML entities**. In this example, there are no such characters.

2. Define the Repeating Content in the XML

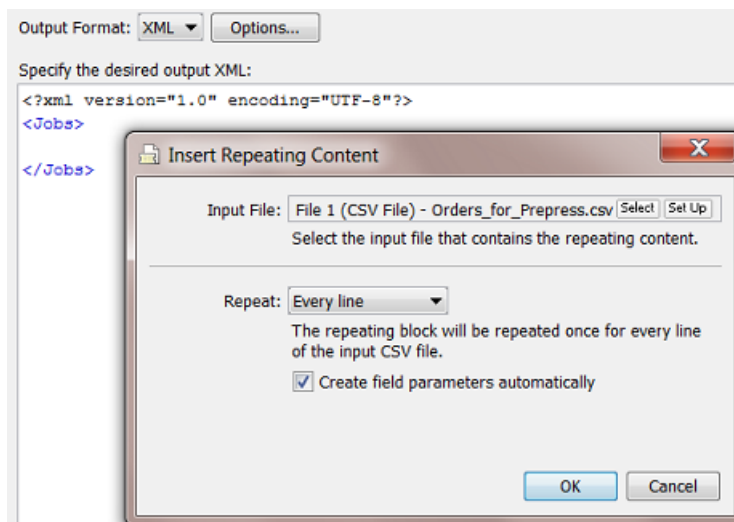
Note: Make sure you have already read [the chapter documenting this advanced technology](#).

This ticket allows many ways to use it. Following steps are specific for this CSV to XML workflow:

In the **Map Data** ticket, in the canvas where you define the output XML,

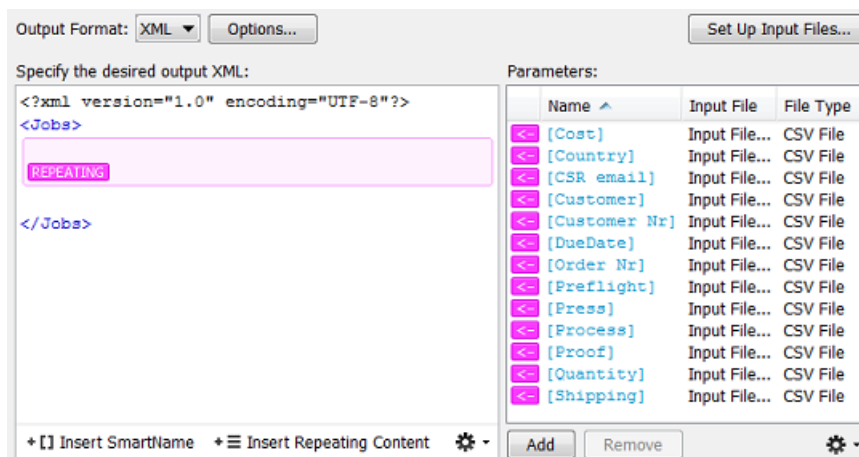
- Right-click and choose **Insert XML Declaration**. This is just good practice when creating XML.
- Right-click and choose **Insert Element**. Give it a logical names, for example `<Jobs>`
- Put your cursor in between the open and closing element, right-click and choose **Insert Repeating Content** (or click on `+≡`).

Remember: You can not make any existing content 'Repeating' *later*; you need to create the 'Repeating Content' block *first* and then add content inside.



Every line (row) in your CSV represents a job order. So you do want **Every line** in your CSV to become the Repeating Content. Remember we already decided to ignore the first line of the CSV.

- Click OK. A pink field for **Repeating Content** is created. Also the field parameters were created automatically. They are colored pink because they are linked to the selected **Repeating Content** field.

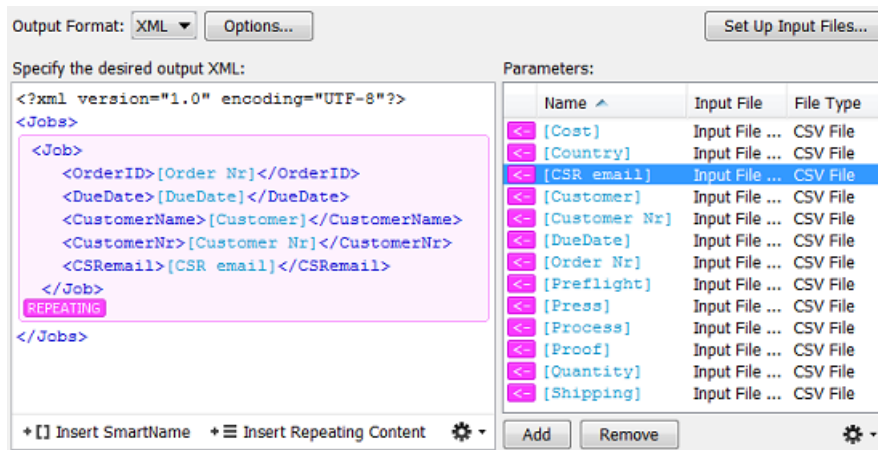


- Now, inside that **Repeating Content** field, continue specifying the content of the XML. Insert Elements and click the pink parameters in between the opening and closing element.



Caution: Make sure your XML elements do not contain spaces!

This is a basic and valid example:



You can see we only use 5 of the parameters in this example. You can of course pick up a lot more.

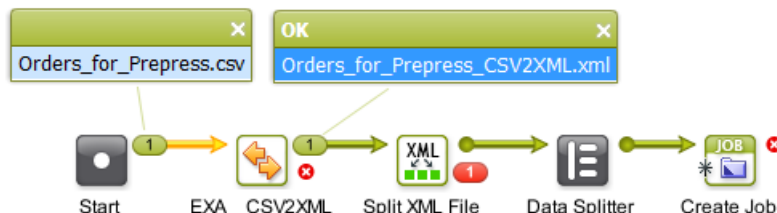
Note: Remember that the XML file will just be an in-between format. You will later, in the **Create Job** ticket, still map the XML elements to the real fields in the Job Setup.

- Click **OK** to close the set up of your **Map Data** ticket.
- Save your Workflow ticket. No need to close it.

Step 4 - Test your CSV to XML Mapping

We advise to already test what we set up in our **Map Data** ticket. Because if that output does not look good, it is better to correct that now instead of using a wrong XML file as input for the next workflow steps.

With the CSV as input file, launch the workflow. Because we haven't defined the settings for the next workflow steps yet, the workflow will surely fail.



You now see that your CSV was mapped to an XML. Double click the output file to open it. It should look like this:


```

<?xml version="1.0" encoding="UTF-8" ?>
- <Jobs>
- <Job>
  <OrderID>EXA_CSV_Job01</OrderID>
  <DueDate>30-09-14</DueDate>
  <CustomerName>FruitCo</CustomerName>
  <CustomerNr>2712</CustomerNr>
  <CSRemail>Franky.fruit@fruitco.com</CSRemail>
</Job>
- <Job>
  <OrderID>EXA_CSV_Job02</OrderID>
  <DueDate>25-10-14</DueDate>
  <CustomerName>Toy Dream</CustomerName>
  <CustomerNr>1965</CustomerNr>
  <CSRemail>Tony.Toy@Toydream.com</CSRemail>
</Job>
</Jobs>

```

You see

- the main structure `/Jobs/Job`
- the description of the 2 Jobs,
- including the 5 parameters that we picked up from their definition in the CSV.

When your XML is not OK, then solve the problem first before defining the next workflow steps.

Step 5 - Set up the Split XML step

- Leave the workflow ticket and go back to your Pilot.
- Go to the folder where you wrote the output of the **Map Data** ticket and select the `Orders_for_Prepress_CSV2XML.XML`
- Open the default **Split XML** ticket. Your XML will be seen as input file.
- Click **Edit** to open the **Xpath Builder**.
- Click on a `Job` node

```

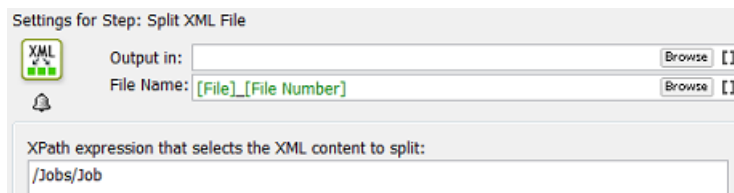
XPath Expression:
/Jobs/Job

2 matches found
Example XML Content:
<?xml version="1.0" encoding="UTF-8"?>
<Jobs>
<Job>
  <OrderID>EXA_CSV_Job01</OrderID>
  <DueDate>30-09-14</DueDate>
  <CustomerName>FruitCo</CustomerName>
  <CustomerNr>2712</CustomerNr>
  <CSRemail>Franky.fruit@fruitco.com</CSRemail>
</Job>
<Job>
  <OrderID>EXA_CSV_Job02</OrderID>
  <DueDate>25-10-14</DueDate>
  <CustomerName>Toy Dream</CustomerName>
  <CustomerNr>1965</CustomerNr>
  <CSRemail>Tony.Toy@Toydream.com</CSRemail>
</Job>
</Jobs>

```

The resulting Expression will be shown above: `/Jobs/Job` .

- Click **OK** to close the dialog.

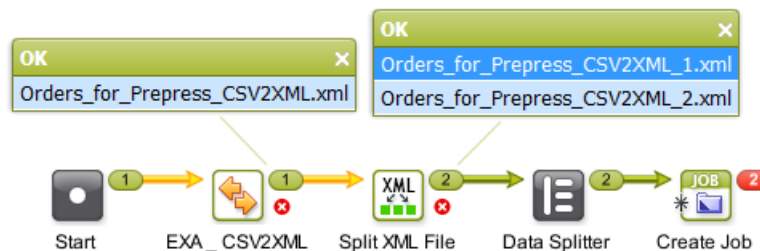


- Your ticket is ready. Save it with a logic name.
- Open your workflow and replace the default **Split XML** step with this specific one.
- Save your workflow.

Step 6 - Test your workflow again

- Select the CSV and launch your workflow. Because we haven't defined the settings for **Create Job** ticket yet, the workflow will surely fail.

What you now need to see is the 2 XMLs as output of the **Split XML** ticket.



- Double-click one of the split off XML files to open it. It should look like this:

```
<?xml version="1.0" encoding="UTF-8" standalone="no" ?>
- <Jobs>
- <Job>
  <OrderID>EXA_CSV_Job01</OrderID>
  <DueDate>30-09-14</DueDate>
  <CustomerName>FruitCo</CustomerName>
  <CustomerNr>2712</CustomerNr>
  <CSRemail>Franky.fruit@fruitco.com</CSRemail>
</Job>
</Jobs>
```

Step 7 - Check the Data Splitter

The **Data Splitter** control in your workflow should have the (default) settings to **'not group'**. Its purpose is identical as in previous example, where we explained [the importance of the Data Splitter](#).

Step 8 - Defining the SmartNames for the Create Job ticket


Making the SmartNames and inserting them into your **Create Job** ticket is not different here than explained in earlier examples, like in [Step 2 - Create your SmartNames](#) on page 1289 and [Step 3 - Insert the SmartNames in the Create Job ticket](#) on page 1291. We will first briefly recapitulate that part and then we will pay extra attention to setting up the SmartName for the Due date.

Creating the necessary SmartNames

Follow these steps to create the SmartNames:

- In your **Pilot**, go to the **SmartNames** tool.
- On the right, in **Resolve all using:**, select **A file:** and browse to one of the split off XML files (for example *Orders_for_Prepress_CSV2XML_1.xml*).
- Create Xpath Query SmartNames for
 - OrderID
 - CustomerName
 - CustomerID
 - CSRemail

Creating the SmartName for the Due Date

- In your **Pilot**, go to the **SmartNames** tool.
- Click on  to create a New SmartName.
- Choose the type : Xpath Query.
- Click on **Edit**. In the XPath Builder, select the value for the element *DueDate*.

```

XPath Expression:
/Job/Job/DueDate/text()

1 match found
Example XML Content:
<?xml version="1.0" encoding="UTF-8" standalone="no"?>
<Jobs>
<Job>
  <OrderID>EXA_CSV_Job01</OrderID>
  <DueDate>30-09-14</DueDate>
  <CustomerName>FruitCo</CustomerName>
  <CustomerNr>2712</CustomerNr>
  <CSRemail>Franky.fruit@fruitco.com</CSRemail>
</Job>
</Jobs>

```

- In the setup of an Automation Engine Job, the due date needs to be specified according the ISO 8601 standard. We therefore need to reformat that date from how it was defined in the CSV ('30-09-14'). Now set the formatting options as follows (see screen shot below):
 - Click on **Formatting Options....**
 - Set the **Data Format** to **Date and Time**.
 - Set the **Format** to **ISO 8601**.
 - Choose your **Time Zone** (the one of your Automation Engine server).

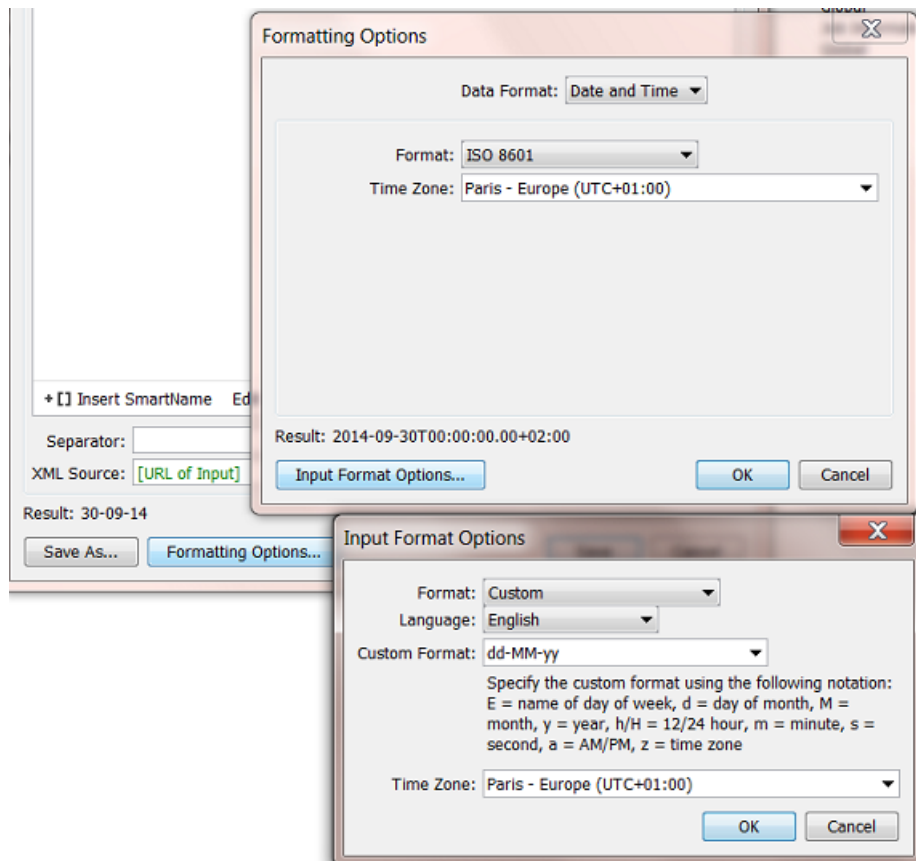
Below left in the dialog, you will see that the **Result** is still not OK.
 - Click on **Input Format Options....**
 - Set the **Format** to **Custom**.
 - Set the **Language**.

- Choose **Custom Format**. Now write how your data was formatted in the CSV: dd-MM-yy.



Caution: The month is here defined by an 'M' in capitals! A lowercase 'm' will be seen as minutes!

- Click **OK**. The correct result should now be visible in the **Formatting Options** dialog.



Now save your SmartName and give it a logic name. For example "*From XML - CSV Due date for ISO*".

The resulting Create Job ticket

- Open the workflow and double-click the **Create Job** ticket.
- Add your SmartNames. It could look like this:

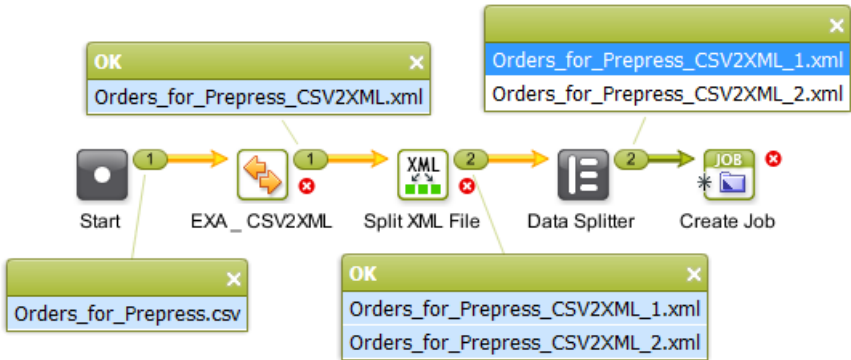
Job	Customer	Inks	Parameters	Bar Codes	Template	Advanced
Order ID:	[EXA XML OrderID]					[]
Sub Order ID:						[]
Name:						[]
Description:						[]
Due Date:	[From XML - CSV Due date for ISO]					[]
Project ID:						[]
Category:						[] ▼
Printing Press:						[] ▼
Substrate:						[] ▼
Job Folder:	it/Automation Engine/Public/demo_container/[EXA XML OrderID]					[Browse] []

Job	Customer	Inks	Parameters	Bar Codes	Template	Advanced
Customer ID:	[EXA - XML Customer ID]					[Select] []
Customer Name:	[EXA - XML Customer]					[]
Customer Description:						[]
Customer's Job Reference:						[]
Customer Service Representative:						[] ▼
E-mail:	[XML CSR Email]					[]

- Now save your workflow ticket.

Step 9 - Workflow Result

In the **Pilot**, select your CSV file and launch your workflow. It will look like this:



It will have created these 2 Jobs:

Jobs			
Name	Order ID	Customer Name	Due Date
EXA_CSV_Job02	EXA_CSV_Job02	Toy Dream	Oct 25, 2014
EXA_CSV_Job01	EXA_CSV_Job01	FruitCo	Sep 30, 2014

9.7.4 Creating Jobs using a Database Access Point

Summary

We will use a **Database Access Point** to query an external database (one from a business system where the job orders originate). The query will, at a regular time interval, ask that database which new Jobs need to be created on Automation Engine. The result of this query are XML files with the description of those Jobs. Those XML files will be used to create the Jobs (as shown in [previous examples](#)).

Tools used in this example:

- An external database
- A **Database Access Point**
- **SmartNames** including **Xpath Builder**
- The **Data Splitter** workflow control
- The **Create Job** task

Step 1 - Analyse the External Database

Tip: Click [this link](#) to find a ZIP with the sample database used in this example.

This is the database we will query in this example:

OrderNr	CustomerName	CustomerID	DueDate	ProofType	Status	Process	Press	Quantity
1234	ToyDream	1965	30-09-14	Web	Shipped	Flexo	F1	200000
1235	GraceLand	6666	27-12-14	FTP-PDF	Ready For Prepress	Screen	S3	250000
1236	FruitCo	2712	11-10-14	HardCopy	Ready For Prepress	Flexo	F1	420000
1237	FruitCo	2712	02-12-14	HardCopy	Ready For Prepress	Flexo	F1	160000
1238	ToyDream	1965	07-09-14	Web	On Hold	Flexo	F1	32000
1239	ToyDream	1965	23-11-14	Web	NULL	Offset	O4	7500
* NULL	NULL	NULL	NULL	NULL	NULL	NULL	NULL	NULL

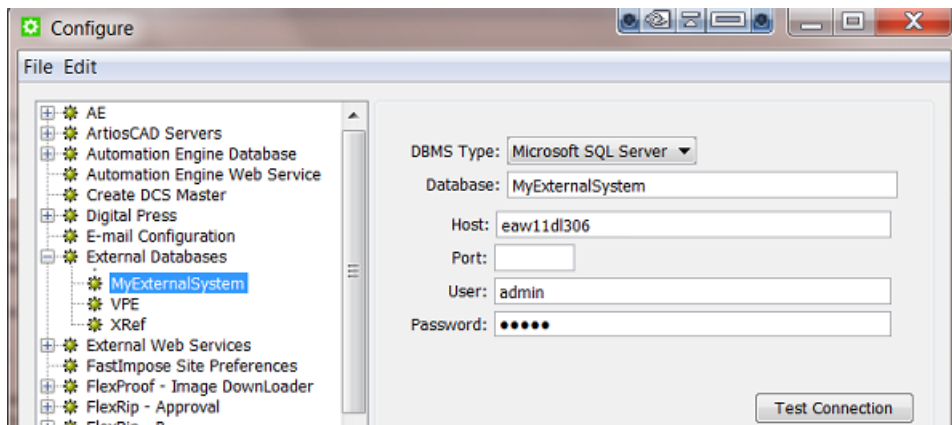
- The database software type is a MS SQL server.
- It runs on a server computer named EAW11DL306.
- The name of the database that we will access is MyExternalSystem (there could be more served by this same software).
- The name of the (opened) table is MyOrders.

- The table contains the description of 6 job orders.
- The `Status` column shows that 3 job orders are 'Ready For Prepress'. Those are the ones that we want to create on Automation Engine.

Step 2 - Configure the Link to the External Database

Set up the link with the external database. This is done in via **Tools > Configure > External Databases**. Read more on such configurations in [External Databases](#) on page 637.

For this example, this is the configuration we need:



Step 3 - Set up the Database Access Point

Note: Setting up **Database Access Points** is documented in [Create or modifying a Database Access Point](#) on page 1246.

This is an example of the Access Point we need:

Modify Database Access Point

Name:

Description:

Database Settings | Monitoring | Automatic Actions

External Database:

SQL Select Query:

After Processing: Delete the records
 Update the records

Column Name	New Value
Status	InPrepress

- In the **Pilot**, create a new **Access Point** and choose the type **Database**.
- Give it a logical **Name** and **Description**.

Database Settings

- Select the **External Database** you configured in the previous step.
- Enter the **SQL Select Query**:
 - 'SELECT' indicates which columns you want to get a value from.
 - 'FROM' indicates the database *table* where these columns are.
 - 'WHERE' indicates an extra filter: you do not want *all* the jobs from that table, but only those with the status 'Ready For Prepress'.

Note: SQL queries are case-sensitive!

Note: It is possible to select values from different tables, for example if you want to get more details about the customer or about production tools. This examples does not do that.

Note: Esko does not offer training or support in database queries. This is mentioned in more detail in [Concept of tasks interacting directly with external systems](#) and [Concept of the Interact with Database task](#).

- **After Processing:** We here use **Update the records** to change the status field in the external database to 'InPrepress'. This is necessary to avoid that, after the next time interval, these jobs would be created/updated again (because they would still be in the status 'Ready For Prepress'.)

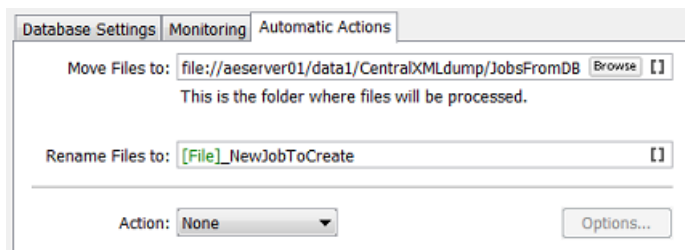
Tip: You can choose to not do this status-update here but rather wait until the workflow really has created the **Job**. In the following workflow, you can then use an extra **Interact with Database** ticket to update that status in the external database with for example a status 'Job Created and Started in Prepress'.

