Contents

1. Location and Copyright...................................................................................................................... 8

2. Welcome.............................................................................................................................................. 10
   Introduction........................................................................................................................................ 10
   Before you begin................................................................................................................................. 10
   Package contents................................................................................................................................. 10
   Important folders, libraries, and files of Esko ArtiosCAD................................................................. 11
   Display adapter optimization in Windows Server™ 2003................................................................. 13
   Operating system notes....................................................................................................................... 13
   Support for VMware® Fusion........................................................................................................... 14

3. Installing and Configuring ArtiosCAD Enterprise............................................................................ 15
   Installing ArtiosCAD Enterprise....................................................................................................... 15
   Step 1 - Installing Esko WebCenter.................................................................................................. 15
   Step 2 - Running the Board Importer Batch File.............................................................................. 22
   Step 3 - Loading the Esko ArtiosCAD Defaults................................................................................ 22
   Step 4 - Creating a New Company and Location........................................................................... 23
   Step 5 - Creating a New Group.......................................................................................................... 23
   Step 6 - Creating New Users............................................................................................................. 24
   Step 7 - Setting a New User's Group Membership.......................................................................... 26
   Step 8 - Inviting New Users to the Shared Defaults Project............................................................ 26
   Step 9 - Installing ArtiosCAD Enterprise on a User's Computer.................................................... 27
   Post-Installation Configuration Tasks................................................................................................ 28
   Configuring ArtiosCAD Enterprise................................................................................................... 33
   Using the Flute Browser.................................................................................................................... 34
   Using the Board Browser.................................................................................................................. 37
   Defaults........................................................................................................................................... 50
   Always Showing All Boards.............................................................................................................. 61
   Defaults Availability overview.......................................................................................................... 62
   Configuring the Automatic Naming Templates............................................................................... 66
   Using Spot Colors in PDF files......................................................................................................... 67

4. Installing Esko ArtiosCAD.................................................................................................................. 70
   Introduction....................................................................................................................................... 70
   Performing a Typical installation......................................................................................................... 71
   Typical Standalone or Server installation......................................................................................... 73
   Typical Standard Client Installation................................................................................................. 74
   Performing an Advanced installation................................................................................................ 74
   Using other databases with Esko ArtiosCAD.................................................................................... 77
   Other database notes.......................................................................................................................... 77
7. Defaults.................................................................................................................................144
   Introduction to Defaults........................................................................................................144
   Creating defaults..................................................................................................................144
   Changing defaults................................................................................................................145
   Saving defaults.....................................................................................................................145
   Defaults performance..........................................................................................................145
   Parameter sets......................................................................................................................145
      Designer parameter sets....................................................................................................145
      Manufacturing parameter sets..........................................................................................146
      Counter parameter sets....................................................................................................149
   Rubber types.......................................................................................................................156
   Rubber parameter sets.........................................................................................................158
      Bolt holes............................................................................................................................158
      Breaker knives....................................................................................................................159
      Crease Mask......................................................................................................................160
      Cut/Crease..........................................................................................................................161
      General...............................................................................................................................161
      Layout................................................................................................................................163
      Plastic cover.......................................................................................................................163
      Rubber types....................................................................................................................164
   Rebuild options....................................................................................................................168
   3D Import options................................................................................................................169
   3D Rendering method..........................................................................................................170
   Database defaults.................................................................................................................171
      File Operations group.......................................................................................................172
      Performance options group..............................................................................................172
      Database Information Dialog Customization groups.......................................................175
      Default selections group..................................................................................................181
      Company selection display group....................................................................................182
   Spell checking.......................................................................................................................182
   Nudge Options and Defaults...............................................................................................183
   Hatch Defaults.....................................................................................................................184
      Hatch Catalog defaults.....................................................................................................184
      Hatch Property defaults...................................................................................................185
      Hatch Startup defaults....................................................................................................186
## Contents

Adding LASERPOINT IQ standards to Style Catalogs ................................................................. 186
  Preparation ............................................................................................................................. 186
Adding the standard to a Style Catalog ................................................................................. 187
Troubleshooting newly-added INTERACT/LASERPOINT/ LASERPOINT IQ standards ...... 188
Adding Designer WorkBench designs to Style Catalogs ....................................................... 189
  Preparation ............................................................................................................................. 189
  Adding a Designer WorkBench standard to a Style Catalog ............................................. 189
Default new design template ................................................................................................. 190
Default layers in Manufacturing .......................................................................................... 191
Toolbars .................................................................................................................................... 194
  Customized toolbars ............................................................................................................. 194
  Colored toolbars ................................................................................................................ 197
Shortcuts ................................................................................................................................... 200
  Assigning a shortcut to a menu command .......................................................................... 201
  Assigning a shortcut to an Output ...................................................................................... 203
  Assigning a shortcut to a geometry macro ......................................................................... 203
  Shortcut uniqueness and priority ....................................................................................... 204
  Notes and warnings for Shortcuts ..................................................................................... 205
Tack Bridging ........................................................................................................................... 206
Bridging Formulae ................................................................................................................... 207
  Evenly spaced bridging formula ......................................................................................... 208
  Inset distance from line end bridging formula ................................................................... 209
Working with tuning catalog entries in Defaults ................................................................... 210
  Creating a PDF vector import tuning catalog entry ......................................................... 210
  Opening a PDF vector import tuning catalog entry ......................................................... 211
  Editing a PDF vector import tuning catalog entry using a template ............................. 212
  Editing a PDF vector import tuning catalog entry manually ......................................... 215
Line types ............................................................................................................................... 216
Subtypes ..................................................................................................................................... 218
Plotting Styles .......................................................................................................................... 219
  Working with simple plotting styles ................................................................................. 219
  Creating a new plotting style by copying from another plotting style ............................ 222
  Working with Advanced plotting styles ........................................................................... 223
Special Rule Types ................................................................................................................... 231
  Manufacturing tab for special rules .................................................................................. 232
  Special rule tab for special rules ....................................................................................... 235
  Multi-Perf Overview .......................................................................................................... 237
  Edge band and Tear Tape Overview ................................................................................. 241
Default line type set ............................................................................................................... 247
CAM Tooling Setup Catalog .................................................................................................. 248
  Tool List tab ....................................................................................................................... 248
  Tool Selection tab .............................................................................................................. 251
  Optimization tab ............................................................................................................... 253
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Counter Tool Widths tab</td>
<td>318</td>
</tr>
<tr>
<td>Counter Cutting Order tab</td>
<td>319</td>
</tr>
<tr>
<td>Grouped Outputs tab</td>
<td>320</td>
</tr>
<tr>
<td>PDF Security tab</td>
<td>322</td>
</tr>
<tr>
<td>Configuring a PDF Output using Adobe® Acrobat® 5</td>
<td>324</td>
</tr>
<tr>
<td>Configuring 3D Outputs</td>
<td>324</td>
</tr>
<tr>
<td>Creating a new 3D Output</td>
<td>325</td>
</tr>
<tr>
<td>Changing Collada defaults</td>
<td>327</td>
</tr>
<tr>
<td>Rule labels</td>
<td>328</td>
</tr>
<tr>
<td>Configuring the Output Scale Table</td>
<td>331</td>
</tr>
<tr>
<td>Notes on using the Output Scale Table</td>
<td>331</td>
</tr>
<tr>
<td>AutoSave / Local Temp Directory</td>
<td>331</td>
</tr>
<tr>
<td>Recovering AutoSave files</td>
<td>332</td>
</tr>
<tr>
<td>Revision Disk File Format in Revision History</td>
<td>333</td>
</tr>
<tr>
<td>Database Browser Reports</td>
<td>334</td>
</tr>
<tr>
<td>Publishing Esko ArtiosCAD documents to WebCenter</td>
<td>336</td>
</tr>
<tr>
<td>WebCenter Project Name group</td>
<td>336</td>
</tr>
<tr>
<td>Publish Options group</td>
<td>337</td>
</tr>
<tr>
<td>WebCenter Folder Name group</td>
<td>337</td>
</tr>
<tr>
<td>WebCenter Login group</td>
<td>337</td>
</tr>
<tr>
<td>WebCenter Connection group</td>
<td>338</td>
</tr>
<tr>
<td>Graphic PDF file rotation import option</td>
<td>338</td>
</tr>
<tr>
<td>Setting CAPE/TOPS Defaults</td>
<td>338</td>
</tr>
<tr>
<td>Setting DWG/DXF Options</td>
<td>340</td>
</tr>
</tbody>
</table>
1. Location and Copyright

Esko
Kortrijksesteenweg 1095
BE-9051 Gent
Belgium
Tel.: (32) (9) 216-92-11
Fax: (32) (9) 216-94-64
Other offices worldwide.

Written and revised by Adam Hartfield, October 2012.
For use with ArtiosCAD Enterprise 12 or greater.

© Copyright 2012 Esko Software BVBA, Gent, Belgium.

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

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

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

This document supersedes all previous dated versions.

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

Adobe, Acrobat, Illustrator, and PostScript are either registered trademarks or trademarks of Adobe Systems Incorporated in the United States and/or other countries.

Apple and QuickTime are registered trademarks of Apple, Inc.

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

The Esko software may contain an implementation of the LZW algorithm licensed under U. S. Patent 4,558,302 and foreign counterparts.

The Esko software may contain 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.

Strip Clip, Strip Fork and Strip Clip® System are products, registered trademarks and patents of Vossen Profitec® GmbH Germany.

OpenGL is a registered trademark of Silicon Graphics, Inc.

The geometry macros contained with this release of Esko ArtiosCAD to facilitate the use of Vossen Profitec components are approved by Vossen Profitec GmbH and are used with their permission. The Vossen Prax® documentation should be consulted and used to ensure correct use and placement of these geometry macros. The shapes and offsets used are in accordance with Vossen Profitec GmbH specifications. Usage and placement of these geometry macros to ensure effective stripping performance however, remains the responsibility of the user. Vossen Profitec GmbH may be contacted for details of worldwide representation at (49) (7771) 920-136 or by e-mail at info@vossen-profitec.de.

This software may use libxml2 - Copyright © 1998-2003 Daniel Veillard - 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
Kortrijksesteenweg 1095
BE-9051 Gent
Belgium
info.eur@esko.com
2. Welcome

Introduction

This book describes how to load Esko ArtiosCAD and its related components, both as a new installation and as an upgrade to a previous version, as well as how to add input and output devices and perform other configuration tasks.

For the latest information on system requirements, please go to the Esko website and search for system requirements.

Before you begin

Before trying to install Esko ArtiosCAD, you should make sure that you know how to:

- view file and directory information with Windows Explorer
- make network connections and create network shares
- use a text editor such as Notepad

If using a network to connect seats, especially on systems using Windows XP or later, make sure the network is up and running correctly so that you can see network resources from all machines. To install Esko ArtiosCAD, you must be able to log in using an account that is a member of the local Administrators group.

Esko ArtiosCAD will work immediately after being installed, but you will most likely want to set up some output devices as described in the Peripherals chapter.

If you have any questions, please do not hesitate to contact your local Support office for help.

If you will be using Esko ArtiosCAD in a language other than American English, make sure your operating system’s regional settings are set to match those of the United States. Specifically, for numbers, the decimal symbol needs to be a period, the decimal grouping symbol a comma, the list separator a comma. For currency, the decimal symbol needs to be a period and the digit grouping symbol a comma. Unpredictable results might occur if these settings are not made.

Package contents

The Esko ArtiosCAD package includes the Esko ArtiosCAD media which contains:

- The installation program
- Release notes - an overview of changes and enhancements and a list by number of bug fixes
- Esko ArtiosCAD Viewer - a program you can give your vendors and customers. It is a stripped down version of Esko ArtiosCAD which allows the user to view and print any Esko ArtiosCAD design or 3D file. Viewer users can directly measure the structure, preview a 3D file with or without graphics from different angles, and fold and unfold it. Viewer is also available for downloading
for free from the Esko ArtiosCAD website (http://www.esko.com) and can be loaded on any 
system which supports Esko ArtiosCAD.

Important folders, libraries, and files of Esko ArtiosCAD

There are several folders that are set up under the \Esko\Artios folder on the drive chosen for the 
installation. One is the folder that contains the Esko ArtiosCAD version that was installed. Others 
hold database, license, or other related information.

Folders underneath the version of Esko ArtiosCAD

Three library folders have unique and important functions in the configuration and running of Esko 
ArtiosCAD. They are: InstLib, ServerLib and ClientLib. These three folders are found under 
the installation directory for Esko ArtiosCAD.

InstLib:

- InstLib contains default files shipped with the software. It is important to not change these files 
  for a number of reasons. NEVER save modified files in InstLib.
- Reloading (upgrade mode) the current revision from media will overwrite any changes made to 
  files in InstLib.
- Some of the files in InstLib (sysdflt.adf and its associated files) are used as reference 
  files when doing an Esko ArtiosCAD version upgrade. These files are checked against the user-
modified versions in ServerLib to determine what and how to move modified information to 
the new version.
- It is useful to have a clean set of files to work from. If a file in ServerLib gets corrupted for 
some reason, you can recover a clean copy without having to reload Esko ArtiosCAD. (The better 
solution is to make regular system backups!)
- There is a searchlist set up for Esko ArtiosCAD which looks at these directories in the order 
  ClientLib, ServerLib, InstLib. Files that are duplicated in ServerLib, for example, will be 
  used in preference to those in InstLib. Duplicate files in ClientLib will be found in preference 
  to files in both ServerLib and InstLib.
- The Symbol sub-folder contains samples of common corrugated symbols such as 
  fragile.ard, recycle.ard, thiswayup.ard.

ServerLib:

- ServerLib contains configurations that have been modified from their initial defaults, standards, 
  logos and other files (other than user data files) that need to be shared by all of the Esko ArtiosCAD 
  seats on the site network.
- When Shared Defaults are modified, the changes are saved to ServerLib in a file called 
  sysdflt.adf (and files that this references).
- New (additional) files that are to be shared by all of the Esko ArtiosCAD seats on the site network 
  should be placed in ServerLib. For example, reports that need to be used by all of the Esko 
  ArtiosCAD seats on the site network should be placed here. A copy of the report should be placed 
  in ServerLib and added to the Reports Catalog in Shared Defaults.
- When upgrades are performed, and you elect to copy existing defaults, sysdflt.adf and the 
  other files in ServerLib are copied into the new ServerLib.

ClientLib:
This directory stores the defaults created in Options > Defaults > User Defaults in a file called clientdflt.adf. The defaults and files saved to the ClientLib directory are only available to the particular workstation on which they were created. It is not intended that they be shared on the network. These defaults are merged with the Shared Defaults. Client defaults take precedence over Shared Defaults.

ClientLib is used for local machine defaults only. It is used only if you wish to limit the function or change to ONE computer. For example, the most common usage is to provide Outputs to equipment like a Laser or a Counter Cutter to the computer system that is located in the room where this equipment resides. If the whole design and estimating departments work on the outside of the box, but the die room wishes to view the layouts Die-side, they might set up a User Default so that layouts are opened Die-side.

If you are working on a live system and are trying to build new defaults, create them as User Defaults. Once happy with the results, copy the results into Shared Defaults and delete the User Defaults.

LpLib and DwLib:
- Contain files that are needed for the support of Standards written using LASERPOINT IQ or The Designer WorkBench. The standards shipped with these systems are also included. These may optionally be loaded during the installation of Esko ArtiosCAD.

Program:
- This is where the executable programs and Help files are stored.

Common:
- On License Server machines, contains and shares the license files for all seats. The license file is stored in this \Esko\Artios\Common folder. If License Manager is not being used, this directory contains the file that controls what features are available in Esko ArtiosCAD. This file is called TUNEOPT.TXT or TUNEOPT7.TXT
- This directory also contains some additional programs in the Program folder that are not Esko ArtiosCAD version-dependent.
- In addition, this directory contains the default board texture pictures.

DataCenter:
- This is the database folder.
- The English (or language folder) is a sub-folder of the DataCenter program folder. It contains several important files such as database export and import programs and databaseID.exe.

User Data Files:
- These can be shared, distributed or stored locally at each workstation.
- A DataCenter resource is automatically created for the User Data File destination chosen during installation. So, for example, if each client chooses a local destination folder during installation, a resource will be created for each directory in the Resource Browser of DataCenter Admin.
- In new installations, the \Esko\Artios\Designs folder is created as the default folder for user files. The destination folder selected here is a master resource in DataCenter Admin. If users add sub-folders underneath this folder after the installation, they automatically become DataCenter Admin resources the first time files are saved in them.
- If directories are added to other locations, they need to be manually added to DataCenter as resources. Files saved to these directories will not be recorded in the database unless the directories are resources.
Display adapter optimization in Windows Server™ 2003

As Microsoft Windows Server™ 2003 is a server-class operating system not meant to run on workstations, its video capabilities are reduced by default so that more processing power is available for server applications. If you will be using a machine using Windows Server 2003 as your primary Esko ArtiosCAD workstation, you should adjust its display adapter properties for full hardware acceleration by doing the following:

1. Log on as Administrator or as a user with administrative privileges.
2. Click Start > Control Panel > Display.
3. Select the Settings tab, and then click Advanced.
4. Select the Troubleshoot tab, and slide the Hardware Acceleration slider to Full.
5. Click OK twice to make the changes and close the Display Properties dialog box.

To verify that all methods of display acceleration are enabled, do the following:

1. Click Start > Run.
2. In the Open: field, type dxdiag and click OK.
3. You may be prompted about downloading new WHQL drivers. Click No.
4. In the DirectX Diagnostic Tool dialog box, select the Display tab.
5. Verify that DirectDraw Acceleration, Direct3D Acceleration, and AGP Texture Acceleration are all set to Enabled (if available). If they are not, click Enable.
6. Click Exit to exit the tool.

Operating system notes

If you are using Microsoft Windows Vista Business as the operating system on your computer, file and utility locations may differ from those referenced later in this book.

Table: Changed items or locations

<table>
<thead>
<tr>
<th>Referenced item or location</th>
<th>Changed item or location</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start &gt; Programs</td>
<td>Start &gt; All Programs</td>
</tr>
<tr>
<td>Start &gt; Settings &gt; Control Panel</td>
<td>Start &gt; Control Panel</td>
</tr>
<tr>
<td>Start &gt; Settings &gt; Control Panel &gt; Administrative Tools &gt; Data Sources (ODBC)</td>
<td>Start &gt; Control Panel &gt; Classic View &gt; Administrative Tools &gt; Data Sources</td>
</tr>
<tr>
<td>Start &gt; Settings &gt; Control Panel &gt; Add/Remove Programs</td>
<td>Start &gt; Control Panel &gt; Programs &gt; Programs and Features</td>
</tr>
<tr>
<td>C:\users\prod (legacy user files location)</td>
<td>Any other folder. Windows Vista Business uses this folder to store operating system files.</td>
</tr>
</tbody>
</table>
Support for VMware® Fusion

ArtiosCAD is now supported for use on Apple Macintosh computers using the VMware Fusion virtualization software to simulate IBM-compatible personal computers. There are some restrictions to what is supported, however:

1. Parallel security keys are not supported as Apple Macintoshes no longer have parallel ports.
2. VMWare Fusion supports DirectX 8 while ArtiosCAD requires DirectX 9c. As a result, some features may show unexpected behavior.
3. 3D rendering in Direct3D mode is not supported. Use OpenGL rendering instead. To use OpenGL, do the following:

   • In ArtiosCAD, click **Options > Defaults > Shared defaults > Startup defaults > 3D Rendering Options**.
   • Select **OpenGL** in the **3D Rendering** group.
   • Select **Use OpenGL for bitmaps**.
   • Click **OK**.
   • Click **File > Exit**.
   • Click **Yes** if asked to save changes to Shared defaults.
   • Click **Yes** if asked to overwrite Shared defaults.

---

Referenced item or location | Changed item or location
---|---
SolidWorks 2007 | SolidWorks 2007 is not supported in Windows Vista (any version).
Esko ArtiosCAD Help file | The Windows Help system supplied with Esko ArtiosCAD does not work until you download a patch from Microsoft. Search for WinHlp32.exe at microsoft.com and download the patch.
3. Installing and Configuring ArtiosCAD Enterprise

ArtiosCAD Enterprise installs and operates differently than non-enterprise versions of ArtiosCAD because of its improved Project and document management framework. Those changes are contained in this chapter, and also in the ArtiosCAD Enterprise chapter in the ArtiosCAD User Guide. If you are using a non-enterprise version of ArtiosCAD, please skip this chapter.

Installing ArtiosCAD Enterprise

The list below is a general guide to the steps required to install ArtiosCAD Enterprise. They will be explained in greater detail in the following sections.

1. Install and configure WebCenter on its dedicated servers. Make sure to have access to the WebCenter and licensing documentation at http://help.esko.com, where you can also download the PDF versions for printing and offline use. WebCenter and Licensing and Activation are listed in the Products section.

2. Run the board importer batch file.

3. Load the ArtiosCAD Enterprise Defaults.

4. In WebCenter, do the following:
   • Configure companies and locations for the initial ArtiosCAD Enterprise users.
   • Create custom groups, if desired, for the initial ArtiosCAD Enterprise users.
   • Create the initial ArtiosCAD Enterprise users as Project Managers.
   • Invite new users to the Shared Defaults Project.

5. Install ArtiosCAD Enterprise on users’ computers.


There are additional configuration tasks to do in WebCenter that will make ArtiosCAD Enterprise easier to use. They include:

• Assigning default boards to a location.
• Creating characteristics, attribute categories, attributes, and restricted sets.

Finally, once everything is configured and working, you can set up a third tier of Defaults called Location Defaults. They sit between Shared Defaults and User Defaults. Location Defaults are configured independently for each location, although you can create a Template on which to base them. For more information about using Location Defaults, see the information starting in the Location Defaults Overview section later in this chapter.

Step 1 - Installing Esko WebCenter

The first step of installing ArtiosCAD Enterprise is to install WebCenter 10.2.
Note: Have the WebCenter Complete Documentation manual handy for easy reference. It is in the User Docs subfolder of the ArtiosCAD Enterprise folder on the media.

Installing WebCenter for use with ArtiosCAD Enterprise varies from the procedure in the WebCenter manual in that there are a few more steps required.

Note:
We do not recommend using an Express version of Microsoft SQL Server 2003 or 2008 if you will have more than a few users or if you will be storing documents in the database itself (known as BLOBs) as well as WebCenter’s FileStore. If you plan on either of those situations, we recommend using a full version of Microsoft SQL Server or Oracle.

The list below is a general overview to installing WebCenter. Refer to the WebCenter Complete Documentation manual for specific instructions. In case of a conflict, the information in the steps below takes precedence over the steps in that manual.

1. Load the database server software (Microsoft SQL Server or Oracle) on the database server if it is not already present. If you will be using a centralized corporate database server instead of a dedicated WebCenter database server, you will either need to load database access and management tools on the Application server, or map a drive to the Application server from the database server, in order to run the database scripts batch file that builds the database schema.

2. If any users will be using the WebCenter Viewer to look at files, load the On-Board Graphics Engine. Note that this will require a separate Automation Engine OBGE Server network license to be loaded on the Application server. For more information, see the WebCenter 10.2 Complete Documentation manual.

3. If you have an advanced database configuration and want to use BLOBs in your database, configure BLOBs in WebCenter. Now is the time to make this decision as migrating from a non-BLOB configuration to a BLOB configuration later is not recommended or supported. For more information, see the Configuring BLOBs section.

4. On the Application server, do the following steps as a member of the local Administrators group:
   a) Load ArtiosCAD Enterprise that is on the same media as WebCenter as an Advanced installation. Do not load a different version. If users will need to output PDF files, make sure you have purchased the PDF option and that when selecting ArtiosCAD Programs to install, you select the PDF option.
   b) Load the WebCenter Application server components. It is important to answer all questions correctly to avoid having to edit files later. You may receive a message about two installation programs running at the same time. Click OK if you see this message.
   c) Run the database scripts batch file appropriate for the type of database software you installed. You need to have the database access and management tools for your database program loaded on the Application server, or you need to do this from the database server with a network drive mapped to the Application server.
      If you are using Oracle, you must have write access to the folder containing the database schema batch file as the batch file creates XML files used to create the database schema. You may want to copy the entire DatabaseSchema folder to the database server and run it locally to ensure proper schema creation.
   d) Start the WebCenter services (WebCenter App-X Application Container, WebCenter JBOSS Application Server, WebCenter CAD-X Server, and WebCenter Search Crawler). You may want to set them to start automatically.

5. On the Web server, do the following steps as a member of the local Administrators group:
a) Make sure that Internet Information Services (IIS) is installed and working correctly. See the next section for more information.
b) Load the WebCenter Web server components. It is important to answer all questions correctly to avoid having to edit files later.
c) Start the WebCenter Tomcat service.
d) Log in to WebCenter and change the admin password.

6. On the Application server, start the WebCenter CAD-X Server service.

Configuring BLOBs

ArtiosCAD Enterprise supports the use of BLOBs in the database. BLOBs are Binary Large OBjects - files stored as data in the database. Using BLOBs is transparent for the user.

When you configure ArtiosCAD Enterprise to use BLOBs, it also stores managed documents in the WebCenter FileStore as a cache to improve performance. If you delete a file from the FileStore, the next time you save it in ArtiosCAD Enterprise, it will be recreated in the FileStore from the BLOB.

Note:
You should decide if you want to use BLOBs when you install WebCenter. Switching to using BLOBs after having used ArtiosCAD Enterprise is not recommended or supported.

Editing appconfig.xml on the Application Server

To enable BLOBs, you must change a line in a configuration file on the Application server. On the Application server:

1. Log in as Administrator or as a user with administrative privileges.
2. Make a backup copy of C:\Artios\WebCenter\config\appconfig.xml. This file will be on the C: drive no matter where you installed WebCenter.
3. Open the file named in the previous step in a text editor such as Notepad.
4. Scroll down to <store_files_in_blob value="false"/> and change false to true.
5. Save the file and exit the text editor.

Restarting Services

Once you have changed appconfig.xml on the Application server, restart all the Application server's WebCenter services to start using BLOBs in ArtiosCAD Enterprise.

1. On the Application server, still logged in as Administrator or as a member of the local Administrators group, open the Services applet in the Administrative Tools group in Control Panel.
2. Restart the following services by clicking each one and then clicking the Restart link:
   a) WebCenter JBOSS Application Server
   b) WebCenter CAD-X
3. Close the Services applet and log off the Application server.

ArtiosCAD Enterprise is now configured to use BLOBs. Note that WebCenter will keep copies of managed documents in its FileStore to improve performance.
Installing IIS7 on the Web Server

Installing IIS7 on the Web server for use with WebCenter and ArtiosCAD Enterprise has three steps:

1. Installing the software.
2. Enabling 32-bit applications (only if the Web server is using a 64-bit operating system).
3. Extending the size limit for uploaded files.

Install IIS on the Web Server Machine

If IIS is not installed on your web server, you need to install it. The procedure below explains how to install IIS7 on Windows Server 2008 R2.

1. Start the server manager.
2. Click Roles in the left panel, then click Add Roles at right.

This opens the Add Roles Wizard.
3. In the Select Server Roles screen, select Web Server (IIS).
If the wizard prompts you to add features required for the web server, do so.

4. In the Select Role Services screen:
a) Select ASP, ISAPI Extensions and ISAPI Filters.
   If the wizard prompts you to add role services required for ASP, do so.
   b) Select Logging Tools and Tracing.
   c) Select IIS 6 Management Compatibility.

5. In the Confirm Installation Selections screen, check the settings and click Install.

This installs the IIS 7 Role with the settings you have selected and the default settings.
The Installation Results screen should show Installation succeeded.

Enable 32 Bit Applications

**Note:** Only do this if you have a 64 bit system!

1. In IIS, go to Application Pools and select the DefaultAppPool.
2. Click Advanced Settings..., then set the Enable 32 Bit Applications option to True and click OK.

3. Click Recycle... at right.

Extend the Upload Limitation

IIS7 has a limitation on how much you can upload at once (30 MB). You can change it to 2GB by following these steps:

1. Open IIS 7 SnapIn.
2. Select the website for which you want enable large file uploads.
3. In the main window, double-click Request filtering.
4. Once the window is opened you may see a list of tabs (file name extensions, rules, hidden segments...).

   Regardless of the tab you select, right-click in the main window and select Edit Feature Settings.
5. Modify the Maximum allowed content length (bytes): change the value to 2000000000 (a 2 and 9 zeros) to be able to upload files close to 2GB.

Step 2 - Running the Board Importer Batch File

After you have loaded WebCenter and changed the admin password, the next step is to run the board importer batch file. On the Web server, do the following:

1. Log in as a member of the local Administrators group.
2. Start a Command Prompt window.
3. Change directories to ..\WebCenter\WebServer\Boards.
4. Run the Board Import utility ImportBoards.bat which takes the following arguments.
   a) -username name of the WebCenter user doing the import, usually admin.
   b) -password password of the WebCenter user doing the import.
   c) -url URL to the WebCenter server.
   d) -dir Directory containing the migration files (same directory as the batch file).
   e) -output Specifies the optional creation of an XML file with the returned status of the procedure.

An average command would be something like this: ImportBoards -username admin -password adminpassword -url http://server/WebCenter_Inst -dir C:\Esko\Artios\WebCenter\WebServer\Boards

5. Press Enter after typing the command. The batch file will run and load the ArtiosCAD board information into WebCenter.

Step 3 - Loading the Esko ArtiosCAD Defaults

After you have run the board import batch file, the next step is to install and run the ArtiosCAD Defaults installer.

Note:
The system you use for loading the Defaults must be able to keep the Defaults on it. You will have to follow the same process to load future Defaults and the new installation will reference the older version's Defaults as part of its installation. We recommend using the Application server for this process.

Note:
This procedure takes a significant amount of time. We recommend starting it and doing other things until it completes.

1. Log in to the Application server as a member of the local Administrators group.
2. Insert the ArtiosCAD Enterprise media into the drive.

   The ArtiosCAD Enterprise installer should appear. If it does not, run setup.exe in the root directory of the media.
3. Click the button next to Install ArtiosCAD Defaults.
a) Choose the language in which the setup program will run and click OK.
b) Click Next in the Welcome dialog box.
c) Click I accept the terms in the license agreement and click Next.
d) In the Defaults Language Choice dialog box, select the language of the Defaults to upload to WebCenter and click Next. Only one language may be uploaded per session, so to install more than one language, repeat this procedure.
e) Enter your WebCenter server URL, admin username, and admin password in the WebCenter Server Information dialog box and click Next.
f) In the Shared Defaults dialog box, you can choose a version of ArtiosCAD Standard Edition that is the same version or earlier from which to merge Shared Defaults into Enterprise. To use the delivered Shared Defaults, choose Use installed shared defaults; otherwise, choose Copy shared defaults from this version and select the version. Click Next.
g) Click Install to begin the installation.

A progress indicator will appear.

4. Click Finish when the installation process is done.

**Step 4 - Creating a New Company and Location**

The next step is to create an entry for your company and location that will contain the users.

1. Log in to WebCenter as admin or as a member of the ADMINS group.
2. Click Admin > Companies > New Company.
3. Enter the information about the new company. Fields with asterisks next to them are required to have information in them.
4. Click Create.

A status message saying the company was created successfully appears at the bottom of the page.

To add more locations to this company, click the successful status message to edit the company, and then click Locations > New Location and enter the pertinent information for the new location. Click Add Location and repeat as desired.

**Step 5 - Creating a New Group**

This step is optional.

Group membership is an important administration and security consideration. Only two groups, ADMINS and USERS, are created by default when you install WebCenter. For efficient classification of users, you may want to create different groups so that different users can easily have different Project permissions. On the other hand, you might find it easier for all users to belong to one group and to invite that one group to your Project templates. It all depends on if you want different groups of multiple users to have different abilities. To create a new group, do the following:

1. Click Admin > Groups > New Group.
2. Enter the name of the group.
3. Choose who will be able to see this group: All Project Managers, Administrators Only, or Selected Project Managers.

4. Click Continue.

5. In Step 2: Select Group Members, since you have not created any users yet, click Continue.

6. Is this new group visible to every Project Manager or only to Administrators:
   - If yes, click Finish. The group is created.
   - If no, click Continue. Then, in Step 3: Assign group visibility to project manager(s), select the Project Managers who can see this group and click Finish.

   Project Managers with Full Visibility is unavailable, as they can see all groups. By default, group members can see the group but you can still de-select the group members so that they don’t know this group exists.

Step 6 - Creating New Users

The next step is to create users who will use ArtiosCAD Enterprise. Refer to the WebCenter documentation for more information.

1. Click Admin > Users > New User.

   Step 1: User Info in the Create New User wizard appears.

2. Type the basic user information into the appropriate fields (Username, First Name, Last Name, ...).

   Note: Required fields are indicated with an asterisk *.

3. Do you want to use LDAP authentication (single sign-on) for this user?
   - If yes, enable the Use LDAP to Authenticate User option. This will allow the user to use his regular Windows network user name and password to log on to WebCenter. Using LDAP requires additional configuration in WebCenter. Please see the WebCenter documentation for more information.
   - If no, provide the user’s initial Password and confirm it.

   For ArtiosCAD Enterprise users, do not enable Require user to change the password at first login.

4. Does the user need guaranteed access by means of a reserved license?
   - If yes, enable the option User Has Guaranteed Access.
   - If no, leave the option disabled and proceed to the next step.

   For more information on guaranteed access and its licensing implications, refer to the WebCenter documentation.

5. Does the user need special permissions in WebCenter? All ArtiosCAD Enterprise users should be at least Project Managers with Limited Visibility of Companies and Groups. If they are not, the administrator will need to do additional configuration work described at the end of this procedure.

<table>
<thead>
<tr>
<th>If the user needs...</th>
<th>Select the option...</th>
</tr>
</thead>
<tbody>
<tr>
<td>no special permissions and will not create Projects</td>
<td>Normal User</td>
</tr>
</tbody>
</table>
If the user needs... | Select the option...
--- | ---
Project Manager permissions, but only for Projects and users belonging to his or her own company | Project Manager with Limited Visibility of Companies and Groups.

Project Manager permissions on the complete system | Project Manager with Full Visibility

6. Does the user need the ability to create new task types in WebCenter? ArtiosCAD Enterprise users do not need to create task types.
   - If yes, enable the option User Can Create Task Type.
   - If no, proceed to the next step.

7. To grant this user the ability to create other users without being a member of the ADMINS group, enable the option User Can Create User checkbox.

8. In Step 2: Assign to a Company, select a company and location to which you want to assign the user. Note that this can be very important to filter the content, users, and Location Defaults the new user will be able to see.

9. If you:
   - are creating a Normal User or a Project Manager with Full Visibility, the wizard is complete. Click Finish to create the user.
   - are creating a Project Manager with Limited Visibility, click Continue and proceed to the next step.

10. In Step 3: Assign Visible Companies, select the companies that this user should be able to see, then click Continue.
    The selection you make here filters the list of users in the next step so that this Project Manager can never see users from the companies you did not select in this step. By default, only the company of the Project Manager himself (the one you selected in the previous step) will be selected.

11. In Step 4: Assign Visible Groups, select the groups the user should be able to see, then click Finish.
    Only groups containing members that belong to one of the companies you assigned in the previous step will show up in the list. In addition, some groups may be set to be visible to all users (such as the ADMINS group).
    A status message saying the user was created will appear. Stay on this page for the next step.

If a new user is not made a Project Manager, the user will not be able to use User Defaults unless the administrator does the following for each language and version of ArtiosCAD Enterprise that the user will be using:
   - Creates a Project for this user named ACadDefaults_N.NN_ll_username (where N.NN is the version number, ll is the language code, and username is the user’s WebCenter username).
   - Invites the user to the Project.
   - Uploads clientdflt.adf to the Project.
   - Grants the user View and Download permissions at the Project level. The user will not be able to change the User Defaults with this level of permissions.
   - If the user needs to change the User Defaults, he or she must have Full permission at the Project level.
• Adds the user to the appropriate Location Defaults Project with at least View and Download permissions. If the user needs to change the Location Defaults, he or she must have Full permission at the Project level.

Step 7 - Setting a New User's Group Membership

Once you have created a new user, the next step is to assign that user to one or more groups. Even if you did not create any custom groups, you should assign a new user to the USERS group.

1. While still on the Create New User page with the successful creation status message, Click Add user to group(s).
2. In the Available Groups column, click each group to which you will add this new user. To select more than one group, hold down CTRL while clicking groups.
3. Click >> to assign the selected available groups to the user.

The user is added to the group(s) immediately.

Step 8 - Inviting New Users to the Shared Defaults Project

The next step is to invite new users to the Shared Defaults Project, which is Shared Defaults in ArtiosCAD Enterprise. You can do this either individually for every user or once through Group membership if you added all users to the same group (such as USERS). Inviting them to the Project means that they will be able to use the Defaults. In order for users to change Shared Defaults, they must have Full permissions on the Project and be members of the ADMINS group.

Note:
All ArtiosCAD Enterprise users **must** be invited to the Shared Defaults Project. Users who are not invited to the project will receive a UFANEX error when they launch ArtiosCAD Enterprise.

1. Click Admin > Project Management > Projects.
2. Click the name of the ACadDefaults_N.nn_II_Shared Project (where N.nn is the version and II is the language) to which you are inviting the user.
3. Click Members on the Project menu.
4. Click Add members.
5. In Step 1 of 2: Select Users/Groups, check the checkboxes next to those users and groups to add to the Project.
6. Click Continue.
7. In Step 2 of 2: Project or folder permission rules, assign at least Download permission for the Project and each folder in it. Invitees to Projects get View permission assigned automatically.
8. Click Finish.

The new invitees should now be able to log into ArtiosCAD Enterprise and start using it.
Step 9 - Installing ArtiosCAD Enterprise on a User’s Computer

To install ArtiosCAD Enterprise on a user’s computer, do the following:

1. Log in as Administrator or as a user with administrative privileges.
2. Put the ArtiosCAD Enterprise media into the computer’s media drive. It should automatically launch the Setup program. If your drive is set to not AutoPlay, start Windows Explorer and double-click setup.exe in the root directory of the media.
3. In the Esko ArtiosCAD Setup dialog box, click the icon next to Install Esko ArtiosCAD N.nnll, where N.nnll is the version number and language.
   The installer will launch and a few messages will display.
4. In the Welcome to the InstallShield Wizard for Esko ArtiosCAD N.nnll dialog box, click Next.
5. Read the Software License Agreement in its entirety and choose I accept the terms of the license agreement if you accept all the terms of the License Agreement. Click Next.
6. Choose a Typical installation and click Next. If you want to keep previous versions of the software or change the default location to save user files, choose Advanced and click Next.
   a) In the User Files Folder dialog box, click Change to change the location for unmanaged user files from the location shown. Enter the new location and click OK.
   b) In the Advanced Options dialog box, Keep Previous Versions is the default for previous version handling. It means that ArtiosCAD is installed using a version-specific folder and icon which match the version and language of the installation, and all previous versions of ArtiosCAD are kept. To change this setting, click Change. In the Previous Version Handling dialog box, Overwrite previous versions simply uses ArtiosCAD as the icon name and ArtiosCAD as the directory with no version number. It overwrites the prior version of ArtiosCAD if this is an upgrade and the previous version was installed to a non-version-specific directory. Once you have chosen the option to use, click OK to return to the Advanced Options dialog box.
   c) Click Next to continue.
7. In the Feature Selection dialog box, choose the features to install on the current machine. For each feature, click the icon in the drop-down list box next to it and choose the appropriate choice. Do not click Next when done, but rather proceed to the next numbered step.
   a) Esko ArtiosCAD Licensing controls whether this machine is a license server or not. For ArtiosCAD Enterprise, the WebCenter Application server should be the licensing server, so this choice should be set to This feature will not be available.
   b) Esko ArtiosCAD Programs are the program files needed to run ArtiosCAD Enterprise. You must install these if you want to run ArtiosCAD Enterprise on the current machine. You must leave this choice selected to install ArtiosCAD itself.
   c) Contained within this group are optional components of ArtiosCAD Enterprise that require purchasing licenses for the corresponding modules: PDF Import/Export and 3D Importer. Refer to the printed license report that came with the media to determine which components to select. Installing software components without having the corresponding licenses wastes disc space, but hinders operation if you purchase a license in the future. Not installing the...
software components but having the licenses for them results in features that do not function within ArtiosCAD.

d) **Esko Configuration Manager** collects data from systems running the Esko Station Information Service and sends the collected data to EskoArtwork automatically for use by Support personnel. It sends information only if you configure it to do so and leave it enabled. It is a great aid to Support personnel as they will have required information about your hardware and licenses available should you need to call for assistance. Only one system per site should have the Esko Configuration Manager installed. Refer to the separate document Scope Configuration Manager 1.0 Software Installation and Configuration Manual on the ArtiosCAD media (SCM_nn_SICFM.PDF in the User Docs folder) for more information.

8. While still in the Feature Selection dialog box, with one of the components to be installed selected, review the folder in which ArtiosCAD Enterprise will be installed, and click **Change** if desired. In the resulting Change Current Destination Folder dialog box, change the drive if desired, and click **OK** to return to the Feature Selection dialog box. ArtiosCAD should always be installed to the \Esko\Artios folder on a local drive. Click **Next**.

9. Choose the type of licensing to use from the drop-down list box. You should have separate information that came with the media that describes your licensing. Click **Next**.

10. In the Ready to Install the Program dialog box, review the settings for correctness. If the settings are incorrect, click **Back** and fix them. If they are correct, click **Install**.

11. If this system has other EskoArtwork products on it, you may be prompted to stop their license services. Click **Yes** to proceed with the installation.

12. In the InstallShield Wizard Completed dialog box, click **Finish**.

   If you are prompted to reboot the system, click **Yes**. **It is very important that you reboot the system if prompted.**

For this user to use ArtiosCAD Enterprise, you should have already followed major installation steps 6 through 8. If not, perform them now. Once done, the user should be ready to use ArtiosCAD Enterprise. Repeat for other users as needed.

### Post-Installation Configuration Tasks

There are two post-installation configuration tasks you should do in WebCenter that make ArtiosCAD Enterprise more useful in terms of proper categorization of Project information:

- Assign default boards to a location.
- Create characteristics, attribute categories, attributes, and restricted sets.

#### Assigning Default Boards to a Location

When users create a design, they have to assign a board to it. Boards that are assigned to their location are shown by default. If no boards are assigned to their location, every time they create a design, they have to click **Show All Boards** to view the actual boards in their folders; otherwise, ArtiosCAD Enterprise shows a list of empty folders.

To assign default boards to a location, do the following:

1. Log in to WebCenter as a member of the ADMINS group.
2. Click **Admin > CAD > Boards**.
3. Expand the board tree by clicking the plus signs (+) in front of the categories.

4. To assign a particular board to a location, do the following:
   a) Click its name.
   b) Check the checkbox to the left of the location to which you are assigning the board. You may check more than one checkbox to assign the board to multiple locations.
   c) Click Finish.
   d) Click Boards to return to the Board tree.

Characteristics
Characteristics are descriptive tags that can be assigned to Projects and documents to make them easier to find.

Characteristics are hierarchical. This means there can be different levels of characteristics for various product lines so that you can drill down through the information to find the exact characteristic desired when searching the database.

For example, you could create a master characteristic for Electronics, and then various sub-characteristics for types of appliances, entertainment equipment, or computers, and so forth. They are presented in a tree format - main categories with expandable branches. The characteristics tree configures the available characteristic names as well as how they are organized.

Create a Characteristic
1. Click Admin > Characteristics.
2. Do you want to add a primary characteristic (a main category) or a secondary characteristic?
   • To add a primary characteristic, select the option button next to Root at the top of the tree and then click New Characteristic.
   • To add a secondary characteristic, select the option button for the characteristic under which to add the new one, and then click New Characteristic.

   There can be up to 17 levels of characteristics or at most 256 characters for the entire characteristics path from the root characteristic to the deepest-level characteristic.
3. Enter the name of the characteristic in the Name field.
4. Click Create. The new characteristic appears in the list.

Edit or Delete a Characteristic
When deleting a characteristic, WebCenter checks if the characteristic or any of its child characteristics are assigned to any Projects or documents. If such a dependency is found, you are prompted to remove the dependency and delete anyway, or cancel the deletion.

1. Click Admin > Characteristics.
2. Do one of the following:
   • To edit a characteristic, click its name in the tree structure. Type a new name in the Edit Characteristic page and click Save to confirm.
   • To delete a characteristic, select it in the tree and click the Delete Characteristic link.

   If there are no dependencies, it is deleted immediately. Note that any sub-characteristics of the selected characteristic will also be deleted.
Project and Document Attributes

Basic concepts

Attributes are a way to assign user-configurable fields and their values to documents and Projects. They can be restricted to a group of preset values (as Restricted Sets), or can be prompted so that the user has to set a value each time they are used. The Restricted Sets must exist before you can use them when creating an Attribute.

Attribute Categories group Attributes together for convenience. An Attribute must be put into an Attribute Category before it can be used; however, the same Attribute can be in multiple categories. You must create an Attribute Category before adding Attributes to it. In ArtiosCAD Enterprise, you can define default Attribute Categories for each type of native design document that are applied to documents as you create them.

Suggested workflow

A suggested workflow is to:

<table>
<thead>
<tr>
<th>Step</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Create Attribute Categories.</td>
</tr>
<tr>
<td>2</td>
<td>Create Restricted Sets (if required).</td>
</tr>
<tr>
<td>3</td>
<td>Create the Attributes.</td>
</tr>
<tr>
<td>4</td>
<td>Assign Restricted Sets to Attributes (if required).</td>
</tr>
<tr>
<td>5</td>
<td>Add the Attributes to the Attribute Categories.</td>
</tr>
<tr>
<td>6</td>
<td>Assign default Attribute Categories to design document types in Defaults in ArtiosCAD Enterprise.</td>
</tr>
</tbody>
</table>

Create an Attribute Category

1. Click Admin > Attributes > Attribute Categories > New Category.
2. Enter the name of the Attribute Category to create, such as Shipping Instructions.
3. Click Create.

Edit or Delete an Attribute Category

WebCenter checks if an Attribute Category is being used before deleting it. If it is in use, you will be prompted to either clear the category from the Projects or documents using it and delete the category, or to cancel the deletion.

1. Click Admin > Attributes > Attribute Categories.
2. Do one of the following:
To edit an attribute category, click its name. Type a new name in the **Edit Category** page and click **Save** to confirm.

To add attributes to a category or remove them from it, click its name. Then use the chevron (<< or >>) buttons to move them between the **Available Attributes** and **Current Attributes** columns.

**Note:** Moving Attributes between columns takes effect immediately.

To delete an attribute category, select it, move the mouse over the menu symbol and click **Delete Category** on the hover menu.

If there are no dependencies, it is deleted immediately. Note that any sub-attributes of the selected attribute will also be deleted.

---

**Create a Restricted Set**

1. Click **Admin > Attributes > Restricted Sets**.
2. Enter information in the fields as appropriate:
   - **Name.** The name of the Restricted Set, up to 35 characters.
   - **Type.** The data type of the Restricted Set. There are six allowed types: Area, Distance, Float, Integer, Text, Volume, and Weight. A float is a number containing a decimal point and an integer is a whole number. The labels in parentheses after the name of the type indicate the units of measurement used in Imperial and metric systems.
   - **Value.** An entry for the restricted set. Text values can be up to 255 characters long. Click **Add** to add the value to the set and clear the Value field. Enter as many values as desired, clicking **Add** after each one.
3. Click **Create** to create the Restricted Set.

**Edit a Restricted Set**

1. Click **Admin > Attributes > Restricted Sets**.
2. Either filter or click **Go** to show the Restricted Sets.
3. Click the name of the Restricted Set to edit.
4. Change the name, or add and remove values as desired. You cannot change the type. If you remove a value, the next time you access a Project using the deleted value, you will be prompted to change it.
5. Click **Change** to make the changes.

