

Plato 18.1

User Manual

01 - 2020

Contents

1. Copyright Notice.....	10
2. Getting Started with Esko Plato.....	12
2.1. Welcome to Esko Plato.....	12
2.2. Starting Esko Plato.....	12
2.3. Concepts and ideas in Esko Plato.....	13
2.3.1. Stations and Layouts.....	13
2.3.2. View modes.....	14
2.3.3. Typical workflows.....	15
2.3.4. Document window elements.....	16
2.3.5. Entering information.....	19
2.3.6. External Files.....	20
2.3.7. Calculator and Parameter Formulas.....	20
2.3.8. Shuttle.....	26
2.3.9. Studio.....	27
2.3.10. Common data resources.....	27
3. Working with Plate Files.....	29
3.1. Creating, opening and saving plates.....	29
3.1.1. Creating a new layout.....	29
3.1.2. Saving a layout.....	31
3.1.3. Saving a layout with a different name.....	32
3.1.4. Opening an existing layout.....	32
3.1.5. Changing the sheet or plate size.....	33
3.2. Creating Layouts.....	33
3.2.1. Layout Assistant.....	33
3.2.2. Grid-based workflow.....	34
3.2.3. CAD workflow.....	36
3.2.4. SmartLayout workflow.....	40
3.2.5. Manual workflow.....	44
3.2.6. Combining workflows.....	45
3.3. Using a CAD file for plate layout (optional).....	45
3.3.1. Supported CAD file formats.....	45
3.3.2. CAD Layout or CAD 1-up?.....	48
3.3.3. Importing a CAD layout.....	48
3.3.4. CAD Styles.....	50
3.3.5. Using Print Item names.....	52
3.3.6. Flipping a CAD layout to print side.....	52
3.3.7. Importing a different CAD layout.....	53
3.3.8. Releasing a CAD layout.....	53

3.4. Using Templates.....	53
3.4.1. What is a template?.....	54
3.4.2. Creating a template.....	54
3.4.3. Saving a template.....	54
3.4.4. Using a template.....	58
4. Working with Stations.....	59
4.1. Creating, editing and saving a station.....	59
4.1.1. Creating a new station.....	59
4.1.2. Creating a new station in a CAD workflow.....	60
4.1.3. Editing an existing station.....	60
4.1.4. Using Select Die Shape.....	61
4.1.5. Saving a station.....	64
4.2. Using a CAD file to create a station.....	64
4.2.1. CAD designs automatically from Esko Plato.....	65
4.2.2. Importing a CAD design.....	65
4.2.3. Importing a different CAD design.....	66
4.3. Adding stations to the plate.....	66
4.3.1. Adding a Station.....	67
4.3.2. Assigning a station.....	69
4.3.3. Placing a station.....	70
4.3.4. Replacing a station.....	71
4.4. Working with empty stations.....	72
4.5. Adjusting overlapping stations.....	72
4.5.1. Using Adjust Masks.....	73
4.5.2. Splitting an Overlap Record.....	76
4.5.3. Resetting masks.....	76
4.5.4. Checking the status of masks.....	76
4.5.5. Adjust Masks tips.....	78
4.6. Nesting stations.....	79
4.7. Numbering Stations.....	80
4.7.1. Using the Place Station / Strip Number Tool.....	81
4.7.2. Numbering stations automatically.....	82
4.7.3. Numbering stations manually.....	84
4.8. Using PDF files as stations.....	85
4.8.1. Die and Mask of a PDF file.....	86
4.8.2. Adjust Masks with PDF files.....	86
4.8.3. Number Stations with PDF files.....	86
4.9. Using GRS files as stations.....	86
4.9.1. Die and Mask of a GRS file.....	87
4.9.2. Adjust Masks with GRS files.....	87
4.9.3. Number Stations with GRS files.....	87
5. Working with Graphics.....	88

5.1. About importing a graphic.....	88
5.2. Replacing a graphic.....	89
5.3. Importing an ArtPro file.....	90
5.4. Importing a PostScript file.....	92
5.4.1. Pages.....	93
5.4.2. Color Management.....	94
5.4.3. Inks.....	97
5.4.4. Overprint.....	98
5.4.5. Images.....	99
5.4.6. Page Size.....	102
5.4.7. Other.....	103
5.4.8. Output.....	104
5.5. Importing a PDF file.....	105
5.5.1. Importing multipage PDF files.....	106
5.6. Importing an EPS file.....	106
5.7. Importing an Illustrator file.....	107
5.8. Positioning a graphic.....	107
5.9. Editing a graphic in Esko PackEdge.....	107
6. Working with Esko Plato Tools.....	108
6.1. Arranging Objects.....	108
6.1.1. Using Layers.....	108
6.1.2. Using To Front/To Back.....	113
6.1.3. Using Group/Ungroup.....	113
6.1.4. Group into Grid.....	114
6.1.5. Using the Transform Window.....	114
6.1.6. Using the Align Window.....	115
6.2. Creating a Repetition (Grid-based Step & Repeat).....	116
6.2.1. Repeating objects with the Grid dialog box.....	116
6.2.2. Step-and-Repeat examples.....	117
6.2.3. Step-and-Repeat with Stagger example.....	122
6.2.4. Step-and-Repeat with Distribute examples.....	124
6.3. Two-sided Printing.....	127
6.3.1. Two-sided plates.....	128
6.3.2. Two-sided stations.....	128
6.4. Managing Inks.....	129
6.4.1. Viewer.....	129
6.4.2. Working with inks.....	130
6.4.3. Using ink libraries.....	136
6.4.4. Using ink mapping.....	136
6.4.5. Using external file ink mapping.....	138
6.4.6. Calculating ink usage.....	138
6.5. Trapping.....	140
6.5.1. PowerTrapper.....	141

6.5.2. The Color Pairs Palette.....	153
6.5.3. Trap Red under Green.....	156
6.5.4. Keep Red away from Green.....	159
6.5.5. Rich Black.....	162
6.5.6. White Underprint.....	163
6.6. Staggered Cutting.....	165
6.6.1. Creating staggered cutting lines.....	165
6.6.2. Changing a staggered cutting line.....	167
7. Working with SmartMarks.....	168
7.1. What are SmartMarks?.....	168
7.2. Using SmartMarks.....	168
7.2.1. Adding a SmartMark.....	169
7.2.2. Deleting a SmartMark.....	170
7.2.3. Editing a SmartMark.....	170
7.2.4. Creating and saving a set.....	170
7.2.5. Using a set.....	171
7.2.6. Replacing and editing a set.....	171
7.2.7. SmartMarks in the Station View.....	171
7.2.8. Custom Inks in SmartMarks.....	171
7.2.9. Custom Color in SmartMarks.....	175
7.3. Supplied SmartMarks.....	179
7.3.1. Setting the location of a SmartMark.....	180
7.3.2. Span Area.....	181
7.3.3. Image Mark.....	181
7.3.4. Center Mark.....	186
7.3.5. Corner Mark.....	191
7.3.6. Crease Mark.....	195
7.3.7. Dimension Line.....	197
7.3.8. Gallus.....	199
7.3.9. Gradation Strip.....	199
7.3.10. MFG Text replace.....	203
7.3.11. Rectangle.....	205
7.3.12. Standard Mark.....	206
7.3.13. Standard Strip.....	208
7.3.14. Text Mark.....	210
7.3.15. Trim Mark.....	216
7.3.16. Bar code.....	217
7.3.17. Ellipse.....	221
7.3.18. Grid Mark.....	223
7.3.19. Cell Numbering.....	225
7.3.20. Grommet Mark.....	227
7.3.21. Position Mark.....	228
7.3.22. Fill Placeholder.....	229

7.3.23. GRQ Gallus.....	230
7.3.24. GRQ Gradation Strip.....	231
7.3.25. GRQ Grid Marks.....	233
7.3.26. GRQ Image.....	235
7.3.27. GRQ Marks.....	237
7.3.28. GRQ Rectangle.....	238
7.3.29. GRQ Strip.....	240
7.3.30. GRQ Text.....	241
7.3.31. GRQ Cellnumbering.....	243
8. Printing and Exporting.....	246
8.1. Printing Esko Plato Files.....	246
8.1.1. Using PostScript.....	246
8.1.2. Printing to a PostScript device.....	246
8.2. Exporting Esko Plato Files.....	247
8.2.1. Exporting to a PostScript File.....	248
8.2.2. Exporting to an EPS File.....	257
8.2.3. Exporting to a PDF File.....	260
8.2.4. Export to an MFG File.....	268
8.2.5. Exporting to a CFF2 File.....	268
8.2.6. Export to a Normalized PDF File.....	269
8.2.7. Exporting to a PLA File.....	271
8.2.8. Exporting to a PDFPLA File.....	272
8.2.9. Exporting to a GRS File.....	273
8.2.10. Exporting to a JDF Layout File.....	274
8.2.11. Exporting to a CIP3 File.....	274
8.2.12. Exporting to a Polar File (Optional).....	276
8.2.13. Exporting to a JDF Cutting file.....	277
8.2.14. Digital Converting Export.....	277
9. Menus and Toolbars.....	281
9.1. Menus.....	281
9.1.1. File menu.....	281
9.1.2. Edit menu.....	282
9.1.3. View menu.....	284
9.1.4. Arrange menu.....	285
9.1.5. Station menu.....	286
9.1.6. Inks menu.....	287
9.1.7. Tools menu.....	287
9.1.8. Production menu.....	288
9.1.9. Trapping menu.....	289
9.1.10. Window menu.....	290
9.1.11. Help menu.....	291
9.2. Toolbars.....	292

9.2.1. Crosshair Position.....	292
9.2.2. Preview Selector.....	293
9.2.3. Layout Type.....	293
9.2.4. Station Selector.....	293
9.2.5. Status Bar.....	294
9.2.6. Tools.....	294
9.2.7. View Selector.....	317
10. Function after function.....	318
10.1. New Layout.....	319
10.2. Open Layout.....	320
10.3. Import CAD Data.....	320
10.4. Export Layout.....	321
10.5. Layout Setup.....	323
10.5.1. General.....	323
10.5.2. Plate and Sheet.....	324
10.6. Document Info.....	326
10.7. XMP Property Info	326
10.8. Print.....	327
10.9. Preferences.....	327
10.9.1. General.....	328
10.9.2. Defaults.....	329
10.9.3. View.....	331
10.9.4. Color.....	332
10.9.5. Editing tab.....	333
10.9.6. Files tab.....	334
10.9.7. Shortcuts tab.....	335
10.9.8. Licenses tab.....	335
10.9.9. Server and Resources.....	336
10.10. Preview.....	338
10.11. View Modes.....	339
10.12. Viewer.....	339
10.12.1. Welcome to Viewer.....	339
10.12.2. Define Press Settings.....	340
10.12.3. The Viewer window.....	343
10.12.4. Viewer Compare Tool.....	350
10.13. Select Die Shape.....	356
10.14. Place Station.....	358
10.15. Replace by.....	359
10.16. Assign Station to Design.....	359
10.17. Import CAD Design.....	360
10.18. Replace by Graphics File.....	362
10.19. Save Station.....	362
10.20. Export Station.....	363

10.21. Station XMP Info.....	363
10.22. Inks.....	364
10.23. Ink Mapping.....	367
10.24. External File Ink Mapping.....	368
10.25. Ink Coverage.....	368
10.26. Create Ink Histogram.....	370
10.27. Create Ink Eater Areas.....	372
10.28. Fill Ink Eaters.....	374
10.29. Fill Ink Eater Areas wizard.....	375
10.30. Compensation Details dialog.....	378
10.31. Open Ink Book.....	381
10.32. Manage Ink Books.....	382
10.33. Number Stations.....	382
10.34. Adjust Masks.....	384
10.35. Create Varnish.....	386
10.36. Manage Plates.....	387
10.37. SmartMarks.....	387
10.38. Staggered Cutting.....	392
10.39. Check Job Parameters.....	394
10.40. Workspace.....	395
10.40.1. Save Workspace.....	395
10.40.2. Manage Workspace.....	396
10.41. Toolbars.....	396
10.42. Align.....	397
10.43. CAD.....	398
10.44. Calculator.....	400
10.45. Colors.....	401
10.46. Color Factory.....	402
10.47. Effects.....	404
10.47.1. Changing the Opacity Gradient.....	405
10.47.2. Creating and Editing PDF Opacity Masks.....	406
10.48. Grids.....	407
10.49. Info Window.....	410
10.50. Layer Browser.....	411
10.51. Pan Window.....	413
10.52. SmartLayout (Optional).....	414
10.53. Search Options.....	415
10.54. Stations.....	417
10.55. Style.....	417
10.55.1. Fill.....	419
10.55.2. Stroke.....	420
10.55.3. Group.....	421
10.56. Transform.....	421

10.57. Color Picker.....	422
10.58. Grid Settings.....	422
10.59. Station Properties.....	426
10.59.1. Front/Back.....	427
10.59.2. Trimming.....	428
10.59.3. SmartLayout.....	428
10.60. Notes.....	430
11. Server and Resources.....	432
12. Appendix.....	434
12.1. QuickStep Compatibility.....	434
12.1.1. Opening QuickStep GRQ files.....	434
12.1.2. Conversion of GRQ data.....	434
12.1.3. Inks marked as 'Tech'.....	435
12.2. Studio.....	436
12.2.1. Introduction.....	436
12.2.2. Studio Designer Basics.....	437
12.2.3. Working with Panels.....	443
12.2.4. 3D Guides.....	450
12.2.5. Exporting.....	453
12.2.6. FAQ.....	457

1. Copyright Notice

© Copyright 2019 Esko Software BV, Gent, Belgium

All rights reserved. This material, information and instructions for use contained herein are the property of Esko Software BV. 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 BV 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 BV 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 BV.

This document supersedes all previous dated versions.

PANTONE® Colors displayed in the software application or in the user documentation may not match PANTONE-identified standards. Consult current PANTONE Color Publications for accurate color. PANTONE® and other Pantone trademarks are the property of Pantone LLC. © Pantone LLC, 2015

Pantone is the copyright owner of color data and/or software which are licensed to Esko to distribute for use only in combination with Studio Visualizer. PANTONE Color Data and/or Software shall not be copied onto another disk or into memory unless as part of the execution of Studio Visualizer.

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 (eay@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.

Part of this software uses technology by Global Vision. ArtProof and ScanProof are registered trademarks of Global Vision Inc.

Part of this software uses technology by Qoppa Software. Copyright © 2002-Present. All Rights Reserved.

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

Correspondence regarding this publication should be forwarded to:

Esko Software BV

Kortrijksesteenweg 1095

B – 9051 Gent

info.eur@esko.com

2. Getting Started with Esko Plato

2.1. Welcome to Esko Plato

Esko Plato™ is a powerful tool for the creation of print-ready layouts. It provides a robust set of automatic and interactive tools to create even the most complex step-and-repeat arrangements. Plato has features such as Input, Output and SmartMarks in common with Esko PackEdge.

The fundamental purpose of Plato is to automate the creation of production-ready printing plates. Speed, productivity and ease-of-use are all essential to providing the best possible product for your customer.

As Plato offers different modules, your system may not have all the features described in this publication. If necessary, contact your Esko salesperson for information about purchasing features not installed on your system. The two main modules are:

Esko Plato for Labels

Esko Plato is a software application for Intel-based PC's dedicated for highly productive and interactive Step and Repeat work. Plato comes as a feature-rich basic application with the option to add a number of Label workflow specific tools like SmartLayout. This is an estimation module that will fill a given page size with a minimum of waste. You are presented with a range of possible sheet layouts with immediate feedback about waste and overrun percentages, based on the desired quantities of each individual one-up. The Layout algorithms are designed to reduce the risk of human mistakes. This will guarantee continuity towards the end process of the printing workflow, making this very productive in large format offset or gravure label workflows.

Esko Plato for Cartons

In a folding carton or corrugated workflow, the die-cutting tools are usually manufactured simultaneously with the pre-press workflow. In other words: the sheet layout is determined by the converter, not by the prepress department. To make sure that the graphics are stepped up to the exact specifications of the converter, Plato comes with a module to open Manufacturing files as generated in ArtiosCAD. With the "print items" option, Plato will even look for the corresponding graphic files, thus highly automating the stepping task for a carton workflow. A wide variety of additional tools to finish the sheet layout are available in Plato: adding Station Numbers, adding print control strips in a dynamic fashion (SmartMarks), adding Ink Eaters etc... Final output is guaranteed consistent with the die shape and is ready for final output on proof or film.

2.2. Starting Esko Plato

To start Plato, log on to your system if necessary and then do the following:

1. Click Start, then point to Programs.
2. Point to the Esko folder and then to PackEdge-Plato 10.1.
3. Click the version of Esko Plato that you have installed.
4. The startup screen will appear.
5. The plate document window will replace the startup screen.



Or you can also simply double-click the desktop icon.

2.3. Concepts and ideas in Esko Plato

- [Stations and Layouts](#)
- [View Modes](#)
- [Typical workflows](#)
- [Document window elements](#)
- [Entering information](#)
- [External Files](#)
- [Calculator](#)

2.3.1. Stations and Layouts

The primary objects that you create in Esko Plato are stations and layouts. A station is the object that you want to repeat on a plate. You may have several copies of the same station or of different stations in one layout.

You may already be familiar with this concept. Sometimes the terms 1-up (the non-repeated object) and layout are used to indicate stations and plates. In ArtiosCAD, a design file is the equivalent of a station and a manufacturing file is the equivalent of a plate.

- [What is a station?](#)
- [What is a layout?](#)

What is a station?

A station can contain a graphic, a die shape, a mask and station marks. You are not required to have a die shape, a mask and station marks in your station. All stations should contain a graphic, but Esko Plato has the notion of 'empty stations' which are stations without a graphic (see further). The concept of empty stations is designed to allow you to assemble plates before the final graphics become available (cost estimating purposes). In a normal workflow, empty stations will be replaced by the actual stations before making plates. The graphics used in a station can either be a (Normalized) PDF file or an Esko GRS file. You can import a die shape from a CAD file or you can create simple dies yourself. All the items that you include in a station will be repeated on your plate. Although stations can be created and manipulated from

the Sheet View, you edit stations in the Station View. The definition of a station is embedded inside the plate file. Optionally stations can be saved as separate files (for re-use in other plates).

Station objects consist of:

- a reference to a graphics. In case of an empty station this will be a rectangle with a dummy color, simply for viewing.
- a die shape. This is the shape on which the actual stations will be cut.
- a bleed mask. This is a gross mask that cuts away all irrelevant graphics. Most of the time, the bleed mask is generated by spreading the die shape over a certain distance.

Advanced attributes are:

- station / strip number definitions
- SmartMarks / geometry marks

The definition of a station (i.e. all attributes enumerated above) is enclosed in the PLA file. In case a station is used in another PLA file, the definition of that station could be copied into a STA file. The PLA file, however, will continue to use the internal definition. Apart from the enumerated attributes, a station can also possess a number of SmartLayout characteristics as e.g. ordered quantity. SmartLayout characteristics are not stored inside the STA file.

What is a layout?

A layout contains a plate size, a sheet size, a sheet-to-plate offset, any number of stations, and marks. The stations contained in a layout may be multiple instances of the same station, single instances of different stations, or some intermediate mix (a combination layout). As plates can be recto/verso, Esko Plato can create the back of a plate layout from the front for two-sided printing. When Esko Plato creates the back side of the plate, the position of the stations is mirrored for accurate front-to-back alignment. The back side of the stations is automatically inserted to create the printing on the back side. Esko Plato maintains a link between the front and back side of a plate: moving stations on the front side will move the corresponding stations on the back side accordingly.

2.3.2. View modes

Esko Plato has two view modes:

- The *Sheet View* shows the plate, the sheet, and all the stations that have been placed. In this view, you can switch between the front and the back of a plate.
- The *Station View* focusses on the individual station.

Switching between those views can be done in one single session.

Sheet View

In the Sheet View, you can assemble the layout by stepping up a number of copies of one or multiple stations. How to step and repeat depends very much on your workflow. In the Sheet View, dedicated dialogs and tools will support different *workflows*. Common to all workflows are the setup dialog, where plate and sheet sizes are defined, and the station dialog, which gives an overview of all stations used in a layout.

Station View

In the Station View, you are basically working on one side of a station. As stated earlier on, a station can contain graphics, a die shape, a mask and station marks. The Station View offers all functionality to create and manipulate these objects:

- import graphics
- create a die shape
- align graphics and die shape
- create a mask (based on the die shape)
- add station marks
- add station numbers
- ...

2.3.3. Typical workflows

Your first task when starting a new plate layout is to determine which of Esko Plato's workflows you need to use. The workflow that you choose will determine how to start building your plate layout. Esko Plato supports different workflows: a CAD-based, SmartLayout and Grid-based Layout Workflow. The fundamental difference between the CAD-based layout on the one hand and the SmartLayout or Grid-based layout on the other hand is that in CAD-based layouts the position of the stations is dictated by the CAD layout whereas in the other workflows, the position of the stations can be defined manually (manual workflows).

The process of creating a plate is usually a logical progression. First you define the sheet and plate size. Next, you create the stations that you plan to include on the plate. You can either use the Station dialog box or the Station View to create stations containing graphics, dies, masks, and station marks. Finally, you add stations and marks to the plate to create a file that is ready for printing.

When creating a new plate or layout, the central dialog for the chosen workflow will automatically be opened. Choosing a particular workflow does not limit the available tools. You will still be able to use grid-based stepping after choosing a CAD based workflow.

Tip:

The [Layout Assistant](#) provides some info on the different workflows. As a result a new layout will be created and the Setup dialog box will be opened.

- **The CAD Workflow:** The CAD Workflow uses an imported CAD file as the basis of the plate layout. First the imported CAD file defines the sheet size, and the placement and orientation of the stations. Next stations are assigned to the CAD layout. The last step is to add the plate marks to the plate and sheet.
- **The Grid-based Workflow:** In a Grid-based layout, stations are stepped in blocks where both the number of repetitions in vertical and horizontal direction and the step distances can be specified numerically.
- **The SmartLayout Workflow:** In a SmartLayout workflow, a new layout will be calculated based on the ordered quantity and the given sheet size will be filled with a minimum of waste. You

are presented a range of possible sheet layouts with immediate feedback about % waste and overrun, based on the desired quantities of each individual one-up.

- A *manual workflow* does not use an imported layout. Instead, you create your stations and then place and arrange them on the sheet.
- Esko Plato also offers you the possibility to *combine workflows*.

2.3.4. Document window elements

The different views all contain a display area and a number of toolbars: see *toolbars*.

Plato 12.0.0 ALPHA - [tequila-test.pdfpla]

File Edit View Arrange Station **4** Tools Product **5** Window **6** Printing Help **7** **8**

View: [Icons] Side: [Icons] Preview: [Icons] 90% [Icon]

[Navigation Icons] **10** designer

0 100 200 300 400 500 600 700 800 900 1000



12 [Layout Settings] 317.500 mm + 482.600 mm 0.000 **13**

[Status Bar] untitled [Masks: up to date!] [Coordinates] -23.040 851.118

1. The **display area** is where you do your work. You can think of it as a blank printing plate on which you are assembling your layout. The white area is the plate, the blue rectangle indicates the borders of the sheet. The size and position of these are defined when you create a new layout.
2. The *Tools toolbar* contains the interactive tools. Some tools are only available in 1 view (Station view or Sheet View).
3. 'File & Edit' offers you the standard save and edit functionality.
4. *View Selector*
5. **Side** indicator tells you which side of the sheet you are working on; front or back and allows to switch from front to back and vice versa.
6. *Preview Selector* indicates the selected preview.
7. **Zoom**: indicates the zoom factor.
8. **Show Trim & Media box**: allows you to switch the visualization of the trim and media box on or off.
9. **Show values / formulas**: adjusts the visualization of formula-enabled fields. You can opt to either show the formula definition or the corresponding value.
10. The *Station Selector* allows you to select a station in the Station View.

Tip:

Click the station button to open the Station dialog box.

11. *Layout type* indicates the selected workflow: grid-based, CAD, SmartLayout or manual.

Tip:

Click the layout button to open the appropriate Layout dialog box (Grids, CAD, or SmartLayout).

12. *Crosshair Position*

13. **Current Orientation / Current Unit**: the orientation and unit can be changed in the *Defaults tab* of the Preferences dialog box. Orientation, units and server name together make up the *Status Bar*.

Note:

The menu bar contains commands that are available in the current module. Not all commands in all menus may be available. Availability depends on the options purchased and the active tool.

Note:

All the toolbars can be opened via the Window menu > Toolbars.

Tip:

The *Workspace Layout Manager* allows you to make some predefined workspaces of the user interface dialog boxes and switch from one to another in one single click.

Tip:

All toolbars of the Window menu can either be used as docked tools or as proper dialog boxes.

2.3.5. Entering information

Many of the dialog boxes expect you to enter information. You usually do this by typing on the keyboard. To edit a value in a dialog box, click the field and enter a new value. Clicking a field once selects the entire contents of the field. Clicking a field twice will insert the cursor into the existing value in the field and allow you to edit it.

There are two other ways to work with dialog boxes that you may not have seen before:

- [Dragging numbers](#)
- [Using smart edit fields](#)

Dragging numbers

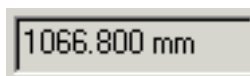
Esko Plato allows you to drag a number between two dialog boxes. Sometimes, you might want to use a value from one dialog box in another. For example, you might want to take some measurements from within your document and then use that measured value as an entry in the Transform dialog box.

To drag a number from one dialog box to another:

1. Click and drag the number that you want to use. The cursor turns into the pointing hand icon.
2. Drag the number to the edit field of another dialog box and release the mouse button. The value is copied from the first dialog box to the second.

Using smart edit fields

Many of Esko Plato's dialog boxes contain smart edit fields. You can recognize smart edit fields by their display: they always contain the current units label after the value in the field.



Smart edit fields have two important properties. First, you can change the current units used to display the field's value at any time. Second, you can use the smart edit field to calculate a new value for the field.

To change the current units that are used to display the field's value:

1. Click the field to select its contents. Everything in the field is selected.
2. Type the units that you would like to use. Type "in" for inches and "mm" for millimeters.
3. Press [Enter]. The field's value is now displayed in the units that you typed.

The field will continue to use the unit of measure that you chose until you change it again. Note that other fields in the same dialog box will be displayed in the previous units until you change

them. To change the default units used throughout the application, see the [Defaults tab](#) in the Preferences dialog box.

To use the smart edit field to calculate a new value:

1. Click the field twice to insert the cursor after the unit of measure. If the cursor is not in the right place, use the arrow keys to move it after the unit of measure.
2. Type the calculation that you'd like to make. For example, if the field currently contains "10 mm" type "+5" to add 5 mm to this value.
3. Press [Enter]. The new value for the field is calculated and displayed. In the example above, the new value will be "15 mm" (10 mm + 5 mm).

You can use these two features together, too. For instance, if the smart edit field contains "10 mm", you can add 2 inches to this value by typing "+2 in". The resulting value will be "60.8 mm", 10 mm + 50.8 mm (2 inches in millimeters).

2.3.6. External Files

Esko Plato uses files as external references. External files are linked to the actual files located on your disk. This means that the high resolution data contained in many graphics files is never included in your plate or station files. Examples of external files are graphics used in stations, and image marks.

The use of external files has two advantages. First, because the real data of the external file is not included in the document, Esko Plato's native plate files are very small. Second, because Esko Plato refers to the external files on disk, stations and plates that use these files are always up-to-date. Changes made to the external files are updated automatically.



Caution:

Note that whereas all graphics used in stations are external references, the station definitions themselves (the link to graphics, die shape, mask, station number, trapping layer, etc...) are stored inside the plate file.

Tip:

When inks are changed in external files, PLA files should be opened and saved.

2.3.7. Calculator and Parameter Formulas

Esko Plato offers you the possibility to use parameters to build expressions or formulas.

- [What are parameters?](#)
- [What are predefined parameters?](#)
- [Using the Calculator](#)
- [How to add a parameter to the parameter\(s\) in the Formula line?](#)
- [Using the Calculator parameter buttons](#)

What are parameters?

Parameters represent properties of a step and repeat job. Every parameter has a unique name. Parameters can be used to build expressions, the so-called formulas. These expressions can be used to define most settings of a step and repeat job. E.g. the parameter called 'PlateHeight' represents the vertical size of the plate. You can define the vertical size of the sheet by the formula 'PlateHeight - 100mm'. In doing so, the vertical size of the sheet will be 100 mm smaller than the vertical size (height) of the plate.

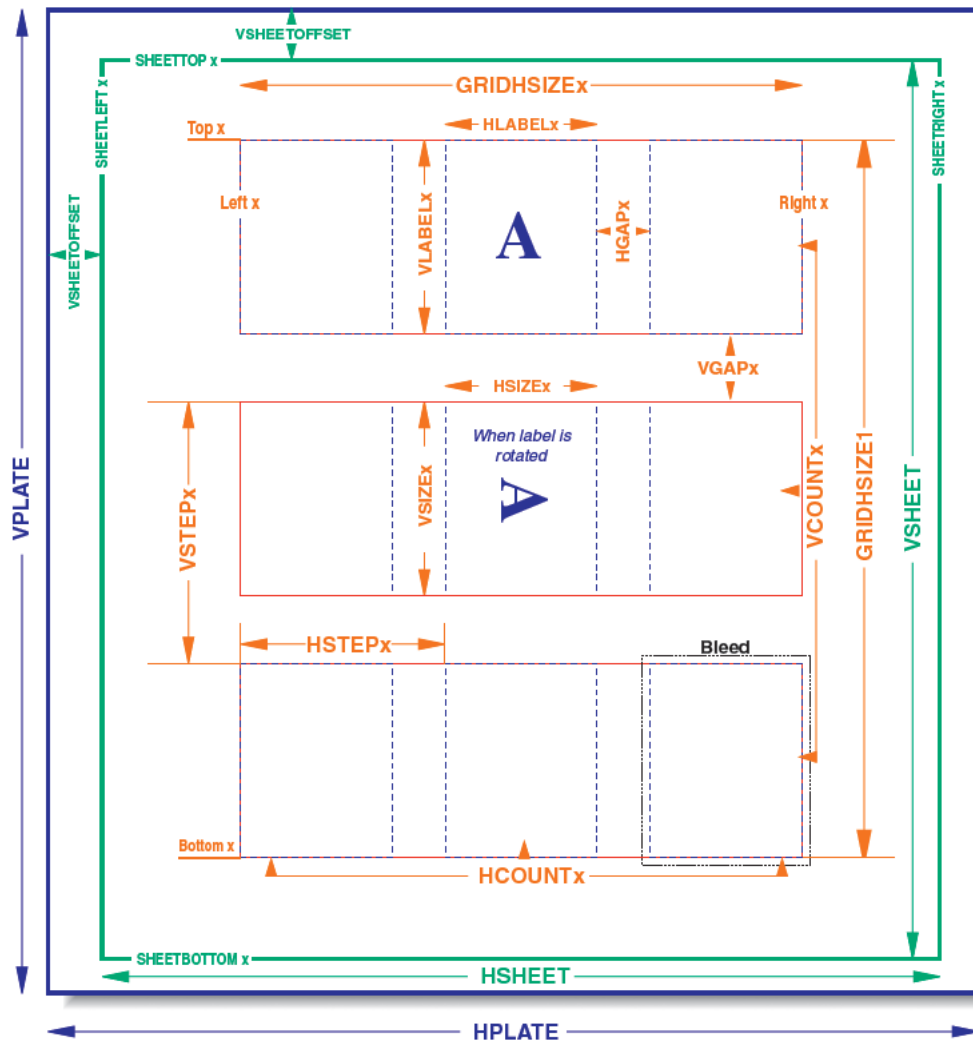
Formulas can be used in different input fields:

- [Layout Setup](#): all fields marked with 'fx'.
- [Grids](#): offset, count, step and angle fields. Fields that support formulas can be visualized in 2 modes: either show the formula definition or show the corresponding value. When moving the mouse over the field, both the definition and the value will be shown in balloon help.

To visualize the parameter formulas, activate Show formulas in the [Layout Setup dialog box](#).

What are predefined parameters?

Esko Plato has a list of predefined parameters. These parameters can be visualized in 2 formats, either as **technical** parameters or as **localized** parameters. Technical formulas correspond with the way in which formulas were visualized in QuickStep. Localized formulas provide a more readable format. The 'Formulas' option in the [Defaults tab](#) of the Preferences dialog box offers you the possibility to switch between technical and localized parameters.



When using parameter values, you are allowed to combine both localized and technical formatting.

Technical Parameter	Localized Parameter	Explanation
VPLATEFIT	PlateHeightFit	vertical plate size
HPLATEFIT	PlateWidthFit	horizontal plate size
VPLATE	PlateHeight	vertical plate size when fitted around all objects in on a plate (so including stations but also SmartMarks and geometry marks)
HPLATE	PlateWidth	horizontal plate size when fitted around all objects in on a plate (so including stations but also SmartMarks and geometry marks)
PLATETOP	PlateTop	position of the top of the plate

Technical Parameter	Localized Parameter	Explanation
PLATELEFT	PlateLeft	
PLATEBOTTOM	PlateBottom	
PLATERIGHT	PlateRight	
GRIPPER	Gripper	
VSHEET	SheetHeight	vertical sheet size
HSHEET	SheetWidth	horizontal sheet size
VSHEETFIT	SheetHeightFit	vertical sheet size that fits around all grid blocks
HSHEETFIT	SheetWidthFit	horizontal sheet size that fits around all grid blocks
VSHEETOFFSET	SheetOffsetHeight	vertical offset of the sheet relative to the plate
HSHEETOFFSET	SheetOffsetWidth	horizontal offset of the sheet relative to the plate
SHEETTOP	SheetTop	top position of the sheet relative to the plate
SHEETLEFT	SheetLeft	left position of the sheet relative to the plate
SHEETBOTTOM	SheetBottom	bottom position of the sheet relative to the plate
SHEETRIGHT	SheetRight	right position of the sheet relative to the plate
NROFINKS	NumberOfInks	number of non-technical inks (technical inks will not be printed)
AREATOP	AreaTop	
AREALEFT	AreaLeft	
AREABOTTOM	AreaBottom	
AREARIGHT	AreaRight	
NROFGRIDS	NumberOfGrids	total number of grids

per grid:

Grid-related formulas are followed by a grid number.

Technical Parameter	Localized Parameter	Explanation
VLABEL	LabelHeight	vertical size of non-rotated job

Technical Parameter	Local Parameter	Explanation
HLABEL	LabelWidth	horizontal size of non-rotated job
VGAP	GapHeight	
HGAP	GapWidth	
VSTEP	StepHeight	vertical step size, from center point to center point
HSTEP	StepWidth	horizontal step size, from center point to center point
VCOUNT	VerticalRepetitions	
HCOUNT	HorizontalRepetitions	
GRIDVSIZE	GridHeight	vertical size of the grid (top-bottom)
GRIDHSIZE	GridWidth	horizontal size of the grid (left-right)
TOP	Top	top position of the grid relative to the sheet
BOTTOM	Bottom	bottom position of the grid relative to the sheet
LEFT	Left	left position of the grid relative to the sheet
RIGHT	Right	right position of the grid relative to the sheet
ANGLE	Angle	rotation angle
VSIZE	StationHeight	vertical size of bounding box of rotated job
HSIZE	StationWidth	horizontal size of bounding box of rotated job

A parameter can represent:

- A value. A value can be either a plain value or the result of a formula.
 GRIDVSIZE1 is a parameter. GRIDVSIZE1 could either be the plain value 25 or the formula $VSTEP1 * VCOUNT1$ equal to 25.
- A grid parameter. 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. Each predefined grid parameter is assigned a number that corresponds to the job of the grid you are currently working on.
 The VSIZE parameter of the first job is called VSIZE1. The VSIZE of the second job will be called VSIZE2 as this parameter refers to the job in the second grid.
- A logical relationship between parameters. The parameter VSTEP1 may amongst others depend on the parameter VSIZE1.
 Example $VSTEP1 = (VSIZE1 + 10)$.

Using the Calculator

To activate the Calculator right-click any formula-enabled input box. The Calculator allows you to compose complex mathematical formulas using numeric values and/or the parameters. You can use predefined parameters, define new parameters, modify existing parameters or remove parameters that you have created previously.

Advantages of the Calculator:

- It has a wide input field, which makes it easier to display long formulas.
- You can create formulas without having to enter parameter names. You can simply select them by clicking the parameter in the parameter column.

How to add a parameter to the parameter(s) in the Formula line?

1. Right-click any formula-enabled input box to activate the Calculator. The input field that is connected to the calculator will be marked with a red rectangle.
2. Put your cursor in the formula line and click once. The cursor appears at the position you clicked.
3. Click the required category of parameters in the left-hand column and select the required parameter or value from the parameter list from the right-hand column. Either click 'Insert' or double-click the parameter.
4. The value from the calculator will be transferred to the selected input field when pressing ENTER or when clicking another input field.

Using the Calculator parameter buttons

Apart from the standard parameters, Esko Plato offers you the possibility to create, modify or remove customized parameters.

- [Add a new parameter](#)
- [Modify a parameter](#)
- [Remove a parameter](#)

Add a new parameter

1. Right-click any formula-enabled input box in the Layout Setup or Grids dialog box. The Calculator appears.
2. Select the category 'Custom' from the left-hand column.
3. Click 'New'. The Add dialog box appears.
4. Enter the name of the new parameter in the Name input box.
5. Enter the parameter formula or parameter in the Formula input box.
6. Select the type of parameter you want to define from the Type drop-down list.
7. Click 'Add'. The new parameter or parameter formula is checked for errors and is added to the parameter list of the Calculator.

Modify a parameter

1. Right-click any formula-enabled input box from the Layout Set or Grid dialog box. The Calculator appears.
2. Select the category 'Custom' from the left-hand column.
3. Click 'Modify'. The Modify dialog box appears.
4. Select the parameter you want to modify from the Name input box.
5. Modify the parameter formula in the Formula input box.



Caution:

The standard predefined parameters, that come with Esko Plato, cannot be altered.

6. Modify the type by accessing the Type drop-down list.
7. Click 'Modify'. The modified parameter is now checked for definition errors and the parameter will become available in the parameter list.

Remove a parameter

1. Right-click any formula-enabled input box in the Layout Setup or Grids dialog box. The Calculator appears.
2. Select the category 'Custom' from the left-hand column.
3. Click 'Remove'. The Remove dialog box appears.
4. Select the name of the parameter you want to remove, from the Name input box. The parameter appears in the Name input box and its formula in the Formula input box.

Note:

You can only remove parameters you have previously defined and which are not longer used in the current repetition.



Caution:

If a parameter is still in use in the current repetition, the Remove dialog does not allow you to select it.

5. Click 'Remove'. The parameter is removed from the Calculator parameter list for this repetition.

2.3.8. Shuttle

Shuttle allows to connect to an EskoArtwork server: Automation Engine 10, Nexus or Odystar. It allows to submit the current job to a workflow queue, and to monitor jobs running on the server.

All Shuttle functionality can be found in **File > Shuttle**

A full explanation on Shuttle can be found in the Shuttle documentation, available on the EskoArtwork Documentation DVD.

2.3.9. Studio

Studio offers a 3D viewing environment where designers can see their artwork applied instantly on the packaging. It also includes navigation and alignment tools, and by writing a 3D PDF file from Studio, designers can share their ideas with clients for review and approval.

Studio reduces time-consuming mock-ups, spots design-errors faster and takes away the guesswork from 2D flats.

For folding carton or corrugated packaging, the shapes can be supplied by ArtiosCAD or Score!.

The Studio tools in Plato can be found in **Production > Studio Designer**

2.3.10. Common data resources

The application needs access to the following Data Resources for correct working:

- **Fonts** data (in folder bg_data_fonts_v040)
- **Color Engine Database** (CMS) (in folder bg_data_cms_v010)
- **Custom data** (in folder bg_data_custom_v010)
- **SmartMarks data** (in folder bg_data_marks_v010)
- **Dot gain Compensation Curves** (DGC) (in folder bg_data_dgc_v040)

The location of these Data Resources can be found from one of the following places:

1. Central Data Resources.

Central Data Resources will be used if the application is connected to an Automation Engine Server. The Common Data Resources need to be installed on the Automation Engine server.

2. Remote Data Resources

The Data Resources are used from a remote system in the network. The Common Data Resources need to be installed on this remote computer. This is an ideal setup when 2 or more stand-alone applications (i.e. not connected to an Automation Engine server) need to share the same Data Resources.

3. Local Data Resources

The Data Resources are used from the current system. The Common Data Resources need to be installed on the current system.

4. Application embedded Data Resources.

A minimal set of Data Resources is embedded in the application. When none of the above Data Resources is found, the embedded Data Resources will be used. These embedded Data Resources can not be shared between different applications

You can set the Central Data Resources in the Preferences: see [Server and Resources](#) on page 336

Note:

If the Central or Remote Data Resources are not available during startup of the application (for example the server is not running), the application will fall back on the Local Data Resources or, if the Local Data Resources are not installed, on the embedded Data Resources.

3. Working with Plate Files

This section will provide you with the basics of creating plate files. You will learn how to define new plates and their parameters. You will also learn how to import CAD layouts for your plate and how to create, save and use templates to automate your work.

- [Creating, opening and saving plates](#)
- [Creating Layouts](#)
- [Using a CAD file for plate layout](#)
- [Using Templates](#)

3.1. Creating, opening and saving plates

When you create and save a plate layout from Esko Plato, you create a layout file. You can save your plate files wherever you like. Esko Plato will maintain its links to the external files that the plate uses. You might like to arrange your plate files into folders that contain the plate file itself and the graphics used in the stations. It is up to you to decide what works best for your company. A plate file is always saved with a pla (or pdfpla) extension. So, saving a file with the name "3Up" will save a file on your disk with the name "3Up.pla", if the graphics that you used are .grs files. Plate files always have the Esko Plato plate file icon



- [Creating a new layout](#)
- [Saving a layout](#)
- [Saving a layout with a different name](#)
- [Opening an existing layout](#)
- [Changing the sheet or plate size](#)

3.1.1. Creating a new layout

When you create a new layout, you need to feed Esko Plato with vital information as e.g. the size of the plate, the size of the sheet and the position of the sheet on the plate. You can do this in the [Layout Setup](#).

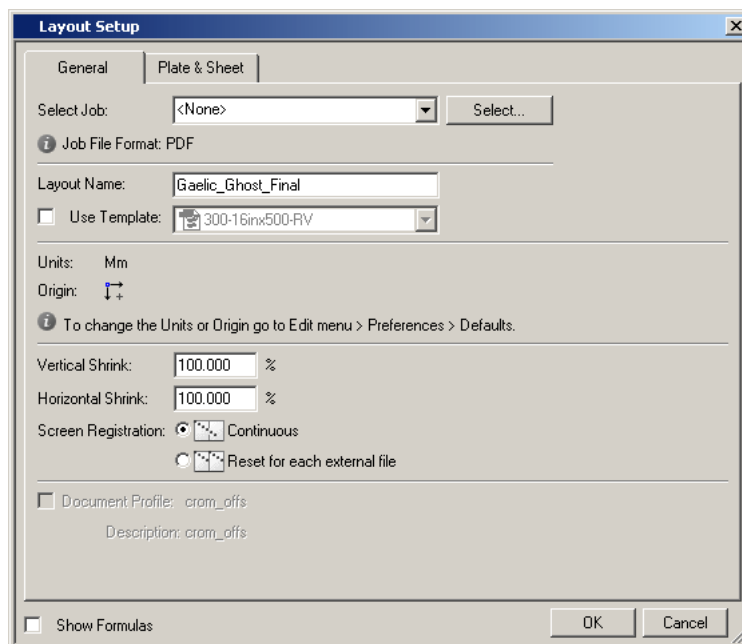
This information will often change for plates with different types of printing processes, plates and presses that you use during production. If you often create the same size of plates and sheets for the same press, you can automate the creation of a plate file by using Templates. See [Using Templates](#) for more information.

Note:

What is the difference between a PDFPLa and a PLA file? A PLA file contains references to GRS files, a PDFPLA file contains references to PDF files. The first type of file that you place on a new plate determines the kind of files you can place after that. Placing a GRS file first means that you cannot place PDF files later on, placing a PDF or a normalized PDF file first means that you cannot place GRS and STA files later on. The mixing of the file types GRS and/or STA with PDF and normalized PDF is not supported.

To create a new layout file:

1. On the File menu, click 'New Layout' and select the type of layout you want to create. The Layout Setup dialog box will appear:



2. Fill in the fields in the dialog box to suit your production needs. Refer to the chapter '[Layout Setup](#)' for detailed information about each field.
3. Click 'OK' to create the new layout. Clicking 'Cancel' will close the Layout Setup dialog box without creating a new plate.

Your new plate is shown in the document window. The white area is the plate, the blue line shows the sheet and its position on the plate.

Tip:

Configure the CTRL+N shortcut to your favorite layout workflow (Edit menu > Preferences > [Shortcuts](#)).

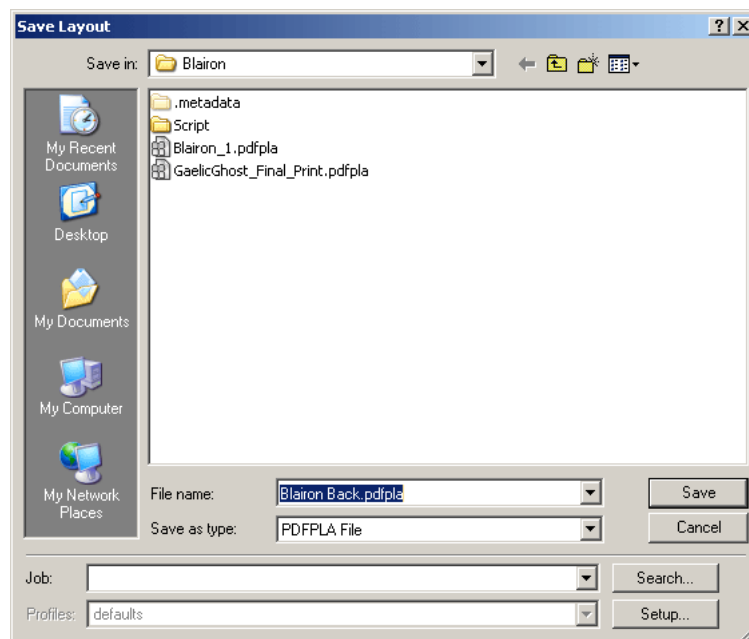
3.1.2. Saving a layout

Once you have created a plate, you need to save it. When you save a plate, and you haven't yet specified a name for the layout in the Layout Setup dialog box (you left it to untitled), you will be prompted to enter a file name and the location where you would like to save it.

It is good practice to save your work at frequent intervals. Thus you know that the file is up-to-date and that your work has been saved.

To save a plate file:

1. On the File menu, click 'Save Layout'. The Save dialog box appears:



2. Enter a name for your plate file.
3. Indicate in which directory you would like to save the file.
4. Click 'Save' to save the contents of your file. Clicking 'Cancel' will close the Save dialog box without saving your file.

Now that your new file is saved with the name that you gave it, you can save it at any time by clicking the File menu and clicking Save. You can also use the keyboard shortcut CTRL+S to save the file as you work.

3.1.3. Saving a layout with a different name

To save an existing file with a new name:

1. On the File menu, click 'Save Layout as'.
2. Type a name and choose a location for the file.
3. Click 'Save' to save the file with the new name.

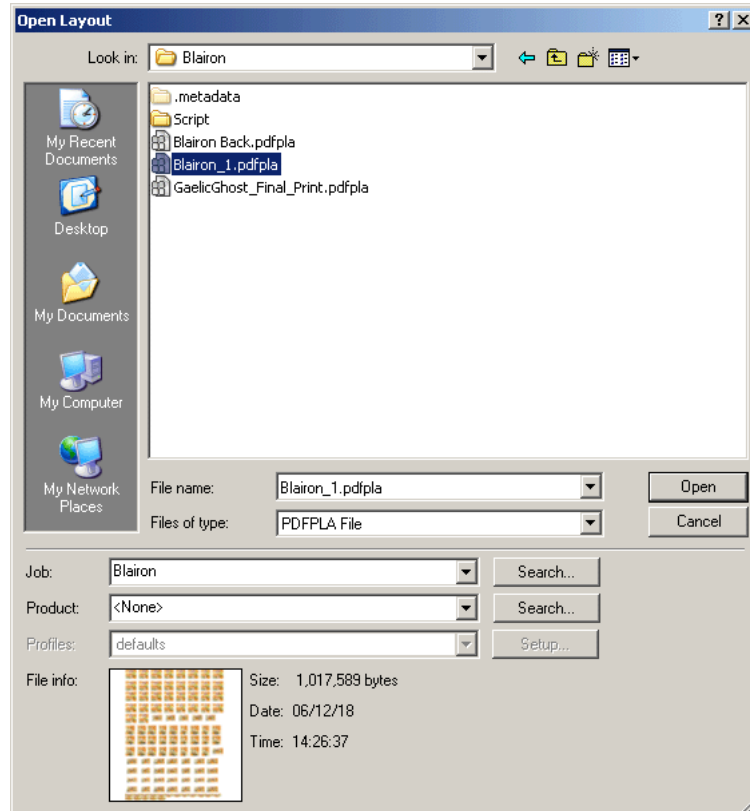
You can use Save As to create a copy of an existing file before you make major modifications. Then you can always go back to the original version of the file if you need to.

3.1.4. Opening an existing layout

You often want to open an existing plate file to make changes to it or to finish a plate that you had started earlier. Use 'Open Layout' to open an existing layout file for editing or printing.

To open an existing plate:

1. On the File menu, click 'Open Layout'. The Open dialog box appears:



2. Locate the file that you would like to open and select its name.
3. Click 'Open'. Clicking 'Cancel' will close the Open dialog box without opening an existing file.

You can only have one plate file open at a time. If you already have a file open when you click 'Open', the current file will be closed. If you have any unsaved changes, you will be asked if you want to save them first.

3.1.5. Changing the sheet or plate size

You might need to make changes to the sheet or plate size of an existing layout for a last minute change.

To change the sheet or plate size of an open file:

1. On the File menu, click 'Layout Setup'. The Layout Setup dialog box appears.
2. Turn to the Plate&Sheet tab to check the values of the file.
3. Edit the fields that you need to change.
4. Click 'OK'. Clicking 'Cancel' will close the Layout Setup dialog box, leaving the current plate with its previous values.

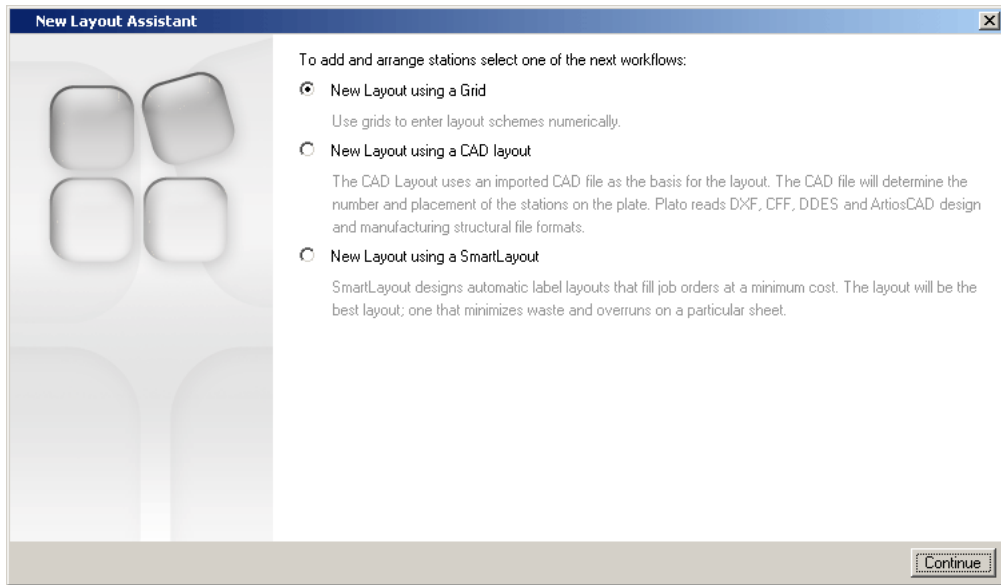
Esko Plato will monitor the position of the stations relative to the sheet when you change size. When moving the sheet-to-plate position, the stations will remain fixed relative to the sheet edge. When moving the sheet, the stations will remain fixed relative to the plate edge. An exception is made for grid objects. As all grid objects have offsets relative to the sheet edge, changing the sheet dimensions will cause these objects to move with the changed sheet size.

3.2. Creating Layouts

- [Layout Assistant](#)
- [Grid-based workflow](#)
- [CAD workflow](#)
- [SmartLayout workflow](#)
- [Manual workflow](#)
- [Combining workflows](#)

3.2.1. Layout Assistant

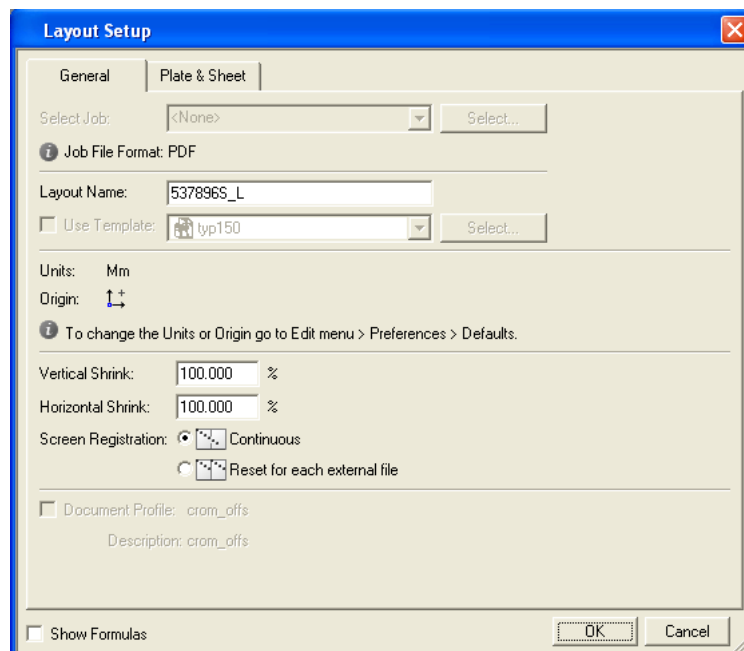
The Layout Assistant provides some info on the different workflows. As a result a new layout will be created and the Setup dialog box will be opened.



3.2.2. Grid-based workflow

A grid-based layout may turn out to be a very easy and practical method if you have to print labels or do straightforward step-and-repeat jobs. You can then organize various one-ups in several grids whereby Esko Plato's set of tools should help you to make optimal use of the size of the sheet.

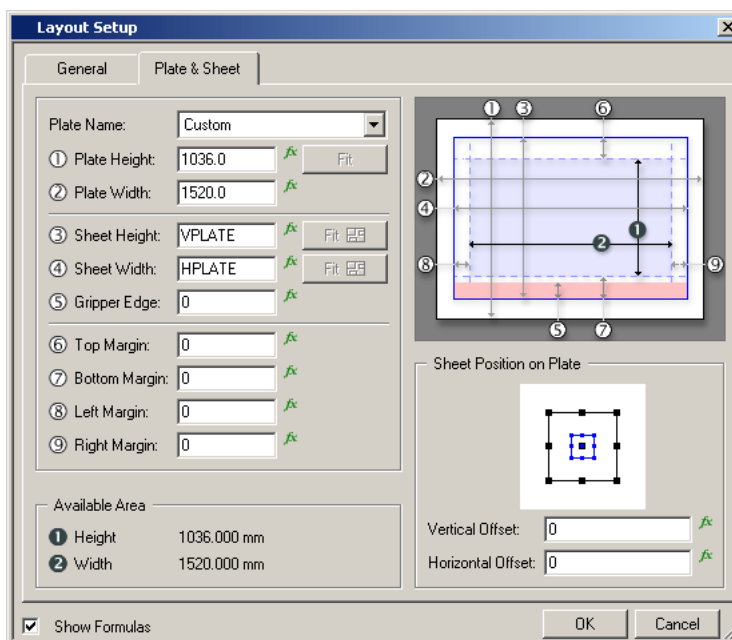
1. Go to File > New Layout > Grid-based Layout... The *Layout Setup dialog box* will pop up.



2. The General tab of the Layout Setup dialog box offers you the possibility to determine some general settings. If your Esko Plato is connected to an Automation Engine server, you can specify in which Automation Engine **Job** the new layout will be created. It indicates the **Job File Format** in which your job will be created: PDF or GRS. It invites you to give a **name** to your layout. Note that templates are at your disposal. You can select one by toggling on 'Use **Template**' and selecting one from the drop-down list.

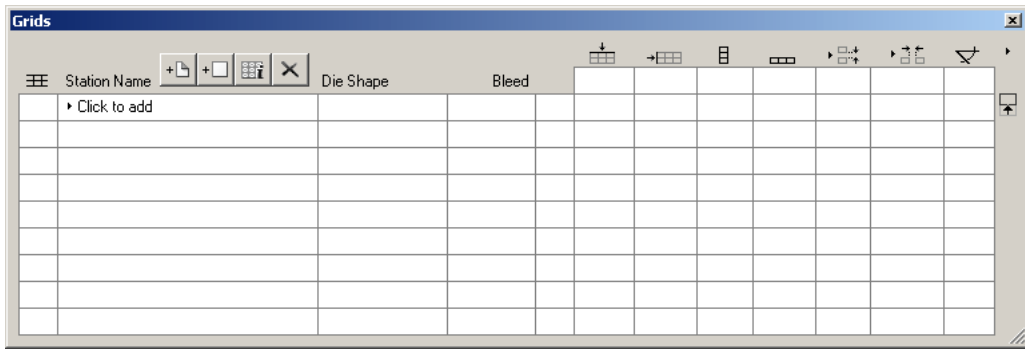
The **Shrink** values are relevant only to particular printing processes such as flexography, in which case the plate tends to expand as it is wrapped around the press cylinder. **Screen registration** is relevant only to Esko FlexRip users. When activated, the origin of the screening will be reset for each station. In doing so, it is guaranteed that exactly the same screening will be applied for each station. When deactivated, screening will be done continuously throughout the complete job.



3. The Plate & Sheet tab allows you to specify the dimensions of your plate and sheet.

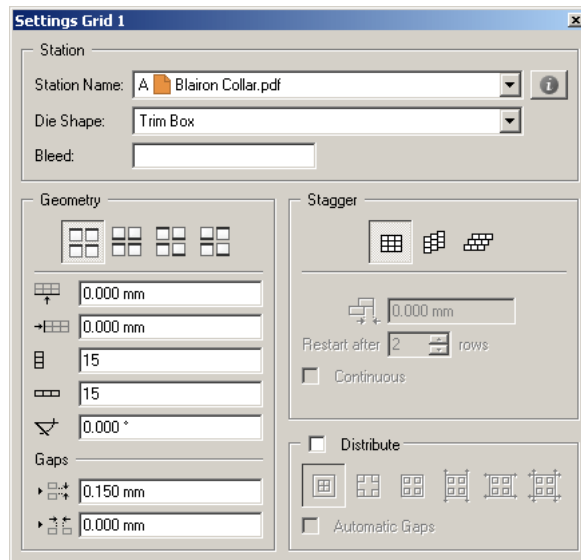


To facilitate this procedure, you have a number of formulas at your disposal, which you can activate by clicking the 'fx' symbol to the right of each entry field. When you click 'fx', the [Calculator](#) pops up.

4. As soon as you have entered all relevant values (or formulas), the [Grids dialog box](#) appears. It serves as the central dialog box for all further actions.



5. Add stations by clicking the 'Add Station' icon  For more information, please refer to the chapter ['Adding a station'](#) or check the function overview in ['Grids'](#).
6. Clicking the grid info button  , will open the [Grid Settings dialog box](#).



The Settings dialog box gives you the opportunity to create an even more complex layout on the sheet, since you can alter geometry, gaps between the various stations, stagger them or distribute them in various ways. You can also determine 'Die Shape' and 'Bleed'.

For more information, please refer to the chapter ['Creating a Repetition'](#) or check the function overview in ['Grid Settings'](#).

3.2.3. CAD workflow

The CAD Workflow uses an imported CAD file as the basis for the layout. The CAD file will determine the number and placement of the stations on the plate. It may also determine the sheet size and the masks that are used for the stations that the CAD file defines.

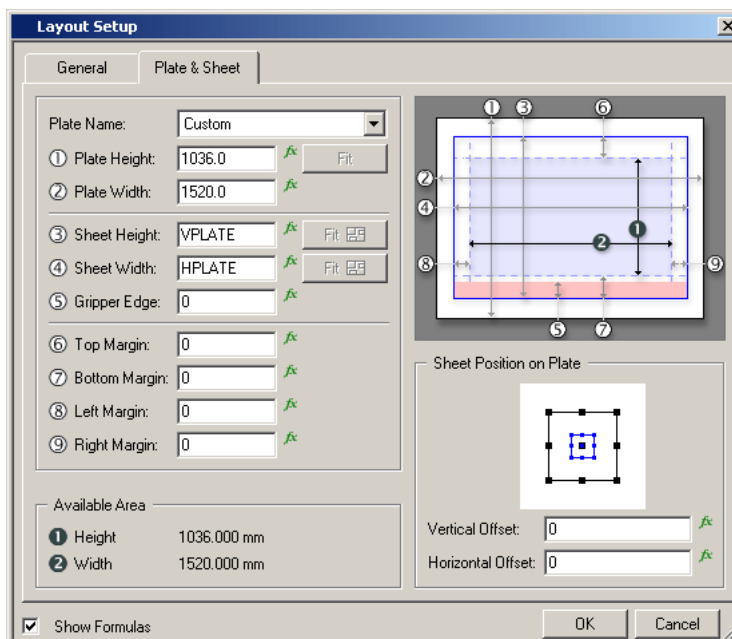
The CAD Workflow is a structured workflow. Esko Plato considers an imported CAD file as its guide to creating the plate layout. Because an imported CAD layout defines the final cutting and folding of a layout, it is not editable in Esko Plato. This means that you will not be allowed to scale or rotate an imported CAD layout. In addition, you cannot change the arrangement or number of the stations defined in the CAD layout. If you need to make these types of changes, you should edit the original CAD file and use the new file in Esko Plato.

There are some parts of an imported CAD layout that you can change. These include the plate and sheet size, the location of the layout on the sheet, and the stations that are assigned to each location on the plate. You can also adjust the mask and bleed sizes of a station to suit your company's needs.

Please remember that it is also possible to release the CAD form from your layout in order to make changes interactively. Once released, the CAD form cannot be connected to your CAD form again.

The basic steps to create a plate layout from a CAD file are:

1. Go to File > New Layout > CAD Layout... The *Layout Setup dialog box* will pop up.

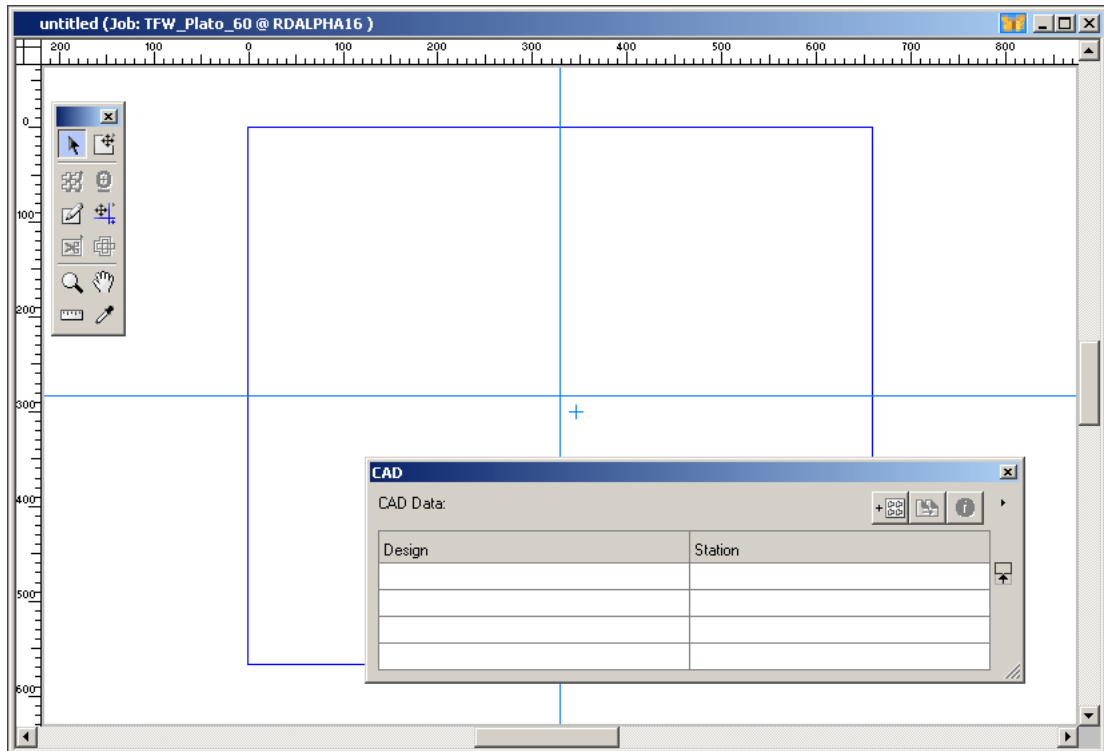



2. Go to the Plate & Sheet tab and select the appropriate plate or enter the plate size manually.

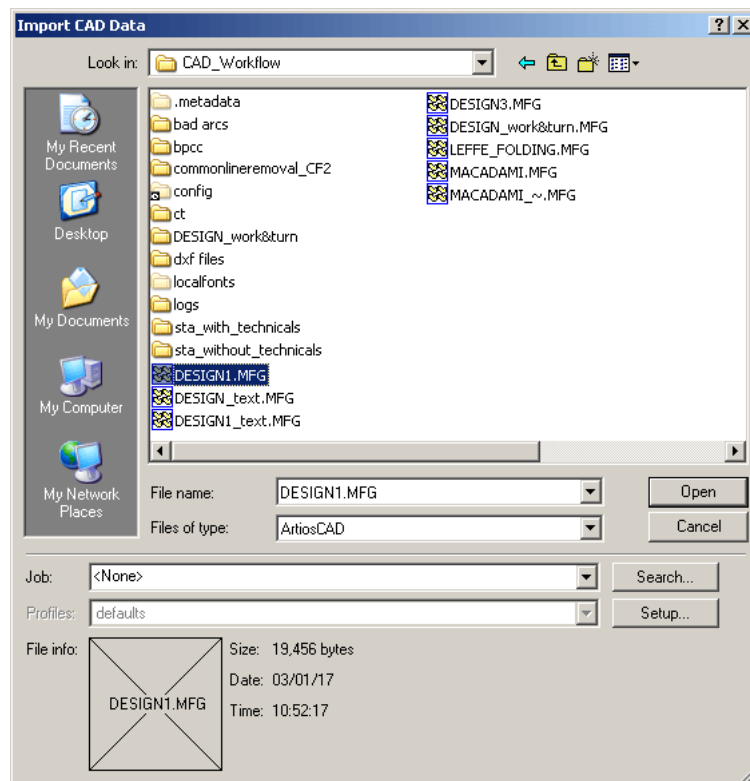
Note:

In a CAD workflow, the sheet size will be extracted from the manufacturing file. The sheet size is taken from the CAD file when importing the CAD data. The sheet size can still be modified later on via the Layout setup dialog box.

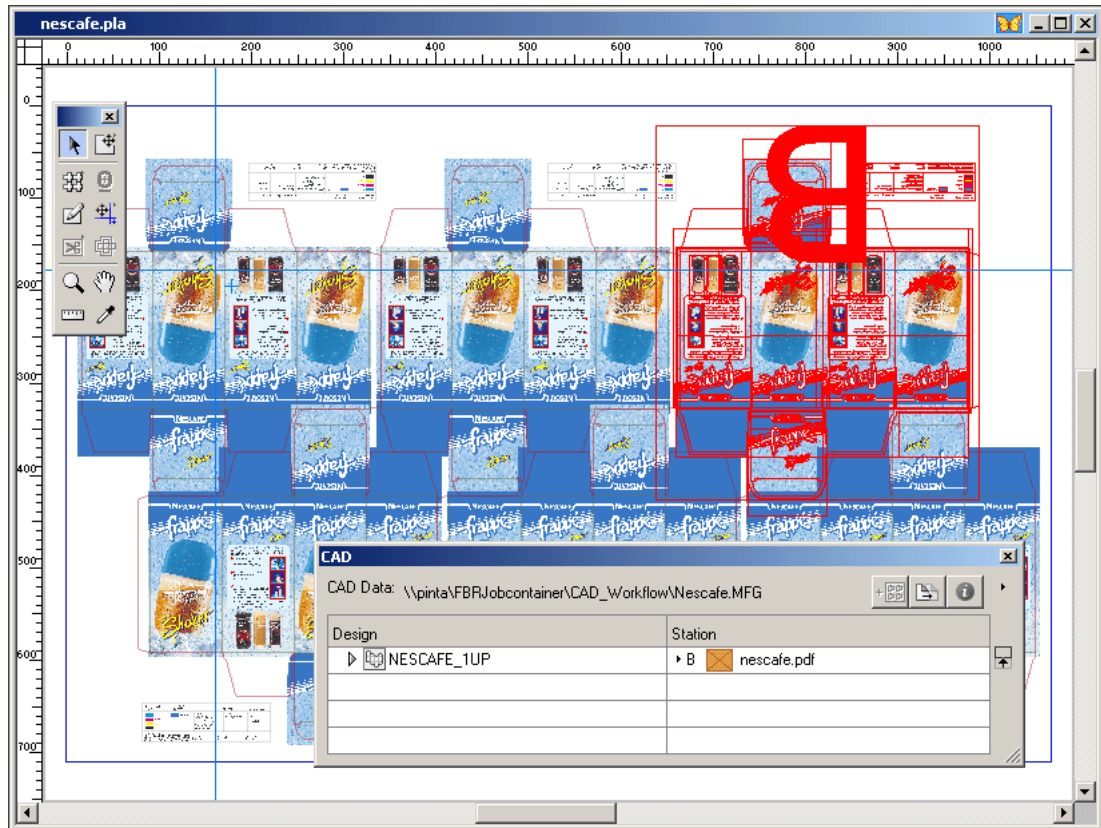
3. Click OK. The workspace indicating your plate and paper will appear, as well as the *CAD dialog box* that will allow you to load a CAD layout.



4. Click the 'Add CAD Layout' icon . The Import CAD Data dialog box will pop up.
5. Browse to your CAD file and click 'Open'.



6. The CAD layout is loaded. Select the station (or all stations) to which you want to assign the graphic(s).

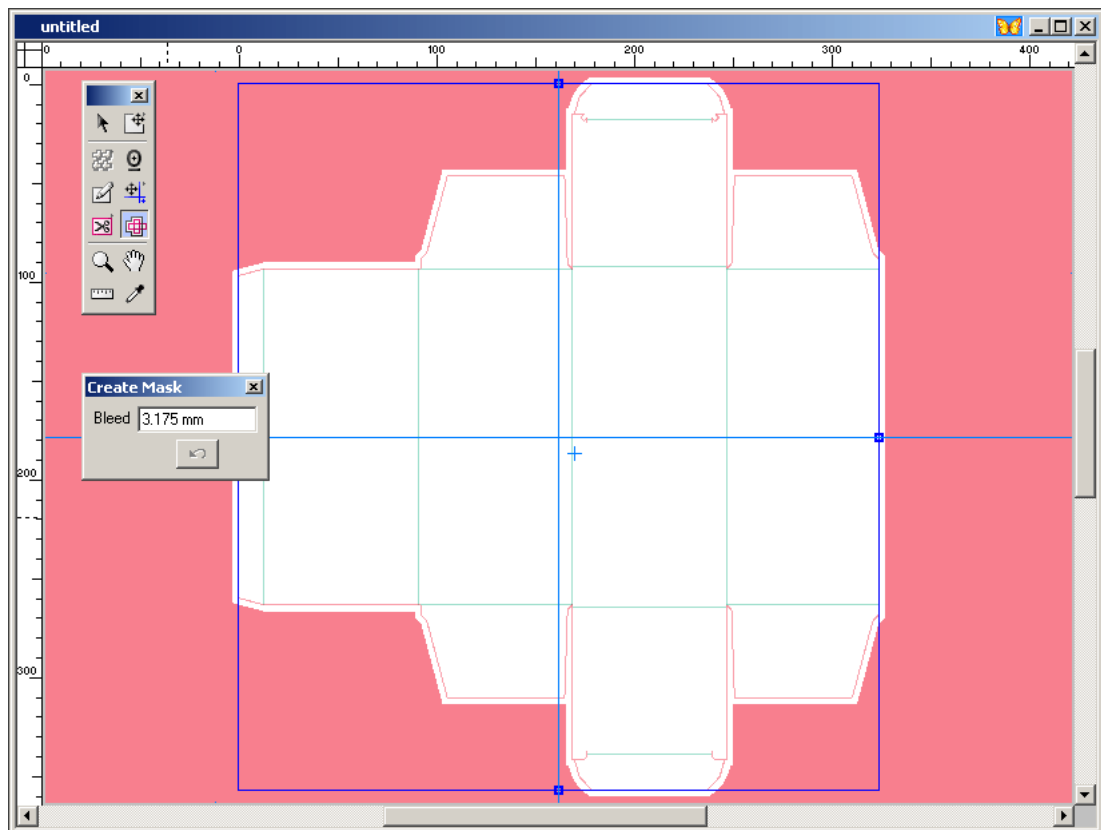


Tip:

You can switch to the Station View in case it is necessary to align the graphics to the die object.

7. You have now a set of tools at your disposal to further finetune the layout. Please refer to the chapter '*Tools*'.

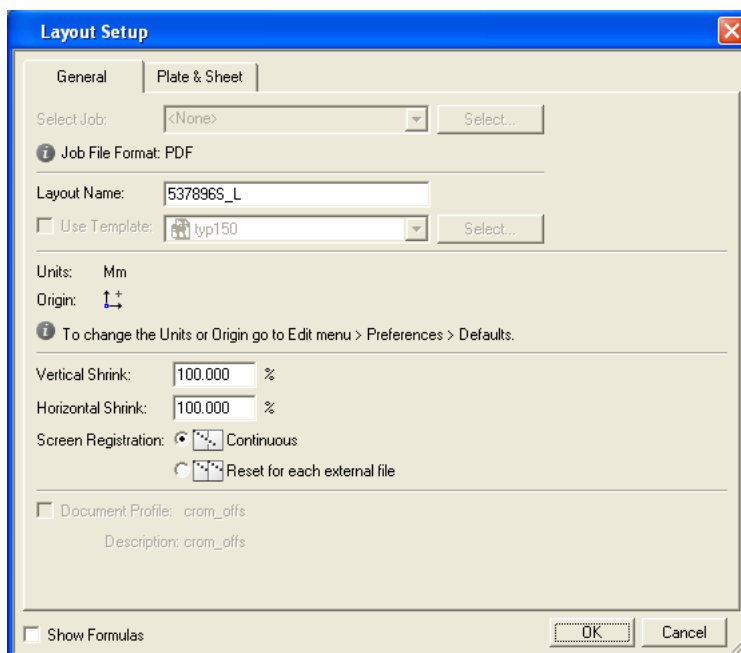
In the Station View, you can use the *Create Bleed Mask tool*  to specify a certain amount of bleed.



3.2.4. SmartLayout workflow

'Smart Layout' is an advanced layout tool that was developed primarily to facilitate the work of operators who have to execute jobs that will be cut by means of a guillotine-cutting device, and who wish to be able to integrate parameters related to order specificities and overrun. Thanks to the SmartLayout option you can optimize the distribution of different labels on your sheet automatically and it takes but a few seconds. It uses different optimizing methods depending on your workflow, it offers dynamic feedback on suggested layouts and operator (inter)actions, e.g. run length, waste, overrun, ...

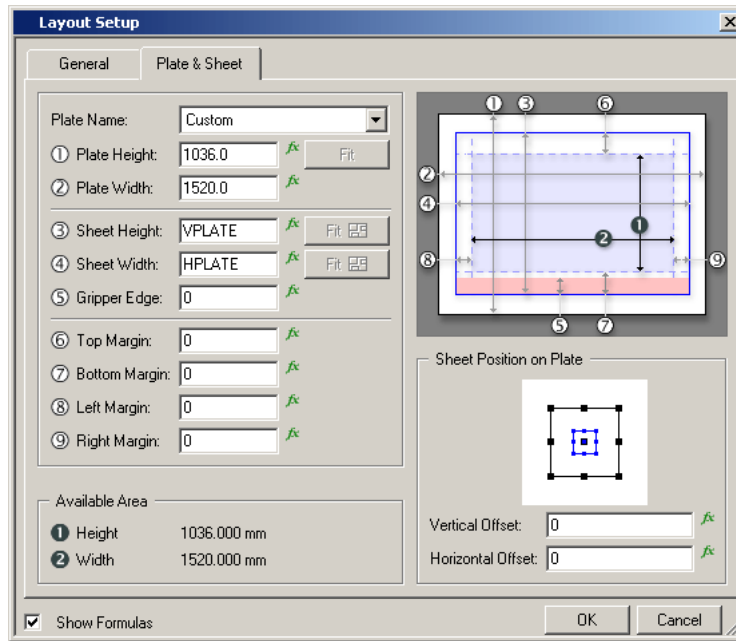
1. Go to File > New Layout > SmartLayout. The *Layout Setup dialog box* will appear:



2. The General tab of the Layout Setup dialog box offers you the possibility to determine some general settings. If your Esko Plato is connected to an Automation Engine server, you can specify in which Automation Engine **Job** the new layout will be created. It indicates the **Job File Format** in which your job will be created: PDF or GRS. It invites you to give a **name** to your layout. Note that templates are at your disposal. You can select one by toggling on '**Use Template**' and selecting one from the drop-down list.

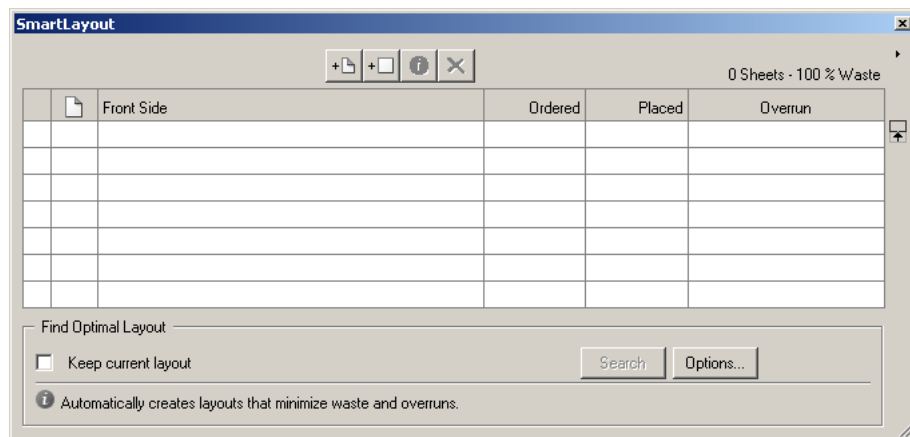
The **Shrink** values are relevant only to particular printing processes such as flexography, in which case the plate tends to expand as it is wrapped around the press cylinder. **Screen registration** is relevant only to Esko FlexRip users. When activated, the origin of the screening will be reset for each station. In doing so, it is guaranteed that exactly the same screening will be applied for each station. When deactivated, screening will be done continuously throughout the complete job.


3. The Plate & Sheet tab allows you to specify the dimensions of your plate and sheet.



To facilitate this procedure, you have a number of formulas at your disposal, which you can activate by clicking the 'fx' symbol to the right of each entry field. When you click 'fx', the [Calculator](#) pops up.

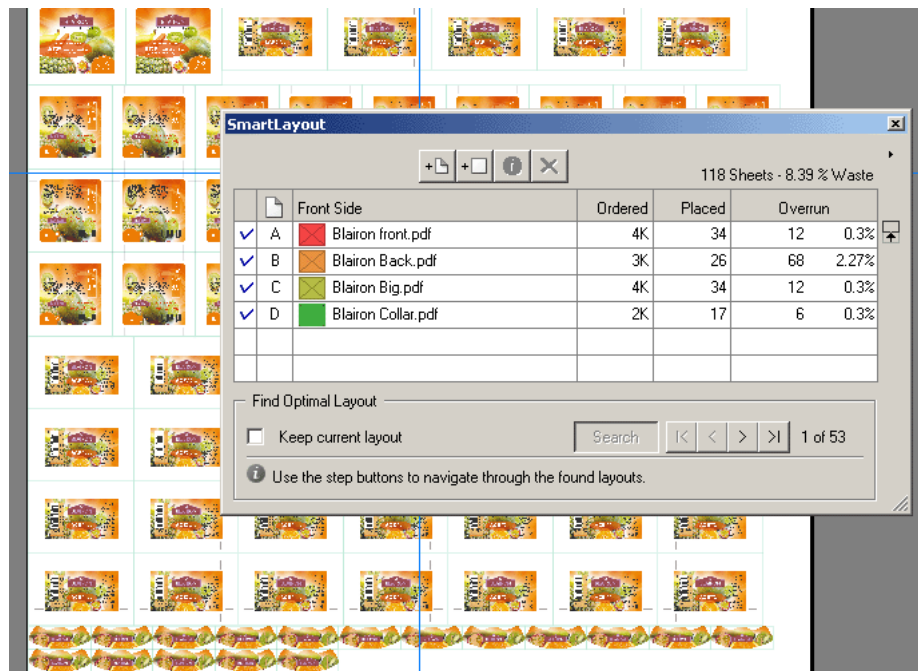
- As soon as you have entered all relevant values (or formulas), the [SmartLayout dialog box](#) appears. It serves as the central dialog box for all further actions.



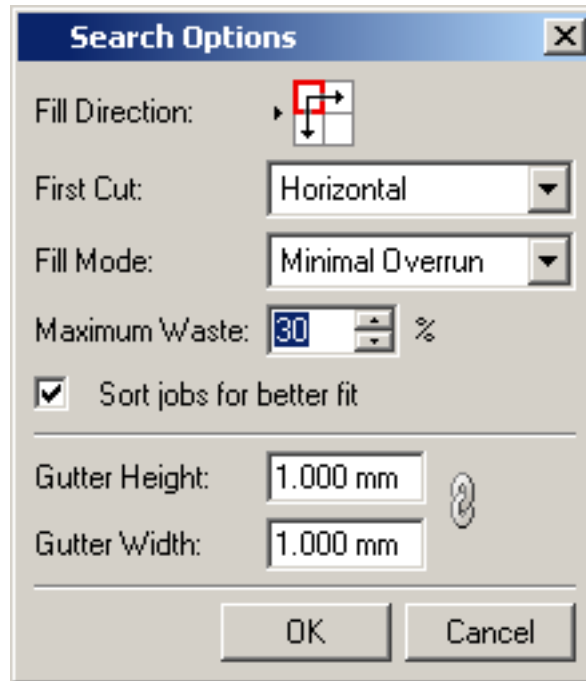
- Add stations by clicking the 'Add Station' icon  Please refer to the chapter [Adding a station](#) for more information.
- For each station, enter the ordered quantity in the 'Ordered' column. This number is expressed in thousands ('K'). 'Placed' indicates the number of times this station is placed in the current layout. This number is read-only.
- You can specify the amount of overrun for each station individually by entering a value in the Station Properties dialog box. For more information, please refer to the chapter [Station Properties](#).

- Click **Search**. The SmartLayout tool will order the selected stations on your sheet in the best possible way in terms of ordered quantity, overrun, waste... The amount of overrun is indicated in actual numbers and as a percentage of the ordered quantity. 'Search' creates a new layout, removing all stations that were placed on the layout before activating the function. A number of layouts will be suggested. The one that is visualized is regarded as the most ideal one. Of course, you can at any time opt for another layout by using the step buttons.

This is an example of a possible layout.



- Click the 'Options' button to further refine your layout.



Fill Direction allows you to determine in which directions the sheet should be filled.

First Cut determines the direction in which the first cut will be made, either horizontally or vertically.

Fill Mode enables you to determine whether you want 'Minimal Overrun', 'Minimal Waste', 'Strips Only' or 'Complete Strips'.

Enter a **Maximum Waste** percentage.

Determine **gutter** height and width.

For more in depth information, please refer to the chapter '[Search Options](#)'.

3.2.5. Manual workflow

A manual workflow does not use an imported layout. Instead, you create your stations and then place and arrange them on the sheet. Esko Plato has a robust set of tools that assist you in this process. You use the Step-and-Repeat, Transform and Align tools to build the layout yourself, or you can use SmartLayout to automatically build a layout. Once you have designed your layout, you finish the plate the same way that you do when using the CAD Layout workflow: by adding marks to the sheet and plate and printing the layout.

The basic steps to create a layout manually are:

1. Create a new layout.
2. Create the stations that will be used on the layout.
3. Place and arrange the stations on the layout.
4. Add marks to the sheet and plate.
5. Print your layout to proofs, films, or plates.

The manual workflow is a fairly unstructured workflow. You can place as many stations as you like on the layout and arrange them to fit your production needs.

3.2.6. Combining workflows

Esko Plato allows you to combine workflows, if you need to. So, you can import a CAD layout and add loose stations via '[Place Station...](#)'. There may be workflows where this is needed. But remember that the final layout still might have to be pre-trimmed to enable die cutting and folding.

3.3. Using a CAD file for plate layout (optional)

Esko Plato can open a CAD file to define the plate layout. When you open a CAD file, you are using the CAD Layout Workflow. This workflow defines the number and locations of the stations that are on the plate. It may also contain a sheet size and masks with bleeds for the stations. If these features are in the CAD file, Esko Plato will use them in your layout.

- [Supported CAD file formats](#)
- [CAD Layout or CAD 1-up?](#)
- [Importing a CAD layout](#)
- [Using Print Item names](#)
- [Flipping a CAD layout to print side](#)
- [Importing a different CAD layout](#)
- [Releasing a CAD layout](#)

3.3.1. Supported CAD file formats

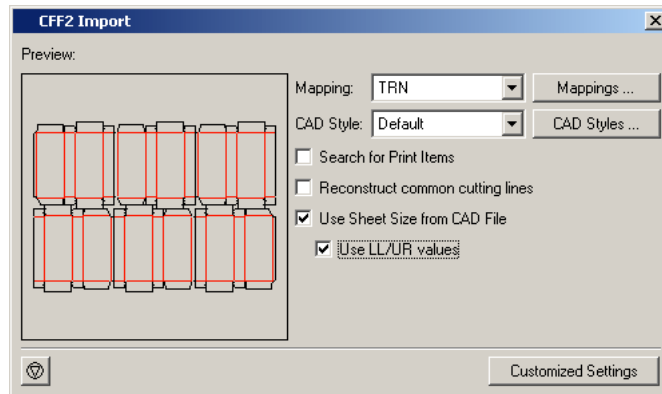
Esko Plato imports [CFF2](#), DDES, [DXF](#) and ArtiosCAD Manufacturing and Design file formats. The CFF2 and DDES CAD file formats are industry standards. Most CAD systems support at least one of these file formats.

Esko Plato knows how to import the native file formats of ArtiosCAD directly. A layout file in ArtiosCAD is called a manufacturing file; a 1-up or single file is called a design file.

Importing CFF2 files

CFF2 import supports the existing CFF2 files: a CFF2 file describing a layout places stations with a rotation combined with a mirroring (negate). The problem is that the CFF2 specification does not specify exactly in which order the rotation and negation have to be performed and results differ depending on the sequence. Some CAD vendors have chosen to rotate first and then negate (e.g. Artios), others do it the other way around (e.g. Elcede).

Click 'Setup', the CFF2 Import dialog box pops up.



Mapping allows you to map CFF2 line attributes (line type, aux line type, pointage) onto line types (ArtiosCAD style). Please refer to the chapter '[DXF Import](#)' for more information.

CAD styles: please refer to the chapter '[CAD styles](#)' for more information.

Click the **Search for Print Item** check box if you want Esko Plato to search for a graphic that matches the Print Item name in the CAD file. Esko Plato will look for a graphics file (GRS/STA/PDF/PDFSTA) that has the same name in the directory where the CAD file is located. If no matching file is found, a browser will appear and ask you to locate the missing file.

In case the supplied CFF2 file has undergone a 'common knife removal' optimization, the option '**reconstruct common cutting lines**' can be used to try to reconstruct common cut lines. This is needed if you want to use the cut lines as the basis for mask creation (Create Bleed mask in Station View and adjust masks in Sheet View). When this option is activated, cut lines from adjacent designs and from the layout level will be used to complete the cut lines from the different designs.

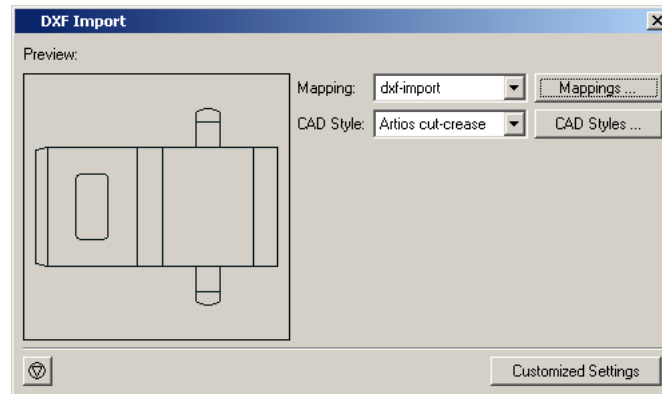
By enabling **Use Sheet Size from CAD File**, you can have the Sheet Size of the open Layout changed to the Sheet Size of the CAD file.

If the **Use LL/UR values** option is enabled, the exact LL and UR values (Lower Left and Upper Right) as defined in the CFF2 file are used to define the sheet size, regardless of the position of the CAD design itself.

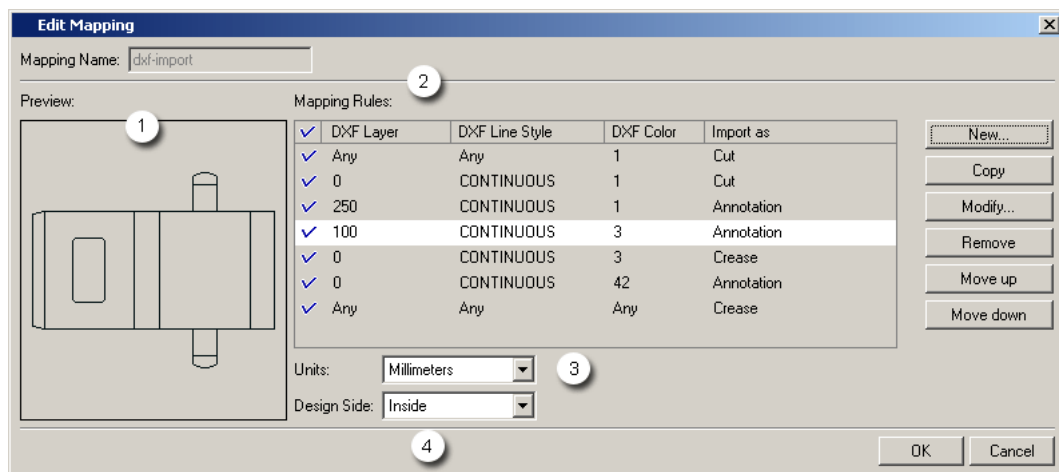
If the **Use LL/UR values** option is disabled, the bounding box of all structural lines is used. However, in that case you can include the LL/UR box in the bounding box by enabling the **LL/UR values contribute to sheet size** option in the Mappings.

DXF Import (Drawing eXchange Format)

Click 'Setup', the DXF Import dialog box pops up.



Click 'Mappings' to check or modify the selected mapping.



1. In the **preview area**, one sees the result of the current settings on the selected file. (Cut lines are visualized in red, crease in green and bleed lines in blue).
2. The **mapping rules** allow to map DXF layer names, DXF line styles and DXF color numbers onto line types (ArtiosCAD style). The mapping rules are a simplified version of the tune options from ArtiosCAD. 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.
3. **Units:** As the DXF file format does not have any indication of the units used in its coordinates, the correct unit must be defined.
4. **Design side:** Select InSide or OutSide. Esko Plato always reads CAD data in 'outside' mode. If the CAD data was designed from the inside, the CAD data will be flipped around the vertical axis.

Tip:

It is very likely that different suppliers of DXF files will use different conventions. Some might use different layers to distinguish between cut and crease lines, some might use different colors. Suppliers of DXF files should be able to indicate how the DXF file is structured. It is good practice to save mapping rules as customized setting for each supplier.

3.3.2. CAD Layout or CAD 1-up?

There are two types of CAD files: layout files and 1-up files. Layout files often contain several stations that are repeated on a sheet or board. 1-up files contain a single station without any repeat information.

ArtiosCAD has two separate formats for these two different types of files, but CFF2 and DDES do not. This difference is important: you can import layout files into a plate but you cannot import 1-up files into a plate. You can, however, import 1-up files into the Station View.

How do you know if your CAD file contains layout information or 1-up information? You can't easily determine this, but Esko Plato knows the difference. If you try to import a CFF2, DXF or DDES file that only contains a 1-up into Esko Plato, you will see an error message telling you that the file is not a layout file.

Note:

Layout CFF2 = subroutined CFF2

Layout DXF = DXF with BLOCK command

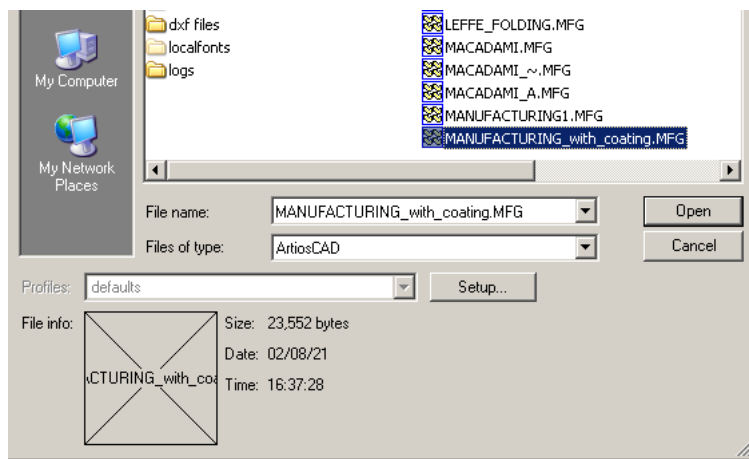
Note:

When trying to import a CFF2 or DXF file that is seen by Esko Plato as a 1-up CAD file, a message will be displayed. However, it is still possible to import the file. In some cases (for instance when common knife removal has been applied), CAD files might have lost their structure which makes it impossible for Esko Plato to extract the position of the stations. After loading these CAD files, you will have to manually align the graphics on the CAD lines.

3.3.3. Importing a CAD layout

You import a CAD layout file into a plate file:

1. On the File menu, click '[Import CAD Data](#)'. The Import CAD Data dialog box appears:



2. Select a file type.
3. Locate and select the file you want to import. A preview of the contents of the file appears in the File Info area of the Import CAD Data dialog box.
4. **Setup** opens the CAD Import dialog box that allows you to do mappings, select or create a certain *CAD style* or search for print items. Esko Plato will then search for stations that match the names of the Print Items listed in the CAD file and will automatically place them in the layout.
5. Click 'Open'. Clicking 'Cancel' will close the Import CAD Data dialog box without importing the file.

The CAD layout is imported into your plate. A lot happens when you import a CAD layout. Here is an overview:

- The cut and crease lines that are in the CAD file are imported in the colors and line widths as set in the selected *CAD style* file.
- If a sheet size is defined in the CAD layout and it does not match your current sheet size, the sheet in your plate is changed to the new size.
- If the sheet size defined in the CAD layout is larger than the plate size in your plate, the plate is enlarged so that the sheet still fits on the plate.
- If you chose to search for Print Items, Esko Plato searches for stations (or GRS files) whose name is the same as those listed in the CAD file. If a matching file is not found, a Browse dialog box will appear to ask you to locate the missing file(s).
- If you chose not to search for Print Items, your layout file now contains a set of "empty" stations or stations that have no graphics. You can select one of these stations by a simple click.

Note: When importing an MFG file, you can choose to import it as "Reference Lines" in the CAD Import setup window. By doing so, the MFG file will not be imported as CAD but only as reference lines, in a separate "CAD Reference " layer. Importing the MFG file will have no other influence on your open document.

If you need to make adjustments to the sheet or plate size, you can use the Layout Setup dialog box to override the values from the CAD layout. See [Changing the sheet or plate size](#).

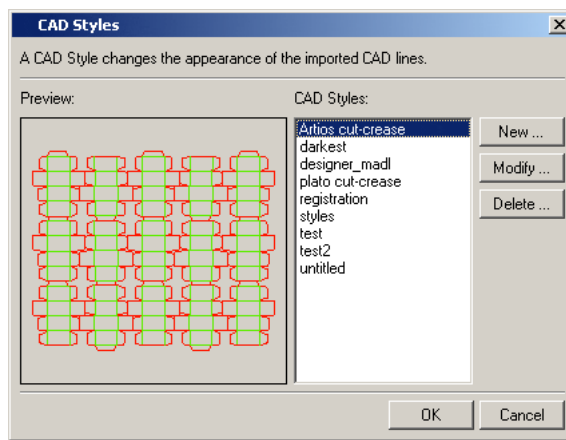
3.3.4. CAD Styles

A CAD Style changes the appearance of the imported CAD lines.

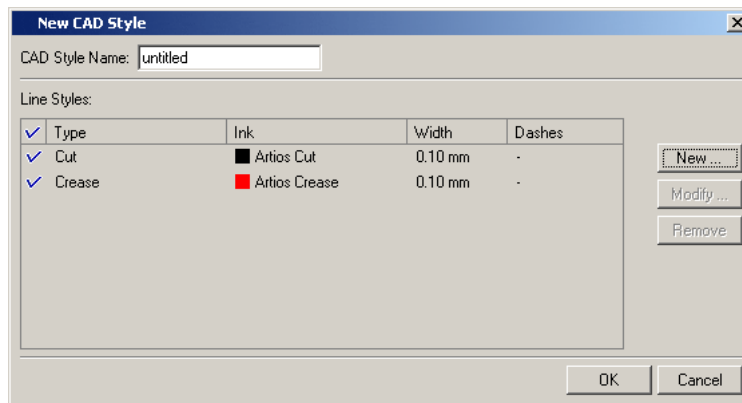
You can define new styles, and edit or delete existing ones.

To create a new CAD style:

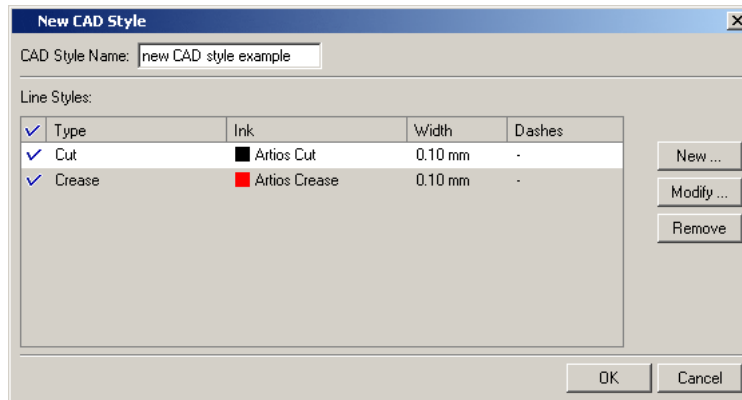
1. Click 'New...'



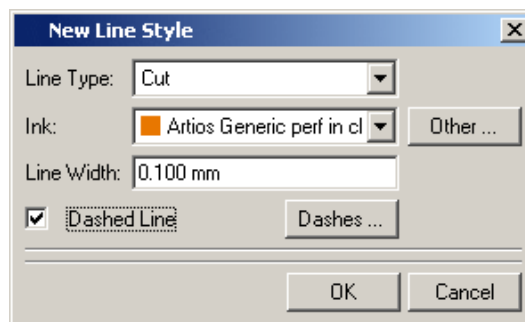
2. The 'New CAD style' dialog box pops up, a default configuration is displayed. You can edit this configuration to fit your needs.



3. Enter a name.
Click 'New...'



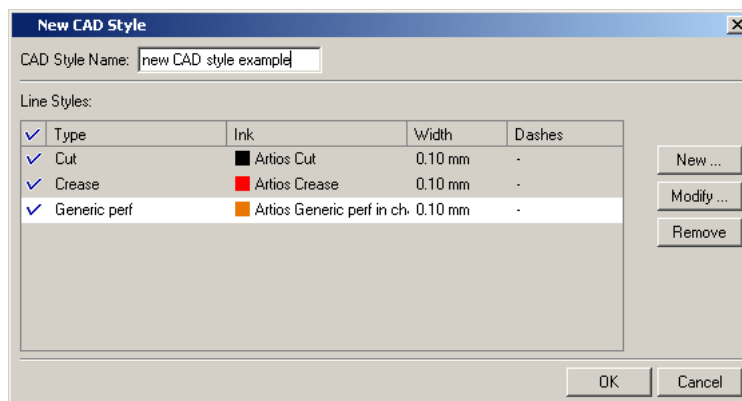
4. The 'New Line Style' dialog box pops up.



Choose a Line type from the drop-down list. Choose an ink for this line type. Define a line width. You can also decide to make it a dashed line by clicking the toggle.

Press [Enter] to add the Line Style to the CAD style.

5. The new Line Style is added. You can edit existing Line Styles by selecting them and clicking 'Modify...'.
Click 'OK' to add the new CAD style to the CAD Style dialog box.



Note:

You can use 'Other Structural' as a wildcard on the last line. This will handle all structural lines types that were not explicitly mentioned in the table of line styles.

3.3.5. Using Print Item names

Esko Plato can search for and automatically place stations based on the Print Item names contained in the CAD file. The following rules are used for searching:

1. Esko Plato looks in the current directory for PDFSTA files that have the same name as the Print Item name specified in the CAD file. The current directory is the one that contains the CAD file. If a PDFSTA file is not found, go to Step 2.
2. Esko Plato looks in the current directory for PDF/GRS files that have the same name as the Print Item name specified in the CAD file. If no PDF/GRS file is found, go to Step 3.
3. A File Browser is opened for you to locate the missing file. Choosing a file adds a new directory to the search list. Now there are 2 search directories : the original one where the CAD file is located, and a new one where you located a file with the File Browser. Go to Step 4.
4. Look in both search directories for the remaining files. If no file can be found, the File Browser appears again asking you to locate the missing file.

This browsing and searching loop continues until all the Print Items specified in the CAD file are found. If all Print Item names can be matched with either (PDF)STA or PDF/GRS files in the directory that contains the CAD file, then the File Browser will never appear and the layout will be automatically created.

These search rules will choose an (PDF)STA file before choosing a PDF/GRS file. For example, if the Print Item name is "job1" and the current directory contains an STA file named "job1.sta" and a GRS file named "job1.grs", the STA file will be assigned. A Station file will always be selected before a GRS file because they contain more information.

With these search rules, your layout may have all (PDF)STA files assigned, all PDF/GRS files assigned or may have a combination of both (PDF)STA and PDF/GRS files.

3.3.6. Flipping a CAD layout to print side

CAD files can show the cutting and creasing information in two orientations: Die Side and Print Side. Die Side displays the layout as it would look if you were looking at the cutting die itself. This is a mirror image of the printed sheet. Print Side shows the layout as it would look if you were looking at the printed sheet. The Print Side orientation is the one you want to use for Esko Plato because it is in the same orientation as your graphics.

You may have CAD files that show Die Side. If this is the case, you will need to "flip" the CAD file over so that you can work with the Print Side.

To flip a CAD layout to print side:

1. Make sure that you have a CAD layout file in your plate. If you do not have a CAD file in the plate, the command will be disabled.

2. On the Tools menu, click 'Flip CAD to Print Side'. The CAD file is flipped over to print side.

3.3.7. Importing a different CAD layout

You can only have one CAD layout on a plate at a time. If you have a CAD layout in your plate, the Import CAD Data command is disabled. If you want to replace it with different CAD layout, you need to delete the current one in the plate.

3.3.8. Releasing a CAD layout

In a CAD Layout Workflow, the stations that are assigned to the imported CAD layout are tied together. The stations cannot be moved away from their positions in the CAD layout or one another. However, there may be times when you need to make adjustments to the locations of the stations with respect to one another. The Release CAD Layout command provides this feature. You can remove the CAD layout and "free" the stations by selecting 'Release CAD Layout' on the Tools menu.

This command will delete the imported CAD layout but keep the stations in their current positions. In reality, you are changing your workflow from the CAD Layout Workflow to the Manual Layout Workflow.

It is a good idea to use this command only after you have most of your layout done. For example, if you don't have the right position for the layout and need to move it after this command, you'll have to select all the stations to keep them together.

Release CAD Layout tips

- You should use this command only if you really need to. After deleting the CAD layout, the stations can be freely moved with respect to one another. So, it's possible for mistakes to occur that make it impossible to die cut the layout later.
- You should have the basic features of your layout set correctly prior to using this command. At the very least, make sure that the plate size, sheet size, sheet offset and layout position are correct.
- If you release the CAD layout and make adjustments to the position of the stations, you should use the Adjust Masks tool after you have made these adjustments. Otherwise, the masks may no longer be correct if the stations have moved.

3.4. Using Templates

You use templates to automate the layout setup, CAD layout and marks features that you often use. If you regularly create layouts for a particular press or customer, you should use a template.

- [What is a template?](#)
- [Creating a template](#)
- [Saving a template](#)
- [Using a template](#)

3.4.1. What is a template?

A template is nothing more than a layout file that has been saved as a template. When you save a layout file as a template, the following features are saved:

- sheet size, plate size and sheet offset
- marks attached to the plate and sheet
- CAD layout, if one is present in the plate file
- Stations that are currently placed on the layout

You assign or place new stations onto the template and then save it to create a finished layout.

The workflow for using templates is:

1. Create a layout file. Add plate and sheet marks; add a CAD layout if desired.
2. Save the file as a template.
3. Select the template when starting a new layout.
4. Add stations to complete the layout.
5. Save as a Esko Plato layout file.

3.4.2. Creating a template

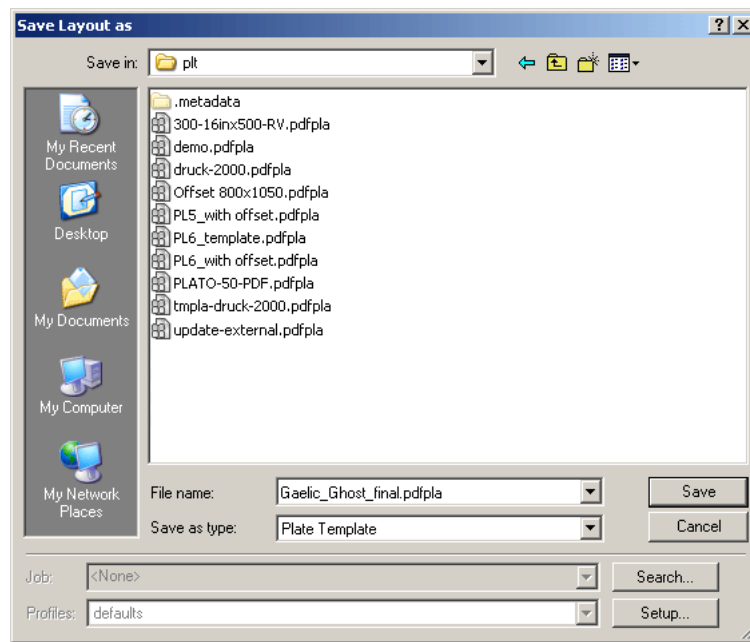
To create a template, you need to create a layout file that contains all the layout information for a particular press. To automate as much as possible, your template should include marks too. Then all the layouts created will be the same.

3.4.3. Saving a template

When you save a layout file, you can choose to save it as a template. This is how templates get created.

To save a template:

1. On the File menu, click 'Save Layout As'.



2. Select 'Plate Template' from the 'Save as type' list.
3. Click 'Save'.

The extension of the saved template files is either .plt or .pdfpla.



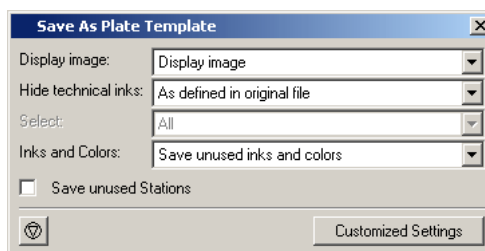
Caution:

Templates are always saved in Esko Plato's template directory, regardless of which directory you choose. If you need to delete a template, you will need to remove it manually from \Esko-Graphics\bg_data_custom_v010\dat\plt OR \Esko\bg_data_custom_v010\dat\plt for new installations.

Click 'Setup...!' to open the set plate template options:

- [Saving a layout as a Plate template](#)
- [Saving a layout as a PDFPLA file](#)
- [Saving a layout as a PLA file](#)

Saving a layout as a Plate template



Display ImageThis option allows you to set the type of display image that will be included in the file. Choices are: No Display , Thumbnail (200 x 200 pixels) or Display image (preview of 72 dpi), which creates a low resolution preview that can be used when the file is placed as an external file.

SelectYou can export your entire job or a part of it.

- All - saves the entire job
- Red Selection - saves objects selected in red only
- Current Layer - saves the currently active layer only
- Current Page - saves the current page

Inks and Colors

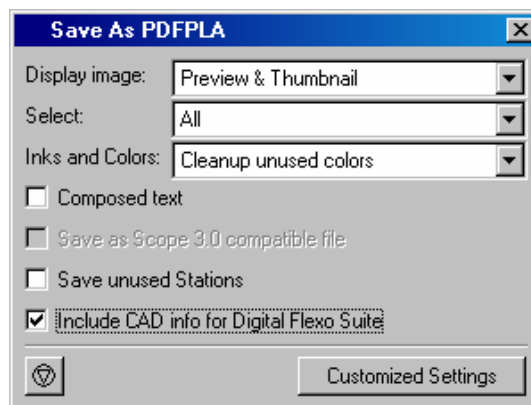
- 'Clean up unused colors' automatically removes all unused inks.
- 'Clean up unused inks and colors' automatically removes all colors and inks that are not used in the job.
- 'Save unused inks and colors' saves unused inks and colors.

Composed textComposed text is non-editable text. It can be converted to editable text by clicking it when you are in text edit mode.

Save unused StationsCheck the box if you want to save stations that have not been used in your design. By default, unused stations will not be saved.

Saving a layout as a PDFPLA file

When working in PDF mode, you can save your layout as a PDFPLA file.



Display ImageThis option allows you to set the type of display image that will be included in the file. Choices are: No Display , Thumbnail (200 x 200 pixels) or Preview & Thumbnail (72 dpi), which creates a low resolution preview that can be used when the file is placed as an external file.

SelectYou can export your entire job or a part of it.

- All - saves the entire job
- Red Selection - saves objects selected in red only
- Current Layer - saves the currently active layer only

- Current Page - saves the current page

Inks and Colors

- 'Clean up unused colors' automatically removes all unused inks.
- 'Clean up unused inks and colors' automatically removes all colors and inks that are not used in the job.
- 'Save unused inks and colors' saves unused inks and colors.

Composed text Composed text is non-editable text. It can be converted to editable text by clicking it when you are in text edit mode.