Monitoring

Define the time interval with which the query should be done.

Note: There is no need to set it to '1 minute' to make testing easier. See further how we will use the **scan now** button instead.

Automatic Actions



The query will write an XML file **for each** job order that it finds in the specified status.

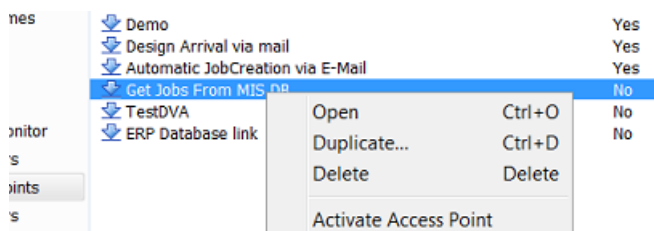
- **Move Files to:** Define where you want those XML files to be written. The Job they represent does not exist yet so we advise to write them in some central folder. See an example in the screen shot.
- **Rename Files to:** Their default name `[File]` actually consists of `[table_name]_[date-and-time]`. You can overrule this or add more SmartNames. Also see later when we show the result.
- **Action:** We will add a workflow in later steps of this example. For the moment, leave this to `None`.

Click OK to confirm your settings


Your **Database Access Point** now exists.

Activate the Access Point

Right-click and choose **Activate Access Point**.

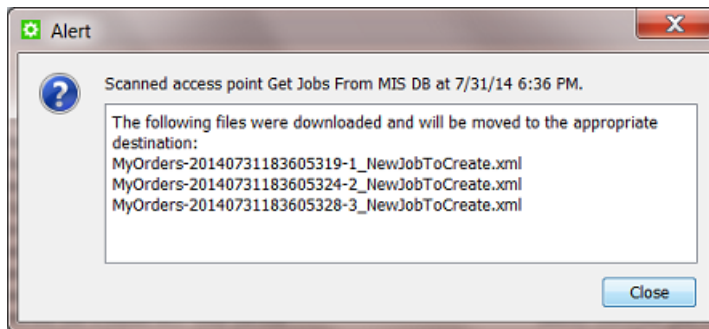


Step 4 - Test your Database Access Point

Select your **Database Access Point** and click the  button to **scan now**.

This will execute the SQL query immediately.

An **Alert** window will appear with the result. In our example, we see



- 3 XML files were created because 3 job orders had the status that we were asking for: 'Ready For Prepress'.
- Their name is the result of the SmartNames as explained in the previous step.

This is one of the resulting XML files:

```
<?xml version="1.0" encoding="UTF-8" ?>
- <MyOrders created="2014-08-01T12:52:56.928+02:00" producer="DBPoller-MyExternalSystem"
  select="SELECT OrderNr, CustomerName, CustomerID, DueDate, ProofType, Process, Status
  FROM MyOrders WHERE Status = 'Ready For Prepress'">
  <OrderNr>1235</OrderNr>
  <CustomerName>Graceland</CustomerName>
  <CustomerID>6666</CustomerID>
  <DueDate>27-12-14</DueDate>
  <ProofType>FTP-PDF</ProofType>
  <Process>Screen</Process>
  <Status>Ready For Prepress</Status>
</MyOrders>
```

See how the XML file contains the query that was used.

These XML files will now be used as input files to create Jobs.

Check the new status in the database

Remember the [Update the records setting in previous step](#). See how, for the 3 job orders that first had the status 'Ready For Prepress', the status has now changed to 'InPrepress':

EAW11DL306.MyEx... - dbo.MyOrders Summary									
	OrderNr	CustomerName	CustomerID	DueDate	ProofType	Status	Process	Press	Quantity
	1234	ToyDream	1965	30-09-14	Web	Shipped	Flexo	F1	200000
▶	1235	Graceland	6666	27-12-14	FTP-PDF	InPrepress	Screen	S3	250000
	1236	FruitCo	2712	11-10-14	HardCopy	InPrepress	Flexo	F1	420000
	1237	FruitCo	2712	02-12-14	HardCopy	InPrepress	Flexo	F1	160000
	1238	ToyDream	1965	07-09-14	Web	On Hold	Flexo	F1	32000
	1239	ToyDream	1965	23-11-14	Web	NULL	Offset	O4	7500

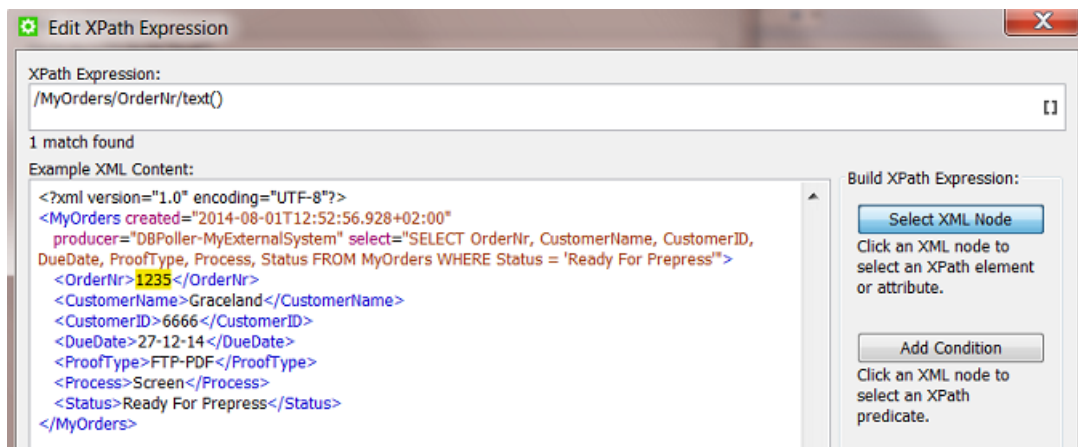


Attention: When testing this **Access Point**, do remember that every time you run it that the status of those job orders will change to 'InPrepress' ! This means that the next time the **Access Point** will do nothing simply because there are no more job orders in that database with the required status 'Ready For Prepress'! So, in between tests, you have to manually reset those statuses to 'Ready For Prepress'. And remember that the queries are case-sensitive!

Step 5 - Create your SmartNames

Identical to how we did this in [step 2 of the first example](#), we will now create the SmartNames that will map the fields in the XML to the Job setup fields in the **Create Job** task.

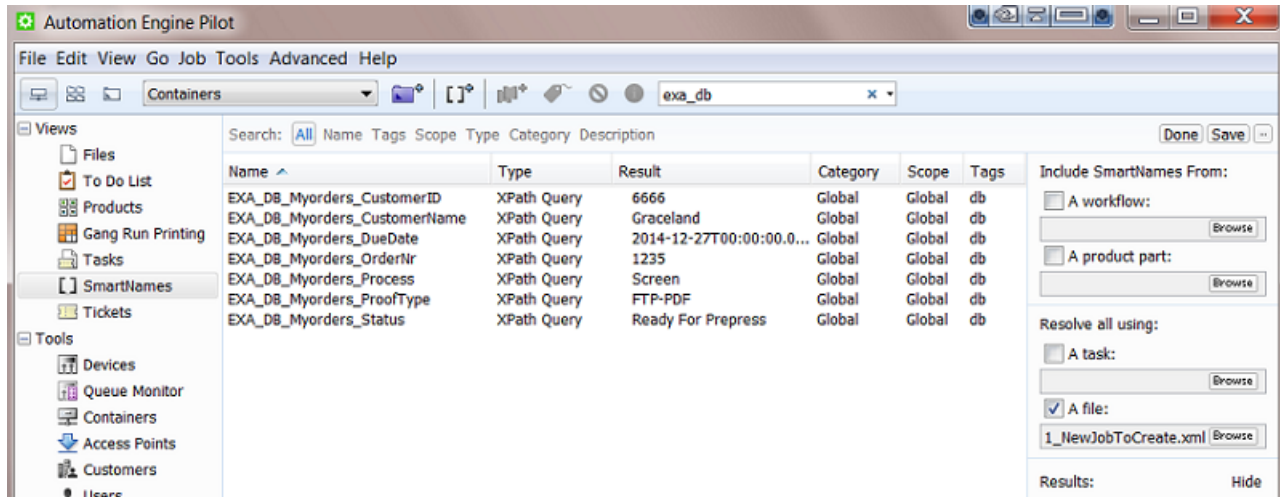
1. In the Pilot, go to the **SmartNames** View.
2. On the right, in **Resolve all using:**, check the button for **A file** and browse to one of the XML files you created in the previous step.
3. Click on **Create a New SmartName**.
4. In the dialog, indicate the type **Xpath Query**.
5. Click on **Edit** to open the **Xpath Builder**.
6. In the XML content, highlight the first value you want to make a **SmartName** for. For example OrderNr.



The resulting **Xpath Expression** will appear.


7. Click **OK**.
You will see the result value 1235 appear bottom left in **Result:** .
8. Click **Save As...** and give the new **SmartName** a logical name. For example EXA_DB_Myorders_OrderNr.
9. Keep the dialog open and click again on **Edit**. Repeat steps 6 to 8 for the next **SmartNames** that you want to make in this example, **except the SmartName for the Due Date**, that one needs an extra formatting. See next step:
10. Creating the **SmartName** for due date is here identical to how we did it [in the previous example](#). You need to make sure the output format is according the ISO standard for this Job setup field. And you need to indicate how the input is constructed. In our example this is dd-MM-yy.

11. When finished saving all **SmartNames**, close the dialog (press **Cancel** or use **Save** for your last **SmartName**)
12. Back in the **SmartNames View**, type a view filter that will only show your 5 new **SmartNames** and then click on **Show** (right panel)
You should now see this:



Step 6 - Create the Workflow with the Create Job task

To have the Jobs created automatically, create this workflow:

1. In the **Pilot**, go back to the **Files** or **Job** mode and click on  to create a new **workflow**.
2. Build a workflow that starts with a **Data Splitter** (keep its default settings 'don't group output'), followed by a **Create Job** ticket.



3. Already save your workflow with a logical name. Remember we will later attach it to the **Database Access Point**.
4. Double click the **Create Job** ticket and add the **SmartNames** to their matching fields.
It should look like this:

Job	Customer	Inks	Parameters	Bar Codes	Template	Advanced
Order ID:	[EXA_DB_Myorders_OrderNr]					[]
Sub Order ID:						[]
Name:						[]
Description:						[]
Due Date:	[EXA_DB_Myorders_DueDate]					[]
Project ID:						[]
Category:	[EXA_DB_Myorders_Process]					[] ▼
Printing Press:						[] ▼
Substrate:						[] ▼
Job Folder:	<A_DB_Myorders_CustomerName>/[EXA_DB_Myorders_OrderNr]					[Browse] []

The **Job Folder** setting in the screen shot is just an example of many ways to define it.

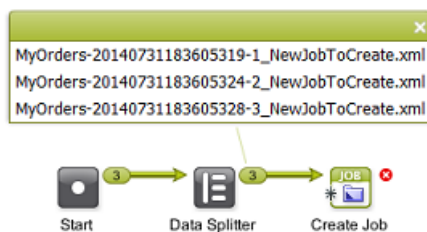
Job	Customer	Inks	Parameters	Bar Codes	Template	Advanced
Customer ID:	[EXA_DB_Myorders_CustomerID]					[Select] []
Customer Name:	[EXA_DB_Myorders_CustomerName]					[]
Customer Description:						[]
Customer's Job Reference:						[]
Customer Service Representative:						[] ▼
E-mail:						[]

Job	Customer	Inks	Parameters	Bar Codes	Template	Advanced
Name		Value				
Type of Proof	[EXA_DB_Myorders_ProofType]					

5. Save your workflow.

Step 7 - Test your workflow

In your **Pilot**, go to that (central) folder where your 3 XML files were written. Select these 3 input files and launch your workflow.



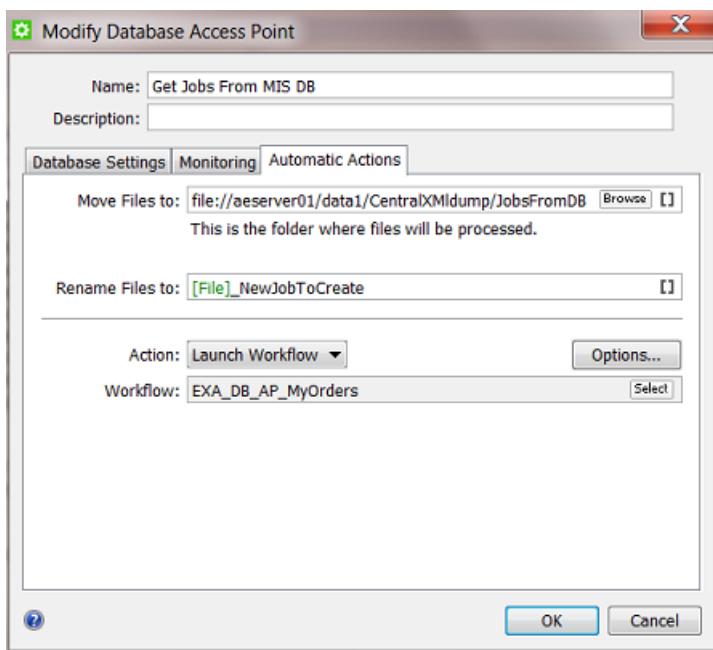
See how the **Data Splitter** receives 3 XML files and hands them over one by one to the **Create Job** ticket that creates these 3 Jobs:

Name	Order ID	Customer Name	Customer ID	Due Date
1235	1235	Graceland	6666	Dec 27, 2014
1236	1236	FruitCo	2712	Oct 11, 2014
1237	1237	FruitCo	2712	Dec 2, 2014

Tip: Also check again if the due dates are formatted correctly.

Step 8 - Add the workflow to the Access Point and test again

Open your **Database Access Point**. In **Action**, choose **Launch Workflow** and select your workflow. Click **OK** to confirm.



According to the set time interval, your **Database Access Point** will run and now also use its resulting XML files as input files for your workflow.

When your **Database Access Point** is selected, you see the workflow task appear:

File Name	Job Name	Task Type	Progress	Phase	State	Launched
MyOrders-20140804...		Workflow	100%		✓	8/4/14 10:38 AM

9.7.5 Launching a workflow with parameters from an XML

Summary

A workflow is automatically launched when the business system writes an XML file on a **Folder Access Point**. The **XML file describes what to do with what file**: it contains a full path to a new design file and describes many parameters that will be used as **Workflow Parameters** in the workflow that will be launched.

Such a concept is used by business systems that trigger a workflow on Automation Engine based on a new status. For example: when the business system notices that the status of a job order changes to 'Design Arrived', it decides to already start a prepress workflow on Automation Engine (the one that is typical for new designs from that customer). It is also common that such setups are not limited to only one status leading to one workflow. Some customers elaborate on this process and have 2 or 3 such links between a status in the business system and an automatically launched workflow on Automation Engine.

Tools used in this example:

- **SmartNames** including the **Xpath Builder**
- **Workflow Parameters**
- **A Folder Access Point**
- These tasks: **Preflight with PitStop, Create Report (ReportMaker), Zip, Send E-mail, Upload via FTP**
- These workflow controls: **Select Referenced File, Router, Data Collector**.

Note: In this example we do **not use** Automation Engine's **Job concept**.

Note: This example assumes you are already comfortable making advanced workflows.

Step 1 - Analyse the incoming XML

Tip: Click [this link](#) to find a ZIP with the sample XML file used in this example.

This is the type of XML file that we receive from the business system:

```

<?xml version="1.0" encoding="UTF-8" ?>
- <Prepress>
- <Workflow>
  <OrderNr>196570</OrderNr>
  <Description>Shrek Promo South America</Description>
  <CustID>1965</CustID>
  <CustName>ToyDream</CustName>
  <CustContact>Tony Sueno</CustContact>
  <CustContactEmail>tony@toydream.com</CustContactEmail>
  <Status>StartPrepress</Status>
  <JobType>Display</JobType>
  <Design>file:///Aserver01/Data1/demo_container/MIS-XML-WFL DATA/Design Arrival/Shrek.pdf</Design>
- <Preflight>
  <PreFlightYesNo>yes</PreFlightYesNo>
  <PreFlightProfile>CMYK Display</PreFlightProfile>
</Preflight>
  <ReportCardYesNo>yes</ReportCardYesNo>
  <SendViaFTPorEMAIL>Email</SendViaFTPorEMAIL>
  <FTPFolder>ToyDreamFTP</FTPFolder>
</Workflow>
</Prepress>

```

In the XML file, notice

- the job order number and some customer details
- a absolute full URL to the design file that just arrived. The design is named 'Shrek.pdf'.
- some decisions if the workflow needs preflighting or creation of a report card.

Step 2 - Create the Workflow

Now create the basic structure of our workflow. Add the steps as in the screen shot. We will add more detail later. It is possible that some of the connections can not be made yet.



Notice these main process steps:

- the workflow needs to run on the PDF, not on the incoming XML file that will land on the **Folder Access Point** (that we will create later in this example). That is why we will also use **Select referenced File** to select that 'Shrek.pdf' (see further).
- the workflow parameters will indicate whether the route to **Preflight with PitStop** and/or route to the **Create Report (ReportMaker)** needs to be taken.
- if the **Router** confirms that preflighting is needed, the step will do that and use a **Profile** defined in the XML.
- if the **Router** confirms that a design report card is needed, the **Create Report (ReportMaker)** will create that.
- the **Data Collector** collects the Preflight Report and/or the Design Report card, after which they are ZIP'ed together.

- the XML will indicate the next router if this ZIP needs to be sent to the customer via E-mail or via FTP.

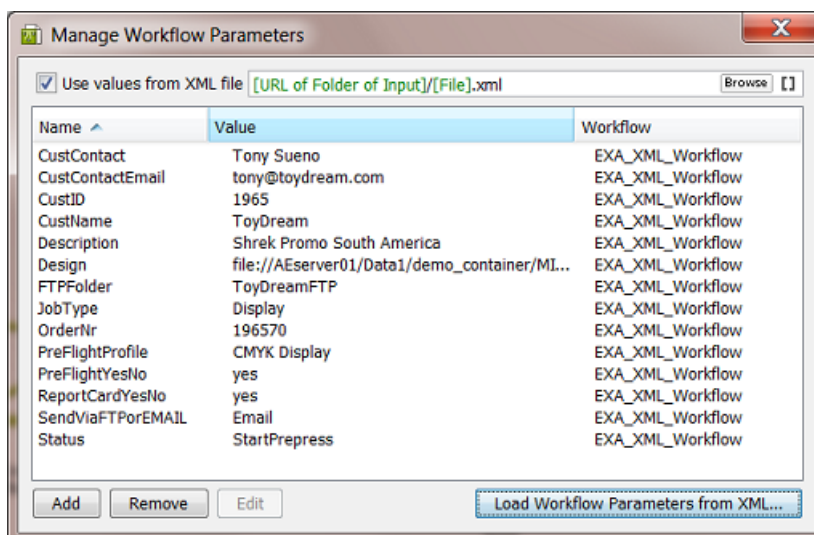
Save your workflow with a logical name, for example EXA_XML_Workflow.

Step 3 - Load Workflow Parameters from the XML

The XML files that the external system sends will always have the same elements, but with mostly different values. This is a typical case to use **Workflow Parameters**.

Note: Read more about **Workflow Parameters** in [Workflow Parameters](#).

- Open the workflow you created and in the menu go to **Advanced > Manage Workflow Parameters**.
- Check **Use values from XML file** and enter these SmartNames [URL of Folder of Input]/[File].xml (see also in below screen shot).
- Click on **Load Workflow Parameters from XML...** (below right). Browse to your XML example file and click **Open**. Choose to **Leave blank to select all elements below the root element**. All the XML element names will be listed, with their value from in this specific example XML:



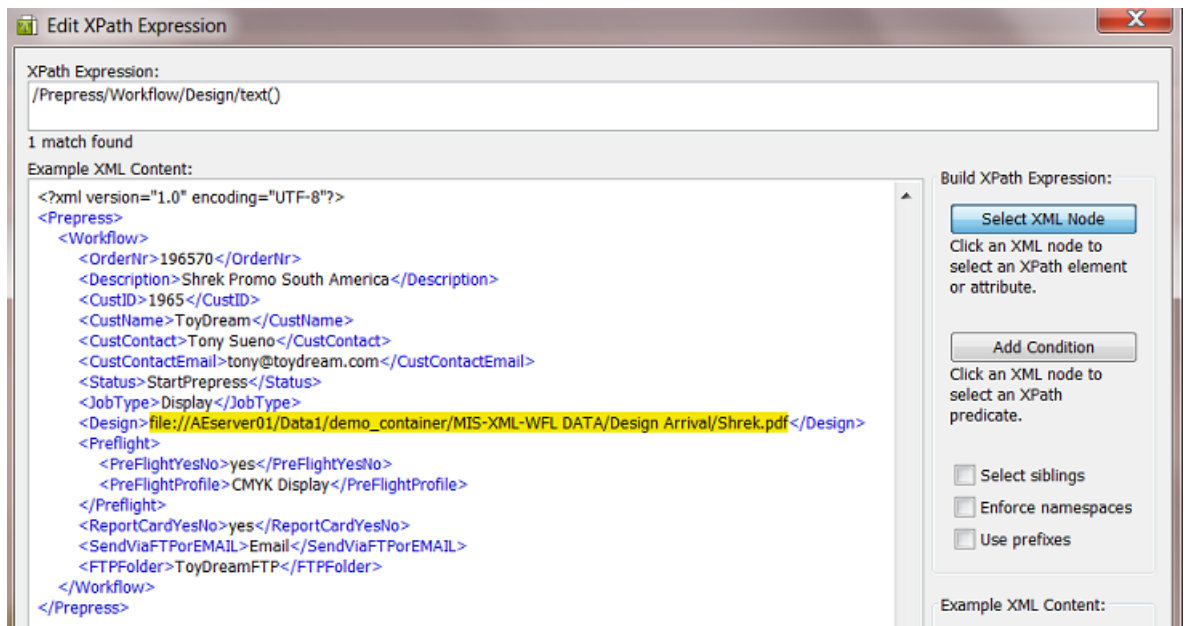
- Click on **X** to close this dialog.
These parameters are now **Workflow Parameters**. Every time this workflow is launched, they will be "loaded" before even starting the first workflow step.
- Save your workflow.

Step 4 - Select the Design file that is Referenced in the XML

The workflow needs to run on the design file, not on the XML file. The design file is referenced in the XML file.

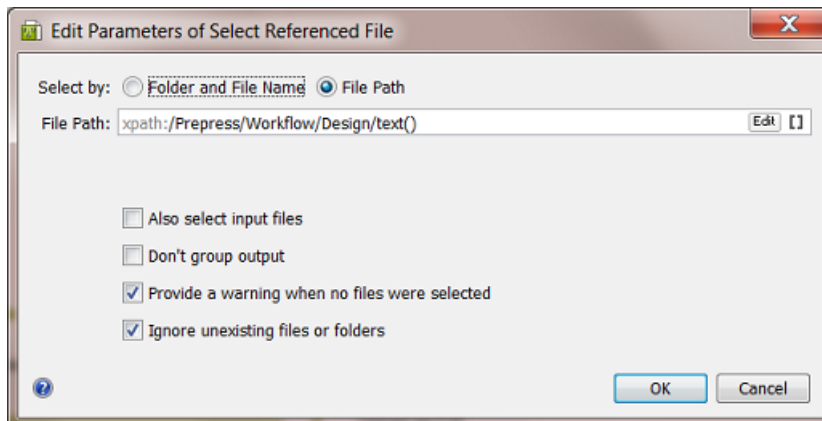
- In the **Pilot**, select the example input XML file and then open the workflow you created.
- In the workflow, open the step **Select Referenced File**.
- In **Select by:**, choose **File Path**.