**Delete a Restricted Set**

WebCenter checks if a Restricted Set is being used before deleting it. If it is in use, you will be prompted to either clear the Restricted Set from the Attribute using it, or to cancel the deletion.

1. Click **Admin > Attributes > Restricted Sets**.
2. Either filter or click **Go** to show the Restricted Sets.
3. Select the checkbox of the Restricted Set to delete.
4. Hover over the menu symbol and click **Delete Restricted Set(s)** from the hover menu.
5. Click **OK** when asked to confirm the deletion.
6. If there are no dependencies, it is deleted immediately. If there are any dependencies, choose the desired option and click OK.

Create an Attribute

1. Click **Admin > Attributes > New Attribute**.
2. Enter information in the fields as appropriate:

<table>
<thead>
<tr>
<th>In the field...</th>
<th>Fill in the...</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>name of the attribute (for example, Billable)</td>
</tr>
<tr>
<td>Description</td>
<td>description of the attribute contents (for example, Indicates whether work on this project should be billed by finance)</td>
</tr>
<tr>
<td>Type</td>
<td>data type of the attribute. Floats are numbers containing a decimal point (3.1415) and integers are whole numbers (42).</td>
</tr>
<tr>
<td>Values</td>
<td>choose Prompted if you want to allow the user to enter any value, Calculated to enter an ArtiosCAD formula for calculating the value of the Attribute when ArtiosCAD opens the document containing the Attribute (the values are read-only in WebCenter), or select a Restricted Set to limit the possible values. To choose a Restricted Set, it must be created before defining the Attribute. Make sure that the type of the Restricted Set is the same as the Attribute type.</td>
</tr>
</tbody>
</table>

3. Click **Create**.

Edit or Delete an Attribute

When deleting an attribute, WebCenter checks if the attribute or any of its child attributes are assigned to any Projects. If such a dependency is found, you are prompted to remove the dependency and delete anyway, or cancel the deletion.

1. Click **Admin > Attributes**.
2. Do one of the following:
   - To edit an attribute, click its name. Type a new name in the **Edit Attribute** page and click **Save** to confirm.
     You cannot change the attribute type.
   - To delete one or more attributes, select the corresponding checkboxes, hover over the menu symbol and click **Delete Attribute(s)** on the hover menu.
     If there are no dependencies, it is deleted immediately.
Assigning Default Attribute Categories to Document Types

ArtiosCAD Enterprise comes with three empty Attribute Categories: DEFAULT A3D CATEGORY, DEFAULT DESIGN CATEGORY, and DEFAULT MANUFACTURING CATEGORY. Create and add Attributes to these categories as described previously, or create your own Attribute Categories as desired. Once you have the Attributes Categories populated as desired, do the following to assign default Attribute Categories to document types:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Options > Defaults.
3. Expand the Database Defaults catalog by clicking the plus sign (+) next to it.
   The Default Attributes Categories dialog box appears.
5. Select the desired categories from the drop-down list boxes and click OK.

6. Save and exit Defaults.

If you have added Attributes to the default categories but the drop-down list boxes are empty, or if you do not see the Attribute Categories you recently created, exit Defaults and use the Cache Manager to update the Attributes table.

Configuring ArtiosCAD Enterprise

ArtiosCAD Enterprise gets its board information from the database. The Board and Flute browsers let you manage paper boards, plastic boards, and foams.
Outputs in ArtiosCAD Enterprise may have their output directed to a Project in addition to a device or a file.

**Using the Flute Browser**

Use the Flute browser to create and edit flutes, which are the wavy papers glued between liners to form corrugated board. ArtiosCAD Enterprise comes with several predefined flutes.

**Opening the Flute Browser**

To open the Flute browser, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click **Database > Manage Boards > Flute browser**.

   The Flute browser opens.

The Flute Browser Toolbar

Shown below is the Flute browser toolbar.
### Working with Flutes

#### Viewing Flute Details

To view details about a flute:

<table>
<thead>
<tr>
<th>You can...</th>
<th>Double-click the flute to select it and open the Details pane.</th>
</tr>
</thead>
<tbody>
<tr>
<td>You can...</td>
<td>Click the Details pane toggle on the toolbar and then select the flute.</td>
</tr>
</tbody>
</table>

The Flute browser will then look like this:
If the Details pane is already open, simply select a flute to see its details.

**Editing a Flute**

To edit a flute, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click **Database** > **Manage Boards** > **Flute browser** to open the flute browser.
3. Select a flute and view its details.
4. Click **Edit**.
5. Change the values as desired.
6. Click **Save**.

**Adding a Flute**

A flute is defined by four pieces of information:

- A unique code to identify the flute. This code must be four characters or shorter.
- The flute pitch, which is the horizontal distance between the lowest parts of the wave. The pitch must be greater than 0.
- The flute height. The height must be greater than 0.
• The take-up factor, which is the ratio between the length of the fluted paper and the length of the board. If the flute were stretched out, it would always be longer than the paper encasing it. The take-up factor must be greater than 1.

To add a flute, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Manage Boards > Flute browser to open the flute browser.
3. Click Insert on the toolbar.
4. In the Details pane, enter the code, the height, the pitch, and the takeup factor in the appropriate fields.
5. Click Insert.

The new flute will appear in the list.

Deleting a Flute
To delete a flute, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Manage Boards > Flute browser to open the flute browser.
3. Select the desired flute.
4. Click Delete on the toolbar.
5. Click Yes to confirm the deletion.

The deleted flute is removed from the list.

Using the Board Browser

Use the Board browser to maintain the corrugated boards, folding carton boards, papers, and foams used in ArtiosCAD Enterprise.

Opening the Board Browser
To open the Board browser, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Manage Boards > Board browser.

The Board browser opens with all the catalogs closed. Click the plus sign next to each folder to open it. Some folders are opened in the following example.
The Board Browser Toolbar

Shown below is the Board browser toolbar.

<table>
<thead>
<tr>
<th>Button</th>
<th>Action</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Insert" /></td>
<td>Inserts a new board.</td>
</tr>
<tr>
<td><img src="image" alt="Delete" /></td>
<td>Deletes the selected board.</td>
</tr>
<tr>
<td><img src="image" alt="In" /> &amp; <img src="image" alt="mm" /></td>
<td>Toggles units between metric and imperial.</td>
</tr>
<tr>
<td><img src="image" alt="Details" /></td>
<td>Toggles display of the Details pane.</td>
</tr>
</tbody>
</table>
Working with Boards

Viewing Board Details
To view details about a board:

You can...

Double-click the board to select it and open the Details pane.

You can...

Click the Details pane toggle on the toolbar and then select the board.

The Board browser will then look like this:

If the Details pane is already open, simply select a board to view its Details.

Editing a Board
To edit a board, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Manage Boards > Board browser to open the board browser.
3. Select a board and view its details.
4. Click Edit.
5. Change the values as desired.
6. Click Save.

Creating a paper

Papers are the building blocks of boards so you must create them before you can create boards.

To create a new paper, first examine an existing paper to determine what information you will need to define the new paper. Papers are in their own folder in the Board browser. Usually the caliper, inside loss, outside gain, rounding value, basis weight, and basis cost are needed. You may also set inside and outside colors and other appearance attributes to the paper that appear in the optional 3D module.

To create a new paper, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Manage Boards > Board browser to open the board browser.
3. Open the Artios folder by clicking the plus sign next to it.
4. Click the word Papers to select it.
5. Click Insert on the toolbar.

   The Details pane will automatically appear with the fields enabled for editing.
6. Enter values in the fields on the Details tab as appropriate.
   a) **Board Code** is the identifier used by the database to identify this new paper; each paper must have a unique code.
   b) **Description** is the description of the paper.
   c) **Caliper** is the thickness of the paper.
   d) **Inside Loss** and **Outside Gain** are the allowances used when folding the paper or when gluing it together (such as in a glue flap). They are generally half the caliper.
   e) **Rounding Value** sets a value to which dimensions may be set to round to when constructing a corrugated design. It is more relevant for boards as a whole rather than individual papers.
   f) **Basis Weight** is the weight of the paper in local currency and units of measurement.
   g) **Basis Cost** is the cost of the paper in local currency and units of measurement.
   h) **Test Value** is the amount of weight needed to break the material when performing very specific standardized tests. It is more relevant for corrugated boards as a whole than individual papers.
   i) **Test Code** is the identifier used in correlation with the Test Value to show the strength of the paper. It is more relevant for corrugated boards than individual papers.
   j) **Weight** and **Cost** in the **Adhesive** group should be set for the complete board definition, not individual paper and flute definitions.

7. Leave the fields on the Papers tab blank, as they are not relevant.

8. On the Material Properties tab are fields to set the shininess, transparency, and inside and outside colors of the paper. These properties are only visible in the optional 3D module.
a) Set the Shininess% and Transparency% fields as desired. The Transparency% field is a separate setting from transparency mode in 3D and is not affected by that setting.

b) Surface colors for papers are split into three components: ambient color, which reflects the ambient light; diffuse color, which reflects the movable lights in an airy, non-shiny way; and specular color, which reflects movable lights for a shiny surface, with a shiny surface having a white specular color and a dull surface a black specular color.

c) By default, the Material Properties tab only displays Diffuse color in Simple color mode. In Simple color mode, choose an RGB color by clicking Select color, or choose a CYMK color by clicking CYMK color. To set the other two colors, click Advanced color, and then click Select color as desired. You may only set a CMYK color for the Diffuse color in Simple color mode.

d) The outermost Outside paper color is used to set the Outside board color if no other color has been manually chosen for the outside of the board. Likewise, the innermost Inside paper color is used to set the Inside board color if no other color has been manually chosen for the inside of the board.

e) The ... button (Browse) is used in board definitions to calculate the color based on the materials comprising the board. It is not relevant for papers.

9. On the Texture tab are fields to specify the graphics that may be used to represent the inside, outside, and edge textures of the paper. These properties are only visible in the optional 3D module. Outside and inside textures are more relevant for boards, but may also be defined for papers, while board edge textures do not apply to papers.
a) To add an inside or outside texture to the paper, click the checkbox for it, and then specify the location of the graphic file for the texture, or click ... (Browse) to the right of the filename field and select it. Some common textures are stored in the **Boards** folder of the Shared Defaults Project. The image should be of the board surface with vertical grain or corrugation direction. If you use a custom image, measure the sample when you take its picture so that you know its size; you may need to touch up the image in a graphic editing program so that the edges blend together when the image is tiled.

b) In the **Image height** field, enter the height of the image. The size needs to be set in order for ArtiosCAD to scale the picture appropriately. If the height is set to 0, the image will be scaled proportional to the width.

c) Set the value in the **Image width as number of flutes** drop-down list box to the number of flutes the image represents. The image should be of a whole number of flutes. It may also be set to **Proportional to height**, in which case the height cannot be set to 0.

d) Shown below is **boardbrown.jpg** from the **Boards** folder of the Shared Defaults Project, an example board texture three flutes wide:
When you have finished defining the paper, click **Insert** to add the new paper definition.

**Note:**
Foams are defined in the same way as papers, just with larger dimensions.

**Adding a Board**

To add a board, do the following:

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click **Database > Manage Boards > Board browser.**
3. Click the plus signs next to folders to open them.
4. Boards are hierarchical. Right-click an item a level above the desired level for the new board (such as **Boards** to add the new board at the top level or **Artios > Corrugated Boards > C** to add a new C flute board) and click **New > Board.** (You may also create a new folder in this manner.)
   - You may also select the parent item and then click **Insert** on the toolbar to create a new board.
   To create a new folder, you must use the right-click method described above.

The Details pane will automatically appear with the fields enabled for editing.

5. Enter values in the fields on the Details tab as appropriate.
a) **Board Code** is the identifier used by the database to identify this new board; each board must have a unique code.

b) **Description** is the description of the paper.

c) **Caliper** is the thickness of the paper.

d) **Inside Loss** and **Outside Gain** are the allowances used when folding the paper or when gluing it together (such as in a glue flap). They are generally half the caliper.

e) **Rounding Value** sets a value to which dimensions may be set to round when constructing a corrugated design. It is more relevant for boards as a whole rather than individual papers.

f) **Basis Weight** is the weight of the board in the selected system of units of measurement. Clicking the ... (Browse) button at the end of the field displays the Calculated Basis Weight dialog box. In this dialog box is an analysis of the calculated weight of the board based on the papers used to make it. To replace a manually-entered weight with the calculated weight, click **Accept**.

g) **Basis Cost** is the cost of the board in local currency and units of measurement. As with the **Basis Weight** field, the ... (Browse) button at the end of the field displays an analysis of the calculated cost of the board based on the papers used to make it. To replace a manually-entered cost with the calculated cost, click **Accept**.

h) **Test Value** is the amount of weight needed to break the material when performing very specific standardized tests.

i) **Test Code** is the identifier used in correlation with the Test Value to show the strength of the paper.

j) The **Weight** and **Cost** fields in the **Adhesive** group are the respective weight and cost of the adhesive for the board as a whole.

6. The Papers tab contains drop-down list boxes to select the liners and mediums that comprise the board.

   a) If you know it already, type the flute code of the new board in the **Board Flute** field. At the end of the field is a ... (Browse) button that opens the Calculated Flute dialog box. Use this to calculate the flute code after defining the liners and mediums for the board. If it is a single board, expect a flute code like C. Double wall could be BC, while triple- and quadruple-wall would comprise four letters.

   b) Boards are defined sequentially from the outside inward. For example, you must select Liner 2 before you may select Medium 2. You may not select Liner 2 and then jump directly to selecting Liner 4.
c) Select liners and mediums in their respective drop-down list boxes. As you select a liner and then a flute, the Flute drop-down list box for that pair becomes available. You must select a flute before the next liner becomes available for selection.

d) Complete the paper selection process with a liner.

7. On the Material Properties tab are fields to set the shininess, transparency, and inside and outside colors of the board. These properties are only visible in the optional 3D module.

![Material Properties Tab]

a) Set the Shininess% and Transparency% fields as desired. The Transparency% field is a separate setting from transparency mode in 3D and is not affected by that setting.

b) Surface colors for papers are split into three components: ambient color, which reflects the ambient light; diffuse color, which reflects the movable lights in an airy, non-shiny way; and specular color, which reflects movable lights for a shiny surface, with a shiny surface having a white specular color and a dull surface a black specular color.

c) By default, the Material Properties tab only displays Diffuse color in Simple color mode. In Simple color mode, choose an RGB color by clicking Select color, or choose a CMYK color by clicking CMYK color. To set the other two colors, click Advanced color, and then click Select color as desired. You may only set a CMYK color for the Diffuse color in Simple color mode.

d) The outermost Outside paper color is used to set the Outside board color if no other color has been manually chosen for the outside of the board. Likewise, the innermost Inside paper color is used to set the Inside board color if no other color has been manually chosen for the inside of the board.

e) The ... button (Browse) calculates the color based on the materials comprising the board. Click Yes to use the color of the paper; click No to use the color of the current board definition.

8. On the Texture tab are fields to specify the graphics that may be used to represent the inside, outside, and edge textures of the board. These properties are visible only in the optional 3D module.
a) To add an inside or outside texture to the board, click the checkbox for it, and then specify the location of the graphic file for the texture, or click ... (Browse) to the right of the filename field and select it. Some common textures are stored in the Boards folder of the Shared Defaults Project. The image should be of the board surface with vertical grain or corrugation direction. If you use a custom image, measure the sample when you take its picture so that you know its size; you may need to touch up the image in a graphic editing program so that the edges blend together when the image is tiled.

b) In the Image height field, enter the height of the image. The size needs to be set in order for ArtiosCAD to scale the picture appropriately. If the height is set to 0, the image will be scaled proportional to the width.

c) Set the value in the Image width as number of flutes drop-down list box to the number of flutes the image represents. The image should be of a whole number of flutes. It may also be set to Proportional to height, in which case the height cannot be set to 0.

d) If using a custom board edge texture on a double, triple, or quadruple wall board, the board edge texture has to be a whole number of all the flutes. The largest flute is used initially, then the smaller flutes are adjusted by a few percent to align a whole number of the smallest flutes with a whole number of the largest flute. Knowing how many flutes the image represents is critical. To determine this, start with the board defined but with no custom edge texture. Export the 3D workspace to VRML and then view the associated PNG files created. One of those will be of the board edge texture. Count the number of flutes for the largest board and set the Image width as number of flutes field to the same number.

e) Shown below is boardbrown.jpg from the Boards folder of the Shared Defaults Project, an example board texture three flutes wide:
f) To specify a graphic for the board edge texture, check the **Board edge texture** checkbox, then specify the filename of the graphic file for the texture, or click ... (Browse) to the right of the filename field and select it. Some common textures, for example paper corrugated edges, plastic corrugated edges, and paper hexacomb flutes, are stored in the **Boards** folder of the Shared Defaults Project. The height of the image for a corrugated board should be the board caliper, and the width should be a whole number of flute pitches. Folding carton boards use an image width proportional to the height.

g) If no texture is specified for a corrugated board edge, a default picture based on the height and pitch of the flutes is used.

h) **Stretch like corrugations** stretches the picture of the board edge texture depending on its angle to the grain/corrugation direction. If this option is not selected, the board edge texture is repeated evenly.

i) Shown below is a picture of the default board edge texture with **Stretch Like Corrugations** turned on in a design in 3D:

![Default Board Edge Texture with Stretch Like Corrugations](image)

j) Shown below is a picture of the BrownB board in the Textured Corrugated folder with **Stretch Like Corrugations** turned off in a design in 3D:

![BrownB Board with Stretch Like Corrugations Turned Off](image)
9. On the Locations tab, set those locations for which this will be a default board. Users at locations that are not checked in this step will have to click Show All Boards every time they create a design in order to see this board.

   a) Check Show All Locations to see all the locations defined.
   b) Click Edit.
   c) Select the checkboxes next to the locations for which this will be a default board. Use the buttons on the right side of the dialog box to select or clear all locations.
   d) Click Save to set the locations in which this board will be visible.

10. When you have finished defining the board, click Insert to add the new board definition.

Renaming, Deleting, Moving, and Copying Board Entries

The Board browser supports multiple item selection, dragging and dropping of entries, cutting and pasting of entries, and right-clicking entries to perform functions on the context menu.

To create a folder, select its parent item (the one a level above its desired position), right-click it, and click New > Folder on the context menu. The folder name must be unique among same-level folders in the same parent folder.
To rename a folder, select it, right-click it, click Rename on the context menu, enter the new name, and press Enter.

To delete a folder, first make sure it is empty, and then select it, right-click it, and click Delete on the context menu. You may not delete a non-empty folder.

To rename a board, edit it and change its description. This change will not be communicated back into unmanaged documents using this board.

To move a folder or a board, drag it from one location to another.

To copy a board definition, select it, right-click it, and click Copy on the context menu. Right-click the folder into which it will be pasted and click Paste on the context menu. When the board is pasted into the new folder, the current system time will be appended to the board code to maintain board code uniqueness.

Defaults

Defaults for ArtiosCAD Enterprise have only a few operational differences from other versions of ArtiosCAD.

- All users must be invited to the Shared Defaults Project for each version and language of ArtiosCAD Enterprise they will be using. See the major Step 8 section of the installation instructions for more information.
- For users other than admin to edit Shared Defaults, they must have Full permissions on the Shared Defaults Project and be members of the ADMINS group. To create or change Location Defaults or User Defaults, they do not have to be members of ADMINS, but they must have View, Add, Update, and Lock permissions on the entire Project.
- As a best practice, we strongly recommend that you place new Defaults reference documents into the ServerLib folder (or a new unique user-defined folder) of the Shared Defaults Project for documents which need to be shared by all users. Please note that document names must be unique within a Project. For new Defaults reference documents which will only be used by you, place these in a new user-defined folder of your User Defaults Project. If you must change an existing ArtiosCAD-installed Defaults reference document, we recommended that you copy it as a new document to the ServerLib folder and then update the Defaults to reflect the new document name. Following these recommendations gives ArtiosCAD Enterprise the best chance of properly managing upgrades.
- If you change the username of a user, you must change the name of that user’s associated Defaults project to match.

Browser Options

You can set browser options by clicking Options > Defaults > Browser Settings > General Information. As admin you can change the Shared Defaults, but in most cases as any user you can change your own User Defaults.
Click OK to return to Defaults once you have set the options as desired.

**General Database Options**

You can set database options by clicking **Options > Defaults > Database Defaults > General Options**. As admin you can change the Shared Defaults and Location Defaults, but in most cases as any user you can change your own User Defaults.
The options in the **Database Information Dialog Options** group control when you are prompted to enter database information about a document: when you create it, when you save it, both, or neither. If you do not select either option, users must click **Database > Information** while working on a document to set its database information.
In the **Board Options** group, you can choose to check to see if a board code has changed when users open documents, and you can choose how they are sorted in the Board Catalog.

The options in the **Cache Options** group control how often ArtiosCAD Enterprise updates the cache. See the discussion on using the cache manager in the User Guide for more information about the cache.

In the **Document Thumbnail Options** group, *Create document thumbnails on server controls* whether ArtiosCAD Enterprise creates document thumbnails (deselected, the default choice) or if WebCenter creates the thumbnails (selected).

Choose the thumbnail format in the **Design Thumbnail Format** group.

The **New Project Options** group has a **Template** drop-down list box. Use this to set the default Template ArtiosCAD will use for new Projects you create through the browser. However, this does not apply to Projects used for Defaults.

The settings in the **Comment Options** group control how ArtiosCAD manages comments when saving a managed document.

To turn on the automatic commenting, check **Preseed comment from previous version**. This also enables the options in the **Initial Comment** group.

**Use document description** copies the document description to the comment field. If there is no document description, ArtiosCAD will prompt you for a comment. This will also cause the Project description to seed the document description when you save a managed document for the first time so that on the next save, the comment will be able to be seeded from the description.

**User defined** lets you enter a custom string that ArtiosCAD will use for the first comment.

**Sending an Output to a Project**

In ArtiosCAD Enterprise, you can direct an Output to go to a Project instead of a device or file.

**Destination Tab in an Output Definition**

Outputs that create files can send those files to Projects rather than directories by choosing the appropriate options on the Destinations tab.
When you select Output To Project, many options become available for setting Project and document names and properties. If you choose any of the prompt options, when you run this Output, the Project Browser appears so you can navigate to the desired Project, set the document and folder names, and enter any prompted information.

If Automatically Open is selected in an Output that goes to a Project, a temporary version of the Output is downloaded to your computer and the appropriate program for the type of Output opens it. In the example above, the PDF file is saved to the Project, and then a temporary copy is downloaded and opened by the application registered to open PDFs on your system.

Setting the Title Bar Display Options

You can choose which revision and version information ArtiosCAD shows in the title bar. The Title Bar Options dialog box in Options > Defaults > Startup defaults presents different formatting options. Choose the preferred option and click OK.
Location Defaults Overview

Location Defaults are an intermediate level of Defaults between User Defaults (used first) and Shared Defaults (used last). Each user must be assigned a location and be invited to the appropriate Location Defaults Project in order for that user to use Location Defaults.

A user who is a member of the ADMINS group must do the initial creation and configuration of the Location Defaults for each location.

Members of ADMINS may also control which categories of Location and User Defaults users may set by using Defaults Availability, covered later in this section.

There are three steps involved in configuring Location Defaults:

1. Perform the Location Defaults Setup on the Options menu logged in as a member of ADMINS with a location set.
2. Use Defaults Availability on the Options menu to set which Defaults are available to users.
3. Add Defaults as desired to Location Defaults in the same way as you would to User Defaults or Shared Defaults.
4. Invite the users of a location to the Location Defaults Project and set their permissions. For non-ADMINS users to be able to create Location Defaults, they must have View, Add, Update, and Lock permissions on the entire Location Defaults Project.

Location Defaults Setup

Only members of ADMINS can set up Location Defaults. Location Defaults may optionally be based on a Template that has the desired members, permissions, and contents already set.

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Database > Change Location and choose the company and location for which you want to set up Location Defaults. This only temporarily assigns your company and location for this ArtiosCAD session.
3. Click Options > Location Defaults Setup.
4. Select the appropriate location and click Create Defaults.
5. In the Create New Project dialog box, change the description and choose a Template, if desired, and click OK. You may not change the name of the Project, and any invalid characters in the company or location name are replaced with underscores (_). Invalid characters for Project names are ", *, ;, /, ?, <, >, =, [ ], ^, @, and a space. See the next section for information about creating a Template to use with Location Defaults.

After it creates the new Location Defaults Project, ArtiosCAD changes the Location Defaults Setup dialog box so that the other commands are available.
Open in Web Browser opens this Location Defaults Project in WebCenter using your default Web browser. You can then invite members, configure permissions, and so forth.

You can also migrate existing ArtiosCAD Standard Edition Shared Defaults or User Defaults into Location Defaults. Note that for User Defaults, you must currently be using the machine holding the User Defaults in order to migrate them.

Now when you open Defaults in ArtiosCAD, you can add items to the Location Defaults pane just as you would for Shared Defaults or User Defaults.
Creating a Location Defaults Project Template

To create a Template to use as the basis for Location Defaults Projects, do the following:

1. Log into WebCenter as a member of ADMINS.
2. Create a new Project to serve as the Location Defaults Template.
3. Invite users to the Project and set their permissions.
4. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
   a) Click Options > Defaults, click User defaults to activate that pane, and then click File > New to create a new set of Defaults.
   b) In the New defaults pane, create new Defaults as desired by dragging and dropping from Shared Defaults or by creating new ones by right-clicking.
   c) Make any desired changes to these new Defaults.
   d) Click File > Save As > File and save the file locally as locationdflt.adf. It must have this exact name.
   e) Click File > Exit to exit Defaults and return to ArtiosCAD.
5. In WebCenter, upload the file you created in the previous step to the root level of the Project serving as the Location Defaults Template.
6. Save this Project as a Template. It is now ready to be selected when you perform Location Defaults Setup for a location.
Migrating Defaults from ArtiosCAD Standard Edition to ArtiosCAD Enterprise

After you have set up Location Defaults, you can choose to migrate Shared Defaults or User Defaults to the new Location Defaults.

Note:

Migrating existing Shared Defaults or User Defaults to Location Defaults will overwrite any existing Location Defaults. It is a good idea if your current Location Defaults have data in them to back up your Location Defaults using the browser. Drag the current Location Defaults Project to a location in the Explorer node before you begin so that you have them as a backup.

1. Still in the Location Defaults Setup dialog box, click either Migrate Shared Defaults to Location or Migrate User Defaults to Location. Note that the potential User Defaults must be from the current machine.

2. Click Yes in the warning dialog box to migrate the Location Defaults.

3. In the Defaults Migration Dialog box, choose the version of ArtiosCAD containing the desired Defaults and click OK. This process may not be canceled or aborted, so proceed with caution if you are unsure and click Cancel if you have any doubts.

4. A progress bar will appear as ArtiosCAD uploads the Defaults to WebCenter.
5. When the upload is complete, you will return to the Location Defaults Setup dialog box. Click **Open in Web Browser** to go to the Location Defaults Project in WebCenter, or click **Close** to return to ArtiosCAD.

Two important notes about the Defaults migration process:

- When you migrate Defaults from ArtiosCAD Standard Edition to Location Defaults in ArtiosCAD Enterprise, data files that the Shared or User Defaults reference remain on the local computer. The Defaults will continue to work as long as the files on the local computer exist. To get them fully into Location Defaults without relying on the local computer, use the Explorer node of the browser to manually drag any referenced files from ServerLib into the appropriate folders inside the Location Defaults Project, and then update the references in Location Defaults to point to the correct documents.

- Likewise, if the User Defaults you migrated reference ArtiosCAD Standard Edition Shared Defaults, the referenced information is not migrated. For example, if you have a custom toolbar defined in User Defaults that reference Outputs that use files (such as Reports) or geometry macros, the toolbar itself will be migrated into Location Defaults but the Report files and geometry macros will not be. Use the Explorer node of the browser to manually drag any referenced files from ServerLib into the appropriate folders inside the Location Defaults Project, and then update the references in Location Defaults to point to the correct documents.

### Migrating User Defaults from ArtiosCAD Standard Edition to ArtiosCAD Enterprise

If users who are Project Managers decline to migrate their User Defaults from ArtiosCAD Standard Edition to ArtiosCAD Enterprise upon their first launch of ArtiosCAD Enterprise, they can manually start the migration afterward.

**Note:** Users who are not Project Managers do not have this menu command available as since they may not create their User Defaults Projects themselves. Instead, they must have a member of ADMINS create their User Defaults Projects, invite them to their User Defaults Projects, and set their permissions to at least Delete and Upload.

1. Logged in as the appropriate user, click **Options > Migrate User Defaults from Conventional ArtiosCAD**.
2. Click **Yes** in the warning dialog box to migrate the User Defaults.
3. In the Defaults Migration Dialog box, choose the version of ArtiosCAD containing the desired Defaults and click **OK**. This process may not be canceled or aborted, so proceed with caution if you are unsure and click **Cancel** if you have any doubts.
4. A progress bar will appear as ArtiosCAD uploads the Defaults to WebCenter and will disappear when the process is complete.

Always Showing All Boards

When users do not have default boards assigned to their locations, when they create a new design or look at the Board Information dialog box, they will not see any boards. To solve this problem, **Always show all in boards selection** in the Default boards catalog in Design defaults is enabled by default.
Turn this option off after you have assigned each location a default board set, else toggling **Show All Boards** will not have much effect.

### Defaults Availability overview

**Defaults Availability** on the Options menu gives members of ADMINS control over the types of Defaults users may create and use in Location Defaults and User Defaults. For example, you may want to force users to use a particular die press parameter set and not let users create new die press parameter sets or modify the existing die press parameter set. Setting the Project permissions appropriately prevents the modification of the die press parameter set in Shared Defaults, and using the settings in Defaults Availability can prevent them from creating a new parameter set in User Defaults that would take precedence over the ones in Location Defaults or Shared Defaults.

Defaults Availability has two modes. As a member of ADMINS, you can:

- set the availability of items in Location Defaults for all locations of all companies
- set the availability of items in User Defaults for multiple locations of your assigned company simultaneously. If you do not have a company assigned, this option will be unavailable; click **Database > Change Location** to set a temporary location before using Defaults Availability.

### Setting Defaults Availability for Location Defaults

To set the availability of defaults in Location Defaults for all companies and all locations, do the following:
### Note:
Any Defaults categories you clear in this procedure become unavailable in all User Defaults as well.

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click **Options > Defaults Availability**.
3. In the Defaults Availability Setup dialog box, click **Set defaults availability for location defaults** if it is not already selected.
4. Clear the checkboxes next to the Location Defaults categories to which you want to deny users access. Right-click the column header to check all the checkboxes, clear all the checkboxes, or invert the selection. Any categories you clear in this step will also become unavailable in all User Defaults as well.

5. Click **Save** when you are done checking and clearing checkboxes.
6. Change the mode selector to **Set defaults availability for user defaults** to make those changes, or click **Close** to return to ArtiosCAD.

#### Setting Defaults Availability for User Defaults
To set the availability of defaults in User Defaults for all users at selected locations of your company, do the following:
Note: If you do not have a company assigned, this option will be unavailable; click Database > Change Location to set a temporary location before using Defaults Availability.

1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.
2. Click Options > Defaults Availability.
3. In the Defaults Availability Setup dialog box, click Set defaults availability for user defaults.
4. Select one or more locations containing the users to whom you are denying access.
5. Clear the checkboxes next to the User Defaults categories to which you want to deny users access. Right-click the column header to check all the checkboxes, clear all the checkboxes, or invert the selection.
6. Click **Save** when you are done checking and clearing checkboxes.

7. Click **Yes** in the warning dialog box about overwriting defaults availability.
   
av) If any the locations you selected do not yet have Location Defaults Projects set up for them, ArtiosCAD will prompt you to create a Project for each as needed.
8. ArtiosCAD displays a progress bar as it makes the changes to Defaults.

9. Click Close in the Defaults Availability Setup dialog box to return to ArtiosCAD.

Configuring the Automatic Naming Templates

There are three places in WebCenter to configure automatic naming - in Admin > Preferences > General for Projects and in Admin > Preferences > Document Types for documents. For Templates, to override the system default autonaming configuration for Projects and documents inside that Template, it is in Admin > Projects > Templates > <template> > Configure > Project Creation or Document Creation.

The naming templates control the prefix, body, and suffix ArtiosCAD Enterprise uses when it names Projects and documents automatically. The body is a counter that increments independently for each Project or document.

To configure the naming templates, do the following:

1. Start a Web browser and log in to WebCenter as a member of the ADMINS group.
2. For Projects, click Admin > Preferences > General.
   a) Check Use Automatic Naming for Creation of Blank Projects.
   b) In the Project Autoname Generation section, enter the building blocks of the prefix and suffix in the appropriate fields. Make sure to use spaces or other delimiters (not commas) between the fields, else they will run together and the counter will not increment correctly.
   c) For a list of the building blocks you can use for the prefix and suffix, click Help at the top right of the page. Make sure to read the notes at the bottom of the page for restrictions. The building blocks are case-sensitive. DO NOT USE COMMAS IN THE PREFIX OR SUFFIX.
3. For documents, click **Admin > Preferences > Document Types**.
   a) Check **Apply Automatic Naming to Documents in Blank Projects**.
   b) In the **Automatic Document Naming** column, change the drop-down list box from **None** to **Custom** for each desired document type.
   c) Enter the building blocks of the prefix and suffix in the appropriate fields. Make sure to use spaces or other delimiters (not commas) between the fields, else they will run together and the counter will not increment correctly.
   d) For a list of the building blocks you can use for the prefix and suffix, click **Help** at the top right of the page. Make sure to read the notes at the bottom of the page for restrictions. **The building blocks are case-sensitive.** **DO NOT USE COMMAS IN THE PREFIX OR SUFFIX.**
   e) Click +[-] to use SmartNames to define the prefix or suffix. If you want to be able to enter your own text in the name at creation time, use the **Edit Field** SmartName as a building block.
   f) In the **Counter Length** field, enter the length of the incremental counter that will form the body of the Project/document names between the prefix and suffix. For example, enter 5 for numbers that will be five digits long.
   g) For **Scope of the Counter**, choose either **Reset per Project** to restart the document counter for each Project or **System wide Counter** to use the same counter for every document.
   h) Repeat for other document types as desired.
   i) Click **Save** to make the changes.

4. For Templates, to use custom autonaming with Projects and documents based on a Template, click **Admin > Project Management > Templates**.
   a) Click the name of the Template to change.
   b) Click **Configure**.
   c) Click **Project Creation** or **Document Creation**.
   d) For Projects, make sure **Use Automatic Naming** is selected, and then click **Use Custom Automatic Naming**. Configure the options as described above in step 2. For documents, for the desired document type, change the **Automatic Document Naming** drop-down list box to **Custom**. Configure the options as described above in step 3 and repeat for other document types as desired.
   e) Click **Save** to make the changes.

5. Log out of WebCenter and close your Web browser.

### Using Spot Colors in PDF files

You can configure PDF Outputs to use spot colors for the line color definitions instead of using plotting style information. To make this change, it is best to copy an existing PDF Output to a new one in Defaults and change its name, and then modify the new entry.

On the Device Type tab of the Output, click the Browse control at the end of the Tuning Filename field and navigate to the Tuning Files folder in the Shared Defaults Project.
1. Launch ArtiosCAD Enterprise and log in as a member of the ADMINS group.

2. Click Options > Defaults.

3. In Shared Defaults, copy an existing PDF Output to an entry with a different name, such as Plot to PDF using spot colors.

4. Double-click the new entry and click the Device tab.

5. Click ... (Browse) at the end of the Tuning Filename field.
   a) In the Project pane of the browser, navigate to the Shared Defaults Project and expand it.
   b) Click the Tuning Files folder.
   c) Change the filetype selector to All files (*.*)
   d) Click Tune.PDF.SpotColors.xml and then click Open. The entry in the Tuning Filename field should change accordingly.

6. Click OK to make the change.

7. Click File > Exit and click Yes when asked to save and then overwrite the Defaults.

Shown below are the Ink Manager palettes for two PDF versions of the same single design, one using the default method and the other using the spot colors method.
To change the ink names, RGB color values, stroke widths or stroke styles used by this method, make the changes in `PDFSpotColorMap.xml`.
4. Installing Esko ArtiosCAD

Introduction

There are three types of installations: Typical, Advanced, and Silent. Each can be used as a new installation or as an upgrade from a previous version of Esko ArtiosCAD.

**Typical** is for most situations. It uses the same machine for Esko ArtiosCAD Defaults, Licensing, and the database. A Typical installation can be Standalone (meaning it relies on no other computers for Esko ArtiosCAD and DataCenter Admin to work), a Server (meaning it shares its configuration information with other Esko ArtiosCAD machines on the network) or a Standard Client (meaning it gets its Esko ArtiosCAD configuration information from one other Esko ArtiosCAD system on the network). It does not install the legacy LASERPOINT IQ and The Designer WorkBench Standards catalogs. You may also choose the type of database to be used with Esko ArtiosCAD. Microsoft® SQL Server 2005 Express Edition, Oracle®, or Microsoft SQL Server may be used as the database engine. However, Esko does not support the installation, operation, or maintenance of Oracle or a full version of Microsoft SQL Server.

**Advanced** is used for more complex customized installations. Different machines can be specified for Esko ArtiosCAD Defaults, Licensing (if applicable), and the database.

Additionally, in an advanced installation, you can specify to install Esko ArtiosCAD so that it is non-version-specific, meaning that the icons and directories are named Esko ArtiosCAD, not Esko ArtiosCAD12 Enterprise, for example, and future upgrades would overwrite the existing installation.

**Silent** installation presents no dialog boxes, and all options must be specified on the command line. This is meant for use only in large corporate WAN environments where one central administrator installs Esko ArtiosCAD on other computers to which the administrator has remote command access. The silent installation program must be run on the computer on which Esko ArtiosCAD will be used. The silent installation program may not be run on one machine resulting in Esko ArtiosCAD being installed on another machine. Contact Support for more information on performing a Silent installation.

**Note:** Keep the original Esko ArtiosCAD media in a safe, easily-accessible place in case you need to modify or repair the installation. You must use the same media or network location to repair or modify Esko ArtiosCAD as when you first installed the software.

The table below summarizes the options available in the three installations for conventional versions. Enterprise versions use WebCenter servers for licensing and database functions.

<table>
<thead>
<tr>
<th>Choice</th>
<th>Typical Installation</th>
<th>Advanced Installation</th>
<th>Silent Installation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ability to use choose database engine</td>
<td>•</td>
<td>•</td>
<td>•</td>
</tr>
</tbody>
</table>

(Configuration batch file must be run manually after Esko ArtiosCAD installation)
### Performing a Typical installation

**Note:** If you receive a message about overwriting read-only files, answer yes to overwrite them. Also, if you receive a message about a control failing to register, proceed normally and reboot when the installation finishes.

To perform a typical Esko ArtiosCAD installation, do the following:

1. Log on as **Administrator**, or as a user with administrative privileges.
2. Ensure there is at least 5 GB of free space on the hard drive or partition where Esko ArtiosCAD will be loaded and 500 MB of free space on the operating system hard drive or partition.
3. Put the Esko ArtiosCAD media into the computer’s media drive. It should automatically launch the Setup program. If your drive is set to not AutoPlay, start Windows Explorer and double-click `setup.exe` in the root directory of the media.
4. The Esko ArtiosCAD Setup dialog box appears. Click the button next to **Install Esko ArtiosCAD <version number>** to start installing Esko ArtiosCAD.

<table>
<thead>
<tr>
<th>Choice</th>
<th>Typical Installation</th>
<th>Advanced Installation</th>
<th>Silent Installation</th>
</tr>
</thead>
<tbody>
<tr>
<td>finishes for Oracle and Microsoft SQL Server)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ability to use same computer for Esko ArtiosCAD Defaults server, License server, and database</td>
<td>•</td>
<td>•</td>
<td>•</td>
</tr>
<tr>
<td>Ability to specify different computers for Esko ArtiosCAD Defaults, Licensing, and database</td>
<td></td>
<td>•</td>
<td>•</td>
</tr>
<tr>
<td>Ability to choose Esko ArtiosCAD installation directory</td>
<td>•</td>
<td>•</td>
<td>•</td>
</tr>
<tr>
<td>Esko ArtiosCAD program executable and Esko ArtiosCAD Standards Catalog automatically selected for loading</td>
<td></td>
<td>•</td>
<td></td>
</tr>
<tr>
<td>Selectively load all components</td>
<td>•</td>
<td>•</td>
<td>•</td>
</tr>
<tr>
<td>Choose to load The Designer WorkBench and LASERPOINT Standards catalog</td>
<td></td>
<td>•</td>
<td></td>
</tr>
<tr>
<td>Automatically keep previous versions (use version-specificness)</td>
<td>•</td>
<td>•</td>
<td>•</td>
</tr>
<tr>
<td>Choose to keep previous versions or overwrite them</td>
<td></td>
<td>•</td>
<td></td>
</tr>
<tr>
<td>Ability to choose default location of Esko ArtiosCAD users’ design files</td>
<td>•</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5. Depending on your system configuration prior to loading Esko ArtiosCAD, the Microsoft Data Access Components and MSXML 6 may be copied to your system. Two status boxes will appear: one showing the copying of the files, and perhaps one showing the configuration of Control Panel. If the system prompts you to reboot, do so; the installation program will resume automatically when you log in after the system comes up again. Do not postpone rebooting if the system requests it.

6. Click Next in the Welcome to the InstallShield Wizard for Esko ArtiosCADx.xx dialog box.

7. Read the Software License Agreement in its entirety and choose I accept the terms of the license agreement if you accept all the terms of the License Agreement. Click Next.

8. If upgrading from a previous version of Esko ArtiosCAD, choose to copy the Defaults from the last version loaded, or choose to use the default configuration which will install a completely unmodified set of Defaults. You should copy the previous configuration if you customized Esko ArtiosCAD in any way, such as adding Output devices and making your own Reports. Click Next. If there is no prior version of Esko ArtiosCAD on this machine, you are not prompted to choose an upgrade version.

9. Choose a Typical installation and click Next. If you are upgrading systems with distributed servers, you must do an Advanced installation.

10. In the Feature Selection dialog box, choose the features to install on the current machine. For each feature, click the icon in the drop-down list box next to it and choose the appropriate choice. Do not click Next when done, but rather proceed to the next numbered step.

   *Esko ArtiosCAD Configuration.* Click the plus sign (+) next to this entry to open it. *Esko ArtiosCAD Licensing, Esko ArtiosCAD Defaults,* and *Esko ArtiosCAD Database* can each be installed in any combination on a machine. For Typical and all Standalone installations, leave them all selected.

   *Esko ArtiosCAD Programs.* *Esko ArtiosCAD Programs* are the program files needed to run Esko ArtiosCAD. You must install these if you want to run Esko ArtiosCAD on the current machine. Not installing these would create a server-only installation provided the servers were selected for installation. You must leave this choice selected to install Esko ArtiosCAD itself.

   Contained within the *Esko ArtiosCAD Programs* group are optional components of Esko ArtiosCAD that require purchasing licenses for the corresponding modules: *PDF Import/Export, 3D Connection and Filters,* and *SolidWorks Import/Export.* Refer to the printed license report that came with the media to determine which components to select. Installing software components without having the corresponding licenses wastes disc space. Not installing the software components but having the licenses for them results in features that do not function within Esko ArtiosCAD.

   *Esko Configuration Manager* collects data from systems running the Esko Station Information Service and sends the collected data to Esko automatically for use by Support personnel. It sends information only if you configure it to do so and leave it enabled. It is a great aid to Support personnel as they will have required information about your hardware and licenses available should you need to call for assistance. Only one system per site should have the Esko Configuration Manager installed. Refer to the separate document *Scope Configuration Manager 1.0 Software Installation and Configuration Manual* on the Esko ArtiosCAD media (*SCM_nn_SICFM.PDF* in the *User Docs* folder) for more information.

   *Laserpoint Standards Catalog* is a legacy Standards catalog for single designs. Only install this if there are LASERPOINT or LASERPOINT IQ standards you want to run.

   *Designer WorkBench Standards Catalog* is a legacy Standards catalog for single designs. Only install this if you want to run standards from The Designer WorkBench.
11. While still in the Feature Selection dialog box, with one of the components to be installed selected, review the folder in which Esko ArtiosCAD will be installed, and click Change if desired. In the resulting Change Current Destination Folder dialog box, change the drive if desired, and click OK to return to the Feature Selection dialog box. Esko ArtiosCAD should always be installed to the \Esko\Artios folder on a local drive. Click Next.

12. If this is a new installation, choose the database program for Esko ArtiosCAD to use. The default and recommended choice is SQL Server 2005 Express Edition, a free desktop database from Microsoft. Choosing Microsoft SQL Server or Oracle is appropriate for workgroup or enterprise use where those databases are already in use. If this is an upgrade, you may not be asked to choose a database type depending on the one already in use.

   If this is an upgrade, and you want to switch from using MSDE to using SQL Server 2005 Express Edition

---

**Note:** If you choose Microsoft SQL Server or Oracle, configuration scripts and a batch file for those databases will be placed in the \Esko\Artios\DataCenter directory. The database administrator MUST run the appropriate batch file before using Esko ArtiosCAD or DataCenter Admin. Other post-installation tasks must be performed as well.

1. Choose the System type. **Standalone** shares nothing - it is entirely self-contained. A **Server** shares its Esko ArtiosCAD configuration with other machines on the network. A **Standard Client** gets its Esko ArtiosCAD configuration from another system on the network; there must be a Server loaded and running before loading any Standard Clients.

   At this point, the Typical installation branches into two parts. If you chose **Standalone** or **Server**, proceed as directed below. If you chose **Standard client**, proceed to the Typical Standard Client installation section.

---

**Typical Standalone or Server installation**

1. Having chosen **Standalone** or **Server** as the system type, click Next.
2. Review the summary of installation settings. If they are correct, click Install to start installing files. If not, click Back to go back and change settings.
3. If this is a new installation, the Esko Station Information Service will be installed automatically. This service collects information about your hardware, software, and Esko licenses. If the Esko Scope Configuration Manager service is installed on a machine on your network, it will forward the information collected to Esko for use by Support personnel if you configure it to do so. Several dialog boxes will appear displaying information; you do not need to click anything as they will automatically continue.
4. A progress indicator shows the status of the copying process. If this system is a Server and this is a new installation of Esko ArtiosCAD, click OK when told that the Artios directory has been shared.
5. The Esko ArtiosCAD Setup program might prompt for a system reboot. Click Yes to reboot the system.
6. Once the system comes back up, Esko ArtiosCAD and DataCenter Admin will be fully functional
7. If you installed a non-English version of Esko ArtiosCAD, run \Esko\Artios\DataCenter\program \<language\>dbtranslate.exe to make DataCenter compatible with your language.
8. Activate the license using the Activate Local Licenses application in the ArtiosCAD folder for a local license, or for a network license, by using the Activate Licenses application in the System
Controller subfolder of the Esko folder of the Start menu. For more more information on license management, refer to the documents available for ArtiosCAD at http://help.esko.com.

8. You are now ready to configure Esko ArtiosCAD.

Typical Standard Client Installation

Note: If you are upgrading and your Esko ArtiosCAD installation is set up so that different machines act as different servers, Typical Standard Client installations will not work, as they require a unified server. If your system uses multiple servers, you must perform an Advanced Client/Server installation.

1. Before doing a Standard Client installation, make sure there is a Server installation on another machine which is running and on the network, and note the name of the Server machine. Also make sure there is at least 5 GB of free space on the hard drive or partition where Esko ArtiosCAD will be loaded and 500 MB free on the system hard drive or partition.

2. Having chosen the Standard Client system type, enter the name of the server in the Esko ArtiosCAD Server: field of the System Type dialog box. Click Next after entering the name.