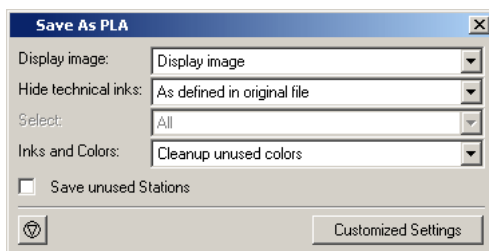
Export to Scope 3 Compatible file Check this box if you want your Normalized PDF file to be Scope 3 compatible.

Save unused Stations Check the box if you want to save stations that have not been used in your design. By default, unused stations will not be saved.

Include CAD info for Digital Flexo Suite: This allows to include staggered cutting & CAD paths in the PDFFLA file. All stroked technical lines can then be forwarded into the LEN file, so that Digital Flexo Suite can pick these up. This can be used for example to plot the die lines on a carrier sheet out of Digital Flexo Suite. See also [Staggered Cutting](#) on page 165

Saving a layout as a PLA file

When working in GRS mode, you can save your layout as a PLA file.



Display Image This option allows you to set the type of display image that will be included in the file. Choices are: No Display , Thumbnail (200 x 200 pixels) or Display image (preview of 72 dpi), which creates a low resolution preview that can be used when the file is placed as an external file.

Hide Technical Inks When selected, technical inks will be stored in the file but will not be visible as exposable inks for the Esko FlexRip.

Select You can export your entire job or a part of it.

- All - saves the entire job
- Red Selection - saves objects selected in red only
- Current Layer - saves the currently active layer only
- Current Page - saves the current page

Inks and Colors

- 'Clean up unused colors' automatically removes all unused inks.

- 'Clean up unused inks and colors' automatically removes all colors and inks that are not used in the job.
- 'Save unused inks and colors' saves unused inks and colors.

Save unused Stations Check the box if you want to save stations that have not been used in your design. By default, unused stations will not be saved.

3.4.4. Using a template

Once a template is saved, it will become available in the Layout Setup dialog box when you create a new layout. To use a template, click the 'Use Template' checkbox and select a template from the list. If QuickStep templates are present on your system, they will also be included in the list.

4. Working with Stations

Arranging stations on the plate is the primary purpose of Esko Plato. This section will provide you with the basics of working with station files. You will learn a.o. how to create stations, how to add them to the plate, how to adjust areas where two stations overlap and how to number stations.


- [Creating, editing and saving a station](#)
- [Using a CAD file to create a station](#)
- [Adding stations to the plate](#)
- [Working with empty stations](#)
- [Adjusting overlapping stations](#)
- [Nesting Stations](#)
- [Numbering stations](#)

4.1. Creating, editing and saving a station

The Station View is a view dedicated to editing a station that you want to use on your plate. You can easily switch between the Sheet View and the Station View by clicking the view icons in the [View Selector](#).


The definition of a station is embedded inside the Esko Plato file. So it is not needed to save your stations. There is however a possibility to save a station file. This is only necessary in case you want to use this station in other Esko Plato files. Important to note is that the only external references that Esko Plato uses are the references to the graphics files. Even if you load a STA file in the Sheet View, the contents of the STA file (i.e. the definition) will be embedded inside the Esko Plato file. In other words, it will keep the link to the graphics file but there is no link to the STA file. For a clear definition of a station, please refer to the chapter ['What is a station?'](#)


A station file is always saved with a STA - or PDFSTA (in the case of Normalized PDF graphics) extension. So, saving a file with the name "3Up" will save a file on your disk with the name

"3Up.sta". Station files always have the station file icon 

- [Creating a new station](#)
- [Creating a new station in a CAD workflow](#)
- [Editing an existing station](#)
- [Using Select Die Shape](#)
- [Saving a station](#)

4.1.1. Creating a new station

You can create new stations by clicking the 'Add station' button  in the Grids, Station or SmartLayout dialog boxes. In all cases a file browser will pop up to select the required graphics. A new station will be created using the (external reference to the) graphics and a die shape on the trim box of the graphics. The first 2 approaches will place the station as a 1 by 1 grid block

on the plate. The 'Add empty station' button  can be used to create a new empty station with a size that can be specified.

Tip:

You can take any entry from the Station dialog box and drag it onto the document window. This will place the station as a 1 by 1 grid block (except when working in a CAD layout). Afterwards, the station can be altered either from the Grids or Station dialog box (setting the die shape and bleed mask) or from the Station View. Once a station is on the plate, you can use the controls in the Grid dialog box, the Transform tool and the Align tool to define your layout.

Note:

Importing a CAD layout will create new (empty) stations for each design that is found in the CAD file. For more information, please refer to the chapter '[Creating a new station in a CAD workflow](#)'.

In the Manual Workflow, you do not start with a CAD layout file. You need to create a new station file and place it on the plate to start building the layout yourself.

4.1.2. Creating a new station in a CAD workflow


In the CAD Workflow, you start by defining a plate and importing a CAD file to create a layout. When you import a CAD file, "empty" stations are created. You then need to create station files to assign to these empty stations. To create a station file for a CAD layout:

1. Import a CAD layout file into your plate.
2. Select an "empty" station that you want to assign a station file to.
3. On the View Selector, click 'Station View'.

The Station View will be activated and the 1-up CAD design for that station is loaded.

You can now complete the station with a [graphic](#), a mask, station marks and [station numbers](#). When you return to the Sheet View, the new station definition is assigned to the CAD layout everywhere this same 1-up CAD design is used.

Tip:

Select Stations in Filled Mode  as it makes them easier to select.

4.1.3. Editing an existing station

Regardless of which workflow you are using, you can always edit an existing station. To edit an existing station on the plate:

1. Select the station that you want to edit (either on the plate or in the Station dialog box).
2. Click the Station View icon on the View Selector toolbar.
3. Edit the station in the Station View mode and switch back to Sheet View mode.

When you return to Sheet View mode, the changes to the station are updated in the plate.

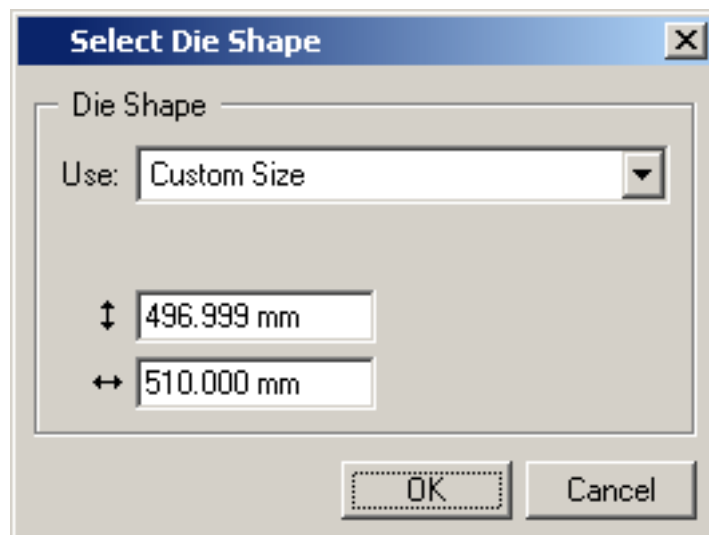
Other ways to edit existing stations:

- Use the 'die shape' and 'bleed' columns in the Grids or Stations dialog box.
- Use the [Station Properties dialog box](#).

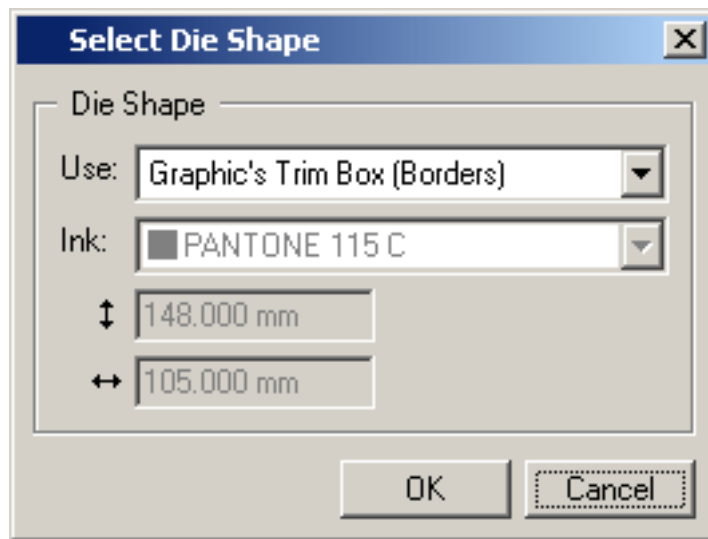
4.1.4. Using Select Die Shape

Use '[Select Die Shape](#)' on the Station menu to modify the default die using several different methods. You can modify the default die on the basis of the trim size of the graphic or you can use the graphics' contours as the default die. 'Select Die Shape' is only available if a graphic has been imported into the station.

You can create a rectangular die shape by using the option **Custom Size**. Simply specify the required height and width of the die shape.



You can fit the die to the **trim size** of an imported graphic. Esko applications, like PackEdge, call the trim size the Borders. Desktop applications often call the trim size the Page Size or the Document Size. If your graphic has the correct trim size to use for the die, the Select Die Shape dialog box can create the station's die from the trim box of the graphic.

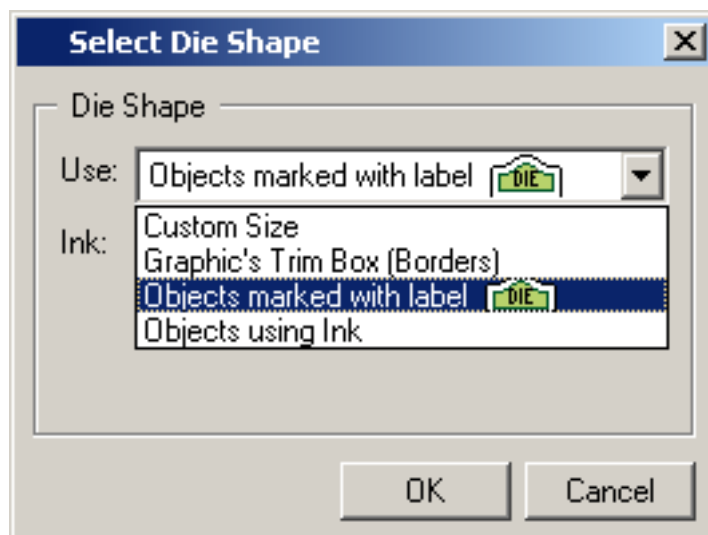


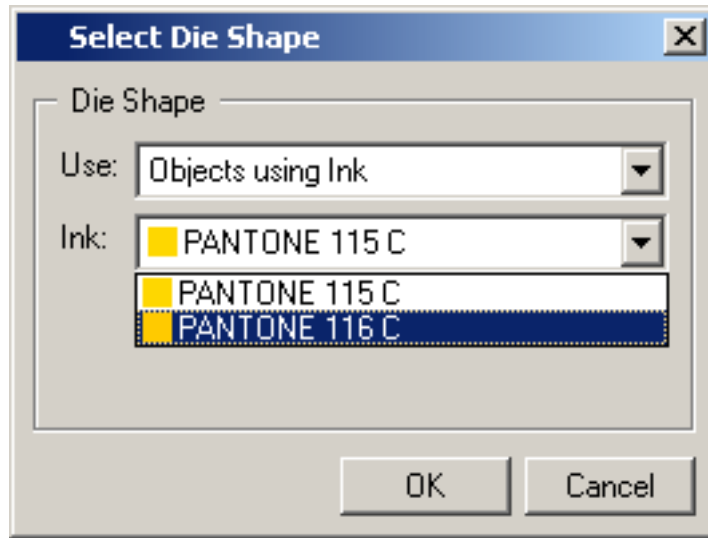
To create a die from the trim size of an imported graphic:

1. On the Station menu, click 'Select Die Shape'.
2. Select the **Graphic's Trim Box (Borders)** from the list.
3. Click 'OK'.

The station's die is modified to match the trim size of the graphic.

You can also choose to use specific contours in the graphic as the die. You can use the other options in the list to choose the contours that you want to use. Use **Objects marked with label Die** to use the contours that have the Esko PackEdge annotation "Die". Use **Objects using ink** to use contours that use a specific ink in the graphic. These options will copy the stroked contours from the graphic into the station.



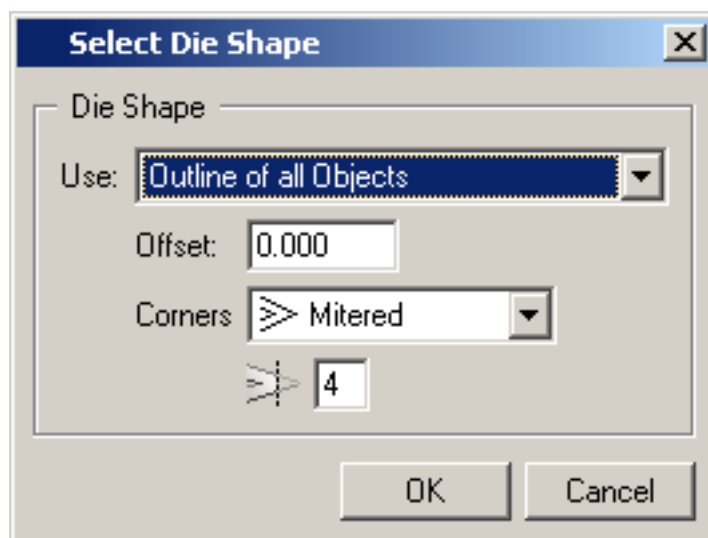


To create a die from contours marked with the Esko PackEdge "Die" annotation in an imported graphic:

1. On the Station menu, click 'Select Die Shape'.
2. Select the objects using 'Objects marked with annotation Die'.
3. Click 'OK'.

If you are using a CAD design in your station, the die shape options are not available. You cannot change the die size of a station with a CAD design. The station's size is always set to the bounding box of the die in this workflow.

Select **Outline of all objects** to create a die shape around objects that have no die shape in the original file. The new die shape will be created around the union of all objects in that particular file. You can specify a particular offset which will make your die shape a little bigger and you can indicate the shape of the corners.

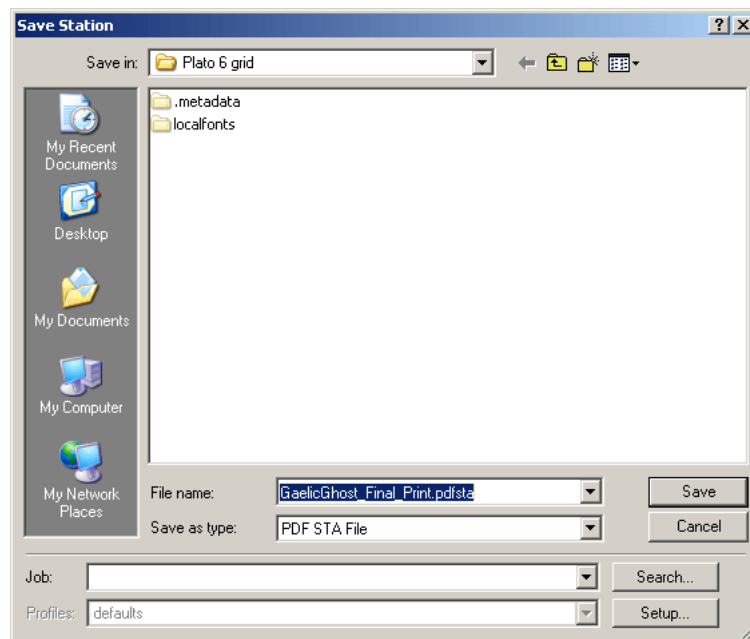


4.1.5. Saving a station

To save a station, click 'Save Station' on the Station menu. Note that saving a station is only needed if the station needs to be used in another Esko Plato file. *Station definitions* are embedded inside the files.

To save a station file:

1. On the Station menu, click 'Save Station'. The Save dialog box appears:



2. Type a name for your station file.
3. Locate the directory where you would like to save the file.
4. Click 'Save' to save the contents of your file. Clicking 'Cancel' will close the Save dialog box without saving your file.

4.2. Using a CAD file to create a station

A CAD file can be used to create the die shape for your station. There are different ways to import CAD designs (1-Up). When importing a CAD layout in the Sheet View, empty stations are created which will have the 1-up CAD design as die shape. Also, when creating a new station when clicking the Add Station button in the Grids or Stations dialog box and selecting a CAD design, an empty station will be created which will have the CAD design as die shape. The design can also be imported from a CAD file from within the Station View.


- *CAD designs automatically from Esko Plato*

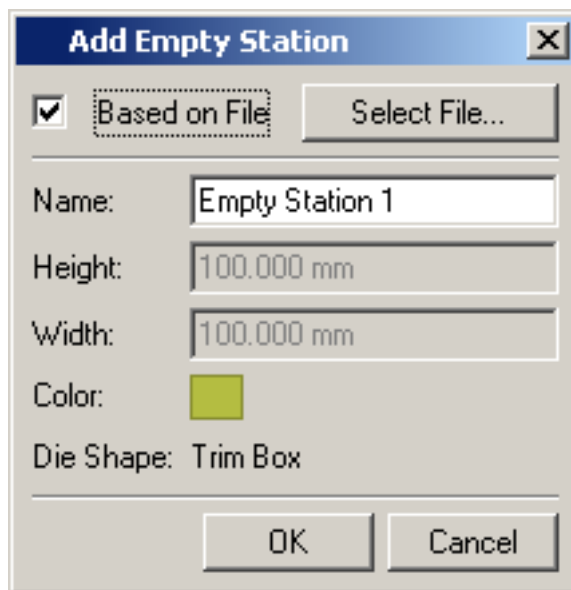
- [Importing a CAD design](#)
- [Importing a different CAD design](#)

4.2.1. CAD designs automatically from Esko Plato

When you import a CAD layout into a plate, "empty" stations are created for each 1-Up design in the layout. If you select one of the "empty" stations and switch to the Station View, the station object is opened and the CAD design for that station is displayed. You then complete the station by adding a graphic, a mask and any marks that are required.

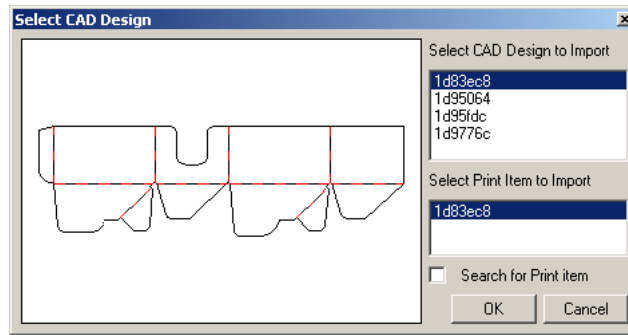
4.2.2. Importing a CAD design

Another way to create a station from a CAD layout is to import a 1-up design directly into the Station dialog box. Click the **Add Empty Station** button  in the Grids or Stations dialog box. Check 'Based on file' and select the required CAD file using the 'Select File' button. This creates an empty station with the CAD file as die shape.



The next step is to select graphics for this station. The reverse sequence is also possible: create a station and afterwards, switch to the Station View and use [Import CAD Data](#) to create the die shape.

1. On the File menu, click '**Import CAD Data...**'. The Import CAD Data dialog box appears.
2. Select a file type.
3. Locate and select the file you want to import.
4. If the selected file turns out to be a layout file (MFG file), an extra dialog will pop up to enable you to select the CAD design from the layout.



This dialog box lists all the 1-up designs that are contained in the CAD layout you selected. The names of the designs are listed on the right. You can choose the design that you want by selecting its name from the list. The preview window on the left previews each of the designs as you select them.

5. Click the **Search for Print Item** checkbox if you want Esko Plato to search for a graphic that matches the print item name in the CAD file. Esko Plato will look for a graphics file (GRS/STA/PDF/PDFSTA) that has the same name in the directory where the CAD file is located. If no matching file is found, a browser will appear to ask you to locate the missing file.
6. Select the design you want and click 'OK'.

The design you selected is imported into your station.

If any other die is currently in the station, even if it is a manual die, it will be replaced by the imported CAD design die. The station's size is set automatically to the same size as the new die.

4.2.3. Importing a different CAD design


You can only have one CAD design in a station at a time. If you want to replace a CAD design with a different one, click **Station > Import CAD Design...** The existing CAD design will be replaced by the new one. If you import a new CAD design and the existing one contained a bleed mask, the mask will only be updated when switching to the Plate View.

4.3. Adding stations to the plate

Above you learned how to create new stations. Esko Plato automatically assigns or places the new stations on the current plate. But there are times that you may want to add an existing station to a new plate. In this workflow, you need to assign or place the station yourself. You may also want to replace a station that is currently on your plate with a different station. This section tells you how to use these important functions.

- [Adding a station](#)
- [Assigning a station](#)
- [Placing a station](#)
- [Replacing a station](#)

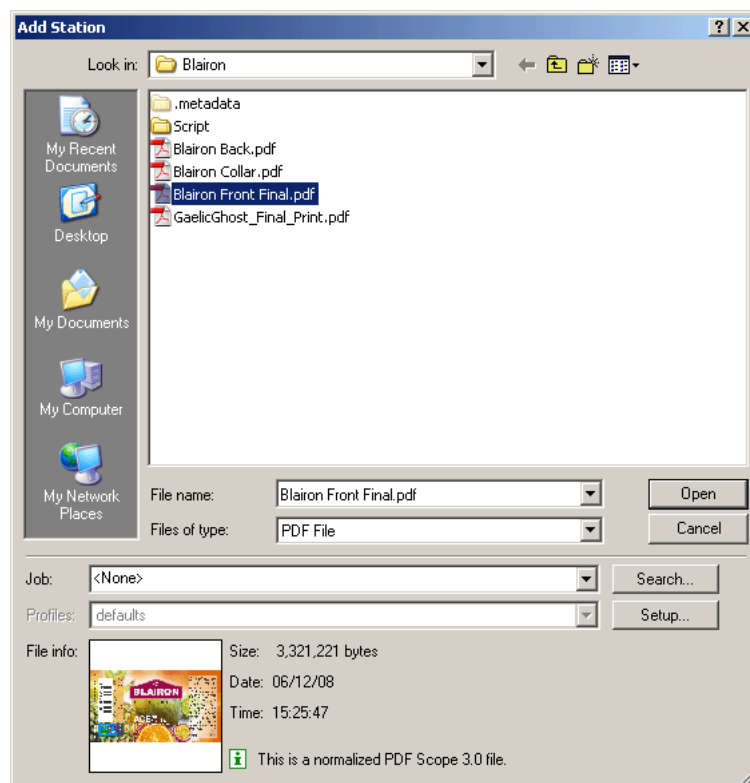
4.3.1. Adding a Station

Click the Add Station icon +  The Add Station dialog box opens.


Tip:

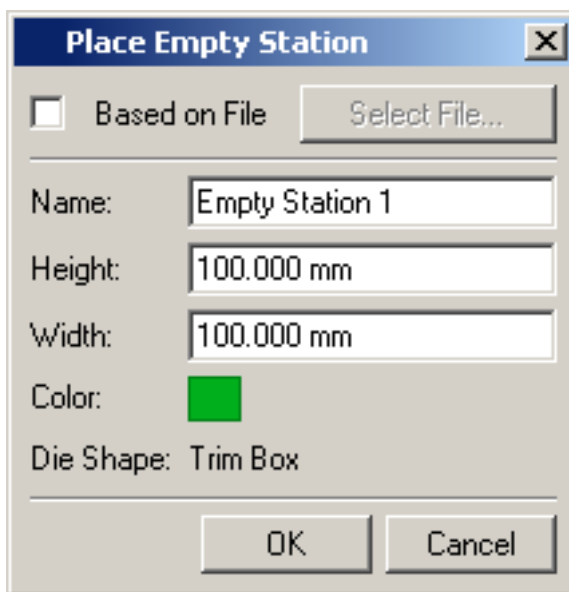
You can select multiple files.

Click 'Open' to add your station to the dialog box. Depending on the workflow chosen, this will be the SmartLayout, Grids or Station dialog box.



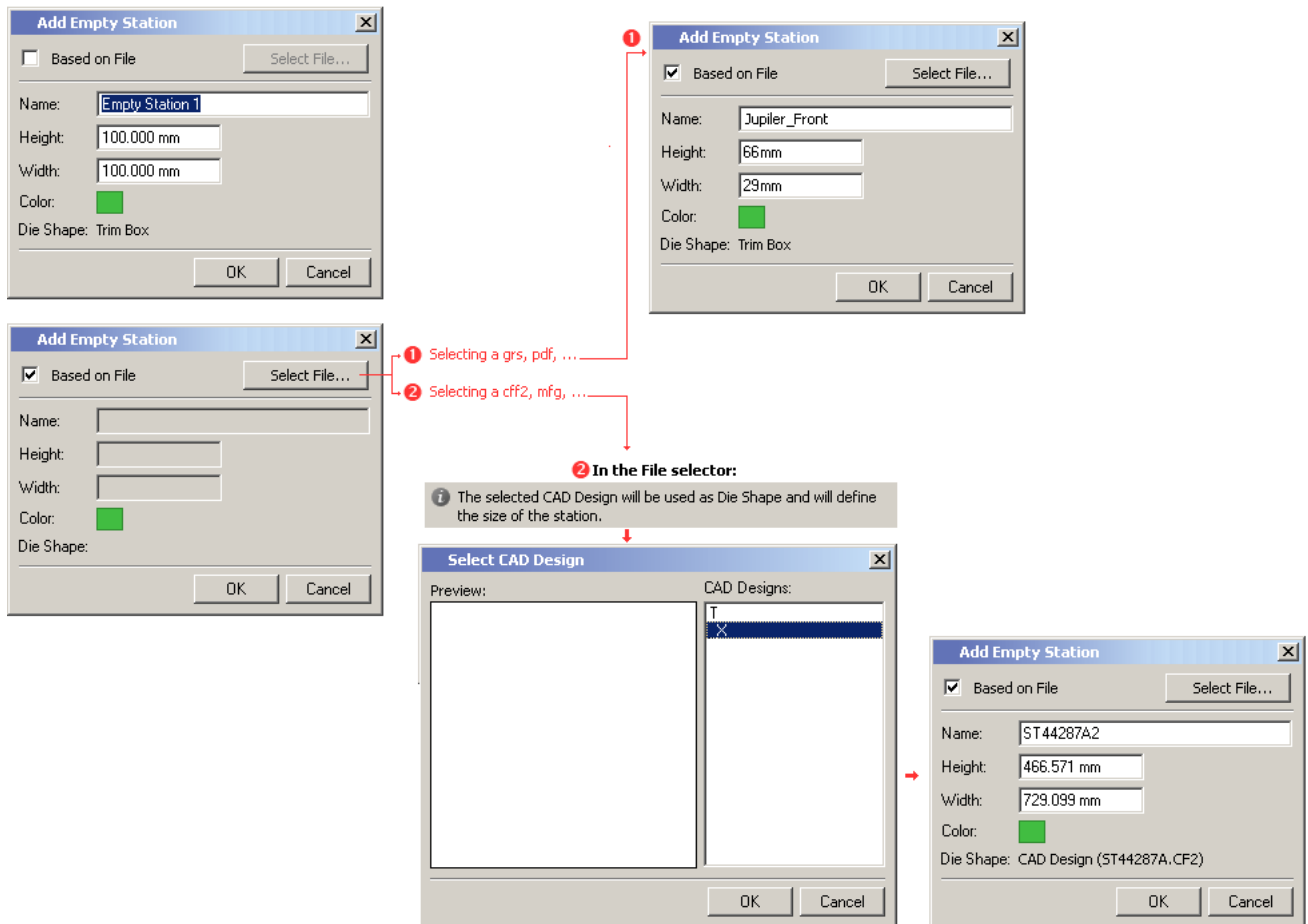
When the station is placed, you can right-click the station or select 'Station Properties...!' from the top-right pop-up menu in the SmartLayout, Grids or Station dialog box to view and adjust the [station properties](#).

Click the Add Empty Station icon +  to add an empty station. An empty station is a dummy station that you can replace with an actual station later on. You would do so when you are designing a layout before having received the actual graphics. However, in that case, you would need the exact measurements of the future graphics, as Esko Plato will ask for them in the Place Empty Station dialog box:



Use the 'based on File' option together with the the 'Select File' button if you want to create an empty station based on an existing file. This offers 2 possibilities:

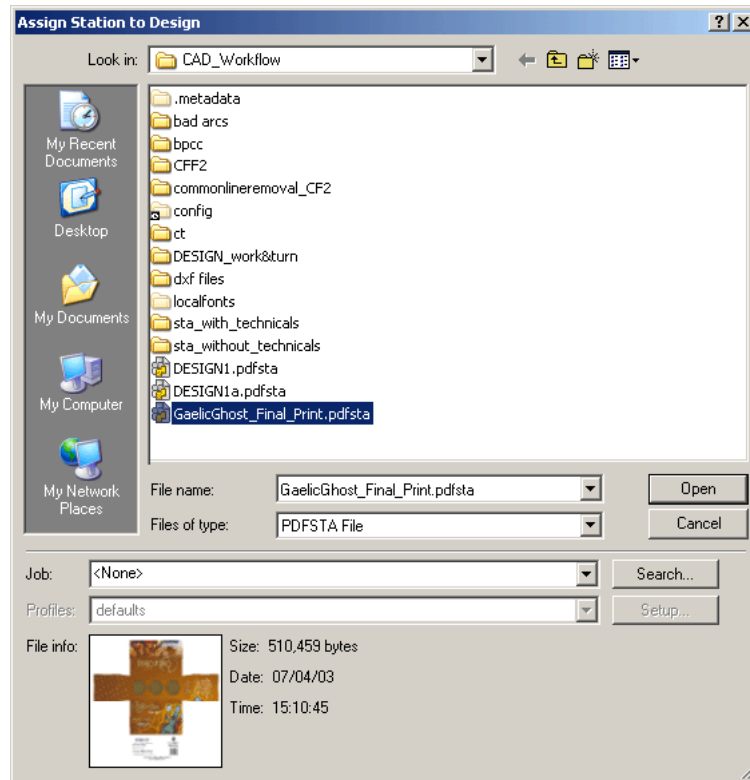
1. select an existing graphics file (grs or normalized PDF). In this case, the size of the selected graphics will be used as the size of the empty station.
2. Select a CAD file. In this case, the CAD data will be used to define the die shape of the empty station and the size of the station will match the bounding box of the CAD data.



4.3.2. Assigning a station

Most of the time, you will be creating new plates and stations starting from a layout. But if you have an existing station that you want to use with a new CAD layout, you will need to assign the station yourself. When assigning a station to a layout, the station is analyzed for a die object in a CAD structure (the CAD file loaded in a STA or GRS file). A dialog box is displayed indicating whether the CAD files match. If you have a CAD layout file in your plate and already have created a station file, you assign the station to the layout. To assign a station:

1. Import a CAD layout file into your plate.
2. Select an empty station to assign the existing station file to.
3. On the Station menu, click 'Assign Station to Design'. The Assign Station dialog box appears:



4. Select the type of file that you would like to assign.
5. Select the file that you want to assign.
6. Click 'Open'.

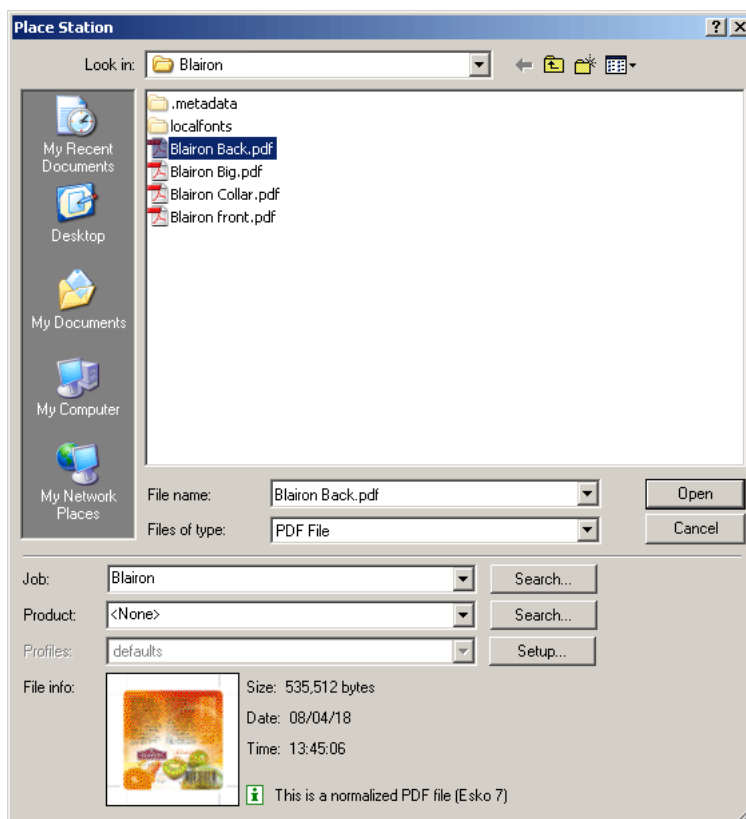
The Assign Station dialog box closes. The station file you selected is assigned to all locations in the CAD layout where the same 1-up CAD design is used. If you want some locations to use a different station file, use the "Replace by..." command. See [Replacing a station](#).

4.3.3. Placing a station

The Place Station command is used to add an existing station to a plate in the Manual Layout Workflow. This command will place an existing station in the lower left corner of the sheet. This is the default situation. The orientation can be set in Edit menu > Preferences > Defaults. Once a station has been placed in the layout, you can use the Transform and Align tools to create your own layout.

To place a station:

1. Create a new layout.
2. On the Station menu, click 'Place Station'. The Place Station dialog box appears:



3. Select the type of file that you would like to assign from the 'Files of type' list.

Tip:
 It is possible to make a page selection. The default is to select page 1 of a PDF file (as well for the front as for the back side). When creating the back side, Plato will automatically select the next page of the PDF file.

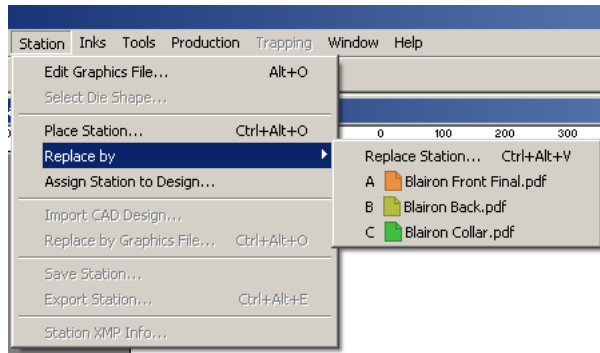
4. Select the station file that you want to place.
5. Click 'Open'.

4.3.4. Replacing a station

You may want to replace certain stations on your plate with a different station. For example, you might want to replace stations in a plate layout that you did earlier with a new version. Or you may want to replace specific stations to create a combination layout .

To replace a station:

1. Select the station(s) to be replaced.
2. On the Station menu, click 'Replace by...'. Click '*Replace Station*' to browse to an already existing station or select a station directly from the drop-down list.



3. Select the type of file that you would like to assign from the 'Files of type' list.
4. Select the new station file to use.
5. Click 'Open'.

The stations that you selected are replaced with the new station. Any stations that were not selected for the Replace Station command remain the same.

4.4. Working with empty stations

Working with empty stations allows you to create layouts even when the graphics aren't available yet. Only the size needs to be known.

When a layout has been generated, empty stations can be replaced by the actual stations, by using the 'Assign Station' or 'Replace Station' functions. While doing so, a check will be performed based either on the CAD design (in case of CAD based layout) or on the size (of the empty station against the chosen graphics file). Another way to replace empty stations is using the Station properties dialog box. Simply browse to the real graphics. Note that this is the equivalent of replacing graphics in the Station View.

The *station properties dialog box* allows you to assign dummy colors to both the actual files and the empty stations. For 'real' stations this color is only used in the *'dummy colors' mode*. For empty stations, a designer ink, representing the chosen color is added.

When replacing a station, the inks used by the old station are automatically cleaned up (if no longer used). As such, the dummy inks created when assigning colors to empty stations will automatically be cleaned up when the empty stations get replaced by the actual files.

4.5. Adjusting overlapping stations

In production, your plate layouts may have stations that partially overlap one another. For example, if your stations include a bleed and they are placed on the layout so that they have a common cut, the bleeds between the stations will overlap. The Adjust Masks tool is used to adjust overlapping stations. The command searches the plate for overlapping stations and presents each overlap to you as an overlap record. You decide how the overlap areas should be adjusted. Decide once and Esko Plato automatically applies your decision to all areas where the same overlap occurs.

- [Using Adjust Masks](#)
- [Splitting an Overlap Record](#)
- [Resetting masks](#)
- [Checking the status of masks](#)
- [Adjust Masks tips](#)

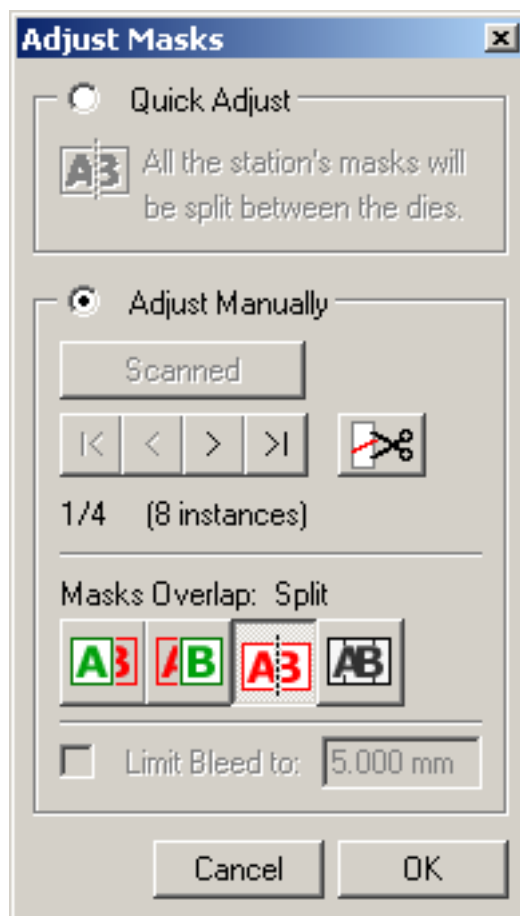
4.5.1. Using Adjust Masks

The Adjust Masks dialog box covers the SmartFlaps™ functionality. It assists you to intelligently adjust overlapping graphic masks on the plate. SmartFlaps™ suggests by default to split masks. Once a decision has been made, it is applied to all similar mask occurrences over the entire plate.

The Adjust Masks tool will create new masks that are applied to the stations on the plate. The stations' original masks are not altered.

To use Adjust Masks:

1. On the Tools menu, click 'Adjust Masks'. The *Adjust Masks dialog box* appears:



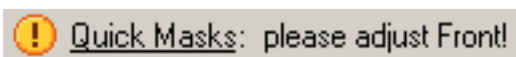
2. You can either opt for a quick adjust, or choose to adjust the masks manually.

When choosing **Quick Masks**, the masks will be split based on the size of the adjoining panels/flaps: large panels are automatically bleeding into smaller ones, assuming that smaller panels are the invisible flaps. The example underneath shows an example: the dark blue panel at the bottom is spread into the light blue glue strip.



The masks are not updated in real-time but they will be updated just before saving the file. To force an update of the Quick Masks, you can use the 'Adjust Mask' dialog box.

The *status* of the masks will be indicated in the Status bar. If the masks are not up to date, a warning will be displayed.



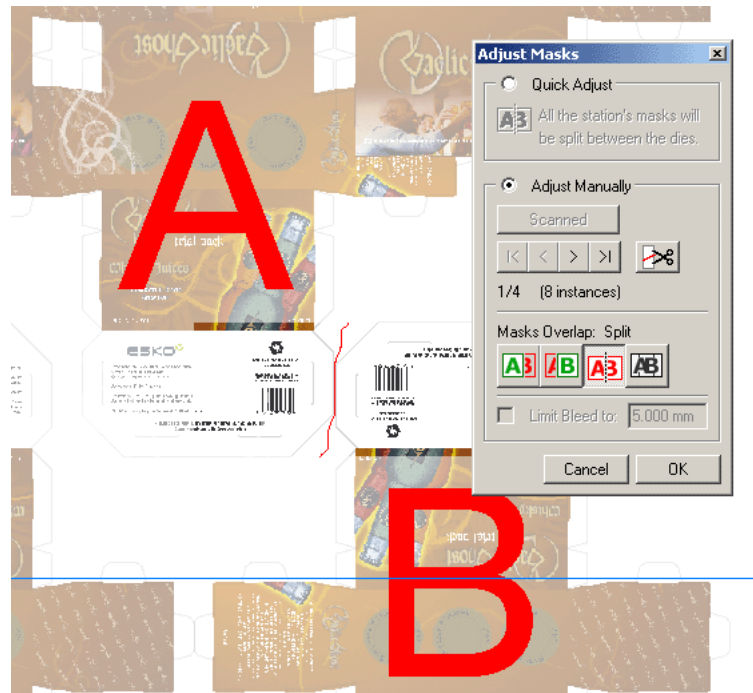
Manual masks (SmartFlaps™) allows you to decide on the overlap direction and bleed limit.

3. If you choose to adjust the masks manually, click the 'Scan...' button. Esko Plato will scan all the stations on the plate looking for overlaps. Depending on the number of stations on

your plate and their complexity, this scanning may take some time. Once Esko Plato has completed the scanning, the overlap records are displayed in the Adjust Masks dialog box.

The document window is zoomed to display the first overlap. The area where these two particular overlaps occur will be displayed normally. The area outside the overlap is dimmed. One of the stations will be labeled "A" and the other will be labeled "B".

When you choose to bleed 1 station into the other (A into B or B into A), you can define the maximum bleed per overlap situation.




4. Examine and set the correct mask for each record. By default, Esko Plato chooses the Split option for each record. If you want all the masks to split between the dies, you do not need to examine each record; you can simply choose 'OK'.
5. When you have examined and set all the overlaps, click 'OK'.

The Adjust Masks dialog box closes and new masks are applied to the stations on the plate. The decision that you made for each record is applied to all instances where that particular overlap occurs on the plate. When you use Adjust Masks, you only need to tell Esko Plato your mask decision once. Esko Plato knows all the locations where that decision should be applied.

Tip:

You can indicate your preferred way of masking by selecting 'Manual' or 'Quick' in the [Defaults tab](#) of the Preferences dialog box. Selecting an option in the adjust mask dialog will change the setting for the current file.

4.5.2. Splitting an Overlap Record

There may be cases in which you need to treat part of an overlap one way and the other part in a different way. Since a single record is modified by a single decision, you cannot do this unless you cut the overlap into two pieces. You use the Split Overlap tool  to create two different records from a single record.

In the following example, the two tuck flaps need to be underneath the two main panels. Since there is one overlap that needs to be treated in two different ways, this overlap needs to be split.

You use the Adjust Masks tool, as shown above, to split one overlap record into two records. To split an overlap:

1. Locate the overlap record you want to cut.
2. Click the Split Overlap tool. The tool is selected in red as a one shot tool; the cursor turns into scissors.
3. Click and drag in the document window where you would like to split the overlap record. The record is cut into two different records.
4. Set the mask decision for each record.

The record is split into two records, which can be adjusted independently. The two records will follow each other in the list. You can split an overlap record as many times as you want. Simply select the Split Overlap tool again to make another cut.

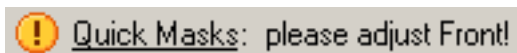
4.5.3. Resetting masks

When you use the Adjust Masks tool, new masks are applied to the stations on the plate. You can remove these masks by selecting 'Reset Masks' on the Tools menu.

When you remove masks from the stations in the plate, the stations revert to using their original masks as they were created in the Station View. The Reset Masks command is disabled if there are no modified masks on the current plate.

4.5.4. Checking the status of masks

How do you know if your masks are up to date? Esko Plato will display masks information in the Status bar. If the masks are not up to date, a warning will be displayed.

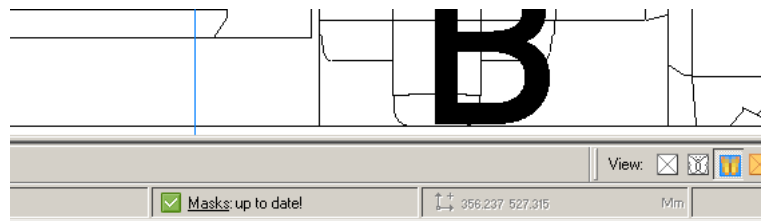


After adjusting, the Quick Masks status will show:

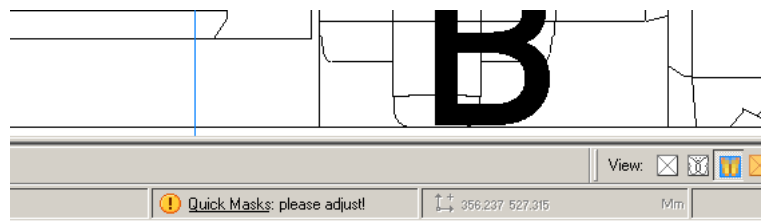


In fact, you can end up in one of these situations:

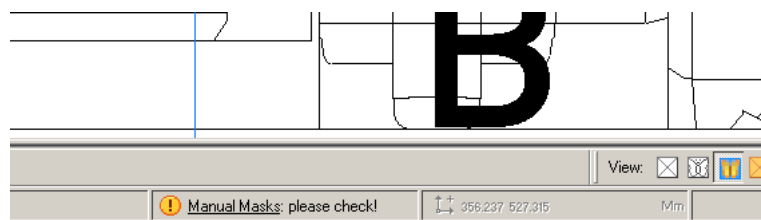
- 1. Masks: up to date!** This situation occurs when masks are generated by one of the methods mentioned above: "Quick Adjust" or "Manual Adjust". As long as no stations are added or moved, this status will not change.



- 2. Quick Masks: please adjust!** This situation occurs when masks were generated via "Quick Adjust". In case stations were added or moved the masks are possibly no longer up to date.



- 3. Manual Masks: please check!** This situation occurs when masks were generated via "Manual Adjust". In case stations were added or moved the masks are possibly no longer up to date.



To bring the masks up to date, go to the Adjust Masks dialog box by clicking the icon or the underlined text in the status bar

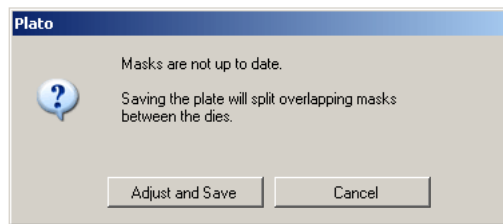


or by selecting '*Adjust Masks*' on the Tools menu.

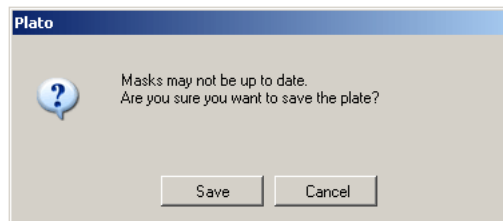
Remarks:

- Masks should be adjusted on both sides of the plate. Only the status of the mask on the current plate side is shown!
- When the masks are not up to date and a user wants to save his work, a warning will be displayed.

In case a plate uses the "Quick Adjust" way:




In case a plate uses the "Manual Adjust" way:



4.5.5. Adjust Masks tips

- If you click 'Cancel', the Adjust Masks dialog box is closed and no masks are adjusted.
- When you use the Adjust Masks tool, new masks are created for the stations on the plate. The new masks are used in the plate only; the original station file and its mask remains unmodified.
- Adjust Masks uses the die outlines and the station masks to make adjustments on the plate. Therefore, it is a good idea to use the mask tool in the Station View to create a mask. If your station doesn't have a mask, Esko Plato doesn't have this information and can only use the bounding box of the station to create the new masks.
- In order to see the 'A' and 'B' labels, make sure the 'Show selection in preview' option is switched on in the *View tab* of the Preferences dialog box.

4.6. Nesting stations

The Esko Plato toolbar boasts a Layout tool . The main purpose is to facilitate stepping and repeating one and the same station in a particular area of the sheet. However, this tool also enables you to create a more intricate and advanced layout whereby labels or even folding cartons can be positioned in such a way that a maximum of space is used within the given area. This is called nesting.

Note:

The Layout tool takes into account the shape of a station. This implies that the shape of the station is correct prior to using the Layout tool. You can adjust the shape via the Properties dialog box.

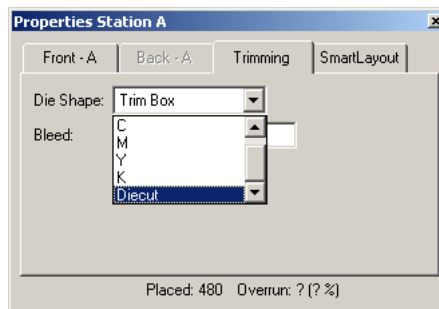
To nest stations, perform the following steps:

1. *Add a station* to your sheet.
2. Right-click the station (depending on the 'Origin', that will be in one of the corners of your sheet) and go to 'Properties Station' or click the Grid Settings button

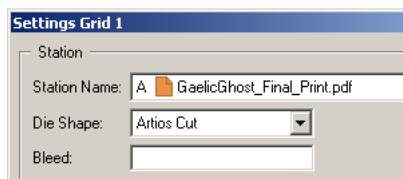


in the Grids dialog box.

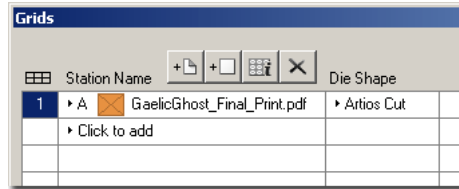
3. **a.** In the Station Properties dialog box, go to the Trimming tab, and select 'Trim box' or 'Die Cut/ArtiosCut' from the 'Die Shape' list.



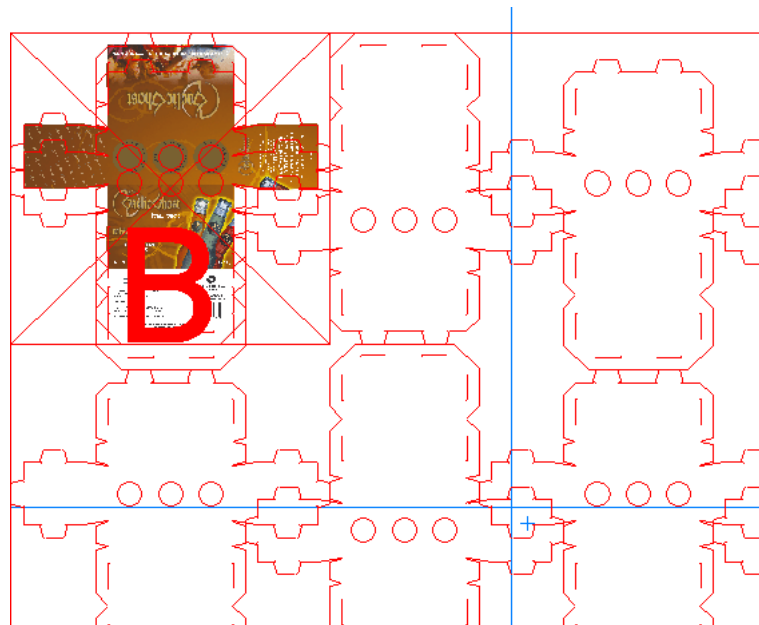
- b.** In the Grid Settings dialog box, select 'Trim box' or 'Die Cut/ArtiosCut' from the 'Die Shape' list.



- c. A third way is to select 'Trim box' or 'Die Cut' in the 'Die Shape' column of the *Grids* or *Stations* dialog box.



- 4. Finally, select the Create Layout tool and nesting mode from the fly-out toolbar and draw a rectangle in which a vertical and horizontal repetition of the station in question will appear, but whereby the stations are organized on the sheet in such a way that they are fitted into one another.



4.7. Numbering Stations

Use the Number Stations command to assign station numbers to the stations in your layout. You must have a station number in your station for the numbers to be displayed on the plate. You can add a station number to a station at any time. The actual numbering of the stations takes place in Esko Plato.

Station numbers of 1 station all have the same number. **Strip numbers** are continuous numbers over all stations.

For more information on the tool, please refer to the chapter '*Place Station / Strip Number tool*'.

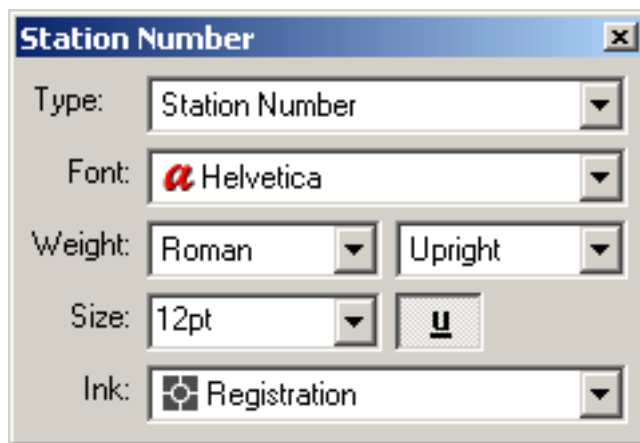
- *Using the Place Station / Strip Number tool*
- *Numbering stations automatically*
- *Numbering stations manually*




4.7.1. Using the Place Station / Strip Number Tool

The Place Station Number tool is used to set the parameters for a station number and to place the number on the station.

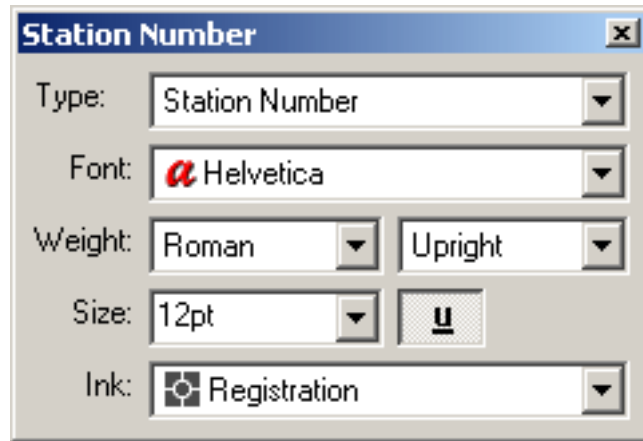
To create a station number:

1. Click the **Place Station / Strip Number** button  on the Tools toolbar to activate the Station Number dialog box:



2. Set the parameters for the station number you want to add to your station. You can also add a pre- and suffix to the station number.
3. Once the 'Place Station Number' tool is selected and you move the mouse over the document the following pointer will be shown 
4. Click the position where you want the station number to be added. The cursor is changed into  (station number) or  (strip number). You can add as many station numbers as you want. However, every number will have the same value when added to the plate.
5. When you have placed all the numbers you want, click any other tool on the toolbar.

A station number always displays as a "0" in the Station View:



When a station with a station number is placed or assigned to a plate, the numbers are automatically replaced with unique numbers. The unique numbers in the plate have the same placement and parameters as the placeholder that you added to the station.

Tip:

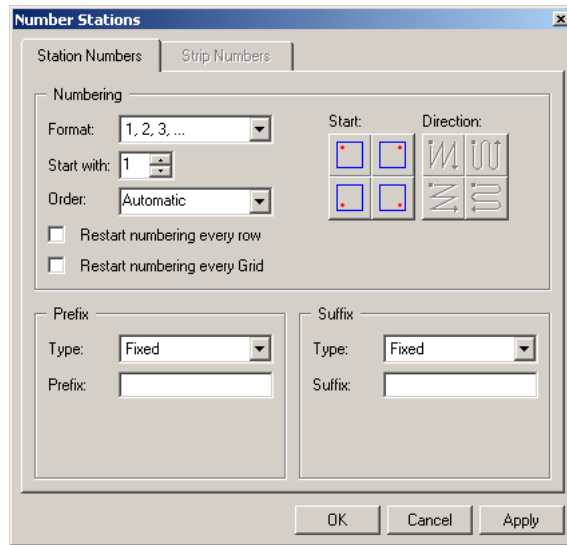
- The Ink list in the Station Number dialog box contains the inks that are used currently in the station. The list also contains some special options for creating a white station number (Knockout), a station number that prints in all inks (Registration), and a station number that prints in the Darkest ink. Registration and Darkest choices will automatically update if the inks in your station change.
- You can edit a station number by double-clicking on the number in the document window. The Station Number window will appear showing you the number's current parameters.
- When you create a station number, a layer named "Station Number" is automatically created. This layer is a special layer that is a non-printing layer. The layer is made non-printing so that the station number placeholders do not appear on the plate.
- A selected number can always be moved by using the arrow keys, even if the Transform tool is not active.

Please refer to the chapter '[Number Stations...](#)' for more information on the various options.

4.7.2. Numbering stations automatically

Esko Plato includes 16 possible number orders from which you can choose.

1. On the Tools menu, click '[Number Stations...](#)'. The Number Stations dialog box appears:

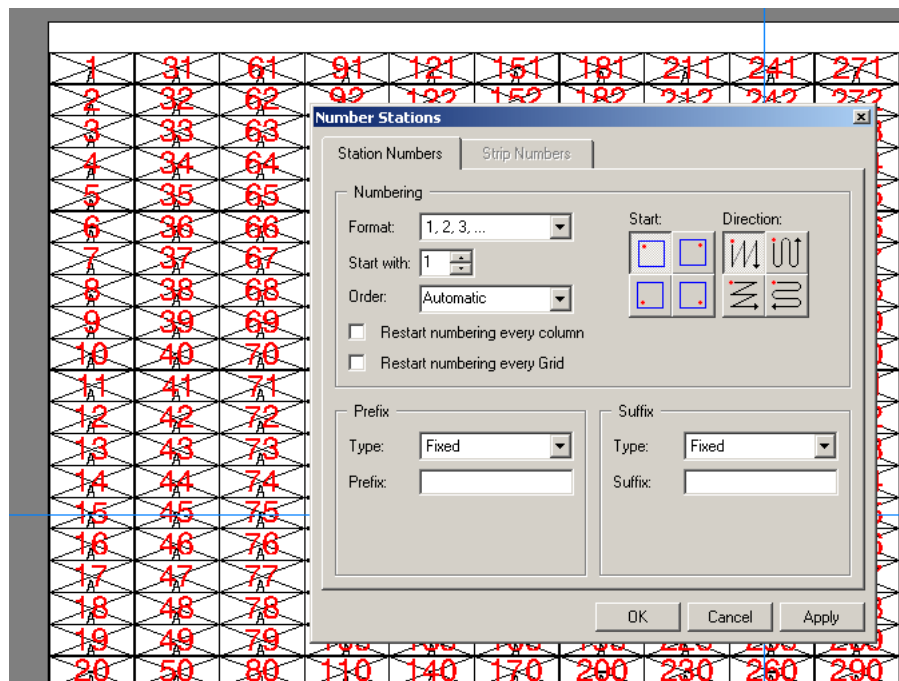


2. Indicate where you want to start numbering and specify a direction. Enter a start number if necessary. Numbers will appear on the stations of the layout.



Caution:

When numbering stations, the position of the center of the label is taken into account.



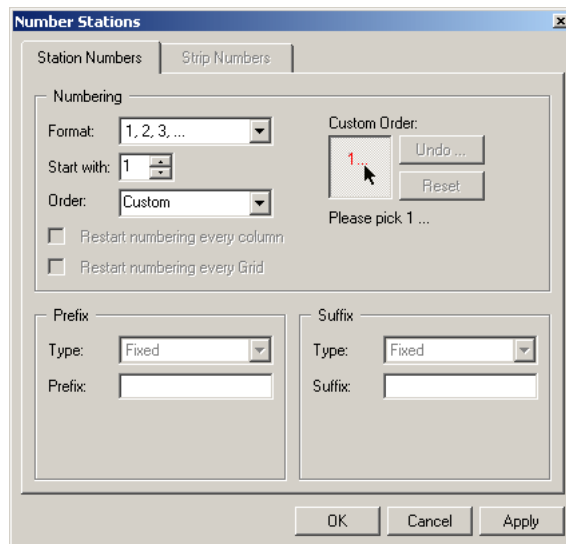
3. When you have selected the order you want for the numbers, click 'OK'.

4.7.3. Numbering stations manually

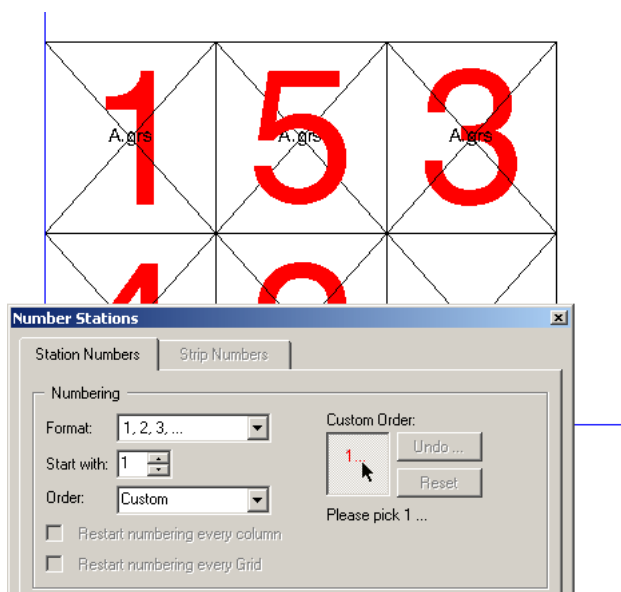
If one of the automatic orders is not what you need, you can define your own Custom Order. With Custom Order, you indicate the number for each station by clicking it. You continue clicking stations until all stations have been numbered. While Custom Order allows you to pick any order, it can be a long task to select every station in a large, complex layout.

To number stations with a custom order:

1. On the Tools menu, click '*Number Stations*'.
2. Enter a start number if necessary.
3. Click the 'Custom Order' button.



4. Click each station to set the order. You can undo the last choice by clicking 'Undo'. You can reset all the numbers and start over by clicking 'Reset'.



5. When you have numbered all the stations, click 'OK'.

You must click all stations to complete the custom numbering. The Number Stations dialog box indicates which number you need to pick next.

Number Stations tips

- When using 'Number Stations', the document display is set to cross view to make it easier to see the numbers. When you exit the Number Stations dialog box, your original display mode is restored.
- The Number Stations tool is disabled when there are less than two stations on the plate.
- Stations can be numbered even if they do not contain a station number. If you add a station number to the station in the Station View mode later, the numbers will appear automatically.
- You can use the zoom, pan and display mode tools to get a better view of the plate. However, using other tools or commands will cancel the Number Stations tool automatically.

4.8. Using PDF files as stations

When you place a PDF file directly in Plato, you are not using a station file (STA). Because PDF files contain less information than PDFSTA files, some of Plato's commands work differently than they do with station files.

- [Die and Mask of a PDF file](#) on page 86
- [Adjust Masks with PDF files](#) on page 86

- [Number Stations with PDF files](#) on page 86

4.8.1. Die and Mask of a PDF file

When a PDF file is used as a station, the Borders of the PDF file are used as the Die and the Margins of the PDF file are used as the Mask. Therefore, all commands that use the Die and Mask, like Step-and-Repeat, Align, Transform, and Adjust Masks, will use these Borders and Margins.

4.8.2. Adjust Masks with PDF files

The Adjust Masks command deserves a special note. Because the Borders and the Margins are simple rectangles, the Adjust Masks command does not have a complex mask for irregularly shaped stations. If your PDF file has an irregular shape, like a folding carton or an odd-shaped label, then Adjust Masks will use the Margin rectangle as the Mask. This is sufficient for those cases when you choose the Split option in Adjust Masks to split between the dies.

However, if you select the A over B or B over A options, then the mask of one station is used to clip the mask of another station. If the masks are simple rectangles, as in the case of PDF files, then one station simply cuts a rectangular hole in the other. This may not be what you want for irregular shapes. For these jobs, you will need to make a station file that contains a complex mask in the Station Editor to get the right overlaps.

4.8.3. Number Stations with PDF files

The Number Stations command will work identically regardless of whether a station is a PDFSTA or PDF file. However, since PDF files do not contain Station Numbers, no number will appear in the layout. Only PDFSTA files that contain a Station Number placeholder will display numbers in the layout. If you need station numbers to appear, you must create a station file (PDFSTA).

4.9. Using GRS files as stations

When you place a GRS file directly in Plato, you are not using a station file (STA). Because GRS files contain less information than STA files, some of Plato's commands work differently than they do with station files.

- [Die and Mask of a GRS file](#)
- [Adjust Masks with GRS files](#)

- [Number Stations with GRS files](#)

4.9.1. Die and Mask of a GRS file

When a GRS file is used as a station, the Borders of the GRS file are used as the Die and the Margins of the GRS file are used as the Mask. Therefore, all commands that use the Die and Mask, like Step-and-Repeat, Align, Transform, and Adjust Masks, will use these Borders and Margins.

4.9.2. Adjust Masks with GRS files

The Adjust Masks command deserves a special note. Because the Borders and the Margins are simple rectangles, the Adjust Masks command does not have a complex mask for irregularly shaped stations. If your GRS file has an irregular shape, like a folding carton or an odd-shaped label, then Adjust Masks will use the Margin rectangle as the Mask. This is sufficient for those cases when you choose the Split option in Adjust Masks to split between the dies.

However, if you select the A over B or B over A options, then the mask of one station is used to clip the mask of another station. If the masks are simple rectangles, as in the case of GRS files, then one station simply cuts a rectangular hole in the other. This may not be what you want for irregular shapes. For these jobs, you will need to make a station file that contains a complex mask in the Station Editor to get the right overlaps.

4.9.3. Number Stations with GRS files

The Number Stations command will work identically regardless of whether a station is a STA or GRS file. However, since GRS files do not contain Station Numbers, no number will appear in the layout. Only STA files that contain a Station Number placeholder will display numbers in the layout. If you need station numbers to appear, you must create a station file (STA).

5. Working with Graphics

Esko Plato was designed to be a replacement for analog step-and-repeat machines. These machines step and expose a film for each item (a station) onto a larger piece of film that is used later to expose the printing plates. Plato uses graphics the same way that a step-and-repeat machine does: imported graphics are assumed to be digital film, ready for stepping.

Esko Plato is capable of doing quite some things a regular step-and-repeat machine cannot. However, like a step-and-repeat machine, Plato assumes that the graphics files are completely ready for imaging. This means that the graphic files should be of the correct size and have the correct content. For this reason, graphics are not directly editable in Plato.

- [About importing a graphic](#)
- [Replacing a graphic](#)
- [Importing an ArtPro file](#)
- [Importing a PostScript file](#)
- [Importing a PDF file](#)
- [Importing an EPS file](#)
- [Importing an Illustrator file](#)
- [Importing an EPF file](#)
- [Positioning a graphic](#)
- [Editing a graphic in Esko PackEdge](#)

5.1. About importing a graphic

When you import a graphic into the Station View, you can position and rotate the graphic just like you can with a piece of film. However, you cannot scale the graphic in the Station View. Scaling a graphic violates the digital film idea and can distort certain features of the graphic, like trapping.

Graphics are always imported as external references. These external references provide the high resolution display of the graphic and contain both size and ink information. Plato uses these features to build station objects.

Station objects use graphics in native Esko-Graphics file formats. Plato can also import many desktop graphic file formats which are automatically converted to an equivalent Esko-Graphics native file format before being placed in the station.

Options windows for importing graphics files

Some file formats have special features that can be accessed by their Options window. The Options window for each imported graphic file type is discussed in the sections below.

You import a graphic into a station by choosing the appropriate file type, selecting the file, and setting options in the Options window, if necessary. The default options for each file type have been chosen carefully so you should change these options only when you really need to.

All of the Options windows for the file types have two useful buttons: the Reset button and the Customized Settings button.

- Reset - The Reset button resets all the options that you may have changed in the Options window to their default values.
- Customized Settings - The Customized Settings button allows you to save the current settings so that they can be used later. You can also save the settings as your own default. This feature is very useful for those file types that have many options that you modify each time you import a graphic of a particular file type.

5.2. Replacing a graphic

There are two ways to replace a graphic in a station with a different graphic.

1. You can use 'Replace by Graphics File' in Station View to discard an existing graphic and put a different one in your station.
2. You can use the Station properties dialog box to replace the graphics.

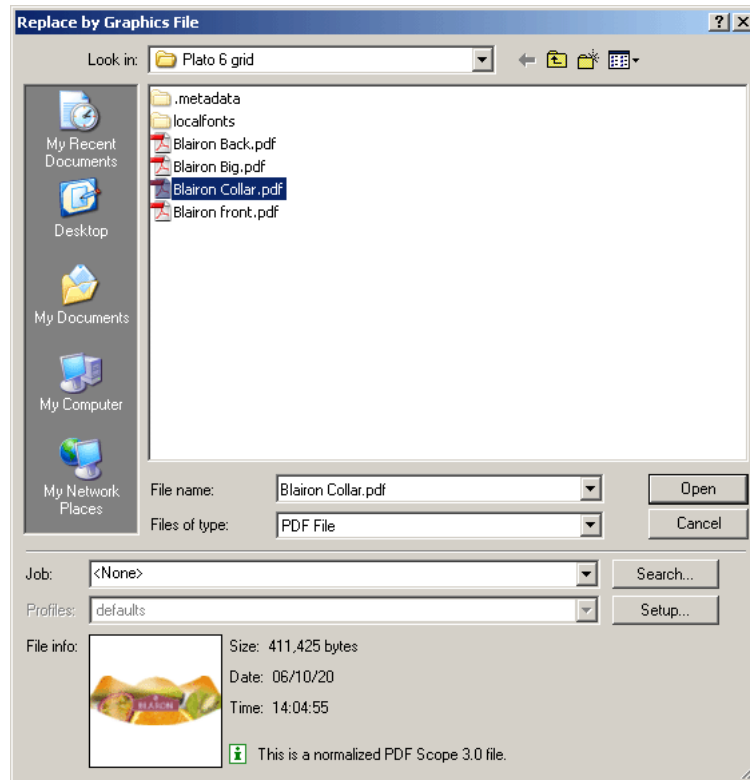
'Replace by Graphics File' will delete the current graphic and place a new graphic in the station. The new graphic is placed, based on the information of the current graphic in the station, which implies that the new graphic will have the same position and rotation as the graphic that it replaces. The center of the graphic is aligned with the center of the current die. If this position is not correct, you can move or rotate the graphic with the Transform and Align tools.

Tip:

This feature is useful for changes that require a new graphic, while all the other parts of the station need to remain exactly the same.

To replace a graphic:

1. On the Station menu, click 'Replace by Graphics File'. The Replace Graphic dialog box appears:



2. Select a file type.
3. Locate and select the file you want to use.
4. Click 'Setup...' to open the options window for the file type you selected.
5. Set any options that are required.
6. Click 'Open'. The graphic is converted to the Esko-Graphics native format, if necessary, and is placed in the station file.

Or you can:

1. Open the Station dialog box.
2. Select the station.
3. Open the station Properties dialog box of the station.
4. Click 'Browse' to browse to a new graphics file.

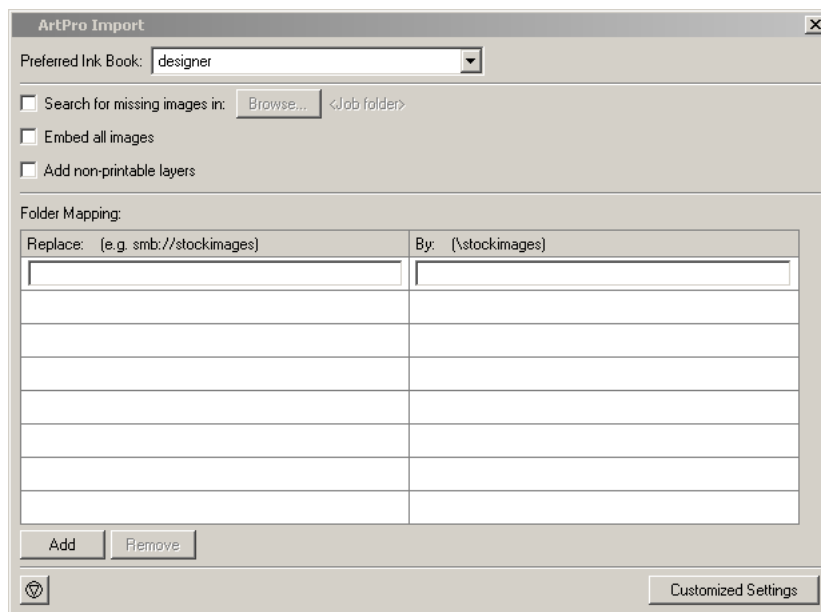
Tip:

If the new graphic uses different inks than the current one, you may need to clean up the inks that have now become unused. For more information, please refer to the chapter ['Inks'](#).

5.3. Importing an ArtPro file

With the ArtPro import filter you can import an (Esko)Artwork ArtPro file into Plato directly. The ArtPro files must have file extension .ap or .art.

Additional options appear in the ArtPro Import dialog.



Search for missing images in:

Use this to search for missing images in a specific folder.

Note: The images are searched: in the place referenced in the links, next to the input file, in the folder defined in this dialog.

Embed all images

Use this to embed all images in the file.

Add non-printable layers

Use this option to include all layers, including the non printing layers.

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 Plato (Windows) workstation and use the Folder Mapping feature in the Export ArtPro to Normalized PDF task to keep the 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 Plato machine.

Example: In our ArtPro file, the path of 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 Plato 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:/".

Limitations

There are some limitations when importing ArtPro files in Plato.

- When you import an ArtPro file into Plato, you will notice that the graphic visualisation of both documents is the same, but the document structure is not.

Example: Linked ArtPro files and external ArtPro CT files will be embedded, and certain special objects in the ArtPro file will have been degraded to normal objects.

Where possible, links to external images will be maintained.

- Step and Repeat or externally linked files in ArtPro to non-normalized PDFs and PostScript files cannot be imported.
- Objects that do not have screening information in the ArtPro file get the following defaults:
 - frequency 120
 - dotshape: simpledot (round)
 - angles C: 15, M: 75, Y: 0, B: 45 and any others: 45
- Inks from the PANTONE and Visualizer ink book are recognized. All other (custom) inks are loaded from the ink book "designer" in Plato.
- Ink attributes "opaque", "varnish" and "normal" are kept. Ink attribute "Technical" (Die) is not kept.
- Nexus screening is not allowed. For Extended Nexus screening, only use angle/ruling.
- The font is subsetted and embedded. This results in real text in Plato. When you import an ArtPro file, two-byte fonts are always vectorized.
- Where possible, images keep their links. This is the case for most Tiff, jpeg and PSD files, and also for mapped images. Some image formats are not supported by or are incompatible with Esko software. This is the case for DCS images, PSD with multiple alpha channels and Tiffs with extra channels which are often generated by ArtPro PS/PDF import. These images are either embedded or they generate a .ct file on Esko software.
- Artlink updates are disabled.
- Barcode information is not kept, nor is the link to BAG and ARD files when the ArtPro file is imported.

5.4. Importing a PostScript file

You can import PostScript files and use them as a graphic in your station. PostScript files are converted to GRS/PDF files. Composite and separated PostScript files can both be imported.

The Options window for importing PostScript and PDF contain the same features. The Options for Illustrator and EPS contain some, but not all the same features as the PostScript Options window. All the features of the PostScript Options window will be described here. The features that are used in PDF, Illustrator and EPS files refer to this section.

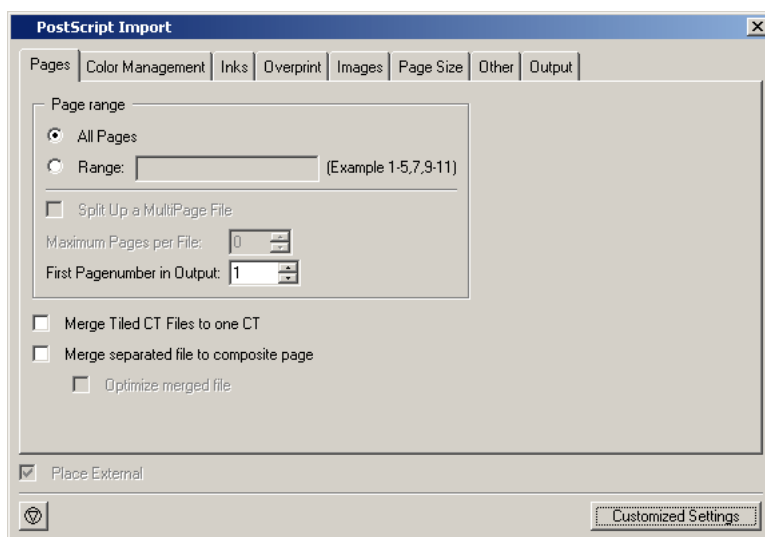
Clicking 'Setup...' in the Replace Station dialog box opens the PostScript Options window. The Options window contains tabs to fine-tune the import of PostScript files.

- [Pages](#)
- [Color Management](#)
- [Inks](#)
- [Overprint](#)
- [Images](#)

- [Page Size](#)
- [Other](#)
- [Output](#)

5.4.1. Pages

Some options of the Pages tab are used by Esko Software Suite applications other than Esko Plato. Some options are disabled, as shown below:



Select **All Pages** or specify a **Range** of pages.

Split up a multipage file To divide the original multipage 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 GRS/` 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.

Split up a multipage file Check the box to divide the original multipage 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.

Merge Tiled CT files to one CT 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 editability of the Esko native file and to optimize further processing. This option is default selected, it doesn't matter if you import a file with no tiled CTs.

Note:

When errors occur when a PostScript file is imported (e.g. a font is not found), the **Merge Tiled CTs to one CT** will not be performed.

Merge Separated PostScript file to Composite page Merges separated single-page and multi-page PostScript files to a composite file.

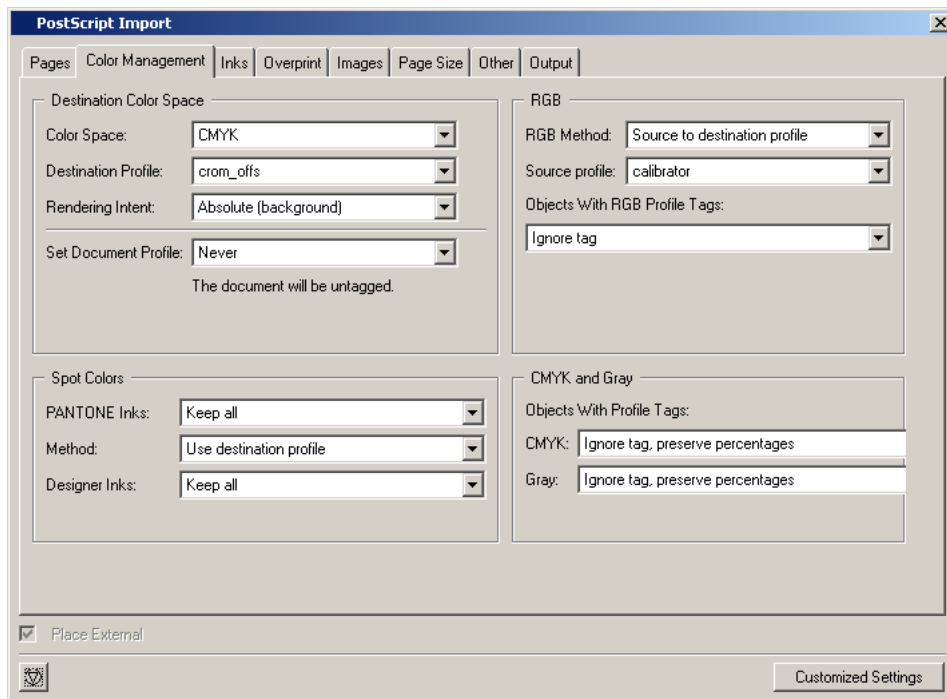
Optimize Merged PostScript File This option optimizes the merged PostScript file. This is useful when you need to edit the imported PostScript file. This option becomes available when 'Merge Separate PostScript File to Composite Page' is activated.

Note:

When errors occur when a PostScript file is imported (e.g. a font is not found), the **Optimize Merged PostScript File** will not be performed.

Password You can enter a password for protected PDF files.

5.4.2. Color Management



- [Destination Color Space](#)
- [Spot Colors](#)
- [RGB](#)
- [CMYK and Gray](#)

Destination Color Space

Here you define what color space the opened/imported document will have.

Color Space

- CMYK
- Grayscale

Destination Profile Choose a color profile from the drop-down list. Currently only CMYK and Grayscale profiles are supported, you can choose between standard ICC profiles and Esko Kaleidoscope profiles.

The 'Use PDF Output intent if available' toggle: in some PDF files, the PDF Output intent is defined (e.g. PDF/X-3). If you want to use the color space as defined in the output intent, use this toggle. If you selected the toggle 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. Any color that falls outside the range that the output device can render is adjusted to the closest color that can be rendered, while all other colors are left unchanged. 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):** Any colors that fall outside the range that the output device can render are adjusted to the closest color that can be rendered, while all other colors are left unchanged. The white point is preserved.
- **Perceptual:** This will result in no background simulation (white of reference color space is mapped on white of destination color space). In other words, if certain colors in an image fall outside of the range of colors that the output device can render, the picture intent will cause all the colors in the image to be adjusted so that the every color in the image falls within the range that can be rendered and so that the relationship between colors is preserved as much as possible. 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.
- **Saturation:** Use this option if you care especially about saturated colors. It preserves the saturation of colors in the image at the possible expense of hue and lightness.

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.
- **Never:** 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** 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.

RGB

RGB will always be converted to process inks during import.

RGB MethodSelect one from the drop-down 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 Kaleidoscope Device Link Profile. this Link Profile implicitly determines the Source and Destination profiles.

Source Profile / Link ProfileSelect one from the drop-down list.

Objects with RGB Profile tagsThe 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.

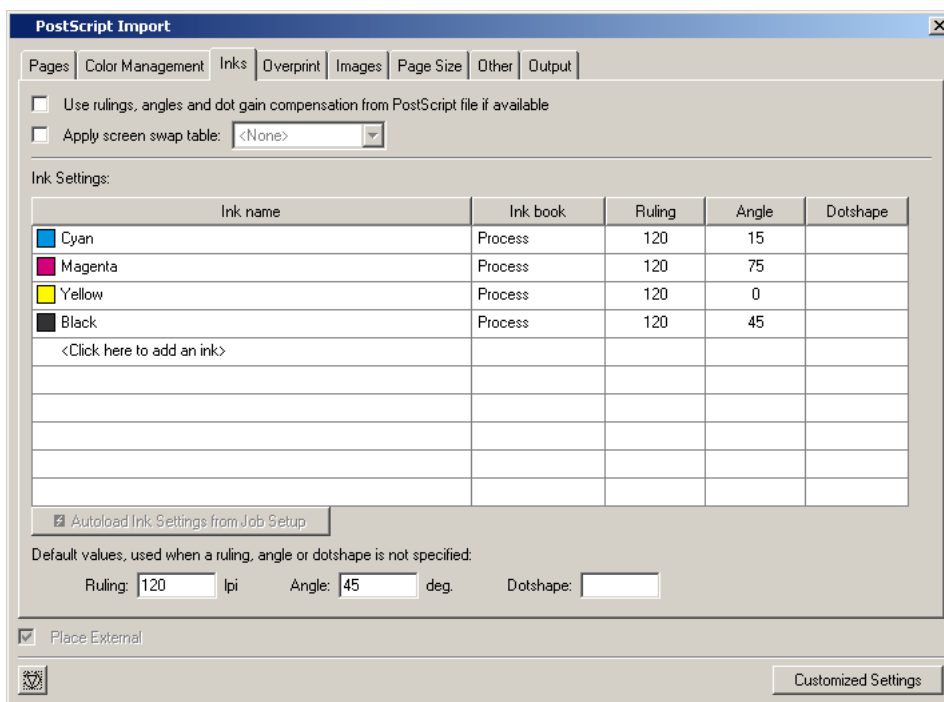
- **Convert, preserve appearance:** the objects' color profile will be taken into account; this means that their color values will change.

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.

5.4.3. Inks

The Inks / Separations tab depends on the output type you selected in the *Color Management 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.



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.

Apply Screen Swap Table Select a screen swap table from the drop-down list.

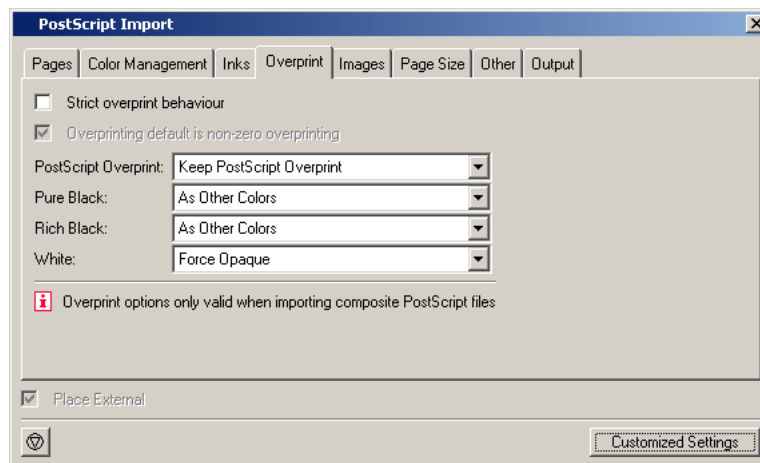
Inks (double-click the fields to edit)

Autoload Ink Settings from Job Setup If Job parameters were defined, you can use this toggle to make sure these inks are used. If you do not click this button, you are free to edit the ink

settings. The default values as defined at the bottom are used for ink attributes that are not set in the ink settings of the job setup.

Default values, used when a ruling, angle or dotshape is not specified. Enter the default values for ruling angle and dotshape, the system will take these values when they are not specified elsewhere (i.e. in the PostScript file, or in job Parameters, ...).

5.4.4. 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 toggle 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).
- 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.

PostScript Overprint To set the overprint mode that the conversion needs to apply when importing the file.

- Convert into Opaque: Ignores any overprint of your job.
- Convert to Darken (Esko): Converts all overprint present in your PostScript job to Darken.

- Keep Postscript Overprint: All PS overprint present in the job is imported correctly. (This is the default setting.)

Note:

The preferred setting is 'Keep PS Overprint'. This will result in faster conversion when outputting the job to PS/EPS/PDF.

Pure BlackTo set the overprint mode for objects containing 100% black only. Possible Pure Black overprint modes in the drop-down list:

- As Other Colors: As it is defined in the job.
- Force Opaque (masking): Forces Opaque for objects in 100% black and knocking out all underlying inks.
- Force >Postscript Overprint: Forces PostScript overprint for objects in 100% black.
- Force Darken (Esko): Forces Darken for ALL objects in 100% black. The black color is printed on top of other colors, instead of erasing the color underneath.



Caution:

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.

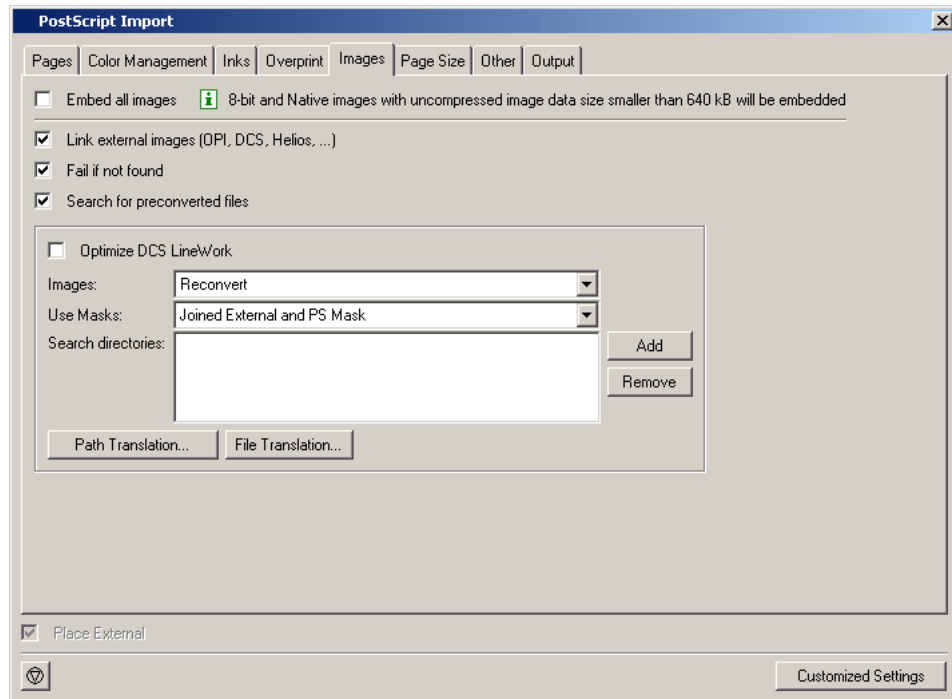
Rich BlackSet the paint mode for objects containing 100% black and at least one other ink.

- As Other Colors: 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 Postscript Overprint: Keeps the PostScript overprint for objects in 100% black and at least one other ink (overprint objects stay in overprint and opaque objects stay opaque).
- Force Darken (Esko): Forces Darken for all objects in 100% black and at least one other ink.

WhitePossible White paint modes to select from the drop-down list:

- Opaque: Forces all objects in white to Opaque.
- Keep PostScript Overprint: Keeps the PostScript overprint for objects in White (overprint objects stay in overprint, opaque objects stay opaque)

5.4.5. Images



Embed all imagesAll images become internal images.

Link External Images (OPI, DCS, Helios, ...)Force the system to look for referenced images.

Fail if not FoundActivate 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.

Search for Preconverted FilesThis 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 in stead.

Optimize DCS LineWorkWithout linework DCS optimization, the result after conversion of linework DCS files can be difficult to edit. 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.

ImagesAvailable options are:

- 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 MasksAvailable options are:

- *PS Mask*: Forces the system to take the PS mask present in the PS file. The mask available in the external image is discarded. If no PS mask is present in the PS 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 PS 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 PS file and the available mask in the external image.

Search directoriesTo specify a list of directories in which to look for referenced images. Use the buttons 'Add' and 'Remove' 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.

Path TranslationTo define an understandable link for paths from a platform to another platform. The Path Translation dialog box opens, it has three columns:

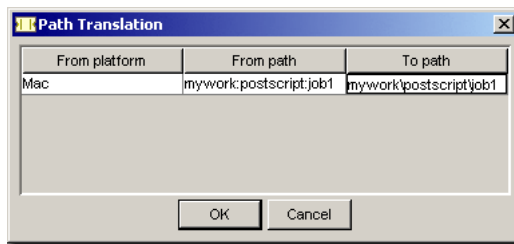
- *From Platform:* Use the drop-down list to select a platform from which the files are to be retrieved: NT, Macintosh, Unix or VMS
- *From path:* Specify the syntax of the path(s) of the platform you want to retrieve files from. e.g.

```
Mywork:PostScript:job1
```

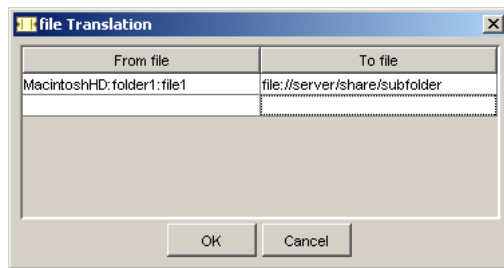
- *To Path:* Define the syntax of the paths so that the destination platform understands. e.g.

```
Mywork/PostScript/job1
```

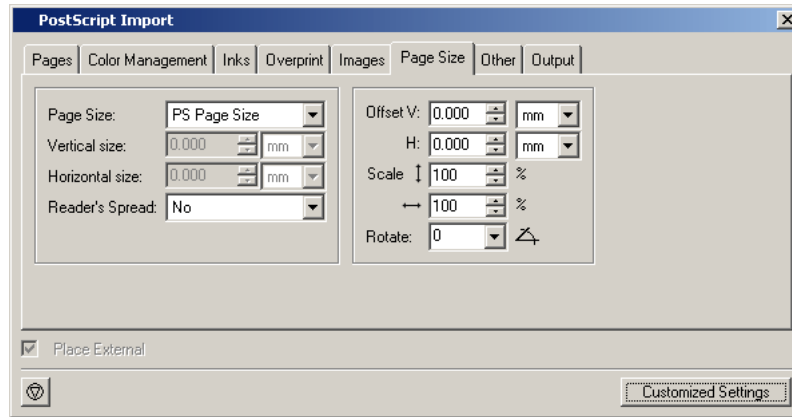
If you were to go from Macintosh to NT, the path translation could look like this:



You can do the same for files: **File Translation.**



5.4.6. Page Size





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 drop-down 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.

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.

Scale vertical  To enter the vertical scale percentage. 100% is the original size of the PostScript file.

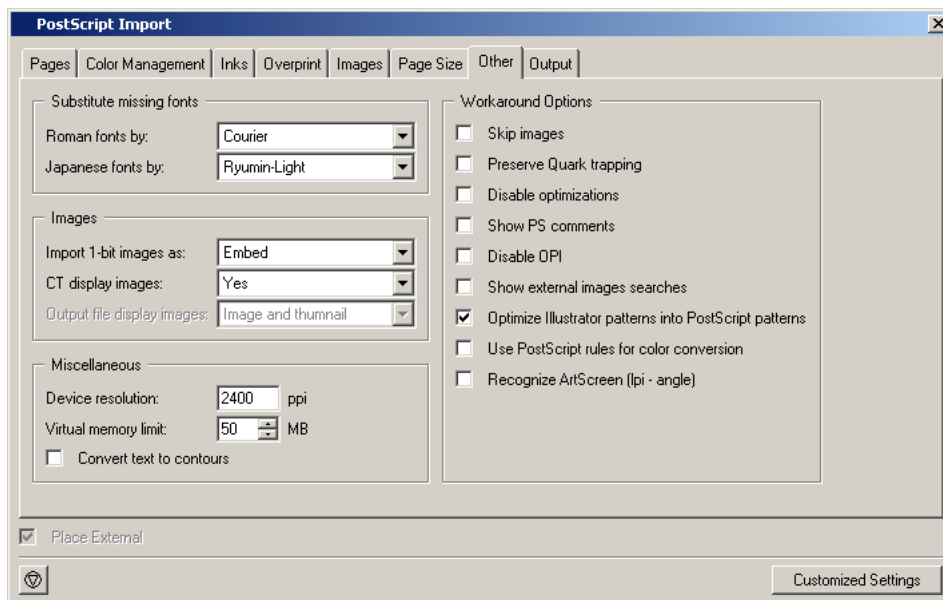
Scale horizontal  To enter the horizontal scale percentage. 100% is the original size of the PostScript file.

Rotate  Forces the system to rotate the PostScript File with the specified angle. The origin for rotation is bottom left. Possible angles in the drop-down 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.

5.4.7. 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

Images Import 1-bit images as:

- Embed: Images become internal black and white bitmaps.
- Linked file: 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 This option is activated when 'Split Up a MultiPage File' was activated. Select the type of display from the drop-down list: None, Thumbnail, Image.

Miscellaneous

Device resolution (xxx ppi) Device resolution is 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 to 1 gigabyte.

Convert text to contours will contourize all text.

Note:

Conversion might increase the file size.

Workaround Options

Skip Images Check this box to open the EPF 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 (page 5) 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.

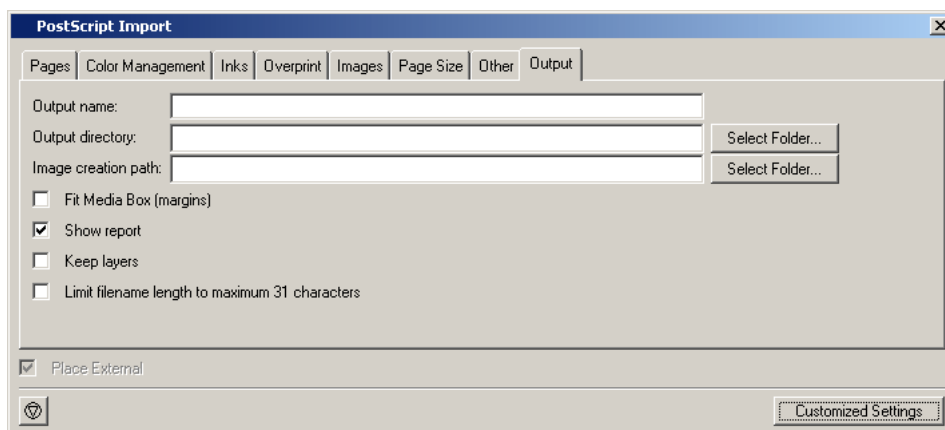
Optimize Illustrator Patterns into PostScript patterns Illustrator writes patterns in a different way than PostScript does, default behavior is that Illustrator patterns will be optimized. Normally this speeds up the further processing (less linework in your file) sometimes, though, the optimization itself may take some time.

Use PostScript rules for color conversion Check this box to make sure that all color management will happen using PostScript rules, in stead of BGCMS. Most options in the [Color Management tabsheet](#) will be greyed 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-Graphics 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.

5.4.8. Output

The Output tab is used to specify the name and directories for converted PostScript files.



Available options on the Output tab are:

Output Name Specify an output name for the files created during conversion. If no name is specified, the original PostScript file name is used.

Output Directory Specify the directory where the converted files should be saved. If no directory is specified, the converted files are saved in the current working directory. Note: If an Image Directory is specified, generated CT images will be saved in the image directory and will not be saved in the Output Directory specified here.

Image Creation Path 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.

Fit Media Box (margins) Check this box to automatically fit the margins to the bounding box of all the objects in the file.

Show Report This option displays valuable information during the import of PostScript files. Information about errors, linked images, search directories,... is displayed.

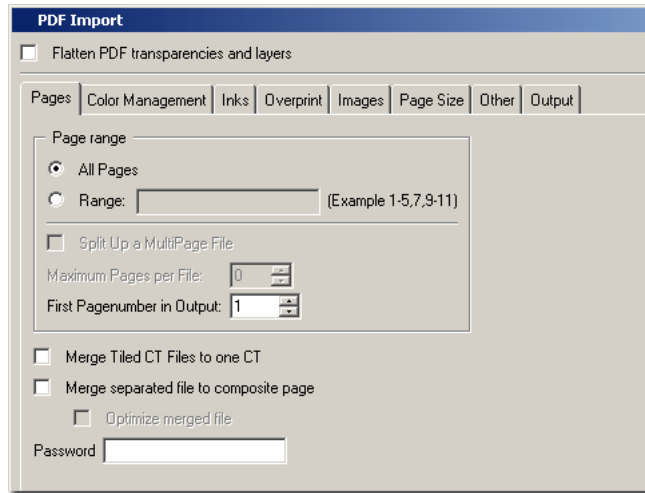
Keep Layers Keeps the Illustrator layer information. This will only work for Illustrator 8 files. It will not work for native Illustrator 9 files (or higher). When normalizing PDF 1.5 files (or higher), the 'Keep Layer' option will also preserve the PDF layers.

Limit Filename Length to Maximum 31 characters. Check this box 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 instance).

5.5. Importing a PDF file

The Options window for importing a PDF file contains the same features as those for PostScript. There are two differences between PostScript and PDF import:

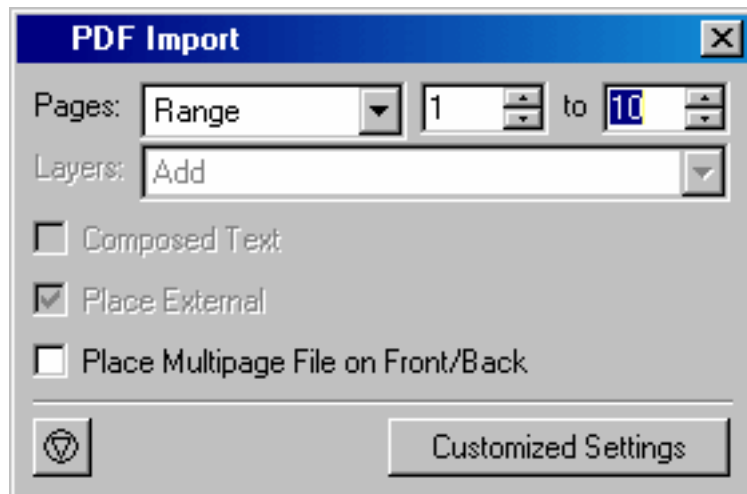
- On the Pages tab, a password field is available. If the PDF file is password-protected, you must enter the password so that Esko Plato is able to convert the PDF file.
- Flatten PDF transparencies and layers Check the box to import PDF files as in Scope 3 (and lower). All transparencies and layers will be flattened. If not checked, a normalized PDF file will be created that contains all typical PDF features (like transparencies and layers).



See [Importing a PostScript file](#) for complete descriptions of the import options.

5.5.1. Importing multipage PDF files

When importing multipage files (without normalization), it is now possible to specify that every other page should be treated as verso graphics. Example below will create 5 stations by loading page 1, 3, 5, 7 and 9, page 2, 4, 6, 8 and 10 will be used as the verso graphics for these stations.



5.6. Importing an EPS file

The Options for importing an EPS file contain some, but not all the same features as the PostScript Options window. In particular, the Pages and the Page Size tabs do not appear.

See [Importing a PostScript file](#) for complete descriptions of the import options.

5.7. Importing an Illustrator file

The Options for importing an Illustrator file contain some, but not all the same features as the PostScript Options window. In particular, the Pages and the Page Size tabs do not appear.

See [Importing a PostScript file](#) for complete descriptions of the import options.

5.8. Positioning a graphic

When you import a graphic, the center of the graphic and the center of the die are aligned with one another. If this position is not correct, you will need to position the graphic in the Station View using the Transform and Align tools.

A graphic can be moved and rotated to precisely position the graphic in relationship to the die. However, you cannot scale a graphic. The scaling feature of the Transform window will be disabled whenever a graphic is included in a selection. If the Scale option of the Transform tool is used on a selection that includes a graphic, scaling will also be disabled.

Tip:

When placing a GRS file in a CAD workflow, Esko Plato will analyze the GRS file and look for die objects. If a die object is found, the GRS will automatically be rotated and moved so that the die object is centered on the current CAD file.

5.9. Editing a graphic in Esko PackEdge

PackEdge can be started up from within the Station View to edit a graphic used in a station. Esko PackEdge must be installed on the same computer for the two applications to exchange files.

To edit a graphic in Esko PackEdge:

1. In Sheet View, select the station that contains the graphics to be edited.
2. On the Station menu, click 'Edit Graphics File...'.

Esko PackEdge is started and the graphic file is opened for editing. Make any changes you like and save the file.

For this command, you must have a graphic in your station. If no graphic is present, 'Edit Graphics File' is disabled.

6. Working with Esko Plato Tools

Using the tools in Esko Plato to organize and layout a plate

- [Arranging Objects](#)
- [Creating a Repetition](#)
- [Creating two-sided plates](#)
- [Managing Inks](#)
- [Trapping](#)

6.1. Arranging Objects

In the previous chapters you learned how to work with stations. You can start with a CAD layout and assign stations to the layout. You can also place stations in the manual layout workflow. You also know how to adjust overlapping stations and how to number stations.

The rest of Esko Plato's tools help you to complete your plate layout. You can organize your work by using Layers. You can transform objects on the plate using the Transform and Align windows. And you can create a layout using the Step-and-Repeat tools. These tools are the subject of this section.

- [Using Layers](#)
- [Using To Front/To Back](#)
- [Using Group/Ungroup](#)
- [Group into Grid](#)
- [Using the Transform Window](#)
- [Using the Align Window](#)

6.1.1. Using Layers

Esko Plato uses the concept of layers to build a file. Layers can be used in stations as well as in plates. You can use layers to organize the objects in your document. Esko Plato automatically creates some layers for you, but you can edit these layers or add your own. You can also create special layers, known as grid and markup layers. You can turn the display of layers on and off, lock layers, indicate that layer must not be printed and rearrange the order of layers. You can use multiple layers to keep similar objects together. For example, you might want to put all your marks on one layer and all stations on another. The [Layer Browser](#) helps you create and organize layers.

Objects in a station can be placed on layers to make them easier to organize and edit. Similarly, objects in the plate can be placed in different layers too.

You can have as many layers as you want. You can also name layers to help you remember what objects are on them and to make it easier for someone else to pick up where you left off. You can also rearrange the order of layers.

- [Types of layers](#)
- [Adding an object layer](#)
- [Deleting layers](#)
- [Changing the name of a layer](#)
- [Changing the order of layers](#)
- [Moving objects to a different layer](#)
- [Using a Grid Layer](#)
- [Using a markup grid layer](#)
- [Automatically created layers](#)

Types of layers

Esko Plato supports three types of layers: object layers, grid layers and markup grid layers. Object layers contain objects in the plate. Grid and markup grid layers contain grids and guides to help you position objects.

Adding an object layer

Even though Esko Plato makes some layers automatically, you will want to create your own layers. You can create your own layers and name them. To create a new object layer:

1. Open the Layer Browser by clicking 'Layer Browser' on the Window menu.
2. Click the Add Layer button +

A new layer appears. The new layer is selected and becomes the active layer.

Deleting layers

You may want to delete a previously created layer. When you delete a layer, the layer and all objects in that layer are deleted. You can delete any layer with the Delete Layer button. To delete a layer:

1. Open the Layer Browser by clicking 'Layer Browser' on the Window menu.
2. Select the layer you want to delete.
3. Click the Delete Layer button -

The selected layer is deleted. You cannot delete the last object layer in a document.

Changing the name of a layer

You can change the name of layers to make their contents easy to remember. To change the name of a layer:

1. Select the layer that you want to change.
2. Click the name of the layer. The name becomes an edit field.
3. Type in the name for the new layer.
4. Press [Enter] on the keyboard to save the new name.

Changing the order of layers

One of the nice things about using layers is that it makes it easy to move groups of objects in front of or behind others. You can change the order of the layers whenever you like and as often as you like. So, you can move objects on top of other temporarily and then easily move them behind again. To change the order of layers:

1. Click and drag the layer to a new location in the list. The location that the layer will be moved to is highlighted.
2. Release the mouse button.

The layer is moved to the new location in the list. Objects in the layer now appear on top of objects on lower layers.

Moving objects to a different layer

To move objects from one layer to another, you use the Cut and Paste commands. When you use the Cut (or Copy) commands and then use Paste, the object is placed on the same location. To move an object to a different layer, you cut the object from its current layer, select a new layer and paste it into the new layer. To move an object to a different layer:


1. Select the object(s) you want in a different layer.
2. On the Edit menu, click 'Cut'.
3. Make another layer the Active Layer by selecting it.
4. On the Edit menu, click 'Paste'.

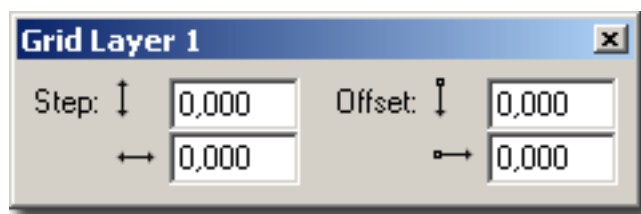
The object is placed in the new layer on the same location from which it was cut.

Using a Grid Layer

Grid layers contain evenly spaced guides that can be used to align objects. You can specify the spacing of a grid layer. You can also create multiple grid layers, each with a different spacing. You can then turn these layers on and off as you need them.

To create a grid layer:

1. Open the Layer Browser by clicking Layer Browser on the Window menu.
2. Click the Add Grid Layer button  The Grid dialog box appears:



3. Enter the spacing for the grid in the vertical and horizontal step fields. For a square grid, enter the same number in each step field. You do not have to enter the same number in both fields; entering different numbers creates a rectangular grid.

4. Enter the start position of the grid in the offset fields. The default start position of the grid is at the current origin. However, you can enter a different start position by entering values other than 0.
5. Click the close box on the Grid dialog box.

The grid layer is added to the Layer Browser. The grid appears in the document window as grey lines on the plate. Depending on the grid size and your current zoom level, it may be hard to see the grid. Like other layers, you can change the order of grid layers in the Layer Browser. If you want to work with the grid layer behind all your objects, move the grid layer to the back. If you want your grid layer to appear on top of all objects, move it to the top. You can also name a grid layer. If you want to work with multiple grid layers, it is a good idea to name them so that you can easily identify each one.

Using a markup grid layer

A markup grid layer is a layer with guides that you can place wherever you want. The guides can be placed numerically or interactively. The guides on markup grid layers are displayed in blue.

If you place a guide interactively, Esko Plato creates a markup grid automatically. If you want to create guides numerically, you need to create a markup grid first and then enter the locations of the guides that you want to create.

To create a guide interactively:

1. Click and drag a guide from either the vertical or horizontal ruler.
2. Release the mouse button to create the guide.

If the Layer Browser is open, you will see that Esko Plato has created a markup grid for this guide if there was none.

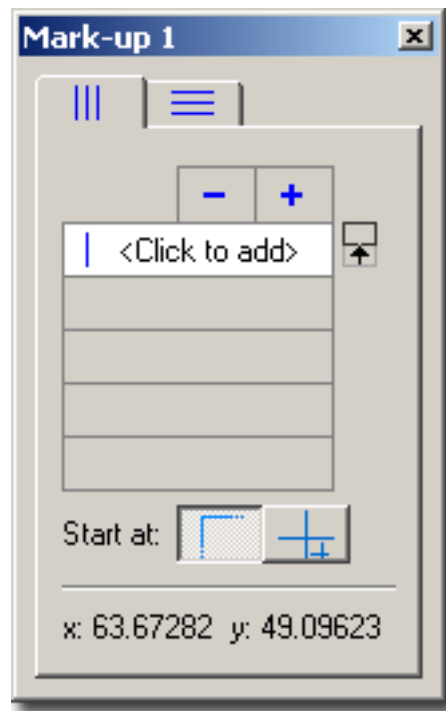
You can also create a markup grid numerically. You use the Mark-up dialog box to enter the exact location that you would like for the guide. You can also use the Mark-up dialog box to fine-tune the position of a guide that you created interactively. Finally, opening the Mark-up dialog box makes it possible to select, move and delete guides.

To create a markup grid numerically:

1. Open the Layer Browser by clicking Layer Browser on the Window menu.
2. Click the Add Markup Grid Layer button



The Mark-up dialog box appears:



3. Select the tab for vertical or horizontal guides based on the type of guide you want to add.
4. Enter the position for the guide in the field below.
5. Press [Enter] to add the guide.
6. Click the appropriate button to position the guide relative to the top-left corner of the sheet



, or relative to a crosshair



The guide you added is selected in the document window. When the Mark-up dialog box is open, you can select, move and delete the guides interactively from the document window. To delete a guide, select it and press [Delete].

Automatically created layers

Esko Plato can create some layers for you automatically. For example, a layer "CAD Layout" will be created when you import a CAD layout into the plate. All the crease and cut lines for the CAD layout are placed in this layer. A "Station" layer will also be created when you place or assign stations on the plate. All stations are placed in this layer.

You are not restricted to using these layers. They are created to help you organize your work. Other objects can be placed in these layers too. For instance, you can place marks in the Station layer.

6.1.2. Using To Front/To Back

Four commands on the Arrange menu help you order the objects in a layer. These are: To Front, Forward, Backward, and To Back.

To move an object to the top of all other objects on the current layer:

1. Select the object(s) that you want in front.
2. On the Arrange menu, click To Front.

To move an object one step up within the current layer:

1. Select the object(s) that you want to move forward.
2. On the Arrange menu, click Forward.

To move a object one step down within the current layer:

1. Select the object(s) that you want to move backward.
2. On the Arrange menu, click Backward.

To move an object to the bottom of all other objects on the current layer:

1. Select the object(s) that you want in the back.
2. On the Arrange menu, click To Back.

Remember that these commands move objects forward and backward in a layer. If you use these commands to bring an object to the front and there still may be objects in front of it, the front objects are located in a different layer.