4. In **File Path**, click on **Edit** to open the **Xpath Builder**.
5. In the **Xpath Builder**, the input XML file of your workflow will be shown as **Example XML Content**.



Select the path (the value of the *Design* element).
On top, the resulting **Xpath Expression** will be shown.

6. Click **OK** to close the **Xpath Builder**.
The **Xpath Expression** is now copied as **File Path**.
7. Check the option **Provide a warning when no files were selected**.



8. Click **OK** to confirm these settings.
9. Save your workflow.

Step 5 - Pick up the Workflow Parameters in your Workflow

We refer to the standard documentation to learn more on [making workflows](#) or on the tasks and workflow controls used in this example. We will here focus on what is specific in our example workflow.

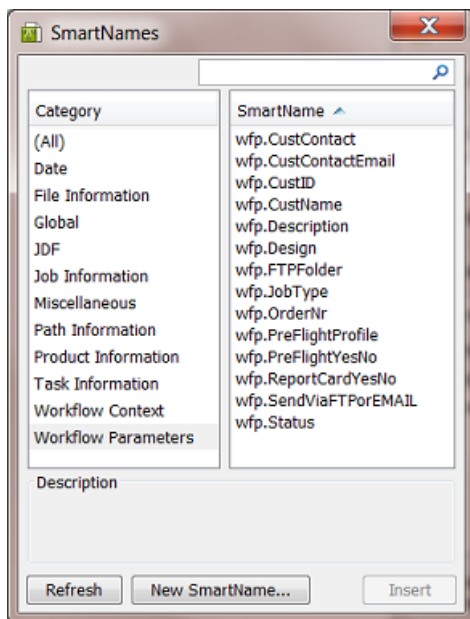
Our workflow does not use the **Job** concept and we (therefore) use **Workflow Parameters**:

- in the **3 Routers**
- in the **Preflight with PitStop** step
- in the **Create Report (ReportMaker)** step
- in the **Send E-mail** step
- in the **Upload via FTP** step.

In the 3 Routers

A value in the XML decides where the workflow will route to. Follow these steps for each **Router** (they are very similar in all 3 routers):

1. In the workflow, open the **Router**.
2. In **Route based on**, choose **SmartName value** from the list.
3. In **SmartName**, pick the appropriate one from the list, category '**Workflow Parameters**' ([WFP.]) and click **Insert**.



As can be seen in below screen shot, the 1st **Router** in the workflow needs the SmartName [wfp.PreFlightYesNo]. The 2nd **Router** needs [wfp.ReportCardYesNo]. The 3rd **Router** needs [wfp.SendViaFTPorEMAIL].

4. Close the list of **SmartNames** by clicking on the **X**.
5. Now set the relation between the SmartName value and the route to follow. See this example for the 1st **Router** deciding if preflight should be done:

Route based on: **SmartName Value**

Routing on SmartNames can seriously affect performance.
 ⚠ If the SmartName only contains job related parameters, it is advised to precede the router with a Data Collector.

SmartName: **[wfp.PreFlightYesNo]**

Treat as numeric value Case sensitive

SmartName Value	Route to
is equal to yes	yes
is equal to no	no
no match	Error

If a value has to be **equal**, it must literally be the same.
 If a value has to **match**, wildcards and [regular expressions](#) can be used.

OK Cancel

See this example for the 2nd **Router** deciding if a report card should be created:

SmartName: **[wfp.ReportCardYesNo]**

Treat as numeric value Case sensitive

SmartName Value	Route to
contains no	no
contains yes	yes
no match	Error

See this example for the 3rd **Router** deciding whether FTP or E-mail should be used:

SmartName: **[wfp.SendViaFTPorEMAIL]**

Treat as numeric value Case sensitive

SmartName Value	Route to
contains Email	Email
contains FTP	FTP
no match	Error

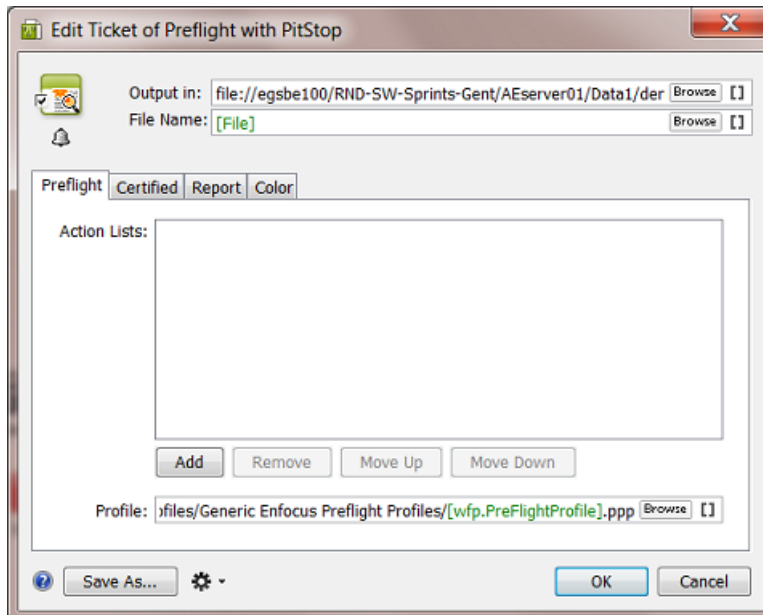
6. Click **OK** to confirm the Router settings and close the workflow step.
7. Save your workflow.

In the Preflight step

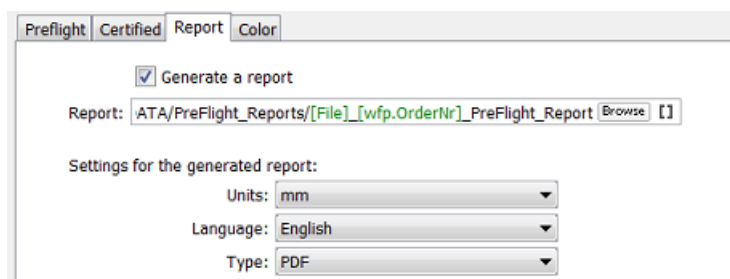
In our example, the XML file also decides which preflight profile needs to be used.

Note: See the full documentation of this workflow step in [Preflight with PitStop](#) on page 407.

1. In the workflow, open the step **Preflight with PitStop**.
2. Define the Output folder and name. Mind that this is not for the preflight report file.
3. In the **Preflight** tab, in **Profile**, insert the **Workflow Parameter SmartName** for the profile as part of the path.



4. In the **Report** tab, define the output folder and name for the preflight report that you will send to the customer.



5. Set any other options as wished.
6. Click **OK** to confirm the settings and close this workflow step.
7. Now drag the logical connections between the steps. The pin that represents the output of the preflight report should link to the **Data Collector**. That file will be sent to the customer, the input design file will not be sent.



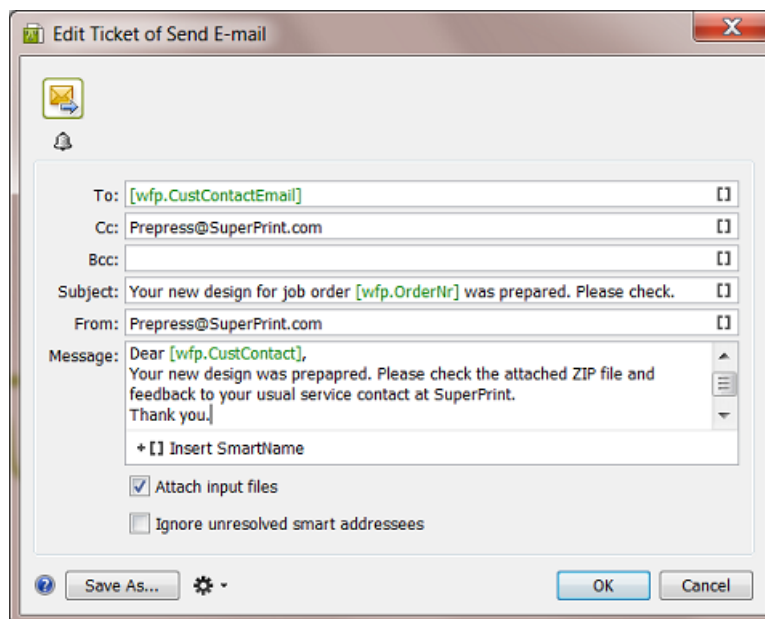
8. Save your workflow.

In the Send E-mail step

In our example, the XML file decides if the resulting files will be e-mailed or FTP'd to the customer.

Note: See the full documentation of this ticket in [Send E-mail](#) on page 471.

1. In the workflow, open the step **Send via E-mail**.
2. In **To**, add the **Workflow Parameter SmartName** for the E-mail of the customer contact for this job order.
3. In the **Subject** line, add the **Workflow Parameter SmartName** for the job order.
4. In **Message**, add the **Workflow Parameter SmartName** for the name of the customer contact.
5. Fill in extra fields as wished



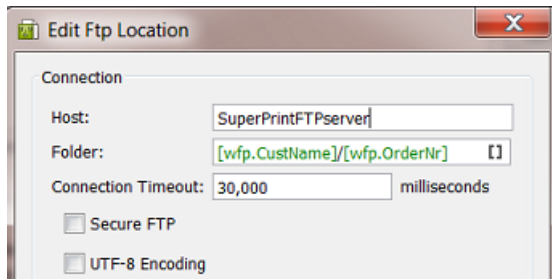
6. Click **OK** to confirm the settings and close this workflow step.
7. Save your workflow.

In the Upload via FTP step

In our example, the XML file decides if the resulting files will be e-mailed or FTP'd to the customer.

Note: See the full documentation of this ticket in [Upload via FTP](#) on page 519.

1. In the workflow, open the step **Upload via FTP**.
2. Enter the name of the **Host**.
3. In **Folder**, use the **Workflow Parameter SmartNames** to send the ZIP into a folder with a logical name. For example:

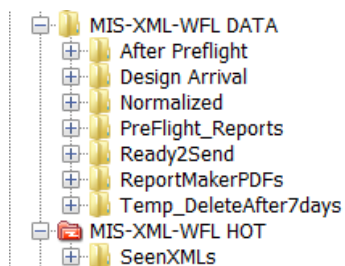


4. Set other settings as wished.
5. Click **OK** to confirm and close this dialog.
6. **TIP:** Optionally, you can send the customer contact an E-mail that his files are on the FTP server. Use the **Notification** tool of this workflow step to do this. Insert the **Workflow Parameter SmartName** for the E-mail and name, similar to how we did it in [the settings of the 'Send E-mail' step](#).
7. Click **OK** to confirm the settings and close this workflow step.
8. Save your workflow.

Step 6 - Data structure of this Workflow

This is a workflow that does not use the **Job** concept, so you can not store all your data in sub folders of a **Job Folder**.

The data structure that this example workflow creates looks like this:



- The folder `MIS-XML-WFL DATA` contains all the data that this workflow creates. See the logical names of the (automatically) created sub folders. These folder names were set in the '**Output in**' fields of the relevant workflow steps.
- The folder `MIS-XML-WFL HOT` will be the folder used for the **Folder Access Point**.

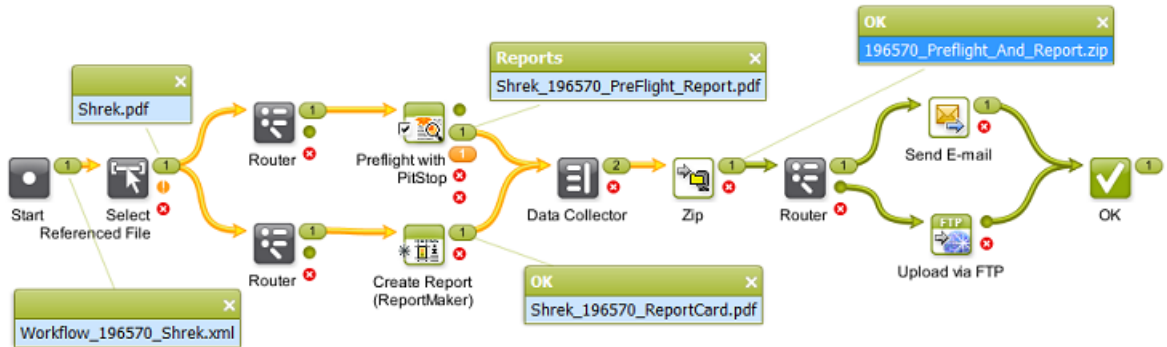
Step 7 - Test the Workflow

1. In the **Pilot**, select your input XML file.
2. Then select the workflow you created and open it (double click).

3.

Launch and Monitor: Press **CTRL-ALT-Enter** or Press **ALT** while clicking .

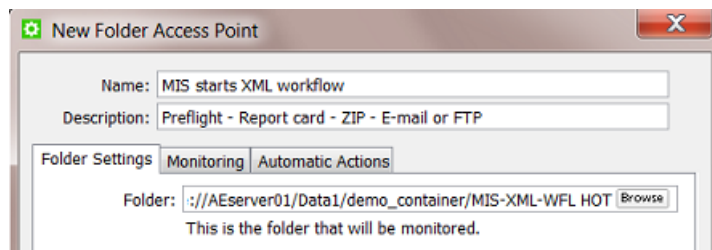
4. Drag the output file numbers so you can see these output files:



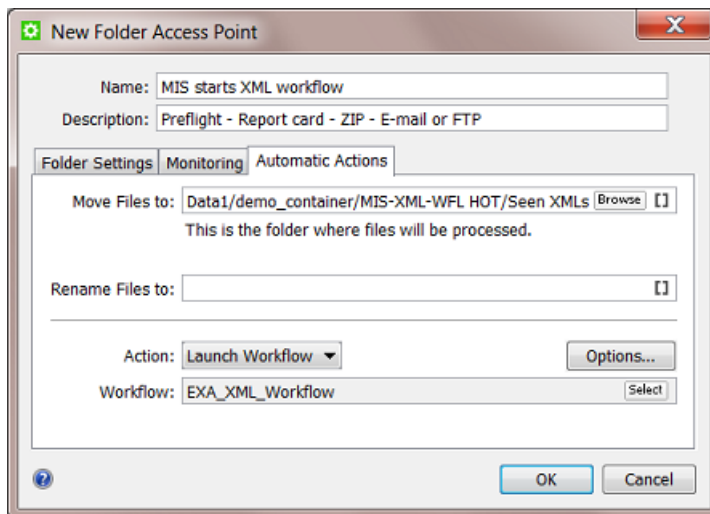
Step 8 - Create the Folder Access Point and Test Again

Note: If you are not familiar yet with **Access Points**, then first read [Access Points](#) on page 1236.

1. In the Pilot, go to the **Access Points** view.
2. Create a new **Access Point** and choose the type **Folder**.
3. Give it a logical **Name** and **Description**.
4. In **Folder Settings**, indicate the folder where the external system will write the XML files.



5. In **Monitoring**, set your preferred time interval.
6. In **Automatic Actions**, indicate the folder where the XML files should be moved to once they are processed.



7. Set the **Action** to **Launch Workflow** and select the workflow of this example.
8. Click **OK** to confirm and close.
9. With the new **Access Point** selected, wait until the time interval passes or press '**Scan now**' to see the resulting tasks appear:

Access Points					
Name	Active	Description	Type	C...	Action
Get Jobs From MIS DB	No		Database		Launch Workflow (EXA_DB_AP_MyOrde...
MIS starts XML workflow	Yes	Preflight - Report card - ZIP - E-mail or FTP	Folder		Launch Workflow (EXA_XML_Workflow)

Tasks							
File Name	Jo...	Task Type	Progress	Phase	State	Launched	
Workflow_196570_Shrek_Processed.xml		Workflow	100%		✓	8/7/14 2:06 PM	
Workflow_196570_Shrek_yes_yes_Processed.xml		Workflow	100%		!	8/7/14 2:01 PM	
Workflow_196570_Shrek_No_No_Processed.xml		Workflow	100%		!	8/7/14 2:01 PM	

The screen shot shows 2 extra tests where the Preflight ended with a warning.

10. Check the workflow in more detail (output files and folders, XMLs with different options).

Step 9 - Possible Workflow Extensions

Here are some ideas to make this workflow even more powerful:

Feedback new status to MIS

It would be logic to inform the (business) system that sent you the XML on the fact that the workflow has finished. Here are some tips on how to do that:

- Adding a **Create XML** step to the workflow and send an XML with the status on a folder that that system will scan.
- Adding a **Interact with Database** step to the workflow and have a database query write that status right back into that system's database.

Note: We can not use **Milestones** here because we are not working in a **Job** context.

Read more on such possibilities in [Automation Engine sending data Back to the External System](#) on page 1235

Adding the Preflight Status already in the Email or Report File name

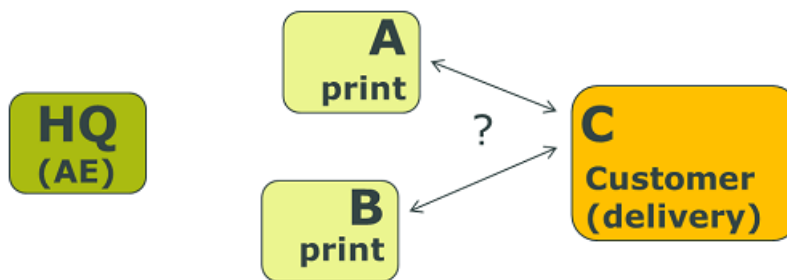
The customer might want to first check the designs where the Preflight showed an error or warning. To make this possible, you could extend the workflow to capture the Preflight status and use it as a SmartName. You could then insert that SmartName

- in the Email subject and or message
- in the FTP folder or name of the ZIP.

9.7.6 Getting info from a Web Service into a Workflow

Summary

Your company has printing plants in site A and site B. Automation Engine is in headquarters ('site HQ') where also the business system is. To enable a cheaper and faster delivery, your planning system asks Automation Engine which printing plant is closest to the customer, where the printed product should be shipped to ('site C').



Automation Engine asks the web service of GoogleMaps the distances and reports the result back to the planning system. The planning system sends its request to Automation Engine via an XML file. Automation Engine will also send a result back as an XML file.

Tools used in this example:

- **SmartNames** using **Xpath Builder**
- **Workflow Parameters**
- **Interact with Web Service** task
- **Map Data** task
- The **Data Collector** and **Sort** workflow control

Step 1 - Analyse the Input XML

Tip: Click [this link](#) to find a ZIP with the sample XML file used in this example.

In our example, this is the XML file that we get from the planning system (`RouteCalcRequest.xml`):

```

<?xml version="1.0"?>
- <RouteCalc>
  <OrderNr>234567</OrderNr>
  <CustomerID>6666</CustomerID>
  <CustomerName>Graceland</CustomerName>
  - <Delivery_Address>
    <Delivery_StreetAndNr>706 Union Avenue </Delivery_StreetAndNr>
    <Delivery_ZIP>TN 38103</Delivery_ZIP>
    <Delivery_City>Memphis</Delivery_City>
    <Delivery_StateCountry>USA</Delivery_StateCountry>
  </Delivery_Address>
  - <PrintSiteA_Address>
    <SiteA_StreetAndNr>2804 Opryland Drive</SiteA_StreetAndNr>
    <SiteA_ZIP>TN 37214</SiteA_ZIP>
    <SiteA_City>Nashville</SiteA_City>
    <SiteA_StateCountry>USA</SiteA_StateCountry>
  </PrintSiteA_Address>
  - <PrintSiteB_Address>
    <SiteB_StreetAndNr>726 Saint Peter Street</SiteB_StreetAndNr>
    <SiteB_ZIP>LA 70116</SiteB_ZIP>
    <SiteB_City>New Orleans</SiteB_City>
    <SiteB_StateCountry>USA</SiteB_StateCountry>
  </PrintSiteB_Address>
</RouteCalc>

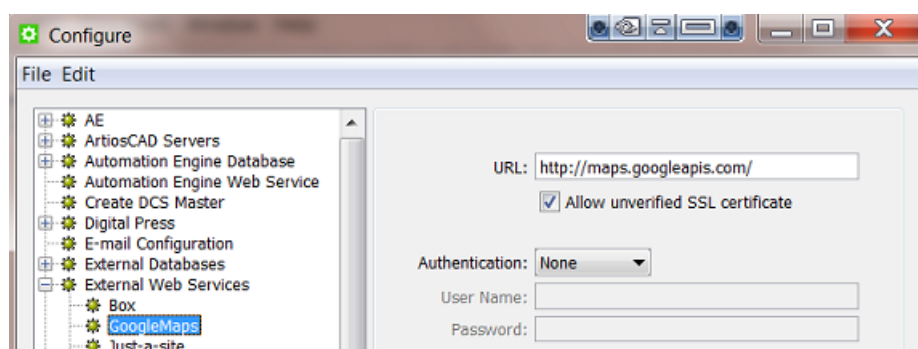
```

For job-order 234567, the planning system asks which printing site is closest to the delivery site in Memphis: A (Nashville) or B (New Orleans).

Note: This example contains address as they are written in the USA.

Step 2 - Configure the Web Service

As documented in [Interact with Web Service task](#) on page 1276, the GoogleMaps API needs to be added in **Configure > External Web Services**.



Step 3 - Create the workflow

Create a workflow with these steps. We will define some details later.

1. In Pilot, select the input XML file and create a workflow with these steps (the 2 'GoogleMaps' steps are **Interact with Web Service** steps.)



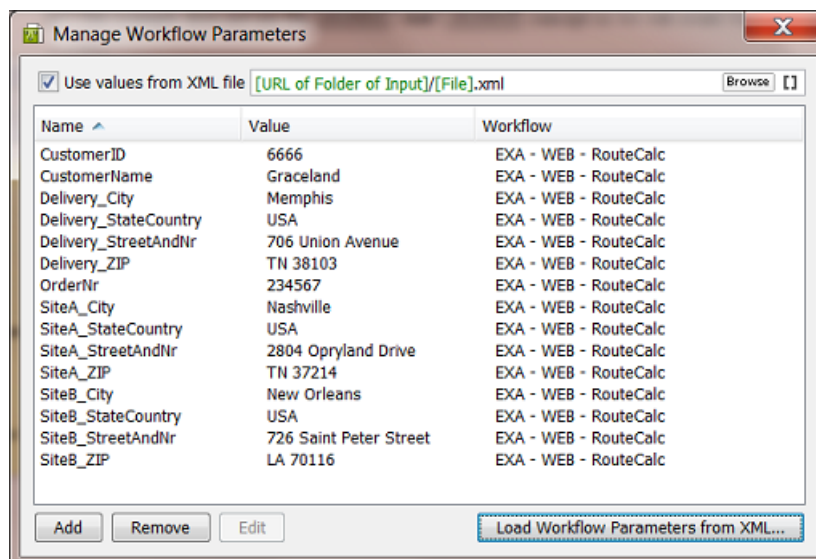
See how the workflow will use a **Map Data** step to join and partially redefine the outputs from the 2 interactions with *GoogleMaps*. Alternatively, you could make this workflow by using a **Map Data** step after each interaction with *GoogleMaps* step, and then later end it with a **Join XML** step.

2. Edit the names of the 2 **Interact with Web Service** steps to those as in the screen shot.
3. Save your workflow with a logical name. For example "EXA - WEB - RouteCalc".