3. Review the installation settings listed in the summary list. If they are correct, click Install to start installing files. If not, click Back to go back and change settings.

4. Depending on your system configuration prior to loading Esko ArtiosCAD, the Microsoft Data Access Components may be copied to your system. Two status boxes may or may not appear: one showing the copying of the files, and perhaps one showing the configuration of Control Panel.

5. A progress indicator shows the status of the copying process.

6. The Esko ArtiosCAD Setup program might prompt for a system reboot. Click Yes to reboot the system.

7. Once the system comes back up, Esko ArtiosCAD and DataCenter Admin will be fully functional. This Esko ArtiosCAD system will use the same configuration files as those on the Server, so configuration of this system is unnecessary.

If you installed a non-English version of Esko ArtiosCAD, run \Esko\Artios\DataCenter\program \<language>\dbtranslate.exe to make DataCenter compatible with your language.

Performing an Advanced installation

To perform an Advanced installation of ArtiosCAD, do the following:

1. Log on as Administrator, or as a user with administrative privileges.

2. Ensure there is at least 5 GB of free space on the hard drive or partition where ArtiosCAD will be loaded and 500 MB of free space on the operating system hard drive or partition.

3. Put the ArtiosCAD media into the computer's media drive. The setup program should start automatically. If it does not, start Windows Explorer and double-click setup.exe in the root directory of the media.

4. The ArtiosCAD Setup dialog box appears. Click the button next to Install ArtiosCAD <version number> to start installing ArtiosCAD.

5. Depending on your system configuration prior to loading ArtiosCAD, the Microsoft Data Access Components and MSXML 6 may be copied to your system. Two status boxes will appear: one showing the copying of the files, and perhaps one showing the configuration of Control Panel. If
the system prompts you to reboot, do so; the installation program will resume automatically when
you log in after the system comes up again. Do not postpone rebooting if the system requests it.

6. Click Next in the Welcome dialog box.
7. Read the License Agreement in its entirety and choose I accept the terms of the license
agreement if you accept all the terms of the License Agreement. Click Next.
8. If upgrading from a previous version of ArtiosCAD, choose to copy the defaults from the
last version loaded, or choose to use the default configuration which will install a completely
unmodified set of Defaults. You should copy the previous configuration if you customized
ArtiosCAD in any way, such as adding Output devices and making your own Reports. Click Next.
If there is no prior version of ArtiosCAD on this machine, you are not prompted to choose an
upgrade version.
9. Choose Advanced in the Setup Type dialog box and click Next.
10. In the Feature Selection dialog box, choose the features to install on the current machine. For each
feature, click the icon in the drop-down list box next to it and choose the appropriate choice. Do
not click Next when done, but rather proceed to the next numbered step.

ArtiosCAD Configuration. Click the plus sign (+) next to this entry to open it. ArtiosCAD Licensing,
ArtiosCAD Defaults, and ArtiosCAD Database Server can each be installed in any combination
on a machine.

ArtiosCAD Programs are the program files needed to run ArtiosCAD. You must install these if
you want to run ArtiosCAD on the current machine. Not installing these would create a server-
only installation provided the servers were selected for installation.

Contained within the ArtiosCAD Programs group are optional components of ArtiosCAD that
require purchasing licenses for the corresponding modules: PDF Import/Export and 3D Importer.
Refer to the printed license report that came with the media to determine which components
to select. Installing software components without having the corresponding licenses will simply
waste disc space. Not installing the software components but having the licenses for them will
lead to features that do not function within ArtiosCAD.

Esko Configuration Manager collects data from systems running the Esko Station Information
Service, and if you configure it to do so, sends the collected data to Esko automatically for use
by Support personnel. It is a great aid to Support personnel as they will have required information
about your hardware and licenses available should you need to call for assistance. Only one
system per site should have the Esko Configuration Manager installed. Refer to the separate
document Scope Configuration Manager 1.0 Software Installation and Configuration Manual
(scm_10_sicfm.pdf) for more information.

Laserpoint Standards Catalog is a legacy Standards catalog for single designs. Only install this
if there are LASERPOINT or LASERPOINT IQ standards you want to run.

Designer WorkBench Standards Catalog is a legacy Standards catalog for single designs. Only
install this if you want to run standards from The Designer WorkBench.

11. While still in the Feature Selection dialog box, review the directory to which ArtiosCAD will be
installed, and click Change if desired. In the resulting Change Current Destination Folder dialog
box, navigate to the desired directory, select it, and click OK to return to the Feature Selection
dialog box. Click Next.
12. In the Database Program Selection dialog box, the database selection defaults to SQL Server
2005 Express Edition, the free desktop database from Microsoft. Microsoft SQL Server and Oracle
both work with ArtiosCAD, but it is your responsibility to purchase, install, configure, and maintain
them. Additionally, if you have MSDE on your system for use with another program, you may
choose to use it, but if you do not already have it loaded, it is not presented as a choice.
Note: If you choose Microsoft SQL Server or Oracle, configuration scripts and a batch file for those databases will be placed in the \Esko\Artios\DataCenter directory. The database administrator **MUST** run the appropriate batch file before using ArtiosCAD or DataCenter Admin. Other post-installation tasks must be performed as well.

13. In the Advanced Options dialog box, **Keep Previous Versions** is the default for previous version handling. It means that ArtiosCAD is installed using a version-specific folder and icon which match the version and language of the installation, and all previous versions of ArtiosCAD are kept. To change this setting, click **Change**. In the Previous Version Handling dialog box, **Overwrite previous versions** simply uses ArtiosCAD as the icon name and ArtiosCAD as the directory with no version number. It overwrites the prior version of ArtiosCAD if this is an upgrade and the previous version was installed to a non-version-specific directory. Once you have chosen the option to use, click **OK** to return to the Advanced Options dialog box.

If you are using SQL Server 2005 Express Edition or Microsoft SQL Server 2005 as the database, you have the choice of database authentication methods. **SQL Server Authentication** requires each user to have a login created within the database; by default, ArtiosCAD uses one database user (programmer) for all ArtiosCAD users. However, your company may have different requirements. **Windows Authentication** passes operating system login credentials to the database. You should use Windows Authentication only if you have a dedicated database administrator who can perform user maintenance tasks.

Click **Next**.

14. In the System Type dialog box, choose the type of system this will be, mindful of the modules chosen for installation in step 10. For example, if you elected not to install ArtiosCAD Programs, this system cannot be a Standalone system, but it can be a Server or Advanced Client/Server.

If this system is to be a **Standard Client**, enter the name of the ArtiosCAD Server in the appropriate field.

If you intend this machine to be a server only, select **Advanced Client/Server**, and enter the system name in the fields that this machine will serve. If it will only be one kind of server, enter the system name in the appropriate field, and enter the names of the other servers in their appropriate fields.

If you choose **Advanced Client/Server**, enter the names of the systems acting as the Defaults server, License Server, and Database Server. If the database is not Microsoft Access, the Database Server field is unavailable; the database server must be configured manually via the ArtiosCAD entry in the ODBC control panel. You can enter the same system name in all three fields, or you can specify a different system in each field. If you enter remote system names, and those remote systems do not have ArtiosCAD loaded on them, you will have to manually enter the exact path to the applicable resources. **ArtiosCAD must be loaded on the server you indicated on these remote systems for ArtiosCAD to work on the current system**! It does not have to be loaded at the time of the current system upgrade, but should be loaded soon afterward.

You can also specify the name of the current system if you want it to act as its own server for any of the functions listed.

Click **Next** when done choosing the system type.

15. If Setup cannot find the appropriate service files for any of the three servers entered in the previous step, for example if you have not loaded ArtiosCAD on those machines yet, it will ask for the UNC locator for the service. The instructions in the dialog box will provide an example to use for
entering the location, but make sure the data you enter is correct for your configuration. Click **Next** when you have entered the full location for the service.

16. The default location for the storage of user files is shown in the User Files Folder dialog box. To change it, click **Change**; to accept it, click **Next**. If you change the location to another system, use a UNC locator (e.g. `\system2\designs`) instead of a mapped drive letter.

17. Review the settings listed in the Ready to Install the Program dialog box. If they are correct, click **Install** to start copying files. If not, click **Back** to go back and change settings.

18. If this is a new installation, the Esko Station Information Service will be installed automatically. This service collects information about your hardware, software, and Esko licenses. If the Esko Configuration Manager service is installed on a machine on your network, it will forward the information collected to Esko for use by Support personnel if you configure it to do so. Several dialog boxes will appear displaying information; you do not need to click anything as they will automatically continue.

19. A progress indicator will show the status of the copying process. Click **OK** if told that the Artios directory has been shared.

20. The ArtiosCAD Setup program might prompt for a system reboot. Click **Finish** to reboot the system.

21. Activate the license using the Activate Local Licenses application in the ArtiosCAD folder for a local license, or for a network license, by using the Activate Licenses application in the System Controller subfolder of the Esko folder of the Start menu. For more information on license management, refer to the documents available for ArtiosCAD at [http://help.esko.com](http://help.esko.com).

22. Reboot the system if you are prompted to do so. Once the system comes back up, ArtiosCAD and DataCenter Admin will be fully functional if the system is using a MSDE or SQL Server 2005 Express Edition database.

If you installed a non-English version of ArtiosCAD, run `\Esko\Artios\DataCenter\program\<language>\dbtranslate.exe` to make DataCenter compatible with your language.

If the system does not use MSDE or SQL Server 2005 Express Edition as the database engine, see [Using other databases with Esko ArtiosCAD](#).

**Note:** If you choose to distribute tasks among remote servers, you must load ArtiosCAD on each of those servers as an Advanced installation, specifying those server names in the appropriate fields of the Advanced Client/Server choice in the System Type dialog box.

---

### Using other databases with Esko ArtiosCAD

If you chose **Microsoft SQL Server** or **Oracle** in the Database Program Selection dialog box, your database administrator must configure the database to work with Esko ArtiosCAD. See [Running the Esko ArtiosCAD database configuration batch file](#) for Microsoft SQL Server 2005 or 2000 or [Using Oracle](#) for Oracle.

### Other database notes

To change the database engine used with Esko ArtiosCAD, delete the ArtiosCAD data source from the System DSN tab of the ODBC Data Sources control panel and reinstall Esko ArtiosCAD.
Oracle databases should either have autoexpand set for the tablespace or should have a very large maximum extent set.

Do not change the database password in a vendor-supplied utility. Rather, change it in DataCenter Admin by using the File > Change Database Password command.

Advanced Upgrading of Esko ArtiosCAD on a server

Notes and warnings before you begin

Because the upgrade process is highly configurable, there are many situations which must be considered before beginning the upgrade process.

If you specified remote servers for ArtiosCAD Defaults, ArtiosCAD Licensing, or ArtiosCAD Database, you must load Esko ArtiosCAD on those servers as Advanced installations, specifying their system names in the appropriate fields in the System Type dialog box on each system.

If you have a remote License server, make sure your licensing information is loaded on that server.

If you are upgrading the current server and specifying a remote Esko ArtiosCAD Defaults server, copy the contents of the current system’s ServerLib to that remote system’s ServerLib directory.

Upgrading other databases with Esko ArtiosCAD

If you use Oracle or Microsoft SQL Server 2000 as the database engine, you must run the configuration batch file after Esko ArtiosCAD finishes loading and the system has rebooted (if applicable). See the appropriate sections further in the manual about running the batch file for more information.

Other database notes

To change the database engine used with Esko ArtiosCAD, delete the ArtiosCAD data source on the System DSN tab from the ODBC control panel and reinstall Esko ArtiosCAD.

Oracle databases should either have autoexpand set for the tablespace or should have a very large maximum extent set.

Do not change the database password in a vendor-supplied utility. Rather, change it in DataCenter Admin by using the File > Change Database Password command.
Changing to SQL Server 2005 Express Edition on the database server

SQL Server 2005 Express Edition is a replacement for Microsoft Access and MSDE which Esko believes offers greater stability and better performance for Esko ArtiosCAD.

If your system currently uses Microsoft Access as the database engine, you must upgrade it to SQL Server 2005 Express Edition. If your system currently uses MSDE, you may continue to use it, but SQL Server 2005 Express Edition offers better performance than MSDE.

If you choose to change to SQL Server 2005 Express Edition, you should export non-design-based database information from the old database before starting to load Esko ArtiosCAD. The migration to SQL Server 2005 Express Edition should be error-free, but if there are problems, you can import this data into a blank database and then do an AutoLoad to create a new database.

Exporting database information from the old database

To export non-design-based data from the existing database, do the following:

1. Ensure that any programs using the database are not running.
2. Log on to the database server as Administrator or as a member of the Administrators group.
3. Click Start > Programs > ArtiosCAD > Migration Tools > ArtiosCAD Database Export.
4. Enter admin as the username with no password, and click OK.
5. The utility will appear similar to the picture shown below.
6. Enter a sensible file name and location in the Migration Filename and Path: group.
7. Select all the available tables.
8. Click Export. The progress bar indicates the progress of the export.
9. When the export is done, click Exit. If the migration to SQL Server 2005 Express Edition is error-free, you can delete the exported file after the migration if desired.

Loading the upgrade

When you install Esko ArtiosCAD as an upgrade and and have been using Microsoft Access or MSDE as the database engine, choose SQL Server 2005 Express Edition as the database type, a checkbox appears asking if you want to migrate prior database data to SQL Server 2005 Express Edition if this system is the database server.
If you choose SQL Server 2005 Express Edition as the database program and this system is the database server, the following actions happen:

- A database schema is built or updated as appropriate.
- The existing ArtiosCAD ODBC datasource is renamed to ArtiosCAD_Access or ArtiosCAD_MSDE as appropriate.
- Empty command prompt windows appear.
- A service called SQL Server (ARTIOSCADEXP2005) is created and started.

If this system is not the database server, the ArtiosCAD ODBC datasource changes to reference the SQL Server 2005 Express Edition database.

If you choose to migrate the data, the setup program tries to export the data from the old database. If the export is successful, the data is migrated to the SQL Server 2005 Express Edition database.

If you do not choose to migrate the data, a blank SQL Server 2005 Express Edition database is created and you have to manually export the data from the old database and manually import the data into SQL Server 2005 Express Edition.

**Automatic migration**

If you choose to have the setup program migrate the data, a series of empty command prompt windows appears during the installation while the migration tasks are executing. Their duration is directly proportional to the size of the old database.

If there are errors during the migration process, you must manually migrate the data as instructed in the next section.

Signs of a successful migration are:

- Data appearing in the design browsers in Esko ArtiosCAD, and also in DataCenter Admin, provided that the Esko ArtiosCAD System DSN uses a Microsoft SQL Server driver.
If the automatic migration fails, you must perform a manual migration.

**Manual migration**

If you do not choose automatic migration, you must perform the data migration manually on the database server for the old database after the Esko ArtiosCAD installation finishes.

The steps to performing the migration manually are as follows:

1. Export the data.
2. Check the log file for errors.
3. Import the data.
4. Check the log file for errors and try Esko ArtiosCAD.

**Exporting the data**

To export the data from the old database, do the following.

1. Log on as Administrator or as a member of the Administrators group.
2. Click **Start > Programs > Esko > ArtiosCAD > Migration Tools > Esko ArtiosCAD Database Migration Export**. The export utility will appear along with a blank command prompt window.
The **Migration Folder** group shows the folder that stores the files created by the export utility. The export files require the same amount of hard drive space as the Microsoft Access database file, so make sure to choose a location with enough free space. Click **Browse** to select a new location.

The **Data Source** group lists the available data sources. **Make sure to select the ArtiosCAD_Access or ArtiosCAD_MSDE as the data source for this migration.**

The **Format** group selects the format of the migration files. **ADTG** is the default; if it errors, try **XML**.

In the **Advanced** group, **Delete design dupes** deletes duplicate design records in the database; preserving the newest record and discarding older duplicates. **Delete orphans** deletes table entries that no longer have corresponding design records on the hard drive. Both options are enabled by default.

The **Migrate** group contains the actual commands. **Export** exports the data from the selected data source in the format selected to the migration folder. **View Log** opens the log file of the export. It is stored in the system temporary directory as `ACadFullDbMigrationExportLog.txt`. **Exit** exits the program.

The **Migration Process** group contains the progress bar showing the table being exported. The length of time required to export the data is directly proportional to the size of the database. **Cancel Migration** cancels the migration. Do not use the migration files created by a cancelled migration.

1. Set the migration folder and format as desired, select the **ArtiosCAD_Access or ArtiosCAD_MSDE** data source, and click **Export**. During the export, all controls in the dialog box are unavailable except for **Cancel Migration**.
2. There should be no errors. If an error appears, delete all the ADTG files created by this migration and try using XML for the migration format. If there are no errors, the next step is to check the log file.

Checking the log file for export errors

1. Even if no errors appeared during export, click View Log to view the log file. Information, warnings, and errors are logged. Check the log file for warnings and errors. Warnings related to tables not existing are usually caused by database schema differences among Esko ArtiosCAD versions; these can be safely ignored. Only errors warrant concern. A sample log file is shown below; yours might vary.

Table: Sample ACadDBFullMigrationExportLog.txt

| Information: [Exporting table] | CODE_TYPE       |
| Information: [Exporting table] | IQ_DEFAULTS    |
| Information: [Exporting table] | CODE           |
| Information: [Exporting table] | HIERARCHICAL_STYLES |
| Information: [Exporting table] | COMPANY        |
| Information: [Exporting table] | PERSON         |
| Information: [Exporting table] | FLUTE          |
| Information: [Exporting table] | BOARD_FOLDER   |
2. Once you have reviewed the log file, close it. If there were errors, delete the XML or ADTG files and AutoLoad in database recovery mode.

3. Click Exit to exit the ArtiosCAD Database Migration Export utility. The next step is to import the data.

Importing the data

1. Click Start > Programs > Esko > ArtiosCAD > Migration Tools > ArtiosCAD Database Migration Import. The import utility will appear along with a blank command prompt window.
2. The Migration Folder Path folder is automatically set to the same folder as the one used previously by the Esko ArtiosCAD Database Migration Export utility. Make sure to set the option button in the Format group to the same format as that used in the export.

Select the ArtiosCAD data source if it is not already selected.

In the Advanced group, Delete Database deletes the contents of the database before importing the data stored in the migration file. For example, to make an exact copy of the database from which you are migrating, select this option, but if you are importing only one table, deselect this option. Stop on Insert error completely stops the migration when there is a problem adding information. When deselected, the process continues regardless of errors, and you should read the log file to see if there are errors.

3. Click Import when you have set the options as desired. During the import, all dialog box controls are unavailable except for Cancel Migration. The progress bar moves to the right as the import progresses. There should be no errors.
Checking the log file for import errors

1. Even if there are no errors, click View Log when the import is done to review the log file for errors. Information, warnings, and errors are logged. Check the log file for warnings and errors. A sample log file is shown below; yours might vary.

Table: Sample ACadDBFullMigrationImportLog.txt

```
****** The ArtiosCAD full database migration began at: 10/09/07, 14:52:17
Information: Deleting data from all tables
Information: [Importing data] code_type [C:\Artios\code_type.adtg]
Information: [Importing data] iq_defaults [C:\Artios\iq_defaults.adtg]
Information: [Importing data] code [C:\Artios\code.adtg]
Information: [Importing data] hierarchical_styles [C:\Artios\hierarchical_styles.adtg]
Information: [Importing data] company [C:\Artios\company.adtg]
Information: [Importing data] person [C:\Artios\person.adtg]
Information: [Importing data] flute [C:\Artios\flute.adtg]
Information: [Importing data] board_folder [C:\Artios\board_folder.adtg]
Information: [Importing data] board [C:\Artios\board.adtg]
.
.
Information: [Importing data] cad_permission [C:\Artios\cad_permission.adtg]
Information: [Importing data] global_permission [C:\Artios\global_permission.adtg]
Information: [Importing data] profile_defaults [C:\Artios\profile_defaults.adtg]
```
2. Once you have verified there are no errors, close the log file. If there were errors, delete the XML or ADTG files and try the complete process again, using the other format for the data files. If that fails as well, delete the data files and use AutoLoad in database recovery mode.

3. Start Esko ArtiosCAD.

4. Click **Database > Browsers > Design browser**. The Design Browser should appear normally.

5. If you are upgrading from a version of Esko ArtiosCAD earlier than version 4, run `\Esko\Artios\DataCenter\Program\Buildkeys.exe` to convert design styles to hierarchical characteristics.

6. The migration to the new database is complete.

Once you have verified that all the data is in the database, delete the XML or ADTG files from the migration folder.

### ArtiosCAD Database Migration utility command-line options

You can run the Esko ArtiosCAD Database Migration utility from the command line if desired. This ability is meant for use by system administrators in large corporations who manage remote computers.

Use a command prompt to go to the `\Esko\Artios\Datacenter\program\language` folder.

Next, run `ACADDbMigration.exe` with the desired switches.

<table>
<thead>
<tr>
<th>Switch</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>-c</td>
<td>Use the command line versus a dialog box.</td>
</tr>
<tr>
<td>-e</td>
<td>Export data (the default.)</td>
</tr>
<tr>
<td>-i</td>
<td>Import data.</td>
</tr>
<tr>
<td>-f folder</td>
<td>Specifies the folder in which the migration files are located.</td>
</tr>
<tr>
<td>-d datasources</td>
<td>The default is whatever is specified in the registry from a previous export else it is unknown. The registry key is HKLM\Software\Artios\Migration. The registry value is Migration_FullDB_Folder.</td>
</tr>
<tr>
<td>-t xml or adtg</td>
<td>Format for the migration files. ADTG is the default.</td>
</tr>
</tbody>
</table>

The default location is the temp directory specified by the TMP variable if it exists, else the TEMP variable if it exists, else the current directory. The file name is `ACADFullDbMigrationExportLog.txt` (or `ImportLog.txt`).

The default is ArtiosCAD. Note that the Microsoft Access datasource name was changed to ArtiosCAD_Access when you changed to MSDE.
Changing to SQL Server 2005 Express Edition on client systems

If you change your database server to SQL Server 2005 Express Edition, you must also load the upgrade on all client systems to configure them to use the new server software. No database software is loaded on clients.
5. Peripherals

Esko ArtiosCAD can output to devices that use a standard Microsoft Windows driver and also to devices simply connected to a port on an Esko ArtiosCAD computer. Devices that do not use a Windows driver use Output Destinations which are configured in the Output Destinations dialog box in Shared Defaults.

Note: This documentation refers to configuring devices in Shared Defaults for all users to access. Configuring a device for use on a single machine in User Defaults follows the same procedure, substituting User Defaults for Shared Defaults.

Installing a Windows printer

Esko ArtiosCAD prints by default to the default Windows printer on the system on which it is running.

For information on installing a printer in the Windows family of operating systems, read the Setting up a Printer topic in the How to…Set Up Hardware section of Help.

If this is the first printer installed on the system, it is automatically designated as the system default printer. The Print command and the Plot to Printer output (File > Outputs > Artios > Printer Output > Plot to Printer) will use it by default.

To add an Output for a newly-installed printer if it is not the system default printer, do the following:

1. Start Esko ArtiosCAD
2. Open Defaults by clicking Options > Defaults.
3. In the Shared Defaults pane, scroll down to Outputs.
4. Right-click Outputs, click New from the context menu, and then click Data.
5. Enter a name for the Output such as the name of the printer; for example, HP DeskJet 560C.
6. Double-click the new Output just created to open its Property page.
7. The Device tab will be selected automatically. Leave the Output type set as Plot and change the Scale to To fit one page.
8. Click the Device tab. If a dialog box appears with the message System printer information could not be read, click OK.

9. Under Windows Driver, click the drop-down list box arrow and select the newly-created printer.

10. Click OK, then File, then Save to accept the printer and save the defaults.

11. Click Yes when asked to overwrite the defaults. The printer is now ready for use.

Output Destinations

Output Destinations enable the ArtiosIO program to carry output data from Esko ArtiosCAD to a serial or parallel port. The port can be on the local machine or it can be on a remote computer on the network. Output Destinations also allow a device to communicate information (such as its size) back to Esko ArtiosCAD.
Output Destinations require the ArtiosIO program to be running on the machine to which the device is connected.

**Starting the ArtiosIO program**

The ArtiosIO program is installed when Esko ArtiosCAD is installed. It is configured using the ArtiosIO applet in Control Panel. Only users with administrative privileges can use this applet and the commands within.

To start the ArtiosIO program and configure it to start automatically each time the computer is rebooted, do the following:

1. Open Control Panel by clicking **Start > Settings > Control Panel**.
2. Double-click the **ArtiosIO** applet. The ArtiosIO Server dialog box will appear.
3. Click **Start** and set the **Startup Type** to **Automatic**.
4. Click **OK**.
5. Do this on each computer which has a port to which a device that does not use a Windows driver will be attached.

Creating an Output Destination

Now that the ArtiosIO program is running, you can configure Output Destinations in Esko ArtiosCAD.

1. Start Esko ArtiosCAD.
2. Open Defaults by clicking Options > Defaults.
3. Right-click Output Destination, click New on the context menu, and click Data.

4. Enter a name for the Output Destination. One good way to name it is to use the system name and port name, e.g. ARTCAD LPT1 or SYSTEM4 COM2.
5. Once you have entered a name for the Output Destination, double-click it to open its Property page.

6. Enter the name of the system the port is on in the Hostname field.
7. Enter the port name in the Port field. Common port names are COM1, COM2, COM3, and LPT1. The COM ports are serial ports (also known as RS-232 ports) and LPT ports are parallel ports.
8. If the port you are adding is a serial port, set its characteristics in the RS 232 Parameters group by choosing the options from the drop-down list boxes. If it is a parallel port, clear the RS 232 Port checkbox.
9. The **Short Name** field is the internal-to-ArtiosIO name of the port and can not be changed.
10. Click **OK** when done.
11. To save these changes, click **File > Save** and click **Yes** to overwrite the Shared defaults.

### Installing CAM Devices

#### Getting started

Use the following steps to configure a CAM (Computer Aided Manufacturing) device with Esko ArtiosCAD. Most CAM devices are equipment such as samplertables, lasers, counter cutters, and so on.

Configuring a CAM device follows this general procedure:

- Pick the output device driver from a master list of known devices. This will tell Esko ArtiosCAD what driver to use and how to tune it. Also check and set the size of the device.
- Specify how the peripheral will receive its instructions. This could be either by creating a disk file or by sending the instructions to a parallel or serial port on one of the computers.
- Configure a plotting style or CAM Tooling Setup catalog entry for the device to use that defines which tools on the device are to be used for each type of line in a design.

You may need to adjust the size and positioning settings copied from the master list.

When this process is complete, you will have an output that can be used by any Esko ArtiosCAD user.

#### Preliminaries

First, you need answers to the following questions.

- Will you be using a Windows driver or a CAM driver?
- If a CAM driver is being used, what port will the output go to and how will it get there?
- Is there a pre-built output and plotting style/CAM Tooling Setup catalog entry for the plotter / samplemaker you need to set up?

**Windows driver or CAM driver?**

Use a Windows driver if possible. These should be used for all drawings if suitable and if drivers are available for your device. Many CAM devices operate correctly when driven using the IBM Graphics standard Microsoft Windows printer driver. A CAM driver is used when there is no suitable Windows driver for a device.

**Is there a pre-built output and style?**

If you have a plotter / samplemaker of a type listed below, there will most likely be a pre-built solution for your plotting and sampplemaking needs. You can probably set up the device without going into

---

94
much of the detail described in this document. If there is not a preconfigured plotting style or CAM Tooling Setup catalog entry, follow the instructions in the Defaults chapter to set it up.

<table>
<thead>
<tr>
<th>Plotter / Samplemaker type</th>
<th>Driver</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Artios Kongsberg samplers</td>
<td>KGB</td>
<td></td>
</tr>
<tr>
<td>Artios Advantage Line samplers</td>
<td>HP</td>
<td></td>
</tr>
<tr>
<td>Data Tech IPC controller</td>
<td>DT</td>
<td>Can not use print spooler</td>
</tr>
<tr>
<td>Data Tech with PC front end</td>
<td>HP / DT</td>
<td></td>
</tr>
<tr>
<td>Elcede Vectocut</td>
<td>ELC / HP</td>
<td></td>
</tr>
<tr>
<td>Elcede IBH controller</td>
<td>ELC</td>
<td>Can not use print spooler</td>
</tr>
<tr>
<td>Grahtec</td>
<td>HP</td>
<td></td>
</tr>
<tr>
<td>Devices driven by HPGL</td>
<td>HP</td>
<td></td>
</tr>
<tr>
<td>LKS Speedplot</td>
<td>PPS</td>
<td></td>
</tr>
<tr>
<td>Alphamerics</td>
<td>HP</td>
<td></td>
</tr>
<tr>
<td>Wild TA2’s and TA10’s</td>
<td>WILD</td>
<td></td>
</tr>
<tr>
<td>Wild TA-100 series</td>
<td>HP</td>
<td></td>
</tr>
<tr>
<td>Zeta / Bruning</td>
<td>HP</td>
<td></td>
</tr>
</tbody>
</table>

**Example 1 - Setting up a Kongsberg XL44 samplemaker**

In this example, you will learn how to open Example output defaults, copy the required entry or entries to your Shared defaults, and save them. Then, if needed, you will learn how to edit the CAM Device Name to complete the setup.

1. Start Esko ArtiosCAD.
2. Click Options > Defaults.
3. Click User defaults.
4. Click File > Open > Example defaults.
5. You will see folders for CAM Tooling Setup Catalog, Customized toolbars, Die press parameter sets, Geometry Macro Catalog, Manufacturing Parameter Sets, NC Export Turning Table, Outputs, Plotting Style Catalog, and Subtype Mapping Catalog. Expand the Outputs folder until you find the Output type you are looking for. (For example: Outputs / Artios / Kongsberg XL22 - FC.)
6. You will see a number of pre-built Outputs. Select and drag Kongsberg XL44 - FC sample cutting and matrix from this side and drop it on the Outputs folder on the Shared defaults side.
7. Double-click the new entry and click the **Directories** tab. Change the directory entries to the directory that the Kongsberg front end computer will look in.

8. Click the **View** tab and note the contents of the **Specify Plotting Style** group. If the plotting style does not exist, its name will be surrounded by parentheses. If the plotting style name is not surrounded by parentheses, click **OK** and proceed to step 12; otherwise, proceed to step 9.
9. Click OK to exit this dialog box.
10. Expand the Plotting Style Catalog in Shared Defaults by clicking the plus sign next to it.
11. Look for the plotting style you noted as being used by the Kongsberg. Copy its definition from the Example defaults in the same manner as you copied the definition for the Kongsberg.
12. Make sure the Shared defaults button is selected. There will be an asterisk next to it indicating that it has been modified and needs saving. Click File and then Save to do this. Confirm by clicking Yes.
13. Click File and then Exit to leave Defaults.

You have successfully set up a Kongsberg XL44 Samplemaker.

Example 2 - Setting up an Alphamerics RS-232 samplemaker

This type of plotter can use a Microsoft shared print spooler to route the data to the particular computer/port to which the device is connected. This is preferred over using a direct connection to a port on a specific computer since it allows the device to be accessed by anyone who has access to the shared printer.

The first thing to do is set up the print spooler. Use a generic printer driver that does not modify data, such as the IBM Graphics printer driver. This driver is part of the standard driver set provided with the operating system.

1. Set up a printer on the appropriate port. Make sure that the baud rate and other RS-232 properties are set up to suit the plotter interface.
2. Let us assume that you have set up a shared printer on the computer called brunnhilde with a share name of alphamerics. Add a networked printer to your system that points to that shared printer.
3. Follow the steps from Example 1, selecting the appropriate entry in Example CAM Outputs for The Alphamerics Corp instead of Artios. Select the appropriate plotting style. (CAM.ALPHAMERICS)
4. Double-click the Output you just copied. Click the Device tab.
5. Click the Browse button (…) at the end of the CAM Device Name: field to open the Output Destinations dialog box.
6. Expand the System Printers folder by clicking the plus sign (+) next to its name.
7. Select the entry matching the printer queue, for example the \brunnhilde\alphamerics entry, and click OK.
8. You are now done. Click OK to exit this dialog.
9. Clear the Out To File checkbox.
10. Make sure the Shared defaults button is still selected. There will be an asterisk next to it indicating that it has been modified and needs saving. Click File and then Save to do this. Confirm by clicking Yes.
11. Click File and then Exit to leave Defaults.

Example 3 - Adding a Calcomp DrawingBoard III digitizer

Digitizers are somewhat different from other CAM devices as they are input-only devices. They generally come in two pieces - a flat tablet and a mouse-like puck with buttons on it.

Note: DO NOT load operating system drivers for the digitizer! They will prohibit Esko ArtiosCAD from accessing the digitizer!

Table: Suggested Digitizer Communication Settings

<table>
<thead>
<tr>
<th>Digitizer settings</th>
<th>Suggested setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baud rate</td>
<td>4800</td>
</tr>
<tr>
<td>Data bits</td>
<td>8</td>
</tr>
<tr>
<td>Parity</td>
<td>None</td>
</tr>
<tr>
<td>Stop bits</td>
<td>0</td>
</tr>
<tr>
<td>Data Mode</td>
<td>Point mode (Required)</td>
</tr>
<tr>
<td>Data format</td>
<td>CALCOMP 9100-1 format compatible</td>
</tr>
<tr>
<td></td>
<td>(APbxxxxxyyyyyy)</td>
</tr>
</tbody>
</table>
Digitizer settings | Suggested setting
---|---
Digitizer step size | 1000 lines/inch or 50 lines/mm

If choosing one of your own, it must be ASCII.

To install the digitizer, do the following:

1. Connect the digitizer to any free serial port (use COM1 for the purposes of this example) and supply it with power. Configure it to match the communications settings listed above.
2. Create an Output Destination for this port and if the serial port is on a remote computer, make sure ArtiosIO is running on the remote machine.
3. Start Esko ArtiosCAD and then click Options > Defaults.
4. Open the Digitizer Setup Catalog and select either Calcomp 16 Button or right-click and create a new data item with the name of your digitizer.
5. Select the Digitizer Parameters tab.
6. Make sure the correct Output Destination is selected in the CAM Device Name field.
7. Make sure the settings in the Digitizer Step Size group match the settings on your digitizer.
8. Press buttons 0, 1 and 2 on the puck. The Decoded Data group should look similar to the picture shown below:

Look carefully at the third character in each line of data. Starting from the top, it should correspond to the button you pressed on the puck.
Repeat this for buttons 3-F. Each digit should match up. If it doesn’t, your digitizer uses a different data format.

The digitizing menu to the left is the default digitizing menu for a Calcomp digitizer. Photocopy it at full scale or plot ..\InstLib\DIGITIZER_MENU.ARD at full size. You will need it for the next step.

1. Click the Command Map tab.
2. Tape the menu to one side of the digitizer surface.

3. Select the Use Menu checkbox so that it is checked, and then click Place Menu.

4. Move the cross-hair on the digitizing puck to the bottom left corner of the menu and click any button.

5. The puck position in the dialog box will change. Move the puck to the bottom right corner of the menu and click any button. The Menu Position and Rotation fields will now have values in them.

6. Click OK to close the Calcomp 16 button dialog box.

7. Click File, and then click Save to save the changes made to Shared Defaults. Click Yes when asked to confirm the operation.
8. Exit Defaults. Your digitizer is now ready for use.

Advanced digitizer setup

Adding a non-Calcomp digitizer

If you are not using a Calcomp digitizer, the digitizer must meet these requirements:

- The digitizer must be connected via an RS-232C (serial) port on the PC.
- The digitizer must be set to Point mode. This means that only one set of positional data is sent when a puck button is pressed.
- The data produced when a button on the puck is pressed must contain one or two characters representing which button was pressed and also strings of numbers indicating the position of the puck when the button was pressed. The data sent must be ASCII, not binary, and must be separated by a carriage return (CR), line feed (LF), or both (CR/LF).

While not required, Esko ArtiosCAD is easier to configure if the digitizer data is in a fixed format.

Once you have the digitizer connected, configured, and turned on, do the following:

1. Start Esko ArtiosCAD and then click Options > Defaults.
2. Add an Output Destination for the port the digitizer is connected to.
3. In the Shared Defaults pane, right-click the Digitizer Setup Catalog and click New, and then click Data. Enter the name of the digitizer, press enter, and then double-click it to open its property page.
4. Select the proper Output Destination in the CAM Device Name field.
5. Press a few buttons on the digitizer puck. You should see some data appear in the Decoded Data group.

The data format should be similar to one of the entries in the Data String column in the table below. The letters b and c indicate button data and should always be a digit between 0–9 or a letter between A–F.

Table: Alternate Digitizer Data Formats

<table>
<thead>
<tr>
<th>Data String</th>
<th>Button ID</th>
<th>X-Data format</th>
<th>Y-Data format</th>
<th>ID position</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before</td>
<td>After</td>
<td>No.</td>
<td>Fixed/Delim</td>
<td>No.</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>1</td>
<td>F</td>
<td>5</td>
</tr>
<tr>
<td>Data String</td>
<td>Button ID</td>
<td>X-Data format</td>
<td>Y-Data format</td>
<td>ID position</td>
</tr>
<tr>
<td>-------------</td>
<td>-----------</td>
<td>---------------</td>
<td>---------------</td>
<td>-------------</td>
</tr>
<tr>
<td></td>
<td>Before</td>
<td>After No.</td>
<td>Fixed/Delim</td>
<td>No.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>APb111112222</td>
<td>2</td>
<td>0</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td>APc555566666</td>
<td>2</td>
<td>0</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>APb 1111 2222</td>
<td>2</td>
<td>1</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td>Apc 5555 6666</td>
<td>2</td>
<td>1</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>APbV +1234 -12</td>
<td>2</td>
<td>0</td>
<td>2</td>
<td>F</td>
</tr>
<tr>
<td>ApcV +5678 +123</td>
<td>2</td>
<td>0</td>
<td>2</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>PQbb0012-00034+</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>F</td>
</tr>
<tr>
<td>PQcc00562+00029+</td>
<td>2</td>
<td>0</td>
<td>0</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>b 123 456</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td>c 197 1276</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>F</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>b 123 345</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>D</td>
</tr>
<tr>
<td>c 1234 567</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>D</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>123,456,b</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>D</td>
</tr>
<tr>
<td>87,10,c</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>D</td>
</tr>
</tbody>
</table>

Once you have determined which format the data is in, set the options in the **Format** group accordingly.
Press buttons on the digitizing puck and check that the appropriate data is appearing in the Decoded Data group. If it is not, adjust the format options until they do.

6. Now that the formatting is set correctly, check the Cursor Key Character Map. As you press a button on the digitizing puck, the corresponding key should appear pressed on screen:

The default configuration assumes the ID character generated matches the name of the button pressed on the digitizing puck.

In most cases, the button named F always produces character F in the data string, but there are digitizers that generate characters other than those expected. For example, some generate the characters ;: and > for keys D, E and F and have a map like the one shown below:
You can manually set the character map by clicking the on-screen button and replacing the value in the Current ID field with the one or two characters produced by the digitizer when the button on the digitizing puck is pressed. Repeat this procedure for each button on the digitizing puck.

You can also set the character map using only the input from the digitizer. First, make sure the formatting is set correctly. Next, click the on-screen button and then click Input from Digitizer. Click the button on the digitizing puck to enter the value for 0. The 1 on-screen button will then be automatically selected, so press the 1 button on the digitizing puck. Repeat for the rest of the buttons. If you make a mistake, simply click the on-screen button you wish to reprogram, and press the appropriate button on the digitizing puck.

7. Now that the character mapping is configured, test it by clicking Test Map. As you press buttons on the digitizing puck, the corresponding on-screen buttons should appear pressed.

Adjusting the step size of the digitizer

Over time, you may notice that your digitized geometry is not as accurate as you desire. While your digitizer may be set to have 1000 lines of resolution per inch, by digitizing a 10-inch square with extreme accuracy which results in a 10.03 by 10.06-inch square in Esko ArtiosCAD, you realize that it is out of calibration. The solution to this is to adjust the step size. Open the entry for the digitizer in Defaults and adjust the entries in the Digitizer Step Size group.

Using the values stated in the previous paragraph, change the 1000 in the drop-down list box to 1003, click the Different in Y checkbox, and change the 1000 in that drop-down list box to 1006.

Configuring the Command Map

Digitizing has 26 commands. Each can be assigned a digitizer puck key sequence and/or a digitizer Menu Item Area. If you wish to use the command while digitizing, you must assign either a Key
Sequence and/or a Menu Item Area to the command. (If you think you will never use some of the
commands, you may choose to leave them unassigned.)

The following abbreviations are used in the table below:

- `<c>` A coordinate on the digitizer.
- `<p>` An existing point. These are the end points (and intersection points) of existing lines in the
current design whether just digitized, or part of the original structure. To select such a point, you
must digitize a point on the digitizer within snap distance of one of these points.
- `<l>` An existing line. To select such a line, you must digitize a point on the digitizer within snap
distance of an existing line in the design.

Table: Digitizer Commands

<table>
<thead>
<tr>
<th>Key sequence</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>MOVE TO &lt;c&gt;</td>
<td>Digitize a coordinate on the digitizer.</td>
</tr>
<tr>
<td>12</td>
<td>MOVE TO &lt;p&gt;</td>
<td>Digitize a point on the digitizer.</td>
</tr>
<tr>
<td>0</td>
<td>LINE TO &lt;c&gt;</td>
<td>Digitize a line, starting at a coordinate.</td>
</tr>
<tr>
<td>10</td>
<td>LINE TO &lt;p&gt;</td>
<td>Digitize a line, starting at an existing point.</td>
</tr>
<tr>
<td>3</td>
<td>ARC THROUGH &lt;c&gt; TO &lt;c&gt;</td>
<td>Digitize an arc through two coordinates.</td>
</tr>
<tr>
<td>41</td>
<td>ARC THROUGH &lt;c&gt; TO &lt;p&gt;</td>
<td>Digitize an arc through a coordinate and an existing point.</td>
</tr>
<tr>
<td>14</td>
<td>ARC THROUGH &lt;p&gt; TO &lt;c&gt;</td>
<td>Digitize an arc through an existing point and a coordinate.</td>
</tr>
<tr>
<td>44</td>
<td>ARC THROUGH &lt;p&gt; TO &lt;p&gt;</td>
<td>Digitize an arc through two existing points.</td>
</tr>
<tr>
<td>77</td>
<td>CENTER &lt;c&gt; RADIUS &lt;c&gt;</td>
<td>Digitize a center and radius for a circle.</td>
</tr>
<tr>
<td>76</td>
<td>CENTER &lt;c&gt; RADIUS &lt;p&gt;</td>
<td>Digitize a center and radius for a circle, using an existing point.</td>
</tr>
<tr>
<td>7A</td>
<td>CENTER &lt;c&gt; RADIUS radius</td>
<td>Digitize a center and radius for a circle, using an existing point and a radius.</td>
</tr>
<tr>
<td>66</td>
<td>CENTER &lt;p&gt; RADIUS &lt;c&gt;</td>
<td>Digitize a center and radius for a circle, using an existing point and a coordinate.</td>
</tr>
<tr>
<td>67</td>
<td>CENTER &lt;p&gt; RADIUS &lt;p&gt;</td>
<td>Digitize a center and radius for a circle, using two existing points.</td>
</tr>
<tr>
<td>6A</td>
<td>CENTER &lt;p&gt; RADIUS radius</td>
<td>Digitize a center and radius for a circle, using two existing points and a radius.</td>
</tr>
<tr>
<td>52</td>
<td>INTERSECT &lt;l&gt; &lt;l&gt;</td>
<td>Digitize the intersection of two lines.</td>
</tr>
<tr>
<td>51</td>
<td>INTERSECT &lt;l&gt; &lt;l&gt; KEEP 1st</td>
<td>Digitize the intersection of two lines, keeping the first intersection point.</td>
</tr>
<tr>
<td>BBB</td>
<td>BLEND &lt;l&gt; &lt;l&gt; THROUGH &lt;c&gt;</td>
<td>Blend two lines through a coordinate.</td>
</tr>
<tr>
<td>BB1</td>
<td>BLEND &lt;l&gt; &lt;l&gt; THROUGH &lt;p&gt;</td>
<td>Blend two lines through an existing point.</td>
</tr>
<tr>
<td>BBA</td>
<td>BLEND &lt;l&gt; &lt;l&gt; RADIUS radius</td>
<td>Blend two lines through an existing point and a radius.</td>
</tr>
<tr>
<td>D</td>
<td>DELETE &lt;l&gt;</td>
<td>Delete a line.</td>
</tr>
<tr>
<td>E</td>
<td>UNDO</td>
<td>Undo the last action.</td>
</tr>
</tbody>
</table>
Key sequence | Command
--- | ---
F | REDO
CD | REPlot - SCALE TO FIT
CC | REPlot - CURRENT SCALE
CA | ZOOM BACK
CF | ZOOM WINDOW <c> <c>

All of these commands are mapped in the Command Map tab of the digitizer's property page in Defaults.

To set a key sequence, select the command you wish to assign to the key, and then type the key sequence to use it with in the **Cursor Key Sequence** field.

The sequence must be comprised of the characters 0–9 A–F. It may be up to six characters long.

A key sequence must be unique and may not mask another command. For example, if you wish to use the Key Sequence 2 for a command, no other Key Sequence may start with 2. This is because 2 by itself matches a command, and therefore masks a key sequence like 21.

The exception to this is when two commands differ only in what is selected — a coordinate, a point, or a line. In this case the commands may share a command sequence. If this is used, the command will snap to a point if one is within snap distance, or to a coordinate if one is not.

While editing this property page, you may temporarily have conflicts as described above. This is so you can easily change Key Sequences, or swap them to different commands. However, you will be warned about them as they are created. You may not exit the property page without resolving such conflicts.
Some commands that are not turned on by default might be very useful for your work — for example, gasket makers may want to turn on the Center tools to make circle creation easier.

Configuring the menu

You can define areas on the digitizer that act as buttons that when clicked perform any of the digitizing commands. A **Menu** is a group of such Menu Item Areas. A Menu Item Area is defined by the X and Y values adjacent to each area and the width and height of each area. These areas are best defined in Quadrant I (+X, +Y) with the origin of the digitizer in the bottom left corner.

The Menu Item Area values for the above menu are shown in the following table.

<table>
<thead>
<tr>
<th>Menu Item Area</th>
<th>X</th>
<th>Y</th>
<th>Width</th>
<th>Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>DELETE &lt;l&gt;</td>
<td>1/4</td>
<td>1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
<tr>
<td>UNDO</td>
<td>1/4</td>
<td>1+1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
<tr>
<td>BLEN[ radius</td>
<td>2+1/4</td>
<td>1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
<tr>
<td>CTR. &lt;p&gt; RAD. radius</td>
<td>2+1/4</td>
<td>1+1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
<tr>
<td>ZOOM</td>
<td>4+1/4</td>
<td>1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
<tr>
<td>PLOT</td>
<td>4+1/4</td>
<td>1+1/4</td>
<td>1</td>
<td>1/2</td>
</tr>
</tbody>
</table>

Make sure that the **Active** checkbox is selected or the Menu Item Area will not be active.

Use ArtiosCAD’s drafting features to draw the menu you have programmed. Print it out at full scale and tape it to the digitizer in the desired position. Then use the **Place Menu** button on the Command Map tab to place the menu.
Configuring Feedback and Snap

The Feedback/Snap tab of the digitizer’s Properties dialog box lets you set the snap tolerance of the digitizer, the default radius, and the sounds heard when digitizing.

When digitizing a point to satisfy a snap point \(<p>\) or a snap line \(<l>\), you need to be within the Snap Tolerance. If you are working on large corrugated designs, you may wish to set this value to something like a quarter-inch or five millimeters. Since the command indicates whether a snap point/line is expected, this value can be considerably larger than the normal screen snap value.