6.1.3. Using Group/Ungroup

You can group individual objects into groups to manipulate and modify a group as a single object. You can also create a group that is composed of other groups, or a group that is a combination of individual objects and groups.

To group objects:

1. Select the objects that you want to group.
2. From the Arrange menu, click Group.

To ungroup objects:

1. Select the grouped object you want to ungroup.
2. From the Arrange menu, click Ungroup.

To Group an object into a Grid:

1. Select the objects that you want to group to a grid.
2. From the Arrange menu, click Group into Grid. Esko Plato will try to detect a grid pattern in the selected stations.

Group and Ungroup Tips

- Grouping objects will rearrange the objects automatically so that no other objects are situated in-between.
- Grouping objects in different layers will place the grouped object in a single layer.
- Ungrouping objects places the individual objects in the same layer, in the same order as previously in the group.

6.1.4. Group into Grid

Use 'Group into Grid' to sort and analyze the red selected stations for a grid based step-and-repeat pattern. Depending on the position of the stations, one or more grid blocks (see the Grid dialog box) will be generated.

6.1.5. Using the Transform Window

You can transform objects numerically with the Transform window. To transform object interactively, use the [Transform tool](#). The Transform window provides two features. First, the Transform window provides you with information about the currently selected object. This information includes the location, size, rotation and scaling of the selected object. Secondly, the Transform window allows you to numerically transform an object by changing this information.

You numerically transform objects by selecting them, choosing a current point if necessary, and entering specific values in the Transform window. To move an object to a specific location:

1. Select the object.
2. In the Transform window, select the current point that you want to locate.
3. Type a new location into the 'Location' fields.
4. Press [Enter] to make the change.

The current point of the selected object is moved to the vertical and horizontal location that you entered.

To enter a specific height and/or width for an object:

1. Select the object.
2. Type the height and/or width for the object in the Size fields.
3. Press [Enter] to make the change.

The selected object is scaled to the size that you entered for the height and/or width.

To rotate an object by a certain angle:

1. Select the object.
2. Type an angle for the object in the Angle field.
3. Press [Enter] to make the change.

The selected object is rotated to the angle that you entered. By default, rotations of objects occur around the center of the object. If you want the object to rotate around a different point, you need to move the point of origin. See [Moving the point of origin](#).

To scale an object as a percentage of current size:

1. Select the object.
2. Type a percentage in the vertical and/or horizontal Scale fields.
3. Press [Enter] to make the change.

The selected object is scaled to the percentage(s) that you entered into the Scale fields. You can scale an object proportionally by entering the same percentage in both fields. You can scale an object non-proportionally by entering a different value in the each field.

6.1.6. Using the Align Window

You use the Align window to align objects to the center of the sheet or to a single other object. Two of the functions of the Align window align objects to the center of the sheet. The other functions of the Align window align objects to one another.

Aligning objects to one another uses both red selection and green selection. A green selection is a "locked"; a red selection is moved to align with the green selection.

Like the Transform window, the Align window uses the die of the current selections, if there is one, to align the objects.

- [Aligning objects with the Align Window](#)
- [Aligning in the Station View](#)

Aligning objects with the Align Window

You align objects with the Align window by selecting objects and pressing the appropriate button. To align an object to the center of the sheet:

1. Select the object that you want to align.
2. Press the one or both of the 'Align to Sheet' buttons in the Align window.

Pressing both buttons one after the other aligns the object to the exact center of the sheet. Pressing only one of the buttons aligns the object in that direction only.

To align two objects by the same attribute:

1. Select the object that you want to align to.
2. Press [Insert] to make the selection green.
3. Select the object that you want to align.
4. Press the appropriate Align by Attribute button in the Align window.

Pressing more than one button after another performs multiple alignment commands. For example, pressing 'Align Middle' followed by 'Align Center' will align the exact center of the red selection to the exact center of the green selection.

To align two objects edge to edge:

1. Select the object that you want to align to.
2. Press [Insert] to make the selection green.
3. Select the object that you want to align.
4. Enter a Gap in the appropriate field, if desired.
5. Press the appropriate Align Edge to Edge button in the Align Window.

The Align Edge to Edge commands use the nearest opposite edge to determine which way to move the red selection. Before using these options, arrange the objects so that the edges that you are aligning are in the relationship that you want. You don't have to be entirely accurate (or even close), Esko Plato just needs to be able to clearly identify which edges that you want to align.

Align Window Tips

- The Align Window uses one object that is locked (the green selection) to align the other objects. It is a good idea to have the green selected object in the right location before starting with the Align options.
- After using the Align commands, you can still move the objects around the sheet and plate. But, if you want these objects to preserve their relationship with one another, you have to make sure that you select and move them together. A good way to maintain the alignment parameters is to use the Group command immediately after. Then the objects that you aligned will always be selected and moved as a single object.
- The Align commands also work with multiple selections. This makes it easy to align several objects at one time, with a single command.

Aligning in the Station View

The Align window also works in the Station View. However, the 2 centering features of Align do not center objects on the sheet, since a station has no sheet. Instead, these 2 options center an object on the current die. You should also remember that Align works with all types of objects in both the Sheet and the Station View. So, you can use Align to align a graphic or a SmartMark with the current die.

6.2. Creating a Repetition (Grid-based Step & Repeat)

You use Esko Plato's Step-and-Repeat command to repeat objects on the plate. You can set up the number of repeats and whether these repeats have gaps or gutters in between them. You can also add head turns and staggers to the repetition to create an entire layout. Step-and-Repeat can be used on any type of object, so you can repeat stations and Geometry Marks.

- [Repeating objects with the Grid dialog box](#)
- [Step-and-Repeat examples](#)
- [Step-and-Repeat with Stagger example](#)
- [Step-and-Repeat with Distribute example](#)
- [Two-sided plates](#)

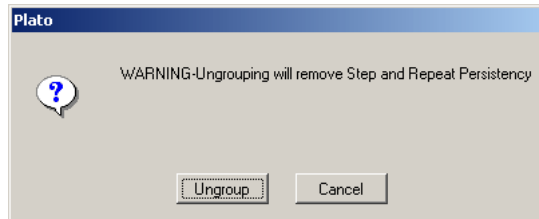
6.2.1. Repeating objects with the Grid dialog box

You step-and-repeat an object by double-clicking the grid number in the [Grids dialog box](#) and entering the parameters you want. The basic Step-and-Repeat command is described below. To see Step-and-Repeat examples, see the next section. To Step-and-Repeat a station:

1. Click 'Grids' on the Window menu to open the Grids dialog box.

2. Create a station on an empty grid line. Click 'Click to add' and select a station from the menu or click 'Add Station...' to browse to a new station.
3. Enter the number of repeats.
4. Enter a gap amount.

The result is a group of stations. This group carries the Step-and-Repeat settings that were used to create it. To modify the settings of the group, double-click the group to open the Grid Settings dialog box with the original parameter settings. When you ungroup a Step-and-Repeat group, these settings will be lost, the system will ask for a confirmation.



After ungrouping, the individual stations can be manipulated.

6.2.2. Step-and-Repeat examples

- [Example one](#)
- [Example two](#)
- [Example three](#)
- [Example four](#)

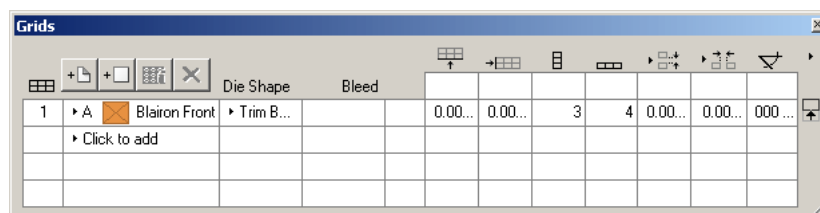
Example One

- Position: lower left corner at (0,0).

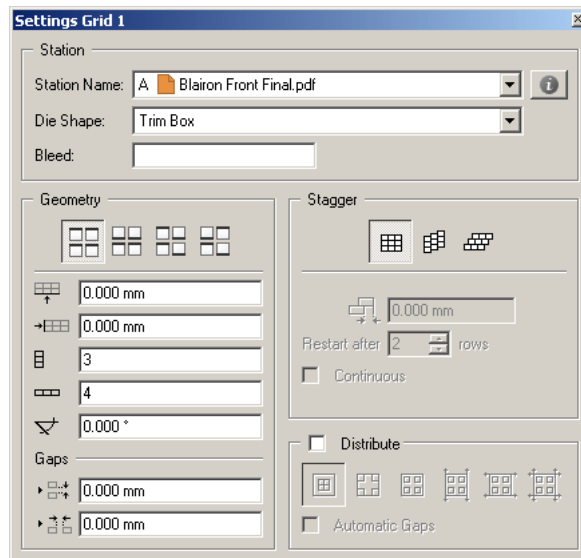
Tip:

Change the Origin (0,0) in the Edit menu > Preferences > Defaults.

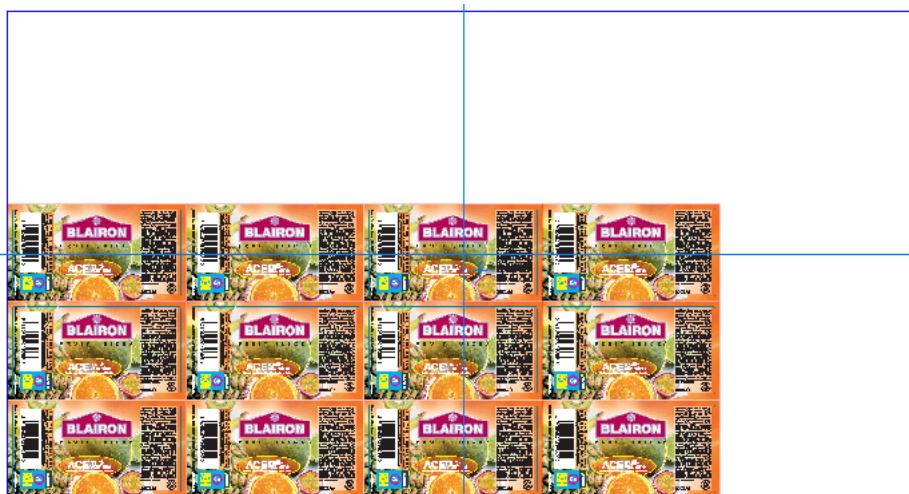
- Gap: no gaps between rows, no gap between columns
- Direction: Up and Right
- Head Turn: No Head Turn
- Number of Rows: 3
- Number of columns: 4



Select the Station and click the Info button. The Grid Settings dialog box appears.

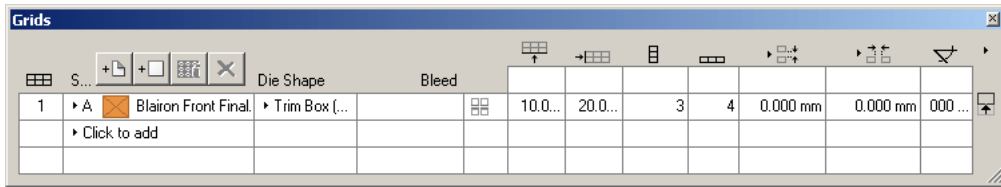


This is a basic step and repeat. A single station is repeated into a 3X4 grid. The grid has no gutters; the dies of each station abut one another. No head turn is used.

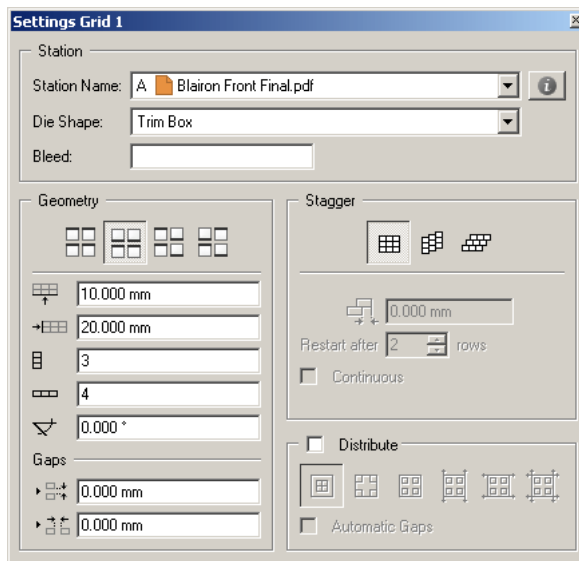


Example Two

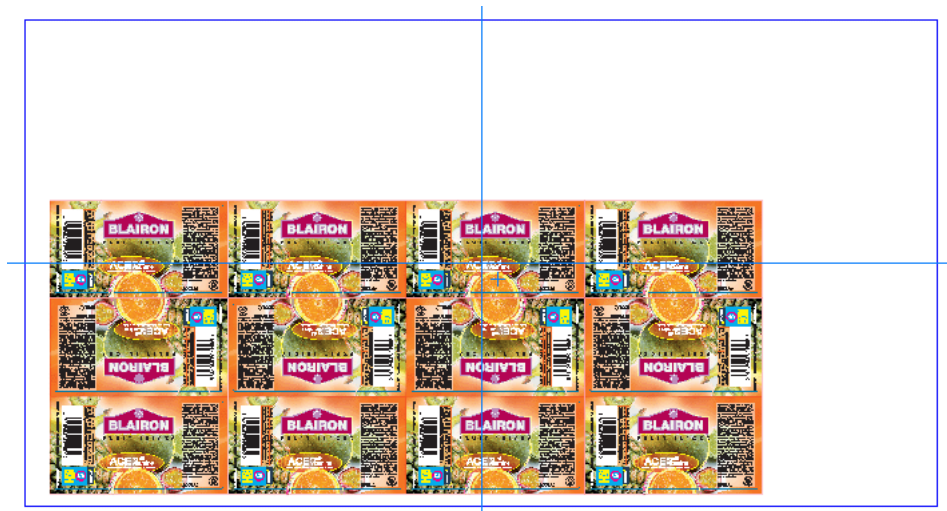
- Position: center at (10, 20)
- Gap: No gap between rows, no gap between columns
- Direction: Up and Right
- Head Turn: Head turn on row
- Number of rows: 3
- Number of Columns: 4



Select the Station and click the Info button. The Grid Settings dialog box appears.

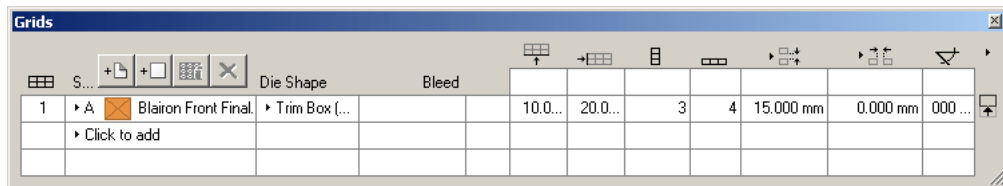


The dialog box positions the station prior to repeating. The station is repeated in a 3x4 grid. The grid has no gaps. A head turn is used on each row.

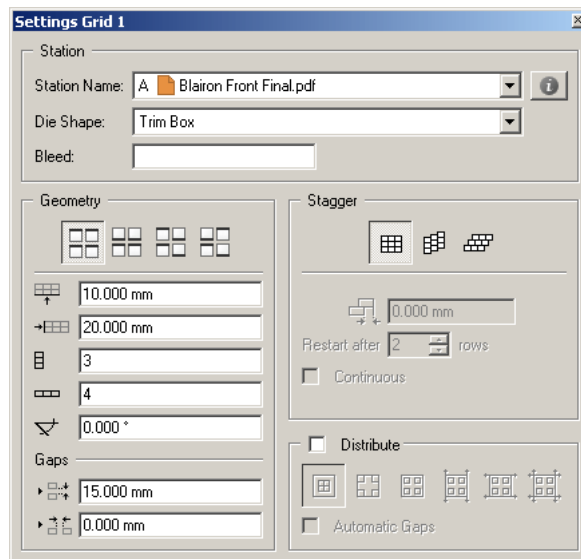


Example Three

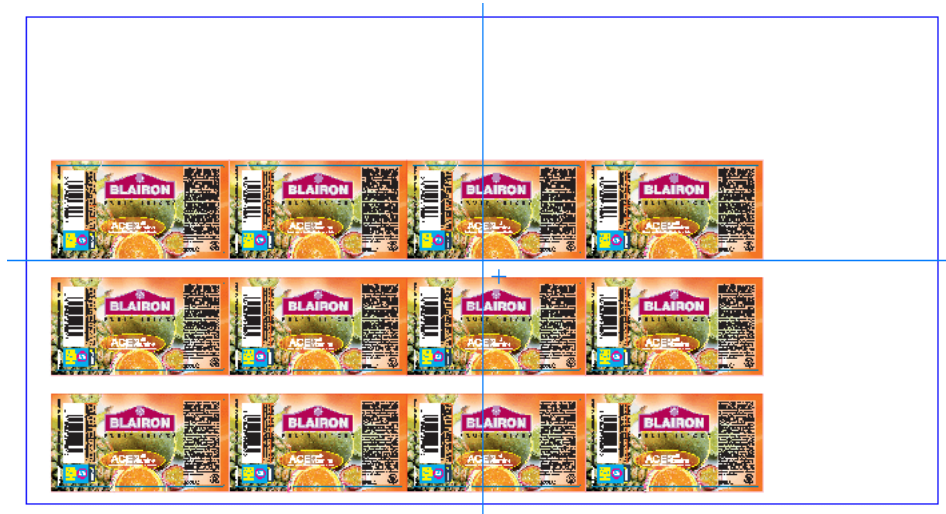
- Position: center at (10, 20)
- Gap: 15 mm gap between rows, no gap between columns
- Direction: Up and Right
- Head Turn: No head turn
- Number of rows: 3
- Number of Columns: 4



Select the Station and click the Info button. The Grid Settings dialog box appears.

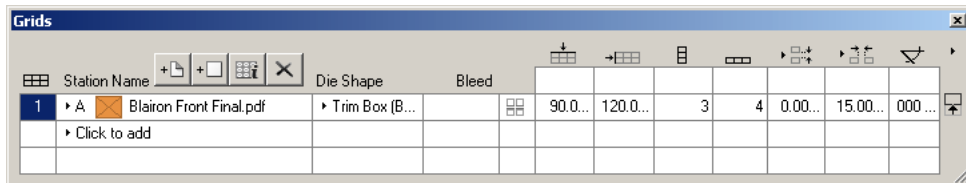


The dialog box positions the station prior to repeating. The station is repeated in a 3X4 grid. The grid has a 15 mm gap between rows; no gap between columns. No head turn is used.

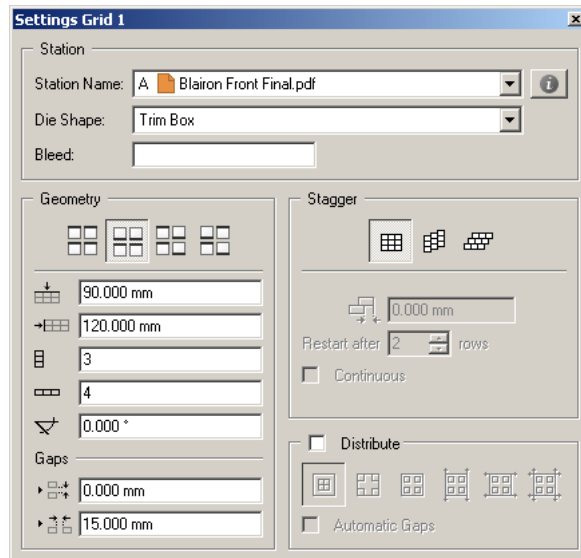


Example Four

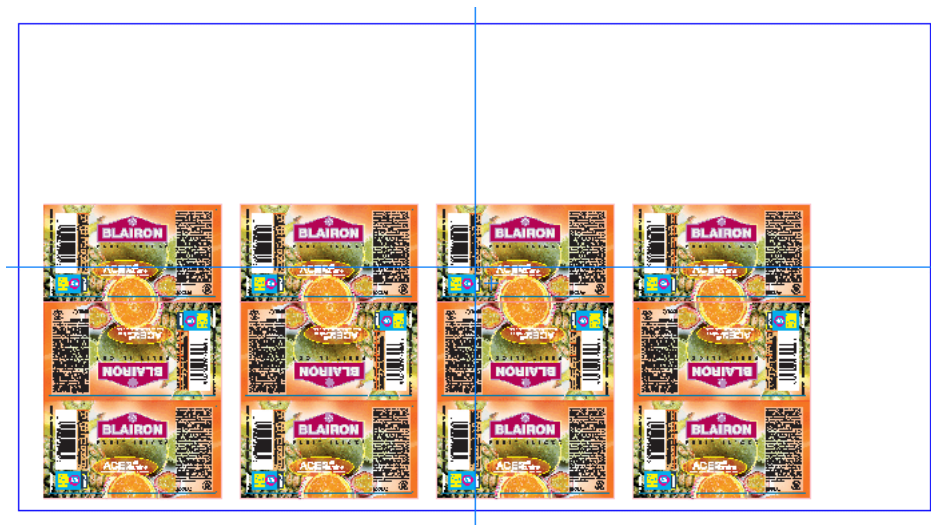
- Position: center at (90, 120)
- Gap: No gap between rows, 15 mm gap between columns
- Direction: Up and Right
- Head Turn: Head turn on row
- Number of rows: 3
- Number of Columns: 4



Select the Station and click the Info button. The Grid Settings dialog box appears.



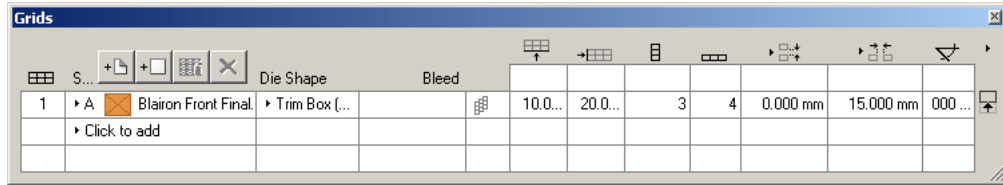
The dialog box positions the station prior to repeating. The station is repeated in a 3X4 grid. The grid has no gap between rows, a 15 mm gap between columns. A head turn is used on each row.



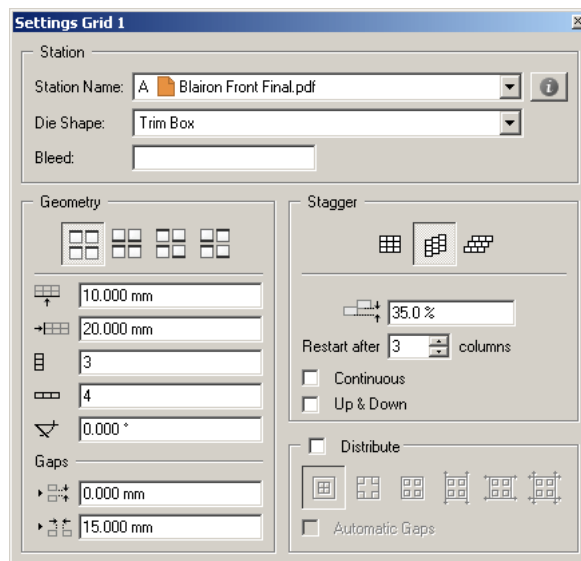
6.2.3. Step-and-Repeat with Stagger example

- Position: center at (10, 20)
- Gap: No gap between rows, 15 mm between columns
- Direction: Up and Right
- Head Turn: No Head turn
- Number of rows: 3
- Number of Columns: 4

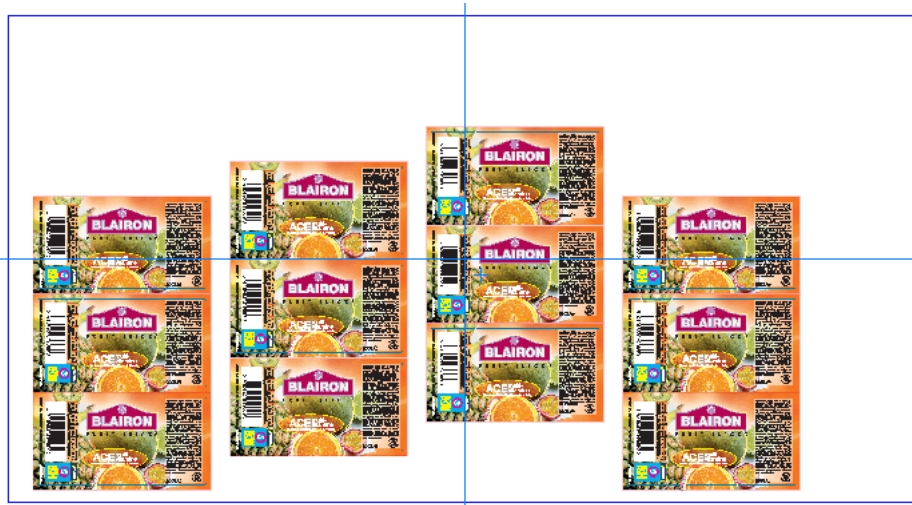
- Stagger: Stagger Columns 3 times
- Up & Down and Continuous staggering disabled
- Stagger amount: 35% of height



Select the Station and click the Info button. The Grid Settings dialog box appears.



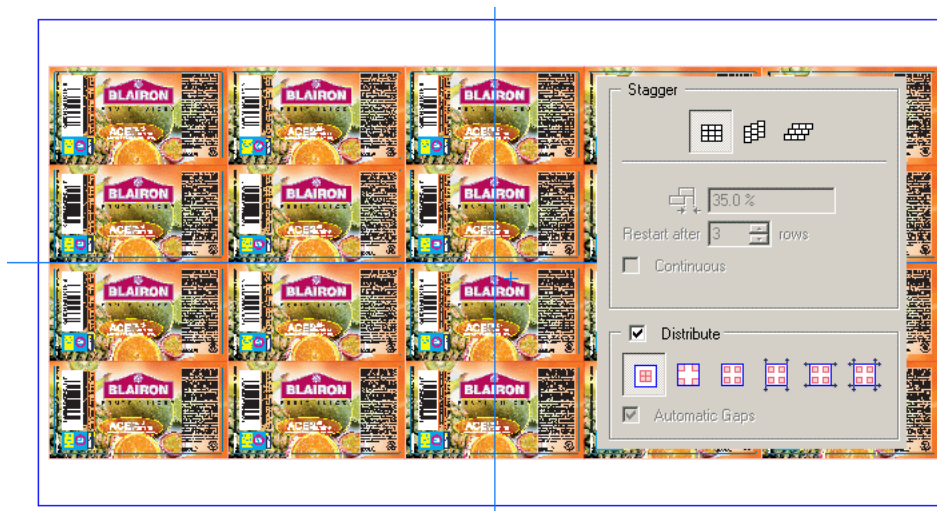
The dialog box positions the station prior to repeating. The station is repeated in a 3X4 grid. The grid has no gap between rows, a 15 mm gap between columns. No head turn is used.



6.2.4. Step-and-Repeat with Distribute examples

Example 1

Type: Single Cuts, no Gap, no Stagger.



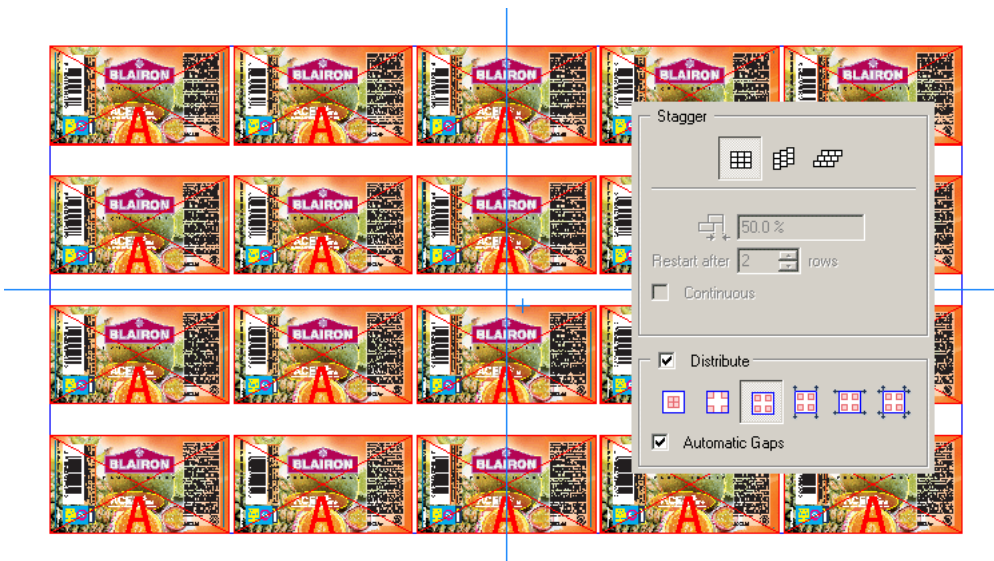
Example 2

Type: Distribute Gaps between Stations, no Gap, no Stagger.



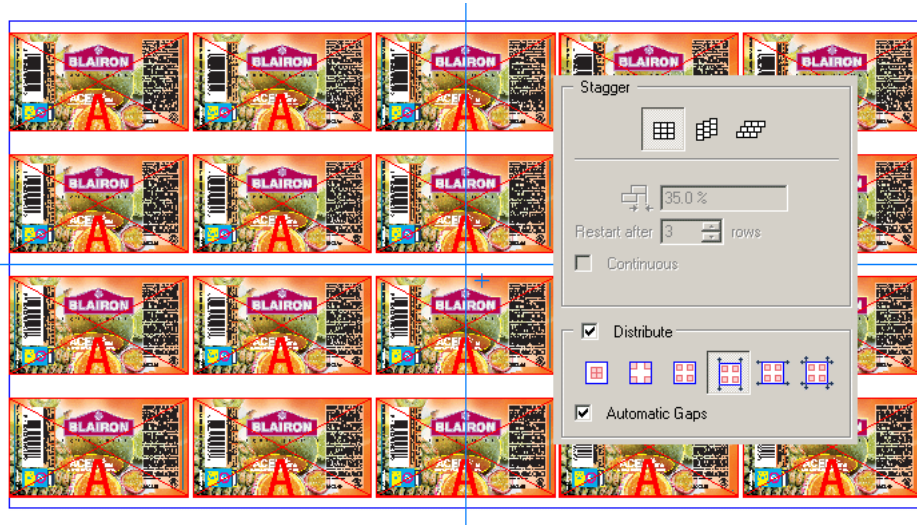
Example 3

Type: Distribute Gaps, no Gap, no Stagger.



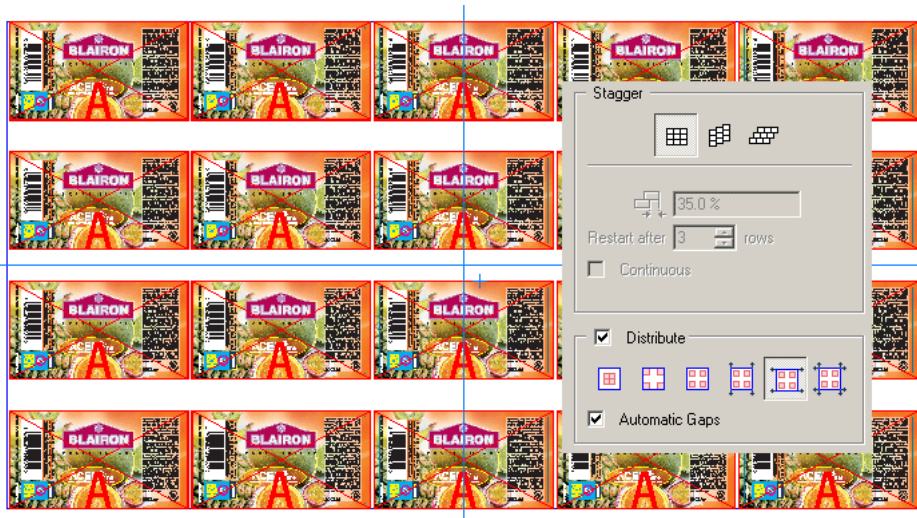
Example 4

Type: Continuous Around, no Gap, no Stagger.



Example 5

Type: Continuous Across, no Gap, no Stagger.



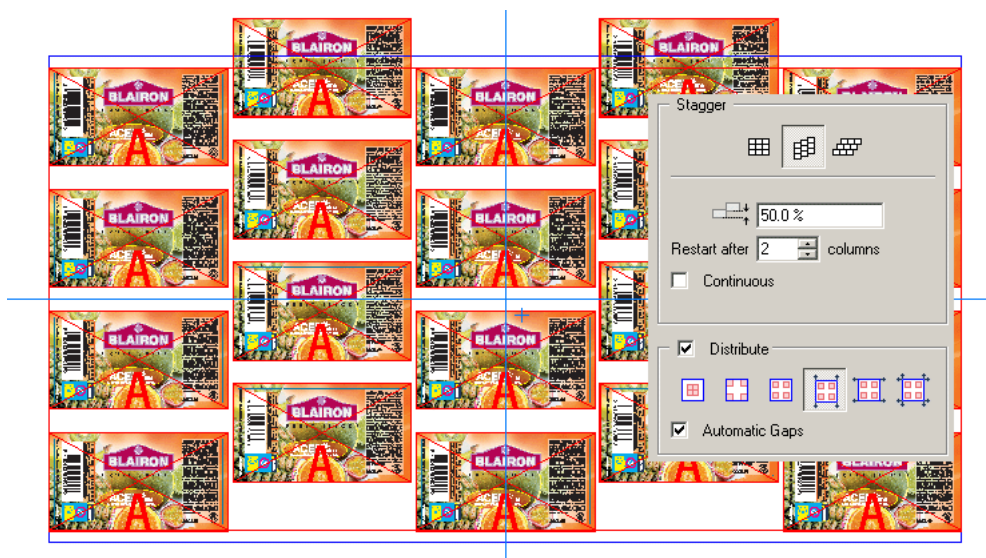
Example 6

Type: Continuous Across and Around, no Gap, no Stagger.



Example 7

Type: Continuous Around, no Gap, Stagger Columns 50%.



6.3. Two-sided Printing

- *Two-sided plates*

- [Two-sided stations](#)

6.3.1. Two-sided plates

Esko Plato can create the back of a plate layout from the front for two-sided printing. You use the **Turn Front to Back** and **Tumble Front to Back** commands on the Tools menu to create the back side.

The 'Turn Front to Back' command rotates the sheet around the vertical center of the sheet. With this command, the left edge of the sheet becomes the right edge of the sheet on the back. The 'Tumble Front to Back' command rotates the sheet around the horizontal center of the sheet. With this command, the top of the sheet becomes the bottom of the sheet on the back. When the back side of the plate is created, the position of the stations is mirrored for accurate front-to-back alignment. The back side of the stations is automatically inserted to create the printing on the back side. If you move a station on the front side, the back side will be updated automatically.

Any marks that you used on the front side of the plate are reapplied to the back side. When the marks are reapplied, their ink usage and mapping are updated based on the inks used on the back of the stations.

To create a back side from the front:

1. On the Tools menu, click 'Turn Front to Back' or 'Tumble Front to Back'.
2. The back side of the plate is automatically created. Plato's view changes to the back side of the plate.

Two-sided plate tips:

- Changes made to the positions of the stations on the front will be automatically applied to the corresponding stations on the back! This only applies to the station positions, all other modifications (SmartMarks, geometry marks, adjust mask on the stations) will not be re-applied on the back side. Please make sure the front side is as complete as possible before using the 'turn/tumble' front to back commands.
- The marks from the front side are reapplied on the back and the ink usage and mapping is updated to reflect differences in the inks used between the two sides. You may have to adjust the positioning of marks on the back side.
- If you have used the Adjust Masks tool to create new masks on the front side, these masks are included on the back side. If you do not want the same mask adjustments on the back, select the Reset Masks command and run Adjust Masks on the back side of the plate, if necessary.

6.3.2. Two-sided stations

Esko Plato can create the back of a plate layout from the front for two-sided printing. When Plato creates the back side of the plate, the back side of each station is inserted automatically. You need to create a two-sided station in the Station View to create a two-sided plate for printing.

To create a back side from the front:

1. Open the Station dialog box
2. Click the 'Add Station' icon that is located on top of the 'Back Side' column.

You complete the back side the same way that you did the front: by importing a graphic, creating a mask and adding marks. The die shape of the front is automatically mirrored to the back.

Tip:

- If the graphic that you are using is multi-page, the back side will be created with the subsequent page. If this is not the graphic that you want, delete it and import a different graphic for the back side.
- Because the back side is created from the front, make sure that your front is complete before using the Turn/Tumble Front to Back command. Changes to the front side will not be made automatically to the back.
- If you have used the Create Bleed Mask tool to create a mask on the front side of the station, this mask is included on the back side. If you do not want the same mask on the back, use the Create Bleed Mask tool to create a different mask for the back side of the station.

6.4. Managing Inks

Esko Plato has several tools that help you manage the inks in your document. You can use these tools to display the individual separations of the document, to add, modify or map inks, and to calculate the ink coverage of a document.

**Caution:**

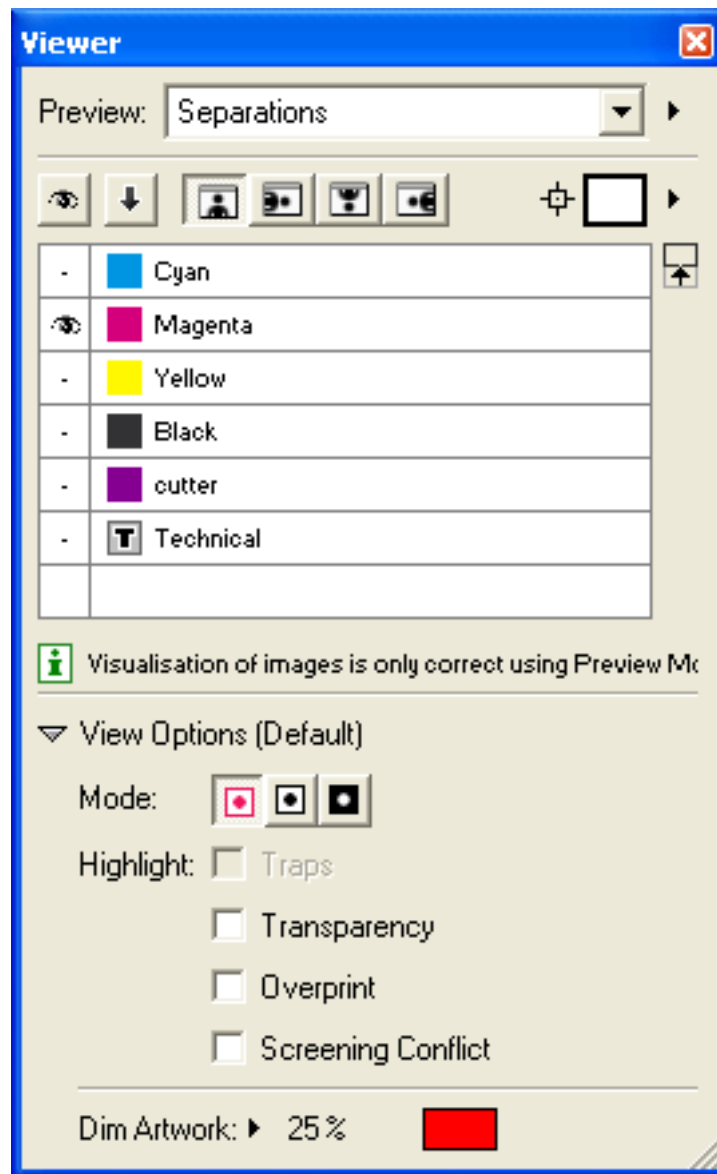
Note that it is impossible to edit or delete inks in Station View mode.

- [ViewX](#)
- [Working with inks](#)
- [Using ink libraries](#)
- [Using ink mapping](#)
- [Using external file ink mapping](#)
- [Calculating ink usage](#)

6.4.1. Viewer

Use **Viewer** to display the separations of a document. You can view individual separations or multiple separations together. To view the separations of a document:

1. From the View menu, click '**Viewer**'. The **Viewer** dialog box appears:



2. Click the Show/Hide buttons to toggle on or off individual separations. Click the expand icon to show all the separations and options.

Please refer to the chapter *'Viewer'* for more information on the **Viewer** options.



6.4.2. Working with inks

Use the Inks window to add and edit inks in a document. You can add inks, delete inks, set the screen ruling, angle and dot shape of the inks, and set the order of the inks in your document.

See also *Inks...* on page 364

- [Adding inks](#)
- [Changing the order of the inks in a document](#)

- [Changing the screen parameters of an ink](#)
- [Cleaning up unused inks in a document](#)
- [Replace Ink](#) on page 132
- [Printing Method](#) on page 135

Note: Plato supports PantoneLIVE ink books. In the Replace Inks, an icon will show if an ink or inkbook is (from) a Color Engine Ink Book  or a PantoneLIVE ink book .

Adding inks

You add and delete inks in the Inks dialog. You can have any number of inks, of any type in a document. There are three groups of inks: Process Inks, PANTONE Inks and Designer Inks. The way in which you add an ink depends on the ink that you want to add.

To add a Process ink:

1. Click in the ink name field for the empty ink.
2. Type the letter for the process ink you want to add. Type C for Cyan, M for Magenta, Y for Yellow and K for Black.
3. Press [Enter].

To add a PANTONE ink:

1. Choose **Inks > Open Ink Book...** and select the appropriate ink book from the list.
2. Locate the PANTONE ink that you would like to add.
3. Drag the PANTONE ink onto the swatch of the empty ink.

To add a Designer ink:

1. Choose **Inks > Open Ink Book...** and select the appropriate ink book from the list.
2. Locate the Designer ink that you would like to add.
3. Drag the Designer ink onto the swatch of the empty ink.

You cannot delete inks that are used in external references, like stations. However, you can map an ink in an external reference to another ink in the job. You can also choose not to print the ink.

Changing the order of the inks in a document

The order of the inks in the document determines how an ink is displayed and the order in which the separations are printed. For these reasons, you may want the inks in a particular order.

To change the order of the inks:

1. Click and drag the ink to a new location.
2. Release the mouse button.

To set the order of the inks based on darkness:

1. Make sure that the Inks window contains all the inks you want in your document.
2. On the fly-out menu, click 'Sort Light to Dark...' or 'Sort Dark to Light...'. The inks are reordered based on the option you choose.

Changing the screen parameters of an ink

You can set the screening parameters for the inks in your document in the Inks dialog box. These parameters include screen ruling, screen angle and dot shape. To set the screen ruling for an ink:

1. Click the screen ruling field for an ink. The contents of the field are highlighted.
2. Enter a new screen ruling.
3. Press [Enter].

To set the screen angle for an ink:

1. Click the screen angle field for an ink. The contents of the field are highlighted.
2. Enter a new screen angle.
3. Press [Enter].

To set the dot shape for an ink:

1. Click the screen dot shape field for an ink. The contents of the field are highlighted.
2. Enter a new screen dot shape.
3. Press [Enter].

Cleaning up unused inks in a document

Your document may contain inks that are no longer being used. For example, you may have replaced stations on the plate with stations that use different inks. The old inks from the previous stations may still be contained in the document even though they are no longer being used.

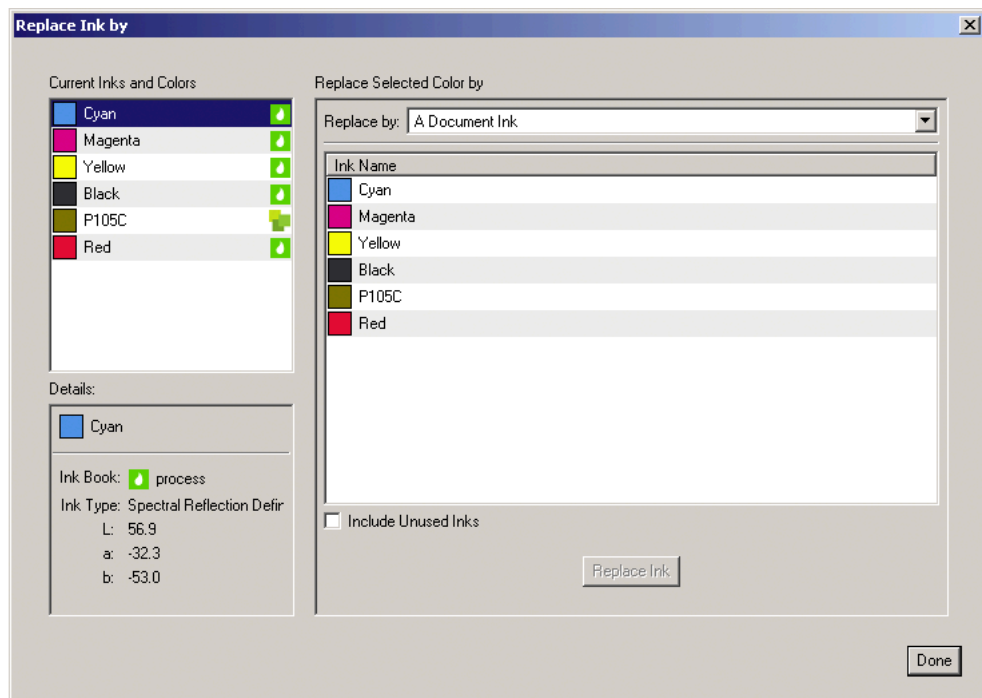
To clean up unused inks:

1. Open the Inks dialog box.
2. On the fly-out menu, click 'Remove Unused Colors and Inks'.

Replace Ink

To replace an Ink:

1. Open the **Replace Ink by** dialog by selecting **Replace Ink...** in the fly-out menu in the **Inks** dialog.



2. In the **Current Inks and Colors**, select the Ink you want to replace.

Inks defined in an Ink book will have a droplet icon next to their name

The **Details** window will show extra information on the selected ink.

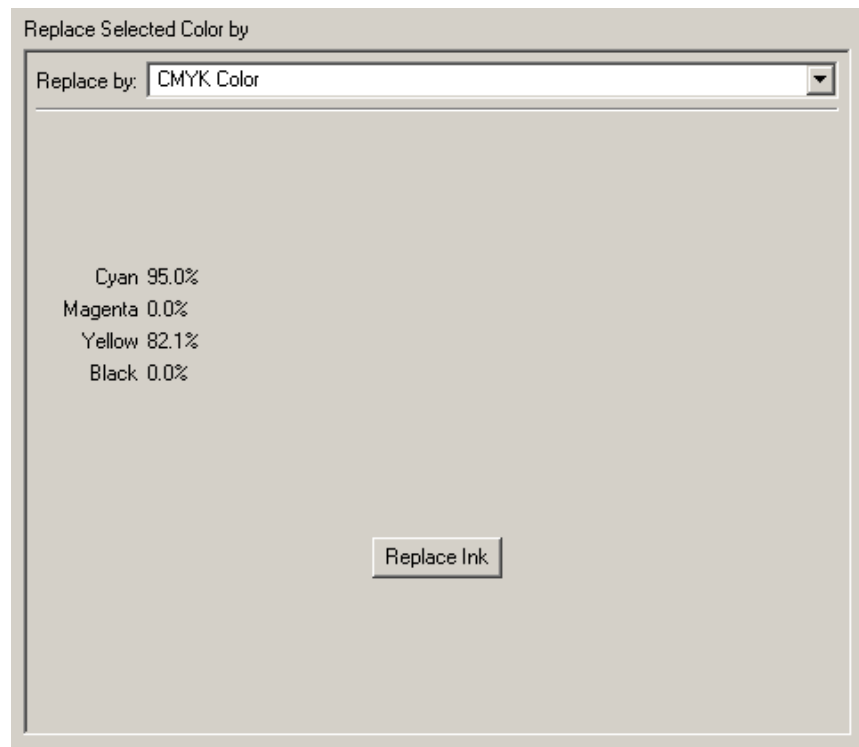
3. Define the **Replace by** type:

- **Replace by A Document Ink** allows to select an Ink from the document inks. By enabling **Include Unused Inks**, all spot color inks will be available, even the inks that aren't used in the job.
- **Replace by CMYK Color** allows to convert the selected ink into its CMYK equivalent. The CMYK values that will be used are shown underneath.

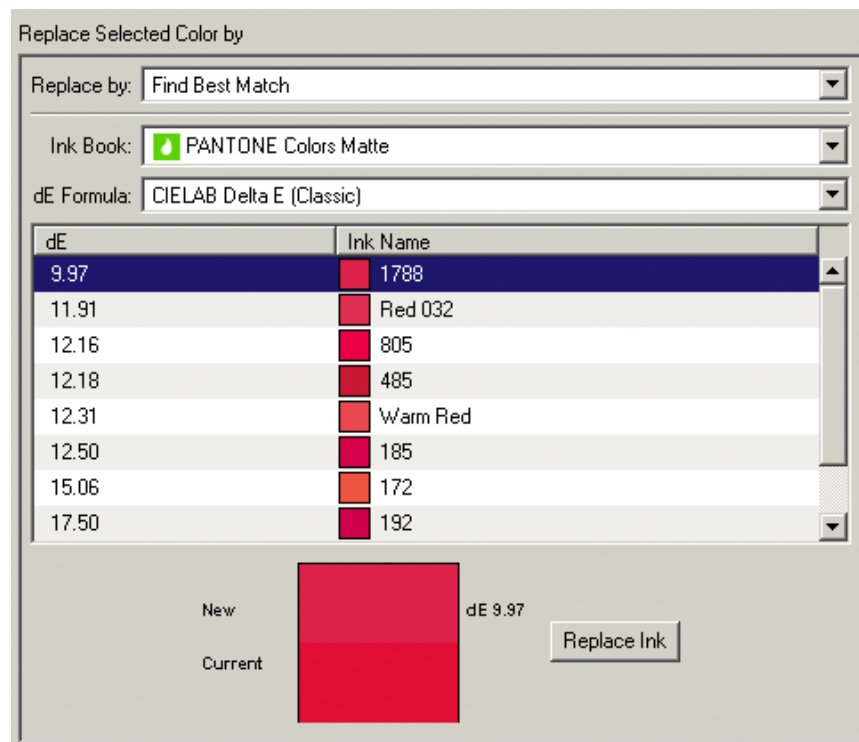
Note:

When converting to CMYK:

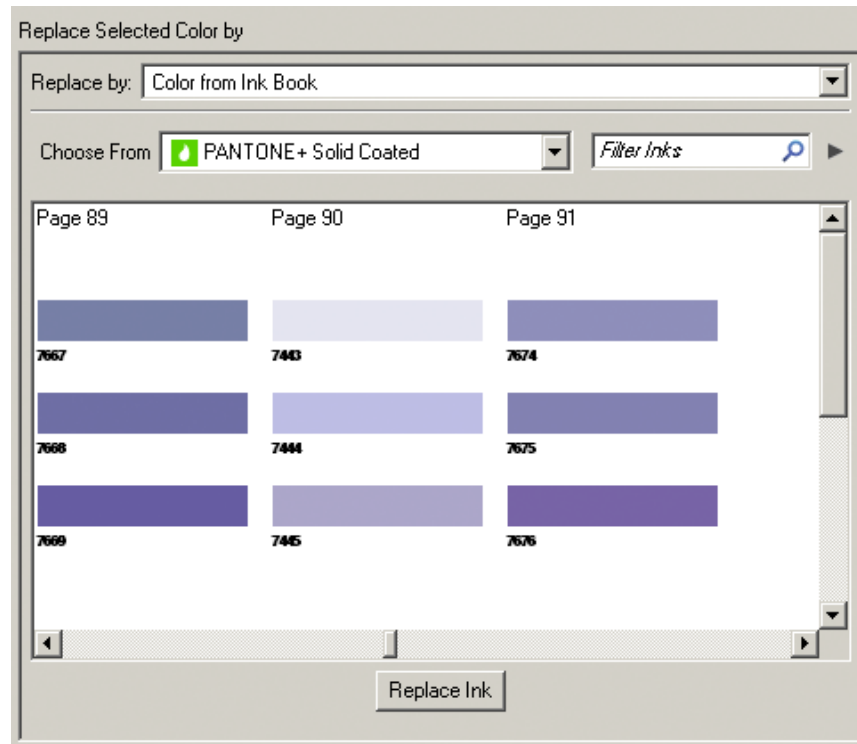
- inks from the document will be replaced by CMYK
- Inks from linked CT files will be planemapped to CMYK paint
- inks from linked PDF files are NOT converted, as this is not supported.



- **Find Best Match** will scan the selected Ink Book, and list the inks with the lowest dE compared to the ink you want to replace. At the bottom, the New and the Current ink are shown side by side.



- **Replace by Color from Ink Book** allows to replace the ink by a new ink from an Ink book.



4. Click the **Replace Ink** button.

Note:

If one or more objects can't be converted, a description of the problem will appear in the Messages dialog, and the converted ink will not disappear from the document.

Printing Method

You can set the **Printing Method** for every .

The **Printing Method** information can be used in SmartNames, or for Automation Engine task that handle the printing method.

By default, the Printing Method column is hidden. in the fly-out menu you can make it visible.

You can select one of the predefined printing methods, or create a new printing method.

Presets

As from version 12.1.2, when connected to an Automation Engine Server, the list of Printing Methods is loaded from the Automation Engine server.

Note: This implies that Printing Methods saved in earlier versions of are no longer visible and will need to be redefined.

Printing Methods can be saved in the Automation Engine Configure tool, or in by selecting **Save to presets**

6.4.3. Using ink libraries

Esko Plato uses a number of default ink libraries: the PANTONE libraries and the Designer Book. Depending on your configuration, you may have additional ink libraries available. Select the ink book you want to use by clicking **Inks > Open Ink Book...**

The PANTONE libraries are designed to look like the pages of the PANTONE formula guide. Each column of inks represents one page in the guide. You can scroll through the inks using the scroll bar at the bottom. You can also search for a particular ink by typing its number (or name) in the search field at the bottom.

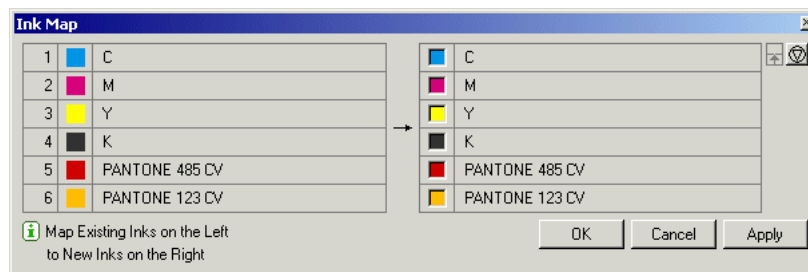
Non-standard (designer) ink books contain inks that are defined by you. Designer inks can be created in two ways: they may be included in an imported graphic or you may define them yourself by using a PANTONE ink. If you open or import a file that contains an unknown designer ink, an information window will appear. In this case the file contains an ink that currently is not in any of Esko Plato's ink libraries. You can add this ink to an ink book so that in the future Esko Plato will recognize this ink. To add an unknown ink to an ink book:

1. Choose **Inks > Open Ink Book...** and select the ink book to which you want to add the ink.
2. Click the lock in the bottom-right corner to "unlock" the library so that you can add a new ink.
3. Click and drag the ink swatch (not the name) of the unknown ink from the Inks dialog box to any of the empty ink patches in the ink book.

6.4.4. Using ink mapping

Your plate may use inks that need to be changed to other inks. For example, your plate contains the ink "Logo Red" and that ink should be "PANTONE 485 CV". You use ink mapping to "map" one ink to a different ink.

You can change ink mapping in the Ink Map dialog box by selecting *'Ink Map'* on the Inks menu. The Ink Map dialog box appears:



The Ink Map dialog box shows the current ink mapping of your document. Before any ink mapping has occurred, all the inks are mapped to themselves, as shown above.

On the left are the inks in your document; the existing inks. On the right are the new inks that you want to use in place of the existing inks. To map an ink, you replace the new ink on the right across from the existing ink.

To map an ink to a new ink:

1. On the Inks menu, click 'Ink Mapping...'
2. Open the ink library that contains the new ink that you want to use.
3. Locate the new ink in the library.
4. Click and drag the color swatch (not the name) for the new ink into the new ink swatch that is across from the ink you want to map.
5. Click 'Apply' to make the change.

Clicking 'Apply' applies the ink map but leaves the Ink Map window open so that you can make more changes. Clicking 'OK' applies the ink map change and closes the Ink Map window. If you click 'Cancel', any changes that you have made since the last Apply are deleted and the Ink Mapping dialog box closes.

You might want to reset the ink mapping table before clicking 'Apply'. This can be done by using the reset button.

Ink mapping will perform ink mapping on all objects on your plate - except for SmartMarks.

- Graphics (external files) will be mapped.
- Geometry marks will be mapped.
- SmartMarks will be regenerated after ink mapping has been applied (to reflect the new ink set).

Ink Map Tips

- You will only see the ink change color if the display mode is set to Medium or High. (The Display Image mode uses the display image contained in the file, not the actual data.)
- After ink mapping, the new inks on the right are the inks that are shown in other windows and dialogs. The Inks window, the View by Ink window and the printing dialog boxes now will show only the inks that appear on the right side of the Ink Map window.
- You can map more than one ink to the same ink. For example, you can map both "Logo Red" and "Magenta" to "PANTONE 485 CV".
- If you have mapped an ink previously and now need to remap it to another ink, you can simply replace the new ink on the right with another new ink.
- The Ink Map window does not show ink mapping used by SmartMarks. SmartMarks automatically map and manage their own inks.

Examples of technical printing problems where Job Ink Mapping brings a solution:

- You have a file where the 4 process inks have a screen ruling of 120 lpi. After placing a PostScript file, you notice that the DTP operator has defined the 4 process inks with a screen ruling of 150 lpi. The result is that you have 2 x 4 process inks. With Job Ink Mapping, you can easily map the screen ruling of 150 lpi to 120 lpi.

- You have a file where the 4 process inks have a screen ruling of 120 lpi. You place a file externally, with screen rulings of 150 lpi. With Job Ink Mapping, you can map all the inks of the external file to 120 lpi, without changing the inks of the externally placed file.

6.4.5. Using external file ink mapping

External file ink mapping is available in the Station View. It allows you to map inks for the graphics that are referenced externally.

6.4.6. Calculating ink usage

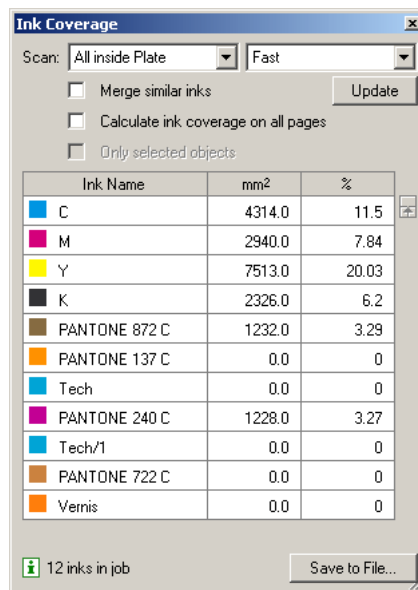
When your plate is complete, Esko Plato has all the information that is needed to calculate its ink usage. You see this information in the *Ink Coverage dialog box*.

- [Using Ink Coverage](#)
- [Saving ink coverage to a file](#)
- [Ink Coverage in the Station View](#)

Using Ink Coverage

To calculate ink coverage:

1. On the Inks menu, click 'Ink Coverage'.
2. Select the Scan Options that you want to use.
3. Click 'Scan'. The Ink Coverage window shows the ink usage for each ink in the document:

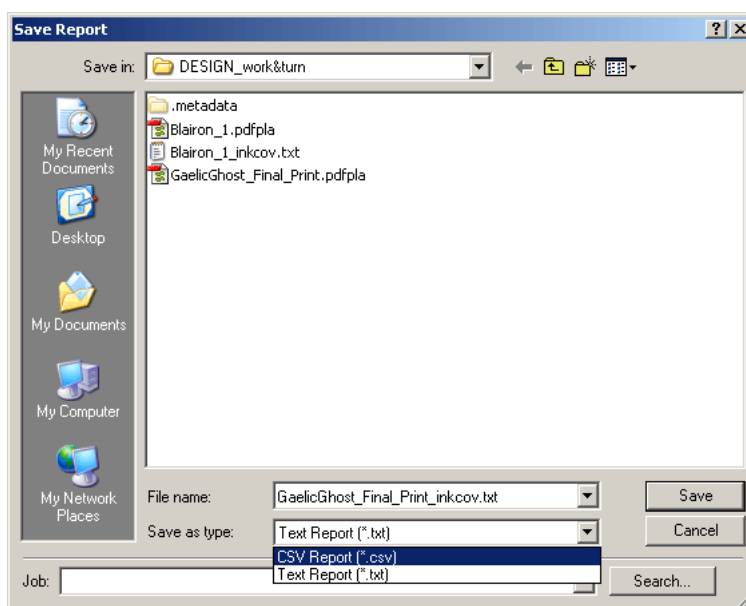


The inks in your document are listed at the bottom of the window. The amount of ink used is provided in the current units and as a percentage of the entire document. If you make changes to the document, these changes may affect the amount of ink used. To update the calculations, click 'Update' in the Ink Coverage window. Esko Plato will scan the document again and update the ink coverage.

Saving ink coverage to a file

You can save the results of the Ink Coverage calculation to a file that can be used later. The file that you save can be used by others to estimate and order ink for printing. You save the ink coverage results in a Report. To save the calculations as a report:

1. Click 'Save To File' in the Ink Coverage window. The Save Report dialog box appears:



2. Select the location where you would like to save the file.
3. Type a name in the File name field.
4. Select the type of report that you would like to save from the pull down list. You can save the file as either a Text Report or a CSV Report. Both types of files can be opened in any text editor. In addition, the CSV Report file can be opened in a spreadsheet application.
5. Click 'Save'.

Ink Coverage in the Station View

The Ink Coverage dialog box in Esko Plato calculates the ink usage for either the sheet or the plate. In the Station View, there is no plate or sheet; instead the options in the Ink Coverage dialog box are 'All Inside Die bounding box' and 'All'.

'All Inside Die bounding box' calculates ink usage inside the die only. Objects placed outside the die are not considered.

The option 'All' calculates ink usage for everything in the station.

6.5. Trapping

Note:

This menu will only be enabled in the Station View.

Trapping creates overlapping areas to compensate for possible misregistration between the different inks on the printed result. It is used whenever two adjoining colors contain different inks in such a way that misregistration during printing results in a white gap or an undesirable line of a different color at the border of the two colors.

There are two ways of looking at trapping:

- Spread trapping is used to trap a lighter area into a bigger darker area. The light object overprints and spreads into the darker background of the darker object.
- Choke trapping is used to trap the lighter into a smaller darker area. The light object overprints the darker area and chokes it.

Trapping can be applied both manually and automatically:

- With PowerTrapper, the user selects (in green) the objects that may need trapping. The system scans these selected objects, indicates where trapping is needed and creates the trapping areas. Here, the Trapping is executed automatically. See [PowerTrapper](#) on page 141
- [Trap Red under Green...](#) is the manual trapping method. The user defines which contours need trapping.

Note:

The trapping areas are placed in a separate 'Trap' layer. This way, you can easily remove the trapping area, without affecting your job.

This trap layer is part of the station definition and is thus embedded in the plate file. Note that the external referenced graphics is not altered! In case you want to reuse trapped stations in other plates, you might want to save your station in a (PDF)STA file (which will also contain the trapping layer).

Another type of trapping is [Keep Red Away from Green...](#), in which case the system introduces an extra distance between two adjoining colors in such a way that when printed, the adjoining colors do not overlap or mix. This is necessary for certain printing processes, in which the nature of the inks or the printing surface cause an unwanted mixing of adjoining inks.

Together with the automatic trapping, you can also define the settings for Rich Black and White Underprint.

- [Rich Black](#) creates a separate (choked) layer on top of the black object with the ink that you defined (by default Cyan), to obtain a darker black.
- [White Underprint](#) places a background under your design, when working with transparent or metallic material. The default ink is Transparent white, but you can select any ink of your job. When printing Yellow on a metal can, for example, the result of the combination will have a goldish appearance. To avoid this, apply White Underprint.

6.5.1. PowerTrapper

The Trap Dialog

The **Trap** dialog is where you define all your trapping settings.

1. Choose your trapping settings in the **Distance & Direction**, **Color & Shape**, and **Processing** tabs.
2. If needed, define trapping rules.
3. Save your trapping settings as a trap preset to be able to reuse them on another document.

Note: The Trap Preset List contains 6 default presets.

Distance and Direction Settings

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 Classic 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 (e.g. metal beverage cans) where overprinting of inks is not allowed as the inks would contaminate each other on the blanket.

Trapping Distance

Enter the width you want to give your traps.

The default trap width is 0.2 mm (or equivalent in your chosen unit).

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 will be 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.

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.

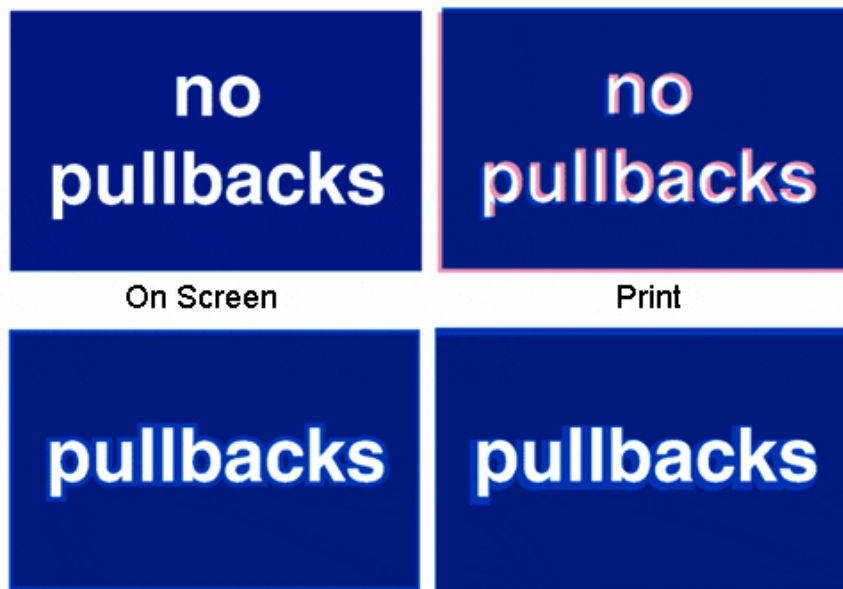
Pull Back

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.

Ink pull back is also known as “Cutbacks” or “Keepaways”.

The example below shows an object with 100% dark blue and 50% magenta on an empty background. 50% magenta is pulled back and a fine line of 100% dark blue remains. The color that remains (dark blue) is the **pull back ink**.



- **Automatic** (default): this creates pull backs on rich black or rich colors, when the color left in the pull back area is not very different from the composed color.

In the example below, the left color is composed of 100% purple ink and 20% cyan. The darkest ink (purple) is not very different from the composed color, so using the **Automatic** pull back option will create a pull back.

The right color is composed of 100% dark red ink and 20% blue ink. The darkest ink (dark red) is very different from the composed color, so using the **Automatic** pull back option will **not** create a pull back.



- **Only Pull Back Rich Black:** this only creates pull backs on rich black.
- **Do Not Pull Back:** this never creates any pull back.

Advanced Settings

Click the triangle beside **Advanced Settings** to show the advanced settings.

Trapping Distance

If you want to use a different distance when trapping into black, spot colors or images or when creating pull backs, enable the corresponding option and enter the trapping distance to use.

Image Trapping

If you don't want images to be trapped with each other, disable **Trap images to images**.

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 **Pull back images and gradients** if you want to create pull backs on images and gradients.

Trap decision

- By default, PowerTrapper Classic uses the color luminance (**Use Lightness**) to define the trapping direction, except for opaque inks, for which the Inks order is used.
 - trap direction: from brightest to darkest
- If the **Treat spot colors as opaque** option is chosen, color luminance is used only for CMYK. For opaque and spot inks, the trap direction is determined by the ink sequence.
 - trap direction for spot colors only: towards the separation listed the lowest in the Separations palette darkest
 - trap direction goes from the process to the spot color if the spot color is below the process color in the Separations list
 - trap direction if the spot color is above the process color: from brightest to darkest
- By selecting **Use separation order**, the trap direction is determined by the ink sequence for all inks, including CMYK.
 - trap direction: towards the separation listed the lowest in the Separations palette darkest
- **Use reverse separation order** does the same, using the reversed ink sequence.
 - trap direction: towards the separation listed the highest in the Separations palette darkest

Note: Using the Separation Order option will give the same result as changing all inks (including CMYK) to Opaque.

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.

An example:



During trapping, this example will generate the following color pairs:

By default (**Automatic**), this color pair list results in the following traps:



If the parameter is set to **Never on Dark Areas**, this will result into:



If the parameter is set to **Never**, this will result into:



Color and Shape Settings

Trap Color Intensity

By default, PowerTrapper Classic 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.

Image Trap Color

Choose how to trap your images:

- 1. Use Original Image Data** (default): when spreading an image, PowerTrapper Classic uses the underlying image data (when the image is clipped) to create the trap (see example 1 below).



Note: When there is no underlying image data (the image isn't clipped), it will create an empty trap.

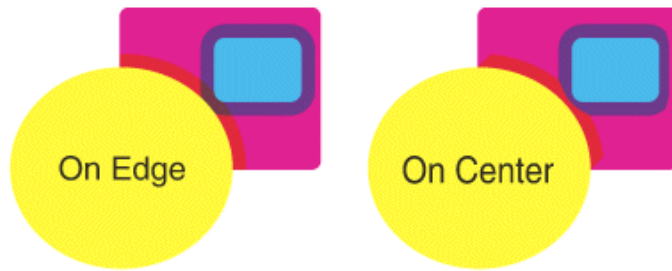
- 2. Extend Image Data:** PowerTrapper Classic will automatically extrapolate the image along its clipping path edge to ensure the trap looks as expected even if there is no clipped image data outside the visible image (see example 2 below).
- 3. Use an Approximate Flat Color:** Instead of using existing or extrapolated image data in the trap, PowerTrapper Classic will use a solid color. A suitable color is determined by averaging the image's colors along its boundary with the object to trap (see example 3 below).



Truncate Traps



PowerTrapper Classic can truncate the trap so that it doesn't 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 Classic 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.



End Caps


This option specifies how to shape the ends of an open trap.

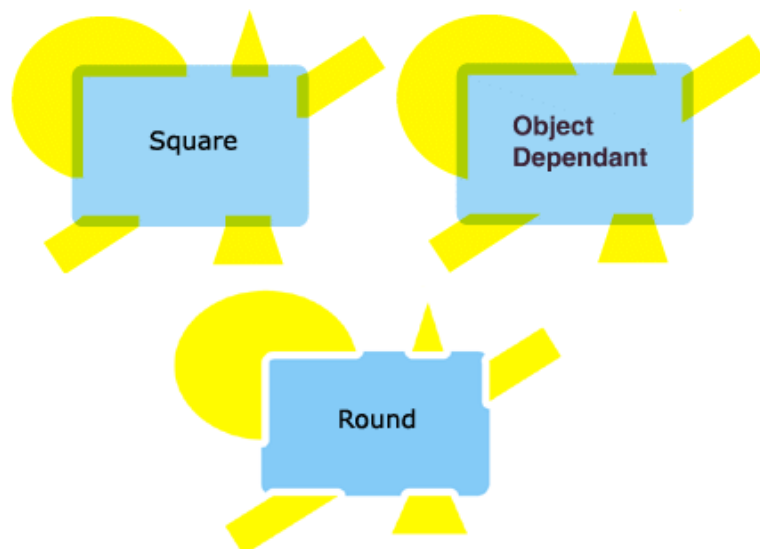
- **Square**  (default): this option ends the trap at right angle to the adjoining object.
- **Round** : this rounds the ends of the trap. This option is typically only used in combination with white knockout (reverse trapping).



Attention:




We recommend not to use **Round End Caps** together with **Truncate Traps On Center**, as this can generate some artefacts (the round caps will be truncated).

- **Object Dependent** : with this option, the trap is a logical continuation of the contour of the spread object.



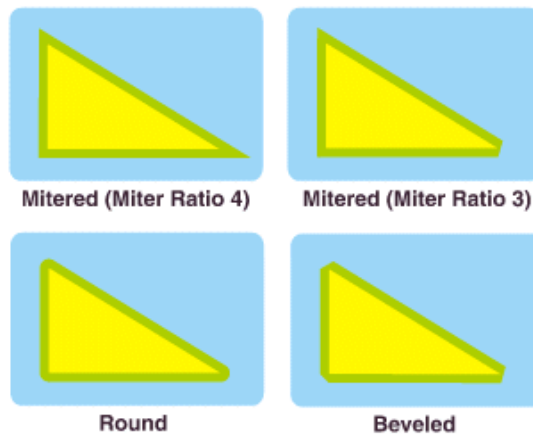
Trap Corners

This option allows you to change how the traps' sharp corners will be handled.

- **Round**  (default): a round cap will be placed at all corners.
- **Beveled**  : this will cut 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 will be cut off (beveled). If it is less than 4 times the **Trapping Distance**, the corner will be left as it is.



Advanced Settings

Click the triangle beside **Advanced Settings** 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**). For more information about truncation modes, see [Truncate Traps](#).

Processing Settings

Color Pairs

PowerTrapper Classic 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.
- **Same decision for all hits**: this traps all hits of the same color pair in the same direction.

- **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 default size is 1 mm.

Restrictions

If the **Respect existing traps** is enabled, traps in the selection will be kept, and no trapping is performed on these locations. If the option is off, selected traps will be removed, and will be re-trapped. This only works for traps created by PowerTrapper Classic or Instant Trapper.

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 Classic can ignore these gaps automatically.

To use this option, select **Close when smaller than** and enter the maximum size small gaps can have.

The default size is 0.01 mm.



Attention: If you set a big gap size, trapping may become slower, and small parts of the job may be ignored during trapping.

Rules

Rules allow you to define custom trapping settings for specific trapping pairs.

When to Use Rules?

In most cases the general trapping settings you enter in the **Trap** dialog will give you excellent results, but in some cases you will need to refine them using rules.

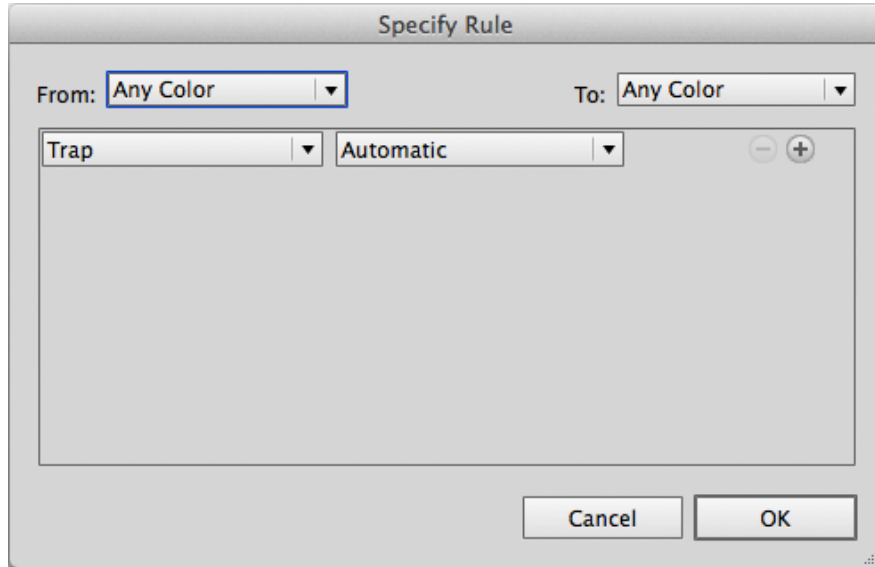
Rules allow you to specify exceptions to the general trapping settings for certain color pairs. They are an advanced feature and should be used only by experienced users.

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 Classic 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.

Creating a Rule

1. Click the **Add Rule** button at the bottom of the **Trap** dialog.



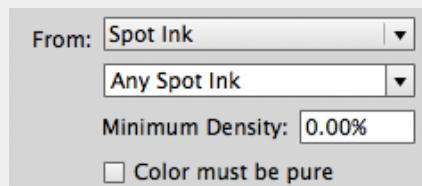
2. In the pop-up that opens, choose the trapping pair to apply the rule to using the **From** and **To** lists.

You can create a rule to trap from/to:

- any color,
- a particular ink (process, spot or opaque),

Note:

In this case, you can choose the ink name, from which **Minimum Density** the rule will apply, and if the color must be pure (no other separation in the color object) to apply the rule.




If **Spot Ink** is selected, the dropdown underneath will contain all Spot Colors used in the document.

- A specific Printing Method
- a particular object (image, gradient or empty background),

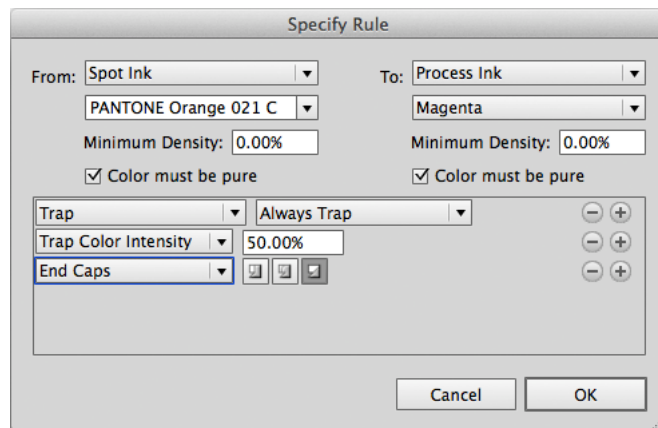
- the registration color.

3. Choose the contents of the rule.

Options	Description
Trap	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Classic calculate the trapping direction. • Always Trap always spreads the “From” color/object on the “To” color/object. • Do Not Trap never traps the “From” and “To” colors/objects together.
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 Classic 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.
Pull Back Fill Mode	<ul style="list-style-type: none"> • Normal gives normal pull backs. • Sharpen allows sharpening on pull backs. This sharpening is used e.g. for white text on images, on CD's or DVD's. Sharpening is done by pulling back the image, and adding 100% black in the trapping area.
Reverse	<ul style="list-style-type: none"> • Automatic lets PowerTrapper Classic 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. See Truncate Traps for details.

Options	Description
End Caps	Choose how to shape the ends of open traps for the trapping pair (square, round or object dependent). See End Caps for details.
Trap Corners	Choose how to shape the trapping pair's trap corners (round, beveled or mitered). See Trap Corners for details.
Direction Mode	Select into both colors to use bidirectional trapping. In case of bidirectional trapping, the trap will go half the trapping distance in each direction.
Horizontal / Vertical distortion	Set the percentage of the trapping distance to be used in vertical / horizontal direction. In the example underneath, the trap was generated with no vertical distortion (so 100%, i.e. using the entered trap distance) and horizontal distortion set to 50% (i.e. half the trap distance). 

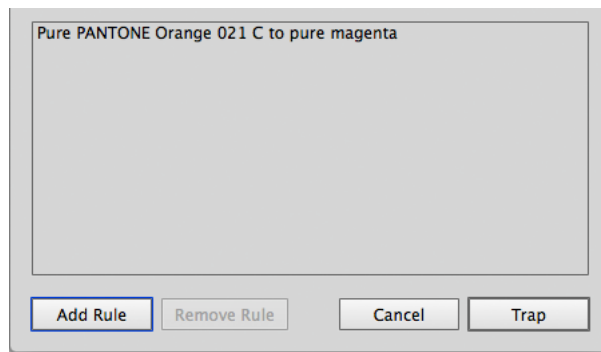
4. If necessary, use the  button to add contents to your rule.



You can use the  button to remove contents.

5. When you are done defining your rule, click **OK**.

You can now see your rule at the bottom of the **Trap** dialog.



When trapping your document, PowerTrapper Classic will trap 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.

Trap Presets

With PowerTrapper Classic, you can save all your trapping settings as a preset to be able to reuse them on other documents.

Trap Presets can be shared between different Esko Applications (PackEdge, ArtPro, DeskPack, ...)

The Trap Presets list contains 6 default Presets that can't be changed or removed:

- Default
- Bi-directional (including a rule to trap into both colors)
- Combipress (including a rule based on Printing Method)
- Offset (optimized for offset printing)
- Reverse
- Sharpening (including a rule optimized for sharpening)

Defining a Trap Preset


1. Define your trapping settings in the **Trap** dialog. Add rules if you need to.
2. In the **Trap Preset** list, choose **Save...**
3. Give your preset a name and click **OK**.

You can now see your preset in the **Trap Preset** list. To reuse it in the future, just select it in the list.

- To delete a preset, select it in the **Trap Preset** list then select **Remove**.
- To edit a preset, select it in the **Trap Preset** list, make your changes in the trapping settings, then save it again with the same name.

The Trap Layer

The Trap Select Tool

You can find the Trap Select tool  in the Tools panel. Use it to select trap areas in your job, or double-click it to open the Trapping dialog.

You can:











- click between two adjoining color areas to highlight the edge and select the trap,
- make a rectangle selection to highlight all edges and select all traps in that rectangle.

Note: Selecting one or more traps in your document using the Trap Select Tool also selects the corresponding color pair(s) in the **Color Pairs** palette. See [The Color Pairs Palette](#) on page 153.

6.5.2. The Color Pairs Palette

After trapping your document, the **Color Pairs** palette shows the trapping settings associated with each of your document's color pairs.

For each color pair, the palette shows:

- the number of **Hits** (occurrences of that color pair) in the document,
- the type of object in the pair (empty background , flat color , image  or gradient ) ,
- the type of trap performed on that color pair (normal trap , pullback , reverse trap ) ,
- the trapping **Distance** used,
- the traps' **Shape** (truncation , caps , corners ) ,
- the **Intensity** of the trap color,
- the **Pullback Ink** (when the trap is a pullback).

Viewing Traps

- To select all the traps of a color pair in your document, click that color pair in the **Color Pairs** palette.



Note: Use **Shift** to select several color pairs, and highlight all the corresponding traps in your document.

- If the tool is active, use the arrow keys to browse through the color pairs. Activate the to browse and visualize your traps one by one in your job.

Note: Use **Arrow left-right** to expand or collapse the trapping pair list.

Viewing Trap Settings


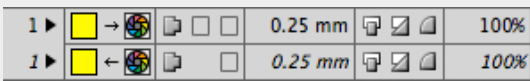



You can hide trapping settings that you are not using to make the **Color Pairs** palette smaller.









- To hide all color pairs that are not trapped with each other, select **Hide Non-trapping Color Pairs** in the palette’s fly-out menu.
To show them again, select **Show All Color Pairs** in the fly-out menu.
- To hide all color pairs that don’t have traps selected in your document, click  or choose in the palette’s fly-out menu.
To show them again, click  again or select in the menu.
- To hide the **Distance, Shape, Intensity** or **Pullback Ink** column, select **Hide Distance/Shape/Intensity/Pullback Ink Column** in the palette’s fly-out menu.
To show the column again, select **Show Distance/Shape/Intensity/Pullback Ink Column** in the fly-out menu.


Refining your Traps

You can use the **Color Pairs** palette to select certain traps and change their trapping settings.

1. Select either:
 - the color pair(s) to edit in the palette,
 - the traps to edit in your document (if you want to only edit certain traps of a color pair).
2. Make your desired changes as explained below:

to change a color pair’s...	do...
<p>trapping direction</p>	<p>click Swap Trap Direction . Trap direction can be from left to right, from right to left, or bidirectional. In case of bidirectional trapping, the trap will go half the trapping distance in each direction.</p> <p>Note: When you change the direction for only certain traps of the color pair, the color pair will be duplicated in the palette (e.g. one entry for spread and one for choke).</p> 
<p>trap type</p>	<p>choose Normal Trap , Pullback  or Reverse Trap  in the Trap column. You can also use this to trap a non-trapping color pair.</p>

to change a color pair's...	do...
trap distance	click the Distance value to edit it.
trap truncation	choose On Center  or On Edge  in the Shape column.
end caps	choose Square  , Round  or Object Dependent  in the Shape column.
trap corners	choose Round  , Beveled  or Mitered  in the Shape column.
trap color intensity	click the Intensity value to edit it.
pullback ink (for pull back traps only)	click the pullback ink to change it to another ink.

3. Click Update Traps  to apply your changes to your document's traps.

Note: To change several color pairs the same way, use **Shift** to select them, then make the changes in one color pair. Clicking Update Traps will apply your changes to all selected color pairs.

Saving and Loading Color Pairs

After trapping a document with PowerTrapper Classic, you can save its color pairs. This ensures you won't need to re-edit the color pairs if you need to change the traps later on.

When to Save Color Pairs?

You can save color pairs as soon as you are finished editing them.

If you haven't saved your color pairs, doing the following will discard them:

- closing your document,
-

Note: This doesn't affect your document's traps.

Where are Color Pairs Saved?

Your color pairs are saved in a file next to your document. This file has the same name as your document and ".tcp" as extension.

Note:

There can be only one color pairs file per document. If you make changes to your color pairs and save them again, your color pairs file will be updated.

If for some reason you need another version of your color pairs file, first save your document under a different name then save the color pairs again.

Saving Color Pairs

- To save your color pairs, select **Save Color Pairs** in the **Color Pairs** palette’s fly-out menu.

Note: This is only available when you have unsaved color pairs.

Loading Color Pairs

- To load color pairs that you saved previously, select **Load Color Pairs** in the fly-out menu.

Note:

This is only available if you have saved color pairs for the current document.

You cannot load another document’s color pairs.

You cannot load color pairs if you have changed the document after trapping.

6.5.3. Trap Red under Green...

To trap adjoining objects manually.

Trap Red under Green is the manual trapping mode. Adjoining objects (contours, text, CT's, blend objects,...) need trapping to avoid the (white) gap that appears between the objects on the printed result.

The adjoining objects, which need a fill attribute, have to touch or overlap. If they overlap and if the color that lies on top masks out (cuts away) the colors below, a trapping area is generated on the boundary of the objects.

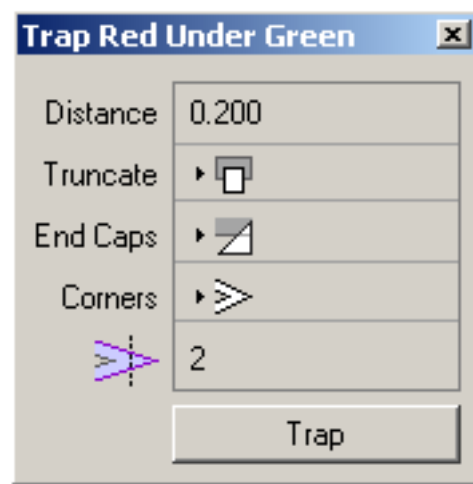
To apply Trap Red under Green, you have to

- Select the objects in red that need to be trapped underneath an adjoining object.
- Select the objects in green under which adjoining objects selected in red are trapped.
- Indicate the distance.
- Indicate the truncation type.

The red objects are trapped underneath the green objects.

Note:

The trapping area is placed in a separate layer. This way, you can easily edit or remove the trapping area, without affecting the rest of your job.



Distance Defines the trapping distance. Only positive values are allowed.

Truncate The trap area is truncated to avoid it exceeding the edge(s) of the color area under which is trapped. There are two different truncation modes:

- On Edge

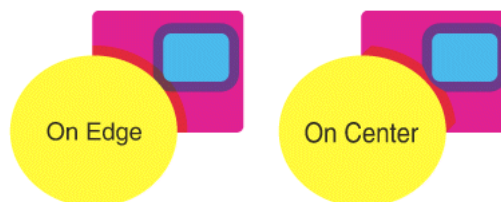


The trapping area is truncated on the object edge, even if a larger distance has been defined. This is to avoid that the trapping area of an object exceeds the edge of the object under which is trapped. This is the default truncation mode. (default)

- 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.

- Square



When this mode is selected, the trapping areas are placed at right angles relative to the boundaries of the abutting color areas.

- Object dependent

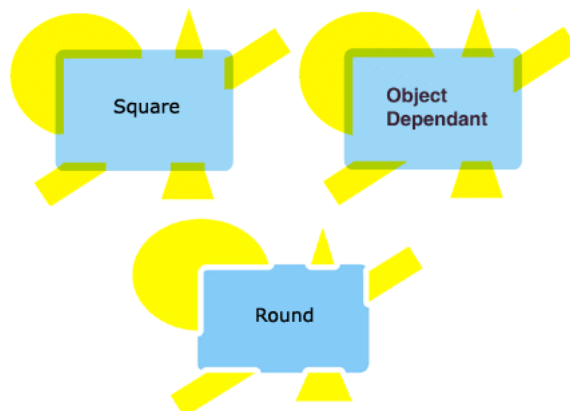


When this mode is selected the trapping areas are a logical continuation of the contour of the object that is spread. (default)

- Round



When this mode is selected the trapping areas are ended with circular arcs. This mode is typically used in combination with white knockout (= reverse trapping) 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

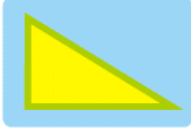
⤵

A round cap will be placed at all corners.


- Beveled

⤵

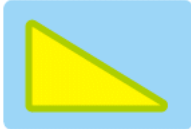
Similar to mitered, but with miter cut off with straight line (= mitered with miter ratio 1).



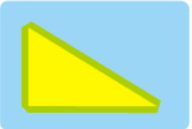
Mitered (Miter Ratio 4)



Mitered (Miter Ratio 3)



Round



Beveled

Trap button To apply the trapping, following the parameters defined in the dialog box.

6.5.4. Keep Red away from Green...

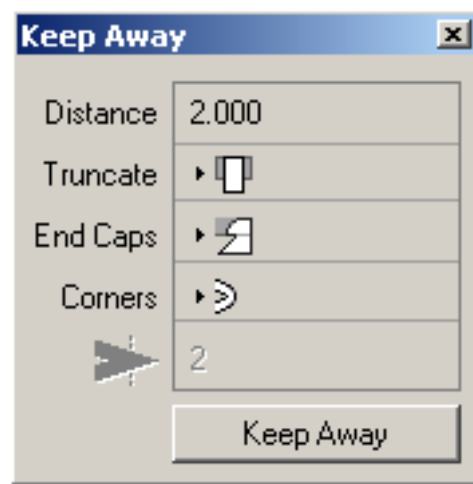
To introduce a distance between two adjoining contours in such a way that when printed, the adjoining inks do not overlap or mix.

In certain printing processes, the nature of the inks or the printing surface cause an unwanted mixing of adjoining inks. Keep Red away from Green compensates these effects.

Keep Red away from Green can also be used to create graphic effects, for example a white outline around text. The red selected object is reduced with the defined distance, to keep it away from the green selected object.



Note:

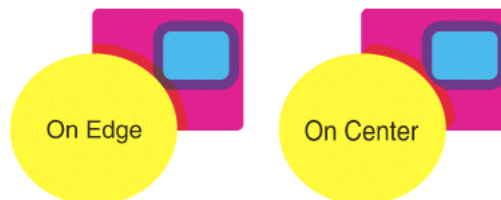
The reduced distance of the red selected object is in fact a trapping area that is placed in a separate layer. This way, you can edit or remove it, without affecting your job.





Distance Defines the trapping distance. Only positive values are allowed.


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:

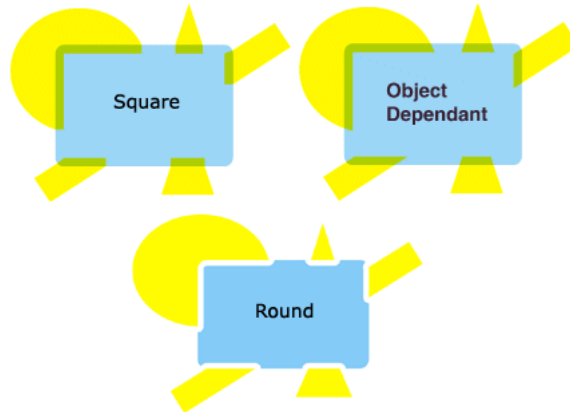
- **On Edge:**  The trapping area is truncated on the object edge, even if a larger distance has been defined. This is to avoid that the trapping area of an object exceeds the edge of the object under which is trapped. This is the default truncation mode. (default)
- **On Center:**  The system recalculates 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.

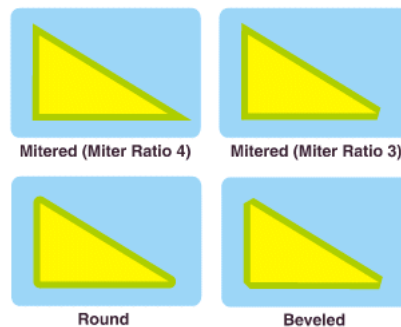
- **Square:**  When this mode is selected, the trapping areas are placed at right angles relative to the boundaries of the abutting color areas.
- **Object dependent:**  When this mode is selected the trapping areas are a logical continuation of the contour of the object that is spread. (default)

- Round:  When this mode is selected the trapping areas are ended with circular arcs. This mode is used typically in combination with white knockout (= reverse trapping) 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:  Similar to mitered, but with miter cut off with straight line (= mitered with miter ratio 1)



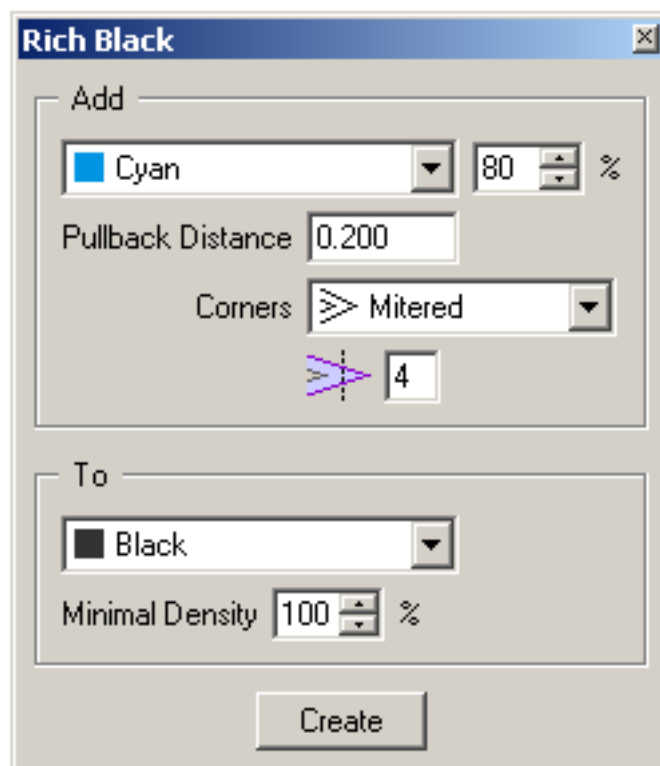


6.5.5. Rich Black...

Use the Rich Black tool to obtain a darker black by overprinting all black areas with the ink that you specify.




Note:

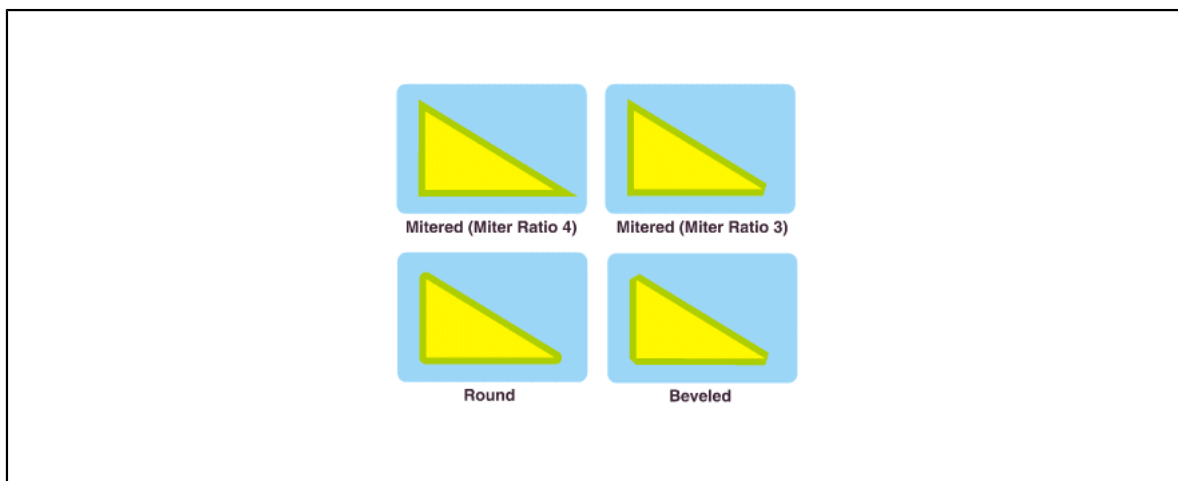
You can use this tool also to make other inks in your job richer.




Add Choose an ink from the ink list, define the percentage and the pullback distance (the distance between the edge of the defined ink and the edge of the black object).

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)

To Select the ink you want to make richer. Also define the minimum density the destination ink must have before the system may enrich it.

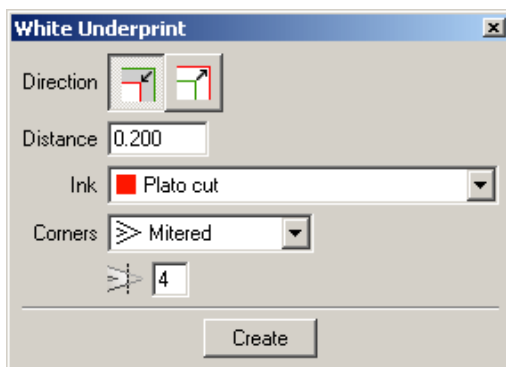
Click 'Create'. The system will generate a 'Rich Ink' layer which contains the added objects.



Note:

If you want to add more than one ink to make another ink richer, you could use the Rich Black tool again, but that takes time. You can also opt to add inks to the ink used in Rich Ink layer (you can do so in the Color Factory).

6.5.6. White Underprint

To create a white background when printing on transparent or metal material. The White Underprint area has the same shape but a smaller size than the object under which it is printed.



Direction: Do you want the White Underprint to be smaller  or bigger  than the original object?




Define the **distance** between the edge of the White Underprint layer and the edge of the object under which White Underprint is applied.

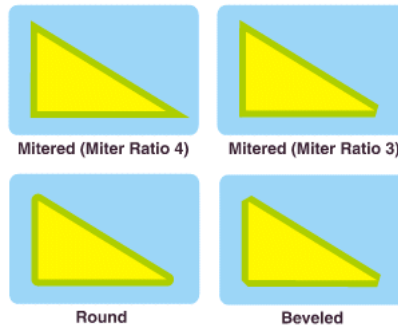
Define an **ink** for the white underprint area. You can define an ink name in the drop-down list. The default ink for White Underprint is TrW (Transparent White).


Tip:

Objects selected in blue will be ignored by the white Underprint tool.

Corners This option allows you to change the way in which 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).

Note:

Trapping output (the trap bananas) should be embedded in the station object, on top of the external reference to the graphics. As a consequence, the graphics will not be changed. In case you want to reuse certain graphics, then either trapping should be reapplied or the station object should be saved as station file.

6.6. Staggered Cutting

Staggered Cutting is designed to facilitate seamless printing in non-sleeve workflows. This means that when web fed flexo printing jobs are not using flexo sleeves the flexo material is cut and wrapped around a cylinder. The unavoidable seam will be positioned in between printable objects. Staggered Cutting is the tool to define the shape of the flexo plates, which will later in the process be sent to a Kongsberg table on which the flexo plates are cut. A cutting path can be used for all separations, or a cutting path per separation can be defined.

Next to these staggered cutting paths, Plato also stores all stroked lines which are in a technical information, in special tags in a PDFPLA file. These tags follow the same track (are forwarded by the FlexRIP into tags in a LEN file, allowing DFS to pick them up).

The workflow is as follows:

1. An operator defines the staggered cutting path in Plato.
2. When ripping the PDFPLA file to a LEN file via FlexRip, the path is copied into the LEN file.
3. The path is then picked up by the Digital Flexo Suite (DFS). From DFS, you can export an ACM file which can be used to cut the plate on a Kongsberg table.

Note:

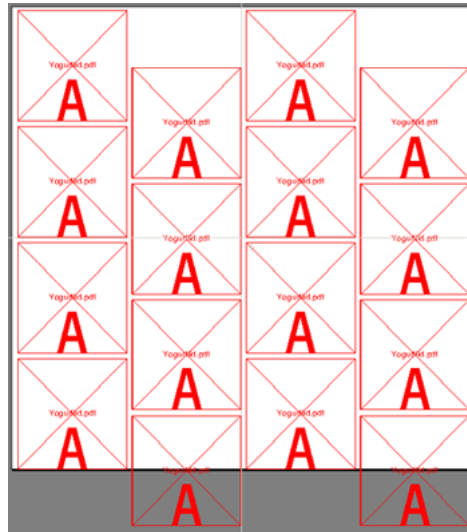
The staggered cutting workflow works only on PDFPLA files (PDF workflow).

The input of this module is a red selected (open) contour. Typically this contour will be created using the [Draw Polygon tool](#).

6.6.1. Creating staggered cutting lines

To create staggered cutting lines:

1. Create a step and repeat layout, with staggering.



A staggered layout will create a plate with a irregular seam which will avoid problems with the press.

2.



Select the *Draw Polygon tool* and draw a path.

If you are going to use the option 'Truncate on Center', the path does not have to be too precise. Even a roughly drawn path can be used to create the ideal seam.



3. Make the appropriate settings in the **Staggered Cutting** dialog. See *Staggered Cutting...* on page 392

4. Click **Create**

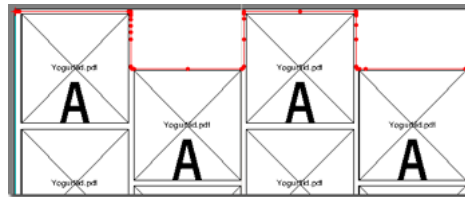
The Staggered Cutting line will be created.



When creating a Staggered Cut, a number of objects are created, based on the selected parameters: the cut path itself, objects knocking out the pixels outside the cut path, a stroked line to visualize where one needs to cut, etc. All these objects are grouped, in a separate layer named "Staggered Path".

The staggered cutting lines will get a stroke, colored in a technical ink. This ink is called "Cutline" if the staggered cut line is used for all separations. If different cut lines are created for different separations, the technical inks used will be called "Cutline <inkname>", e.g. "Cutline Cyan".

When selecting one of these objects, the **Staggered Cut** dialog will show the parameters that were used to generate it. You can change the parameters and click **Modify**: the staggered cut path will be recreated using the new parameters, for example to apply **Truncate on Center**. The path generated will run along the center line between the different die shapes. See [Staggered Cutting...](#) on page 392 to get an overview of all the parameters.



6.6.2. Changing a staggered cutting line

You can change the parameters of a staggered cutting line by selecting it, changing the parameters and clicking the **Modify** button. However, for more elaborate editing, you can use the **Extract for Editing** button.

1. Select the staggered cutting line you want to modify
2. In the **Staggered Cutting** dialog, click the **Extract for Editing** button.
A copy of the top part section of the staggered cutting line is created, and placed as a regular open path in a layer called "Staggered Path Edit".
3. Edit the path using the transformation and path editing tools, e.g. the Redraw Path tool. See [Redraw Path Tool](#) on page 309
4. Select the modified path, and create a Staggered Cutting line as described in [Creating staggered cutting lines](#) on page 165

The newly created staggered cutting lines will replace the existing cutting lines. If you only select one or a few separations when creating the new cutting lines, only the staggered cutting lines for those inks will be replaced, while the cutting lines for other separations remain unchanged.

7. Working with SmartMarks

- [What are SmartMarks?](#)
- [Using SmartMarks](#)
- [Supplied SmartMarks](#)

7.1. What are SmartMarks?

SmartMarks are marks used as production controls that automatically update based on changes to the sheet size, the plate size and the number of inks on the plate.

You add SmartMarks to your plate or station using the SmartMarks tool on the Production menu. You can add as many marks as you like to your job. You can also add multiple copies of a mark to the same job. If you have marks that you add to every job, you can save the collection as a SmartMark Set. A set can be added to a job to place several marks all at once.

Marks and setsThe SmartMark framework uses two concepts that you should understand when working with SmartMarks: Marks and Sets. The combination of these two concepts enable you to create production controls efficiently and accurately.

What is a Mark? All of the SmartMarks that you add using the SmartMark tool are based on mark macros. Macros are scripts that are executed to create the marks that you have specified. So instead of simply drawing an object, you select a script (e.g. a rectangle mark) and specify some parameter settings (e.g. draw in center of sheet in registration ink).The mark scripts are executed at certain times to make sure that they are completely up-to-date with the current job. For example, when you save or open a file containing SmartMarks, the marks are executed. Other actions also cause marks to be executed, like adding an ink or changing an ink's definition.

What is a Set? A set is a collection of marks that you save using the SmartMarks window. A set can contain any number of SmartMark mark definitions. When you save a set, Esko Plato will ask you for a name for the new set. In the future, you can apply all the marks in that set by picking its name from a list of available sets. Like the marks themselves, you can create and save as many sets as you like. This way, you can create a customized set of marks that are used for different customers, different printing presses and different printing processes.

7.2. Using SmartMarks

SmartMarks are easy to use. Click SmartMarks on the Production menu and can begin to add marks.


- [Adding a SmartMark](#)
- [Deleting a SmartMark](#)
- [Editing a SmartMark](#)

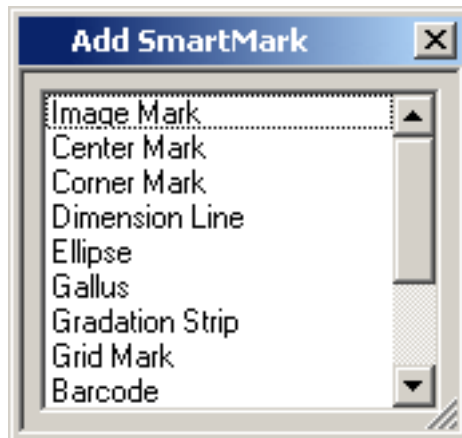
- [Creating and saving a set](#)
- [Using a set](#)
- [Replacing and editing a set](#)
- [SmartMarks in the Station View](#)
- [Custom Color in SmartMarks](#)

7.2.1. Adding a SmartMark

Adding a SmartMark is easy. Even though each individual mark contains its own set of parameters, the basic procedure for adding a mark to a job is the same.

To add a SmartMark:

1. On the Production menu, click 'SmartMarks'. The SmartMarks window will appear.
2. Select the desired reference point (general, plate, sheet, visible objects, Position Mark,...)
3. Click the 'Add Mark' button +  The Add SmartMark dialog box appears, listing the available marks:



Note:

The list of available SmartMarks will change according to the 'object' you want to attach the SmartMark to.

4. Click the name of the mark that you want to use. The options dialog box for the mark you selected will appear.
5. Set the options as required for your job.
6. Click 'OK'.

You can use 'Apply' to preview the mark before clicking OK. If the mark is not exactly what you want, you can make changes to the dialog box and click 'Apply' again.

The mark's name appears in the 'Current Marks' list. The name will be selected in the list.

Whenever a mark name is selected in the SmartMarks window, the mark that is created by that mark will also be selected in the document window. This helps you to identify which mark definition creates a particular mark in the job.

7.2.2. Deleting a SmartMark

You can delete a SmartMark in various ways. First, you can simply select it in the document window and delete it like you would any other object. When you delete a SmartMark from the document window, its definition is deleted automatically from the SmartMark list of Current Marks.

You can also delete a SmartMark from the SmartMarks window:

1. In the SmartMarks window, click the name of the mark that you want to delete.
2. Choose 'Delete' from the fly-out menu.

Tip:

You can delete more than one mark from the 'Current Mark' list by holding [CTRL] to select more than one name at a time.

Tip:

You can also delete the SmartMark directly by selecting the object and deleting it.

7.2.3. Editing a SmartMark

After you have created a SmartMark, you may want to edit it later. There are two ways that you can edit an existing SmartMark.

First, you can open the mark's options dialog box by double-clicking the mark in the document window. The mark options dialog box will appear. Make any changes and click 'OK' to update the mark in the document window.

You can also edit a SmartMark from the SmartMarks window:

1. Select 'SmartMarks' on the Production menu. The SmartMarks window appears.
2. In the SmartMarks window, click the name of the mark that you want to edit.
3. On the fly-out menu, click 'Options'. The mark options dialog box will appear.
4. Edit the parameters and click 'OK'.

You can also double-click a mark name in the SmartMarks window or the mark object in the drawing window to open the option dialog box for that specific mark.

7.2.4. Creating and saving a set

If you regularly create jobs that use the same marks, you can save time by creating a set that contains all these marks. Each mark of a set can be applied in one step by selecting it from the Mark Set list.

To create and save a Mark Set:

1. In the SmartMarks dialog box, add all the marks that you would like to have included in your set.
2. On the fly-out menu, click 'Save Set'.
3. Enter a name for the Set and click 'OK'.

Note:

Position Marks (except for SmartMarks objects) are not saved in a set.

7.2.5. Using a set

Once you have defined a SmartMark set, its name will appear in the Mark Set list. To use it, simply select its name from the list. The marks in that set will be applied to the current job. However, it might occur that you want to keep existing marks that are already present in the job while adding marks from a set. If this is the case, you should use 'Load Set' to add the marks of a set to existing marks that are already in the job.

7.2.6. Replacing and editing a set

Replacing a set If you have used a set in a job and want to use a completely different set, simply open another set. Opening a different set will remove all the marks from the previous set and replace them with marks from the new set.

Editing a set Editing a set is done by loading the set, making the required changes, and saving the set again. If you want to use an existing set as the basis for a new set, you can load the existing set, make the required changes and use 'Save As' on the fly-out menu. Using 'Save As' creates a new set with a different name.

7.2.7. SmartMarks in the Station View

When using SmartMarks in Esko Plato, the reference point in the marks dialog boxes is based on the plate or the sheet. In the Station View, there is no plate nor sheet; in this case the reference point is based on the borders or the margins.

In the Station View, the borders are set to the bounding box of the current die. So, you can attach SmartMarks to the die of the station.

The Margins of a station are set to the bounding box of all the objects in the station, including those that are outside the die. So, if you have other elements, like Geometry Marks that are located outside the die, the Margins are fit to include them. See [Setting the location of a SmartMark](#).

7.2.8. Custom Inks in SmartMarks

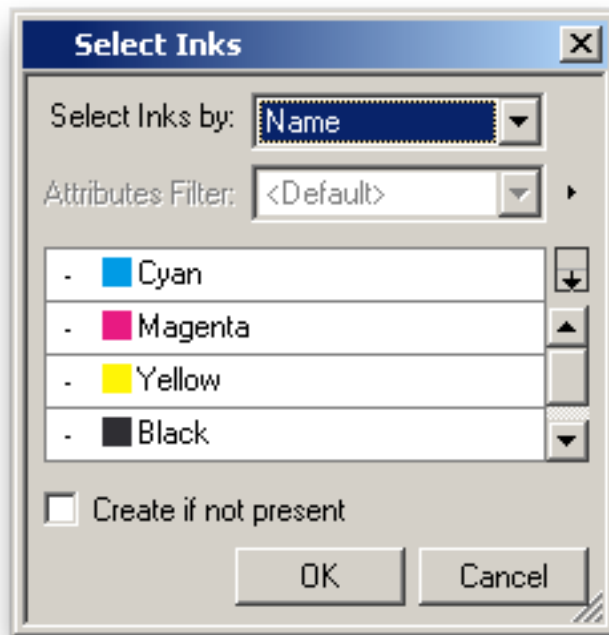
Custom Ink allows to pick any number of inks, by name, number or attribute.