Step 4 - Load Workflow Parameters from the XML

This example does not use the **Job** concept so we will create **Workflow Parameters** and pick those up throughout the workflow.

1. Open the workflow you created and in the menu go to **Advanced > Manage Workflow Parameters**.
2. Check **Use values from XML file** and enter this combination of **SmartNames**: [URL of Folder of Input]/[File].xml (see also in below screen shot).
3. Click on **Load Workflow Parameters from XML...** (below right). Browse to your XML example file and click **Open**. Choose to **Leave blank to select all elements below the root element**. All the XML element names will be listed, with their value from in this specific example XML:

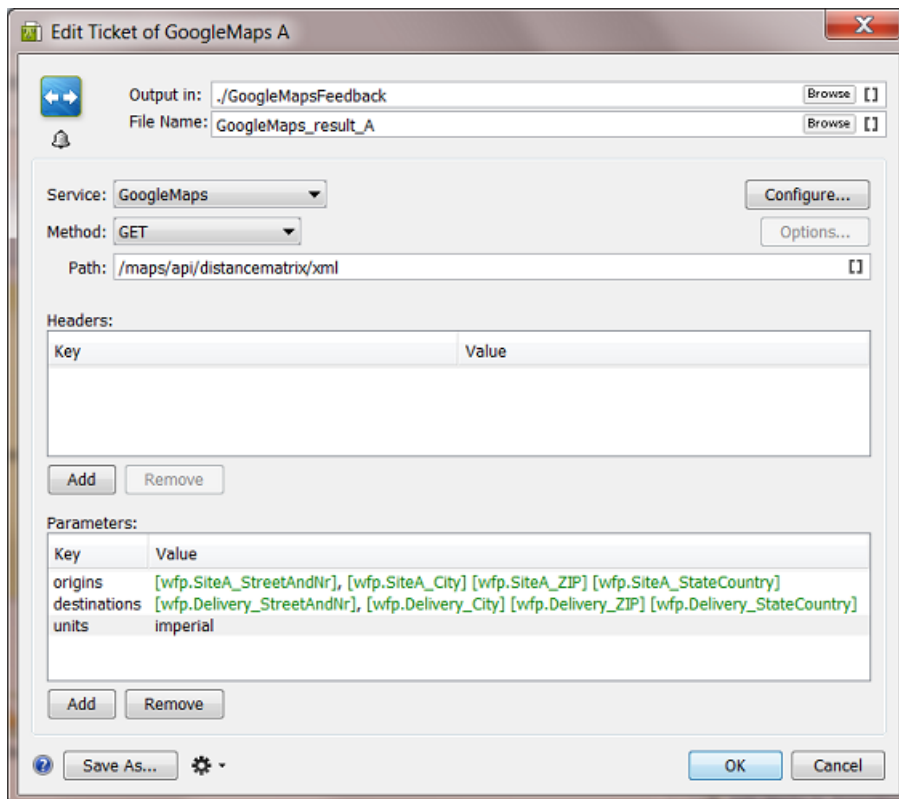


4. Click on **X** to close this dialog.
These parameters are now **Workflow Parameters**. Every time this workflow is launched, they will be "loaded" before even starting the first workflow step.
5. Save your workflow.

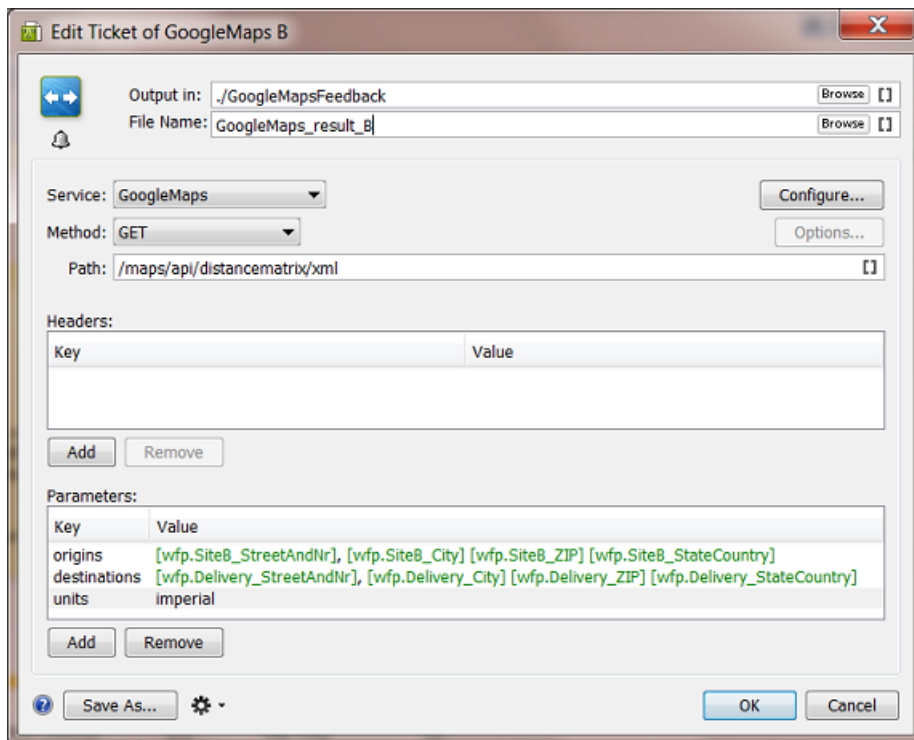
Step 5 - Define the 2 Interact with Web Service steps

The settings for these 2 steps are almost identical. We will explain the ones for step 'A' and at the end show the differences with those for 'B'.

1. Double-click the step 'GoogleMaps A' to open it.
2. Set logical values for the **Output in** and **File Name** fields.



3. Choose the web **Service** you configured for Google Maps and choose the **Method GET**.
4. Enter the **Path** as defined in the API documentation of Google Maps.
Because we are only interested in the distances, we here use Google Map's API 'distancematrix' and not the API 'directions'.
5. In **Parameters**, use **Workflow parameters** to define the API's key **origins** and **destinations**.
Unlike in Google Map's API 'directions', the keys 'origin' and 'destination' are both in plural here! The comma or space between the parts of the address needs to lead to a valid address specification for Google Maps.
6. The key **units** is optional. The default already is 'imperial' (miles, feet, inches ..)
7. Press **OK** to confirm and close this dialog.
8. Now double-click the other step 'GoogleMaps B' to open it.
9. Make sure the **File name** of the output XML file is different! And for the key **origins**, use the **Workflow parameters** that match the printing site 'B':



10. Press **OK** to confirm and close this dialog.

11. Save your workflow.

Step 6 - Settings to Collect and Sort the 2 Responses

It is important in this workflow to keep control over the order of the 2 XML files that Google Maps will return us. You do not want to risk mixing up the distances for site A or B.

Settings for the Data Collector step:

Keep the default settings: **Group all collected data**

Settings for the Sort step:

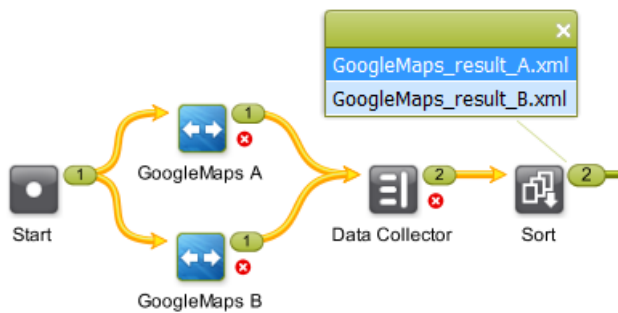
Keep the default settings: **Sort by file name in ascending order**

Save your workflow.

Step 7 - First test to get the Google Maps result files

To define the settings for the **Map Data** step, we need the result XML files from the interactions with Google Maps.

1. In **Pilot**, select the input XML file `RouteCalcRequest.xml` and launch your workflow `EXA - WEB - RouteCalc`. It will probably fail, because we haven't defined the **Map Data** step yet.
2. You should get these 2 files:



Right-click and choose '**Open**' to inspect them.

This is what Google Maps returns us when we ask the distance between site **A** and the delivery site:

```
<?xml version="1.0" encoding="UTF-8" ?>
- <DistanceMatrixResponse>
  <status>OK</status>
  <origin_address>2804 Opryland Drive, Nashville, TN 37214, USA</origin_address>
  <destination_address>706 Union Avenue, Memphis, TN 38103, USA</destination_address>
  - <row>
    - <element>
      <status>OK</status>
      - <duration>
        <value>11860</value>
        <text>3 hours 18 mins</text>
      </duration>
      - <distance>
        <value>358003</value>
        <text>222 mi</text>
      </distance>
    </element>
  </row>
</DistanceMatrixResponse>
```

And this is what Google Maps returns us when asking the distance between site **B** and the delivery site:

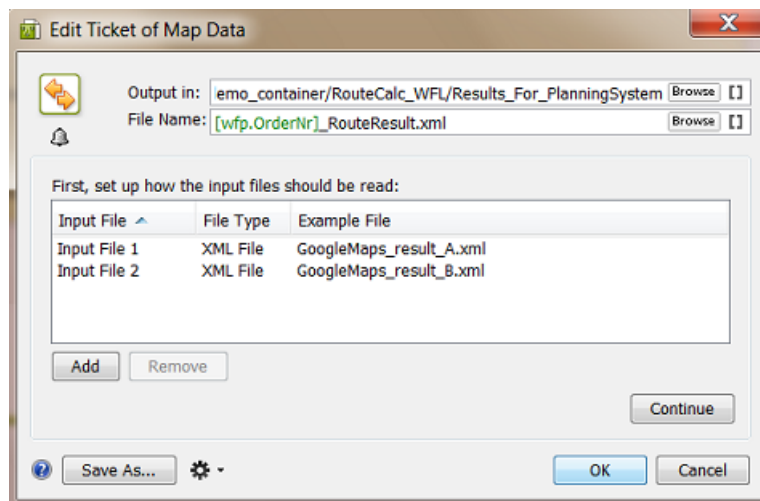
```
<?xml version="1.0" encoding="UTF-8" ?>
- <DistanceMatrixResponse>
  <status>OK</status>
  <origin_address>726 Saint Peter Street, New Orleans, LA 70116, USA</origin_address>
  <destination_address>706 Union Avenue, Memphis, TN 38103, USA</destination_address>
  - <row>
    - <element>
      <status>OK</status>
      - <duration>
        <value>20309</value>
        <text>5 hours 38 mins</text>
      </duration>
      - <distance>
        <value>634574</value>
        <text>394 mi</text>
      </distance>
    </element>
  </row>
</DistanceMatrixResponse>
```

Note: In Google Maps API, the default value unit for time is seconds and the default value unit for distance is meters.

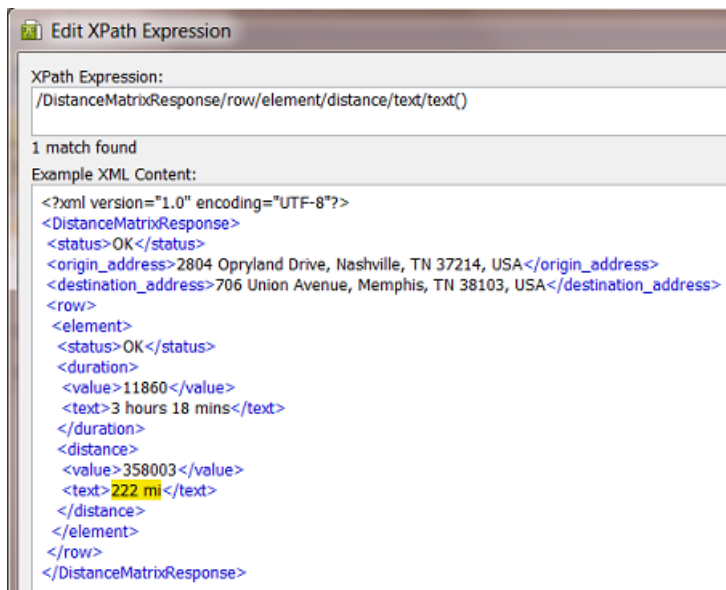
Step 8 - Settings in the Map Data step

Next thing to do in this workflow is merge this information into one XML file.

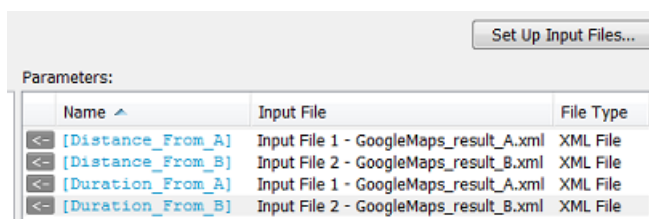
1. In your workflow (**edit mode!**), open the **Map Data** step.
You will see that it proposes the workflow's input file (`RouteCalcRequest.xml`) as example input file. This is **not** what we want here!
2. Select the line with that default example **Input File** and double-click to open it. Browse to the first input file we really want as example: `GoogleMaps_result_A.xml`, which we wrote in a sub folder `GoogleMapsFeedback`. Click **OK** to confirm.
3. Click **Add** and browse to the 2nd file we got back from Google Maps and add it as 2nd example **Input File**. Click **OK** to confirm.
4. This also a good time to set a logical folder and file name for this output XML file that we will feed back to the planning system. Let's assume that our output folder is a hot folder of that planning system. Use the **Workflow Parameter** `[wfp.OrderNr]` in the file name.



5. Click **Continue** to start defining the XML content that we want to send back.
6. We will first pick up some parameters from the example input files. On the right, in **Parameters**, click **Add**.
7. Click **Edit** to open the **Xpath Builder** and highlight the value of the `<text>` version of the element `<distance>` (222 mi).



8. Click **OK** to close. In the **Add Parameter** dialog, enter the name 'Distance_From_A' and click **OK** to close.
9. Do the same for the value of the <text> version of the element <duration> (3 hours 18 mins).
10. Then create the same 2 parameters but using the other example input file 'GoogleMaps_result_B.xml' ('394 mi' and '5 hours 38 mins').
You end up with these 4:



11. Now specify the output XML as shown in below screen shot. See how we use a mix of **Workflow Parameters** and **Parameters** from the example **Input Files**.



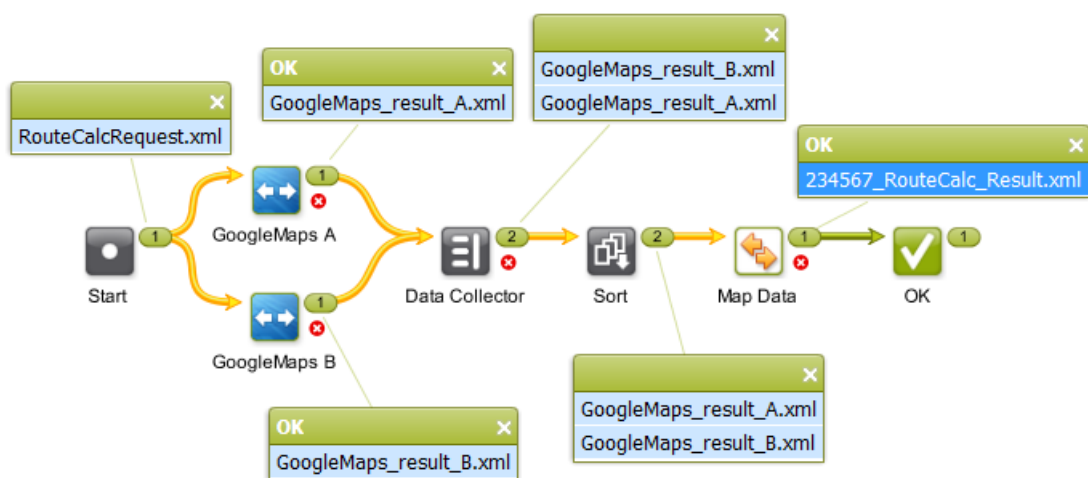
12. Click **OK** to close this workflow step.

13. Save your workflow.

Step 9 - Test the complete workflow

Our workflow is now complete (we assume that the place where the **Map Data** step writes the output XML is monitored by the planning system).

1. In **Pilot**, select the input XML file `RouteCalcRequest.xml` and launch your workflow 'EXA - WEB - RouteCalc'.
2. Check the flow and names of the output files. It should look like this:



3. Right-click and open the final output file '`2345657_RouteCalc_Result.xml`'. It should look like this:

```
<?xml version="1.0" encoding="UTF-8" ?>
- <RouteCalcResult>
  <OrderNr>234567</OrderNr>
  <CustomerID>6666</CustomerID>
  <CustomerName>Graceland</CustomerName>
  <Delivery_City>Memphis</Delivery_City>
  <PrintSiteA_City>Nashville</PrintSiteA_City>
  <PrintSiteA_Distance>222 mi</PrintSiteA_Distance>
  <PrintSiteA_Duration>3 hours 18 mins</PrintSiteA_Duration>
  <PrintSiteB_City>New Orleans</PrintSiteB_City>
  <PrintSiteB_Distance>394 mi</PrintSiteB_Distance>
  <PrintSiteB_Duration>5 hours 38 mins</PrintSiteB_Duration>
</RouteCalcResult>
```

The planning system will conclude that, for this job order, the printing site in Nashville is closest.

9.7.7 Updating a Product Status in an External System via Interact with Database

Summary

Our workflow includes changing the status of a **Product Part** to **Waiting for Approval**. Because this information is important for the service and sales staff, we will also inform the business system of this new status.

Note: In this example, we assume you are familiar with the **Products** tool. Read more on this tool in [Working with Products \(old\)](#).

Tools used in this example:

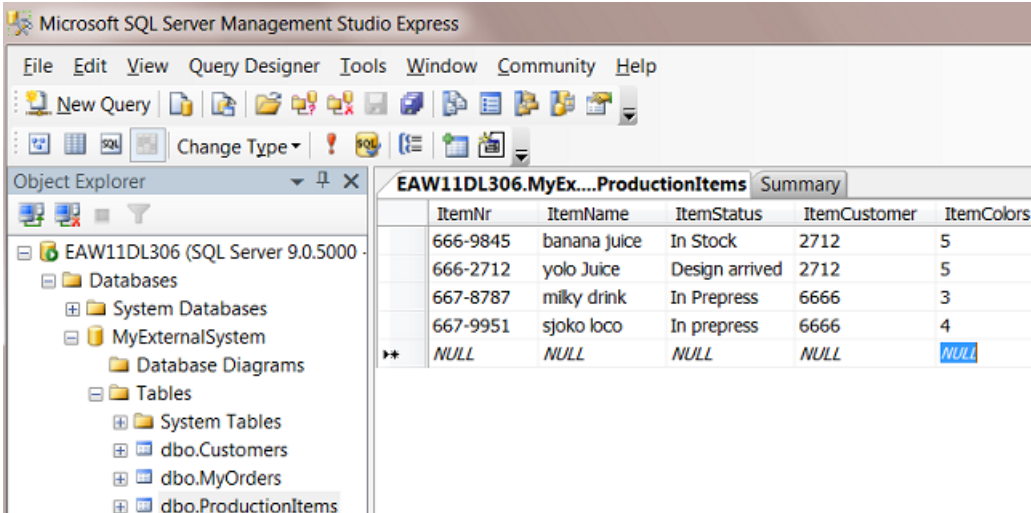
- **Products** tool
- **Manage Product Status** task
- **Interact with Database** task, to an external MS SQL Server
- **Mark File** and **Select Marked File** task
- **ReportMaker** task
- **Send E-mail** task

Step 1 - Analysis and Setup of the External Database

Tip: Click [this link](#) to find a ZIP with the sample database file used in this example.

Structure of the External Database Table

This is the database we will interact with:



The screenshot shows the Microsoft SQL Server Management Studio Express interface. The Object Explorer on the left displays the database structure for 'EAW11DL306 (SQL Server 9.0.5000)'. The 'MyExternalSystem' database is expanded, showing 'Database Diagrams', 'Tables', and 'System Tables'. The 'ProductionItems' table is selected. The main window displays the 'Summary' of the 'ProductionItems' table, showing the following data:

ItemNr	ItemName	ItemStatus	ItemCustomer	ItemColors
666-9845	banana juice	In Stock	2712	5
666-2712	yolo Juice	Design arrived	2712	5
667-8787	milky drink	In Prepress	6666	3
667-9951	sjoko loco	In prepress	6666	4
NULL	NULL	NULL	NULL	NULL

- The database software type is a MS SQL server.
- It runs on a server computer named EAW11DL306.

- The name of the database that we will access is `MyExternalSystem`.
- The name of the (opened) table is `ProductionItems`.
- The table contains the description of 4 production items.
- The item 'yolo juice' shows its 'ItemStatus' on 'Design arrived'. That is the database record that we will change in this example.

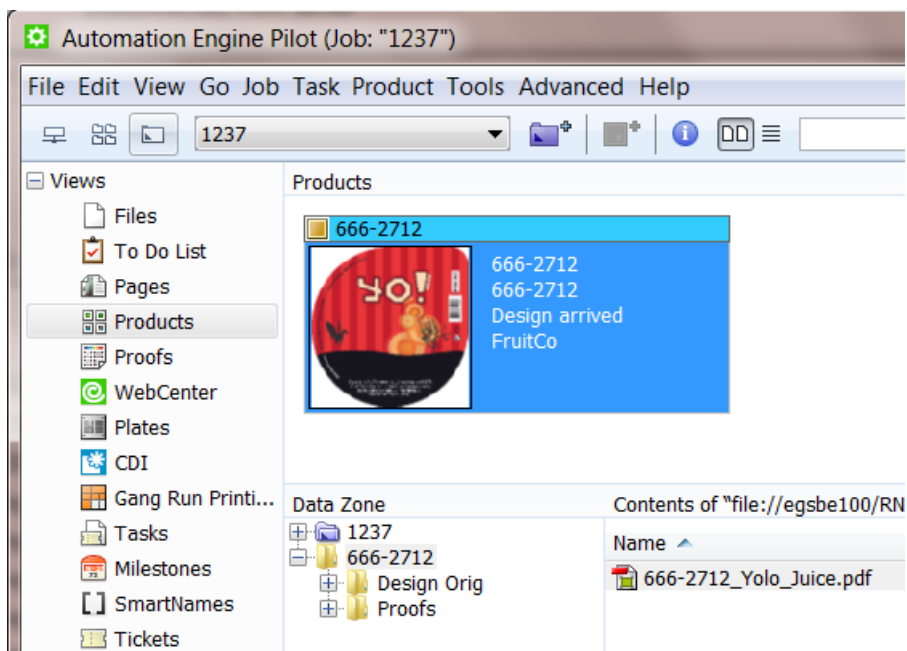
Configuring the Link to this External Database

Setting up the link to this **External Database** is identical to how we did it for a previous example in [Step 2 - Configure the Link to the External Database](#) on page 1311.

Step 2 - Initial Product Status and Job Setup

This is the starting configuration we need on Automation Engine:

- In **Pilot > Products > Manage Product Statuses**, create a **Status 'Design arrived'** with a blue color and create a status '**Waiting For Approval**' with a yellow color.
- Create a **Product 666-2712**. Give it the status 'Design arrived' and link it to the **Job 1237**.



- The **Job 1237** is for customer `FruitCo`.