The Default radius field is used by some of the BLEND and CENTER commands. Set a sensible default; it is easily changed while digitizing.

The Feedback / Sounds group controls the audible feedback generated while digitizing. There are five types of feedback. Windows Sounds require a sound card in the computer; beeps can be made by most computers.

<table>
<thead>
<tr>
<th>Type of feedback</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>Data Received</td>
<td>Data received from the digitizer. If your digitizer beeps when a puck button is pressed, you may wish to turn this off.</td>
</tr>
<tr>
<td>Command OK</td>
<td>The command was successful.</td>
</tr>
<tr>
<td>Snap Point/Line error</td>
<td>You were too far away for a snap point (&lt;p&gt;) or snap line (&lt;l&gt;) to be found.</td>
</tr>
<tr>
<td>Command not recognized</td>
<td>The key sequence does not match any of those defined.</td>
</tr>
<tr>
<td>Type of feedback</td>
<td>Meaning</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Geometry Error</td>
<td>The command failed because the geometry could not be performed. (e.g. A blend would not fit.)</td>
</tr>
</tbody>
</table>

The fields in the Windows Sounds column designate which sound to use as defined in the Sounds control panel applet.

- ![Play](#) When clicked, the triangle button plays a sample of the sound or beep.
- ![Ellipsis](#) The ellipsis button lets you browse for a different sound file.

The frequency and the duration of the beeps can be set using the appropriate fields.
6. Other tasks

Uninstalling Esko ArtiosCAD

Uninstall Esko ArtiosCAD by using the Add/Remove Programs applet in Control Panel. Select it in the list of programs, click Change, click Next, choose the Remove option, and follow the prompts.

Removing individual Esko ArtiosCAD Programs

To remove individual parts of Esko ArtiosCAD such as the PDF Import/Export feature, perform a Modify and de-select the ArtiosCAD Programs to remove. After uninstallation there may be files from these components left behind in the \Esko\Artios and \Esko\Artios\ArtiosCADn.dll folders; these files may be safely deleted if desired after all Esko ArtiosCAD versions are removed.

Uninstalling SolidWorks separately from Esko ArtiosCAD

If you have purchased the SolidWorks option and need to remove the SolidWorks Original Equipment Manufacturer software without uninstalling Esko ArtiosCAD, run \ArtiosCAD\SolidWorksRemove.bat on disk 1 of the Esko ArtiosCAD media. SolidWorks OEM software is normally removed when the version of Esko ArtiosCAD that installed it is removed and no other versions of Esko ArtiosCAD are using it. This is different from removing the SolidWorks Import/Export component of ArtiosCAD in that it actually deletes the SolidWorks software from the system, not just the portions of ArtiosCAD that interface with it.

Removing MSDE

Uninstall the Esko ArtiosCAD MSDE database program files by using the Add/Remove Programs applet in Control Panel. Select Microsoft SQL Server Desktop Engine (ARTIOSCADDDB) from the list of installed programs and click Remove.

Uninstalling the database software does not remove the database data files stored in ..\Esko\Artios\DataCenter\MSDE. This is intentional in case you want to resume using MSDE. If you
will not be using MSDE again, you may delete the ..\Esko\Artios\DataCenter\MSDE folder and its contents.

Removing the Esko ArtiosCAD ODBC entry

Once all versions of Esko ArtiosCAD are removed, open the ODBC/Data Sources applet in Control Panel > Administrative Tools, click the System DSN tab, and remove the Esko ArtiosCAD datasource entry by selecting ArtiosCAD and clicking Remove.

You must remove the existing Esko ArtiosCAD ODBC entry after uninstalling all versions of Esko ArtiosCAD if you wish to change Esko ArtiosCAD database types in a subsequent installation.

Uninstalling a patch

To uninstall a patch when using Windows XP Service Pack 2 or greater, remove that particular build through Add/Remove Programs (making sure Show Updates is checked).

To uninstall a patch when using an operating system earlier than Windows XP Service Pack 2, click the Remove ArtiosCAD N.nnnn_bbb choice in the ArtiosCAD folder on the Start menu.

Esko ArtiosCAD will revert to the build that was present before the patch was loaded.

Modifying and Repairing Esko ArtiosCAD

You can modify or repair the Esko ArtiosCAD installation by clicking Change in the Add/Remove Programs applet in Control Panel, and then by choosing Modify or Repair. You must use the original Esko ArtiosCAD media, or use the same network location, as when you first installed the software.

Modify allows you to change the installed features and options as if you were performing an Advanced installation.

Repair looks for missing or corrupt files and replaces them as necessary. It will not overwrite Defaults. You will be given the same License Disk options as when you first installed the software. If you are trying to repair the licensing information choose Yes, I have licensing information to apply and use the licensing diskette, otherwise choose License information already applied or unavailable to keep your current licensing information.

Loading an Esko ArtiosCAD patch

Esko ArtiosCAD patches are installed using a single executable file. To install a patch, do the following:

1. Log on as Administrator, or as a user with local administrative privileges.
2. Using Windows Explorer, double-click the patch file. Patch files are named ArtiosCAD<version and build number>.exe, such as ArtiosCAD7.1Build497.exe.
3. In the Welcome dialog box, click Update >.
4. A status bar will appear showing the progress of the patch installation.
5. Click Finish in the InstallShield Wizard Completed dialog box. The patched version of Esko ArtiosCAD is now ready for use.
6. Log off the system.

Working with the ArtiosIO Status program

The ArtiosIO Status program is on the Start menu in the ArtiosCAD folder. It queries the ArtiosIO program on a given machine and displays the active connections to that machine. Using it, you can close a connection that is no longer responding.

To view the connections on a machine, click Query Server and either enter the name of the system to query, or click the drop-down list button to view a list of available machines on the network. Only machines in the same domain will be shown.

To close a connection, highlight it and click Close Connection.

Using Floating Licenses

Floating licenses allow Esko ArtiosCAD users to share software licenses among computers. Each license is accessed on an as-needed basis.

Esko ArtiosCAD allocates licenses based on the active tools and toolbars. Tools and toolbars for which licenses are unavailable will either be unavailable or a dialog box similar to the one shown below will appear. The dialog box lists which users and systems have allocated the licenses you
need. You can either go ask someone to give up the license you need and click **Wait for License**, or you can click **Cancel** and return to the previous tool you were using.

Release licenses by exiting the design, manufacturing file, or 3D workspace which took the license. If you click **Wait for License**, a dialog box showing the needed license appears:

When a license becomes available, this dialog box disappears and the desired feature is activated.

**Configuring floating licenses**

The **Floating License Configuration** program in the **Esko > ArtiosCAD** program group on the Esko ArtiosCAD License server lets you allocate floating licenses. You can reserve or deny licenses to certain systems. When started, the program initially appears similar to the following dialog box (but with your specific licenses listed):
Double-click a license to configure it. A dialog box will appear in which you can reserve and deny licenses:

Reserving a license means that the system you specify will always have the license reserved for it. Denying a license means that the system you specify will never be able to have a license. To specify a system, click Add. Enter the short name of the system exactly as it is shown on the Computer Name tab of the System applet in Control Panel. The short name is the first word up to the period - the full name minus the domain name. For example, if the full name of the system is ArtCad1.plasticangelcontainers.com, the short name is ArtCad1. Make sure to match the case exactly, as it will not work otherwise. Once you have entered the name, click OK.
To remove a system from either the Reserved or Denied lists, select it, click Remove, and click OK.

When you have finished configuring the license as desired, click OK to return to the Floating License Configuration dialog box, or click Cancel to ignore the changes you made.

To exit the Floating License Configuration dialog box, click License, and then click Exit.

**Note:** If you have made changes to the configuration, the License Manager will be shut down and restarted when you exit the Floating License Configuration program. It takes the Elan License Manager three minutes to restart after shutting down.

**Troubleshooting**

If you have trouble using a desired feature and get a dialog box like the one below, use either the ArtiosLM applet in Control Panel or the ArtiosCAD License Management utility on the Esko ArtiosCAD License server to make sure the server is running.

If the Floating License Utility does not start, it is because there are no floating licenses to configure.
Upgrading Esko ArtiosCAD Licenses

Esko ArtiosCAD licenses enable you to use the features you have bought and disable the features you have not bought. As your needs change, you may have to change your licenses accordingly.

When you upgrade a license, you usually receive a licensing file by e-mail. You then have to install that file properly as described in the following section.

You must log on to your system with administrative privileges to upgrade licenses.

**Note:** NO SYSTEM MAY HAVE INSTANCES OF WebCenter, Esko ArtiosCAD, DataCenter Admin, OR Esko SpecLink OPEN WHEN UPGRADING LICENSES!

License Manager: keyless

To upgrade a keyless license for the FlexNet license manager, do the following:

1. Save the licensing file sent to you by Esko in a place where you can access it from the machine whose license you will be upgrading, for example a network location, on a USB thumb drive, and so forth.

2. On the machine with the license to upgrade, log in as Administrator or as a user with administrative privileges.

3. Click Start > Programs/All Programs > Esko > ArtiosCAD > Activate Local Licenses/Activate Server Licenses.

4. In the Esko (Local or Server) License Manager dialog box, click Activate.

5. Click Next in the Activate License Wizard dialog box.

6. In the On-line activation or off-line activation dialog box, choose either On-line activation or Off-line activation as appropriate for your system and click Next.

7. In the Your product keys dialog box, choose I have a file containing one or more product keys and click Next.

8. In the File containing your product keys dialog box, browse for the file you saved in step 1 and click Next.

9. In the Esko Account dialog box, enter your Esko licensing account name and password and click Next.

10. In the Activation completed successfully dialog box, the license(s) you activated should appear. Click Finish to complete the license upgrade.

11. Optionally, click Save Licenses Report to save a report of the license upgrade for your records.

12. Close the license manager dialog box by clicking the X in the upper right corner and test the new license as desired.
Changing the Esko ArtiosCAD licensing method

In the Esko > ArtiosCAD folder on the Start menu is the Esko ArtiosCAD License Type Configuration utility. As an administrative user, use it to view the current licensing method, switch between local and network licenses, or change the license server if you are using a network license.

DataCenter database administration when using MSDE or SQL Server 2005 Express Edition

All database products require some amount of administration regardless of vendor. Failure to perform this administration will ultimately lead to a loss of some or all of your DataCenter data.

If you have any questions on these tasks, contact Support.

Use DataCenter Admin to backup and restore the MSDE or SQL Server 2005 Express Edition database.

Backing up the database

The database backup procedure stops the database service, copies the database files to \Esko \Artios\DataCenter\MSDE\MSSQL$ ARTIOSCADDDB\Data\backup for MSDE or \Esko \Artios\DataCenter\SQL2005Express\MSSQL.1\MSSQL\Data\backup for SQL Server.
2005 Express Edition, and then restarts the service. You must then copy that folder to backup media, or include it in your backup program, to fully backup the database.

To backup the MSDE database, do the following:

1. Ensure that any programs using the database are not running.
2. Log on to the database server as Administrator or as a member of the Administrators group.
3. Start DataCenter Admin by clicking Start > Programs > (Esko) > ArtiosCAD > DataCenter Admin, or by using the icon on the desktop.
4. Click File > Backup database.
5. In the Login as the Database Administrator dialog box, enter the name and password of the database administrator and click OK. The defaults are sa and a blank password for MSDE and sa and oicu812! for SQL Server 2005 Express Edition. Click OK when done.
6. The Status bar indicates the current operation. When the Status bar is clear, the backup is complete.

Restoring the database

The database restore procedure stops the database service, replaces the data in the database with data from the backed-up database files from \Esko\Artios\DataCenter\MSDE\ MSSQL $ARTIOSCADDDB\Data\backup for MSDE and \Esko\Artios\DataCenter\ SQL2005Express \MSSQL.1\MSSQL\Data\backup for SQL Server 2005 Express Edition, and then restarts the database service.

To restore the database, do the following:

1. Ensure that any programs using the database are not running.
2. Log on to the database server as Administrator or as a member of the Administrators group.
3. Start DataCenter Admin by clicking Start > Programs > (Esko) > ArtiosCAD > DataCenter Admin, or by using the icon on the desktop.
4. Click File > Restore database.
5. In the Login as the Database Administrator dialog box, enter the name and password of the database administrator and click OK. The defaults are sa and a blank password for MSDE and sa and oicu812! for SQL Server 2005 Express Edition. Click OK when done.
6. The Status bar indicates the current operation. When the Status bar is clear, the restore is complete.

Database space management for MSDE

The MSDE database files automatically expand to accommodate new data. After a few months, to compact the database, do the following:

1. Make sure no programs which use the database are active. This includes Esko ArtiosCAD, Esko SpecLink, DataCenter Admin, and any products using CAD-X.
2. Log on to the database server as Administrator or as a member of the Administrators group.
3. Start a command prompt, and change directories to C:\Program Files\Microsoft SQL Server\80\Tools\Binn.
4. Type osql -Usa -P"" -Sdatabase_server_name\ArtiosCADdb and press enter.
5. At the 1> prompt, type `use ArtiosCADdb` and press enter.
6. At the 2> prompt, type `go` and press enter.
7. At the 1> prompt, type `dbcc shrinkfile (acaddata)` and press enter.
8. At the 2> prompt, type `go` and press enter.
9. At the 1> prompt, type `dbcc shrinkfile (acadlog)` and press enter.
10. At the 2> prompt, type `go` and press enter.
11. At the 1> prompt, type `exit` and press enter.

Your screen should look similar to the one shown below.

**MSDE-related Application Log entries in Event Viewer**

On the database server, the MSSQL\ARTIOSCADDB service writes warning entries of category (8) to the Application Log in Event Viewer. They are about connecting to Active Directory and can be ignored.

**Using Microsoft SQL Server 2005**

Microsoft SQL Server 2005 is a large, robust database which can require complex administration. The installation documentation provided below is provided as a convenience only for initial installation. A dedicated Database Administrator (DBA) is required to administer and maintain the database, and is free to choose options other than those documented to meet local site requirements.
Make sure to load Esko ArtiosCAD specifying **Microsoft SQL Server or Oracle** as the database type before installing Microsoft SQL Server 2005.

After installing Esko ArtiosCAD, the steps to using Microsoft SQL Server 2005 with a new installation of Esko ArtiosCAD are installing Microsoft SQL Server 2005, configuring Microsoft SQL Server 2005, configuring the ODBC driver, and running the database configuration batch file for Esko ArtiosCAD.

Once all that is done, you can configure servers and resources in DataCenter Admin and start using Esko ArtiosCAD.

### Installing Microsoft SQL Server 2005

Perform the following steps to install Microsoft SQL Server 2005 for the first time to use with Esko ArtiosCAD.

1. Make sure the computer which you intend to make the database server meets Esko’s recommended hardware specifications.
2. Log on as **Administrator** or as a user with administrative privileges.
3. Put the Microsoft SQL Server 2005 media into the media drive. It should automatically start. If it does not, navigate to its root directory using Windows Explorer and double-click `Autoplay.exe`.
4. In the dialog box that appears, read the license agreement. If you agree with it, select **Yes, I accept the licensing terms and conditions** and then click Next to continue the installation.
5. In the Installing Prerequisites dialog box, click **Install**. A progress bar shows the progress of the task.
6. Click **Next** when the prerequisite components are installed.
7. In the Welcome to the Microsoft SQL Server Installation Wizard dialog box, click **Next**.

8. The installation wizard will perform a system configuration check and install any needed components. Click **Next** when it finishes.
9. The Installation Wizard will continue and the Registration Information dialog box will appear. Enter your name, company, and Product Key as appropriate. Click Next when done.
10. In the Components to Install dialog box, click **Advanced**.
11. In the Feature Selection dialog box, click the box next to Database Services and click **Entire feature will be installed on the hard drive.**

![Microsoft SQL Server 2005 Setup dialog box](image)

12. Still in the Feature Selection dialog box, expand the Client Components catalog, click the box box next to Connectivity Components, and click **Entire feature will be installed on the hard drive.** Do the same for Management Tools and click **Next.**
13. In the Instance Name dialog box, click **Named instance** and enter a sensible name in the field. You will need to know this instance name later, so referencing Esko ArtiosCAD would be prudent.
14. In the Service Account dialog box, click **Use the built-in System account** and change the drop-down list box entry to **Network service**. Click **Next**.

15. In the Authentication Mode dialog box, click **Mixed Mode**, and enter the password of **oicu812!** in both password fields. Click **Next** after entering both passwords.
Note: You may change the sa password as desired, but it must be set to oicu812! when running the database scripts batch file when installing or upgrading Esko ArtiosCAD. If the password is not set correctly, the configuration scripts will fail.

16. In the Error and Usage Report Settings dialog box, leave both checkboxes clear and click Next.
In the Ready to Install dialog box, click **Install** to begin the installation.
18. The Setup Progress dialog box shows the status of the installation. Click Next when the tasks have completed.

19. In the Completing Microsoft SQL Server 2005 Setup dialog box, click Surface Area Configuration tool.
20. In the Minimize SQL Server 2005 Surface Area dialog box, click **Surface Area Connection for Services and Connections**.

21. In the resulting dialog box, click **Remote Connections** under Database Engine, click **Local and remote connections**, and click **Using TCP/IP only**. Click **Apply** when done.
22. In the Connection Settings Change Alert dialog box, click OK.

23. Click Service under Database Engine.
24. Click Stop, then Start to restart the database engine service.
25. Click OK to exit the Surface Area Configuration for Services and Connections dialog box.
27. In the Microsoft SQL Server 2005 Setup dialog box, click Finish. Reboot the system if prompted to do so. Do not postpone this reboot.

Note: At this point, apply the latest Microsoft SQL Server 2005 service packs and patches available from the Microsoft web site at http://www.microsoft.com/sql/default.mspx.

Configuring Microsoft SQL Server 2005

Once you have updated and patched Microsoft SQL Server 2005 appropriately, continue as follows. You may have to start the database by hand by right-clicking the SQL Server icon in the system tray and selecting MSSQLServer - Start.
Creating the ArtiosCADdb database

1. If you rebooted the machine, log in as a user with administrative privileges.
2. Click Start > Programs > Microsoft SQL Server 2005 > SQL Server Management Studio.
3. In the Connect to Server dialog box, ensure that the Server name: field is set to the instance created in the previous procedure, change the Authentication type: drop-down list box to SQL Server Authentication, and enter sa for the username and oicu812! for the password. Click Connect.

4. Right-click Databases and click New Database on the context menu.

5. In the New Database dialog box, in the Database Name: field, enter ArtiosCADdb. Do not click OK.
6. Click the 3 in the Initial size (MB) column and change it to 10. Do not click OK.
7. If this is a normal computer with one hard drive and two hard drive controllers, click **OK**.

If this is a high-end database server and has multiple hard drives and multiple controllers, use the scroll bar to scroll to the right, click the Browse (...) button for the ArtiosCADdb Log file, and locate the transaction log on a separate disk on a separate controller. This will minimize data loss when a hard drive crashes. Click **OK** when done.

The ArtiosCADdb database should appear in the group of databases as shown below.

![Database creation screen](image)

The database is now created and the next step is configuring the ODBC driver.

**Configuring the ODBC driver**

Microsoft SQL Server 2005 requires the ODBC driver provided by MDAC 2.8 or later. It is installed on the server when you install the database, and on client systems by Esko ArtiosCAD when it is loaded.

If the system will not have Esko ArtiosCAD on it, load MDAC 2.8 on it by selecting the Connectivity Components element of the Client Components catalog when running the Microsoft SQL Server 2005 setup program.

To configure the ODBC driver on both the server and on clients, do the following:
1. You should still be logged in as Administrator or as a member of the Administrators group. If not, log in as such.

2. Open the Data Sources (ODBC) applet in Control Panel. Depending on the operating system, it may be in Administrative Tools.

3. Click the System DSN tab, and then click Add.

4. Select SQL Server from the list of available drivers and click Finish.

5. In the Create a New Data Source to SQL Server dialog box, enter ArtiosCAD in the Name: field and select the server and instance name in the Server: drop-down list box.
   
   If this is a client, specify the name of the database server.
   
   Click Next.

6. When asked about authenticity, select the With SQL Server authentication... choice. In that same dialog box, deselect the Connect to SQL Server... checkbox. Do not click Next.
7. Click Client Configuration. The Add Network Library Configuration dialog box appears.

Make sure TCP/IP is selected in the Network libraries group and ensure the server name is correct. If (local) was selected in the Which SQL Server do you want to connect to? field in Step 5, only a period (.) appears in the Server alias field and Server name field.

Ensure that Dynamically determine port is checked and that the Port number is set to 1433. Click OK.

Click Next in the Create a New Data Source to SQL Server dialog box.

8. In the next dialog box, leave the Create temporary stored procedures... and Use ANSI... choices selected and click Next.
9. In the next dialog box, leave **Perform translation for character data** selected.
   
   If the system is using non-U.S. locale settings, check the **Change the language of SQL Server system messages to:** checkbox and set the value to **English**.
   
   Click Finish.

10. The ODBC Microsoft SQL Server Setup dialog box will appear showing a summary of the configuration. Click **OK**. Do not test the configuration; it will fail because the proper database user is not configured because the database configuration batch file has not yet been run.
11. Click OK in the ODBC Microsoft SQL Server Setup dialog box.
12. Click OK in the ODBC Data Source Administrator dialog box.

The ODBC connection on the server is now configured properly. The next step is running the database configuration batch file.

Running the Esko ArtiosCAD database configuration batch file

Make sure the ODBC connection to the Esko ArtiosCAD database is configured as described in the previous section before running the batch file.

To run the Esko ArtiosCAD database scripts, do the following:

1. You should still be logged in as Administrator or as a member of the Administrators group. If not, log in as such.
2. Start a command prompt and change the directory to \Esko\Artios\DataCenter.
3. Build the database schema. The schema build command is in the form

   Build_MSSQLServerSchema "<sa_user_password>" "<instance name>" "<DataCenter root directory>" "<DataCenter program language directory>".

   The command should be similar to the one shown below, substituting your installation information and language as needed.

   Build_MSSQLServerSchema "oicu812!" "MBCCADSERVER\SQL2K5_ArtiosCAD" "C:\Esko\Artios\datacenter" "C:\Esko\Artios\datacenter\program\english"

   When you have typed the command, press enter. Comments and commands will appear in the command prompt similar to example shown below.
4. When the command prompt returns, type `exit` and press enter.

The database is now ready for use with Esko ArtiosCAD and DataCenter Admin. Add servers and resources as described in the Working with servers and resources section of the DataCenter chapter of the Esko ArtiosCAD User Guide.

Notes and troubleshooting

Microsoft Data Access Components (MDAC) version 2.8 or higher must be used with Microsoft SQL Server 2005.

The following notes deal with using SQL Server Management Studio in the Microsoft SQL Server folder on the Start menu.

- If error messages appear about not being associated with a trusted SQL Server connection, right-click the server instance name, and then click Properties > Security. Make sure SQL Server and Windows Authentication mode is selected in the Server authentication group.

- If many script errors appear, make sure the Implicit transactions checkbox is clear on the Properties > Connections page in the Default connection options list.

- If error messages appear about the programmer user not existing or not having permission, do the following:
  - Expand the database instance in the left pane so the Security folder is open and double-click Logins.
  - Right-click Programmer and click Properties.
  - On the General page, there should be a password set in the Password: field, the Default database: should be set to ArtiosCADdb, and the Default language: should be set to English. If this is not so, click the Server Roles page and ensure that public is checked.
If you are using SQL Server 2005 Express Edition and you need to run the database script batch file, use `Build_ExpressSchema.bat` with the same syntax as `Build_MSSQLServerSchema.bat`.

### Using Oracle

Instruction on installing Oracle is beyond the scope of this document. It should only be done by experienced database administrators. Here is a rough overview of the steps required:

- Install Oracle and create an instance. Within that instance create a database. Set up network access to it using Net Configuration Assistant to connect to that database.
- Install Esko ArtiosCAD on the database server, making sure to do an **Advanced** installation and selecting **Microsoft SQL Server 2000 or Oracle** as the database choice.
- Set up an ODBC Data Source using the Microsoft ODBC for Oracle driver. Set the **Data Source Name**: to **ArtiosCAD**, the **Description**: to **ArtiosCAD**, and the **Server**: to `TNS:Name_of_database_defined_in_Net_Configuration_Assistant` that you configured when installing Oracle, such as `TNS:orcl`.
- Run the database configuration batch file to build the database schema.

Start a command prompt and change the directory to `\Esko\Artios\DataCenter`.

The schema build command is in the form `Build_OracleSchema "<sys Password>" "<TNS name>" "<DataCenter root directory>" "<DataCenter program language directory>"`.

If the **sys** database user has no password, use a double set of quotation marks (""").

The final command you type at the command prompt should be similar to the one shown below, substituting your installation information and language as needed.

```
Build_OracleSchema "administrator" "orcl" "C:\Artios\datacenter" "C:\Artios\datacenter\program\english"
```

- Add servers and resources as described in the **Working with servers and resources** section of the **DataCenter** chapter of the **Esko ArtiosCAD User Guide**.

**Note:** If WebCenter and your production installation of Esko ArtiosCAD (not the one required for WebCenter's Application server) are using the same database server with Oracle 9i, and you upgrade the version of Oracle on the database server, it may break the Microsoft ODBC for Oracle drivers required by Esko ArtiosCAD. Please note that this is NOT a supported configuration and you alone are responsible for ensuring that it works. If you upgrade Oracle and start getting `SQLSetConnectAttr` errors, try changing two entries in the Registry on the database server. Specifically, in `HKEY_LOCAL_MACHINE > Software > Microsoft > MsDTC > MTxOCI`, change the `OracleSqlLib` key to `orasql9.dll` and change the `OracleXaLib` key to `oraclient9.dll`.

### Replacing an Esko ArtiosCAD server

As time passes, you may need to replace an existing Esko ArtiosCAD server with a newer, faster model. Listed below are the basic steps required to replace a machine acting as the **Defaults server**,
Database server using MSDE, and License server using a security key. Contact Esko Support for more details.

1. Make a complete backup of the system that is being replaced and verify that the data is readable. Keep this backup in a safe, easily-accessible place.

2. Install the same version of Esko ArtiosCAD on the new server as is currently on the existing server. Make sure to specify the same licensing method.

3. Copy the ..\Esko\Artios\ArtiosCADn.nn\ServerLib folder from the existing server to the new server.

4. Copy ..\Esko\Artios\ArtiosCADn.nn\Common\tuneopt.txt and/or tuneopt7.txt on the old server to the corresponding directory on the new server.

5. Move the security key to the new server.


7. Go through the various entries and adjust them as necessary, specifically the entries for Output devices and other things which may contain system names or paths.

8. An exact image copy of the database from the old server will be made on the new server. Therefore, put the user files (if any) in the same location in the file system on the new machine as on the old machine.

9. On the old system, detach the database using the following procedure:
   - Log in as a member of the Administrators group.
   - Start a command prompt.
   - At the prompt, type `osql -Usa -P"" -Ssystemname\ArtiosCADdb` and press enter. (If the `sa` password is not blank, insert it between the quotation marks in the `-P` switch.)
   - At the 1> prompt, type `sp_detach_db 'ArtiosCADdb'` and press enter.
   - At the 2> prompt, type `go` and press enter.
   - At the 1> prompt, type `quit` and press enter.
   - Type `exit` and press enter to close the command prompt.

10. On the new server, do the same thing to detach the empty, just-installed MSDE database:
    - Log in as a member of the Administrators group.
    - Start a command prompt.
    - At the prompt, type `osql -Usa -P"" -Ssystemname\ArtiosCADdb` and press enter. (If the `sa` password is not blank, insert it between the quotation marks in the `-P` switch.)
    - At the 1> prompt, type `sp_detach_db 'ArtiosCADdb'` and press enter.
    - At the 2> prompt, type `go` and press enter.
    - Do not close this program as it is needed for a future step.

11. Copy the database files from the old server to the new server. They are in the ..\Esko\Artios\DataCenter\MSDE\MSSQL\SARTIOSCADDB\Data directory. Copy ArtiosCADdb_data.mdf and ArtiosCADdb_log.ldf to the corresponding directory on the new server. If prompted to overwrite existing files, say yes.

12. On the new server, attach the new database files to the instance:
    - In the command prompt window running osql, at the 1> prompt, type `sp_attach_db "ArtiosCADdb", "C:\Esko\Artios\DataCenter\MSDE\MSSQL\SARTIOSCADDB\Data\ArtiosCADdb.mdf"` and press enter. Note: Do not omit the comma. Also, change the path to the .mdf file as needed if you did not install Esko ArtiosCAD to the C: drive.
    - At the 2> prompt, type `go` and press enter.
13. Reconnect the programmer database login to the database instance:

- At the 1> prompt, type `use ArtiosCADdb` and press enter.
- At the 2> prompt, type `go` and press enter.
- At the 1> prompt, type `sp_change_users_login update_one, programmer, programmer, null` and press enter.
- At the 2> prompt, type `go` and press enter.
- At the 1> prompt, type `quit` and press enter.
- Type `exit` and press enter to close the command prompt.

14. In DataCenter Admin, go through the entries for servers and resources and adjust them as necessary.

15. If the old server used ArtiosIO, configure and start it on the new server.

16. On each client, use Add/Remove Programs in Control Panel to Change and then Modify the installation of Esko ArtiosCAD. Specify the name of the server as appropriate.

Esko ArtiosCAD should now be fully functional on the new server and any clients.
7. Defaults

Introduction to Defaults

Defaults store all the configuration information used by Esko ArtiosCAD. This is where you change options, add custom standards to the Standards Catalog, define Outputs, and so forth. You access defaults by clicking **Defaults** on the Options menu. There are many options in Defaults that are not covered in this chapter; you should open each Defaults catalog or folder and review its contents.

There are two types of defaults: **Shared defaults** and **User defaults**. Shared defaults are shared by all users served by the same Esko ArtiosCAD server. User defaults affect only the machine being used.

Creating defaults

There are three ways to create new defaults:

1. In the Shared defaults pane, open the folder containing the Defaults you want to add to and right-click. Clicking **Folder** adds a new folder; clicking **Data** adds a new default setting.
2. In the User defaults pane, make sure nothing is selected, and right-click. Choose the type of default to add from the pop-up context menu. When naming the new item, do not use a backslash (\).
3. Drag a pre-existing default from one pane into the other and modify as desired.
Changing defaults

To change a default, open the folder it is in and double-click it. A dialog box will appear showing all the settings you can change.

If you want to change all the defaults in a folder at once, double-click the folder instead of an individual item, and a tabbed dialog box will appear containing all the individual items.

Saving defaults

It is very important to save the defaults after you make changes. The option button above the two panes indicates which set of defaults is being affected by the commands on the File menu. Make sure the option button is set to the pane of defaults you have changed, click File, and then click Save.

Defaults performance

ArtiosCAD loads Defaults into system memory when you launch it for more robust performance.

If you make changes to Defaults, click Options > Clear Defaults Cache to re-read Defaults into memory without having to restart ArtiosCAD.

ArtiosCAD re-reads the Defaults automatically when you click Options > Defaults, so you only need to clear the cache when you make a change and exit Defaults and then continue working in ArtiosCAD. Another time to clear the cache would be when you changed Defaults on one computer and other users were sharing those Defaults on other computers and did not want to restart ArtiosCAD; they would have to click Clear Defaults Cache to see your changes.

Parameter sets

Designer parameter sets

Parameter sets are collections of settings that govern how you work with designs or manufacturing files. The collection can be based on anything – by customers, by machines, or however else you want to group information.

Single design parameter sets allow you to assign different values to different sets for the following entries:

- Default bridging formulae to use
- Flute/grain setting
- Side up
You select a parameter set when creating a design. You can change the parameter set being used by a design by clicking Change Parameter Set on the Options menu.

To create a parameter set, do the following:

1. Start Esko ArtiosCAD.
2. Click Options, and then click Defaults.
3. To add a parameter set to Shared Defaults, open the Single Design Parameter Set entry. Right click it, select New, and then click Data.
4. Enter the name of this new parameter set and press Enter.
5. Click the plus sign (+) next to the parameter set you created.
6. Double-click Startup defaults. This will bring up a tabbed dialog box where you can set the initial side of the design, a default flute/grain direction, and default bridging formulae.
7. Click OK when you are done setting options for this new parameter set.
8. Click File, and then click Save to save the changes.

Manufacturing parameter sets

Manufacturing parameter sets, in conjunction with machine parameter sets, allow you to customize various settings per set or per machine. A partial list of the values you can set are:

- Die board size
- Automatic stripping parameters
- Default bridging formulae
- Hole mounting pattern workspaces and the side of the die used to determine if they are mirrored or not
- The Machine Reference Point, used by many things for alignment
- Settings for scrap/chop knives
- Mounting alignment hole parameters

A manufacturing parameter set references a die press parameter set and a printing press parameter set in addition to containing its own values.

To create a die press or printing press parameter set, do the following:

1. Start Esko ArtiosCAD.
2. Click Options, and then click Defaults.
3. To add a parameter set to Shared Defaults, open the Die Press Parameter Sets or Printing Press Parameter Sets entry. Right click it, select New, and then click Data.
4. Enter the name of this new parameter set and press Enter.
5. Click the plus sign (+) next to the parameter set you created. The parameter set expands to show the top-level categories. Click each plus sign to open each category.
6. Double-click each entry, verify or change the settings within the dialog box that opens, and click OK when done. Repeat as desired for other settings.

7. Click File, and then click Save to save the changes, clicking Yes to overwrite previous Defaults.

Once the die press and printing press parameter sets are created, they can be incorporated into a manufacturing parameter set.

To create a manufacturing parameter set, follow the same procedure as creating other parameter sets: right-click Manufacturing parameter sets, click New, and so forth. Next, do the following:
1. Double-click the name of the parameter set to associate die and printing press parameter sets with it. Click OK once you have chosen sets.

![Parameter Set Example](image)

2. Open the parameter set and enter values for all the fields. Your Defaults pane with all catalogs expanded should look similar to the picture shown below.
3. Click OK in each parameter’s dialog box to save the changes.
4. Click File, click Save to save the changes, and then click Yes to overwrite previous Defaults.

**Note:** Changing the default die press parameter set and default printing press parameter set has no effect when choosing a manufacturing parameter set. To change the die press and printing press parameter sets used by a manufacturing parameter set, change the manufacturing parameter set itself, and not the default die press and printing press selections.

**Counter parameter sets**

As with single design and manufacturing parameter sets, Counter parameter sets group counter parameters together for easy manipulation. You can copy the Artios counter parameter set and
modify it to meet your unique needs, or you can create a new parameter set by right-clicking and selecting New on the context menu.

Double-click a parameter set to open it for editing.

**Tool Angles, Widths tab**

Shown below is Tool Angles, Width tab of the Artios counter parameter set.

Fields followed by f(x) indicate that the parameter being set is dependent on the board. Clicking f(x) leads to the Edit Expression dialog box, where the following parameters should be used to build expressions:

- #CWIDTHWG: Width of with grain creases
- #CWIDTHXG: Width of cross grain creases
- CAL: Board caliper

The five parameters in this dialog box are configured like this:

<table>
<thead>
<tr>
<th>Description</th>
<th>Workspace Variable</th>
<th>Default Expression</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Counter thickness</td>
<td>CHGT</td>
<td>ROUNDUP(CAL+0.007*#i, 0.002*#i)</td>
<td>Affects the periphery and chamfer tool offset.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Description</th>
<th>Workspace Variable</th>
<th>Default Expression</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Counter thickness</td>
<td>CHGT</td>
<td>ROUNDUP(CAL+0.007*#i, 0.002*#i)</td>
<td>Affects the periphery and chamfer tool offset.</td>
</tr>
<tr>
<td>Description</td>
<td>Workspace Variable</td>
<td>Default Expression</td>
<td>Notes</td>
</tr>
<tr>
<td>---------------------------</td>
<td>--------------------</td>
<td>--------------------------</td>
<td>----------------------------------------------------------------------</td>
</tr>
<tr>
<td>Crease channel depth</td>
<td>CRECHD</td>
<td>CAL+0.002*#I</td>
<td>For information only. Does not affect tool paths.</td>
</tr>
<tr>
<td>Chamfer depth</td>
<td>CHAMFD</td>
<td>CHGT-0.004*#I</td>
<td>Automatically limited by maximum chamfer width.</td>
</tr>
<tr>
<td>With grain channel width</td>
<td>CHWWG</td>
<td>#CWIDTHWG+2*CAL</td>
<td></td>
</tr>
<tr>
<td>Cross grain channel width</td>
<td>CHWXG</td>
<td>#CWIDTHXG+2<em>CAL+0.004</em>#I</td>
<td></td>
</tr>
</tbody>
</table>

The **Maximum Chamfer Diameter** is usually the diameter of the chamfer tool head.

**Periphery tab**

The **Periphery** tab of the counter parameter set dialog box contains settings which affect the periphery tool.

The parameters on this page of the dialog box are all measurements whose functions are illustrated in the pictures next to them except for the Periphery Tool Angle. The angle specified in the **Periphery Tool Angle** field, along with the thickness of the counter material, determines the width of the periphery.
Crease End tab

The Crease End tab of the counter parameter set dialog box controls how the crease ends are machined.

The options in the Crease End Option group, Use Periphery Tool Only and Use Crease End and Periphery Tools affect the display of the rest of the tab. If Use Periphery Tool Only is selected, all the pictures change to omit any reference to the Crease End tool's existence.

Partial cut tab

Partial cuts and glue assist lines can have the counter extend underneath them if you so desire.
By default, Partial Cuts are set to be On the counter. Glue assist lines are set conditionally by default. They are off the counter if the distance between the crease and the closer end of the line is greater than the minimum specified in the Minimum field. Of course you can specify them to be On or Off all the time.

Cut and Crease tab

The options on the Cut and Crease tab affect the creation of bowties, which are special constructs made necessary by having cuts and creases of such short length. All of the options are explained in the pictures.

Chamfer tab

The options on the Chamfer tab affect the counter at those places in the design where cut lines and the ends of crease lines meet. The chamfer mostly follows the periphery except it always stays a specified distance away from crease channels.
The **Chamfer Gap** is the distance between the inside edge of a chamfer and the crease channel. The **Pickup** and **No Pickup** options control whether the chamfer is continuous around the end of the crease channel. The **Chamfer Tool Angle** is the angle of the tool, which, together with the thickness of the counter material, determines the width of the chamfer channel.

**Reverse Creases tab**

Reverse creases are used to make a panel fold outward instead of inward.

The **Insert** option indicates that a piece of foreign material will be attached to the counter to create the reverse crease. This is shown in the top picture on the tab. **Counter material** means that an area around the reverse crease will be milled out so that the board gets pushed into the depressed area. This is shown in the bottom picture on the tab. The **Crease Width** field becomes available when this option is selected. **Off the counter** disables all the fields.
Reverse Cuts tab

Reverse cuts are made on the counter in the same way as the Insert method for reverse creases. The parameters on the Reverse Cuts tab are explained in the picture.

Embossing tab

An area of the board can be embossed by milling out the part of the counter surrounding the embossed area. As with other tabs, the parameters on the Embossing tab are explained in the picture.

Position Holes tab

The Position Holes tab contains one parameter - the workspace name for the position hole. Click Specify to browse for a new workspace.
Tack Bridges tab

The **Tack Bridge Width** field on the **Tack Bridges** tab sets the width of tack bridges in the periphery of the counter.

Name tab

The choices on the **Name** tab affect the output of the Name tool on the Counter toolbar. These choices include the font to use, its size, and its position, as shown in the picture.

Rubber types

Esko ArtiosCAD has a default Rubber types catalog. Review its entries and modify or add to them as necessary to meet your needs.
A rubber type is defined by its unique name, thickness, and stock sheet size. If there are rubber types defined that are different thicknesses of the same material, include the thickness in the name, as each name must be unique.

To edit a rubber type, double-click its name. To create a new rubber type, right-click the Rubber types catalog and click New on the context menu.

Shown below are the properties for the Red Neoprene rubber type.

Reversible indicates if the top and bottom sides of the rubber are the same, so that a piece of rubber and its mirror image could be considered the same when making layouts. (This is different from the mirror options in Auto-Repeat.)

The value in the Rubber code: field is what is used on the Sheet Filename Generation page of the rubber sheets Output. Use letter and numbers only with no spaces.

Set the Sheet width: and Sheet height: fields to the appropriate sizes of the sheets for this type of rubber.

The Thickness: field is currently unused, but to prepare for its possible use in the future, enter the thickness of the sheet.
Select color sets the color of the rubber when Rubber Fill is selected in the Rubber View Options dialog box. Clicking it opens the standard operating system color selection dialog box.

The four option buttons set the type of the rubber.

- **Isotropic** means it is the same in all directions, so can be rotated as necessary.
- **Wave** means it is rippled, and the long dimension of the rubber must cross the ripples. This type of rubber is generally used in long, narrow slots. Wave rubber pieces are put on the sheet with their longest dimension horizontally regardless of the state of Fill from left.
- **Strip/Profile** means that it is premanufactured rubber that is not calculated for layouts, but can be drawn on the dieboard so that it can be placed correctly after manually being cut.
- **Plastic cover** causes Esko ArtiosCAD to nest the pieces of the plastic cover (if one is used) as if they were rubber.

**Width** is the width of the rubber when it is cut.

**Offset from side of rule** is the distance from the edge of the rule to the edge of the rubber which allows for pointage.

Once you have finished modifying an existing rubber type or creating a new one, click OK to return to Defaults, and then click File > Save to save the changes.

### Rubber parameter sets

As with single design and manufacturing parameter sets, rubber parameter sets group rubber parameters together for easy manipulation. We recommend copying the RubParamExample parameter set and modifying it to meet your needs, but you can create a new empty parameter set by right-clicking Rubber parameter sets and selecting New > Data on the context menu.

### Bolt holes

After the rubber is made, it needs to be trimmed to allow for bolt holes, but because of the placement of the holes, the trimming may affect how pieces are auto-repeated. Therefore, this dialog box controls whether the trimming is automatic or not by the state of Subtract bolt holes. If bolt holes are not automatically subtracted from the rubber, the Subtract Bolt Holes tool allows for manual trimming after the rubber is created.
The rest of the fields in the dialog box determine how much rubber will be trimmed. If the hole in the dieboard is almost circular and less or equal to the specified size, a circular hole of the specified size will be made in the rubber. If not, the hole in the dieboard will be copied into the rubber.

Laser position holes can be made using lines of linetype Laser position hole or Unknifed diecut. They can be triangular, square, hexagonal, octagonal, must be a reasonable size of less than 16mm, and fit around a circular pin. If a laser position hole is recognized, the hole in the rubber will be offset by the specified offset. If the laser position hole is not recognized, the hole will be copied without an offset.

Click OK when done to return to Defaults.

**Breaker knives**

Breaker knives are also known as stripping rules. When used in Rubbering, they are the stripping rules on the outside of the die. Sometimes they are rubbered to prevent injury to the people handling the dieboard. The options in the Breaker Knives section of the parameter set control how far the rubber extends past the sheet edge. Choose the desired option button and click OK to return to Defaults.
Crease Mask

The options in the Crease Mask section of the parameter set control how the rubber is cut back near creases.
The options in this dialog box are made available or unavailable by the state of **Make Plastic Cover** in the General section of the parameter set. When that option is not selected, the controls in the bottom half of this dialog box are unavailable, and when it is selected, the controls in the top half are unavailable.

For normal areas of the design, the options along the left side of the top quarter of the dialog box are the distance that is cut back from the crease, and the values on the right side are the angles at which the cuts are made. Each end of the cutback is the same as it is symmetrical.

For thin areas of the design, the rubber pieces are smaller so there are fewer cutbacks.

When a plastic cover is used, set the offsets from the plastic cover edge and from the edge of the crease rule as desired.

Click **OK** when done to return to Defaults.

**Cut/Crease**

Esko ArtiosCAD can treat cut/crease rules as if they were cuts, or creases, or a mixture of the two. They need to have different parameters than the standard crease mask so that the angle of the cutbacks can be greater, so these are set in the Cut/Crease section of the parameter set. Also, rubber pieces for cut/crease rule are made thicker (to the thickness you specify in this dialog box) so they do not break at the thin points.

In the **Treat cut/crease** group, **Like cuts**, **Like creases**, and **Use parameters below** all behave as their names imply. **Use parameters below** enables the distance and angle fields for the crease cutback.

Click **OK** when done to return to Defaults.

**General**

The General section of the parameter set contains settings that affect the overall creation of the rubber.
Add blend causes the Rubber Area, Add Rubber, and Subtract Rubber tools to add blends to the rubber of the size specified in the Blend radius: field.

Fixed width, Open out, and Fill set the style of rubber Esko ArtiosCAD makes.

The Open out tack width: sets the width of the connecting piece of rubber left at the end of an open-out.

In the Make splits group, the title control determines if Esko ArtiosCAD limits the length of pieces of rubber or splits them. The default is to split them. This option also controls if ArtiosCAD creates pseudo temporary stripping rules to break up large pieces of rubber for easier manufacture.

- If they are split on bridges, there is better airflow and less suction. If they are split at corners, they nest better and are easier to replace when worn out. Both creates splits at both locations.
- Maximum length: sets the largest size allowed for pieces of rubber placed outside the designs.
- Maximum length inside: sets the largest size allowed for pieces of rubber placed inside the designs.
- Maximum length straight: is a separate length for straight pieces, and should be set to longer than the Maximum length: value as straight pieces nest better.
- Split width: sets the size of automatically-created splits and also sets the default value for the Add Split tool.

Minimum length: sets the value at which small pieces of rubber are merged with longer adjacent pieces.

The Trim gap: field sets the size of the gap when two pieces of differing-types of rubber overlap. It can be as small as 0.001 inch or 0.05mm, but set this to a value larger than 0 so that the Adjust Outline tools can work with each piece individually.

Trim against counter controls whether the rubber is trimmed against the counter (if one is defined) and if so, the gap is set in the Counter trim gap: field. If this option is not selected, the rubber can be trimmed later with the Trim Rubber tool.

Trim lead edge, when selected, makes it so that no rubber goes lower than the lead edge.
Make plastic cover, when selected, causes a plastic cover to be made in its own layer when the Rubber Area tool is used. Click OK when done to return to Defaults.

Layout

If you have the Rubber Design and Layout option, Esko ArtiosCAD can automatically group pieces of rubber on the sheets based on their shapes.

Trim is the distance from the edge of the rubber sheet to the edges of the rubber elements, and Gutter is the distance between each rubber element.

Increasing the value in the Nesting tries: field causes Esko ArtiosCAD to try more nests, but this takes more time.

The five choices in the Grouping group control how the instances of each rubber element are grouped together on the sheets. Manual means that the grouping will be taken from the values set in the List Rubber Elements dialog box. None means that each item will be considered its own group. Low means only straight items will be grouped together. Medium will do an intermediate level of grouping, but High will group together as many instances of the element as possible. These controls are meant to let you find the balance between making pieces easy to find and wasting the least rubber.

Click OK when done to return to Defaults.

Plastic cover

Some diemakers glue a plastic cover on top of the pieces of rubber to keep the board from sticking to the rubber as the press operates. The plastic cover pieces are supported by pieces of rubber on all sides, including along creases. The plastic cover extends slightly past the pieces of rubber underneath. Plastic cover pieces are not generated for slots.
When **Only in waste areas** is selected, the plastic cover is used only to cover waste, not designs. When it is not selected, it covers both waste and designs, but not slots. Also, when it is selected, it is nested as if it were a type of rubber.

**Offset from cut:** and **Offset from crease:** both control how close the plastic cover gets to rule. Remember that it needs to project slightly over the rubber but not touch the rule.

**Subtract bolt holes** and its associated fields work the same way as they do in the Bolt holes section of the parameter set.

**Left allowance:**, **Right allowance:**, **Top allowance:**, **Lead edge:**, and the options in the **Relative to** group all determine how the plastic cover is positioned in reference to the edges of the layout.