Click **Select Inks...** to select the Inks. You can select inks in three ways:

- [Select Custom Ink by Name](#) on page 172
- [Select Custom Ink by Number](#) on page 172
- [Select Custom Ink by Attribute](#) on page 173

Select Custom Ink by Name

To select Custom Inks by Name, select "Name" from the dropdown list.

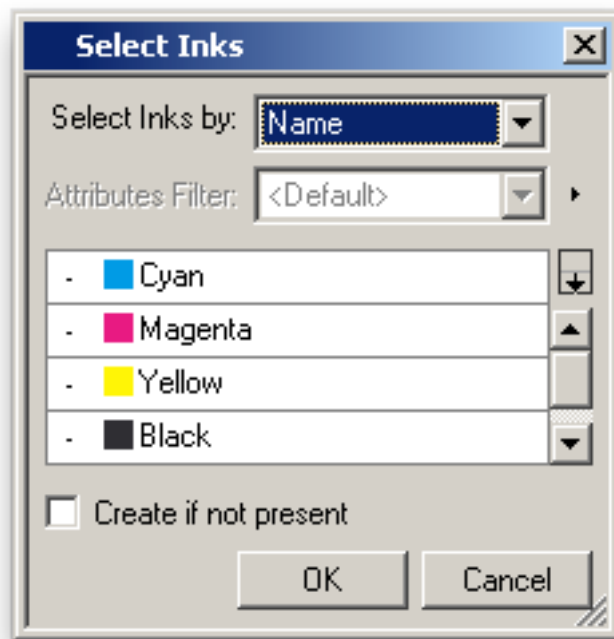


Only the inks that are used in the current document are displayed. Activate an ink by clicking the "check" area in front of the ink.

If the "Create if not present" option is selected, the ink will be created if it is not available in the document you want to apply the mark in.

Select Custom Ink by Number

To select Custom Inks by number, select "Number" from the dropdown list.



The inks that are used in the current document are displayed. Activate an ink by clicking the "check" area in front of the ink.

When building SmartMarks using a custom color in an empty document, or in the list below the used inks, the list shows the color "unknown [n]" whereas [n] is the number of the ink.

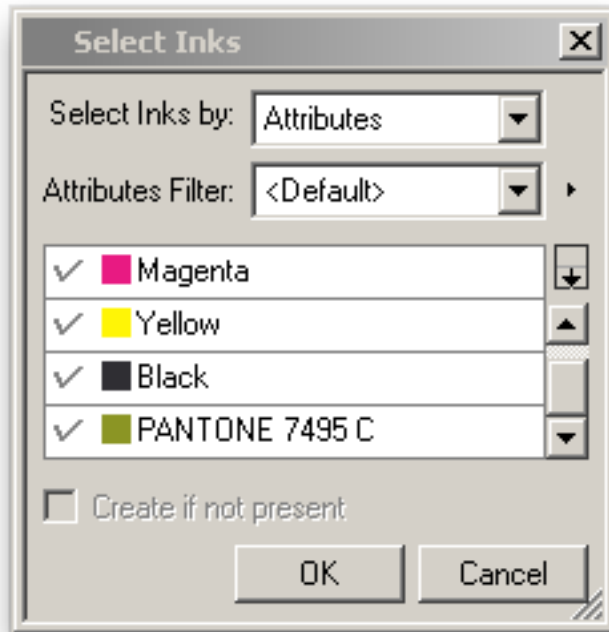
Select Custom Ink by Attribute

Selecting Custom Inks by Attribute allows to apply a filter on all inks.

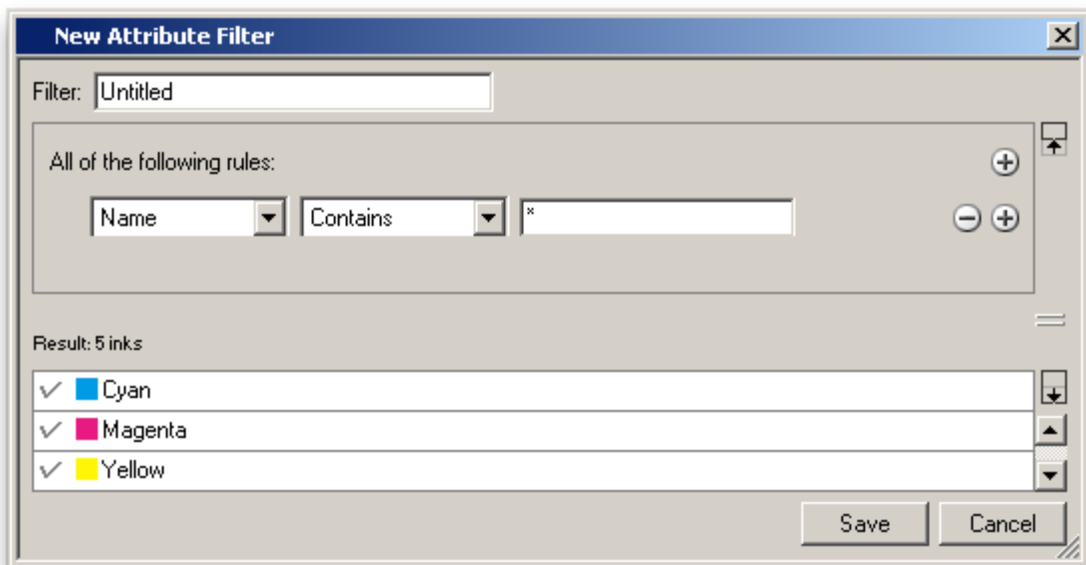
You can select an Attributes Filter from the dropdown.

Note:

If no Attributes Filter is defined, Plato will make a "Default" filter, selecting all Inks.



Using the fly-out next to the Attributes Filter dropdown, you can create a new filter, edit an existing filter or delete one.



A filter is defined by one or more "rules". You can add or remove rules by clicking the + or - button. A rule can select inks based on the name, ink book, ruling, angle, dot shape, type or printing method.

An ink will be selected if it matches all of the highest level rules (a logical "AND" function: the ink is selected if rule 1 AND rule 2 is fulfilled).

By holding down the ALT-key, the + button will change to "...". Clicking this button will add a logical operator, allowing to combine rules using logical expressions: "All of the following rules" (AND), "Any of the following rules (OR), or "None of the following rules" (NOT).

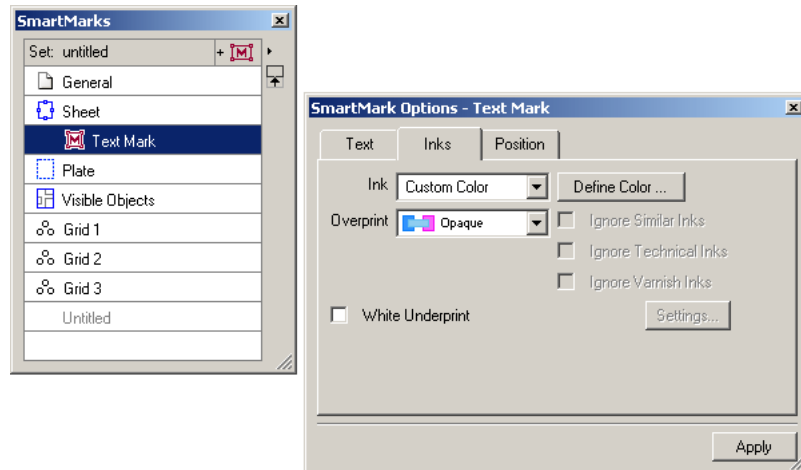
The top example will select inks with "Pantone" in their ink name, and that are either have "Offset" as printing method, or come from an ink book which name begins with "Custom". The second example will select inks with "Pantone" in their ink name, except if they have "Offset" as printing method

The "Result" underneath shows what inks from the current document would be selected using the current filter.

Note: Filters are stored on the Automation Engine server (when connected), so they can be used in all connected PackEdge / Plato stations, and in the SmartMarks Automation Engine tasks.

7.2.9. Custom Color in SmartMarks

When you select a Custom Color ink from the Ink dropdown list, you can define a color and use it afterwards in a SmartMark. Custom color inks can be applied e.g. in text marks.



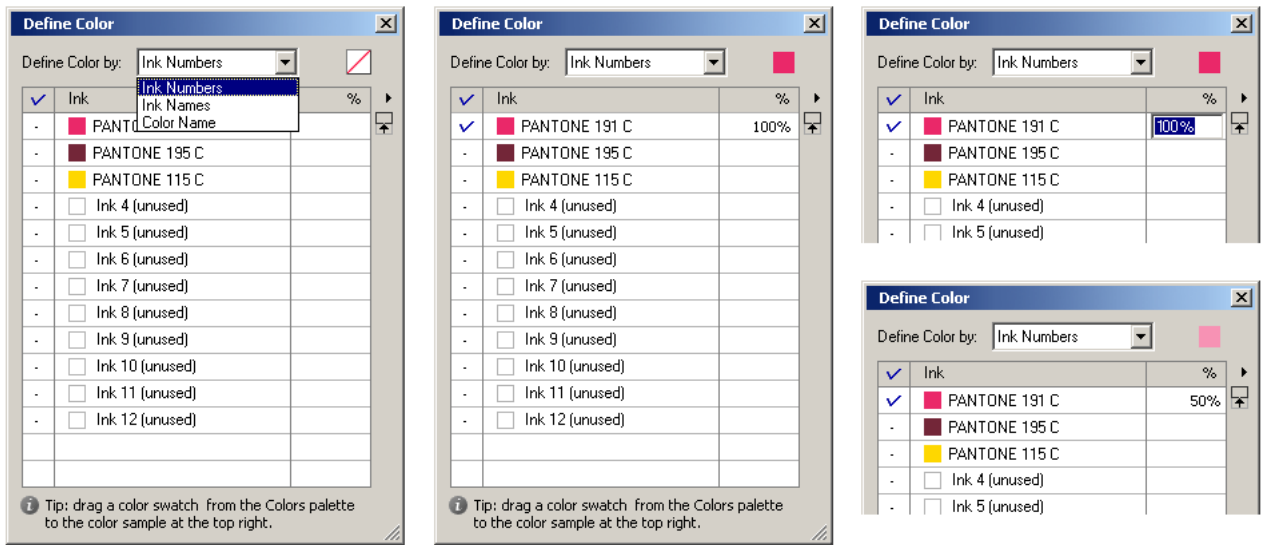
Click 'Define Color...!' to define a custom color. Custom colors can be defined in three ways:

- *By Ink Number*
- *By Ink Name*
- *By Color Name*

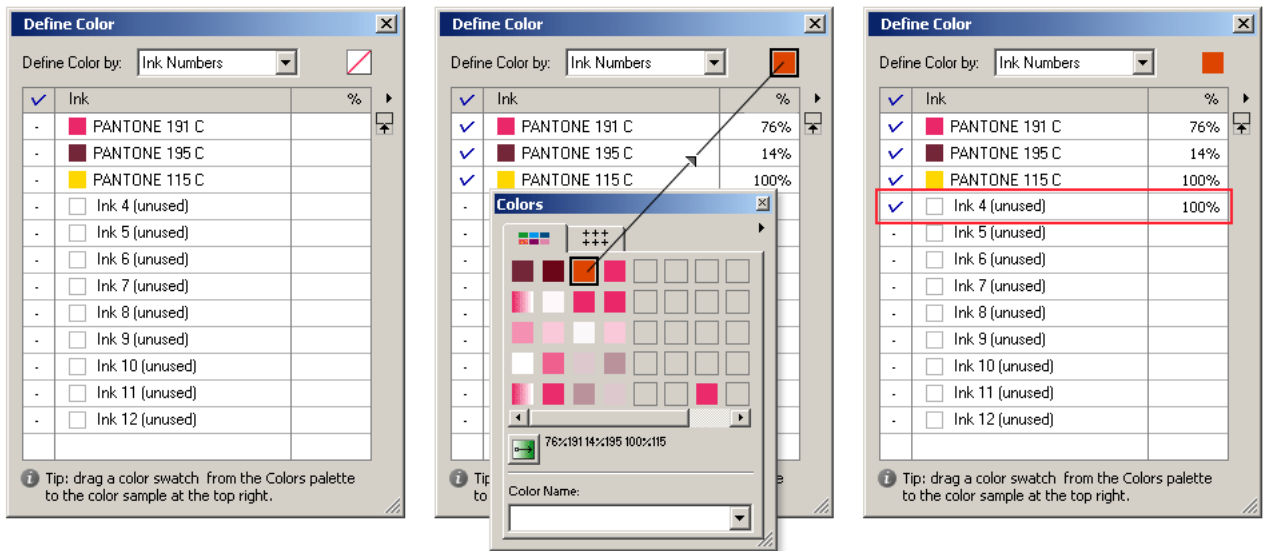


Ink Numbers

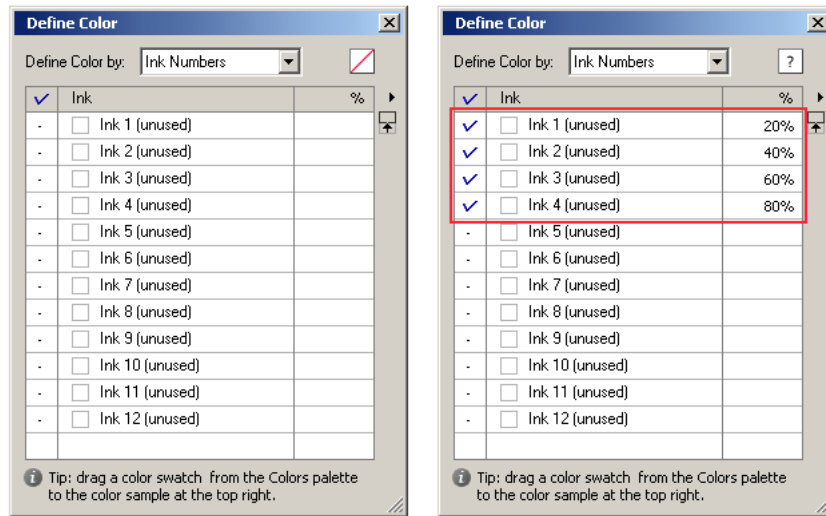
To define a Custom Color by means of an ink number, select 'Ink Numbers' from the dropdown list. The inks that are used in the current document are displayed. Activate the ink by clicking the "check" area in front of the ink and its value will be set to 100%. You can click the value if you want to change the percentage. The color patch will show you the resulting color.



You can drag a color swatch from the Colors palette to the color sample at the top right of the dialog box and the list in the "Define Color" dialog box will be updated immediately.

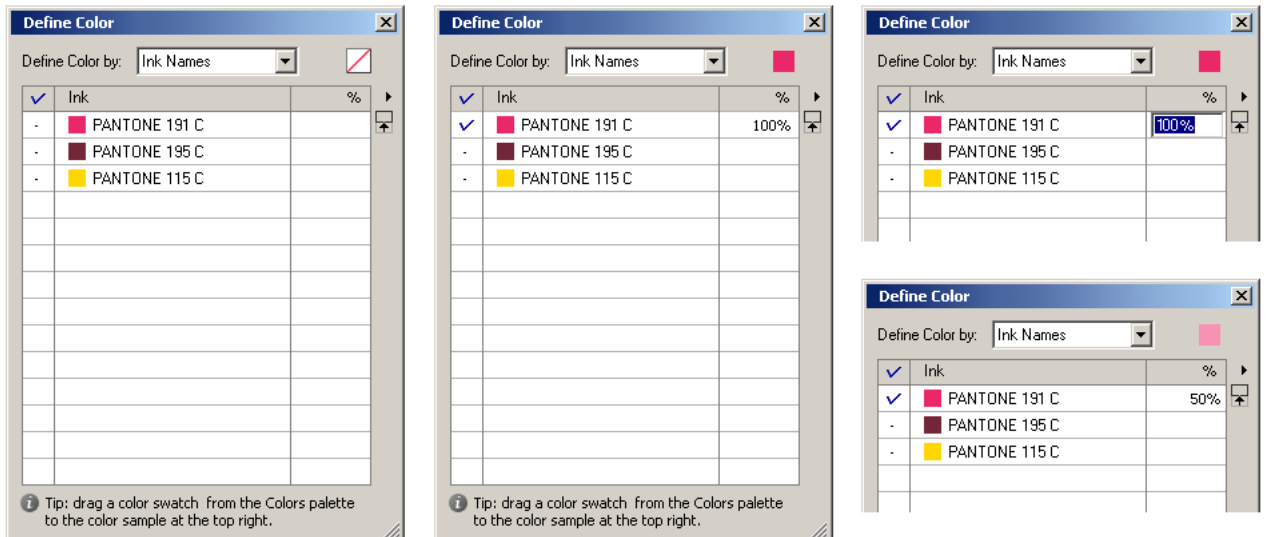


When building SmartMarks using a custom color in an empty document, the color patch shows the color "unknown".



Ink Name

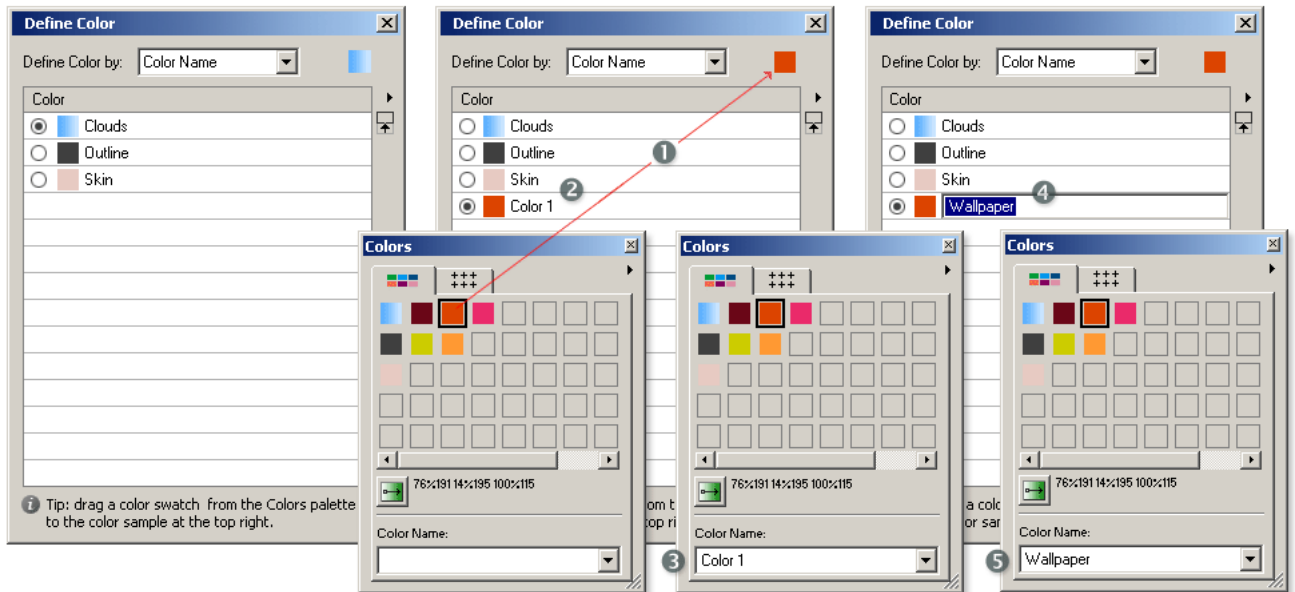
When you select Ink Name from the dropdown list, only the inks that are used in the current document are displayed. Activate the ink by clicking the "check" area in front of the ink and its value will be set to 100%. You can click the value if you want to change the percentage. The color patch will show you the resulting color.



Color Name

All named colors are displayed in the list. Select one of them by selecting the button in front of the color. To create a new color, you can simply drop a color from the Colors palette on the patch. This color will be added to the list and gets a default name (Color 1, Color 2, ...). The color name will also be displayed in the Colors palette. At any time you are able to change this

name in the "Define Color" dialog box. When changing the name in the 'Define Color' dialog box, the Colors palette will updated automatically.



7.3. Supplied SmartMarks

SmartMarks are very easy to use. The mark Options dialog boxes allow you to set the location and ink usage for the mark. This section describes each of the marks that are supplied with Esko Plato.

- [Setting the location of a SmartMark](#)
- [Span Area](#)
- [Image Mark](#)
- [Center Mark](#)
- [Corner Mark](#)
- [Crease Mark](#)
- [Dimension Line](#)
- [Gallus](#)
- [Gradation Strip](#)
- [MFG Text replace](#)
- [Rectangle](#)
- [Standard Mark](#)
- [Standard Strip](#)
- [Text Mark](#)
- [Trim Mark](#)
- [Bar code](#)

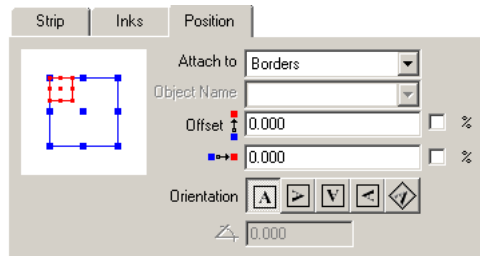
- *Ellipse*
- *Grid Mark*
- *Cell Numbering*
- *Grommet*
- *Position Mark*
- *GRQ Gallus*
- *GRQ Gradation Strip*
- *GRQ Grid Marks*
- *GRQ Image*
- *GRQ Marks*
- *GRQ Rectangle*
- *GRQ Strip*
- *GRQ Text*
- *GRQ Cellnumbering*

Note:
 If QuickStep was installed on your workstation, the list of registration marks that were available in QuickStep will also be available as SmartMarks.

Note:
 The list of available SmartMarks will depend on the chosen anchor point. Marks that do not have an anchor point (like standard mark), will only pop up when 'General' is selected in the SmartMarks dialog box.

7.3.1. Setting the location of a SmartMark

You specify the location for most SmartMarks using a 9-point selector similar to the one that is used in the Transform dialog box. Here is an example of the Standard Strip mark:



In the left half of the dialog box, you set the location for the mark. The red selector represents the reference point on the mark; the blue selector represents the reference point in the job.

The reference point in the job is selected by position. In Esko Plato, a SmartMark can be **attached to** a sheet, a plate, visible objects (= bounding box of all printable objects except

SmartMarks), or 'Position Marks'. In the Station View, a SmartMark can be attached to the borders, the margins, to visible objects' (= bounding box of all printable objects except SmartMarks), 'Position Marks', or to the 'General' level.

You click and drag the mark's red selector to align any of its 9 points to any of the 9 points on the job. In the dialog box above, the upper left corner of the mark will be aligned with the upper left corner of the sheet.

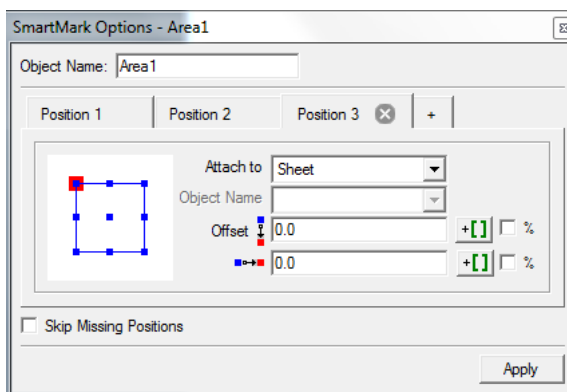
Once you have set the reference point for a mark, you can specify an **offset** from this point using the vertical and horizontal offset fields. Both positive and negative offsets are allowed.

You set the **orientation** of the mark by selecting one of the 5 orientation options.

7.3.2. Span Area

The Span Area SmartMark allows to create new named objects which span a (rectangular) area using the bounding box of two or more points.

This can be used to create named objects, that can be used as anchor points to position other SmartMarks.



The SmartMark is defined by

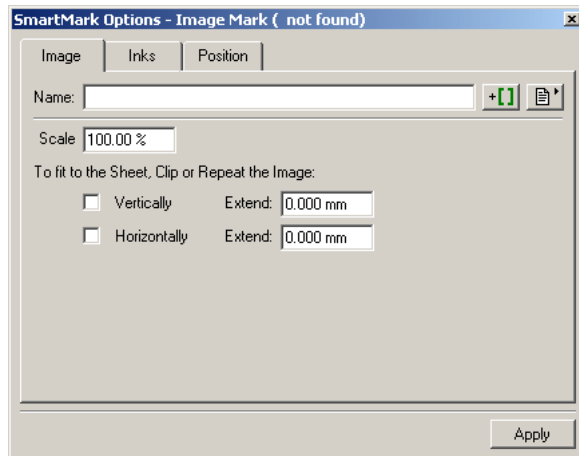
- The **Name** for the named object
- The Position of the different points. Position is defined as for regular SmartMarks. See [Setting the location of a SmartMark](#) on page 180

You can click the + to add an extra point, or x to remove a point.

If the **Skip Missing Positions** option is enabled, any missing position (e.g. because it's attached to a non existing named object) will be ignored. If a position is missing and the option is off, or if there are less than two positions defining the Span Area, the Span Area SmartMark will fail.

7.3.3. Image Mark

The image mark allows you to enter a CT, LP, LC , GRS or Normalized PDF file as a mark.




- Monochrome images:(Can be placed in one or all inks of the job, with or without step.)
 - LP, LC, monochrome CT
 - GRS files containing only one ink
 - Normalized PDF files containing more than one ink
- Colored images:(Can be placed in their original inks, or can be inkmapped to the job's inks.)
 - CMYK and multichannel CT files
 - GRS files containing more than one ink

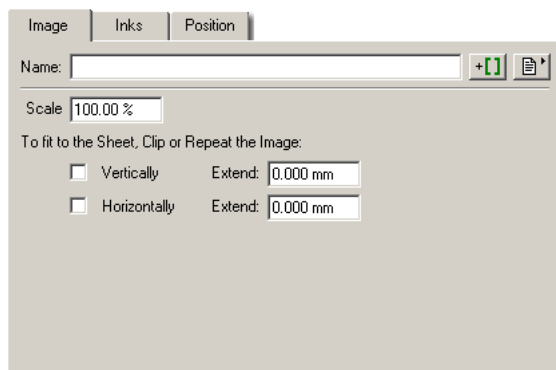
The Image Mark dialog box has three tabs:

- *Image*
- *Inks*
- *Position*

Tip:

The Image Mark can be moved in your job by simply using the *Move tool*  in the Tools toolbar.

Image



Name: Enter the name of the Image Mark you want to use or click the browse button to browse to a Normalized PDF, GRS, LC or LP file.

Tip:

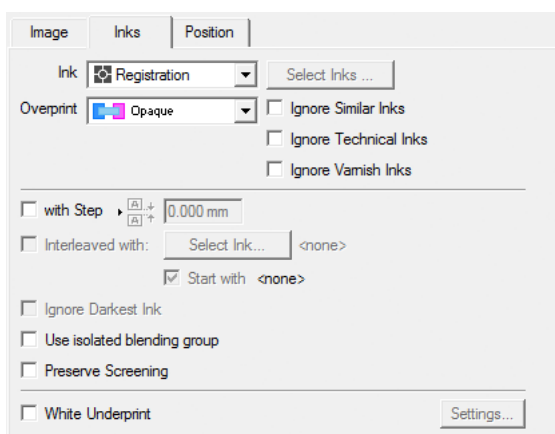
If you want to incorporate SmartNames in your file name, click the 'SmartNames' button.

Scale: You can scale the Image Mark. If the Image Mark is attached to a Position Mark, other scale options are possible: Scale to fit, Downscale to fit, Autorotate and scale to fit.

To fit to the Sheet, Clip or Repeat the Image: This option will copy the image a number of times so that its width and/or height matches the width and/or height of the anchor point (sheet/plate/visible objects). If the size is too large, a clip path will be applied to make sure that the image exactly matches the required size.

When placing **[taskinputfile]**, you have the option to **Hide trap layer(s)** . This will remove all trapping layers which were generated in PackEdge or Automation Engine. If this option is on, the [taskinputfile] will no longer be placed as an external reference, so the resulting output file can be larger.

Inks



Ink

- Registration
- Darkest
- Black or Darkest

Note:

Black or Darkest selects any black (process, pantone) if it exists. If they dont exist, the darkest ink is taken. This option ensures that a black ink is used for the marks, even if the job contains a (darker) ink like Reflex Blue

- Black

Note:

This option will select any black (process or pantone). If no such ink exists, process black is added to the ink table.

- Knock Out
- Custom Ink allows to pick any number of inks, either by name or by position (in the inks list).
- Custom Color allows you to pick a color, based on the ink name, ink number or color name.

Overprint

- Opaque
- Darken
- Postscript
- Add
- Reverse: Knocks out the selected ink in the underlying separation. If you select Magenta for instance, it will be knocked out in the Magenta separation.

Ignore Similar Inks becomes available when you select Registration ink.

Tip:

This is useful when you have similar inks in your job, e.g. two cyans with a different angle, and you want to step all the inks but you only want to have one cyan.

Ignore Technical Inks will not include technical inks in the SmartMark. This option becomes available when you select Registration ink.

Ignore Varnish Inks will not include varnish inks in the SmartMark.

With Step is active when the ink is set to Registration or Custom. The mark object will be repeated in each of the selected inks. The repetition is done using the specified step parameters (direction + step/gap).

Interleaved with can be used in combination with 'With Step'. Inbetween each stepped instance of the mark object, an additional instance of the mark object will be present in the specified ink.

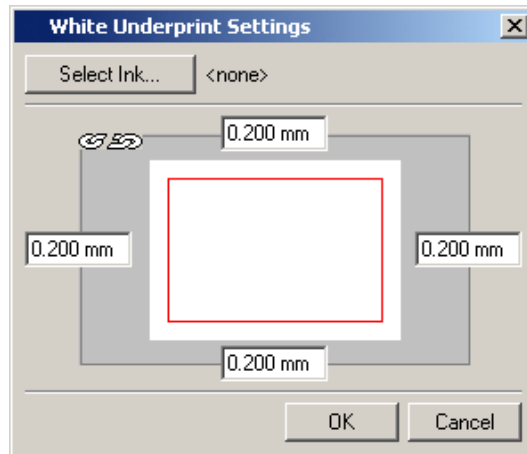
Start with can be used in combination with the 'Interleaved with' and 'With Step' options. It determines if the extra objects generated by activating the 'Interleaved with' option are placed before or after the stepped instances of the mark object.

Ignore Darkest Ink is used in combination with 'With Step'. It determines whether or not the darkest ink should be excluded from the stepped instances of the mark object.

Use isolated blending group will put the mark in a group with the **Isolated blending** option enabled. This will ensure that blend modes used in the mark will not affect underlying objects

The **Preserve Screening** option allows to place the mark in all separations of the current job, but keep the screening (dot shape, lpi, angle) from the original image.

White underprint allows you to add white underprint to the mark. Click the 'Setting' button, the White Underprint Settings dialog box pops up.



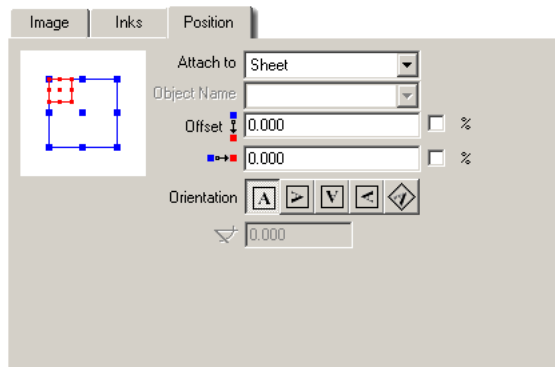
Select **Keep original inks** if you want to use the original inks of the image when placing this mark. Image inks that do not yet appear in the job's ink list will be added to that list. Inks that do appear in the job's ink list, but which have different ink settings will also be added to that list.

Select **Map inks** if you want the inks of the image mark to be mapped to the job's inks.


Keep ink names is almost the same as 'Keep original inks' except that image inks that already appear in the job's ink list with the same name (but with a different ruling, angle or dotshape) will be replaced by the corresponding ink from the current job. Inks that do not yet appear in the job's ink list will be added.

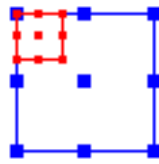
Position

The Position tab allows you to select the position of the mark by moving the red rectangle (the mark) in the blue rectangle (the job borders).




Attach to: Plate, Sheet, Visible Objects, Objects.

Use the offset options  to apply an extra offset to the chosen position using



Orientation Click one of the buttons to obtain the desired orientation for your image.

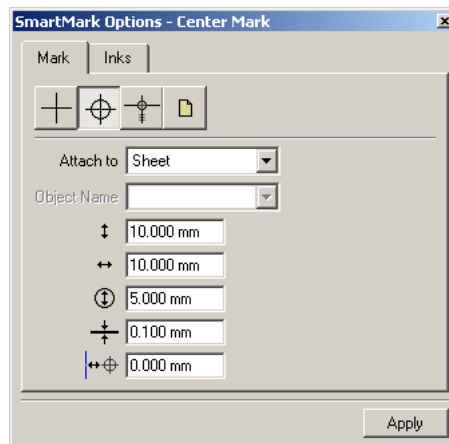


Angle  allows you to specify an angle in case you selected the rotation button in the 'Orientation' group.

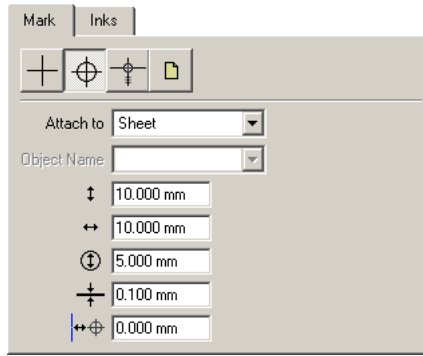
7.3.4. Center Mark

A center mark places marks at the centers of the 4 sides of the anchor point (plate/sheet/ visible objects/ Position Mark).

The Center Mark dialog box has two tabs: *Mark* and *Inks*.



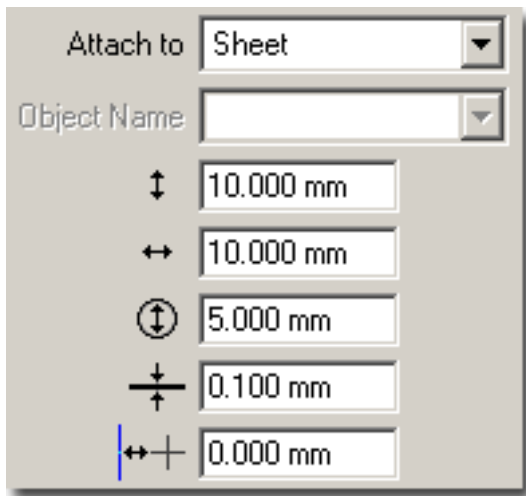
Mark



There are four kinds of Center Marks:

- [Center mark 1](#) on page 187
- [Center mark 2](#) on page 188
- [Center mark 3](#) on page 189
- [From file:](#)

Center mark 1



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



Define the vertical and horizontal dimensions .



Define the diameter.

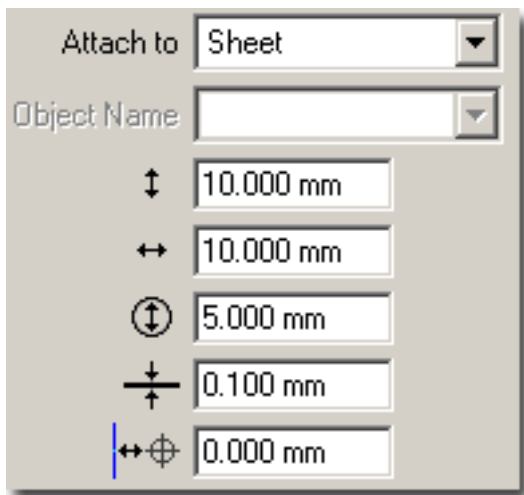


Define the thickness of the mark's lines.



Defines the distance between the mark and the object the Center Mark is attached to.

Center mark 2



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



Define the vertical and horizontal dimensions .



Define the diameter.

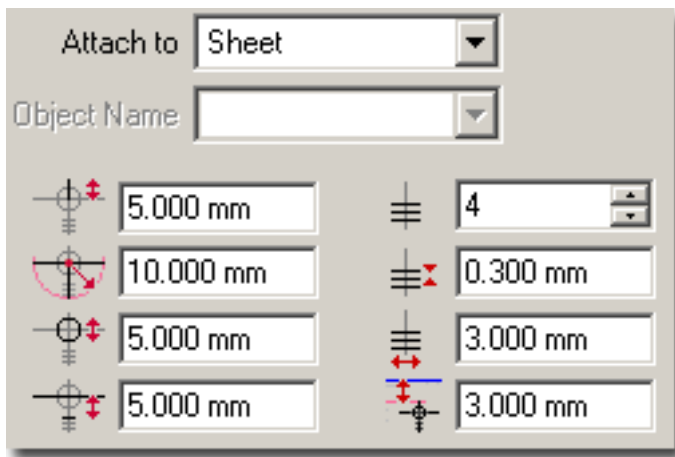


Define the thickness of the mark's lines.



Defines the distance between the mark and the object the Center Mark is attached to.

Center mark 3



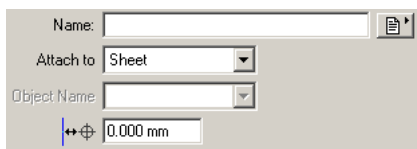
Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.

Every aspect of this Center Mark can be defined.

From file:

If you really have specific needs, you can always design your own mark and use this option to place it as a Center Mark.



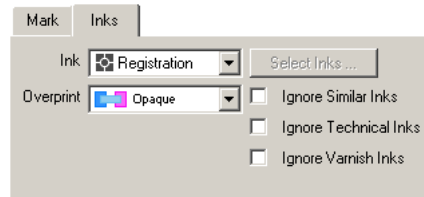
Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



Defines the distance between the mark and the job borders.

Inks



Ink

- Registration
- Darkest
- Black or Darkest

Note:

Black or Darkest selects any black (process, pantone) if it exists. If they dont exist, the darkest ink is taken. This option ensures that a black ink is used for the marks, even if the job contains a (darker) ink like Reflex Blue

- Black

Note:

This option will select any black (process or pantone). If no such ink exists, process black is added to the ink table.

- Knock Out
- Custom Ink allows to pick any number of inks, either by name or by position (in the inks list).
- Custom Color allows you to pick a color, based on the ink name, ink number or color name.

Overprint

- Opaque
- Darken
- Postscript
- Add
- Reverse: Knocks out the selected ink in the underlying separation. If you select for instance Magenta, it will be knocked out in the Magenta separation.

Ignore Similar Inks: This option becomes available when you selected Registration ink

Tip:

This is useful when you have similar inks in your job, e.g. two cyans with a different angle, and you want to step all the inks but you only want to have one cyan.

Ignore Technical Inks: Technical inks will not be used in the SmartMark. This option becomes available when you selected Registration ink.

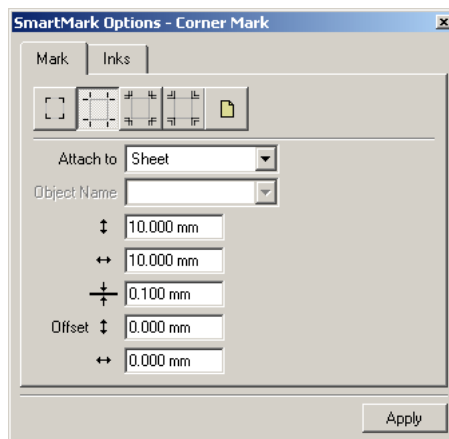
Ignore Varnish Inks: Varnish inks will not be used in the SmartMark.

7.3.5. Corner Mark

A corner mark places marks at the 4 corners of the anchor point (plate/sheet/visible objects/ Position Mark).





The Corner Mark dialog box has two tabs:

- [Mark](#)
- [Inks](#)

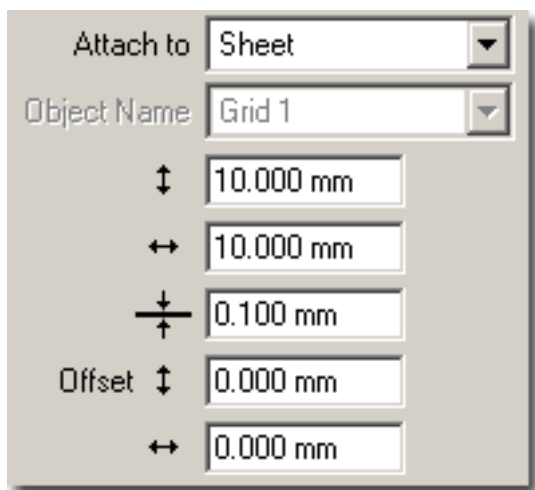


Mark

There are five kinds of Corner Marks:

- [Corner mark 1](#) on page 192 
- [Corner mark 2](#) on page 192 
- [Corner mark 3](#) on page 193 
- [Corner mark 4](#) on page 194 
- [From file:](#)

Corner mark 1



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



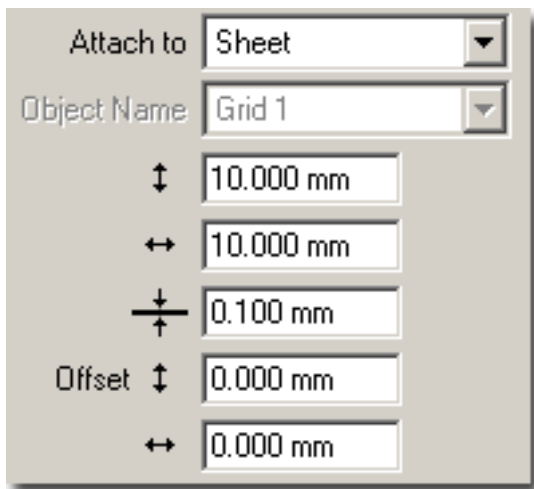
Define the vertical and horizontal dimensions .



Define the thickness of the mark's lines.

Vertical and horizontal **Offset** allows you to change the position of the mark.

Corner mark 2



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



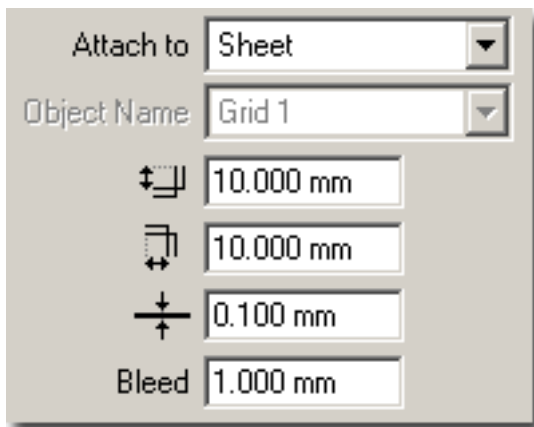
Define the vertical and horizontal dimensions .



Define the thickness of the mark's lines.

Vertical and horizontal **Offset** allows you to change the position of the mark.

Corner mark 3



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



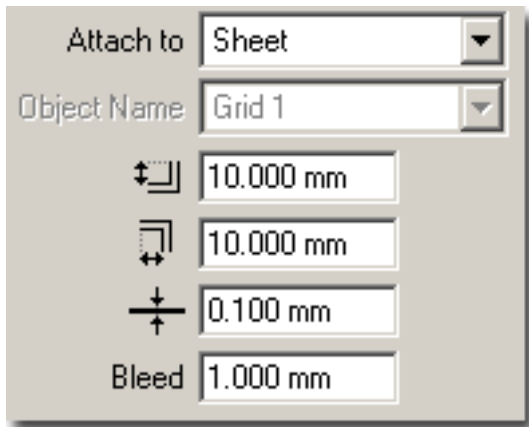
Allows you to define the vertical / horizontal distance between multiple corner lines.



Define the thickness of the mark's lines.

Bleed: Enter the bleed distance where you want to position the marks.

Corner mark 4



Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.



Allows you to define the vertical / horizontal distance between multiple corner lines.

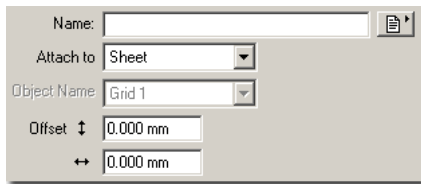


Define the thickness of the mark's lines.

Bleed: Enter the bleed distance where you want to position the marks.

From file:

If you really have specific needs, you can always design your own mark and use this option to place it as a Corner Mark.



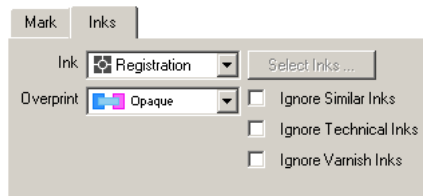
Enter the **Name** of the object you want to attach the mark to.

Attach to: Sheet, Plate, Visible objects, Objects.

Vertical and horizontal **Offset** allows you to change the position of the mark.

Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.

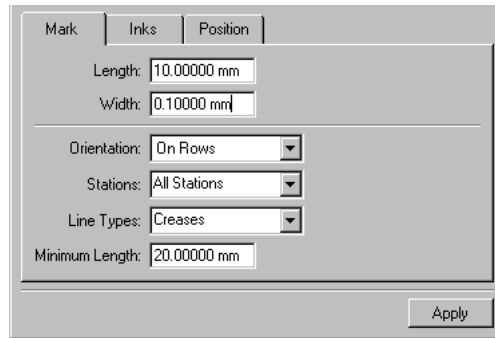


7.3.6. Crease Mark

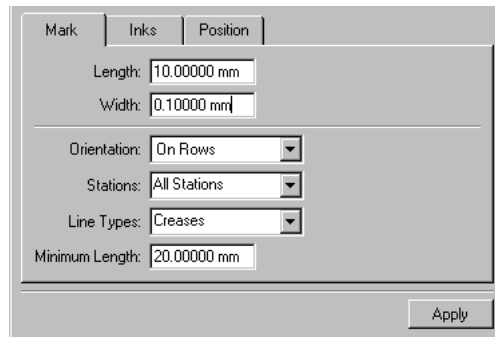
A crease mark places registration lines which line up with the crease lines in the layout..

The Crease Mark dialog box has three tabs:

- *Mark*
- *Inks*
- *Position*



Mark



Enter the settings for the crease marks:

- The Length and Width define the size of the crease lines
- The orientation: either on Rows (horizontal) or on columns (vertical)
- On what stations to base the crease marks : **All Stations** will generate crease marks for every crease line in every station. By setting to Left Column, Right Column, Top Row or Bottom Row, only the crease lines from the corresponding stations will be used.
- The line type: Creases, Cuts or Creases and Cuts
- The minimum length of a crease or cut line to be taken into account. Crease or cut lines below the minimum length will be disregarded.

Inks

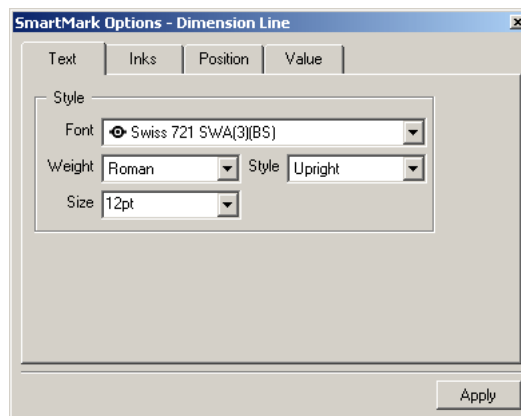
The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.

Position

The Position tab is identical to the *Position tab* of the Image Mark dialog box.

7.3.7. Dimension Line

The Dimension Line measures and displays the horizontal and/or vertical dimensions of an (named) object, job borders and/or margins.

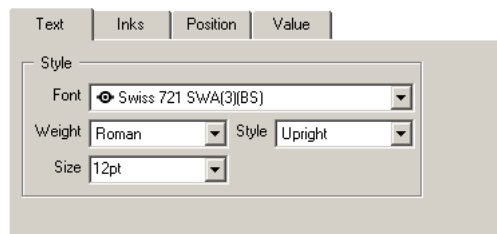


The Dimension Line dialog box has four tabs:

- *Text*
- *Inks*
- *Position*
- *Value*

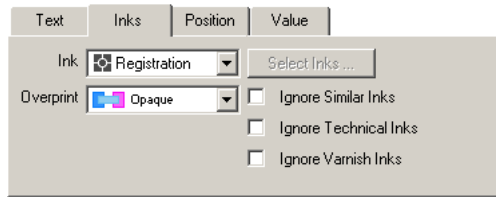
Text

Allows you to define the font, weight, size, style.

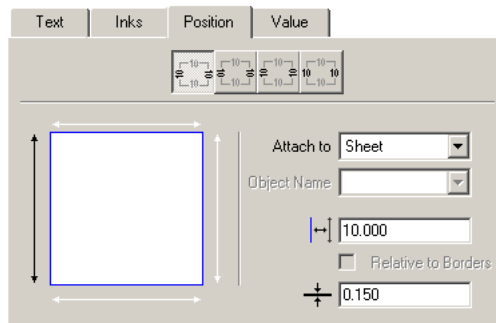


Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



Position



The area on the left allows you to indicate where you want to place the Dimension Line. Click the white arrow to position the Dimension Line, the arrows turn black.

Enter the **Name** of the object you want to attach the mark to.

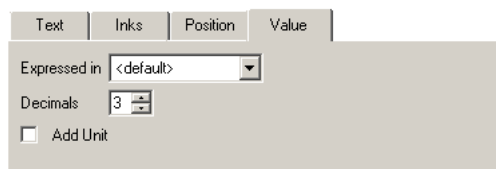
Attach to: Sheet, Plate, Visible objects, Objects.

 Define the distance between the Dimension Line and the border, margin, object.

Check **Relative to Borders** if you want to position the dimension line relative to the border. The distance specified above will be taken into account.

 Define the thickness of the mark's lines.

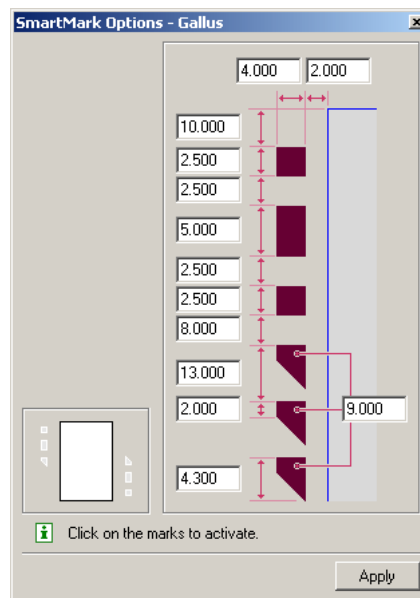
Value



Expressed in defines the units that you want use to to express the length of the dimension line. **Decimals** defines the number of decimals you want to put behind the comma. Select **Add Unit** if you want to put the unit behind the value.

7.3.8. Gallus

The gallus mark allows you to define the positioning of gallus strips left and right with your specific values.



The gallus marks can be put in two directions. On the left you see that the gallus mark starts with a square box and on the right with a triangle. If you click the white marks, they turn black, indicating they have now become active.


Tip:

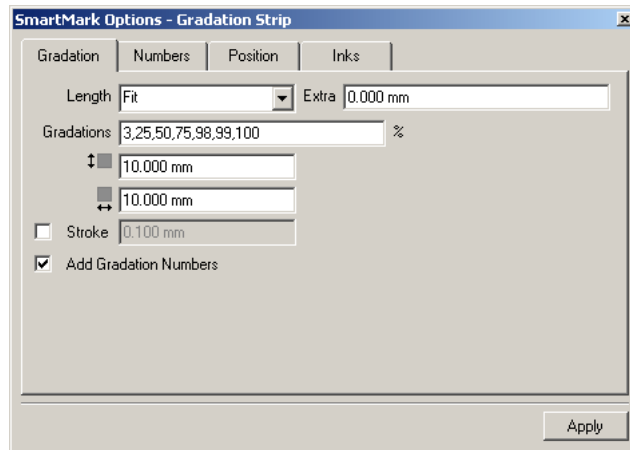
Enter negative values if you want the gallus marks to appear inside the borders.

7.3.9. Gradation Strip

The Gradation Strip allows you to create a strip for all the inks in the job.

Tip:

The Gradation Strip can be moved in your job by simply using the *Move tool*  in the Tools toolbar.

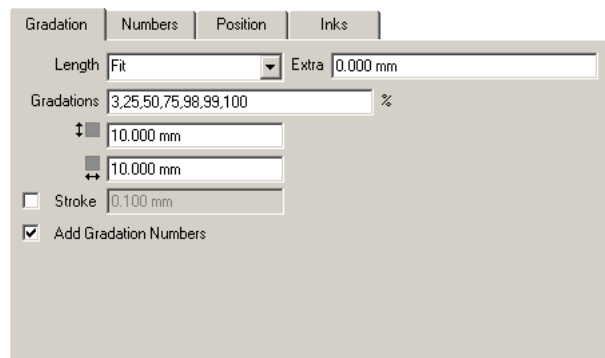


The Gradation Strip dialog box has four tabs:

- [Gradation](#)
- [Numbers](#)
- [Position](#)
- [Inks](#)

Gradation

The size of the boxes can be specified as well as the gradations for each ink. The different gradations have to be separated by a comma.



Length Choose the length of your mark from the drop-down list.



- Specific: define the specific length of the strip
- Fit: the strip fits in your job. You can add an *extra* length.
- Single copy: puts the strip once in your job, no matter how big the job is.

Gradations Allows you to specify the list of comma separated percentages and at the same time the amount of gradations boxes per ink.

In the menu the sizes of the gradation boxes are 5x5 and the gradations are 3, 25, 50, 75, 98, 99, 100. If the job has 4 inks this will generate a strip with 4 parts, a part for each ink, and per

part you will have 7 gradation boxes. These 4 parts will be repeated until the length of the strip specified in the menu is reached.

Gradation specifications   To specify the height of your gradation box.

  To specify the width of your gradation box.

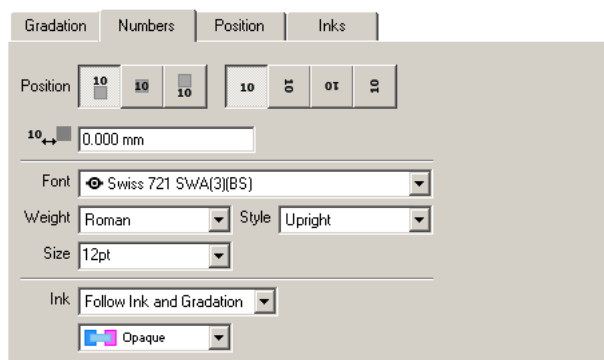
Enable **Stroke** to be able to specify a stroke width for the box.

Enable '**Add Gradation Numbers**' to be able to change the settings for gradation numbers.

Note:


As long as 'Add Gradation Numbers' is not enabled, the Numbers tab will appear dimmed.

Numbers



When you activate the 'Add Gradation Numbers' toggle in the *Gradation tab* you can access the 'Numbers' tab where you can define the Gradation Numbers' position, font and ink.

PositionThe buttons allow you to position your text to the left, on top or to the right of the gradation boxes and make it possible to select the correct orientation.

 The distance input field allows you to specify a distance from the gradation boxes.

StyleSelect the *Font* from the drop-down list. Select the *weight*: 'Roman' or 'Bold'. Select the font *style*: 'Upright' or 'Italic'. Specify the point *size*.

InkYou can choose the color of the ink an the overprint mode.

The *color* of the ink:

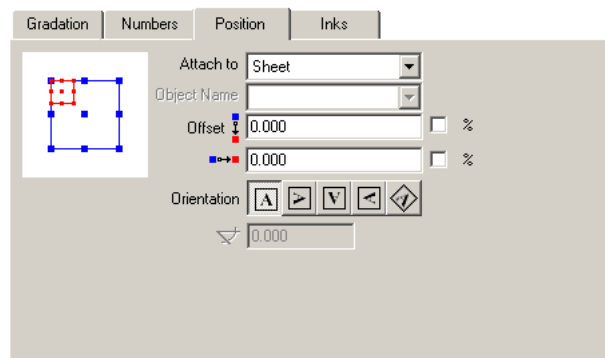
- Follow ink and gradation: the number has the same percentage of the ink of the corresponding gradation box.
- Follow ink: the number has 100% of the ink of the corresponding gradation box.
- Registration
- Darkest Ink

The *Overprint Mode*:


- Opaque
- Darken
- Postscript
- Add

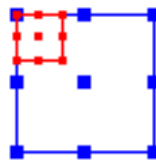
Position

The Position tab allows you to select the position of the mark by moving the red rectangle (the mark) in the blue rectangle (the job borders).




Attach to: Plate, Sheet, Visible Objects, Objects.

Use the offset options  to apply an extra offset to the chosen position using



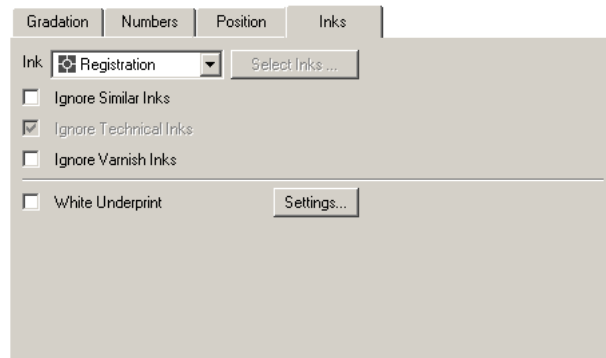
Orientation Click one of the buttons to obtain the desired orientation for your image.



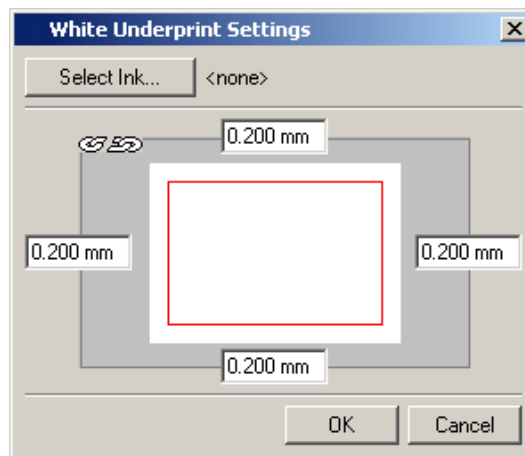
Angle  allows you to specify an angle in case you selected the rotation button in the 'Orientation' group.

Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



White underprint allows you to add white underprint to the mark. Click the 'Setting' button, the White Underprint settings dialog box pops up.



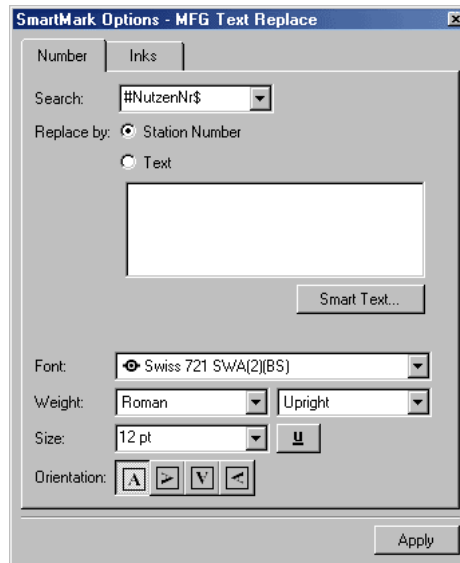
7.3.10. MFG Text replace

MFG Text Replace scans an ArtiosCAD MFG file for text objects with a specific content. This content will then be replaced by a station's number, or by user defined SmartName enabled text string.

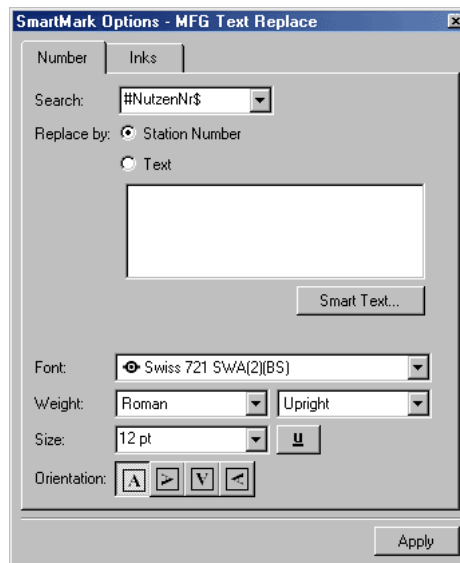
When scanning for ArtiosCAD text objects, the position and rotation angle of the text is kept, and will be used for the replacement text.

The MFG Text Replace dialog box has two tabs:

- *Number*
- *Inks*



Number



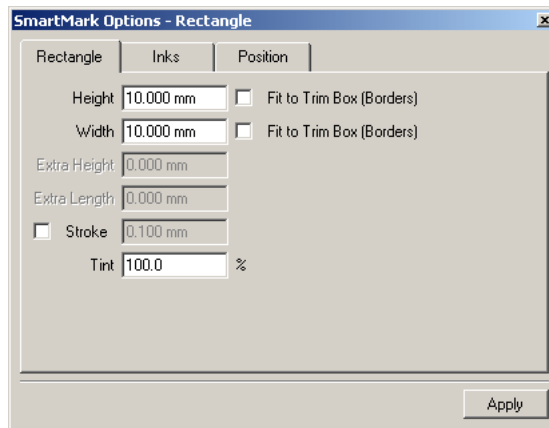
- Set the **Search** string to define the MFG text that needs to be replaced
- Set **Replace by Station Number** to replace the found text by the Station's number, or **Replace by text** to replace it by user defined text. When defining the text, you can use Smart Text by clicking the button.
- Set the Font, Weight, Size and Orientation for the text.

Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.

7.3.11. Rectangle


A rectangle mark generates a rectangle with the specified dimensions in the required ink(s).



The Rectangle Mark dialog box has three tabs:

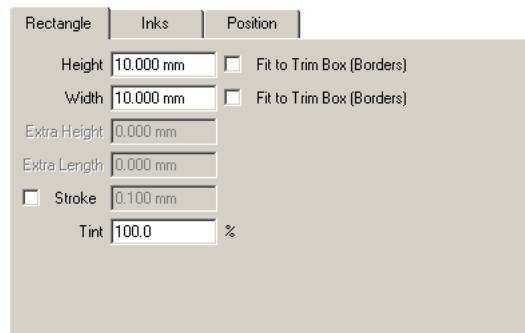
- *Rectangle*
- *Inks*
- *Position*

Tip:

The Rectangle can be moved in your job by simply using the *Move tool*  in the Tools toolbar.

Rectangle

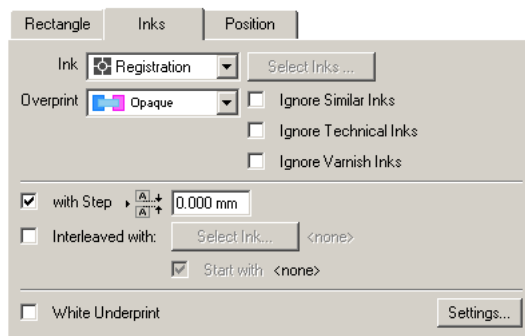
Enter the Rectangle's vertical and horizontal dimensions.



Enter the **height** and **width** of the ellipse. Select the **Fit to Trim Box (Borders)** toggle if you want the width and height to be fitted to the trim box dimensions. This will activate the **Extra Height/Length** input boxes and will allow you to make the mark as long (or wide) as the anchor object (borders, margins, Position Mark,...) of your job. You may opt to give the mark a stroke instead of a fill. Select the **Stroke** toggle and enter a width. **Tint**: Enter a percentage. If you enter 50 %, the mark will be shown using 50 percent of the inks.

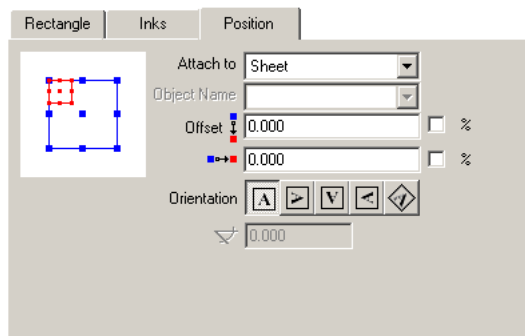
Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



Position

The Position tab is identical to the *Position tab* of the Gradation Strip Mark dialog box.



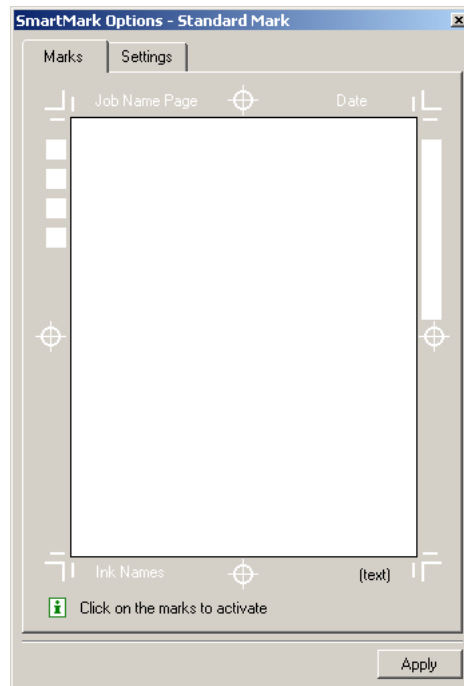
7.3.12. Standard Mark

The Standard Mark dialog box allows you to define the display of cutting marks, centermarks, bleed, ink squares and other job information.

The Standard Mark dialog box has two tabs:

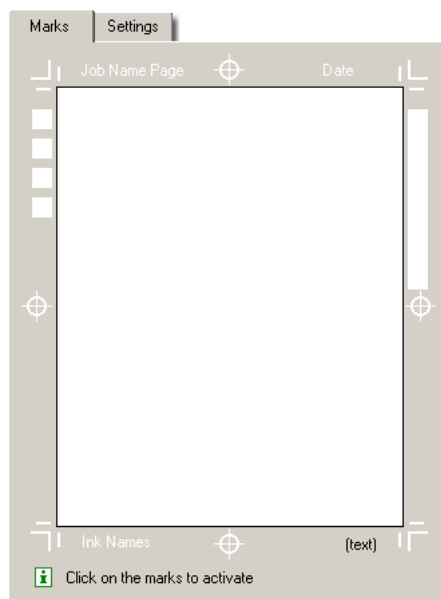
- *Marks*

- [Settings](#)



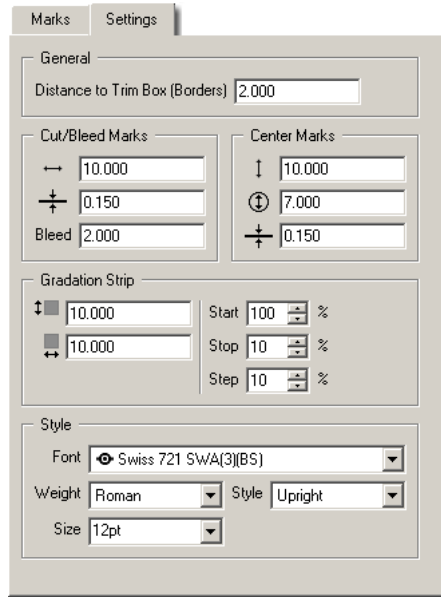
Marks

To activate a specific mark, just click it and the mark will turn black. Only the marks displayed in black will be added to your repetition. This is not the case for marks displayed in white. You can insert (black) text in the text field.



Settings

The Settings tab allows you to define size, fonts, distances of the Standard Marks.



General: Define the 'Distance to borders'.

Cut/Bleed Marks: To select the size of cut/bleed marks. To select the thickness of the cut/bleed marks. Enter a *Bleed* value if you want to change the distance between the marks and the job borders.

Center Marks To change the size of the center marks. To change the diameter of the center mark. To define the thickness of the cut/bleed marks.

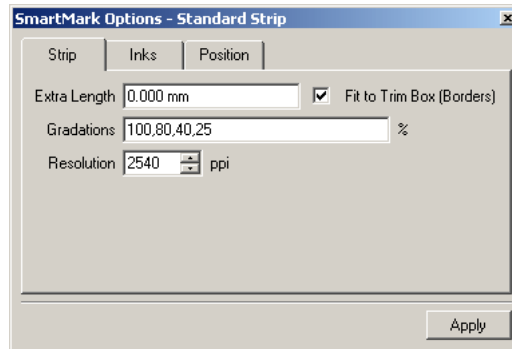
Gradation Strip To specify the height of your gradation box.

To specify the width of your gradation box. *Start*: define the percentage of the first (top) grayscale box. The default value for the start is 100%. *Stop*: define the percentage of the last (bottom) grayscale box. *Step*: define the percentage step between two grayscale boxes.

Style Select the *Font* from the drop-down list. Select the *weight*: 'Roman' or 'Bold'. Select the font *style*: 'Upright' or 'Italic'. Specify the point *size*.

7.3.13. Standard Strip


A Standard Strip mark generates an Esko-defined strip containing adjustable gradation patches in a selectable set of inks.



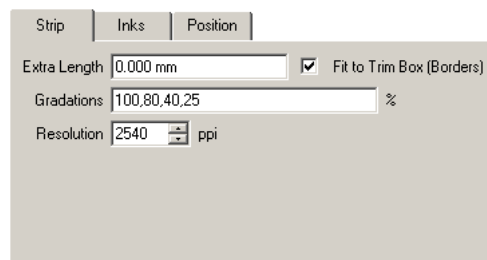
The Standard Strip dialog box has three tabs:

- [Strip](#)
- [Inks](#)
- [Position](#)

Tip:

Note that the Strip can be moved in your job by simply using the [Move tool](#)  in the Tools toolbar.

Strip



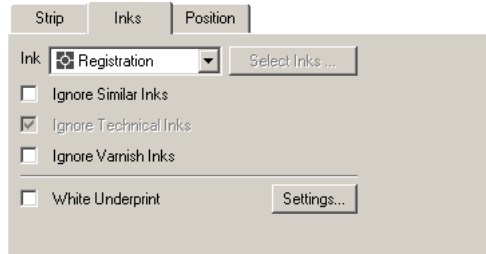
If the 'Fit to Trim Box (Borders)' box is checked, you can define **extra length**. **Gradations** allows you to specify the list of comma separated percentages and at the same time the amount of gradations boxes per ink.

In the menu the sizes of the gradation boxes are 5x5 and the gradations are 3, 25, 50, 75, 98, 99, 100. If the job has 4 inks this will generate a strip with 4 parts, a part for each ink, and per part you will have 7 gradation boxes. These 4 parts will be repeated until the length of the strip specified in the menu is reached.

Resolution allows you to specify the resolution of the mark in ppi.

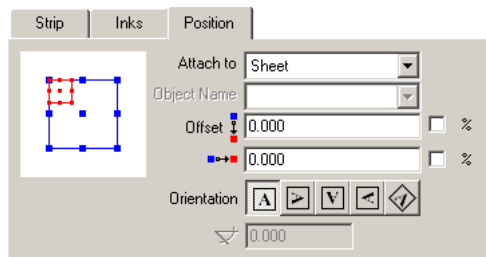
Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



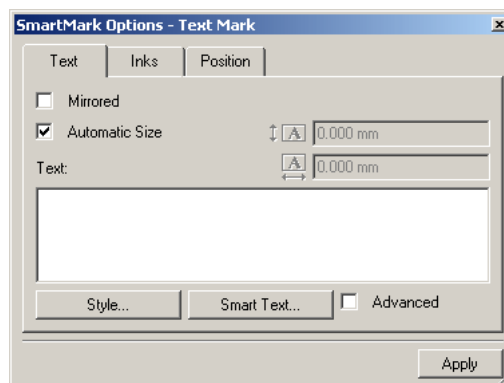
Position

The Position tab is identical to the *Position tab* of the Gradation Strip Mark dialog box.



7.3.14. Text Mark

The Text dialog box allows you to create additional text on a specified position and a certain orientation. The additional text fields will be put next to each other on the specified position. Text fields can contain any additional information that you type in and will be created in the ink selected.




The text Mark has three tabs:

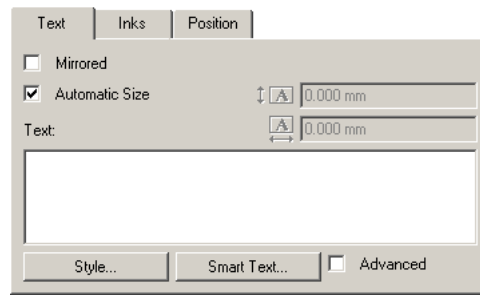
- [Text](#)
- [Inks](#)
- [Position](#)

Text

To create additional text on a specified position and orientation.

Tip:

The Text Mark can be moved in your job by simply using the [Move tool](#)  in the Tools toolbar.

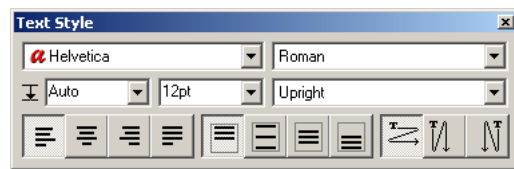


Check the **Mirrored** box if you want your text to appear mirrored. When choosing '**Automatic Size**', quadding modes will be ignored and the size of the box will depend on the size of the text. Deselect 'Automatic Size' to be able to enter text box dimensions yourself.

Click '[Style](#)' to format your text. Click '[Smart Text...](#)' if you want to use smart text.

Style...


Clicking 'Style...' opens the Text Style dialog box:



It allows you to change the:

- Font
- Font Weight
- Leading
- Size
- Style

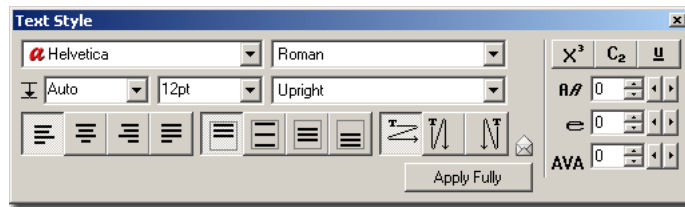
- Horizontal Quadding
- Vertical Quadding
- Text Orientation




When you select 'Advanced', the Text Style dialog box expands.  and 'Apply Fully' become available.

Note:

In 'normal' mode, style changes are always applied to the complete text. In 'Advanced' mode, style changes are inserted at the current cursor position, except when you click 'Apply Fully'. When 'Apply Fully' is clicked, the settings are applied to the mark.

When you click the envelope, you are presented with a number of extra formatting options.



- **X³** Superscript
- **C₂** Subscript
- **U** Underscore
- **R/**  Slant
- **e**  Distort
- **AVA**  Track

Smart Text...

Smart text fields are text codes that correspond with certain attributes of the job. During text lay out, these text codes are replaced by their value. If during editing these values should happen to change, the smart text field will be recalculated.

Select a category from the left column and a topic from the right column.

The categories are:

- *Ink/Color*
- *Document*
- *Date/Time*
- *Typographic*
- *User information*
- ReportMaker is used when creating ReportMaker templates (Esko PackEdge).
- SmartNames: Lists all the SmartNames known on the Automation Engine server

- [Output Parameter](#)
- [CAD data](#)
- Job SmartNames: lists all the SmartNames which are defined in the Automation Engine job on which the editor is currently working.
- [XMP data](#)

Ink / Color

Ink name	Inserts ink names, you can select the inks by the ink colors.
All ink names	Inserts the names of all the inks in your job
Ink attribute	<p>Inserts attributes of inks. You select the ink by the ink number.</p> <p>Available attributes are</p> <ul style="list-style-type: none"> • name: the name of the ink as shown in the inks dialog box • shortname: e.g. Reb for Reflex Blue • fullname: the full ink name • angle: the angle of the ink • lpi / lpmm / lpcm: the lineature of the ink • inkbook: (only for Pantone Ink books) returns C, M or U for coated, matte and uncoated inkbooks. • dot: the dotshape of the ink • group: the group (Process, PANTONE) to which the ink belongs • type: the type of ink (Normal, Opaque, Varnish, Technical) • number: the number of the ink • LAB values: the LAB value of the ink
Attributes of all inks	Inserts attributes of all the inks.
Ink coverage	Inserts the Ink coverage. Select the ink by entering the number. Choose a unit (percent, mm, cm, inches).

Document

Job name	Inserts the name of the job.
Job V size	Inserts the vertical size of the job.
Job H size	Inserts the horizontal size of the job.
Job sizes	Inserts the vertical and horizontal job sizes

Page number	Inserts the page number.
Localized Modification Date	Inserts the job's localized modification date. Choose a format: date, long date, time, ...

Date / Time

Date	Inserts the current date (dd-MMM-yy)
Time	Inserts the current time (hh:mm:ss)
Localized Date and Time	Inserts localized date and time. Choose a format: date, long date, time, ...

Typographic

Superscript	Puts text in superscript.
Subscript	Puts text in subscript.
Normal	Makes text normal (without characteristics).
Underline On	Underlines the text.
Underline Off	Counters the Underline On option.
Elastic	Inserts an elastic whitespace.
Space	Inserts an fixed white space. You can define the fixed space.
Vertical space	Inserts a vertical fixed space. You can define this space yourself.

User Information

User Name	Inserts the user name.
Workstation Name	Inserts the workstation name.

Output Parameter

These strings of information are derived from the output settings in the editor, or from the ticket used on the server. these work only when applied in a set used during output (PostScript or PDF output or during Esko FlexRip output.)

Resolution	Inserts the resolution.
Color Strategy	Inserts the used color strategy .
Source Profile	Inserts the source profile in case it is used in the marks set that is applied during proofer output.
Destination Profile	Inserts the destination profile in case it is used in the marks set that is applied during proofer output.
Screen Based DGC	Inserts the used color strategy.
DGC	Inserts the DGC settings.

Dotshape	Inserts the dotshape used in the job.
V /H Distortion	Inserts the used vertical/horizontal distortion settings used.
V/H Scale	Inserts the used vertical/horizontal scale settings used.
Device	Inserts the device name.
Configuration	Inserts the color strategy used.
Emulsion	Inserts the emulsion settings.
Image	Inserts the used image settings..
Screen Angle Direction	Inserts "CCW" (counterclockwise) when used in an expose ticket on imaging engine when screen angles are set to counterclockwise. In all other situations, it will be "CW" (clockwise).

CAD data

CAD data SmartMarks allow you to add CAD-related design, customer or board information.

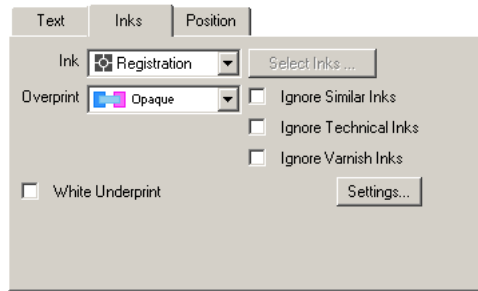
Design	Inserts data from the CAD design (if available): Description, Name, Length, Width, Depth
Customer	Inserts customer information (if available): Name, Adress1, Adress2, City
Board	Inserts data from the CAD database (if available): Code, Description, Flute, Caliper

XMP data

Artwork	Inserts Artwork information, choose from the drop-down list.
Bar codes	Inserts bar code xmp information, toggle the ones you need: type, code, bar width reduction, device compensation, magnification, magnification in percentage, X- dimension, ratio.
External References	Inserts all external references, choose what information you want (File Name, File URL, Count, Page, SmartID) a separator: dash, space, new line.
Fonts	Inserts all font names, choose a separator: dash, space, new line.
Inks	Inserts XMP ink data, toggle the ones you want: name, group, lpi, angle, dotshape, type. also choose a separator: dash, space, new line.
Layers	Inserts all layer names, choose a separator: dash, space, new line.
Job	Inserts XMP information on a Job, choose between Description, Order ID, SubOrder ID.

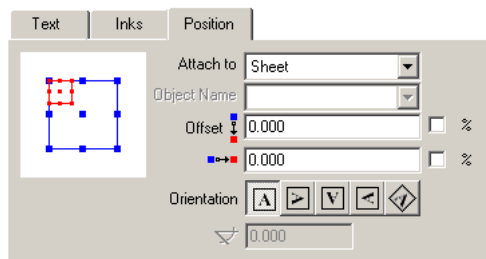
Inks

The Inks tab is identical to the [Inks tab](#) of the Center Mark dialog box.



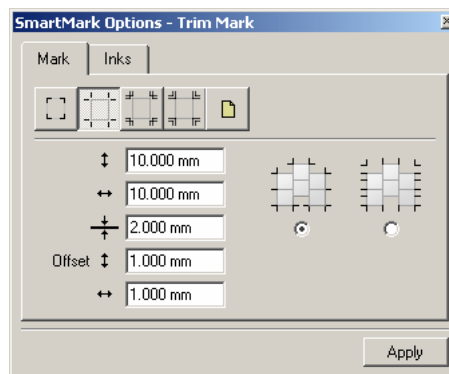
Position


The Position tab is identical to the *Position tab* of the Image Mark dialog box.



7.3.15. Trim Mark

The Trim Mark places itself only around externally placed jobs (GRS, PLA, STA). The Trim Mark has the same features as the *Corner Mark*.



When you select the second type of corner mark , an extra option becomes available. Check the left option to put trim marks around the stations, taking into account only the outer stations. Check the right option to put trim marks around the stations, but taking all stations

into account. Trim marks will be put outside the bounding box, aligned to the outer edge of the stations..

7.3.16. Bar code

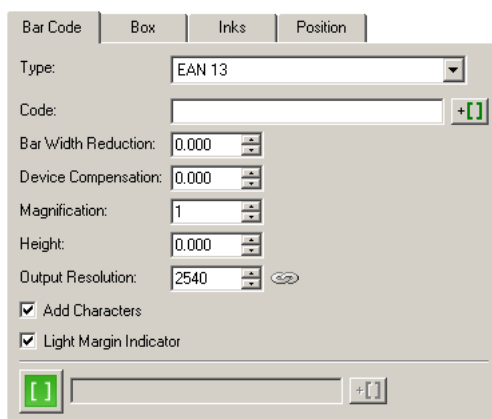
A Bar Code mark generates a bar code object.

Please check the barX manual (on the Documentation DVD or via Help) for more information on the different bar code types.

The Bar code dialog box has four tabs:

- [Bar Code](#)
- [Box](#)
- [Inks](#)
- [Position](#)

Bar Code



TypeSelect a bar code type from the list.

CodeTo enter the digits of the bar code.

Note:

When entering a wrong number of digits or wrong begin or end digits, the system displays a warning message, informing you on the required number of digits or displaying the correct code. The code is corrected automatically.

Bar Width ReductionTo adapt the bar width of the bar code. This is necessary to adapt the bar code to printing processes where the ink tends to 'bleed'. To compensate for this ink bleeding, the bars have to be a little thinner. The exact value will be provided by your printer or customer. If the printer specifies for example that the line width will increase with 0.01mm, then specify a bar width reduction of 0.01 mm. By default, the value 0 is entered for all bar codes. This means that the bar width is not adapted. A negative value in this input box results in thicker bars, a positive value in thinner bars.

Device CompensationThe in/decrease in bar width has 2 causes: the printing process (printed line is wider than what is on the plate) and the plate-making process. Here you can enter a device compensation value, The net compensation is the sum of both the device compensation and the **Bar Width Reduction**. You can leave on '0' if you do not want to compensate.

MagnificationRepresents a reduction or enlargement factor for proportional reduction or enlargement of the bar code. Entering "1" creates a 100% bar code. Entering "1.5" creates a 150% bar code. The value is an absolute factor which is automatically rounded depending on the type of bar code. As bar codes are magnified or reduced in size, the relationship between the bars remains constant The widths of individual bars and spaces however, are increased or reduced.

Note:

Since all the values in this dialog box are related proportionally, specifying a number in the Magnification input box will influence the other values in the dialog box.

HeightThe height of the bar code. The height covers the height of the bars and the digits (if any). Although the height does not contain information, it has to be sufficient to allow an efficient reading of the code.



Caution:

Do not change the height of the bar code before entering a Magnification value. The height will change again according to the Magnification factor that you define.

Characters per unitDefines the number of characters that you want to print. It is necessary to specify the number of characters per unit, because the character width depends on the printing process and the surface on which the bar code is printed.

RatioDefines the ratio between the narrowest bar, which is used as reference unit, and the other bars in the bar code.

Narrow barIndicates the width of the smallest unit of the bar code.

Wide BarDetermines the width of the wide line within the symbol.

GapDetermines the distance between lines within the symbol and interacts with the ratio: gap value = (1 + ratio) * narrow bar width / 2.

Output resolutionDefines the resolution for the exposure of the bar code. The unit is ppi. If you unlock the chain, the system will no longer take the Output resolution in account.

Add CharactersBy default, the characters are added underneath the bar code when it is created. If there is no need to add them, uncheck the box.

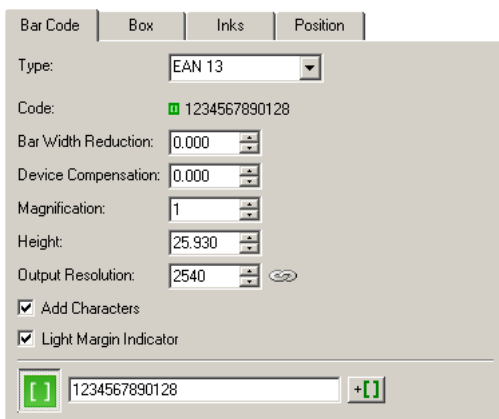
Light Margin IndicatorCheck the box to have a greater than (>) or less than (<) character printed in the human readable field of the bar code symbol, with the tip aligned with the outer edge of the light margin.

The SmartMark bar code can get (part of) its parameters from an external source through SmartNames. If this is the case, then the parameter is no longer editable and the value is preceded by a green SmartMarks icon

You have two options for using SmartNames in the SmartMark bar code dialog box:

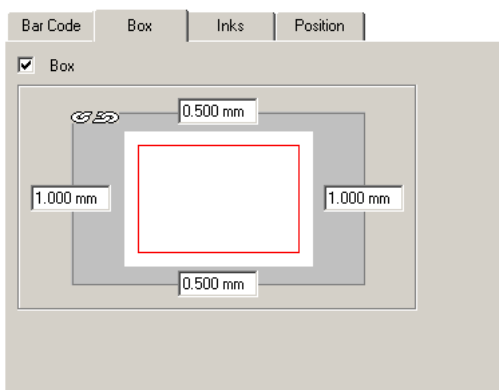
1. If the code field in the bar code SmartMark is SmartName enabled, you can immediately make this field dependant of external data sources.
2. If you want to vary other settings (bar code type, magnification...) as well, you have to use the bottom field in the bar code SmartMark dialog box. You can enter an XML line which will transfer its values to the different other fields of the bar code dialog box. The field is SmartName enabled, so you can get your XML information from an external data source.

Make sure that the syntax of this piece of XML is correct. The syntax is the same as used when specifying bar code parameters in JDF tickets. E.g. to specify type 'datamatrix' and code string 'abcdef', your data source has to return the following XML string:
`<eg:BarcodeParam egBarc:Type="DATAMATRIX" egBarc:Code="abcdef">`



Box

Check the **Box** checkbox to automatically create a box behind the bar code. The box has the size of the required light margin around the bar code. After creating the bar code with the box, they both appear selected in red. The bar code and the box can have different color properties.



Inks



Ink

- Registration
- Darkest
- Knock Out
- Custom Ink allows to pick any number of inks, either by name or by position (in the inks list).
- Custom Color allows you to pick a color, based on the ink name, ink number or color name.

Overprint

- Opaque
- Darken
- Postscript
- Add
- Reverse: Knocks out the selected ink in the underlying separation. If you select Magenta for instance, it will be knocked out in the Magenta separation.

Ignore Similar Inks: This option becomes available when you selected Registration ink

Tip:

This is useful when you have similar inks in your job, e.g. two cyans with a different angle, and you want to step all the inks but you only want to have one cyan.

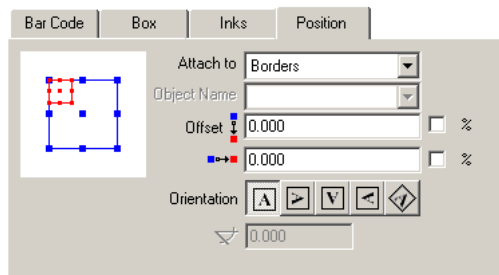
Ignore Technical Inks: Technical inks will not be used in the SmartMark. This option becomes available when you selected Registration ink.

Ignore Varnish Inks: Varnish inks will not be used in the SmartMark.


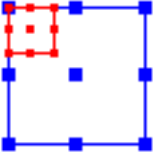
With Step repeats the mark in various inks with a step between each repetition.

Position


The Position tab allows you to select the position of the mark by moving the red rectangle (the mark) in the blue rectangle (the job borders).



Attach to: Plate, Sheet, Visible Objects, Objects.

Use the offset options  to apply an extra offset to the chosen position using  **Orientation** Click one of the buttons to obtain the desired orientation for your image.



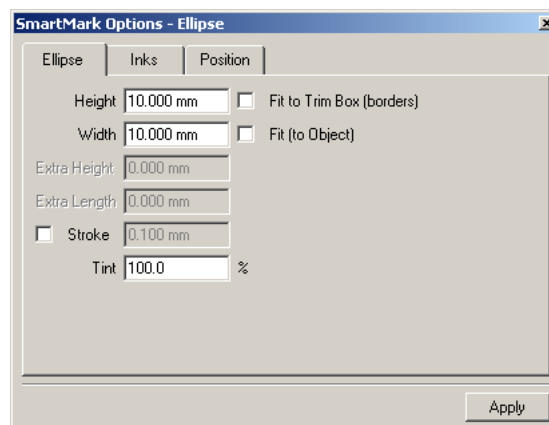
Angle  allows you to specify an angle in case you selected the rotation button in the 'Orientation' group.

7.3.17. Ellipse

An Ellipse mark will generate an oval shape (or circle when height equals width).

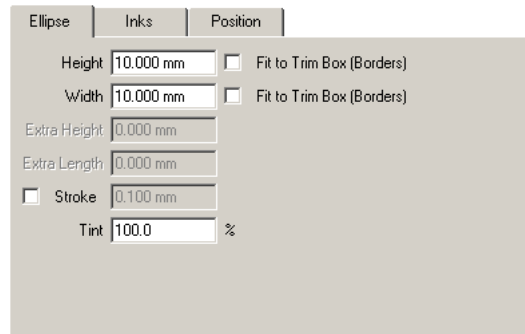
The Ellipse dialog box has three tabs:

- [Ellipse](#)
- [Inks](#)
- [Position](#)



Ellipse

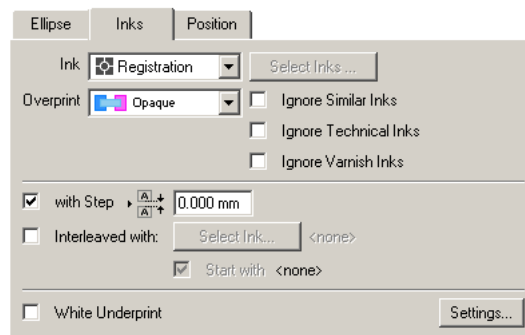
Enter the Ellipse's vertical and horizontal dimensions.



Enter the **height** and **width** of the ellipse. Select the **Fit to Trim Box (Borders)** toggle if you want the width and height to be fitted to the trim box dimensions. This will activate the **Extra Height/Length** input boxes and will allow you to make the mark as long (or wide) as the anchor object (borders, margins, Position Mark,...) of your job. You may opt to give the mark a stroke instead of a fill. Select the **Stroke** toggle and enter a width. **Tint**: Enter a percentage. If you enter 50 %, the mark will be shown using 50 percent of the inks.

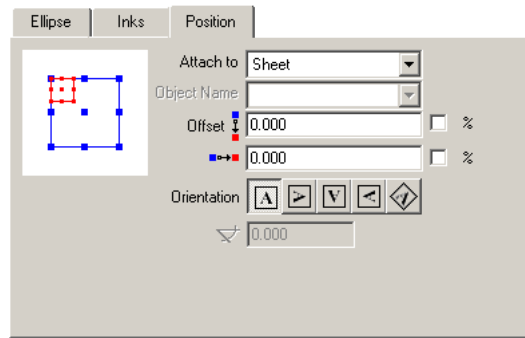
Inks

The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



Position

The Position tab is identical to the *Position tab* of the Gradation Strip Mark dialog box.



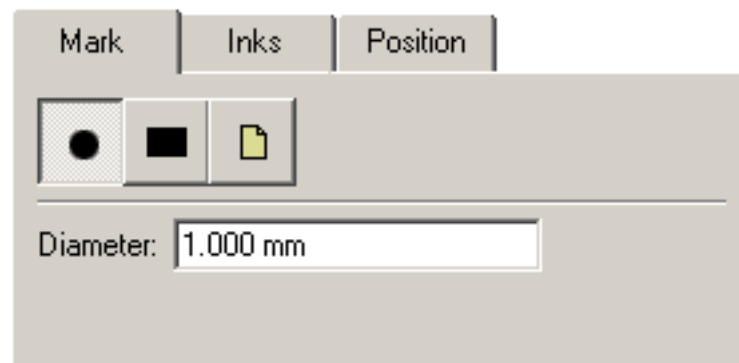
7.3.18. Grid Mark

Places marks on the vertical sides of external references. Is meant mainly for repetition blocks (microdots...).The Grid marks has three tabs:

- *Mark*
- *Inks*
- *Position*

Mark

Select the shape of the grid mark: a circle, square or an image from file. Enter the diameter, the dimensions or the scale.

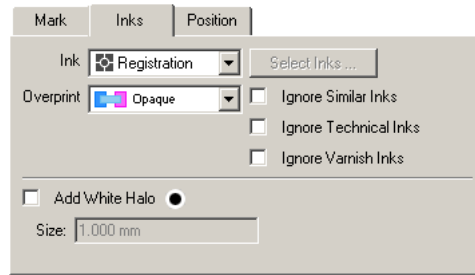


When using an image from file, you can define the name for the file and the **Preserve Screening** option.

The **Preserve Screening** option allows to place the mark in all separations of the current job, but keep the screening (dot shape, lpi, angle) from the original image.

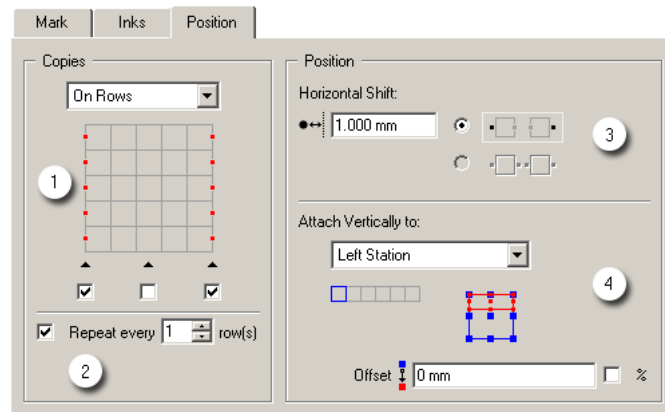
Inks



The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



Check **Add White Halo** to create a white halo effect around the grid mark.

Position



1. **a.** Define the horizontal or vertical copies. Select 'On Rows' or 'On Columns' from the drop-down list. Depending on what you select, the appearance of the other options will change.
 - b.** Indicate if the mark should be put on the left/ right and / or middle stations.
2. Indicate how frequent the mark should be repeated. Should it appear on every row, or only every every x rows?
3. Specify the position of the marks. Define the horizontal or vertical shift. The interface will change according to the selection in the 'Copies' group.
 -  only affects the marks attached to the left and right stations.
 -  a shift is also applied on the marks on the middle stations.
4. The vertical position of the marks: attach them to the left station/column, to the right station/column, to each station relative to the bounding box of each individual station (particularly useful in the case of staggered grids) or to the sheet, in the scan direction relative to the sheet..

Select the relative position of the mark by moving the red rectangle (the mark) in the blue rectangle (the stations). Define an offset if needed.

You can define the Offset from center of the mark to the edge of the station (default), or from the edge of the mark to the edge of the station.



Tip:
Check the box to be able to specify an offset in percentage.

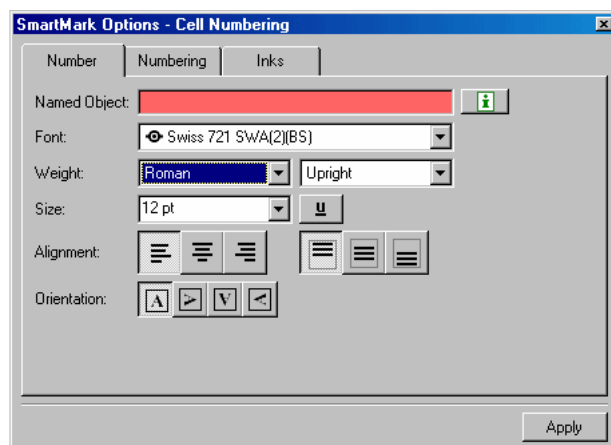
7.3.19. Cell Numbering

Places station numbers on the position defined by the Position Mark. The Cell Numbering has three tabs:

- *Number*
- *Numbering*
- *Inks*

Number

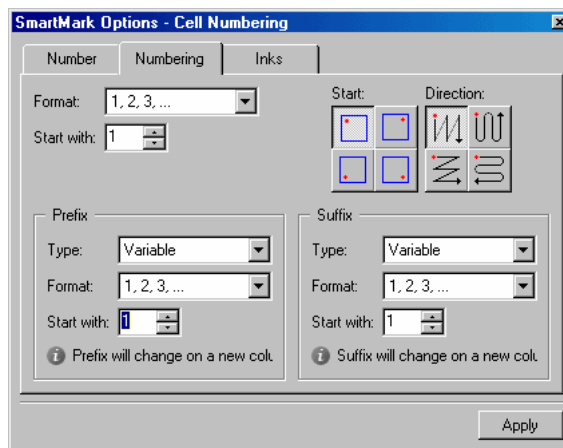
Select the Position Mark (Named Object) to define the position of the Cell Number. The info button allows to scan all placed external references for Position Marks, and all the names of the Position Marks will become available in a dropdown list to easily select the correct Position Mark. Enter the font, weight, size, alignment and orientation for the number.



Numbering

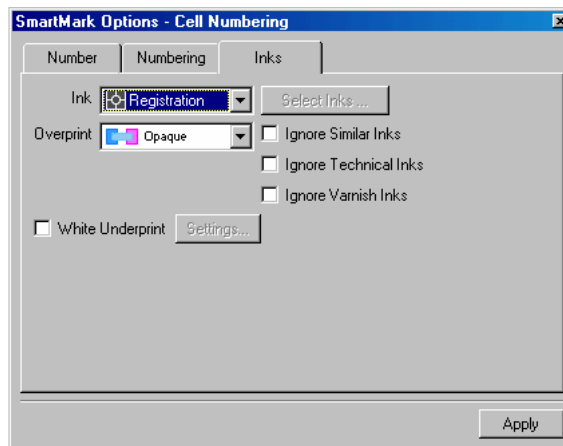
Define the Format and start value for the Cell numbering. Use the buttons to define the start point and direction.

You can add a Prefix and a Suffix.



Inks

Use the Inks tab to define the color of the Cell Numbering. The Inks tab is identical to the *Inks tab* of the Center Mark dialog box.



Check **White Underprint** to create a white underprint underneath the Cell Number.

7.3.20. Grommet Mark

Places marks around the anchor point (mostly the plate) by dividing the height and width..The Grommet Mark has three tabs:

- [Marks](#)
- [Inks](#)
- [Position](#)

Marks

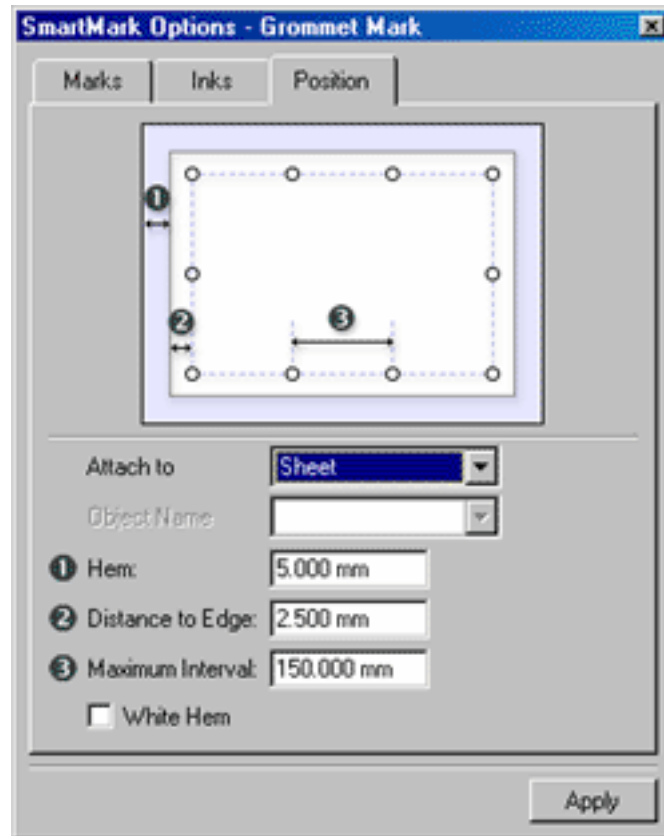
Select the type of mark, and its settings. The Marks tab is similar to the [Marks tab](#) of the Center Mark dialog box..

Inks

Use the Inks tab to define the color of the Cell Numbering. The Inks tab is identical to the [Inks tab](#) of the Center Mark dialog box.

Position

Define the position of the marks:



1. Define the “hem” (the width of the border)
2. Define the distance of the marks to the edge
3. Define the Maximum interval.

Grommet Marks can be used to print marks where to put grommets on a poster. These grommets can be used to mount a poster on a frame.

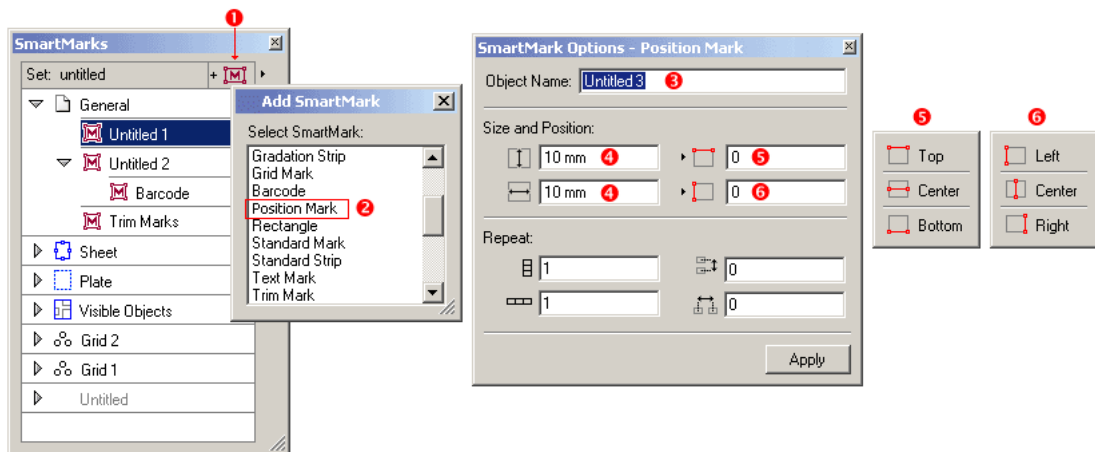
7.3.21. Position Mark

A Position Mark generates uncolored objects which have a name assigned (Position Marks). It generates a placeholder object that will be used as an anchor point for other SmartMarks. Position Marks can be positioned by means of formulas. In other words, any SmartMark can be positioned using formulas in a 2 step approach:

1. Create a Position Mark using the requested formulas.
2. Place the requested SmartMark relative to this Position Mark.

Note:

Position Marks will only appear in Esko Plato.



1. Click the "Add SmartMark" icon



2. Select the "Position Mark" SmartMark. The Position Mark dialog box will appear.
3. A default name is suggested but you are prompted to specify a meaningful name.
4. A default size is specified.
5. Define the vertical position of the Position Mark (default is 0).
6. Define the horizontal position of the object (default is 0).

7.3.22. Fill Placeholder

The **Fill Placeholder** SmartMark will search for placeholders in a CAD based layout, and generate objects, either a SmartMark set or Station Numbers, on the locations of these placeholders.

The Placeholder Types

- **Named Objects** inside the PDF graphics used in the stations.
 - The **Object Name** is used to search the Placeholder. The name of all Named Objects used in the PDF graphics in one or more stations will be shown in the dropdown list.
 - The bounding box of the Named Object is used as frame for the placeholder. In case the object is a text object or a SmartMark object, the rotation angle will be used for the placeholder.
- **Text Objects** in the MFG CAD object.
 - The **Text Content** is used to search the Placeholder. The content of all text objects from the embedded designs will be shown in the drop down list.
 - The frame of the placeholder will be a single point, the anchor point of the text, with a frame size of zero. The rotation angle of the ArtiosCAD text object will be used.
- **Artwork Panels** in the embedded designs of the MFG CAD object.

- The **Panel Name** is used to search the Placeholder. All the artwork panels from the embedded designs will be shown in the drop down list.
- The bounding box of the Artwork Panel is used as frame for the placeholder, taking in account the offset for the artwork pale, and the orientation of the artwork panel.
- **Dynamic Art** in the (embedded designs of the) MFG CAD object.
 - The **Dynamic Art** name is used to search the Placeholder. A concatenation of both text labels used in the properties of all dynamic art object will be shown in the drop down list.
 - The bounding box and rotation of the Dynamic Art object will be used for the frame and rotation of the placeholder.

Fill with Station Numbers

The color and text format of the station number can be set in the **Inks** and **Number formats** tabs, using the same settings as for the cell numbering SmartMark. See [Cell Numbering](#) on page 225

The sequence of the station numbers can be set in the **Numbering** tab. These settings are also the same as for the [Cell Numbering](#) SmartMark.

The 'Default Ordering' is the station sequence as set in the current layout. When **Use Sequence from MFG Layout** is enabled, the sequence as defined in the MFG file is used.

Note: If certain stations don't have a matching placeholder (e.g. the PDF doesn't contain the Named Object), no text object will be generated, but the station number sequence will include those stations, so the corresponding station numbers will be missing.

Fill with SmartMark set

When filling the placeholder with a SmartMark set, you can select any of the existing SmartMark sets from the dropdown list.

The SmartMark set is loaded on the location of each of the selected placeholders. The frame of the placeholder is used as borders for the SmartMark set, using all inks and SmartNames of the complete document.

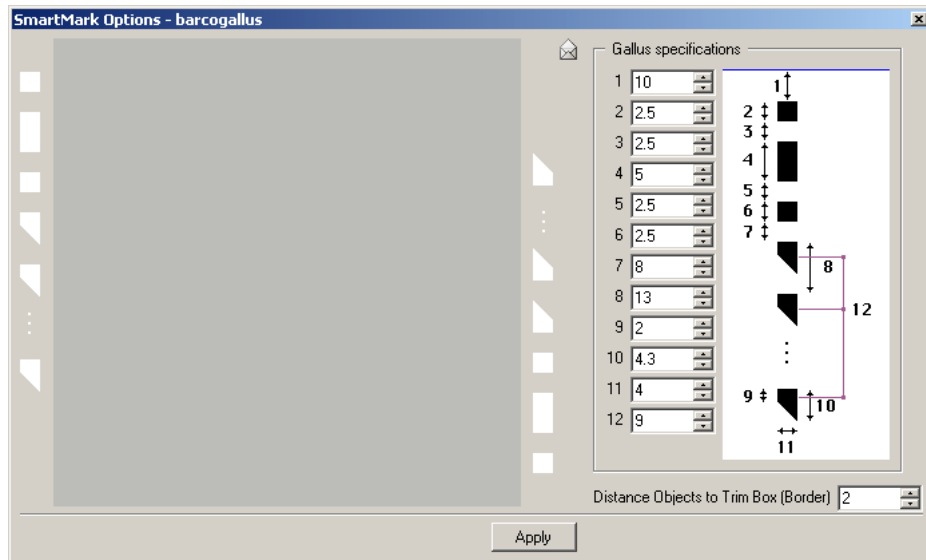
Note: When using text objects as placeholder, there is no frame, so the borders as seen by the SmartMark Set will be zero.

For SmartNames, the SmartMark set will use the context of the graphic in which the mark is placed. This means that for example the "File" SmartName will resolve into the name of the graphic of each station, and not the name of the PDFPLA file.

7.3.23. GRQ Gallus

The Barcogallus mark allows you to define the positioning of gallus strips left and right with your specific values. Barcogallus marks can be applied on Sheet and Plate level.

The gallus marks can be put in two directions. On the left you see that the gallus mark starts with a square box and on the right with a triangle. If you click the white marks, they turn black, indicating they have now become active.



If you click the Barcogallus envelope the menu extends and gives you the possibility to change the values.

The numbered list is explained by the drawing next to it.

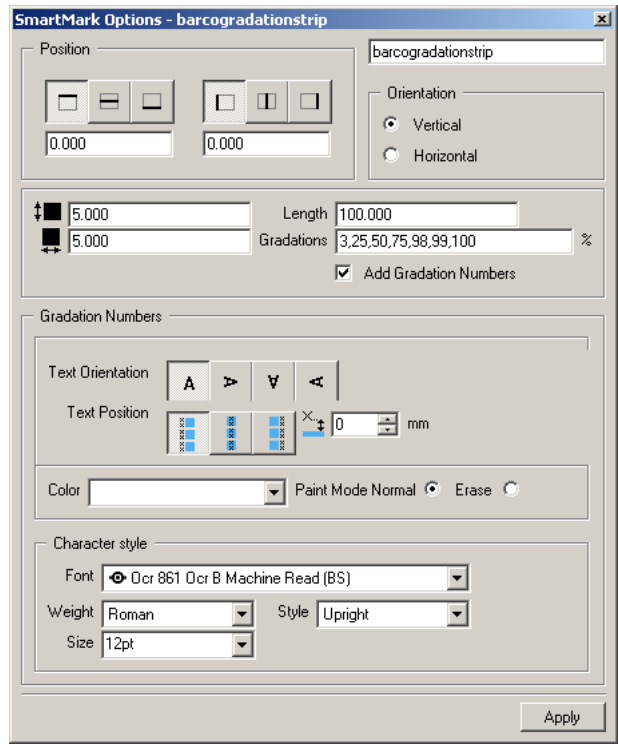
Distance objects to Trim Box (Border) allows you to define the distance of all the objects displayed to the borders.

Tip:

Enter negative values if you want the gallus marks to appear inside the borders.

7.3.24. GRQ Gradation Strip

The Barcogradationstrip allows you to create a strip for all inks in the job. The size of the boxes can be specified as well as the gradations for each ink. The different gradations have to be separated by a comma. The Barcogradationstrip can be customized. Enter your values and enter a name for it in the Customized name input box.



Position Puts the top of the mark against the vertical origin of your repetition, providing the orientation is set to zero.

Puts the center of the mark on the vertical origin of your repetition, providing the orientation is set to zero.

Puts the bottom of the mark against the vertical origin of your repetition, providing the orientation is set to zero.

Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition

Puts the left side of the mark against the horizontal origin of the repetition, providing the orientation is set to zero.



Puts the center of the mark on the horizontal origin of the repetition, providing the orientation is set to zero.