Job Setup

Job Customer Inks Parameters Bar Codes Step&Repeat Imposition Automation Page Assignment Advanced

Customer ID: 2712

Customer Name: FruitCo

Customer Description: Yummy 'n fruity

Customer's Job Reference:

Customer Service Representative: Chantal DeLaService

E-mail: chantal@superprint.com

Step 3 - Create the Workflow

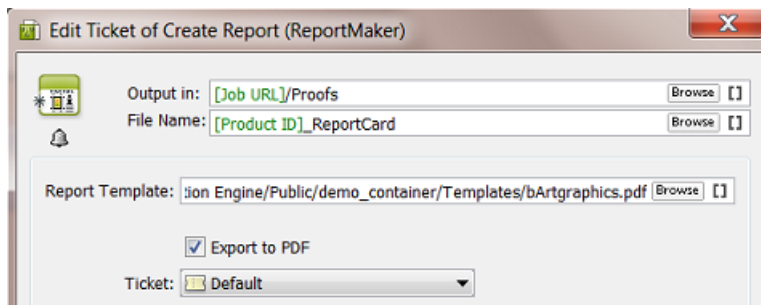
In our example, we assume that the design was already preflighted and made ready for print. Our workflow starts by creating a proof report and e-mailing it to the customer. It then changes the **Product** status to **Waiting For Approval** and also updates the external database of this new status.

1. Create this workflow and save it as 'EXA - PRODSTAT' :

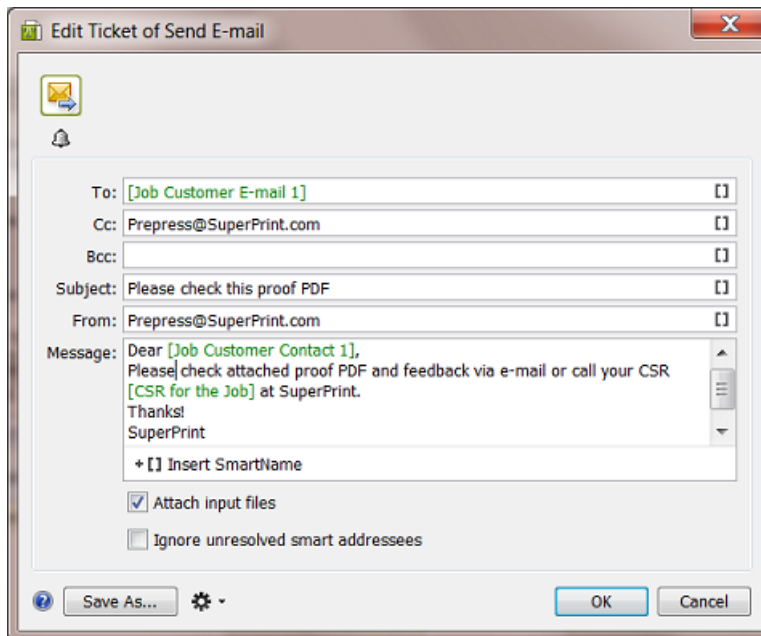


2. In the **Mark File** step, set the **tag** 'Product file'.
3. In the **Create Report** step, we advise these logical settings:

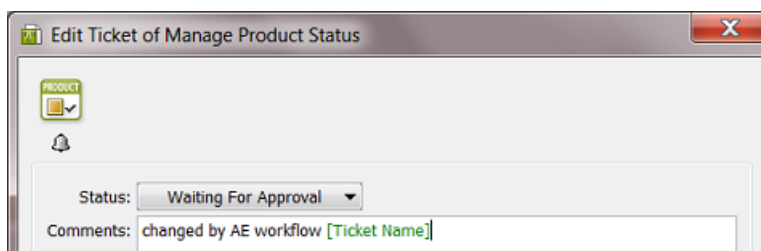
- choose any **Report Template**
- choose to **Export** to an extra PDF (we will e-mail the report PDF so you do not want the PDF to refer to external files!)
- choose a to **Output** the file somewhere in the **Job Folder** (because the report will probably contain Job specific info)
- choose a logical **File Name** which will make sense to the customer when he receives it via e-mail.



4. In the **Send E-mail** step, we advise these logical settings:



5. In the **Select Mark File** step, use 'Product file' as the tag to be selected.
 6. In the **Manage Product Status** step, select the **Status 'Waiting For Approval'**.

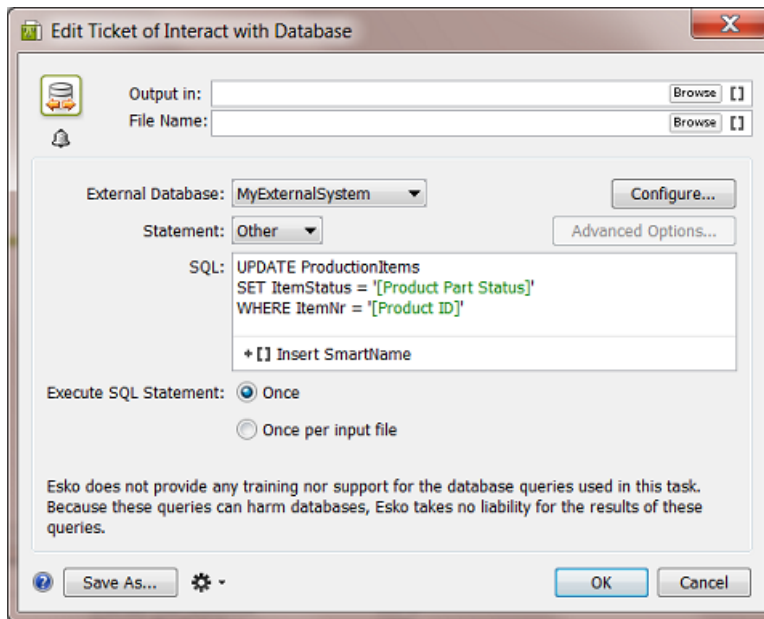


7. Save your workflow.

The **Interact with Database** step is described in the next step of this example.

Step 4 - Setup of the Interact with Database Step

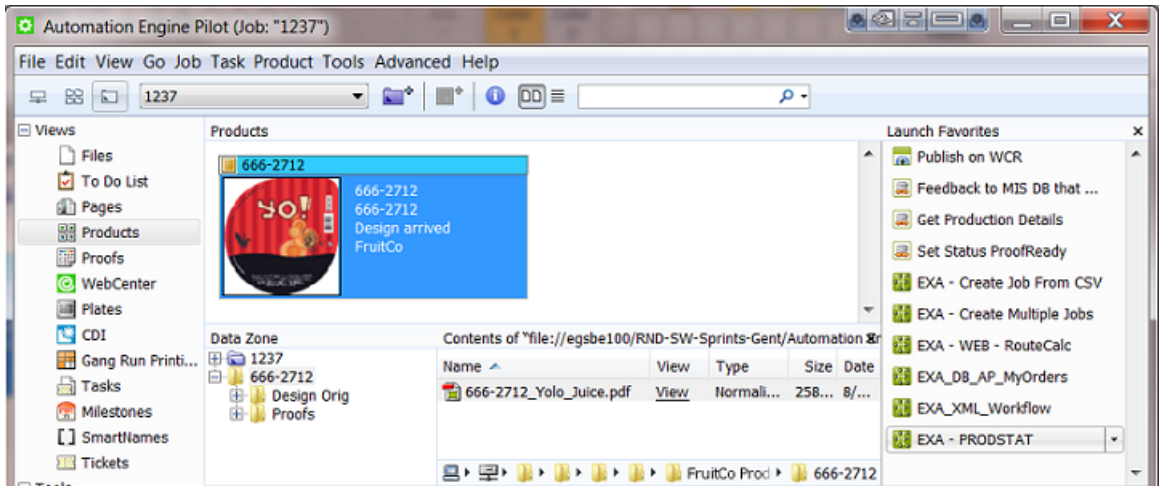
These are settings we need:



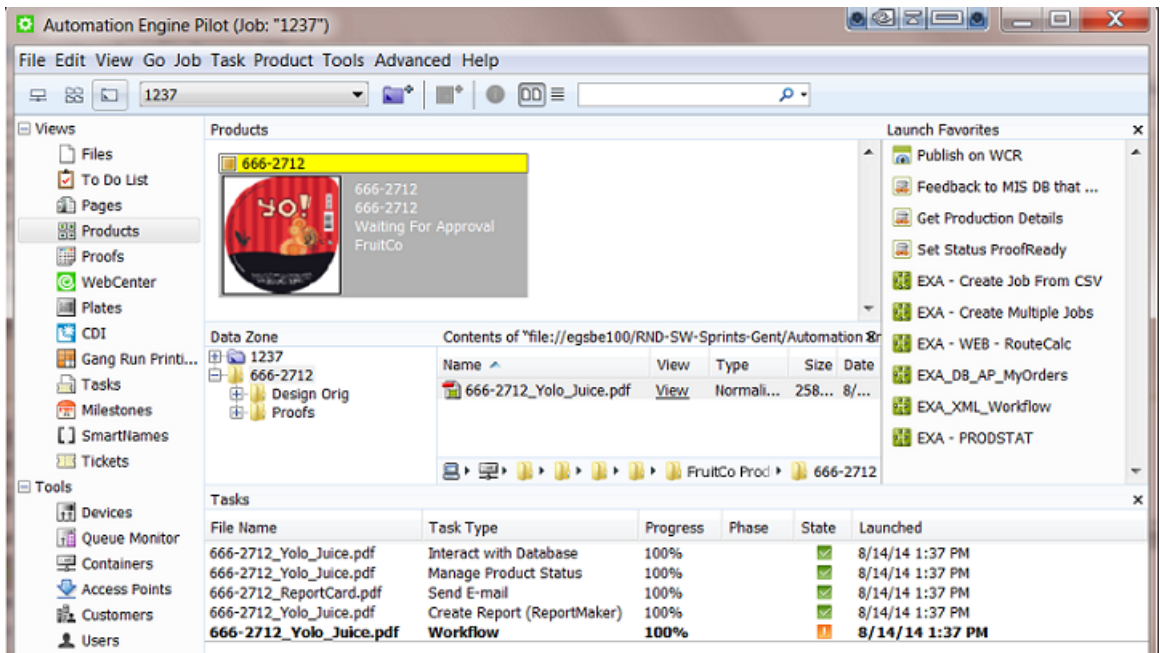
- **Output in** and **File Name**: because we are changing an existing database record, the type of **SQL** statement in this example is 'UPDATE'. As mentioned in [the documentation of this task](#), this task only outputs an XML file for 'SELECT' statements.
- **External Database**: select the one you set up for the database in this example.
- **Statement**: use **Other**.
- **SQL**: use the query as shown in the screen shot. The 2 **SmartNames** make this query work for all **Product (Parts)**.
- Click **OK** to close.
- Save your workflow.

Step 5 - Test the Workflow

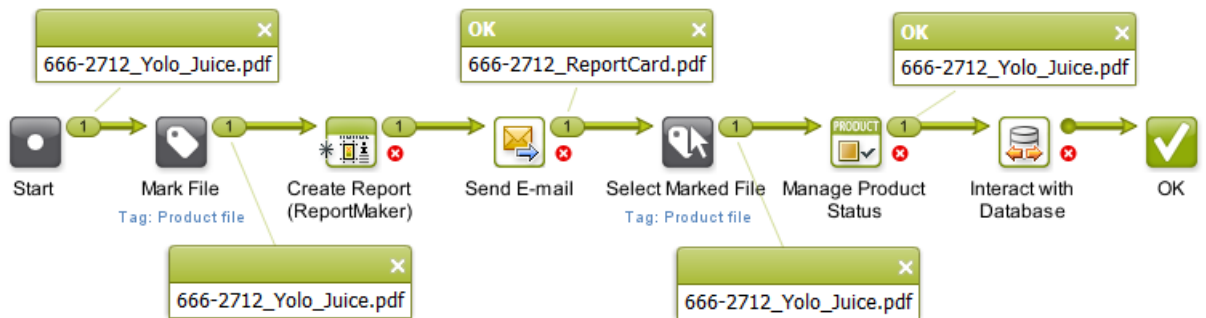
1. In **Pilot**, open the **Job** '1237'.
2. In **Products View**, select the linked in **Product** '666-2712'.
3. Launch the workflow 'EXA - PRODSTAT' that you created (using a **Favorite** or right-click **Launch with** or...).



4. This is the result in the Pilot ; notice the Product's change in status and color:



5. This is the resulting finished workflow:



6. This is how the external database should have changed ; notice the new status 'Waiting For Approval' for item '666-2712':

EAW11DL306.MyEx....ProductionItems					Summary
ItemNr	ItemName	ItemStatus	ItemCusto...	ItemColors	
666-9845	banana juice	In Stock	2712	5	
666-2712	yolo Juice	Waiting For Approval	2712	5	
667-8787	milky drink	In Prepress	6666	3	
667-9951	sjoko loco	In prepress	6666	4	

10. Integrating with WebCenter

10.1 Introduction

Automation Engine is not available outside its intranet (LAN). You should see it as an *in-house* system. Automation Engine can of course be connected with many other systems. And those systems can be web-enabled. Those systems will also have a different kind of users.

WebCenter is such a system. It is typically used by

- your colleagues working in places outside the intranet (in other plants, other countries, sales people on the road etc.)
- your customers or other business partners.

If you look at **what is communicated** between Automation Engine and WebCenter, we distinguish these types:

- exchanging files: exchanging files from Automation Engine to WebCenter and vice versa.
- exchanging information: exchanging the meta data from a Job to a Project and vice versa.
- sending direct commands to the WebCenter application (advanced setups).

Let's have a look at the **main integration concepts** between these 2 Esko products:

- WebCenter users want to send instructions or data to the Automation Engine users
- WebCenter workflows contain steps that are prepress actions. These are 'sub-contracted' to Automation Engine. For example: WebCenter can not pre-flight designs, so it will ask Automation Engine to do that.
- Automation Engine sending commands to WebCenter.

Publish and Approval

New or changed designs that were made print-ready need approval from the customer. Because the customer cannot directly access Automation Engine's Jobs or Containers, the prepress department will have Automation Engine publish the document to WebCenter. WebCenter then manages this approval process with the customer and delivers the (final) answer back to Automation Engine.

Using WebCenter as a centralized product management system

WebCenter can be used to store all the prepress data centrally. Local prepress sites then download to their Automation Engine the data that they need for prepress or production tasks. This means that the files in Automation Engine Containers are actually temporary data whereas the long term data storage is then on WebCenter.

Important: Automation Engine does not keep track of versions of files. WebCenter does and that is important when you use WebCenter as your central storage of prepress data.

Note: Mind that you can choose to do this for only a *part* of your prepress data. For example only for the files of some of your largest customers where prepress production is done in different countries.

Note: When Automation Engine users from different sites need data from the central WebCenter storage, then several types of access rights will define what data they can see and what they can do with it.

Note: The parameters of a Automation Engine Product (Part) can be pushed as document attributes on WebCenter (when the document is part of a BOM).

Apart from storing the prepress data centrally on the web, WebCenter can also use Automation Engine as background server of some prepress functionality. A WebCenter workflow where your customer is working with your sales rep could include a workflow step where Automation Engine is asked to preflight the PDFs.

Automation Engine sending commands to WebCenter

All the above is possible through a handful of tasks and features that handle such communication. Automation Engine can go a lot further than this and instruct WebCenter in much more detail. Some examples:

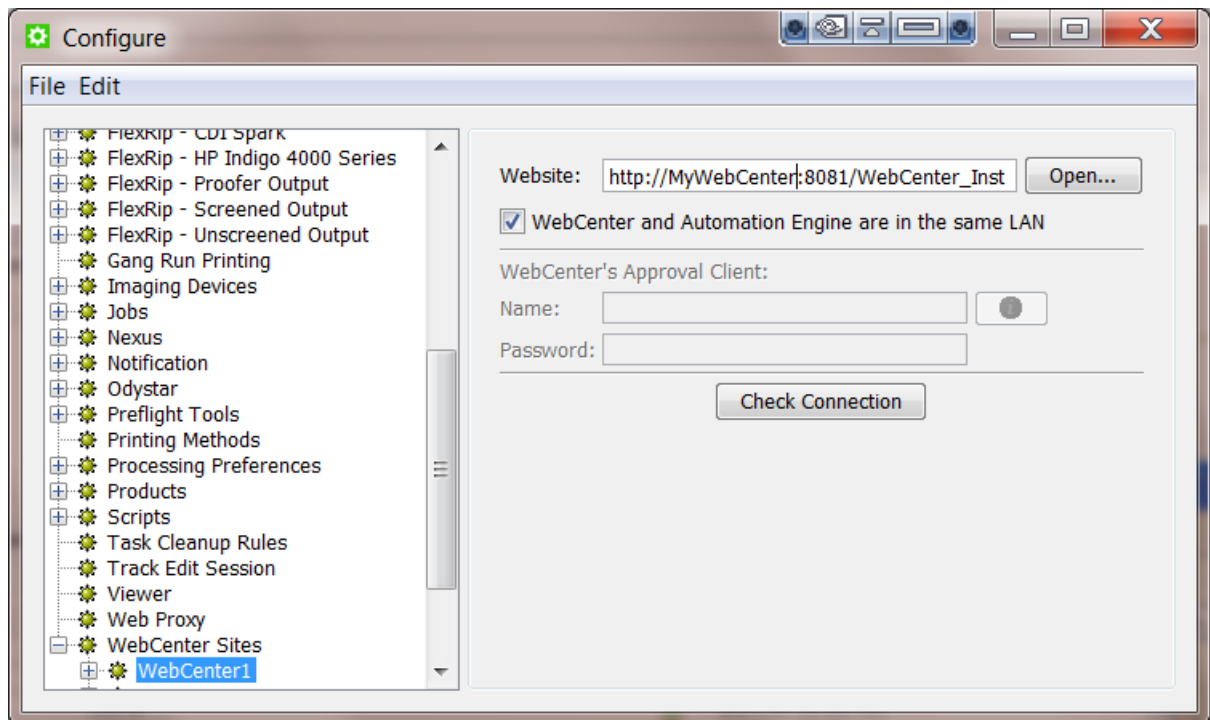
- Automation Engine managing WebCenter users, groups, customers (create, delete...).
- Automation Engine managing WebCenter documents (download, download metadata, delete, set attributes...).
- Automation Engine managing WebCenter projects (create, manage members, get lists, get document lists...).
- and much more.

This advanced technique is enabled by the [Integrate with WebCenter task](#) on page 1363 in Automation Engine. It offers over 50 different commands to send to WebCenter.

Note: Automation Engine can also be told by an external system to send these commands to WebCenter. For more on this topic, see the chapter on **Integrating with External Systems**.

10.2 WebCenter Sites

To configure links from Automation Engine to WebCenter sites, go to the item **WebCenter sites** in the **Configure** tool (**Pilot > Tools > Configure**).



Website:

Specify the web address of the WebCenter site. After you entered the WebCenter address, click **Open...** to check if the WebCenter site is up and running. If you are redirected to the login page, it means that the address is correct and that the site is up and running.

Tip: Be aware that this address is also the address that will be included in the notification e-mails when adding links to documents (approval cycle start e-mail). You should therefore make sure that the address can be accessed from the outside. Make sure to not use an internal address, but one that you want your users to use when *they* use WebCenter.

WebCenter and Automation Engine are in the same LAN

Check this toggle if this is true. This will most likely be the case. Un-check this toggle if WebCenter and Automation Engine are remote.

Note: How can you verify that WebCenter and Automation Engine are in the same LAN? WebCenter and Automation Engine are in the same LAN when you can connect from WebCenter's application server, this is the system where JBOSS is installed, to an Automation Engine Job Folder with the credentials of the user 'BGSystem'.

WebCenter's Approval Client:

If WebCenter and Automation Engine are not in the same LAN, you first need to create an Automation Engine **Approval Client** in WebCenter. This is necessary to give Automation Engine permissions to listen to the approval messages that are stored in WebCenter.

Tip: If the **Approval Client** does not exist yet, follow these steps:

1. In WebCenter, log in as `Administrator`.
2. Go to the **Admin** section. In **Configurations**, select **Workflow Approval Clients**.
3. Click the link "**New Workflow Approval Client**".
4. Enter a **User name** and a **Password** for the Automation Engine client. Mind that this is also the user name and password that have to be entered when configuring the WebCenter site in Automation Engine.

Fill out the client **Name** and **Password** and click the '**Check Connection**' button. This will check if that user can log on successfully.

The **Info** button displays the following information: *"In case Automation Engine and WebCenter are not in the same local area network (LAN), the retrieval of the approval results is secured by a name and password. The name and password specified in Automation Engine must match the name and password specified in WebCenter by the WebCenter administrator. When clicking the 'Check' button, the WebCenter version will be checked and the name and password will be validated by WebCenter."*

Check Connection

When you click this button, you should, after a few seconds, get a window saying "**Check OK**". The software version of that WebCenter will also be mentioned.

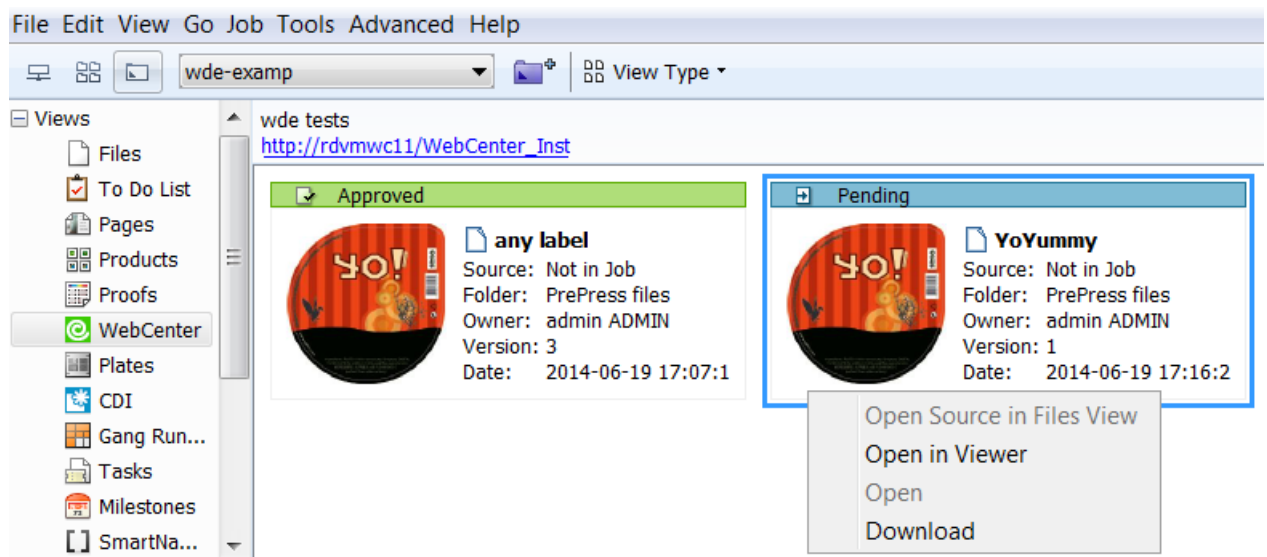
Note: The button will check the validity of the user name or password to connect to that WebCenter but **not** of your WebCenter's **Approval Client**.

Note: When you connect to a secured WebCenter site (for example one that uses HTTPS://), then you have to export its security certificate and import it on the Automation Engine server. If this is not done, publishing from Automation Engine to WebCenter will result in an error. Read more on how to do this in the chapter **Install SSL Certificates on the Workflow Production Server** of the WebCenter documentation.

Tip: After configuring a new WebCenter site, it is recommended to close your Pilot and start it again in order to see the updated list of WebCenter sites in your tickets.