Click **OK** when done to return to ** Defaults**.

### Rubber types

The **Rubber types** section of the parameter set is how Esko ArtiosCAD determines which type of rubber to use in a given location and also how to place it.
Rubber is assigned to lines on the basis of line type, knife location, knife side, slots, rule direction, and nicks. Use this dialog box to modify existing selections, create new selections, or delete existing selections.

The line entries in the Rubber Parameters dialog are applied in order, starting with the first (top) one and moving down through the list to find a match; each rule is compared with each selection in turn. The first selection that the rule matches defines the rubber type for that rule. The more critical selections with high shore rubber, such as for slots and nicks, should be first in the list.

To add a selection list entry, do the following:

1. Select an existing selection and click Insert.
2. A new selection is added to the table with the name of the rule from step 1 with (2) appended to its name.
3. Delete the text in the Selection name: field and enter the name for the new selection.
4. Enter the width of the rubber pieces in the Width: field, and the desired offset in the Offset from side of rule: field. Select Use selection and set the type of rubber from the drop-down list box. (The type of rubber must be defined in the Rubber Types catalog before it can be used in the parameter set.)
5. Click Selection. The Rubber Type Selection dialog box opens as shown below.
In this dialog box you select the conditions that a piece of rule must meet in order to be rubbered by the type of rubber selected. The checkboxes in the Line types: group and the Knife Location: group use OR logic, but the two groups together use AND logic: Cut/knife OR Scrap knife OR Balance knife, and so forth, AND of type Leading edge knife OR Exterior waste OR Interior slit, and so forth.

If the type of rubber you are defining is for slots, select Slots and enter the maximum width of the slot in the Width limit: field. If you define more than one type of rubber for slots, the narrowest slot should be highest in the list in the Rubber types dialog box, followed by the wider slots.

The other groups allow you to add more conditions to the rules that will receive this type of rubber; set them as desired. Nicks have the following restrictions: For **Short piece centered on nick**, the Distance past nick: field should be more than the width of the rubber. Also, for both of the first two options, the Minimum length should be less than the remaining length lines which you do not want to have the nick rubber type.

For the Knife Location: group, refer to the picture and table below.
Table: Knife locations for rubbering types

<table>
<thead>
<tr>
<th>Number</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Side</td>
</tr>
<tr>
<td>2</td>
<td>Trailing edge</td>
</tr>
<tr>
<td>3</td>
<td>Breaker knife (stripping rule)</td>
</tr>
<tr>
<td>4</td>
<td>Interior slit</td>
</tr>
<tr>
<td>5</td>
<td>Product side</td>
</tr>
<tr>
<td>6</td>
<td>Waste side</td>
</tr>
<tr>
<td>7</td>
<td>Common knife</td>
</tr>
<tr>
<td>8</td>
<td>Common waste</td>
</tr>
<tr>
<td>9</td>
<td>Interior waste</td>
</tr>
<tr>
<td>10</td>
<td>Leading edge</td>
</tr>
</tbody>
</table>

Once you have finished selecting the criteria for this rule, click OK to return to the Rubber Types dialog box.

1. Use the arrows at the right end of the list to move the selected rubber type selection either up or down in the priority list.
2. Repeat the process as desired to add selection list entries.
3. Click OK when done to return to Defaults.

All rule types except crease and balance knife should select a rubber type (rubber on balance knives is optional because this does not interfere with rubber on other rules). If you want no rubber on a particular rule type, because you are going to put rubber there manually, you should select rubber type None for this rule type, and still define an offset and a width, so Esko ArtiosCAD knows how much space to leave for it.

For example, suppose you want no rubber on cut/crease. Copy the Cuts selection on the Rubber types tab, change the name to Cut/crease, change the rubber to None, and change the selection to select Cut crease lines. You can adjust the width and offset from rule for the rubber you are going to add manually.

If there are rules on the die with no rubber type defined, Esko ArtiosCAD displays a warning dialog when you start the Rubber Area tool. You can either show the rules with no rubber type defined, or you can ignore the error, in which case surrounding rubber along nearby rules will cross the rules for which no rubber type is defined.
The Rebuild Options entry in the Design defaults catalog lets you change behavior when rebuilding a single design or running a standard.

Automatically reinitialize subsequent variables reinitializes variables in subsequent menus which currently match their default values. When this option is selected and you change a variable, any
subsequent variable depending on that variable which matched its default value before the rebuild started will get updated with the new value. For example, with this option selected, if you change L, any variable in any subsequent menu which depends on L gets automatically updated with the new value for L.

When this option is off, dependent variables are not automatically updated and are flagged for review in the Rebuild Conflicts dialog box.

Start rebuild at board menu controls whether you may choose a new board when rebuilding a design instead of starting at the first visible menu.

Show full screen sets the Rebuild, Run Standard, and Reports dialog boxes to the maximum size possible.

The options in the Variable labels... groups control what is shown to the left of the fields when running a standard, rebuilding a design, using a geometry macro, or outputting a report. Choose from Variable name only, Description only, Description and variable name, and Variable name and description. If Description only is the chosen method and the variable description is blank, the variable name is used in parentheses. If there is no documentation plot for the menu, the description is shown with the variable name in parentheses. These options can be changed on the fly in the dialog boxes by clicking the tiny triangle above the variable list and selecting a different display option.

Click OK to make the changes and return to Defaults, or click Cancel to discard the changes.

3D Import options

Use the options in Startup defaults > 3D Import to control how ArtiosCAD imports solids.

Render both sides shows the inside and the outside of the solid.
Preserve origin on adding new solid keeps the 3D workspace origin set so that when you import solids, they stitch together properly. When this option is off, ArtiosCAD adds solids centered behind existing solids.

Interactive preview shows the preview window, allowing you to choose which parts of multi-part solids to import. When this option is off, ArtiosCAD imports all parts of multi-part solids.

The SolidWorks Image Quality slider controls the fineness of curves by controlling the length of chords made from arcs when opening the file. Set the slider left for longer chords (resulting in a more angular appearance for curves) or right for shorter chords (resulting in a smoother appearance for curves).

Click OK to make the changes and return to Defaults, or click Cancel to discard the changes.

3D Rendering method

The 3D module can use two different rendering (display) methods, OpenGL (the default) and Direct3D. Which method will perform better for you depends on the display adapter in your system. The main difference between the two methods is how data is transferred between main system memory and the display adapter. Other differences include:

- Shininess appears to work better when using Direct3D.
- Esko ArtiosCAD using Direct3D takes longer to open a 3D workspace initially but once open draws it faster than when using OpenGL.
- Using OpenGL allows the export of bitmaps with more pixels than with Direct3D
- In wireframe view mode, OpenGL seems to be able to display more lines.
- When Esko ArtiosCAD and the Cortona VRML viewer both use Direct3D and are open at the same time, they compete for resources and unpredictable results may occur. Change the rendering method in one of them for better results.

To change the rendering method in Defaults, open the Startup Defaults catalog and double-click 3D Rendering Options.

Choose the option button for the method to use and click OK.

Use OpenGL for bitmaps enables Esko ArtiosCAD to output bitmaps using OpenGL regardless of the screen rendering method chosen. OpenGL supports the output of larger bitmaps than Direct3D.
Save and exit Defaults as usual. The change is effective immediately. You can also change it at any time by clicking **Options > 3D Rendering Options**.

### Database defaults

Database defaults configure the operation of the Esko ArtiosCAD database. Click **Options > Defaults**, and in the Shared Defaults pane, click **Design Defaults > Database** to open the Database dialog box.

![Database dialog box](image)

**Enable Database**, when checked, lets Esko ArtiosCAD save information to the database. When the checkbox is clear, no new information is saved to the database. Database information that already exists is retained.

**Create Thumbnail in Format** controls if Esko ArtiosCAD creates thumbnail pictures of workspaces when designs are saved. The thumbnails are stored in the database and are used by the database browsers.

The drop-down list box next to the checkbox controls the type of thumbnail created.

**BMP with RLE compression** is the default choice. It creates thumbnails in BMP format using run length encoding compression.

**JPEG format** creates thumbnails in JPG format, which is compressed but lossy (meaning some detail is lost).
**Folded JPEG format** first tries to create a folded view for the thumbnail based on saved angles in a single design. If there are no saved angles, a flat thumbnail is created. In each case JPEG format is used.

**Artios3D format** causes Esko ArtiosCAD to look for an .A3D file by the same name as the workspace being saved in the same directory, and if found, that is used for the thumbnail. If it is not found, Folded JPEG format is used instead.

### File Operations group

The checkboxes in the **File Operations** group control database activity when creating or saving a workspace.

- **Use Resource View on Open and Save As**, when checked, directs Esko ArtiosCAD to use Resource View when opening workspaces or saving them using Save As. Directory View is used when this option is not checked.

- **Show database dialog box when creating a new design**, when checked, instructs Esko ArtiosCAD to prompt you for database information when you create a new design.

- **Show Create As dialog box when creating a new design**, when checked, when checked, instructs Esko ArtiosCAD to prompt you for the filename and resource or directory in which the design is to be saved when creating a new design.

- **Show database dialog box when saving**, when checked, tells Esko ArtiosCAD to prompt you for database information when you save a workspace in a resource.

- **Synchronize disk file when saving in browsers**, when checked, causes Esko ArtiosCAD to modify the disk file for the record you change in a database browser. This is independent of saving the record, which only affects the database.

- **Add new characteristics when synchronizing**, when checked, instructs Esko ArtiosCAD to compare the characteristics in a workspace being opened with those already in the database, and if there are new ones in the file, prompts you to add them to the database.

- **Enable Skip DB button**, when checked, lets you click **Skip DB** when saving a workspace so that you can save a workspace with no corresponding database information.

### Performance options group

The checkboxes in the **Performance Options** group control how Esko ArtiosCAD retrieves certain types of information from the database.
Typically, databases which run over WANs (Wide Area Networks), for example between plants, experience slower network performance compared to those which run over faster LANs (Local Area Networks). By default, Esko ArtiosCAD requests a complete set of database entities each time the Database Information dialog box is activated, which can lead to wait times if there are many records to download from the database server.

To address this, Esko ArtiosCAD can store the Customer, Owner, Designer, Salesperson, Characteristics, and User Fields entities from the Database Information dialog box in memory, as well as the boards, companies, company types, and Resources. This is called caching.

Both Esko ArtiosCAD and CAD-X can use this feature, and each will use more memory if it is enabled. Once the database dialog box entries or other entities are loaded into memory, Esko ArtiosCAD does not re-request them from the database. If changes are made to the entities in the database itself, Esko ArtiosCAD will not see them until the cache is refreshed or a new Esko ArtiosCAD session is started. To refresh the cache, click Database > Refresh cache > Item.

Esko ArtiosCAD can also search the database for specific records which match a pattern thus returning only a few records instead of thousands. This feature is called pattern searching.

Note: These performance options do not apply to customized database information dialog boxes. To use them, you must use the standard database information dialog boxes.

Caching

Toggle the Cache dialog box entries checkbox to enable and disable caching.

The table below lists how the fields in the Database Information dialog box behave when database dialog box entry caching is turned on. The Database Information dialog box behaves the same for Single Design and Manufacturing.

Table: Dialog box behavior when database dialog entity caching is enabled

<table>
<thead>
<tr>
<th>Field</th>
<th>Behavior</th>
</tr>
</thead>
<tbody>
<tr>
<td>Customer and Owner</td>
<td>All Customer and Owner entities are requested when the dialog box is first accessed, and then they are cached.</td>
</tr>
<tr>
<td>Salesperson and Designer</td>
<td>All Salesperson and Designer entities are requested when the dropdown list box arrow for either field is first clicked, and then they are cached. They share the same cache.</td>
</tr>
<tr>
<td>Characteristics button</td>
<td>Characteristics are requested the first time the button is clicked, and then cached. Only applicable to Single Design.</td>
</tr>
<tr>
<td>User Fields button</td>
<td>Same as Characteristics.</td>
</tr>
<tr>
<td>Field</td>
<td>Behavior</td>
</tr>
<tr>
<td>------------------------------</td>
<td>--------------------------------------------------------------------------</td>
</tr>
<tr>
<td>New button in Customer dialog box</td>
<td>Causes a refresh of the Customer cache.</td>
</tr>
</tbody>
</table>

In addition to caching Database Information dialog box entries, when caching is enabled, Esko ArtiosCAD also caches boards, companies, company types, and Resources.

Once the database dialog box entries or other entities are loaded into memory, Esko ArtiosCAD does not re-request them from the database. If changes are made to the entities in the database itself, Esko ArtiosCAD will not see them until the cache is refreshed or a new Esko ArtiosCAD session is started. To refresh the cache, click **Database > Refresh cache > Item.**

**Pattern searching**

Another way to ensure robust database performance over a slow Wide Area Network for the **Customer**, **Owner**, **Designer**, and **Salesperson** fields is to enable pattern searching by checking the **Use pattern searching for customers and owners** and **Use pattern searching for designers and salespeople** checkboxes. To maintain compatibility with previous versions, these checkboxes are both cleared by default.

When pattern searching is enabled, the Oneup Design Information and Manufacturing Database Information dialog boxes change. The drop-down list boxes for the **Customer**, **Owner**, **Designer**, and **Salesperson** fields disappear and the fields become disabled. In addition, there are ... buttons leading to dialog boxes which let you search for records and then display details about the selected records, or display details if there is already a selected record. Shown below is the Customer dialog box; all four dialog boxes act similarly, but only the Customer dialog box has the **New** button.
Pattern searching works by entering a few letters of a word, or a whole word, or a complete phrase, then choosing a method of searching, and then clicking Find. The results of the search populate the drop-down list box.

Mixing database performance modes

Database dialog box entry caching and pattern searching can be used simultaneously, but not for all four types of records. Only a pair of records (Customer and owners or designers and salespeople) for which pattern searching is turned off can be cached.

Database Information Dialog Customization groups

The options in the Single Design and Layout Database Information Dialog Customization groups allow you to choose what happens when entering database information for a single design or layout.
Use standard dialog uses the dialog box that is included with Esko ArtiosCAD.

Use custom dialog definition file enables the Customize button, which in turn leads to the Control Layout Manager described in the next section. Using this feature requires the Information Enhancement option.

Use custom external application enables a field to specify the path to a custom program (such as an ERM program) that controls information storage instead of the Esko ArtiosCAD database. This custom application is not provided or supported by Esko unless it was provided by the Professional Services department. Using this feature requires the Information Enhancement option.

Designing a custom Database Information dialog box

When you choose Use custom dialog definition file and click Customize, the Control Layout Manager dialog box opens showing the current dialog box.

On the left is the dialog box itself showing the groups, labels, fields, and buttons. On the right are the size controls, the control catalogs, the Properties fields for the selected item, and the OK, Test, Cancel, Open, and Save As buttons.

There is no undo functionality in the Control Layout Manager, but you can click Cancel to discard all changes and then start again.

To change the size of the dialog box, adjust the values in the Dialog Size: fields, or select the edge of the dialog box within the definition window and drag it to the desired position. To make a dialog box larger than the current one, resize the Control Layout Manager dialog box itself using standard Windows methods, and then increase the size of the dialog box window.

To add a control, select it from the catalog and drag it to the desired position. You may also double-click it; it will appear at the top left of the dialog box, and you may then select it and drag it to the desired position.

To remove a control, select it and press Delete on the keyboard.
To move a control, select it and drag it to the desired position, or select it and use the **Position**: fields in the **Properties** group to adjust its position. You can also select it and use the arrow keys to move it in one-pixel increments.

To resize a control, select it, and use the handle points to change its size as desired, or select it and use the **Size**: fields in the **Properties** group to adjust its size.

To change text in a control, select it, and then change the contents of the **Text**: field in the **Properties** group.

When editing a field in the **Properties** group, or a **Dialog Size**: field, click in another field or select a different object to activate the changes made to the first object. Do not press enter as that is interpreted as clicking **OK** and will thus close the dialog box.

Click **OK** to save the changes to the dialog box and close the Control Layout Manager. The single design information is saved as **DBDlgDesignDef.xml**, and the manufacturing data is saved as **DBDlgMFGDef.xml**, both in **ServerLib** or **ClientLib** depending on which set of defaults was changed.

Click **Cancel** to discard the changes to the dialog box and close the Control Layout Manager.

Click **Test** to open a sample dialog box that previews the final result.

Click **Save As** to save the dialog box layout to a file.

Click **Open** to open a previously-saved dialog box layout file.

Shown below is a modified **Database** group in the Control Layout Manager and then in the test dialog box.
The tab order of the resultant dialog box follows the order in which controls were added. To change the tab order after the dialog box is completed, make a backup copy, and then edit DBDlgDesignDef.xml or DBDlgMFGDef.xml as appropriate using a text editor such as Wordpad. Change the order of the <FORM_CTRL>entities</FORM_CTRL> blocks as desired and save the file.

**Note:** Database performance options do not apply to customized database information dialog boxes. To use them, you must use the standard database information dialog boxes.

### Using a custom external application

**Use a custom external application** allows a separate standalone application to display and gather database information. Specify the absolute path name of the application in the associated field, or use the Browse button (…) to locate it. This custom application is invoked whenever the Database Information dialog box would otherwise be shown.

The exact sequence is:

1. Esko ArtiosCAD requests database information (by clicking Database > Database Information or during a Save or Open).
2. The current database information is packaged into a randomly-named XML file in the directory defined as the TEMP directory in Options > Defaults > Startup Defaults.
3. The external application is invoked with the input XML file as the first parameter.
4. Esko ArtiosCAD waits until the external application is done.
5. Esko ArtiosCAD reads the XML file specified as the second parameter on the external application’s command line and uses it to populate the database.

Shown below is the dialog box created by a sample application.

![DBInfoApp](image)

The exact command executed is:

```bash
Application.exe -iInputXMLDBinfo.xml -oOutputXMLDBinfo.xml
```

where `Application.exe` is the name of the custom application, `InputXMLDBinfo.xml` is the full path name of the temporary input XML file that Esko ArtiosCAD creates, and `OutputXMLDBinfo.xml` is the full path name to the output XML file created by the custom application. The names of the parameters are generated randomly by Esko ArtiosCAD; the custom application should read them from the command line.
Esko ArtiosCAD checks for a return code from the custom application of 1 for OK and 2 for Cancel.

A sample database definition XML file is shown below.

```xml
<ARTIOSDBDOC>
  <UNITS>
    IMPERIAL</UNITS>
  <DESIGN>
    <NAME>
      SAMPLE1.ARD</NAME>
    <REV>
      0</REV>
    <REVNUM>
      0</REVNUM>
    <PATH>
      C:\Artios\Designs\</PATH>
    <REVDATE>
      2004-04-14</REVDATE>
    <SAVEDATE>
      2004-04-14</SAVEDATE>
    <DESC>
      This is a sample long description</DESC>
    <SHORTDESC1>
      Short 1</SHORTDESC1>
    <SHORTDESC2>
      Short 2</SHORTDESC2>
    <SHORTDESC3>
      Short 3</SHORTDESC3>
    <AUTH>
      Authorization</AUTH>
    <REVDESC>
      Revision Description</REVDESC>
    <LENGTH>
      8.000000</LENGTH>
    <WIDTH>
      7.000000</WIDTH>
    <DEPTH>
      6.000000</DEPTH>
    <GRAINDIR>
      2</GRAINDIR>
    <CAL>
      0.016000</CAL>
    <BLANKLEN>
      780.256226</BLANKLEN>
    <BLANKHEIGHT>
      558.799927</BLANKHEIGHT>
    <AREA>
      267241.718750</AREA>
    <RULELEN>
      5269.554688</RULELEN>
    <RESOURCE>
      <ID>
        1</ID>
      <NAME>
        ArtiosCAD</NAME>
      <RELPATH>
        Artios\Designs</RELPATH>
      <PATH>
        C:\Artios\Designs</PATH>
      <MASTER>
```
ArtiosCAD Enterprise

1</MASTER>
<ACTIVE>
1</ACTIVE>
<INHERITAUTONUM>
1</INHERITAUTONUM>
<SERVER>
<ID>
1</ID>
<NAME>
ArtiosCAD</NAME>
<Path>
C:\</Path>
</SERVER>
</RESOURCE>

<BOARD>
<ID>
39</ID>
<CODE>
I-SBS-16</CODE>
<DESC>
I-SBS-16</DESC>
<FLUTE/>
<TestCode/>
<Cal>0.016000</Cal>
<TestVal>0.000000</TestVal>
(IL>0.008000</IL>
<OG>0.008000</OG>
<CRRV>0.031250</CRRV>
<BasisWeight>64.000000</BasisWeight>
<BasisCost>72.000000</BasisCost>
<Workspace>0</Workspace>
</Board>
<CUSTOMER>
<WORKSPACE>
0</WORKSPACE>
</CUSTOMER>
<OWNER>
<WORKSPACE>
0</WORKSPACE>
</OWNER>
<SALESPERSON>
<WORKSPACE>
0</WORKSPACE>
</SALESPERSON>

1</ID>
<PUBID>
-1</PUBID>
</CUSTOMER>
<OWNER>
<WORKSPACE>
0</WORKSPACE>
</OWNER>
<SALESPERSON>
<WORKSPACE>
0</WORKSPACE>
</SALESPERSON>

1</ID>
<PUBID>
-1</PUBID>
</OWNER>

1</ID>
<PUBID>
For more information about using a custom application, contact Professional Services.

**Default selections group**

The drop-down list boxes in the **Default Selections** group set the owner, salesperson, and designer for newly-created workspaces. These fields get their lists of possible values from the Company and Person browsers in DataCenter Admin. Set the values as desired.
Click **OK** to save the changes made to Database defaults and return to the main Defaults dialog box. Click **Apply** to save the changes but remain in the Database defaults dialog box. To discard all changes and return to the main Defaults dialog box, click **Cancel**.

**Company selection display group**

The option buttons in the Company Selection Display group control how company name and identification information is shown in Esko ArtiosCAD. **Name** is the default choice.

- Name
- Number
- Name, Number
- Number, Name

**Name** and **Number** are fields you set for a company in the Company browser in DataCenter Admin. Note that **Number** can be any number you set; it is not calculated. Choose the display format as desired.

Any changes made in this group will not appear in the Default Selections group in this dialog box until you click **OK** and then open the Database defaults again.

Shown below is an example in the Oneup Database Information dialog box when **Number, Name** is the chosen option in Company Selection Display.

**Spell checking**

If you have Microsoft Word installed on your system, Esko ArtiosCAD can use its spell checking functionality in Single Design and Manufacturing in most places where you enter text using a field in a dialog box. This feature is turned on by default. You may turn it off in the Paragraph Text tool and in text fields by using the Spelling dialog box in the Startup Defaults catalog.
Toggle the Enable spell checking checkbox as desired, and click OK to save the change and return to the main Defaults dialog box.

**Note:** This setting does not affect document-wide spell check using Tools > Check Spelling. It is available regardless of the state of this checkbox.

## Nudge Options and Defaults

Click *Options > Nudge* to set the active nudge modes per session and the distances they use. This command is available when a single design or a manufacturing file is open. Essentially the same dialog box is used in Defaults in *Startup defaults > Nudge options*.

Shown below is the **Nudge options** dialog box.
Show nudge options in select tool controls the availability of the nudge modes. When it is selected, they are active. When it is cleared, nudge moves by a pixel.

Set the three mode selectors as desired. They determine which nudge modes are active when a tool which supports nudge is started.

The values in the Fixed values group control the distance used by the Nudge by a Fixed Value mode. Set them as desired.

**Note:**
If you change the nudge options or Defaults while a Select tool is active, the tool will be restarted to update it and the current selection will be lost.

---

Hatch Defaults

ArtiosCAD stores Defaults for the Hatch tool in three locations: the Hatch Catalog, Property defaults > Hatch, and Startup defaults > Hatch tool options.

**Hatch Catalog defaults**

The Hatch Catalog in Defaults is where you add, remove, and change entries in the Hatch Catalog. ArtiosCAD comes with sample entries in the Misc folder.

To add a folder to the catalog, right-click the folder which will contain the new folder, then click New > Folder. To add a new entry to a folder, right-click the folder and then click New > Data.

To remove a folder or an entry, right-click it and click Delete on the context menu. ArtiosCAD will ask you to confirm the deletion.

To change an entry, double-click it and change the properties as desired. The Hatch Properties dialog box is the same as it is in drawing mode.
Hatch Property defaults

In Property defaults > Hatch, you set the Defaults for the <custom> Hatch Catalog entry as shown below.
The Hatch Properties are the same as in drawing mode; set them as desired and click OK to return to Defaults.

**Hatch Startup defaults**

In *Startup defaults > Hatch tool options*, you set the options for how the Hatch tool will initially work.

In the **Tool select option** and **Offsets** groups, set the options as desired.

- **Prompt before changing layers**, when checked, makes you confirm changing layers to the layer specified in a Hatch Catalog entry’s **Placement Options** dialog box. **Restore layer on tool exit**, when checked, returns you to the layer you were in if the layer changed due to using a Hatch Catalog entry with a layer assignment.

When you are done setting the options as desired, click OK to return to Defaults.

**Adding LASERPOINT IQ standards to Style Catalogs**

**Preparation**

Before adding a LASERPOINT IQ standard to Esko ArtiosCAD, it is necessary to locate the LASERPOINT IQ files needed to run the standard. The name can be determined using Menu Management. Locate the standard to add and select the **Edit** option. This will show a standard name which looks like one of the following entries:

- **EXE STDMENU WITH ‘XYZ’** This is an entry for a LASERPOINT IQ 2.0 format standard whose name is XYZ. You will need the files XYZGEOM and XYZW.
• XYZ. This is an entry for a pre-LASERPOINT IQ 2.0 standard whose name is XYZ. You will need the file XYZ and there may also be a file called XYZW containing documentation plots.

The files for the standard you are migrating are probably in your \LP\USER_LIBI directory. Copy them to the \Artios\ArtiosCADn.nnll\ServerLib directory (where n.nn is the version number of Esko ArtiosCAD and l is the language.

Adding the standard to a Style Catalog

Add the LASERPOINT IQ standard to the Esko ArtiosCAD style catalog using Defaults in a similar way to normal Esko ArtiosCAD standards. Start Esko ArtiosCAD, and then open the style catalog as follows:

1. Click Options and Defaults to open the Defaults dialog box.
2. Use the right mouse button to click Style Catalog in the left area of the dialog box and then click New and Data.
3. Type the descriptive name for the standard as you want it displayed in Esko ArtiosCAD.
4. Double-click the icon beside the standard name to open the dialog box for specifying the standard.

5. For LASERPOINT IQ 2-4 standards:
   Type the name of the standard file without the W or GEOM suffix. In the example above with files XYZW and XYZGEOM, type XYZ.
   For pre-LASERPOINT IQ2 standards:
   Type the name of the standard.
   If you don’t know the name of the standard, click the … button at the end of the Filename field. This will open the standard Open dialog box which you can use to navigate to the proper directory and open the file.
6. Click OK to close the dialog and save changes. Esko ArtiosCAD will check the file and set the standard type automatically.

You are now ready to run the standard in Esko ArtiosCAD.
Troubleshooting newly-added INTERACT/LASERPOINT/ LASERPOINT IQ standards

Missing files

LASERPOINT IQ standards may be created based on other standards or attachments. You must copy the files for the standards to run in the Esko ArtiosCAD environment. Any missing files will be shown in error dialog boxes when a standard is run. Remember to copy all the files – for example, if ATTACHGEOM is missing, you will also need the associated ATTACHW file.

Changing the framework

Esko ArtiosCAD sets up a framework to execute the various types of INTERACT/LP/LASERPOINT IQ 1 standards. The framework settings are available by clicking the Advanced button.

![Selecting framework settings](image)

LASERPOINT IQ 2–4 standards may need to be set up using the INTERACT type if they used elements from before LASERPOINT IQ 2.0. This is set by changing to the standard type INTERACT/LP/IQ1, clicking Advanced, and selecting the framework Non-standard IQ2-4.

INTERACT/LP/IQ1 standards that fail to run properly can be tried with alternative frameworks by clicking Advanced and selecting one of:

- LP/IQ1
- INTERACT design standard
- Designer Edition
Adding Designer WorkBench designs to Style Catalogs

Preparation

Before you can add Designer WorkBench designs to a Style Catalog, Esko ArtiosCAD needs to be told the location of the DWB resources. You will need to know the directories used for the DWB resources and then add these to the Esko ArtiosCAD environment. The steps to do this are:

1. Start DataCenter Admin and double-click Server.
2. Check that there is a server set up for the drive letter where your DWB designs are located. Add a new server as necessary. For example, if your DWB designs are on the D: drive, click the Insert icon, give the new server a name and select D: as the server path. Click Insert to add the server.
4. Do the following for each DWB resource:
   - Click the Insert icon, give the resource a name, pick the server and the path for the resource.
   - Click Insert to add the resource.
5. When complete, save the changes and close DataCenter Admin.

Adding a Designer WorkBench standard to a Style Catalog

Any DWB design can be added to the Esko ArtiosCAD Style Catalog by using the following procedure:

1. Start Esko ArtiosCAD.
2. Click Options and Defaults to open the Defaults dialog box.
3. Use the right mouse button to click Style Catalog in Shared defaults and then click New and Data.
4. Type the descriptive name for the standard as you want it displayed in Esko ArtiosCAD.
5. Double-click the icon beside the standard name to open the dialog for specifying the standard.
6. Click the … button beside the data entry area for the design name. This will show the Open dialog.
7. Change to resource view, select the resource, and check you are showing designs of type either All files or Designer Workbench Designs.
8. Select the design and click Open.
9. Close the Defaults dialog box and save changes.

You are now ready to run the standard in Esko ArtiosCAD.

Note: The standard can be moved to sub-folders in your style catalog, if desired, to give a more structured organization.
Default new design template

Each parameter set can now have a default single design template defined. This means that when you create a new single design by clicking **New Design** on the File menu, instead of getting a blank design, the standard you have associated with the parameter set will run and the new workspace will inherit the results.

Double-clicking **New Design Template** opens the New Design Template dialog box for the current parameter set. Select the **Use Design Template** checkbox and choose a standard from one of the catalogs shown.
Only standards of type ArtiosCAD can be used as new design templates, so only those are listed in the dialog box. INTERACT, LASERPOINT, and LASERPOINT IQ standards may not be used as design templates.

The design template does not have to draw any geometry - you can simply set default variables, menus, and layers in it, and they will be copied into each new workspace made by the template.

Settings in the parameter set for the board code, bridging formula, sideness, and property defaults take precedence over those settings in the template.

The design template is used only when explicitly creating a new single design. It is not used when creating a new embedded design in Manufacturing or when moving lines to a new design with the Move To Layer tool.

### Default layers in Manufacturing

You can specify layers that will be created in new manufacturing files at creation and in existing manufacturing files when they are opened.

To do this, do the following:

1. Start Esko ArtiosCAD and click Options > Defaults > Startup defaults > Default MFG User Layers. The empty Default MFG User Layers dialog box will open.
2. Click Add and enter the name of the new default layer in the New Default Manufacturing Layer dialog box. Click OK when done.

3. The new layer will appear in the dialog box already selected.
4. Continue adding layers as desired, clicking OK when done.
5. Save and exit Defaults.

Layers which are not selected are not created automatically; however, they are available for picking in the drop-down list box in the Create Layer dialog box in Manufacturing.

All of the default layers, whether created in the manufacturing file or not, will also be available for selection on the View tab of any Output when the User Defined layer set selection is active.
In Esko ArtiosCAD you can change the background of toolbars and create customized toolbars to suit your needs. As with other Defaults, you can define toolbars either on a per-machine basis (User defaults) or on a server basis (Shared defaults). Customized toolbars can contain buttons for Esko ArtiosCAD tools, geometry macros, Outputs, and menu commands.

Customized toolbars

Customized toolbars are managed by the Customized Toolbars catalog in Defaults. Esko ArtiosCAD displays up to 30 customized toolbars in Single Design and 30 customized toolbars in Manufacturing, with each toolbar holding a maximum of 15 items. You can define more than 30 customized toolbars...
per module, but Esko ArtiosCAD will display a random assortment in the Customized Toolbars Master Control.

3D does not offer customized toolbars.

Each item on a customized toolbar must have a unique name. If there are items with the same name (for instance, one in Shared Defaults and one in User Defaults), Esko ArtiosCAD executes the first one it finds when you click the button on the toolbar.

Shown below is the Customized Toolbars catalog in Server defaults. The individual folders are empty until you create customized toolbars within them.

Icons for customized toolbars

By default, customized toolbars use the icon of a person’s head. You may design your own icons for customized toolbars using the 16 primary system colors. They must be 15 pixels high by 16 pixels wide in BMP format. Icons for Shared defaults must be saved in $ServerLib$, while icons for User defaults must be saved in $ClientLib$.

Each geometry macro and Output can have a customized icon as well. Follow the same rules for them as those listed above for a customized toolbar icon. Specify the icon in the Icon for custom toolbar group in the Properties dialog box for the geometry macro, and in the Icon for custom toolbar group on the Output Type tab of the Output’s Properties dialog box. If you changed the definition of an Output to specify a custom icon, save the changes, and then exit and re-enter Defaults for the changes to be shown.

You should create any desired icons before entering Defaults to create the customized toolbar.
Making a customized toolbar

Add custom geometry macros and Outputs to catalogs and save Defaults before making a customized toolbar. To make a shared customized toolbar, do the following:

1. Start Esko ArtiosCAD and click **Options > Defaults**.
2. In the Shared defaults pane, click the plus sign (+) next to the Customized Toolbars catalog to open it.
3. Right-click the appropriate catalog, click **New**, and then click **Data** on the context menu.

4. Replace **New Data 1** with the name of the new custom toolbar. Press **Enter** when you have entered the name and then double-click the new entry to access its Properties dialog box.

5. The Properties dialog box for the custom toolbar will appear.
6. In the top half of the dialog box, choose the tab containing the item that will be replicated on the customized toolbar. Items from each category can be added to the same customized toolbar. On the Toolbars tab, select the toolbar to copy from by using the ArtiosCAD Toolbars drop-down list box. On the Geometry Macros and Outputs tabs, expand the catalogs to access the individual geometry macros and Outputs. On the Commands tab, simply select the desired command.

7. Drag the desired tool, geometry macro, Output, or command from the top pane to the Tools in the Custom Toolbar group. Repeat as desired, putting up to 15 items on the customized toolbar. Shown below is act of dragging a command to the Tools in the Custom Toolbar group.

8. To change the icon used for this customized toolbar in the Customized Toolbars Master Control, enter the name of the icon file you created previously and placed in ServerLib in the Toolbar Icon group.

To create a customized toolbar for users of the current machine only, create a new Customized Toolbars catalog by right-clicking in the User defaults pane, clicking New, selecting Customized toolbars, and following the procedure above for Shared defaults.

To delete an item from a customized toolbar, drag it to the Recycle Bin icon at the bottom right of the Tools in the Custom Toolbar group.

To delete a custom toolbar, delete its entry from the catalog.

When adding an Output or a geometry macro to a toolbar, the number Esko ArtiosCAD assigns the button will remain static and cannot be changed by deleting other buttons.

**Colored toolbars**

Each toolbar in Esko ArtiosCAD can have a different background color to make it easier to identify. Color definitions in User defaults override those set in Shared defaults.
Colors on standard toolbars

To change the background color of a standard Esko ArtiosCAD toolbar, do the following:

1. Start Esko ArtiosCAD and click **Options > Defaults**.
2. Open the appropriate Customized toolbars catalog (either the one in Shared defaults or one in User Defaults) by clicking the plus sign (+) next to it.
3. Double-click **ArtiosCAD Toolbars colors**. The ArtiosCAD Toolbars colors dialog box will appear.

4. Click the button of the toolbar to change and click **Toolbar color** to open the Toolbar color dialog box.
5. Click the color to set as the toolbar background and click OK. To reset the toolbar background color to the default, click Set to System Color and then click OK.

6. The Toolbar color button in the ArtiosCAD Toolbars colors dialog box will change to the color you selected.

7. Repeat steps 4 and 5 as desired. When done, click OK to return to Defaults, or click Cancel to discard the changes.

Colors on customized toolbars

To change the background color of a customized toolbar, do the following:

1. Start Esko ArtiosCAD and click Options > Defaults.
2. Open the Customized toolbars catalog (either the one in Shared defaults or one in User Defaults) by clicking the plus sign (+) next to it.
3. Open the Designer customized toolbars catalog or the Manufacturing customized toolbars catalog by clicking the plus sign (+) next to it.
4. Double-click the customized toolbar to change to open its Properties dialog box.
5. Click Toolbar color.
6. The Toolbar color dialog box will appear. Click the color to set as the toolbar background and click OK.

To reset the toolbar background color to the default, click Set to System Color and then click OK.

7. The Toolbar color button in the customized toolbar's Properties dialog box will change to the color you selected.

8. Repeat as desired for other customized toolbars. When done, click OK to return to Defaults, or click Cancel to discard the changes.

**Shortcuts**

**Shortcuts** are keyboard sequences that activate Esko ArtiosCAD commands, such as pressing **CTRL-N** for a new single design workspace. Shortcuts are organized by where they are used: 3D, Design, and Manufacturing. Shortcuts can also be assigned to Outputs and geometry macros.

Every menu command can have a shortcut assigned to it.

Shown below are the categories for menu commands in the Shortcuts Catalog in Shared Defaults.

To create menu command shortcuts in User Defaults, right-click in the User Defaults pane, click New, and then click Shortcuts. A Shortcuts Catalog is created with the three entries for the types of
menu command shortcuts. Save the User Defaults after creating the Shortcuts Catalog but before adding to it.

Assigning a shortcut to a menu command

To assign a shortcut to a menu command, do the following:

1. Start Esko ArtiosCAD and click Options > Defaults.
2. Scroll down the list of Shared Defaults, and open the Shortcuts Catalog by clicking the plus sign (+) next to Shortcuts.
3. Double-click the category for the desired new shortcut. In this example, the Direction menu command on the View Options sub-menu of the View menu will be assigned a shortcut, so double-click the Design Shortcuts category. The Design Shortcuts dialog box opens as shown below.

![Design Shortcuts dialog box](image)

4. Scroll down to the View menu, and double-click Direction under View Options.
5. An empty Shortcut dialog box appears. Press the key combination to assign to the Direction command. For this example, press ALT-D. The key sequence and the command to which it will be assigned appear as you press it. The command is followed by the type of shortcut (system or user) and the area of Esko ArtiosCAD in which it is active.

6. Click OK to return to the Design Shortcuts dialog box.

7. Note that the newly-assigned command now appears in the Key column of the dialog box.

8. Click OK in the Design Shortcuts dialog box.
9. Click File > Save to save the Shared Defaults. The new shortcut is now ready for use in Builder and Designer.

Assigning a shortcut to an Output

To assign a shortcut to an Output, open the Properties dialog box for that Output. Enter the shortcut in the Shortcut: group on the Output Type tab as shown below.

Assigning a shortcut to a geometry macro

To assign a shortcut to a geometry macro, open the Properties dialog box for that geometry macro and enter the shortcut in the Shortcut: group as shown below.
Shortcut uniqueness and priority

When a shortcut in Shared Defaults is assigned a key sequence, any other shortcuts having the same key sequence are shown. Only shortcuts in Shared Defaults and those in the current machine’s User Defaults are compared. Shortcuts in other machines’ User Defaults are not compared with those in the current machine’s User Defaults.
The shortcut at the top of the list has the highest priority and will prevent other shortcuts assigned the same key sequence in the same module from working. However, if the other key sequences are in other modules, there will be no conflict when the shortcut is used.

Shortcuts in User Defaults take priority over those in Shared Defaults. Within User Defaults, menu items take priority over geometry macros, and geometry macros take priority over Outputs. If a shortcut is assigned to the same command in all three subcatalogs, the one in Design is listed first, then the one in Manufacturing, and finally the one in 3D.

Notes and warnings for Shortcuts

Only the first 300 shortcuts defined for geometry macros are active.

Counter layouts use the same shortcuts as Manufacturing.

When a geometry macro has a shortcut, and this geometry macro is present on a customized toolbar, and the geometry macro opens a dialog box, the button on the customized toolbar is not depressed when the geometry macro is activated by the shortcut.

There is only one set of user shortcuts per machine.

If you change shortcuts and exit without saving the changes, the uniqueness tests for successive changes will not be accurate.

enter can not be used as any part of a shortcut.

Some shortcuts are permanent as shown in the table below.

Table: Permanent Shortcuts

<table>
<thead>
<tr>
<th>Command</th>
<th>Shortcut</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run Standard</td>
<td>CTRL-2</td>
</tr>
<tr>
<td>New 3D</td>
<td>CTRL-3</td>
</tr>
<tr>
<td>New Design</td>
<td>CTRL-N</td>
</tr>
<tr>
<td>Command</td>
<td>Shortcut</td>
</tr>
<tr>
<td>---------------------------------</td>
<td>------------</td>
</tr>
<tr>
<td>File Open</td>
<td>CTRL-O</td>
</tr>
<tr>
<td>Help</td>
<td>F1</td>
</tr>
<tr>
<td>Next Pane</td>
<td>F6</td>
</tr>
<tr>
<td>Previous Pane</td>
<td>SHIFT-F6</td>
</tr>
<tr>
<td>Properties</td>
<td>ALT-Enter</td>
</tr>
<tr>
<td>Close CloseUp Window</td>
<td>SHIFT-Space</td>
</tr>
<tr>
<td>Restart current tool</td>
<td>ESC</td>
</tr>
</tbody>
</table>

**Tack Bridging**

Tack bridging formulae are stored in the Tack Bridging Catalog.

![Tack Bridging Diagram]

Shown below is a sample exterior metric formula.
Set the distance between tack bridges in the Tack distance: field and set the width of each tack bridge in the Tack width: field.

The One-sided and Two-sided option buttons control if the tack bridges are cut only toward the interior of the carton or toward the interior of the carton and the exterior of the carton.

The Simple, 90 degree, 120 degree, 135 degree, and Arc tack bridge option buttons all set the style of tack bridge as shown in the pictures in the dialog box.

Click OK to save the changes to the Tack Bridging formula and return to Defaults, or click Cancel to return to Defaults, discarding the changes.

Bridging Formulae

Esko ArtiosCAD’s default bridging formulae are stored in the Artios subfolder of the Bridging Catalog. There are corrugated and folding carton bridging formulae in Imperial and metric units. Shown below is the Corrugated - Metric bridging formula set for evenly spaced bridges.
The drop-down list box controls all the other settings in this dialog box. The No bridges selection removes all other settings from the dialog box. Evenly spaced and Inset distance from line end are explained in the following sections.

**Evenly spaced bridging formula**

An evenly spaced bridging formula applies bridges based on the length of the line or arc. The size of the bridges is determined by the length of the line.

There are three groups of parameters in an evenly spaced bridging formula - Bridges on lines, Bridges on arcs, and Width of Bridges.

**Bridges on lines and Bridges on arcs groups**

The Bridges on lines and Bridges on arcs groups work similarly. A list of limits determines the number of bridges applied to lines or arcs. Shown below are the two groups in the Corrugated - Metric formula.

If the length of the line or arc is less than or equal to the value in the field, the integer to the left of the field is the number of bridges applied to the line or arc. If the length of the line or arc is greater than the value in the field, the limit is skipped and the next limit is applied. Click Add limit to add a limit; click Delete limit to delete a limit.
After all the limits are met, the lines and arcs are bridged according to the value shown after then bridge every X.XX where X.XX is the difference between the last two limits. There must be at least one limit per group.

**Width of Bridges group**

The **Width of Bridges** group works slightly differently than the others in that there are two values to set per limit - the width of the bridge and length of the bridge for the limit.

```
<table>
<thead>
<tr>
<th>Width of Bridges</th>
<th>1.25</th>
<th>if length &lt;=</th>
<th>3.50</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2.00</td>
<td>if length &lt;=</td>
<td>15.00</td>
</tr>
<tr>
<td></td>
<td>5.00</td>
<td>if length &lt;=</td>
<td>25.00</td>
</tr>
<tr>
<td></td>
<td>10.00</td>
<td>if longer</td>
<td></td>
</tr>
</tbody>
</table>
```

The length of the bridges will be the value on the left if the line or arc length is less than or equal to the length on the right.

Click **Add limit** to add a limit; click **Delete limit** to delete a limit. There must always be at least two limits - one **less than or equal** limit, and one if **longer** limit.

**Inset distance from line end bridging formula**

An **Inset distance from line end** bridging formula insets a bridge at each end of a line and then evenly spaces the bridges between the two that were inset.

```
<table>
<thead>
<tr>
<th>Inset properties</th>
<th>Minimum rule length to bridge:</th>
<th>1/2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inset distance:</td>
<td>3/4</td>
</tr>
<tr>
<td></td>
<td>Minimum distance between bridges:</td>
<td>11/2</td>
</tr>
<tr>
<td></td>
<td>Maximum distance between bridges:</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>Min rule length for full inset distance:</td>
<td>311/2</td>
</tr>
</tbody>
</table>
```

Rules shorter than the **Minimum rule length to bridge** will not be bridged. Rules whose length is between the **Minimum rule length to bridge** value and the **Min rule length for full inset distance** will have only one bridge. Rules longer than the value in **Min rule length for full inset distance** will have the two inset bridges, plus more bridges if the two inset bridges are further apart than the value in **Maximum distance between bridges**. The **Inset distance** is the distance from the ends of the
line to the centers of the inset bridges. Bridge centers can be no closer than the Minimum distance between bridges, and cannot be further apart than the Maximum distance between bridges.

The Advanced tab adds two parameters to the group:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inset factor limit</td>
<td>2</td>
</tr>
<tr>
<td>Inset factor</td>
<td>0.285</td>
</tr>
</tbody>
</table>

These two fields are used when migrating bridging formulae from The Designer WorkBench into Esko ArtiosCAD. Set the Inset factor to 0.285. The One Bridge Minimum equals the Minimum distance to bridge. The Inset Factor Limit is half the Maximum distance between bridges.

Inset Bridge Formula

- If line length < One Bridge Minimum, no bridges
- Calculate BiSep = LineLength - 2*InsetDistance
- If BiSep < Minimum Distance to Bridge, 1 bridge centered
- If BiSep < Inset Factor Limit, 2 bridges at InsetFactor * length
- If BiSep > Maximum, add additional bridges, evenly spaced
- Else two bridges at Inset distance

The options in the Width of Bridges group work the same for Inset formulae as they do for Evenly spaced formulae.

Working with tuning catalog entries in Defaults

You will most likely need to create your own tuning catalog entries for PDF or DWG/DXF files as each supplier may produce them differently. The procedures for working with DWG/DXF tuning files are the same as they are for PDF files; substitute the type of file you are working with as appropriate. Follow the instructions below to work with tuning catalog entries.

Creating a PDF vector import tuning catalog entry

ArtiosCAD comes with two example tuning files for importing PDF files as lines. However, since people can create PDF files in many different ways, you may need many different tuning catalog entries to accommodate them.

To create a new PDF vector import tuning file, do the following:
1. Log in as Administrator or as a user with administrative privileges.
2. Start ArtiosCAD.
3. Click Options > Defaults.
4. In the Shared defaults pane, scroll down the list to Import Tuning Table. Optionally, expand it by clicking the plus sign (+) next to it.
5. Right-click Import Tuning Table (or a folder underneath it if you expanded the catalog) and click New > Data on the context menu.
6. In the Select import type list box, select PDF and click OK.
7. Enter a name for the new tuning file and press Enter.

A tuning catalog entry has been created with specified name.

**Opening a PDF vector import tuning catalog entry**

To open a PDF vector import tuning file for editing, do the following:

1. Start ArtiosCAD.
2. Click Options > Defaults.
3. In the Shared defaults pane, scroll down the list to Import Tuning Table.
4. Expand the catalog and double-click the name of the tuning entry to open for editing.