Puts the right side of the mark against the horizontal origin of the repetition, providing the orientation is set to zero.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition

Orientation The *Horizontal* and *Vertical* option buttons allow you to position the mark horizontally or vertically.

Gradation specifications   To specify the height of your gradation box.

  To specify the width of your gradation box.

Specify the '*Length*' of the gradation strip. Use '*Gradations*' to specify a list of comma separated percentages and at the same time the amount of gradations boxes per ink.

In the menu the sizes of the gradation boxes are 5x5 and the gradations are 3, 25, 50, 75, 98, 99, 100. If the job has 4 inks this will generate a strip with 4 parts, a part for each ink, and per part you will have 7 gradation boxes. These 4 parts will be repeated until the length of the strip specified in the menu is reached.

Enable '*Add Gradation Numbers*' to be able to change the settings for gradation numbers.

Note:

As long as '*Add Gradation Numbers*' is not enabled, all options for gradation numbers appear dimmed.

Gradation Numbers Select the correct *Text Orientation*. *Text Position*: The buttons allow you to position your text to the left, on top or to the right of the gradation boxes. The distance input field allows you to specify a distance from the gradation boxes. To specify the gradation numbers *Color*.

- Follow ink + %: the number has the same percentage of the ink of the corresponding gradation box.
- Follow ink: the number has 100% of the ink of the corresponding gradation box.
- Registration Black
- Darkest Ink
- Custom Ink: an ink that is used in the file.

Paint Mode Select 'Normal' to set the paint mode for the image to normal (overprint). Select 'Erase' to set the paint mode for the image to erase (erase=knocking out white).

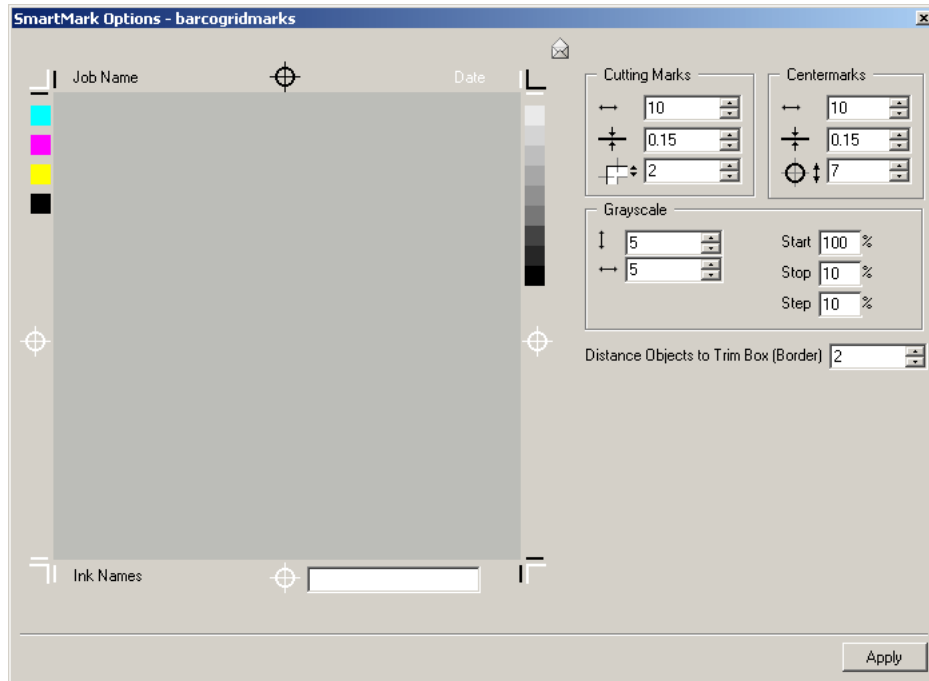
Tip:


Use 'erase' in combination with the Text Position button on top.


Character Style Select the *Font* from the drop-down list. Select the *weight*: 'Roman' or 'Bold'. Select the font *style*: 'Upright' or 'Italic'. Specify the point *size*.


7.3.25. GRQ Grid Marks

Grid marks allow you to define the display of cutting marks, centermarks, bleed, ink squares and other job information. To activate a specific mark, just click it and the mark will turn black. Only the marks displayed in black will be added to your repetition. This is not the case for marks displayed in white. You can insert (black) text in the text field. If you click the envelope the Extended marks dialog box appears, which allows you to influence the settings for the marks that are displayed in black.



Bleed  Click the 90 degree corner mark to display bleed marks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks.


Cutting marks  Click the Cutting marks to display the cutting marks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks. Click the Barcomarks envelope to display the Extended Barcomarks dialog box which allows you to influence the settings for Cutting Marks. You can define the length and thickness of the cutting marks and specify the distance from the cutting marks to the borders.


Centermarks  Click the Centermarks to display the centermarks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks. Click the Barcomarks envelope to display the Extended Barcomarks dialog box which allows you to influence the settings for Centermarks Marks. You can define the length of the rules of the centermarks as well as the the rule thickness. You can also specify the diameter of the circle of the centermarks.

Click **Job Name** to add the Job Name to the repetition. The mark turns black. The marks displayed in black will be added to you repetition.

Click **Date** to display the Date on your job. The mark turns black. The marks displayed in black will be added to you repetition.

Use the **Information Input box** to add any extra information you want to display on your job.

Inks  Click the white ink strip (left) to display a 100% color box of all the inks used in the job. Only if the strip displays color information it will be added to the repetition

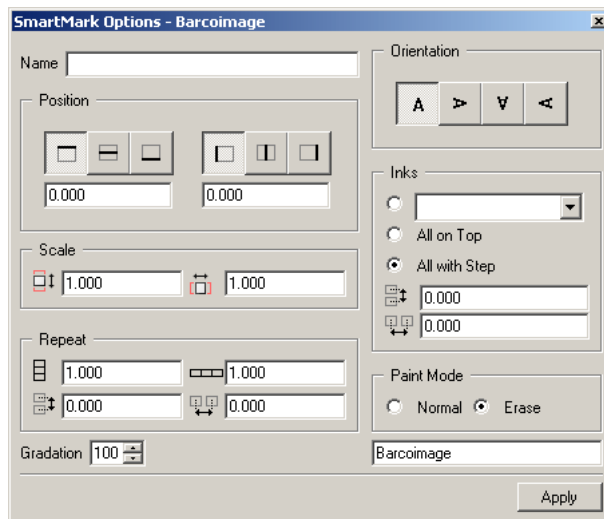
Grayscale  Click the white grayscale strip (right) to add it to your job. Only if the strip displays grayscale information it will be added to the repetition. Click the Barcomarks envelope to display the Extended Barcomarks dialog box which allows you to influence the percentages of the grayscale strip.

Specify the height and width of the grayscale boxes. *Start* allows you to define the black percentage of the first (top) grayscale box. The default value for the start is 100%. *Stop* allows you to define the black percentage of the last (bottom) grayscale box. The default value for the stop is 10%. *Step* allows you to define the percentage step between two grayscale boxes.

Distance objects to Trim Box (Border) allows you to define the distance of all the objects displayed to the borders.

7.3.26. GRQ Image

The Barcoimage dialog box allows you to specify an image as a mark and to define amongst others, its position and orientation.





Name Enter the name of the file that you want to use as a mark.


- Type the file name of the image directly, provided that the file exists in the current directory.
- Right-click the input box to open the file selector, allowing you to select the path and the name of your image.

An lp, lc, grs, gro, grc and gra file can be used. If you want to use gr* files in this menu, the inks from the gr* files will be the inks for this marks, no matter what you specify in the inks part of the menu. If the gr* file has no ink, the inks from the Barcoimage menu will be taken.


If you want a different image name for all inks, create as many images as there are inks: e.g. test1.lc, test2.lc, test3.lc, ... Then enter in the Name input box 'test*.lc'. The first ink will get test1.lc, the second ink gets test2.lc, the third ink gets test3.lc,... Keep in mind that 'test.lc' has to exist. If not, it is not possible to enter test*.lc in the Name input box.


Position  Puts the top of the mark against the vertical origin of the repetition.


 Puts the center of the mark on the vertical origin of the repetition.

 Puts the bottom of the mark against the vertical origin of the repetition.


Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition.

 Puts the left of the mark against the horizontal origin of the repetition.

 Puts the center of the mark against the horizontal origin of the repetition.


 Puts the right of the mark against the horizontal origin of the repetition.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition.

Scale  Enter a scale factor in the corresponding input box to scale your image vertically.

Note:

1 is equal to 100% of the image size. 0.5 is equal to 50% of the image size.

 Enter a scale factor in the corresponding input box to scale your image horizontally.

Note:


1 is equal to 100% of the image size. 0.5 is equal to 50% of the image size.

Repeat Use the Repeat options to specify the repeat count and step value for the Barcoimage. The Esko Plato parameters and formulas of the calculator can be used.

 To specify the vertical count of your image.

 To specify the vertical step distance between the stepped images.


 To specify the horizontal count of your image.


 To specify the horizontal step distance between the stepped images.

Gradation Gradation allows you to specify the ink percentage for the Barcoimage.

Orientation Click one of the buttons to obtain the desired orientation for your image.

Inks The '*Ink input box*' option button allows you to select one ink by its name or ink number. The drop-down list displays all inks available in the QuickStep file. When you select ink number 1, the first ink of the ink menu of QuickStep is chosen. The '*All on Top*' option button allows you to specify the selected image 'All on Top' in all inks. The '*All with Step*' option button will force QuickStep to first execute the menu settings and then step the complete ink block.

 To specify the vertical step distance between the stepped colors.

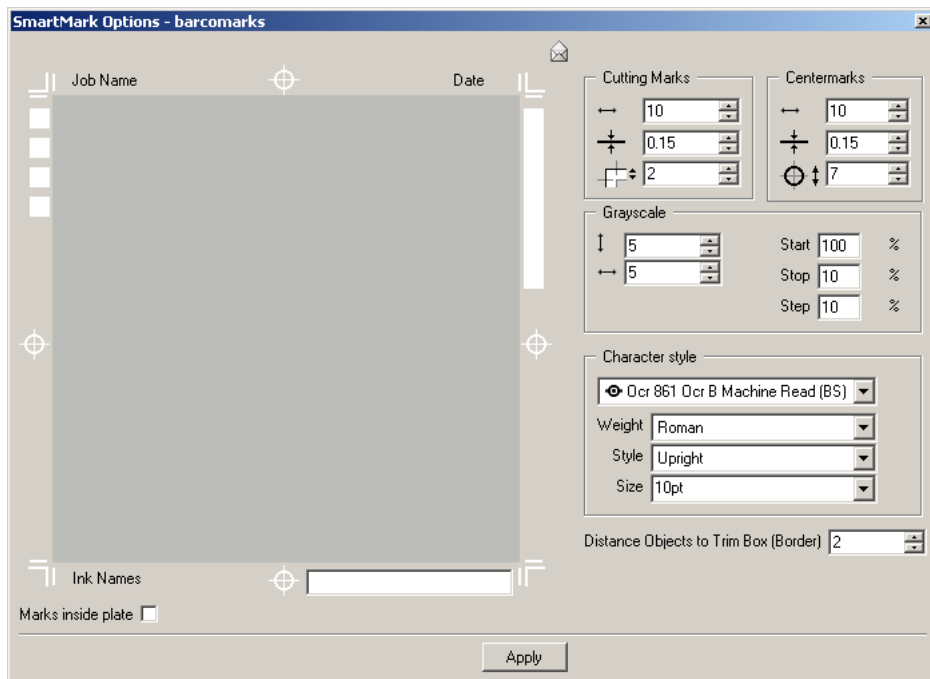
 To specify the horizontal step distance between the stepped colors.


Paint Mode Select 'Normal' to set the paint mode for the image to normal (overprint). Select 'Erase' to set the paint mode for the image to erase (erase=knocking out white).


Use the **Customized Name Input box** to give a customized name to the Barcoimage mark. Enter the customized name, click 'Apply' and the new mark name appears in the list of marks.

7.3.27. GRQ Marks


GRQ marks allow you to define the display of cutting marks, centermarks, bleed, ink squares and other job information. To activate a specific mark, just click it and the mark will turn black. Only the marks displayed in black will be added to your repetition. This is not the case for marks displayed in white. You can insert (black) text in the text field. If you click the envelope the Extended marks dialog box appears, which allows you to influence the settings for the marks that are displayed in black.



Bleed  Click the 90 degree corner mark to display bleed marks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks.

Cutting marks  Click the Cutting marks to display the cutting marks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks. Click the Barcomarks envelope to display the Extended Barcomarks dialog box which


allows you to influence the settings for Cutting Marks. You can define the length and thickness of the cutting marks and specify the distance from the cutting marks to the borders.


Centermarks  Click the Centermarks to display the centermarks on your job. The marks displayed in black will be added to you repetition. This is not the case for white marks. Click the Barcomarks envelope to display the Extended Barcomarks dialog box which allows you to influence the settings for Centermarks Marks. You can define the length of the rules of the centermarks as well as the the rule thickness. You can also specify the diameter of the circle of the centermarks.

Click **Job Name** to add the Job Name to the repetition. The mark turns black. The marks displayed in black will be added to you repetition.

Click **Date** to display the Date on your job. The mark turns black. The marks displayed in black will be added to you repetition.

Use the **Information Input box** to add any extra information you want to display on your job.

Inks  Click the white ink strip (left) to display a 100% color box of all the inks used in the job. Only if the strip displays color information it will be added to the repetition

Grayscale  Click the white grayscale strip (right) to add it to your job. Only if the strip displays grayscale information it will be added to the repetition Click the Barcomarks envelope to display the Extended Barcomarks dialog box which allows you to influence the percentages of the grayscale strip.

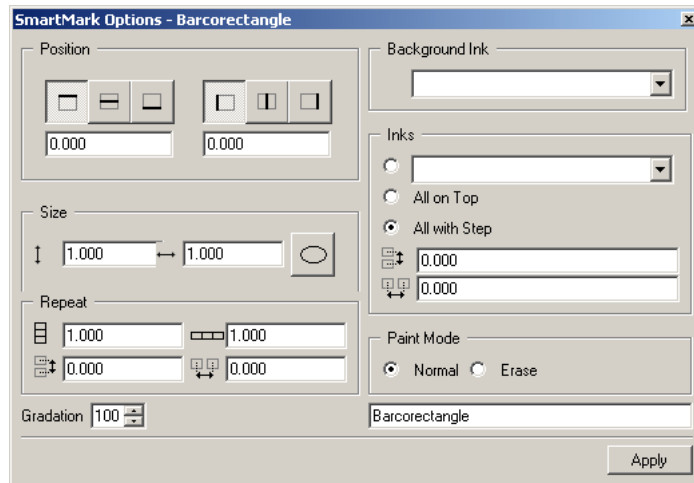
Specify the height and width of the grayscale boxes. *Start* allows you to define the black percentage of the first (top) grayscale box. The default value for the start is 100%. *Stop* allows you to define the black percentage of the last (bottom) grayscale box. The default value for the stop is 10%. *Step* allows you to define the percentage step between two grayscale boxes.

Distance objects to Trim Box (Border) allows you to define the distance of all the objects displayed to the borders.


Enable **Marks inside plate** to put the marks inside the plate.

7.3.28. GRQ Rectangle

The Barcorectangle dialog box is similar to the barcoimages dialog box. You can specify the size of the rectangle or ellipse. The Esko Plato parameters and formulas of the calculator can be used in all relevant input boxes.





Position  Puts the top of the mark against the vertical origin of the repetition.


 Puts the center of the mark on the vertical origin of the repetition.

 Puts the bottom of the mark against the vertical origin of the repetition.


Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition.


 Puts the left of the mark against the horizontal origin of the repetition.

 Puts the center of the mark against the horizontal origin of the repetition.

 Puts the right of the mark against the horizontal origin of the repetition.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition.

Size  Enter the vertical and horizontal size of the rectangle.


 Press the ellipse button if you want to use an ellipse instead of a rectangle.

Repeat Use the Repeat options to specify the repeat count and step value for the Barcoimage. The Esko Plato parameters and formulas of the calculator can be used.

 To specify the vertical count of your image.

 To specify the vertical step distance between the stepped images.

 To specify the horizontal count of your image.

 To specify the horizontal step distance between the stepped images.

Gradation Gradation allows you to specify the ink percentage for the Barcoimage.

Background Ink Select an ink by its name or its number in order to define a background color for each rectangle that you create.

Inks The 'Ink input box' option button allows you to select one ink by its name or ink number. The drop-down list displays all inks available in the QuickStep file. When you select ink number 1, the first ink of the ink menu of QuickStep is chosen. The 'All on Top' option button allows you to specify the selected image 'All on Top' in all inks. The 'All with Step' option button will force QuickStep to first execute the menu settings and then step the complete ink block.



To specify the vertical step distance between the stepped colors.



To specify the horizontal step distance between the stepped colors.

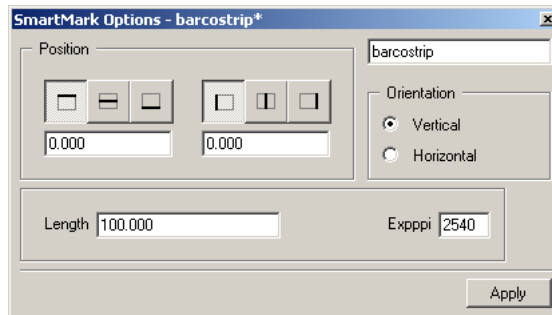
Paint Mode Select 'Normal' to set the paint mode for the image to normal (overprint). Select 'Erase' to set the paint mode for the image to erase (erase=knocking out white).

Use the **Customized Name Input box** to give a customized name to the Barcorectangle mark. Enter the customized name, click 'Apply' and the new mark name appears in the list of marks.

Example: Enter the new name for the Barcorectangle, e.g. Ellipse. When the dialog box closes, the original name in the list of marks has changed to: Ellipse [barcorectangle].

7.3.29. GRQ Strip

The Barcostrip dialog box allows you to position a predefined control strip.



Position Puts the top of the mark against the vertical origin of the repetition.



Puts the center of the mark on the vertical origin of the repetition.



Puts the bottom of the mark against the vertical origin of the repetition.

Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition.



Puts the left of the mark against the horizontal origin of the repetition.



Puts the center of the mark against the horizontal origin of the repetition.

Puts the right of the mark against the horizontal origin of the repetition.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition.

Orientation The *Horizontal* and *Vertical* option buttons allow you to position the mark horizontally or vertically.

Length allows you to specify the length of your gradation strip.

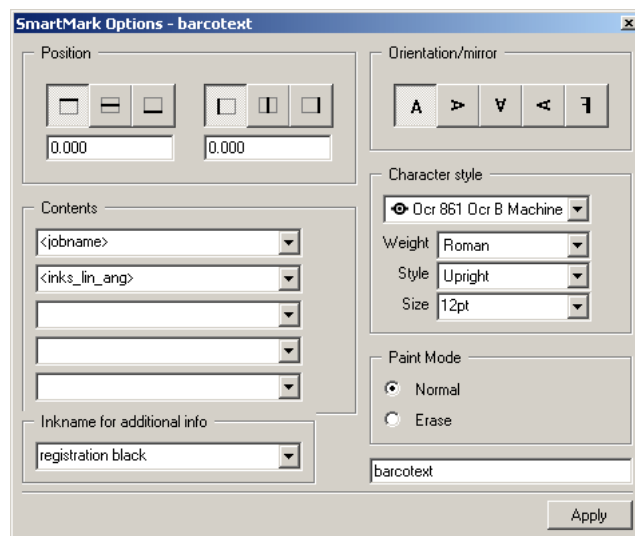
Exppi allows you to specify the Expose resolution.

Use the **Customized Name Input box** to give a customized name to the Barcostrip mark. Enter the customized name, click 'Apply' and the new mark name appears in the list of marks.

Example: Enter the new name for the Barcostrip name, e.g. Mycontrolstrip. When the dialog box closes, the original name in the list of marks has changed to: Mycontrolstrip [barcostrip].

7.3.30. GRQ Text

The Barcotext dialog box allows you to create additional text on a specified position and a certain orientation. The additional text fields will be put next to each other on the specified position. Text fields can contain any additional information that you type in and will be created in the ink selected.



Position Puts the top of the mark against the vertical origin of the repetition.

Puts the center of the mark on the vertical origin of the repetition.

Puts the bottom of the mark against the vertical origin of the repetition.

Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition.

- Puts the left of the mark against the horizontal origin of the repetition.
- Puts the center of the mark against the horizontal origin of the repetition.
- Puts the right of the mark against the horizontal origin of the repetition.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition.

ContentsThe Contents input boxes allow you to use predefined settings or simply enter any text that you want to see displayed as additional information. This information will be created in the selected ink. Predefined names can be selected from the drop-down list.The following predefined names are available:

- #jobname# will display the name of the GRQ file in all inks.
- #jobinfo# will display the Document Info of the file (as in File menu > Document Info).
- #inks# will display all inks that are present in the job. The order of the inks in the mark is based on the order of the ink palette.



Caution:

Note that technical inks will not be included in this list.

Tip:

#inks# can be used to put your additional text information in erase mode on a 100% bearer bar that you previously created. As a result, only the ink name of the corresponding film will be in erase mode in the bearer bar.

- #inks_lin_ang# will display information about screen ruling and angle.
- #ink_1#... will display individual inks.
- #date# will display the date in all inks.
- #time# will display the time in all inks
- #format# will display the plate size in all inks.

Inkname for additional info Select the ink by name or number in which your additional text should be created.This ink setting is applied on all 5 contents fields. The drop-down list displays all inks present in the QuickStep file and an extra ink, Registration Black. Registration Black puts your additional text in all the inks of your QuickStep file.

Example: When you choose another ink like Cyan, the additional text will only be visible on your Cyan film.

Orientation/mirrorClick one of the buttons to obtain the desired orientation or mirroring for the image.

Character StyleSelect the *Font* from the drop-down list.Select the *weight*: 'Roman' or 'Bold'.Select the font *style*: 'Upright' or 'Italic'. Specify the point *size*.

Paint ModeSelect 'Normal' to set the paint mode for the image to normal (overprint).Select 'Erase' to set the paint mode for the image to erase (erase=knocking out white).

Tip:

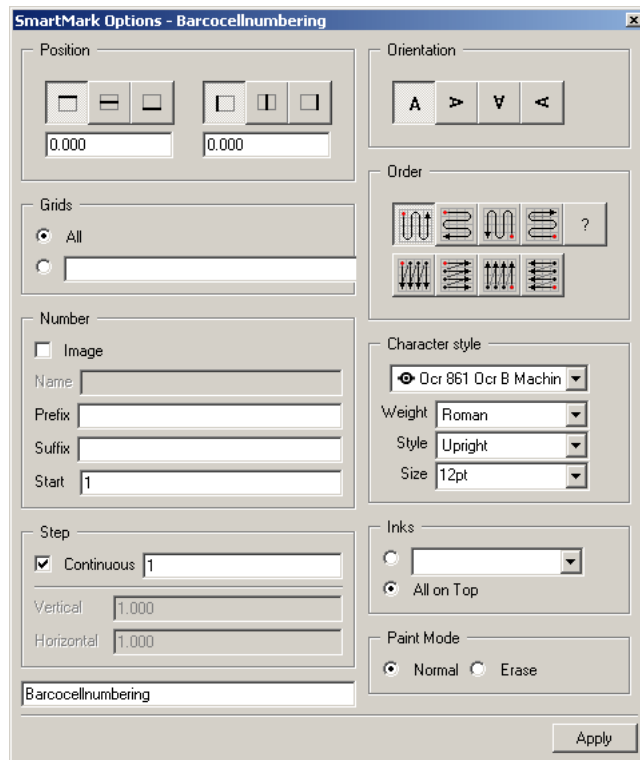
If you want all the inks in the bearer bar, provided the bearer bar is 100% and in all inks, you can give in : ?#inks#. As paint mode you choose 'Erase'. As a result you will only see C on the cyan film, Y on the yellow film, etc. Be sure that your text is positioned on the bearer bar, which has to have a 100% gradation.


Use the **Customized Name Input box** to give a customized name to the Barcotext mark. Enter the customized name, click 'Apply' and the new mark name appears in the list of marks.


Example: Enter the new name for the Barcotext Mark name, e.g. Mytext. When the dialog box closes, the original name in the list of marks has changed to: Mytext [barcotext].

7.3.31. GRQ Cellnumbering

You can use the GRQ Cellnumbering mark to add a number to each stepped element.



Position  Puts the top of the mark against the vertical origin of the repetition.

 Puts the center of the mark on the vertical origin of the repetition.

 Puts the bottom of the mark against the vertical origin of the repetition.

Use the *Vertical offset input box* to define the vertical offset distance from the borders of the repetition.

Puts the left of the mark against the horizontal origin of the repetition.

Puts the center of the mark against the horizontal origin of the repetition.

Puts the right of the mark against the horizontal origin of the repetition.

Use the *Horizontal offset input box* to define the horizontal offset distance from the borders of the repetition.

Grids Enable *All* to select all grids of the QuickStep file Use the *Input box* to select 1 or more grids to which you want to add the cell numbers. The different grids have to be separated by a comma.

Number/Image To use images for your numbering instead of digits. When *Image* is activated *Name* becomes available. Type the name of your image or use your right mouse button to select the image from the file selector. If you select Num.lc the system will look for Num1.lc, Num2.lc, Num3.lc,... . Each image used for numbering will have a different name.

Prefix If you want to add something more before the number then enter the text here. Example: When you enter A then the numbering is: A1, A2, A3, ...

Suffix If you want to add something more behind the number then enter the text here. Example: When you enter z then the numbering is: 1z, Az, Az, ...

Use *Start* to define the starting number. Example: When you enter 3 then the numbering starts with 3 and then 3, 4, 5, ...

Step When *Continuous* is activated and you defined a number in the input box next to it, the vertical and horizontal numbering will increase with that value.

Note:

e When *Continuous* is not activated, a different Vertical and Horizontal step can be defined.


Example: Put *Start* on 0, activate *Continuous* and put its value to 5. Your first stepped element gets the number 0, the second element gets 5, the third element gets 10,

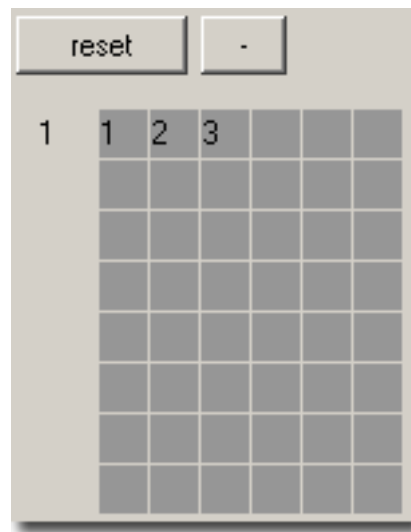
Use *Vertical* to define the Vertical Step numbering when it has to be different to the horizontal step. Use *Horizontal* to define the Horizontal step numbering when it has to be different to the vertical step.

Use the **Customized Name Input box** to give a customized name to the Barcotext mark. Enter the customized name, click 'Apply' and the new mark name appears in the list of marks.

Enter the a name for the Barcocellnumbering, e.g. Mynumbering. When the dialog box closes, the original name in the list of marks has changed to: Mybumbering [Barcocellnumbering].

Orientation Click one of the buttons to obtain the desired orientation for the image.

Order To define the direction for the cellnumbering. There are 8 available predefined directions buttons. Click the corresponding button to select the requested direction. If the available directions do not correspond to what you want, click the question mark button . The Barcocellnumbering dialog box expands and allows you to define the complete cellnumbering for the selected grid(s) yourself.



On the left side of the grid you can see a number. This is the grid indication. The grey boxes represent the vertical and horizontal count of the grid. By clicking the squares, you can assign numbers and so determine the order. The numbers that appear are a result of the values set for *Start* and *Continuous*. Click the minus button to remove the number that was assigned last. Click *Reset* to clear all numbers.

Character StyleSelect the *Font* from the drop-down list.Select the *weight*: 'Roman' or 'Bold'.Select the font *style*: 'Upright' or 'Italic'. Specify the point *size*.

InksThe '*Ink input box*' option button allows you to select one ink by its name or ink number. The drop-down list displays all inks available in the QuickStep file. When you select ink number 1, the first ink of the ink menu of QuickStep is chosen.The '*All on Top*' option button allows you to specify the selected image 'All on Top' in all inks.

Paint ModeSelect 'Normal' to set the paint mode for the image to normal (overprint).Select 'Erase' to set the paint mode for the image to erase (erase=knocking out white).

8. Printing and Exporting

Print your plates and stations or export them to other file formats.

You can use the printing and exporting features to create proofs, films or printing plates for your production process. The station and plate files that you create in Plato can be printed to a variety of PostScript output devices. Station and plate files can be exported to e.g. PostScript, EPS and GRS file formats for proofing or printing at a remote site.

- [Printing Esko Plato Files](#)
- [Exporting Esko Plato Files](#)

8.1. Printing Esko Plato Files

You can print to a PostScript output device directly from Esko Plato. Both composite and separated PostScript can be used. Typically, a composite PostScript file is used for proofing. A separated PostScript file is used typically for exposing films or plates on a PostScript imagesetter or platesetter.

- [Using PostScript](#)
- [Printing to a PostScript device](#)

8.1.1. Using PostScript

Before discussing the printing and exporting options of Esko Plato, you should know some things about Esko file formats and their conversion to PostScript file formats. Esko file formats may contain advanced features that are not directly supported in the PostScript model. Examples of these features include the Esko Overprint modes and Translucency.

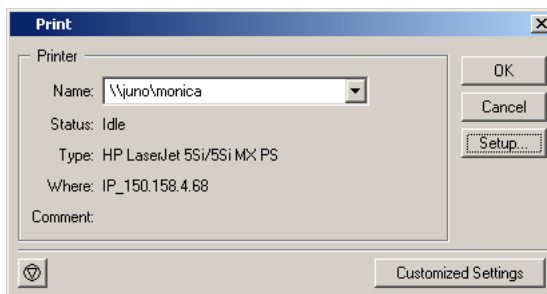
For example, an Esko file may have objects that overprint one another in one separation. Printing or exporting such a file to a PostScript or EPS format could result in the loss of the overprint due to the known PostScript limitations.

Esko Plato uses the Esko OutRight Technology to overcome this limitation. This technology pre-rips these objects into images that can be rendered accurately in PostScript. OutRight guarantees that what you see in Esko Plato is what you get when you print.

8.1.2. Printing to a PostScript device

To print to a PostScript output device:

1. On the File menu, click 'Print'. The Print dialog box appears:



2. Select the printer and the options you need.

Note:

Click 'Setup' to open the Setup dialog box. For more on these options, please refer to the chapter *'Exporting to a PostScript File'*.

3. Click 'Print'.

8.2. Exporting Esko Plato Files

You can export Esko Plato files to other file formats for remote viewing, proofing and printing. Click 'Export' on the File menu to choose an output format and to specify options.

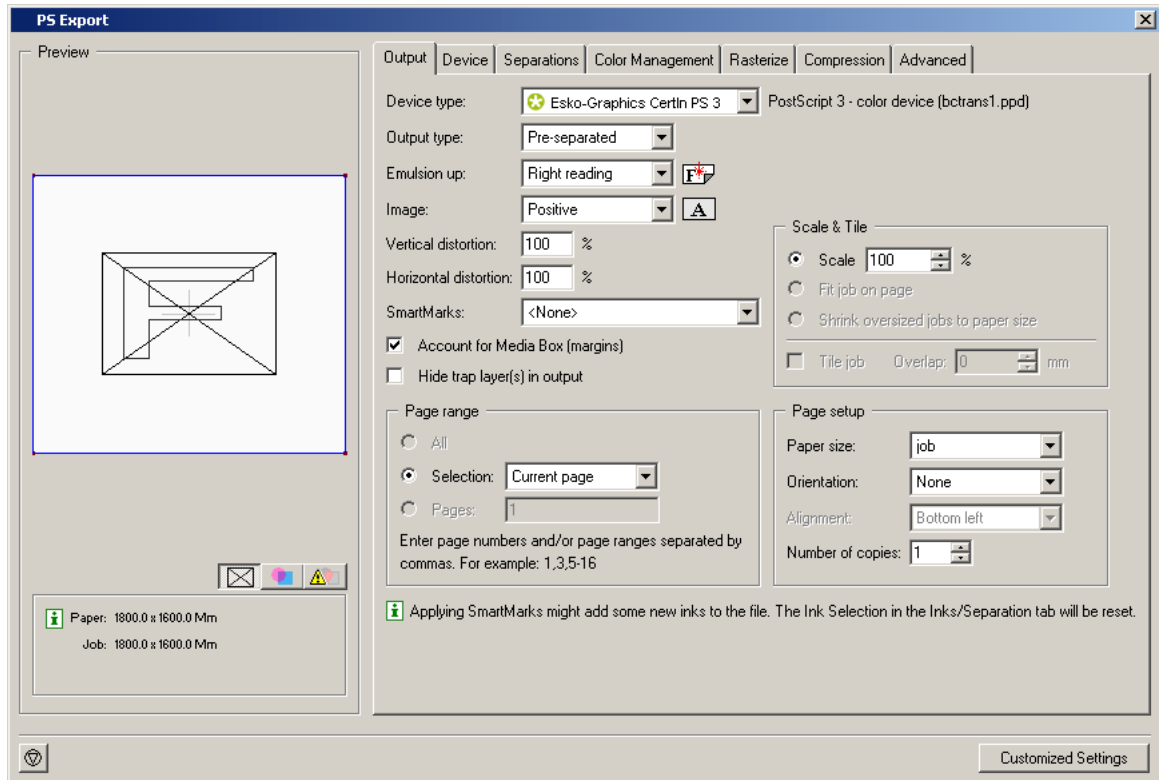
When you export PostScript and EPS file formats, the options contain many of the same options used for printing. The differences between exporting and printing will be indicated in the sections below. If a format uses the same options as for printing PostScript, you will be referred to the appropriate section for Printing.

- [Exporting to a PostScript File](#)
- [Exporting to an EPS File](#)
- [Exporting to a PDF File](#)
- [Export to an MFG file](#)
- [Exporting to a CFF2 File](#)
- [Exporting to a Normalized PDF File](#)
- [Exporting to a PLA File](#)
- [Exporting to a PDFPLA File](#)
- [Exporting to a GRS File](#)
- [Exporting to a JDF Layout File](#)
- [Exporting to a CIP3 File](#)
- [Exporting to a Polar File](#)
- [Exporting to a JDF Cutting file](#)

- [Digital Converting Export](#) on page 277

8.2.1. Exporting to a PostScript File

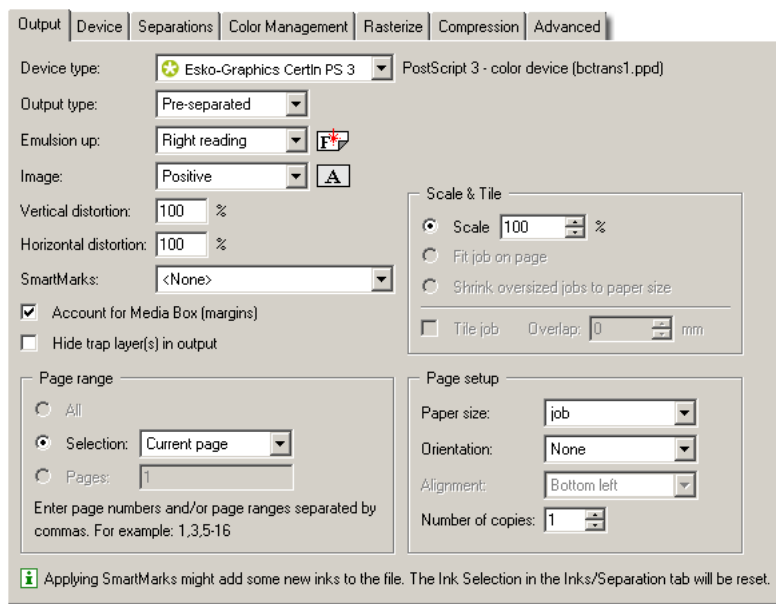
You can export your station and plate files to a PostScript file. You can indicate whether the PostScript file is either a Composite or Pre-Separated file.



The PostScript Export ticket has the following tabs:

- [Output](#)
- [Device](#)
- [Inks / Separations](#)
- [Color Management](#)
- [Rasterize](#)
- [Compression](#)
- [Advanced](#)

Output



Device typeTo 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 typeTo 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 colors are converted to CMYK.

- Pre-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 color space 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 UpTo obtain Right Reading (= emulsion up) or Wrong Reading (=emulsion down).

ImageTo 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 drop-down list.

Account for Media Box (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 page numbers or page ranges separated by commas, e.g. 1,3,5-8.

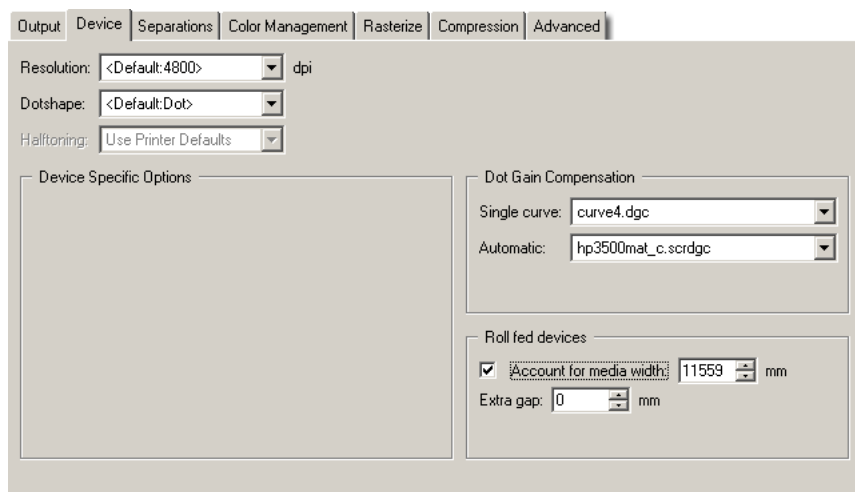
Scale & Tile

- 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 **Dotshape**.

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 info 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.

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.

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.

Inks / Separations

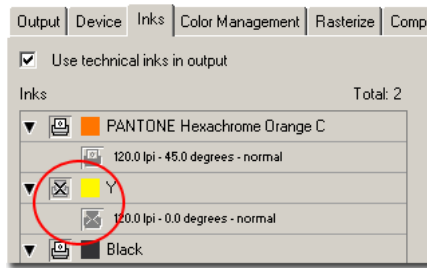
The Inks / Separations tab depends on the output type you selected in the *Output tab*. If you choose CMYK Composite or Composite, the Inks tab will be visible, if you choose separated, the Separations tab will be visible.

Inks Tab

Use technical inks in output Normally technical inks are hidden when you save a job in an Esko editor. But, there's a possibility to save the Technical inks as visible. This toggle here makes it possible to output the visible (non hidden) technical inks.

The **Inks** are listed. You can deselect the ones you want to exclude from export by clicking the

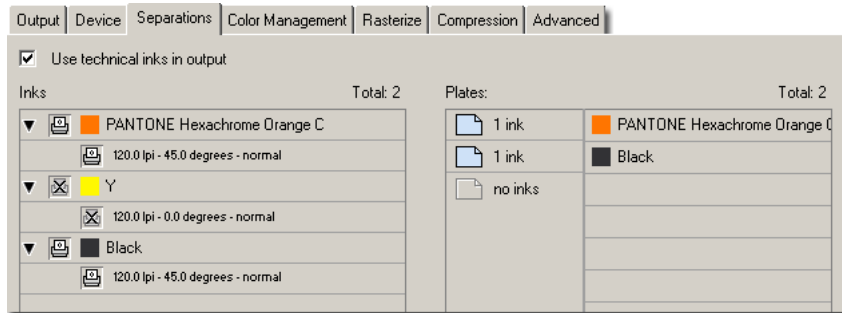
print icon  



Separations Tab

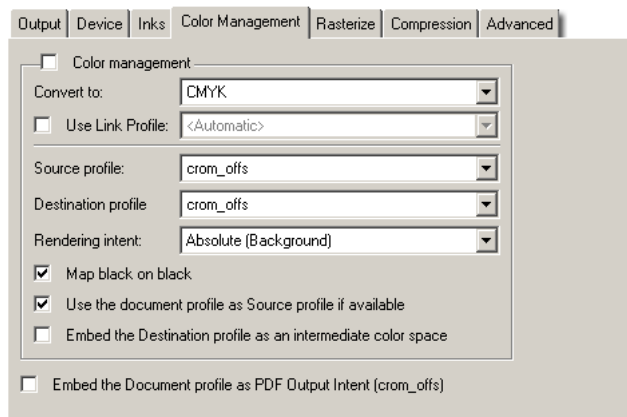
Use technical inks in output Normally technical inks are hidden when you save a job in an Esko editor. But, there's a possibility to save the Technical inks as visible. This toggle here makes it possible to output the visible (non hidden) technical inks.

The inks and separations are listed. You can deselect one or more inks, the according separation will disappear from the Plates list.



Use the 'Convert to CMYK' button to convert PANTONE inks to CMYK values.

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 Kaleidoscope 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 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 there is a document profile defined, that will be used as a source profile. The one chosen from the Source Profile drop-down list will be overruled.

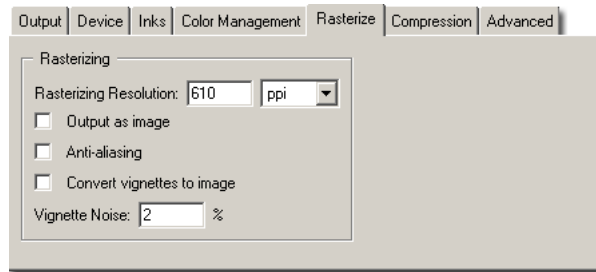
Embed the Destination profile as an intermediate color space

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 color space 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 Kaleidoscope profile, it will first be converted to an ICC profile.

Rasterize



Output as image When this toggle is activated an image is made of the job. Enter the resolution.

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.

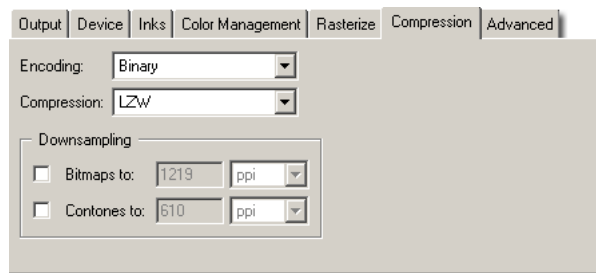
Tip:

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



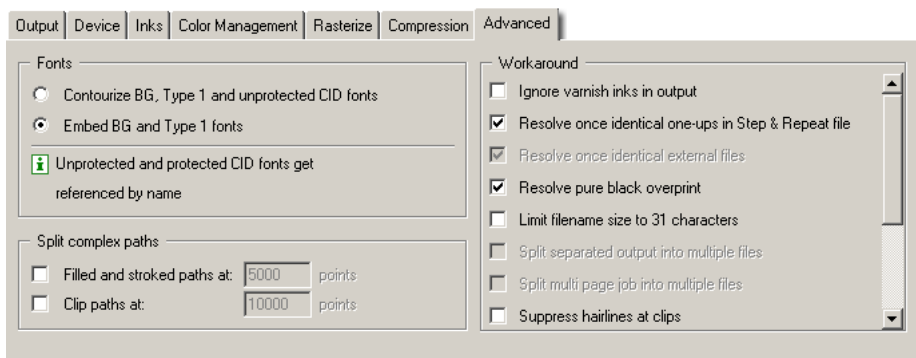
Encoding

- Binary: The image data is represented as binary data. This results in compact files that are not editable in a common ASCII editor however. 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 can also 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 to speed up generation of the PostScript file. Select the unit for subsampling from the drop-down list: lpm or ppi. You can subsample both bitmaps and contones.

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



Caution:

These are workarounds for typical PostScript level 1 path limitations. Only use these when **really** necessary.

- Use *Filled and stroked paths at X points* 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.
- Use *Clip paths at X points* 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

- *Ignore varnish inks* Varnish inks will not be included while outputting to PostScript.
- *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.
- *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 pure black overprint* Overprints are normally resolved, but 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 options in the Color Management tab are activated.
- *Limit filename size to 31 characters* Check this toggle if you want to make sure your imported files can be seen/read from/by all Apple operating systems.
- *Split separated output into multiple files* Instead of writing one file containing all separations, the system will write one file per separation.
- *Split multi page jobs into multiple files* Every page will be output as a separate file.
- *Suppress Hairlines At Clips* Due to the PostScript limitation of maximum 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 may become visible. To avoid this problem, activate Suppress Hairlines at Clips.
- *Image To Paper Edge* To move the origin of the converted job to the lower left corner of the paper or film. Activate this toggle to align your job to the edge of the paper or film to use the full area.



Caution:

Exporting to a paper PostScript device: For most of these devices, there is always a border on the output page where nothing can be imaged. Parts of your design in that border will be lost.

- *Avoid Tabular Number Problems* 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 Problems.
- *Disable path clipping* By default, paths are clipped by the borders of the job, but there are instances where you do not want the paths to be clipped, if so, select this toggle.
- *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 (e.g. Scitex Brisque Workflow). This option is available when Pre-Separated was selected. Spot color separations can be handled only on such systems by remapping them to a specific process separation. When this toggle is checked, spot colors will be exported as a C, Y, M or K separation. The mapping is done based on the lpi and angle of the spot color.

If the process Cyan ink has a given lpi of 120 and angle of 15, a spot color Reflex blue may be mapped to Cyan, if in the Paint menu it is given the same lpi and angle of process Cyan.

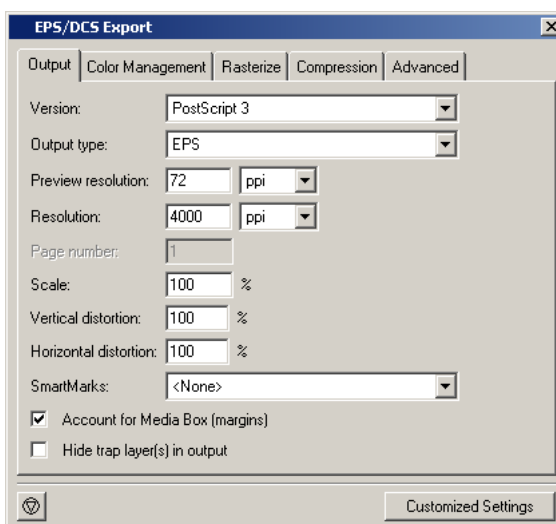
- *Suffix for PANTONE ink names* Enter a suffix, the default is C.
- *Maximum deviation of resolved curves* During the resolving process Bezier curves are sometimes approximated by vectors. The maximum allowed deviation of the curves can be entered here.

Note:

When a Esko native file is exported to PostScript, certain elements in the job which cannot be translated into PostScript will be resolved in order to get an exact result. The same happens when exporting to PDF. PDF files however have several destinations and don't always need the high quality resolving because they may only serve as soft proof. If you want to obtain even smaller files than what you can get with the "PDF for RGB/CMYK proofing" settings, then this workaround option might help. When we resolve objects splines are converted to vectors (e.g. line mask). The spline only has a 2 points but you need a lot of vector points to describe a curve. The resolver default deviation is 0.002 mm but you can reduce the amount of vector points by raising the deviation value and thus reducing the size of the PDF. Which values? When comparing the default (0.002 = 1360 kb PDF) with 0.02 (326 kb PDF) you will end up with an acceptable result but it is clear that high values will alter the PDF too much. Keep in mind that this option will only modify resolved vector curves, it will not "straighten" any other objects while exporting.

8.2.2. Exporting to an EPS File

You can export your station and plate files to an EPS file.

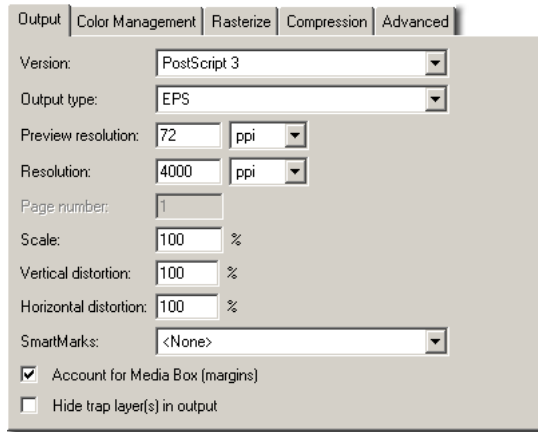


The EPS Export Setup dialog box has five tabs.

- [Output](#)
- [Color Management](#)

- [Rasterize](#)
- [Compression](#)
- [Advanced](#)

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 and 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 for composite printing. *Multiple DCS file* This will create a master EPS DCS file referring to multiple separations (one file per ink). These separations may contain linework as well as images. 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 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 on 16.000 ppi. If you do not know where the EPS will be placed, you might want to make the resolution a high one.

Page Number: To select the page of the job you want to export.

Scale, you can scale using percentages.

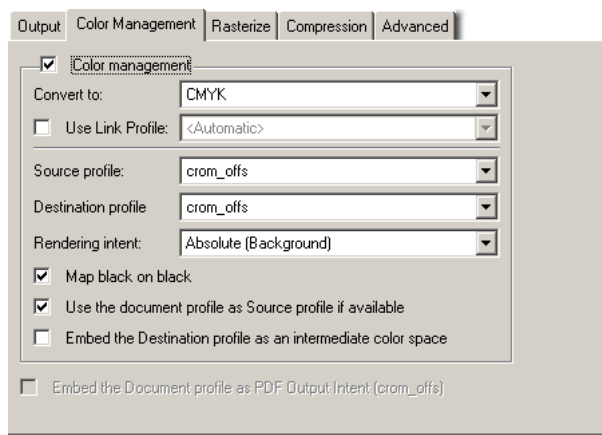
Add SmartMarks sets by selecting one from the drop-down list.

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 also 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.

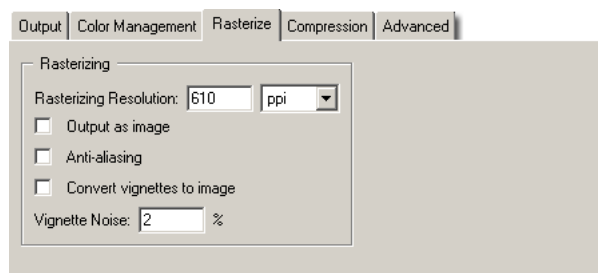
Color Management

The Color Management tab is identical to the *Color Management tab* of the PostScript Export dialog box.



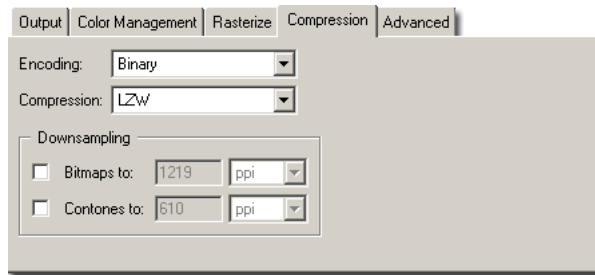
Rasterize

The Rasterize tab is identical to the *Rasterize tab* of the PostScript Export dialog box.



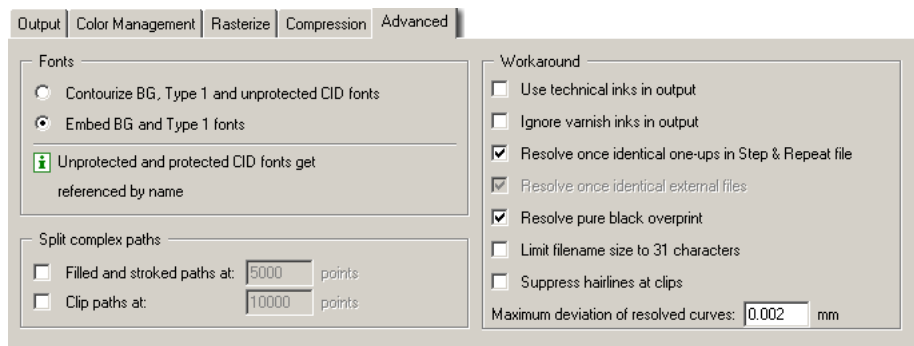
Compression

The Compression tab is identical to the *Compression tab* of the PostScript Export dialog box.



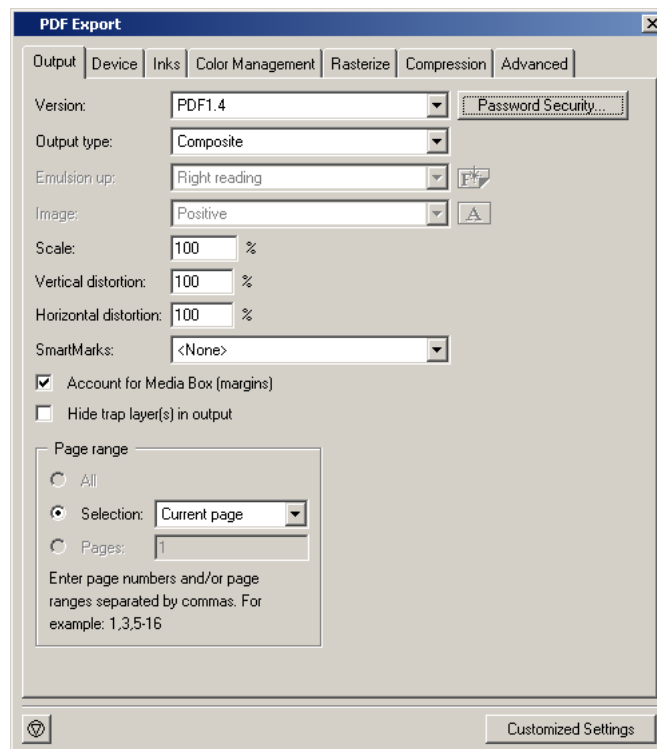
Advanced

The Advanced tab based on the *Advanced tab* of the PostScript Export dialog box.



8.2.3. Exporting to a PDF File

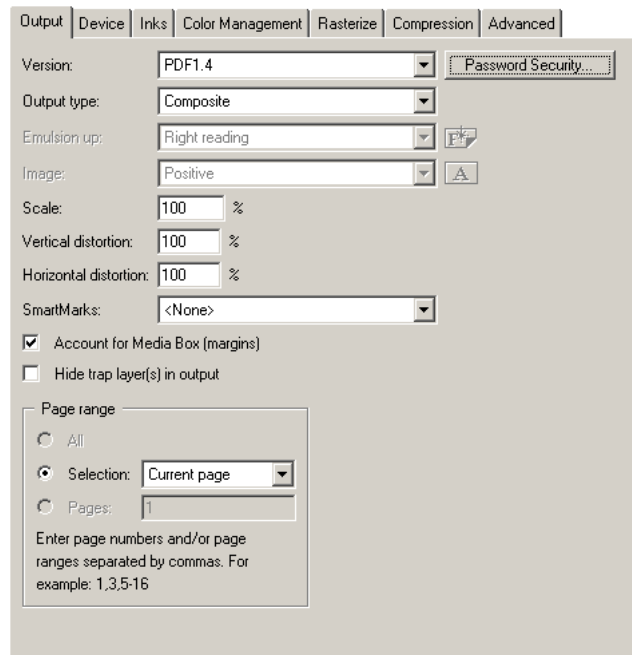
Exporting to PDF 1.4 will use PDF forms to represent the different one-ups. This should result in compact PDF representations of PLA files.



The PDF Export dialog box has the following tabs.

- *Output*
- *Device*
- *Inks*
- *Color Management*
- *Rasterize*
- *Compression*
- *Advanced*

Output



PDF Version Select a PDF version from the drop-down list.

- PDF for Illustrator 10 or higher: Select this version if you want to open the generated PDF 1.4 file in Adobe Illustrator 10 or higher. Because Adobe Illustrator only supports a (undocumented) subset of PDF 1.4 features some objects and/or color spaces must be split up, transformed and/or prepped in order to correctly represent them in Illustrator.
- PDF/X-1 a:2001 (ISO /DIS 15930-1)
- PDF/X-1:2001 (ISO /DIS 15930-1)
- PDF 1.4: the Acrobat 5 file format.
- PDF 1.3: the Acrobat 4 file format.

Output type To export or print your design as CMYK Composite, Separated or Composite.

- 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 color space 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 drop-down list.

Account for Media Box (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 outputTrap layers will not be included in the output.

Page range

- Output all pages.
- Output a number of pages. Enter page numbers or page ranges separated by commas, e.g. 1,3,5-8.

ScaleDefine the scale in percentages. You might want to scale your job to make it easier to softproof via e mail.

Select **SmartMarks** from the drop-down list.

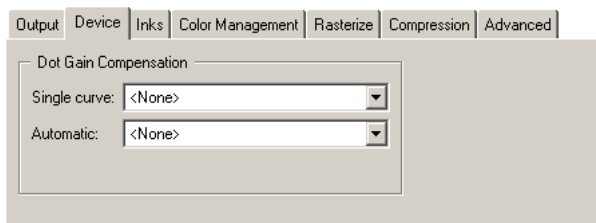
Put marks with ink coverage on a hardcopy proof (or view it in Acrobat).

Account for job marginsTo 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.

Output range

- Output All pages.
- Output a number of pages. Enter pagenumbers or page ranges separated by commas. e.g. 1,3,5-8

Device





Dot Gain CompensationYou 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.

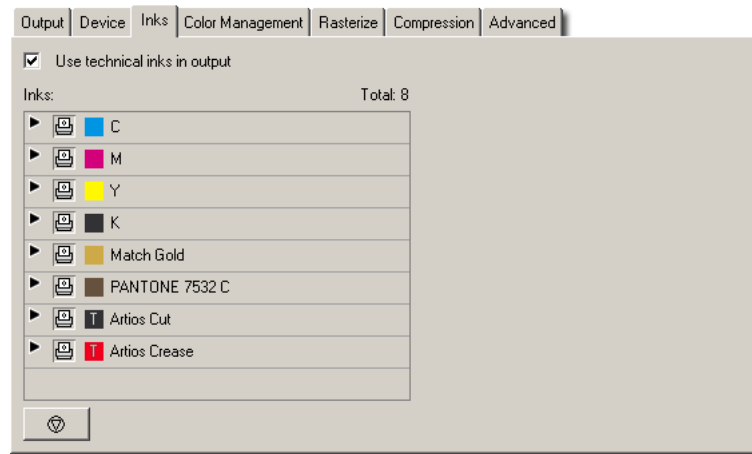
Inks

The **inks** are listed, you can (de)select them by clicking the print icon  

Check the box to **use technical inks in the output job.**

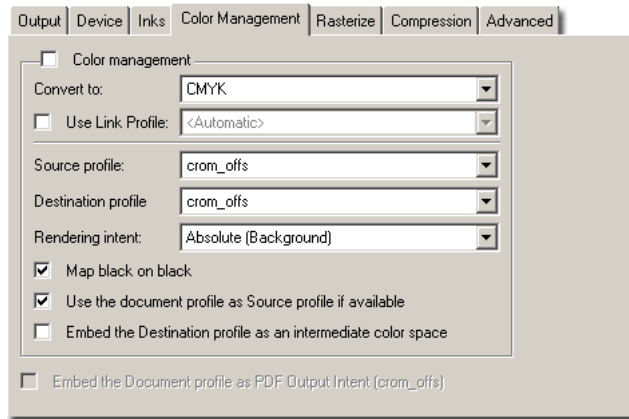
Note:

The inks tab becomes available when exporting to a separated file. The separation tabsheet becomes available when exporting to CMKY and Composite. The Separation tab offers the possibility to put multiple inks on the same separation/plate.



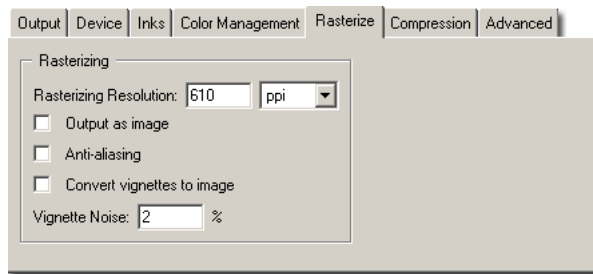
Color Management

The Color Management tab is identical to the *Color Management tab* of the PostScript Export dialog box.

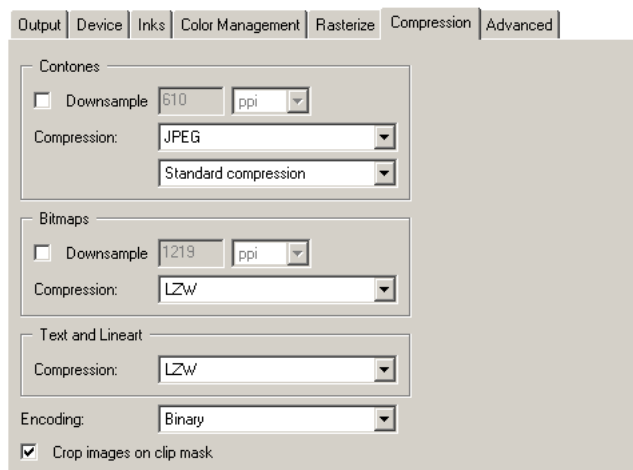


Rasterize

The Rasterize tab is identical to the *Rasterize tab* of the PostScript Export dialog box.



Compression



Encoding:

- Binary: The image data is represented as binary data. This results in compact files that are not editable in a common ASCII editor however. 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: They can be sent over a communication line using whatever communication protocol is available. The drawback however is that hexadecimal image data takes 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

Bitmaps

- None
- CCITT group 3 and CCITT group 4
- Run Length
- LZW
- LZW Diff
- Flate
- Flate Diff

Contones

- None
- LZW
- LZW Diff
- JPEG Compressions (from Limited over Standard to Low compression)
- Flate
- Flate Diff

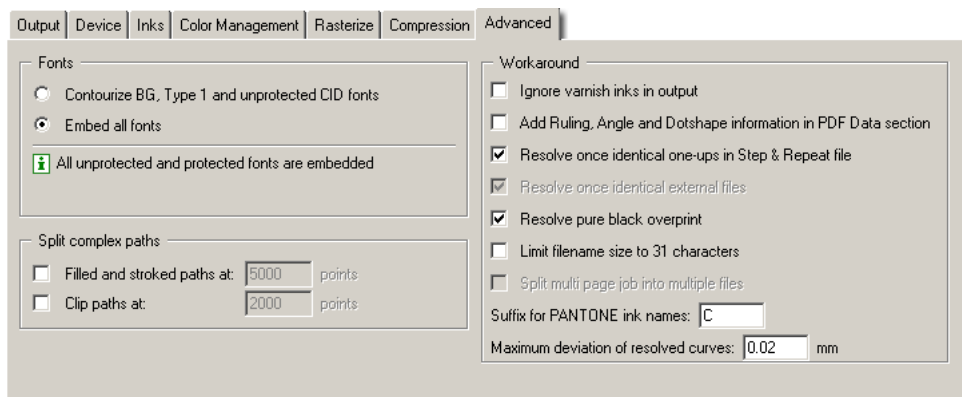
Text and Lineart

- None
- LZW
- Flate

Downsampling 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 drop-down list: lpmm or ppi. You can subsample both bitmaps and contones.

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.

Advanced



Fonts

- Contourize fonts: protected CID fonts are embedded.
- Embed BG and Type1 fonts: Allows you to include the used BG and Type 1 fonts of your design in your PDF file. A subset of the font will be embedded, not the whole font.

Note:

In case the BG font does not have a corresponding PostScript Name, it will be included in the PostScript file automatically.



Caution:

When you switch off these functions, take into account that the font downloaded on the PostScript device may be different from the font used in your job. Consequently, the output result can be different to your design.

Split Complex Paths



Caution:

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

- *Ignore varnish inks in output* Varnish inks will not be included while outputting to PostScript.
- *Add ruling, angle and dotshape information in PDF Data section*
- *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.
- *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 pure black overprint* Overprints are normally resolved, but 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 options in the Color Management tab are activated.
- *Limit filename size to 31 characters* Check this toggle if you want to make sure your imported files can be seen/read from/by all Apple operating systems.
- *Split multi page jobs into multiple files* Every page will be output as a separate file.
- *Suffix for PANTONE ink names* Enter a suffix, the default is C.
- *Maximum deviation of resolved curves* During the resolving process Bezier curves are sometimes approximated by vectors. The maximum allowed deviation of the curves can be entered here.

Note:

When a Esko native file is exported to PostScript, certain elements in the job which cannot be translated into PostScript will be resolved in order to get an exact result. The same happens when exporting to PDF. PDF files however have several destinations and don't always need the high quality resolving because they may only serve as soft proof. If you want to obtain even smaller files than what you can get with the "PDF for RGB/CMYK proofing" settings, then this workaround option might help. When we resolve objects splines are converted to vectors (e.g. line mask). The spline only has a 2 points but you need a lot of vector points to describe a curve. The resolver default deviation is 0.002 mm but you can reduce the amount of vector points by raising the deviation value and thus reducing the size of the PDF. Which values? When comparing the default (0.002 = 1360 kb PDF) with 0.02 (326 kb PDF) you will end up with an acceptable result but it is clear that high values will alter the PDF too much. Keep in mind that this option will only modify resolved vector curves, it will not "straighten" any other objects while exporting.

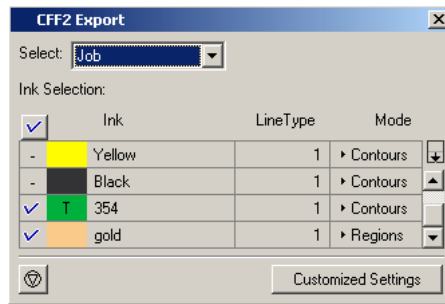
8.2.4. Export to an MFG File

Exporting to MFG will export the die shape of each station to ARD format and place all these designs into a layout (MFG) file. Similar exports are: Export to layout CFF2 file and Export to layout DDEs3 file. In these cases, the die shapes are exported to CFF2/ DDEs3 file.

8.2.5. Exporting to a CFF2 File

CFF2 is short for the Common File Format version 2.

CFF2 Export allows to export one or more separations to a CAD system. This can be used to export varnish or embossing shapes to a CAD system.



Use the drop-down list to **select** whether you want to export the whole 'job', the 'red selection' or the 'current layer'.

You can also (de)select which ink you (do not) want to output by clicking the tick: . Define the CFF2 **LineType**: Setting the CFF2 linetype allows the CAD operator to distinguish between objects in different separations. Common CFF2 linetypes: 1 normally used for cutlines, 2 for crease lines

Determine the **Mode**:

- Output as Contours.

Two objects (filled with Cyan) will be outputted as two squares:



- Output as Regions.

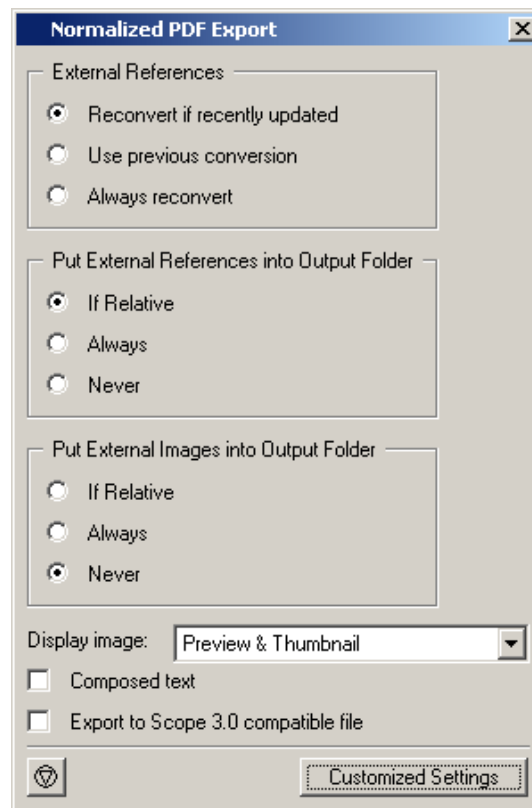
Two objects (Cyan) will be outputted as one 'region'



8.2.6. Export to a Normalized PDF File

Note:

This option is only available in case of a **PDF** Job.



External References: External references have to be converted.

- Reconvert if recently updated: only updated references will be reconverted, all other conversions will be used as they were.
- Use previous conversion: the existing conversions will be used, potential updates will not be considered.
- Always reconvert: all external references will be converted at all times.

Put External References / Images into Output Folder: Referenced files in GRS Job File Format will be converted to referenced external files in Normalized PDF mode, but where will they end up? There are two options:

- External Normalized PDF files will be written in the output folder, next to the main Normalized PDF file.
- External Normalized PDF files will be written in the original folder, next to the original external GRS files.
 - Always: the converted external files will be saved next to the main file.
 - Never: the converted external files will be saved in the folder of the original files.
 - If relative: if the external GRS files were close to the main GRS file (subfolder, neighboring folder) the converted files will be saved in the output folder. If the external GRS files were not 'close' to the main file (central smartmarks, logos, ...) the converted files will be saved next to the original files.

Display image: You can save the file with 'no display' image, 'thumbnail' (200 x 200 pixels) or low-resolution 'display image' (To save the file with a 72 dpi image. This image can be used when the file is placed as an external file).

Composed text: Composed text is non-editable text. It can be converted to editable text by clicking it when you are in text edit mode.

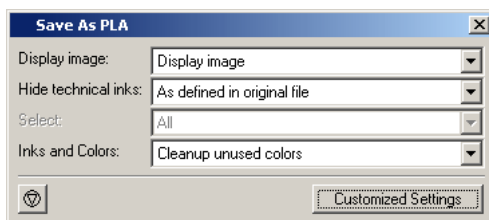
Export to Scope 3 Compatible file: Check this box if you want your Normalized PDF file to be Scope 3 compatible.

8.2.7. Exporting to a PLA File

Can be used to export a PDFPLA file (which can only be processed by a BackStage 7.0 or Flexrip 7.0) to a Scope 3.0 compatible file.

Note:

This option is only available in case of a **PDF** Job.



Display ImageThis option allows you to set the type of display image that will be included in the GRS file. Choices are: No Display , Thumbnail (200 x 200 pixels) or Display Image (72 dpi), which creates a low resolution preview that can be used when the file is placed as an external file.

Hide Technical InksWhen selected, technical inks will be stored in the file but will not be visible as exposable inks for the Esko FlexRip.

SelectYou can export your entire job or a part of it.

- All - saves the entire job
- Red Selection - saves objects selected in red only
- Current Layer - saves the currently active layer only
- Current Page - saves the current page

Inks and Colors

- 'Clean up unused colors' automatically removes all unused inks.
- 'Clean up unused inks and colors' automatically removes all colors and inks that are not used in the job.
- 'Save unused inks and colors' saves unused inks and colors.

ResetThe Reset button resets all the options that you may have changed to their default values.

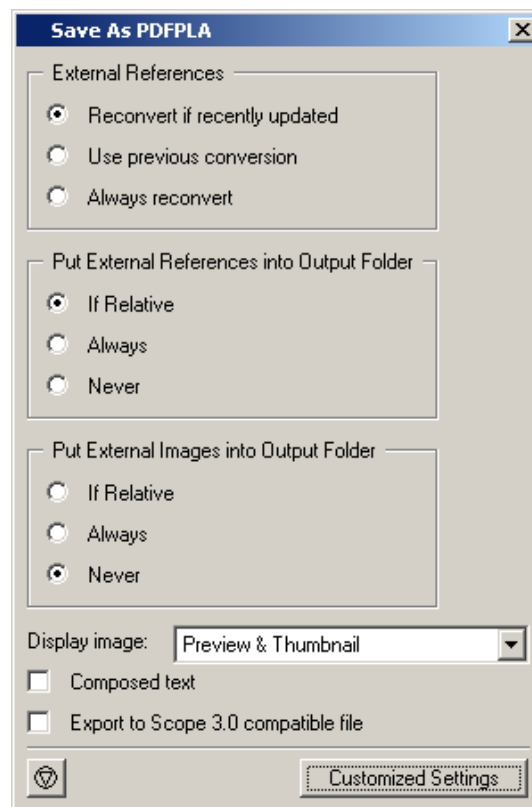
Customized SettingsThe Customized Settings button allows you to save the current settings so that they can be used later or can be set as your own defaults. The feature is very useful for

those export file types that have many options that you modify each time you export to that file type.

8.2.8. Exporting to a PDFPLA File

Note:

This option is only available in case of a **GRS** Job.



External References

- Reconvert if recently updated: only updated references will be reconverted, all other conversions will be used as they were.
- Use previous conversion: the existing conversions will be used, potential updates will not be considered.
- Always reconvert: all external references will be converted at all times.

Put External References / Images into Output Folder: Referenced files in GRS mode will be converted to referenced external files in Normalized PDF mode, but where will they end up? There are two options:

- External Normalized PDF files will be written in the output folder, next to the main Normalized PDF file.

- External Normalized PDF files will be written in the original folder, next to the original external GRS files.
 - Always: the converted external files will be saved next to the main file.
 - Never: the converted external files will be saved in the folder of the original files.
 - If relative: if the external GRS files were close to the main GRS file (subfolder, neighboring folder) the converted files will be saved in the output folder. If the external GRS files were not 'close' to the main file (central smartmarks, logos, ...) the converted files will be saved next to the original files.

Display image: You can save the file with 'no display' image, 'thumbnail' (200 x 200 pixels) or low-resolution 'display image' (To save the file with a 72 dpi image. This image can be used when the file is placed as an external file).

Composed text: Composed text is non-editable text. It can be converted to editable text by checking this box.

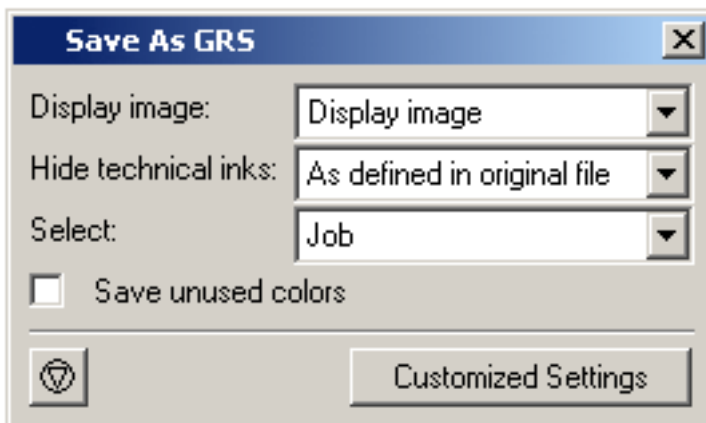
Export to Scope 3 Compatible file: Check this box if you want your Normalized PDF file to be Scope 3 compatible.

8.2.9. Exporting to a GRS File

You can export your station and plate files to a GRS file.

Note:

This option is only available in case of a **GRS** Job.



The options for exporting to a GRS file are: **Display Image**This option allows you to set the type of display image that will be included in the GRS file. Choices are: No Display, Thumbnail (200 x 200 pixels) or Display Image (72 dpi), which creates a low resolution preview that can be used when the file is placed as an external file.

Hide Technical InksWhen selected, technical inks will be stored in the file but will not be visible as exposable inks for the Esko FlexRip.

SelectYou can export your entire job or a part of it.

- All - saves the entire job
- Red Selection - saves objects selected in red only
- Current Layer - saves the currently active layer only
- Current Page - saves the current page

Save Unused Colors saves any unused colors with the job. If some colors in the file are no longer used, by default they will be removed when saved. To save unused colors, check this option.

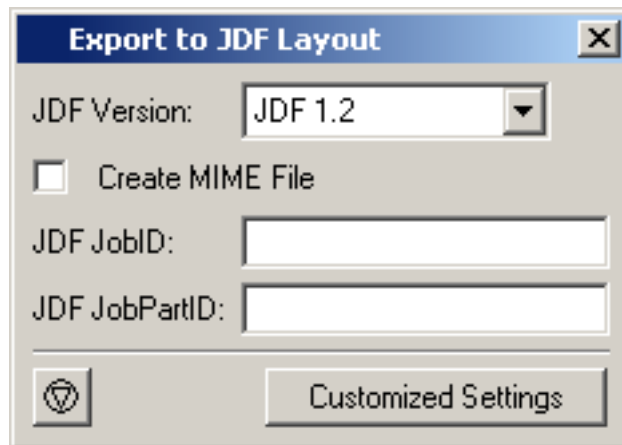
ResetThe Reset button resets all the options that you may have changed to their default values.

Customized SettingsThe Customized Settings button allows you to save the current settings so that they can be used later or can be set as your own defaults. The feature is very useful for those export file types that have many options that you modify each time you export to that file type.

8.2.10. Exporting to a JDF Layout File

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.

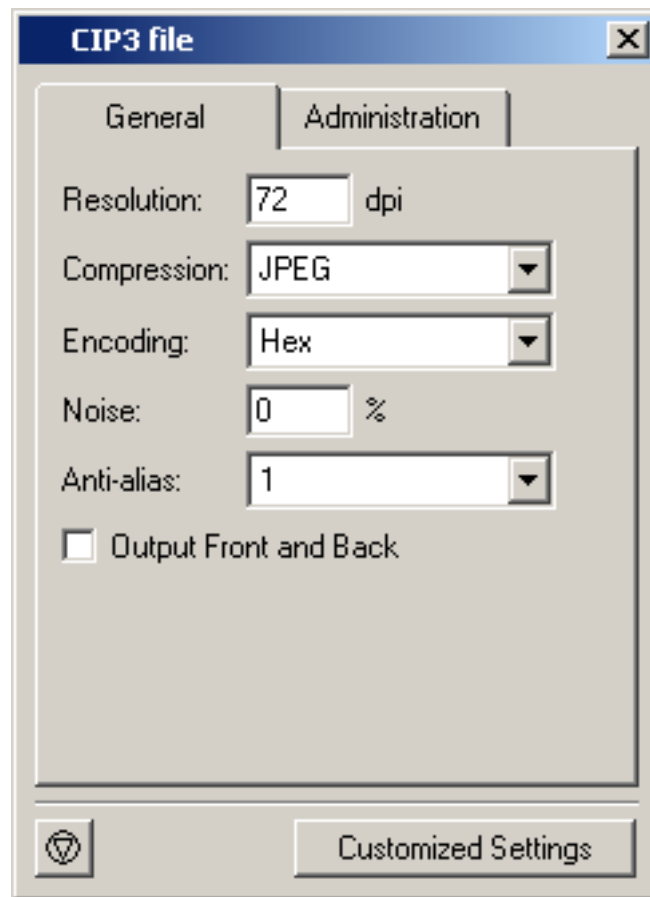
Check 'Create MIME file' to put the JDF and the referenced PDF files together in a MIME packed file.



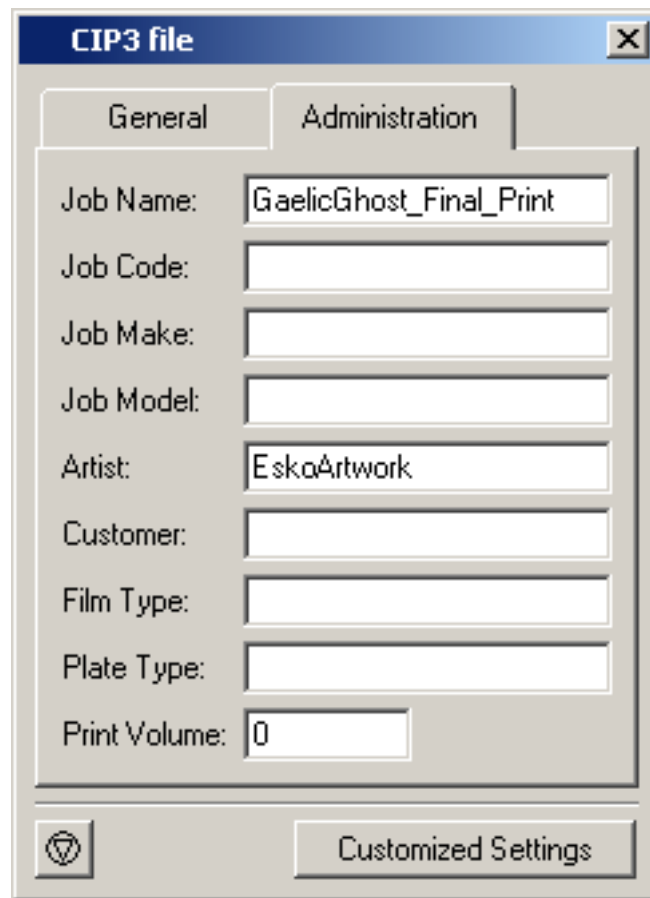
8.2.11. Exporting to a CIP3 File

A CIP3 file contains information that can be used on press, pre-press and post-press systems. CIP3 file generated from PackEdge will only include information regarding the ink key settings on the press. A low resolution image of all separations will be stored in the file. A CIP3 file has a PostScript syntax, but cannot be opened as a PostScript file. A stand-alone Plato will benefit most of CIP3 export, because there is already a CIP3 export module on the FlexRIP.

The **General** tab allows you to define some general settings regarding the generated PPF file.



The **Administration** tab offers the most commonly used keys in PPF files. You can enter data depending on what your press actually needs.



Caution:
Exporting to CIP3 files only works for single page documents.

8.2.12. Exporting to a Polar File (Optional)

It is now possible to export cutting information to Polar CompuCut stations. This speeds up the setup times of guillotine cut sheets. When the bounding box stations overlap, the stations will be exported as 1 block, so no cutting errors occur.

Click the Setup button to open the Export to Polar dialog box.



Use the drop-down lists to define the position of the Side Guide (left/right) and Gripper (top/bottom).

8.2.13. Exporting to a JDF Cutting file

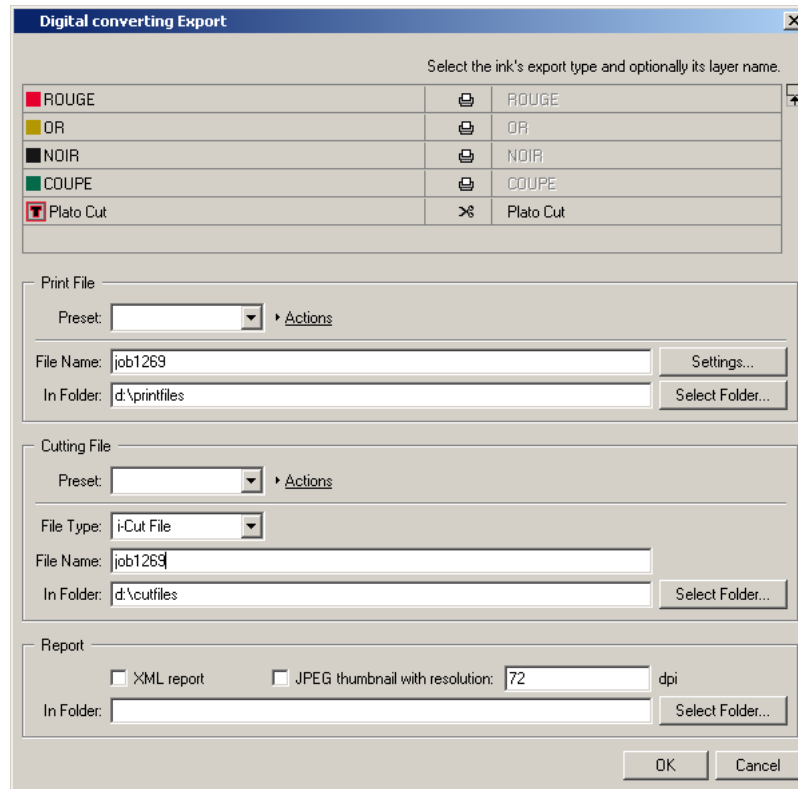
This task makes it possible to export a CIP4 JDF cutting file. This file contains cutting information for guillotine cutting, saved in a CIP4 JDF file format.

8.2.14. Digital Converting Export

Digital Converting Export is an optional module for Plato users that wish to export cut files for digital finishing on a Kongsberg table, for sample making or short run production purposes.

Note:

To place i-cut marks around the layout, the file "icutmark.dtl" should be copied manually from X:\Esko\bg_prog_marks_v010\dtl\gmfk2_optional to X:\Esko\bg_data_marks_v010\dtl\gmfk2_v013



It allows to save all files needed to produce the current layouts:

- a PDF file which will be sent to the proofer or digital press
- a cutting file which will be sent to the cutting table.

The Digital Converting Export dialog contains different parts:




- The **Inks export type**
- **Export PDF File:** to create the PDF files containing all graphics for the layouts
- **Export Cutting File:** to create the file (i-cut file, HPGL, PDF, MFG or ACM file) containing all cutting information

Inks

The Inks shows all inks used in the layouts.

The first column lists the name of all inks

The second column shows how the inks are assigned:

-  : these inks need to be printed, so they will be included in the PDF file
-  : these inks contain cutting information, and thus will be included in the cutting file
-  : these inks will be excluded from export.

You can change the assignment of an ink by clicking the icon.

Note:

If you only want to output the PDF file, you can switch the cutting inks to "exclude". If you want to output only the i-Cut file, you can switch all printing inks to "exclude". When e.g. outputting PDFPLA to a FlexRip, the PDF is not needed, so all print separations can be excluded.

The third column is only available for inks assigned to the Cutting file. What you can define depends on the selected cutting file type (see below).

If the Cutting File type is set to i-Cut file or PDF file, you can enter (or change) the name of the layer the ink will be placed in.

If the Cutting File type is set to HPGL, you can define the HPGL line type to be used.

If the Cutting File type is set to MFG, you can select the ArtiosCad line types . If an ArtiosCAD file was imported for the cut shape, the complete ARD file will be put in the MFG file.

If the Cutting File type is set to ACM, you can either select ArtiosCad line types, or you can select the native line types (P1, P2, P3, ...). When selecting ArtiosCad line types, the conversion from MFG to ACM will include a number of optimizations and manipulations. When using P1, P2, ..., the cut lines are dumped straight into the ACM file.

Export PDF File

- Set the **File Name**
- Define in what folder the file must be saved, either by entering the path, or by clicking the **Select Folder** button and browse to it.
- Click the **Export Settings** to set extra PDF options

Note:

In the Settings, you can choose to output as a multipage PDF. This will result in one PDF file containing one page for every layout needed, instead of one PDF for every layout. This option is only available if all layouts have the same dimensions.

Export Cutting File

- Select the **File Type**: i-Cut file, HPGL file, PDF file, MFG file, or ACM file.
- Set the **File Name**
- Define in what folder the file must be saved, either by entering the path, or by clicking the **Select Folder** button and browse to it.

Note:

When exporting to i-cut, HPGL or ACM, common line removal is applied, avoiding double lines in the output.

Depending on the selected cutting file type, you can add information in the third column of the Inks list:

- If the Cutting File type is set to i-Cut file or PDF file, you can enter (or change) the name of the layer the ink will be placed in.
- If the Cutting File type is set to HPGL, you can define the HPGL line type to be used.

- MFG is the native ArtiosCAD file type. If the Cutting File type is set to MFG, you can select the ArtiosCad line types . If an ArtiosCAD file was imported for the cut shape, the complete ARD file will be put in the MFG file.
- ACM is the native language of a Kongsberg table. If the Cutting File type is set to ACM, you can either select ArtiosCad line types, or you can select the native line types (P1, P2, P3, ...). When selecting ArtiosCad line types, the conversion from MFG to ACM will include a number of optimizations and manipulations. When using P1, P2, ..., the cut lines are simply copied into the ACM file.

Presets

Presets can be used if you want to store and reuse the settings you make.

To save a preset:

1. Enter the settings (folder, file type, settings) you want to save.
2. Click the **Actions** button and select **Save Preset**
3. Enter a name for the preset, and click **Save**

in the Presets dialog, which can be opened by clicking **Actions > Manage Presets**, you can load a preset, or delete an existing preset.

Report

- **Export XML report** will generate an XML file describing which graphics are placed on which layout.
- **Export JPEG thumbnail** will generate an RGB JPEG image of each layout. The resolution for the JPEG file can be defined.

9. Menus and Toolbars

This chapters describes all toolbars and menu items.

- [Menus](#)
- [Toolbars](#)

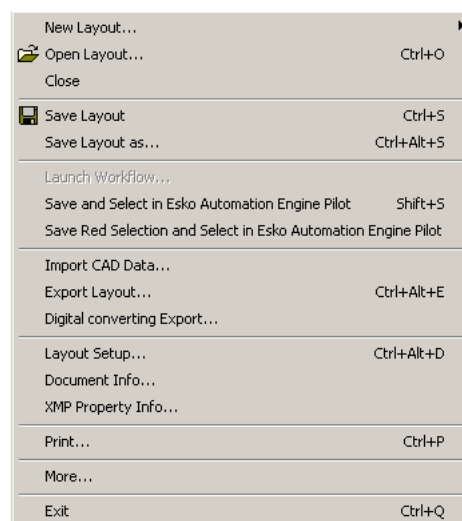
9.1. Menus

The menu bar contains commands available in the current module. Not all commands on all menus may be available; availability depends on the options purchased and the active tool.

- [File](#)
- [Edit](#)
- [View](#)
- [Arrange](#)
- [Station](#)
- [Inks](#)
- [Tools](#)
- [Production](#)
- [Trapping](#)
- [Window](#)
- [Help](#)

9.1.1. File menu

The File menu contains following items:



- [New Layout...](#) offers you the possibility to create a [Grid-based Layout](#), a [CAD Layout](#) or a [SmartLayout](#). You can also set up a new layout with the help of the [Layout Assistant](#).
- [Open Layout...](#) allows you to open an existing layout.
- **Close** closes the job you are currently working on.
- **Save Layout** saves the layout you are currently working on.
- **Save Layout as...** saves the layout under another name.
- **Save and Select in Automation Engine Pilot** automatically selects the open file in the Automation Engine Pilot. When the document has been changed, you will be prompted to save the changes.

Note:

The Automation Engine Pilot needs to be running.

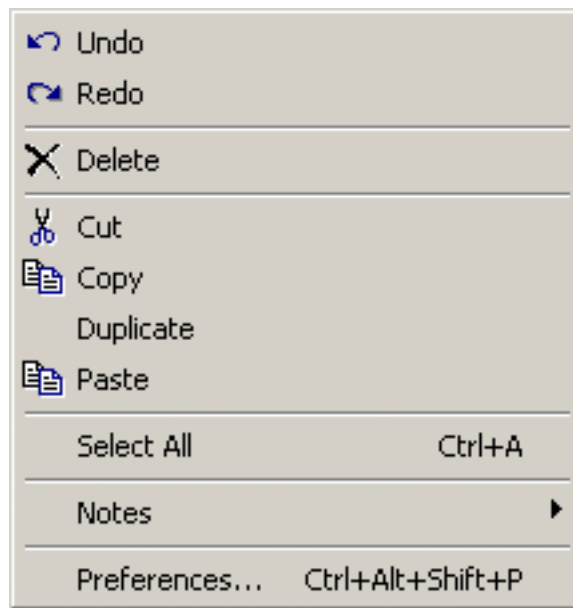
- **Save Red Selection and Select in Automation Engine Pilot:** saves the red selection and automatically selects the open file in the Automation Engine Pilot. When the document has been changed, you will be prompted to save the changes.
- [Import CAD Data...](#) allows you to import a CAD layout.
- [Export Layout...](#) allows you to export a layout.
- [Digital Converting Export](#) on page 277 allows you to export both a print file and a cut file.
- [Layout Setup...](#) opens the Layout Setup dialog box.
- In the [Document Info](#) dialog box you can enter all (administrative) information related to the job. The info can also be used as smart text in a text mark.
- [XMP Property Info...](#) opens the XMP Metadata Properties dialog box.
- [Print...](#) allows you to proof or expose the design opened in Esko Plato.
- **Exit:** Closes the application.

Note:

The history list of file names provides fast access to recently used files. Simply click one of the names to open the file.

9.1.2. Edit menu

The Edit menu contains following items:



- **Undo:** To cancel the last editing function (or a series of editing functions) you performed. The editing functions are undone one editing function at a time, starting from the most recent one. The last modification you made is undone and the job appears as it was before you performed the editing function. Continue to choose Undo to cancel other modifications that you made to the job.

Note:

The number of undos you can perform depends on the number of *Undo Levels* entered in Preferences.

Notice that 'remembering' the previous actions consumes memory that cannot be used for your current job anymore. If you are working with extremely complex jobs, it may be necessary to enter a lower number of undo levels.

- **Redo:** To cancel the last undo (or series of undos) you performed. Redo cancels one undo at a time, starting from the most recent occurrence and displaying the job as it was before you performed an undo. Continue to choose Redo to cancel other undos that you made to the job.

Note:

The number of redos you can perform depends on the number of *Undo Levels* entered in Preferences.

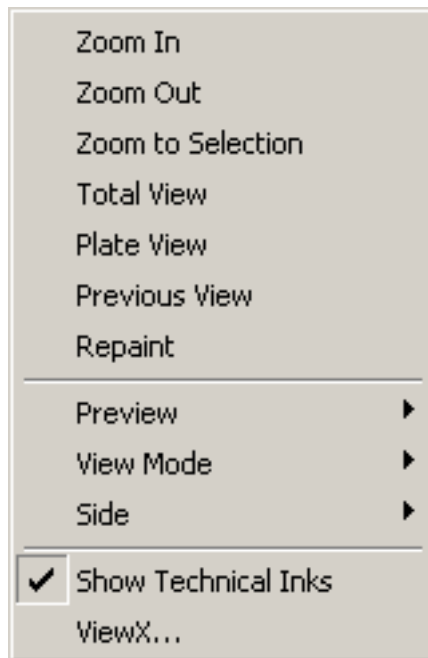
If you change something to your job after an undo action, you cannot redo it anymore, or - in other words - redo only works immediately after an undo action.

- **Delete:** To delete an object or a group of objects.
- **Cut:** To delete selected red objects from their current position in order to paste them into another position.
- **Copy:** To copy objects selected in red in order to paste them into another position.

- **Duplicate:** To duplicate any (group of) object(s) at its exact location. You can move the duplicate to a new location. The original object is kept.
- **Paste:** To paste cut or copied objects from the clipboard.
- **Select All:** To select all objects within a job. The objects are displayed in red to indicate selection, and can be manipulated or modified.
- **Notes:** see [Notes](#) on page 430
- [Preferences...](#): To define standard settings to be used by the application as you work on a job.

9.1.3. View menu

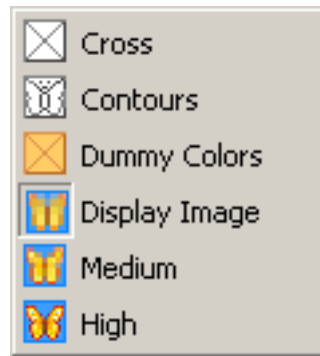
The View menu gives access to commands that determine how the imposition job is previewed.



The View menu contains following items:

- **Zoom In:** To zoom in (by a factor of 2) on the displayed job in the active window.
- **Zoom Out:** To zoom out (by a factor of 2) on the displayed size of the job in the active window.
- **Zoom to Selection:** This function allows you to automatically zoom in on the selected area in your job.
- **Total View:** To display the entire job, including objects outside the plate, after you have used any of the other View options.
- **Plate View:** To zoom to the plate.
- **Previous View:** To display the previous view of the active window after you have used any of the other View options.
- **Repaint:** To repaint the active window to remove stray pixels or to finish painting a large file on screen.

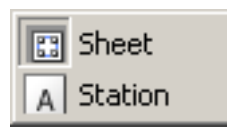
- **Preview** opens the *Preview menu*.



Tip:

The Preview Mode buttons also appear in the Preview Selector toolbar that can be added to the work environment via Window > Toolbars > Preview Selector.

- **View Mode** opens the *View Mode submenu*.



- **Side** allows you to switch between the front and back side of the plate.
- **Show Technical Inks** allows you to view the Technical inks in the job window.
- *Viewer* opens **Viewer**, a very accurate separation viewer with a lot of extra production tools.

9.1.4. Arrange menu

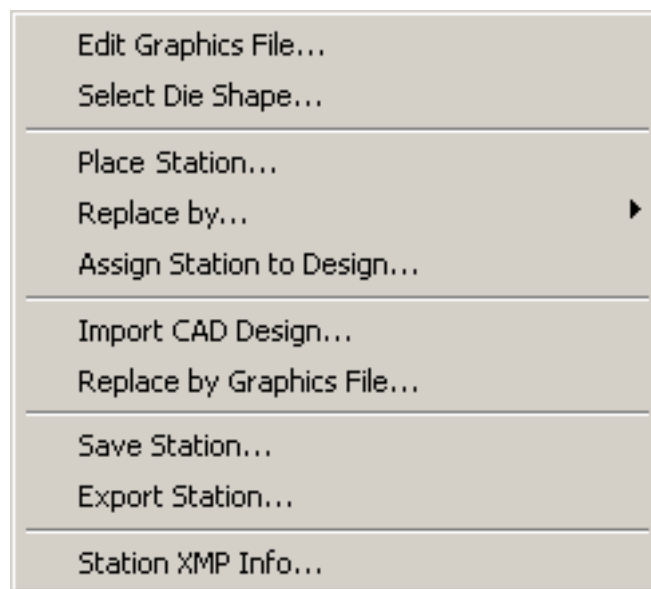
The Arrange menu contains following items:



- **To Front:** To move an object on top of all other objects within the current layer.
- **To Back:** To move an object underneath all other objects within the current layer.
- **Forward:** To move an object one step up within the current layer.
- **Backward:** To move an object one step down within the current layer.
- **Group:** To group objects selected in red.
- **Group into Grid:** Tries to recognize individual stations as grid blocks.
- **Ungroup:** To ungroup objects after they have been grouped.
- **Snap Crosshair:** To make the crosshair jump to the last selected point.

9.1.5. Station menu

The Station menu contains following items:



- **Edit Graphics File...** opens the selected file in PackEdge.
- **Select Die Shape...** allows you to select a *die shape*.
- **Place Station...** allows you to *place a station* in the layout you are currently working on.
- **Replace by...** allows you to *replace a station*.
- **Assign Station to Design...** allows you to *assign a station to a CAD design*.
- **Import CAD Design...** allows you to *import a CAD Design*.
- **Replace by Graphics File...** allows you to *replace a graphic*.
- **Save Station...** allows you to *save the selected station*.

- **Export Station...** allows you to *export the station*.
- **Station XMP Info...** shows you the *XMP info of the station*.

9.1.6. Inks menu

The Inks menu contains following items:

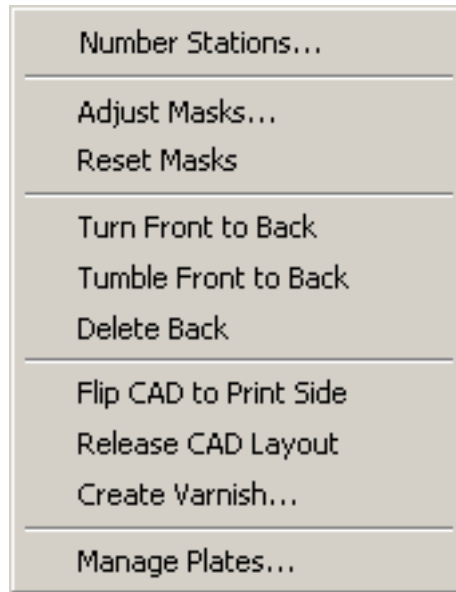
Inks...	Alt+Ctrl+I
Ink Mapping...	Ctrl+Alt+J
External File Ink Mapping...	Alt+Shift+X
<hr/>	
Ink Coverage...	Alt+Ctrl+K
Create Ink Histogram...	
Create Ink Eater Areas...	
Fill Ink Eaters...	
<hr/>	
PANTONE® Solid Coated...	Alt+Shift+L
Ink Books...	Alt+Shift+B

- *Inks...*: To add, modify or remove Inks and to display the current inks in your job with their properties.
- *Ink Mapping...*: To map inks in the job.
- *External File Ink Mapping...*: To replace the ink separations of a external job by any color from the color palette or by any ink from the Inks dialog box.
- *Ink Coverage...*: To estimate the needed ink coverage for the job.
- *Create Ink Histogram...*: To add an Ink Histogram to the plate.
- *Create Ink Eater Areas...*: To create areas which you can use for ink compensation.
- *Fill Ink Eaters...*
- *Open Ink Book*: To open the ink book of your choice.
- *Manage Ink Books...*: To manage the content of the ink books.

9.1.7. Tools menu

Note:

This menu will only be enabled in the Sheet View.

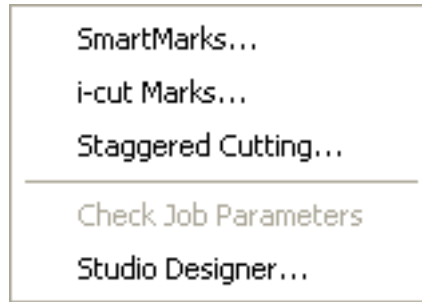


The Tools menu contains following items:

- **Number Stations...:** opens the *Number Stations dialog box* .
- The *Adjust Masks* tool is used to adjust overlapping stations.
- **Reset Masks** removes all masks that were generated using the 'Adjust Masks' tool.
- **Turn Front to Back** will rotate the sheet around the vertical center of the sheet and create the back of the file. With this command, the left edge of the sheet becomes the right edge of the sheet on the back.
- **Tumble Front to Back** will rotate the sheet around the horizontal center of the sheet and create the back of the file. With this command, the top of the sheet becomes the bottom of the sheet on the back.
- **Delete Back** deletes the back side of the sheet.
- **Flip CAD to Print Side**
 You may have CAD designs that show Die Side. If this is the case, you will need to "flip" the CAD design over so that you can work with the Print Side.
 For more information, please refer to the chapter '*Flipping a CAD Layout to Print Side*'.
- **Release CAD Layout** removes the CAD layout but keeps all associated stations. After releasing the CAD layout, the stations no longer are tied up together and can be moved individually.
- *Create Varnish...* creates a varnish layer.
- **Manage Plates...** opens the *Manage Plates dialog box*.

9.1.8. Production menu

The Production menu contains following items:

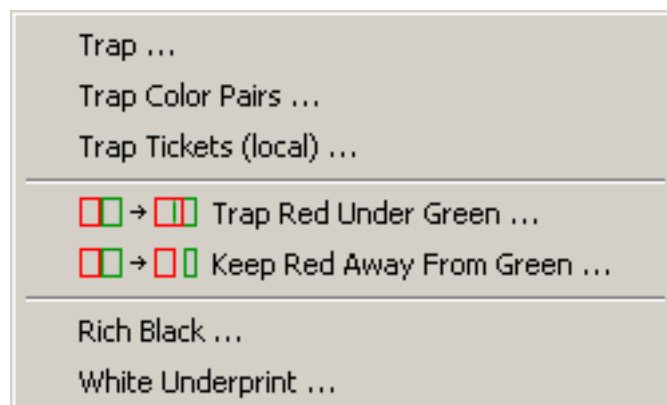


- [SmartMarks.....](#) opens the SmartMarks dialog box .
- [i-cut Marks...](#) opens the *i*-cut Marks dialog box.
- [Staggered Cutting](#) facilitates seamless printing in non-sleeve workflows.
- [Check Job Parameters](#) allows you to check the inks in your files for inconsistencies regarding Job parameters that are predefined for a Automation Engine Job and allows you to correct them (if necessary).

9.1.9. Trapping menu

Note:

This menu will only be enabled in the Station View and requires a dedicated trapping licence for Esko Plato.



Trapping creates overlapping areas to compensate for possible misregistration between the different inks on the printed result. It is used whenever two adjoining colors contain different inks in such a way that misregistration during printing results in a white gap or an undesirable line of a different color at the border of the two colors.

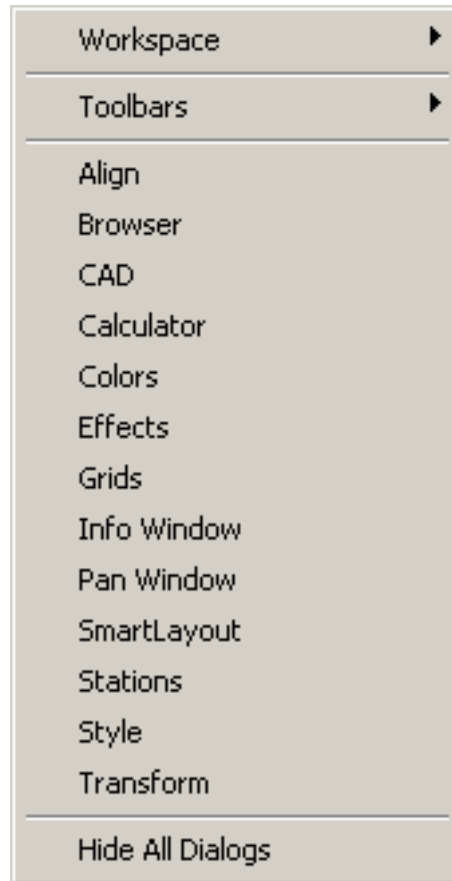
Please refer to the chapter '[Trapping](#)' for more information.

The Trapping menu contains following items:

- [PowerTrapper...](#)
- [Trap Color Pairs...](#)
- [Trap Red Under Green...](#)
- [Keep Red Away From Green...](#)
- [Rich Black...](#)
- [White Underprint...](#)

9.1.10. Window menu

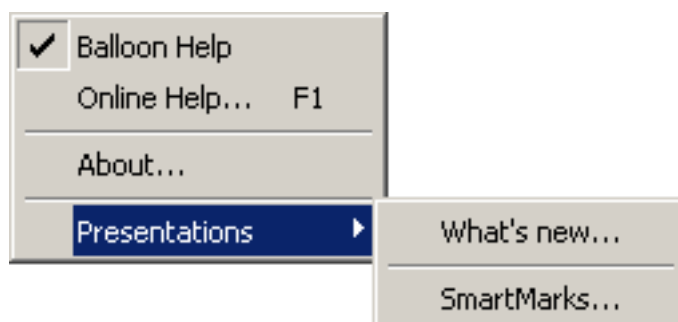
The Window menu contains following items:



- **Workspace** allows you to create predefined [workspaces](#).
- **Toolbars** allows you to view and hide individual [toolbars](#).
- **Align** opens the [Align dialog box](#).
- **CAD** opens the [CAD dialog box](#).
- **Calculator** opens the [Calculator](#).

- **Colors** opens the *Colors dialog box*.
- **Effects** opens the *Effects dialog box*.
- **Grids** opens the *Grids dialog box*.
- **Info Window** opens the *Info Window*.
- **Layer Browser** opens the *Layer Browser*.
- **Pan Window** opens the *Pan window*.
- **SmartLayout** opens the *SmartLayout dialog box*.
- **Stations** opens the *Stations dialog box*.
- **Style** opens the *Style dialog box*.
- **Transform** opens the *Transform dialog box*.
- **Hide all Dialogs** will hide all opened dialog boxes.

9.1.11. Help menu



While you are learning or working with ESKO Plato, you may want to consult other Help features.

Balloon Help

ESKO Plato has Balloon Help to get you up and running as quickly as possible. Balloons appear when you point the cursor at an icon or command. If a keyboard shortcut is available for the item, it will also be shown in the balloon help.

Formula-enabled fields also have balloon help: it shows both the formula and the corresponding value.

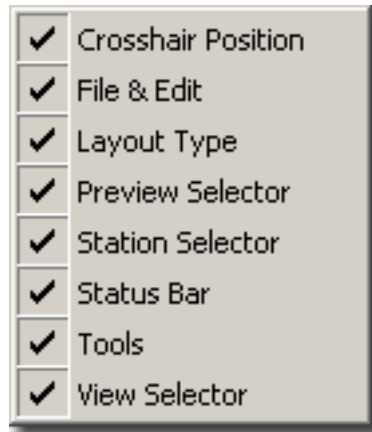
Balloon Help is turned on by default. As you become more familiar with ESKO Plato, you may want to turn the balloons off. Simply click the Help menu and select Balloon Help. The balloons can be turned on and off with this command.

ESKO Plato's **Online Help** contains the entire contents of the User's Guide in electronic form. You access Online Help by selecting 'Online Help' on the Help menu. Selecting a topic will jump directly to that section of the User's Guide. You also open Online Help using the keyboard shortcut F1.

Presentations leads you to the What's new in ESKO Plato and the SmartMarks presentations.

9.2. Toolbars

All toolbars can easily be shown or hidden easily via the toolbars list in the Window menu:



- [Crosshair Position](#)
- File & Edit
- [Layout Type](#)
- [Preview Selector](#)
- [Station Selector](#)
- [Status Bar](#)
- [Tools](#)
- [View Selector](#)

All toolbars can be docked around your drawing window.

Tip:

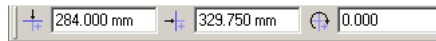
The [Workspace Manager](#) allows you to make some predefined workspaces of the user interface dialog boxes and switch from one to another in one single click.

Note:

For information on the Shuttle toolbar, we refer to [Shuttle](#) on page 26

9.2.1. Crosshair Position

This toolbar indicates the vertical and horizontal position of the crosshair and the rotation angle. It also allows you to set the position or rotation of the crosshair numerically. Simply enter the requested position or angle in the proper field.



Note:

Manual crosshair manipulations, such as moving and rotating, are also possible whenever the Select or Transform tool is activated.

9.2.2. Preview Selector

The Preview Selector contains a number of elements that offer a different visualization of the job.



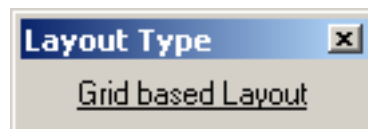
- [Preview selector buttons](#)
- Zoom: indicates and sets the zoom factor.
- Show Trim & Media box: allows you to switch the visualization of the trim and media box (respectively sheet and plate edge) on or off. Switching off the blue or black lines indicating the trim/media box can sometimes be useful to verify objects that are placed near or on the trim/media box.
- Show values / formulas: adjusts the visualization of formula-enabled fields. You can opt to either show the formula definition or the corresponding value.

9.2.3. Layout Type

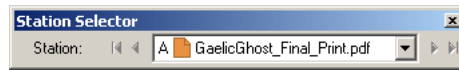
Layout type indicates the selected workflow: grid-based, CAD, SmartLayout, or manual.

Tip:

Click the layout button to open the appropriate Layout dialog box (Grids, CAD, or SmartLayout).



9.2.4. Station Selector

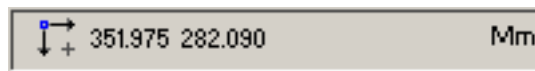


The Station Selector is active in the Station View. It lists all available stations for the current job. Selecting a station (from the drop-down list or using the arrows) opens the selected station in Station View.

Tip:

Click the station button to open the Station dialog box in the Sheet View.

9.2.5. Status Bar



The Status Bar indicates the position of the cursor and displays the unit in which your job is set. The server icon



indicates to which Esko Automation Engine server you are connected.

Note:

You cannot change units while working in a job.

In order to change units, select 'Preferences' on the Edit menu and select the correct unit in the Defaults tab. Changes will become effective when creating a new layout.

9.2.6. Tools

These tools are the interactive tools that you use to build a plate. You can open and close the toolbar via the Toolbars > Tools selection on the Window menu.