10.3 The WebCenter View in Automation Engine Pilot

The Automation Engine **Pilot** offers a specific **WebCenter View** that will show you all documents in a WebCenter project without having to open a browser to look at the Project in WebCenter itself. The Pilot user can also see the documents' approval status in WebCenter and can also download those documents from this View.



How to set up

In the Pilot, in the list of Views, click on the WebCenter icon. If the WebCenter View is not configured yet, you will get a configuration page where you can fill out your WebCenter address and the user name/password that will be used to connect to WebCenter. If the WebCenter View is already configured, but you want to change the configuration, you can right-click in the window and you will get the configuration window again.

Tip: In the configuration, you need to specify the Project name. It is common practice to map Jobs to WebCenter Projects using the SmartNames [Job Name] / [Project Name]. On the configuration page, use the SmartName [Job Name] for the Project name.

Troubleshooting

If you do not see anything appear in your WebCenter View, right-click on the View canvas and select Configure. Copy the value of the URL, open a browser window and go to this URL. You should get the WebCenter login window. To log in, use the same credentials as in the configure page. If you can log in with the user name of the configure page, retype the password in the configure page that you just used for logging in to WebCenter. This ensures that the password is still up-to-date. Try again to connect to WebCenter. If it still does not work, verify that the Project name you try to access really exists in WebCenter. Ensure that the SmartNames resolve correctly to the project in WebCenter.

10.4 WebCenter Integration Tasks

10.4.1 Create or Modify WebCenter Project task

Concept

This task allows to create a WebCenter Project from Automation Engine. If the project already exists, it will be updated with new or changed parameters.

Tip: Some users want a WebCenter Project for every new Job in Automation Engine.

- When you manually create a Job, you can (in the advanced tab) select this task to be launched when you create the Job.
- In a workflow, you can insert this task as a step after the **Create Job** task.

The WebCenter Project that you create is not limited to having the attributes that you defined here in this task. The Project can also get extra attributes from the selected Project Template or the Customer Company.

Requirements

To use this task successfully, you need

- **in Automation Engine:**
 - to be in a Job context, a Job with minimally an **Order ID**.
 - the user name and password of a WebCenter **Project Manager**.
- **in WebCenter:**
 - a **Project template** with **attributes matching the Automation Engine Job parameters (Order ID, SubOrder ID, and - if used - any custom parameter defined in the Parameters tab of the Job setup)**.

Note: Make sure the WebCenter attributes have the correct **type** for your Job parameter data. For example, when passing on a *date* parameter, the corresponding WebCenter attribute should have the type 'Date and Time'. When passing on a *numeric* value, the WebCenter attribute should have the type 'Numeric' ('Float' or 'Integer'). For more information about attributes, see the WebCenter documentation.



Attention: If the WebCenter attributes do not have the exact same name as the Automation Engine Job parameters, the Job parameters will not be passed on.

- Optionally, check the WebCenter naming of **Customer Company** and **Customer Location**.

Task Options

Most fields allow both direct input and/or using SmartNames.

- **WebCenter Site:** Select it from the list of Configured WebCenter sites. Optionally, Click **Configure...** to access the Configure panel
- **Project:**The name of the WebCenter Project
- **Project Template:** Define if you want the new Project to be based on a Project template.

Note: Make sure you select a Project template with attributes that match the Automation Engine Job parameters

- **Project Status, Customer Company, Customer Location:** Click the refresh button to get an updated list.
- **Project Description:** If you want, define a description for this Project.
- **Project Manager User Name:** Define which WebCenter user will be the manager of this Project.

Note:

- This user must already exist in WebCenter.
- This *can* be the same user as the one you use to contact WebCenter with (the user and password fields further below). Also, that user that contacts WebCenter in this ticket must have **at least as much visibility** in WebCenter as the Project Manager user that you enter here.

- **User Name:** This the user that you contact WebCenter with and who will create or modify the Project.

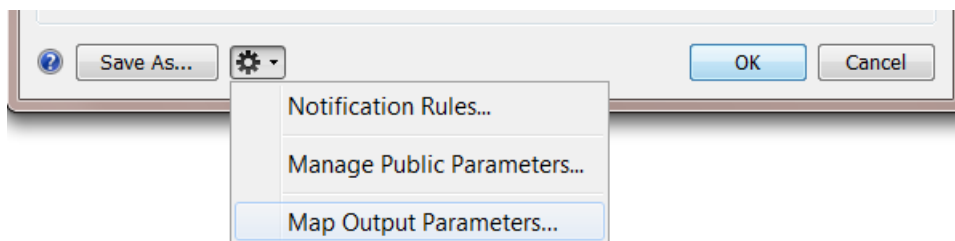
Note: The user you enter here must be a Project Manager in WebCenter to be able to *create* Projects.

- **Password:**
 - Encrypted. Type it in. It will not be shown visible.
 - Use SmartNames. Some customers prefer a SmartName because they do not want Automation Engine users to know this password or because they want the latest changes in that password to be automatically picked up here.

Modifying workflow parameters using this task

As mentioned in [Modifying Workflow Parameters During a Workflow](#) on page 1075, this task can also be used to modify **Workflow Parameters**.

This task allows you to modify the value of a specific Workflow Parameter with the value of the Project ID of the created or modified WebCenter Project. The mapping of the Project ID to a specific Workflow Parameter can be done using the **Map Output Parameters** dialog.



10.4.2 Publish on WebCenter task

The **Publish on WebCenter** task allows to upload files from Automation Engine to a WebCenter Project.

Optionally, you can request WebCenter to start an approval cycle on those files. In that case, this **Publish on WebCenter** task will remain in a waiting status until the WebCenter sends back the final answer of that cycle ('approved' or 'rejected').

Note: An approval cycle in WebCenter ends when all approval members have answered or when someone used a 'forced approve' or 'forced reject' answer.

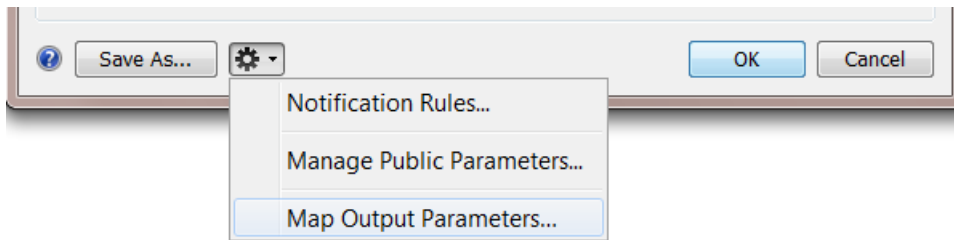
This ticket has three tabs:

- **Destination:** Define in which WebCenter Project you want to publish the file.
- **Publish:** Define what version of the file you want to make public on WebCenter
- **Approval:** Define if and how you want to start an approval cycle.

Modifying workflow parameters using this task

As mentioned in [Modifying Workflow Parameters During a Workflow](#) on page 1075, this task can also be used to modify **Workflow Parameters**.

This task allows you to modify the value of a specific Workflow Parameter with the value of the **Document Version ID** of the document you publish in WebCenter. The mapping of the Document Version ID to a specific Workflow Parameter can be done using the **Map Output Parameters** Dialog.



Destination

Note: You can use SmartNames in most of the fields.

Destination | Publish | Approval

WebCenter Site: WebCenter1 Configure...

Project: [Job Name] []

Folder: PrePress files []

Project Template: [Job Customer Name]_template []

Document Name: [File] []

Document Description: []

Document Attributes:

Name ^	Value
[Bar code from XMP]	bar code
[Trim Box Height]	Vsize

Add Remove

User Name: Elvis []

Password: Encrypted Use SmartNames

.....

- **WebCenter Site:** Select the WebCenter site you want to publish the document to. Click the **Configure...** button if you want to add or change WebCenter sites.
- **Project:** Define in which WebCenter Project you want to publish the document.

Tip: Use [JobName] as Project name to map your Jobs in Automation Engine to the Projects in WebCenter.

Important: If the WebCenter Project does not yet exist at publish time, it will be created. Mind that the user that does the publish needs to be **Project Manager** in WebCenter to be able to create a Project in WebCenter.

Remember: The recommended method to create a WebCenter Project is by using the **Create or Modify WebCenter Project** task.

Note: When this task creates the Project, the Project manager will be :

- the Project Manager of the Template group if the WebCenter user executing the task is part of that group.
- or the WebCenter user executing this task.

- **Folder:** Define in which folder of that Project that the documents should be stored.

Note: If the folder does not exist yet, it will be created if the user is a **Project Manager** in WebCenter. With user we here mean the WebCenter user that you define further below in this ticket.

- **Project Template:** If the task needs to create the WebCenter Project, define here which existing Project template should be used.

Note: Some WebCenter users do not see all Project templates. Templates are filtered according the user's visibility. When you define a user's visibility, you can list a set of companies and/or a set of groups. The effect on this is the following:

1. If a template has a customer defined, this customer (= company), needs to be in the list of companies that the user can see. If the company is not in the list of visible companies of this user, the template is not visible for this user. This means that this user can not use this template to create a new Project.
2. When the template has group members, WebCenter checks whether the user can see all the group members. As soon as there is one group in the template that is not visible to this the user, the template is considered as 'not visible' for this user. Again, the user will not be able to create a new project based on this template.

- **Document Name:** This field allows you to set a specific name of the published document. If left blank, it takes the original name of the document.

Note: If the document already exists, the task will create a new version of the document.

Note: If the document already exists in a different folder of that Project, the task will fail.

- **Document Description:** This description will be available in WebCenter as **Version Comment**.
- **Document Attributes:** This allows to set specific attributes for the document in WebCenter. These may have different names than what you call them in Automation Engine. For example you can map some metadata from the PDF to a Document Attribute.
- **User Name and Password:** Enter the name and password of the WebCenter user that will connect to WebCenter and upload the document.

Note: The task will fail in error if this WebCenter user lacks upload rights, even if the Project exists.

Publish

In the Publish tab you decide what version of the input file you want to publish and/or if you want to prepare special raster data for the WebCenter Viewer.

Allow Viewing and Annotating in WebCenter

If the **Prepare for viewing and annotating** toggle is checked, Automation Engine will rasterize the graphic into files readable for the (WebCenter) **Viewer**. These rasterized data will then be published to WebCenter.

Note: This toggle can not be turned on when WebCenter and Automation Engine are not in the same LAN.

Note: If this option is not on, WebCenter will create the raster data for viewing.

The **Resolution for non-images** (linework, text ...) will be the maximum resolution you will have in the WebCenter Viewer. You can choose between::

- **Optimal:** Automation Engine calculates the best resolution, mainly based on the size of the file. It is **recommended** to select this option. For small files, it will rasterize with a high resolution. For large files, it will be a lower resolution. This balance is to ensure you always have a fast viewing experience in the Viewer, both when you first open the file and when you start zooming in.
- **Custom:** Allows you to specify a resolution yourself. If you set a high resolution, it will create large rasterized files. Set a low resolution to have a lower quality view of the document in the WebCenter Viewer.

Important: For images, the maximum available resolution is used.

Anti-aliasing: This is relevant for the linework objects in your design that need pre-processing and for which extra images are generated (CTs). Anti-Aliasing smoothes the staircase effect you get in these CTs. Text will be more readable, also when zooming in.

Automatic grouping of input file into digital film sets: In a workflow, the publish task can be used right after a RIP task that creates digital films (for example LEN or TIFF files). In such cases, you may want to publish these files as one document. Enable this option to bundle that set of digital film files and publish them as one document to WebCenter. Each digital film will make up one separation of the document.

Note: When digital film files were rasterized as one group, the Viewer tool will also recognize them as 1 set of separations.

Allow Downloading in WebCenter

Make available for download:

- When this option is **off**, the **Publish on WebCenter** task will not publish the task's input file to WebCenter but only the 'prepared' rasterized files. Choose this option when you do not want any form of your production files to be downloaded by WebCenter users.
- When this option is **on**, you can choose which files you will make available for download on WebCenter (if the user has such permissions):
- **Input files.** For example: your task input file is a production PDF, and that is the PDF you want the WebCenter users to have access to. Or you already had made low resolution PDFs, and you choose to publish those instead.
- **PDF files for proofing.** In this case, you choose not to publish the input file but a **PDF file for proofing** that does not exist yet. Select from the list of PDF export tickets which kind of PDF you wish to create and then publish to the WebCenter user. This way, you offer the outside world a 'PDF proxy' file instead of your real input production file.



Attention: Always choose this option when the task input files are not self-contained. For example when your PDF refers to artwork, fonts or other PDFs.

Note: If you select **PDF files for proofing** the viewing quality will be limited to the resolution of the PDF files for viewing.

When disabling both **Prepare for viewing and annotating** and **Make available for download**, a warning box is displayed indicating that no data will be published. You need to select at least one type of data to publish.

Approval

The Approval tab allows you to select if and when an approval cycle should be started on WebCenter.

Important: Make sure you already have an approval cycle set up in WebCenter! This approval cycle setup will indicate who to invite, what their role is and more. Mind that the approval cycle can be part of a Project template that you here selected.

Start the Approval Cycle and wait for approval. Select this if you want the approval cycle to start automatically as soon as the file is uploaded to WebCenter. For this, you need to have an approval rule specified in the Project or Project folder where you upload the document to. By default, WebCenter will apply the folder approval setup if it exists. If not, the Project approval setup will be applied.

Due Date: If the WebCenter user that you use to log in with ('Destination' tab) has the **Change Approval Setup** permission in WebCenter, you can also set the approval due date.

Note: The due date specified here will take priority over the due date specified in WebCenter.

You can choose one of the following:

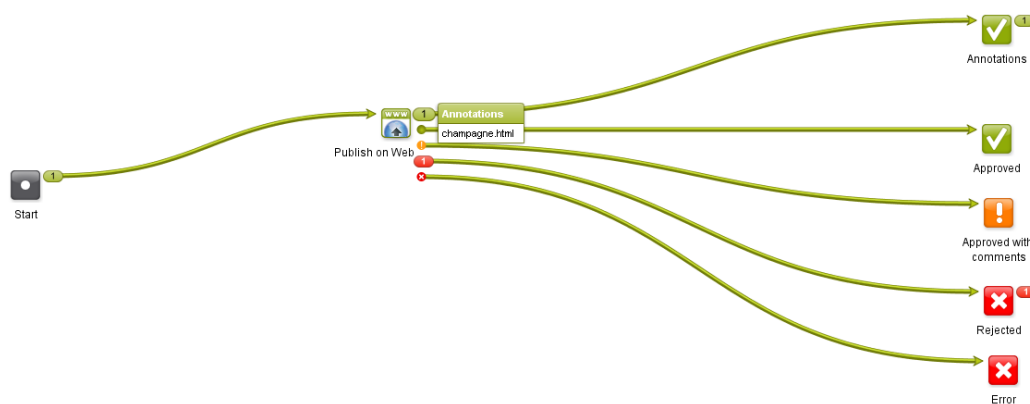
Choice	Explanation
Immediate	This will set the due date to the time that the task was launched.
Today	This will set the due date to today, at a user specified time.
Tomorrow	This will set the due date to tomorrow at a user specified time.
In ... hours	This will set the due date to a specified number hours from the time the task was launched.
In ... days	This will set the due date to a specified number of days from the time the task was launched and

Choice	Explanation
<p>On</p> <p>Job due date</p> <p>Select SmartName...</p>	<p>at a specified time. A SmartName can be used to set the number of days.</p> <p>For example "in 2 days at 5:43 PM" is the same as "the day after tomorrow at 5:43 PM".</p> <p>Use the calendar to set the due date and time. This will set the due date to the one specified in the Job setup.</p> <p>If you use a SmartName, it will also be added to the above menu items.</p> <p>Note: The SmartName should be conform to the ISO 8601 date string format. For example 2014-12-27T11:30:00+02:00</p>

Annotations:

If any annotations were made during the WebCenter approval cycle, the **Publish on WebCenter** task generates a annotations summary. At the end of the approval cycle, an HTML file will be generated. This HTML file redirects the Automation Engine user directly to the WebCenter annotations page in a browser. He will be asked to login to WebCenter.

You can here decide the folder and name where this 'shortcut' HTML will be written.



What if you get conditional approvals?

Documents that are approved conditionally in WebCenter will go through the **Approved with a Condition** output of the **Publish on Web** task (standard of the type 'warning').



It is good practice to use a **Wait for Action (Checkpoint)** task for those documents, so the Automation Engine operator can fix them manually. For more details about conditional approval, see the WebCenter documentation.

What if you have multiple input files?

When your **Publish on WebCenter** task has multiple input files, and your ticket requests an approval cycle, then an approval cycle will be started for each of those input files. When the **Publish on WebCenter** task has received an answer for all files, it will then show their specific status in the task's output pins: which one was approved or rejected etc...

When you use the **Split** task in front of the **Publish on WebCenter** task, then Automation Engine will start up a Publish task **per** input file.

Troubleshooting tip

If you get errors in this publish task and the log file does not give you an indication what might go wrong, then try to do the same steps interactively inside WebCenter, using the user account that you had used in your Publish ticket. If the error is due to permissions, you will quickly figure out what goes wrong.

Publishing Automation Engine Products to WebCenter



Attention: This page addresses customers who use the Automation Engine **Products** tool. Learn more about this tool in the chapter [Products](#) on page 876.

When you publish an Automation Engine **Product** to WebCenter, the Product Properties and **Product (Part) Parameters** will be taken as **WebCenter Attributes**. If there are existing WebCenter attributes with corresponding names, their values will be updated during publishing.



Attention: For this to work, WebCenter needs to be enabled to work with **BOM - Bill Of Materials**. This is a setting in WebCenter -> Admin -> General Preferences.

The following Product (Part) properties and Product (Part) Parameters are synchronized with WebCenter Attributes (see further how to rename these internal names to more readable ones):

Automation Engine Properties/Parameter	WebCenter Attribute (Internal name)
Product Customer Reference	ae.p.cusref
Product Custom Field 1	ae.p.category1
Product Custom Field 2	ae.p.category2
Product Custom Field 3	ae.p.category3
Product Internal ID	ae.p.id
Product ID	ae.p.prodid
Product Name	ae.p.name
Product Description	ae.p.desc

Automation Engine Properties/Parameter	WebCenter Attribute (Internal name)
Product Part Internal ID	ae.ppid
Part Default Angle	ae.pp.defaultangle
Part Default Dotshape	ae.pp.defaultdotshape
Part URL	ae.pp.url
Part Default Ruling	ae.pp.defaultruling
Part Preferred InkBook	ae.pp.preferredinkbook
Part Preferred InkBook UI	ae.pp.preferredinkbookui
Part Internal ID	ae.pp.id
Part Custom Field 3	ae.pp.category3
Part Custom Field 2	ae.pp.category2
Part File	ae.pp.file
Part Status	ae.pp.status
Part Status Nickname	ae.pp.status_nickname
Part Status User	ae.pp.status_user
Part Status Notes	ae.pp.status_notes
Part Status Date	ae.pp.status_date
Part Folder	ae.pp.folder
Part Custom Field 1	ae.pp.category1
Part Name	ae.pp.name
Part Inks	ae.pp.inks
Part Barcodes	ae.pp.barcodes
Product Customer Name	ae.c.name
Product Customer Description	ae.c.desc
Product Customer ID	ae.c.id
MyProductParameter1	ae.p.par.MyProductParameter1
MyProductPartParameter2	ae.pp.par.MyProductPartParameter1

When a **Product** or **Product Part** is published, the corresponding WebCenter Documents are automatically added to the **Bill Of Materials** of the WebCenter Project. The **Product Amount** for the Job (if given) is updated as the **Count** property of the **Bill of Materials**.

Tip: You can rename the internal name of WebCenter Attributes to a more readable name by first adding it to a **Category**:

Edit Category AETestCategory

Name:

Current View: **Default** [Add Creation View](#) [Add Header View](#) [Add View](#) [Edit Terminology](#)

Alternating Background
 Show also attribute types and restricted set types
 Keep session alive (avoid that people get logged off due to time out)

[Remove Grid Layout](#)

Attributes:

- \$dl\$Dieline
- \$dl\$DocumentLink1
- \$dl\$DocumentLinkRich
- \$dl\$Material
- \$dl\$POA
- \$dl\$Search
- *tema_Rich_Text
- 00 Change Category
- 00 Change Order Number
- 00 Change Type
- 00 Description

[HeaderBar]

ae.p.name
ae.c.desc
ae.p.prodId

Details for selected cell:

Attribute	ae.p.name
Attribute Type:	Text
Label:	Text: Product Name
Label:	Image:
Label:	Image:
Position:	Left
Span:	1

Learn more about this in the PDF named **Connection between Automation Engine and WebCenter 14** that you can download from <http://help.esko.com/products/WebCenter>.

10.4.3 Download WebCenter Document task

This task allows to download documents from WebCenter into an Automation Engine Container (or Job Folder). Optionally, downloaded documents including their document attributes can be immediately added to the Products database.

The following **SmartNames** are useful in a **Download WebCenter Document** task:

- **WebCenter Document Folder:** The folder in the WebCenter project that holds the document that you are downloading.
- **WebCenter Document Name:** The name of the WebCenter document that you are downloading.
- **WebCenter Document Original File name:** The original name of the file that was uploaded to WebCenter and there made a document.

- **Output in** and **File name** The above screen-shot shows how the default ticket proposes you a logic place and name for the file you are downloading.
- **Site:** The WebCenter site to download the document from. Click **Configure...** to add or change WebCenter site configurations (enabled only for administrator users).
- **Project Name:** The name of the WebCenter Project that you want to download a document from.
- **Document Name:** The name of the WebCenter document that you want to download.
- **Lock document:** Check this option if you want to lock the document in WebCenter. When the document is locked, other WebCenter users will not be allowed to upload new versions. Only the WebCenter user that locked the document can publish a new version and/or unlock the document.
- **User Name** and **Password:** of the WebCenter user that downloads and/or locks the document.
- **Create or update Job Product:** Check this option if you want to add the downloaded document as an Automation Engine Product for the Job you are working in. If the file is already a Product, the WebCenter document attributes are merged with the existing Product (Part) parameters. If the attribute already exists as Product (Part) parameter, that Product (Part) parameter's value is *updated*. If the attribute was not a Product (Part) parameter yet, it is *added* as a Product parameter.

Important: In order to use the option **Create or update Job Product**, the task must run in a Job context!

10.5 Advanced WebCenter Integration

10.5.1 Integrate with WebCenter task

Concept

WebCenter can be used

- interactively, in a browser, by a WebCenter user
- by an external system, through a special connection setup
- by Automation Engine, by using the **Integrate with WebCenter** task.

This Automation Engine task goes a lot further than the ones we documented earlier: those other tasks focus on exchanging files and creating projects in WebCenter. This **Integrate with WebCenter** task enables Automation Engine to perform more than 50 different actions on WebCenter. Some of these actions are very elaborate commands or requests for information and some even change configurations.

Note: The technology behind this task has historically often been referred to as the "WebCenter SDK"



Attention: This task requires advanced knowledge of both WebCenter and Automation Engine.

Task Options

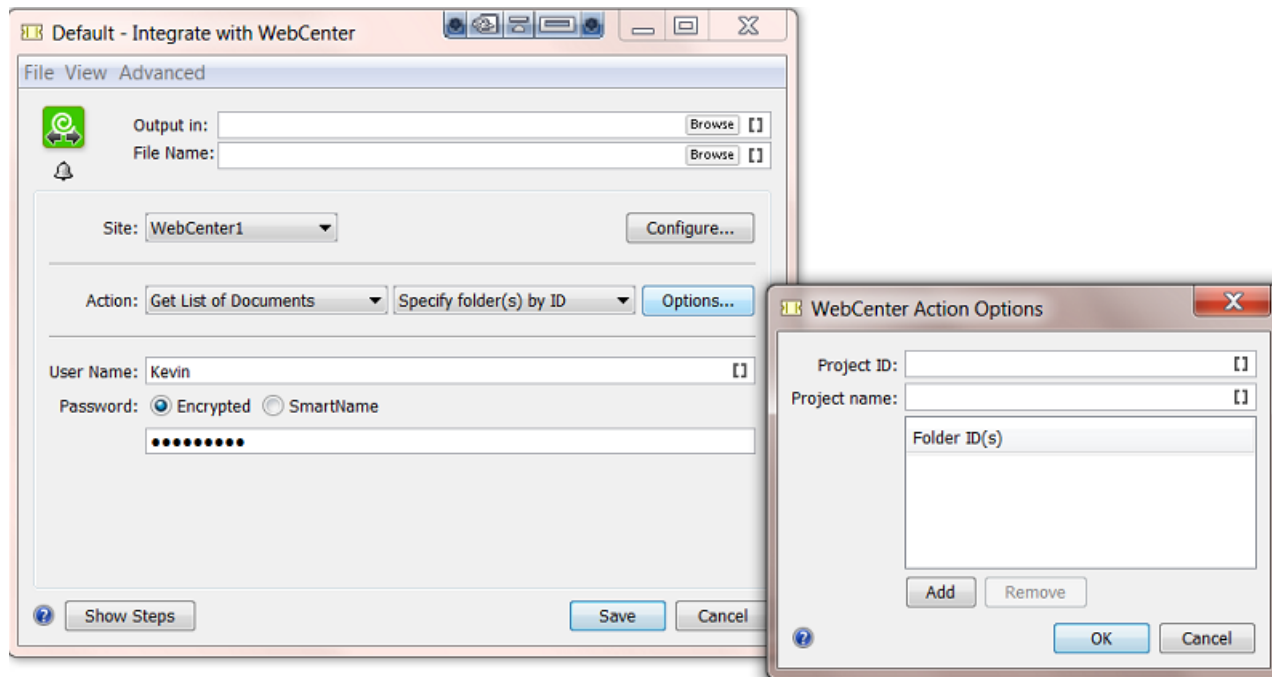
The screenshot shows a configuration window titled "Default - Integrate with WebCenter". It has three tabs: "File", "View", and "Advanced". The "Advanced" tab is active. The window contains the following elements:

- Output in:** A text field with a "Browse" button.
- File Name:** A text field with a "Browse" button.
- Site:** A dropdown menu currently showing "WebCenter1" and a "Configure..." button.
- Action:** A dropdown menu with "None" selected. The menu is open, showing a list of actions: "None", "Approve Document", "Create Company", "Create Group", "Create Project", "Create User", "Create WebCenter Task", and "Delete WebCenter Task".
- User Name:** A text field.
- Password:** A text field.
- Options...:** A button next to the Action dropdown.
- Show Steps:** A button with a question mark icon.
- Save** and **Cancel** buttons at the bottom right.

- **Output in** and **File name:** Depending on the **Action** chosen below, these fields might be relevant or not. If the action triggers this task to have an output file, then you can here define their folder and name.
- **Site:** Select the WebCenter site from the list. Click **Configure...** to add or change WebCenter site configurations (enabled only for administrator users).

Note: A connection will be made with the WebCenter that you select. It will not just test the connection but will also get the list of possible commands from that specific WebCenter server. That list then becomes the list of **Actions** (for this WebCenter).

- **Action:** Select the action. When the selected action offers options, the **Options...** button will become active. The options dialog is specific per action. Here is an example:



See how selecting the action "Get List of Documents" automatically offers a sub-type of action to choose, here "Specify folder(s) by ID" (or "by Name"). The button **Options...** offers another specific dialog with more options for this Action.



Attention: Most option fields offer **Tool Tips** which extra information, conditions and general tips. Tool tips appear when you hold your mouse-cursor over the field.

- **User Name** and **Password.** Define which WebCenter user will execute the selected action on WebCenter. Make sure this user has the required privileges.

More info

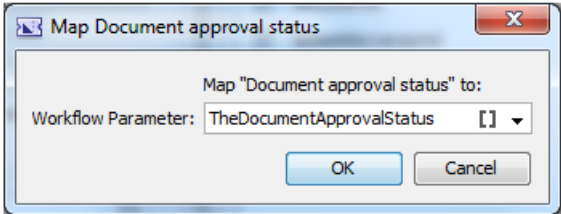
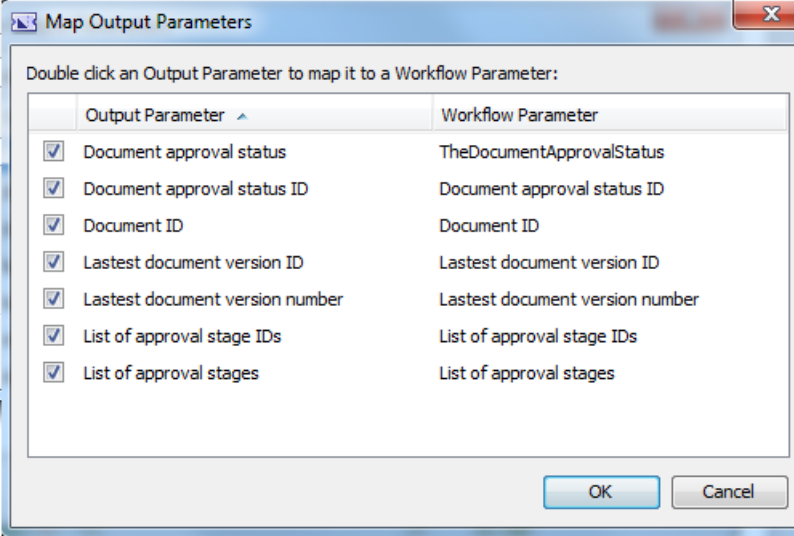
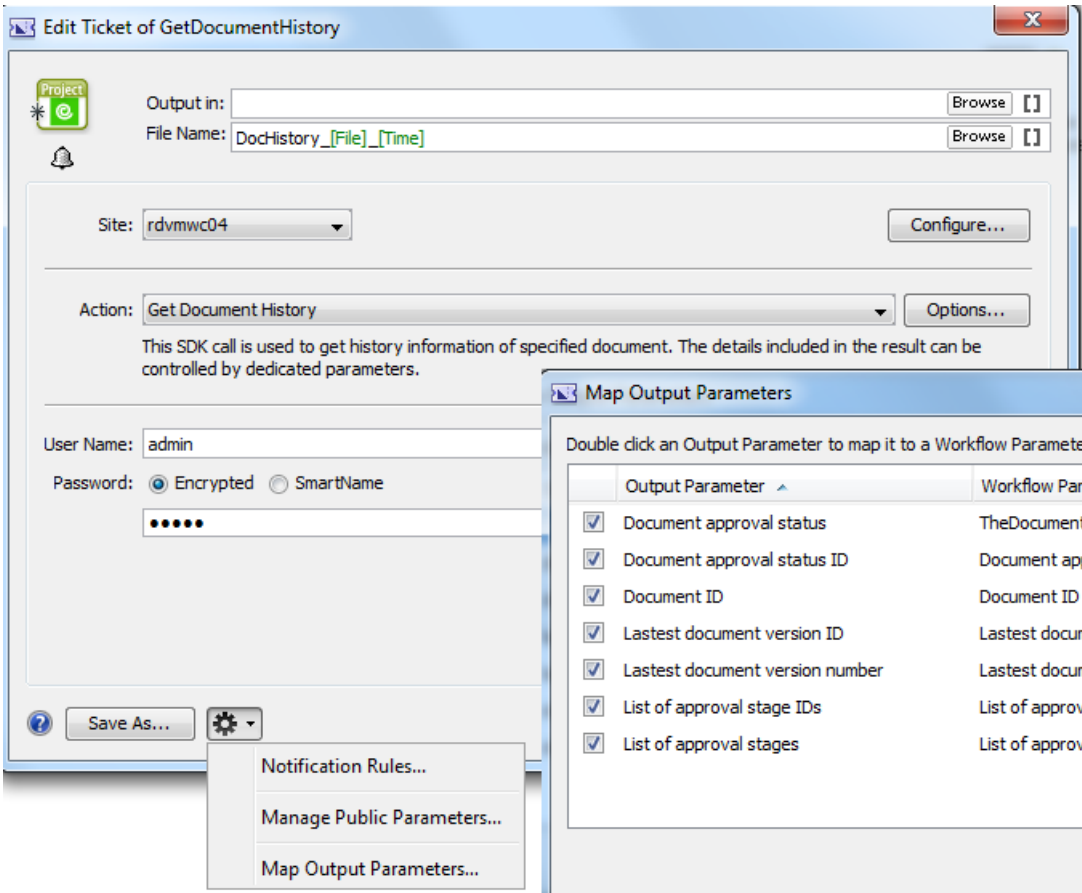
More detailed technical info is available in the "**WebCenter SDK documentation**". Read more about this document in this Esko Knowledge Base article: [KB 92180925](#).

Modifying workflow parameters using this task

As mentioned in [Modifying Workflow Parameters During a Workflow](#) on page 1075, this task can also be used to modify **Workflow Parameters**.

The **Integrate with WebCenter** task allows you to modify the value of a specific Workflow Parameter with the result of the action executed by the task.

For example (as shown in the screen shot): when you select the action "Get Document History", you can update the value of a specific Workflow Parameter with the value of the approval status of the document. First you need to define to which Workflow Parameter a specific Output Parameter needs to be mapped to. You can do this using the Map Output Parameters Dialog.



11. Troubleshooting

11.1 Troubleshooting the Automation Engine Server

As an Automation Engine administrator, you can run diagnostic tests on the Automation Engine server when something seems wrong. With the provided tools, you can attempt to fix the problems yourself without the need to call Esko Customer Support.

11.1.1 How to check the status of the AE databases

Proceed as follows:

1. Open the **Server Admin Tool** by entering `http://<name of your server>` or `http://<name of your server>:9999` in your browser.
2. Click on **Diagnostics**.
3. Log in using an administrator account or using an account that is a member of the **BGADMIN** user group on the Automation Engine server.
4. Click on **Databases**.
When the **Databases** page is loaded for the first time, the database diagnostics tests will be executed automatically for every Automation Engine database (see the following image).

The screenshot shows the 'Automation Engine Server' web interface. The server name is 'RDVMAEDEMO' and it is running. The 'Databases' section is expanded, showing a table of database statuses. All databases are listed as 'online and accessible'.

Status	Database	Database System	Details
●	Jobs	Microsoft SQL Server	Database is online and accessible. Info
●	Tasks	Microsoft SQL Server	Database is online and accessible. Info
●	Task History	Microsoft SQL Server	Database is online and accessible. Info
●	JDF	Microsoft SQL Server	Database is online and accessible. Info
●	FlexoPlates	Microsoft SQL Server	Database is online and accessible. Info
●	Gang Run Printing	Microsoft SQL Server	Database is online and accessible. Info
●	Application Server	Microsoft SQL Server	Database is online and accessible. Info

[CHECK AGAIN](#)





5. If you want to execute the diagnostics tests for the Automation Engine databases again, click the **Check Again** button.

For every Automation Engine database, you can see the following information:

- The **Status** of the database.
- The **Database** name.
- The **Database System**.

- The status **Details** of the database.
- A link with detailed information on the database.

The Status of the database can be one of the following:

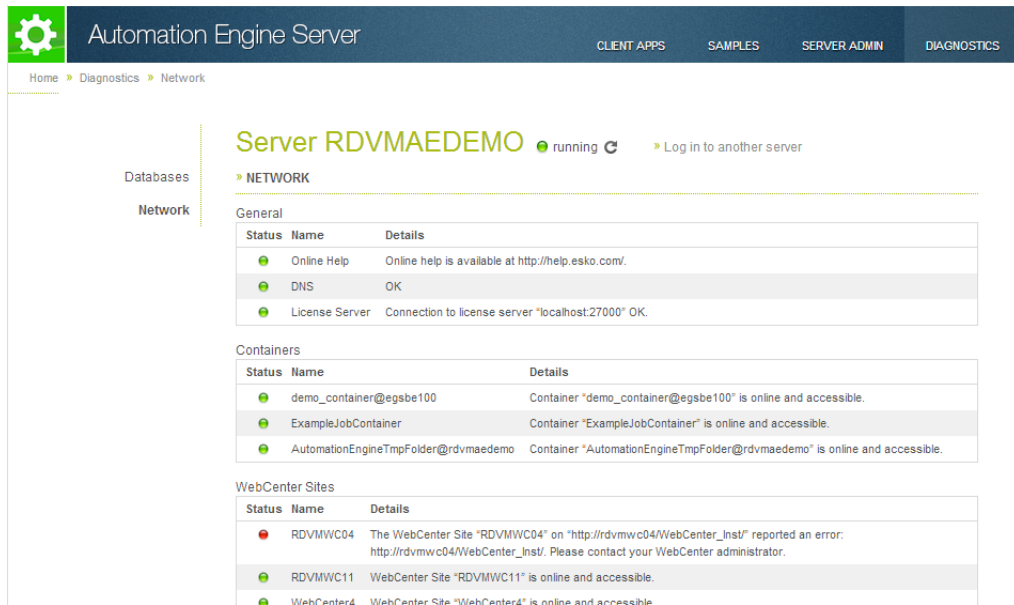
Status	Definition
	The database is online and accessible.
	The database is online and accessible, but there are one or more warnings (for example the database is near its size limit).
	The database is not online or is not accessible (for example no connection could be made to the database system, or the login failed).
	The status of the database is unknown. This is the case if the DBMS type of your Application Server database is an Oracle database or any of the other databases is an Embedded database. There are no diagnostic tests available for these two types.
<p>Note: In the Configure tool, you can change the DBMS type of the database. For more info, see: Automation Engine Databases.</p>	

Note: If the Automation Engine server is not running, you can still check the status of the Automation Engine databases. In this case however, the **Database System** field will be empty.

11.1.2 How to run network diagnostics tests on the AE server

To check if your network is set up properly, proceed as follows:

1. Open the **Server Admin Tool** by entering `http://<name of your server>` or `http://<name of your server>:9999` in your browser.
2. Click on **Diagnostics**.
3. Log in using an administrator account or using an account that is a member of the **BGADMIN** user group on the Automation Engine server.
4. Click on **Network**.
When the **Network** page is loaded for the first time, the network diagnostics tests will be executed automatically for every Automation Engine database (see the following image).







5. If you want to execute the network diagnostics tests again, click the **Check Again** button.

For every Automation Engine network diagnostics test, you can see the following information:

- The status of the network test.
- The name of the network test.
- The status details of the network test.

The Status of the database can be one of the following:

Status	Definition
	The network test succeeded.
	The network test succeeded, but there are one or more warnings.
	The network test failed.
	The network test was not applicable or was skipped.

What do the network diagnostics tests on the AE server do?

Test name	Explanation
Online help	A test to verify if the server can access http://help.esko.com . the online source of help, with a reference guide and a plethora of knowledge base articles.
DNS	A basic networking test to verify if the server name for this computer corresponds uniquely to what the Domain Name Server will return to other systems trying to connect to this system.
License Server	A connection test from the server to the License Server. In case of failure, basic network tests are done to verify DNS configuration and TCP/IP connectivity.

Test name	Explanation
Assistant Server	A connection test from the server to each configured assistant server. In case of failure, basic network tests are done to verify DNS configuration and TCP/IP connectivity. In case no assistant servers as configured, this test will not show up.
Container Server	A test to see if the server can connect to all configured Container servers.
Webcenter Server	A test to see if the server can connect to the configured WebCenter Sites.
Device Server	A test to see if the server can connect to each configured subsystem. In case of failure, basic network tests are done to verify DNS configuration and TCP/IP connectivity. The tested subsystems are RIP stations (FlexRIP), Imaging Engine, Digital Front End for digital presses (DFEs), Nexus and Odystar servers.

11.2 Troubleshooting the Automation Engine Pilot

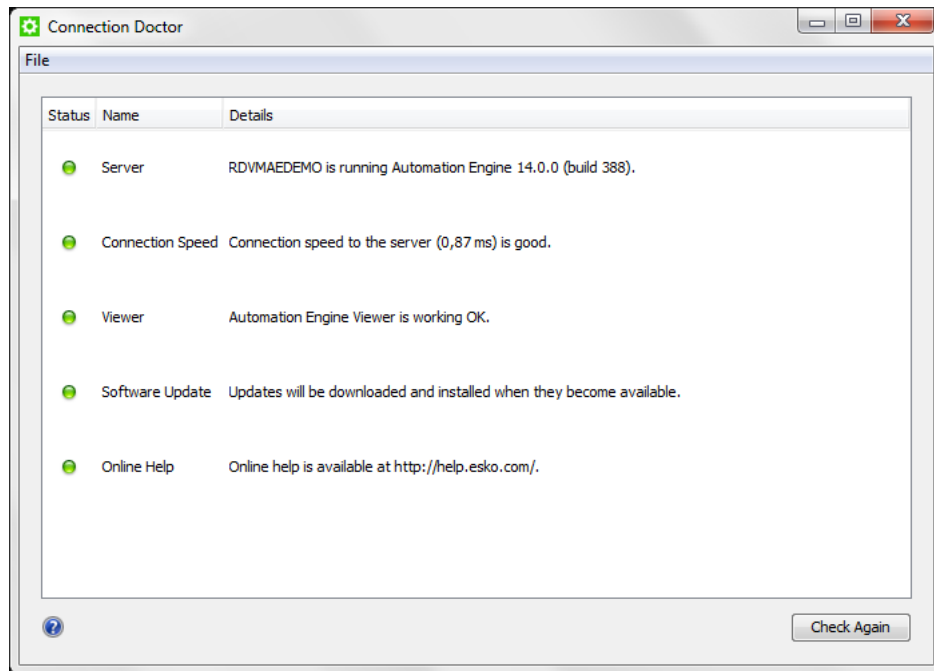
As an Automation Engine administrator, you can run diagnostic tests from an operator's workstation to verify that the Pilot's connection to the Automation Engine server works properly. With the provided tools, you can attempt to fix the problems yourself without the need to call Esko Customer Support.

11.2.1 Connection Doctor

Opening the Connection Doctor

You can run diagnostic tests from an operator's workstation to verify if the Pilot's connection to the Automation Engine server works properly. This can be done via the **Connection Doctor** tool in the Automation Engine Pilot.

To view the **Connection Doctor** in the Pilot, choose **Tools > Connection Doctor** .



Alternatively, if you have trouble logging into the Pilot, an extra **Connection Doctor...** button will be available to go directly to the **Connection Doctor**.



The Connection Doctor runs a series of tests that check the connection from Pilot to the Automation Engine server. These tests are the following:

- Server access test
- Connection Speed test
- Automation Engine Viewer access test
- Software Update access test
- Online Help access test

Status Messages

The tests in the **Connection Manager** can have the following statuses:

Table: Server

Message	Clarification
The Automation Engine server is down for maintenance. Please try again later or contact your administrator.	The maintenance status includes: server updates, server upgrades or the installation of hot fixes. Try again on a later moment, or contact your administrator if necessary.

Message	Clarification
The Automation Engine server is restarting. Please try again in a moment.	This can take a few second to a few minutes.
The server you're connecting to is an assistant server. Please connect to the master server <host_name of the master>.	In setups where Automation Engine runs on multiple servers, one server is designated the role of master. The other servers take up the role of assistants. The master server is the server you need to log in to.
The server is currently set to idle. Please try again later or contact your administrator.	If a server's current role is set to idle, it will accept no connections. Please try again later or contact your administrator if necessary.
There was a problem connecting to <host_name>. Please check that the host name is correct and <host_name> is on.	This computer is unable to connect to the host. Please check that the host name is filled in correctly and that the computer is switched on. Please contact your IT department should you need assistance.
There was a problem connecting to <host_name>. Please check that the server is running.	<p>This computer is unable to connect to the Automation Engine server. The most common reason for this is that the server is not running.</p> <p>If the server is running, verify if your computer is correctly connected to the network.</p> <p>If you require assistance with this problem, contact your IT department.</p>
There was a problem connecting to <host_name>. Please check that the server is running, and verify with your IT administrator if the "<host_name>'s" file is correct.	<p>This computer is unable to connect to the Automation Engine server. The most common reason for this is that the server is not running.</p> <p>If the server is running, verify if the hosts file of your operating system is correct. The hosts configuration of this computer contains explicit references to the Automation Engine server's host name and its IP addresses. A configuration error could be the cause of this connection problem.</p> <p>If you require assistance with this problem, contact your IT department.</p>
There was a problem connecting to <host_name>. Please check that your computer is connected to the network.	<p>This computer is unable to access the network. You can try and diagnose the problem by using one of the following methods:</p> <ul style="list-style-type: none"> • On Mac, go to: Applications > System Preferences > Network > Assist me... to test your connection . • On Windows, go to: Start > Control Panel > Network and Internet > Network and Sharing Center > Troubleshoot Problems > Network Adapter to test your connection.

Table: Viewer

Message	Clarification
There was a problem connecting to the Application Server. Please check that the server is running. Contact your administrator if the problem persists.	The Application Server could not be reached. Check if the server is running. Contact your administrator if the problem persists.
The Application Server <host_name> is not running on port <port_number>. Please contact your administrator.	The Application Server is not running on the designated port. Contact your Automation Engine administrator to fix this issue.

Table: Connection Speed

Message	Clarification
The connection speed to the server is good.	The response time between this computer and the server is less than 20 milliseconds, which is considered good.
The connection speed to the server is acceptable.	The response time between this computer and the server is between 20 milliseconds and 100 milliseconds. This is OK for normal operations.
The connection speed to the server is bad.	The response time between this computer and the server is more than 100 milliseconds. This is considered too slow for normal operation. Consult with your IT department what can be done to improve the speed.

Table: Software Updates

Message	Clarification
Software updates will be downloaded and installed when they become available.	The Automation Engine Web Server is accessible from this computer. Software updates will be downloaded and installed when they become available.
The Automation Engine Web Server could not be reached. Software updates will not be installed on this machine.	The Automation Engine Web Server could not be reached. This is most likely due to firewall restrictions. Contact your administrator to resolve this issue.

Note: The Automation Engine server hosts software updates for Automation Engine Pilot. The **Software Updates** test checks if software updates can be downloaded to the client computer. The test will fail if the Automation Engine Web Server is not correctly hosted on port 9999. For example, if another web service has claimed this port on the server. Please contact your Automation Engine administrator to resolve this issue.

Table: Online Help

Message	Clarification
Esko online help is available at http://help.esko.com/.	Esko online help is available.
Pilot was unable to connect to the Internet. Please check the Internet proxy preferences of your operating system if you have	Automation Engine Pilot was unable to connect to the Internet. Check the Internet proxy preferences of your operating system. Contact

Message	Clarification
experience problems accessing Esko online help.	your IT department if you require assistance with fixing this issue.

Note: This test verifies if the computer can access <http://help.esko.com>, your online source of help, with a reference guide and a plethora of knowledge base articles. A warning is issued when the online help is unreachable. This does not necessarily mean online help is unavailable. Your Internet browser, which is used to display the help pages, may be configured with an Internet proxy that enables you to access the online help even if the Pilot is unable to connect to it. If unsure, please contact your IT administrator for more information.