A dialog box appears showing the tuning for this catalog entry.
Editing a PDF vector import tuning catalog entry using a template

This procedure requires knowledge of the existence and design of the layer structure of your incoming PDF files. PDF files can have layers but also can be flattened (when all elements are in one layer). Ask the senders of PDF files about the layer structure they use, or examine them in a program that can edit PDFs, such as Adobe Illustrator. If the files are coming from Adobe Illustrator, and you want the layer information to be preserved, the Create Acrobat Layers from Top-Level Layers option must be selected when saving the file in PDF format, otherwise the file will be flattened and no layer information will be preserved.

To edit a PDF vector import tuning catalog entry, start ArtiosCAD and follow the instructions in Opening a PDF vector import tuning catalog entry to open it.

Once you have the tuning catalog entry open, set the options according to the procedure described below.

1. In the Import As group, select either Oneup, Layout, or Auto-Detect. Oneup means that this tuning catalog entry will always be used with PDF files containing single designs. Layout always processes the PDF files as if they contain multiple single designs. Auto-Detect reads the geometry of the vectors and process the file appropriately.

2. Set the Design Side and Die Side list boxes to appropriate values (Inside or Outside and Die Back Side or Die Knife Side) for the incoming PDF data.

3. In the Flute or Grain group, choose either Vertical or Horizontal for the direction of the material used to manufacture the workspace.

4. The controls in the Mapping group determine how ArtiosCAD converts vectors from the PDF data into design lines. The easiest way to set this up is to use an existing PDF file as a template. You may need to do this for each supplier of PDF files (in separate tuning files) depending on the software they use to create them.
5. Click Template.
6. Select the PDF file to use as a template and click Open. If the PDF file contains more than one page, select the page to use as the template.
7. In the preview image, select a line and then set what it should be in the ArtiosCAD Design Attributes group. In the example below, the edge was selected and set to be cut lines in the Main Design layer at pointage 2. The Zoom controls above Apply and Help let you change the view so you can select the lines you need to tune more easily.

8. Continue selecting lines in the preview and defining their properties in the ArtiosCAD Design Attributes group. If you make a mistake, select a line of tuning and click Delete. ArtiosCAD applies the tuning entries in a top-down order. In the example above, it will set all the lines with color Artios_cut to be Cut lines, and then it will set all the lines with color Artios_creste to be Crease lines. Shown below is the same dialog box as above with more line types tuned. You can change the processing order by selecting a line of tuning and clicking the up and down arrows at the edge of the group control.
9. After you have finished selecting all the lines in the preview to tune, click Insert to add a final line of tuning. This final line of tuning will send all lines that are not explicitly tuned to the Annotation layer as 2-point Annotation lines.
   a) For this final line of tuning, set the controls in the PDF attributes group to any, any, and any.
   b) In the ArtiosCAD Design Attributes group, set the Layer to Annotation, the Line type to Annotation, and the Pointage to 2.

10. Click Color Names to change the names of the colors coming in from the tuning template. In the Color Names dialog box, you can rename an existing color by selecting it and changing the Name: field. For example, if you have a color defined as 100% magenta, 30% yellow, and 15% black, you could select it and rename it reddish. You can also insert and delete color definitions as desired using Insert and Delete, but selecting lines in the template is the intended way of inserting color names initially.
   a) There are two colors with special meaning. White matches all paths that have no color assigned or that have 0% for all components. Uncolored matches all paths that are not rendered in the PDF file, such as guides and clipping paths.

11. Click Options to open the PDF Import Options dialog box.

   a) Import text as outlines controls if text is imported or ignored and also controls the availability of the two sub-options. Add to annotation layer adds the text as outlines to an Annotation layer using the Annotation line type. Add as fills to graphics layer adds the text to a Graphics layer and fills it with the font color specified in the PDF file.
   b) Change beziers to arcs changes beziers to arcs during the import process instead of having to use the tool later.


c) **Unfiltered items to graphics layer** sends strokes and fills that were not processed by a tuning entry to the Graphics layer.

d) Click **OK** to return to the tuning dialog box.

12. Click **OK** to complete the editing of the tuning catalog entry and return to Defaults.

13. Click **File > Save** to save the changes to Defaults.

14. Click **Yes** to confirm overwriting Defaults.

---

**Editing a PDF vector import tuning catalog entry manually**

If you do not configure the PDF line type mapping using a template as described in the previous section, configure it manually by following the instructions below.

Once you have the tuning catalog entry open, set the options according to the procedure described below.

1. In the **Import As** group, select either **Oneup**, **Layout**, or **Auto-Detect**. **Oneup** means that this tuning catalog entry will always be used with PDF files containing single designs. **Layout** always processes the PDF files as if they contain multiple single designs. **Auto-Detect** reads the geometry of the vectors and process the file appropriately.

![Import As Options](image)

2. Set the **Design Side** and **Die Side** list boxes to appropriate values (Inside or Outside and Die Back Side or Die Knife Side) for the incoming PDF data.

3. In the **Flute or Grain** group, choose either **Vertical** or **Horizontal** for the direction of the material used to manufacture the workspace.

![Flute or Grain Options](image)

4. The controls in the **Mapping** group determine how ArtiosCAD converts vectors from the PDF file into design lines.

   a) In the **PDF Attributes** group, **Layer**: selects the layer in the PDF file containing the lines to process for import into ArtiosCAD.

   b) **Graphic Type**: can be a few different choices. **Any** is any stroke, fill, or bitmap. **Any_stroke** matches any stroke and **fill** matches any fill. Any positive number is a stroke width.

   c) The **Color Names**: field and corresponding dialog box are meant for use when defining tuning using a template. Ignore them when setting tuning manually.

   d) In the **ArtiosCAD Design Attributes** group, set the **Layer**, **Line Type**, and **Pointage**: fields to the type of ArtiosCAD line that will be created for each line that matches the parameters.
set in the PDF Attributes group. Use PDF layer name names the ArtiosCAD layer the same as the PDF layer.
e) The mapping entries are processed in a top-down order. To change the order, select an entry, then use the up and down arrows at the side of the control to move the selected entry.

5. After you have finished selecting all the lines in the preview to tune, click Insert to add a final line of tuning. This final line of tuning will send all lines that are not explicitly tuned to the Annotation layer as 2-point Annotation lines.
a) For this final line of tuning, set the controls in the PDF attributes group to any, any, and any.
b) In the ArtiosCAD Design Attributes group, set the Layer to Annotation, the Line type to Annotation, and the Pointage to 2.

6. Click Options to open the PDF Import Options dialog box.

![PDF Import Options dialog box]

a) **Import text as outlines** controls if text is imported or ignored and also controls the availability of the two sub-options. **Add to annotation layer** adds the text as outlines to an Annotation layer using the Annotation line type. **Add as fills to graphics layer** adds the text to a Graphics layer and fills it with the font color specified in the PDF file.
b) **Change beziers to arcs** changes beziers to arcs during the import process instead of having to use the tool later.
c) **Unfiltered items to graphics layer** sends strokes and fills that were not processed by a tuning entry to the Graphics layer.
d) Click OK to return to the tuning dialog box.

7. Click OK to complete the editing of the tuning catalog entry and return to Defaults.

8. Click File > Save to save the changes to Defaults.

9. Click Yes to confirm overwriting Defaults.

### Line types

- Distinguish between rules for flat dies and rules for rotary dies.
- Distinguish between lines which should be burnt on their outer edge (such as the die wood edge) and those which should be burnt on their inner edge (such as lower stripping holes).
• Support pointage of 1.5.

Line information in older workspaces will be automatically converted with no loss of information when the workspaces are opened in Esko ArtiosCAD.

The category in which a line type resides determines its properties and also how it is processed during output. Of particular note are the Inside Edge and Outside edge categories. When line types belonging to these categories are output to a laser, they are offset to compensate for their thickness. Tool outlines must form complete loops for the direction of the offset to be well defined. Line types in the Center Edge category have no offset - the laser beam burns down their middle.

Only rules should have bridges defined, and only tool outline line types should have tack bridges. Other line types should be created without bridges. Only cut lines should have nicks.

Table: Line Type Categories

<table>
<thead>
<tr>
<th>Line type category</th>
<th>Examples</th>
<th>Properties / Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Generic rules</td>
<td>Cut, crease, reverse crease, glue assist, balance rules, chop knives</td>
<td>Width, bridges, subtype</td>
</tr>
<tr>
<td>Rules</td>
<td>1/8 x 1/8 perf, 1/4 x 1/4 cut/crease, zipper rule, punch hole, side bevel cut, specific rotary cut rule. Defined in the rule table.</td>
<td>Width, Bridges, Generic type, height, name, rule geometry, rotary teeth spacing</td>
</tr>
<tr>
<td>Center edge</td>
<td>Unknifed diecut, unknifed lower, unknifed upper</td>
<td>Width, tack bridges, subtype</td>
</tr>
<tr>
<td>Inside edge</td>
<td>Lower stripping hole, Die registration hole</td>
<td>Tack bridges, subtype. Offset is applied inward during laser output.</td>
</tr>
<tr>
<td>Outside edge</td>
<td>Die edge, Lower stripping edge, stripping block</td>
<td>Tack bridges, subtype. Offset is applied outward during laser output.</td>
</tr>
<tr>
<td>Etch</td>
<td>Die etch, lower stripping etch, upper stripping etch</td>
<td>Subtype. No offset applied during laser output.</td>
</tr>
<tr>
<td>Laser output</td>
<td>Tool 1, Tool 2, Tool 3, .. increased from 20 to 30 laser line types.</td>
<td>None. Laser output will pass beam width for DDES,CFF2</td>
</tr>
<tr>
<td>Annotation</td>
<td>Annotation, Print image, Print registration, Bleed, Varnish</td>
<td>None</td>
</tr>
<tr>
<td>Sample output</td>
<td>Sample cut, sample partial cut, sample with grain crease, sample cross grain crease</td>
<td>None</td>
</tr>
<tr>
<td>Counter output</td>
<td>With grain channel, Cross grain channel, Crease end tool, Periphery tool</td>
<td>None</td>
</tr>
</tbody>
</table>
Subtypes

A subtype is a definition to further classify a type of line - it is a label only and is used for sorting the tool assignment data when performing a CAM Output. Subtypes can have their own entries in Advanced plotting styles. Up to 100 subtypes can be defined in the Subtype Mapping Catalog.

To define a subtype, select it, and enter the description of the subtype in the Description: field. Repeat as desired, and when done, click OK to save the definitions or Cancel to ignore them and return to Defaults.
Plotting Styles

Plotting styles control the appearance of lines on screen and how lines are drawn or made on printers, plotters, samplers, and countercutters. Plotting styles control the colors, widths, and styles used. They also control which physical tool an output device uses to construct the line.

Note: Plotting styles are not used to configure CAM outputs such as lasers and Kongsberg DieSaws; output to those types of devices is configured using the CAM Tooling Setup catalog.

Each plotter, sampler, and countercutter should have its own plotting style configured as each has a different tool configuration. Most known output devices have plotting styles defined in the example CAM Outputs configuration file, located in ..\InstLib\eg_cam_outputs.adf. To view the example CAM Outputs, simply open that file in Defaults, open the Plotting Style Catalog it contains, and drag the appropriate entry into the Plotting Style Catalog in the Shared defaults pane. The names may contain CAM, but rest assured these are for plotters, samplers, and countercutters.

Plotting styles may be either simple (the default) or advanced. Simple plotting styles configure the line color, line width, line style, and tool number based solely on a line type. Advanced plotting styles do the same configuration, but can be based on line type, subtype, and pointage instead of just line type. An Advanced plotting style can not revert to a Simple plotting style.

Working with simple plotting styles

When the Advanced checkbox in the Line Types group is clear, the current plotting style is considered simple. In a simple plotting style, the line types are organized on five tabs - Design, Manufacturing, Sample, Tooling, and Counter.
On each tab are the line types relevant to each label. For examples, the Design tab lists such types as Cut, Crease, Partial cut, and so forth. The Manufacturing tab contains line types used in the Manufacturing module, such as Stripping Rule, Die board edge, and Unknifed blanker. The same is true for the other tabs.

When looking at the line types, shown first is the line type name, then an example of the line type using the attributes selected in the right pane, and finally the tool number. When creating a new plotting style, all the tool numbers are set to zero by default.

Creating a new simple plotting style

To create a completely new simple plotting style, do the following:

1. Right-click the Plotting Style Catalog entry in Defaults and click New > Data on the context menus that appear.

2. Enter a name for the new plotting style and press ENTER.

3. Double-click the style to access its property page in order to change its defaults. The window should appear similar to the one shown below.
On the left is a set of five tabs that correspond to the different categories of lines. Each tab has a control showing the line name, the line style, the line color, and the currently-assigned tool number. On the right are controls to set those attributes.

Checking the **Output Only** checkbox makes this plotting style available only when configuring an Output - it is hidden when changing the plotting style in the View Mode.

**Save as XML** saves the plotting style as an XML file for use with the WebCenter Viewer.

For outputs, the **tool number** applies only when this plotting style will be used in an Output definition using a CAM driver.

For outputs, the **Color** and **Line Width** apply only when this plotting style will be used in an Output definition using either a Windows driver or a graphic CAM driver such as the PCL driver used for sample printing.

**Copy From Style** copies attributes of another plotting style into the current one.

4. Click each line style and set the line attributes as desired.
5. Click **OK** to save the changes to return to Defaults, or click **Cancel** to discard them and return to Defaults.
Changing the line attributes in a simple plotting style

To change the attributes of a line in a simple plotting style, do the following:

1. Open Defaults and double-click the plotting style to change.
2. Click the tab for the class containing the lines you wish to modify.
3. Select the line to modify in the Line Types group.
4. Select the new attributes - Color, Line Width, Line Style - from their respective lists in the Attributes group.
5. If desired, change the Tool Number.
6. Click OK to save the changes and return to the Plotting Style catalog, or click Cancel to discard the changes.

Creating a new plotting style by copying from another plotting style

To base a new plotting style on an existing plotting style, do the following:

1. Right-click the Plotting Style Catalog entry in Defaults and click New > Data on the context menus that appear.
2. Enter a name for the new plotting style and press enter.
3. Double-click the style to access its property page in order to change its defaults.
4. Click Copy From Style.
5. Select the plotting style to copy from on the left.
6. Select the line type categories to copy by checking the appropriate checkboxes in the Line Type Subset group None are selected by default.
7. Select or clear the Copy color and style and Copy tool numbers checkboxes as appropriate.
8. Click OK to copy the styles and attributes or click Cancel to return to the Plotting Style dialog box.

Working with Advanced plotting styles

When the Advanced checkbox is checked, the plotting style is considered Advanced. Configuring an Advanced plotting style takes slightly more time than a Simple plotting style but it allows greater flexibility.

Creating a new Advanced plotting style

To create a completely new Advanced plotting style, do the following:

1. Right-click the Plotting Style Catalog entry in Defaults and click New > Data on the context menus that appear.

2. Enter a name for the new plotting style and press enter.
3. Double-click the style to access its property page in order to change its defaults and click the Advanced checkbox. You will be asked to choose whether to keep the definitions from the simple style or to create a blank style. For the purposes of this workflow, choose Initialize to blank style and click OK.

![Convert simple plotting style to advanced dialog box]

4. The blank Advanced style appears as shown below.

![Advanced plotting style dialog box]

The Attributes group for an Advanced plotting style works in the same way as it does for a Simple plotting style. The Line Types group, however, works differently as explained below.

5. In a newly-created advanced plotting style, as seen in the Line Types group, all lines are configured to be shown as solid line style, black color, 1.00 width, and tool 0.
Advanced plotting styles work by assigning line attributes based on conditions defined in a line style statement. If a line meets a line style condition, it is assigned whatever attributes are assigned in the line style statement. The (Remainder) statement applies to line styles which meet none of the conditions. There will always be a (Remainder) statement; it cannot be deleted, but its attributes can be modified as desired.

6. To add line style statements, do the following:

   - Click Insert. A new entry appears in the table and the corresponding Esko ArtiosCAD syntax is shown in the Selection: field. By default, this new entry is solid line style, black color, 1.00 width, and tool 0.
Click **Types** to open the Types dialog box. All line types other than **Annotation** are selected by default. Select the line types to designate in this line style condition. Click category tabs (Design, Sample, Tooling, Counter, and Manufacturing) to choose from lists of line types in those areas of Esko ArtiosCAD. **Select All** selects all the line types on each tab. **Invert Selection** reverses whatever is selected - each style that is checked becomes cleared and each style that was clear becomes checked. Your selection will appear in Esko ArtiosCAD syntax in the field below the checkboxes.
Click **OK** to save this selection and return to the Plotting Style properties dialog box or **Cancel** to ignore this selection and return to the Plotting Style properties dialog box.

Once back at the Plotting Style properties dialog box, click **Subtypes** to open the Subtypes dialog box and choose which subtypes to include in this line style definition. The Esko ArtiosCAD syntax for your selection will appear in the field beneath the checkboxes.
Click **OK** to save this selection and return to the Plotting Style properties dialog box or **Cancel** to ignore this selection and return to the Plotting Style properties dialog box.

Click **Pointages** to open the Pointages dialog box. Check or clear the checkboxes as desired. The Esko ArtiosCAD syntax for your selection will appear in the field beneath the checkboxes.
• Click OK to save this selection and return to the Plotting Style properties dialog box or Cancel to ignore this selection and return to the Plotting Style properties dialog box.

• Do not forget to also change the line attributes (color, line width, line style, and tool number) if you do not want them to be the default of black, 1.00, solid, and 0. Shown below is an example of crease line subtypes 0 through 2 and 4 through 5, pointage 0 through 1 and 2 and greater made to be red, have a line width of 1.00, a style of dot-dash, and output on tool 2.

• If you are familiar with Esko ArtiosCAD syntax, you may edit the Selection: field directly.

7. Repeat step 5 for each line style statement you wish to make. Shown below is the Artios.Pointage style which is based on line pointage; it uses a different color and tool number for each pointage.
8. When you have finished adding line style statements, click OK to save the changes to the plotting style or click Cancel to ignore them and return to Defaults.

9. Save the Defaults by clicking File > Save and clicking Yes when prompted to overwrite the existing Defaults.

Converting a simple plotting style to an Advanced plotting style

When you convert a simple plotting style to an Advanced plotting style by checking the Advanced checkbox in a simple plotting style definition, you can choose to preserve the line style definitions, or you can initialize the Advanced plotting style with no values. Preserving the settings of the simple plotting style results in an Advanced plotting style similar to the one shown below. Initializing a blank plotting style results in having to define every line type as described in the previous section. In either case, edit the properties as desired. When done, click OK to accept the changes, or click Cancel to discard them.
Special Rule Types

Esko ArtiosCAD provides a series of generic rule types such as cut and crease. These are used at the design phase when how the line is cut or creased is unimportant or not known. There are other generic rule types, for example, for cut-crease lines. It is often necessary to specify that a cut-crease line is in fact ½” cut by ½” crease with crease used as the end land. In this case, a special rule should be used.

Special rules may also be used for simple cut and crease rules to allow more specific control during manufacturing. For example:

- Helmold 3pt 0.937” cut rule
- Böhler 3pt 23.6mm SN1 notched cutting rule

Specific bridging methods, rule widths, subtypes (for special cutting characteristics), and labels can be associated with special rules which aid in the flat and rotary die manufacturing process. Shown below is the Special Rule Types Catalog in Defaults.
When editing a special rule, the Properties dialog box will have one or two tabs depending on the class of rule. All special rules have a **Manufacturing** tab with options such as pointage, subtype, rule height, flat rule or rotary rule, and so forth. Special rules that require more information to manufacture, such as perf, have a **Special Rule** tab in which rule-specific parameters are set, such as length of cut and crease, land length, land options, and so forth.

**Manufacturing tab for special rules**

Shown below is an example of the Manufacturing tab for 3-point rotary cut rule in Imperial format.
Pointage sets the pointage of the rule. You can either select a value from the drop-down list box or type your own.

Subtype sets the subtype of the rule. The subtype is used to configure specific manufacturing options in a CAM Tooling Setup.

Rule height and Rule Cost set the height and cost of the rule respectively. This information is used in line type legends and DDES-3 files only.

The text in the Rule Label field is printed when rule labels are enabled in the View Mode dialog box or in Outputs.

The Generic Type field is used to determine if more configuration options are needed on a second tab. For example, the cut and crease generic types do not require more configuration information, but the perfs and cut & crease types do.

The options in the Die Representation group determine how the rule is manufactured. Use pointage sets the slot width to the pointage * 0.014 inches. Design sets the slot in the die so that it follows the design of the special rule, such as waves in a wave rule. The slot width is the pointage * 0.014 inches. Slot lets you specify a custom slot width independent of the pointage.

Bridge formula for the rule path tells the rule which bridging formula to use. Bridge on teeth controls whether or not the rule has teeth, and what the tooth size, tab width, and which bridging formula to use.

Nominal tooth spacing refers to the customary size of the teeth on rotary rule. The Actual spacing is the distance between each tooth measured around the cylinder at cylinder level. The easiest way to measure this value is to mark off the distance between 10 teeth on a cylinder, and measure around the cylinder with an accurate tape measure, making sure to use the same point for the start and end. Divide by 10 to get the Actual spacing.

The Cylinder diameter is the cylinder for which this rule was made. It is used ONLY to compute tooth spacing when other-sized cylinders use this rule.
If you enter a cylinder diameter and an actual spacing value, Esko ArtiosCAD can compute the tooth spacing when other die cylinders are used. This means that you do not have to enter separate special rules for each diameter cylinder!

Table: Rule Tooth Values

<table>
<thead>
<tr>
<th>Common cylinder diameters</th>
<th>Tooth Spacing</th>
<th>Result from</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nominal</td>
<td>Actual</td>
</tr>
<tr>
<td>Inch</td>
<td></td>
<td></td>
</tr>
<tr>
<td>21.000</td>
<td>0.500</td>
<td>0.471</td>
</tr>
<tr>
<td>19.1875</td>
<td>0.500</td>
<td>0.469</td>
</tr>
<tr>
<td>14.125</td>
<td>0.500</td>
<td>0.458</td>
</tr>
<tr>
<td>11.500</td>
<td>0.500</td>
<td>0.450</td>
</tr>
<tr>
<td>10.1875</td>
<td>0.500</td>
<td>0.444</td>
</tr>
<tr>
<td>7.000</td>
<td>0.500</td>
<td>0.423</td>
</tr>
<tr>
<td>Metric</td>
<td></td>
<td></td>
</tr>
<tr>
<td>530.00</td>
<td>11.00</td>
<td>10.36</td>
</tr>
<tr>
<td>487.36</td>
<td>11.00</td>
<td>10.31</td>
</tr>
<tr>
<td>300.00</td>
<td>11.00</td>
<td>9.92</td>
</tr>
<tr>
<td>250.00</td>
<td>11.00</td>
<td>9.73</td>
</tr>
</tbody>
</table>

The tooth spacing is different for different cylinder diameters because when the rule is bent, it bends around the neutral axis. The neutral axis is the point in the rule that neither gets longer nor shorter when the rule is bent into a circular arc. The position of the neutral axis will be different for different types of rule. For example, some 6pt or 3pt rule will have the neutral axis a little closer to the top of the rule than simple 3pt rule. This is because the 6pt part is harder to bend than the 3pt part towards the bottom. Of course, the bottom of the rule has teeth cut out of it, so it is very easy to bend. Normal 3pt toothed rule would have the neutral axis about half way between the top of the tooth notch and the top of the rule.

As the cylinder diameter gets smaller, the teeth will get closer together, but the distance along the neutral axis will remain unchanged. The position of the neutral axis can be computed from the values given. It can then be used to calculate the tooth spacing for any diameter.
Length of adjustment for blend is used to calculate tooth positions after bending the rule. The entry in this field is derived from the table accessed by clicking the f(x) button.

The Flat rule and Rotary rule option buttons designate the kind of rule. When Flat Rule is chosen, click the Browse button (...) at the end of the Associated rotary rule: field to choose the associated rotary rule. Rotary to flat rule and Flat to rotary rule determine which is bent when a rule path bends around a corner when making a rotary die. Extension past corner sets the amount of rule to keep after the bend. Angle at which to use rotary rule sets the angle at which to switch from using flat rule to using rotary rule. The angle is measured from the horizontal axis.

Special rule tab for special rules

The Special Rules tab is different for each kind of special rule. For example, for a perf rule, it looks like this:
But for a scallop rule it looks like this:

For wave and zig zag rules, the **Nick on top** option is available:
You probably already know the parameters for the rule you are configuring; enter the values and click **OK** to save the changes to the special rule and return to Defaults, or click **Cancel** to discard the changes.

**Multi-Perf Overview**

*Multi-perf* is perforation rule with a repeating pattern of differently sized teeth and gaps. Shown below are some examples of multi-perf rules.

![Multi-perf patterns](image)

The pattern for the multi-perf is defined in a workspace.

Save your pattern definition workspace to **ServerLib** (conventional version) or the **ServerLib** folder in the Shared Defaults Project (Enterprise version).

**Designing a multi-perf pattern**

In each pattern definition workspace, design a combination of horizontally aligned cuts, creases, and gaps that define the pattern. You can add optional vertical annotation lines marking the land length and the ends of the pattern. Shown below are sample patterns.
Special rule type

1/8 1/8 perf-in-crease

1/4 1/4 perf crease

1/4 1/4 3/4 perf

1/8 1/8 3/8 perf

Multi-perf1

Perf pattern workspace

---

The vertical annotation line in the 1/8 1/8 perf-in-crease example instructs ArtiosCAD to treat the crease as the land. For the 1/4 1/4 3/4 perf, the land is the space between the two annotation lines, which ArtiosCAD uses to center the perf pattern along design lines. The Multi-perf1 pattern is too long for a land, so the vertical annotation line marks the end of the pattern.

Use the following line types when designing your own pattern definition workspace:

- Cut
- Partial cut
- Reverse partial cut
- Crease
- Second height crease
- Matrix crease

**Defining the land**

An annotation line in the middle of a pattern marks the right end of a land.

If the pattern has a land, ArtiosCAD centers the perf pattern on the line with the same land at both ends. Shown below is an example of how the pattern is centered on lines of different length.
If the pattern has no land, ArtiosCAD aligns the pattern with the start of the line, and the value in the Minimum land/pattern offset field of the special rule definition defines the offset of the pattern along the line as shown below.

---

Offset 0

---

Offset 2mm

---

Offset 4mm

---

Offset 6mm

---

Offset 8mm

To make the pattern center correctly on the rule, make the land symmetrical, so it has the same cut or gap at both ends.

| Land |  | Land |  |  | Symmetrical, correct
| Land |  | Land |  |  | Symmetrical, correct
| Land |  |  | Asymmetrical, wrong

Adding a Multi-Perf Definition to the Special Rule Types Catalog

Once you have designed a pattern definition workspace for a multi-perf and saved it to ServerLib, do the following to add it to the Special Rule Types Catalog.

1. Start ArtiosCAD. If you are using an Enterprise version, log in as a member of ADMINS.
2. Click Options > Defaults.
3. In the Shared Defaults pane, expand the Special Rule Types Catalog.
4. Right-click Multi-perf and then click New > Data on the context menu.
5. Enter a name for the new multi-perf type and press Enter.

6. On the Manufacturing tab of the special rule type dialog box, select Multi-perf from the Generic type drop-down list box.

7. On the Special rule tab of the special rule type dialog box, enter or browse for the pattern definition file. Enter a value in the Minimum land/pattern offset field if the pattern definition workspace does not have a land defined.
8. Click **OK** to save the new multi-perf catalog entry.

9. Save and exit Defaults as usual.

**Edge band and Tear Tape Overview**

*Edge band* is a cover that hides the edge of thick material, such as Re-board®, usually used for displays.
*Tear tape* is the thin plastic strip on the inside of box-opening features.
Both edge band and tear tape are special rules.

**Defining an Edge Band**

To define an edge band, do the following:

1. Design a cross-section of the edge band as a single design. Make sure it forms a loop and orient it horizontally so that its lower middle section aligns with the board edge.
2. Copy the cross-section workspace to ServerLib.

3. In Defaults, add a new entry to the Edgeband folder of the Special Rule Types catalog by right-clicking it and then clicking New > Data on the context menu.

4. On the Manufacturing tab, set its Generic type to Edge band.

5. On the Special rule tab, enter the cross-section workspace's name or browse for it.

6. Click Select Color to set the color of the edge band.
7. Click OK to return to Defaults.
8. Modify a plotting style to give the Edge band type a thick dashed line.

<table>
<thead>
<tr>
<th>Tear tape</th>
<th>............................. 0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Edge band</td>
<td>- - - - - - - - - - - - - - -</td>
</tr>
</tbody>
</table>

9. Save and exit Defaults normally.

**Defining a Tear Tape**

A tear tape has a thickness and a color.

1. In Defaults, add a new entry to the Tear tape folder of the Special Rule Types catalog by right-clicking it and then clicking New > Data on the context menu.
2. On the Manufacturing tab, set the Generic type to Tear tape.
3. On the Special rule tab, enter a value in the Width field.

4. Click Select Color to set the color of the tear tape.
5. Click OK to return to Defaults. Save and exit Defaults normally.

Default line type set

When you first launch Esko ArtiosCAD, cut and crease are the two line types available from the Line type drop-down list box in the View bar. To change the available lines at startup, double-click the Default Line Type Set entry in Design defaults. The Default Line Type Set dialog box will appear.
To add a line type or special rule type to the default set available at startup, select it and click **Add to default set**. To delete a line type or special rule type from the default set, select it in the **Default Line Type Set** group and click **Delete**.

When you have finished modifying the Default Line Type Set, click **OK** to save your changes and return to Defaults, or click **Cancel** to return to Defaults and discard your changes.

### CAM Tooling Setup Catalog

The CAM Tooling Setup Catalog contains the configuration information for CAM output devices such as Kongsberg DieSaws and lasers. Configuration information is separated onto three tabs in each Catalog entry - **Tool List**, **Tool Selection**, and **Optimization**.

#### Tool List tab

The Tool List tab contains the configuration information for the tool type, width, depth (if applicable), step height, overlap, lead hole (if applicable), and name of the tool. Each combination of tool, width, and depth needs to be assigned to a different tooling line type. To use an expression to define the tool width, select **Use Expression**.
To set up the tool list, choose the tool number to modify and then set the tool type. Set the available options and enter a name for the tool in the **Name** field.

The table below explains the function of each tool type.

**Table: Tool Types, Their Properties, And Intended Uses**

<table>
<thead>
<tr>
<th>Tool type</th>
<th>Properties</th>
<th>Description</th>
<th>Intended uses</th>
</tr>
</thead>
<tbody>
<tr>
<td>line</td>
<td>none</td>
<td>Uses the original lines; performs no processing.</td>
<td>Drawing with a pen.</td>
</tr>
<tr>
<td>slot</td>
<td>Width - effective width of the tool</td>
<td>Makes a slot of a defined width. Will envelope if the slot width is wider than the tool width.</td>
<td>Laser-cut slot for a rule in a dieboard</td>
</tr>
<tr>
<td></td>
<td>Overlap - overlap between multiple passes of the tool. Available when Partial Depth is selected.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>drill</td>
<td>Width - effective diameter of the drill</td>
<td>Drills a hole at a point whose diameter is defined by the width in the CAD setup.</td>
<td>Mounting holes, leader holes</td>
</tr>
<tr>
<td></td>
<td>Full Depth - Depth is the full depth value, defined in the output</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Partial Depth - Depth is defined in the CAM setup</td>
<td></td>
<td></td>
</tr>
<tr>
<td>any drill</td>
<td>Full Depth - Depth is the full depth value, defined in the output</td>
<td>Drills a hole at a point whose diameter is defined by the diameter of the circle in the workspace.</td>
<td>Mounting holes</td>
</tr>
<tr>
<td></td>
<td>Partial Depth - Depth is defined in the CAM setup</td>
<td></td>
<td></td>
</tr>
<tr>
<td>counter drill</td>
<td>Width - effective diameter of the drill</td>
<td>Drills a hole with a counter sink drill, same as the drill tool type, but sends different commands to the Kongsberg driver.</td>
<td>Countersink holes.</td>
</tr>
<tr>
<td></td>
<td>Full Depth - Depth is the full depth value, defined in the output</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Partial Depth - Depth is defined in the CAM setup</td>
<td></td>
<td></td>
</tr>
<tr>
<td>mill</td>
<td>Width - Effective diameter of the tool.</td>
<td>Mills inside or outside the outline which is usually a loop.</td>
<td>Holes in a dieboard that are too small to saw; the dieboard alignment notch.</td>
</tr>
<tr>
<td></td>
<td>Full depth - Depth is the full depth value, defined in the output.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Partial depth - Depth is defined in the CAM setup.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Include fill (off) - One pass inside or outside the outline.</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Include fill (on) - Multiple loops, starting in the middle of a hole, and working outwards</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tool type</td>
<td>Properties</td>
<td>Description</td>
<td>Intended uses</td>
</tr>
<tr>
<td>-----------</td>
<td>------------</td>
<td>-------------</td>
<td>---------------</td>
</tr>
<tr>
<td>saw</td>
<td>Step height - Amount by which the tool penetrates into the material if the tool requires multiple passes at increasing depth to complete the hole.</td>
<td>Makes a slot of a specified width, same as the slot tool type, but adds a leader hole.</td>
<td>Lines sawn on a DieSaw.</td>
</tr>
<tr>
<td></td>
<td>Width - Effective width of the saw blade</td>
<td>Makes a slot of a specified width, same as the slot tool type, but adds a leader hole.</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Lead hole - Which leader hole to use.</td>
<td>Makes a slot of a specified width, same as the slot tool type, but adds a leader hole.</td>
<td></td>
</tr>
</tbody>
</table>

### Tool Processing and Offsets when Outputting

A combination of the Tool type in the CAM Tooling Setup and the line type category of lines in the workspace determines how the lines are offset on Output. For example, a mill, saw, or slot tool will be offset by half the tool width for inside-edge or outside-edge lines such as Dieboard edge or Lower stripping hole. The same mill, saw, or slot tool will cut on a center-edge line (such as Unknifed diecut or a rule such as Cut) without an offset.

Inside-edge and outside-edge line types are defined as the edge of a loop and do not have a slot width. When output through a CAM Output, their offset is half the tool width in the CAM Tooling Setup. The pointage property of inside- and outside-edge line types can be used for selection purposes but does not affect the processing in a CAM Output.

### Table: Offsets when Outputting

<table>
<thead>
<tr>
<th>Tool Type</th>
<th>Line type</th>
<th>Processing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line</td>
<td>All</td>
<td>Unchanged</td>
</tr>
<tr>
<td>Slot or Saw</td>
<td>Draw</td>
<td>Unchanged</td>
</tr>
<tr>
<td>Inside edge</td>
<td>Cuts path inside by half tool width</td>
<td></td>
</tr>
<tr>
<td>Outside edge</td>
<td>Cuts path outside by half tool width</td>
<td></td>
</tr>
<tr>
<td>Center edge, fill off</td>
<td>If width is less than slot width, cuts envelope, else unchanged</td>
<td></td>
</tr>
<tr>
<td>Center edge, fill on</td>
<td>If width is less than pointage or slot width, multipass, else unchanged</td>
<td></td>
</tr>
<tr>
<td></td>
<td>If saw, adds leader holes as necessary.</td>
<td></td>
</tr>
<tr>
<td>Drill, any drill, or</td>
<td>All</td>
<td>Unchanged</td>
</tr>
<tr>
<td>countersink drill</td>
<td>Draw</td>
<td>Unchanged</td>
</tr>
<tr>
<td>Mill</td>
<td>Draw</td>
<td>Unchanged</td>
</tr>
<tr>
<td>Inside edge</td>
<td>Cuts path inside by half tool width</td>
<td></td>
</tr>
<tr>
<td>Tool Type</td>
<td>Line type</td>
<td>Processing</td>
</tr>
<tr>
<td>-----------------</td>
<td>----------------------------------</td>
<td>-----------------------------------------------------------------------------</td>
</tr>
<tr>
<td>Outside edge</td>
<td>Cuts path outside by half tool width</td>
<td></td>
</tr>
<tr>
<td>Center edge, fill off</td>
<td>If width less than slot width, cuts envelope, else unchanged</td>
<td></td>
</tr>
<tr>
<td>Center Edge, fill on</td>
<td>Series of loops starting in the middle and working outward</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Repeats with the next depth necessary depending on the step height</td>
</tr>
</tbody>
</table>

**Tool Selection tab**

The Tool Selection tab maps line types, pointages, and subtypes to tools on the output device. It is also the place where the output priority is assigned. During output, a line is processed according to the first selection in the list that matches the line's properties. When a line matches the selection criteria, it is mapped to the specified CAM tool and then assigned to a group. Once a line has been mapped, the process begins again with the next piece of line data; lines are processed by one selection and one selection only. When all the data has been processed, the groups are internally optimized and output to the machine in ascending order by group number. Lines that do not match any of the selections in the list are processed using the (Remainder) selection, which usually are not output.

In the example above, drawn lines, etch lines, and lines with subtype Countersink ID are output first as they are in group 1. The Countersink ID lines are not actually created on the machine as they are assigned to the Do not output tool. Once the group 1 lines are output, then the group 10 lines are output, then 20, then 30, and so forth.
The **Types**, **Subtypes**, and **Pointages** buttons lead to dialog boxes where you can select exactly what will be output in this selection. These dialog boxes are similar to those used for selections in plotting styles.

**Note:** The **Text and dimensions** checkbox in **Types > Design** only applies to text and dimensions that have not had a color applied. Those with a color applied are considered graphics and are not output to CAM devices.

By default, all items are turned on when you enter the dialog box for the first time. Click **Invert Selection** to clear each checkbox. Clicking **Select All** selects all the items in the list. If you make selections other than having all selected or having none selected, the base Esko ArtiosCAD syntax is shown in the field beneath the list. When you have finished selecting, click **OK** to return to the Setup dialog box saving your changes or **Cancel** to return, ignoring your changes.

![Types dialog box](image)

The **Slot Width** button opens the Slot Width dialog box in which you can specify a minimum slot width and a maximum slot width. Any line with a slot width between those values will be selected. #I or #M is automatically appended to the slot width selection to allow for conversion between units of measurement.
The up and down arrows to the side of the selection list move the selected entry either up or down in the list depending on which button is clicked.

**Insert** adds a new entry above the current selection; **Delete** deletes the current selection. The (Remainder) selection may not be deleted.

The **Tool number** field selects the CAM tool to map to the line selection. Entries for this drop-down list come from the Tool List tab. The **Size only** tool is meant to be used for easier registration. Lines sent to the **Size only** tool show in a preview and are used to calculate the output size, but are only actually output when outputting to a layer.

The **Width adjustment** field allows you to add to the slot width that is cut. Usually the slot width is defined by the slot width of the rule if so defined or by the pointage of the rule. However, the preferred way to adjust the slot width for laser output is to use a special rule with a defined slot width.

The **Group number** defines which group the selected lines are put into. Group 1 is output first, then group 2, group 3, and so on.

If the **Separate bridge tool** checkbox is checked, another line is added beneath the selection in the list to allow the processing of bridges for that selection to be defined separately. This means that bridges can be made with a different tool.

The **Number of tools** field allows for more than one tool to manufacture the lines selected by a selection criteria line. Configure the extra lines as you would any of the other selection criteria lines.

Enter the desired name for the selection criteria line in the **Name** field.

**Note:** Wave rules with nick width of 0 are treated as cuts and processed by the GENTYPE 1 tool selection.

### Optimization tab

The **Optimization** tab controls the internal optimization of the groups of output. Optimizing is on for all groups by default.
To turn off optimization for a group, select it and clear the **Optimize** checkbox. To turn it on, select it and check the **Optimization** checkbox.

**Separate subroutine** controls if each group will be in its own subroutine. If the box is checked, the selected group is independent. If the box is clear, the selected group shares the same subroutine as the previous group.

### CAM Output of Rubber Sheets

A CAM Tooling Setup catalog entry should be set up for the following line types for output to (for example) a waterjet rubber-cutting machine.

<table>
<thead>
<tr>
<th>Line type</th>
<th>Name</th>
<th>Usage</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type 220</td>
<td>Rubber inside edge</td>
<td>If using different logical tools for the inside edge and the outside edge. Otherwise make them the same.</td>
</tr>
<tr>
<td>Type 221</td>
<td>Rubber outline</td>
<td></td>
</tr>
<tr>
<td>Type 222</td>
<td>Rubber name</td>
<td>Cuts the numbers in the rubber pieces, if <strong>Show element numbers</strong> is selected on the Processing page in the Output.</td>
</tr>
<tr>
<td>Type 97</td>
<td>Usable limit of sheet</td>
<td>If required, cuts the outline of the sheet area used.</td>
</tr>
<tr>
<td>Type 98</td>
<td>Sheet size</td>
<td>Map to Size only to see the sheet size.</td>
</tr>
</tbody>
</table>

Shown below is an example CAM Tooling Setup catalog entry.
Specifying a rotary die in a press definition

When you first create a press, Esko ArtiosCAD assumes it is flat by default. To specify a rotary dieboard configuration, double-click the **Rotary Shrink Stretch Factors** entry in the parameter set for that press.

In the Rotary Shrink Stretch Factors dialog box, check the **Rotary Dieboard** checkbox and fill in the values for the **Wood Thickness**, **Cylinder Diameter**, **Design to Wood Level Shrink**, and **Cylinder to Wood Level Stretch** fields with the proper values.

Once the fields are complete, click **OK** to return to Defaults or click **Cancel** to cancel.
Hole Catalog

Holes are used in manufacturing to attach the boards to the presses. There are three types of holes: General holes, Drill holes, and Oblong holes. As holes utilize subtypes, configure subtypes before adding holes.

Adding a hole to the Hole Catalog

To add a hole, right-click Hole Catalog, and click New > Data from the context menu. Give the new hole a name and double-click its entry to open its Properties dialog box. Edit the values as appropriate. The Diameter Y field is available when the hole type is set to Oblong Hole. Counter Sink Diameter is available only when the Counter Sink checkbox is checked. Set the Subtype fields as appropriate.
Nicks

Nick formulas

Nick formulas are stored in the Nick Formulas Catalog. Sample entries are contained in the Artios folder. The default nick formula is chosen in either the Manufacturing or Single Design parameter set. Shown below is a Kongsberg nick formula.
The five placement buttons at the upper left of the tab set the default placement option for nicks when the Add Nick tool is activated.

- **Offset** sets the nicks so that they offset from the nearest endpoint of the line or arc.
- Center places the nick in the center of a line or arc.
- Anywhere places the nick at the point you click.
- Fill line fills the line or arc with evenly spaced nicks.
- **At the end of the line** places the nick at the end of a line.

The fields in the Nick sizes:, Offsets:, and Between nicks: columns set the values for the drop-down list boxes when the Add Nick tool is activated.

The **Number** field sets the default number of nicks to add when the Add Nick tool is activated.
Set the default nick size to use by entering either an absolute value or a formula in the Default nick size: field. If no default size is set, the Add Nick tool and the Bridging properties dialog box default to the first defined size.

Nick styles

Esko ArtiosCAD can represent nicks as a single line, a double line, a text label, or as any workspace you designate. You can configure different ways for nicks to be represented based on their size. Nick Styles are configured in the Nick Styles Catalog. You can either modify the default nick styles in the Artios folder or you can create your own. A sample nick style is shown below.

By default, a newly-created nick style has only one entry in it. This one entry defines how all nicks are shown.
ArtiosCAD Enterprise

Insert and Delete add conditions for nick sizes. Select an entry in the list before clicking either.

The option buttons in the Nick Output Type group - Single line, Pair of lines, Text, and Workspace - control the type of nick made.

The values in the Start and End fields set the length of the line(s) representing the nick. The value in the Start field sets the distance between the line containing the nick and the start of the line representing the nick. Use a negative offset to have the nick representation cross the design line. The value in the End field sets the distance between the line containing the nick and the end of the line representing the nick. The line(s) representing the nick are always drawn perpendicular to the line containing the nick.

The Workspace name: field is available only when the Nick Output Type is set to Workspace. In this field, enter the name of the workspace to use to indicate a nick, or use the Browse button (...) to browse for the nick workspace. The workspace must have the center of the nick representation as the origin of the workspace (where the two construction line axes meet). The line type used to draw the nick workspace is ignored; the nick is drawn as a nick line type. Anything other than lines and arcs in the nick workspace is ignored.

Add the size of the nick as text to the nick by using the Text nick output type in a selected Nick Style in Defaults. Set the size of the text in the Text Size: drop-down list box, which is available only when the Nick output type is set to Text.

When two nicks are close together, and the nicks are represented by workspaces, if the two workspaces would overlap, the second nick is shown as a line.
When zooming in on nicks, the mark size and text size increase. If two nicks are closer together than the width of the text or workspace label, the text or label is shown only once.

When a text nick style is used in a CAM Output, the text for the nick size may be converted to lines according to how text is handled by the Output. If the Line text option is selected, the nick size text uses the same tool number as the nick line type.

To have Esko ArtiosCAD display the nick sizes on a printed or plotted output, make a Nick Style of all workspaces which are based on the nick size. In each workspace, enter the number, and then convert it to lines using the Line Text tool.

Table: Nick Examples

<table>
<thead>
<tr>
<th>Output Type</th>
<th>Parameters</th>
<th>Output</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single line</td>
<td>Start = 0, End = 1/8</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Single line</td>
<td>Start = 1/32, End = 5/32</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Single line</td>
<td>Start = -1/8, End = 1/8</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Double lines</td>
<td>Start = -1/8, End = 1/8</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Workspace</td>
<td>Workspace: ![Diagram]</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Workspace</td>
<td>Workspace: ![Diagram], Start = 1/16, End = 5/16</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Workspace, with two nicks close together</td>
<td>Workspace: ![Diagram], Start = 1/16, End = 5/16</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Workspace</td>
<td>Workspace: ![Diagram], Start = 1/16, End = 5/16</td>
<td>![Example Image]</td>
</tr>
<tr>
<td>Workspace</td>
<td>Workspace: ![Diagram]</td>
<td>![Example Image]</td>
</tr>
</tbody>
</table>
When you have finished modifying the nick style, click OK to return to Defaults keeping your changes, or click Cancel to return to Defaults ignoring your changes.

Standard Sheet Layout

Parameters and standard sheet sizes for Standard Sheet Layout are configured in the Sheet Utilization catalog in Defaults.

Sheet Layout Parameters

Double-click Sheet Layout Parameters to set the defaults for the trim, gutters, nest types, and other options to use in Standard Sheet Layout as shown below.
In the Trim group, the fields set the trim tolerances at the edge of the sheet. Set them to be less than the maximum values for the presses used to produce the layout.

Use gutters inserts gutters between designs in the nest when it is checked. When it is clear, the single designs touch. The values in the X and Y fields set the gutters in X and Y for the first and second rows and columns of single designs in the layout. The values in the X2 and Y2 drop-down list boxes set the gutters between the second and third rows and columns. These are repeated as appropriate between all the rows and columns in the layout.

In the Nest type group, the checkboxes control the kinds of nests made automatically. Select the checkboxes next to the type of nests to use; clear those next to undesired nests.

Touch bleed outlines controls whether the designs touch using the bleed outlines or the edges of the designs. If Use gutters is enabled in conjunction with Touch bleed outlines, the gutters are inserted between the bleed outlines instead of the edges of the designs.

In the Design options group, Allow 90 degree rotation lets designs rotate so they are cross-grain; the sheet grain runs in the longer direction. Maximum allowable reduction is the maximum distance that could be shaved off each design in order for another row or column to fit on the sheet. If such a cutback is possible in each design, a row or column will overlap the edge of the sheet when a layout is suggested and there will be values in the various Reduction fields of the Sheet Layout Results dialog box.
Click **OK** to save the changes, or click **Cancel** to ignore them and return to Defaults.