The toolbar contains following tools:



- [Select Tool](#)
- [Transform Tool](#)
- [Create Layout Tool](#)
- [Place Station/Strip Number Tool](#)

- [Geometry Marks Tool](#)
- [Redraw Path Tool](#) on page 309
- [Modify](#)
- [Crosshair Tool](#)
- [Draw Die Shape Tool](#)
- [Create Bleed Mask Tool](#)
- [Zoom Tool](#)
- [Pan Tool](#)
- [Measure Tool](#)
- [Densitometer Tool](#)
- [Notes tool](#)
- [Working with tools and dialog boxes](#)

Select Tool

Use the Select tool to select objects in the document window. You must select objects before you can transform them. Selected objects are always highlighted in red. If you select objects but they are not highlighted, make sure that the "[Show Selection in Preview](#)" box is checked in Preferences.

Esko Plato supports two types of selection: red selection and green selection. Objects selected in red can be transformed and modified. Objects selected in green cannot be transformed or modified. Esko Plato's Align window uses green selections to align objects to one another.

- [Selecting an object](#)
- [Selecting an object in green](#)
- [Selecting multiple objects](#)
- [Selecting all objects](#)
- [Deselecting an object](#)

Selecting an object

To select individual objects:

1. Click Select on the toolbar. The Select button changes to a blue button showing that it is the active tool.
2. Click the object you want to select.

The object is selected in red, indicating that it can be transformed or modified.

Selecting an object in green

Use green selection to select an object while keeping it from being transformed or modified. To select an object in green:

1. Click Select on the toolbar.
2. Click the object that you want to select.
3. Press the INSERT key to change the selection from red to green.

If you have some objects selected in red and some selected in green, pressing the INSERT key will toggle the objects between the two selections. That is, objects selected in red become selected in green and objects selected in green become selected in red.

**Caution:**

The Insert key is an Esko shortcut. If you are using DTP shortcuts, or created your own shortcuts, the key could be different.

For more information about shortcuts, please refer to the chapter ['Shortcuts'](#).

Selecting multiple objects

To add an object to the current selection:

1. Click Select on the toolbar and click the first object.
2. Hold SHIFT and click each additional object that you would like to add.

**Caution:**

The Shift key is an Esko shortcut. If you are using DTP shortcuts, or created your own shortcuts, the key could be different.

For more information about shortcuts, please refer to the chapter ['Shortcuts'](#).

To select all objects within an area:

1. Click Select on the toolbar.
2. Position the cursor outside the objects you want to select, and click, hold and drag the mouse to create a box around the objects. Release the mouse button.

You must entirely enclose an object with the selection box for it to be selected.

Selecting all objects

To select all objects in the document:

On the Edit menu, click 'Select All'.

Deselecting an object

You can only deselect objects that are selected in red. To deselect objects selected in green, you must first change them to objects selected in red by pressing the INSERT key.

**Caution:**

The Insert key is an Esko shortcut. If you are using DTP shortcuts, or created your own shortcuts, the key could be different.

For more information about shortcuts, please refer to the chapter ['Shortcuts'](#).

To deselect every object in the red selection:

1. Click 'Select' on the toolbar.
2. Click anywhere in the document window outside the selected objects. All objects selected in red become deselected.

To deselect a single object: right-click the object.

Tip:

There is a [preference setting](#) for reversing the selection of an object: SHIFT+select the object by dragging.

Transform Tool

Use the Transform tool to interactively move, scale or rotate a selected object. You can also transform objects numerically using the [Transform dialog box](#). Some objects in Esko Plato cannot be transformed or can be transformed only partially. The following table shows the objects that may be in your document and their transformation properties:

Object Type	Move	Scale	Rotate
CAD Layout	Yes	No	No
Grid blocks	Yes	No	No
Stations	Yes	No	Yes
SmartMarks	Yes *	No	No
Geometry Marks	Yes	Yes	Yes

Note:

* Certain SmartMarks cannot be moved, e.g. the grid mark.

Click the Move tool to activate the fly-out toolbar. Select the appropriate tool and to enter a step value if needed.



- [Using the Transform Reference box](#)
- [Moving the point of origin](#)
- [Using constrain](#)
- [Moving an object](#)
- [Scaling an object](#)
- [Rotating an object](#)

- [Transforming an object by a step value](#)
- [Transforming an object numerically](#)
- [Transforming in the Station View](#)

Using the Transform Reference box

When you use the Transform tools, a red reference box with selection handles appears around the objects that have been selected in red. The individual objects in this Transform reference box are modified as one object with the Transform tools.

You can make transformations using the selection handles or the outline of the reference box. If you select one of the handles or the outline of the reference box, this becomes your point of reference, the point that you are transforming. The point of reference does not have to be on the reference box; you can also pick any location on a selected object too.

Moving the point of origin

The point of origin is the point around which transformations will occur. When you select an object, the point of origin is in the center of the object. It is shown as a small circle with a point in it.

You can move the point of origin for the scale and rotate transformations. For example, when you rotate an object, it rotates around the point of origin, or the center in the default case. If you move the point of origin to some other location, Esko Plato will rotate the object around the new point of origin.

If you cannot locate the point of origin when you use the Transform tools to modify objects (for example, if you zoomed in on the document window and the origin is located outside the display area), press [F11] and click a point. The point of origin appears where you clicked.

Using constrain

The transform tools can be set to constrain their transformations. For example, you might want to constrain rotations to 45 and 90 degrees. Or you may want to move an object vertically. These are examples of constraints on the Transform tools.

Once you turn constrain on, it is active until you turn it off again. This is important to remember because some tools, like the Geometry tool, also use constrain.

To turn constrain on (or off), press F3 .

You can also hold the Shift key whilst dragging.

Moving an object

Use the Move option of the Transform tool to move an object to a new location.

To move an object:

1. Select the object or objects you want to move.
2. Click Transform on the toolbar. The fly-out toolbar appears.
3. Click the Move button.
4. Click and hold on the reference box (not one of the handles) and drag the box to a new location.

Scaling an object

Use the Scale option of the Transform tool to scale an object relative to its point of origin. To scale an object proportionally:

1. Select the object or objects you want to scale.
2. Click Transform on the toolbar. The fly-out toolbar appears.
3. Click the Scale button.
4. Move the point of origin to the appropriate location, if necessary.
5. Click and drag one of the four corner points on the Transform reference box.

To scale an object horizontally:

1. Change to the Move button on the fly-out toolbar.
2. Click and drag one of the horizontal points on the Transform reference box.

To scale an object vertically:

1. Change to the Move button on the fly-out toolbar.
2. Click and drag one of the vertical points on the Transform reference box.

Rotating an object

Use the Rotate option of the Transform tool to rotate an object relative to its point of origin. To rotate an object:

1. Select the object or objects you want to rotate.
2. Click Transform on the toolbar. The fly-out toolbar appears.
3. Click the Rotate button on the fly-out toolbar.
4. Click and hold on the reference box or on one of the objects in the reference box and drag it to a new rotation.

Transforming an object by a step value

You may want to transform an object in a number of steps. This is nudging. You use the arrow keys on the keyboard to nudge objects by a step amount. To transform an object by a step value:

1. Select the object or objects you want to modify.
2. Click Transform on the toolbar. The fly-out toolbar appears.
3. Click the appropriate transform button (Move, Scale or Rotate).
4. Enter a step value in the Step Amount field.
5. Use the arrow keys on the keyboard to transform the selected objects by the step value.

You can apply the step value to transform the objects in the following ways:

- Press CTRL+ [an arrow key] to apply 0.1 times the step value.
- Press CTRL+ ALT+[an arrow key] to apply 0.01 times the step value.

Transforming an object numerically

With the Transform tool, you interactively transform objects by selecting them and applying the transformation yourself. You can also transform objects numerically, using the Transform window. The Transform window allows you to enter exact values for the placement, scaling and rotation of selected objects. See Using the Transform Window.

Transforming in the Station View

The Transform tool in the Station View works the same way as in the Sheet View. Remember, however, that the measurements for position are relative to the current origin. Since the borders of the station are always fit to the current die, the Transform window's position measurements usually are relative to the die. If the origin of the station has been moved manually, the measurements in the Transform window will be relative to the new origin, which may not be the die.

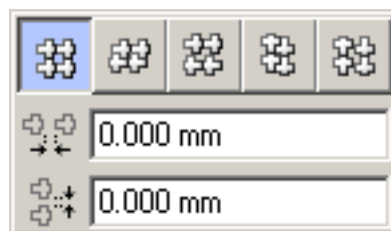
Create Layout Tool

This tool is subject to a dedicated license. If you do not have the full license for this tool, only the first tool of the fly-out toolbar will be available.

The distance, or *gutter*, between stations is set to zero by default. You can change it before using the Create Layout tool. When reversing rows or columns, it is possible to specify secondary gutter values. This option is only supported in CADX 7.17 or higher.

Alternatively, you can change it while using the Layout tool by specifying different step values between the rows and columns in the edit boxes on the Status bar.

Click the Straight Nest tool or choose one of the other tools from the fly-out toolbar shown below.



The first tool on the fly-out toolbar is the **Straight Nest** tool, which performs no rotation or alignment on the single designs as it arranges them.

The second tool on the fly-out toolbar is the **Reverse Second Row Nest** tool, which flips the design horizontally every other row.

The third tool on the fly-out toolbar is the **Reverse Second Row Align Nest** tool, which works similarly to the Reverse 2nd Row Nest tool, except that it butts each row of designs against the edge of the first station instead of nesting them as close together as possible.

The fourth tool on the fly-out toolbar is the **Reverse Second Column Nest** tool, which flips the design vertically every other column.

The fifth tool on the fly-out toolbar is the **Reverse Second Column Align Nest** tool, which works similarly to the Reverse 2nd Column Nest tool, except that it butts each column of designs against the edge of the first station instead of nesting them as close together as possible.

Once you select a Nest tool, as you move the drag with the mouse, ghost images of the design appear. Next to the mouse pointer, the total number of nested stations is indicated.

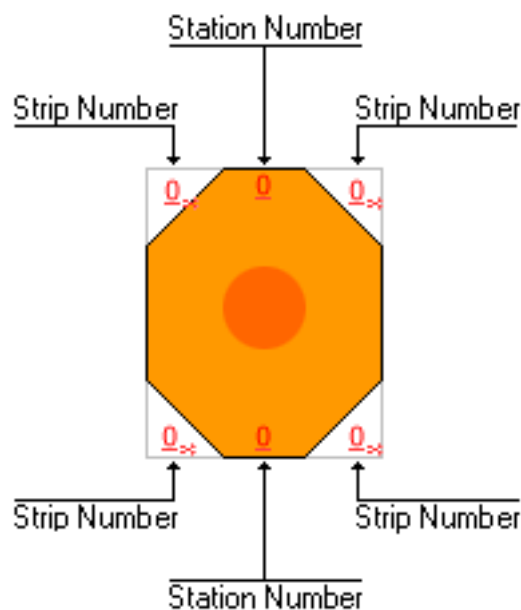
Please refer to the chapter '[Nesting stations](#)'.

Place Station/Strip Number Tool

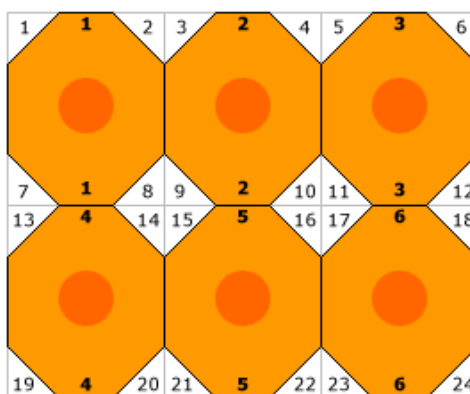
This tool is active only in Station View mode. Click the tool button to activate the Strip/Station Number dialog box.

The Place Station/Strip Number tool will create station or strip number placeholders for your station. A **Station Number** is a small number that is often placed on a hidden area of the package to identify the location of the package on the sheet. Station Numbers are often called die numbers or cell numbers. The placeholder specifies attributes such as position, font, size and the ink(s) in which the number is to be printed. If multiple station numbers are placed on a single station, they all get the same value. **Strip numbers** on the other hand are numbered in a continuous way. If multiple strip numbers are placed on a single station, they all get a different value.

In the Station View you can define the position and appearance of the station/strip number.



Expansion will happen in the Sheet View.



Geometry Marks Tool

Use the Geometry Marks tool to create marks that are simple rectangles, ovals and polygons. Unlike SmartMarks, Geometry Marks do not update automatically. However, Geometry Marks that are designated as Registration color or darkest color will update when inks are added to the plate. You can use Geometry Marks to add new inks to the plate. Geometry Marks can be transformed like any other object. There is no limit to the number of geometry marks in one plate.

- [Choosing the shape of Geometry Marks](#)
- [Setting the style of Geometry Marks](#)
- [Adding Geometry Marks](#)
- [Geometry Marks in the Station View](#)

Choosing the shape of Geometry Marks

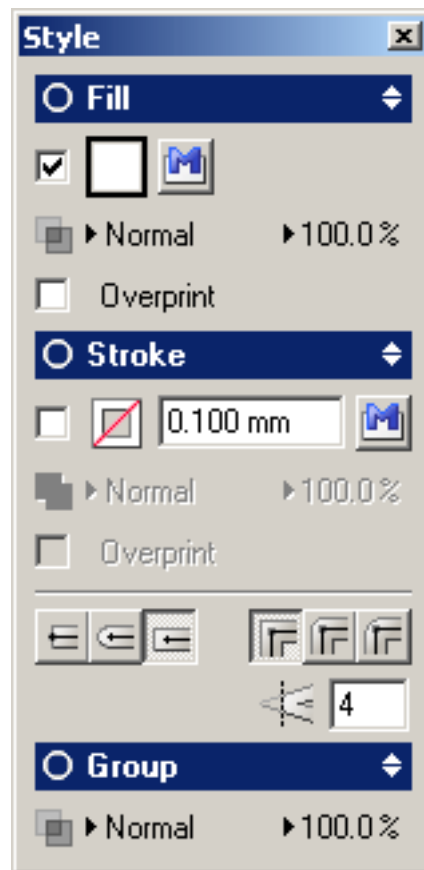
Click the Draw Rectangle button on the Tools toolbar to activate the fly-out toolbar. Select the appropriate tool; draw rectangle, draw ellipse or draw polygon.



Setting the style of Geometry Marks

The Style window allows you to set the parameters for the mark that you create. You can choose whether your mark has a Fill, a Stroke or both. You can choose the colors of the fill and stroke as well as the line width for the stroke. You can also set colors in Geometry Marks to overprint other objects. Finally, you can choose for the color to be Registration color, which means that the color contains 100% of all inks in the document.

To set the Style of a Geometry Mark, use the Style dialog box:



Please refer to the chapter *'Style'* for more information on the various options.

Adding Geometry Marks

To add a Geometry Mark:

1. Click the 'Draw Rectangle' button on the toolbar.
2. Select the type of mark you would like to create.
3. Set the parameters for the mark in the Style dialog box.
4. Draw the mark.

The way that you draw the mark depends on the type of mark you are creating.

To draw a rectangle:

1. Click the 'Draw Rectangle' button on the toolbar.
2. Starting where you want the top left corner of the rectangle, click, hold and drag to create the rectangle. You can also simply click where you would like the top left corner of the rectangle and the bottom right corner of the rectangle. Holding the ALT key while dragging will draw the rectangle from the center. You also use the rectangle tool to create a square. You constrain the rectangle to a square by turning on constrain. See *Using constrain*.

Tip:

Note that a *preference setting* allows you to draw a rectangle in a single click.

To draw an ellipse:

1. Select the 'Draw Ellipse' button from the fly-out toolbar.
2. Starting where you want the top left corner of the ellipse, click, hold and drag to create the oval. You can also simply click where you would like the top left corner of the ellipse and the bottom right corner of the ellipse. Holding the ALT key while dragging will draw the ellipse from the center. You also use the ellipse tool to create a circle. You constrain the ellipse to a circle by turning on constrain. See [Using constrain](#).

Tip:

Note that a [preference setting](#) allows you to draw an ellipse in a single click.

To draw a polygon:

1. Select the 'Draw Polygon' button from the fly-out toolbar.
2. Click where you want the first point of the polygon.
3. Click to create each additional point in the polygon.
4. Type 2 to close and end the contour. Type 3 to end the contour without closing it.

Note:

The Geometry Marks tool also allows you to draw splines. A simple click will create a new line segment. If you click, hold and drag the mouse spline segments will be added.

Geometry Marks tool tips

- Once you have set the parameters for the Geometry Mark tool, you can close the Style dialog. All marks that you create with the tool will use the parameters that you set in the dialog.
- Geometry Marks can be transformed using the [Transform tool](#) on the toolbar or the [Transform window](#).
- If you want to change the parameters for a Geometry Mark that you have created already, double-click it. The [Style dialog box](#) for the selected mark will appear.

Geometry Marks in the Station View

Geometry Marks also work in the Station View. However, remember that the available inks shown in the Geometry Marks Style window include inks used on the current side of the station as well as any unused inks.

If you have a two-sided station with graphics placed on both sides, the inks for the graphic on the current side are available for use in your Geometry Mark. If you need another ink, add the ink in the Inks dialog box before creating the Geometry Mark. See [Geometry Marks tool](#).

Modify (toolbar)



To determine the modification you want to execute.

Tip:

Modify in combination with Numbers.

If the Modify tool is activated, you can move points numerically. In the absolute value part, only the input boxes for vertical, horizontal and diagonal distance and for rotation are present. In the relative value part, the values for diagonal distance, rotation and vertical and horizontal move are present. All Modify actions are performed immediately (no Apply).

Click Modify on the toolbar. The Modify Rosette appears. The highlighted button indicates which Modify tool is activated.

Click another Modify tool in the Modify Rosette or press the corresponding numeric keypad key (with [NumLock] activated) to activate the desired Modify function.

You can hide the Modify Rosette while you are working with Modify. To toggle between displaying and hiding the Modify Rosette press the [Enter] key while Modify remains activated.

- Press z to convert to Bézier mode
- Press x to convert to free curve mode
- Press c to convert to frame mode

Modify Rosette

Move Point



To move points and anchor points on a segment or an object. When you move the points, you can change the size and shape of the segment or object.

Slide Anchor



To slide anchor points along the lines of a frame (on a curved segment), or along a line (on a line).

Trim Line



To move the segment, if you are working in frame mode. The corners with the adjacent segments remain the same.

Cut by Cursor



To divide a segment or object. The divided pieces of the segment or object can be manipulated or modified individually.



Attention:

- You must click a point on the segment or object, not a point on the frame or bounding box.
- The newly created objects have no properties.

Make Corner Smooth



To change a curve into a corner, and vice versa.



Attention:

When you click a point, it toggles between Make Corner and Corner Smooth.

Delete Point



To delete points from a segment or object. By deleting the points, you can change the shape of the segment or object.

Direction Points



To display all handles on the selected segment or object when you use the Modify tools.

Handles can be displayed (in red) to enable you to easily move and manipulate the tension of the curve.

The type of handle depends upon the Digitize tool that was used to create the segment.

For example:

- If you created the segment with Add Segment, anchor points and handles display.
- If you created the segment with Add Arc or Add Curve, only points display.
- In frame mode extra handles appear on the frame to manipulate the tension of the curve.

Crosshair Tool

The sixth button on the Tools toolbar activates the Move Crosshair tool. Click to activate the Crosshair fly-out toolbar.



When you click this button , you can move the crosshair manually. You can move the vertical or horizontal axis by clicking and dragging either one of them. If you want to move the crosshair vertically and horizontally at once, you click and drag the center. If you click the left mouse button anywhere in your job, the center of the crosshair jumps to that point. With the right mouse button you can rotate the crosshair manually by clicking and dragging it. If you click the right mouse button anywhere in your job, the crosshair is rotated to that point.

You can align the crosshair on a rotated object (in this case, NOT on the bounding box of the rotated object). To align the vertical axis, you click a point on the object with the left mouse button. To align the horizontal axis, you click a point on the object with the right mouse button.

Note:

Even if the object is not selected, there is gravity.

Rotate 

When you click this button, you can rotate the crosshair manually by clicking and dragging it. When you click anywhere in your job, the crosshair will be rotated to that point.

Note:

Manual crosshair manipulations, such as moving and rotating, are also possible whenever the Select or Transform tool is activated.

Tip:

To position or rotate the crosshair numerically, use the crosshair toolbar.

Note:

The rotation angle strictly refers to the rotation of the crosshair itself and is in no way related to the distance by nor to the direction in which it was moved. Click Apply to perform the Crosshair action.

Note:

Objets snap to the crosshair. Please find the adjustable value in the preferences.

Draw Die Shape Tool

In Station View mode, use the Draw Die Shape tool to draw die shape contours for your station. The Station View's default die is a rectangle (or a square, if the height and width are the same). But some jobs are not this shape. You can use the Draw Die Shape tool to create die contours that are rectangular or square, oval or round, or a polygon. You are not limited to a single die contour; you can create multiple contours to create a more complex die. All contours created with the Draw Die Shape tool are die cut lines. Die contours can be transformed with the Transform tools to create the exact shape and size that you need.

- [Choosing the shape of the Draw Die Shape tool](#)
- [Drawing Die Shape contours](#)

Choosing the shape of the Draw Die Shape tool

The seventh button on the Tools toolbar activates the Draw Die Shape fly-out toolbar.



Drawing Die Shape contours

To add a die shape contour:

1. From the toolbar, click 'Draw Die Shape'.
2. Select the shape of the contour you want to create from the fly-out toolbar.
3. Draw the die contour.
4. The way that you draw the contour depends on the shape of the contour you are creating.

To draw a rectangle:

1. Select the 'Draw Die Shape - Rectangle' button from the fly-out toolbar.
2. Click where you want the top left corner of the rectangle.
3. Click, hold and drag to create the rectangle. You can also simply click where you would like the top left corner of the rectangle and the bottom right corner of the rectangle. Holding the ALT key while dragging will draw the rectangle from the center. You also use the rectangle tool to create a square. You constrain the rectangle to a square by turning on constrain. See [Using constrain](#). Note that a preference setting allows you to draw a rectangle with a single click.

Tip:

Note that a [preference setting](#) allows you to draw a rectangle in a single click.

To draw an ellipse:

1. Select the 'Draw Die Shape - Ellipse' button from the fly-out toolbar.
2. Click where you want the top left corner of the ellipse.
3. Click, hold and drag to create the ellipse. You can also simply click where you would like the top left corner of the ellipse and the bottom right corner of the ellipse. Holding the ALT key while dragging will draw the ellipse from the center. You also use the ellipse tool to create a circle. You constrain the ellipse to a circle by turning on constrain. See [Using constrain](#). Note that a preference setting allows you to draw an ellipse with a single click.

Tip:

Note that a [preference setting](#) allows you to draw an ellipse in a single click.

To draw a polygon:

1. Select the 'Draw Die Shape - Polygon' button from the fly-out toolbar.
2. Click where you want to be the first point of the polygon.
3. Click to create each additional point in the polygon.
4. Type 2 to close and end the contour. Type 3 to end the contour without closing it.

Tip:

- Die contours can be transformed using the Transform tool on the toolbar or the Transform window.
- Be aware of the number and complexity of die contours that you create. Since the Create Bleed Mask tool is used to create a mask from the die contours, you may see unexpected results in a station with many overlapping and complex die contours.
- For masking purposes, it is sometimes useful to create additional die contours to force the mask to be a certain shape or size. After the mask is created you can then select and delete the die contours that you no longer need.
- The color settings for manually drawn die contours can be set as a preference (Edit menu > Preferences > [View](#) > 'Show Manual Cut Lines' and 'Line Width').

Redraw Path Tool



Redraw Path Tool

To cut away the "smallest" part of the contour and to add the contour segment(s) that you draw to the "largest" part. Click to indicate a point on or close to the die shape (it will snap) and redraw a segment (which can consist of different points). The end point has to be on or close to the contour so it will snap again.

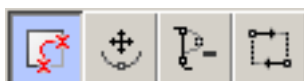
Tip:

When you want to put an intermediate point of your line or curve segment near a contour, click the middle mouse button instead of the left mouse button, to prevent that contour from being affected.

Repeat

When editing a Staggered Cutting Path, you can do an automatic redo of the redraw operation on all top labels using the **Repeat** button. This Repeat only works if the redrawn section is inside the width of 1 label. It moves the start and end point of the redrawn section with an offset equal to the distance between 2 labels on the top row, so these 2 points have to match with points on the currently selected path.

Contour Cleanup Tool





Cut Path and Redraw Shortest Part To cut away the "smallest" part of the contour and to add the contour segment(s) that you draw to the "largest" part. Click to indicate a point on or close to the die shape (it will snap) and redraw a segment (which can consist of different points). The end point has to be on or close to the contour so it will snap again.

Tip:

When you want to put an intermediate point of your line or curve segment near a contour, click the middle mouse button instead of the left mouse button, to prevent that contour from being affected.



Move Point To move points and anchor points on a segment or an object. When you move the points, you can change the size and shape of the segment or object.



Delete Point To delete points from a segment or object. By deleting the points, you can change the shape of the segment or object.



Change Start Point and Direction Click a point to define it as the starting point. Click the point again if you want to change the direction of the curve.

Nick Tool

Nicks are small pieces of substrate that are not cut by the table and thus create interruptions in the die shape when exporting to *i*-cut. As a result, the cut design will stay attached to the substrate. Nicks may be present in the design, but the nick tool allows you to add or remove extra nicks that guarantee tiles to stay connected. You can specify the size of the nick in the box. Click the die shape and add or remove nicks as you wish. When exporting, the position and size of the nicks will be saved in the .cut file.



Tip:

Press the Alt key to switch between the Add Nick and Remove Nick tools.

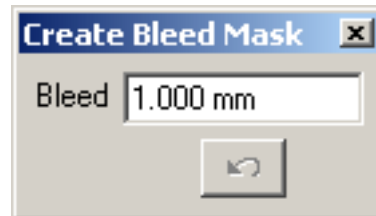
Create Bleed Mask Tool

Use the Create Bleed Mask tool to create a mask for the graphic. The mask is based on the station die. You can add a bleed amount to the mask. If the die in your station includes panels,

you can also specify which panels to include in the mask to create non-printing areas on the graphic.

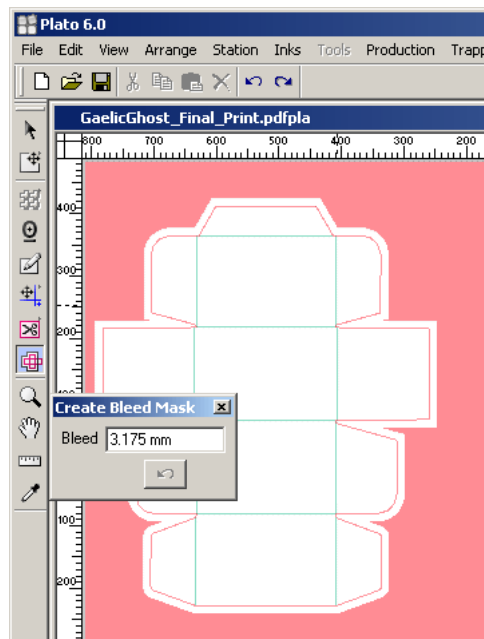
Masks created with the Create Bleed Mask tool are applied to the graphic only; other objects, like marks are not masked by the Create Bleed Mask tool.

The Create Bleed Mask tool includes an option window:



The option window allows you to specify the Bleed amount that you would like on the mask. The value that you enter is the amount of bleed outside the die. The option window also includes an Undo button. You use the Undo button to undo any panel selections that you have made.

When you use the Create Bleed Mask tool, the mask is displayed in a transparent color over the entire station. The color of the mask is the same as the color used for the Die Cut lines. By default, the Die Cut lines in Esko Plato are red, so the mask usually appears as a transparent red that simulates RubyLith masking material:

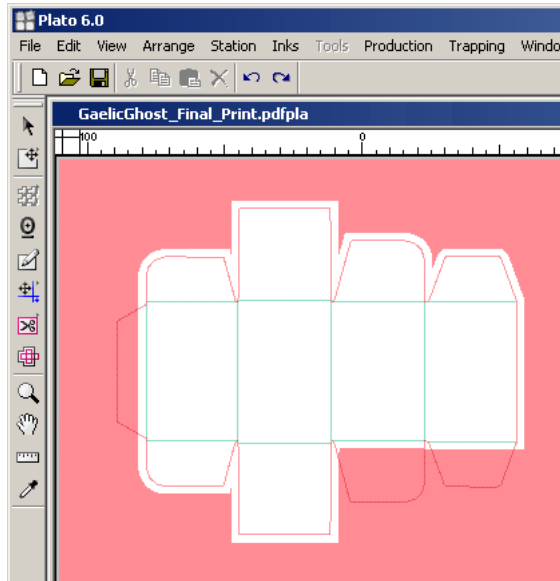


The transparent red area indicates the part of the station that will not be shown. Any part of the graphic that is under the red transparent area will be not be visible or printable. The inside of the mask will allow the graphic to show through.

You can add specific panels to the mask by clicking on them. Each time you click inside a panel, it is added to the mask.

The selection of a panel can be undone by clicking the Undo button on the Create Bleed Mask window. The selection of panels is undone in the reverse order in which they were selected. When there are no panels to undo, the Undo button is disabled.

In the following example, the glue flap and the two dust flaps at the bottom of the package have been included in the mask:



To complete a mask, click the close box of the Create Bleed Mask options window or simply click any other tool in the toolbar. You can redo the mask at any time by selecting the Create Bleed Mask tool again. The previous mask will be discarded and a new mask will be created.

- [Drawing a mask](#)
- [Deleting a mask](#)
- [Replacing a mask](#)

Drawing a mask

To mask the graphic:

1. Click the 'Create Bleed Mask' button on the toolbar.
2. Enter a bleed amount.
3. Click in the document window on any panels that you want to exclude from the mask.
4. Click the Close button in the title bar.

You can use the Create Bleed Mask tool at any time, even if you have not imported a graphic into your station. In the example above, there is no graphic in the station. If you import a graphic later, the mask will be applied automatically to the imported graphic.

Deleting a mask

Because the mask is an unfilled and uncolored contour, it is difficult to select and delete it in the color view modes. However, you can easily identify and select the mask when you set the preview mode to '*Cross*'. Masks are always displayed as blue outlines in Cross and Contour modes, making them easy to identify and select.

To delete a mask:

1. Set the display mode to Cross.
2. Locate and select the blue mask outline.
3. Press [Delete].

Replacing a mask

If you have an imported CAD die in your station and then use the Draw Die Shape tool to create a new die, the imported CAD die is replaced with the contours that you create with the tool. This also applies if the die shape is on the trim box of the job. If the die is created in another way (e.g. by extracting contours in a specific ink), this **die is kept and should be deleted manually if it is no longer wanted**.

Zoom Tool

Use the Zoom tool to magnify a portion of your workspace at an increased scale.

To zoom in, click with the left mouse button. To zoom out, click with the right mouse button. Or you can move the mouse cursor to a corner of the area to zoom in upon, hold down the mouse button, and drag to the diagonally opposite corner.

Tip:

Use the Zoom In and Zoom Out commands in the View menu to zoom one zoom level at a time.

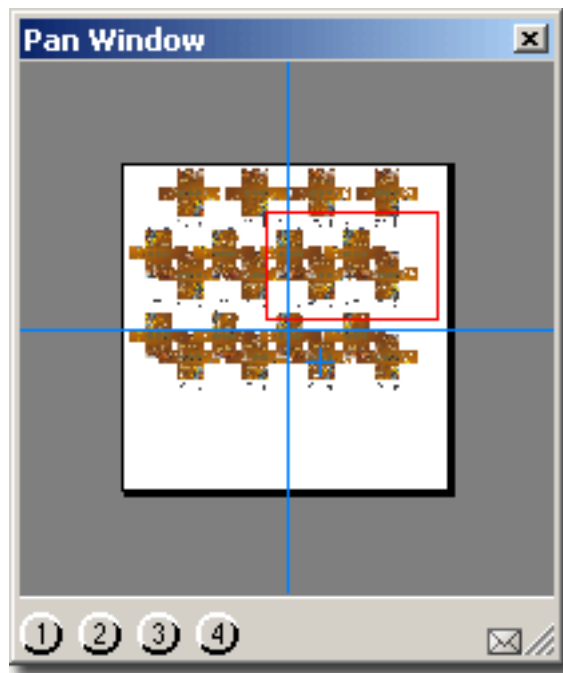
Pan Tool

Use the Pan tool to move around the document. You can change your view of the current document by using the Pan tool or the Pan Window (see [Using the Pan Window](#)). The Pan tool is located on the Tools toolbar; the Pan Window is opened from the Window menu.

To use the Pan tool, select the tool and use the cursor to drag the document around in the document window. You can also pan around the document using the window scroll bars.

Use the Pan Window to see an overview of your document and the current view. The document is displayed as a thumbnail with a rectangle indicating the current view.

1. On the Window menu, click 'Pan Window'. The Pan Window opens:

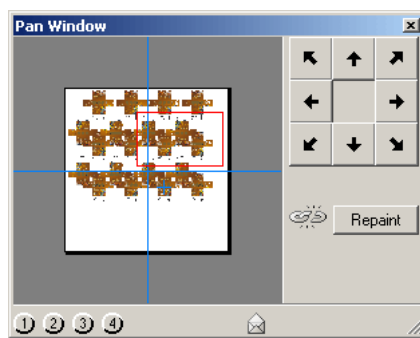


2. You change the view area by dragging the red view rectangle to other locations on the thumbnail.

You can also set the four Memory buttons at the bottom of the dialog box to remember previous views. To store the current view in a Memory button:

1. Click and hold one of the memory buttons. The button will turn pink.
2. An Info dialog box will be displayed to inform you that the current view has been stored.

The envelope in the Pan Window is used to expand the dialog box to show more options. Click the envelope to expand the dialog:



The expanded Pan Window dialog box contains other features. You can use the arrow buttons to move the view rectangle over the document. You can also force the application to repaint the display by clicking the Repaint button.

Measure Tool

Use the Measure tool to measure one or two points in the document window. The measurements for the points are shown relative to the current origin of your document.

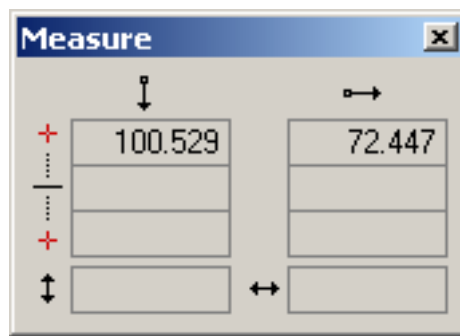
- [Measuring a single point](#)
- [Measuring the distance between two points](#)

Measuring a single point

To measure a single point:

1. Click the Measure icon on the toolbar. The Measure window appears.
2. Click in the document window on the point that you want to measure.

The point you clicked is highlighted in the document window. The location of this point is shown in the Measure window:

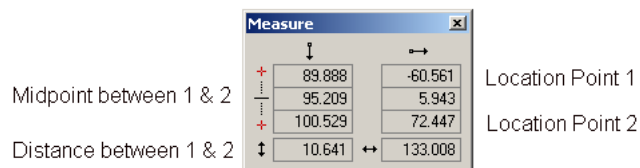


Measuring the distance between two points

The Measure window shows the location of the two points, and the midpoint and the distance between them. To measure two points:

1. Click the Measure icon on the toolbar. The Measure window appears.
2. Click in the document window on the first point that you want to measure.
3. Click in the document window on the second point that you want to measure.

The two points that you clicked are highlighted in the document window. The location of the two points, the midpoint between them and their distance are shown in the Measure window:



When you selected two points, the values for the first point you clicked were entered into the top line of the Measure window. The values for the second point you clicked were entered into the position for the second point (the third line of the window).

Measure tool tips

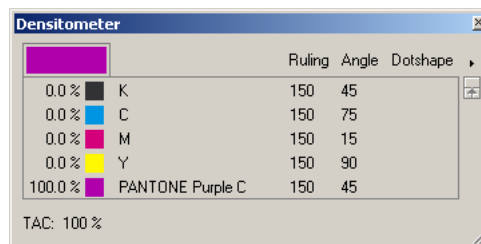
- If you click another point, the values for the new point are entered into the second point position and the previous values are moved up to be the first point.
- The Measure tool will "snap" to contours when in Contour Display mode, allowing you to accurately measure the distance between two objects. When in Cross or Dummy color mode, it will snap to the bounding box of the stations.
- The values in the Measure tool are copied to the Measure Window. You can open the Measure Window to use these values in other dialog boxes or windows.

Densitometer Tool

Use the densitometer tool to view ink percentages for locations in your document. You may want to use the measured values from the densitometer tool to create other objects, such as Geometry Marks. You can also use the densitometer tool to check the ink values for stations and SmartMarks to make sure that the correct values are assigned to them.

To measure ink percentages in your document:

1. Click Densitometer on the toolbar. The densitometer window appears:



2. Click in the document window to display ink percentages for a specific location.

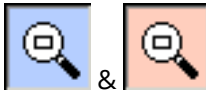
Densitometer tool Tips


- TAC (Total Area Coverage) is the sum of the dot percentages of the different inks that will be found on the paper after printing.
- The display mode must be set to Medium or High for the densitometer to measure correctly.
- If you click and then drag the densitometer, the values in the densitometer dialog box will continuously update.
- The fly-out menu offers the possibility to choose between exact (92.2%) and rounded (92%) readings. you can also set the color of the densitometer's cursor.
- To close the densitometer dialog, click the close box on the dialog box or select another tool.

Working with tools and dialog boxes

Esko Plato contains some unique tool features and icons that you may not have seen before. These features are designed to provide you with just the right degree of productivity and simplicity.

Some tools can be used as **"one-shot" tools**. A one-shot tool is a tool that can be used once during other commands. After this one-time use, the current tool is reset to the tool that you were using originally. One shot tools are highlighted in red instead of blue, as shown below:



 **Envelope:** Some dialog boxes contain an envelope icon. The envelope is used to expand a dialog box to show more options. Clicking on a closed envelope will expand the dialog box to show more options for the particular feature. Clicking an open envelope will collapse a dialog box to show fewer options.



Link: In some dialog boxes you will see a link. You use the link to enable or disable immediate refreshing of the screen. When the link is broken, as shown above, you can make several changes or selections in the dialog box without waiting for the document window to refresh the display after each change. When the link is intact, each change to the dialog box is immediately displayed. Clicking the icon toggles between these two settings.

Tip:

A quick and easy way to hide or show all dialog boxes is to select **Hide / Show All** on the Window menu.


9.2.7. View Selector



The View Selector indicates or sets:

- The View in which you are working: Sheet or Station View.
- The side of the sheet on which you are working: front or back.

Tip:

Click the 'Show Trim and Media box' button  in the *Preview Selector toolbar* to switch the visualization of the trim and media box on or off.

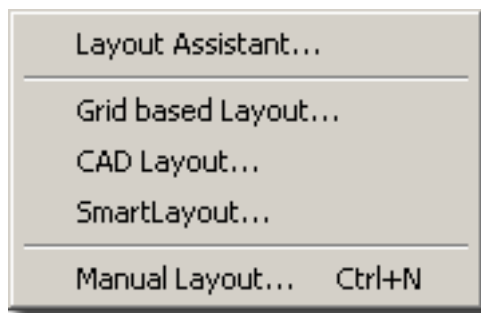
10. Function after function

- *New Layout...*
- *Open Layout...*
- *Import CAD Data...*
- *Export Layout...*
- *Layout Setup...*
- *Document Info...*
- *XMP Properties Info...*
- *Print...*
- *Preferences...*
- *Preview*
- *View Modes*
- *ViewX...*
- *Select Die Shape...*
- *Place Station...*
- *Replace by...*
- *Assign Station to Design...*
- *Import CAD Design*
- *Replace by Graphics File...*
- *Save Station*
- *Export Station*
- *Station XMP Info...*
- *Inks...*
- *Ink Mapping...*
- *External File Ink Mapping*
- *Ink Coverage...*
- *Create Ink Histogram...*
- *Create Ink Eater Areas...*
- *Fill Ink Eaters...*
- *Open Ink Book*
- *Manage Ink Books...*
- *Number Stations...*
- *Adjust Masks...*
- *Create Varnish*
- *Manage Plates*
- *SmartMarks...*
- *Staggered Cutting...*
- *Check Job Parameters*
- *Workspace*

- [Toolbars](#)
- [Align](#)
- [CAD](#)
- [Calculator](#)
- [Colors](#)
- [Color Factory](#)
- [Effects](#)
- [Grids](#)
- [Info Window](#)
- [Layer Browser](#)
- [Pan Window](#)
- [SmartLayout](#)
- [Search Options](#)
- [Stations](#)
- [Style](#)
- [Transform](#)
- [Color Picker](#)
- [Grid Settings](#)
- [Station Properties](#)

10.1. New Layout...

New Layout creates a new document. There are 3 subitems under the 'New Layout' command. These correspond to the different workflows that Esko Plato offers. The only difference between these subitems is that they will open the appropriate workflow dialog automatically. In other words: there are no different flavors of (PDF)PLA documents. For example, even after choosing 'New SmartLayout', one can still add a step and repeat grids.



- [Layout Assistant...](#)
- [Grid based Layout...](#)
- [CAD Layout...](#)

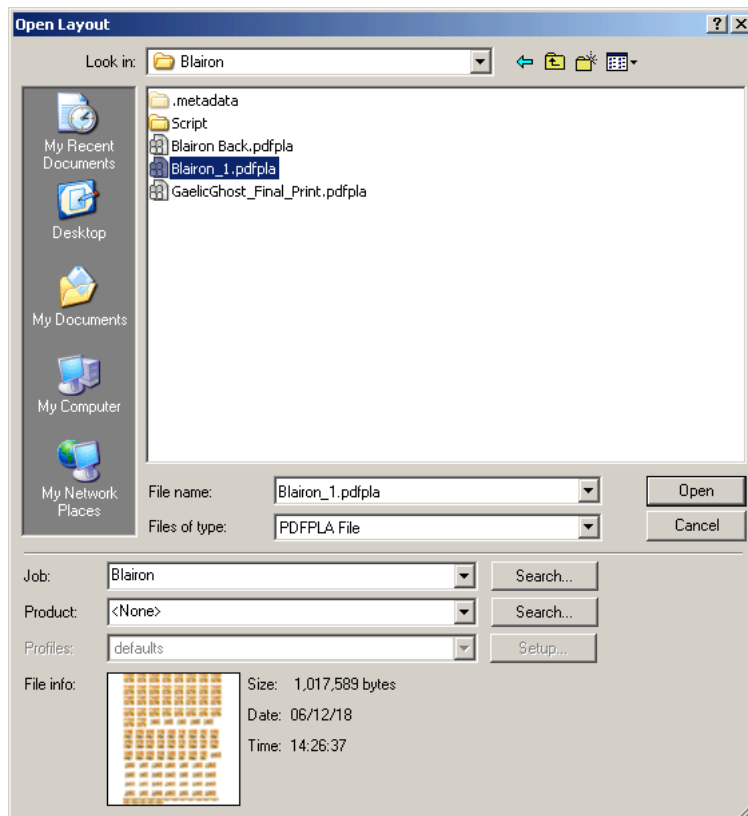
- [SmartLayout...](#)
- [Manual Layout...](#)

Tip:

If you always use the same workflow, you can configure the Ctrl-N shortcut to start your favorite new operation immediately.

10.2. Open Layout...

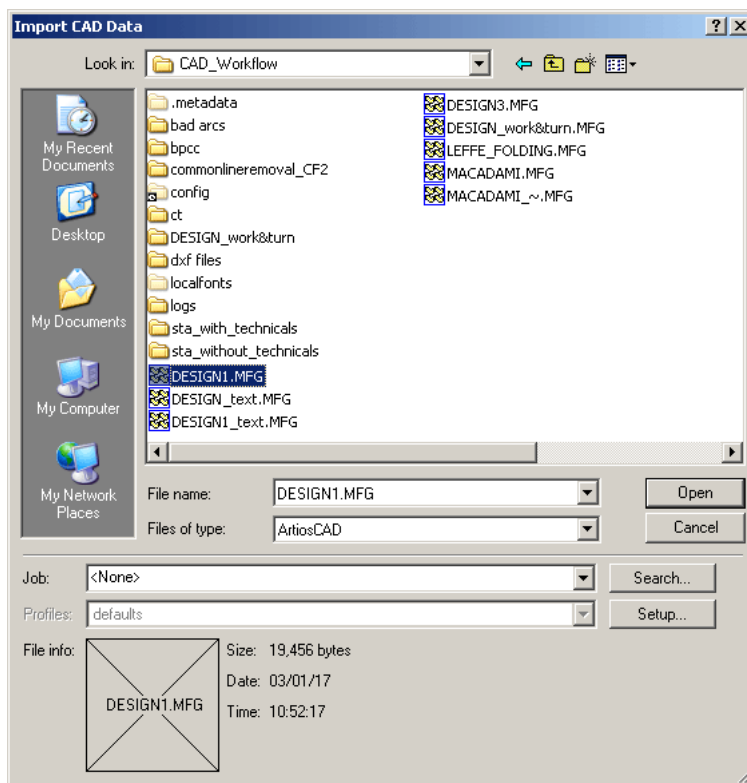
Open Layout opens an existing Esko Plato document. If the document is created by Plato 5.0 or lower, it will automatically be converted into an Esko Plato 7.0 structure. Opening of GRQ files and CAD files is also supported.



10.3. Import CAD Data...

Import CAD Data opens a CAD layout file, extracts the sheet size and the position and orientation of the stations.

The CAD information can be retrieved from a CAD file such as ArtiosCAD, or from a PDF file.



There are different ways to import CAD designs (1-Up). When importing CAD data in the Sheet View, empty stations are being created which will have the 1-up CAD design as die shape. For more information, please refer to the chapter ['Using a CAD file to create a station'](#).

Esko Plato can open a CAD file to define the plate layout. When you open a CAD file, you are using the CAD Layout Workflow. This workflow defines the number and locations of the stations that are on the plate. It may also contain a sheet size and masks with bleeds for the stations. If these features are in the CAD file, Esko Plato will use them in your layout. For more information, please refer to the chapter ['Using a CAD file for plate layout \(optional\)'](#).

Setup opens the CAD Import dialog box that allows you to do mappings, select or create a certain [CAD style](#) or search for print items.

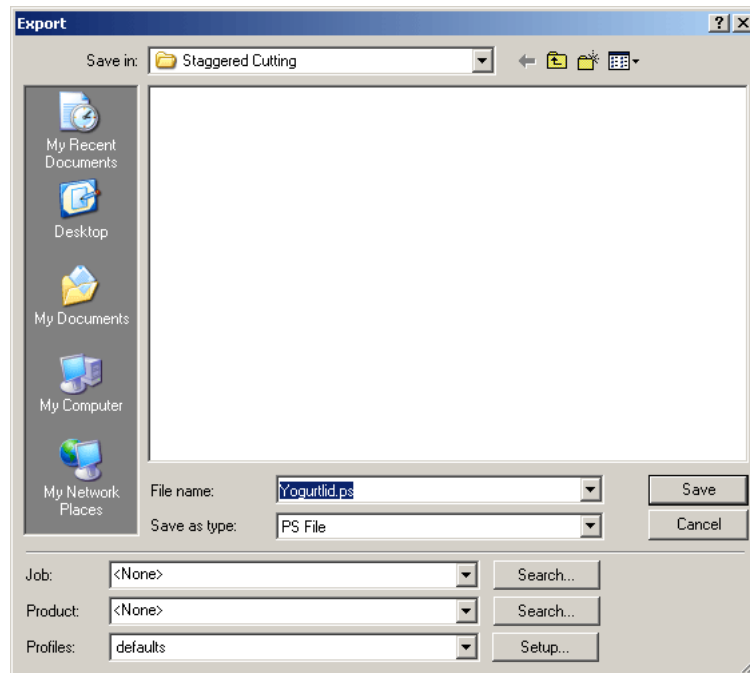
Note: When importing an MFG file, you can choose to import it as "Reference Lines" in the CAD Import setup window. By doing so, the MFG file will not be imported as CAD but only as reference lines, in a separate "CAD Reference" layer. Importing the MFG file will have no other influence on your open document.

10.4. Export Layout...

Export Layout allows to export to various formats: PS, EPS, PDF files. All conversions take the complete layout (graphics + die shapes) and convert the information to the selected format.

Tip:

Note that even for PDFPLA files, exporting to PDF makes sense. Esko PDFPLA files contain references to PDF files. When exporting a PDFPLA file to PDF, an all-in-one PDF file will be created (embedding the referenced PDF files).



Click 'Setup' to open the Setup dialog box. For more on these options, please refer to the chapter ['Exporting to a PostScript File'](#).

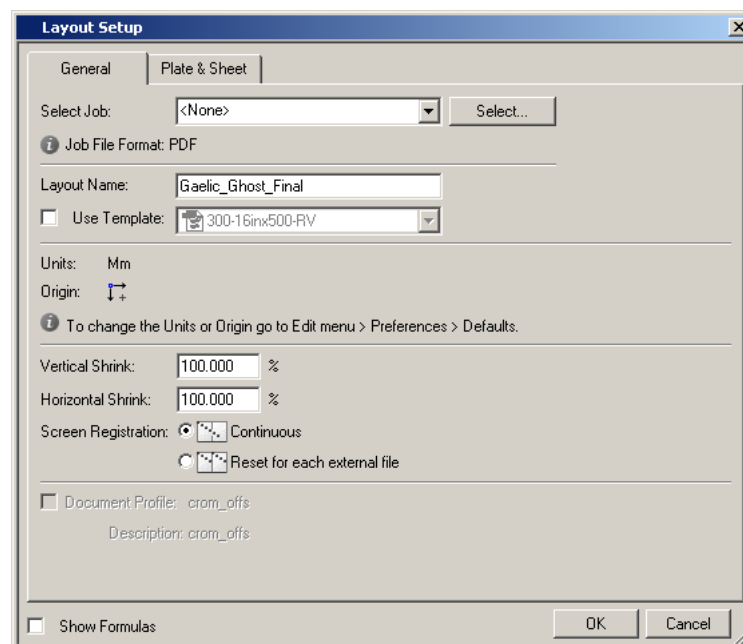
- **Normalized PDF:** Only available if the current format is a PDFPLA file. It allows you to export an Esko Software Suite 7 PDFPLA file into a Scope 3 compatible Normalized PDF file.
- **PLA or PDFPLA:** Only available if the current format is PDFPLA or PLA. It allows you to change the format from Esko native PLA to Normalized PDFPLA and vice versa.
- **Polar** (requires a dedicated license) allows you to output to Polar Compucut cutting stations.
- **FFF2** allows you to export to FFF2 format, based on separations. You can either export the contour lines or the outline of a region. This option is mainly intended to export data to cut varnish blankets, embossing or foil stamping areas.
- **JDF Layout** exports the graphics into PDF files and links these PDF files together through a JDF layout ticket.
- **MFG, Layout FFF2, Layout DDES3** converts die shapes into a layout CAD file of the requested format (ArtiosCAD MFG, FFF2 or DDES3).
- **CIP3** exports the file to a CIP3 compatible .PPF file.
- **JDF Cutting File:** see [Exporting to a JDF Cutting file](#) on page 277

10.5. Layout Setup...

The Layout Setup dialog box allows you to set layout settings. It is the dialog box that appears when you intend to start creating a new layout. For more information on creating a new layout, please refer to the chapters on [grid-based](#), [CAD](#), or [SmartLayout](#) workflows.

The Layout Setup dialog box consists of two tabs:

- [General](#)
- [Plate and Sheet](#)



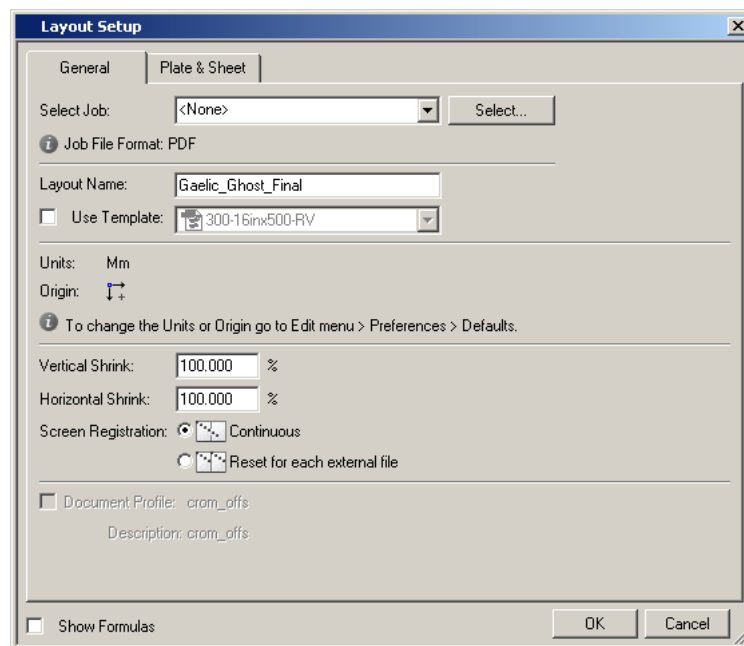
10.5.1. General

The General tab of the Layout Setup dialog box offers you the possibility to determine some general settings.

- If Esko Plato is connected to an Esko Automation Engine server, you can specify in which Automation Engine **Job** the new layout will be created. Specifying the Job is important to:
 - set up SmartNames
 - determine the default file format (important when normalizing foreign file formats)
- It indicates the **Job File Format** in which your job will be created: PDF or GRS.
- You can enter a **name** for your new plate in the Layout Name field. If you don't enter a name, Esko Plato will ask for a name when you save the file. If you want to use a **template** file that you created previously, you can click the check box next to "Use Template" and select one from the drop-down list. If you haven't created any plate templates, the check

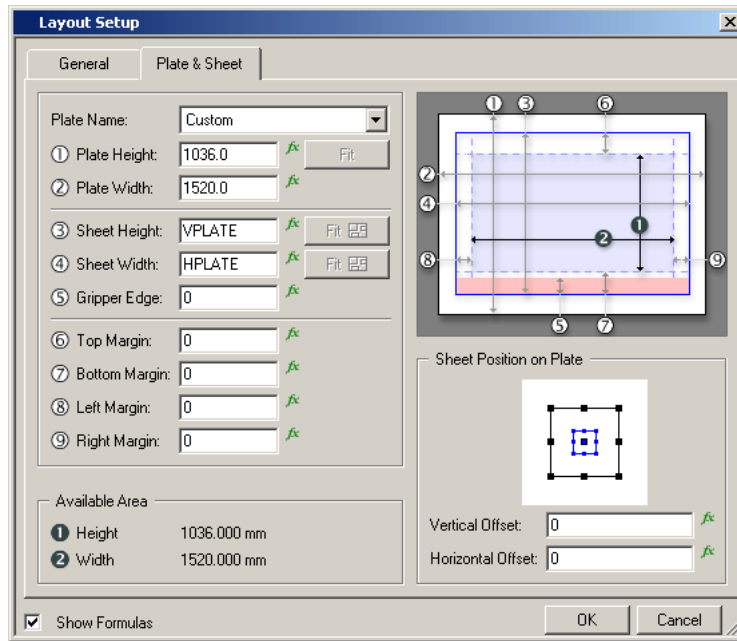
box and field will be disabled, as they are in the dialog box below. See [Using Templates](#) for more information about using templates. If you have QuickStep templates on your system, these will also show up in the list. When selecting an Esko Plato template (not for QuickStep templates), the template will be searched for information like plate and sheet size and these values will be shown in the corresponding fields in the Plate & Sheet tab.

- **Units** and **Origin** can be changed in the [Defaults tab](#) of the Preferences dialog box.
- For some printing processes, like flexo printing, the plate must be distorted to compensate for the expansion of the plate when it is wrapped around the cylinder of the press. The **Shrink** fields allow you to set a distortion factor that is used when printing or exporting. You can enter Shrink values between 90 and 110. The default Shrink value is 100; the plate will be printed at 100%. If you enter a value other than 100, the Distortion option in the Print dialog box is disabled.
- **Screen registration** is relevant only to Esko FlexRip users. When activated, the origin of the screening will be reset for each station. In doing so, it is guaranteed that exactly the same screening will be applied for each station. When deactivated, screening will be done continuously throughout the complete job.



10.5.2. Plate and Sheet

The Plate and Sheet tab allows you to specify the **dimensions** of your plate and sheet.



To facilitate this procedure, you have a number of formulas at your disposal, which you can activate by clicking the 'fx' symbol to the right of each entry field. When you click 'fx', the [Calculator](#) pops up. Right-clicking in the entry field will make the calculator pop up as well.

Note:

You can toggle 'Show Formulas' to either work with formulas, or with numeric values.

Plate Size - Enter the height and width of the plate in the Size fields. You can create a plate with any dimensions; the plate can be wider than it is tall, taller than it is wide, or square.

Sheet Size - Enter the height and width of the sheet in the Size fields. You can create a sheet with any dimensions; the sheet can be wider than it is tall, taller than it is wide, or square. You cannot set the sheet to be larger than the plate but the sheet and plate sizes can be equal.

Fit - Use the fit buttons to fit the sheet size (vertically and horizontally) to the current red selection. The 'Plate Fit' button will fit the plate size around all current objects.

Note:

There is a big difference between using the fit buttons and using the plate and sheet fit parameters (V/HPlateFit and V/HSheetFit). The fit buttons will do a one time fit. They measure the current objects and store the corresponding values as sheet or plate settings. The fit parameters on the other hand will make sure that the plate or sheet settings are automatically updated whenever the station grids are updated (V/HSheetFit parameter) or whenever the bounding box of all objects changes (V/HPlatefit parameter).

Margins allows you to indicate which zones on the 4 sides of the sheet should not be used to place stations. The sheet minus these margins equals the available area (available to place stations).

Gripper Edge allows you to specify the width of the gripper zone (in sheetfed offset). The value specified here will be used to define the parameter called 'Gripper'.

Sheet Position on Plate allows you to indicate how the sheet should be positioned on the plate. You can use the 9-point selector and/or specify an offset. The small, blue square represents the outline of the sheet. The large, black square represents the plate. The parameters you enter in the vertical and horizontal offset input box are added to the position of the sheet on the plate. Drag or click to position your sheet on the plate. The offset value is based on the current origin. See the [Defaults tab](#) in the Preferences dialog box for information about origin.

10.6. Document Info...

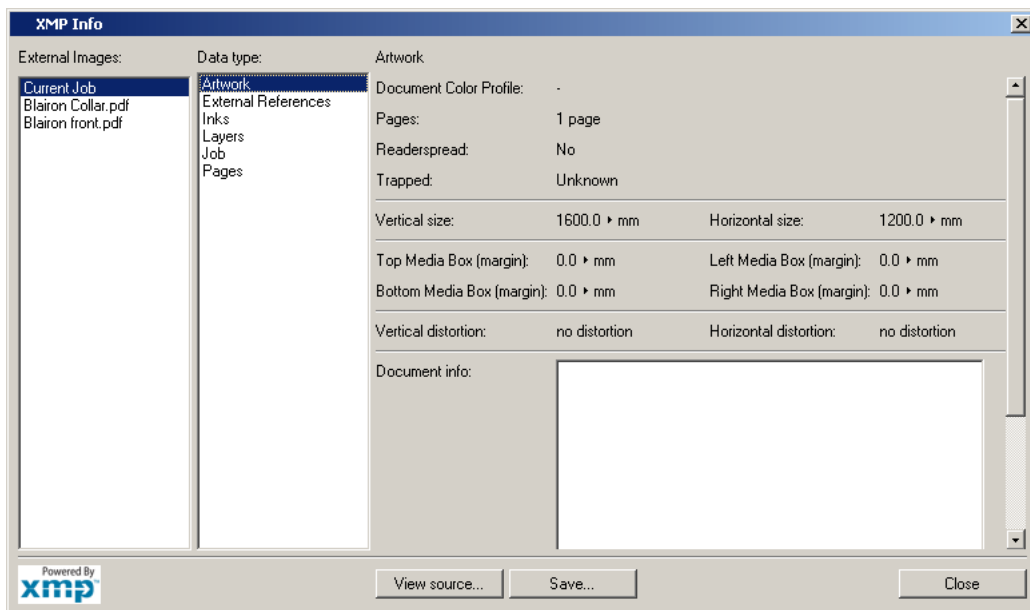
The Document Info dialog box can be used to write job-dependent production information. The info can also be used as smart text in a text mark. The 'Document Info' dialog opens after clicking 'Document Info...' on the File menu.

10.7. XMP Property Info ...

XMP stands for eXtensible Metadata Platform, it is a standard for metadata. The editor understands and works with XMP.

When you create a file, all general job information will be stored the moment you save the file. Additionally ink information (name, lpi, dotshape, inkcoverage), bar code information, fonts, CADfiles, etc... used in the file is listed in the XMP.

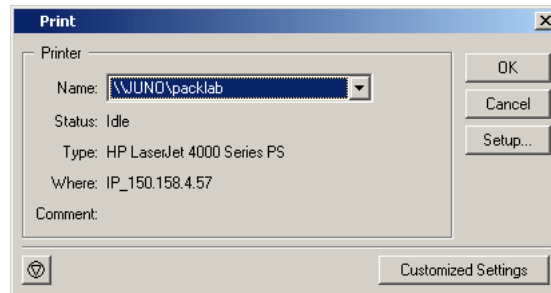
Choosing 'XMP Property Info' from the File menu opens the XMP Metadata Properties dialog box.



10.8. Print...

Print allows you to proof or expose the design opened in Esko Plato.

Clicking 'Print' on the File menu opens the Print dialog box.



Click 'Setup' to open the Setup dialog box. For more on these options, please refer to the chapter ['Exporting to a PostScript File'](#).

10.9. Preferences

All Esko Plato modules have various preferences that you can set. You set or change your preferences by clicking 'Preferences' on the Edit menu.

1. On the Edit menu, click 'Preferences'.
2. Select the tab that contains the settings that you would like to change.
3. Click 'OK' to save your changes.

Selecting 'Cancel' will discard any changes you have made since the last 'OK' and will close the Preferences dialog box. Selecting 'Reset' will reset all settings in all tabs to the application defaults.

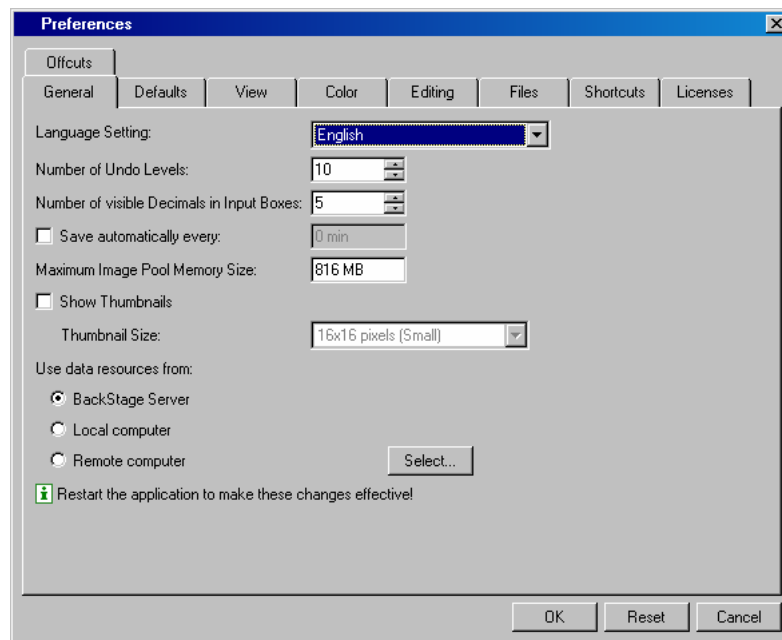
The Preferences dialog box has seven tabs:

1. [General](#)
2. [Defaults](#)
3. [View](#)
4. [Color](#)
5. [Editing](#)
6. [Files](#)
7. [Shortcuts](#)
8. [Licenses](#)

9. [Server and Resources](#) on page 336

10.9.1. General

To define general standard settings to be used by the application as you work on a job.



Language Setting: You can change the language of the user interface by selecting one of the following languages from the list box: English, French, German, Spanish, Japanese, Chinese Simplified or Chinese Traditional. You no longer need to install the language packs from the DVD to have a non-English user interface. In case you prefer to have the localized online help documentation, however, you still need to install the language packs.

Note:

The editor needs to be restarted before the changes will be effective.

Number of Undo Levels: Defines the minimum number of steps the system remembers for Undo and Redo. Enter a value between 0 and 1000.

Tip:

For complex jobs, it is best to keep the number low to save memory.

Number of visible Decimals in Input Boxes: Defines the number of decimals displayed in the input boxes.

Enter a value between 0 and 8.

Values are rounded when you enter more digits (into an input box) than the number defined in this Decimals input box.

Save automatically every : Defines how often you want the application to save your work automatically. The interval between 2 automatic saves is expressed in minutes. Your work is saved in a file called 'autosave.pla' or 'autosave.pdfpla' in the Temporary directory you have selected in the Files tab sheet.

A value of 0 disables automatic save.

Maximum Image Pool Memory Size: Defines the amount of RAM allocated for storing Image display images.

The higher the value, the more RAM the system has at its disposal for display image(s), reducing the need to reload display images from the hard disk.

The default value is 25% of the available RAM, the minimum value is 16 Mb, the maximum value is 80% of available RAM.

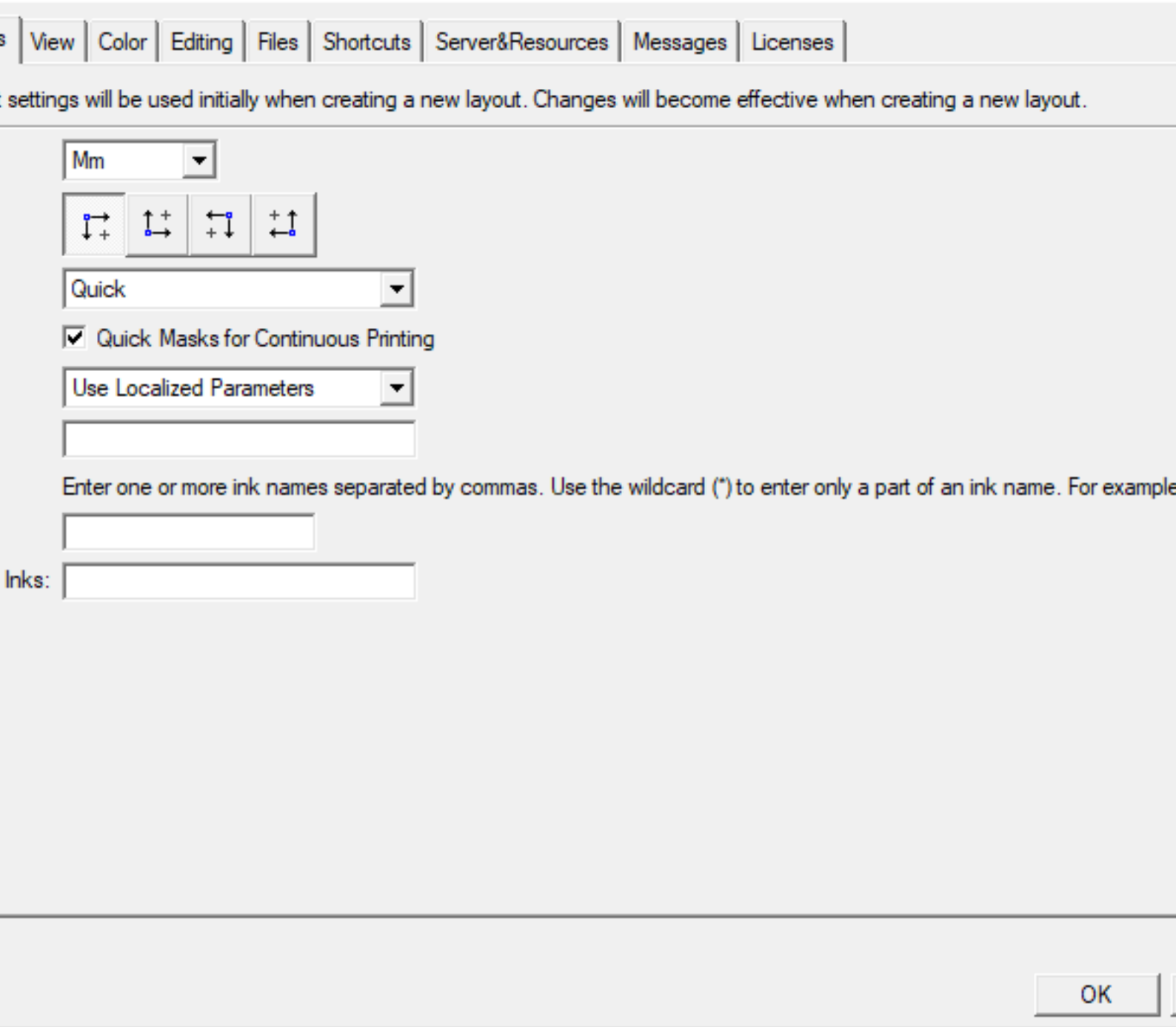
Note:

In general, the system performs quicker when the application can hold Image display images in memory; note, however, that the memory allocated for Image caching, reduces the amount of memory available for other purposes.

Use data resources from : If Plato is connected to a Automation Engine server, the resources will be taken by default from the Automation Engine server. Other options are Local computer or Remote Computer. This can allow multiple Plato seats to share resources even when no Automation Engine server is available, for example to exchange SmartMark sets. Use the Select button to define the location of the remote computer.

10.9.2. Defaults

Use the Defaults tab to set the units and the default origin for new documents.



Units Specify whether you want your measurements and calculations displayed in millimeters or inches.

Origin Choose the orientation of the axes in your document. Your preference will be stored inside the file. Opening the file later will set the orientation in the same situation as in which the file was created.



Caution:

It is not possible to change the units or orientation while working at the job. New preference settings will only be applied when creating a new job.

Adjust Masks is used to adjust overlapping stations. The command searches the plate for overlapping stations and presents each overlap to you as an overlap record. You choose how these overlap areas should be adjusted: manually or quick. 'Manual masks' allow you to make a decision on the priority of overlapping flaps and panels per case. 'Quick masks' always split overlapping masks halfway between the two selected die shapes.

If the layout is by default meant for web printing, you can enable the **Quick Masks for Continuous Printing** option. The Quick Adjust of the stations at the top and bottom of the layout will take into account that the layout is repeated, using the vertical sheet size as Repeat Length.

Formulas Select the format in which the formulas are visualized: technical formulas or localized formulas. Technical formulas correspond with the way in which formulas were visualized in QuickStep. Localized formulas provide a more readable format.

Default Die Ink In case you always use the same die ink, you can specify it here and it will automatically be detected as the default die ink in the file.

10.9.3. View

The View tab changes the way that your document is displayed. Changes made here do not affect the structure of your document.

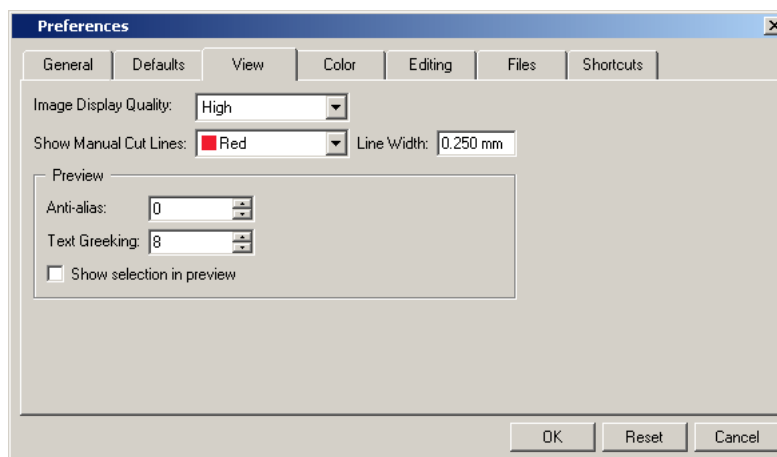


Image Display Quality defines the resolution used by the application to display images.

- Very High: the application creates a display image at the highest possible resolution, regardless of the display information in the image file itself.
- High: the application creates a display image at a high resolution (72 dpi), regardless of the display information in the image itself. This can take very long since all data of the images

are read. If you have, however, a large job that takes more than half of the CT Cache Size, the display image will be displayed at a resolution lower than 72 dpi.

- Default: the application uses the display image of the CT file itself or, if none is available, creates a display image;
- Low: the application uses a display image only if it is available in the CT image itself. If you choose this possibility, the CT will not be visible in the *Extended Preview mode*.



Caution:

The option Very High is introduced to try to enable real resolution views of CTs. Due to the large amounts of data this involves, setting the Image Display Quality to Very High will considerably slow down the visualization. Consequently, we would like to recommend you not to play with it and only use these possibilities when you really need them.

Show Manual Cut Lines defines color and width of the manual cut lines. Manual Cut Lines are cut lines drawn with the *'Draw Die Shape'* tool in the Station View, or created by applying 'Outline of all objects' in the *Select Die Shape dialog box*. All other lines follow the CAD styles.

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.

Text Greeking defines the minimum size of the text before it is displayed as a filled box rather than readable text. The boxes will turn into text again when zooming in.

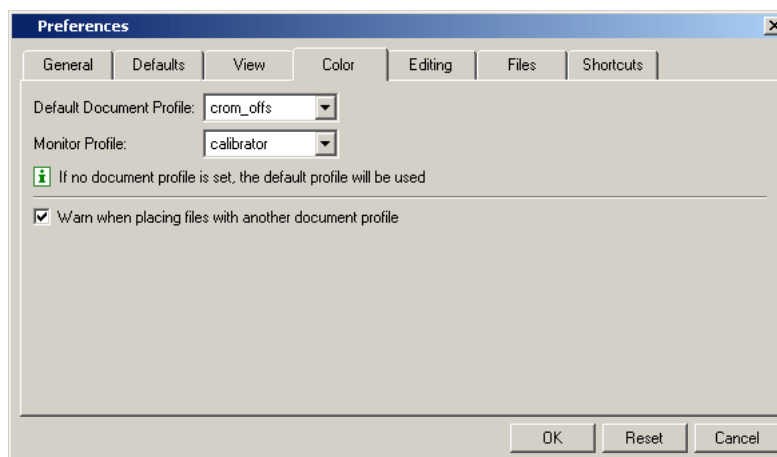
Show Selection in Preview: When objects are selected, they are displayed with a red outline. You can choose not to show the selection in the three "color" display modes by clearing this box. (Selections are always shown in Cross and Contour display modes, regardless of this setting.)

10.9.4. Color

Default Document Profile: choose the default profile from the list.

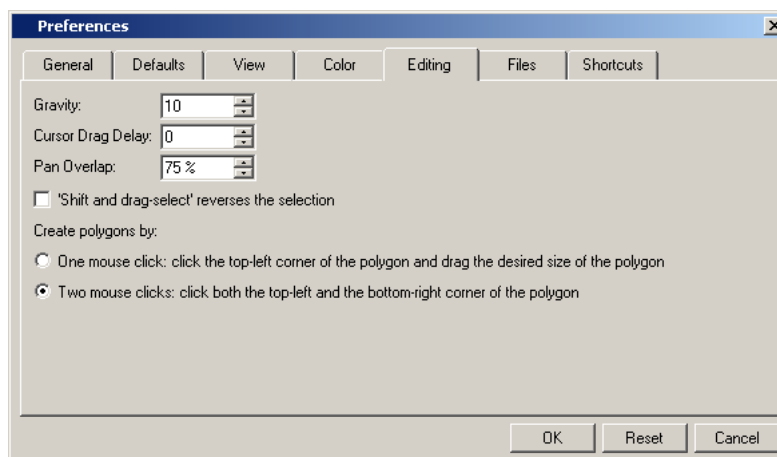
Monitor Profile: choose a monitor profile from the list.

Check **Warn when placing files with another document profile** to have the system display a warning when placing PDF files that have other document profiles embedded.



10.9.5. Editing tab

The Editing tab sets Esko Plato's defaults for editing. These defaults do not change the structure of your document; they only change your interaction with the application when editing files.



Gravity defines the distance in screen pixels where gravity (or snap) is in effect. The higher the value, the easier an object will snap to guides or other objects. Enter a value between 0 and 16.

Cursor Drag Delay defines the minimum distance in screen pixels the cursor must be moved before any transformation of an object takes place. Enter a value between 0 and 16.

Pan Overlap defines the percentage of overlap between the previous and new view when panning. For example, with the Pan Overlap set to "75", the new view after panning contains 75% of the previous view and 25% of a new view.

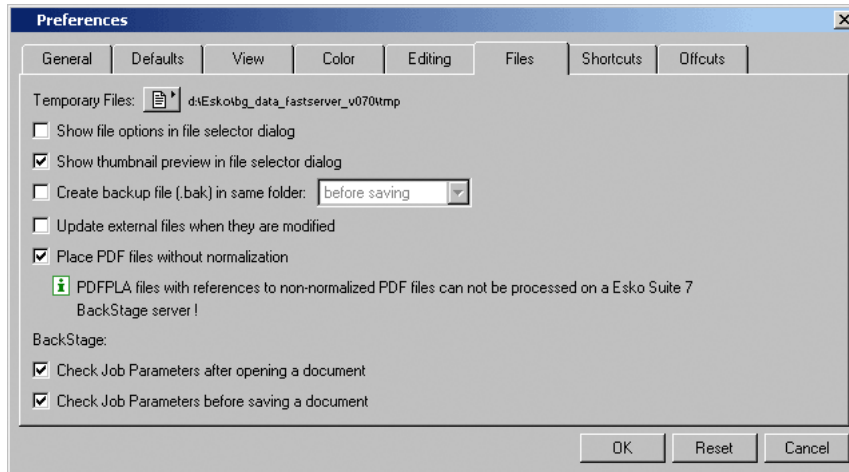
'Shift and drag-select' reverses the selection: Dragging a selection over an object while holding SHIFT actually deselects selected objects and selects unselected ones.

Create polygons by:

- One mouse click: click the top-left corner of the polygon (depending on the chosen orientation) and draw a polygon of the desired size.
- Two mouse clicks: click both the top-left and the bottom-right corner of the polygon.

10.9.6. Files tab

Use the Files tab to define file settings to be used by Esko Plato as you work on a document.



Temporary Files: The default temporary files directory is the \temp folder on your home folder. Click the Browse icon to choose another directory. Select a directory on the fastest hard disk of the workstation.

Show file options in file selector dialog: Checking this checkbox will open the File Options dialog box every time you open or import a file.

Show thumbnail preview in file selector dialog: The file selector shows File Info. Check the checkbox to turn the preview on or off.

Create backup file (.bak) in same folder: Creates a backup file in the same folder. Indicate whether you want to create a backup file before saving or before loading. You need to change the .bak into .pla in order to be able to read the file into the editor.

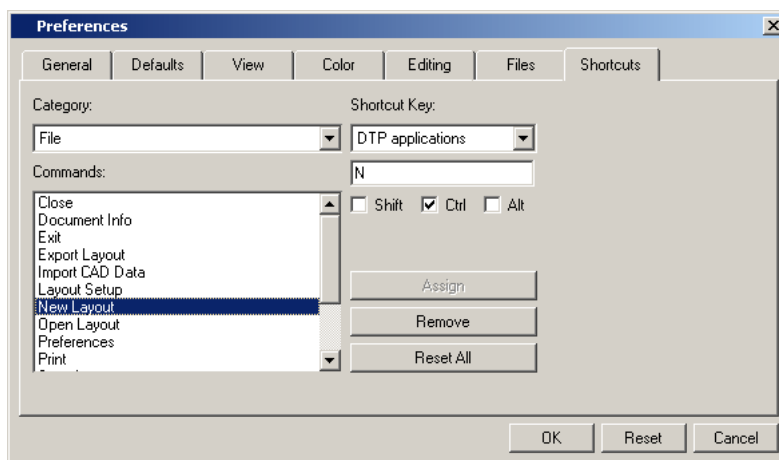
Update external files when they are modified checks whether the referenced files were changed since the file has been loaded. Only happens when the workstation is idle (e.g. when a GRS is placed in Esko Plato, but is edited in Esko PackEdge at the same time).

Place PDF files without normalization It is possible to place non-normalized PDF files, using this preference option. When placing non-normalized PDF files, this will result in a non-rippable PDFPLA file. This means that such a file will need pre-processing when further operations are applied (expose to FlexRIP / Automation Engine operations) – during pre-processing a normalization ‘on-the-fly’ will happen. As such, this workflow is only advisable when working in a standalone PDF environment (placing PDF files, exporting to PDF). Furthermore, this option is only allowed, if Plato is not connected to a Automation Engine server (running in standalone mode). When switching to a mode where the real data is needed (for example switching to extended preview, applying trapping,...), a normalisation ‘on-the-fly’ is done..

Check Job Parameters It is possible to define Job Parameters on the Automation Engine server. Job Parameters are settings (type of bar code, inks, certain RIP parameters,...) that are predefined for a Automation Engine Job.The Check Job Parameters options allows you to check your files for inconsistencies and to correct them (if necessary) when opening or saving a document.

10.9.7. Shortcuts tab

There are 2 sets of shortcuts: Esko shortcuts and DTP applications shortcuts. You can switch between both sets by selecting the appropriate set from the Shortcut Key drop-down list.



You can, however, modify whichever set of shortcuts to fit your personal needs.

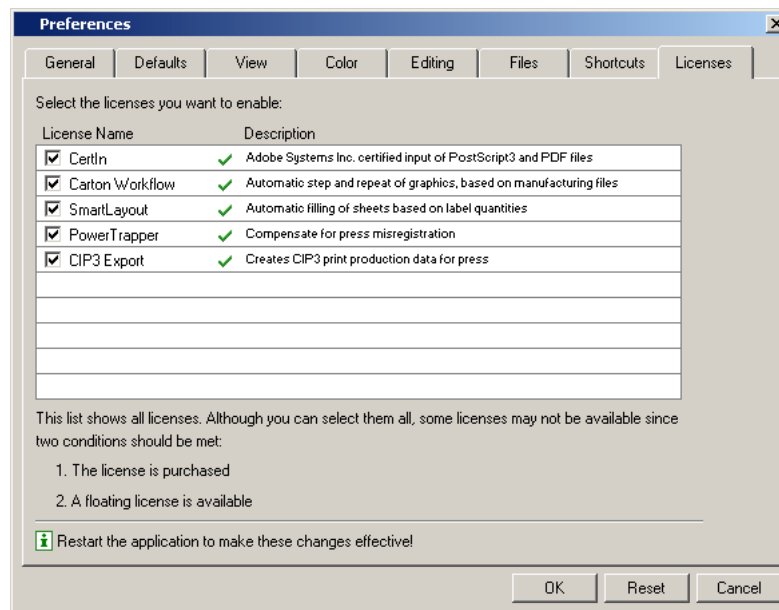
1. Select the right set of shortcuts, if it is not yet selected.
2. Select the 'Category' (e.g., 'File') of which you want to personalize a shortcut.
3. Select the command you wish to personalize (e.g., 'New')
4. Now personalize your shortcut. Enter a key and click 'Assign'.

Note:

If you want to remove a shortcut , click 'Remove'.If you want to return to the original shortcuts, click 'Reset All'.

10.9.8. Licenses tab

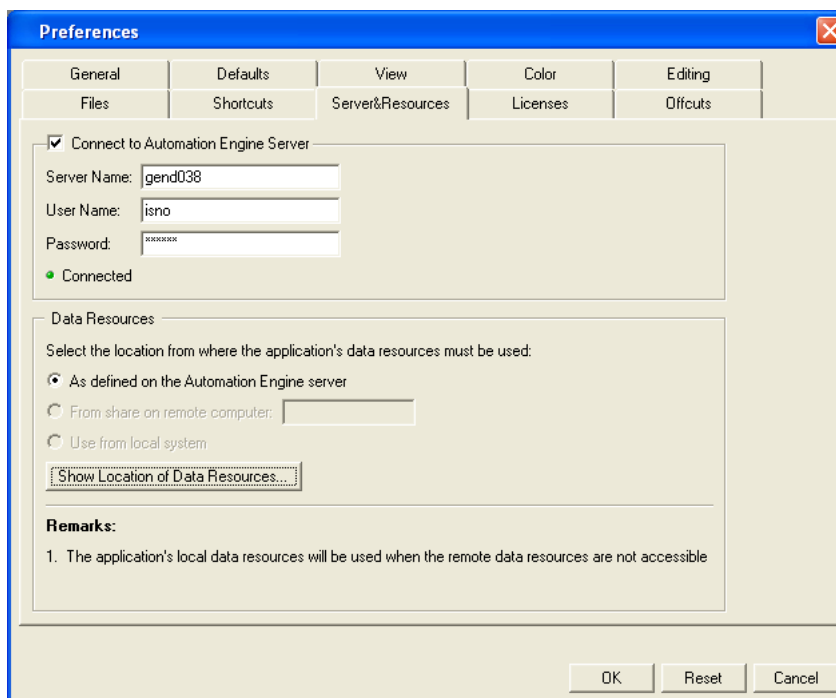
The Licenses tab allows you to activate the licenses that you want to enable.For more information, please refer to the appendix: [License Management](#).



10.9.9. Server and Resources

To define the Automation Engine server to connect to, and the Data Resources to be used by the application as you work on a job.

Note:
 In case the application is connected to an Automation Engine or BackStage server, the Data Resources are always used from this server. You can use the emote or local Data resources if the application is not connected to an Automation Engine or BackStage server.



Automation Engine Server

You can set the Server Name, User Name and Password to connect to Automation Engine.

The connection to an Automation Engine Server has an influence on the following features in the application:

- Submitting to Shuttle: you can only submit to the Automation Engine server the application is connected to
- The user name for the Track Edit Session task on the Automation Engine server
- The list of Jobs and Products in the File Selector dialogs
- The parameters for the 'Check Job Parameters' as available on the Automation Engine server

Note:

If you are connected to an Automation Engine, a green notification will be shown.

Data Resources

The common Data Resources used by the application are Color Engine Database, Marks, Fonts, Custom and DGC.

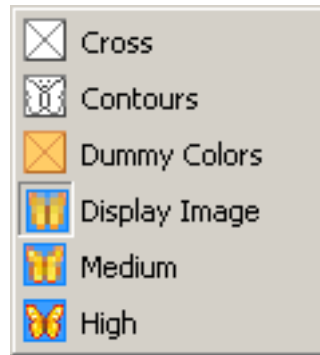
By default, these Data Resources are taken from the Automation Engine server. However, it is possible for 2 or more stand-alone applications (without an Automation Engine server) to share the same data resources. This requires that the Data Resources are shared on a remote computer.

Note:

Changing the Data Resource settings requires a restart of the application.

10.10. Preview

The Preview menu makes it possible to easily switch between the different preview modes.



Cross: Cross mode shows all external files as a box displaying the file name. This mode displays your sheet very quickly but shows no detail.

Contours: Contours mode shows all objects as outlines. This mode is useful for aligning two contour edges precisely.

Dummy Colors: This mode is particularly useful if complex layouts needs inspection (e.g. the ones generated by SmartLayout). When placing a station, a dummy color is assigned automatically. This color can be modified from within the station properties. In case of 'real' stations, this representation color is used only for this visualization mode! The [color picker](#) in the station properties dialog box also allows to assign a color to empty stations. When doing so, a designer ink is added to the ink list representing the chosen color. When replacing a station, the inks (from the dummy station) are cleaned up automatically (if no longer used). As such, the dummy inks created when assigning colors to empty stations will be cleaned up automatically when the empty stations get replaced by the actual stations.

Display Image mode displays all objects in color. External files, like stations and image marks, will be shown using their Display Image. If an external file does not have a display image, it will be shown as a color cross.

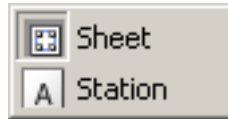
Medium mode shows the true objects in color. It does not use the display image so the detail is very good.

High mode shows all objects in color and with ink-based color management applied. This mode is extremely accurate. However, the display speed may be slow due to the high level of detail being calculated and displayed.

Please refer to the chapter '[Preview Selector](#)' for additional information.

10.11. View Modes


The View mode menu makes it possible to easily switch between the two different view modes



The **Sheet View** shows the plate, the sheet, and all stations that have been placed. In this view, you can switch between the front and the back of a plate.

The **Station View** shows individual stations.

Tip:

Click the 'Show Trim and Media box' button  in the [Preview Selector toolbar](#) to switch the visualization of the trim and media box on or off.

Please refer to the chapter '[View Selector](#)' for more information.

10.12. Viewer

10.12.1. Welcome to Viewer

Benefits of using Viewer

Viewer is a very accurate separation viewer with a lot of extra production tools. It eliminates the need for separated proofs and will help you to avoid bad plate-making.

Fly-out menu options

A number of settings can be found in the Fly-out menu, by clicking the arrow at the top right of the dialog.

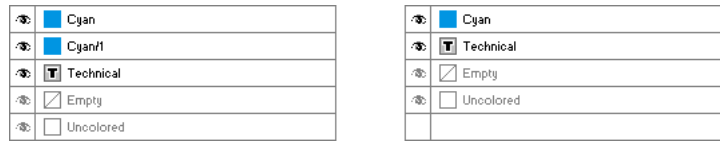
Show in All Windows

Use this to apply the **Viewer** preview settings to all the open windows (when you used several windows).

Merge Similar Inks

This option affects how a separation containing objects with different halftone information is displayed in the separation list.

Leave it off to display a separate entry for each different halftone setting (as shown below at left), or select it to display different halftone settings of the same separation as one entry (as shown below at right).



Default

Click **Default** to use the Default Press Setting. For more information, see [Define Press Settings](#) on page 340

Manage Press Settings...

Click this option to create, duplicate, edit or delete Press Settings. See [Define Press Settings](#) on page 340.

10.12.2. Define Press Settings

Before you can start working with Viewer in a meaningful way, you should define your **Press Settings**. Without basic information on the properties of the press you are using, Viewer cannot simulate accurately.

You can open the Press Settings window by selecting **Press Settings...** in the fly-out menu

The Default Press Settings

In the list of Press Settings, there is always one Press Setting called **Default**, which you can edit but not delete.

You can select it by clicking on **Default** in the **Viewer** dialog’s fly-out menu, and edit it using **Press Settings...**

Creating a Press Setting

To create a Press Setting in **Viewer**:

1. Select **Press Settings...** in the fly-out menu .
2. Click **New...** in the Press Settings dialog.
3. Enter the name of the new press setting in the **New** dialog.
4. Click **OK**

Copying a Press Setting

To copy (duplicate) a Press Setting:

1. Select it in the **Press Settings** dialog.
2. Click the **Copy...** button.
3. Give it a new **Name**.
4. Change its parameters as you wish (see [The Press Settings in Detail](#) on page 341).

5. Click **OK**.

Editing a Press Setting

To edit a Press Setting:

1. Select it in the **Press Settings** dialog.
2. Click the **Modify...** button.
3. Change its parameters as you wish (see [The Press Settings in Detail](#) on page 341).
4. Click **OK**.

Deleting a Press Setting

To delete a Press Setting:

1. Select it in the **Press Settings** dialog.
2. Click the **Delete** button.
3. Confirm that you want to delete it.



Attention:

Deleting Press Settings might also affect other workstations.

Choosing a Press Setting

- To select a Press Setting, click on it in the **Viewer** dialog's fly-out menu.

The Press Settings in Detail

First Dot (%)

Specifies which minimal gray value (expressed in %) in the file still will yield a dot on the plate, provided the RIP and platemaking occur under normal conditions.

Note:

Normal conditions indicate that the DGC must have been set up completely. The value does not correspond with any measurement that has been made prior or during DGC set up, nor does it correspond with any densitometer measurement on plate or on print. In fact, this value is (implicitly) chosen by the person who set up the DGC and plate-making. Typical values range from 0.4% to 1.6%.

Plato uses 8-bit precision, yielding 256 different possible gray values. A single gray value in an image may make the difference between a dot or no dot on a flexo plate. To allow for this precision, Viewer uses percentages with one decimal digit.

8-bit gray value in Plato	Percentage
0	0.0%
1	0.4%
2	0.8%

8-bit gray value in Plato	Percentage
3	1.2%
...	...
254	99.6%
255	100%

Note:

If you do not specify a first dot, you will not be able to use the following Viewer view modes for this press setting:

- The **Flexo Plate Preview**
- The **Flexo Print Preview**

Dot Gain in Highlights

When you have specified a First Dot job percentage, you can also specify how dark it should actually print. Enable Dot Gain In Highlights and specify the dot gain parameters:

- **First Dot Prints As:** indicates the dot gain for the first dot. Typical values range from 5% to 15% depending on the substrate. The value is a job percentage, that will print on an ideal reference press as dark as the first dot prints on this press.
- **Range:** while the previous value measured the amount of dot gain in the highlights, this value expresses how far this effect penetrates into the midtones. Typically plate making is configured in such a way that the mid-tones print like an offset press.

Note:

If you do not specify Dot Gain In Highlights, you will not be able to use the following Viewer viewing mode: **Flexo Print Preview**.

Limited Total Area Coverage

The **Total Area Coverage** is the sum of percentages of the different channels on the same location. A higher total area coverage means more of the various inks is printed on top of each other.

If you want, you can specify the Maximum Total Area Coverage to specify the maximum amount of ink that the substrate can take on the same spot on your press.

Note:

Exceeding the Maximum Total Area Coverage for the press may cause drying problems, unexpected color results, and other problems.

Registration Error

Specifies the average maximum Registration Error on your press.

If you enable this option in the Press Settings, you can use the **Registration Errors** preview in Viewer to check whether your trapping is sufficient to compensate for possible registration errors on the press.

Moiré Settings

In the Moiré Detection settings of the Press Settings, you can define the default values for the settings in the Moiré Detection window.

For more info on the different settings, see [Moire Detection](#) on page 350

10.12.3. The Viewer window

Viewing options

Options particular to certain **Viewer** previews are grouped under **View Options**. You can expand or collapse them using the triangle beside **View Options**.


The following options are available in all Previews:

Separation visibility

In the Ink list, you can:

- Click the eye icon in front of a separation/color name to show or hide it.
- Click the eye button above the separations/colors list to show or hide all separations/colors.
- Click the ink name (not the eye button) to quickly hide all other separations and go in single-separation mode.

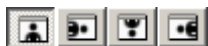
Reverse Printing Order

Use the **Reverse Printing Order button**  to view how your file would look like with its ink order reversed.

When Reverse Printing Ink Order is enabled () , it is always used in all windows, and a red arrow icon is visible in the title bar of all the windows.

Orientation

The orientation buttons allow to change the orientation of the Preview of the job.



This is applied to all windows when **Show in All Windows** is active, or only to the current window when **Show in All Windows** is inactive.

Note:

Keep in mind that this only changes the rotation of the preview, it does NOT actually rotate the job.

Separations

The first viewing mode shows a list of all used inks.

Mode

By default a separation is displayed in color. In order to see a separation as it would appear on film, you can switch to “Show as Film: Positive”.

By switching to “Show as Film: Negative”, the visible separation will be shown in negative.

Note:

As both “Show as Film: Positive” and “Show as Film: Negative” are more useful on a single separation, switching to one of those will show only one separation.

Highlight

Traps

This is not available in Plato

Transparency

This will highlight all objects that have an opacity percentage, a blend mode or an opacity mask (regardless of what is underneath).

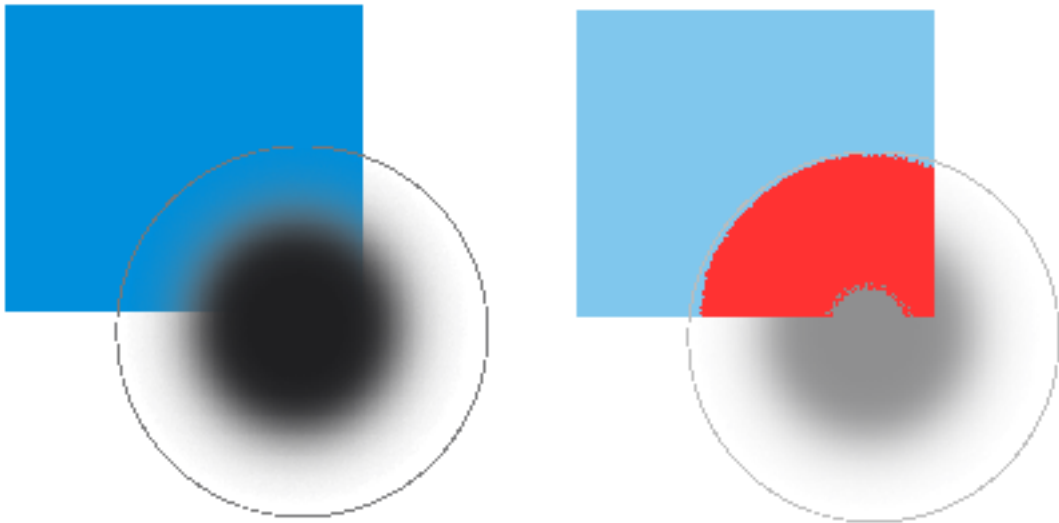
Overprint

Highlight Overprint will highlight the areas where an “overprint” setting in the paint style takes effect.

Screening Conflict

The Screening Conflict checkbox highlights areas where a screening conflict can occur. They can occur if an object with transparency (opacity mask, opacity percentage or blend mode) is overlapping on an object with another screening. The PDF definition clearly states that screening is always taken from the topmost object, even if that object is completely transparent in some areas. In most cases, this is not what you would expect.

In the example underneath, you can see a cyan rectangle, and a circle on top, with an opacity mask. The objects have a different screening. Where the circle is on top of the cyan rectangle, the cyan will take the screening of the blended circle. Therefore, as shown on the right, these areas will be highlighted as Screen Conflict.



Dim Artwork

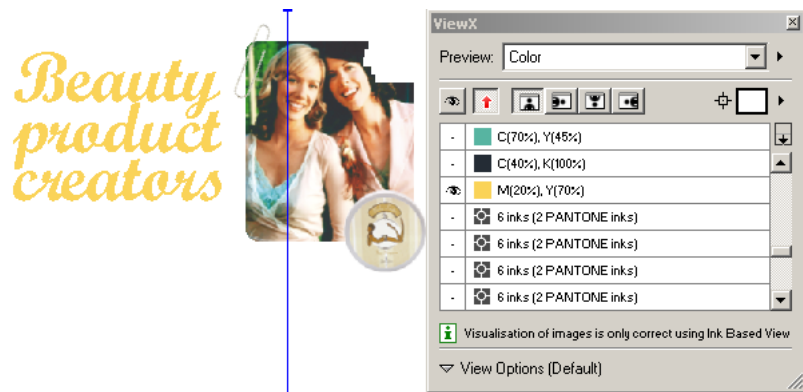
When you enable an Highlight option, the document artwork is faded to the percentage you choose in **Dim Artwork**

Color

When you enable a Highlight option, the objects are highlighted in the color you chose in the Color Picker.

Color

This viewing mode shows a list of used colors.



Click the eye-icons in front of the colors to show or hide them.

Click the eye button above the colors list to show or hide all colors.

Total Area Coverage

The Total Area Coverage (TAC) is the sum of all separation-densities at a certain point in your document.

TAC Limit

The **TAC 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. See [Define Press Settings](#) on page 340). By default the TAC Limit is set to 270%.

Viewing areas over the TAC Limit

When you switch to the **Total Area Coverage** preview, the image in the preview pane will be dimmed and all areas where the sum of the densities is higher than the TAC limit specified in the current Press Setting will be displayed in 100% black.



Note:

You can press the “D” key to activate the Densitometer Tool, and measure the actual TAC amount in a specific area (see [Measure Ink Densities](#)).

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.

Note:

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 (see [Define Press Settings](#) on page 340).

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

The Flexo Plate 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 (see [Define Press Settings](#) on page 340).

Going from light to dark:

- The Flexo Print Preview 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 the **Range** value are displayed unchanged.

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 [Define Press Settings](#) on page 340). 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.



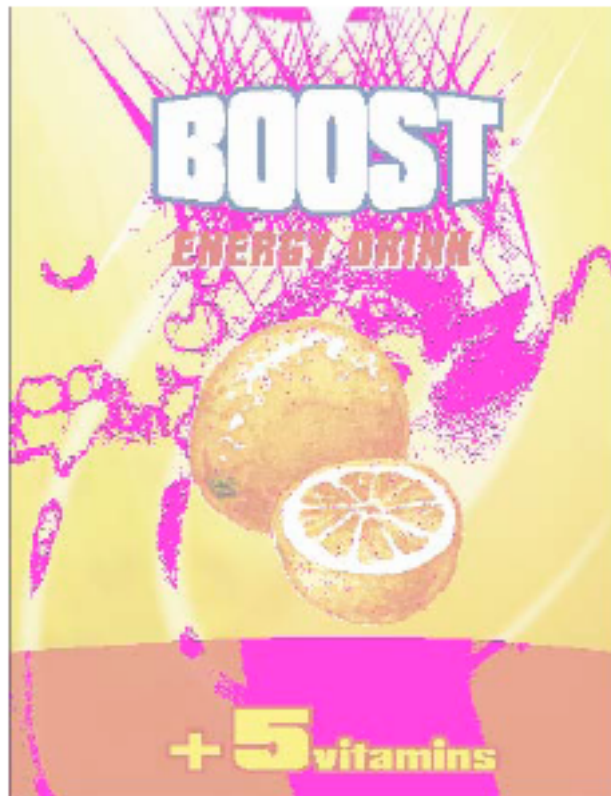
In the example above, you can see a small area of a job in regular preview (left) and with registration errors (right). In this case, you can clearly see no trapping was applied, so white lines appear at the registration shifts.

Breakout

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: see [Define Press Settings](#) on page 340. 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 (see [Measure Ink Densities](#)).



Moire Detection

Moire Detection allows to detect possible screening problems (Moire) in the file.

You can set the options to **highlight overlapping areas**, to detect situations in which Moire might occur:

- when the two overlapping areas have screening angles that do not differ 30 degrees.
- when the two overlapping areas have the same angle
- when the two overlapping areas have different rulings
- when the two overlapping areas have different dot shapes.

You can use the **Ignore inks** option to ignore inks that have

- a density lower than a certain value,
- a density higher than a certain value
- a Luminance higher than a certain value

as in all those cases, the Moire patterning will hardly be visible.

All these Moire Detection options are stored in the Press Settings. See [Define Press Settings](#) on page 340

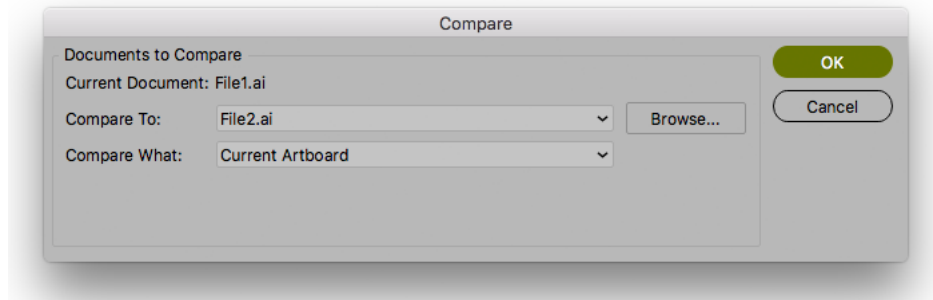
10.12.4. Viewer Compare Tool

You can use the Viewer Compare tool to compare the active document with another document.

To compare the active job:

1. Go to **Window > Esko > Viewer > Compare...**

The Compare dialog opens, and the name for the Current Document is shown.



2. Select the file you want to compare the active document to:

- If you want to compare it to an open document, select its file name from the dropdown list
- If you want to compare it to a document on disk, click the browse button and select the file

3. Define what to compare:

- all artwork
- the current Artboard
- only Trim Box or Media Box (if they are defined)
- only the selected area.

4. Click **OK**

A **Viewer Compare** window will open, showing a comparison between the two jobs.

Note:

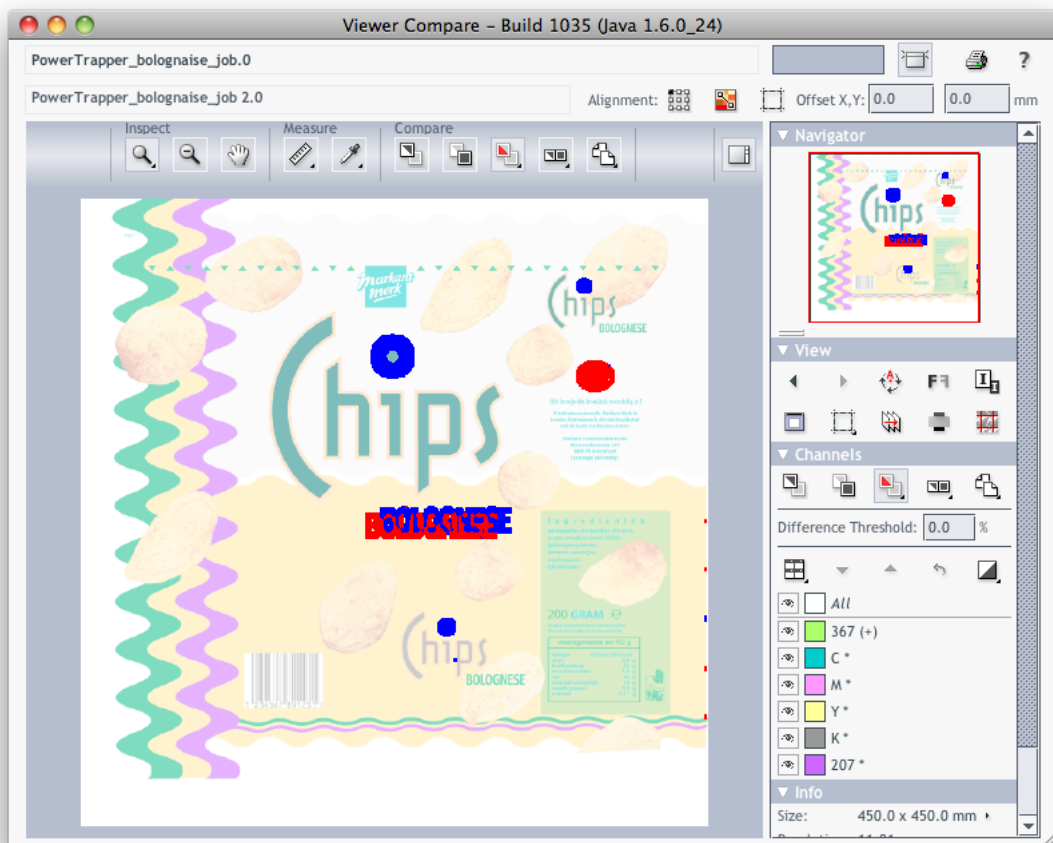
If you have a Trim Box defined in the Illustrator job, the Viewer Compare tool will only show what's inside the trim box.

The Viewer Compare window

The Viewer Compare window shows the two jobs and their differences, and contains a number of tools and buttons to inspect these differences in various ways.

Move over a button with the cursor and the tooltip will indicate what it stands for.

Buttons with a triangle in the lower right corner contain additional options: Control-click (Windows) or Command-click (Mac) the button, or click and hold the button to see the extra options.



Navigating the Viewer Compare window

The Viewer Compare window contains different ways to navigate:

The Inspect tools



- The **Zoom** tool. Under the zoom tool you find
 - Zoom out
 - Fit document in view
 - Fit document width / height
 - Zoom to 1:1 view
 - Show Zoom Options (showing all these options as individual buttons)
- The **Zoom out** tool

- The **Pan** tool

The Navigator

By dragging the red rectangle (when zoomed in), you can navigate through the job.



The View options

The **View** options allow to go back to the previous view, or use additional settings, such as rotate, invert or mirror the view.



Compare modes

Using the **Compare** buttons, you can choose different Compare modes / views



- **View Current version** only
- **View Reference version** only
- **View Both** documents on top of each other.

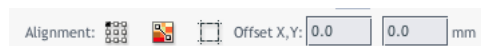
Additional options:

- **View Both** in their original colors

- **View Both - Show difference:** the differences will be shown in the "Added" and "Substracted" highlight colors.
- **View Both - highlight difference:** same as above, but the identical parts are dimmed.
- **View Both - only difference:** same as above, but the identical parts are hidden.
- **Change "Added" Highlight Color, Change "Substracted" Highlight Color and Change Both Highlight Color,** to change the colors in which differences are shown
- **Reset Difference Highlight Colors** to reset the default colors
- **Channel Matching,** to match corresponding channels/inks for the two documents.
- **View Both - side by side** shows both versions next to each other. You can choose between
 - Automatic tiling
 - Horizontal tiling
 - Vertical tiling.
- **View both - mirror reference** shows the current document on top of the mirrored reference document.

Alignment

Comparing files with different sizes may seem like a challenge as they are not aligned in the Viewer Compare window, but it provides some tools that make it very easy.




- **Alignment**
 - on one of 9 corner / center points
 - **Align From Ruler:** see [Align from Ruler](#) on page 354
 - **Auto Align From Ruler:** see [Auto Align from Ruler](#) on page 355
 - **Auto Align:** see [Auto Align](#) on page 356
- **Auto Align:** see [Auto Align](#) on page 356
- **Use trim box for alignment**
- **Offset,** Horizontal and Vertical

Align from Ruler

The ruler tool works by defining two points (one in each file) that have to be aligned.

1.

Select the reference version of the file by clicking the  icon.

2.

Select the Ruler tool



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 **Alignment** button



and select **Align From Ruler**. 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.

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).

Viewer Compare Preferences

You can open the Compare Preferences by choosing **Illustrator > Preferences > Esko > Viewer Compare Preferences** (on Mac OS) or **Edit > Preferences > Esko > Viewer Compare Preferences** (on Windows).

You can define the resolution for the Viewer Compare previews: either based on the document size, or at a fixed resolution. "Based on Document Size" will result in a low resolution for big documents, so e.g. aligning will become more difficult when zooming in. On the other hand, a "Fixed Resolution" can result in a very heavy preview calculation if the fixed resolution is high and the document is large.

You can define if anti-aliasing should be applied.

10.13. Select Die Shape

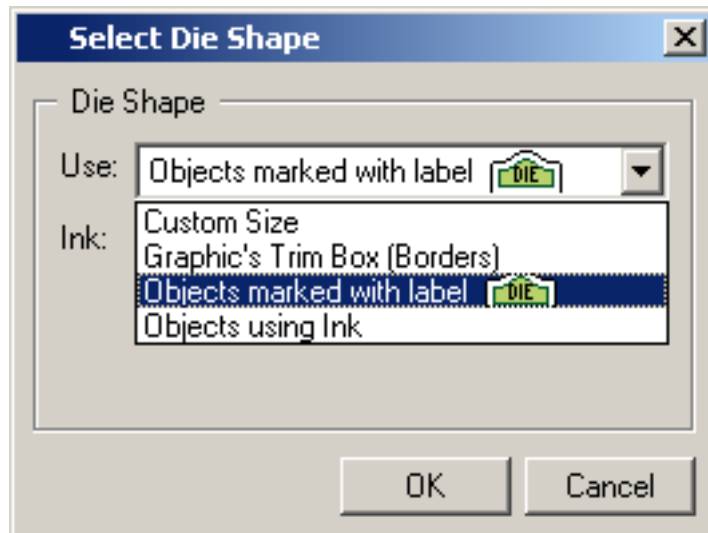
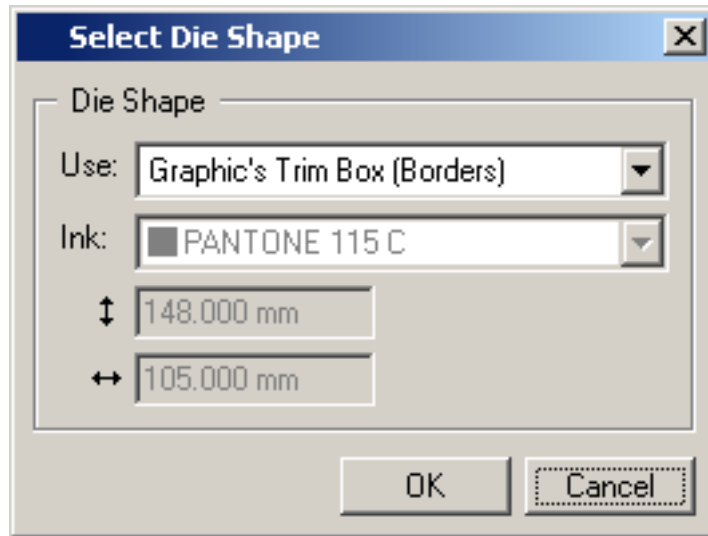
Select Die Shape allows you to create a new die shape contour based on specific settings.