12. IT requirements and FAQs

12.1 Introduction

This document describes the information needed to facilitate the installation of an Automation Engine. It also answers typical IT related questions.

General system requirements for the involved computers can be found on <http://www.esko.com/en/SystemRequirements/Automation%20Engine/>

Note: This document mentions components and features that are (commercially) optional.

Note: CP is used as abbreviation for 'Commercial Print'.

There are two sections in this document :

1. Technical details of an Automation Engine setup.
2. Guidelines and good practices in setting up an Automation Engine Environment.

12.2 Technical Details

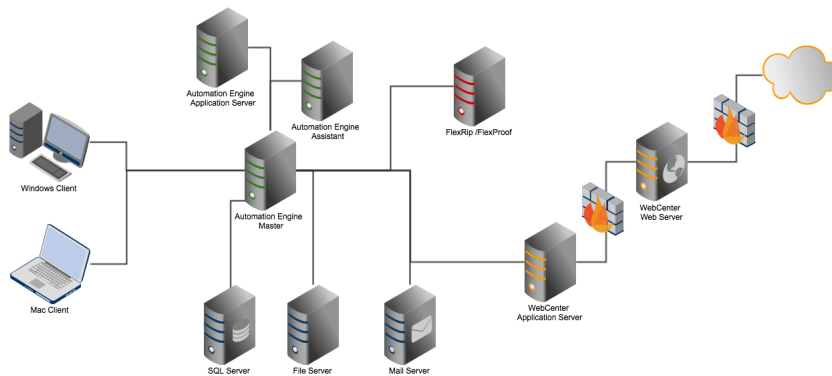
12.2.1 Automation Engine in its environment

Overview

Overview

Component	Description
Automation Engine Master server	Main server where all clients are connecting to. This server manages the Jobs and tasks database. This server also executes the actual server tasks.
Automation Engine Assistant server	Assistant server that can execute tasks to take away load from the master server.
Application server	Component serving the Automation Engine Viewer and CP (page) workflow tools. Typically this component is found on the Automation Engine Master server.

Database server	Server where the databases are located. Typically this component is found on the Automation Engine Master server.
Client	A client workstation is connecting to the Automation Engine Master server via Pilot or Shuttle. Editors can also connect to the server (ArtPro, PackEdge, Adobe Illustrator).
FlexRip	Esko RIP that generates digital files for proofs and/or plates. The Automation Engine Master typically connects to one or more RIP devices.
WebCenter	Esko's online collaboration tool.
File server	Where the actual data is. Files are stored in 'Containers'.
Mail server	A mail server can be used by Automation Engine to send out e-mails.



12.2.2 Automation Engine installation

Automation Engine System User

Automation Engine needs a system user to run its service and connect to other network components. The Automation Engine System User account must meet following requirements on the Automation Engine server:

- should be a member of the **Administrators group**
- should have the **Logon as Batch** right
- should have the **Logon as Service** right

An installation of Automation Engine creates a local user **BGSYSTEM** on the Automation Engine Master according to these requirements. This user then serves as the Automation Engine **System User**.

Services

An installation of Automation Engine will install these services:

Service name	Service running under	Purpose
BGMD	Automation Engine System User	Automation Engine's main service
EG APPLICATION SERVER (*)	Automation Engine System User	Application server service for Viewer and CP Layout tools
EG STATION INFORMATION SERVICE	Local System User	System info collection service
EGWEBSRV	Automation Engine System User	Webserver for server admin and client updates
FLEXlm License Manager	Local System User	License manager
FLEXnet licensing service	Local System User	License manager
MSSQLSERVER	Network Service	Default database server
MSSQL\$EG_APPSRV_JBOSS (*)	Network Service	Default database server for the application server
MSSQLServerADHelper	Network Service	SQL Server Active Directory Helper
SQLWriter	Network Service	SQL Server VSS Writer
SQLBrowser	Network Service	SQL Server Browser

(*) Installed optionally on systems having Viewer and/or CP Layout Tools.



Attention: All the services and processes of Automation Engine run as 32 bit.

Shares

These shares are created during the installation of Automation Engine:

Sharename	Purpose	Access
AutomationEngineImgFolder	Temporary folder for image data	server only
AutomationEngineLogging	Automation Engine log files	server only
AutomationEngineTmpFolder	Temporary folder for Automation Engine tasks	server only
bg_data_cms_v010	Central resources - Color management data	server and clients
bg_data_custom_v010	Central resources - Automation Engine presets	server and clients
bg_data_dgc_v010	Central resources - Dot gain curves	server and clients
bg_data_fonts_v040	Central resources - Fonts data	server and clients
bg_data_ipldata_v010	Central resources - CIP3/ Inkplanner presets	server and clients
bg_data_marks_v010	Central resources - SmartMarks sets	server and clients
bg_data_system_v010	Central resources - License configuration	server and clients

DeskPackContainer	Container for temporary DeskPack files	server and clients
EG_APPSRV_BINDEPLOY	Application server configuration folder	server only
EG_APPSRV_CONFIGURATION	Application server configuration folder	server only
EG_APPSRV_DEPLOY	Application server configuration folder	server only
ExampleJobContainer	Default data container created at first install	server and clients
FastRipTask	FlexRip temporary folder	server and clients
Impose	FastImpose templates, marks and defaults	server and clients
PitStop	Enfocus PitStop templates and configuration data	server and clients

Note: Access right is set to: **Read/Write for 'Everyone'**

Installation Folder

Automation Engine is by default installed in a folder `Esko` on the root drive. Automation Engine can not be installed in the `Program Files` or **Program Files (x86)** folder.

Every Automation Engine installation will create and use a temporary folder in the Esko folder (`BG_TMP`).

Tip: It is good practice to defer that folder to a drive other than the system drive. How to do that is described in the **Installation Guide**".

12.2.3 Network Details

Network Configuration

Automation Engine requires TCP and IPv4 to work.

Note: IPv6 is unsupported.

You need to configure DNS correctly. Links to other network should be configured using host names, not IP addresses.

If a DNS and/or proxy server is used, make sure all components in the equation (Automation Engine Server, Automation Engine Client, Proxy Server) are able to do an `nslookup` on `hostname` of all other components.

Danger: After installation of the Automation Engine software, the host name of that server cannot be changed.

Ports

These are the ports used by Automation Engine and the client software:

TCP :

- 8000-9000 for BGMD and the Automation Engine servers
- 5182 for ShuttleServer, unless configured otherwise
- 4401 for the Automation Engine JDF Server (service port)
- 1433 for the SQL server (default installation)
- 27100 and a dynamic port for Station Information Service
- 1098 RMI port for the EG Application server
- 1099 JNDI Naming Service for the EG Application server
- 4444 RMI Object for the EG Application server
- 4445 Transaction Manager for the EG Application server
- 27000 for the License Manager service

HTTP:

- 80 + 9999 for Tomcat/ServerAdmin (egwebsrv.exe)
- 8080 for EG Application Server
- 4411 for the Automation Engine JDFServer (JMF)
- 4412 for the Automation Engine Frip2JDF Server : process incoming feedback (JMF) from DFS
- 4415 for the Automation Engine ElemServer (experimental)

Connection with clients:

- The server sometimes relies on ephemeral ports, for example for uploading files in Pilot. These ports are picked randomly between 49152 and 65535.

LAN/WAN Network Performance

Automation Engine client-server connection requires decent network performance.

Latency simulations in a lab environment show that:

- With delays below 20ms client (Automation Engine Pilot) performance is optimal
- With delays till 50ms client performance is relatively acceptable
- With delays around 100ms (and higher) client performance is poor to unacceptable

Therefore Automation Engine is not suited to run in a WAN environment.



Attention: Using tools such as remote desktop services is in violation with the EULA (End User License Agreement).

12.2.4 Database server

By default, Automation Engine is installed with an SQL Express database engine. However, for scaling and performance reasons, one can choose to use a Full SQL server. Read more about this in [Using a Full SQLServer](#) on page 1383.

Installation Details

Mandatory components:

- SQL Server Database Engine
- Replication and Fulltext components

Mandatory installation parameters:

- Collation settings: SQL_Latin1_General_CP1_CI_AS
- Security Mode: SQL
- TCP protocol for the SQL Server service: enabled

Note: These are the installation parameters used to install the default MSSQL Express server:

```
/ACTION=INSTALL
/FEATURES=SQL
/BROWSERSVCSTARTUPTYPE=Automatic
/SQLSVCSTARTUPTYPE=Automatic
/SQLSVCACCOUNT="NT AUTHORITY\SYSTEM"
/SQLSYSADMINACCOUNTS="BUILTIN\ADMINISTRATORS"
/SQLCOLLATION=SQL_Latin1_General_CP1_CI_AS
/IACCEPTSQLSERVERLICENSETERMS=1
/INSTANCENAME=<instancename> of MSSQLSERVER for unnamed instance
/SECURITYMODE=SQL
/SAPWD=<default password>
/TCPENABLED=1
```

Configuration details

Automation Engine requires 4 databases to run:

- BSJobs
- FastLane_TM
- FastLane
- EG_Appserver_JBOSS

Note: The terms BS and FastLane refer to names of product that Automation Engine's architecture originates from.

To access and modify these databases, Automation Engine needs a user who can:

- Login to the Database server:
- Database roles:
 - db_owner: Members have full access.
 - db_accessadmin: Members can manage Windows groups and SQL Server logins.
 - db_datareader: Members can read all data.
 - db_datawriter: Members can add, delete, or modify data in the tables.
 - db_ddladmin: Members can run dynamic-link library (DLL) statements.
 - db_securityadmin: Members can modify role membership and manage permissions.
 - db_backupoperator: Members can back up the database.

Oracle as Database Server for the Application Server component

An Oracle database server can be used to run the EG_Appserver_JBOSS database of the Application Server component.

This will increase performance of the Pilot modes **Pages**, **Proofs** and **Plates** module. This so means that the use of Oracle is only relevant in a production workflow for books and magazines. It has no effect on the performance of the **Viewer** (which also uses the application server component).

Following database server versions are supported:

- Oracle 10g Standard Edition, Enterprise Edition
- Oracle 11g Standard Edition, Enterprise Edition

Important: using Oracle as a database server for BSJobs, FastLane_TM and FastLane is not supported.

12.2.5 File Server

Server Types

Automation Engine can use data that is available on network shares. These shares need to be on Windows or Unix/Linux (using SMB) file servers.

: We recommend to use Windows Server 2003 or Windows Server 2008 as file server.

Restriction: Due to increasing connectivity issues between Mac OS and Windows Server, Mac OS X (version 10.5.x, 10.6.x, 10.7.x, 10.8.x, 10.9.x) data servers are no longer supported.

Supported Access Protocols

Esko does not support fileserver access over protocols other than SMB/SMB2.

File server access

The Automation Engine system user must have read and write access on the shares. read more on this in [Automation Engine System User](#) on page 1375.

12.2.6 Mail Server

Automation Engine can send e-mail to communication with your colleagues or customers, but also with Esko Customer Service, to report problems.

Supported protocols are :

- smtp
- smtps (secure)

You can set up the Corporate Mail server or the Google servers to be used by the Automation Engine to send e-mails. For setting up a Google Server, you need a Google mail account. You can configure Automation Engine to use a secure connection to smtp.gmail.com on port 465 using that account.

12.2.7 Users and Groups

Automation Engine Users

Users in Automation Engine map directly to Windows users. The Windows users are organized in two local Windows groups on the Automation Engine master. Regular users are in the **BGUSERS** group, administrators are in the **BGADMIN** group.

These Windows users can be local users on the Automation Engine master server or Windows domain users. If you create a new user via the Automation Engine Pilot, a local Windows user is created for you.

If you want to use domain users, then assign these domain users to either the BGUSERS or the BGADMIN local Windows group.

Automation Engine Groups

User groups in Automation Engine have no correlation with Windows groups.

Domain Users

Domain users can be used in Automation Engine. This requires that the Automation Engine system user can retrieve user info from the Active Directory. Read more in [Automation Engine System User](#) on page 1375

Restriction: Automation Engine does not offer support for Domain Groups.

Single Sign-On

The Automation Engine client software does not offer any implementation for "single sign-on" .

12.2.8 The Automation Engine Pilot

Java

On Windows, a Java Runtime Environment (JRE version 1.7) is distributed with the Pilot. The user does not need to install Java on the client machine.

On MAC, a Java Runtime Environment 1.6 is required. However, Apple does not allow Esko to distribute this version, so the installation of Java has to happen separately from the Automation Engine client. The installation of Java should start automatically when the Java Runtime Environment is needed by the Automation Engine client installer. This automatic installation is currently provided by Apple and could change in the future.

12.3 General Guidelines

12.3.1 Scalability and Performance

Adding an Automation Engine Assistant

Adding an Assistant to Increase Throughput

Automation Engine comes with a built-in limit on the amount of concurrent tasks (tasks that are processed simultaneously). The number of concurrent tasks that can be processed depends on the licenses purchased. Evaluating and defining the amount of concurrent tasks needed should take into account the number of files processed and the number of task run on these files over a certain period.

The amount of concurrent tasks needs to be in line with the capacity (RAM/CPU) of the Automation Engine server. More concurrent tasks will increase the load on the system. If the capacity of the Automation Engine server has reached it's limits, it can be increased by adding one or more Assistant servers. One Automation Engine server will take the role of Master, the other(s) will take the role of Assistant. The Master will monitor its own load and the load of the assistants to decide on which server a task should be executed.

Adding an Assistant for Redundancy

An Automation Engine Assistant is not an out of the box fail safe solution.

However, it could be used in a fail safe scenario. In case the Master breaks down, an Automation Engine server in an Assistant role, can be promoted to a Master server. A full backup package of the Master (Configuration, Resources, Databases) can then be restored on the Assistant to rebuild the initial configuration. Note: the backup package of the Master generated by the scheduled backup procedure of Automation Engine can be automatically copied to the Assistant.

Fail safe solutions based on proper backup tools or on snapshots of virtual images are a valid alternative, and might be seen as a more effective solution.

Adding an Assistant versus Adding an extra Master

When throughput needs to be increased, adding an assistant is not necessarily the best option. If the production/organization can be split into logical components, then it could make sense to add an extra master.

One master-assistant(s) combination has one main processing queue.

Redirecting Central Resources

Following shares are considered as central resources:

- bg_data_cms_v010
- bg_data_custom_v010

- bg_data_dgc_v010
- bg_data_fonts_v040
- bg_data_ipldata_v010
- bg_data_marks_v010

It is allowed to redirect these resources to a central file server. Valid reasons for doing this are:

- Access: these shares need to be accessed from other applications/servers than Automation Engine (PackEdge, ArtPro, FlexRip,...)
- Backup strategy: these shares contain critical files such as color profiles, marks, curves,... Remember that these shares are also included in Automation Engine's scheduled backup.

Note: Esko recommends keeping the resources on the Automation Engine server to avoid extra network traffic.

Using a Full SQL Server

Express edition versus Full SQL Server

Automation Engine requires a working DBMS (Database Management System) for its proper operation and persistent storage.

Microsoft's SQL Server Express Edition is delivered and installed together with Automation Engine. Automation Engine is configured to work with this SQL Server as default DBMS in this out-of-the-box setup. New and empty databases and users are created during installation.

The SQL Server Express Edition software has the following limitations :

- Database sizes are limited to 10 GB (as of Express Edition 2008 R2).
- The SQL Server service will only use 1 CPU and will restrict its memory usage to 1 GB.

In many cases, the SQL Server Express Edition will be sufficient. However, from a certain load or performance expectation onwards, it is highly recommended to upgrade your Express edition to a full SQL Server. This section gives recommendations as to when the DBMS for an Automation Engine installation should be upgraded from an Express Edition to a full SQL server.

Database Size Limitations

SQL Server Express Edition 2008 R2, delivered along with Automation Engine (since 12.1.1 onwards), has a size limitation of 10 GB per database. However, you can scale up to a full version of SQL Server to increase the size limit to the disk size. Given that Automation Engine has three main databases, consider the following points:

- **Jobs database:** Automation Engine does not function as an archiving system. To keep the Jobs database size under the limits, we recommend:
 - keeping a Job online only for its lifetime. The concept of a Job is temporary by nature: Jobs, after a while, are done. If you have many re-prints of products, then consider using the Products tool (Database) as well.
 - archiving unused Jobs and Products.

Note: Theoretical limits regarding the number of Jobs and Products exceed the practical Express Edition server demands (<50.000 jobs / <250.000 products) considerably.

- **Task Monitor database:** To keep the size of the Task Monitor database under limits, we recommend:
 - setting up adequate **Task Clean Up rules**.

Note: The theoretical limit on the number of tasks in the Task Monitor in terms of size (>100.000 tasks) is overruled by what the Express Edition server can handle in terms of performance (<100.000 tasks).

- **Task History database:** We recommend archiving the tasks in the **Task History** database regularly to keep the database size under limits.

Note: For the 2005 and 2008 Express Edition databases, you can estimate the limits by reducing the limits by a factor 2.5.

Read more in:

- [Performance Considerations](#) on page 1384
- [Recommendations](#) on page 1385

Performance Considerations

The Automation Engine server performance depends on the performance of the used DBMS. This may be noticeable when searching a Job in the Pilot (for example while filtering), when monitoring tasks (while filtering or scrolling), when executing tasks (while persisting their state, progress, ticket, ...), and so on.

The number of tasks ran per day and the average number of connected operators (using a Pilot or Shuttle) determines the load put on the Automation Engine server and its DBMS. In addition, the setup of the DBMS and the environment in which it is running will also have an important effect on performance :

- The SQL Server software can be installed on the same computer as the Automation Engine software or on another computer.
- If the SQL Server is installed on another computer, then the network is involved. Under-performing network components may seriously influence DBMS performance. Other software running on that computer may influence performance as well.
- If the SQL Server is installed on the same computer as the Automation Engine (master or assistant), that may have an impact on the amount of memory and CPU resources available to the Automation Engine.
- An SQL Server can be set up to run 1 instance (unnamed or the default), or even multiple (named) instances. In case of an Express Edition, each instance then has its own limitations (as described above), independent of the others.
- Each Automation Engine database may be configured independently of the others (to potentially point to different SQL Server instances on potentially different computers).

Issues with database performance can be noticed in the following situations (but are not limited to):

- a decline in average response time as experienced by operators of the Pilot.
- a decline in average time between the moment of submitting a task and the moment the submitted task starts processing.

Recommendations

We recommend upgrading the SQL Server Express Edition to a full SQL Server if one or more of the following conditions apply when:

- more than 30.000 tasks are processed per day- this typically corresponds to 1.000 Jobs or files per day.
- more than 100.000 processed tasks are constantly present in the Task Monitor (assuming that cleanup rules are set up adequately).
- more than 20 active Pilots are connected simultaneously.
- Automation Engine is licensed and configured to run more than 7 concurrent tasks.
- the database size limitation risks to be exceeded.
- you use the functionality of the Pages, Proofs and Plates views (in commercial print imposition workflows).
- there is a continuous load on the Automation Engine server (when production runs 24 hours per day).

When upgrading to a full SQL Server, special attention must go to provide sufficiently powerful hardware. When the SQL Server is installed on a dedicated computer, we recommend:

- SQL Server 2012 Standard Edition
- a powerful machine (4-8 CPU)
- 64-bit OS (Windows 2008 R2 / Windows 2012)
- with at least 16 GB RAM (preferably 32 GB)
- with a high performance network card

Note: While SQL Server Express Edition is expected to be adequate, problems due to database hiccups can happen (for example when peak load happens frequently, which might cause some of the numbers mentioned above to be surpassed temporarily, but not on average). In such cases, we recommend consulting Esko Customer Support and switching to a full SQL server.

Local versus Remote SQLServer

Esko recommends having the database server(s) locally on the Automation Engine Master (to avoid extra network traffic). However, when a full SQL Server is required, the licensing model of SQLServer needs to be taken into account:

- Per CPU: high cost if high number of CPU
- Per Client: high cost if high number of end users (end users = amount of users connecting via Pilot, Shuttle, Access Points, so not just the BGMD service)

A full SQL server is needed in cases where both CPU usage and number of Automation Engine users will be high. In such cases, a dedicated server with 2 CPU's can be a cost efficient alternative.

Example Configurations

An Automation Engine setup is scalable and modular. How to set up depends on the load that your server needs to carry and on the fail safe requirements.

Small Size Configuration

This includes:

- One Master Server : setup as Automation Engine Master. The role of this server is managing Jobs and tasks as well as processing tasks.
- One File Server: storing the data processed by the Automation Engine.

Note: Esko recommends to have a dedicated file server in every case, even for small size configurations.

Medium Size Configuration

This includes:

- One Master Server: setup as Automation Engine Master. The role of this server is managing Jobs and tasks as well as processing tasks.
- One Assistant Server: configured as an Automation Engine Assistant. This server will only process tasks.
- One File Server: this stores the data processed by the Automation Engine.

Large Configuration

This includes:

- One Master Server: setup as Automation Engine Master. The role of this server is managing Jobs and tasks as well as processing tasks.
- One Assistant Server: configured as an Automation Engine Assistant. This server will only process tasks.
- One File Server: stores the data processed by the Automation Engine
- One Viewing Assistant: configured to handle the Viewing sessions of all the clients.
- One Database Server: The Master server connects to a dedicated database server.

Very Large configuration

This includes two separate server groups consisting of one Master server and two Assistant servers. Both server groups are completely independent units.

- Two Master Servers: setup as Automation Engine Master. The role of this server is managing Jobs and tasks as well as processing tasks.
- Four Assistant Servers: configured as an Automation Engine Assistant. These servers will only process tasks.
- One File Server: stores the data processed by the Automation Engine

One server per group is setup as Automation Engine Master. The role of this server is reduced to only managing Jobs and tasks. The server has a relatively low amount of CPU and RAM. This server runs a full SQLServer database engine.

Two servers are configured as Assistant. Their role is to process tasks. These servers have a high amount of CPU and RAM.

The data processed by both server groups is stored on the file server.

12.3.2 Snapshots/Cloning

Taking a snapshot or cloning for fail safe purposes can be done under the condition that no licenses are activated on the server itself. When failing to do so, after restoring a snapshot of a virtual server or restoring a backup clone, the licenses will be broken.

This can be overcome by:

- Temporarily deactivating the licenses before creating a snapshot or a clone.
- Installing the licenses on a computer other than the Automation Engine server

12.3.3 Remote Desktop Services



Attention: Running the Automation Engine client software via Remote Desktop Services is in violation with the EULA (End User License Agreement).

12.3.4 Sharing the Automation Engine server with Other Applications

Automation Engine requires a dedicated server to run on. Installing other software might influence the functioning of Automation Engine and therefore is **strongly discouraged**.

Sharing the Automation Engine Server with other Esko Applications

1. **WebCenter:** Automation Engine and WebCenter can not be installed on the same server. Both applications will malfunction if installed on the same server.
2. **FlexRip:** FlexRip can be installed on the same server as Automation Engine. However, Esko advises to have these components on different servers.
3. **PackEdge / Plato / i-cut Layout:** Editors such as PackEdge / Plato / i-cut Layout are typically installed on a client station but can be installed on the same server as Automation Engine (for example to build templates). However, Esko does not recommend to use the server as a client workstation.
4. **Nexus:** Automation Engine and Nexus cannot be installed on the same server. Both applications will malfunction if installed on the same server.

Virus Scanners and Firewalls

Make sure that virus scanners and firewalls (including OS firewalls) do not interfere with the Automation Engine software (blocking temp files, processes).