**Standard Sheet Sizes**

Double-click **Standard Sheet Sizes** to configure the defaults for standard sheets used by Standard Sheet Layout.

To use a sheet for possible layout solutions, check its box in the **Use** column. To disable its use, clear the checkbox. To toggle the usage of all sheets at once, click **All on/off**.

The arrows on the side of the dialog box move the currently-selected sheet up and down in the list of sheets. The order of sheets on this tab is the same order used to present layout results.

To edit a sheet, select it, and change the fields in the **Edit Sheet Properties** group. The fields change as soon as you click another field.

To add a sheet, click **Insert**, and then edit the fields in the **Edit Sheet Properties** group. Set the checkbox in the **Use** column as desired.

To delete a sheet, select it, and click **Delete**.

To save the changes to the standard sheets, click **OK**. To discard the changes and return to Defaults, click **Cancel**.
Configuring Intelligent Layout defaults and Costing/Estimating

In order to work optimally, Intelligent Layout and Costing/Estimating must have their defaults configured accurately. These defaults should match the real-life manufacturing conditions used to make the containers.

Configuring cost centers

Costing/Estimating uses the general concept of a cost center to allow costs to be configured. A cost center represents a machine such as a printing press, die press or folder-gluer, and/or a material cost such as gold stamping or shipping. Machine costs are calculated per hour and material costs are calculated per sheet or per carton. Esko ArtiosCAD supplies some example cost centers in Intelligent Layout Defaults, and you are expected to modify these and add your own cost centers to suit your needs.

To view or edit cost centers, do the following.

1. Start Esko ArtiosCAD.
2. Click Options and Defaults to open the Defaults dialog box.
3. Click the plus sign next to Intelligent Layout defaults to view its contents.
4. Double-click Cost centers. The first cost center will be selected by default.

The groups and options shown depend on the option button selected in the Cost specification method group.

5. Make the desired changes and then click OK to save them or Cancel to ignore them.
The items in the Cost center dialog are explained in the following list:

### Table: Cost Center Dialog Box

<table>
<thead>
<tr>
<th>Item</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>List of cost centers</td>
<td>The list of cost centers must be arranged in the order they are used. The output of one cost center is the input to the next. When the checkbox is off, the cost center is an option, available to be selected but is not turned on by default.</td>
</tr>
<tr>
<td>Up arrow, down arrow</td>
<td>Moves the selected cost center up or down in the list</td>
</tr>
<tr>
<td>Name</td>
<td>Changes the name of the selected cost center</td>
</tr>
<tr>
<td>Insert</td>
<td>Inserts a new cost center above the selected one.</td>
</tr>
<tr>
<td>Delete</td>
<td>Deletes the selected cost center</td>
</tr>
<tr>
<td>Per sheet</td>
<td>The cost center applies to sheets, for example a sheeter.</td>
</tr>
<tr>
<td>Printing press</td>
<td>Works the same as a sheet cost center, but identifies the printing press cost center to be reported separately.</td>
</tr>
<tr>
<td>Die press</td>
<td>Works the same as a sheet cost center, but identifies the die press cost center to be reported separately.</td>
</tr>
<tr>
<td>Per design</td>
<td>Cost center such as a folder-gluer, where make ready is needed for each different design, and a smaller make ready is needed for a different print item of the same design.</td>
</tr>
<tr>
<td>Per print item</td>
<td>Cost center such as windowing, if the windows were different on each print item. There is a separate make ready for each print item.</td>
</tr>
<tr>
<td>Use prompted variable</td>
<td>Only if a variable is to be associated with this cost center. For example FGD = folder-gluer degree of difficulty, or NCOL2 = number of colors for a second printing press, see note. The variable is added to the Prompted Variables tab of the Quantities and Costs dialog if this cost center is included.</td>
</tr>
<tr>
<td>Prompt</td>
<td>Prompt/description of this variable</td>
</tr>
<tr>
<td>Variable name</td>
<td>Name of the prompted variable to be used in expressions. A prompted variable can be used in expressions such as make ready and speed, inside the PVAR function. For example, the make ready time of the printing press is defined to be $1 + PVAR(NCOLOR)/2$, that is, 1 hour plus $1/2$ hour for each color. The name of the variable can use up to 6 letters and digits and must start with a letter.</td>
</tr>
<tr>
<td>Default value</td>
<td>Default value of the prompted variable</td>
</tr>
<tr>
<td>Make ready time</td>
<td>Expression for the make ready time in hours</td>
</tr>
<tr>
<td>Sheets/hour</td>
<td>The normal speed of the machine when it has reached its optimal speed.</td>
</tr>
</tbody>
</table>
The theory is that the machine runs for a while at a slower speed for the first few sheets while it is brought into alignment, and then the speed is increased to its optimal speed. For a multi-step ramp up in speed, use the SPEEDSTEP function.

Run waste %
The percent waste while running. A machine such as a printing press may periodically go out of alignment and waste some sheets.

Cost/hour
Cost of the machine per hour while running. Does not include the cost of the machine during make ready, which is separate.

Cost/sheet
Additional material cost per sheet apart from the board cost. If the cost specification method is per print item, this label changes to Cost/carton, for example the cost of a window in each carton.

Opens an expression configuration dialog box. See the following section.

Changing Cost Center expressions
Clicking the f(x) button in a cost center expression leads to an expression configuration dialog box. For example, clicking f(x) for the Make ready time expression:

(\text{STEP} (#N\text{COLOR}, 1, 2, 1.25, 3, 1.5, 4, 1.75, 5, 2.25, 6, 2.75, 7, 3, 8, 3.5))

leads to:

This dialog box lets you choose how the value for the selected field is determined - by a table of conditions or by an expression; in this example, the method is set to table. Only when the method is set to table is there a Table tab in the dialog box.
To set the value using an expression, select the Expression option and type the expression in the field. Click the f(x) button to access the Edit Expression dialog box.

Creating a table to set the value

When configuring a cost center expression, you can create a table of values to use when certain conditions are met. In the picture below, the make-ready time is determined by the number of one-ups and the board caliper.

![Table example](image)

The One parameter and Two parameters option buttons control how many conditions must be met when setting a value. When One parameter is selected, everything relating to columns is unavailable; you may only add and delete rows.

The Row parameter and Column parameter drop-down list boxes set what is being used to determine the conditions of the table. You may type expressions of your own choosing in this box. If Esko ArtiosCAD recognizes the expression (CAL, for example), it translates the expression to its text equivalent (“Board caliper”).

Add row, Add column, Delete row, and Delete Column do what they say. There can be up to 18 rows and 14 columns. If the cursor is in any of the edit boxes, Delete row deletes that row; otherwise the bottom row is deleted. Delete Column deletes the column the cursor is in; otherwise, it deletes the rightmost column.
The f(x) button for the Sheets/hour field leads to an expression configuration dialog box which has three entries which behave exactly as those described above.

Modifiers for cost center expressions

A modifier allows adjustments to be made to an expression in certain situations. For example, a modifier could use this logic: if the caliper is 0.016” or less, add 10%. Modifiers can do the following to the output of the expression:

- Add a percentage
- Subtract a percentage
- Add a value
- Subtract a value
- Multiply by a value
- Divide by a value
- Set a minimum value
- Set a maximum value

To add a modifier to a cost center expression, do the following:

1. Click the f(x) button for that expression and then click the Modifiers tab.

2. Click Add to add a modifier. A list of modifiers appears:
3. Choose a modifier to add to the expression (for this example, choose Large caliper) and click OK. Esko ArtiosCAD will return to the Modifiers dialog box.

4. Click Edit to open the Edit Modifier dialog box.
The option buttons in the Condition type group let you quickly change the condition with a single mouse click. If you select No condition, the Limit value and Condition fields are unavailable. Changing the condition type option button automatically updates the expression in the Condition field. The Condition type and the Limit value combine into the Condition.

The option buttons in the Modification group determine the operation of the modifier. The amount of the modification is set in the Modification value field.

5. Click OK to accept the changes to the modifier or Cancel to ignore them. Esko ArtiosCAD will return to the expression configuration dialog box.

Notes on Intelligent Layout Defaults

Most defaults have the option of User defaults overriding Shared defaults. This does not work well for Intelligent Layout because the cost centers from Shared Defaults and User Defaults are mixed together. We recommend that you only use Shared Defaults for Intelligent Layout.

The order of the cost centers is important. The Per design and Per print item cost centers must be after the printing press and the die press.

If you set up a cost center with a prompted variable that has a default value, for example a printing press with 2 colors, and then want to run a layout with three colors, your first impulse might be to double-pass the sheet and use the cost center twice. This approach, however, will result in 4 colors being used in the calculations. Make sure to set up a cost center with prompted variables that have defaults specific for each case you may encounter.

Overview: Setting up a multi-color printing press cost center using a table

The following overview illustrates how to set up a cost center containing variable information. We are going to set up the following information:

<table>
<thead>
<tr>
<th>Number of colors</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Make ready time in hours</td>
<td>1.5</td>
<td>1.75</td>
<td>2.0</td>
<td>2.5</td>
<td>2.75</td>
<td>3.5</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Note that you must enter time in hours, not hours:minutes. For example, 2 hours 45 minutes is represented as 2.75 hours.

1. Start Esko ArtiosCAD.
2. Click Options and then click Defaults to open the Defaults dialog box.
3. Click the plus sign next to Intelligent Layout defaults to view its contents.
4. Double-click Cost centers. The first cost center will be selected by default.
5. Click Insert to create a new cost center called New cost center.
6. Change the name in the Name: field from New cost center to My 8 color press.
7. Set the Cost specification method to Printing press.

The Intelligent Layout Defaults dialog should look as follows:
8. Click f(x) at the end of the Make ready time: field to set up a table of conditions for the number of colors. The expression configuration dialog box appears:

9. Click the Table option button in the Make ready time group.
10. Click the Table tab.

11. Select One parameter and choose Number of colors for the Row parameter.
12. Enter 2 and 1.5 in the first and second fields, respectively, under **Number of colors**. Esko ArtiosCAD will change the 1.5 into 1+1/2.

13. Click **Add row**. Enter 3 and 1.75 in the first and second fields of the new row.

14. Repeat for the rest of the values in the table.

15. When the table is complete it should look like this:

<table>
<thead>
<tr>
<th>Number of colors</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1+1/2</td>
</tr>
<tr>
<td>3</td>
<td>1-3/4</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
</tr>
<tr>
<td>5</td>
<td>2-1/2</td>
</tr>
<tr>
<td>6</td>
<td>2-3/4</td>
</tr>
<tr>
<td>7</td>
<td>3-1/2</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
</tr>
</tbody>
</table>

16. Click **OK** to finish configuring the table for the make ready time, and proceed to configuring the rest of the cost center.

**Useful Functions for Cost Center Expressions**

**STEP(variable, value1, limit1, value2, limit2,..., limitn, valuen)**

- Returns value1 when variable < limit1
- value2 when limit1 <= variable < limit2
- value3 when limit2 <= variable < limit3
- ...
- valuen when limitn <= variable

The STEP function and all its variations must have an even number of parameters, at least 4.
Note that we would not recommend that a step function be used for the speed of a machine because the run time of a machine using a step function like this would have discontinuities in its run time:

**STEPUP**(variable, value1, limit1, value2, limit2, ..., limitn, valuen)

Same as STEP except < and <= are reversed:

Returns: value1 when variable <= limit1
value2 when limit1 < variable <= limit2
...
valuen when limitn < variable

**LINESTEP**(variable, value1, limit1, value2, limit2, ..., limitn, valuen)

As a more sensible alternative to the STEP function for the speed of a machine, this function does a linear interpolation between data points, and avoids the discontinuities in the STEP function.

Returns: value1 when variable <= limit1
Interpolation between value1 and value2 when limit1 <= variable <= limit2
Interpolation between value2 and value3 when limit2 <= variable <= limit3
...
valuen when variable >= limitn
Note that to prevent a discontinuity at the last limit, the last two values should be the same, or the last value might be an expression.

**SPEEDSTEP(#RUN,speed1,run1,speed2,run2, ..., runn, speedn)**

The SPEEDSTEP function is an alternative for the speed of a machine (depends on how the speed is specified). The machine runs at speed1 for the first run1 sheets, runs at speed speed2 up to a run length of run2, and so on, and runs at speed speedn after the run length reaches runn. The function returns an average speed of the machine given the run length.

### List of functions for configuring cost centers

This list of functions is accessible from the Cost Centers dialog box by clicking f(x) and then Insert Library Function.

**Table: Functions for configuring cost centers**

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>#SHTSIZEX</td>
<td>Sheet size X, current units (inches or mm).</td>
</tr>
<tr>
<td>#SHTSIZEY</td>
<td>Sheet size Y, current units (inches or mm).</td>
</tr>
<tr>
<td>#SHTSIZEWG</td>
<td>Sheet size with grain, current units (inches or mm).</td>
</tr>
<tr>
<td>#SHTSIZEXG</td>
<td>Sheet size cross grain, current units (inches or mm).</td>
</tr>
<tr>
<td>#CUTSIZEX</td>
<td>Knife-to-knife X of designs in the layout, in current units (inches or mm).</td>
</tr>
<tr>
<td>#CUTSIZEY</td>
<td>Knife-to-knife Y of designs in the layout, in current units (inches or mm).</td>
</tr>
<tr>
<td>#MANSIZEX</td>
<td>Design blank size X (manufacturing lines), in current units (inches or mm)</td>
</tr>
<tr>
<td>#MANSIZEY</td>
<td>Design blank size Y (manufacturing lines), in current units (inches or mm)</td>
</tr>
<tr>
<td>#MANSIZEWG</td>
<td>Design blank size with grain (manufacturing lines), in current units (inches or mm)</td>
</tr>
<tr>
<td>#MANSIZEXG</td>
<td>Design blank size cross grain (manufacturing lines), in current units (inches or mm)</td>
</tr>
<tr>
<td>#AREA</td>
<td>Area of design ignoring holes (zero if area fails) in current units (square inches or square mm)</td>
</tr>
<tr>
<td>Function</td>
<td>Description</td>
</tr>
<tr>
<td>-----------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>#AREAH</td>
<td>Area of design minus holes (zero if area fails) in current units (square inches or square mm) of the current design. If sheet cost center, of the first design.</td>
</tr>
<tr>
<td>#RUN</td>
<td>(In the Intelligent Layout Parameters dialog). Run length for this cost center/item in number of sheets or number of cartons.</td>
</tr>
<tr>
<td></td>
<td>(In a form or OCX control). Run length of the first cost center, usually the printing press.</td>
</tr>
<tr>
<td>#NUPD</td>
<td>(Design, item cost center) How many of this design on the sheet (Sheet cost center) 0</td>
</tr>
<tr>
<td>#NUPP</td>
<td>(Item cost center) How many of this print item on the sheet (Sheet or design cost center) 0</td>
</tr>
<tr>
<td>#NUP</td>
<td>Total number of designs on the sheet</td>
</tr>
<tr>
<td>#MKRT</td>
<td>Make ready time for this cost center/item</td>
</tr>
<tr>
<td>#MKRTA</td>
<td>(Design cost center) Make ready time for this cost center/item for additional items.</td>
</tr>
<tr>
<td></td>
<td>(Sheet or item cost center) 0.</td>
</tr>
<tr>
<td>#BRDWTG</td>
<td>Board weight grams/square meter.</td>
</tr>
<tr>
<td>#BRDWTLBS</td>
<td>Board weight lbs/1000 square feet</td>
</tr>
<tr>
<td>#BRDCSTM</td>
<td>Board cost/1000 square meters</td>
</tr>
<tr>
<td>#BRDCSTFT</td>
<td>Board cost/1000 square feet</td>
</tr>
<tr>
<td>#BRDCSTKG</td>
<td>Board cost per kg.</td>
</tr>
<tr>
<td>#BRDCSTLBS</td>
<td>Board cost per lb.</td>
</tr>
<tr>
<td>#WEIGHTKG</td>
<td>Weight of one item (board only) in kg.</td>
</tr>
<tr>
<td>#WEIGHTLBS</td>
<td>Weight of one item (board only) in lbs.</td>
</tr>
<tr>
<td>#ITEMORD</td>
<td>(Item cost center) Ordered quantity of this item.</td>
</tr>
<tr>
<td></td>
<td>(Sheet cost center) Ordered quantity of the first item.</td>
</tr>
<tr>
<td></td>
<td>(Design cost center) 0.</td>
</tr>
<tr>
<td>RULELEN(&quot;TYPE 1&quot;)</td>
<td>(Design or Item cost center) Length of cuts in the design.</td>
</tr>
<tr>
<td>Function</td>
<td>Description</td>
</tr>
<tr>
<td>-------------------</td>
<td>---------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>RULELEN(&quot;TYPE 2&quot;)</td>
<td>Length of creases in the layout. Also supports more general selection such as RULELEN(&quot;TYPE 1,3 RAD ..1/8&quot;)</td>
</tr>
<tr>
<td>PVAR(varname)</td>
<td>Value of prompted variable, e.g. PVAR(NCOLOR) or PVAR(CRW). Actual variable is CCEN.NCOLOR (per sheet) or CCEN.D[i].CRW (per design) or CCEN.P[i].MYVAR (per print item)</td>
</tr>
<tr>
<td>DESVAR(varname)</td>
<td>(Design or item cost center) Value of variable from design, e.g. DESVAR(L) (Sheet cost center) Value of variable from the first design. (If variable does not exist) 0</td>
</tr>
<tr>
<td>#NCOLOR</td>
<td>Number of colors for the printing press set in the Ink Coverage page.</td>
</tr>
<tr>
<td>#LBSKG</td>
<td>Number of pounds in a kilogram = 0.45359237</td>
</tr>
<tr>
<td>#KGSSTON</td>
<td>Number of kilograms in a short ton = 907.18474</td>
</tr>
<tr>
<td>#KGSFLTTON</td>
<td>Number of kilograms in a long ton = 1016.0469088</td>
</tr>
<tr>
<td>#SQMSQFT</td>
<td>Number of square meters in a square foot = 0.09290304</td>
</tr>
<tr>
<td>#LBSFTG</td>
<td>Number of (lbs/1000 sq. ft) in (g/sq. m) = 4.882427636383</td>
</tr>
</tbody>
</table>

**Example Folder-Gluer Formula**

To calculate the speed of a folder-gluer machine, suppose the belts on the machine run at 240,000 inches per hour. The cartons are carried through the folder-gluer side-by-side with a 1/2-inch gap between them. The blank size of a carton is given by the function #MANSIZEX and 1/2 inch is represented by #I/2, so the step distance between cartons is #MANSIZEX+#I/2. Dividing the speed of the belts by the step distance gives a speed (in cartons per hour) of:

\[
240000 \times \frac{#I}{(#MANSIZEX+#I/2)}
\]

However, suppose some cartons are more difficult to fold up and the machine has to run slower for these. Let us introduce a compensation factor in a prompted variable Folder difficulty which has values:

1 = Easy, Full speed 240000 inches/hour  
2 = Moderate, Moderate speed 200000 inches/hour  
3 = Hard, Slow speed 160000 inches/hour

To allow for this, the modified formula uses a step function of the Folder difficulty variable FGD (Folder Gluer Difficulty):

\[
\text{STEP}(\text{PVAR}(\text{FGD}),240000,2;200000,3,160000)*\frac{#I}{(#MANSIZEX+#I/2)}
\]

To use this, the estimator should enter the “Folder difficulty” for each design along with the number of colors for the job in the Prompted Variables tab in the Quantities and Costs dialog box:
Run Length Calculation

The costing model used in Esko ArtiosCAD has a list of cost centers in the order that the cartons are processed through the plant. Once the sheet has been processed in the die press, you no longer have sheets, and the remaining cost centers process print items (=cartons). Therefore, the sheet cost centers (printing press and die press) come before all the item cost centers (e.g. windowing, folder-gluer, shipping). Between the last sheet cost center (the die press), and the first item cost center, the sheet is divided up into print items depending on the number up of each item.

For the run length calculation, each cost center has an attribute of the number of sheets or cartons used in make ready, and a run-time waste percentage. For example, suppose you start with 100000 sheets, and the printing press uses 5000 for make ready, and has a run time waste of 4%, because it periodically goes out of registration. Then the run length of the printing press is 100000-5000 = 95000, but because of the run-time waste the output from the printing press that is available for the next cost center is 96% of this, which equals 91200.

For each cost center in the list, the input to this cost center (= make ready + run length) is equal to the output of the previous cost center. When you enter ordered quantities, the calculation starts with the required delivered quantity which is the output from the last cost center, and works backwards, adding the run-time waste and make ready for each cost center, to come up with the number of sheets you need to start with.

To illustrate the run length calculation in Intelligent Layout, suppose you have ordered quantities 100000 of A and 200000 of B, and the sheet has 5 up of A and 8 up of B. The first thing we see is that with these proportions, we are going to get a larger overrun of A than of B. Suppose there was no make ready waste. There would be a run length of 12500 giving the required 200000 (100%) of B but 125000 (125%) of A. Therefore, for the run length calculation, we can ignore A with the larger overrun, and calculate the run length needed for B.

In our simple example, there are 3 cost centers:
Cost Center Name | Make Ready | Run Time Waste %
--- | --- | ---
Printing Press | 5000 sheets | 4%
Die Press | 3000 sheets | 2%
Folder-gluer | 500 cartons | 0%

This can be represented in a diagram as follows:

![Diagram of process flow]

Working backwards, the delivered quantity of B is 200000. This is the output from the folder-gluer. The folder-gluer needs 500 cartons make ready, so the input to the folder-gluer is:

run length = 200000, make ready = 500, total = 200500

There are 8 of B on the sheet, so

Die Press output = 200500/8 = (rounded up) 25063.

To allow for the 2% run time waste, the input to the Die Press is:
run length = 25063 * 100 / (100-2) = 25564, make ready = 3000, total = 28564

To allow for the 4% run time waste, the input to the Printing Press is:
run length = 28564 * 100 / (100-4) = 29754, make ready = 5000, total = 34754

Therefore, the total number of sheets needed for this job is 34754.

**Print Items and Folder-Gluer Make Ready**

If you have several print items of the same design, you only need to set up the folder-gluer for the first item. The make ready for subsequent items is just the time needed to move them with the forklift and restart the machine. However, which is the “first item”? If the run lengths of the print items are different, Intelligent Layout picks the item with the largest overrun, and assumes that this is the “first item” you will use to set up the folder-gluer.
Configuring Outputs

Esko ArtiosCAD provides many ways to configure Outputs to get the results your company’s workflow requires.

The tabs shown on an Output properties dialog box depend on the selection in the Output Type group. For example, in the picture above, there is no Reports tab, but if you set the Output Type to Plot, the samplmaking-related tabs disappear and the Report tab appears.

Note: Do not use a backward slash (\) in any Output name.
Note:
There are many sample Outputs in the Example Defaults catalog, accessed by clicking one of the panes in Defaults, then clicking File > Open > Example Defaults. For example, to see how to configure ArtiosCAD to make a sample with Re-board®, you could look at the Kongsberg XL44 - V-notch sample cutting entry under Outputs > Artios > Kongsberg XL44 - Corrugated.

Output Type tab

Output Type group
The choice in the Output Type group of the Output Type page determines which other tabs are available. In the above example, because the Output Type is set to Sample, there are tabs in the dialog box which only appear for samples.

When the Output Type is set to Plot, the tabs which appear only for plots become visible.

The selection in the Output Type group also determines what will be on the Output Type page itself. For example, when the Output Type is set to Grouped Output, the Scale group disappears from the page.

Shown In group
The Shown In group determines how this Output will be used - in the Output menu, the Send menu, the Export menu, or in a Multiple Workspace output or for use in WebCenter. At least one of these choices must be selected.
The Export menu is meant to be used to group commonly-used file exports together. The Send menu is meant to be used to group together those Outputs which will be emailed. A Multiple Workspace output lets the same Output be used on a group of workspaces both in Esko ArtiosCAD and in the Database browsers.

When the WebCenter checkbox is checked, this Output can be used for downloading files from WebCenter. This is meant only for file outputs such as DDES, DXF, and EPSF. Outputs for WebCenter must have unique names; there may not be an Output used with WebCenter with the same name in both Shared Defaults and User Defaults.

Scale group

The options in the Scale group determine the size of the output.

To fit one page modifies the size of the output so that it fits on one page.

Specify lets you set the size of the output. Setting a scale of one makes a full-sized output. Tiles, which are individual sections of the design, are printed if necessary.

Differential Scale makes the Vertical Scale field appear and changes the Scale field to Horizontal Scale.

Preview control group

The options in the Preview Control group control how and if the Output Preview dialog box appears when running the Output.

Auto Preview, when checked, causes the Output Preview dialog box to automatically open when running the Output. The options in the Layout Type and Window Size groups determine how the contents of the dialog box are shown. Shown below is the Output Preview dialog box.
Device tab

The Device tab lets you specify the hardware device (if applicable) to which the Output will be sent.
When entering this tab for the first time, you may see a dialog box informing you that Esko ArtiosCAD was unable to find a certain printer. Ignore this and click **OK** to proceed.

In the **Driver Type** group, the **Windows Driver** option instructs Esko ArtiosCAD to send the output information to the operating system mechanism of driving the device. The **CAM Driver** option instructs Esko ArtiosCAD to use its internal methods to drive the device. The selected option in this group determines how the rest of the tab looks. **Bitmap** creates a picture file of the output. **Workspace Layer** creates a new layer in the workspace and draws the output data in that.

**Note:** When outputting to a layer, you must re-perform the Output after each rebuild to maintain correct information in the layer.

**Out to file** sends the output data to a disc file instead of physically out to a device through a port or network connection. Some devices such as samplers make a network connection to the CAD workstation and pick up the file automatically.

**Number of copies** allows you to set the default number of copies made when performing this Output.

The value selected in the **High graphics memory use** drop-down list box determines how much memory Esko ArtiosCAD uses when outputting a bitmap image (typically a referenced PDF file when the PDF option has been purchased). This option is available only for Outputs of type Windows Driver or Bitmap.

A bitmap may have a maximum of 15 million pixels. The size of the bitmap is calculated dynamically during the output; system performance may be affected if a large bitmap is required. You can set the maximum number of pixels for the Output to use to balance system speed and output.

<table>
<thead>
<tr>
<th>Output Type</th>
<th>Device</th>
<th>Reports</th>
</tr>
</thead>
<tbody>
<tr>
<td>Output Device</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Driver Type</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Windows Driver</td>
<td></td>
<td></td>
</tr>
<tr>
<td>CAM Driver</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bitmap</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Workspace Layer</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| Windows Driver | | |
| LOCAL Canon iR330-400 PS Ver 1.0 | |
| | |

| Orientation | | |
| Portrait | | |
| Landscape | | |
| Optimum | | |

| Number of Copies | | |
| 1 | | |

| High graphics memory use | | |
| Upto 15 million pixels | | |
resolution. There are four choices: **High graphics mode off**, **Upto 5 million pixels**, **Upto 10 million pixels**, and **Upto 15 million pixels**. The fewer the number of pixels used, the coarser the resolution of the output.

**Windows driver**

When **Windows Driver** is selected, there is a drop-down list box allowing you to choose the device that will perform the output. Devices must be set up in the operating system beforehand for Esko ArtiosCAD to list them here. You can also choose the orientation of the output on the page by choosing the appropriate choice in the Orientation group: **Portrait**, which makes the long side of the paper the vertical axis, **Landscape**, which makes the long side of the paper the horizontal axis, or **Optimum**, which sets the orientation based on which one can output at the largest scale.

You can also set the **High graphics memory use** when using a Windows driver.

**CAM driver**

When **CAM driver** is selected, the Output Device group changes to look similar to the following picture, except no data will be shown in the various fields:

The **Tuning Entry** field indicates the type of device receiving the output data from Esko ArtiosCAD. **CAM driver** specifies the internal Esko ArtiosCAD method for creating the data. **Tuning Filename** lets you choose the configuration file for the device. **CAM Device Name** lets you set the Output Destination for the data if it is not going out to file. Refer to the **Peripherals** chapter for more information about installing CAM devices.
Bitmap outputs

The Bitmap driver type lets Esko ArtiosCAD output a bitmap picture of a workspace.

The option buttons in the Bitmap Format group set the type of bitmap created. JPEG output (Joint Photographic Experts Group) makes compressed pictures for efficient use in other programs, but some color information is lost. PNG output (Portable Network Graphic) is used for vector graphics (lines and arcs) shown on the World Wide Web. TIFF output (Tagged Image File Format) is uncompressed and is useful for other graphic programs such as Adobe Photoshop. Note, however, that TIFF files are usually quite large.

The Shrink size to remove white space checkbox, when checked, causes Esko ArtiosCAD to crop the extra white space around the perimeter of the design upon Output.

The JPEG Quality field adjusts the quality and the file size of the resulting JPG file. A value of 0 makes the smallest file with the worst quality, while a value of 100 makes the largest file with the best quality. Esko ArtiosCAD defaults to 70. Set the value in this field between 80 and 90 if you will be making Animations in 3D. This field is available only when the Bitmap Format is set to JPEG.

High graphics memory use determines how much memory Esko ArtiosCAD uses to render PDF graphics being processed by the Output.

Getting bitmap outputs just right can take some experimentation.
Output to layer

Workspace Layer writes the results of an Output to a layer in a design.

For example, you could define a Report output to go to a layer, and send the resulting file to a customer. The customer could then use the Esko ArtiosCAD Viewer to view both the design and your report.

The Automatically overwrite checkbox controls whether the layer will be overwritten each time the Output is performed. If the checkbox is cleared, a number will be appended to the layer name in a sequential fashion.

Note: When outputting to a layer, you must re-perform the Output after each rebuild. When a design is rebuilt, the layer that was output to is automatically deleted as the information that was in it does not reflect the current state of the workspace.

Outputs using high graphics mode

Outputs that support bitmaps can use high graphics mode. These must be actual Outputs, as opposed to using the Print command on the File menu.

A bitmap may have a maximum of 15 million pixels. The size of the bitmap is calculated dynamically during the output; system performance may be affected if a large bitmap is required. You can set the maximum number of pixels for the Output to use to balance system speed and output resolution. The pixel usage is configured in the High graphics memory usage drop-down list box in the Output method’s settings in Defaults, or may be configured each time in the Properties dialog box. There are four choices: High graphics mode off, Upto 5 million pixels, Upto 10 million pixels, and Upto 15 million pixels. The fewer the number of pixels used, the coarser the resolution of the output.

The memory usage for high graphics mode when using a 3D Output is fixed at 15 million pixels.

Shown below is the Properties dialog box for a Kongsberg CM1930 samplemaker with the Illustrator 2 printing head:
Shown below is the Plot to Printer dialog box.

Position tab

The settings on the Position tab indicate the physical placement of the output on the output medium.
Device Size and Margin groups

The fields in the Device Size group let you set the size of the device (if you are using a CAM driver; otherwise, the size fields are unavailable) and the starting position as measured from the bottom left corner of the medium.

The settings in the Margin group set the distance between the edge of the medium and the physical area that can be printed by the device. When you set the device size to 0 (for unlimited size), you can still set the margins.

Orientation and Position groups

The commands in the Orientation group let you specify the relative position of the output on the device.

The options in the Position group control the point to which the output is snapped on the device and the side of the design which is output. The nine buttons snap the design to their respective location on the page - top left corner, top center, top right corner, right center, bottom right corner, bottom center, bottom left corner, left center, and center.

The Side Up list box lets you choose which side of the design is output: the Inside, Outside, Die Knife side, Die back side, or the default side, which is the current side of the design.

Reports tab

The Reports tab lets you specify a report to output using the data in the design. Check the Use Report checkbox and choose the report to use.
If the desired Report is not in the Reports Catalog, use the Specify button to browse for it.

**Directories tab**

The **Directories** tab lets you specify if the output data is written to a disc file instead of going to the physical device. The **Out to file** checkbox on the **Device** tab must be checked in order for these options to work. The contents of this dialog box will vary depending on what Output type is selected on the Output Type tab. The examples shown below are for plots. If the Output Type were set to Sample output, there would be many more fields for the different kinds of files that sample output can generate.

**Make DOS Compatible** truncates the filename so that it conforms to the 8.3 file-naming convention used in MS-DOS. When this checkbox is checked, the **Strip front** and **Strip back** option buttons become available. These control which part of the filename is stripped off to make it eight characters.

**Automatically Overwrite** controls if you are prompted to overwrite (delete) an old output file with the same name as the output file you will create when performing the output. This option is usually checked by default in pre-defined Outputs that create files. If this option is checked for an Output that
is added to a Grouped Output, the current filename with the default extension is used. The current directory is also used; you are not prompted to select a directory. You might have to search for the file created by the Output as it may not be where you expect it. Use this option with care.

**Automatically Open** opens the file in the associated Windows application after being output. For example, if you output a .txt file, Windows would recognize it as a Notepad document and open it in Notepad.

**File groups**

The names and amount of data fields in this dialog box depends on the type of Output selected on the Output Type tab. When the Output type is set to plot, for example, there is one file group, **Plot File**, but when the Output type is set to Sample, there are four file groups - **Make Counter File**, **Partial Cuts File**, **Make Sample File**, and **Print Sample File**. In each group, however, the fields and checkboxes are all the same and work the same.

The **Extension** field lets you use the same filename extension, which is the part of the file name which follows the last period, each time this Output is performed. By default, the filename is the same as the workspace name. Leave this field blank to specify an extension when creating an output file.

The **Directory** field lets you type the location of the directory in which to store the output file. Use the ellipsis button to browse for a directory.

<table>
<thead>
<tr>
<th>Make Sample File</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extension: MAP</td>
</tr>
<tr>
<td>Directory: C:\CAM</td>
</tr>
</tbody>
</table>

The **Fixed filename** checkbox lets you specify the same filename for all files created using this Output. When this checkbox is on, and the directory specified in the Directory field is valid, the filename selection dialog box is suppressed.

<table>
<thead>
<tr>
<th>Make Sample File</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filename: job1.map</td>
</tr>
<tr>
<td>Directory: C:\CAM</td>
</tr>
</tbody>
</table>

**From command file** indicates that the filename is generated by a command file. When this option is checked, the output file name will be the result of executing the function `TLIBRARY(GenSName.txt)`.

Since **Fixed Filename** is not checked, the resulting filename is the default filename when the output is run.

<table>
<thead>
<tr>
<th>Make Sample File</th>
</tr>
</thead>
<tbody>
<tr>
<td>TLIBRARY GenSName.txt</td>
</tr>
<tr>
<td>Directory: C:\CAM</td>
</tr>
</tbody>
</table>
View tab

The options on the View tab of the Output dialog box control how the layers and line types are output.

Style Selection group

Specify Plotting Style overrides the plotting style at time of output with the plotting style specified in the Available Plotting Styles drop-down list box. The list box is available only when this option is checked. Checking Rule labels turns on rule labels for this Output.
Dimension Position group

Auto-adjust overlapping text prevents rogue text items from being printed on top of each other. You can specify the range of text sizes to consider when Esko ArtiosCAD looks for overlapping text by entering point values in the Minimum text size and Maximum text size fields.

The Remove extension lines checkbox controls whether extension lines are removed if text has to be moved to make it not overlap.

Override Dimension Format group

The options in the Override Dimension Format group allow you to override the workspace settings for the dimension format with those that you choose in this group. Set them as desired.

View Selection group

The options in the View Selection group control which layers are output when running the output.

Current zoomed in view Outputs the objects in the current zoomed in area. In this case the Layer set selection is set to Current View and the Available Layers list is unavailable.

Current selection uses the currently selected items as the Output. All non-selected items are ignored. If no items are selected, the Output defaults to the Current View Layer set selection.

The Use selection for report calculations checkbox is only available with Current selection and is only applicable if the Output uses a Report. This option forces the calculated text commands in the Report to be evaluated against a temporary document which contains a copy of the selected item.

Choose the layer to output from the Layer Set Selection drop-down list box.
Choosing a layer set from the drop-down list box toggles different layers in the Available Layers group. Select or deselect layers as desired. To choose specific layers without starting from a preconfigured layer set, select the User Defined layer set, and then select layers as desired.

**Note:** Do not select both the Designs layer and Rule Path layer for any Output.

The options in the Rotary Diemaking View Level group control how the size of the Output is manipulated when the Layer Set Selection is set to Dieboard. Design Level, Wood Level, and Cylinder Level apply the appropriate shrink or stretch factors to the output data so that the output is sized correctly.

**Processing tab**

The commands on the Processing tab control how the data is manipulated between when you click Output and when the data arrives at the output device or disc file.
Note: The options on the Processing tab depend on the type of output chosen on the Output Type tab. Different outputs have different options.

The Browse button at the end of the CAM Tooling Setup field lets you choose the CAM Tooling Setup to use in this Output. The CAM Tooling Setup maps which line types are mapped to which physical tools on the output device. Click it to choose an entry from the Select Cam Tooling dialog box.

Optimization options group

Depending on the Output Type chosen, items in this group may be unavailable.

**Optimization options**
- **No optimization** makes no changes to the output data.
- **Plotter Pen Optimization** examines each 100 lines of output code, groups together lines made with the same pen, outputs them, and then examines the next 100 lines.
Enable subroutined output means support subroutines in the NC (numeric control) output, which only works with the PUNCH command and certain SAVE commands. Turning this checkbox off combines all elements of the design or manufacturing file into one set of data on output.

Enable subroutined output also works and should normally be checked for CFF2-Save and DDES-Save and Elcede exports. For a driver that supports this option, checking this checkbox causes each different design to be output as a subroutine, with a call to that subroutine for each instance of the design on the sheet. This saves memory in the controller for an NC output, and differentiates each design from the layout in DDES and CFF2 output. When this checkbox is clear, the entire layout is output flat, as if it were one big design. This option must be checked for the Layout Sequencing group to be available.

Line processing options group

The option buttons in this group determine how lines are physically output.

- **No processing** ignores bridges and outputs each special rule as a single line.
- **Design representation** means that special rules are output as “actual” lines; for example, a wave rule would be output as a series of arcs.
- **Partially-cut bridges** instructs the output device to cut into the material at a partial depth to make room for bridges in the rule.
- **Leave gaps for bridges** splits the line output data into pieces and leaves gaps in the output data for bridges, meaning that no slot or line is created in the output material where the bridges occur.
- **Software line styles** processes all line styles before they are sent to the driver. For example, when this option is selected, a perf would be broken up into a series of short cut lines and gaps instead of one line marked perf that the driver would then process. This option is available only for Plot outputs.

Layout Sequencing group

The options in this group are shown only when the output type is set to **Plot**, **CAM**, **Sample**, or **Counter** on the Output Type tab. Enable subroutined output in the Optimization options group must be turned on for this group to be available in **Plot** and **CAM** outputs.
The **Sequence one-ups in layout** checkbox determines if Esko ArtiosCAD automatically resequences the single designs in the layout when they are output. Clear this checkbox if you plan to always use the **Layout Manual Sequencing** tool with this Output.

The first group of four buttons sets the starting point for the sequencing. The first button starts the sequencing at the bottom left of the layout. The second button starts the sequencing at the bottom right of the layout. The third button starts the sequencing at the top left of the layout. The fourth button starts the sequencing at the top right of the layout.

The second group of four buttons controls the sequencing direction and is not available until a button in the first group is pressed.

The first button sequences each horizontal row, starting each row on the same side of the layout so that each row is sequenced in the same direction, as shown in the picture above, with the starting point at the bottom left.

The second button sequences each vertical column, starting each column on the same side of the layout so that each column is sequenced in the same direction.

The third button sequences each row, starting on opposite sides of the layout so that each row is sequenced in the opposite direction, forming a horizontal snake pattern.

The fourth button sequences each column, starting on opposite sides of the layout so that each column is sequenced in the opposite direction, forming a vertical snake pattern.

**Nicks group**

The **Draw nicks** checkbox controls whether nicks are drawn on the output, and the drop-down list box lets you choose the style to use from the Nick Styles Catalog.

**Rubber group**

The options in this group affect how rubber is shown on the Output.
If Fill - Color by is selected, the rubber is drawn filled with color; choose to color the rubber by Rubber type or Element number. Esko ArtiosCAD cycles through about 10 colors when drawing by element number. If Fill - Color by is not selected, the rubber is drawn as outlines.

To draw the element number inside each piece of rubber, select Show element numbers, and then set the Size: field (in points) and Italic checkbox as desired.

Dieboard group

The options in this group control how the dieboard is created for a relevant Output type.

Ignore die splits (full dieboard) makes a full dieboard regardless of any splits.

Undercut at die splits by instructs the output device to cut back each line leading up to a die split from the split by the amount specified. Bridge positions are correctly maintained.

Undercut at crease ends by instructs the output device to put a bridge of the size specified at each end of all creases. If a crease line is bridged, any cutback may not exceed the size of the bridge width. If this option is chosen, set the minimum size of the line that will be undercut in the Minimum crease length: field.

Text options group

Plotter font tells the output device to use its built-in font for rendering the text. Use this option for pen plotters or for a driver that supports simple text entities, but not fonts, such as CFF2, DDES, Elcede DieCAD, and DXF.

Line text converts the text to geometry, which is then affected by transformations on output. Line text is suitable for etching text on a die. This is particularly useful for Asian languages, because the Name tool in Manufacturing only supports ASCII characters. To use Line text on a die or stripping tool, add a text item in a suitable font to the die or stripping board layer, with no color, and use an output configured with Line text. A suitable font for line text should be simple, have parallel edges, and not be too thick.

Normal text leaves the text alone. Normal text will always be output so that it is readable, regardless of dieboard orientation. This is suitable for output formats that support fonts, such as all Windows drivers, EPSF, and bitmap outputs.
Text is always output right side up, for all three of the above options. The only way to output text backwards is to convert the text to lines in the design using the **Outline Text** or **Line Text** tools.

**Diesaw tab**

**Kongsberg DieSaw group**

This tab appears only when the output is to a Kongsberg DieSaw.

<table>
<thead>
<tr>
<th>Output Type</th>
<th>Device</th>
<th>Position</th>
<th>Director</th>
<th>View</th>
<th>Processing</th>
<th>DieSaw</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kongsberg DieSaw</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Full board depth</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Allowance reduction</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>No hole length</td>
<td>1/8</td>
<td>Length of leader</td>
<td>1/8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>One hole length</td>
<td>3/8</td>
<td>Length of leader</td>
<td>3/8</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Full board depth** is the thickness of the dieboard.

**Generate saw leader holes** controls whether Esko ArtiosCAD creates leader holes. **Leader holes** are drilled holes in the dieboard which allow the saw to position itself to start cutting in the proper position. They are made before any saw cuts.

**No leader holes in corners**, when checked, prevents Esko ArtiosCAD from automatically creating leader holes in corners. Leader holes are instead put on bridges of the lines that meet to form a corner.

**Adjust leader holes on bridges** allows Esko ArtiosCAD to move the leader holes further apart from each other if they occur at either end of a bridge. When this happens, the amount of material left between the two holes is not strong enough and could break; therefore, the centers of the leader holes are offset from the ends of the lines to prevent bridge breakage. The picture below illustrates this process.
The **Allowance reduction** specifies by how much the edge of the leader hole is offset from the maximum adjustment of the hole position. The maximum adjustment would position the edge of the leader hole tangent with the end of the rule path. The Allowance reduction is the distance by which the leader hole is moved away from that tangent position.

Lines shorter than the length specified in the **No hole length** field do not have leader holes drilled for them; rather, they have a lead-in line cut for them with a hole at the beginning of the lead-in line. The length of the lead-in line is specified in the **Length of lead-in** field.

Lines shorter than the value in the **One Hole Length** field (but longer than the value of the **No hole length** field) get only one leader hole drilled.

The value in the **Long saw length** field sets the minimum length of lines that are optimized for the quickest sawing time. For lines longer than the long saw length, the direction of the line and the leader hole is determined by optimizing the sawing time. For lines shorter than this value, the direction of the line is determined to use the minimum number of leader holes. Connected lines that are longer than this value will not share leader holes.
Tiling tab

The settings on the Tiling tab control what happens when the scale of the output is too big to fit on one sheet of output medium.

- **Registration**
  - Registration cross size: 0
  - Tile overlap: 0
  - Samplemaking registration offset: 0
  - Samplemaking registration hole diameter: 1/4
  - Just draw registration marks
  - Using sample counter

- **Output Filename Generation**
  - When outputting to file, insert the following before the extension:
    - Tile separator character: _
  - Tile representation:
    - Represent tile as number (1...) (on by default)
    - Represent tile as a character (A...) (off by default)
    - Use row/column position (1A, 1B ...)
  - Example filename: Design_4.txt

- **Skip empty tiles.**
- **Confirmation required for each tile, when outputting directly to a device.**

**Skip empty tiles** tells Esko ArtiosCAD to not output blank tiles. If you will be assembling all the tiles together into one big sheet, you may want to leave this option unchecked so as not to confuse the assembler.

**Confirmation required for each tile, when outputting directly to a device** tells Esko ArtiosCAD to prompt you to click OK before outputting each tile. This is handy for when you have to reload media on the output device in between tiles.
Tile registration group

The options in this group control how the tiles are marked for assembly.

<table>
<thead>
<tr>
<th>Tile Registration Options</th>
</tr>
</thead>
<tbody>
<tr>
<td>Registration cross size:</td>
</tr>
<tr>
<td>Tile overlap:</td>
</tr>
<tr>
<td>Samplemaking registration offset:</td>
</tr>
<tr>
<td>Samplemaking registration hole diameter:</td>
</tr>
<tr>
<td>Just draw registration marks:</td>
</tr>
<tr>
<td>Using sample counter:</td>
</tr>
</tbody>
</table>

Registration cross size is the overall height and width of the cross mark. Setting this option to zero turns off the drawing of registration crosses in the output. Also, registration crosses are only drawn in the corner of tiles which align with other tiles; a registration mark will never be drawn in the bottom left corner of the first tile.

Tile overlap is the amount of overlap from one tile to the next. This value will be used for both the horizontal and vertical overlap. A value of zero means no overlap, and when used with registration crosses, turns the cross marks into corner marks.

Samplemaking registration hole diameter is only available for Sample outputs and is the diameter of the hole cut into the sample material to aid in registering oversized samples. The diameter value must be between zero and the size of the registration crosses. The hole is centered on the registration mark.

Using sample counter and Just draw registration marks are also only available for Sample outputs. These options indicate if a counter is being used when this Output is run. If Just draw registration marks is selected, then the Make Counter button on the main sample output dialog (what you see when you perform the Output) is replaced with a Draw Registration button. This button is unavailable if tiling is not required. Selecting this option will generate output files (using the Make Counter file designations) with just the tile registration marks drawn using a sample pen.

Output filename generation group

The options in the Output Filename Generation group automatically constructs a unique file name for each tile when the output is set to go out to file. When both conditions are met (Out to file and a scale larger than can fit on the output device), the options in this group change the name out of the output file to indicate the separate tiles.
Sample line types tab

You configure the line types and tools used in samplemaking on the Sample line types tab.

To configure how the line type is handled when performing this Output, select the line type in the Line type: group and choose the tool in the Samplemaking tool: group. There may be settings for the tool that appear in the bottom right corner; if there are, set them as desired.

As with other tabs, the information displayed on this tab is conditional; that is, the information requested on the right side of the tab depends on the selected item on the left side of the tab. What you see will not necessarily look exactly like the picture above.

To make nicks in the sample whose size based on the caliper of the board, enter a multiplier in the field next to + CAL °.
To cut stripping rules on the sample, check the **Cut stripping rules** checkbox. Stripping rules are cut outward from the design to avoid overcuts.

The **Size only** samplemaking tool is used to calculate the output size. Use it only when outputting to a layer.

When using Re-board®, samplers cannot cut small holes in the material and use a drill tool instead. Therefore, enter an appropriate value in **Drill holes smaller than diameter** for holes that are drilled and not cut. If you will be using drill holes, you should update the **Sample drill hole** line type on the Sample tab of your plotting style definition so that it uses a thicker solid line instead of the default.