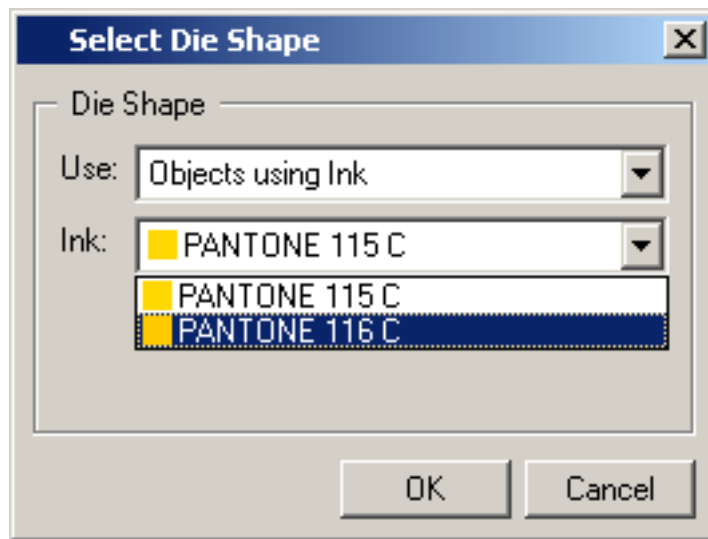
Note:

This option will only be enabled in the Station View.

- **Custom size** creates a rectangular die shape with an adjustable size.
- **Graphic's Trim Box (Borders)** creates a die shape that matches the trim size of the graphic.
- **Objects marked with annotation 'Die'** will use the contours that have the Esko PackEdge annotation "Die".
- **Objects using ink** will use the contours that are placed in a specific separation in the graphic.
- **Outline of all objects** is perfect for creating a die shape around objects that have no die shape in the original file. The new die shape will be created around the union of all objects in that particular file. The Grids dialog box will indicate this die shape by indicating "Manual" in the column Die Shape. The ink and stroke width that will be used to indicate the new die shape are the ones that are selected in the 'Show Manual Cut Lines' option of the [Preferences' View tab](#).

All choices except 'Objects using ink' will copy the stroked contours from the graphic into the station and assign them the Plato Cut ink. 'Objects using ink' will simply copy the contours from the graphic into the station but keep the original separation.

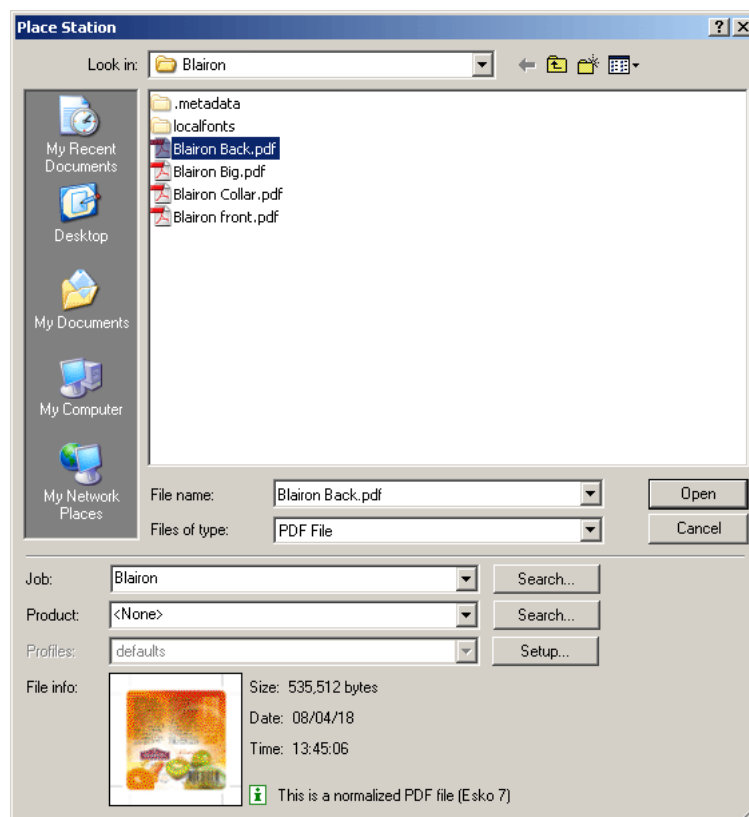




For more information, please refer to the chapter *'Using Select Die Shape'*.

10.14. Place Station...

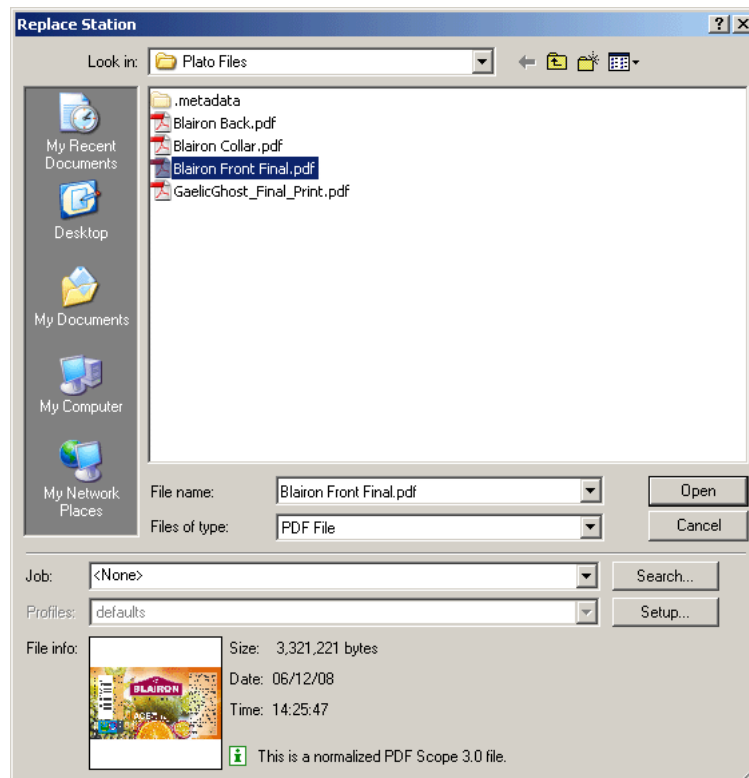
Place Station allows you to open a graphics file and convert it into an internal station.



Depending on the selected file type, clicking **Setup...** opens a dialog box with additional file type related settings.

10.15. Replace by...

The Replace Station dialog box offers you the possibility to select a station that will replace the selected station.

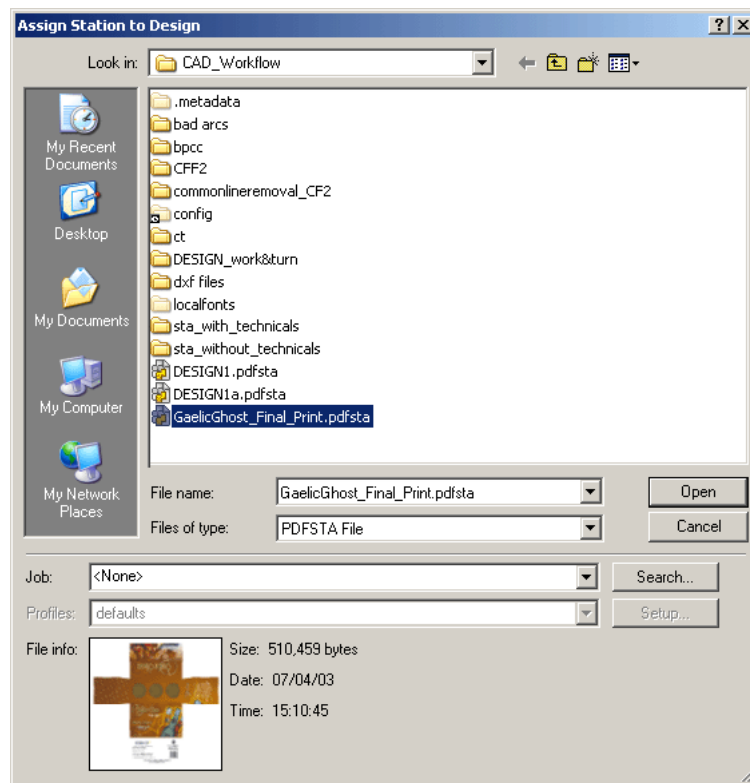


You may want to replace certain stations on your plate with a different station. For example, you might want to replace stations in a plate layout that you did earlier with a new version. Or you may want to replace specific stations to create a combination layout .

Please refer to the chapter *'Replacing a station'* for additional information.

10.16. Assign Station to Design...

The Assign Station dialog box offers you the possibility to select a station and assign it to the selected CAD layout.

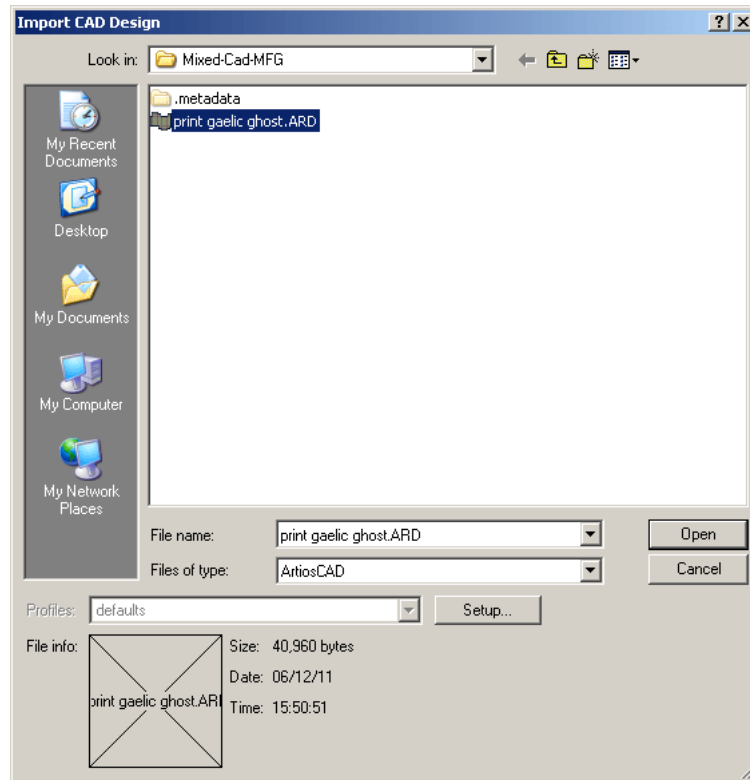


When assigning a station to a layout, the station is analyzed for a die objective in a CAD structure (the CAD file loaded in a STA or GRS file). A dialog box is displayed indicating whether the CAD files match.

Please refer to the chapter *'Assigning a station'* for additional information.

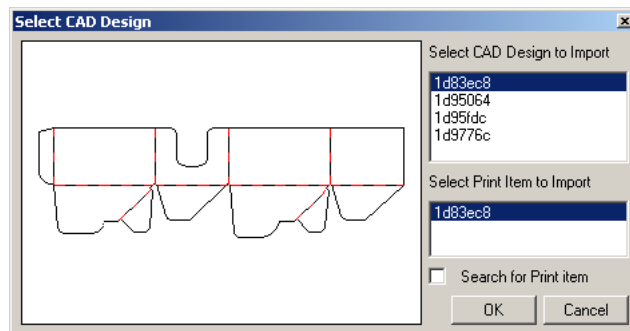
10.17. Import CAD Design...

Import CAD Design allows you to import 1-up CAD designs (ARD files) in the Station View. Also DXF, CFF2, DDES, PDF and LC masks are supported.

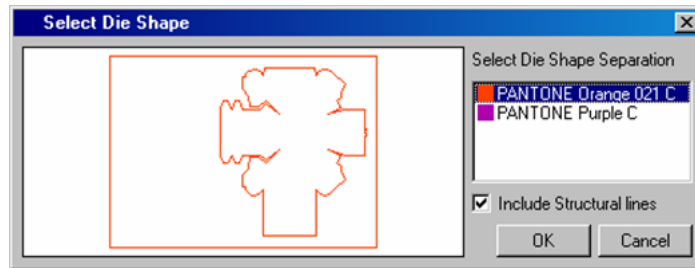


Setup opens the CAD Import dialog box that allows you to do mappings, select or create a certain *CAD style* or search for print items.

If the selected file turns out to be a layout file (MFG file), an extra dialog will pop up to enable you to select the CAD design from the layout.

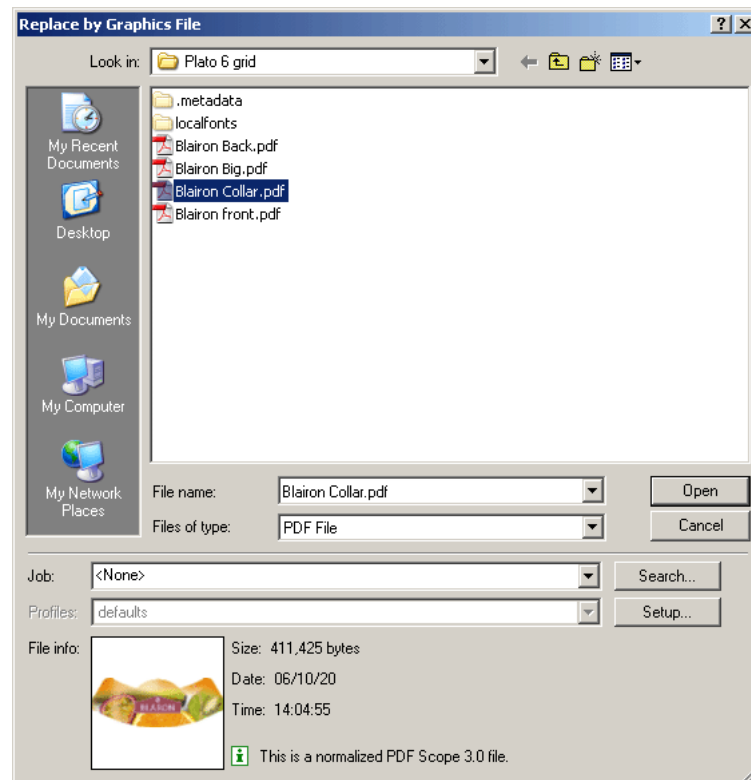


If the selected file turns out to be a PDF file, an extra dialog will pop up to enable you to select the separation(s) that contain the cutting lines.



10.18. Replace by Graphics File...

Replace by Graphics File allows you to replace the existing graphics from a station.



Click **Setup** to open the Import Settings dialog box.

10.19. Save Station

Saves the current station as (PDF)STA file. Saving stations is useful if a station needs to be used in other layouts. The (PDF)PLA layout will still refer to the embedded internal station.



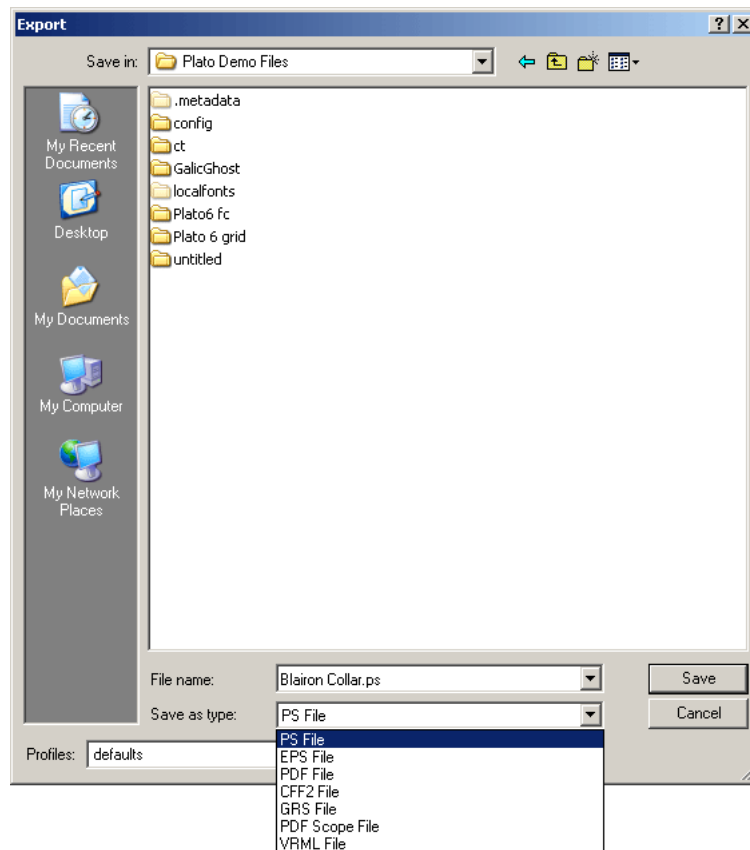
Caution:

Note that a (PDF)PLA file never keeps a reference to a station file, but does keep its reference to the underlying GRS or PDF file. When placing a (PDF)STA file, this file is embedded as an internal station.

10.20. Export Station

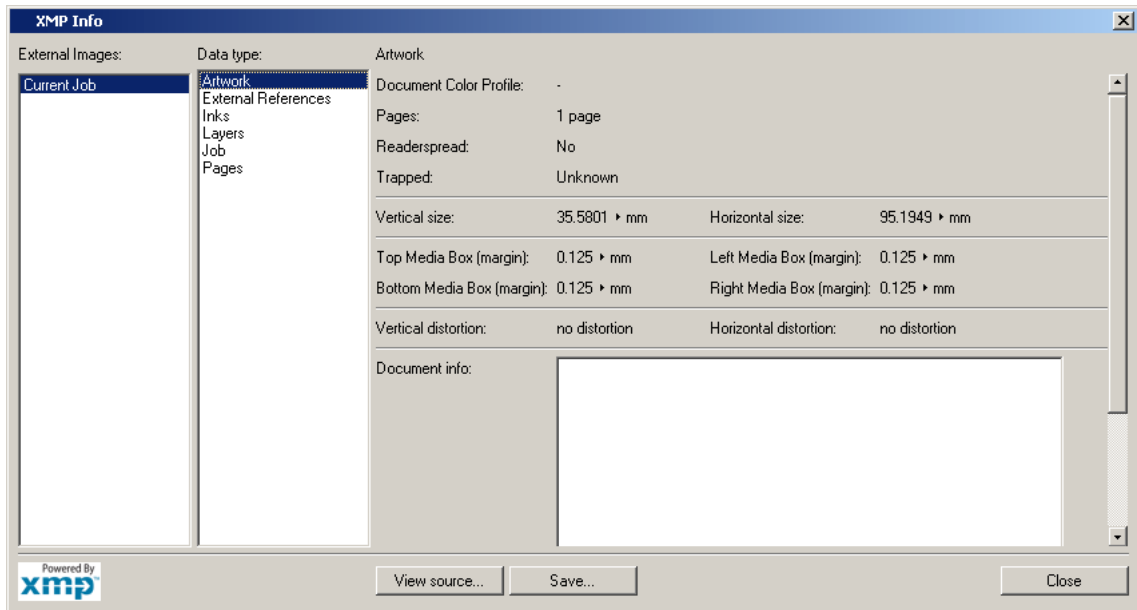
Export Station allows you to export the station data to an external format (PS, PDF,...)

For more information on the different export formats, please refer to the chapter *'Exporting Esko Plato Files'*.



10.21. Station XMP Info...

The XMP Info dialog box provides an overview of all XMP information that is stored inside the station file.



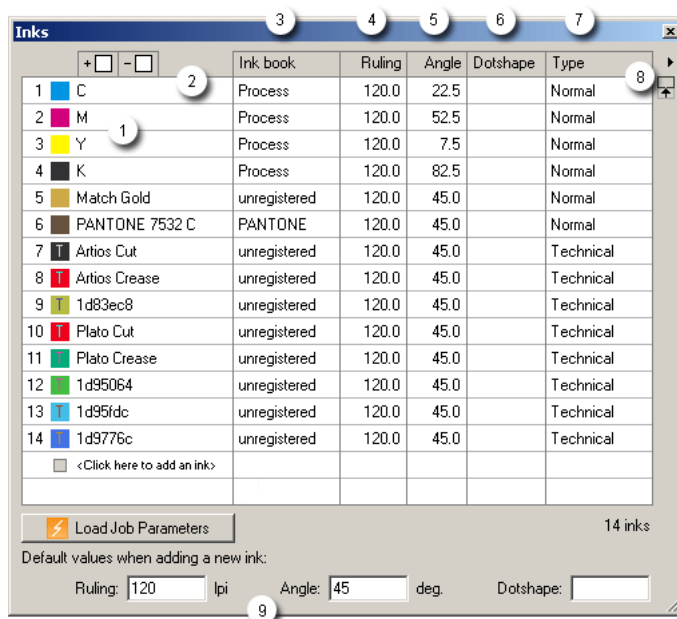
10.22. Inks...

Click 'Inks' to add, modify or remove inks and to display the current inks in your job with their properties.

See also [Working with inks](#) on page 130

Note:

The Inks dialog box in the Station View is purely informative.



1. Ink Number

This number, which appears automatically in front of the Ink box when the ink is defined, identifies the ink by its place in the inks table.

Tip:

If you want to change the order of the inks in the table, drag the number of the ink towards the number where you want to insert it.

Ink Patch

Displays the ink.

When you define an ink by entering its name, the Ink patch automatically displays the defined ink.

When you create new inks, they are added immediately in the Color Palette of the Color/ Style box as a color swatch with 100% of that ink.

When you double-click an empty ink box, the PANTONE Book appears, from which you can select inks.

2. + []

Allows you to add an ink. You can give the name or the number of the ink.


- []

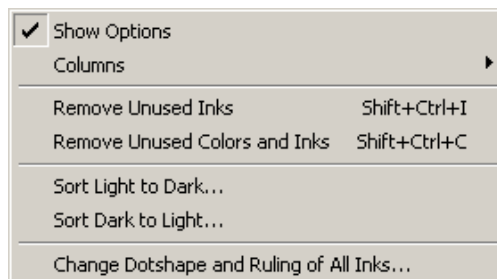
Allows you to remove an ink.



Caution:

You cannot delete inks that are used in external references, nor are you allowed to remove the Plato Cut and Plato Crease inks.

3. **Ink Book:** Lists to which ink book the ink belongs. Information on the Ink Book is automatically assigned by Esko Plato and is for your information only.
4. **Ruling:** To define the screen ruling for the ink (to obtain the appropriate printing quality).
The default value is 120 lines per inch. For more specific information on screen ruling, we refer you to the manual on Screens.
5. **Angle:** To define the screen angle for the ink (to avoid moiré patterns, especially when using process inks). For more specific information on screen angles, we refer you to the manual on Screens.
6. **Dotshape:** To define the dot shape for the ink.
M for Monet, E for Elliptical, ...
For more specific information on dot shapes, we refer you to the manual on Screens
7. The Inks **Type:** choose Normal, Opaque, Varnish or Technical from the drop-down list.
 - Normal - Normal inks are inks that appear transparent when printed over other inks. Normal is the default type for all inks.
 - Opaque - Opaque inks are inks that appear opaque when printed over other inks. Setting an ink to the Opaque type affects only the display of the ink.
 - Varnish - Varnish inks are inks that are used to create varnish plates. An ink set to Varnish does not display its color. Setting an ink to the Varnish type affects only the display of the ink.
 - Technical - Technical inks are inks that are used for layout purposes only. Plato's Cut Ink and Crease Ink are examples of Technical inks. Setting an ink to the Technical type affects the display (always shown as opaque ink) and affects their use in marks. Station Numbers, Geometry Marks and SmartMarks ignore technical inks.
8.  Opens a new menu:



- Show Options: displays / hides the options at the bottom of the Inks dialog box.
- Columns: select the columns you want to see.
- Remove Unused Inks - removes inks that no longer are used in the document.
- Remove Unused Colors and Inks - removes inks and colors that no longer are used in the document. This option will combine colors in Esko Plato's internal palette that are the same and then remove inks and colors that no longer are used.

- Sort Light to Dark - sorts the inks in the document so that the lightest is first and the darkest is last.
- Sort Dark to Light - sorts the inks in the document so that the darkest is first and the lightest is last.
- Change Dotshape and ruling of All Inks... opens a dialog box which allows you to define the ruling and dotshape for all the inks.

9. Default values when adding a new ink: define the default settings for ruling, angle and dotshape, these settings will be used when new inks are added.

Note: If enabled in the fly-out menu, you can also see (and define) the Printing Method. See [Printing Method](#) on page 135

10.23. Ink Mapping...

Note:

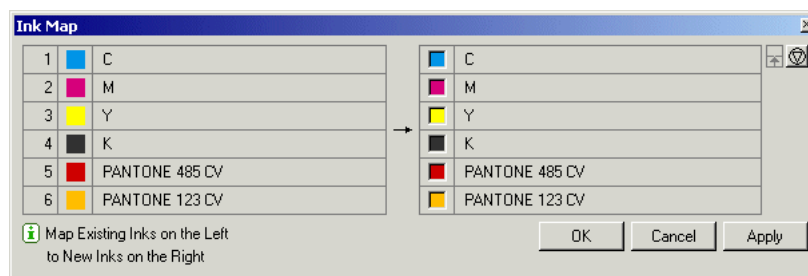
Ink mapping is applied on the complete document and therefore it is only available in Sheet View.

The Ink Map dialog box shows the current ink mapping of your document. Before any ink mapping has occurred, all the inks are mapped to themselves, as shown below.

On the left are the inks in your document; the existing inks. On the right are the new inks that you want to use in place of the existing inks. To map an ink, you replace the new ink on the right across from the existing ink.

Your plate may use inks that need to be changed to other inks. For example, your plate contains the ink "Logo Red" and that ink should be "PANTONE 485 CV". You use ink mapping to "map" one ink to a different ink.

To open the Ink Map window, click 'Ink Map' on the Inks menu.



The Job Ink Mapping dialog box displays the current inks of the job. These are not only the inks of the job objects, but also the inks of the external jobs, for example externally placed Esko native files or CT files.

Note:

After you have applied Job Ink Mapping, the inks that are not used anymore in your job are cleaned up automatically.



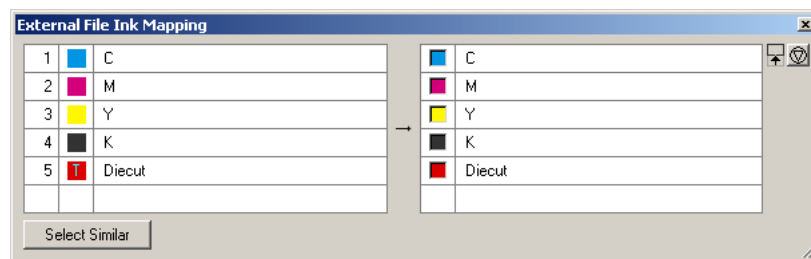
Caution:

It is possible to map two inks to the same destination ink. If, however, the mapped file contains patches in Postscript overprint where the color is defined by both inks, make sure the percentage of the inks is not 0%. If this is the case, due to known limitations, the 0% wins over the other value.

Please refer to the chapter *'Using Ink Mapping'* for additional information.

10.24. External File Ink Mapping...

External file ink mapping is only active in the Station View. It is used to replace the ink separations of the external graphics of a station by any ink from the Inks dialog box.



Ink Map rows: To map the original ink in the left column to the destination ink in the right column.

Name input box: Instead of dragging an ink to an ink box in the right column, you can also fill out the name of an ink in the name input box.

Select Similar: Allows you to select all External Files, this feature makes it easier to perform inkmapping on all externally placed files at once.

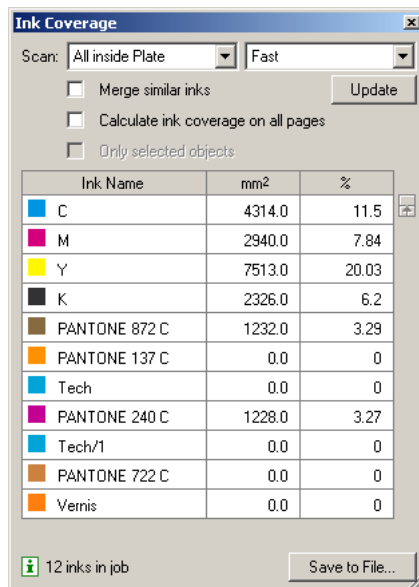


Allows you to reset.

10.25. Ink Coverage...

The **Ink Coverage** dialog box gives you an estimate of the area that the inks on a particular sheet side will cover. The accuracy of the result depends on the calculation procedure you choose from the drop-down list. Either you'll select **Fast** or you'll opt for an **Accurate** calculation.


If you open the Ink Coverage dialog box and there is no data in the list, select the calculation procedure and click the **Scan** button. If the ink coverage of this job has been calculated before, you can ask for a more recent overview by clicking the **Update** button.



- **Fast / Accurate**

Fast indicates that the calculation is faster but you only get a rough estimate of the ink coverage.

Accurate indicates that the calculation might take a while but you get an accurate estimate of the ink coverage.

- **Scan / Update:** The first time, the ink coverage will be scanned. A message, saying: "Calculating Ink Coverage. Please wait." appears. The information in the Ink Coverage dialog box is now updated.
 - All Inside Sheet (Sheet View) scans the area inside the sheet only. Objects that are outside the sheet are not included.
 - All Inside Plate (Sheet View) scans the entire plate. All objects on the sheet and plate are included. If you have objects that extend or are outside the plate, they are not included.
 - All (Station View)
 - All inside Die bounding box (Station View)
- **Merge Similar Inks** allows you to display all objects with the same ink but different ruling and dotshape.
- **Calculate Ink Coverage on all pages** will scan both the front and the back of the plate. The total ink coverage for both sides is then displayed in Ink Coverage.
- **Only selected objects** scans only in selected objects.
- **Update** allows you to re-calculate the ink coverage after some changes have been made in the design to update the values or another type of scan was asked for.
- **mm²** indicates the area that the ink will cover.
- **%** indicates what percentage of the plate is covered with a particular ink.
-  Informs you about the number of inks in your job or it displays the message 'Database has changed'. Click Update to get a new estimation after the changes.

Note:

If the message 'Database has changed' is displayed, you will notice that the 'Save To File' button can be unavailable and/or the separations are dimmed as well. It means that either the design and/or the number of inks has changed.

- **Save To File...** Allows you to save the results of the calculation to a Text Report or a CSV Report which can be printed or edited. The dialog box allows to select a destination directory, to specify a file name and the type of file. The drop-down list allows to choose between Text Report or CSV Report. Both types can be opened in any text editor (e.g. Wordpad, Notepad...) A CSV file can be opened in a spreadsheet program.

Please refer to the chapter '*Calculating Ink Usage*' for additional information.

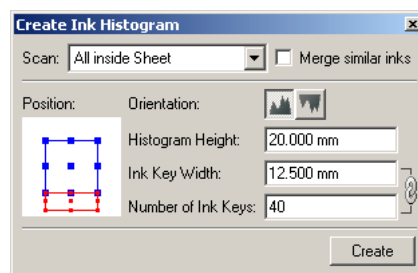
10.26. Create Ink Histogram...

By selecting Create Ink Histogram you can calculate the ink key histogram from the current document and add it as a database object in the document.

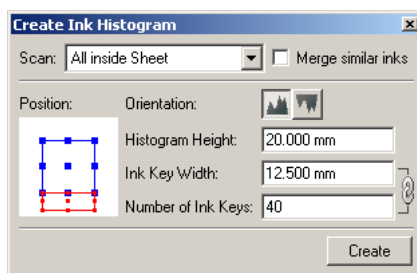
The ink key histogram is an important factor in setting up the press. It allows you to **regulate the ink consumption in each printing zone** before printing. This will significantly reduce the press setup time. It is also useful to help you to decide whether or not to modify the layout of your designs on to plate or to have a more homogeneous distribution of the ink consumption along the plate.

This ink key histogram is a database object and will be saved in the file. Position, orientation, number of ink keys and scan borders/margins can be defined.

You can adjust the ink consumption by manually **creating ink eater areas in Plato**, based on the ink key histogram. Use the Fill Ink Eater Area tool in Plato to automatically adjust the ink distribution.



Create Ink Histogram dialog box



Scan: Determine the area the scan has to cover.

- *All inside Trim box* will take into account all information within the trim box.
- *All inside Media Box* will take into account the information inside the media box.

Merge Similar Inks: the Ink Histogram tool merges the results for inks that are the same but have different ruling and/or dotshape.

Position: specify the position of the ink histogram with respect to the trim box. To do so, select the Ink Histogram object and click on the Position control to align the Ink Histogram on the top, bottom, left or right side of the document. The Ink Histogram object in your document will be updated automatically.

Tip:

When the Ink Histogram is aligned on the top or bottom, the ink histogram zones are calculated in vertical zones of the document. In case the Ink Histogram is aligned on the right or left side of the document, the histogram is calculated in horizontal zones.

Orientation: specify the orientation of the ink histogram bars in the ink histogram object.

Histogram Height: specify the height of the ink histogram object in the document.

Ink Key Width: specify the width of a single ink key bar in the histogram. The Ink Key Width is normally set to the same width as the width of a single ink key zone on the press.

Number of Ink Keys: specify the number of ink histogram bars or ink key zones on the press.



Caution:

The ink key histogram will not be updated automatically when the Ink Histogram Object is moved, or when the document is changed or saved. You have to select the Ink Histogram Object and click on the **Update** button to update the Ink Histogram.

The Ink Histogram in the image below is attached to the bottom of the plate. It displays 40 ink keys of 12.5 mm each. This represents the settings in the Create Ink Histogram dialog box.

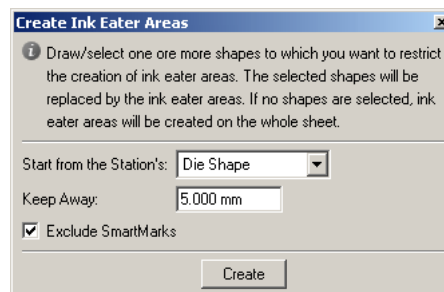


Caution:

The ink key histogram will not be updated automatically when the document is changed or saved.

10.27. Create Ink Eater Areas...

In Plato you can automatically create Ink Eater Areas. These 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. Choose Inks > Create Ink Eater Areas to do so.



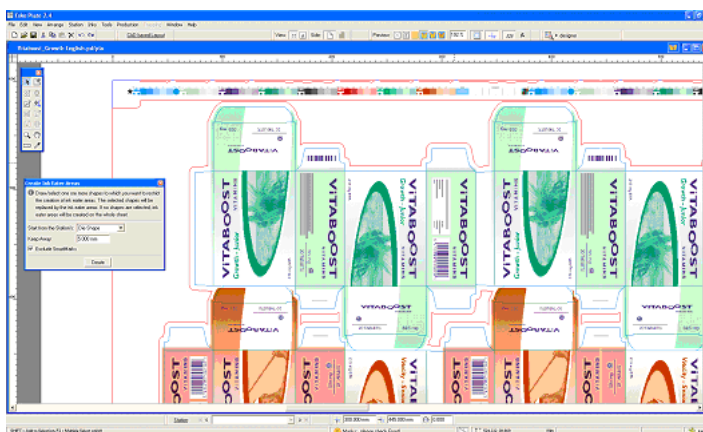
In the Start from the Station's field you can determine the edge of the Ink Eater object. Choose one of the following:

- Die Shape: when the Die Shape is defined in the Station's dialog 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.

In the Keep Away field you can set the offset for the Ink Eater object in relation to the Die Shape or Trim Box.

By enabling or disabling the Exclude SmartMarks option you can set whether or not existing SmartMarks are included in the Ink Eater object or not.

Click Create to create the Ink Eater object. This is an empty contour that is placed in the document and can be used by the Fill Ink Eater functionality.

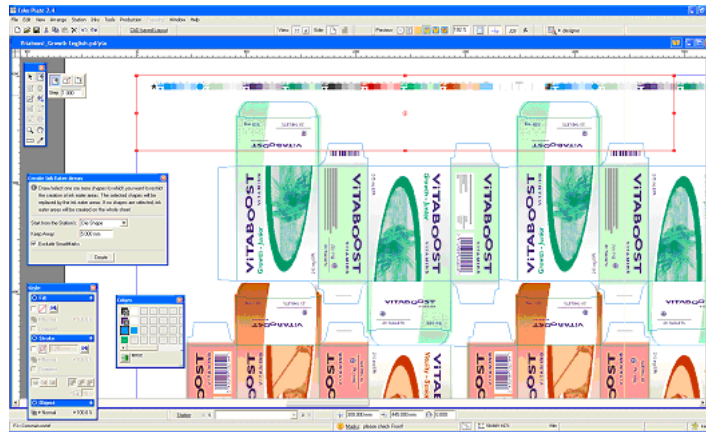


You can clearly see that the created Ink Eater object (red selection) has an offset in relation to the used Die Cut of the one-ups (blue). The SmartMarks are excluded from this Ink Eater object. You can now use the zone in this object to compensate the ink key histogram.

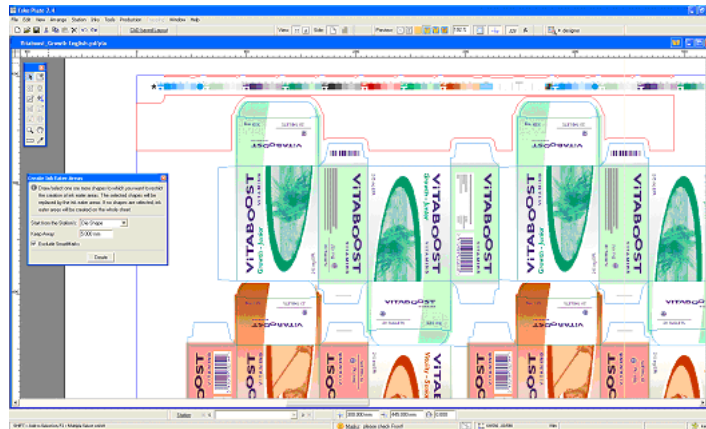
Tip:

When you select an contour object that already exists, this contour is used as the maximum surface from which the Ink Eater Area can be created.

Determine the zone where you want to compensate by creating a contour object. In the case below we are only going to compensate the upper part of the plate.



Click Create to turn the selected object into a compensation zone. The Die Shape is set as edge for the ink eater object. The SmartMarks are excluded in this case.



10.28. Fill Ink Eaters...

Fill Ink Eater Areas is an ink eater adjustment tool that allows you to compensate the ink key histogram for all separations based on the visualization of the ink key histogram.

The **ink consumption on the press** is determined by the design of the plate: the design of the plate dictates which ink percentages are used where. Ink consumption is regulated by ink keys on the press. Every ink key determines the flow of ink to a specific zone on the press. The goal of the Ink Eater Areas feature is 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.

Avoiding "peaks" and "lows" in the ink key histogram also avoids an excess of ink building up on the press cylinders, which might otherwise spill onto non-printed zones. **Adding and filling Ink Eater Areas to your plate** flattens the ink key histogram by filling non-printed zones with a vignette (gradient). A gradient is used to avoid hard edges in the compensation object,

and extra options are available to set a minimum ink density, minimum ink key value and to indicate when the maximum total area coverage (TAC) is too high.

Note:

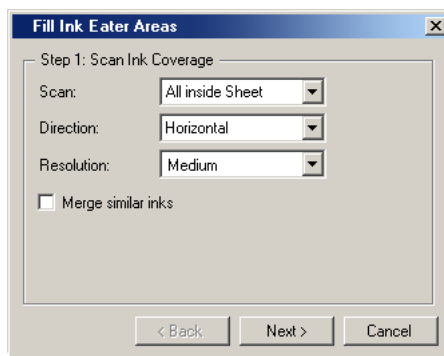
Total ink consumption on the press will be higher when using ink eaters.

Note:

In PackEdge, you have to create Ink Eater Areas manually with contour objects. The Create Ink Eater Area tool in Plato can automatically determine suitable areas for use as ink eater areas.

10.29. Fill Ink Eater Areas wizard

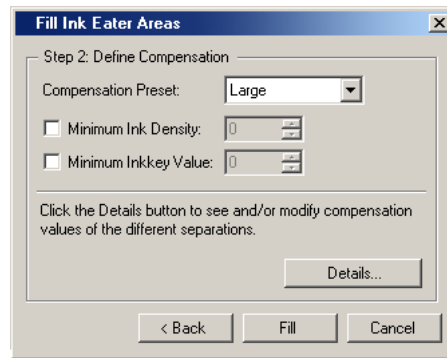
1. Select a contour object or a group of contour objects in the document that can be used for the compensation. Note that a group of contour objects are linked to one object when using the Fill Ink Eater Area tool.
2. Choose Fill Ink Eater Areas from the Paint menu to start the wizard.



3. Select the appropriate options:

- **Scan:** Determine the area the scan has to cover. "All inside Trim box" will take into account all information within the trim box. "All inside Media Box" will take into account the information inside the media box.
- **Direction:** indicate the direction for the compensation. Horizontal means 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:** indicate the number of intermediate steps of the gradient to be used as compensation. Use 'Very Low' for 5 steps, 'Low' for 15 steps, 'Medium' for 30 steps, 'High' for 45 steps and 'Very High' for 60 steps. The compensation is more local when using fewer steps.
- **Merge Similar Inks:** the Ink Histogram merges the results for inks that are the same but have different ruling and/or dotshape.

4. Click **Next**.



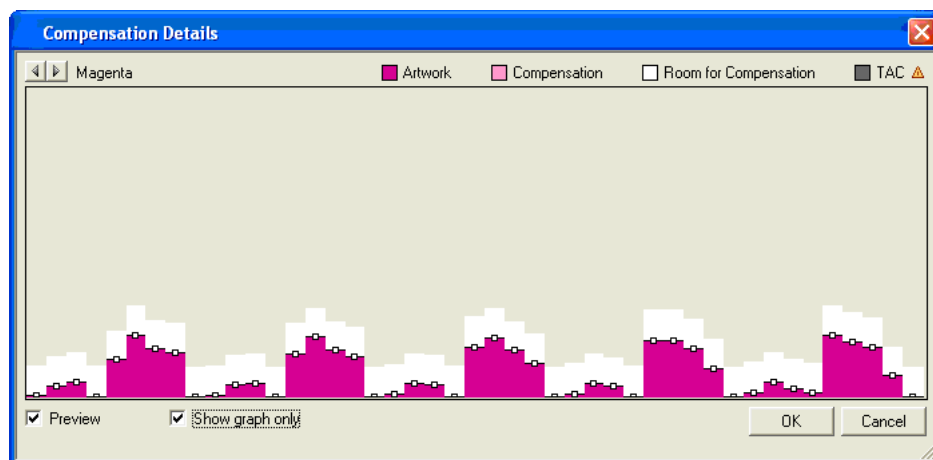
5. In Step 2, define the compensation parameters:

- **Preset:** indicate the intensity of the compensation gradient.
- **Minimum Ink Density:** set the minimum ink density of the ink separation in the ink eater area.
- **Minimum Inkkey Value:** indicate the minimum ink key histogram value for each ink key zone. The ink density of the gradient is adjusted to make sure the ink key histogram value is higher than this settings. Keep in mind that it is not possible to have this minimum inkkey value for all ink key zones when the ink eater area is not large enough.
- **Details:** click to open the Compensation Details dialog. Modifying the Preset, Minimum Ink Density and Mimumum Inkkey Value settings in Step 2 of the Fill Ink Eater Areas wizard will adjust these settings for all separations. Use the Compensation Details dialog to adjust the individual separations.

6. Click **OK** to fill the ink eater area.

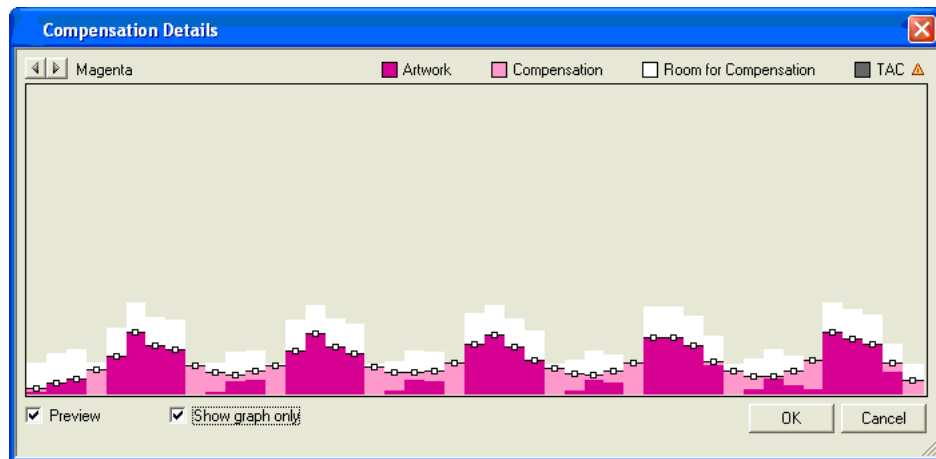
The Compensation Presets in detail

- Choose the **Preset "Small"** when you want a light compensation to softly smoothen the Ink key historgram in a local zone.

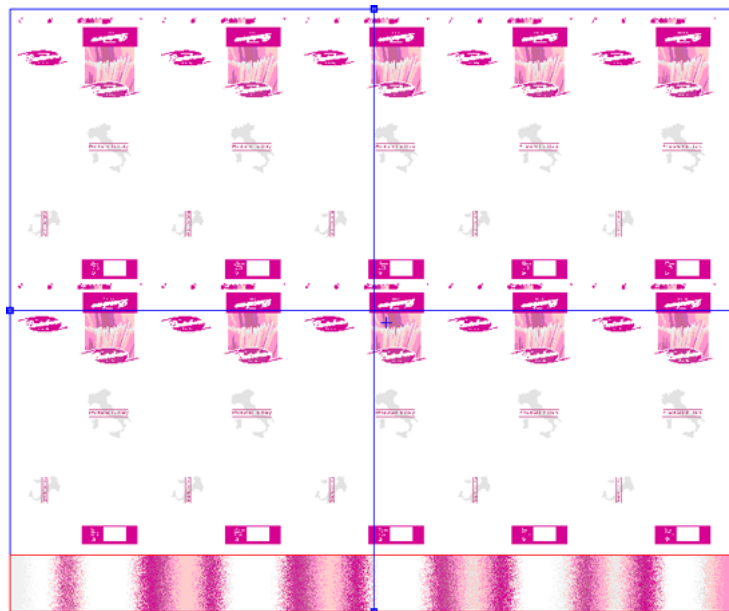


The dark magenta area in the compensation graph is the current ink histogram of the (uncompensated) document. The white area in the graph is the area that can be used for the compensation as defined by the ink eater area(s). Light magenta areas have been compensated by the ink eater.

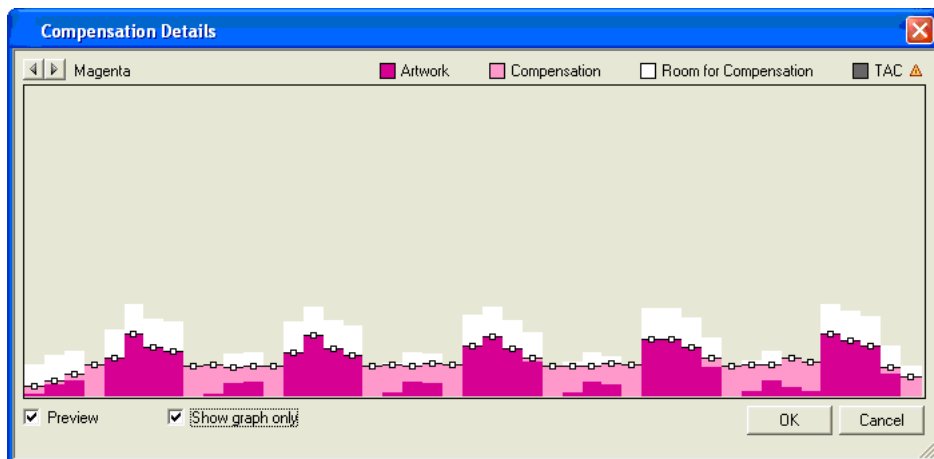
- Choose the **Preset "Medium"** to fill the compensation area with a gradient, resulting in a smooth ink histogram.



The light magenta area indicates the zones that have been compensated. The ink eater area would look as follows:



- Choose the **Preset "Very Large"** to fill the ink eater area with with a gradient to equalize the ink histogram as much as possible. It is clear that doing this will use more ink.

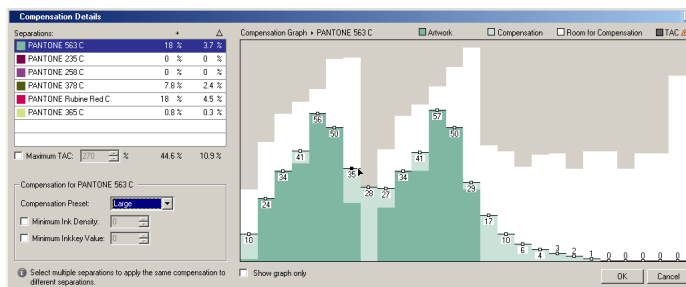


As you can see, the compensation is quite strong in this example. The ink eater area is filled with an ink-heavy gradient.



10.30. Compensation Details dialog

Use the Compensation Details dialog to adjust how the ink eater area is filled for individual separations.



Tip:

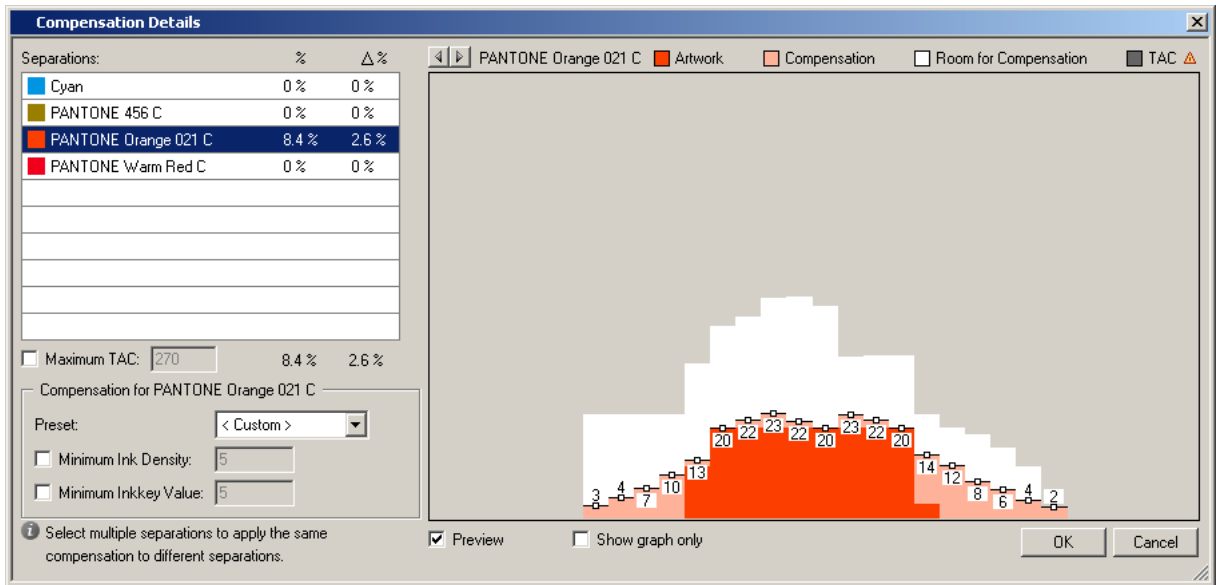
Turn on the Preview option

To fine-tune the gradient in the ink eater area, do one or more of the following:

- **Click a separation** in the Separations list to view the ink histogram and compensation room for that separation.
- Select a Compensation Preset to apply one of the standard compensation algorithms to the ink histogram.
- Set the minimum ink density and/or ink key values for every separation.
- Drag the handles on a bar in the ink histogram to **manually tweak the ink key value**. The white bars in the histogram show the allowed compensation room for every bar. You cannot drag a bar higher than the white area, or below the original uncompensated artwork value.
- Enable the option **Maximum TAC** 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. The offending bars in the histogram will be highlighted with a blinking effect.

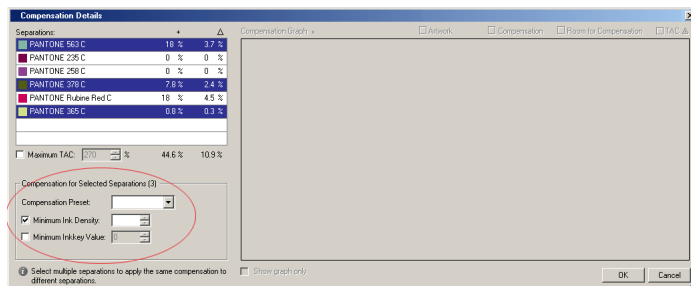
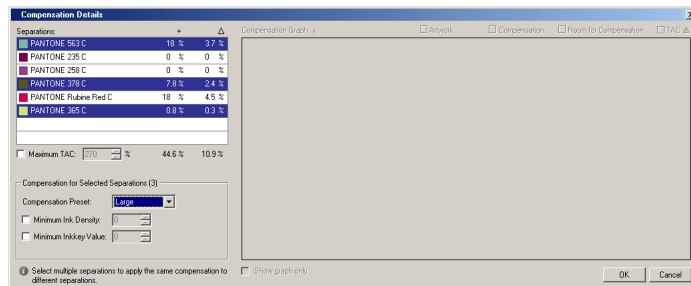
Tip:

You can select multiple separations to apply the same compensation to multiple separations simultaneously.



Tip:

Select multiple separations to apply the same compensation to different separations.



When there are more separations selected with conflicting settings, the read out of the Compensation user interface is not defined.

10.31. Open Ink Book

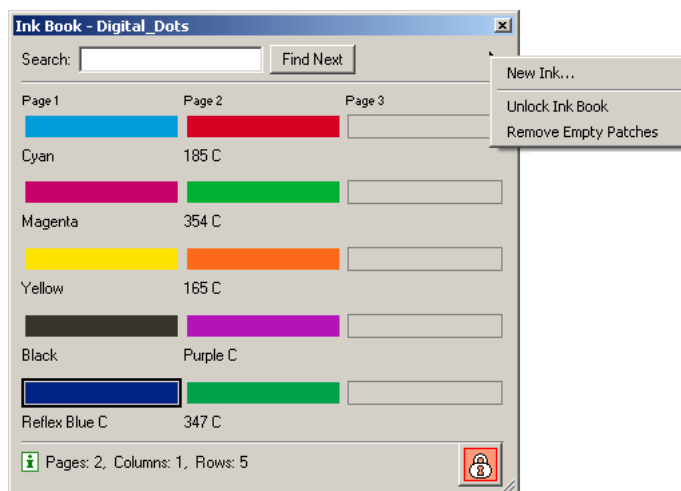
Opens the selected ink book.

Note:

Inks in standard ink books (PANTONE, Toyo, HKS, NCS,...) or PantoneLIVE ink books cannot be altered. In custom ink books you can rename or delete inks and you can create new inks based on Lab or RGB values.

Tip:

It is advised not to create custom ink books that contain standard inks as those will not be recognized as being an ink from the custom ink book, but from the standard ink book.

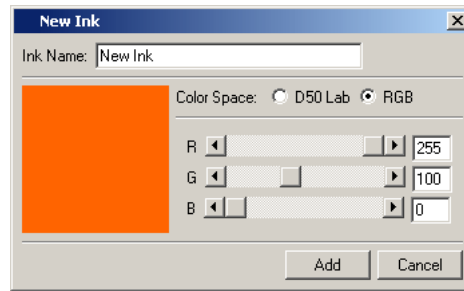


The info icon provides extra information on the number of pages, rows and columns of the selected ink book. Empty ink patches in the book are left open, which implies that the layout of the ink book appears unaltered the Ink Books dialog box. You can remove empty ink patches by clicking **Remove Empty Patches** from the fly-out menu.

You first have to unlock the ink book before you can do certain modifications, like removing empty ink patches or renaming inks. Creating a **New Ink** will automatically unlock/lock the ink book. You can either specify the ink by entering Lab values or RGB values.

Enter a value in the **Search** box to search for a specific ink in the selected book.

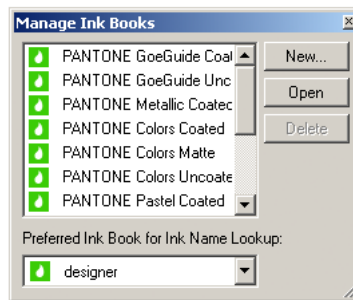
New Ink... allows you to create a new ink by entering Lab or RGB values.



You can remove inks by selecting the ink patch and selecting **Clean Ink** from the context menu.

10.32. Manage Ink Books...



Shows you an overview of all ink books that are installed on the system and allows you to create, delete and view individual ink books.



The selected ink book is regarded as the preferred ink book and will be checked first when inks are entered in the Inks dialog box.

Click **New...** to create a new ink book on the CMS data tree. All you have to do is enter a name.

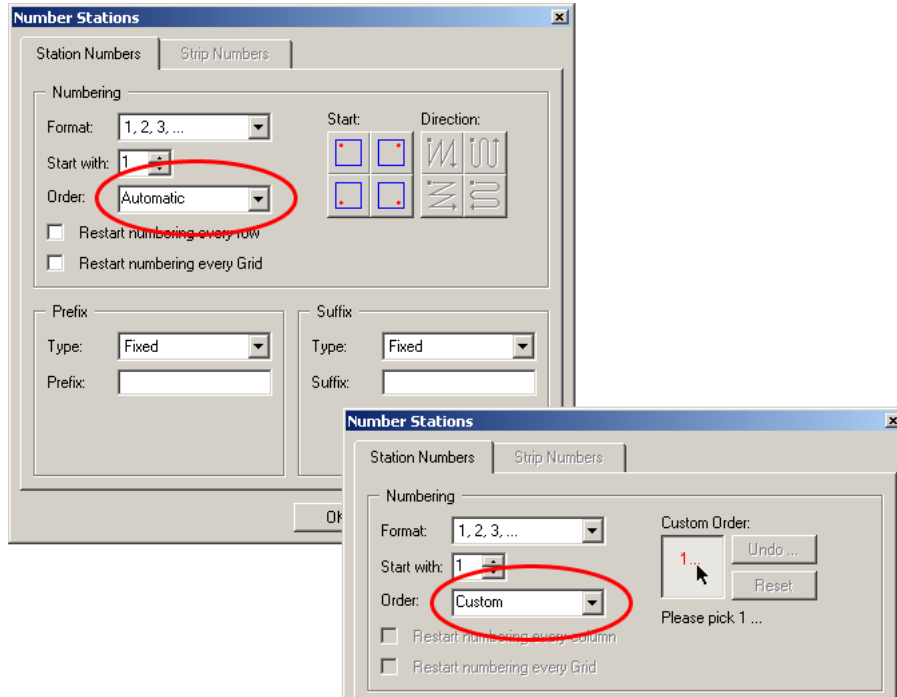
Please refer to the chapter '[Open Ink Book](#)' for additional information on how to add and create inks.

Note: Plato supports PantoneLIVE ink books. In Manage Ink Books, an icon will show if the inkbook is a Color Engine Ink Book  or a PantoneLIVE ink book .

10.33. Number Stations...

Use the Number Stations command to assign station numbers to the stations in your layout. You must have a station number definition in your station for the numbers to be displayed on the plate. You can add a station number to a station at any time (in Station View mode). The actual numbering of the stations takes place in Esko Plato's Sheet View.

Exactly the same mechanism is used for strip numbers. A strip number is defined in Station View, the actual numbering is done in Sheet View. The Number Stations dialog box allows to determine how the actual numbering will be executed (format, sequence, ...)



Numbering

- Format: indicate the number format; numbers or lower or upper case latin characters.
- Start with: indicate the number or character that will be used for the first station.
- Order: select either automatic (by setting start position and direction) or custom (define the order of the stations by clicking in each station).
- Start: indicate the starting point for automatic numbering.
- Direction: indicate the direction in which the automatic numbering should be carried out.
- Restart numbering every row/column: This applies to automatic numbering. Whenever a new row (or column, depending on the chosen direction) is started, the numbering restarts using the starting number (defined under 'start with').
- Restart numbering every Grid: This applies to automatic numbering. The chosen sequence will be applied to each grid individually.

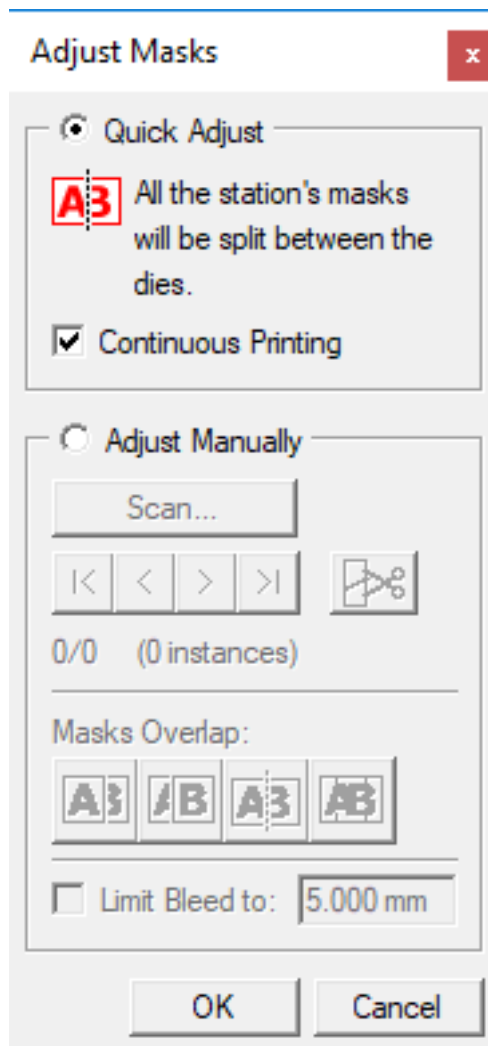
Prefix / Suffix

- Type:
 - Fixed: prefix/suffix will be used for each station number.
 - Variable: prefix/suffix will change for each new row (or column, depending on the chosen direction).
- Prefix/Suffix: in case of fixed prefix/suffix, type in the prefix/suffix that should be used.

Please refer to the chapters '[Numbering stations](#)' and '[Station/Strip Number tool](#)' for additional information.

10.34. Adjust Masks

The Adjust Masks tool is used to adjust overlapping stations. It will create new masks that are applied to the stations on the plate. The stations' original masks are not altered. The newly applied masks will further reduce the bleed of the stations in areas where there is overlap between two stations.








After scanning, the Adjust Masks dialog box looks like the one above. All the overlapping stations have been located and these overlaps have been grouped into records. Each record represents one kind of overlap which may occur any number of times on the plate. All overlaps of the same "kind" are grouped together into one record. Choosing a mask decision for a record will apply that decision to all instances where that overlap occurs. While a plate may contain many records, only one record is shown at a time in the Adjust Masks dialog box.

1. Quick Adjust splits overlapping masks halfway between the two die shapes.

If the layout is meant for web printing, you can enable the **Continuous Printing** option. The Quick Adjust of the stations at the top and bottom of the layout will take into account that the layout is repeated, using the vertical sheet size as Repeat Length.

If you do not wish to use the Quick Adjust option, you work in manual mode. This allows you to make a decision on the priority of overlapping flaps and panels per case.

2. Scan - Click the Scan button to scan the stations on your plate. The Scan button is disabled, if scanning was done already.
3. Forward/Back - Use the Forward/Back buttons to see each record. The first button moves to the first record in the list. The second button moves to the previous record in the list. The third button moves to the next record in the list. Finally, the fourth button moves to the last record in the list of overlaps. Some of the buttons may be disabled depending on which record you are currently viewing.
4. Split Overlap tool  The Split Overlap tool allows you to cut the current record into two records, each of which can have a different mask decision. This is particularly useful for plates that contain folding cartons where two flaps on the same carton must be masked differently. See [Splitting an Overlap Record](#).
5. Record Number - The Record Number show you where you are in the list. In the dialog box shown above, you are looking at the second record in a list of five records. The overlap represented by this record occurs twice on the plate (2 Instances).
6. Mask Decision - The Mask Decision Buttons determine how the masks are created. Each button performs a different specific action on the current record:
 -  A over B - The station marked "A" will overlap the station marked "B". The mask from station A will be used to cut back station B. If station A does not contain a mask, the bounding box of station A will be used.
 -  B over A - The station marked "B" will overlap the station marked "A". The mask from station B will be used to cut back station A. If station B does not contain a mask, the bounding box of station B will be used.
 -  Split - The masks of both stations will be split between the dies. This is Esko Plato's default Mask Decision. After scanning, all records will be set to Split initially.
 -  Ignore - A record set to "Ignore" will not be altered by the Adjust Masks tool. This overlap will be ignored and will remain the way that it is now in the plate.

As you view each Overlap Record, you can choose one of these four Mask Decisions. The currently selected Mask Decision is displayed in the document window so that you can see the result of this decision. This way, you can view the effect of all the choices to make the right decision.

7. Limit Bleed - Check the Limit Bleed box and enter the maximum distance over which one station 'bleeds' into the adjacent station.

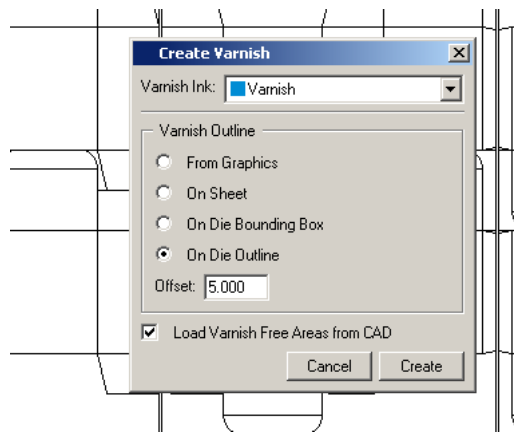
Please refer to the chapter ['Adjusting overlapping stations'](#) for additional information.

10.35. Create Varnish

Create Varnish loads a varnish layer. When a CAD layout is loaded, the function 'Create Varnish' will be enabled. This allows to create varnish areas based on the loaded CAD layout. If the CAD layout contains coating free areas, these can be taken into account.

Note:

The varnish ink must be defined in the ink table already.



The varnish areas can be assembled by creating a contour that defines the outer limits of the varnish. This contour can be:

- From Graphics: Present in the graphics.
- On Sheet: Based on the boundaries of the sheet.
- On Die Bounding Box: Based on the bounding boxes of the individual dies.
- On Die Outline: Based on the union of all die lines.

Excluding from the above mentioned contour, are the areas defined in the CAD layout as 'varnish free' areas (Artios MFG files only).

Note:

The 'Load Varnish Free Areas for CAD' becomes available only when the file contains varnish information.

Note:

The 'From Graphics' option becomes available only when the CAD file contains varnish (or rather 'coating free') areas.

When this option is selected, no real varnish area will be created (as the varnish info is already part of the graphics). Only the varnish free areas from the CAD file are used to selectively knock out areas in the varnish separation.

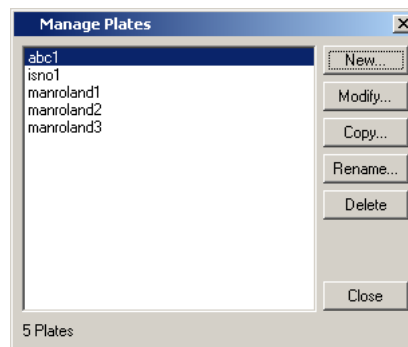
You can define an offset.

Note:

The offset direction depends on whether you create a varnish based on sheet (offset to the inside, a negative offset) or based on die shape (to the outside, a positive offset).

10.36. Manage Plates

'Manage Plates' allows you to set up a collection of pre-defined plate sizes. The list of plate sizes will be available in the Layout Setup.



10.37. SmartMarks

Note:

While marks are common to Esko PackEdge and Esko Plato, a certain number of SmartMarks are specific for Esko Plato (and are not supported in Esko PackEdge): the '*Position Marks*' and the complete range of QuickStep regmarks. The QuickStep regmarks are only available if the QuickStep license is active.

The SmartMarks dialog box allows you to create, save, open and edit sets of SmartMarks. Sets can be saved to disk and can be used during output. SmartMarks are regenerated as soon as something in the job changes: job size, ink definitions...

SmartMarks can be attached to

- the general level



- a sheet



- a plate



- visible objects

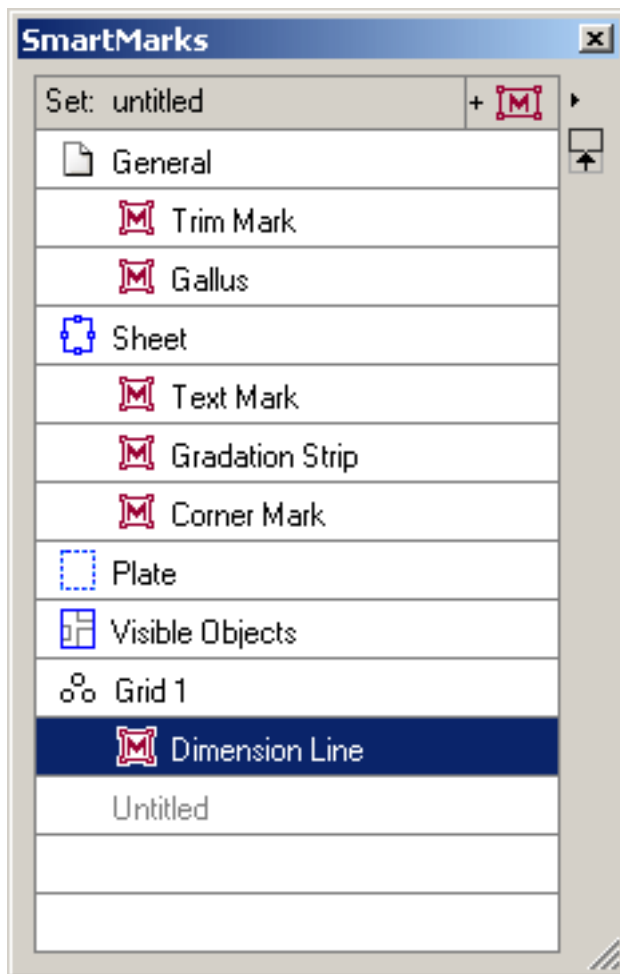


- a grid



- Position Marks (objects in your job which have an object name)

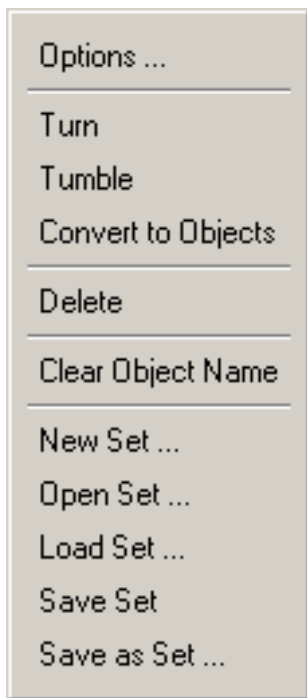




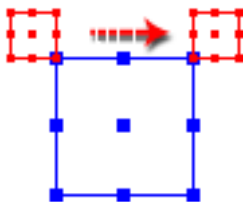
Click the black arrow



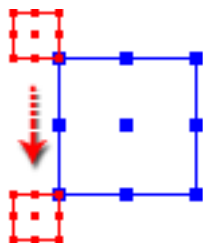
to activate the fly-out menu.



- **Options...:** Opens the SmartMarks dialog box for the selected mark. Please refer to the chapter '*Supplied SmartMarks*' for an elaborate explanation of the individual marks.
- **Turn:** Relocates the SmartMark, using the vertical axis of your job.



- **Tumble:** Relocates the SmartMark, using the horizontal axis of your job.



- **Convert to Objects:** This converts the selected SmartMarks to objects. These objects will look like marks in your job, but they are in fact dynamic objects and will behave like ordinary objects: they can be edited, rotated, scaled...
- **Delete** Removes the selected mark from your list.

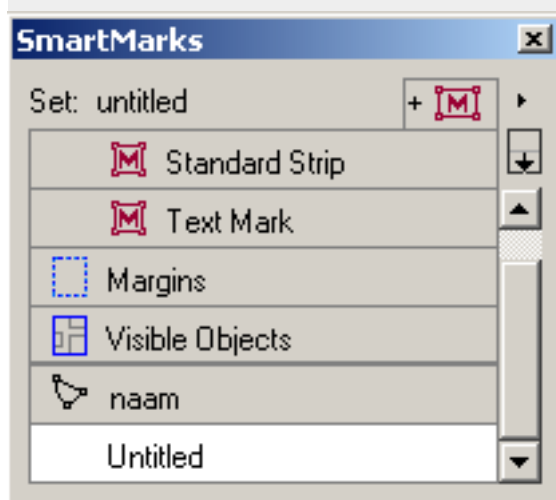
- **Clear Object Name:** If the SmartMark has an object name, you can clear it here.
- **New Set...:** empties your list of marks.
- **Open Set...:** opens a saved set.
- **Load Set...:** adds a saved set to your list.
- **Save Set...:** saves your list.
- **Save as Set...:** saves your list under a new name.
- A list of **recently used sets**.

Tip:

To change the name of a SmartMark, click it in the list and edit the field.

Tip:

An easy way to identify the elements in your job is to use *Position Marks*. Select the object. The SmartMarks dialog box highlights 'Untitled'.



Click and hold this item, the field will become editable, enter a new name.

Note:

It is important to know that all sets will be saved in the data directory of Esko Plato: `bg_data_marks_v010/dat`.

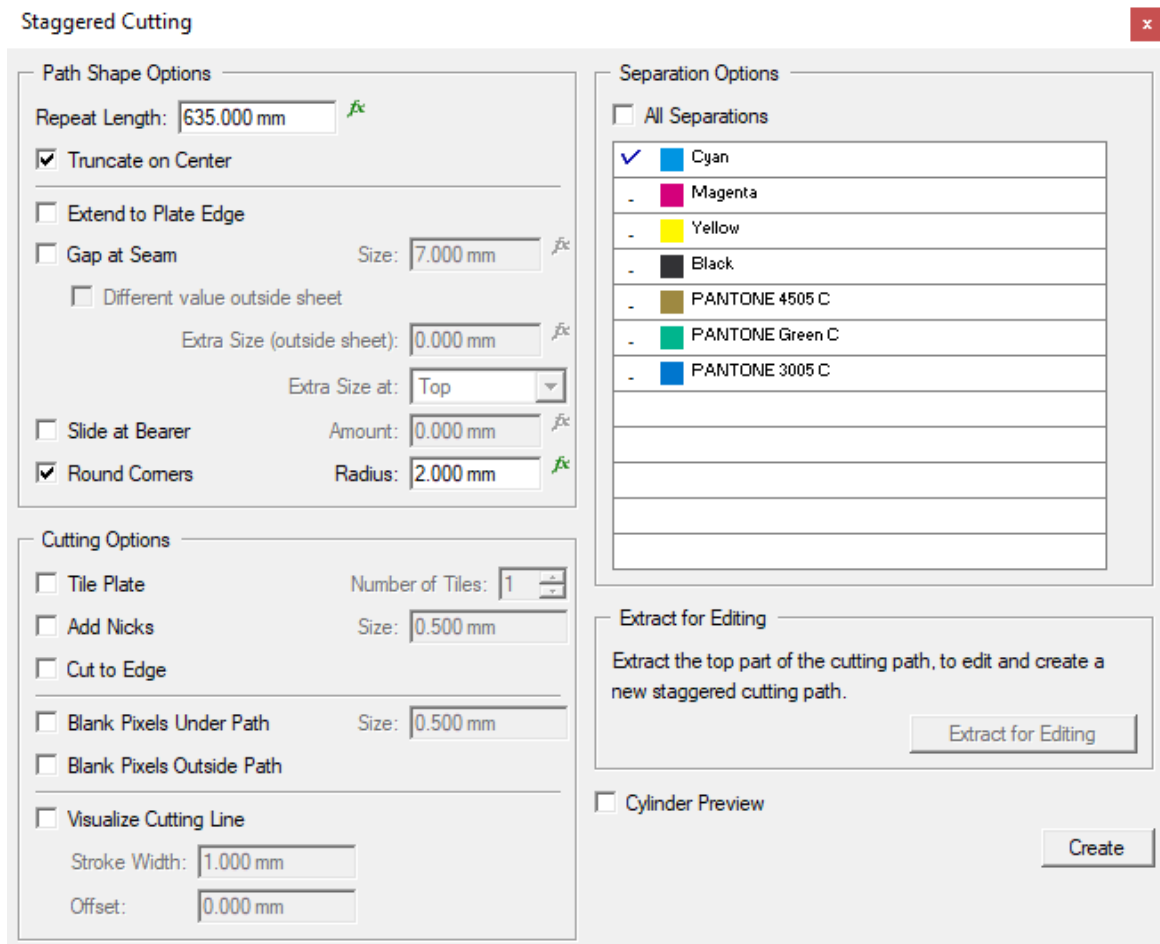
Note:

Although the extension of a SmartMark set is `.GRS`, sets can both be used in PLA files as in PDFPLA documents.

For additional information on how to work with SmartMarks, please refer to the chapter *'Using SmartMarks'*.

10.38. Staggered Cutting...

For more information on the concept of Staggered Cutting and how to use it, see [Staggered Cutting](#) on page 165



Path Shape Options define how the shape of the cutting path will be generated from the red selected contour.

- *Repeat Length* specifies the circumference of the cylinder. In normal situations, this equals the height of the sheet (which is also the default value). Note that formulas can be used. Basically, the cutting path will contain two identical horizontal segments (generated by taking the red selected input contour) which will create the seam. The distance between these two segments is the 'Repeat Length'. These 2 segments are connected through vertical segments which correspond with the plate edges.
- *Truncate on Center* will modify the shape of the red selected object so it is pulled to the nearest centerline between the die shape of the stations. When calculating the center line, stations are virtually copied over the repeat length. Stations that are heigher than the repeat length are ignored (typically these stations contain an overlay layer with marks).

If there are any other objects that should be taken into account when calculating the centerline (ex bearer bars), select these in green.

- *Extend to Plate Edge*: will extend the selected path to the width of the Plato plate (default this option is on). If this option is deselected, the left and right point from the red selected contour are important – these will determine the width of the staggered cutting path..
- Specify a *Gap at Seam* when it is wishful to make the flexo plate a bit smaller than the repeat length. The distance between the two identical segments will be the Repeat Length minus the Gap.
- If the *Different value outside sheet* option is enabled, the Gap at Seam is not applied outside of the sheet. This can be used e.g. to apply no gap in between bearer bars. By using a positive *Extra Size (outside sheet)* value, you can make the flexo plate a bit larger outside of the sheet, e.g. to make bearer bars overlap, so these can be cut with a knife when mounting. When setting an *Extra Size*, you can choose to add this extra size at the top or the bottom.
- If you enable *Slide at Bearer*, the staggered cut module will recognize bearer bars (elements the same height of the sheet, outside of the sheet) and cut through these by a diagonal line instead of a horizontal line. This is to avoid a hard gap, that could result in bouncing effects on the press. Use **Amount** to define how much the cut is slanted.
- *Round Corners* will modify the red input contour so that corners are rounded to the specified radius. Rounded corners facilitate cutting and damaging the plate when taking it off the cylinder.

Separation Options allows to define if the cutting path is to be used for every separation or for the selected separation(s) only. If an existing staggered cutting path is selected, the Separation Options are updated to reflect in which separations this path is active.

Cutting Options specify how the staggered cutting path will be cut on the Kongsberg table.

- *Tile Plate* will generate a number of additional vertical cutting lines. The horizontal positions where these cut lines are created is defined by vertical markup lines. These markup lines can be adjusted manually or numerically. The number of vertical cutting lines is defined by the selected number of Tiles.
- *Add Nicks* leaves small interruptions (nicks) in the cutting path to make sure the flexo material remains connected (and is better kept by the vacuum on the cutting table). The nicks are added on the 4 corners of the cutting path.
- *Cut to Edge* will split up the cutting path so that the 2 horizontal segments are cut first, followed by the vertical segments. Each vertical segment is cut in 2 parts, each time starting from the center going to the corner.
- *Blank Pixels Under Path* will generate a knockout stroke on the staggered cutting line, to make sure there is no imaged area under the staggered cutting line. Since cutting can slightly increase the thickness of the plate around the edges, cutting through imaged areas could cause problems on the press.
- *Blank Pixels Outside Path* will generate a knockout area which covers all objects that lay outside the cutting path. This object can be used to knockout excessive bleed.
- *Visualize cutting line* will generate a small stroked line indicating the cutting path. This can provide a visual indication for an operator to manually cut the plate. The width of the stroke and the offset from the actual cutting path can be set.

Extract for Editing creates a regular path that can be edited, in order to modify the staggered cutting line. See [Changing a staggered cutting line](#) on page 167

The *Create* button becomes active when there is one contour selected in red (the input contour). It will create a cutting path in a dedicated layer (named 'Staggered Path').

If an existing cutting path was Selected, the button changes to *Modify*. In that case, the existing cutting path will be replaced by the new one as there can only be 1 cutting path at a time.

See [Staggered Cutting](#) for a workflow example.

If the *Cylinder preview* option is selected, an extra row of graphics above and below the layout is shown, simulating the graphics from the previous or next cylinder impression. Note that this is only a visual aid for drawing cutting lines. When cutting through graphics above or below the layout, you should use the **Continuous** option in the grid dialog.

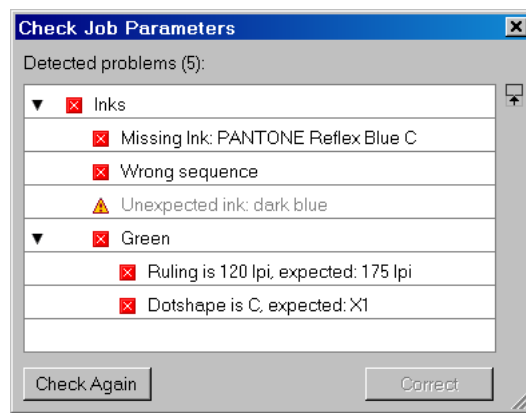
10.39. Check Job Parameters

In Esko Automation Engine, 'Check Job Parameters' allows you to check your files for inconsistencies regarding Job parameters (type of bar code, inks, certain RIP parameters,...) that are predefined for a Automation Engine Job and allows you to correct them (if necessary). Esko Plato only checks the ink job parameters.

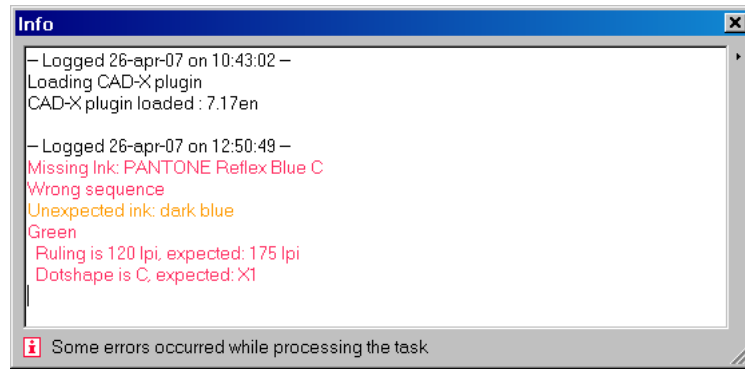
Note:

'Check Job Parameters' will only be active if your Esko Plato station is connected to an Esko Automation Engine server.

The Check Job Parameters function lists all conflicts. For example:



Job Parameters are automatically checked when opening or saving a file. Information regarding Job Parameters is also displayed in the Info window.



10.40. Workspace

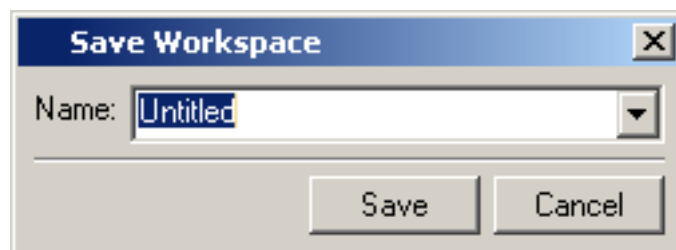
It is possible to save, for each opened dialog box and palette, its position, size and some dialog-specific settings like e.g.the width and visibility of the columns in the Grid dialog box. This kind of information is saved in a 'workspace'.

- [Default]: switches back to the factory default.
- [Save](#)
- [Manage](#)

10.40.1. Save Workspace

Save a workspace and use [Manage](#) to rename or delete previously saved workspaces.Saving stores the positions of all open palettes and dialog boxes when exiting the application.

After starting up Esko Plato, all user interface dialog boxes will take the position they had when exiting.

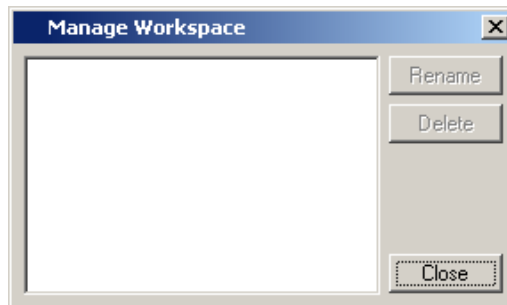


Tip:

All toolbars on the Window menu can either be used as docked tools or as proper dialog boxes.

10.40.2. Manage Workspace

The Workspace Layout Manager allows you to make some predefined workspaces of the user interface dialog boxes and to switch from one to another with a single click. The positions of all open palettes and dialog boxes are saved when exiting the application. You can start with default palette positions or restore default positions. You can also save multiple layouts as different workspaces.



10.41. Toolbars

All toolbars can easily be shown or hidden easily via the toolbars list on the Window menu:

- [Crosshair Position](#)
- [Preview Selector](#)
- File & Edit
- [Layout Type](#)
- [Station Selector](#)
- [Status Bar](#)
- [Tools](#)
- [View Selector](#)

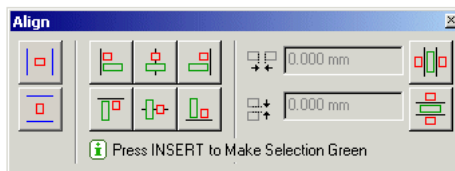
Tip:

All toolbars on the Window menu can either be used as docked tools or as proper dialog boxes.

For a detailed overview of the various tools and toolbars, please refer to the chapter '[Toolbars](#)'.

10.42. Align

Click 'Align' on the Window menu to open the Align dialog box.



Align to Sheet Align by Attribute Align Edge to Edge

Align to Sheet - The Align to Sheet section aligns the center of a red selection to the center of the sheet. These options do not use green selection.



Center horizontally on Sheet - Aligns the center of the red selection to the center of the sheet from top to bottom.



Center vertically on Sheet - Aligns the center of the red selection to the center of the sheet from left to right.

Align by Attribute - The Align by Attribute section aligns the attribute of the red selection to the same attribute of the green selection. The attributes that can be aligned are: left, right, top and bottom edges, and the center (vertical alignment) and middle (horizontal alignment) of the objects.



Align Left - Aligns the left edge of the red selection to the left edge of the green selection. Movement is horizontal only; vertical position of the red selection remains unchanged.



Align Center - Aligns the center of the red selection to the center of the green selection. Movement is horizontal only; vertical position of the red selection remains unchanged. Align Right - Aligns the right edge of the red selection to the right edge of the green selection. Movement is horizontal only; vertical position of the red selection remains unchanged.



Align Top - Aligns the top of the red selection to the top of the green selection. Movement is vertical only; horizontal position of the red selection remains unchanged.



Align Middle - Aligns the middle of the red selection to the middle of the green selection. Movement is vertical only; horizontal position of the red selection remains unchanged



Align Bottom - Aligns the bottom of the red selection to the bottom of the green selection. Movement is vertical only; horizontal position of the red selection remains unchanged.

Align Edge to Edge - The Align Edge to Edge section is used to align the nearest opposite edge of the red selection to the green selection. For example, the Align Left to Right button on the top aligns the left edge of the red selection to the right edge of the green selection. The Align Edge to Edge commands can also be used with a Gap. Entering a gap, aligns the opposing edges with the specified gap in between them.



Align Left-to-Right - Aligns the nearest opposite left or right edge of the red selection to the right or left edge of the green selection. The opposing edges are aligned leaving the Gap amount specified between them. If the Gap amount is zero, the opposing edges abut. Movement is horizontal only; vertical position of the red selection remains unchanged.

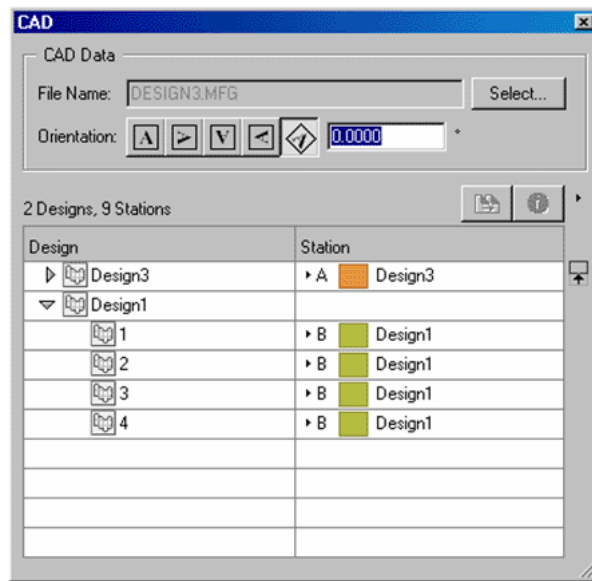


Align Top-to-Bottom - Aligns the nearest opposite top or bottom edge of the red selection to the bottom or top edge of the green selection. The opposing edges are aligned leaving the Gap amount specified between them. If the Gap amount is zero, the opposing edges abut. Movement is vertical only; horizontal position of the red selection remains unchanged.

For all information on how to use the Align tool, please refer to the chapter ['Using the Align Window'](#).

10.43. CAD


The CAD dialog centralizes all functionality needed in a CAD-based workflow.

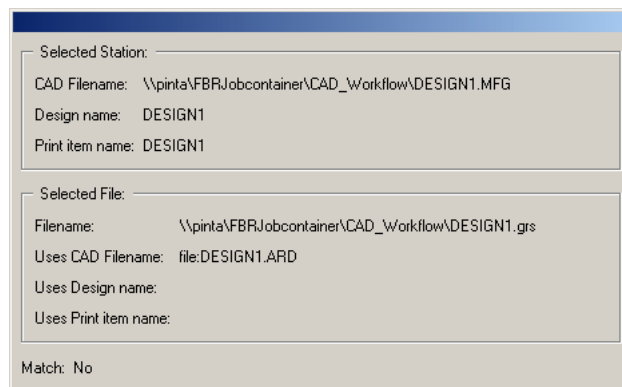



The CAD dialog box is built up of a number of **columns**: Design and Station.

If CAD data is loaded the name of the CAD file will be indicated on the first line of the CAD dialog. Click the **'Select'** button to import or replace the CAD layout. The [Import CAD Data dialog box](#) will pop up. If CAD data is loaded, this will replace the current CAD data.

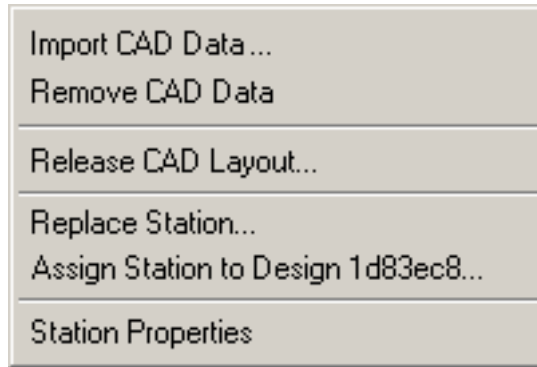
Using the Orientation buttons or by entering an angle, you can define the rotation angle of the CAD layout.

Click the 'Assign/Replace Stations' icon  to replace the selected station(s). The Replace Station dialog box will pop up, together with an extra dialog box that indicates whether or not the selected station matches the selected design.

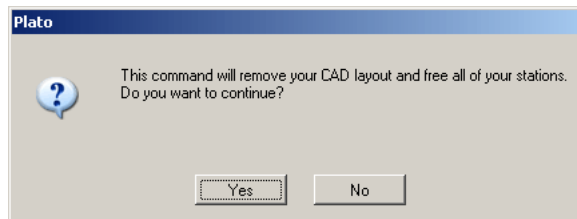


Click the **'Info'** button  to view or adjust the properties of the selected station. For more information about the options in the Properties dialog box, please refer to the chapter ['Station Properties'](#).

Click the arrow in the top-right corner to select one of the following items from the fly-out menu.



- 'Import CAD Data...' has the same function as [Import CAD Data](#) on the File menu.
- 'Remove CAD Data' removes that CAD layout and all associated stations.
- 'Release CAD Layout' removes the CAD layout but keeps all associated stations. After releasing the CAD layout, the stations no longer are tied up together and can be moved individually.

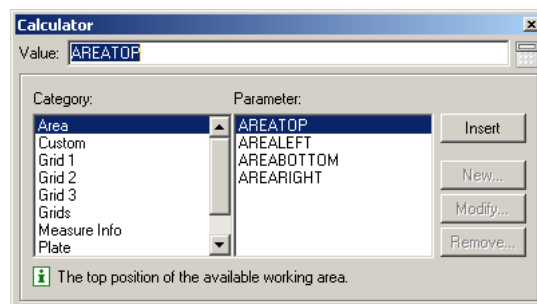


- Replace Station... opens the [Replace Station dialog box](#).
- Assign Station to Design... opens the [Assign Station dialog box](#).
- Station Properties... opens the [Station Properties dialog box](#).

For more information on creating a CAD workflow, please refer to the chapter '[CAD workflow](#)'.

10.44. Calculator

The Calculator allows you to create, modify or remove parameters.



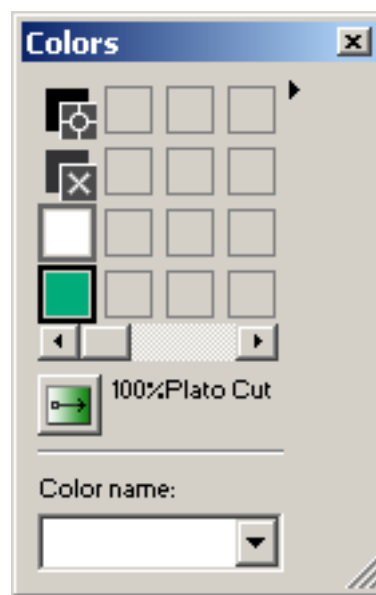
For more information on formulas and working with the calculator, please refer to the chapter '[Calculator and Parameter Formulas](#)'.

10.45. Colors

In the Colors dialog box you define, add or modify colors and vignette colors to be applied to objects.

Note:

The Colors dialog box can only be used in combination with the Geometry Mark tool.



Color swatchesTo define, add or modify colors or vignette colors to be applied to objects. When you click a color swatch to add a color, it is automatically filled with an empty color (white).

Note:

You can also resize the Style Bar horizontally, to display more color swatches.

Color Name input boxTo define a “work” name for the selected color swatch in the Color Palette, to help you to identify the color for a category of objects.

You created a drawing with a blue color for the sky and a blue color for the ocean. For the first blue color, you can enter the name “sky” and for the second blue color, you can enter the name “Ocean”.

When you select the name, the color with that name becomes selected in the Color Palette.



Caution:

The name you enter in the Color Name input box is job related and is only a tool to quickly retrieve colors. It is NOT a designer ink that you create here.

There are 2 special entries in the color palette:

- registration color
- darkest color

Vignette

To change the selected color from a plain color to a vignette color or vice versa. The vignette options display in the Color factory. For more information about the vignette options, we refer you to the [Vignette Color Factory](#). The highlighted button indicates that a vignette color is active for the selected color swatch in the Color Palette.



Caution:

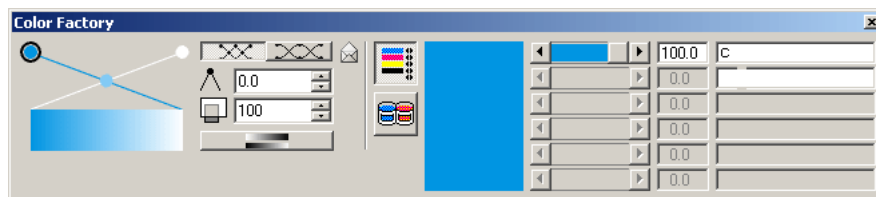
White, which is the empty color that automatically appears in the color swatches when you click them, is also a color. When making a vignette from a color to the empty color white, be aware that white is 100% opaque.

10.46. Color Factory

Use the Color Factory to display or modify the selected color.

Open the Color Factory by:

- Double-clicking a color swatch in the Color palette,
- activating Vignette in the Color palette.



You can display or modify the selected color in the Color Factory.



Color Factory

You can modify the percentages of inks with the slider or the spin boxes. You can add an ink by entering the initial of the ink name in the input box, next to the ink value input box. This adds a new ink (if still non-existing) to the Inks dialog box.

2.



Ink Factory

To display or modify the inks that will be used to print a color when a job is printed or processed.


Vignette Color Factory To create color vignettes.

When you select an (empty) color swatch in the Color Palette and then click the vignette button, the Vignette Color Factory appears. The Vignette Color Factory is an extension of the Color Factory.

Vignette Points and Curve To determine how the color will be averaged between the colors assigned to the points on both sides.

- The vignette points are used to determine the color for the vignette, and the vignette curve is used to determine how the colors (between two or more points) will be graded.
- Two curves (and their corresponding points) are displayed by default for the vignette. You may add curves and points by clicking the intersecting points and dragging the points to the top of the vignette curve, or by dragging and dropping additional colors to the intersecting points.
- Delete colors from the vignette by clicking the point that represents the color you want to delete and dragging it downward, outside of the Vignette dialog box


Vignette Envelope The envelope in the vignette Color Factory contains the Color Factory or the Ink Factory.

Smooth Vignette Button  Essentially, the smooth vignette button activates an automatic mechanism that determines the course of the colors in such a way that banding is no longer visible in the vignette.

Position of Vignette Color 

Enter a numeric value in the input box to change the position of the vignette color.

The default is 50 for intersecting points and 100 for starting and ending points. The intersecting point remains between 13 and 87.

The icon  before the input box will change, depending on the vignette point you clicked on, into an intersecting point or a beginning or ending point.

Opacity 

Determine the opacity of the selected vignette color by entering a value in the input box.

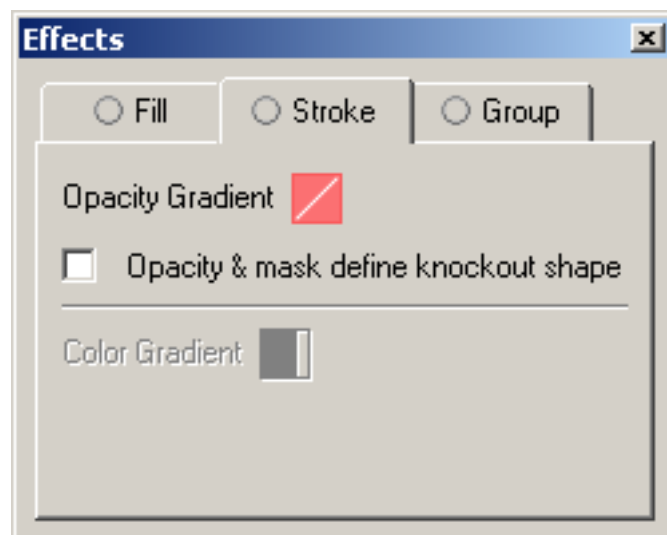
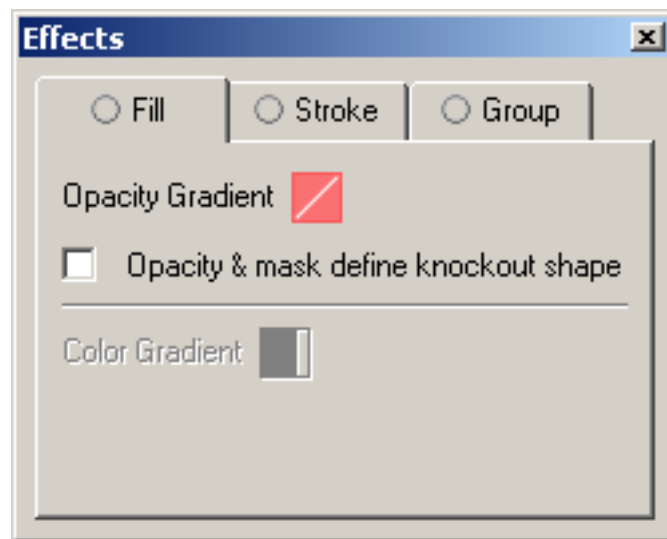
Vignette Direction 

Click this button to indicate the starting and ending point for the vignette in the job. The point you click first is the point where the vignette starts. The second point you click is the point where the vignette ends. The points can be selected and replaced. If you replace the points within the vignette the color for this vignette will also be adapted. Consequently also, *the color swatch* in the Color Palette, displaying the vignette color, will be adapted.

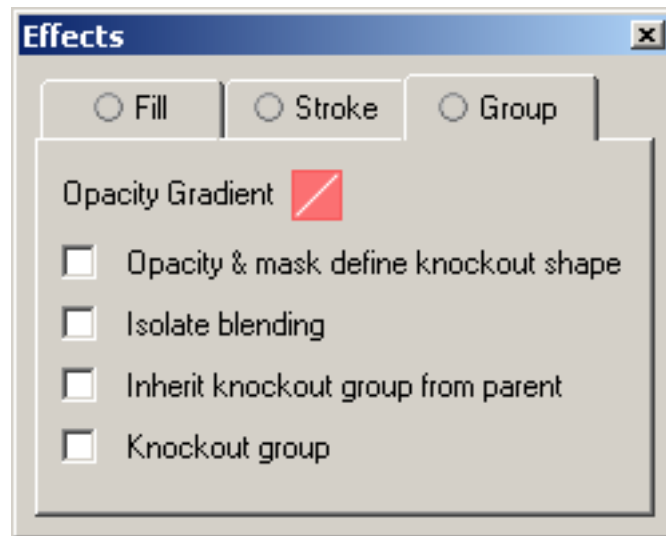
10.47. Effects

The Effects dialog box can appear when setting the *style* of a geometry mark and allows you to add more advanced transparency settings. In the Fill and Stroke tab entry you can apply transparency settings on the fill and stroke attribute of the selected objects from the document, in the Group tab entry (when a group or combi-object is targeted) you can apply transparency effects to the object or group it self.

Fill and Stroke tab entry contain the same options.



The Object/Group tab contains specific transparency options for PDF Transparency groups.



Opacity & Mask Define Knockout Shape checkbox keeps a knockout effect proportional to the masking object's opacity. The knockout effect will be strong in areas of the mask that are close to 100%; the knockout effect will be weaker in areas with less opacity.

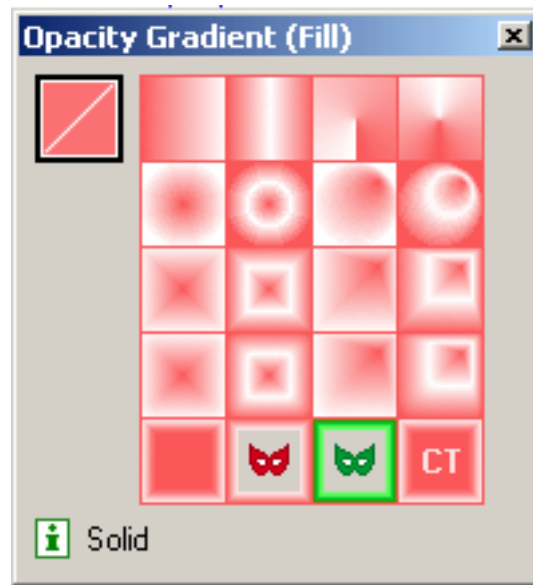
The **Isolate Blending** checkbox prevents blending modes applied to objects within a group from being applied to objects beneath the group. This option is only useful when used on groups that contain at least one object with a blend mode other than Normal applied to them or when used on individual objects that have a blending mode other than Normal applied on the stroke and fill attribute of the object.

Check the **Inherit Knockout Group from Parent** checkbox if you want to group objects without interfering with the knockout behavior determined by the enclosed layer or group. Clear the option and check the **Knockout Group** checkbox to prevent transparency from revealing underlying objects within the group. To make sure that a layer or group of transparency objects will never knock each other out, clear both the 'Inherit Knockout Group from Parent' and 'Knockout Group' checkboxes.

10.47.1. Changing the Opacity Gradient

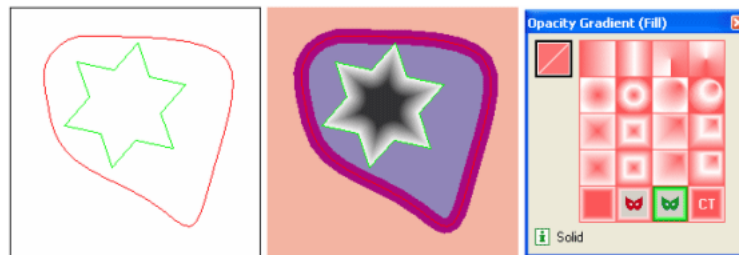
The Opacity Gradient offers a wide range of opacity gradients to be applied to the object, fill and/or stroke attribute of an object. The Opacity value lets you vary the transparency with a constant value, but the Opacity Gradient tool allows you to set varying levels of transparency.

The Opacity Gradient dialog still contains the alpha modes that were available in PackEdge 5.0 and earlier versions, but make them clearly visualized by a small thumbnail preview: linear, linear updown, conical, conical updown, elliptical, elliptical updown, elliptical eccentric, elliptical updown eccentric, rectangle, rectangle updown, rectangle eccentric, rectangle updown eccentric, parallelogramic, parallelogramic updown, parallelogramic eccentric, parallelogramic updown eccentric, edge, softmask and CT mask.

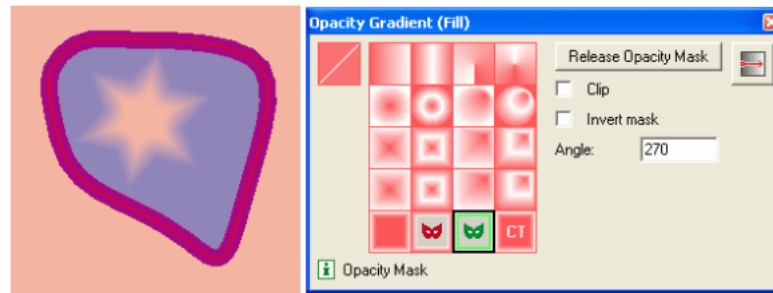


10.47.2. Creating and Editing PDF Opacity Masks

The Opacity Gradient dialog allows you to create and edit PDF Opacity Masks in PackEdge 6.0. Any object in a document, even linked files, can be used as a mask for another object. The mask overlays the selected object, covering entirely where the mask is solid or partially showing through based upon the shades of colors on the mask. The level of opacity is based on the difference of the luminance from the colors of the mask and the object: white is 100% opaque, black is 100% transparent and grays (colors change to grayscale on a mask) have varying levels.



Create a PDF Opacity Mask by double-clicking on the Opacity Gradient icon in the Effects dialog to open the Opacity Gradient dialog. Select the opacity mask into a green selection and the object on which the opacity mask has to be applied into the red selection. Press the Opacity Mask button (👤) to apply the green selected object as an opacity mask to the targeted object(s). When the green selection contains more than one object (a group with all individual objects selected is considered as one object), the objects of the green selection will be grouped before to be applied as an opacity mask.



The Clip checkbox makes an opacity mask double as a clipping mask, which caused the masked objects to be invisible outside the boundaries of the masking object.

The Invert Mask checkbox reverse the luminosity values of the masking object, which reverse the opacity of the masked object. For example, areas that are 15% transparent become 85% transparent after inversion.

By using the Transform Opacity Gradient tool you are able to transform the mask object. The mask object can be moved to another position relative of the masked object, scaled and transformed. Press a tool from the toolbar dialog to stop modifying the Opacity Mask transformation (or press the spacebar to make the selection tool active).

Release the Opacity Mask from an object by selecting the object mask object into the red selection and clicking on the Release Opacity Mask button in the Opacity Gradient dialog. The Opacity Mask object will be put into the green selection.

Edit the Opacity Mask by first releasing the Opacity Mask from the masked object. After you have modified the masked object, you have to recreate the Opacity Mask by clicking on the Opacity Mask button to apply the green selected object as an opacity mask to the red selected object(s).

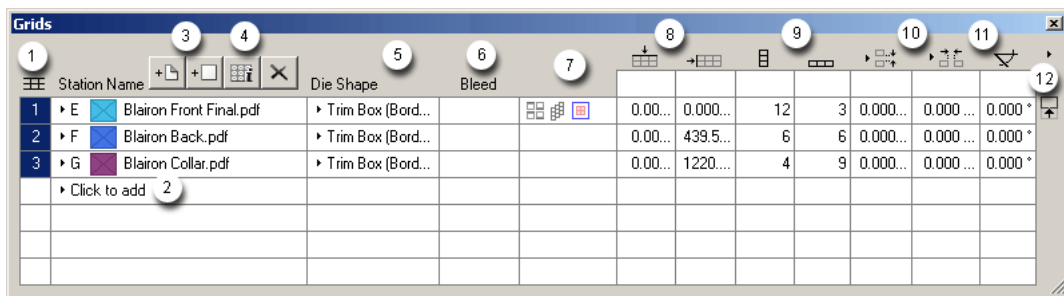
Note:

when an object having an opacity mask is transformed (moved, scaled, rotated, shear) in the document, you have the choice to determine how the opacity mask has to behave:

- Check the Scale Vignette toggle in the Transform tool options dialog to apply the same transformation on the mask object.
- Uncheck the Scale Vignette toggle to only transform the masked object. The mask object will not be transformed


10.48. Grids

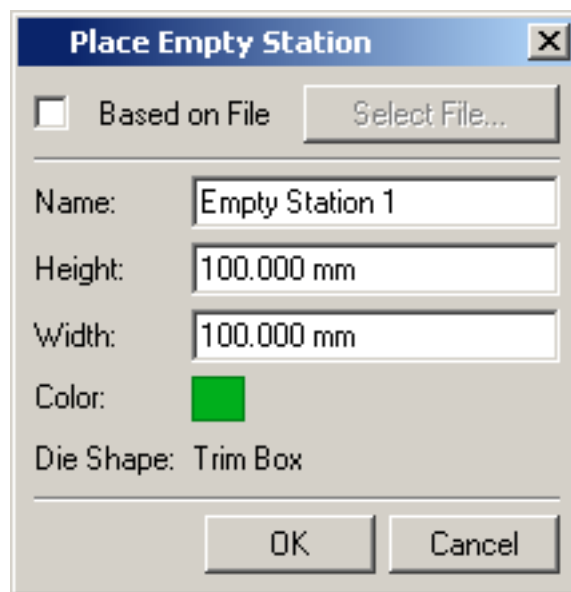
The Grids dialog box allows you to create step and repeat blocks set their numeric parameters.