**Text options group**

<table>
<thead>
<tr>
<th>Text options</th>
<th>Plotter font</th>
<th>Line text</th>
<th>Normal text</th>
</tr>
</thead>
</table>

**Plotter font** tells the output device to use its built-in font for rendering the text. Use this option for pen plotters or for a driver that supports simple text entities, but not fonts, such as CFF2, DDES, Elcede DieCAD, and DXF.

**Line text** converts the text to geometry, which is then affected by transformations on output. Line text is suitable for etching text on a die. This is particularly useful for Asian languages, because the **Name** tool in Manufacturing only supports ASCII characters. To use Line text on a die or stripping tool, add a text item in a suitable font to the die or stripping board layer, with no color, and use an output configured with Line text. A suitable font for line text should be simple, have parallel edges, and not be too thick.

**Normal text** leaves the text alone. Normal text will always be output so that it is readable, regardless of dieboard orientation. This is suitable for output formats that support fonts, such as all Windows drivers, EPSF, and bitmap outputs.

Text is always output right side up, for all three of the above options. The only way to output text backwards is to convert the text to lines in the design using the **Outline Text** or **Line Text** tools.

**Make nicks in sample group**

The **Make nicks in sample** checkbox controls if nicks are made when cutting a sample. Check the box to have the samplecutter make the nicks as defined in the design being output, or clear it to
ignore nicks. When the checkbox is checked, the Adjustment for nick overcut: field is available. This is the offset distance between the tip of the blade and the point where the blade exits the material.

Sample Sequencing tab

The settings on the Sample Sequencing tab control how some problematic scenarios are dealt with when making a sample, as well as controlling layout sequencing.

Split lines connecting corners and Automatically round corners may not both be selected.

Round slot method group

Cutting a sharp corner can tear the material being cut. The options in the Round slot method group control how sharp corners are cut. Sharp corners are identified using Tear Distance and Tear Angle.
If a corner is narrower than the distance defined for the Tear distance, it is a sharp corner. If a corner has an angle less than the angle specified in Tear angle, it is a sharp corner.

Option A, Continuous cut, cuts all the lines in the slot in one direction.

Option B, Reverse the last line, reverses the last line to cut into the slot.

Option C, Cut short lines last, cuts the long lines first and then the connecting lines.

**Overcut method group**

The mechanics of samplemaking require that the knife cuts a little past the end of a line. If the direction of the line is into the sample, this will leave a mark on the sample. On a sample table such as a Kongsberg with a vertical rear edge to the knife, this problem can be fixed by reversing these lines (but only where this does not conflict with the line order required for sharp corners).

In the Overcut method group, you can set the overcut method to reverse at perpendicular intersections (called T-junctions). You can also specify to Split lines connecting corners.
Automatically round corners group

Some samplers can automatically replace a corner with a radius, reducing the number of knife lifts.

![Automatically round corners](image)

The Radius: field sets the size of the radius. The shortest lines intersecting to form a corner must be at least the length specified in the Minimum length: field to replace a corner with a radius.

Over-crease group

Over-crease is when the ceasing wheel makes a continuous crease through gaps and on top of cuts. In the Over-crease group, specify the maximum sizes of gaps and cuts to run the creasing wheel over without it lifting.

![Maximum length of cut to over-crease](image)

Set the value in the Maximum length of cut to over-crease: field in the Over-crease group to set limits on what is output as one line and what is output as separate lines.

The value in the Maximum length of cut to over-crease: field sets a limit so that short cuts and creases are output as a sequence while long cuts and creases are output separately. Adjust this field as needed to result in a series of short cuts with one long crease.

Set the value in the Maximum gap to over-crease: field to the maximum size of the gap the creasing wheel will pass through without lifting.

Sample creases tab

The commands on this tab control how creases are made on the samplers making machine.
Single pass creasing and Double pass creasing control the number of times the creasing tool passes over the material to make a crease. Double pass with grain crease and Double pass cross grain crease only perform double pass creasing for creases in the specified direction, and use the options in the Double pass crease offset group.

Double pass reverse crease, Second tool for cross grain, and Second tool for thicker creases enable their namesake actions when checked.

The Crease cut back is the amount by which creases are shortened at both ends.

**Double pass crease offset group**

The options in this group are available only when Double pass creasing is selected.

The offset can be a fixed distance (in current units), a multiple of the caliper, or an allowance for the pointage, or any combination of the three.

If Double pass for reverse creases is selected, the same crease offset applies as for creases, but no counter channel is made for reverse creases.

A fixed offset is specified by checking the Fixed offset checkbox and entering the value. A multiple of the caliper is entered by checking the Multiple of caliper checkbox and entering the value. To allow different pointage creases to have different crease offsets, check the Allow for pointage checkbox.
and enter the width of the scoring wheel. This defaults to 0.028 inches = 2 points; this adds (pointage - value) to the crease offset.

**Sample Counter tab**

The options on this tab control how sample counters are made. For each type of crease in the Parameter for: group, you can set the type of crease channel as well as the widths, counter cut back, and the channel angle. Routed counters invoke conditional text by deleting the field for the channel angle and inserting fields for the routing tool widths. The grain angle tolerance is the same for all crease types.

**Print Sample Position tab**

The Print Sample Position tab lets you specify how to print sample graphics on your sample printer.
This tab contains the exact same controls as the **Position** tab.

**Send to tab**

The commands on the **Send to** tab let you send the results of the Output via email. The design file itself is not sent when an Output with this option enabled is run. You must have a MAPI-compliant mail program such as Microsoft Outlook installed on your computer for this to work. If you do not have a MAPI-compliant email program installed, you will receive error messages when you attempt any email-related functions.

<table>
<thead>
<tr>
<th>Tiling</th>
<th>Send to</th>
<th>Advanced</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="check" alt="Use email" /></td>
<td><img src="field" alt="To..." /></td>
<td><img src="check" alt="Attach to e-mail" /></td>
</tr>
<tr>
<td><img src="field" alt="E-mail options" /></td>
<td><img src="field" alt="Cc..." /></td>
<td><img src="checkbox" alt="Put contents in email" /></td>
</tr>
<tr>
<td><img src="field" alt="Subject" /></td>
<td><img src="checkbox" alt="Prompt for message" /></td>
<td><img src="checkbox" alt="Delete output disk file" /></td>
</tr>
</tbody>
</table>

The **Use email** checkbox turns on this feature.

The **To** and **Cc** buttons let you choose recipients from your mail program’s address book.

**Prompt for message** opens the message in your email program. If this option is not checked, the message is sent with no user intervention.

**Attach to email** sends the output as an attachment, while **Put contents in email** inserts the text of the output into the message body.

**Delete output disk file** deletes the file created by the Output after it is added to the email. To preserve the file, clear this checkbox.

**Advanced tab**

The commands on the **Advanced** tab are meant to be used by people with very specific needs.
Show Command for Diagnostics displays the instructions Esko ArtiosCAD follows when making the output. Esko ArtiosCAD then instructs you to click OK to perform the Output or Cancel to cancel it as shown below:

Use prompting report indicates that a special report is used with this Output. Prompting reports prompt you for workspace variables and do not produce a printed output. Once this report is run, the variables are copied into the workspace. This is useful for setting variables that you want to track in DataCenter.

Use post processing command file lets you specify the name of a command file that modifies the geometry in the :TEMPOUTPUT section. You should only use this feature if you have extensive knowledge of the INTERACT programming language. The command file you specify must not have any menu windows and must not crash.

Output through command file directs the output data through a command file specified in the EXE field with parameters specified in the entry box after OUTPT.

Kongsberg Interface group

Select Job Setup File prompts you to select a Material Setup File when running this Output. Material Setup files are usually configured when your Kongsberg sample table is installed. Shown below is the Specify A Kongsberg Job Setup File dialog box when the Output is performed.
Record Variable Definition

These records and variables are created in the Output that is acted upon by the command file you specify in the Output through command file checkbox. Use these records to get information from the design to the command file.

<table>
<thead>
<tr>
<th>Record</th>
<th>Variable</th>
<th>Sub-variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>&amp;OUTCM.</td>
<td>CMD$</td>
<td></td>
<td>the actual command that would be issued</td>
</tr>
<tr>
<td>DEBUG</td>
<td></td>
<td></td>
<td>1 if “Show Command for Diagnostics” is ON, else 0</td>
</tr>
<tr>
<td>CMDTY$</td>
<td></td>
<td></td>
<td>the command type “PLOT”, “PUNCH” or “SAVE”</td>
</tr>
<tr>
<td>CMDTYP</td>
<td></td>
<td></td>
<td>command type - if (OTPTYP = 0 or 2) = 0, else (OTPTYP = 1) = 0 for Make Counter, 1 = Make Sample, 2 = Partial Cuts, 3 = Print Sample</td>
</tr>
<tr>
<td>DRV$</td>
<td></td>
<td></td>
<td>the driver name e.g. “PRTMAN”, “HP”</td>
</tr>
<tr>
<td>Record</td>
<td>Variable</td>
<td>Sub-variable</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>----------</td>
<td>--------------</td>
<td>------------</td>
</tr>
<tr>
<td></td>
<td>DEV$</td>
<td></td>
<td>the device name - when going directly to the device - not defined when going out to a file</td>
</tr>
<tr>
<td></td>
<td>USRTX$</td>
<td></td>
<td>additional user defined command parameters</td>
</tr>
<tr>
<td></td>
<td>OPT$</td>
<td></td>
<td>driver “opt” flag - either “OPT” or empty (“”)</td>
</tr>
<tr>
<td></td>
<td>DRVOP$</td>
<td></td>
<td>additional driver options</td>
</tr>
<tr>
<td></td>
<td>KGBMF$</td>
<td></td>
<td>KGB Material file name - not defined if not used or empty</td>
</tr>
<tr>
<td></td>
<td>OUTFL$</td>
<td></td>
<td>the output file name - not defined when going directly to the device</td>
</tr>
<tr>
<td></td>
<td>TUNE$</td>
<td></td>
<td>the tuning file name</td>
</tr>
<tr>
<td></td>
<td>DVSZLX</td>
<td></td>
<td>the lower X of the device size (in current units)</td>
</tr>
<tr>
<td></td>
<td>DVSZLY</td>
<td></td>
<td>the lower Y of the device size (in current units)</td>
</tr>
<tr>
<td></td>
<td>DVSZUX</td>
<td></td>
<td>the upper X of the device size (in current units)</td>
</tr>
<tr>
<td></td>
<td>DVSZUY</td>
<td></td>
<td>the upper Y of the device size (in current units)</td>
</tr>
<tr>
<td></td>
<td>WINLX</td>
<td></td>
<td>the lower X of the window - not defined if not tiled output</td>
</tr>
<tr>
<td></td>
<td>WINLY</td>
<td></td>
<td>the lower Y of the window - not defined if not tiled output</td>
</tr>
<tr>
<td></td>
<td>WINUX</td>
<td></td>
<td>the upper X of the window - not defined if not tiled output</td>
</tr>
<tr>
<td></td>
<td>WINUY</td>
<td></td>
<td>the upper Y of the window - not defined if not tiled output</td>
</tr>
<tr>
<td></td>
<td>SEC$</td>
<td></td>
<td>the section to output e.g. “:TEMPOUTPUT” or “:TEMPOUTPUT:GRAPHOUT” (for print sample)</td>
</tr>
<tr>
<td></td>
<td>SCLSTR</td>
<td></td>
<td>scale or stretch - 1 for “SCALE” - 2 for “STRETCH”</td>
</tr>
<tr>
<td></td>
<td>XSCALE</td>
<td></td>
<td>x scale factor</td>
</tr>
<tr>
<td>Record</td>
<td>Variable</td>
<td>Sub-variable</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>----------</td>
<td>--------------</td>
<td>------------</td>
</tr>
<tr>
<td>YSCALE</td>
<td></td>
<td></td>
<td>y scale factor (set to XSCALE when SCLSTR = 1)</td>
</tr>
<tr>
<td>NEGX$</td>
<td></td>
<td></td>
<td>either “NEG X” or EMPTY (“”)</td>
</tr>
<tr>
<td>ROT</td>
<td></td>
<td></td>
<td>rotation in degrees</td>
</tr>
<tr>
<td>POSX</td>
<td></td>
<td></td>
<td>X position in current units</td>
</tr>
<tr>
<td>POSY</td>
<td></td>
<td></td>
<td>Y position in current units</td>
</tr>
<tr>
<td>LTYPS$</td>
<td></td>
<td></td>
<td>the parameters to the TYPE keyword e.g. “TYPE #SCUT, #SCREASE..#SRXCREASE NOGRAPH” or empty(“”)</td>
</tr>
<tr>
<td>STY$</td>
<td></td>
<td></td>
<td>the plotting style to use e.g. “STY artios1” or empty (“”)</td>
</tr>
<tr>
<td>TILEX</td>
<td></td>
<td></td>
<td>for tiling - the row of this tile</td>
</tr>
<tr>
<td>TILEY</td>
<td></td>
<td></td>
<td>for tiling - the column of this tile</td>
</tr>
<tr>
<td>&amp;OUTPT.</td>
<td></td>
<td></td>
<td>output properties</td>
</tr>
<tr>
<td>OTPTYP</td>
<td></td>
<td></td>
<td>the output type 0 = PLOT, 1 = SAMPLE or 2 = MFG</td>
</tr>
<tr>
<td>OTPNM$</td>
<td></td>
<td></td>
<td>the name of the output</td>
</tr>
<tr>
<td>USESCL</td>
<td></td>
<td></td>
<td>1 to use the scale given - else automatically calculate it</td>
</tr>
<tr>
<td>SCALE</td>
<td></td>
<td></td>
<td>the scale factor given</td>
</tr>
<tr>
<td>USESTR</td>
<td></td>
<td></td>
<td>1 to use “STRETCH” instead of “SCALE”</td>
</tr>
<tr>
<td>YSCALE</td>
<td></td>
<td></td>
<td>the y-scale factor given</td>
</tr>
<tr>
<td>USEFRM</td>
<td></td>
<td></td>
<td>1 - this output uses a form</td>
</tr>
<tr>
<td>FRMNMS</td>
<td></td>
<td></td>
<td>the name of the form</td>
</tr>
<tr>
<td>DRV.</td>
<td></td>
<td></td>
<td>the driver/positioning data</td>
</tr>
<tr>
<td></td>
<td>DRVTPY</td>
<td></td>
<td>0 - Windows driver, 1 = CAM driver</td>
</tr>
<tr>
<td></td>
<td>PRTR$</td>
<td></td>
<td>name of the windows printer</td>
</tr>
<tr>
<td></td>
<td>TOFIL</td>
<td></td>
<td>1 - when output is set to file</td>
</tr>
<tr>
<td></td>
<td>ORIENT</td>
<td></td>
<td>0=Portrait, 1=Landscape, 2=Optimum -only used for a windows driver always portrait for CAM</td>
</tr>
<tr>
<td></td>
<td>TNDES$</td>
<td></td>
<td>the description as read from TUNEFILES</td>
</tr>
<tr>
<td>Record</td>
<td>Variable</td>
<td>Sub-variable</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>----------</td>
<td>--------------</td>
<td>------------</td>
</tr>
<tr>
<td></td>
<td>TNFIL$</td>
<td></td>
<td>tuning file name</td>
</tr>
<tr>
<td></td>
<td>DRVNM$</td>
<td></td>
<td>the driver name as read from TUNEFILES e.g. HP</td>
</tr>
<tr>
<td></td>
<td>DVNAM$</td>
<td></td>
<td>name of CAM device for LP devices; e.g. HP1, DDES etc.</td>
</tr>
<tr>
<td></td>
<td>DROPT$</td>
<td></td>
<td>the CAM driver options</td>
</tr>
<tr>
<td></td>
<td>DEVSZX</td>
<td></td>
<td>CAM device size in X in current units</td>
</tr>
<tr>
<td></td>
<td>DEVSZY</td>
<td></td>
<td>CAM device size in Y in current units</td>
</tr>
<tr>
<td></td>
<td>MGTOP</td>
<td></td>
<td>CAM top margin in current units</td>
</tr>
<tr>
<td></td>
<td>MGBOT</td>
<td></td>
<td>CAM bottom margin in current units</td>
</tr>
<tr>
<td></td>
<td>MGLEFT</td>
<td></td>
<td>CAM left margin in current units</td>
</tr>
<tr>
<td></td>
<td>MGRGHT</td>
<td></td>
<td>CAM right margin in current units</td>
</tr>
<tr>
<td></td>
<td>OFFSTX</td>
<td></td>
<td>X offset from the justification position in current units</td>
</tr>
<tr>
<td></td>
<td>OFFSTY</td>
<td></td>
<td>Y offset from the justification position in current units</td>
</tr>
<tr>
<td></td>
<td>STARTX</td>
<td></td>
<td>X origin shift</td>
</tr>
<tr>
<td></td>
<td>STARTY</td>
<td></td>
<td>Y origin shift</td>
</tr>
<tr>
<td></td>
<td>ROT</td>
<td></td>
<td>Rotation flag: 0=rotate 0, 1=rotate 90, 2=rotate 180, 3=rotate -90</td>
</tr>
<tr>
<td></td>
<td>MIRROR</td>
<td></td>
<td>Mirror flag: 0=no mirror, 1=mirror horiz., 2=mirror vert</td>
</tr>
<tr>
<td></td>
<td>JUST</td>
<td></td>
<td>Justification flag: 0=top left, 1=top center, 1=top right, 3=center left, 4=center, 5=center right, 6=bottom left, 7=bottom center, 8=bottom right</td>
</tr>
<tr>
<td></td>
<td>SIDE</td>
<td></td>
<td>Side flag: 0 = PRINTEDSIDE, 1 = UNPRINTEDSIDE, 2 = default side</td>
</tr>
<tr>
<td></td>
<td>TILED</td>
<td></td>
<td>0 if we don’t need tiling</td>
</tr>
<tr>
<td></td>
<td>TILESX</td>
<td></td>
<td>number of tile rows</td>
</tr>
<tr>
<td></td>
<td>TILESY</td>
<td></td>
<td>number of tile columns</td>
</tr>
<tr>
<td></td>
<td>TLOVLP</td>
<td></td>
<td>tile overlap</td>
</tr>
<tr>
<td>Record</td>
<td>Variable</td>
<td>Sub-variable</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>----------</td>
<td>--------------</td>
<td>------------</td>
</tr>
<tr>
<td>PSDRV.</td>
<td></td>
<td></td>
<td>the optional print sample driver/positioning data</td>
</tr>
<tr>
<td></td>
<td>DRVTyp</td>
<td></td>
<td>0 = Windows driver, 1 = CAM driver, 2 = no printer</td>
</tr>
<tr>
<td></td>
<td>PRTR$</td>
<td></td>
<td>name of the windows printer</td>
</tr>
<tr>
<td></td>
<td>TOFIL</td>
<td></td>
<td>1 = when output is set to file</td>
</tr>
<tr>
<td></td>
<td>ORIENT</td>
<td></td>
<td>0 = Portrait, 1 = Landscape, 2 = Optimum (2 is only used for a Windows driver) always portrait for CAM</td>
</tr>
<tr>
<td></td>
<td>TNDES$</td>
<td></td>
<td>the description as read from TUNEFILES</td>
</tr>
<tr>
<td></td>
<td>TNFIL$</td>
<td></td>
<td>tuning file name</td>
</tr>
<tr>
<td></td>
<td>DRVNM$</td>
<td></td>
<td>the driver name as read from TUNEFILES e.g. HP</td>
</tr>
<tr>
<td></td>
<td>DVNAM$</td>
<td></td>
<td>name of CAM device for LP devices; e.g. HP1, DDES etc.</td>
</tr>
<tr>
<td></td>
<td>DROPT$</td>
<td></td>
<td>the CAM driver options</td>
</tr>
<tr>
<td></td>
<td>DEVSZX</td>
<td></td>
<td>CAM device size in X in current units</td>
</tr>
<tr>
<td></td>
<td>DEVSZY</td>
<td></td>
<td>CAM device size in Y in current units</td>
</tr>
<tr>
<td></td>
<td>MGTOP</td>
<td></td>
<td>CAM top margin in current units</td>
</tr>
<tr>
<td></td>
<td>MGBOT</td>
<td></td>
<td>CAM left margin in current units</td>
</tr>
<tr>
<td></td>
<td>MLEFT</td>
<td></td>
<td>CAM right margin in current units</td>
</tr>
<tr>
<td></td>
<td>MGRGHT</td>
<td></td>
<td>CAM right margin in current units</td>
</tr>
<tr>
<td></td>
<td>OFFSTX</td>
<td></td>
<td>X offset from the justification position in current units</td>
</tr>
<tr>
<td></td>
<td>OFFSTY</td>
<td></td>
<td>Y offset from the justification position in current units</td>
</tr>
<tr>
<td></td>
<td>STARTX</td>
<td></td>
<td>X origin shift</td>
</tr>
<tr>
<td></td>
<td>STARTY</td>
<td></td>
<td>Y origin shift</td>
</tr>
<tr>
<td></td>
<td>ROT</td>
<td></td>
<td>Rotation flag: 0 = rotate 0, 1 = rotate 90, 2 = rotate 180, 3 = rotate -90</td>
</tr>
<tr>
<td></td>
<td>MIRROR</td>
<td></td>
<td>Mirror flag: 0 = no mirror, 1 = mirror horiz., 2 = mirror vert</td>
</tr>
</tbody>
</table>
### Information Filter tab

The commands on the Information Filter tab let you select the Esko ArtiosCAD data included in the Output. The Output type must be set to *Information Filter* to make this tab appear.

<table>
<thead>
<tr>
<th>Record</th>
<th>Variable</th>
<th>Sub-variable</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>JUST</td>
<td></td>
<td>Justification flag: 0 = top left, 1 = top center, 1 = top right, 3 = center left, 4 = center, 5 = center right, 6 = bottom left, 7 = bottom center, 8 = bottom right</td>
</tr>
<tr>
<td></td>
<td>SIDE</td>
<td></td>
<td>Side flag: 0 = PRINTEDSIDE, 1 = UNPRINTEDSIDE, 2 = default side</td>
</tr>
</tbody>
</table>

**Note:** The settings on this tab supersede any view settings you have set in Esko ArtiosCAD when you select this Output. This means you may send information you do not mean to send, or vice-versa.
All layers must be turned on for the resulting workspace to be rebuildable. If all the layers are not turned on, the **Parametrics(Rebuild)** checkbox is cleared and cannot be checked until all layers are turned on. Click **All On** to turn on all the layers.

If the **Manufacturing parameter sets** checkbox is not selected, the Information Filter will remove manufacturing parameter set information from Manufacturing files and copies of parameter sets that are embedded in design files.

When using the Information Filter to strip prompted text variables from an Output, only those variables defined in the Prompted Text Catalog are stripped from the Output.

The output format for an information filter is usually an Esko ArtiosCAD workspace. The allowed formats for an information filter output are the current version of Esko ArtiosCAD, the major previous Esko ArtiosCAD releases, a **LASERPOINT IQ 3** or **4** workspace, and a **LASERPOINT IQ 2** workspace.

### Counter Tool Widths tab

All the fields which are unavailable in this tab in Defaults become available when performing the actual Output. The values for them are dependent upon the board used and parameter set. These fields are calculated automatically when this Output is invoked and cannot be changed during an Output.

The fields in the **Tool Widths** group are measured in current units and are enabled only when their respective checkboxes are checked.

When the **Multi-Pass** checkbox is off, the corresponding field is automatically set to the crease channel width and disabled. When the **Multi-Pass** checkbox is on, the edit box is enabled. Enter a tool width greater than zero and it will multi-pass with this tool.
For the Position Holes tool, the Allow for Tool Diameter checkbox follows the design lines when it is not selected, and allows for the width of the tool when the checkbox is selected.

If the specified tool width is too large to fit within the programmed lines, it will always cut something rather than nothing.

If the tool width of the crease tool is wider than the required crease channel it will cut with a single pass.

If the crease end shape is narrower than the crease end tool, it will cut a path that is the same distance from the crease end.

**Counter Cutting Order tab**

As the name implies, the tools on Counter Cutting Order tab control the direction in which the counter lines are made on the counter cutter.

The four option buttons in the Milling Direction group control the milling order.

The tools listed in the Tool Cutting Order group cut from the top of the list to the bottom of the list. To change the cutting order, select a tool to move, and then click the arrows to the right of the list.

The options in the Multi-Pass for Creases group control how the multiple passes for creases are made if that option is turned on. Optimize Tool Path makes a small rectangular crease channel out of four passes, while Account for Deflection makes two parallel passes in the same direction.
The value you set in the Milling Overlap field indicates the overlap the milling tools will use when the machine mills lines close to each other. This is to ensure all material is milled correctly.

**Grouped Outputs tab**

The Grouped Outputs tab, which appears only when the Output Type is set to Grouped Output, lets you group already-defined Outputs into one. For example, with one click, you could save the workspace, print a spec sheet, export a DDES file, send an email containing a DXF file, and cut a sample.

If any of the component Outputs you select go out to a file, they do not need to be configured to automatically overwrite the file if necessary; Esko ArtiosCAD will prompt for a filename when necessary.

To add an Output to this Grouped Output, click Add and select it from the list of Outputs; to delete it, click Delete.

Also, the Scale group on the Output Type tab is unavailable for Grouped Outputs as the scales are set for each component Output.

The Outputs listed are processed in top-down order. To change the position of an Output, select it, and then move it by using the arrows at the side of the list box.

Save before performing outputs uses the current name of the workspace if one is defined. If it is not defined, Esko ArtiosCAD prompts for the information needed. Save-As before performing outputs lets you select a new folder or resource as well as a new filename. Esko ArtiosCAD works with the new file as soon as it is saved, so any Outputs in this grouped Output will use the new file.
In both cases, any dialog boxes normally opened by the save process will be opened during the Output, such as the Database Information dialog box. Use caution when using the Save or Save-As options with CAD-X as it may lead to unpredictable results.

Both Save before performing outputs and Save-As before performing outputs may be checked at the same time to allow saving the workspace with its original name and then saving it as a new name.

Grouped Outputs can also be added to a compressed archive in ZIP format; select Zip in the Grouped Output Type group and Insert into Zip archive in the E-mail/Zip options group.

Delete output disk file deletes the working files that the Grouped Output creates after inserting them into the ZIP archive. In this example, those would be the DXF and DDES files.

As the E-mail and Zip Grouped Output types are mutually exclusive, in order to e-mail a zipped archive, you have to create a Zip Grouped Output first and then include that in an E-mail Grouped Output.

E-mail options group

A Grouped Output can also have its own top level email output (configured on the Send to tab) in addition to each individual Output going to email.
No change to destination indicates that the selected individual Output runs without any changes.

Attach to e-mail intercepts the output file for this component Output and embeds it as part of the group mail output. For example, if the output is to a Kongsberg samplemaker, the .MAP file will be attached to the email message instead of being sent to the table.

Put contents in e-mail means take the contents of a text file created by this component Output and put it in the e-mail message rather than attaching the file. It is intended for inserting text such as ordering information in the message. This should only be used with outputs through command files which generate text files.

Delete output disk file deletes any temporary files needed to add the output to an email message.

Select which sample output group

This group is conditional; it appears only when a sample output is selected in the list of Outputs.

Select the option button for the type of sample output you wish to create.

PDF Security tab

The PDF security options available in an Esko ArtiosCAD Output correspond to those used in Adobe Acrobat.
Checking **Require a password to open the document** requires the recipient of the file to enter a password when opening the PDF file using Adobe Acrobat Viewer. Enter this password in the **Open password** and **Repeat password** fields; they must match exactly. There is no way to recover this password if it is lost or forgotten.

Checking **Use a password to restrict permissions** requires the recipient of the file to enter a password to change how the file is protected. Enter this password in the **Open password** and **Repeat password** fields; they must match exactly. There is no way to recover this password if it is lost or forgotten.

If both password options are used, the two sets of passwords must be different.

**Note:** If you forget a password, change the password in Defaults, save the changed Defaults, and then regenerate the PDF document. It will use the new password.

In the **Printing allowed** drop-down list box, there are three settings: **None**, **Low Resolution**, and **High Resolution**. **None** means the recipient may not print the document. **Low Resolution** allows printing
as a bitmap up to 150 dpi. **High Resolution** allows printing the document at any resolution, and to use high-quality vector output and other advanced printing features if the printer supports them.

In the **Changes allowed** drop-down list box, there are three choices: **None**, **Page content changes only**, and **All changes allowed**. **None** allows no changes in Adobe Acrobat. **Page content changes only** restricts changes to what is shown on the page, but prevents other changes such as page rotation. **All changes allowed** allows any change.

The recipient of a PDF file created by Esko ArtiosCAD with security options set must use Adobe Acrobat 5.0 or later to open the file.

**Note:** Esko ArtiosCAD cannot open PDF files with an Open password, nor can it open PDF files that have restricted permissions and that do not have content copying enabled.

### Configuring a PDF Output using Adobe® Acrobat® 5

To configure an Esko ArtiosCAD Output that creates a PDF file using Adobe Acrobat 5, do the following:

1. Install Adobe Acrobat 5 as a custom installation, making sure to install the *Acrobat PDFWriter*.
2. Start Esko ArtiosCAD, open Defaults, and create a new Plot output using the procedures described earlier in the chapter.
3. In the **Scale** group on the **Output Type** tab, click **To fit one page**.
4. On the **Device** tab, set the **Driver Type** to **Windows Driver** and select **Acrobat PDFWriter** in the **Windows Driver** drop-down list box. Also set the **Orientation** to **Optimum**. The orientation in Esko ArtiosCAD works independently from the orientation in the printer driver; you may have to adjust the settings manually to achieve the desired output.
5. In the **Plot File** group on the **Directories** tab, set the extension to `.pdf`. Make sure to include the period before pdf.
6. Click **OK** and save the defaults.

Later versions of Adobe Acrobat will not work correctly with Esko ArtiosCAD.

**Note:** Esko ArtiosCAD also has a predefined PDF export and Output that use its native PDF method without requiring Adobe Acrobat 5 to be loaded on the system. These require no special configuration for use, but the option must be purchased and installed for them to work. If you have Adobe Acrobat 5, you can make PDF files in the method described above without having to purchase the PDF option in Esko ArtiosCAD. However, without purchasing the PDF option, you will not be able to import PDF files or use high graphics mode.

### Configuring 3D Outputs

3D has its own table in Defaults for Outputs. Shown below is the Artios catalog in **Shared Defaults > Outputs-3D**.
For detailed information about the options available in these Outputs, see 3D Outputs in the Esko ArtiosCAD User Guide.

Creating a new 3D Output

To create a new 3D Output, do the following:

1. Start Esko ArtiosCAD and click Options > Defaults.
2. In the Shared defaults pane, scroll down to the Outputs-3D catalog, right-click it, click New, and then click Data on the context menu.
3. Replace New Data 1 with the name of the new Output.
4. Press enter when you have entered the name and then double-click the new entry to access its Properties dialog box.
5. On the Output Type tab, choose the type of Output to make. Plot can either make a PDF file (if you have purchased that option), an Illustrator 8.0-compatible EPSF file, or the data can be sent to a device using a Windows printer driver. Bitmap makes a JPEG or PNG file. VRML, SolidWorks Export, and U3D all export to those file types. Animation Snapshots makes a series of bitmaps images of an animation and can output those to Microsoft Word, Microsoft PowerPoint, or PDF document, or to individual bitmap files. For this example, choose Bitmap.
6. Click the Bitmap tab.
7. In the **Save bitmap as** group, choose **JPEG**. Set the other options as desired.

8. Click the **General** tab and enter a directory in the **Output directory**: field. Set the other options as desired.

9. Click **OK** to finish defining the Output.

10. Click **File** > **Save**, and then click **Yes** to confirm overwriting shared Defaults.

11. The new Output in 3D is ready to use.

There are some slight differences between the options on the tabs in Defaults and those in the Outputs at time of use. For example, the **Use OpenGL for bitmaps** checkbox on the Bitmap tab when using an Output is set in the 3D Rendering Options entry in Startup defaults, not the Output defaults.

There is a fifth type of 3D Output available; **Animation Snapshots** exports each frame to individual PNG or JPEG files. For example, if you have an 8-frame animation, an Output of this type would produce 8 .JPEG or .PNG files in the same directory. When configuring an Output of this type, on the Animation tab with **Export animation as**: set to **Bitmaps**, the **Bitmap Filename Generation** group appears. The options in this group configure how the different filenames are named. The root of each filename is the workspace name, and then the **Frame separator character** is inserted after the name, and either a number or letter is added as a suffix depending on the option chosen.
Changing Collada defaults

Using the Collada entry in the Outputs-3D catalog in Defaults creates a .DAE file with .PNG images at the current graphics resolution.

To change the settings, do the following:

1. Start ArtiosCAD.
2. Click Options > Defaults.
3. In the Shared Defaults pane, scroll down to the Outputs-3D catalog and open it by clicking the plus sign (+) next to it.
4. Double-click the Collada entry.
5. Click the Collada Options tab.
   - In the Graphics resolution group, Use View Settings uses the resolution set in the View Mode dialog box. Deselect it to enable the drop-down list box that sets the resolution setting manually.
   - PNG and JPEG in the Include bitmaps as group control how the images are stored. Select JPEG to enable the quality setting.
   - In the Save as group, choose to save Collada files as standalone .DAE files or .ZAE archive files.
6. Click OK when done setting the options to return to Defaults.
7. Click File > Save to save the changes.
8. Click Yes to overwrite the existing Defaults.
Note:
You may want to create additional Collada entries with different options set for ease of output. Make sure to set the Output Type to Collada on the Output Type tab of the dialog box.

Rule labels

You can define rule labels for special rules that appear in the design when viewing the design, or when printing the design, or both. Rule labels display the specific rule type as the generic rule type plus whatever label you specify.

To configure this, you must add a rule label to the rule definition in the Special Rule Types catalog in Defaults.

Enclosing the fraction in braces causes Esko ArtiosCAD to display it as a real fraction, such as $\frac{1}{4}$.

To turn on rule labels when working in Esko ArtiosCAD, open the View Mode dialog box, select the Rule labels checkbox, and click OK.

Shown below is a hanger hole made from $\frac{1}{4}$ by $\frac{1}{4}$ perf with rule labels turned off.
Here is the same hanger hole with rule labels turned on:

To turn on rule labels as a function of an output, check the Rule Labels checkbox on the View tab of the output's properties page.

The defaults for the size and default appearance of the rule labels are stored in the Rule Labels entry in Design defaults.
The **Size**: of the rule label is in points. The value in the **Space between labels** controls how often along the line the label is repeated. **Rotate parallel to line** determines if the label is parallel to the rule or always shown so that it is easily readable. **Place above line** controls if the label is in the line or above the line. **Draw generic type** controls if ArtiosCAD is shown.

Shown below is the same hanger hole with **Rotate parallel to line** and **Place above line** turned off.
Configuring the Output Scale Table

To add or remove values from the Output Scale Table, do the following:

1. Start Esko ArtiosCAD and click **Options > Defaults**.
2. Open the **Startup Defaults** catalog by clicking the plus sign preceding it (+) and double-click **Output Scale Table**. It should look similar to the picture below:

3. To add a value to the table, click **Add** and enter the value in the **Scale value** field. The value you enter must be at least 0.005 units of measurement different from any other value.
4. To delete a value from the table, select the value and click **Delete**. There must always be at least one value in the table; the last value can not be deleted.

**Notes on using the Output Scale Table**

When using the Output Scale Table, the scale that is actually used is stored in the **Plotting Scale** variable in the General Information calculated text catalog. Insert this item of calculated text on the report to show the scale used.

If the report itself is scaled when it is output, that scale will be applied to the file window. In addition, all calculated text fields that use the Plotting Scale system variable (#FWSCALE) will be reevaluated using the combined file window scale and report scale.

**AutoSave / Local Temp Directory**

The **AutoSave** default in Startup defaults automatically saves a backup copy of open workspaces in the **Local Temp Directory** at the interval defined in the **AutoSave every** field. Esko ArtiosCAD AutoSaves only when the **Activate AutoSave** checkbox is selected.
The Local Temp Directory is configured in the Local Temp Directory default in Startup Defaults.

The system TEMP directory is defined by an environment variable. See your system administrator, or refer to your operating system documentation for more information on changing the system TEMP directory.

To set your own TEMP directory, select the Specify option and browse for the desired directory. Everyone must have write access to that directory. The Local Temp Directory must be on the current machine - it may not be set to a network drive.

The AutoSave timer is NOT active when:

- a dialog box is open
- when the Rebuild Playback tool is in use
- when the Interact window is open.

The AutoSave file is named BCKxxxxx.AAS, where xxxx is a random number, and AAS stands for ArtiosCAD Auto Save.

No database synchronization is done with the AutoSave file, nor is it saved in the database.

Recovering AutoSave files

The first time Esko ArtiosCAD starts after an improper termination, the AutoSave Recovery dialog box will appear listing the workspaces for which AutoSave files exist.
Check the checkboxes next to each file to recover and click **OK**. The AutoSave files will be restored for the selected files; **SAVE** the workspaces once they are open. AutoSave files that are not selected for restoration are deleted automatically.

### Revision Disk File Format in Revision History

The Revision History feature of Esko ArtiosCAD relies on changing the name of files as they are stored on the hard drive. How file name is changed is controlled by **Options > Defaults > Startup defaults > Revision Disk File Format**.

For example, consider a file called **FILE**. If you choose the first option, **Don’t put revision extension on latest revision**, and create a new revision, then the latest revision becomes **FILE** and the original becomes **FILE_~**. If you then make a new revision, then the latest file will still be **FILE**, the original will be **FILE_~**, and the first revision from the original will be **FILE_A**. In this case the latest **FILE** would also be **FILE_B**, but the _B extension will not be visible; if you open other revisions they will warn you that the latest revision is **FILE_B**.

If you choose the second option, **Don’t put revision extension on first revision**, then each new revision gets appended in sequence: the original is and will always be **FILE**, the second revision will
be FILE_A, the third will be FILE_B and so on. The most recent revision will be the one with the highest extension.

**Default to update all projects when saving revisions**, when checked, causes Esko ArtiosCAD to update all Projects that contain the current design when you use **Save As Revision**. When cleared, Esko ArtiosCAD updates only the current Project when you use **Save As Revision**. Also when this option is cleared, If the design is not in a Project, or the design is not in the active Project, no Projects are updated with information about this revision. This option is checked by default.

**Database Browser Reports**

This information is supplied for reference only. Only the most advanced users of Esko ArtiosCAD who are very familiar with XSLT and XML technology should consider modifying the database reports.

There are two types of Database browser reports - those used in the Esko ArtiosCAD browsers and those used in DataCenter Admin.

Esko ArtiosCAD Database browser reports are configured in the Database Browser Reports Catalog in Defaults.

There are two reports for each Esko ArtiosCAD browser. The **AutoCol Browser List format** automatically detects which data are selected for inclusion in Defaults and includes them on the report.

The **SimpleEmb Browser List format** presents just the information shown in the browsers.

The reports use three formats:

- **CSV** - produces a CSV (Comma Separated Values) file suitable for importing into a spreadsheet.
- **HTML** - makes an HTML (Hyper Text Markup Language) document which can be viewed in a Web browser.
- **XML** - makes an XML (eXtensible Markup Language) file suitable for delivering data to other applications in a rich, structured way. Only the Simple reports use this format.
Shown below is the Properties dialog box for the AutoColDesignListHTML report for illustrative purposes. For the reports with Simple in their name, any modifications made to the data included in the report require modifying the XSLT transformation file to put that data into the output file. The reports with Auto in the name automatically recognize the data included so the items in Select Data to Include can be added and removed as desired.

The option buttons in the Selection group control which database records are included in the report when it is output.

The Template File: field controls which XSLT template is used with the report. For the Esko ArtiosCAD browsers, the template files are stored in InstLib. For the DataCenter Admin browsers, the template files are stored in the language directory of the DataCenter program folder.

The options in the Output File group control where the output file is created and how it is treated after it is created.

The Select Data to Include group controls the data going into the raw XML file before it is translated into the output file by the XSLT file. If Thumbnail is selected, a JPG thumbnail for each record is saved in the same directory as the report file.

DataCenter Admin reports use hard-coded report names with extension .XSL that are in \Esko\Artios\DataCenter\Program\<language>.

The reports use XSLT files that are not meant for modification except by the most advanced users who are already familiar with XSLT and XML. Modifying the catalog entries in Defaults only modifies the information going into the raw XML file - the XSLT transformation template must also be updated to use the information. It is your responsibility to edit or create a new XSL template file used to format the data into the output file.
Publishing Esko ArtiosCAD documents to WebCenter

To configure Esko ArtiosCAD for publishing documents to WebCenter, double-click **Publish to WebCenter** in the WebCenter catalog in Shared Defaults.

**Note:** The WebCenter servers must be running version 6.1 or greater. Esko ArtiosCAD can publish to a single WebCenter site. Change the **WebCenter address** field in the Publish to WebCenter dialog box in Defaults as needed to publish to more than one WebCenter site.

Shown below is the default Publish to WebCenter dialog box.

![Publish to WebCenter dialog box](image)

**WebCenter Project Name group**

The options in the WebCenter Project Name group control which WebCenter Project receives the Esko ArtiosCAD document.

**Use document name** uses the Esko ArtiosCAD document name as the WebCenter Project name.

**Calculate using a library function** lets users with advanced Esko ArtiosCAD knowledge use a library function to determine the WebCenter Project name. Enter the name of the library function in the **TLIBRARY()** field.

**Use ArtiosCAD project name (ARD only)** names the WebCenter Project after the Esko ArtiosCAD project name if one was defined. This option overrides **Use document name** and **Calculate using a library function**.

**Prompt for project name** prompts for the Project name. If any Prompt option is selected, the Publish to WebCenter dialog box appears when publishing. They all must be deselected in order to hide that dialog box when publishing.
Create project if it does not exist creates the WebCenter Project named using the options selected in the previous fields if it does not already exist. If this option is deselected and the WebCenter Project does not exist, when Esko ArtiosCAD tries to publish a document into that WebCenter Project, an error will occur.

Use template lets you specify the template to use to create the Project. Enter its name in the Name field or enter the name of the library function used to calculate the name in the TLIBRARY() field.

Use document thumbnail causes WebCenter to use the thumbnail for the single design or 3D document as the Project thumbnail.

**Publish Options group**

If document already exists then publish as new version controls what happens when a document with the same name as the document being published already exists in the same folder in the WebCenter Project. When this option is selected, the document being published is treated as a new revision of the document already in the WebCenter Project. When this option is not selected and a document exists with the same name as one being published, an error will occur.

**WebCenter Folder Name group**

The options in the WebCenter Folder Name group control where in the WebCenter Project the Esko ArtiosCAD document is placed once it is published.

Publish to root level puts the Esko ArtiosCAD document in the default folder in the Project. Security on this folder is inherited from the WebCenter Project security settings.

Use standard ARD, A3D, MFG folders uses folders with those names for Esko ArtiosCAD documents with those filename extensions. Single designs would go into a folder called ARD, 3D documents into A3D, and manufacturing files into MFG.

Calculate using a library function lets users with advanced Esko ArtiosCAD knowledge use a library function to determine the folder name. Enter the name of the library function in the TLIBRARY() field.

Prompt for folder name prompts for the folder name. If any Prompt option is selected, the Publish to WebCenter dialog box appears when publishing. They all must be deselected in order to hide that dialog box when publishing.

Create folder if it does not exist creates the folder named using the options selected in the previous fields if it does not already exist. If this option is deselected and the folder does not exist, when Esko ArtiosCAD tries to publish a document into that folder, an error will occur.

**WebCenter Login group**

The WebCenter system administrator must provide a username and password for the account Esko ArtiosCAD will use to publish documents to WebCenter. Enter the username in the Username field. Prompt for password causes the Esko ArtiosCAD user to have to enter a password when publishing. Deselecting this option causes the Password field to become available. If you specify a username
and a password, they will be automatically entered in the Publish to WebCenter dialog box when publishing.

**WebCenter Connection group**

Enter the URL for the WebCenter server in the **WebCenter address** field.

**Note:** Esko ArtiosCAD may only publish to the WebCenter site listed in this field. To publish to another WebCenter site, change this field as needed.

**Test** tests the connection to the WebCenter server. The test is successful only if you receive a message that the WebCenter Web-API was properly installed. Any other response means the test failed.

**Graphic PDF file rotation import option**

A PDF file can contain an internal flag to indicate if the document is rotated or not. ArtiosCAD has ignored this setting until version 7.35. In **Options > Defaults > Shared Defaults > Startup defaults > PDF Options** is the **Honor Acrobat Rotation** checkbox. To have ArtiosCAD respect this flag when importing a PDF file as graphics, enable this checkbox. The checkbox is not selected by default, and has no effect for PDF files imported as lines.

**Setting CAPE/TOPS Defaults**

Before you use the palletizing features, you must set the CAPE/TOPS Defaults by doing the following:

1. Start ArtiosCAD.
2. Click **Options > Defaults**.
3. In the list of Shared defaults, scroll down to the Palletization catalog and open it by clicking the plus sign (+) next to it.
4. Double-click **CAPE/TOPS Defaults**.

The CAPE Defaults dialog box opens as shown below.
5. Set the options in the dialog box as desired.
   a) If you are using CAPE/TOPS in client/server mode, select Client/Server and enter the User ID for CAPE/TOPS in the field provided. If you are using it in workstation/standalone mode, leave this field empty.
b) Choose a default pallet from the drop-down list box in the Default Pallet group.
c) Set the Maximum pallet height and Maximum pallet weight appropriately using inches and pounds or millimeters and kilograms.
d) To not see the waste areas of the layout and see only the designs in CAPE/TOPS, check Output to CAPE/TOPS as stripped.
e) To calculate the caliper from the board elements instead of using the caliper defined by the board code, check Calculate True Caliper from Medium and Liners.
f) Set an Import pallet tolerance for use when exchanging design data between ArtiosCAD and CAPE/TOPS. As the outer dimensions of the CAPE/TOPS pack type and the ArtiosCAD standard may not match, the tolerance will make it easier to generate pallet loads.
g) In the Arrange/Design Options group, set the default amount of slack and the default maximum package size.
h) In the Number of primary packages group, set the minimum and maximum number of primary packages in the secondary package.
i) Set Round to nearest 1/16th as desired. This works in Inches mode only.
j) Click OK to return to Defaults when you are done setting the CAPE/TOPS Defaults.

6. Click File > Save to save the changes.
7. Click Yes to confirm the saving of the changes to Defaults and return to ArtiosCAD.

Setting DWG/DXF Options

**DWG/DXF Options** in the Startup defaults catalog is where you:

- set whether or not to use third-party import libraries
- map font names in DWG/DXF files to Windows TrueType fonts.
Import DXF files using third party libraries controls whether or not ArtiosCAD uses a more robust method to open DWG or DXF files than its built-in method. If you have trouble importing DWG or DXF files, try deselecting this checkbox.

ArtiosCAD does not display the fonts that DWG or DXF files use. Therefore, those fonts have to be mapped to the Windows TrueType (TTF) fonts that ArtiosCAD does display. In the default map shown above, **Width Factor** means the kerning and **Height Factor** adjusts the height of the text. To change any of the fields other than **Imported Font**, click in the field. ArtiosCAD maps any unmapped fonts using the *Default setting.*

If you have a DWG/DXF file containing unmapped fonts (text will be the wrong size), click **Get font definitions from file**, then browse for the file. ArtiosCAD scans the file and sets up new map entries.
with which you can then configure as desired. It then stores `FontMap.XML` in `ClientLIb` which you can copy between machines as needed.

**Note:** ArtiosCAD does not use font mapping if **Import DXF files using third party libraries** is not checked.