1. Grid Number
2. Station Name
3. [Add Station / Add Empty Station](#)

+  Click the 'Place Station' icon to add stations to the sheet.

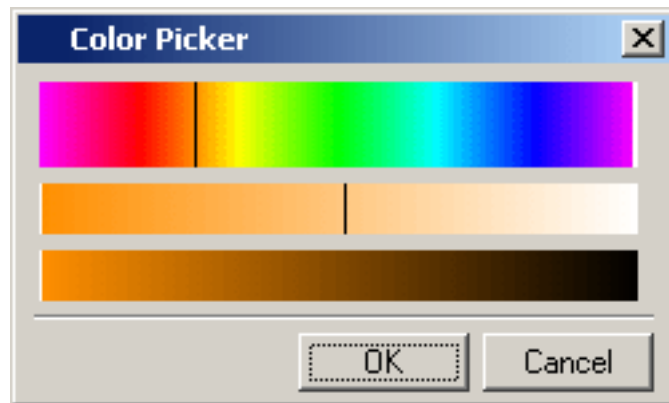
+  It is possible to place so-called empty stations by clicking the 'Place Empty Stations' icon. You would do so when you are designing a layout before having received the actual graphics. However, in that case, you would need the exact measurements of the future graphics, as Esko Plato will ask for them in the Place Empty Station dialog box:



Use the 'based on File' option together with the the 'Select File' button if you want to create an empty station based on an existing file. This offers 2 possibilities:

- a. select an existing graphics file (grs or normalized PDF). In this case, the size of the selected graphics will be used as the size of the empty station.
- b. Select a CAD file. In this case, the CAD data will be used to define the die shape of the empty station and the size of the station will match the bounding box of the CAD data.

You can change the standard color by clicking the color patch, which will make the [Color Picker](#) pop up. Simply click in the color area that you wish to use for your empty station.




✕ Click the 'Remove Grid' icon to delete the selected grid.

Tip:

Stations that were used previously in the current layout can be recovered by clicking the little arrow in the Station Name column.

4. You can view or adjust the *Grid Settings* in a number of ways:

- Click the Info button 
- Double-click the grid number.
- Select Grid Settings from the top-right fly-out menu.
- Select Grid Settings from the fly-out menu in front of the station.

5. **Die Shape**

6. **Bleed**

7. **Advanced Settings** indicate if stagger, geometry or distribute settings have been changed.

8. **Vertical / Horizontal Offset from Sheet** (formula-enabled)

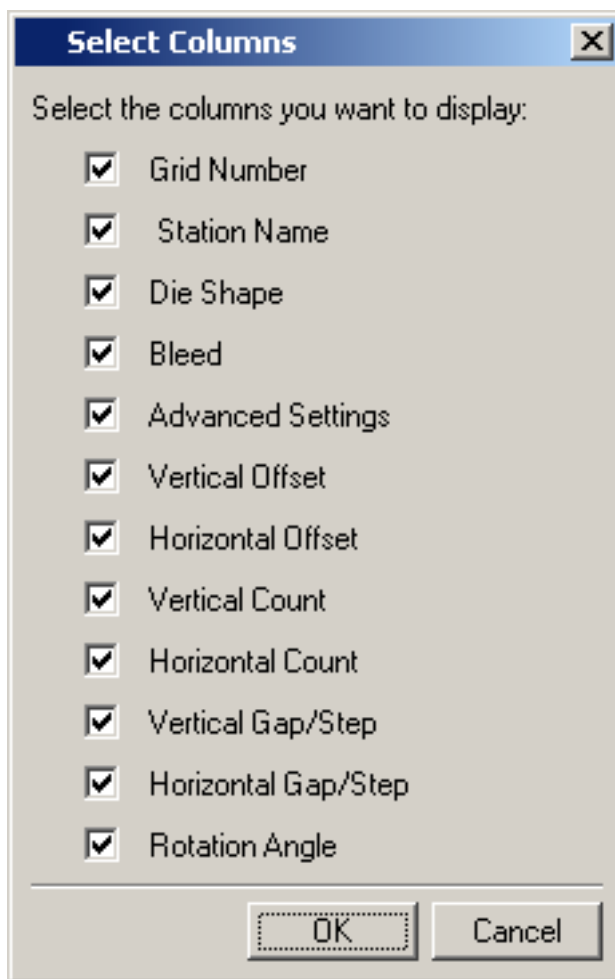
9. **Vertical / Horizontal Count** (formula-enabled)

10. **Vertical / Horizontal Gap / Step** (formula-enabled)

11. **Rotation Angle** (formula-enabled)

12. fly-out menu

The Grids dialog box is built up of a number of **columns**. To show or hide columns, click the arrow in the top-right corner and select 'Select Columns' from the fly-out menu.

**Tip:**

The width of the different column can be adjusted by dragging the vertical separators between them.

Tip:

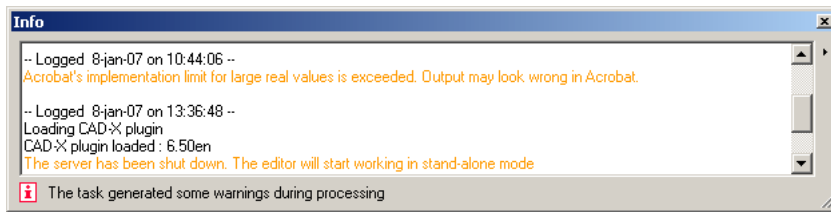
Stations that were previously used in the current layout can be recovered by right-clicking an empty cell in the 'Station Name' column. By repeating the previous steps, you can add, delete or modify the stations that you intend to use in the current layout.

For more information on creating a grid-based workflow, please refer to the chapter '[Grid-based workflow](#)'.

10.49. Info Window

The info window displays information that the editor generates about the job.

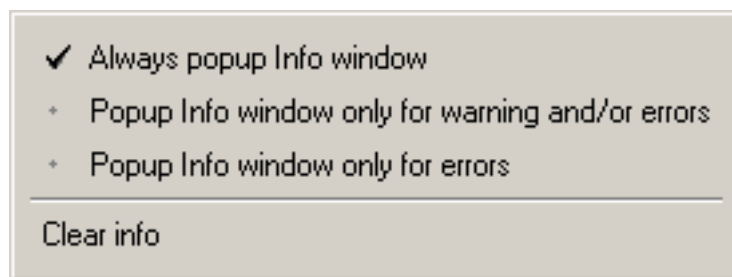
Warnings are displayed in orange, errors are red.



You can define the Info window's behavior. Click the black arrow and a the different possibilities pop up.

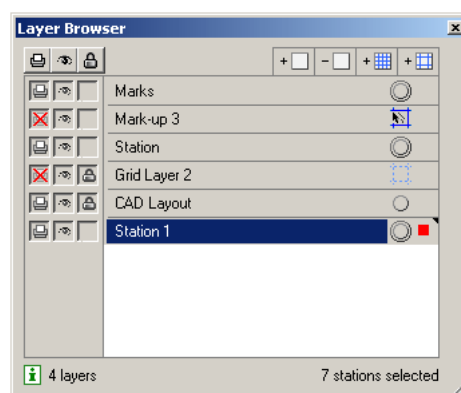
- Always pop up Info window
- Pop up info window only for errors/warnings
- Pop up info window only for errors.

Clear Info, clears all messages in the dialog box.





10.50. Layer Browser


You work with layers with the Layer Browser. Click 'Layer Browser' on the Window menu.






The features of the Layer Browser are detailed below.


Print  Layers can be set to be printable or non-printable. Click the Print icon next to each layer to toggle between the two choices. Clicking the icon at the top of the Print column toggles all the layers to printable.

Visibility  Layers can be displayed or hidden. An eye icon in the box indicates the layer is turned on. Click the box to turn the layer on or off. The eye button at the top of the column turns all layers off except the current layer. A layer that is not displayed cannot be edited and is locked.



Lock  Layers can be locked or unlocked. A padlock icon indicates the layer is locked and its contents may not be selected or changed. Click the box to lock or unlock the layer. The padlock button at the top of the column locks or unlocks all layers.


Add/Delete Layer +  -  Layers can be added or deleted from the document. From left to right, these icons are add a grid layer, add a markup grid layer, add an object layer and delete a layer. The delete layer button can be used to delete any layer in the list. Deleting a layer deletes the layer and all the objects on the layer.



+  Allows you to add a new Grid Layer.

+  Allows you to add a new Markup Layer.

Active Layer The Active Layer is the layer to which all new objects will be added. The current Active Layer is shown with a black triangle at the top-right corner of the layer. To make a layer the active layer, click its name in the layer list. There is a difference between a layer being selected and a layer being the active layer. For example, you can select a grid layer but it will not become the active layer because objects cannot be placed on grid layers.

  A little red square indicates the layer contains red selected objects. A big red square indicates that all objects of a layer are selected in red.

A double ring () indicates the object is targeted (or is the current active). The Style and Effects dialog boxes show the properties of the object that is targeted.

Snap On/Off   Snap can be turned on or off for grid and markup layers. Click the 'Snap On/Off' icon for the layer to toggle between the two choices. In the window shown above, objects will not snap to the layer named 'Markup Grid' but will snap to objects on the layers named 'Grid Layer'.

Note:

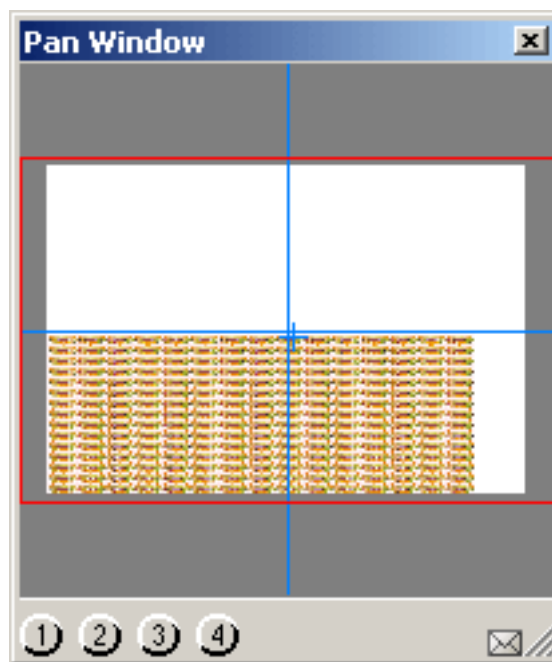
Clicking the object name in the Layer Browser does not select the object in the document, but selects the object in the list from the Layer Browser.

- The drag and drop functionality to move object to another position or into another group works on the objects selected in the list.
- Clicking the remove button will delete the objects selected in the list and not the objects that are selected in the document.

Setting up several grid layers and then choosing which ones to snap to is useful when creating a complex layout. See below for details about creating grid and markup grid layers.

10.51. Pan Window

Use the Pan Window to see an overview of your document and the current view. The document is displayed as a thumbnail with a rectangle indicating the current view.



You can program views that you intend to use more than once with this job. The job is represented as a thumbnail over which lies a representation of the viewport rectangle. If you select another job that is on the screen, the dialog box will be updated.

Viewport rectangle (selection of the thumbnail view) The viewport rectangle lies over the thumbnail view.

You can see the viewport rectangle in the image window and follow its movements in the thumbnail view. The viewport rectangle will be updated each time you make a change to your

window view, and conversely, if you move this small viewport rectangle over the Pan box thumbnail view, the view within your current window will be updated at the same time.

To move the viewport:

- Press the arrow keys on the keyboard.
- Press the buttons that appear in the envelope.
- Drag the viewport rectangle in the thumbnail view of the image.

If you click and hold the cursor over the thumbnail view, the viewport rectangle follows the point of the cursor until you release the mouse button and drop the viewport rectangle at the desired part.

- Or click the program buttons (if they have been programmed yet).

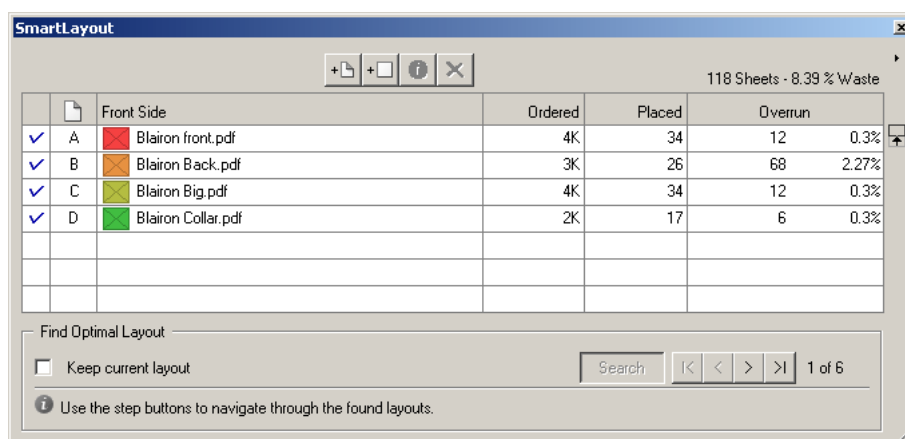
Program buttonsTo preset some view selections for each image on the screen. These settings will be saved, to reuse them during your current session. The settings are kept as long as the image for which they were programmed stays on the screen.

RepaintTo refresh the thumbnail view. If you moved the viewport rectangle in the thumbnail view and you click Repaint, the thumbnail view is resized until it contains the complete viewport rectangle and the image in total view. Use the **Repaint plug** to automatically repaint the thumbnail view each time your job changes.

Please refer to the *Pan tool chapter* for more information on how to use the tool.

10.52. SmartLayout (Optional)


Thanks to the SmartLayout option you can optimize the distribution of different labels on your sheet automatically and it takes but a few seconds. It uses different optimizing methods depending on your workflow, it offers dynamic feedback on suggested layouts and operator (inter)actions, e.g. run length, waste, overrun, ...




The SmartLayout dialog box is built up of a number of **columns**.

- The 'Check' column indicates whether or not the station is indicated in the SmartLayout search. If a station is ignored, it will not be taken into account when creating the optimal layout.
- Front Side short name
- Front side graphics
- Click the 'Ordered' edit field to enter number of stations. This number is expressed in thousands ('K').
- 'Placed' indicates the number of times this station is placed in the current layout. This number is read-only.
- 'Overrun' indicates the amount of overrun in actual numbers and in percentage of the ordered quantity.

You can show and hide columns by clicking the top-right arrow and selecting 'Select Columns...' from the fly-out menu.

Add stations by clicking the '**Add Station**' icon 

Add empty stations by clicking the '**Add Empty Station**' icon  Please refer to the chapter '[Adding a station](#)' for more information.

Click the '**Info**' button  to view or adjust the properties of the selected station. For more information about the options in the Properties dialog box, please refer to the chapter '[Station Properties](#)'.

Find Optimal Layout

Keep current layout: Keeps the current layout but replaces the stations from the layout by taking matching stations from the list of selected stations in the SmartLayout dialog. While replacing, the ordered quantities are taken into account to optimize the run length.

Click **Search**. The SmartLayout tool will order the selected stations on your sheet in the best possible way in terms of ordered quantity, overrun, waste... The amount of overrun is indicated in actual numbers and as a percentage of the ordered quantity. 'Search' creates a new layout, removing all stations that were placed on the layout before activating the function. Esko Plato will suggest a number of layouts. The one that is visualized is regarded as the most optimal one. Of course, you can at any time opt for another layout by using the step buttons.

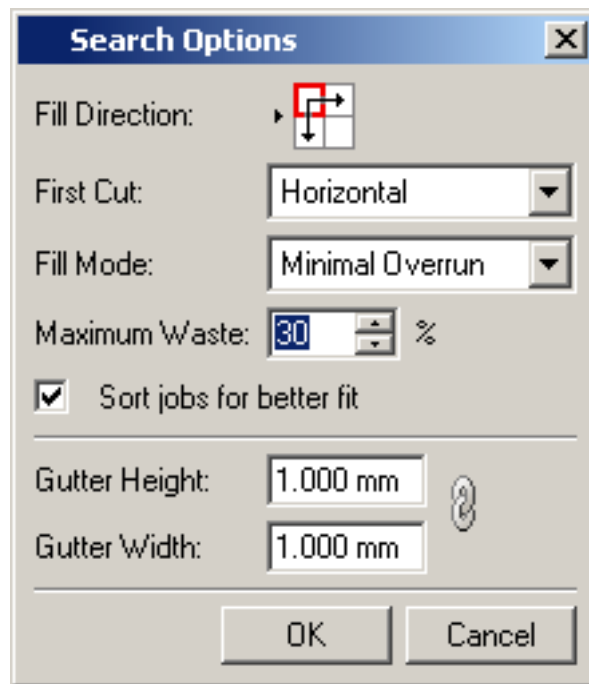
Click the '[Options](#)' button to further refine your layout.

Click '**Load All Products...**' from the fly-out menu to load all Products of the Job in which you are working.

For more information on creating a SmartLayout workflow, please refer to the chapter '[SmartLayout workflow](#)'.

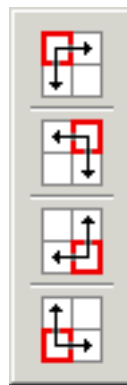
10.53. Search Options

The Search Options dialog box appears when you click the 'Options' button in the [SmartLayout dialog box](#).



Fill Settings

- Fill Direction: defines the fill direction. You basically choose the corner from which the SmartLayout starts the repetition.



- First Cut: define the direction of the first cut: horizontal and vertical
- Fill Mode: Choose a fill mode
 - Minimal Overrun: this option will try and place stations on a plate so that the requested number will be matched as closely as possible.
 - Minimal Waste: this option will try to keep the waste as low as possible. Waste is the combination of overrun and those parts of your paper you did not print on.
 - Strips Only: the fill will be as such that one strip will contain the repetition of only one Station
 - Complete Strips: this mode is the same as Strips Only, but the strips will be full. The Overrun parameter will be ignored.



- Maximum Waste: define the maximum allowed waste in percentages.
- Sort jobs for better fit: if this option is disabled, the order of stations from the SheetInspector will be replaced when placing stations on the sheet.
- Gutter Height and Gutter Width: define the gutter dimensions.


10.54. Stations


The Stations dialog gives an overview of all currently defined stations. It shows:

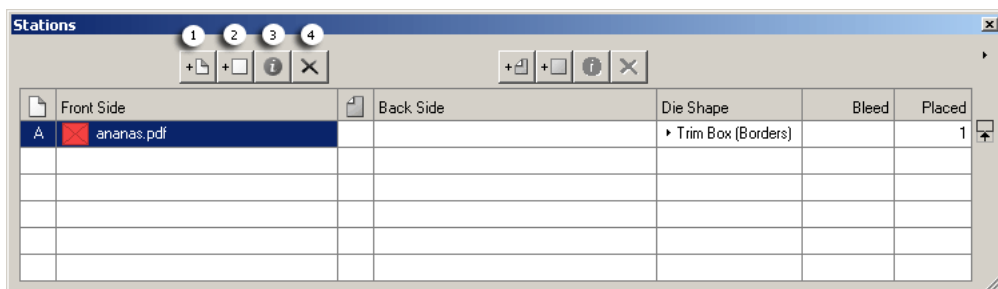
- the station's pseudo name
- the graphics used on the front side
- the graphics used on the back side
- the die shape. You can select a different die shape by clicking the arrow.
- the bleed mask. The bleed field is editable.
- the total number of copies placed on the layout
- Removing a station (only possible if station is not used on a layout)

You can show and hide columns by clicking the top-right arrow and selecting 'Select Columns...' from the fly-out menu.

1. Add stations by clicking the 'Add Station' icon  2. Add empty stations by clicking the 'Add Empty Station' icon  Please refer to the chapter ['Adding a station'](#) for more information.

3. Click the 'Info' button  to view or adjust the properties of the selected station. For more information about the options in the Properties dialog box, please refer to the chapter ['Station Properties'](#).

4. You can remove a station by clicking  You can only remove a station if the station has not been used on a layout.

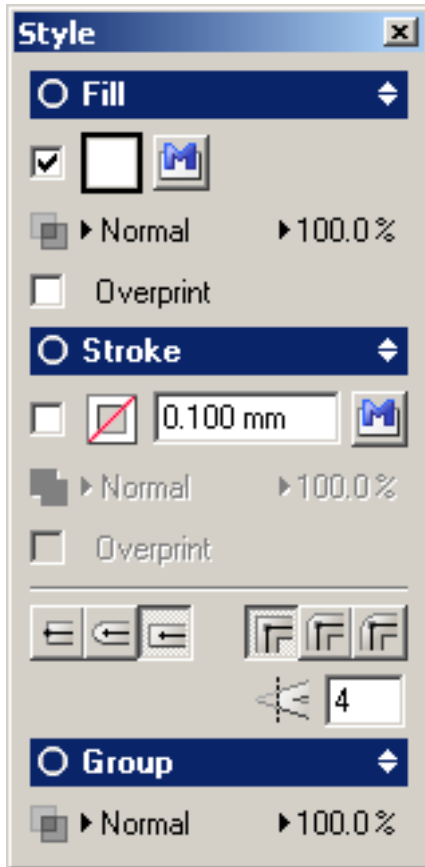


10.55. Style

The Style window allows you to set the parameters for the mark that you create. You can choose whether your mark has a Fill, a Stroke or both. You can choose the colors of the fill and

stroke as well as the line width for the stroke. You can also set colors in Geometry Marks to overprint other objects. Finally, you can choose for the color to be Registration color, which means that the color contains 100% of all inks in the document.

To set the Style of a Geometry Mark, use the Style dialog box:



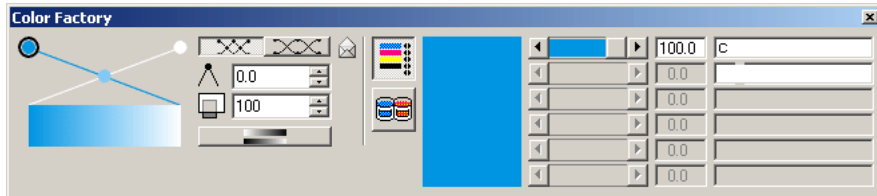
The Style dialog allows you to add a color for the fill and stroke and to apply some transparency effects to the Fill and Stroke attribute. Additional stroke options can be set when the Stroke group is expanded in the user interface. New in the Style dialog is the possibility to apply a blend mode or opacity to an object or group. With the latter you create the so called PDF Transparency groups. A PDF transparency group is a group of objects that have a transparency effect (blend mode or opacity). These PDF transparency groups are treated as a single object relative to underlying objects.

A shaded circle in the title bar of the groups indicates when a Fill, Stroke or Object has a transparency applied. Double click his icon to show the [Effects Dialog](#) in which you can apply more transparency to the object.

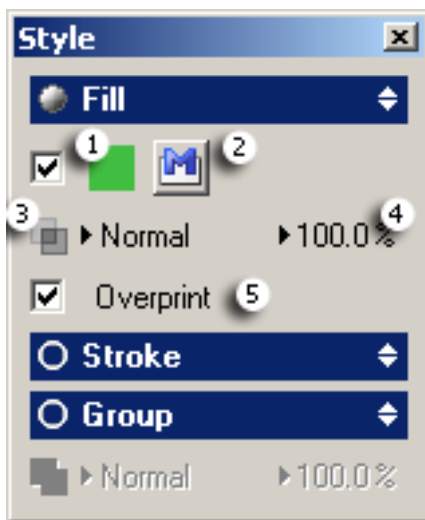
Specifying default settings for transparency features: if you change transparency settings when no objects are selected, the settings are applied to newly created objects.

Note:

When 2 or more objects in the document with a different value for the same attribute are selected, an unknown state is displayed in the user interface. In this example 2 objects with a different blend mode for the fill are selected in the document. Also the title bar shows an unknown state because one of the selected objects still has the default transparency settings (blend mode "Normal"), while the other object has a blend mode different from "Normal".



10.55.1. Fill



1. The **color swatch** shows the current color for the fill of the selected mark or one that you are about to create. The color displayed is based on the settings of the *Color Factory*. Check the box to activate the Fill attribute.
2. Click the **Clipping path** icon to knock out all data outside the borders of the path.
 A clipping path is a vector path which allows part of an image to show while hiding the rest (usually its background) effectively rendering part of the image transparent. It is a means to make parts of an image opaque and parts of an image transparent. Usually it is used to "knock out" the background. It is also a way of changing the rectangular-shaped boundary of a bitmap image into a shape of your choice.
3. Click the arrow to select a **blend mode** from the list.
4. Click the arrow to adjust the **opacity** using a slider.

5. Turn **overprint** on or off by using the overprint checkbox.

The shaded circle



indicates that certain attributes have a transparency applied.

Tip:

Clicking the arrows

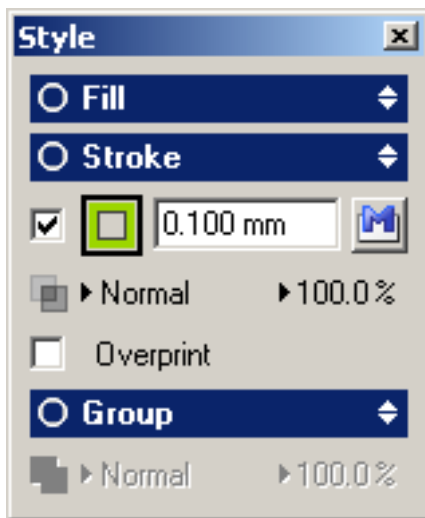


expands or contracts specific parts of the Style window.

Tip:

Clicking the Fill title bar opens the *Effects dialog box*.

10.55.2. Stroke



The **Stroke Color** box shows the current color for the stroke of the selected mark or one that you are about to create. The color displayed is based on the settings of the Ink Sliders.

The **Stroke Width** field shows the stroke width in the current units.

Click the arrow to adjust the **transparency** using a slider.

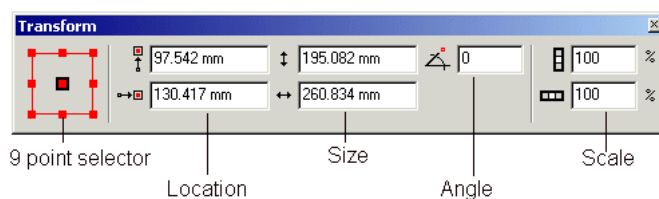
Turn **overprint** on or off by using the overprint checkbox.

10.55.3. Group

Adjust the **opacity**, click the arrow to adjust the **transparency** using a slider.

10.56. Transform

Click 'Transform' on the Window menu. The Transform Window appears:



9-point Selector - Use 9-point Selector to see information about a particular point in the current selection. The nine points that you can select are the four corners, the midpoints between the four corners and the center. The current point is highlighted with a black line around it; in the window above, the center point is the current point.

Location - The Location fields show the vertical and horizontal location of the current point. In the above window, changing the values in these fields will place the center of the selection at a new location.

Size - The Size fields show the height and width of the current selection. Changing values in these fields will scale the object so that it should have the height and/or width that you entered.

Angle - The Angle field shows the angle of the current selection.

Scale - The Scale fields show the vertical and horizontal scale of the current selection.

Transform Window Tips

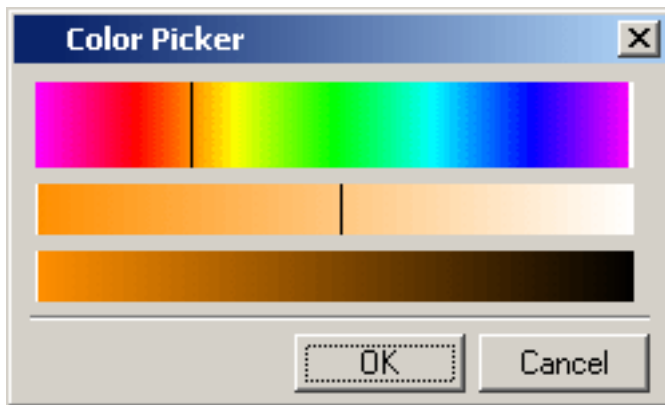
- Depending on the current selection, some of the fields may be disabled. See Transform tool for transformations that are allowed on Esko Plato objects. If no object is selected, all fields are disabled.
- The Location and Size fields display information about the die if either a CAD layout or a station are selected. For example, if a station is selected, the size in the Transform Window will display the location and size of the die, regardless of any bleed included in the station. The same is true for a CAD layout. This functionality makes it easy to accurately locate the first cut of the die at a specific location.
- If the current selection includes multiple objects, some that contain dies and some that do not, the location and size fields display values for the bounding box that encloses all the selected objects.
- SmartMarks can be moved, either interactively or numerically. However, the rotation (angle) of SmartMarks cannot be changed with either of these methods. However, the orientation

of most SmartMarks can be determined by means of their parameters. If you need to rotate a SmartMark, double-Click the mark and change its parameters in the corresponding dialog.


Please refer to the chapter *'Using the Transform Window'* for more information on how to use the Transform options.

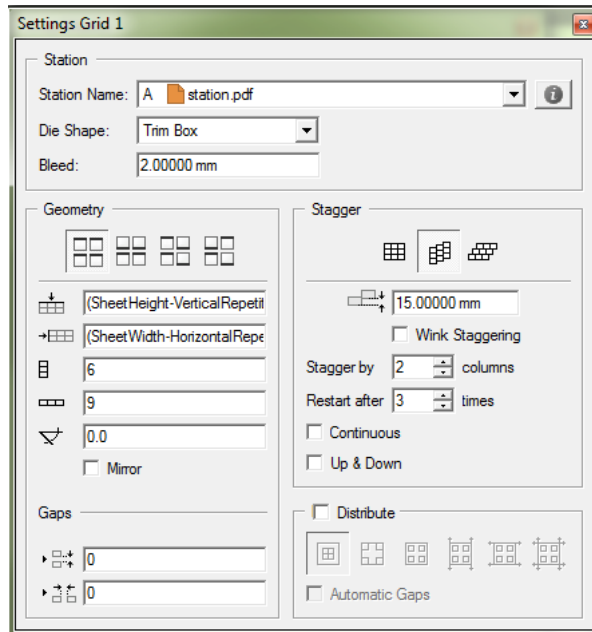
10.57. Color Picker

The Color Picker allows you to pick a color to represent a station in 'dummy color' mode.



10.58. Grid Settings

The Grid Settings dialog box offers all options to create your ideal step-and-repeat layout. You can enter numeric values, but you can also opt to work with formulas. Right-click an edit field and the *Calculator* will pop up. In the *Grids dialog box*, click the 'Grid Settings' icon  or double-click the grid number. The Grid Settings dialog box appears:



Geometry

The Geometry section allows you to specify settings regarding head turns, offset from sheet, number of repetitions, angle and gaps.



The default is a step with no head turns (180 degree rotations). The choices, from top to bottom, are: No Head Turn, Head Turn on Row, Head Turn on Column, Head Turn on Row and Column.



Enter the horizontal offset of a grid block relative to the sheet edge.



Enter the vertical offset of a grid block relative to the sheet edge.



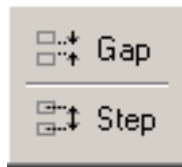
Set the number of repeated objects you want vertically and horizontally. Entering 1 in either field means that the object will not be repeated in that direction.



Enter a degree to specify the rotation angle of the station.

Mirror allows to mirror the station.

Gap Gap sets the parameters for stepping the current selection. You can step by gap or by step. Click the arrow to switch from step to gap and vice versa.



Vertical Gap

Horizontal Gap

The default is a step based on a gap. For Gap, the value specifies a gap between the stepped objects. If Gap is selected, Esko Plato determines how far to step the selection so that the requested gap is created between the objects. A gap of zero makes the stepped objects abut one another.

Vertical Step

Horizontal Step

For **Step**, the value specifies the distance between the centers of the stepped objects. If Step is selected, Esko Plato steps the selection by the distance entered into the 'Step Amount' field. The value that you enter must be a positive number that is at least the size of the object. This option is useful if you know exactly how far apart the objects need to be but do not want to figure out the step gap yourself.

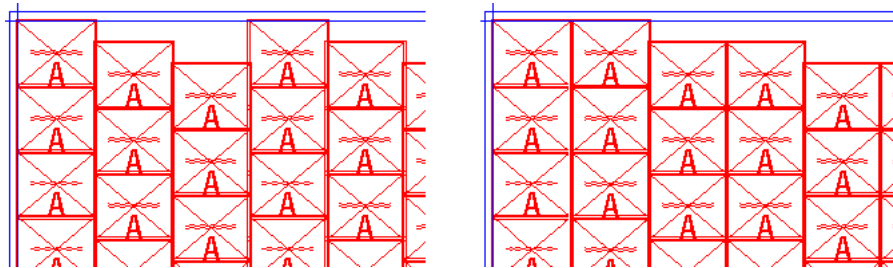
Stagger

Select the **Direction** in which you want to stagger the selected object. From left to right, the choices are: No Stagger, Stagger Columns, and Stagger Rows.

The **Stagger Distance** field indicates how much to stagger the object on each repeat. You can enter a value in the current units or you can enter a percentage of the object's size. In the dialog box shown above, the stagger distance is 50% of the height of the object. The amount to stagger that must be applied will be calculated on-the-fly. To use a value as a percentage, enter the percent sign (%) after the value.

Use the **Stagger by** setting to add the Stagger distance every n rows or columns. By default, this is set to 1, so that every row or column is offset from the previous.

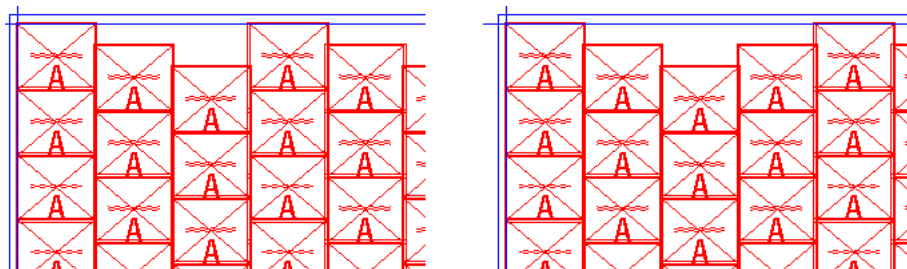
In the example underneath you see a vertical stagger, restarting after 3 columns, with the **Stagger by** set to 1 (left) and 2 (right)



Activating **Continuous** will add 1 extra station per row/column to fill the 'hole' created by making the indent.

The stagger will be repeated for a number of rows or columns as defined in **Restart after n columns**. After that, the next object will be at the begin position again. By setting the **Up and down** option, after reaching the number of columns, the next object will be staggered in the opposite direction.

In the example underneath you see a vertical stagger, restarting after 3 columns, with the **Up and down** option disabled (left) and enabled (right)



Distribute sets the method used to fill the sheet. The Distribute option offers you the possibility to fill a sheet in an automated way with a single station. Esko Plato calculates the number of stations that will fit on the sheet on the basis of the size of the station. The station size is determined by its die; objects outside the die are ignored. Stepping-and-Repeating via 'Distribute' is based on the sheet size. You can fill the sheet horizontally, vertically or in both directions. You can also use gap and stagger to create a seamless repeat.



Single Cuts - Fills the sheet with as many stations as will fit with no gaps between the dies of the stations. The filled repeat is centered both vertically and horizontally on the sheet. The extra space around the repeat is equally divided between the top and bottom and the left and right of the sheet. Gap and stagger cannot be used.



Distribute Gaps between Stations - Fills the sheet with as many stations as will fit with no gaps between the stations and the borders of the sheet, but with equal gaps between the stations themselves.



Distribute Gaps - Fills the sheet with as many stations as will fit with equal gaps between the dies of the station. The extra space is evenly divided between the stations; the dies of the outermost stations align with the edges of the sheet. Gap and Stagger cannot be used.



Continuous Around - Fills the sheet horizontally like Distribute Gaps but makes the around (vertical) repeat continuous. This creates a continuous and seamless repeat in the around direction. Gap can be set in the across direction; stagger can be set in the around direction.



Continuous Across - Fills the sheet vertically like Distribute Gaps but makes the across (horizontal) repeat continuous. This creates a continuous and seamless repeat in the across direction. Gap can be set in the around direction; stagger can be set in the across direction.



Continuous Around and Across - Fills the sheet so that the stations are continuous in both the across and around directions. This creates a completely continuous and seamless repeat in both directions. Gap cannot be used, stagger can be set in either the across or around direction.

Check **Automatic gaps** and Esko Plato will automatically determine the gap.

10.59. Station Properties

You can check the properties of the selected station by:

- Clicking the info button



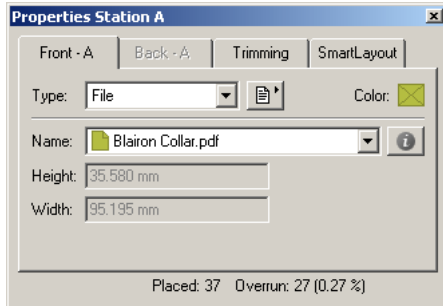
in the Grids/CAD/SmartLayout dialog box.

- Clicking right on the selected station and selecting 'Station Properties' from the popup menu.

The Station Properties dialog box provides you with some elementary information about the station in question.

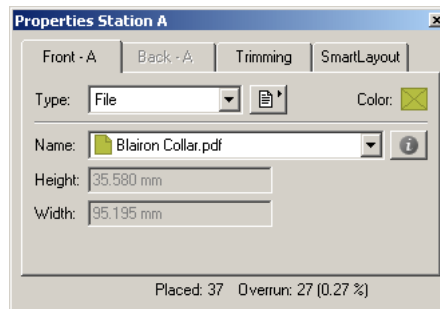
- [Front/Back](#)


- *Trimming*
- *SmartLayout*




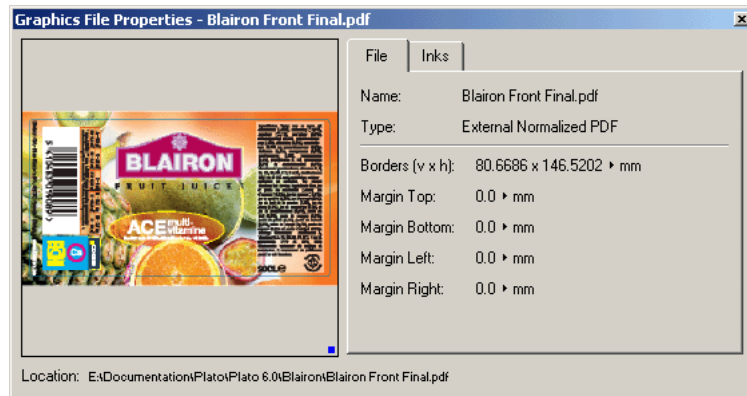
10.59.1. Front/Back

The Front and Back tab sheets provide information about the type of station (either a file or an empty station), its color, name, height, and width.



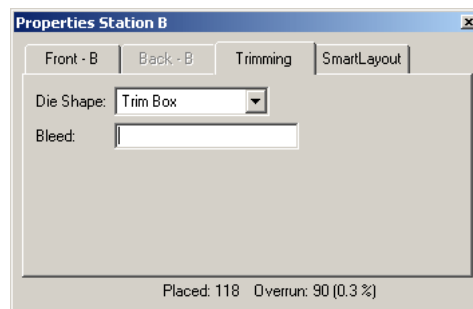
Click the 'Select Graphics' icon  to assign a different station to the design. The Assign Station to Design dialog box will appear.

Click the info icon  for more information about the file, transformations, and inks. The Image Properties dialog box will appear:



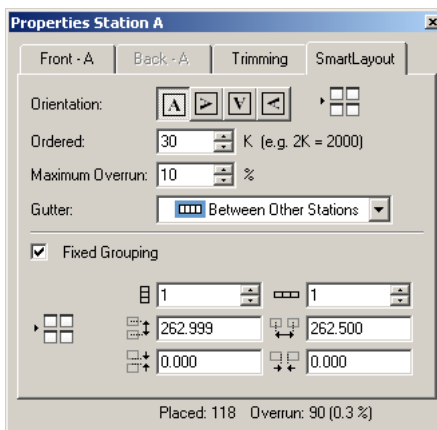
10.59.2. Trimming

In the Trimming tab sheet, you can determine the die shape that should be taken into account and the amount of bleed for the selected station.



10.59.3. SmartLayout

The SmartLayout tab sheet allows you to set the parameters regarding overrun, orders, maximum waste, ... This tab also allows you to determine orientation and gutter. These values will only be taken into account in the SmartLayout algorithm and not for instance when creating step-and-repeat grids.



Orientation: Indicate the orientation of the station.



Click to indicate how the stations should be placed in respect to one another. Possible options are:

- No Head Turn
- Head Turn on Row
- Head Turn on Column
- Head Turn on Row & Column

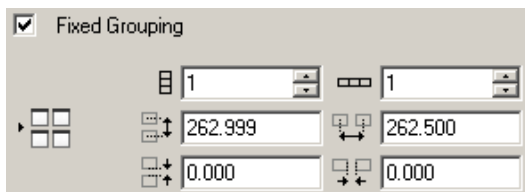
Ordered: enter the number of Labels you need (in thousands).

Maximum Overrun: enter the percentage of overrun (waste) you're willing to allow for.

Gutter distance can be entered in the 'Search Options' dialog box (SmartLayout dialog box > Options button).

- Always
- Between Other Stations (puts white between the selected station and every where it touches other stations).
- None
- Customized: You can create a customized gutter setting that inserts a gutter e.g. every second or third station.

Fixed grouping: Use the toggle to get extra settings:



If you activate the fixed grouping options, then every time SmartLayout places this station it will be placed in a group as indicated in this option (i.e. with the given number of copies in vertical/horizontal and with the given step distances). You can define the geometry of

the group, enter the amount of station in horizontal and vertical directions and specify the distance or gap between them.

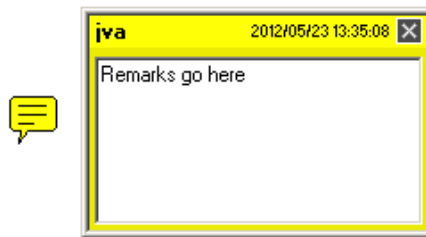
Note:

Rotation angle, head turn, grouping, etc ... are all parameters which are only used by the SmartLayout algorithm. These settings have no effect when a station is used in a grid-based workflow.

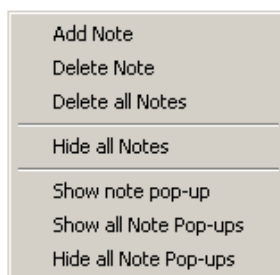
10.60. Notes

Using the Notes tool and Notes function in the Edit menu, you can add and edit notes in the Plato document.

Notes are saved in the Normalized PDF file as regular PDF Comments, and will show up as "Sticky notes" in Adobe Acrobat. Sticky notes created in Acrobat will also show up in Plato, while other types of comments (text corrections, highlights, ...), and note properties or statuses are disregarded when opening a PDF file in Plato.



A note consists of two parts: the **Note** itself, the yellow balloon indicating the location of the note, and the **Note Pop-up**, containing text, the name of the user that created it, and the time of creation.



- To create a note, select the **Note tool** and click in your document, or select **Edit > Notes > Add Note**, or select **Add note** from the contextual menu using the Note tool
- To move a note, click and drag the note pop-up
- To delete a note, select it and choose **Edit > Notes > Delete Note** or select **Delete Note** from the contextual menu using the Note tool.

- To delete all notes, choose **Edit > Notes > Delete all Notes** or select **Delete all Notes** from the contextual menu using the Note tool.
- To hide or show all notes, choose **Edit > Notes > Hide/Show all Notes** or select **Hide/Show all Notes** from the contextual menu using the Note tool.

Note: When using **Hide all Notes**, both the notes and the note pop-ups will be hidden. Only **Show all Notes** makes notes visible again. Using **Show all Note Pop-ups** will not have any affect.

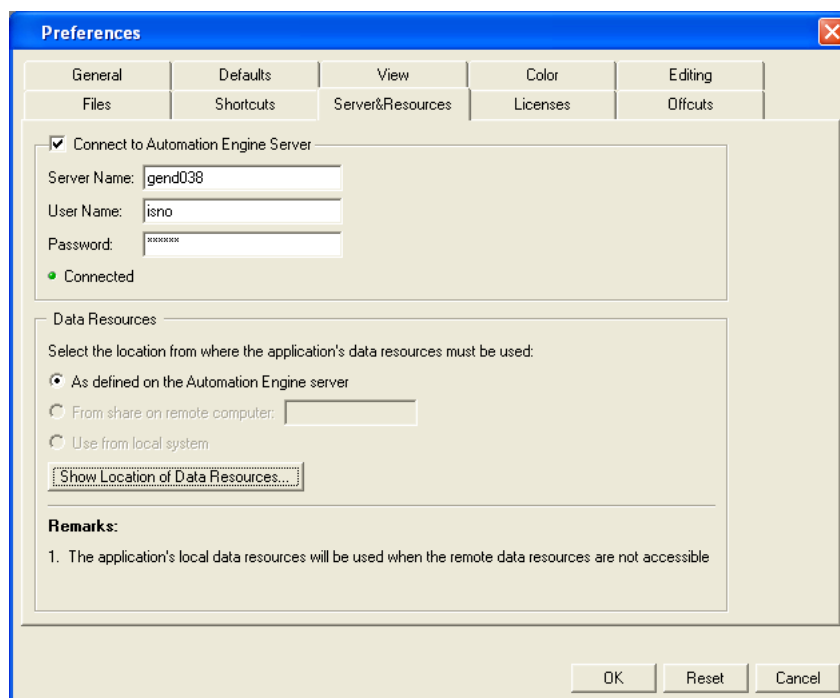
- To hide a note pop-up, click the X in the top right corner. This doesn't affect the note itself, it only hides the pop-up
- To hide all note pop-ups, select **Edit > Notes > Hide all Note Pop-ups** or select **Hide all Note Pop-ups** from the contextual menu using the Note tool
- To show all note pop-ups, select **Edit > Notes > Show all Note Pop-ups** or select **Show all Note Pop-ups** from the contextual menu using the Note tool

11. Server and Resources

To define the Automation Engine server to connect to, and the Data Resources to be used by the application as you work on a job.

Note:

In case the application is connected to an Automation Engine or BackStage server, the Data Resources are always used from this server. You can use the emote or local Data resources if the application is not connected to an Automation Engine or BackStage server.



Automation Engine Server

You can set the Server Name, User Name and Password to connect to Automation Engine.

The connection to an Automation Engine Server has an influence on the following features in the application:

- Submitting to Shuttle: you can only submit to the Automation Engine server the application is connected to
- The user name for the Track Edit Session task on the Automation Engine server
- The list of Jobs and Products in the File Selector dialogs
- The parameters for the 'Check Job Parameters' as available on the Automation Engine server

Note:

If you are connected to an Automation Engine, a green notification will be shown.

Data Resources

The common Data Resources used by the application are Color Engine Database, Marks, Fonts, Custom and DGC.

By default, these Data Resources are taken from the Automation Engine server. However, it is possible for 2 or more stand-alone applications (without an Automation Engine server) to share the same data resources. This requires that the Data Resources are shared on a remote computer.

Note:

Changing the Data Resource settings requires a restart of the application.

12. Appendix

- [QuickStep Compatibility](#)

12.1. QuickStep Compatibility

12.1.1. Opening QuickStep GRQ files

Esko Plato allows you to open QuickStep GRQ files which are converted into an Esko Plato compatible format. While special attention was paid to maximize the compatibility, this import conversion has some pitfalls, which are outlined in this section.

12.1.2. Conversion of GRQ data

The grid-based workflow of Esko Plato closely resembles the QuickStep approach of creating step-and-repeat layouts. When opening a GRQ file (or a Plato 5.0 file containing QuickStep objects) the following conversions happen:

1. QuickStep library objects (graphics) are converted into stations.
2. Plate and sheet settings are copied into the Esko Plato layout setup.
3. Formulas are preserved, including the user variables (if any).
4. QuickStep grids are converted into Esko Plato grids.
5. Registration marks are kept

1. Converting graphics The QuickStep graphics are converted into stations that preserve the die shape and bleed settings. In doing so, each QuickStep grid line is converted into an Esko Plato station. If the same graphics is used in multiple grid lines, these will result in multiple stations referring to the same graphics. When an external LC or GRS file is used as die shape, the contour data is read and embedded inside the Esko Plato station resulting in a 'manual' die shape.

2. Plate and sheet settings There is a fundamental difference in the notion of plate between QuickStep and Esko Plato. In Esko Plato the plate and sheet correspond with respectively the media box and trim box (PDF terminology). The QuickStep sheet was also the trim box (borders) of a GRQ file, which is a perfect match. The QuickStep sheet is translated into the Esko Plato sheet. The plate definition poses a problem. Inherent to the definition of media box, no visible graphics can be positioned outside the media box. This implies that graphics are always clipped on the plate edges. This was not the case in QuickStep, where you could for instance have marks outside the plate. QuickStep had a notion of margins (media box) which were fitted around all graphics. As a consequence, mapping the QuickStep plate onto the Esko Plato plate is not always possible. The following approach is taken during conversion:

1. The sheet definition is taken over.
2. If the sheet and plate definition in the GRQ file are identical:

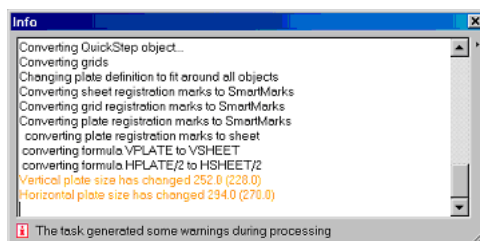
- a. the plate definition is set to VPLATEFIT and HPLATEFIT.
- b. all marks attached to the plate are moved to the sheet. In all formulas, the parameters VPLATE/HPLATE are replaced by VSHEET/HSHEET. By doing so, the marks should keep their exact positions (sheet and plate were identical in QuickStep) and the Esko Plato plate will be fitted around all objects (and will thus correspond to the QuickStep media box).

Esko Plato has no notion of 'clip on sheet'. If you used this feature in QuickStep, it will be lost during conversion. If, during conversion, the sheet or plate size changes, a message will be displayed after conversion.

3. Formulas Formulas are preserved, including user variables (if any). Note that plate parameters might be replaced by sheet parameters (see previous topic).

4. Grids Grids and grid parameters exist in both QuickStep and Esko Plato. All QuickStep grids are simply converted into Esko Plato grids.

5. Registration marks The QuickStep set of registration marks is available under SmartMarks (Esko Plato and step-and-repeat tasks only - not available in Esko PackEdge). This means that QuickStep registration marks are converted into SmartMarks while preserving their functionality, including customised QuickStep registration marks (if any). If the GRQ data is still using old GRR macros, these are not converted and a message will be displayed.



- The plate definition is set to 'platefit'.
- All marks are converted into SmartMarks.
- Plate parameters using formulas used in grids/marks,... are replaced by the corresponding sheet parameters.
- As the plate is now fitted around all objects, the plate size of the Esko Plato file no longer matches the plate size of the QuickStep file (but it should match the margins).

12.1.3. Inks marked as 'Tech'

QuickStep allowed to mark inks as technical but it does not completely match with the notion of technical inks in Esko Plato. In Esko Plato, technical inks are mainly used for die shapes and special non-printing separations. Technical inks are always rendered as opaque. Marking inks as 'Tech' in QuickStep was merely used to exclude inks from registration marks. Inks marked as 'Tech' will be converted into technical inks but note that this is not always possible (for instance, process inks can not be set to technical in Esko Plato). Whenever such a situation is encountered during conversion, a message will appear.

12.2. Studio

Studio offers a 3D viewing environment where designers can see their artwork applied instantly on the packaging. It also includes navigation and alignment tools, and by writing a 3D PDF file from Studio, designers can share their ideas with clients for review and approval.

Studio reduces time-consuming mock-ups, spots design-errors faster and takes away the guesswork from 2D flats.

For folding carton or corrugated packaging, the shapes can be supplied by ArtiosCAD or Score!.

The Studio tools in Plato can be found in **Production > Studio Designer**

12.2.1. Introduction

Welcome to the Studio Designer documentation.

This documentation describes Studio Designer as a plugin in Illustrator as well as Studio integrated in other Esko applications such as PackEdge. Although the documentation is based on Studio Designer in Adobe Illustrator, the functionality is the same in other versions.

About Studio Designer

Studio Designer is a plug-in for Adobe Illustrator, ArtPro, PackEdge and Plato, for interactive 3D packaging design which helps designers make graphic designs or do pre-press for folding carton, bag files, cans, corrugated boxes, flexible packages and so on. Studio Designer works with structural design files, created in ArtiosCAD or Studio Toolkit. These can be ARD files, .bag files or .dae/.zae files (Collada files with a printable part defined). Once a structural design file is placed in a document, Studio Designer can display a three-dimensional preview of the packaging with the artwork rendered on it. This allows designers to see how their graphics will look on the final package. Studio Designer also lets you load a multi-part structural design file as a scene. Each part of the scene can get its graphics from another document. Studio Designer also contains productive tools for quick and accurate aligning and duplicating graphic elements.

Studio Designer is able to export a three-dimensional package to PDF and it can export snapshots of the three-dimensional package to an image file.

About Structural Design files

A structural design file should always contain 2D information, and possibly even 3D information. If the structural design file contains 3D information, Studio Designer can show a folded 3D representation of the structural design file.

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.

Supported structural design file formats:

- ArtiosCAD or Studio Toolkit for Boxes (.ard files) for boxes
- Studio Toolkit .bag files
- Collada files with a printable part (.dae or .zae files)

In case a Collada file is placed, a check is performed to see if the Collada file has a printable part defined. A printable part is an Esko extension to the Collada format.

Collada files with one or more printable parts can also be created in Studio Toolkit and ArtiosCAD.

You can also load a structural design file as an extra scene in the Studio window. Each part of the scene can get its graphics from another illustrator document. See [Scenes and Multiple Documents](#)

Licenses

This software is protected against unauthorised use.

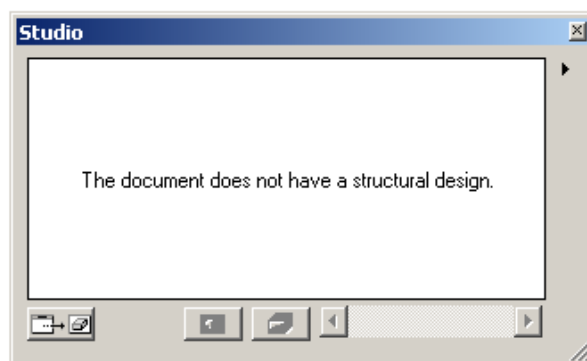
To purchase a license or a subscription, please visit <http://www.esko.com/store> or contact your Esko sales representative

If you have a license for Studio Designer in PackEdge, ArtPro or Plato, this license is included in the PackEdge, ArtPro or Plato licensing. For more information, see the licensing chapter in the corresponding manual.

12.2.2. Studio Designer Basics

To get started with Studio Designer, you need a document with a structural design file placed into it.

If the current document does not contain a structural design file, the Studio window will look like this:







Above you can see how a structural design file is show as a 2D representation. Open the Studio window to see the 3D view.

The Studio Window

The Studio window is the heart of Studio Designer. It is a floating palette where you can see the three-dimensional shape of the package.

To view the Studio window, choose **Production > Studio Designer...**



1.  The Maximize button . Click it to make the Studio window options panel visible. See [The Studio Window options panel](#)
2. The **Scene**. This can be the structural design file used in the current document, or a scene combining multiple documents. See [Scenes and Multiple Documents](#)
3. The **View** dropdown. You can select a View preset from the dropdown, or click **View** to open the View settings. See [View](#) on page 441
4. The **Orientation**. See [Changing the Document Orientation](#)
5. The Fly-out button, opening the fly-out menu.
6. The 3D preview of the current document or scene.
7. The **Refresh** button. See [Refreshing](#) on page 440
8. The **Full Screen Preview** button. This will open the Studio Window in full screen preview, until you hit the **esc** key. See [Full Screen Mode](#) on page 440
9. **Show/Hide transparent panels**. See [Working with Panels](#) on page 443
10. The **Show/Hide Outlines** button adds the outlines of the structural design to the 3D preview.
11. The **Show/Hide 3D Guides** button. See [3D Guides](#) on page 450
12. The **Zoom Slider**. See [Changing the viewing Angle and Distance](#) on page 440

Note: If the placed ARD file does not contain folding angles, the plug-in cannot know how the box has to be folded. In that case 90-degree foldings are assumed on all folding lines. If the resulting shape is not the intended one, the proper folding angles need to be added to the ARD file, for example using ArtiosCAD or Studio Toolkit for Boxes.

Refreshing

By clicking the **Refresh** button in the bottom-left corner of the Studio Window, the graphics will be drawn on the box. Every time the graphics change, the Studio Window will only update if you click the Refresh button.



Refresh is also available via **refresh** in the fly-out window.

If **Auto-Refresh** is enabled in the fly-out window, the Studio Designer window will automatically be refreshed every time your document is changed.

Changing the viewing Angle and Distance

In the Studio Window, you can look at the package from different angles.

- You can click and drag to rotate the 3D object.
- You can hold the shift key and click and drag to move the 3D object left, right, up or down (Panning).
- By moving the slider in the bottom-right corner you can look at the box from a closer or further distance. Another way to do this zooming, is by using the mouse wheel.
- You can reset the camera position by double-clicking while holding the shift key.

To indicate that the package is not floating in mid-air, the **floor** concept was introduced. A package in the Studio Window is always positioned relative to the floor, giving the package a top and bottom definition. If the package (currently only boxes) is showing the wrong panel as top panel, then you can use the **Turn Box** option. See [Turn Box](#) on page 446

Note:

When your box is facing the wrong way and when you have difficulty to turn your box upright, try using Turn Box... The turning around functionality has been designed in such a way that it works fine if you have selected the right bottom panel.

Full Screen Mode

There are different ways to switch your Studio window to full screen mode:

- By clicking the "Full Screen Mode" button  at the bottom of the Studio window
- By choosing **Enable Full Screen Mode** from the Studio Window fly-out menu

- By choosing **Window > Esko > Studio Designer > Enable Full Screen Mode**
- By using the shortcut **F11** on Windows or **Cmd+Alt+-** (minus) on Mac

When full screen mode is activated, you can only change the camera rotation, panning and environment rotation. You can get out of full screen mode by pressing the **esc** key.

Scene Settings

In the **Scene Settings** you can finetune the way your work looks in the Studio window.

You can open the Scene Settings dialog by choosing **Scene Settings** from the fly-out menu, or by clicking the **Scene Settings** link in the Studio Options.

If the **Preview** option is enabled, the Studio window will immediately reflect the changes you make in the Scene Settings.

All settings from the Scene Settings dialog are used in the Export dialog, except Floor Reflection and Background Image for PDF file with 3D design and UD3.

Environment

In the **Environment** tab you can set the **Lighting Environment**, the amount of **Shadow Contrast**, and the background, either a **color** or an **image**.

Floor Reflection

The second tab of the Scene Settings defines the **Floor Reflection** by setting the amount of **Reflection Opacity** and the **Fade** effect.

Note: Lighting Environment and Floor Reflection Fade can only be set when using Visualizer Quality. See [Studio Visualizer in Studio Designer](#) on page 442

View

In the **View** settings tab you can numerically set the view angles, pan and camera distance.

- Set the **Object Heading** and **Object Tilt** to define the position of the object
- Set the **Pan** and **Camera Distance** to define the position of the camera, or click **Fit in Window** to reset the pan to zero and automatically adjust the camera position so that the 3D view fits in the Studio Window.
- Set the **Perspective** by either defining the **Focal Length** (in mm) or **Field of view** (in degrees) of the camera.
- Enable **Keep vertical lines parallel** if you want to get the effect of a tilt shift lens. If enabled, the vertical lines in the pack shots remain vertical (without vantage point). You can use this to create images that are easier to combine afterwards in a 'line-up' image. If the tilt angle is very large it can lead to very unrealistic distorted perspective.

The **View Preset** dropdown allows to select one of the default settings, or any previously saved Preset.

You can save the current settings as a new preset by selecting **Save as** in the dropdown.

Select **Manage...** to open a separate dialog in which you can delete unnecessary presets.

The Scene Presets and View Presets folders can be found here:

- On Macintosh: Macintosh/Users/user/Library/Application Support/Esko/Studio/
- On Windows: C:\Users\user\AppData\Roaming\Esko\Studio\

You can exchange the Presets between different operators by copying the files to another computer.

If you remove all Presets and restart Adobe Illustrator, the default Presets will be recreated.

Scene Presets

The **Scene Preset** dropdown allows to select one of the default settings, or any previously saved Preset.

Note: Scene Settings contain the Environment, Floor Reflection and View.

You can save the current settings as a new preset by selecting **Save as** in the dropdown.

Select **Manage...** to open a separate dialog in which you can delete unnecessary presets.

The Scene Presets and View Presets folders can be found here:

- On Macintosh: Macintosh/Users/user/Library/Application Support/Esko/Studio/
- On Windows: C:\Users\user\AppData\Roaming\Esko\Studio\

You can exchange the Presets between different operators by copying the files to another computer.

If you remove all Presets and restart Adobe Illustrator, the default Presets will be recreated.

Selecting Parts and Panels

You can select parts and panels by clicking on them in the Studio window.

By holding the alt key and clicking a panel or part one or several times, you can select a panel or part that is behind another one.

A Printable part or panel will be selected in red. When selecting a non printable part, all non printable parts sharing the same material will be highlighted in orange. For multiple instances (e.g. multiple cans on a tray), only the printable parts of the original instance will be highlighted.

See also:

- [Zooming in on Panels in the document window](#) on page 445
- [About Hidden Panels](#) on page 444
- [Scenes and Multiple Documents](#)

Studio Visualizer in Studio Designer

Besides the Studio Designer functionality, the Studio window contains a number of functions and options that are related to Studio Visualizer for Adobe Illustrator. These options require a Studio Visualizer license.

By default, Studio Designer shows a preview in its own Studio Designer quality. You can switch to the enhanced Visualizer Quality preview, by selecting **Visualizer Quality** in the fly-out menu

of the Studio palette, or by selecting **Visualizer Quality** in the Studio Window settings panel, if no part is selected.

If **Visualizer Quality** is checked, a number of extra options, such as changing the lighting environment, will also become available.

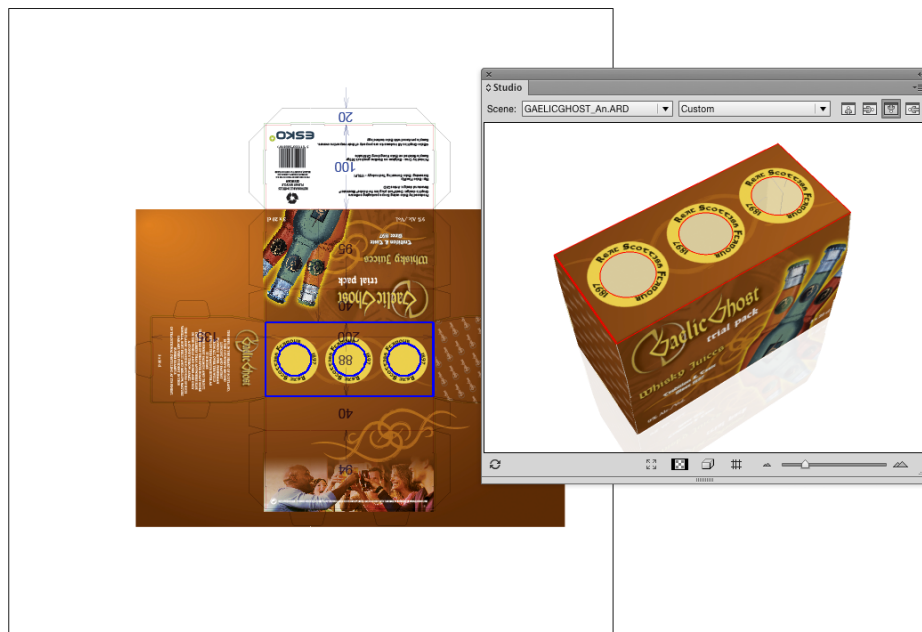
Next to these options in the Studio window, Visualizer for Adobe Illustrator also contains specific palettes such as the Substrate and the Finishing Operations. For more information on these, we refer to the Studio Visualizer for Adobe Illustrator documentation: <http://help.esko.com/products/Studio%20Visualizer%20for%20Adobe%20Illustrator>

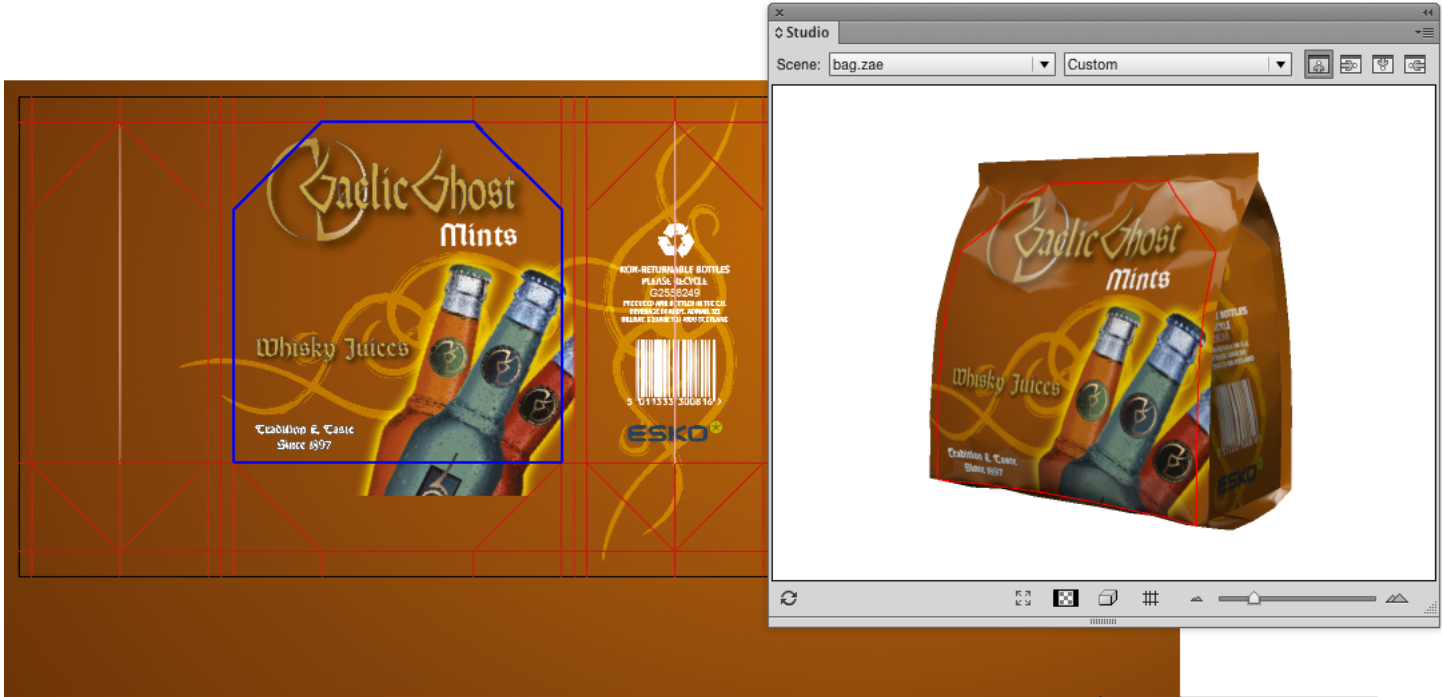
12.2.3. Working with Panels

A box design and a bag design consist of a set of panels which are folded relative to each other.

Selecting Panels

In the Studio Window you can select a panel, simply by clicking on it. The selected panel is used by certain Studio Designer features and does not affect the selected artwork.





Tip:

Show Face Outline (in the fly-out menu) ensures that the selected panel is also shown in the your document as a blue solid line.

Note:

Sometimes a dotted line appears next to the solid one. This is the case when you select a panel that is also part of a **composed face**.

About Hidden Panels

Some panels cannot be seen in the Studio Window because they are covered by another panel no matter which angle you are viewing the package from. To see those panels, you can choose **Show Transparent Panels**, which is available at the bottom and from the Studio Window fly-out menu. In this mode the package is drawn in a semi transparent way, allowing you to see covered panels. This will help you to understand how the package's 2D unfolded view and the 3D view relate to each other.



By holding the alt key and clicking a panel one or several times, you can select a panel that is behind another panel.

Zooming in on Panels in the document window

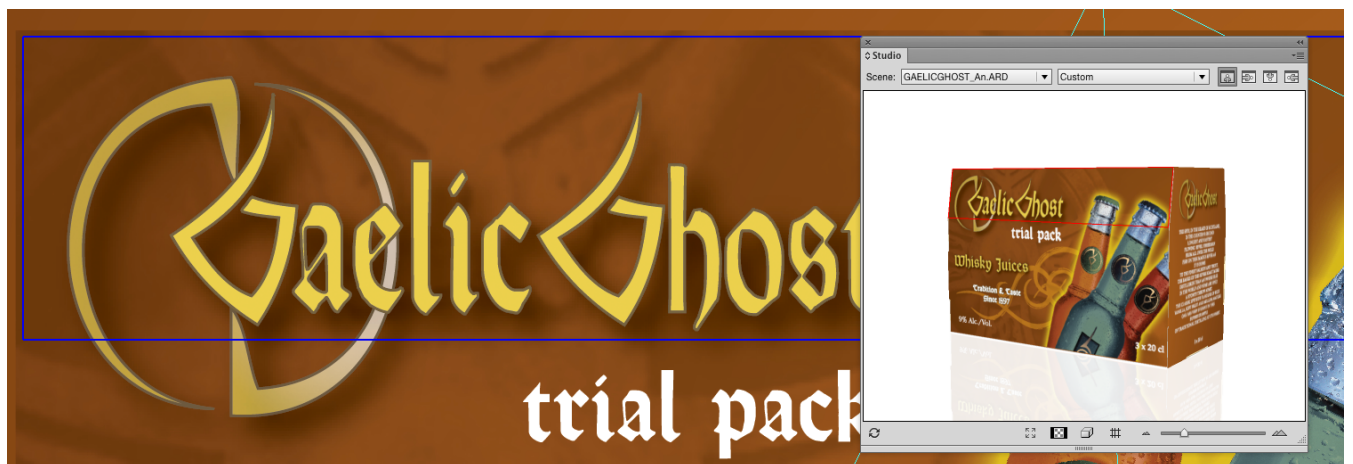
If you double-click a panel (or if you select **Fit Panel in Document Window** from the Studio Window's fly-out menu), the document window will zoom in on the corresponding part of the graphics.

If your graphics are in another document, then this document will become active.

Since only one document can be open at once, this will close the open document. If the open document was changed, you will be asked if you want to save the document first.

Furthermore, the orientation of the document window will change to match the orientation in the Studio Window as closely as possible.

You can also zoom in by holding down the Ctrl key and dragging a rectangle around the area you want to zoom in on.



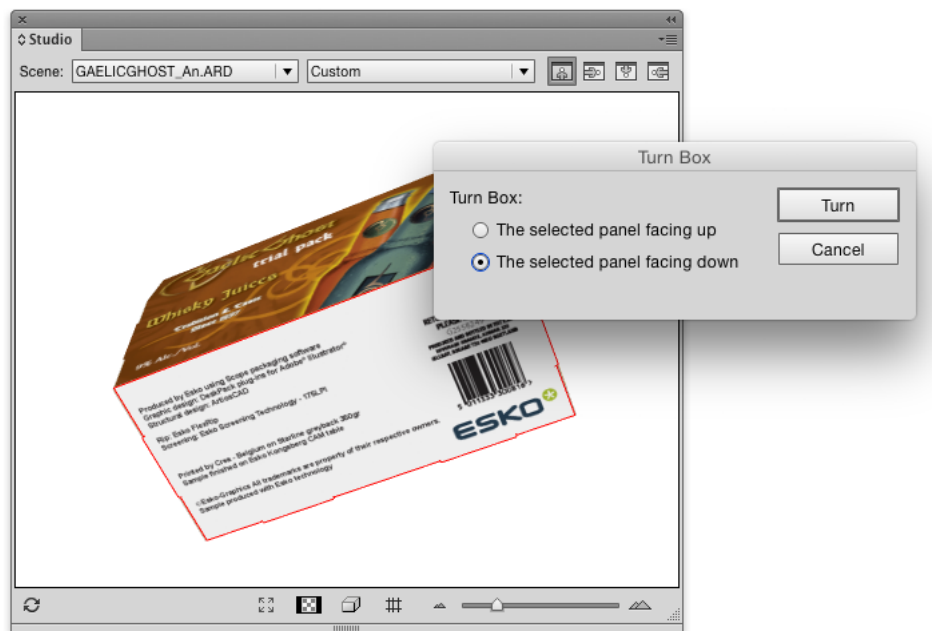
By spinning the package and double-clicking the different panels, the Studio Window can be used to navigate through the document.

Turn Box

When working with designs for folding carton or corrugated boxes, you might not be able to position the box in the desired viewing angle in the Studio Window. This happens when the package has the wrong panel(s) facing up. You can change this e.g. in the structural design file (by selecting another Base Panel in ArtiosCAD or Studio Toolkit for Boxes), but you can also turn the box in the Studio Window. Just select a panel that should be facing up or down and select **Turn Box** from the Studio fly-out menu. The Turn Box functionality is linked with the Floor concept, so e.g. **The selected panel facing up** means that the selected panel will become the top panel relative to the floor.

Note:

The Turn Box option is only available for boxes (ard files)



Bring Panel Forward and Send Panel Backward

This functionality is only available for boxes and composed faces of which the panels are all in the same plane. Overlapping panels in the same plane can occur in any file. Sometimes the order of these overlapping panels is incorrect. Most of the time, this will happen when the panels have been folded 90 degrees on top of each other. With **Bring Panel Forward** and **Send Panel Backward**, you can correct this order.

If you wish to view another panel on top, you can select the panel, select **Bring Panel Forward** from the Studio fly-out window and bring it forward.

Creating Copies with the Distribute command

Most of the faces of a package consist of a single panel. So there is just one rectangular area in the document that will make up this face. However, sometimes a face is made of several panels. The panels of such a “composed face” are spread across the document. They can even have a different orientation. So for composed faces it is very difficult to create graphics that run continuous from one panel into the other. Studio Designer can help you in this process with a feature called **Distribute**.

Note:

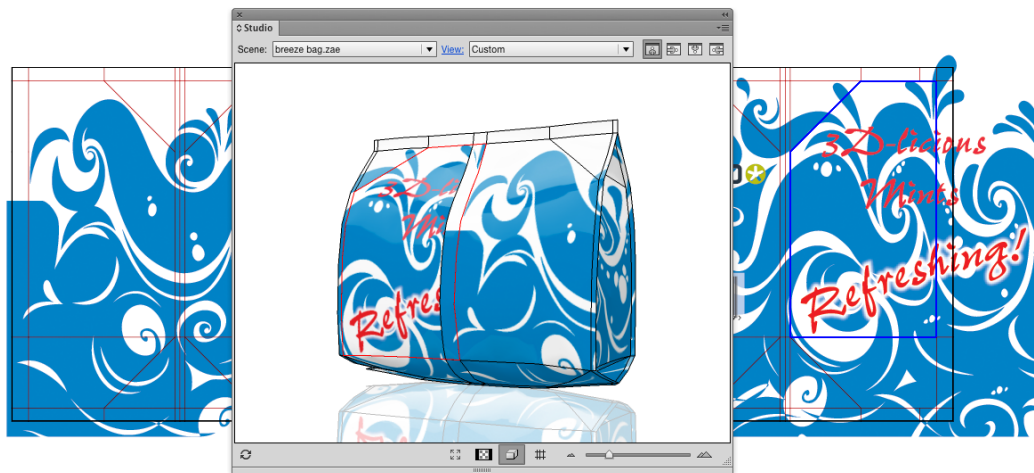
You can use Distribute on .bag and .ard files but not on Collada files.

Composed Faces

This is an example of a box with composed faces.-



This is an example of a bag.



Viewing the Composed Face

If you select **Show Face Outlines** in the Studio fly-out menu and you click one panel that is part of a composed face, 'ghost' outlines will be drawn of the other panels of that face. These ghost outlines can give you a good impression of the size and shape of the composed face.



Distribute

If you draw some art within the boundaries of such a composed face, it will only appear on one panel, since the other panels are actually somewhere else in the Illustrator document. However, with **Distribute**, you can create copies of the art over several panels that are perfectly aligned to make it look continuous on the folded composed face.

This is how the folded package will look without applying Distribute.



Distribute is available from Studio's fly-out menu. First you need to select art in the document and you need to select the corresponding panel in the Studio Window. The selected art is supposed to be positioned correctly on the selected panel. If you choose **Distribute**, copies will be made of the selected art for the other panels (of the same composed face). In the Studio Window (which is automatically refreshed after a Distribute) you can see how the graphics are running seamless from one panel into another.

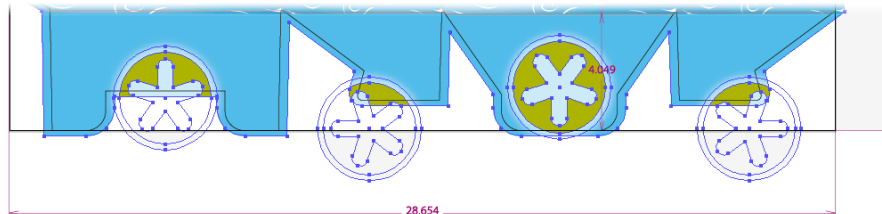
Normally, distribute will make a copy of the selected objects for every panel in the composed face (unless there is no overlap). If you do not need all these copies, you can simply delete the ones you do not need.

This is how the folded package will look when having applied Distribute.



Making Clipping Masks

It is very likely that one or more of the copies lay partially outside of the panel they were created for. Sometimes this is a desired effect (having the graphics run continuously over a folding line). Sometimes this is not a desired effect and you will have to create a clipping mask to contain the copy within its panel.



Working With Distributed Art

You can update the copied distributed art, for example after you have edited the original art; or you can undistribute and expand distributed art if you want to delete the copies.

Undistribute and Expand

The options **Undistribute** and **Expand**, which can be found in Studio's fly-out menu also apply to distributed art. If you select distributed art and you **Undistribute**, all other (non-selected) copies are deleted. If you have distributed art selected and you **Expand**, nothing will change visually yet the selected art will loose its 'intelligence' and become regular artwork, insensitive to actions like **Update Copies** or **Undistribute**.

You will need to use **Expand** if you start copying and pasting distributed art. The copy will still be considered as part of a distribution, by expanding it, it will no longer be seen as a distribution, but as an ordinary object.

Note:

When the distributed art is a group, ungrouping it is equivalent to applying **Expand**. The art does not longer know that it was distributed, even if you group it again.

12.2.4. 3D Guides

In Illustrator, PackEdge, ... you can create horizontal and vertical Guides to align art. Studio Designer offers another type of guides: 3D Guides for boxes and Collada files.

3D guides appear both in the document window and in the Studio Window.

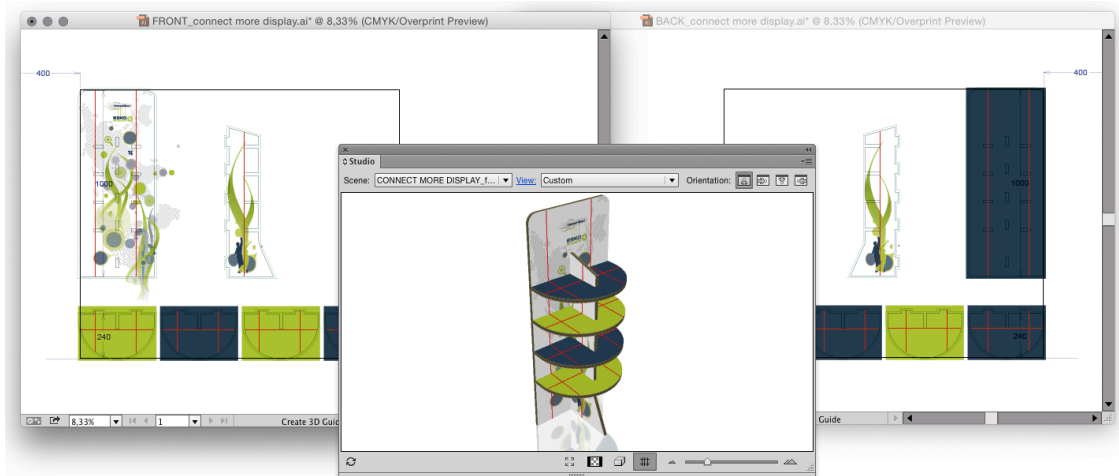
Note: 3D guides are temporary visual aids, and are NOT saved.

You can hide or show the 3D guides in the document window by switching the visibility of the 3D Guides layer on or off in the Layer window.

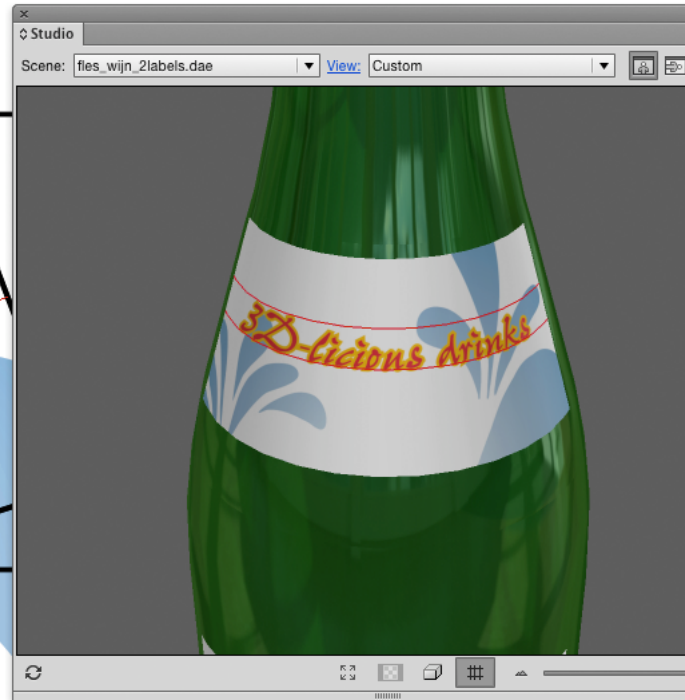
Select Show/Hide 3D Guides from the Studio fly-out menu (or click the button at the bottom) to hide or show the 3D guides in the Studio Window.

In the Studio Window, the 3D guides appear like a horizontal or vertical plane that is intersecting the shape. In the document, a 3D Guide appears like a set of horizontal, vertical or even slanted line segments clipped inside panel boundaries. These line segments show where the plane is intersecting with the panels. Graphics can snap to these segments just as with regular guides.

This is how the 3D guides are presented in the 2D and 3D view:



For Collada files made with Studio Toolkit For Labels, you can get curved 3D Guides in the document if your printable part is a conical label:



Creating and Editing 3D Guides

To create or move a 3D guide, use the **3D Guide** tool (in the applications toolbar). Click and drag somewhere in the shape in the document to create or move a guide. By holding the option (alt) key, you can switch between vertical and horizontal guides.



To delete a 3D guide, use the 3D guide tool and simply drag it outside the shape in the document.

To delete all 3D guides, you can delete the “3D Guides” layer that contains all 3D guides.

12.2.5. Exporting

Studio Designer offers a variety of export options to a number of common formats.

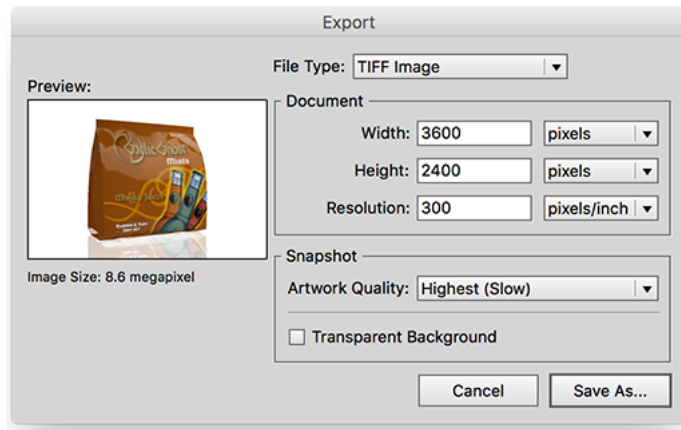
The background color and other settings defined in the Scene Settings will show up in your exported file.

Note: If you have the **WebCenter Connector** plug-in installed, you can use **Save to WebCenter** to upload the exported file directly to WebCenter. See [Using Studio Designer and WebCenter Connector](#)

Writing TIFF Images

Studio Designer can export the 3D design to an RGB TIFF image with the graphics on top (much like the Studio window but at higher resolutions).

1. Choose **Export...** from Studio's fly-out menu. Choose **TIFF Image** as file type.



2. Define the **Width**, **Height** and **Resolution** for the image that you want to export.
3. Define the **Artwork Quality**. Depending on what the image is needed for, you can choose between 4 presets for the level of detail. Choosing Low (fast) will export to an image fast but with less detail, while choosing Medium, High or Highest (slow) will produce greater detail but will take more time.

Tip:
 The "Low" and "Medium" presets correspondent to the "High" and "Very High" Image Quality found under "Model" in the "old" Studio Visualizer 14 application.

4. The **Transparent Background** option allows you to set the background transparent instead of the background (color or image) defined in the Scene Settings.
5. Click **Save As...** to save the TIFF file.

The **Scene Settings** are used in the exported TIFF image: Object heading and Tilt, Pan, Perspective, etc. See [Scene Settings](#) on page 441

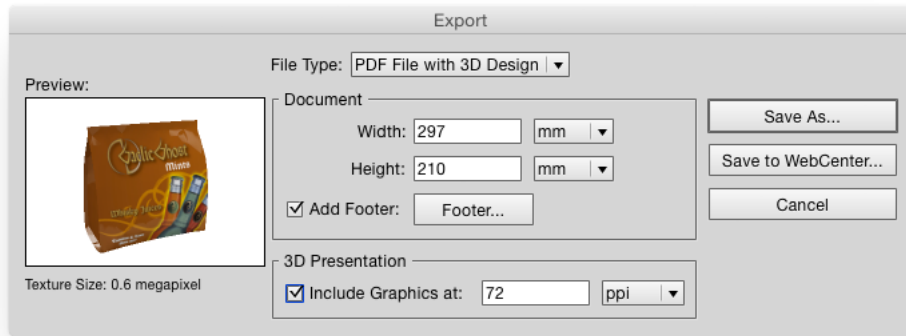
Export to a 3D PDF File

In Adobe Reader or Acrobat Professional (version 8.0.1 or later) you can view / rotate 3D objects in PDF files. Studio Designer can write such PDF files that can be used to view a package in 3D on a computer that does not have Illustrator or Studio installed.

Choose **Export...** from Studio's fly-out menu. Choose **PDF File with 3D Design** as file type.

You can define the **Size** of the PDF file. Since the PDF file is only meant for on-screen viewing, this size is of little importance.

The **Add Footer** toggle allows you to switch on or off the footer text. Instead of the default EskoArtwork text you can also personalize the footer text and include your own message.



Note:

To avoid z-fighting problems in the exported PDF files, you can use the **Avoid Artefacts** option. This will make sure that whenever it is unclear which panel is on top (resulting in flickering graphics), a calculated choice will be made, cutting out the underlying part and giving the top part carte blanche.



Export to a U3D File

U3D is a common exchange format for 3D objects. For example, in Acrobat Professional 8.0.1 or higher you can place U3D files in PDF files.

Choose **Export...** from Studio's fly-out menu. Choose **U3D File** as file type.



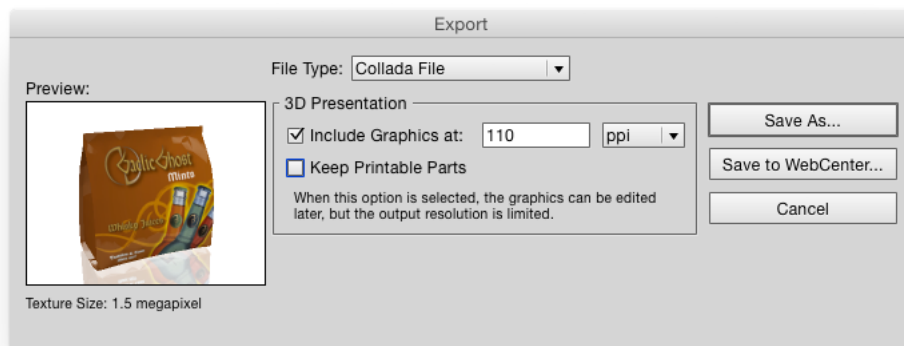
Note:

To avoid z-fighting problems in the exported U3D files, you can use the **Avoid Artefacts** option. This will make sure that whenever it is unclear which panel is on top (resulting in flickering graphics), a calculated choice will be made, cutting out the underlying part and giving the top part carte blanche.

Export Collada File

Studio Designer can also export to a Collada file (*.zae), a common exchange format for 3D models.

1. Choose **Export...** from Studio's fly-out menu.
2. Choose **Collada File** (*.zae) as file type.



3. Select **Include Graphics at...** to include the graphics at the desired resolution.

4. If you want to further use the Collada file in Studio Designer you need to select **Keep Printable Parts**.

If **Keep Printable Parts** is enabled, the artwork is always exported at the default resolution.

5. Click the **Save As** button, and define name and location for the file.

Note:

The exported Collada files (.zae) can be further used in other EskoArtwork Studio applications like Studio Visualizer, Studio Toolkit for Shrink Sleeves or Esko Store Visualizer and ArtiosCAD or uploaded to the online Esko Studio Viewer.

Save Graphics Preview as PNG...

From within the Studio window, you can save a graphics preview file as a .png file, mainly for use in **Studio Toolkit for Flexibles**.



12.2.6. FAQ

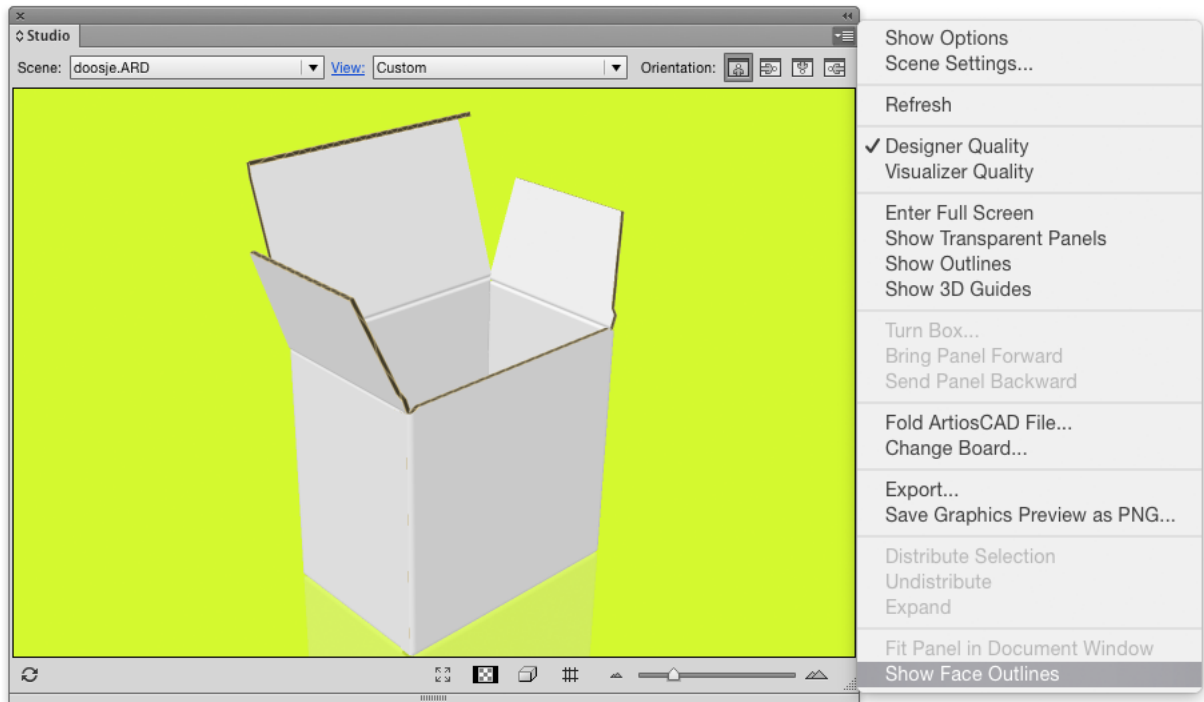
This section contains the answers to a number of Frequently Asked Questions.

There is no overlap between the selected objects and the selected panel

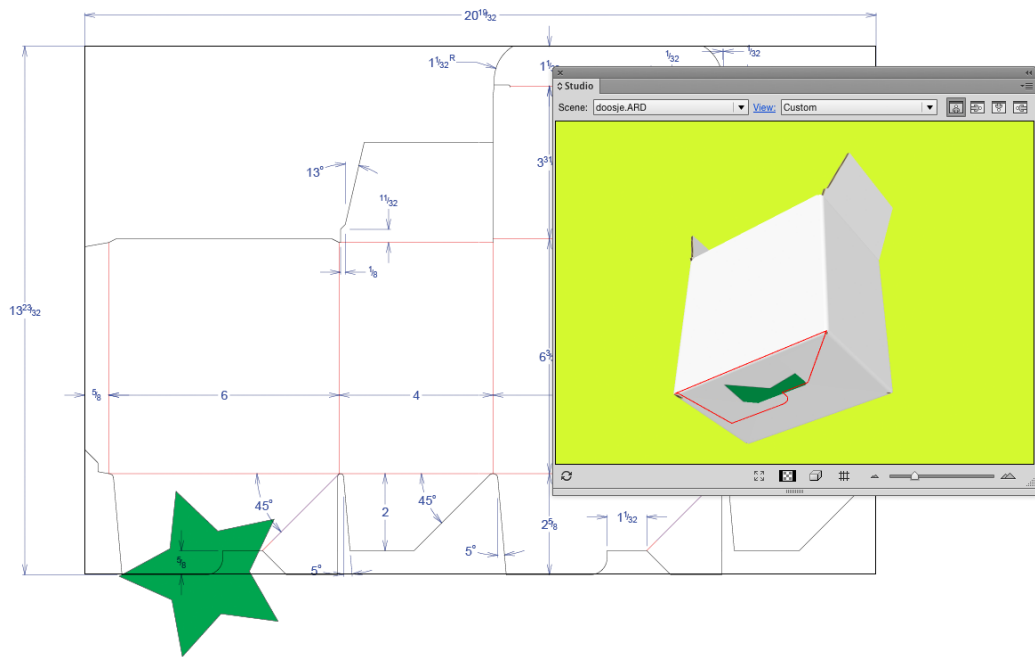
I try to distribute a logo, and I am getting the error message There is no overlap between the selected objects and the selected panel. Please select a panel that overlaps with the selected objects.

Whenever distributing for the first time, you will need to select two things: the graphics that you want to distribute in the 2D view AND the source panel in the Studio window. The 'no overlap' error message indicates that you did select a panel in the Studio window, but in 2D there is no overlap between the selected panel and the selected art.

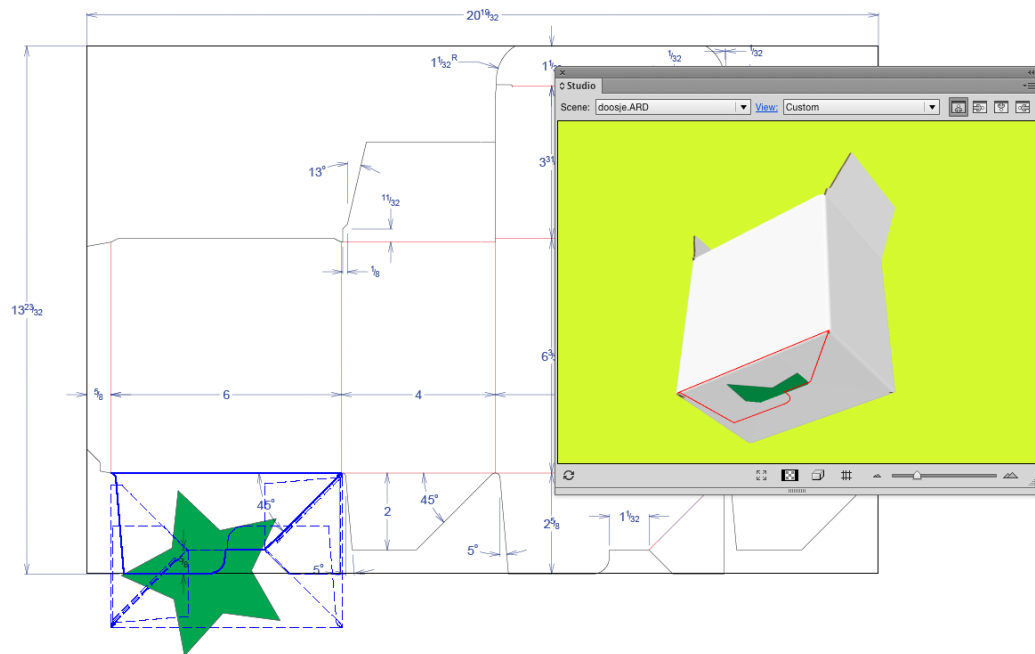
It is a good idea to switch on **Show Face Outlines** in Studio's fly-out menu.



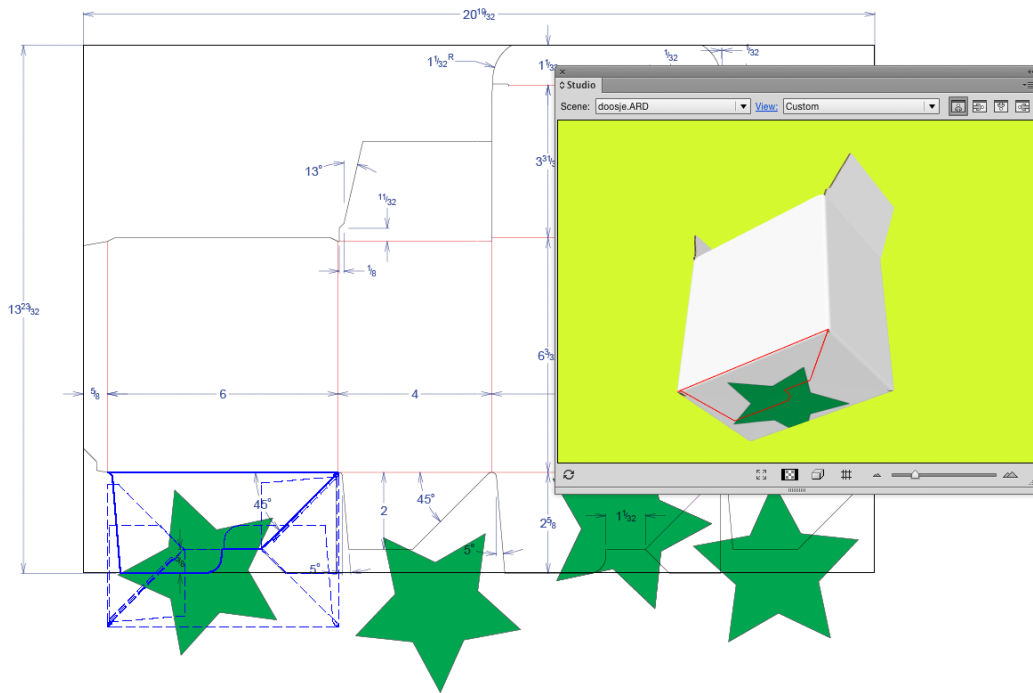
When **Show Face Outlines** is switched on, the selected panel in the Studio window will be shown in the 2D View:



You will immediately see that there is no overlap between the selected panel (the blue rectangle) and the selected art (the star), so you basically selected the wrong panel. You need to select a panel that has an overlap with the selected art:



Now you see that there definitely is an overlap between the blue rectangle and the star. You will notice that **Distribute Selection** will work, which will give you the following result:



Please note that you never had to specify to which 'target panel' you had to distribute, so how can you know which target panels will be taken?

In this version of Studio Designer, the distribute functionality only works on coplanar panels. The bottom face consists of four panels coming together: two glue panels and two larger panels. The selected panel is drawn with a solid blue line in the 2D View, the other panels that are coplanar with it are drawn with dashed lines. Distribution happens from the panel drawn with a solid blue line to the panels drawn in dashed blue lines.

That is why **Show Face Outlines** is so interesting, as it immediately gives you an indication whether or not distributing is at all possible and to which panels the distribution will happen.

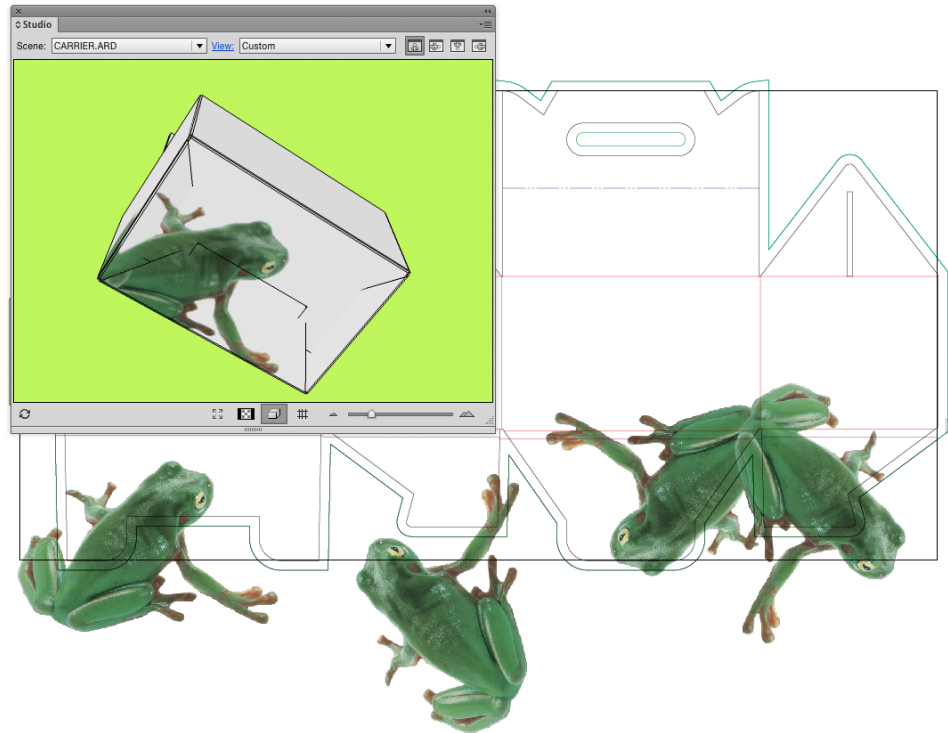
If we move one of the stars and do an update, you will notice that the three other stars need to move as well. It is now enough to select the moved star and click **Update Copies**. You will notice that **Distribute Selection** is now called **Update Copies**. There is no need anymore to select a panel in the Studio window.

The selected art knows that it has been distributed before. An indication that you have selected "distributed art" is given in the info message in the Studio window: `The selection contains distributed art.` This piece of distributed art has become a smart object that knows to which panel it belongs and which other art belongs to the same distribution. When clicking **Update Copies**, the other art that belongs to the same distribution (in our example, the other stars) will be replaced with a new updated copy of the selected distributed art.

In the layer browser, you also have an indication which art is "distributed art" and hence smart. The name of distributed art is always "Distribution".

What can I do if the distribution looks wrong?

The graphics have been distributed to the correct coplanar panels, but the result of the distribution in the Studio window is still not correct.



In this case it is not that the distribution has positioned the copies in the wrong place, but that in the 2D document window, the different copies are overlapping one another. This can happen whenever the graphic that you want to distribute is rather large. We will need to make clipping masks in this case, using the standard tools.

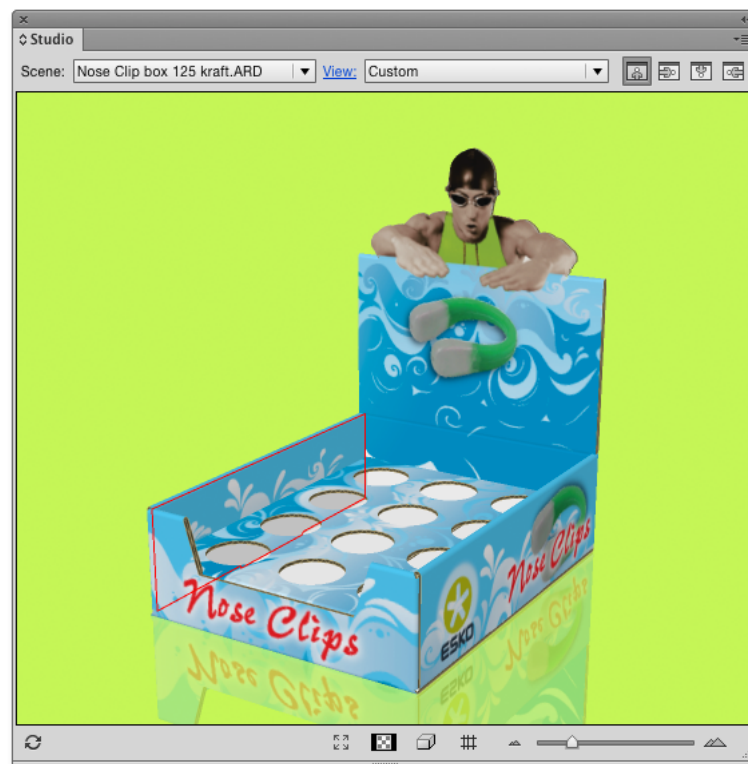
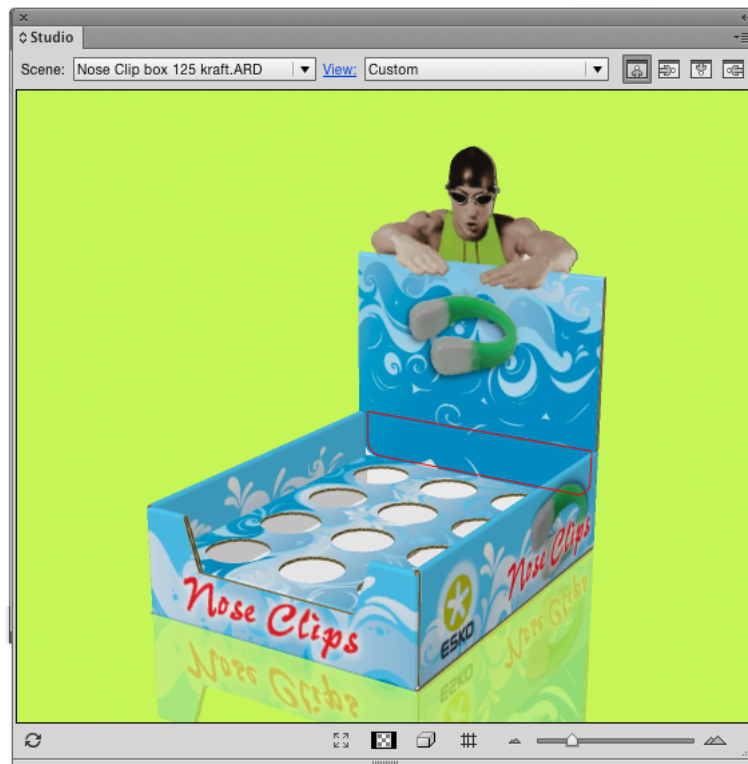
If we do this for every distributed copies and then refresh the Studio window, we will get the correct result:

You only need to make the clipping masks once. Assume you would like to move the frog a little, then simply move one copy and click **Update Copies**. The other copies will also be updated and they will keep their position under their clipping masks.

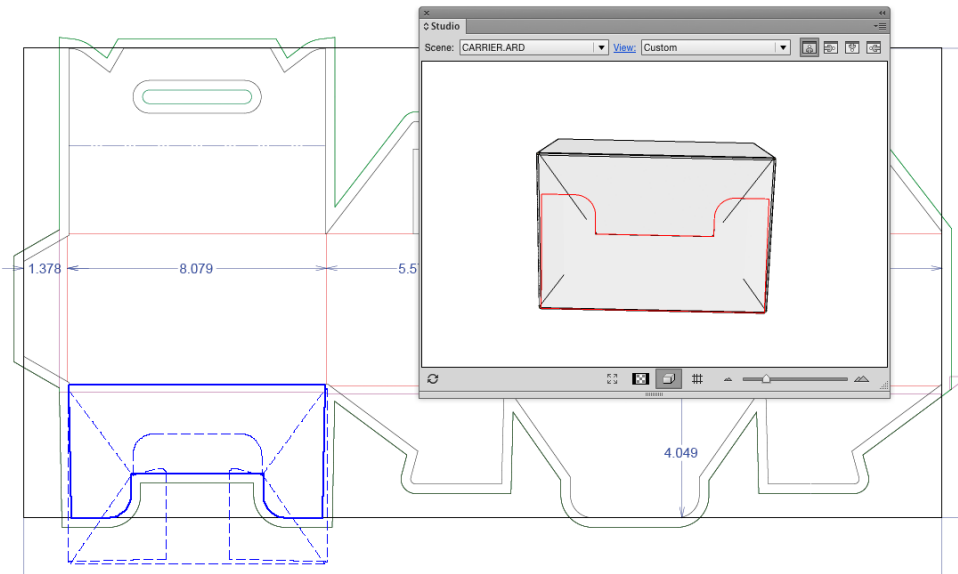
How can I distribute around a corner?

In this version of Studio Designer, distributing is only possible to other coplanar panels. Sometimes a distribution appears to go around the corner, but this only works if a panel that is attached to the panel around the corner is coplanar with the selected panel.

This is not the case with the box in the next example.



In this version of the software, we are not capable of doing this with the automatic distribute. We have limited the distribution functionality to panels that are coplanar. A face in the Studio window can be made up of different panels coming together. A beautiful example is the bottom of the box in the picture below.



The bottom of this box consists of four panels coming together. If you select **Show Face Outlines** in the Studio's fly-out menu, you will see blue lines showing up in your 2D Document Window. The solid blue line is a reflection of the selected red panel in the Studio Window. The other panels that are coplanar with the selected panel and that together form the bottom face, are drawn on top of that in dashed lines. If you were to draw some graphics and click **Distribute Selection** then the graphics would be copied and pasted to the three other panels that are coplanar with the selected one. Those three panels are shown with dashed blue lines in the 2D Document Window.

In the following example, we would like a distribute from one panel to another that is adjacent to the first in the Studio Window, but they are not at all coplanar. On the contrary, the angle between the two panels is 90 degrees. As automatically distributing will be impossible in this case we will have to do the copying, rotating and pasting of the graphics ourselves. The **Create 3D Guide** option can be a solution.

You could create a 3D Guide in the back inner panel and that is immediately going to create extra guides in the other panels that the guide plane intersects with.

You could now design your graphics in the back inner panel. You do not need to do this upside down as double-clicking on the panel in the Studio Window will rotate the whole job. You can also rotate your job manually using the orientation buttons in the right top of the Studio Window.

We will add some text to this panel that also needs to extend into the other panel.



We will now copy the text, double-click the other panel in the Studio Window where you would like the text to be pasted, paste the text, rotate it and position it correctly on the guide:

