ArtiosCAD

User Guide

Contents

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For use with ArtiosCAD 12.0.10 or greater.

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2. Overview of Changes in ArtiosCAD 12.0.10

ArtiosCAD 12.0.10 changes existing features. Improvements include:

- Changes to imports and exports using the Spatial library in 3D
- Changes to Collada imports and exports in 3D
- Support for animations in U3D Outputs
- Support for squishing a container inside another in 3D
- Defaults for the Light Source tool in 3D
- Support for `CTRL-C` and `CTRL-V` copy and paste in 3D
- Easier samplemaking for labels
- Rule Length dialog box changes in Single Design
- Using spot colors in PDF files

Changes to 3D and 3D File Formats

This section covers changes to importing and exporting files into and from 3D. The changes are in 3D or Defaults.

Note: The example **SolidWorks export** Output in the Outputs-3D catalog has been removed. If you have upgraded from a previous version and have never modified the Output, the installation process removes it. You can easily create a new one as the SolidWorks output type still exists.

Importing Other Solids

3D Designer also can import many different 3D models from other programs. Collada and VRML come with 3D, but other formats require the 3D Importer option.

Table: Importable Solid File Formats

File Format	File Extensions	Versions Supported
ACIS	.sat, .sab, .asat, .asab	R1 - R23
CGM	.xcgm	R2012 - R2013
CATIA version 4	.model, .exp., session	4.1.9 - 4.2.4
CATIA version 5	.CATPart, .CATProduct	R6 - R22 (V5-6 R2012)
IGES	.igs, .iges	Up to 5.3
Inventor	.ipt	6 - 2013
	.iam	11 - 2013
Parasolid	.x_t, .xmt_txt., .x_b, .xmt_bin	10 - 25.0.155
ProEngineer	.prt, .prt., .asm, .asm.	16 - Creo 2.0

File Format	File Extensions	Versions Supported
Solidworks	.sldprt, .sldasm	98 - 2013
STEP	.stp, .step	AP203, AP214 (geometry only)
Unigraphics	.prt	11 to NX8, NX to NX 7.5

Importing non-VRML files

When you import an ACIS, STEP, IGES, CATIA, ProEngineer, or SolidWorks file, ArtiosCAD performs three steps before the workspace opens in the design window:

1. The file is preprocessed to convert the 3D data into polygons.
2. The polygons are written to file `<system temp directory>\TEMP.MSH.LOG`.
3. ArtiosCAD reads the temporary file and opens it in a 3D workspace.

SolidWorks files can contain multiple configurations of how to present the model. If opening such a file, ArtiosCAD chooses the default configuration.

SolidWorks assembly files (ending in **.sldasm**) may also contain references to other parts files (ending in **.sldprt**). If any parts file is missing, ArtiosCAD ignores the missing parts and continues the import, but will tell you the name of the missing files.

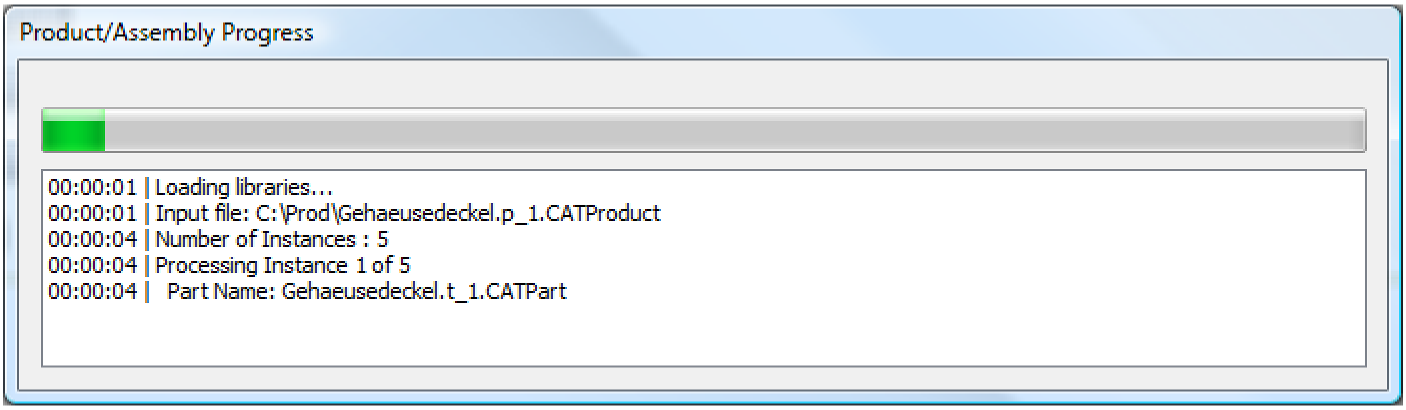
Opening large files of 50 to 100 MB or hundreds of thousands of polygons can take a lot of time (up to a half hour) depending on the capabilities of your computer. **Save the 3D workspace as soon as it opens**. If the import or save fails for any reason, try opening the `TEMP.MSH.LOG` file to avoid repeating the initial conversion. Files larger than 50 to 100 MB may fail due to lack of memory or too many polygons for the display adapter to process. To show the number of polygons, click **Help** > **Diagnostics** > **List Embedded Designs** .

Use the **Select Labels or Parts** tool to select parts of solids to change their position or properties, or to delete selected parts by selecting them and pressing `Delete` on the keyboard.

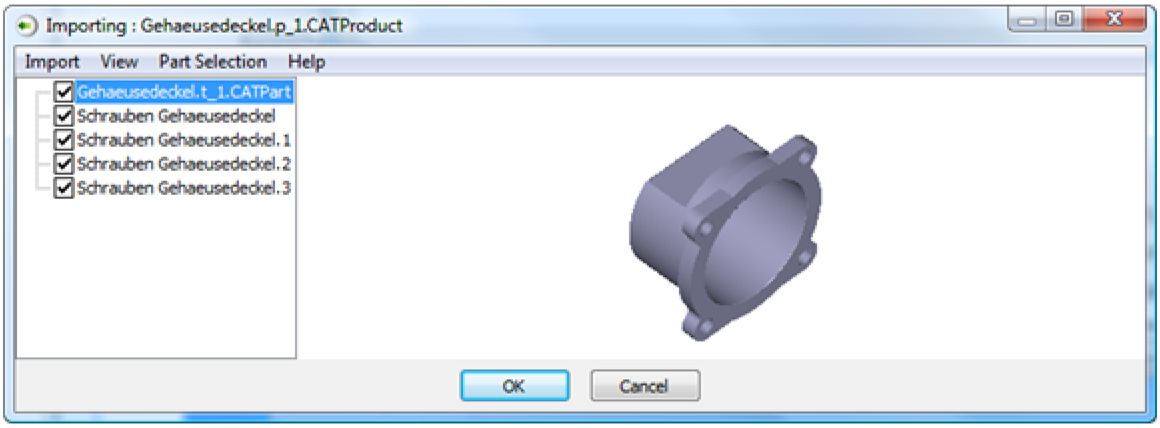
Importing a Solid

Do the following to import a solid:

1. Import the solid into an open 3D workspace, or open it directly. A progress bar appears.



2. A preview of the solid appears. If it has parts defined, you can turn them on and off with the checkboxes next to their names, or on the **Part Selection** menu. To change to a defined angle in the view, choose an option on the **View** menu. You can also use the left mouse button to drag and rotate the camera, the scroll wheel to zoom in and out, and the right mouse button to drag and move the camera. Changing the view in the preview does not change the initial view of the solid when it opens in ArtiosCAD.



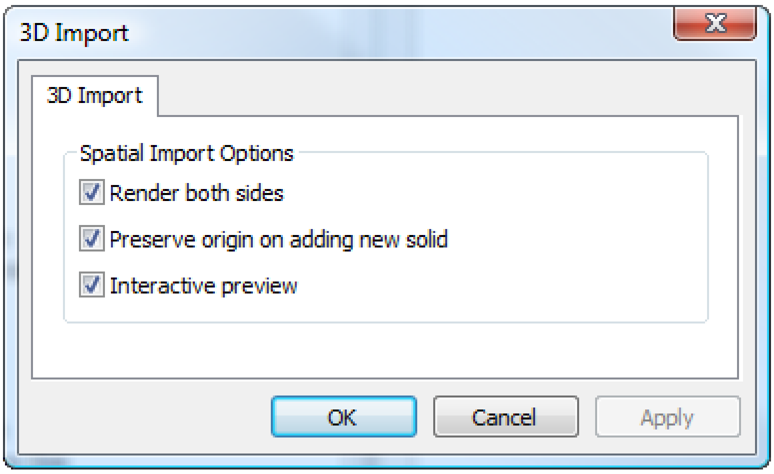
   If there is no preview, that means there is no data ArtiosCAD can break into triangles, such as 3D arcs or 3D points. Also, there is no preview for SolidWorks files.

3. Click **OK** to finish importing the solid.

If you get an error message about an unsupported format, it could be because there are accented characters in the filename. Rename the file and try again.

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.

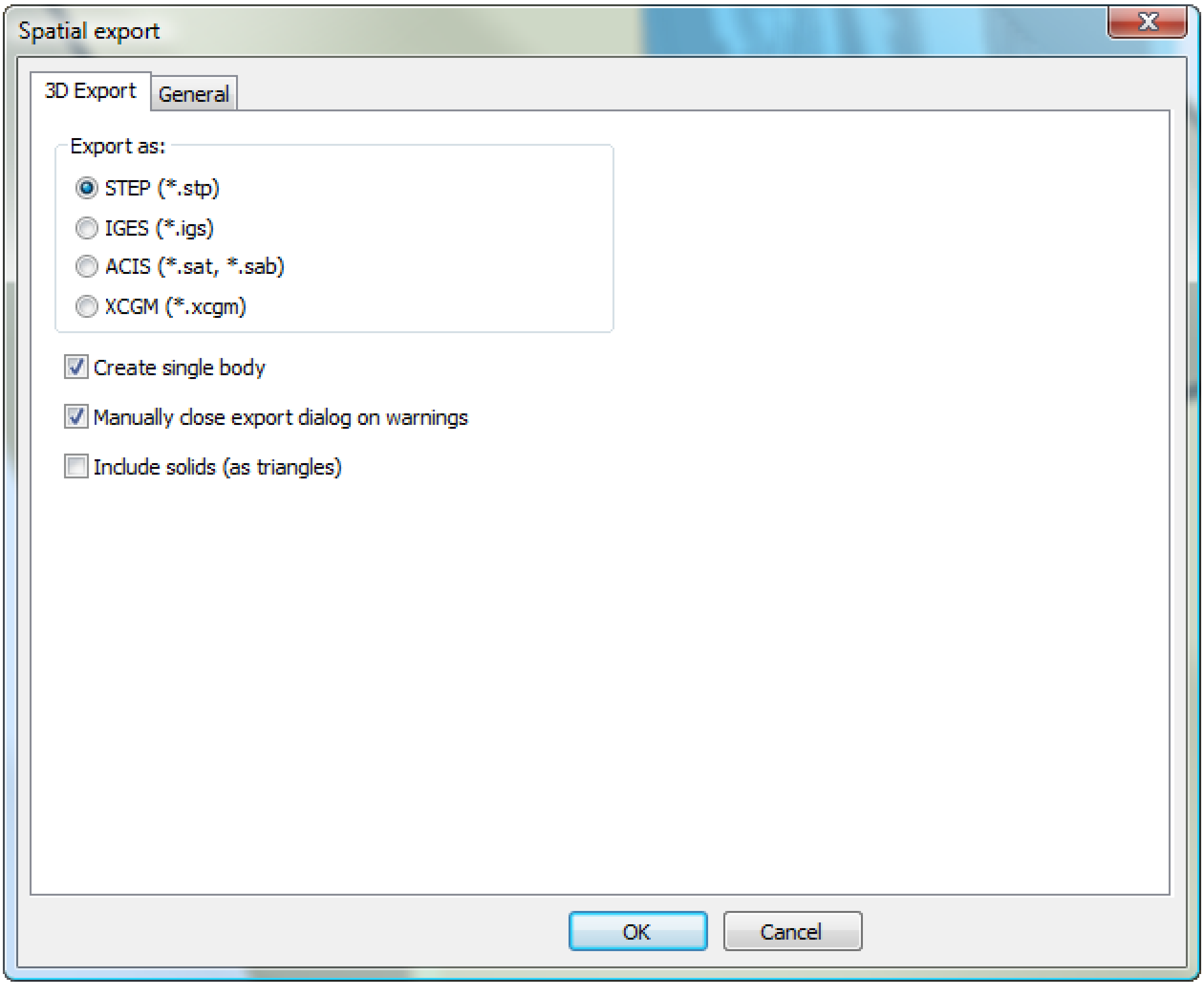
Click **OK** to make the changes and return to Defaults, or click **Cancel** to discard the changes.

Outputting a 3D Workspace as a STEP, IGES, ACIS, or XCGM File

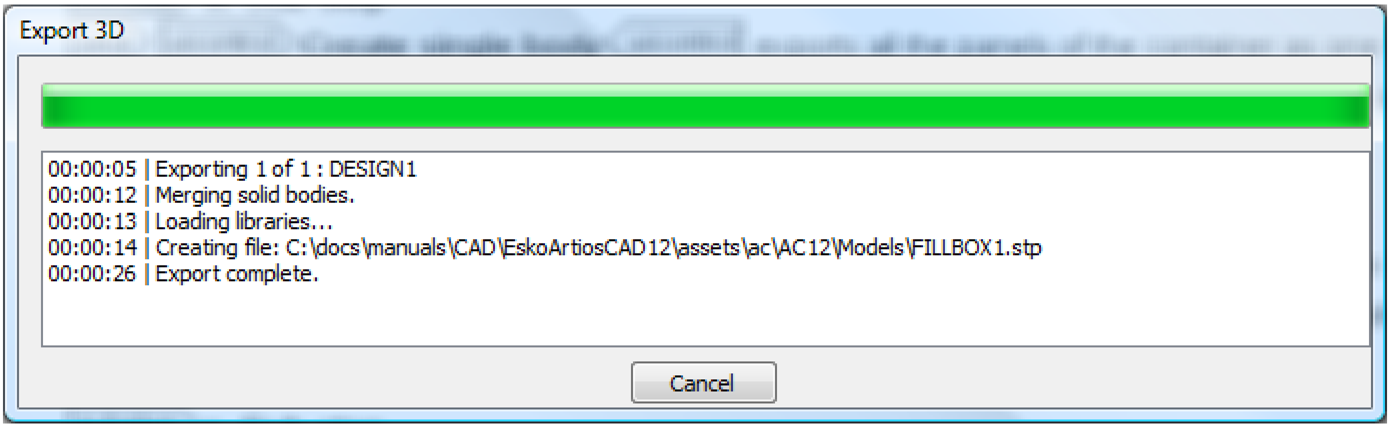
Use the **Spatial export** Output in 3D to export a 3D workspace as a STEP, IGES, ACIS, or XCGM file for use in other programs. This Output makes a solid body from each panel and crease of an ArtiosCAD design instead of thousands of triangles. If there are multiple parts in a workspace, each part is a separate body.

To perform the export, do the following:

1. Position the items in the 3D workspace as desired and save the workspace.
2. Click **File** > **Outputs-3D** > **Spatial export** .



3. Choose the desired file type in the Export as: group.
4. Set the checkboxes as desired:
   a) **Create single body** exports all the panels of the container as one solid body for easier use in other applications. When this checkbox is not checked, each individual panel is exported as a part.
   b) **Manually close export dialog on warnings** keeps the progress bar dialog box open if there are any warnings so you can see them. Click **Close** to close the dialog box.
   c) **Include solids (as triangles)** includes any solids in the 3D workspace in the Output as triangles. When this checkbox is not selected, ArtiosCAD only includes containers in the exported file. This checkbox is not available if there are no solids in the workspace.
5. Click **OK** to start the Output.
6. Set the filename and folder as desired and click **Save**.
7. ArtiosCAD displays a progress bar as it exports the file.



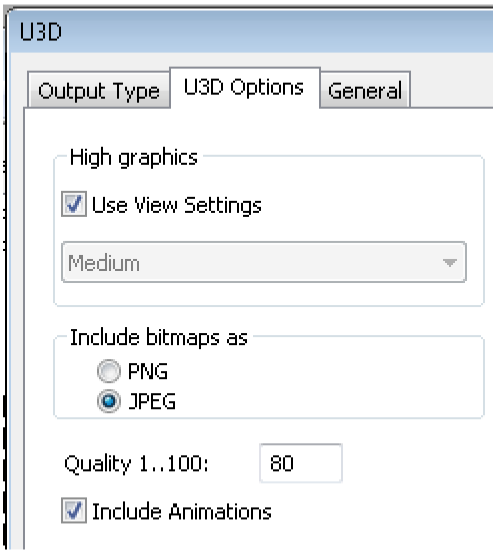
Spatial Export Notes

Notes about using the **Spatial export** 3D Output:

- ArtiosCAD includes no graphics in the exported file due to file format restrictions.
- There can be only one color per single design. ArtiosCAD sets the color to the outside color of the single design(s).
- Creases are represented by cylinders. When their ends are not flat, ArtiosCAD renders their ends with triangles for a more realistic appearance.
- Slots and indented perfs have square edges and are cut through completely.
- There is no representation for edge crushing.
- ArtiosCAD exports imported solids as triangles. More intricate solids may cause the export to take longer.

U3D Output Changes

ArtiosCAD now includes animations in exported U3D files. **Include Animations** on the U3D Options tab of a U3D Output is checked by default.



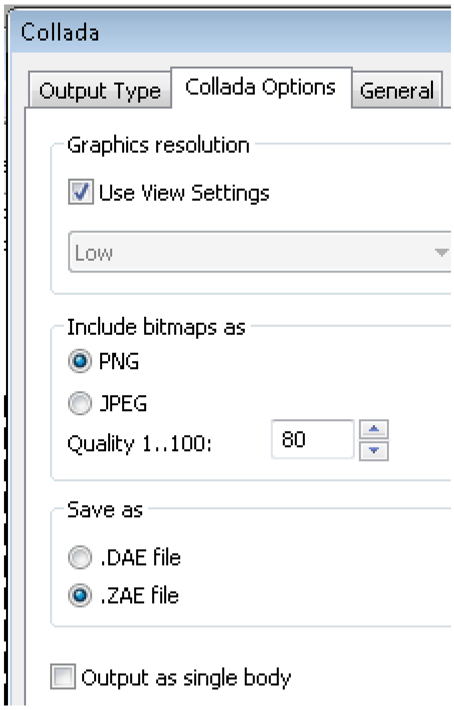
U3D Animation Notes

Some notes about animations in exported U3D files:

- ArtiosCAD will instruct Acrobat Reader to repeat the animation 1000 times because it does not recognize when you click **Play** after the animation finishes. This way, you may pause and resume the animation as desired. If you have a piece that appears or disappears at the end of the animation, make sure to add a frame afterward so that this action is visible.
- U3D animations showing moving bend panels will not exactly match the ArtiosCAD animations due to U3D format restrictions; however, they will be a close approximation.
- Surface smoothness in an exported U3D file can vary due to variances in the U3D algorithm.
- **Scale to Fit** is not supported.

Collada Changes

ArtiosCAD can now export 3D workspaces as single-body Collada files. On the Collada Options tab of a Collada 3D Output, **Output as single body** defaults to unchecked to preserve the established workflow, but you may check it as desired.



Selecting this checkbox turns off graphics in the exported Collada file as it may have one color and texture. ArtiosCAD will use the outside color for the body color.

Collada Notes

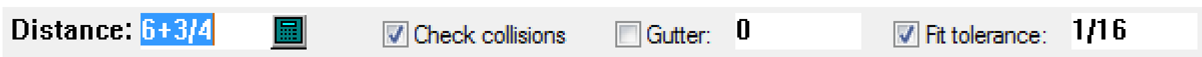
To preserve round-trip abilities between Esko applications, ArtiosCAD warns you when you try to adjust an object that originally came from a Collada file. When you import a Collada file, ArtiosCAD creates a link between the contents of its native workspace and the Collada file. If you then re-export that workspace as a .ZAE, ArtiosCAD bundles the original Collada file into the archive to carry over its information into the new file. Changing the properties of the imported objects in ArtiosCAD breaks that link, and that is the data loss referred to in the warning.

If the Collada file contains more than one object, ArtiosCAD groups the contents of the Collada file together upon import to keep them together.

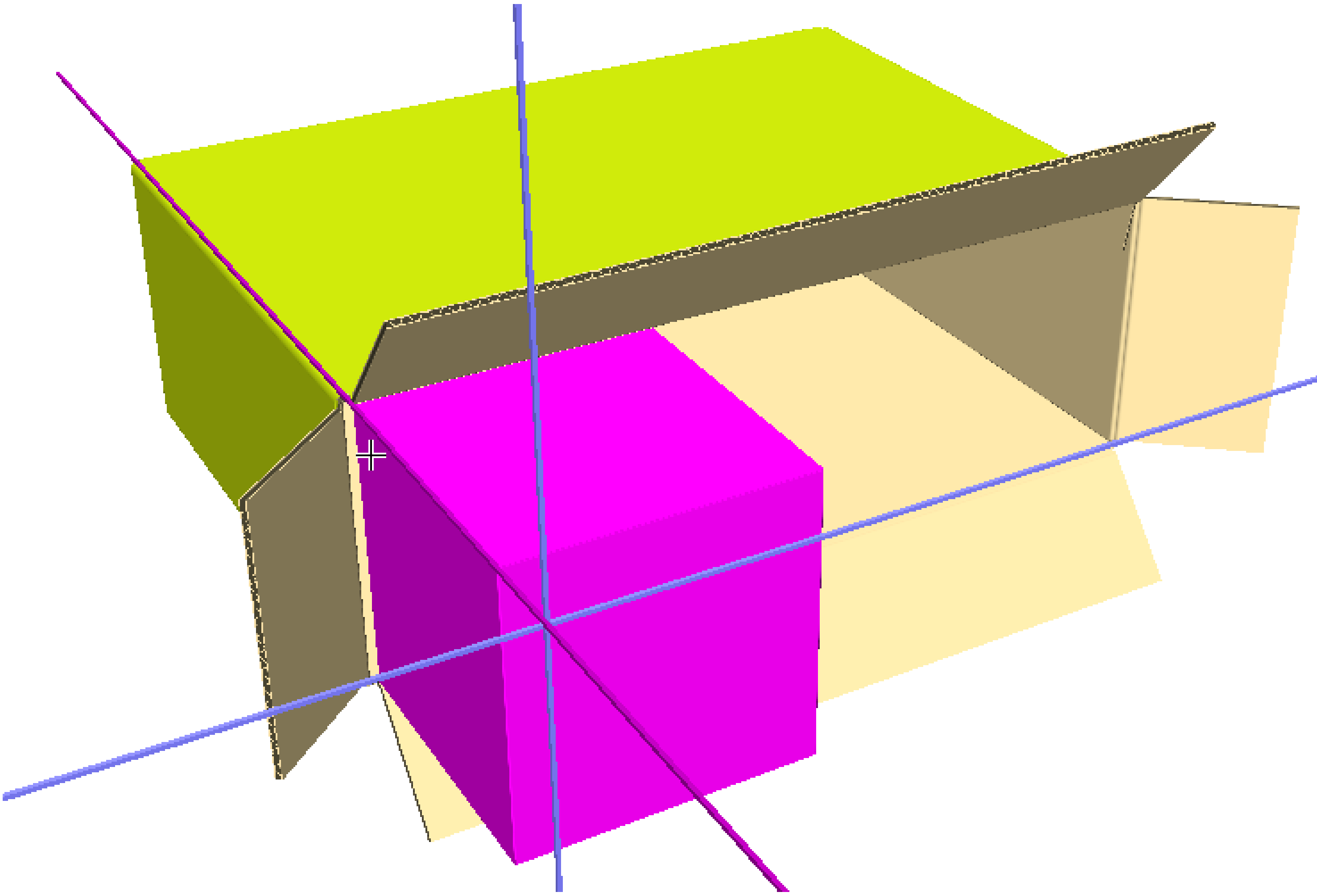
If you will be exporting the workspace to Collada again, carefully consider the desired changes. It may be better to arrange the object(s) as desired in the source application before you export to Collada and import into ArtiosCAD.

Move Design Tool Changes

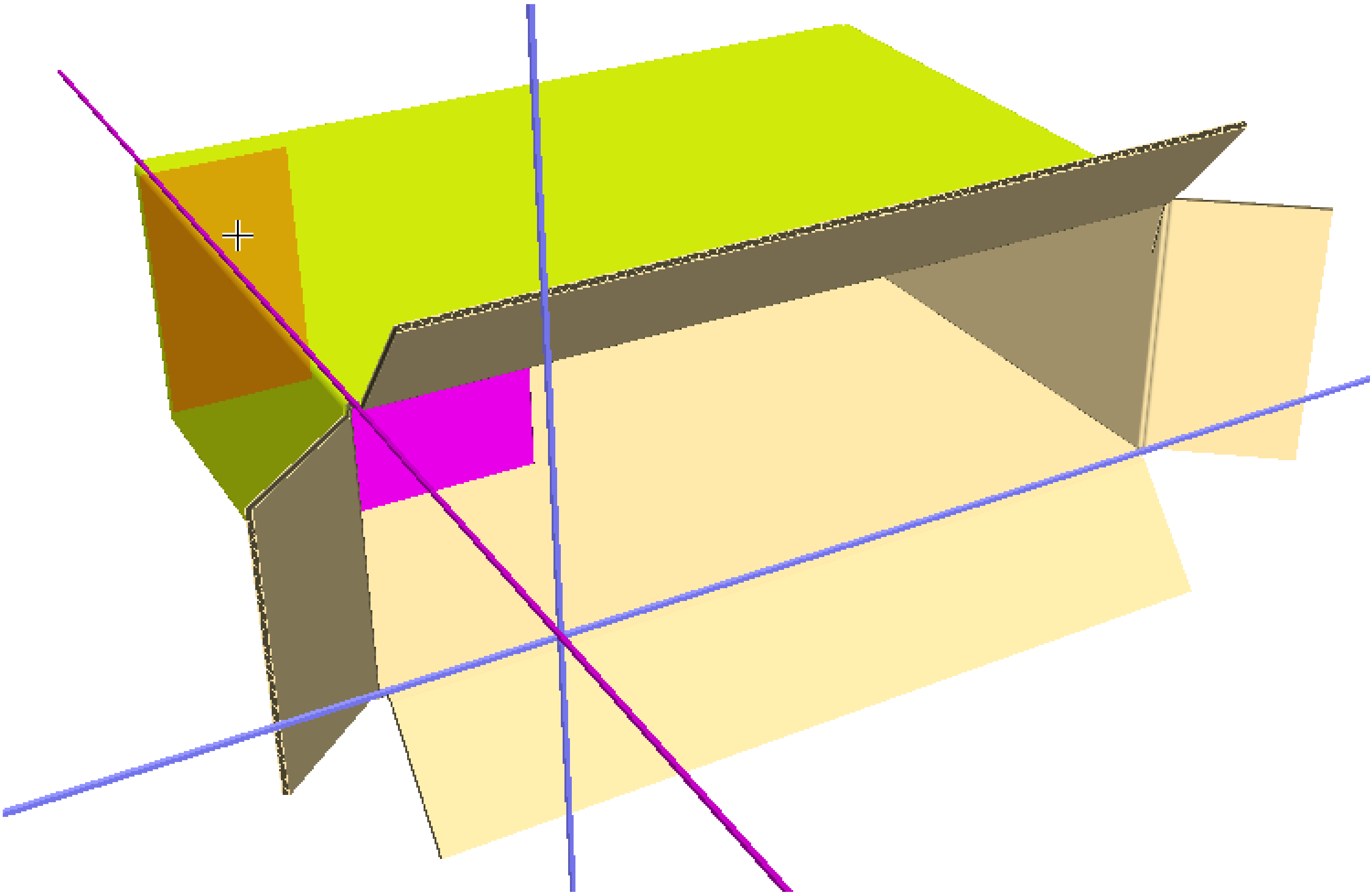
Fit tolerance on the **Move Design** tool's Status bar in 3D lets you squish together designs which would take some effort to fit together in the real world. The option works by shrinking the moving design by the amount you specify in the Fit tolerance field in both directions perpendicular to the movement axis. ArtiosCAD does not change the size of the selections in the direction in which you are moving them; that is controlled by the gutter.



1. Create a 3D workspace with components that do not quite fit together.
2.  Use **Move Design** to fit them together and discover the inner carton is slightly too large to fit into the outer box, but you know you could shove it in the real world and it would fit.



3. Check **Fit Tolerance** on the Status bar and enter a value in the field. The default is 1/16" or 1.5mm.
4. The inner box will now move inside the outer box. Shown below is it colliding with the farther edge of the outer box.



Light Source Tool Changes

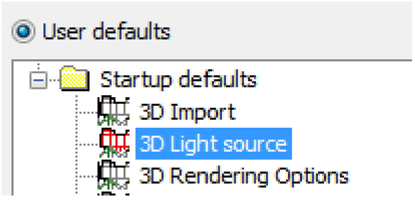
You can now set default colors and placement of the light sources in the **Light Source** tool. This information is also saved in the 3D workspace so that if someone opens the design on another system, the light source(s) appear as you intended if that person has his/her Defaults set to use the settings saved in the file.

If you close a 3D workspace while you still have its associated single design workspace open, or use **Update 2D**, ArtiosCAD saves the light source information in the single design workspace in addition to saving it in the 3D workspace.

Setting Light Source Defaults

To set the defaults for the Light Source tool, do the following:

1. Create a 3D workspace and use **Light Source** to arrange the lights and their colors as desired.
2. Click **Save lights default** on the Status bar. This saves the current lights configuration to the Startup defaults catalog in User Defaults.



Copy that entry to Shared Defaults if desired. However, you cannot change its contents inside Defaults; you must do that while working in a 3D workspace.

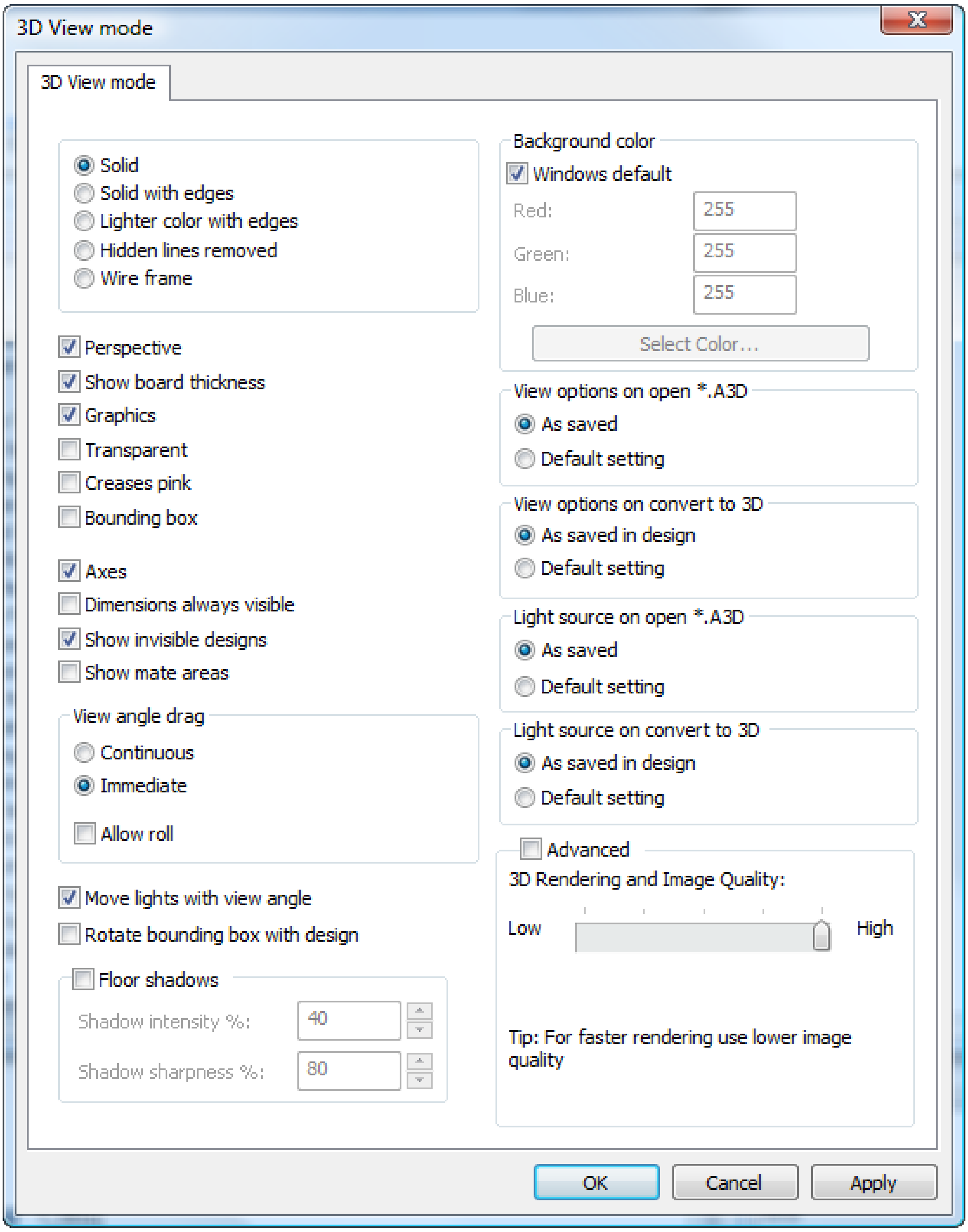
Getting Light Source Defaults

To restore the Light Source settings in a workspace to their defaults, do the following:

1.  Open the 3D workspace and click **Light Source**.
2.  Click **Get lights default** on the Status bar. This resets the workspace's current Light Source settings to the defaults.

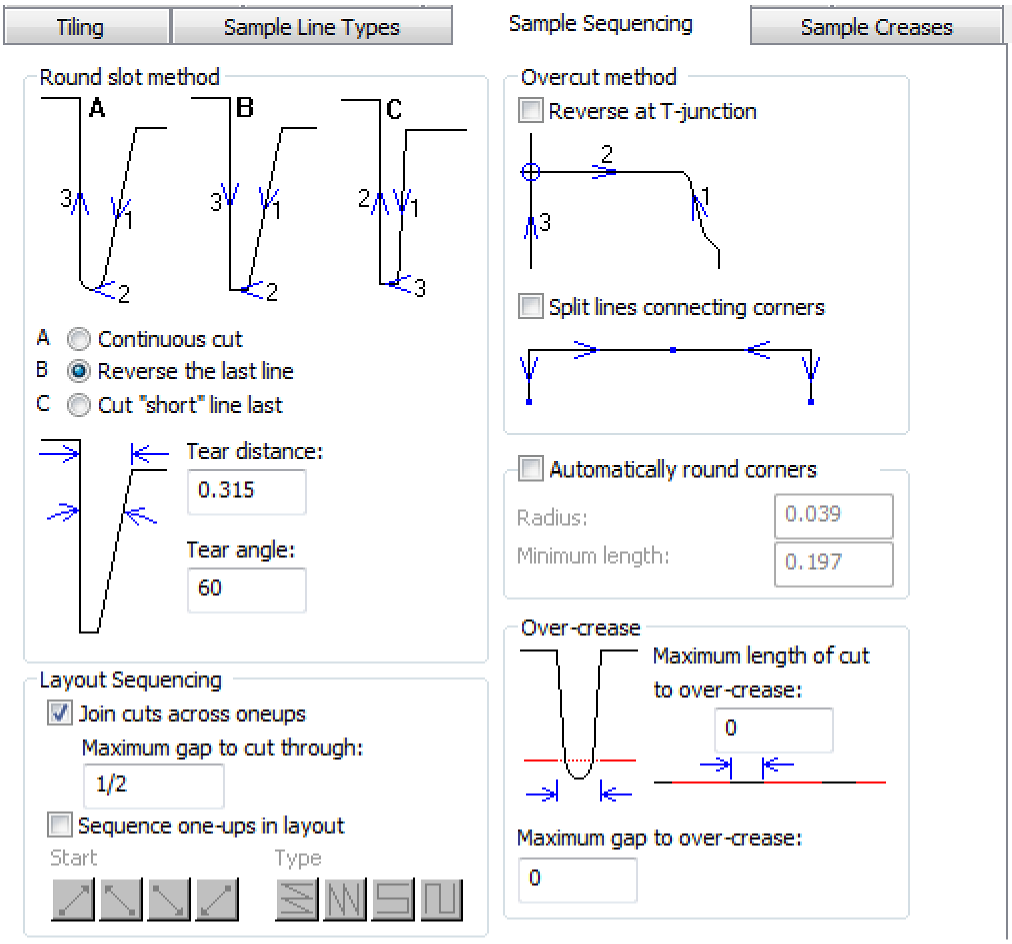
Default Settings for Light Source Defaults Behavior

When you restore a 3D workspace, ArtiosCAD normally uses the Light Source settings saved in the workspace. However if you would prefer to always use your own settings, change the settings in the **Light source on open *.A3D** and **Light source on convert to 3D** groups in the **3D View mode** entry in the Startup defaults catalog.



Faster Cutting of Labels on Samplemakers

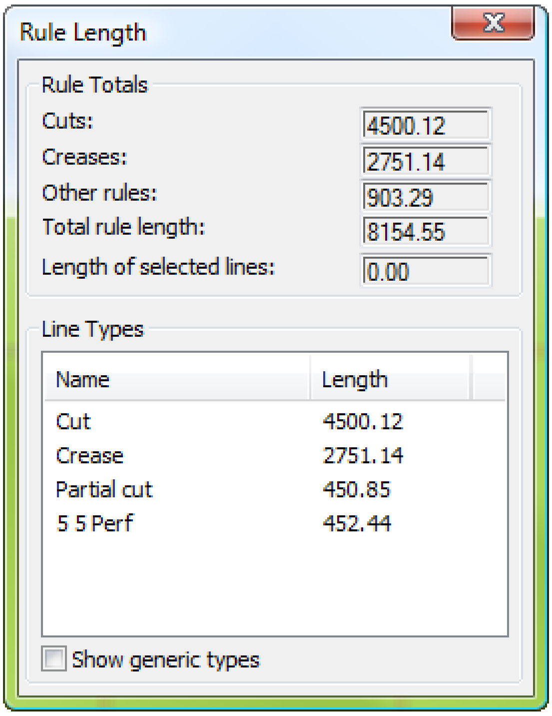
When you cut labels on a samplemaker, **Join cuts across oneups** on the Sample Sequencing tab of an Output causes the samplemaker to keep cutting through the gaps between aligned labels for faster progress instead of having to lift the knife and turn to cut label by label. **Maximum gap to cut through** sets the size limit for this behavior.

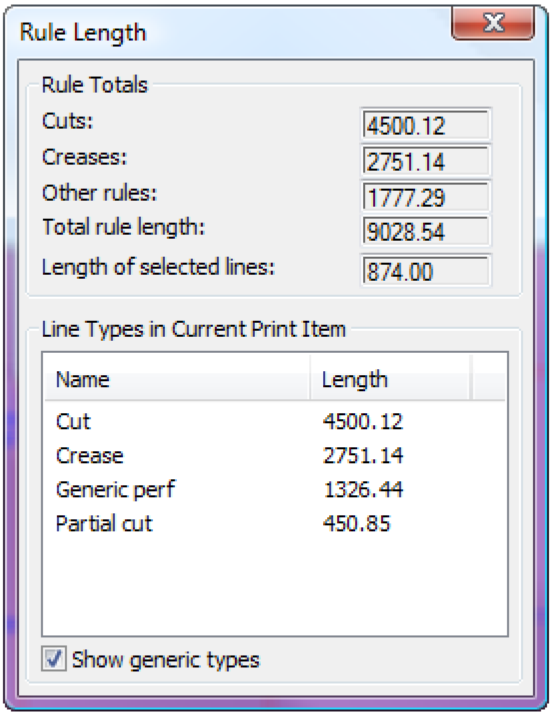


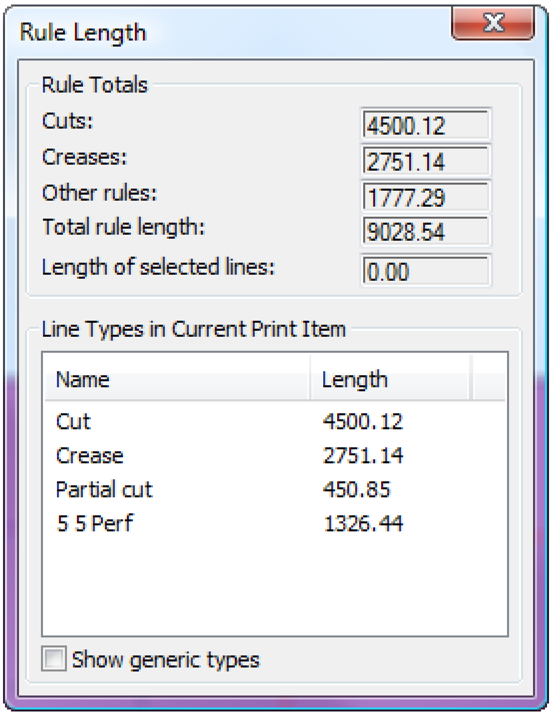
Note: When you select **Join cuts across oneups**, **Sequence one-ups in layout** is unavailable.

Rule Length Dialog Box Changes

The Rule Length dialog box (**Info** > **Rule Length)** now shows more information, similar to what is available in the line type legend. Additionally, it can show information for the current print item, as well as differentiate between specific and generic rule using **Show generic types**.


Rule Length
Rule Totals
Cuts: 4500.12
Creases: 2751.14
Other rules: 903.29
Total rule length: 8154.55
Length of selected lines: 0.00
Line Types
Name Length
Cut 4500.12
Crease 2751.14
Partial cut 450.85
5 5 Perf 452.44
Show generic types

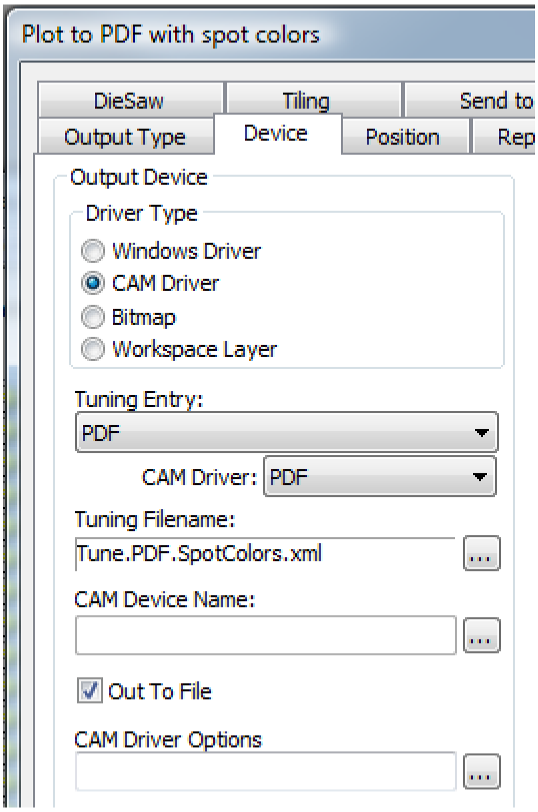

Rule Length
Rule Totals
Cuts: 4500.12
Creases: 2751.14
Other rules: 1777.29
Total rule length: 9028.54
Length of selected lines: 874.00
Line Types in Current Print Item
Name Length
Cut 4500.12
Crease 2751.14
Generic perf 1326.44
Partial cut 450.85
Show generic types



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.

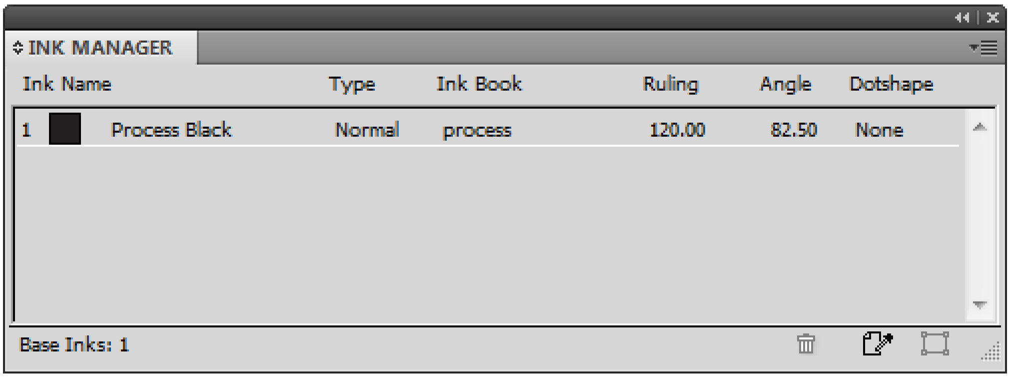
1. Click **Options** > **Defaults** .
2. In Shared Defaults, copy an existing PDF Output to an entry with a different name, such as `Plot to PDF using spot colors`.
3. Double-click the new entry and click the **Device** tab.
4. Click **...** (Browse) at the end of the Tuning Filename field.
   a) Navigate to `InstLib`.
   b) Change the filetype selector to **All files (*.*)**.
   c) Click `Tune.PDF.SpotColors.xml` and then click **Open**. The entry in the Tuning Filename field should change accordingly.

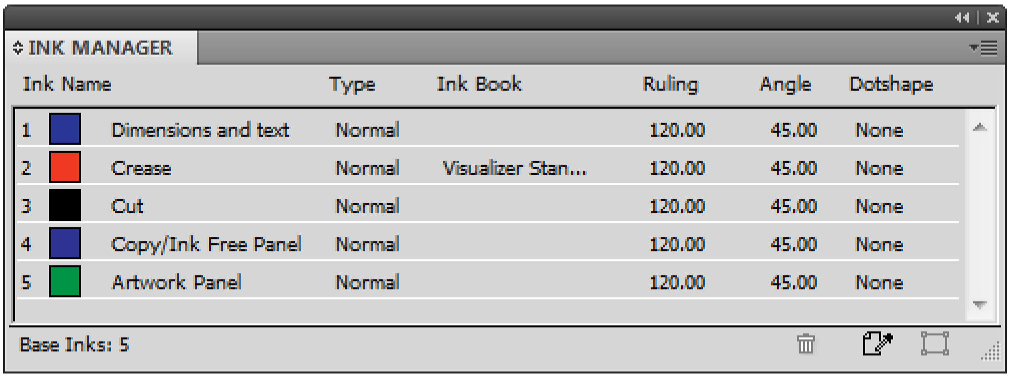


5. Click **OK** to make the change.

6. 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`.

3. Getting started with ArtiosCAD

Welcome to ArtiosCAD

The fundamental purpose of ArtiosCAD is to turn your creative thoughts into business for your company. Speed, efficiency, and accuracy are all essential to providing the best product to your customers.

This book is intended for people with experience using the Microsoft Windows family of operating systems who are just starting to use ArtiosCAD.

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

Your system may not have all of the features described in this publication. Contact your Esko salesperson for information about purchasing features not installed on your system. Also refer to `http://www.esko.com` for more information.

You must have local administrative privileges to fully configure ArtiosCAD.

Multiple monitor settings

If your system uses multiple monitors, to prevent unpredictable behavior, make sure they are both set to the same performance settings in the operating system.

Multiple monitors in Windows XP

In Windows XP, do the following to check the monitor settings:

1. Log in as `Administrator` or as a user with administrative privileges.
2. Go to Control Panel and open the **Display** applet.
3. Click **Settings**.
4. Click the monitor to adjust.
5. Click **Advanced** and then click the **Troubleshooting** tab.
6. Slide the **Hardware acceleration** slider as far right as possible.
7. Check the **Enable write combining** checkbox if it is not checked.
8. Click **OK** twice to make the changes and dismiss the dialog boxes.

Multiple monitors in Windows Vista

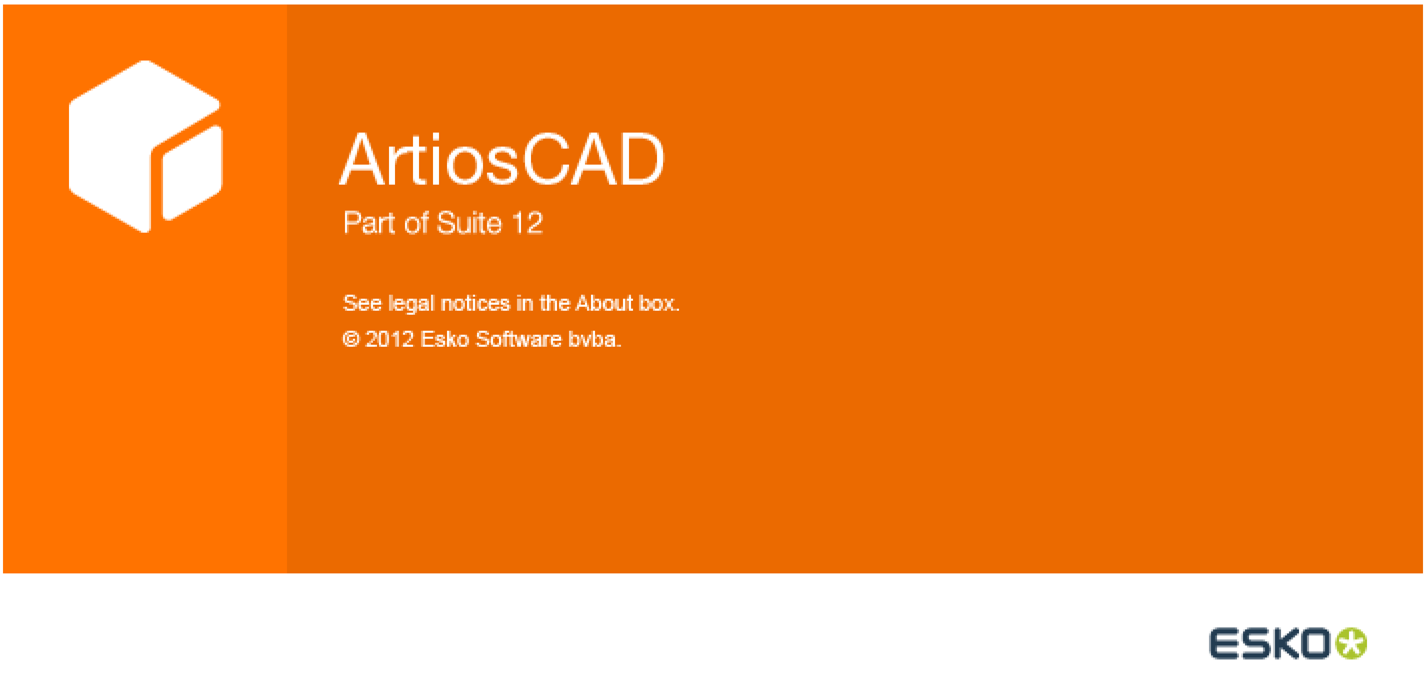
In Windows Vista, do the following to check the monitor settings:

1. Log in as `Administrator` or as a user with administrative privileges.
2. Go to Control Panel.
3. In the **Appearance and Personalization** category, click **Adjust screen resolution**.
4. In the **Display Settings** dialog box, click **Advanced Settings**.
5. Click the **Troubleshooting** tab.
6. If available, click **Change settings**, set the **Hardware acceleration** slider to **Full** and check the **Enable write combining** checkbox. In Windows Vista Business, these controls may not be available due to the display adapter vendor's utilities taking precedence. In that case, use those utilities to adjust the display adapter performance.
7. Click **OK** twice to make the changes and close the dialog boxes.

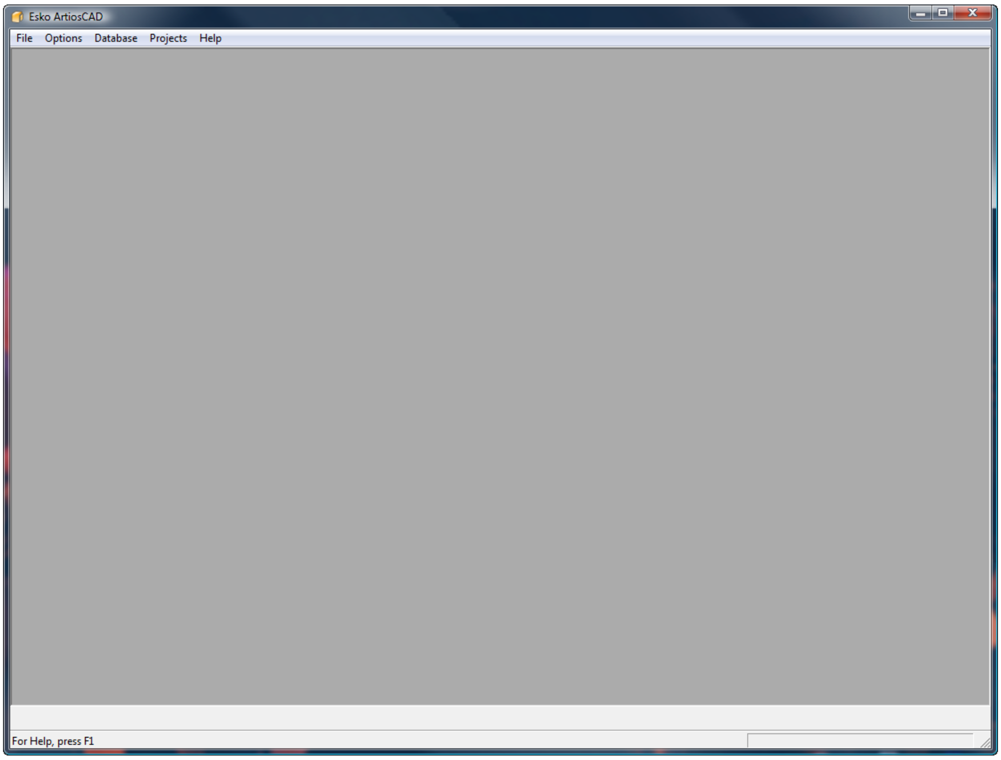
Starting ArtiosCAD

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

1. Click **Start**, then point to **All Programs**.
2. Point to the **Esko** > **Esko ArtiosCAD** folder in the Start menu, and then click the version of ArtiosCAD you have installed.
3. The ArtiosCAD startup screen will appear.



4. The ArtiosCAD main window will replace the startup screen.



ArtiosCAD is now ready for use – you can create or open a single design, fold a design in three dimensions, create a set of manufacturing tools, set system defaults, or use database functions.

Concepts and Ideas in ArtiosCAD

A typical workflow

The process of creating boxes, either folding carton or corru-gated, is usually a logical progression. First you design the box and embellish it with graphics from your art department, from the customer, or graphics that you designed yourself. Then you print it on a desktop printer for review. At this point, you may export it to a different format for use by others. You could also fold the box in 3D to make sure all the dimensions work together. Sometimes a specification sheet of the folded design is created to send to the customer for approval. If ap-proved, samples are made. If the samples work and are ap-proved, manufacturing tools for the box are made. These manu-facturing tools then are used in production of the box, and the box is shipped to the customer.

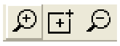
Tools

Work in ArtiosCAD is completed using tools found on toolbars. Select a tool by clicking it with the mouse cursor. The selected tool will prompt for information in the Status bar and guide you using graphical feedback (drag). The toolbar master controls determine which toolbars are visible.

Changing the view

The way a design is viewed in ArtiosCAD can be customized to show different information. The structural properties of the design remain the same throughout - only the presentation changes. Three elements control the display – the **Zoom** and **Pan** commands, the *View mode*, and the *plotting style*.

Zooming in and out

 ArtiosCAD lets you view your design from different distances. This distance is set using the **Zoom** tools. You can zoom in on a particular area so that you can see minute tolerances, or you can zoom back so far that the design becomes a dot on the screen.

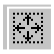
 Zooming in with the **Zoom In** tool means the design is displayed in more detail. Either click and drag a rectangle to indicate an area to zoom into, or click a coordinate to zoom in on that coordinate.

 Zooming out with the **Zoom Out** tool means the design is viewed in less detail.

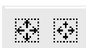
 The **Center-Point Zoom** tool, when clicked, prompts you to click the center of an area to be zoomed in upon, and then to drag to the corner of that same area. Release the mouse button to perform the zoom.

If you zoom in on an imported Adobe Acrobat PDF graphic and you have purchased the PDF option, **High graphics mode** in the View Mode dialog box enables sharper resolution as you zoom in. High graphics mode is covered in depth later in this manual.

Scale to fit tools

 The **Scale to Fit** command on the View menu (and on the View toolbar) zooms in as close as possible without having any edge of the design outside of the window. This is especially useful when you are either zoomed way in or zoomed way out and want to see the complete design quickly.

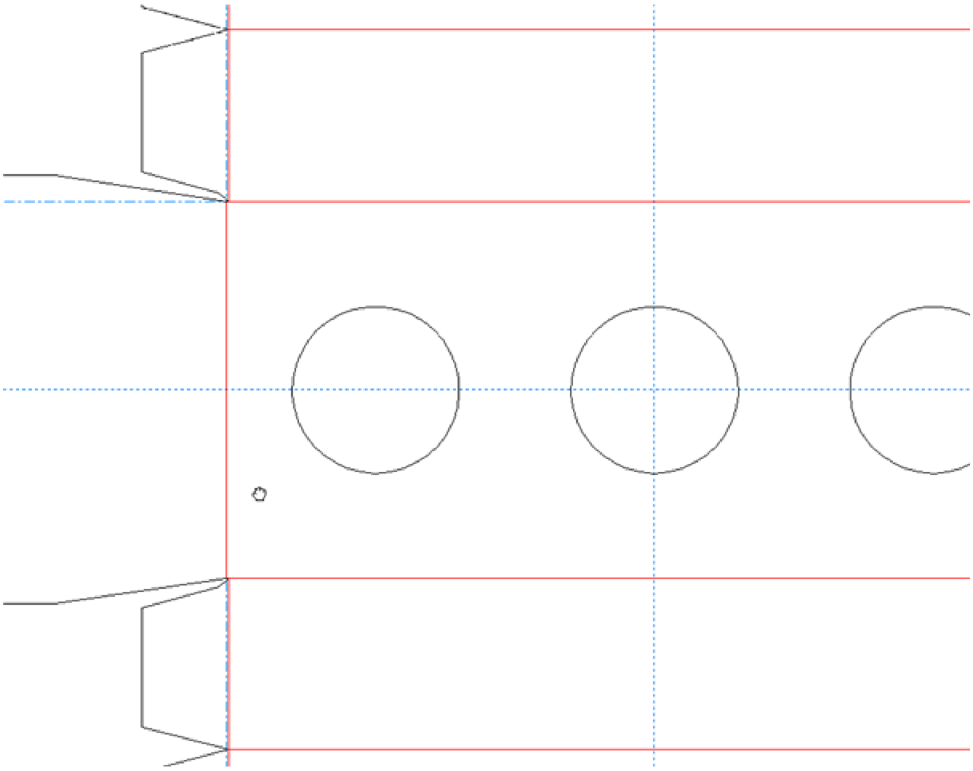
When held down, it activates the Scale to Fit tools flyout toolbar.



 Click the**Scale to Fit with Border** tool to put a border around the scaled-to-fit view. The size of the border is configured in **Options** > **Defaults** > **Startup defaults** > **View Tools options**.

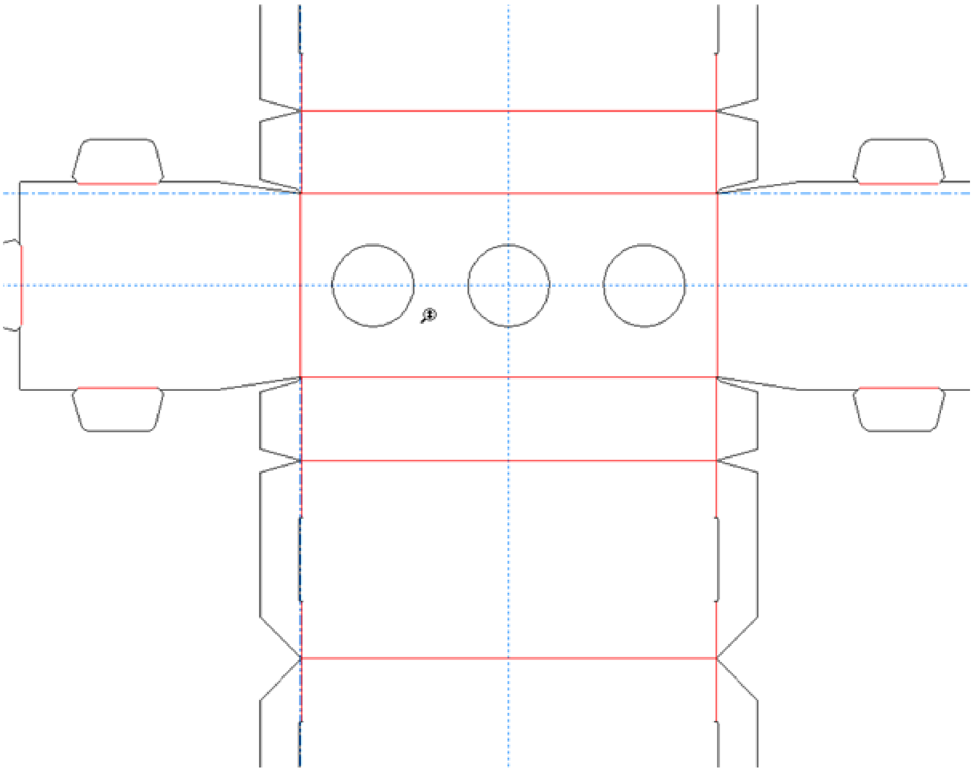
Pan/Zoom tool

The **Pan/Zoom** tool lets you grab a location on the screen and drag it as desired in Pan mode, or zooms in and out according to the direction you drag (up or down) when in Zoom mode. To use it in Pan mode, click it, click and hold a location on the screen, and drag to the new location. Click the right mouse button or press ESC to exit the tool and return to the previous tool used. The cursor looks like a clenched hand while dragging.



Note: If you use the Pan/Zoom tool in high graphics mode, you must refresh the screen by pressing F2 to re-render the graphics.

To use the Pan/Zoom tool in Zoom mode, click it, click **Zoom** on the Status bar, position the cursor in the desired area, and then drag the cursor up to zoom in or down to zoom out. Click the right mouse button or press ESC to exit the tool and return to the previous tool used. Shown below is a design in process of being zoomed in upon.



Press tab on the keyboard to change between Pan and Zoom modes without clicking their respective option buttons.

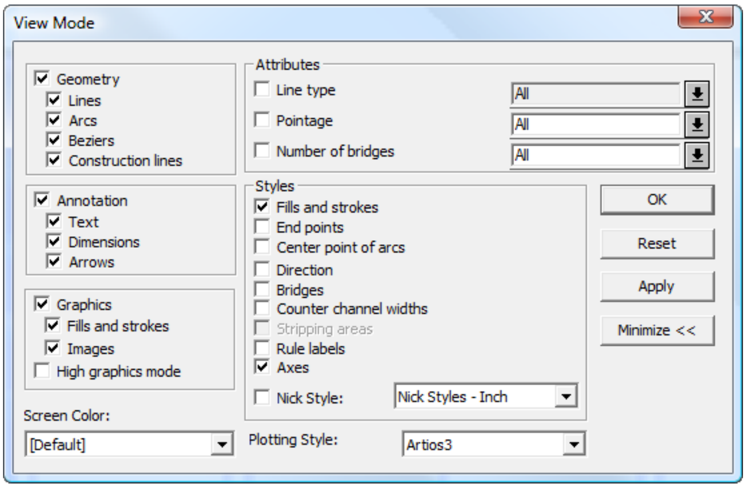
To exit the Pan/Zoom tool in either mode, either click the mouse button, press ESC on the keyboard, or activate another tool. The previously-used tool will resume operation.

If you have a three-button USB mouse with a scroll wheel, using the scroll wheel at any time zooms in or zooms out as if you were using the Pan/Zoom tool in Zoom mode. Holding the middle mouse button down and dragging the cursor pans the view as if you were using the Pan/Zoom tool in Pan mode.

The amount of zoom controlled by the scroll wheel is configured in **Options** > **Defaults** > **Startup defaults** > **View tools options**.

View mode

 The **View mode** command on the View menu (and on the View toolbar) lets you choose the types of geometry and graphics ArtiosCAD displays.



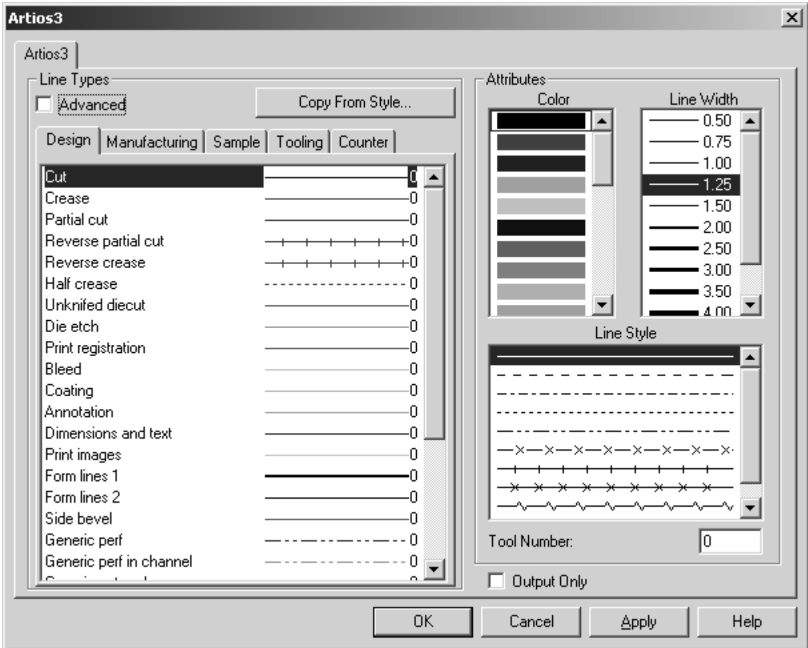
Check or clear the checkboxes as desired. Click the arrow at the end of the **Plotting Style** field to select a new plotting style from the drop-down list.

The arrows at the ends of the items in the **Attributes** group let you choose predefined values from a drop-down list. You can type your own values (or a range of values by typing **starting value .. ending value**) in the **Pointage** and **Number of bridges** fields but not in the **Line type** field.

The default view mode styles can be set by clicking **Options** > **Defaults** > **Design Defaults** > **Default View Mode**. The default screen color is set in the **Screen color** element of the Design Defaults catalog.

Plotting styles

Plotting styles control the display attributes of all objects on the screen. Using plotting styles, you can control the width, color, and pattern of design lines, and the tools a samplemaker or other machine uses to cut, burn, or otherwise create the lines. The current plotting style can be changed in the View Mode menu, and plotting styles in general are configured in Defaults.



Construction lines

Construction lines are special guide lines used as drafting aids only. They aid in the construction of geometry that relies on points not in the actual design. Construction lines are not manufac-tured, and are not exported to different formats.

Layers

ArtiosCAD uses the concept of layers to build a complete design. Layers are used in single design and manufacturing. Think of a piece of paper with a drawing of a basic container. Then on top of that, add transparencies. One transparency could contain dimensions, another could contain art, and yet another could contain additional design geometry such as a window or an optional flap. A design may have up to 100 layers.

ArtiosCAD works with the layers that are turned on. Only layers that are visible are printed or exported to a file. However, all layers are saved when the design is saved regardless of their viewing status.

Properties

Most objects in ArtiosCAD have properties associated with them, such as line type, pointage, font, color, and so forth that can be changed by double-clicking the object(s) with the Select tool. This invokes the Properties dialog box. If more than one type of object is selected, for instance a line and some text, the Properties dialog box will have tabs across the top that allow you to change the properties of each type of item selected. Close the dialog box to apply the changes to the properties.

Transformations

Transformations such as moving, copying, or rotating design elements are two-step processes. The first step is to select the item(s) to be transformed; selected items turn magenta. The second step is to select a tool on the Edit toolbar. The Edit tools (other than Select) are available only when something is selected.

Current position

The current position is the point in a workspace where creation tools start creating. When a new blank design is created, the current position defaults to the intersection of the construction line axes. When the first line is drawn from the current position, the current position then moves to the end of that line. What-ever is created next will begin at the end of the first line. Then when the second object is created, the current position will make the third object start from the end of the second object.

The current position can be moved by using the  **Move to Point** tool (`CTRL-W`) on the Geometry Tools toolbar or **Move By** (`CTRL-Q`) on the **Tools** > **Current Point** menu. **Move to Point** moves the current position to an established point. **Move by** prompts for an angle and offset at which to move the current position.

Entering information in ArtiosCAD

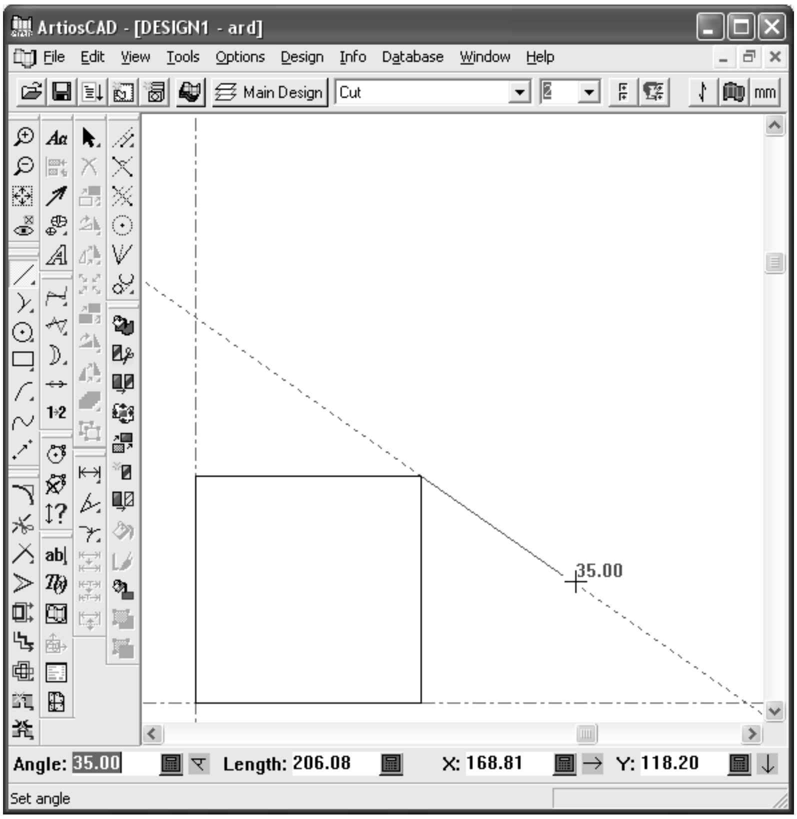
There are three ways to enter information in response to a prompt. The first way is to move the pointer until the desired value is displayed on the screen (called *drag*). The second way is to type information on the keyboard into fields on the Status bar. The third way is to use the keypad on the screen in conjunction with the mouse.

Using drag

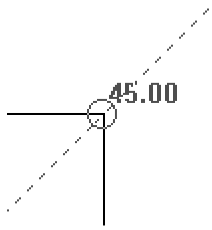
Drag is a basic concept of ArtiosCAD. Drag is an image and numbers that appear when ArtiosCAD prompts for a physical dimension, such as specifying the angle at which to move an object. Moving the pointer changes this image and numbers. Drag can be tuned to use the increments you want in the Snap Options dialog box on the Options menu.

Freehand dragallows you to move objects by clicking and dragging them. By default, this behavior is enabled only for text, dimensions, and annotations; however, it can apply to lines, arcs, and beziers by clicking that option in the Snap dialog box in Options.

You must select a line, arc, or bezier by one of its endpoints in order to use it with freehand drag. This reduces the potential for accidental moves.



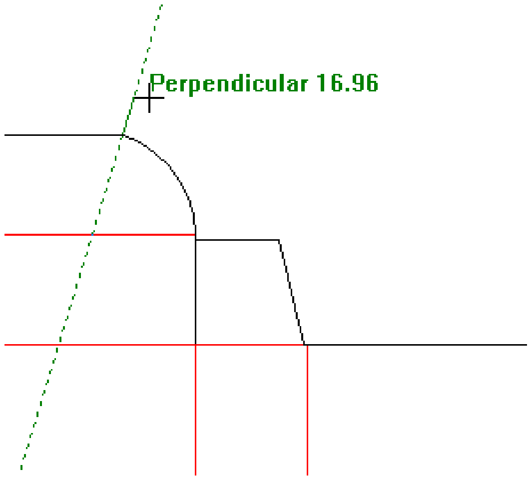
A small circle appears when you have created an object by aligning it with an established point (called snapping to a point). If the circle does not appear around the point you think you snapped to, then you did not snap to that point.

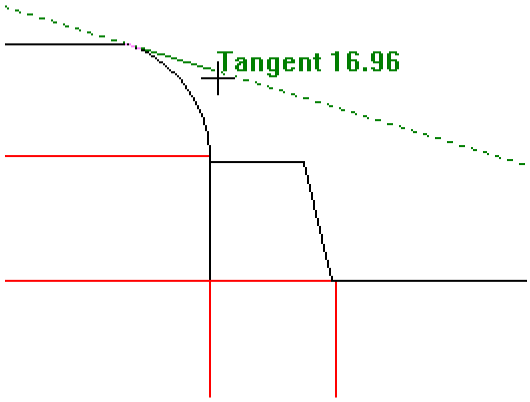


When using the following tools, holding down `SHIFT` while using drag to set an angle adds perpendicular and tangential angles to the snap.

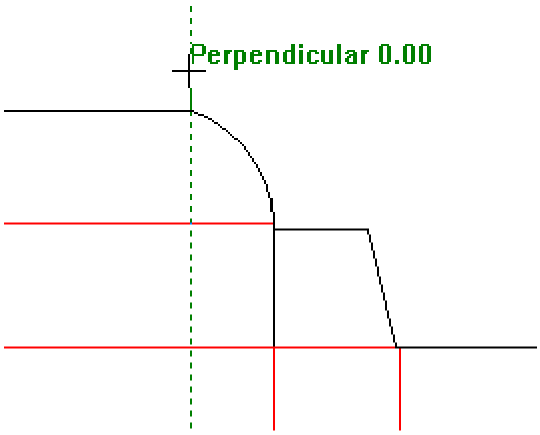
- Line Angle/Offset
- Line Horiz/Vert
- Line Angle/Length
- Arc Start Angle
- Arc End Point
- Straight Curve
- Curve Straight
- Conline Offset/Angle

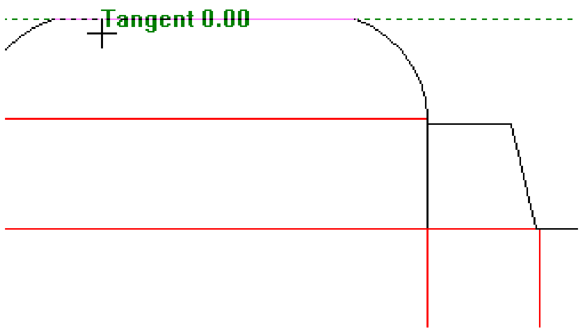
In the examples below, the end of the arc is the point against which the perpendicular and tangential angles are measured.





However, since the endpoint is shared, ArtiosCAD also shows the perpendicular and tangential angles for the line as shown below.





Keyboard Shortcuts in ArtiosCAD

You can press the keys in the Shortcut columns in the table below to perform the commands in the Function columns. Not all commands are available at all times; for example, if you have not cut or copied anything, you can not paste anything. You can change Shortcuts in Defaults.

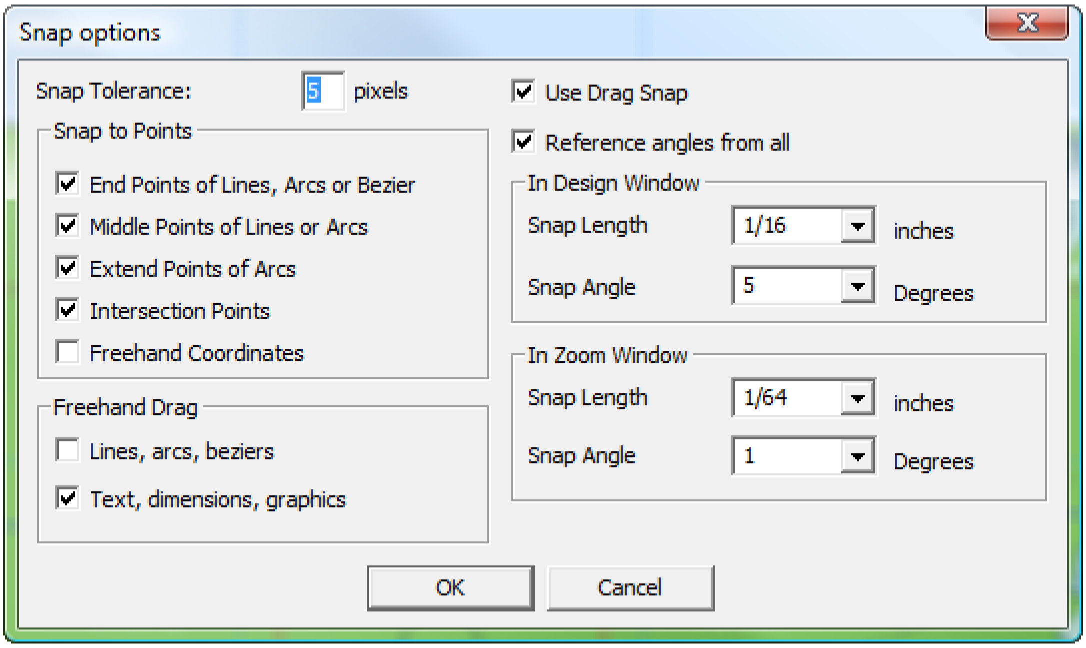
Table: Keyboard Shortcuts

Function	Shortcut	Function	Shortcut
New Design	CTRL + N	Zoom Rectangle	CTRL + R
Run a Standard	CTRL + 2	Zoom Out	CTRL + A
Convert to 3D	CTRL + 3	CloseUp Window	CTRL + Space
New Layout	CTRL + L	Extend	ALT + X
Open	CTRL + O	Clear Extend	CTRL + ALT + X
Save	CTRL + S	Move to Point	CTRL + W
Print	CTRL + P	Move By	CTRL + Q
Undo	CTRL + Z	Line Angle/Offset	CTRL + 1
Redo	ALT + Z	Conline Offset/Angle	CTRL + F
Cut	CTRL + X	Keypad	CTRL + 4
Copy	CTRL + C	Rebuild	F5
Paste	CTRL + V	Measure Tool	CTRL + G
Select	CTRL + E	Copy Mode	CTRL + INSERT
Move	CTRL + M	Move Mode	CTRL + M
Properties	ALT + Enter	Split Line	CTRL + L
Refresh	F2		

Function	Shortcut	Function	Shortcut
Scale to Fit	CTRL + D		

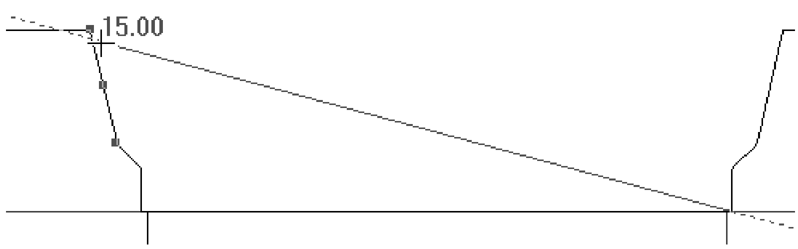
Snap

Snap accommodates non-precision when selecting points. If you click close enough to a point, ArtiosCAD interprets your click as selecting that point, even if you are a few pixels off. The area around a point that you can click in and still select it is called **Snap Tolerance**.

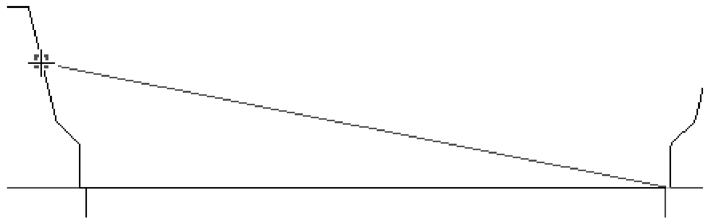


Some tools, such as the Bezier tool, require **Freehand Coordinates** to be turned on. If you try to click a point and **Cannot select coord** appears in the Status bar, make sure Freehand Coordinates is turned on.

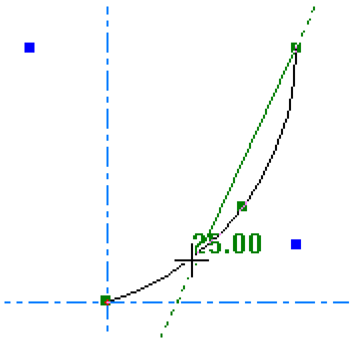
If **Middle Points of Lines and Arcs** is enabled, once you are within the snap tolerance of a line or arc, the endpoints and midpoint appear.



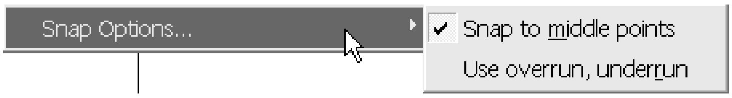
Highlighting appears around such a point when the cursor is over it.



Extend Points of Arcs causes blue squares to appear around center, corner, and quadrant points of arcs, and green squares highlight the start, end, and midpoints of arcs. Blue squares also appear for the center and quadrant points of a circle. When this checkbox is cleared, only the start, end, and midpoints of arcs appear. Zoom in on small geometry, else you will see overlapping squares. There are no snap points on construction line circles.

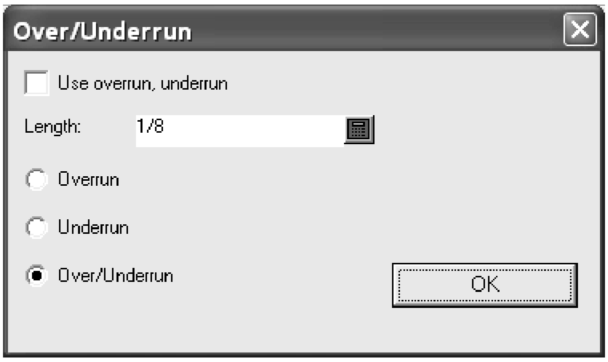


On the View menu, the **Snap Options** submenu lets you quickly turn on snapping to middle points. You can also assign this to a Shortcut if desired.



Overrun/Underrun

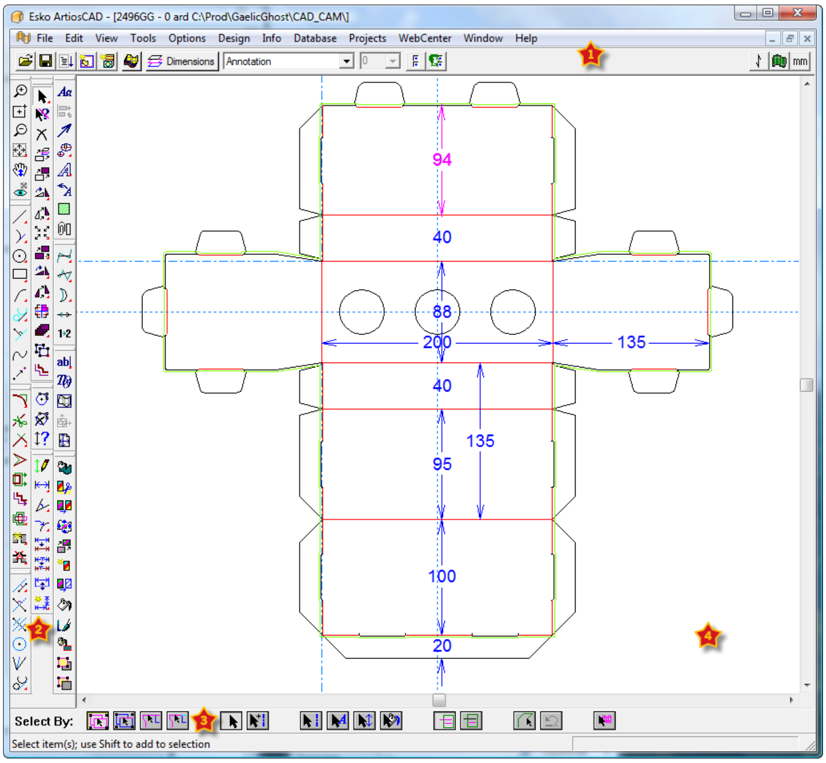
Also on the **View** > **Snap Options** menu is **Use overrun, underrun** which extends, shortens, or does both to every line by a set amount and is configured in **Options** > **Over/Underrun**.



Design window elements

The ArtiosCAD Design window contains four functional areas:

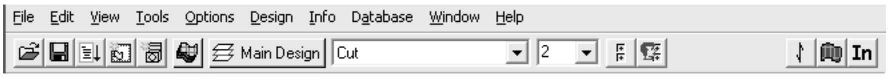
1. The menu bar, the view bar, and the toolbar master controls
2. The toolbars
3. The status bar
4. The drawing area



The menu bar, view bar, and toolbars all control the action in the drawing area. The status bar displays information about the tool being used, and if applicable, prompts for specific information.

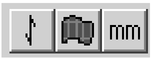
The menu bar

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



The View bar

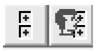
The View bar provides one-click access to commonly used features.



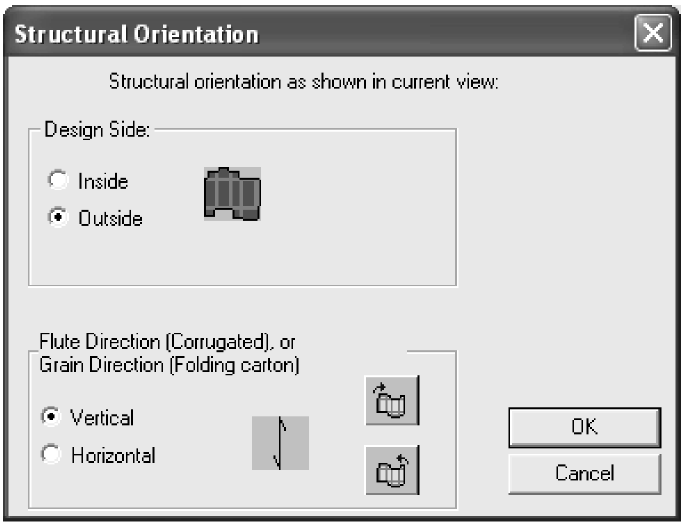
Each button on the View bar displays a ToolTip if the pointer hovers over it for a few seconds.

In the first group of controls on the View bar, the first button opens the Open dialog box. The second button saves the current design. The third button rebuilds the current design. The fourth converts the current design to a Manufacturing file. The fifth button converts the current design to a 3D file. The sixth button creates a print item. The seventh button adds, deletes, and modifies layers.

The second group of controls on the View bar controls line type and pointage. They are described in the next section.


The third group of buttons on the View bar are the toolbar master control buttons. They are described after line types.

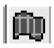

In the fourth group of buttons on the View bar, the first button displays the Structural Orientation dialog box.



The **Inside** and **Outside** option buttons in the **Design Side** group show the current side up of the design. Change which side is up by choosing the other option button and clicking **OK** to perform the change.

In the **Flute Direction/Grain Direction** group, the **Vertical** and **Horizontal** option buttons show the current fiber orientation. Change the current direction by choosing the other option button and clicking **OK** to perform the change.

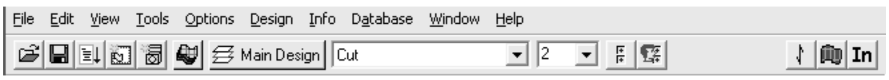
The two buttons labelled with a carton and an arrow change the physical orientation of the current design. Clicking the top button rotates the design 90 degrees to the right and changes the grain/corrugation direction. Clicking the bottom button rotates the design 90 degrees to the left and changes the grain/corrugation direction. Clicking **OK** accepts these changes; clicking Cancel undoes them.

 The second button displays the current side up (printed or unprinted), and changes it when clicked.

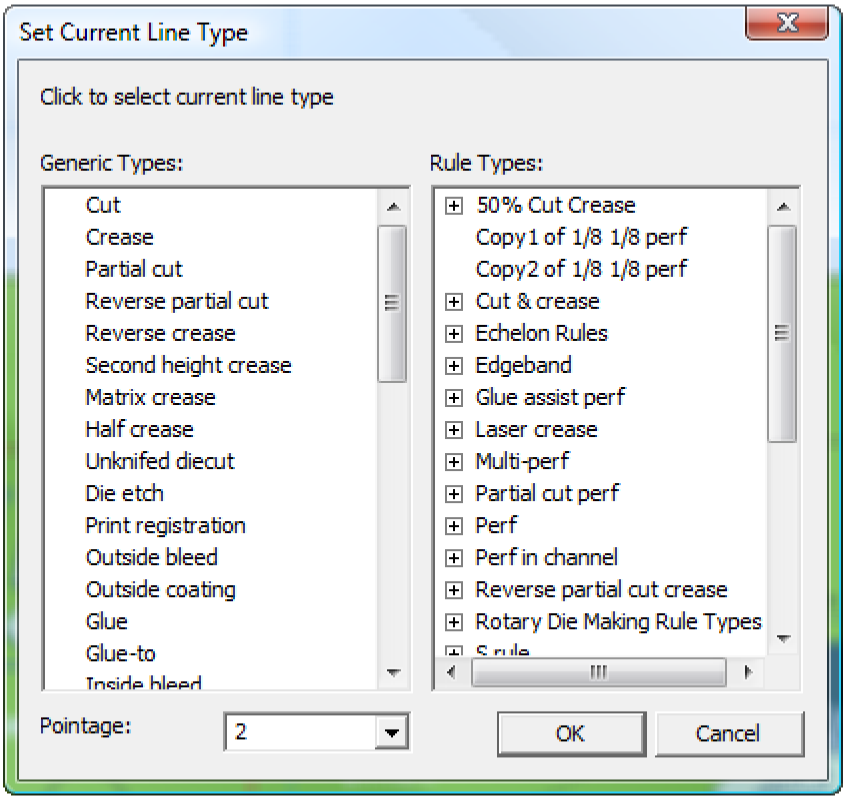
The third button displays the units of the current design - In for imperial format and mm for metric format.

Changing the line type and pointage

The second group of controls on the View bar controls the line type and pointage. With nothing selected, they show the line type and pointage used to create new lines. If any lines are selected and the controls are changed, the lines change accordingly. If lines of differing attributes are selected, the respective drop-down list boxes are empty. If nothing is selected and these controls are changed, then any lines made from that point forward will reflect the changes made in the controls.



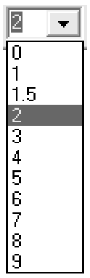
Each control is a drop-down list box. The **Line Type** drop-down list box displays all line types currently in the workspace and an **Others** entry which opens the Set Current Line Type dialog box when clicked.



Select the type of line to create and click **OK** to continue.

When selecting special rules to use for manufacturing, make sure to select the special rule from the **Rule Types** group, and not the **Generic Types** group. If you select a special rule type, the special rule type will use its default pointage and the Pointage drop-down list box will be disabled.

The **Pointage** drop down list box displays a list of the most commonly-used pointages.



Select the new pointage from the list. It will affect new lines created that are of a generic line type.

When you change the line type and then make a line, ArtiosCAD checks to make sure that the line type is appropriate for the current layer. Listed below is a list of line types and their appropriate layers.

Table: Line types and appropriate layers

Line type	Appropriate layer classes
Annotation, print image, dimensions and text, form windows, construction lines, Tooling line types	Any (no warning)
Tear	3D Assist
Print registration	Print registration, Horizontal cross section
Outside bleed	Outside bleed
Inside bleed	Inside bleed
Outside coating	Outside coating (1-3)
Inside coating	Inside coating
Cut	Main, Manufacturing, Windows and cutouts, Horizontal cross section
Crease, Glue assist	Main, Manufacturing, Windows and cutouts
Side bevel, generic perf, all special and generic rules	Main, Manufacturing, Windows and cutouts
Unknifed diecut, Die etch, Die registration hole, Laser position hole, Die bolt holes	Main, Manufacturing, Windows and cutouts
Counter line types	Counter
Stripping boards, other manufacturing	Main, Manufacturing
Dynamic Art	Dynamic Art
Copy area, Copy free	Artwork Panels

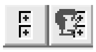
The default line types for layer classes are listed below.

Table: Default line types for layer classes

Layer class(es)	Default line type
Main, Manufacturing, Windows and cutouts, Horizontal cross section	Cut, or last valid line type used
Counter	Counter periphery tool, or last valid line type used
Print registration	Print registration
Outside bleed	Outside bleed
Outside coating (1-3)	Outside coating

Layer class(es)	Default line type
Inside bleed	Inside bleed
Inside coating	Inside coating
Dimensions, Overall dimensions, Annotation	Annotation
Dynamic Art	Dynamic Art
Artwork Panels	Copy area
3D Assist	Tear
Graphics, Outside graphics, Inside graphics, Label graphics, Shrink-wrap graphics	Print image
All others	None; any type is appropriate

The Toolbar Master Controls and toolbars

 The Toolbar Master Controls let you turn toolbars on and off in Designer and Manufacturing. The first button controls which standard toolbars are shown. The second button controls which customized toolbars are shown.

Initially the toolbars appear to the left of the drawing area, but each toolbar can be independently detached, or docked in a different position if desired.

Note: When a toolbar is activated for the first time, it will appear undocked outside the ArtiosCAD window. Drag it to the desired position inside ArtiosCAD.

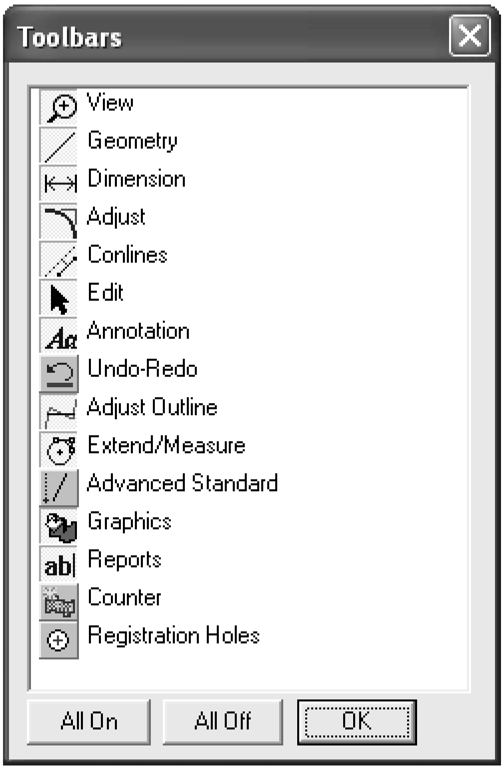
Fifteen pre-defined toolbars are controlled by the Toolbar Master Control for standard toolbars in Designer. Up to 10 customized toolbars are controlled by the Customized Toolbar Master Control; it depends how many are defined in Defaults.

To turn a toolbar on or off, click the button preceding the name of the toolbar. Each button depresses when clicked for the first time to indicate the toolbar is on. Click the button again to return it to its raised position, indicating the toolbar is turned off.

Availability of toolbars controlled by the Toolbar Master Controls depends on the options purchased. If you cannot click a toolbar activation button, the tools or features controlled by that toolbar have not been purchased.

Designer toolbars

Shown below is the Toolbar Master Control for standard toolbars in Design.



The first button turns the Zoom toolbar on and off. The tools on the Zoom toolbar are used to change the view of the current window.

The second button turns the Geometry toolbar on and off. The tools on the Geometry toolbar construct lines and shapes.

The third button turns the Dimension toolbar on and off. The tools on the Dimension toolbar construct and modify dimensional annotations of lines, angles, and arcs.

The fourth button turns the Adjust toolbar on and off. The tools on the Adjust toolbar change the linear attributes of existing lines.

The fifth button turns the Construction Lines toolbar on and off. Construction lines are drawing aids that are not plotted or manufactured. The tools on the Construction Lines toolbar manipulate construction lines.

The sixth button turns the Edit toolbar on and off. The tools on the Edit toolbar perform physical transformations on items such as moving or copying them.

The seventh button turns the Annotations toolbar on and off. The tools on the Annotations toolbar add text, legends, and details to the current design.

The eighth button turns the Undo-Redo toolbar on and off. Tools on the Undo-Redo toolbar undo and redo actions, and manipulate the list of already-executed actions which can be undone and redone.

The ninth button turns the Adjust Outline toolbar on and off. The tools on the Adjust Outline toolbar adjust the non-linear physical properties of lines such as parallelism with an axis and direction.

The tenth button turns the Extend toolbar on and off. Tools on this toolbar create construction line extensions that extend lines into infinity and arcs into circles. These extensions contain points on which you can reference new geometry.

The eleventh button turns the Advanced Standard toolbar on and off. These tools create geometry based on other points in the design and are intended for use by the most advanced users.

The twelfth button turns the Graphics toolbar on and off. The tools on the Graphics toolbar manipulate pasted images from outside sources, the stock color, fills, strokes, and graphic placement.

The thirteenth button turns the Reports toolbar on and off. The tools on the Reports toolbar convert the current design into a report.

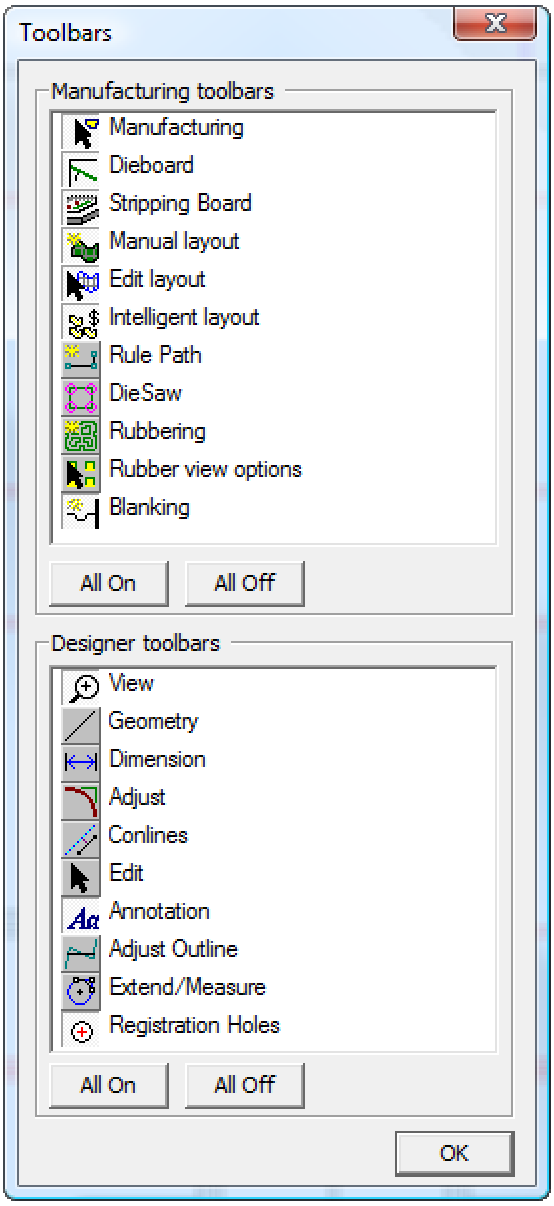
The fourteenth button turns the Counters toolbar on and off. The tools on the Counters toolbar operate the Intelligent Counters module. This module must be purchased in order for the Toolbar Master Control button to work.

The fifteenth button turns the Registration Holes toolbar on and off. Registration holes are used by the DieSaw and other die-making machines to manufacture dies.

The **All On** and **All Off** buttons turn on or turn off all the toolbars, respectively.

Manufacturing toolbars

The standard Toolbar Master Control in Manufacturing works similarly to the one in Designer, but it has additional c;ontrols for the Manufacturing toolbars.



 The first button in the Manufacturing Toolbars group controls the appearance of the **Manufacturing** toolbar. The tools on this toolbar select, delete, and repeat elements, as well as creating the dieboard edge, adding a burn name, and working with bridges and nicks.

 The second button controls the **Dieboard** toolbar. Tools on the Dieboard toolbar make stripping rules, evenly spaced scrap knives, balance knives, dieboard splits, and so on.

 The third button controls the **Stripping Board** toolbar. Tools on the Stripping Board toolbar make automatic area stripping, alignment holes, air holes, interference, and so on.

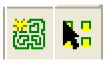
 The fourth button controls the **Manual Layout** toolbar. Tools on this toolbar control which single designs are used in the layout, the gutter distance between designs, and the position of the layout on the sheet.

 The fifth button controls the **Edit Layout** toolbar. Tools on this toolbar manipulate single designs and their position on the sheet.

 The sixth button turns the **Intelligent Layout** toolbar on and off. Tools on this toolbar let you use and configure Intelligent Layout and Costing/Estimating.

 The seventh button controls the **Rule Path** toolbar. Tools on this toolbar create, manipulate, and delete rule paths.

 The eighth button controls the **DieSaw** toolbar. Tools on this toolbar control drill holes and other DieSaw-specific features.

 The ninth and tenth buttons control the **Rubbering** and **Rubbering View** toolbars respectively. These tools let you build rubber along the lines in a layout, nest it, and lay it out in sheets for production, as well as letting you view different aspects of the rubber easily.

 The eleventh button controls the **Blanking** toolbar. Tools on this toolbar create upper and lower blanking boards.

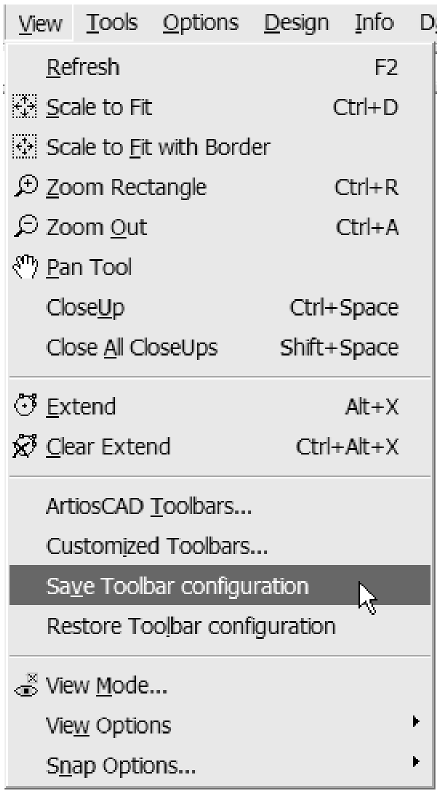
Availability of toolbars controlled by the Toolbar Master Controls depends on the options purchased. If you cannot click a toolbar activation button, the tools or features controlled by that toolbar have not been purchased.

Customized toolbars

The Customized Toolbar Master Control works the same way as the standard Master Toolbar Control. It will be empty until customized toolbars are created in Defaults. Once customized toolbars are created, use this control to turn them on and off like standard toolbars.

Toolbar positions

ArtiosCAD toolbar positions are saved automatically when ArtiosCAD is exited, but you can save them manually by clicking **View** > **Save Toolbar configuration**. You can then click **View** > **Restore Toolbar configuration** if they ever appear in the wrong place. Toolbar positions are stored in the Registry on a per-user basis.

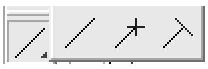


Flyout toolbars

Some toolbar buttons have small red triangles in the lower right corner.

This indicates that there are related tools available on a flyout toolbar. Clicking and holding the mouse button will make the flyout toolbar appear, and dragging and releasing over a tool will select it.

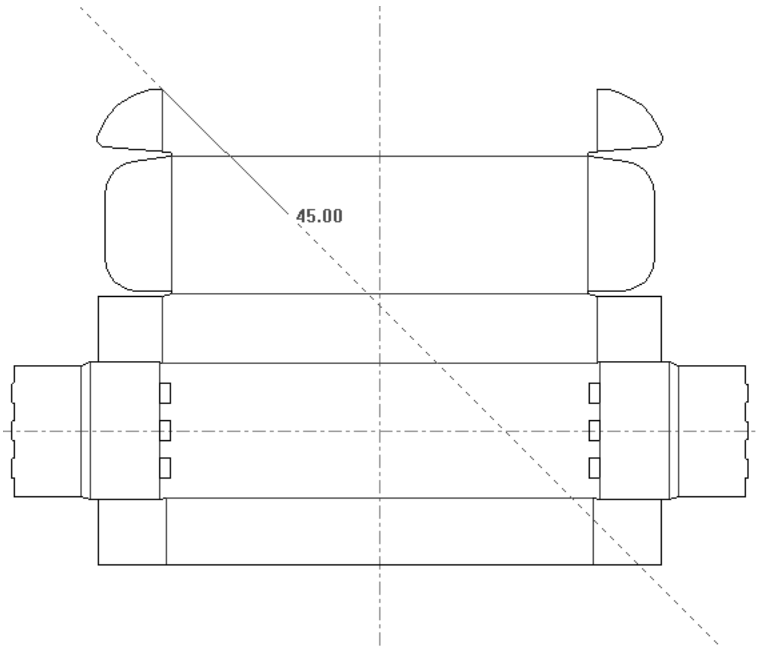
Shown below is the **Line** tool with its flyout toolbar activated.



When a tool on a flyout toolbar is selected, it becomes the tool shown on the regular toolbar. For that reason, all tools on flyout toolbars have the flyout indicator, but they do not lead to another level of flyout toolbars.

The drawing area

The drawing area is where the work is done. Think of it as a piece of paper onto which you are drawing a design. The **View Mode** command on the View menu (and on the Zoom toolbar) controls what objects are displayed and how they are displayed in the drawing area. Shown below is a picture of a design in the drawing area.



The Status bar

The fields and messages on the Status bar are the primary ways ArtiosCAD prompts for information needed to complete a task.

The picture of the Status bar above shows the prompts for a Line tool. The angle of the line, the length of the line, and the X and Y offsets of the endpoint from the current position can all be used to construct the geometry.

Move among the fields using the arrow keys on the keyboard, or click in them with the mouse. Next to each field is a symbol showing the direction in which the object being prompted for will be constructed.

Also next to each field is a keypad icon . Clicking the keypad displays an angle or length keypad and the variable keypad. Keypads are further described in the next section.

The Status bar also gives instructions. For example, when the Select tool is active, the Status bar tells you to hold down shift to add items to the current selection.

ArtiosCAD also uses a combination of prompts and instructions to gather information.

When using drag, the Status bar indicates what the drag will set. In the picture below, the drag is setting the value for an angle, and the Status bar indicates that by highlighting the field being set and by telling you what to do – set the angle.

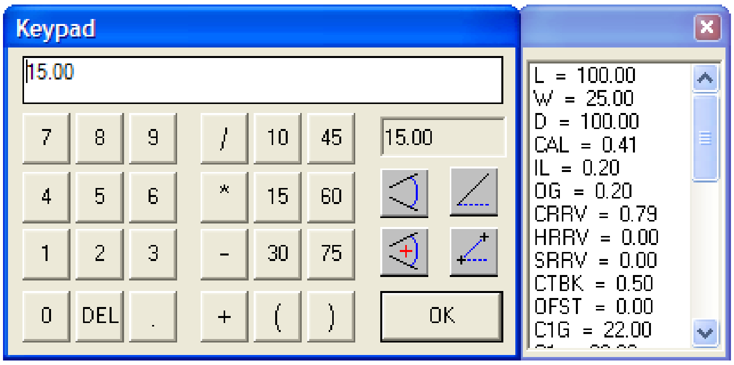
Keypads

In addition to typing values directly into fields on the Status bar, you can use a keypad to click number buttons or use the field to enter expressions longer than the short fields on the Status bar. ArtiosCAD uses one keypad for angles and another for lengths, radii, and so on.

The keypads follow the order of operations; expressions are evaluated from left to right, in this order: items in parentheses, powers, multiplication and division, and then addition and subtraction being computed last.

When entering information in the calculation field by clicking buttons or variables, ArtiosCAD adds the plus sign automatically if the cursor is at the end of the calculation field. When starting the field or operator by clicking a denominator button, ArtiosCAD automatically inserts 1 before the slash to act as a numerator.

Shown below is the angle keypad, with a list of current workspace variables and their values in a separate but attached dialog box to its right.

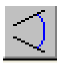


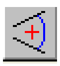
Common to both keypads, the number keys on the left side of the dialog box enter their numbers in the calculation field running across the top of the dialog box. Click inside the calculation field to type numbers, letters, and expressions. del deletes the last number, letter, or operator entered.

In the center group of buttons are the operators and common angle measurements.

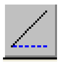
Along the right side of the angle keypad are the results field and the special angle measurement buttons. The result field displays the current results of the expression in the calculation field. If the expression cannot be evaluated (for example, if it ends with an operator), ArtiosCAD displays `err` in the results field.

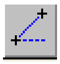
In the following feature descriptions, **absolute** means measured from the positive X axis in a counterclockwise direction. Also, using these four buttons replaces any values in the calculation field.

 **Angle between two lines** prompts you to select two lines and then inserts the resulting angle between them into the calculation field.

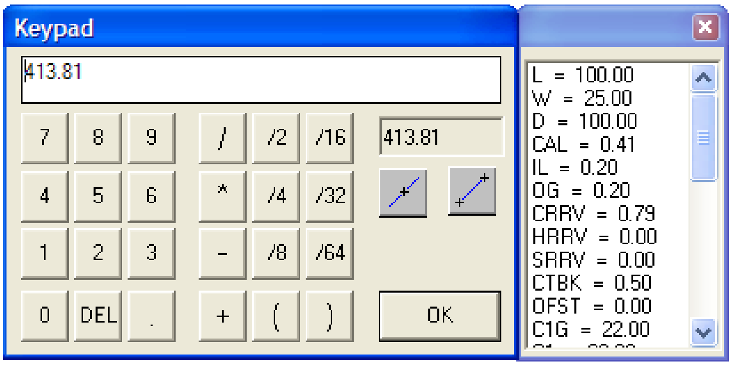
 **Absolute angle between two lines with reference** prompts you to select an initial line or point for reference, and then two lines. The angle of those two lines is either added or subtracted to the

reference line's angle depending on which side of the reference point or line was clicked. The concept is that you click a line and then add the angle of two other lines to it. This added angle is what appears in the calculation field. Note that the new angle is not absolute to the reference line; it was absolute when it was measured between the two other lines.

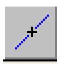
 **Absolute angle of a line** sets the value in the calculation field to the absolute angle of the line or endpoint clicked.

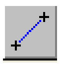
 **Absolute angle between two points** sets the value in the calculation field to the absolute angle of the selected two points.

Shown below is the length keypad, with a list of current workspace variables and their values in a separate but attached dialog box to its right.



As on the angle keypad, the left portion of the keypad has buttons for each numeral and del. The center of the keypad contains the operators plus commonly-used denominators for fractions. Along the right side of the length keypad are the results field and the special line measurement buttons. The result field displays the current results of the expression in the calculation field. If the expression cannot be evaluated (for example, if it ends with an operator), ArtiosCAD displays `err` in the results field.

 The **Absolute length of a line** button prompts you to click a line and then enters the expression for its length in the calculation field. The actual length is shown in the results field.

 The **Distance between two points** button in uses dimensions of existing geometry to set the value of a new piece of geometry. This is useful when the distance between two points is unknown, or if you want to make a rebuildable design.

Parameter sets

Parameter sets are collections of settings that govern aspects of the current design or manufacturing file. 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
- Bleed and Varnish offsets

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

- Die board size
- Default bridging formulae
- Settings for chop knives
- Mounting hole parameters

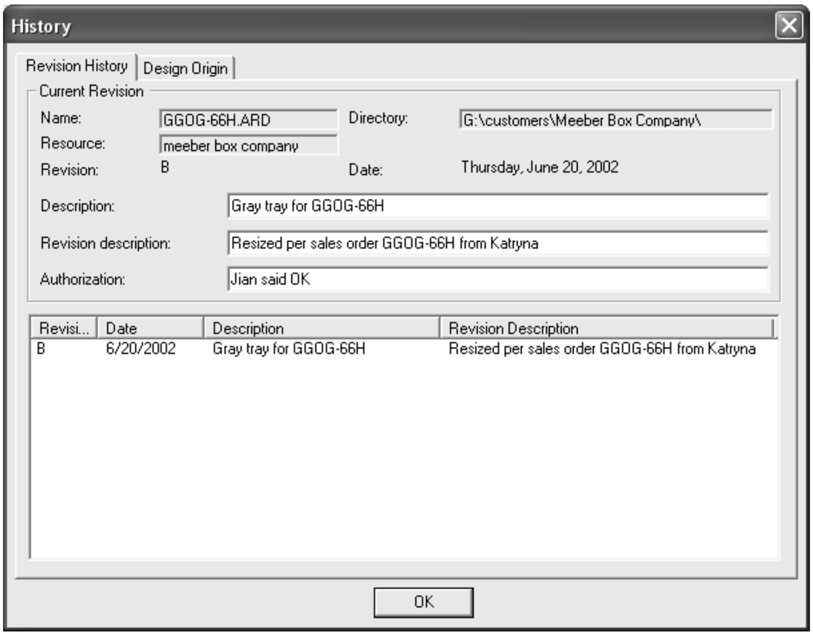
Many more options can be set using parameter sets.

Working with design files

Although ArtiosCAD has different modules, they manipulate design files in the same way.

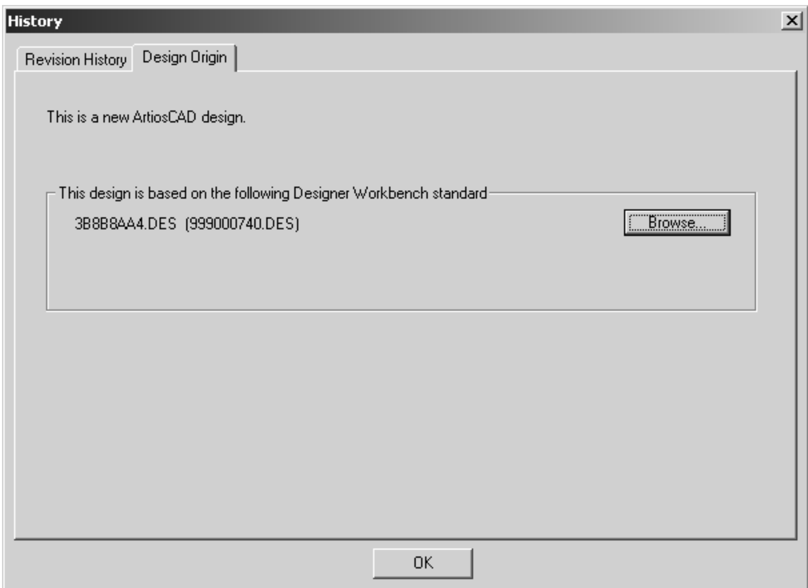
How to view revision history

To view the revision history of a file, click **Info** > **Revision History**. The History dialog box will open and show the revision history.



How to view the design origin

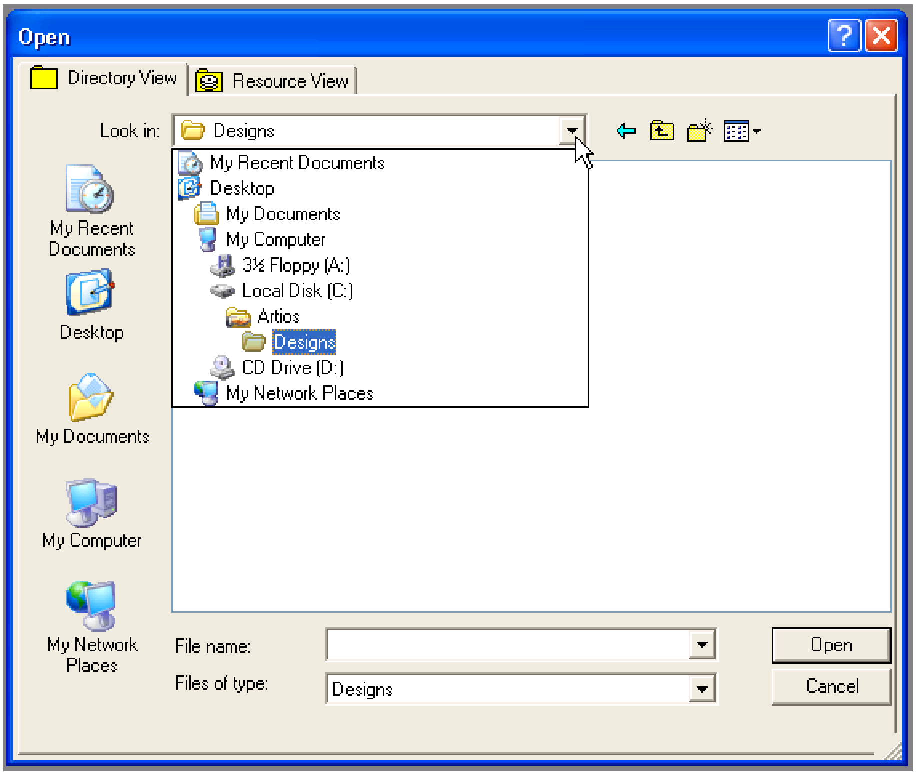
The **Design Origin** tab in the Revision History dialog box shows the name of the standard used to create a new design, if the design was made from a standard.



File views

ArtiosCAD's tight integration with DataCenter lets you open or save files in file system directories or database resources. There are two tabs in each dialog box that control where the file is saved to or opened from.

Directory view lets you navigate among the directories, hard drives, and network connections to choose the location for the file.

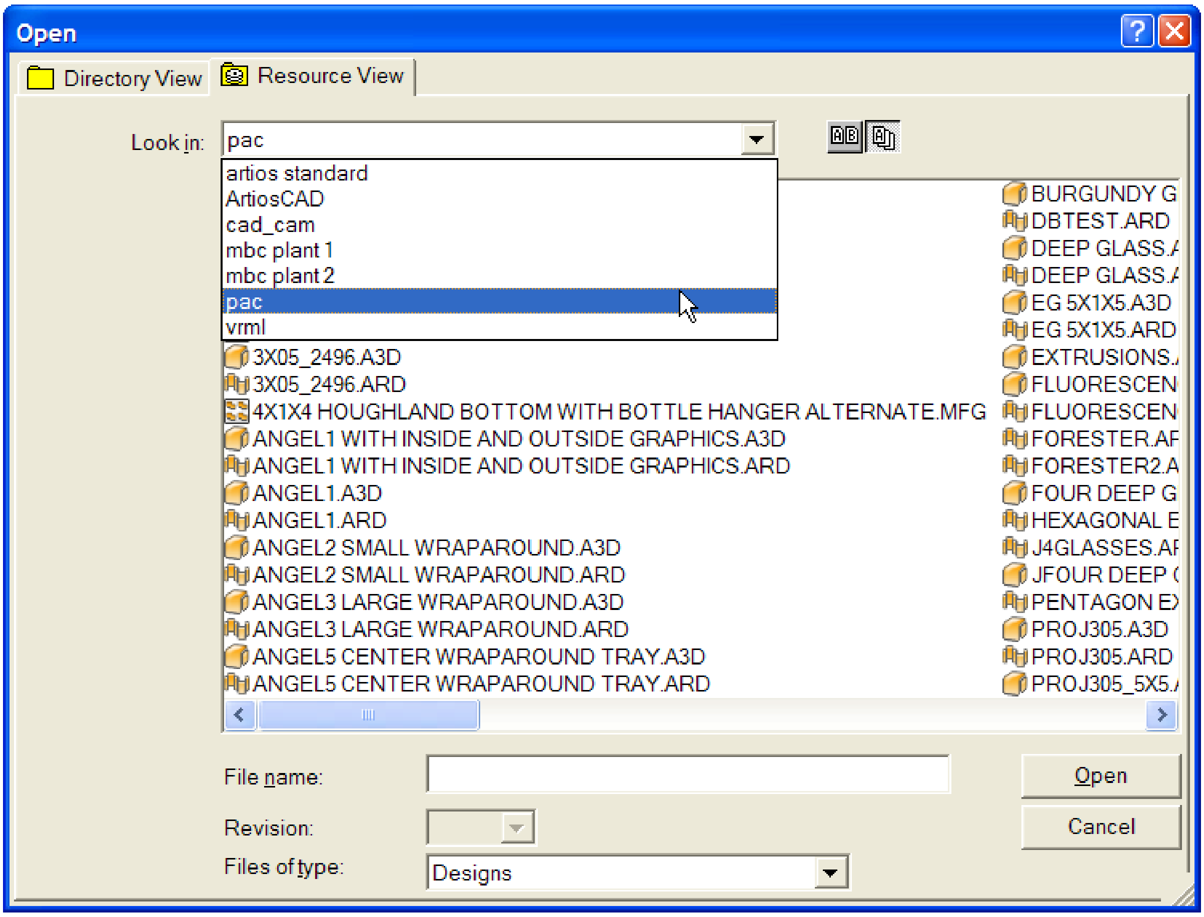


Resource view lets you navigate among database resources to choose the location for the file. Configure resources using DataCenter Admin; see the DataCenter chapter for more information.

Resource view also allows you to view all revisions of a file, or only the most recent version of the file.

Click  to view all versions of a design.

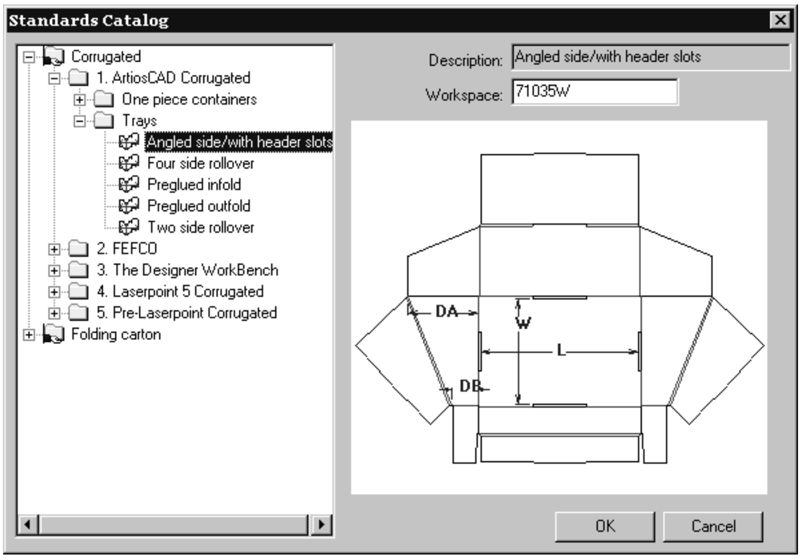
Click  to view only the most recent revision.



It may take some time for all filenames to display in resource view, particularly if there are Designer WorkBench files in that resource. If you know the name of a design, you may type it in the **File name**: field at any point without having to wait for the full list to appear.

Creating, opening, and importing designs

Creating, opening, or importing a design use the creative portions of ArtiosCAD. Running a standard is one way to create a new design. To run a standard, click **File**, then click **Run a standard**. The Standards Catalog will appear from which you may choose the standard on which your new design will be based.



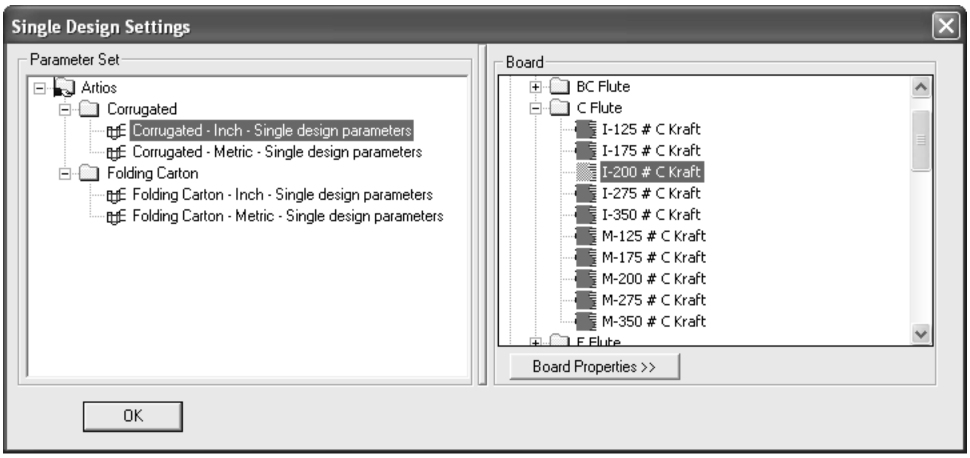
How to create a new blank design

Another way to create a new design is to click **File**, and then click **New Design**. Choose a parameter set and a board code, and click **OK**. This creates a new blank workspace; use the Designer tools to create geometry.

How to assign a parameter set and board code

Whether creating a design from a standard or from scratch, you must choose a parameter set and a board code. A board code is a group of settings that tells ArtiosCAD various things about the material used to manufacture the design. This can include information such as the thickness of the board, its weight, cost, inside loss, outside gain, scant, and allowance.

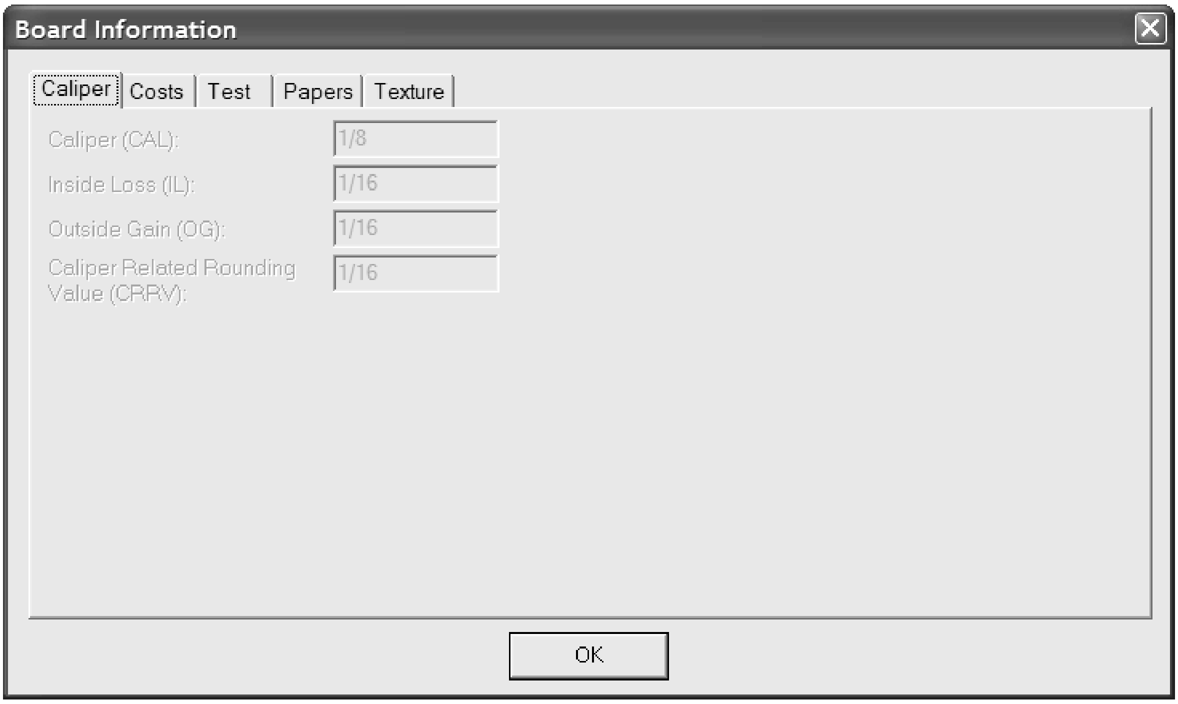
Select a board code at the same time you select a parameter set. Select the board code you want to use and then click **OK** (if creating a blank design) or **Next** (if running a standard) to continue creating the design.



Drag the splitter between panes to adjust their sizes.

Clicking **Board Properties** shows information about the selected board. Board information is extracted from the database and thus cannot be changed in ArtiosCAD if the database is available.

Only when the database is unavailable are the fields available for changing. Click **OK** to close the dialog box.



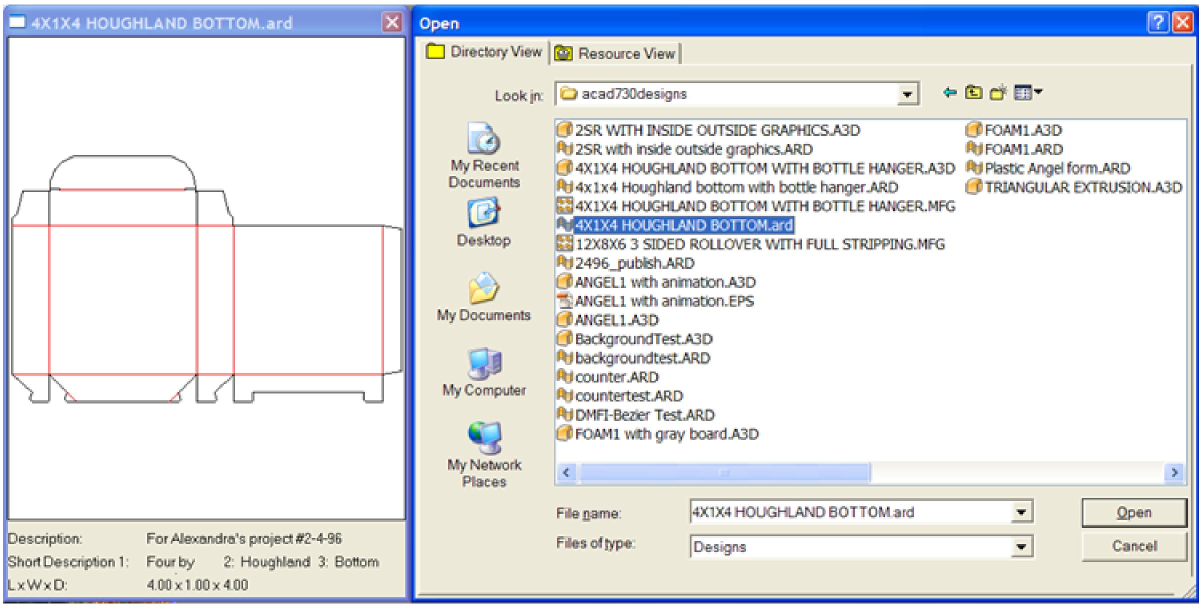
How to open an existing design

To open an existing design, click **File**, and then click **Open**. Use the **Directory View** tab to navigate using directories, or use the **Resource View** tab to navigate using database resources. Click the filename of the design to open, and click **Open**.

To open more than one design at once, use Directory View, click the first design to open, hold down `CTRL`, click the other files to open, and then click **Open**.

There is a limit of 2562 characters in the **File name**: field when opening designs.

The design preview is shown next to the Open dialog; the **Description:**, **Short Description 1**:, **Short Description 2**:, and **Short Description 3**: fields and the dimensions (if known) are shown beneath the preview. The design preview can be resized as desired, but if it is too small, some of the text shown will be truncated. When selecting multiple designs to open, only the first design selected is previewed.



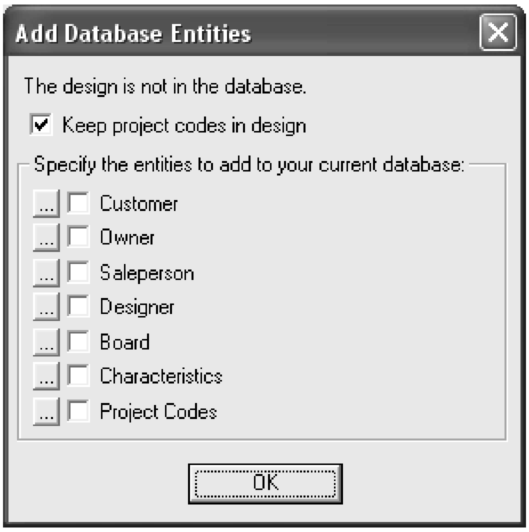
Upon opening, the design is initially displayed according to the View Mode defaults in **Options** > **Defaults** > **Design Defaults** > **Default View Mode**. In that dialog box are settings for line attributes, nick style, high graphics mode, screen background color, plotting style, and whether the layers shown as are saved or the default settings.

It is easiest to open Designer WorkBench designs in Resource View. That way you can navigate to the proper resource and the DWB hexadecimal disc filename will be displayed as the DWB design number.

Designer WorkBench single designs which contain stepping information will use that data to create a layout when the single design is opened and converted to a manufacturing file.

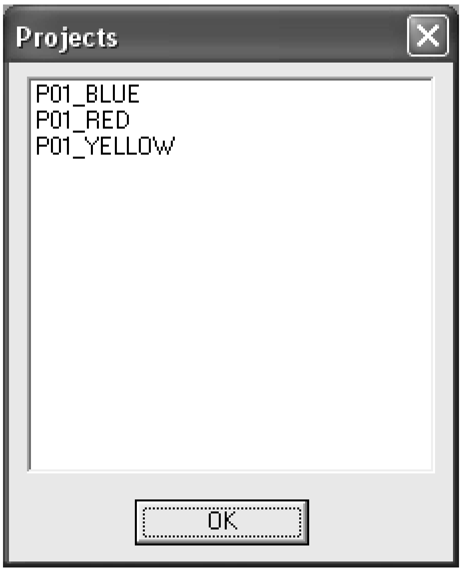
The Scant variable in workspaces originally created in Designer WorkBench is mapped to either the Outside Gain (OG) or Inside Loss (IL) variable via the setting in **Options** > **Defaults** > **Design Defaults** > **DWB Options**.

When you open a single design containing records from another database, the Add Database Entities dialog box appears. In this dialog box, choose which entities to add to your database by checking the checkbox next to each available category. The enabled categories depend on which entities are set in the single design.

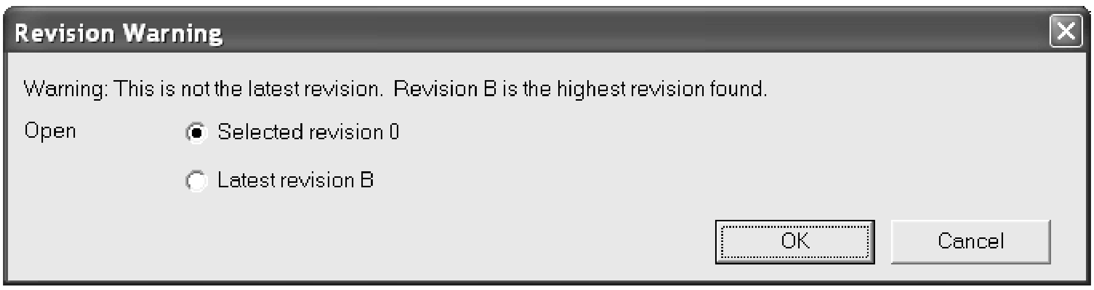


The **Keep project codes in design** checkbox controls whether or not ArtiosCAD leaves the Project codes from the foreign database in the single design or if it strips them out.

To see the specific entities for each category of database entities, click **...** next to the checkbox. All entities will be added if you check the checkbox; there is no way to choose which specific entities are added to the database. Specific Project entities are shown below; click **OK** to return to the Add Database Entities dialog box.



If a later revision of the design exists, you are prompted to choose to open the selected revision, or the latest revision. Choose the desired choice and click **OK**.



Opening a design that is already open

If you open a design that is open in another session of ArtiosCAD, ArtiosCAD warns you that it is already open, and also who opened it and when it was opened.

Click **Open Read-Only** to open the file in read-only mode. Using **File** > **Save** on a read-only file generates a warning message. You must use **Save As** to save the file with a new name.

Click **Cancel** to not open the file.

If you open a file as read-only, the ArtiosCAD window title bar displays (READ_ONLY) as a reminder.

ArtiosCAD tracks open files by creating a hidden **filename.alck** file in the same directory as the open file. If you do not have write permission in the directory, the lock file cannot be created and the file will not protected against opening by other sessions.

To open a file directly in read-only mode, click **File** > **Open Read-Only**.

Note: If you choose the original filename when doing a **Save As**, ArtiosCAD will overwrite the original file with the new file if the file is no longer locked. If the file is still locked by another instance of ArtiosCAD, you may not save the file with the same name.

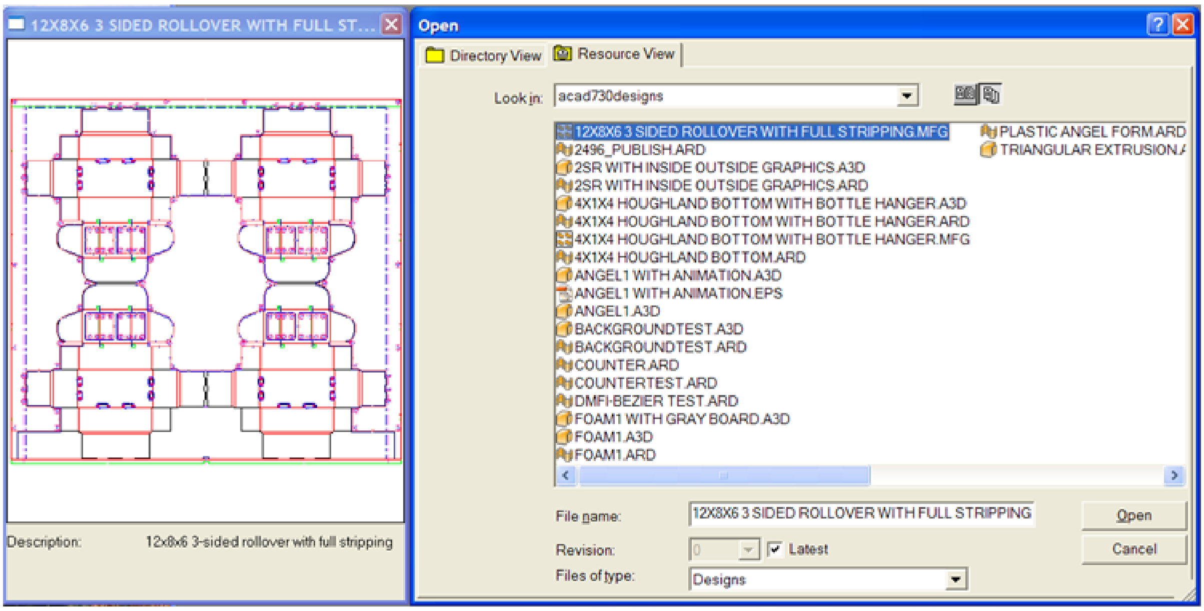
How to open a design on a Web server

To open a design that is saved on a Web server, click **File** > **Open URL** and enter the URL for that design, e.g. `http://artcad.mycompany.com/myfilename.ARD`.

How to open an existing manufacturing file

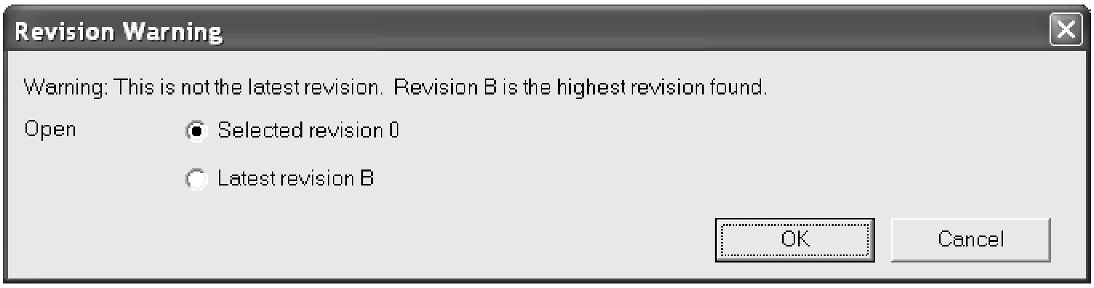
To open an existing manufacturing file, do the following:

1. Start ArtiosCAD.
2. Click **File** > **Open**.
3. Using either Directory view or Resource view, navigate to the location in which the manufacturing file is saved.
4. Click the name of the manufacturing file to select it. A preview of the manufacturing file appears just as it does for a single design.

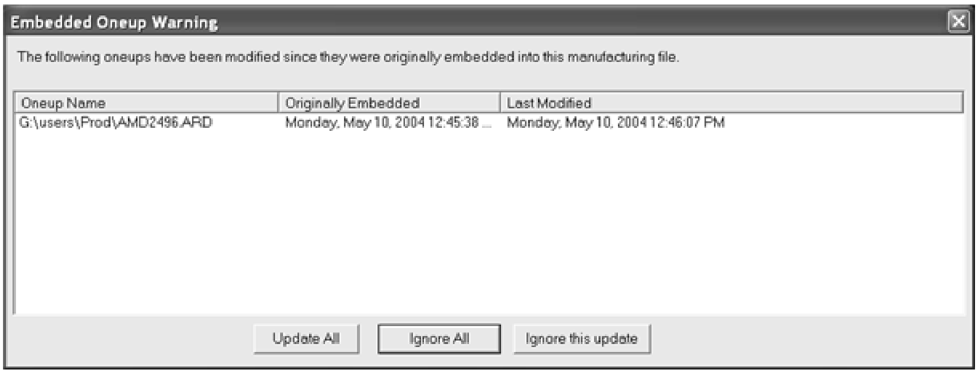


5. Click **Open**. The manufacturing file is opened.

If a later revision of the manufacturing file exists, you are prompted to choose to open the selected revision, or the latest revision. Choose the desired choice and click **OK**.



When opening a manufacturing file, ArtiosCAD checks if a disk file exists with the same name as the embedded single design(s). If such a file exists, its creation date and time are compared with the embedded single design(s). If the two are different, ArtiosCAD prompts you to select the one to use.



Update all replaces the older embedded single designs with the newer versions. ArtiosCAD checks for double knives in the manufacturing file, double lines in the single designs, and to see if the single designs have different origins, sizes, sides up, or grain directions from those they are replacing.

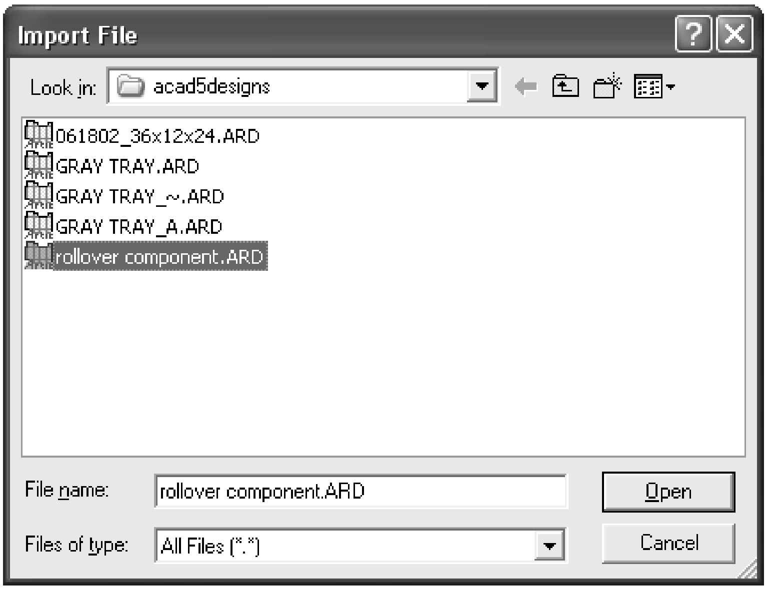
Ignore all keeps the embedded single designs and ignores the newer versions. The warning will appear again the next time the manufacturing file is opened.

Ignore this update keeps the embedded single designs and saves the new modification date in the manufacturing file so no warning occurs the next time the manufacturing file is opened.

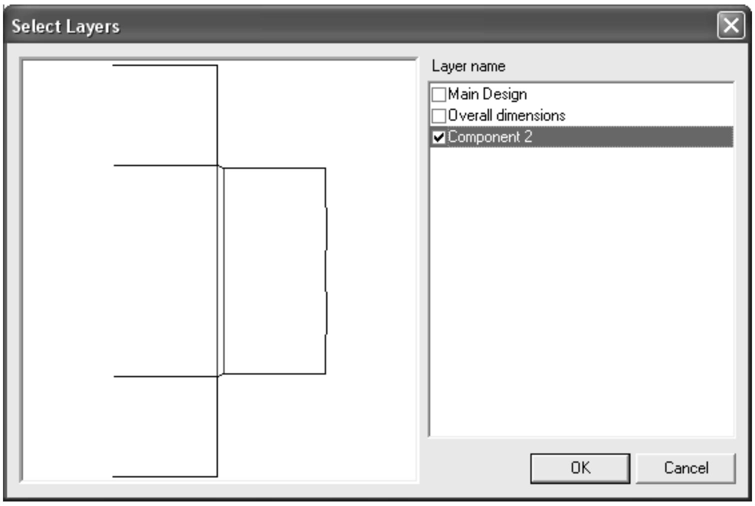
How to import an existing ArtiosCAD design into another

To import an existing ArtiosCAD design into another, do the following:

1. Start ArtiosCAD and open the design into which the other design will be imported.
2. Click **File** > **Import File**.
3. Select the file to import and click **Open**.



4. In the Select Layers dialog box, select the layers to import into the design by checking or clearing the checkboxes next to the layer names as appropriate. Click **OK** when you have chosen the layers of the design to import.



5. The selected layers of the design will be imported into the current workspace. All of the imported objects will be selected and the **Move** tool will be active. Click to set a pick-up point and click again to set a put-down point. To cancel the move, select another tool.

How to import a design in a different format

Opening a design in a different format, such as DDES, DXF, EPSF, or CFF2, is done in the same way as opening a native ArtiosCAD design file. Click **File**, and then click **Open**. Navigate to the proper directory using the standard controls, click the filename of the design to open, and click **OK**.

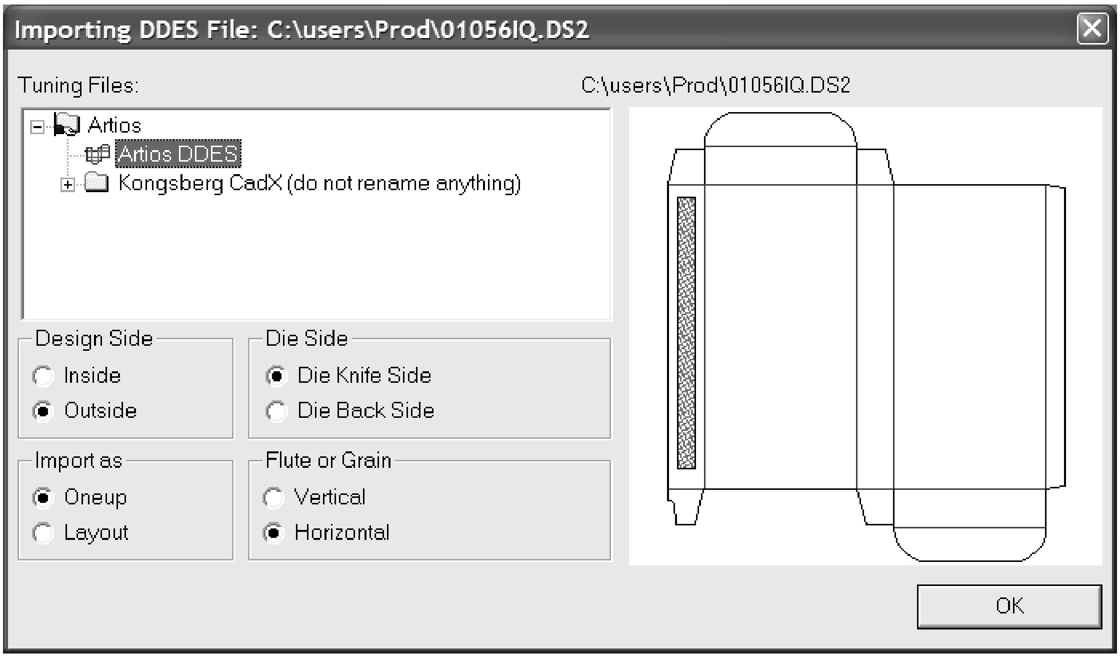
If your filename does not have one of the standard extensions (.DS2, .DXF, .EPS, .CFF, .N, etc.), then you must change the entry in the **Files of type** list box to **All files** in the Open dialog box. If your file does not have one of the standard extensions, it will not appear in the list of files available unless you change the listing to **All files**.

Line type mappings and other translations for each file type are configured in **Defaults Shared defaults Import Tuning Table**.

A preview of the file selected for import appears next to the File Open dialog box.

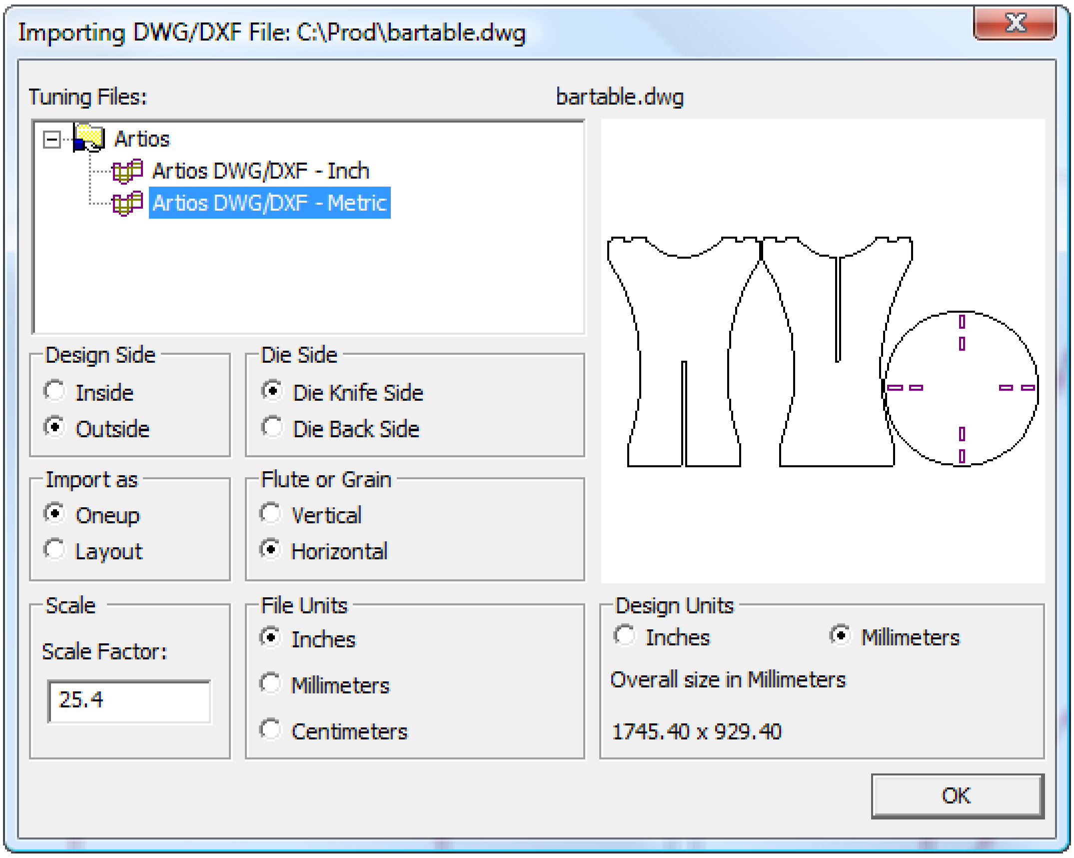
Importing a DDES file

When you open a DDES2 or DDES3 file, the Importing DDES File dialog box (or Importing DDES3 file dialog box) appears as shown below. Set the option buttons as desired, click **OK**, and choose the desired parameter set and board code to import the file.



Importing a DWG file

When you open a DWG file, the Importing DWG/DXF File dialog box appears. Set the option buttons as desired and click **OK** to import the file.

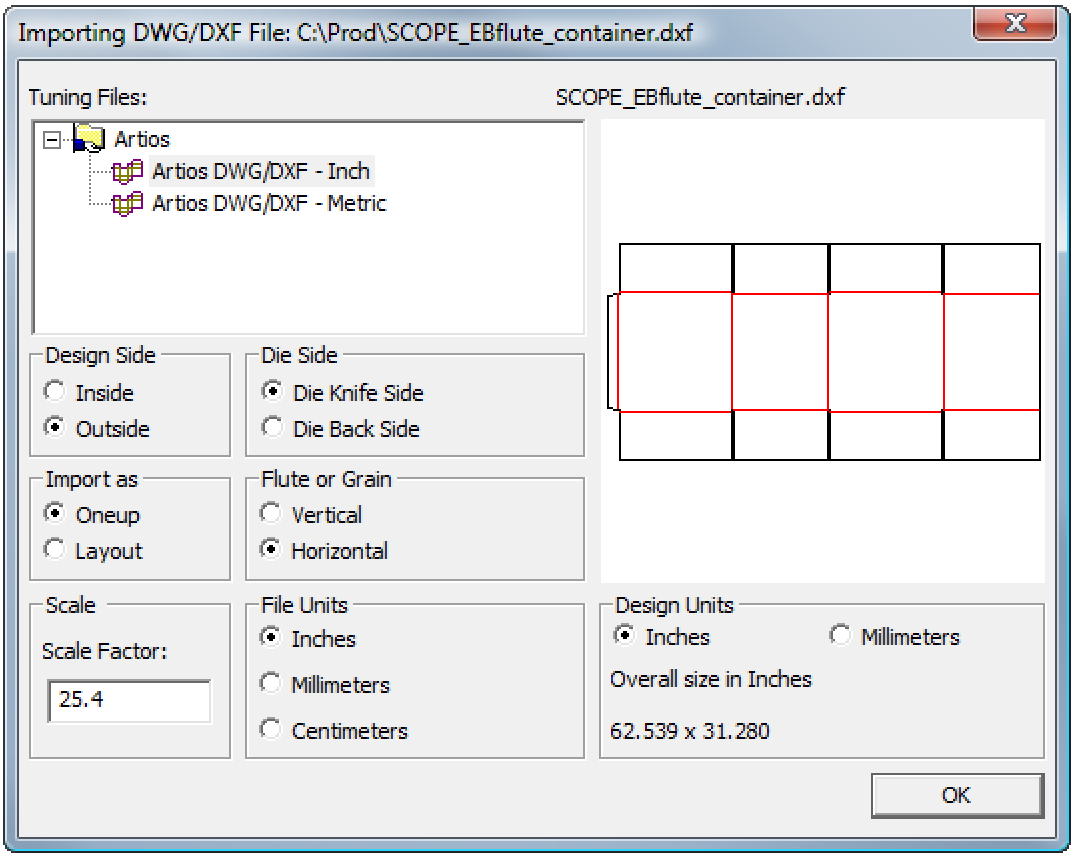


Note: ArtiosCAD ignores any 3D data in the file.

Importing a DXF file

When you open a DXF file, the Importing DWG/DXF File dialog box appears.

Set the option buttons as desired and click **OK** to import the file.

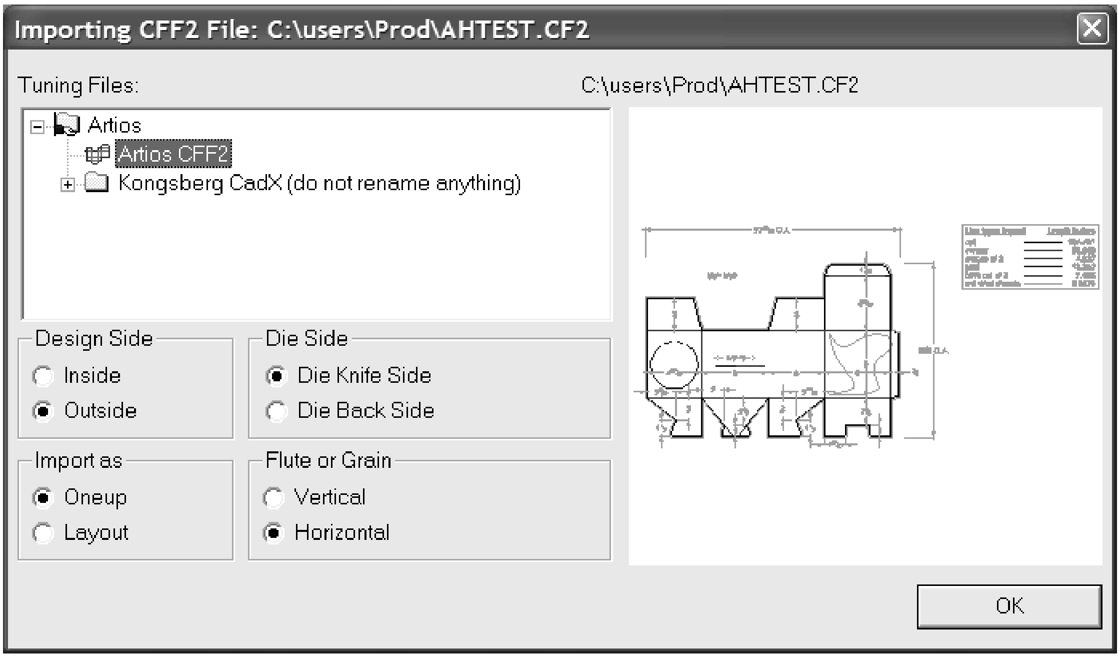


Note: ArtiosCAD ignores any 3D data in the file.

Note: If you have trouble importing a DXF file, try clearing **Import DXF files using third party libraries** in **DWG/DXF Options** in Startup defaults.

Importing a CFF2 file

When you open a CFF2 file, the Import CFF2 File dialog box appears. Set the option buttons as desired and click **OK** to import the file.



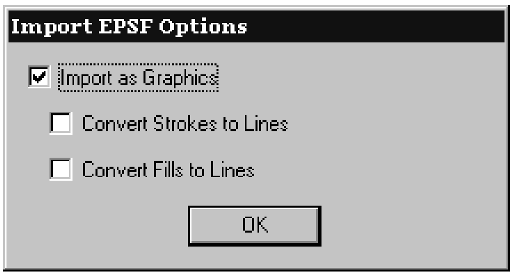
When opening a CFF2 layout, ArtiosCAD can base the size of the sheet on the lower left and upper right values contained within the file. This is configured by the **LL/UR values to contribute to sheet size** checkbox in **Defaults** > **Shared defaults** > **Import Tuning Table** > **Artios CFF2** > **Options**. Rotation and mirroring options are also set in this dialog box, as is the transformation order and the minimum blank area needed to consider the CFF2 file a layout. The default standard transformation is to rotate and then negate, but you can choose to negate then rotate in the Elcede style.

Importing an EPSF/AI file

When you open an .EPSF or .AI file, the Import EPSF Options dialog box appears.

Import as Graphics imports strokes and fills unchanged. **Convert Strokes to Lines** and **Convert Fills to Lines** change fills and strokes to print image lines and places them into overlays whose names are the color of the fills/stroke and the thickness of the strokes in points.

Set the checkboxes as desired and click **OK** to import the file.



You can import only Illustrator 3.2 and Illustrator 8.0 EPSF files. TIFF previews are ignored. Other EPSF file types are not supported. Some .AI files are supported using EPSF import functionality while others require the PDF option to open. If you receive a Feature not available message while trying to open an .AI file, the file is likely in the format that needs the PDF option.

If the EPS file has more than 8 MB of vector data, it will be lost when the design is converted to 3D. Use the JPEG format or PDF instead of EPS if working with large files.

Adobe® Illustrator® ArtiosCAD Import plug-in

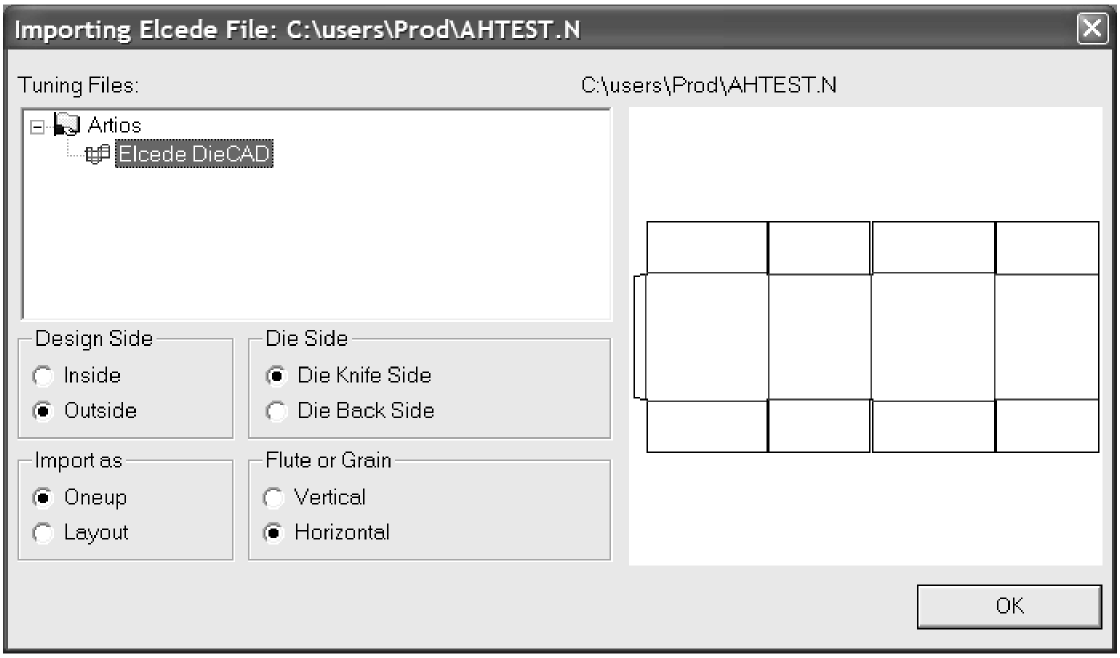
The free Adobe® Illustrator® ArtiosCAD Import plug-in lets Adobe Illustrator users place ArtiosCAD workspaces in Illustrator documents. It does not include placed EPS objects or other pixel-based objects. The plug-in can be downloaded from Esko's Web site at `http://www.esko.com` and is also on the ArtiosCAD media.

Contact your local Esko salesperson for information about purchasing the ArtiosCAD Export plug-in for Adobe Illustrator.

For more information on the plug-in, refer to the documentation that is included in its installation package.

Importing Elcede .N files

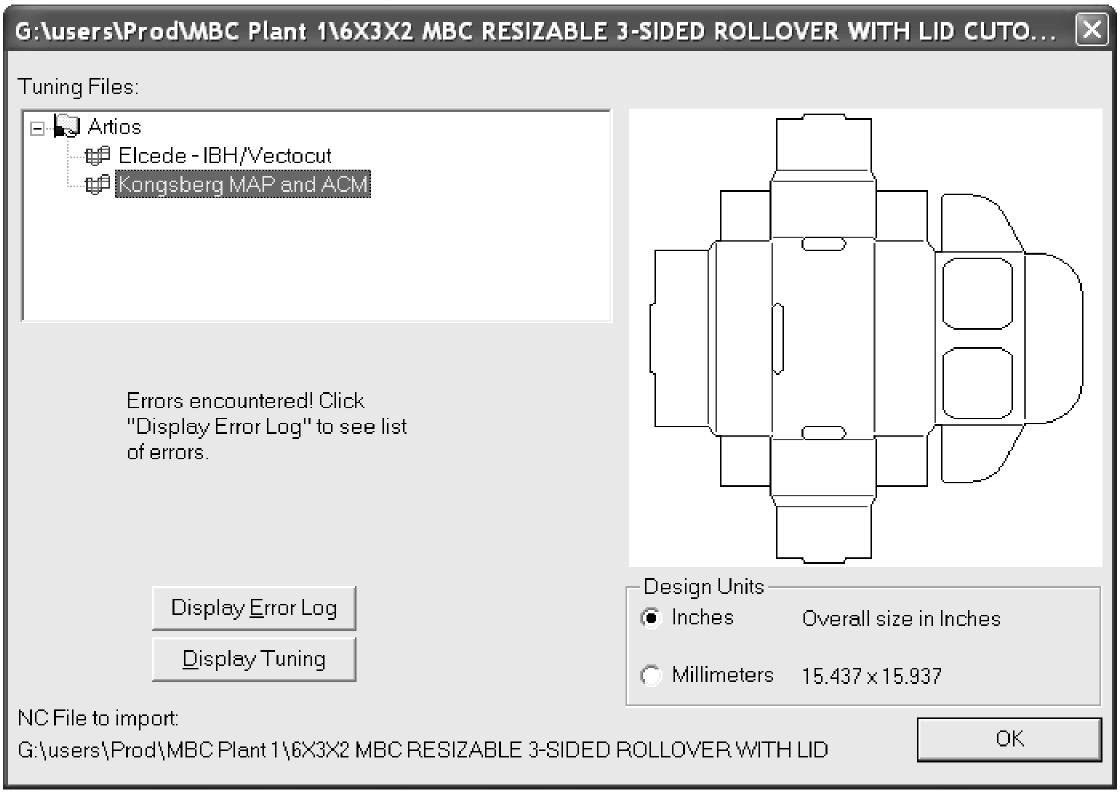
When you open an Elcede Diecad .N file, the Import Elcede File dialog box appears. Set the option buttons as desired and click **OK** to import the file.



Note: ArtiosCAD can not open Diecad files with more than approximately 1000 lines in a section. If you encounter such a file, export the file from Diecad in CFF2 format, and read the CFF2 file into ArtiosCAD.

Working with other NC files

If you have Numeric Control files for which there are no native ArtiosCAD import filters, such as Elcede IBH/Vectocut files or Kongsberg MAP and ACM files, you can try using the **Open NC file** command on the Diagnostics menu in Single Design. This command is not supported; use it at your own risk. For help importing files of these types, contact Professional Services.



Opening a PDF file as lines

ArtiosCAD can now import the lines (also called *vectors*) in a PDF file. Previous versions could only import a PDF file as a graphic entity. Once the PDF file is opened, you can then work on the lines as you would any other structural workspace.

Before you can import a PDF file as lines, you must make sure there is at least one tuning catalog entry set up in Defaults to process PDF files. ArtiosCAD comes with two different tuning catalog entries as examples, but it is likely you will need to make a tuning catalog entry for each supplier of PDF files based on how they create the files.

Opening the PDF file

To open the PDF file, do the following:

1. Start ArtiosCAD.
2. Click **File** > **Open**.
3. Navigate to the directory or resource containing the PDF file.
4. Select the desired file.

   A preview of the file will appear.
5. Click **Open**.

   The **Importing PDF File** dialog box appears.

Choosing Import Options

In the **Importing PDF File** dialog box, do the following:

1. In the **Tuning Files** group, select the tuning file to use for this PDF file. The preview of the design may change depending on the tuning file you choose.
2. In the **Design Side** group, choose whether the side up is the **Inside** or **Outside**.
3. Set the options similarly in the **Die Side**, **Import as**, **Flute or Grain**, and **Design Units** groups.
4. Click **OK**.

Choosing Single Design Settings

In the **Single Design Settings** dialog box, do the following:

1. Select a parameter set and board to use in the new workspace.
2. Click **OK**.

The PDF file opens in ArtiosCAD according to the tuning you selected. You can work on it as you would any other single design.

PDF vector import notes and warnings

Please consider the following notes and warnings when importing PDF files as lines.

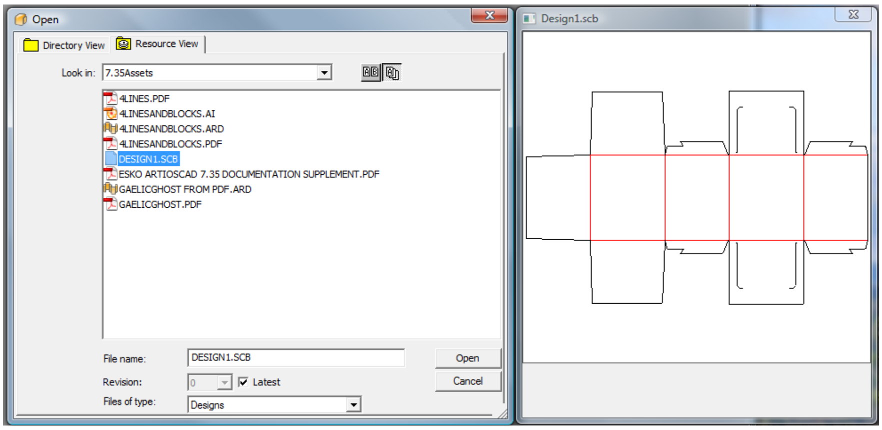
- Bitmaps in PDF objects come in as their net shape of their clip path (the outline of the part of the image that is rendered) and have graphic type **bitmap**. To preserve the bitmap image, import the PDF file as graphics rather than lines. The actual pixels of the image are not imported. Note: some bitmaps in PDF are not defined with a clipping path that is the shape of the thing in the bitmap. An example is the bitmap for the shadow in an ArtiosCAD Plot to PDF with shadows turned on. In these cases, the bitmap outline comes in as a rectangle.
- ArtiosCAD has a 99 layer limit. Therefore, complicated PDF files (those with more than 99 unique sets of import attributes) will typically reach this limit if they use a tuning that maps “any any any” to New Layer.
- When **Unfiltered items to graphics layer** in the **PDF Import Options** dialog box is enabled, a stroke's cap style, miter style, and miter limit are not preserved. All strokes in this situation appear with rounded join style and square cap style.
- When **Unfiltered items to graphics layer** in the **PDF Import Options** dialog box is enabled, the strokes and fills may appear underneath things (masked) even though they are visible in the original PDF file.
- When **Unfiltered items to graphics layer** in the **PDF Import Options** dialog box is enabled, objects with any coloration other than a single color (such as gradient fills, mesh objects, and so forth) appear as a solid color fill which depends on how the file was created. Gradient fills from Adobe Illustrator-created files generally come in as black fills.

How to open a Score! file

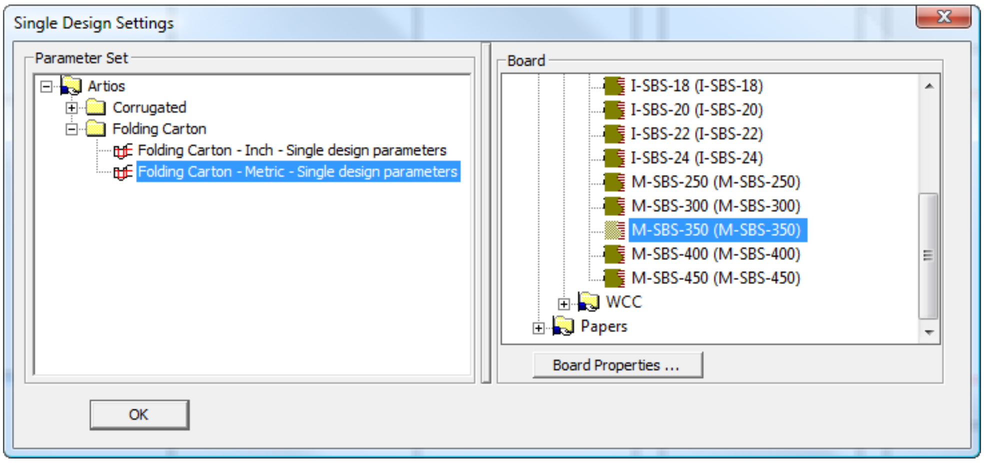
To open a Score! or Vellum file in ArtiosCAD, do the following:

1. Start ArtiosCAD.
2. Click **File** > **Open**.
3. Using either **Directory view** or **Resource view**, navigate to where the Score! file is saved.
4. **Files of type** should be set to **Designs**. If it is not, click the drop-down arrow and set it to either **Designs** or **Score!**.
5. Click the name of the Score! file to select it.

A preview of the Score! file appears.



6. Click **Open**.
7. In the **Single Design Settings** dialog box, select the desired parameter set and board code, then click **OK**.



ArtiosCAD opens the file and you can work on the file as you would any other design.

More information about opening a Score! file

The following sections contain reference information to consider when opening a Score! or Vellum file in ArtiosCAD.

Score! file import restrictions

ArtiosCAD can open Score! and Vellum files with the following restrictions:

- ArtiosCAD does not import data from the Score! database about the file being opened.
- ArtiosCAD maps bridges on ellipses onto the arcs as closely as possible to the original file, but ArtiosCAD tries to keep the count and position of the bridges even across the series of arcs forming an ellipse.

Score! file model structure

A Score! file generally contains at least two models: `Model 1` for the flat design and `Foldup` for the folded design. The contents of `Layer1` in `Model 1` go into the `Main Design` layer of the single design. All other layers in the Score! file go into layers of the same name in ArtiosCAD of layer class `Annotation`. If there is a layer named `Dimensions` in the Score! file, ArtiosCAD makes a `Dimensions` layer of class `Dimensions` in the ArtiosCAD workspace.

When ArtiosCAD opens a Vellum (.vlm) file containing multiple models, those models are imported into separate layers.

Score! line type mapping

ArtiosCAD automatically converts Score! line types to native linetypes per the entries in the following table.

Score! line type and name	ArtiosCAD line type and name
0, Annotation 10, Crosses 113, Chopper 115, Tester	0, Annotation
101, Knife	1, Cut
102, Crease	2, Crease
103, Perforation	3, Perf
104, Cutscore	4, Cut-Score
105, Cutcrease	5, Cut-Crease
106, Cutscore_Crease	15, Partial cut and crease
107, Reverse Cutscore	9, Reverse partial cut
108, Emboss	16, Half crease
109, Zipper 110, Wave	13, Generic special rule
112, Stripping	93, Stripping rule
80, Construction	34, Construction line
All others	0, Annotation

Score! bridging

A Score! file may have one of two different types of bridging: simple or complex.

For a file with simple bridging, ArtiosCAD takes the number of bridges specified and their width and uses those as the parameters for applying its `Evenly spaced` bridging formula.

For a file with complex bridging, ArtiosCAD uses the `Specify bridge positions` bridging formula. The first bridge width defined in the Score! file sets the width for all the bridges in the ArtiosCAD workspace. If the Score! file contains any double-width bridges, they are converted to two bridges separated by a slight gap.

Score! text

ArtiosCAD imports text from Score! files as text with the following limitations:

- Line wrap is ignored; a line will be extended as long as necessary.
- Text color is ignored.
- Line spacing is ignored.
- Underlined text is not supported.

Score! dimensions

ArtiosCAD imports the following types of basic dimensions from Score! files: `Horizontal`, `Vertical`, `Parallel`, `Radial`, `Diameter`, and `Angle`. `Arc Length` dimensions will be ignored.

Score! uses `#` to represent the real value of a dimension; ArtiosCAD changes this to `{V}` (also the real value of the dimension) except in the following cases:

- For a `Distance` or `Angle` dimension with just `#` in the dimension text.
- For a `Radius` dimension with exactly `R #` in the dimension text,ArtiosCAD will not show any dimension text, but the format will be set to **R50**.
- For a `Diameter` dimension with exactly `0 #` in the dimension text, ArtiosCAD will not show any dimension text, but the format will be set to **Ø50**.

The table below shows how ArtiosCAD imports other aspects of Score! dimensions.

Score! dimensional element	How ArtiosCAD imports it
Text positioning	Uses closest matching positioning options
Arrowhead style	Uses closest matching
Arrowhead thickness	Imported
Dimension font and size	Imported
Dimension color	Ignored and set by the plotting style
Extension length and gap	Calculated from Score! settings
Units and formatting	Set by ArtiosCAD property defaults
Internal dimension extension lines	Not drawn when dimension text is outside the measured object

Score! fold angles

Most Score! standards are saved with a flattened `Foldup` model and with fold angles assigned to the creases.

During the import, ArtiosCAD copies fold angles and the base face from either the `Foldup` model or the **Foldup** layer from `Model 1`, to the creases in the **Main Design** layer.

The import of fold angles will not work if the `Foldup` model or layer does not exist or if the `Foldup` model has been folded up in 3D space.

Note:

if there are gaps in the design perimeter, ArtiosCAD cannot automatically convert the design to 3D.

Saving and exporting designs

Saving a design means that the work you have done in ArtiosCAD is written to a file on the hard drive for future use in ArtiosCAD. ArtiosCAD files are binary files that can be read only by computers running select Esko applications.

Exporting a design means that you are essentially taking a snapshot of the design and converting it to a different format for use by someone who does not have ArtiosCAD. Most common export formats are text formats.

How to save a design

To save a design, click **File**, and then click **Save**. If you are saving the design for the first time, ArtiosCAD will prompt you for the name of the file by displaying the Save As dialog box (see the next section). Do not use an underscore (_) as the last character of the filename, for that is used to determine revision history. If you have already saved the file, ArtiosCAD will save the design using the name shown in the title bar of the window.

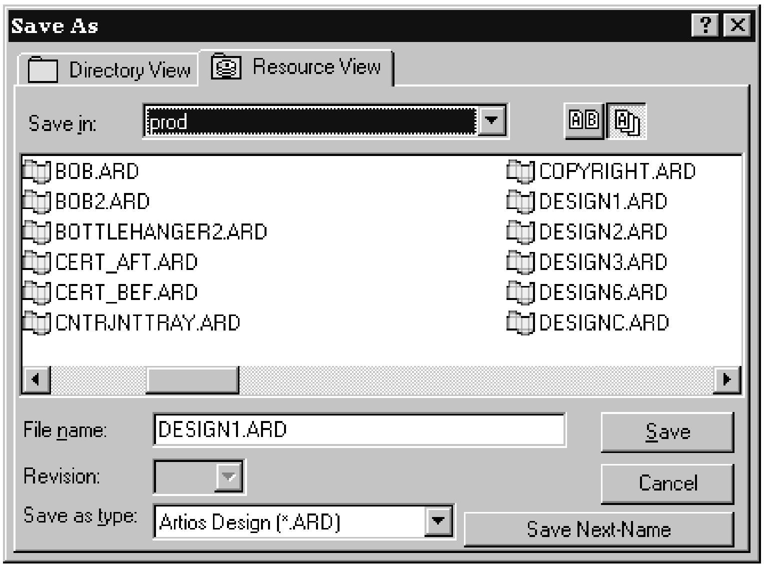
If **Design check before Save, Save as** is checked in **Options** > **Defaults** > **Shared Defaults** > **Design Defaults** > **Save Options**, the design is checked for double lines and gaps in the perimeter of the design when it is saved. This option is off by default.

You will also be prompted to enter database information for this design. When you save a design, ArtiosCAD automatically supplies DataCenter with some information about the design, but you must specify other information such as the customer name and the designer. Your system administrator may choose to customize the information requested when saving a design. If the design is not rebuildable, specify the length, width, and depth if desired. When you are done specifying information, click **OK** to save the database information and the design.

If you have opened an existing file, when you save it, ArtiosCAD checks to see if the original file has been modified before it overwrites it. This will warn you if another user has been working on the same file at the same time to prevent accidental loss of work.

How to save a design with a different name

Click **Save As** on the **File** menu to save the current design with a new name and start working with the new workspace. Navigate to the desired folder or resource, type the new name in the **File name**: field (do not use an underscore as the last character) and click **Save**.



Files saved using **Save As** use the date and time at save for the creation date and revision date in the database record, regardless of the date and time at which the original file was created.

How to save a copy of a design

To save a copy of the current workspace with a new name but continue working with the current design, click **File** > **Save Copy As**. The Save As dialog box will appear. Navigate to the desired folder or resource, type the name for the copy, and click **Save**. ArtiosCAD will save the copy and you may resume working on the current workspace.

How to save a design using automatic numbering

To save a design using automatic numbering, you must have configured automatic numbering as described in the *DataCenter* chapter. You must also be saving the design (or manufacturing file) in a resource for which automatic numbering is configured.

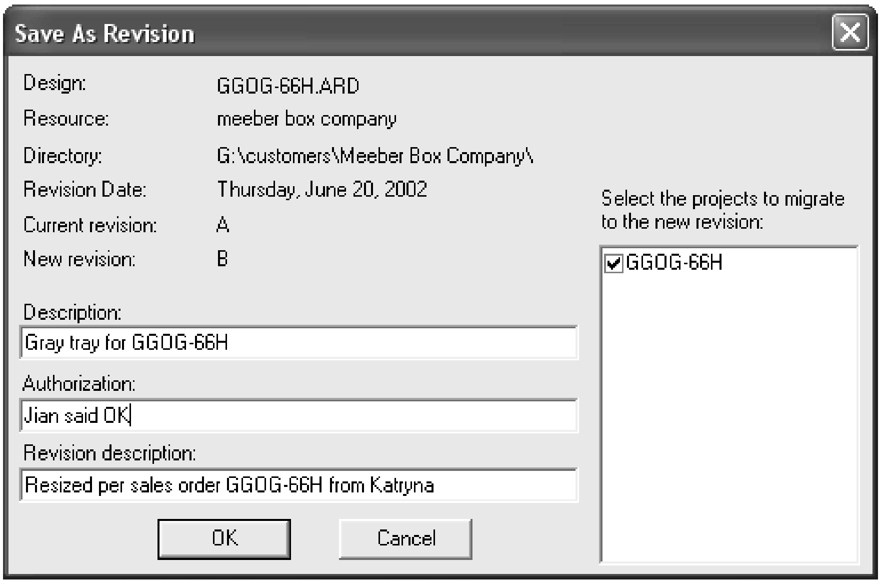
Once that is set up, the **Save Next Name** button in the Save dialog box is available as shown a few sections previously. It is unavailable for resources which do not have Automatic Numbering configured. When you click **Save Next Name**, ArtiosCAD will automatically save the file with the next available number according to the configuration in DataCenter.

How to save a revision of a design

Use the **Save As Revision** command on the File menu to save different versions of the same design. Each revision of the design file shares the same basic name, but there is a progress indicator at the end of the filename before the .ARD extension. How the file name is changed is controlled by **Options** > **Defaults** > **Startup defaults** > **Revision Disk File Format**; see the Defaults chapter for more information.

For example, the first time you save design REVTEST, it is saved as REVTEST.ARD. The next revision of the same design would be saved automatically as REVTEST_~.ARD, and then REVTEST_A.ARD, REVTEST_B.ARD, and so forth.

If you change the file somewhat, but still want to keep the original for reference, click **Save As Revision** on the File menu. The Save As Revision dialog box will appear.



Enter a description of the file, and enter any authorization message, such as `Approved by Mike` or `OK per JG`. If desired, enter a revision description indicating what changed from the previous revision. Migrate the Projects information to the new revision by selecting the checkbox next to the Project name. To keep the Project information in the old revision, clear the checkbox next to the Project name. Click **OK** to save the file.

When it is time to open the design again, use Resource view. Resource view also allows you to view all revisions of a file, or only the most recent.

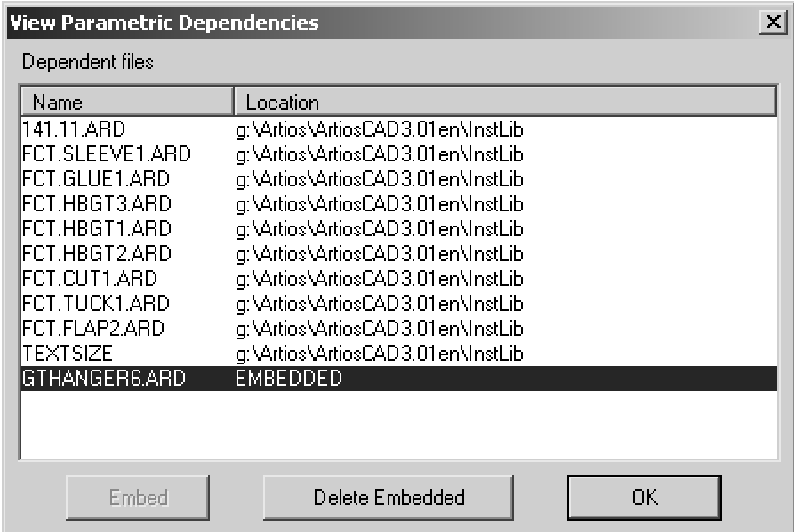
Click  to view all versions of a design.

Click  to view only the most recent revision in the list of files.

How to save a self-contained parametric design

Rebuildable ArtiosCAD designs can be made of many components. To save a rebuildable design with everything comprising it saved in the same workspace, define an Output in Defaults that has **Embed dependent files** checked on the Information Filter tab. Then use that Output to create the self-contained parametric file.

To find out which files will be included, click **Design** > **View Parametric Dependencies**. This menu option is available only for rebuildable designs. The View Parametric Dependencies dialog box will open and show the included components, if any, of the current rebuildable workspace. Files that are not currently marked as embedded will be automatically embedded in the workspace when the Output is run.



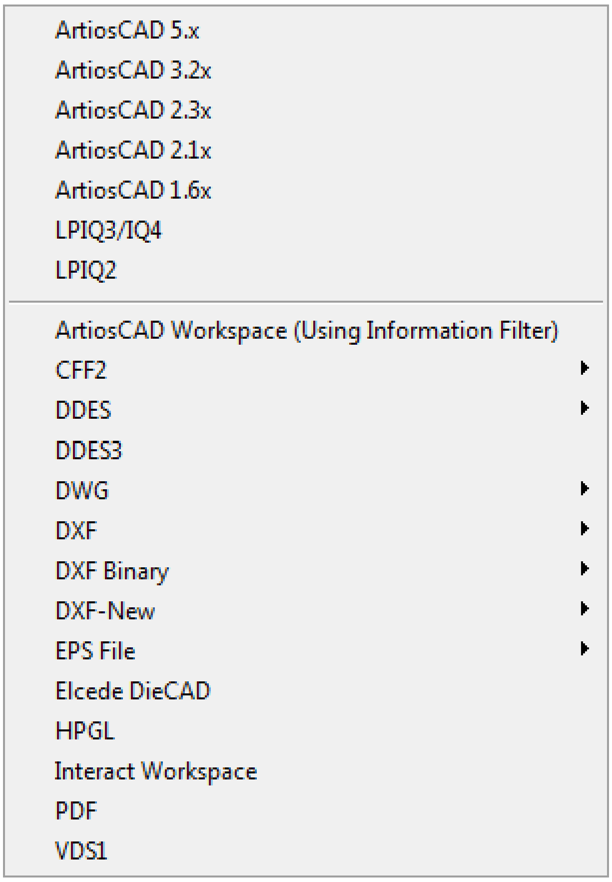
When the parametric design is rebuilt by the recipient, embedded components take precedence over local files with the same name as the embedded component.

To delete an embedded component and use the local version (if one exists - it must have the same name as the embedded component), select it and click **Delete Embedded**. You can only delete an embedded component if there is a workspace with the same name in the ArtiosCAD program directories. When you delete an embedded component, the EMBEDDED designation is replaced by the location of the component on the local system.

Self-contained parametric designs may be added to a Style Catalog and run as a standard. Any embedded components in the standard will be embedded in the resulting single design.

How to export a design to a different format

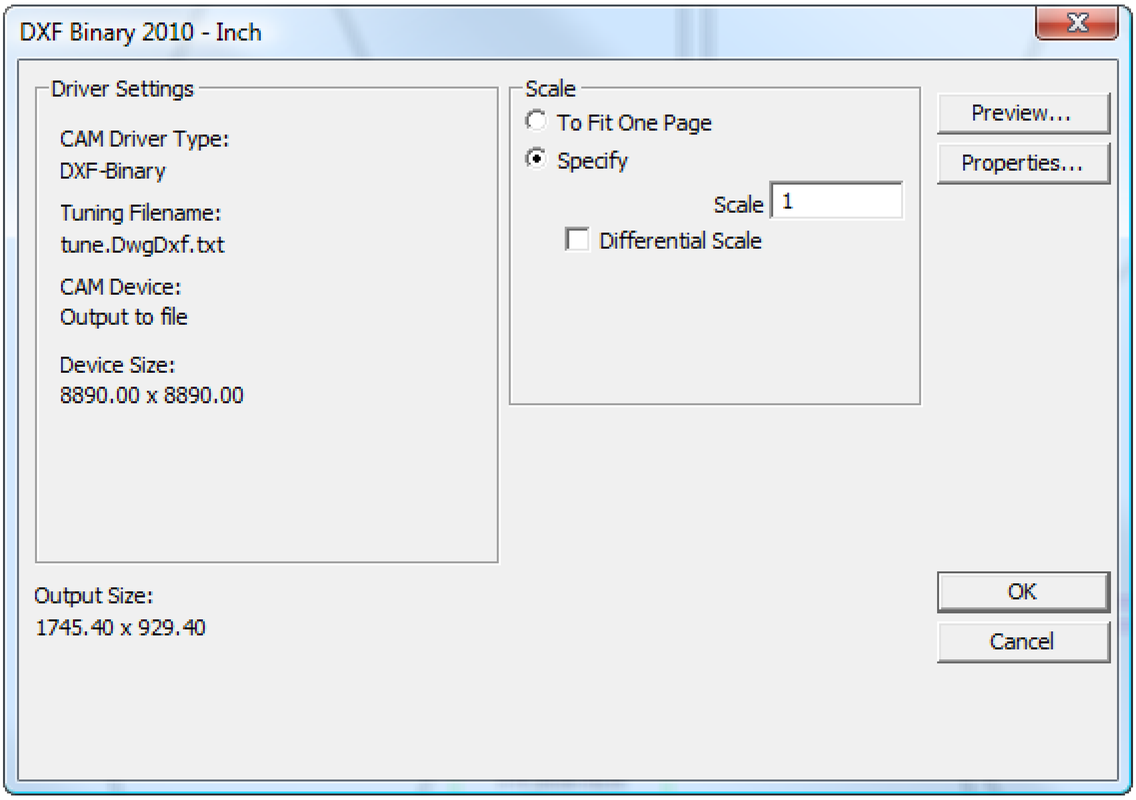
To export a design to a different file format, click **File** > **Export** , and then click the format in which to export the design.



VDS1 stands for **Verein Deutsches Stanzformbauer**, a format used in Europe.

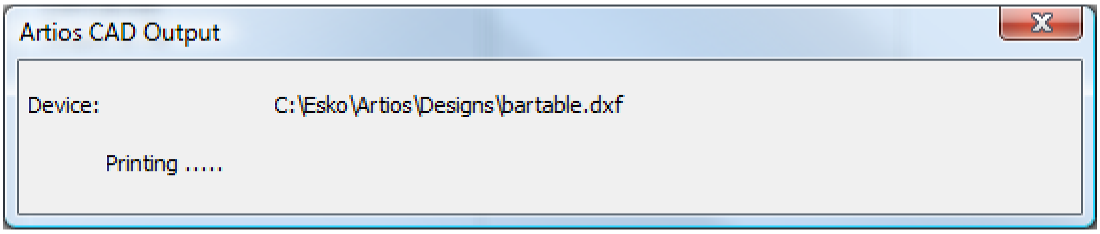
Once you have chosen a format, an Exports dialog box will appear allowing you to specify the directory/resource and filename.

All files are exported to the current directory by default. This is usually `\Esko\Artios\Designs.`



Click **OK** if the settings are correct. You may want to go through all the property sheets (by clicking **Properties**) to make sure the settings are appropriate for your needs.

After you have clicked **OK**, you will be prompted for a name for the file in the Save As dialog box. Enter a name without spaces and click **Save**. The file will be created according to the settings in the Outputs dialog box.



When exporting a DDES file, the `Job Name` is set to the first 7 characters of the file name, and A for the first subroutine, B for the second, and so on. When exporting a DDES3 file, print items are not supported, and decimal places should be set to 4 or less to ensure accuracy.

ArtiosCAD can output DXF files in different ways:

- **DXF**. This option uses the built-in method to create a DXF file.
- **DXF Binary**. This option creates a binary DXF file using a third-party library that has more features than the built-in method.
- **DXF-New**. This option creates a text DXF file using the third-party library.

When exporting an EPSF file, you can either export it as an Illustrator 3-compatible file or an Illustrator 8-compatible file. Exporting it as version 3 or 8 makes those respective versions of Illustrator treat it as a native file.

To lock a layer in EPSF files when they are output, copy `..\InstLib\tune.epsf.txt` to `..\ServerLib` and change the ninth parameter of each group of lines to 1. The tuning file may not have a ninth parameter; in that case, simply add the numeral 1 to the end of each layer's line pair if necessary. Other values such as the line color, layer name, stroke width, and so forth are controlled by the EPSF tuning file, not by the plotting style. The plotting style affects only the Output preview.

To export an EPSF file with all elements colored black, on the Device tab of the Properties dialog box of the Output, browse for the `\Esko\Artios\ArtiosCADn.nnll\InstLib\TUNE.EPSF.BLACK.TXT` tuning file in the **Tuning Filename**: field.

When exporting a 3D workspace to Illustrator 8, and there are imported PDF graphics on the design, and you have the PDF option, you can set the resolution of the bitmaps in the **High graphics memory use** drop-down list box on the Device tab of the Properties dialog box for the Illustrator export.

Font notes and warnings when exporting as PDF

Only TrueType fonts are exported to PDF. TrueType fonts have this symbol before their names. If text uses a non-TrueType font, the default font from Property Defaults is used. If the default font cannot be found, ArtiosCAD will use Arial. The output preview will show non-TrueType fonts correctly but they will be replaced in the actual PDF file.

Graphic fonts such as Symbol and WingDings are not supported.

Fonts defined in `.ttc` files are not supported.

Fonts with vertical text (those with @ preceding their names) are not supported.

ArtiosCAD Reader Illustrator Plug-In

The ArtiosCAD Reader Adobe Illustrator Plug-In lets Adobe Illustrator users open or place native ArtiosCAD single design workspaces without the need for the ArtiosCAD user to convert to EPSF. There also are dedicated tools for moving the entire CAD structure within Illustrator and aligning graphic elements to it. The plug-in can be downloaded from Esko 's Web site at `http://www.esko.com`, and is also on the ArtiosCAD CD.

The plug-in works with structural data only. It does not read graphics, text, dimensions, fills, strokes, or manufacturing line types into the Illustrator workspace. It reads the Main Design layer and the Outside Bleed layer; Outside Coating layers are combined into one coating layer in Illustrator, as are all Annotation layers. Other layers are not read.

For more information on the plug-in, refer to the documentation that is included in its installation package.

How to export files to the ArtiosCAD Viewer

No special export is needed for the ArtiosCAD Viewer. It can open native `.ARD` and `.A3D` files.

ArtiosCADworkspaces are binary, not text.

How to export files to previous versions of ArtiosCAD

If you need to send designs to people using earlier versions of ArtiosCAD, you need to export them so that their versions will read your files properly. In some cases, information like manufacturing parameter sets will have to be reselected by the recipients.

To export the file, click **File** > **Export**, and choose the software version that is less than or equal to the recipient's software.

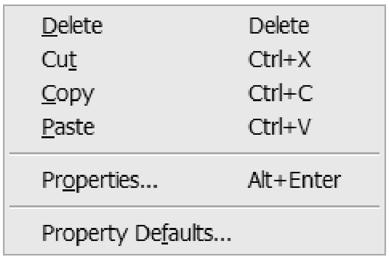
If you need to strip information out of the files, use the Information Filter.

Right mouse button functionality

Clicking the right mouse button activates a context menu from which you can perform common tasks.

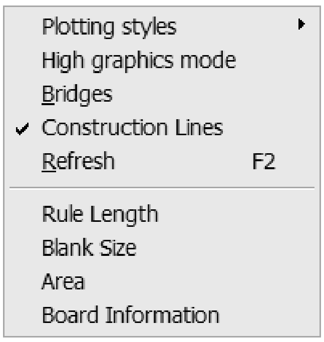
Single Design

In Single Design, the first click of the right mouse button ends the current tool and activates the Select tool. If there is an object under the cursor, the second click of the right mouse button selects the object and opens a context menu as shown below.



The **Paste** command appears only if something has been previously cut or copied.

If there is nothing under the mouse cursor when the right mouse button is clicked, and nothing is selected, a general context menu is opened similar to the one shown below.



Plotting styles which have the **Output only** checkbox selected in their definition in Defaults will not appear in this context menu.

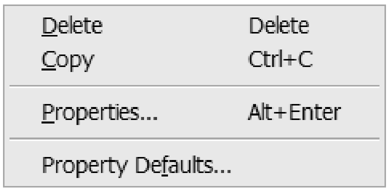
Manufacturing

Clicking the right mouse button in Manufacturing stops the current tool but does not automatically start the **Select** tool.

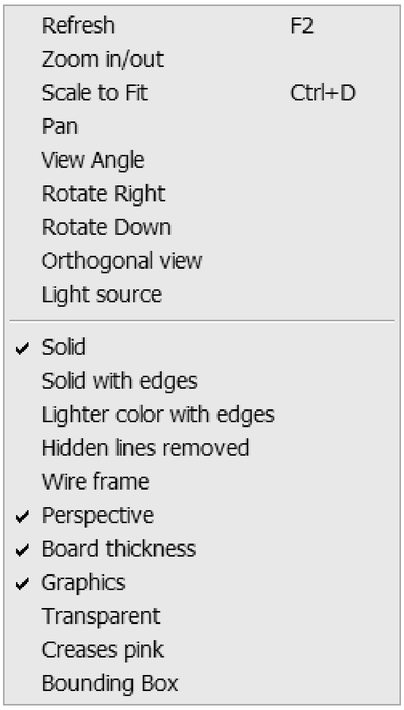
When the **Select** tool is active in Manufacturing, clicking the right mouse button has the same functionality as it does in Single Design.

3D

Clicking the right mouse button in 3D when the cursor is over a design line stops the current tool, selects the object under the cursor, and opens the context menu shown below.



When there is no current selection and the cursor is not over a design line, a context menu for accessing common commands appears as shown below.



Using the spell checker

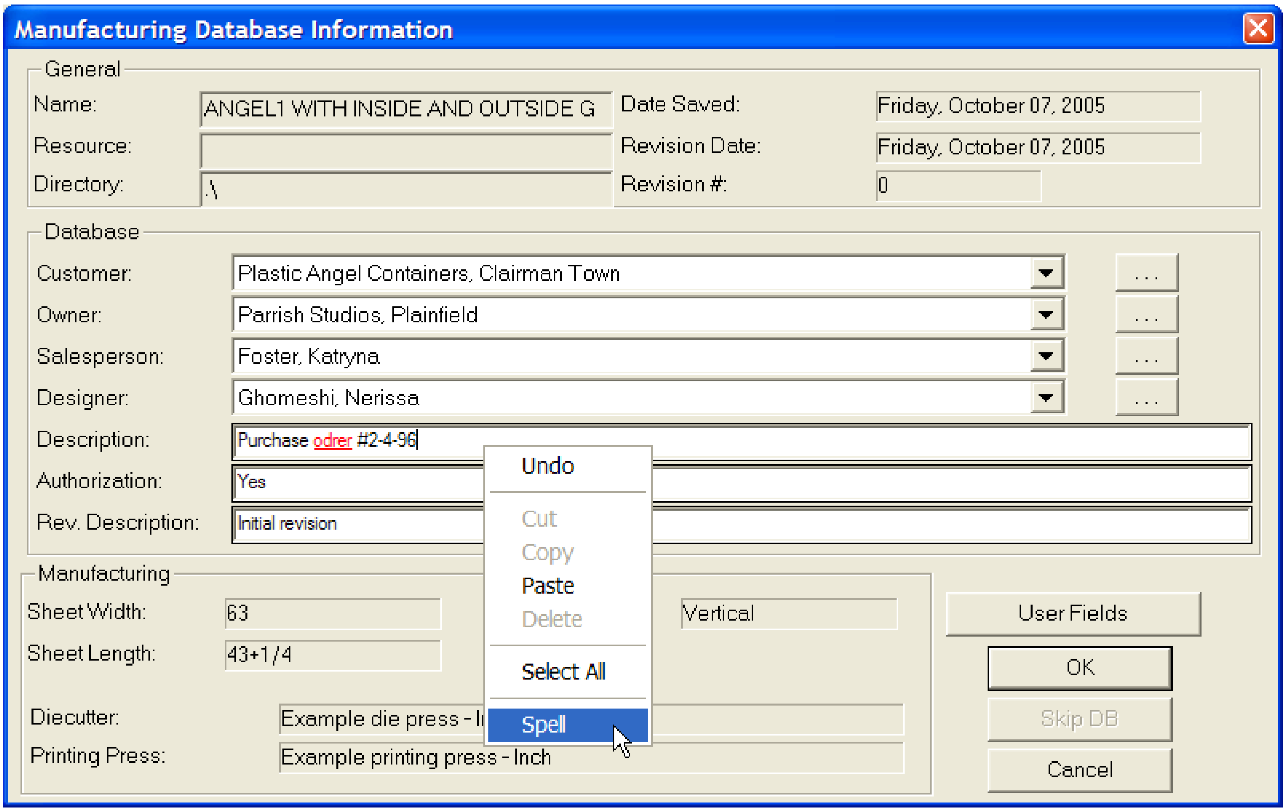
If you have Microsoft Word 2000 or later installed on your system, ArtiosCAD can use its spell checking functionality in Single Design and Manufacturing in most places where you enter text using a field in a dialog box. As you type, specific keystrokes trigger the spell check: `Backspace`, `Enter`, `Space`, `exclamation point`, `close parenthesis`, `comma`, `period`, `semicolon`, `question mark`, and `close brace`.

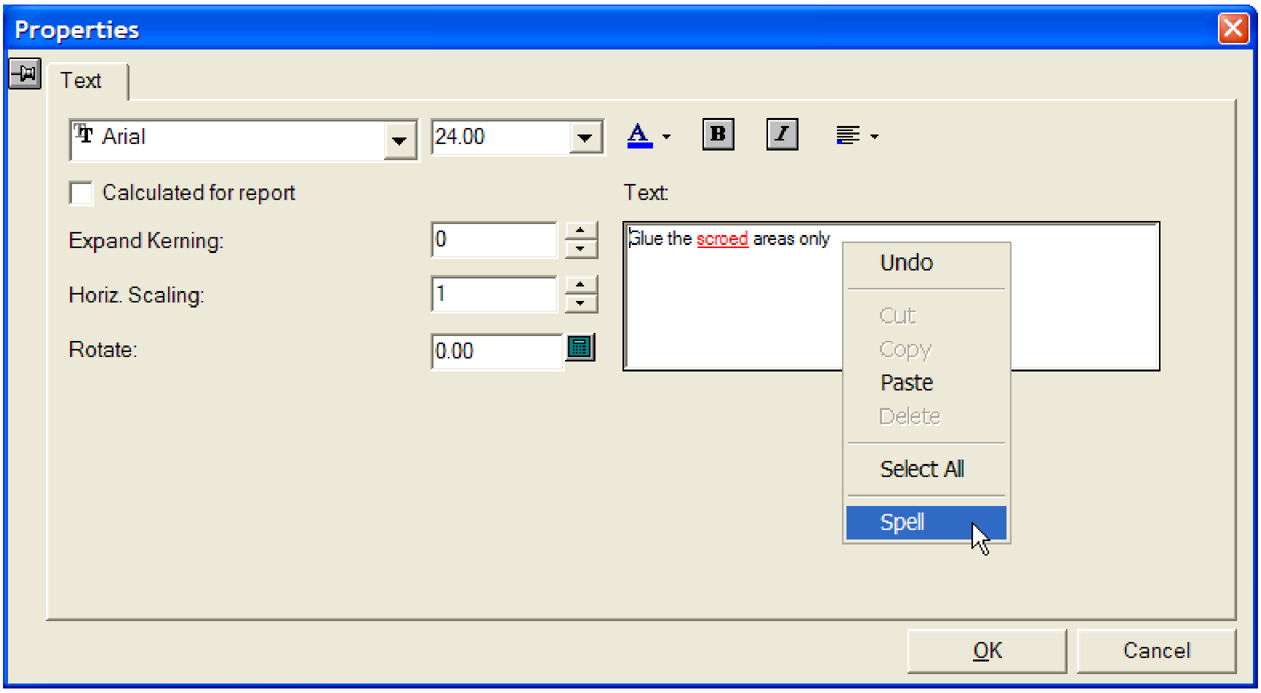
There are three ways to use the spell checker. The first is by using the `Paragraph Text` tool. The second is by entering text in fields in the Database Information dialog boxes, both default and custom, and other text-entry field such as those in the Properties dialog boxes for text and dimensions. The third is by executing a document-wide spell check by clicking **Tools** > **Check spelling**.

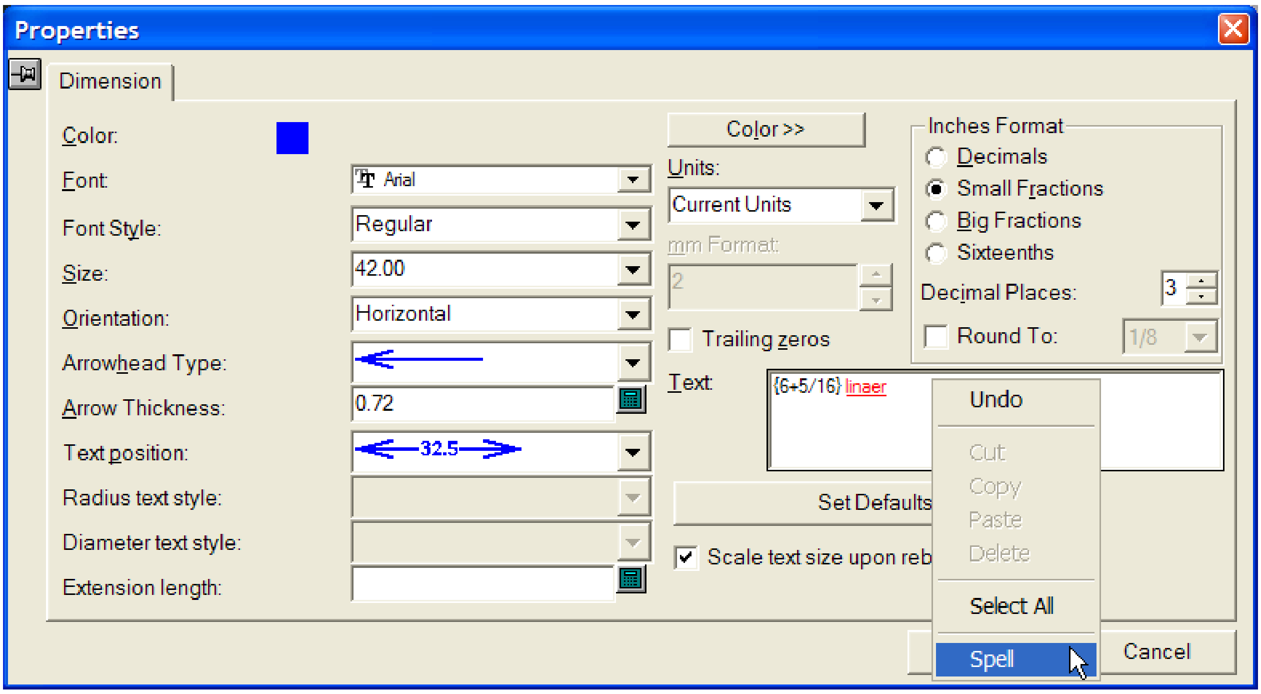
To learn more about using the spell checker in the Paragraph Text tool, see *Spell checking in the Paragraph Text* tool for more information.

Spell checking in fields

Fields that support spell checking have an emphasized border, and any misspelled words within them are underlined and shown in red. Right-click in the field and choose **Spell** from the context menu to open the Spelling dialog box.

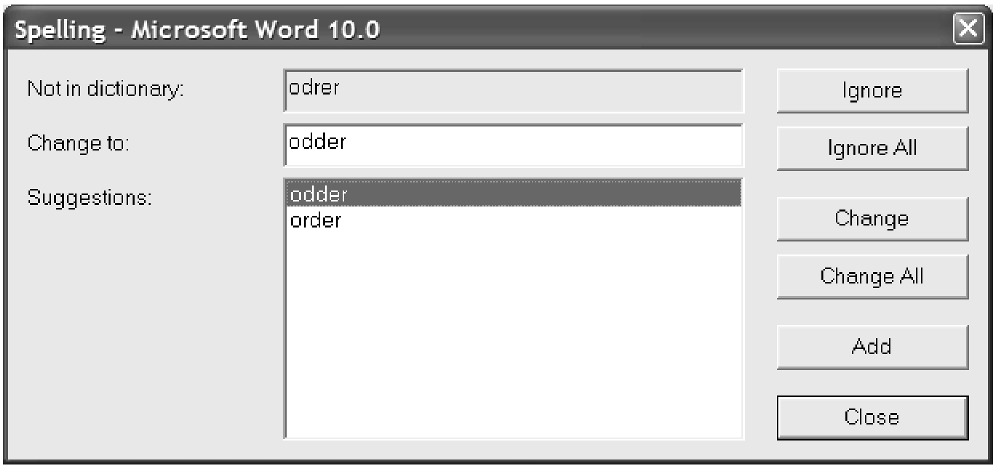






The first time you activate a tool or dialog box in ArtiosCAD that supports spell check, ArtiosCAD launches Microsoft Word in the background and communicates with it as needed during the ArtiosCAD session. This initial launch of Microsoft Word may cause the ArtiosCAD dialog box to take a few extra moments to open, depending on the speed of your system. Once Microsoft Word is running in the background, dialog boxes supporting spell check appear at their usual speed.

When you right-click a field and choose **Spell**, the standard Microsoft Word Spelling dialog box appears as shown below.



The unrecognized word is shown in the **Not in dictionary:** field. The **Suggestions:** field contains the list of suggested words, and the top one is selected and seeded to the **Change to:** field. If there are no suggestions, the unrecognized word is repeated. You can edit the contents of the **Change to:** field as desired.

Ignore and **Ignore All** respectively ignore this instance and all subsequent instances of the unrecognized word for this invocation of the spell checker. The next unrecognized item is then displayed.

Change changes the unrecognized word to the selected suggestion or the word you entered in the **Change to:** field. **Change All** changes all subsequent instances of the unrecognized word in the workspace to the selected suggestion or the word entered in the **Change to:** field; previously-ignored instances are not changed. The next unrecognized item is then displayed for review.

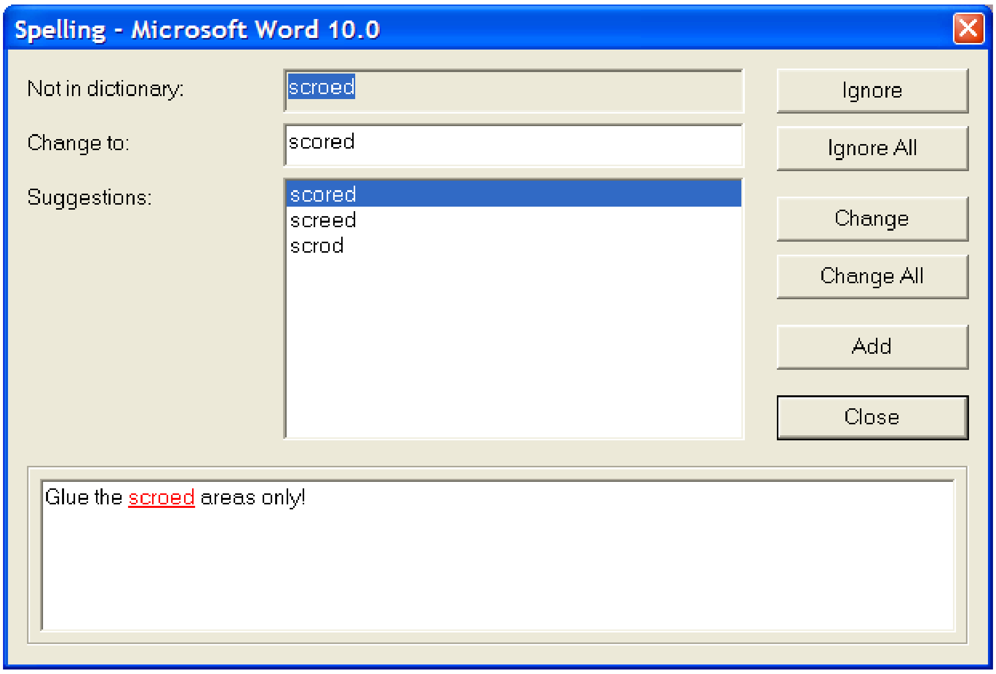
Add adds the unrecognized word to the ArtiosCAD custom dictionary file so that it will be recognized in the future. The next unrecognized item is then displayed. This command is disabled if the custom dictionary is not installed or is not editable.

Close closes the Spelling dialog box.

Spell checking through the whole workspace

You may also spell check all text items in the workspace by clicking **Tools** > **Check Spelling**. Note that this method does not check database entries for unrecognized words. This tool is available regardless of the state of the **Enable spell checking** option in Defaults.

When this tool is activated, each text item in the workspace is checked for unrecognized words, including text items in locked layers. If an unrecognized word is found, the entire text item is shown with the unrecognized word underlined and shown in red as shown below.



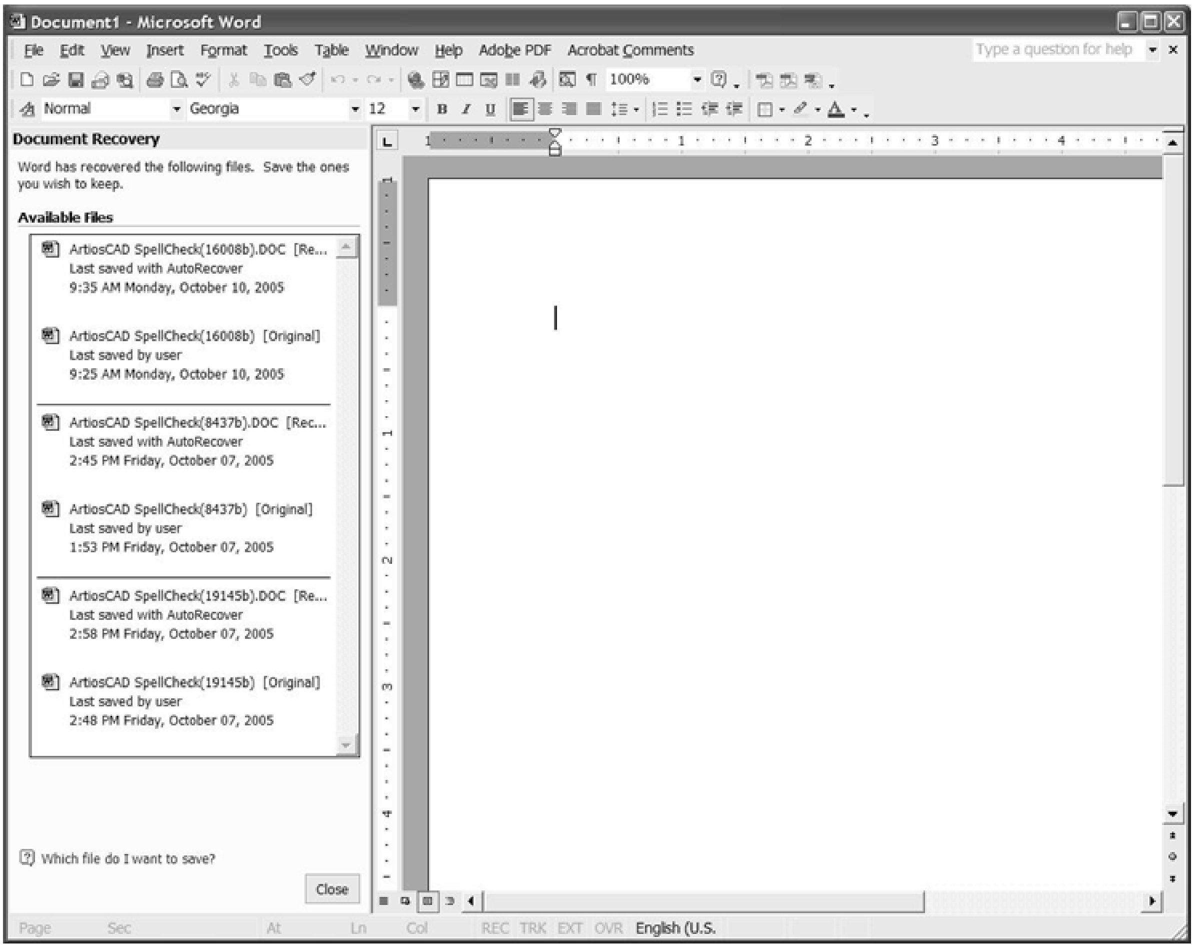
The options in this dialog box work the same way as described in the previous section.

Once you ignore, change, or add an unrecognized word, the next text item is checked. When all text items have been checked, **Close** is the only available option in the Spelling dialog box.

Notes about spell checking

The spell checker does not flag repeated words.

When you exit ArtiosCAD using **File** > **Exit**, ArtiosCAD terminates the associated Microsoft Word process. If ArtiosCAD terminates abnormally, the Microsoft Word program **WINWORD.EXE** may still be left running; use Task Manager to end it. If you then launch Microsoft Word using the Start menu, it may show `ArtiosCAD Spell Check(number).doc` as being available for auto-recovery as shown below. No such temporary files need to be recovered or saved, and you may delete them from the hard drive.

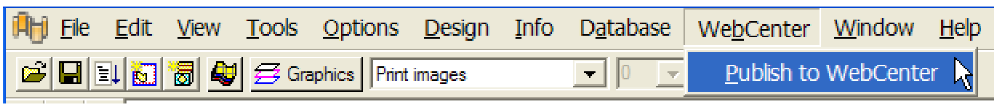


These temporary files are the means by which ArtiosCAD interfaces with Microsoft Word to check spelling. If you are using Microsoft Word at the same time that ArtiosCAD is running, the temporary file may be shown on the Window menu in Microsoft Word. Do not open the temporary file, as this will break the connection between ArtiosCAD and Microsoft Word, and spell checking in ArtiosCAD will be disabled until you reactivate the tool which invoked it.

The custom dictionary file `CUSTOM.DIC` is stored in `\ArtiosCAD installation folder\Common\Proof`. To edit it, close ArtiosCAD and use WordPad (not Notepad) to edit it. Put each word on a separate line and save the file as type **Text Document**. Do not change this file while ArtiosCAD is running, as it will not reflect the changes.

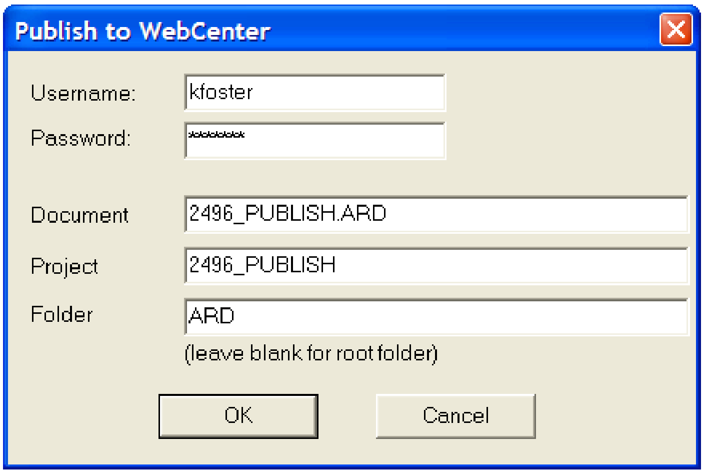
Publishing ArtiosCAD documents to WebCenter

Publish to WebCenter on the **WebCenter** menu allows you to publish a single design, a manufacturing file, or a 3D workspace to WebCenter, Esko's design lifecycle management software. Shown below is the menu in Single Design.



Before you can publish to WebCenter, the ArtiosCAD system administrator must have configured an WebCenter site in the WebCenter catalog in Defaults, and the WebCenter system administrator must have created a user account with sufficient permissions for you to use. When ready, publish to WebCenter by doing the following:

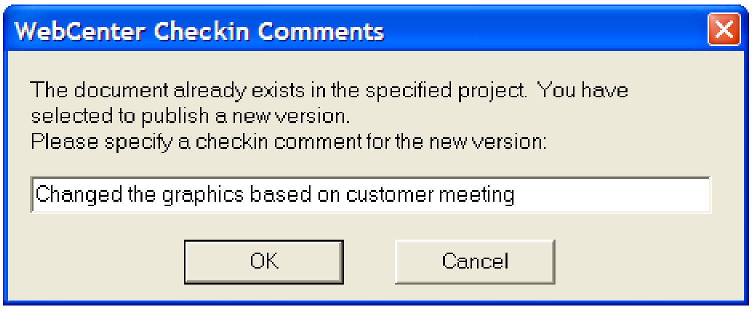
1. Open the desired document in ArtiosCAD.
2. Click **WebCenter** > **Publish to WebCenter**.
3. If ArtiosCAD is configured to prompt for information, the Publish to WebCenter dialog box appears. If prompted, enter the WebCenter username and password supplied to you by the WebCenter system administrator. The **Document**, **Project**, and **Folder** fields are completed by default to be based on the name and type of the ArtiosCAD document.



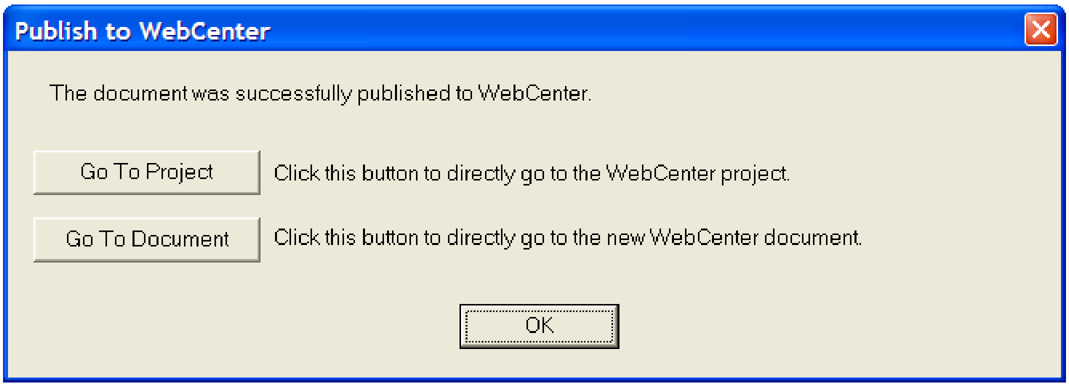
The ArtiosCAD system administrator can define the **Publish to WebCenter** entry in Defaults so that all, some, or none of these fields are pre-completed.

Click **OK** to publish the document to WebCenter once all the fields are completed.

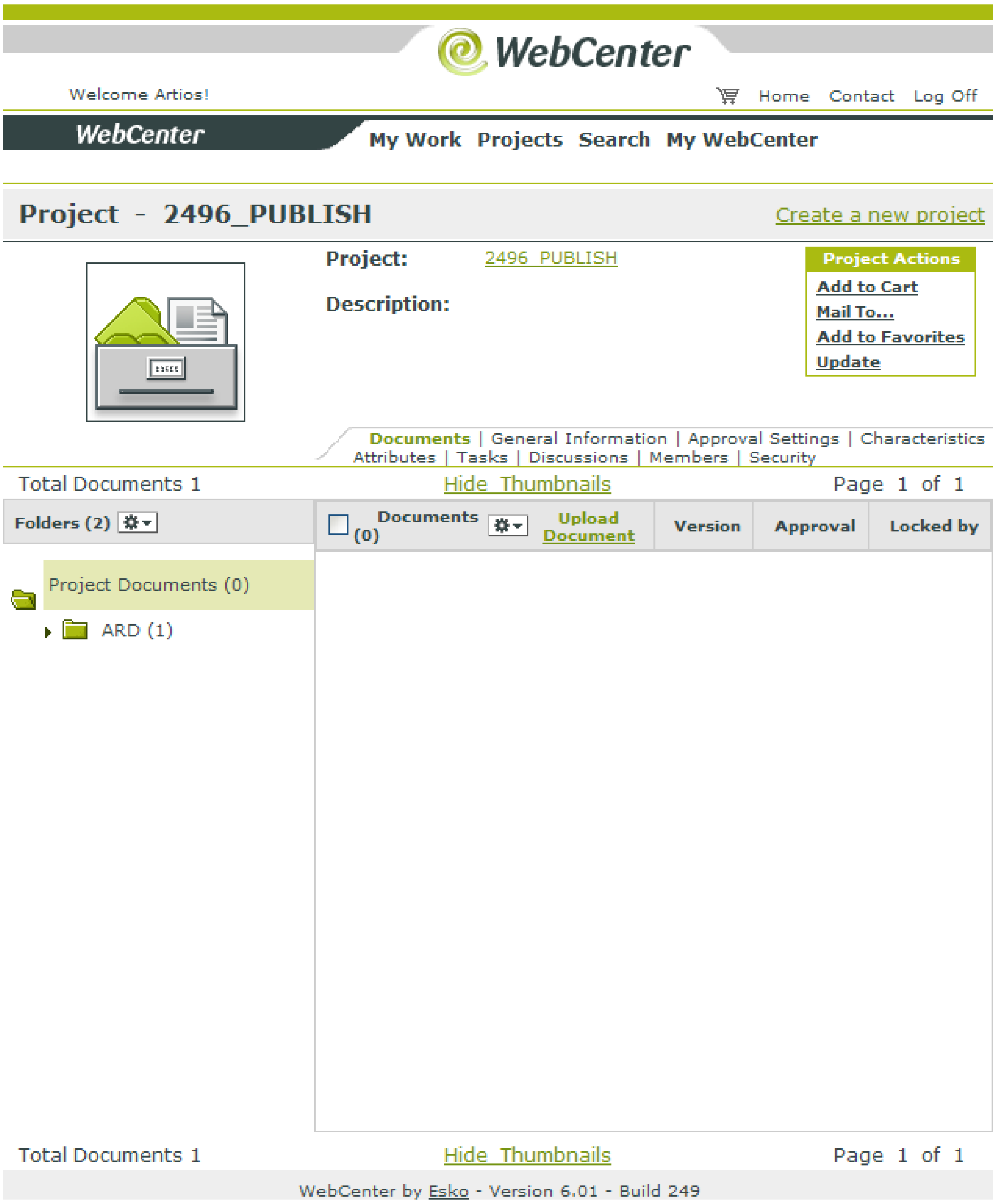
4. If the document already exists in the specified Project, you will be prompted to enter a comment about the new version as shown below. Enter a comment and click **OK**.



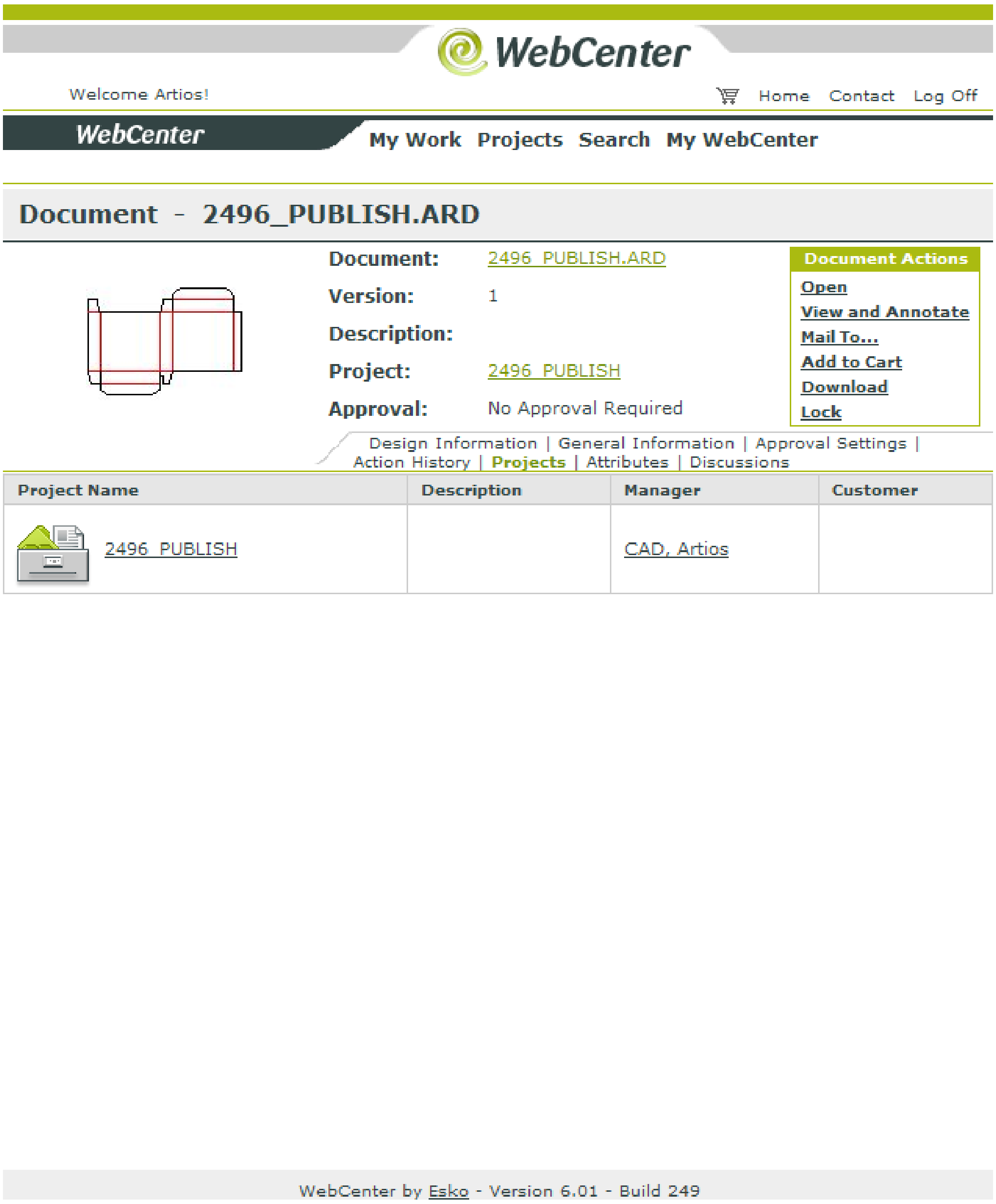
5. The Publish to WebCenter dialog box should change to match the one below.



Clicking **Go To Project** opens the Project in WebCenter. You must log in to WebCenter if you are not already logged in.



Clicking **Go To Document** opens the document. You must log in to WebCenter if you are not already logged in.



Click **OK** to stay in ArtiosCAD.

In the rare event that the document does not appear in WebCenter and no error is shown, check that the **WebCenter address** field in the **Publish to WebCenter** entry of the WebCenter catalog in Defaults is correct.

Portable Workstation mode

Portable Workstation mode lets an ArtiosCAD system using shared network resources such as the database or Shared Defaults detach from the network and operate as if it were standalone with no database. When you enter Portable Workstation mode, ArtiosCAD copies necessary elements from the network servers to the local machine (called **synchronizing**) and disables the database.

Note: Detaching ArtiosCAD from the network servers does not actually break the network connection for other applications in the operating system.

There are important restrictions when using Portable Workstation mode:

- Using Portable Workstation mode requires local licensing; that is, a License Manager client may not use Portable Workstation mode. Upon attempting to enter Portable Workstation mode, the licensing will be checked and you will be prevented from entering it if the licensing method does not support it.
- Any workspaces needed that normally reside on network resources must be manually copied to the local system. Likewise, anything extra data needed (symbols, graphics, Reports, and so forth) that are not in `ServerLib` or `Common` must also be copied manually. Subdirectories of `ServerLib` and `Common` are copied (except for `Common\Program`).
- If you changed the search list in the Windows Registry, the changes are ignored.
- When reconnected to the network, you must resave any workspaces created while in Portable Workstation mode in resources to put them into the database.
- Detaching from the network is done on a per-user and per-ArtiosCAD-version basis. For example, if you plan to use more than one version of ArtiosCAD while disconnected from the network, you must synchronize and detach each version before disconnecting from the network. Likewise, each user on the machine must detach each version of ArtiosCAD that he or she will use while detached.
- If the **Start in** property for any ArtiosCAD icon is set to a network location, it must be changed to a local drive. Right-click the icon, click **Properties** on the context menu, examine the contents of the **Start in** field, and change it to a directory on the portable workstation's drive (such as `C:\temp`) if it points to a network drive (`S:\users\wmeeber\CAD`) or a UNC location (`\\mbcfileserv\users\wmeeber\CAD`).
- DataCenter Admin uses the database specified by the System DSN ODBC data source regardless of the state of Portable Workstation mode. If you use a network database and detach from the network, DataCenter Admin will continue to function until the network is disconnected. If you detach and disconnect from the network and use a local database, DataCenter Admin will continue to function using your local database if you change the System DSN to point to your local database. If you do not use a local database after detaching and disconnecting from the network, DataCenter Admin will not function correctly as it will be searching for the networked database. Do not start DataCenter Admin if you will not be using a local database when disconnected from the network.

Using Portable Workstation mode for the first time

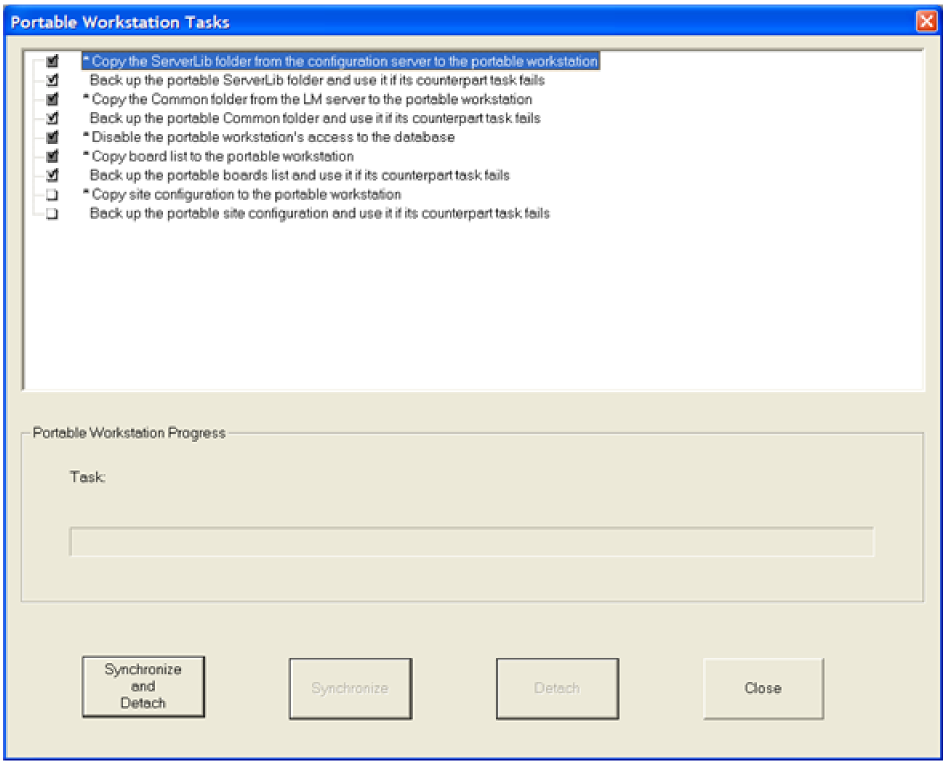
Using Portable Workstation mode for the first time involves performing required tasks that are not required with subsequent uses.

Detaching from the network

To use Portable Workstation mode and detach from the network for the first time, do the following:

1. Make sure the network is connected to the computer.

2. If necessary, change the **Start in** property for the ArtiosCAD icon.
3. Start ArtiosCAD.
4. Click **Options** > **Portable Workstation** > **Detach from Network**.
5. The Portable Workstation Tasks dialog box will open as shown below.



6. A list of tasks is displayed.
   - Selected tasks have checks in their associated checkboxes.
   - Grey checkboxes may not have their state changed.
   - White checkboxes may be changed as desired.
   - Tasks with an asterisk (*) before their names must complete successfully for the workstation to enter portable mode.
   - Tasks have different default states for the first time they are run versus being run after a successful detach.

In general, there are two kinds of tasks, **copy** tasks and **back up** tasks. Copy tasks copy data from the network servers to the local machine. Back up tasks make backups of the copied data on the portable workstation in case it becomes corrupted and unusable. Each copy task has a corresponding backup task. The **Disable the portable workstation's access to the database** task neither copies nor backs up.

Table: Tasks and Their Actions

Task	Action	Can be skipped if
Copy the ServerLib folder from the configuration	Copies the Shared Defaults and shared files (such as custom Reports) from the Configuration server. If the portable workstation is configured as its own	You have recently synchronized and you are confident it meets your needs.

Task	Action	Can be skipped if
server to the portable workstation	standalone Configuration server, the information and files are copied to another location on the portable workstation.	ArtiosCAD is installed as standalone or as a Configuration server and thus uses its own set of Defaults and shared files.
Copy the Common folder from the LM server to the portable workstation	Copies the files from `..\Artios\Common` on the License server to the portable workstation. This copies licensing information and board textures. This task must be selected if the workstation is not its own License server and uses a security key. No subdirectories are copied. Previous copies are deleted each time this task runs. **Note**: The only kind of licensing that will work with Portable Workstation mode and a networked License server is when there is a hardware key physically attached to the portable workstation and the `TUNEOPT7.TXT` is stored on the License server. If you are in this mode, you must run this task.	You have recently synchronized and you are confident it meets your needs. ArtiosCAD is installed as standalone or as a License server and thus uses its own set of Common files.
Disable the portable workstation's access to the database	Ensures no database connection is attempted when opening or saving a workspace, and disables all database-related fields and dialog boxes except for boards. This includes the Database dialog box in Defaults, the Database Information dialog box in Single Design, Layout, and Manufacturing modules, AutoLoad, the three database browsers, Projects, Search on the File menu, and the Resource View tab in the Open dialog box.	You have a local database you want to use after detaching from the network, which would require changing the ArtiosCAD System DSN ODBC entry.
Copy board list to the portable workstation	Copies the boards, board hierarchy, mediums, and flutes from the Database and License servers to the portable workstation.	You have recently synchronized and you are confident it meets your needs. ArtiosCAD is installed as standalone or as a License server and a Database server.

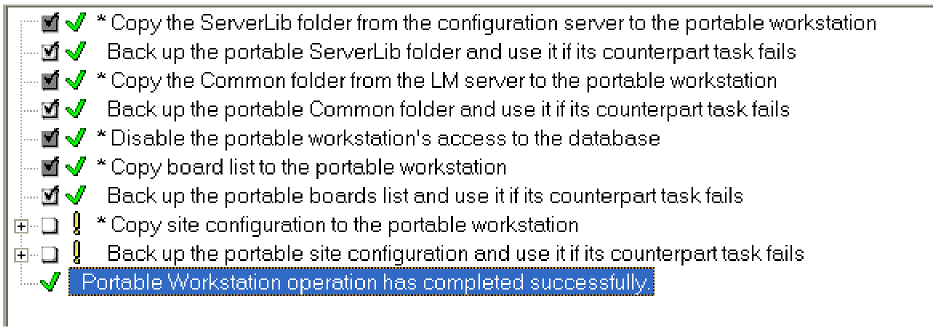
Task	Action	Can be skipped if
Copy site configuration to the portable workstation	Copies custom User Defaults used for “site” defaults that are stored in a network location to the portable workstation.	You do not use networked User Defaults in such a manner.
Back up...	Makes a backup copy of the information being copied from the network. These backups can then be used in the event of a failure of a future synchronization. Each time there is successful synchronization, the previous backups are deleted if the corresponding backup task was selected.	You have previously successfully synchronized and backed up the copy tasks.

Beneath the list of tasks is a progress indicator for the current task.

Beneath the task progress indicator are four buttons.

- **Synchronize and Detach** performs the selected tasks and prepares ArtiosCAD for the network to be disconnected. Note that the actual network connection is not changed; that you must do outside of ArtiosCAD. We strongly recommend running all the selected tasks the first time you enter Portable Workstation.
- **Synchronize** performs the selected tasks but does not prepare ArtiosCAD for the network to be disconnected. This command is available only after the first Synchronization and Detach.
- **Detach** detaches the computer from the network. You must perform a Synchronize before using Detach.
- **Close** closes the dialog box. Close changes to **Cancel** when a task is running. If you click Cancel, all tasks are cancelled and the state of ArtiosCAD is rolled back to how it was when the dialog box opened.

7. Click **Synchronize and Detach** once you have reviewed the tasks to run. The selected tasks will run and result indicators will appear next to each task similar to those shown below.



8. ArtiosCAD is now running in Portable Workstation mode. Use it as desired. To use a local database, change the ArtiosCAD ODBC entry and restart ArtiosCAD.

Some tasks may have different result indicators than those shown in the preceding example.

Table: Task Result Indicators

Icon	Meaning
	A green arrow means the task is running.
	A green checkmark means the task succeeded.
	A red X means the required task failed and the workstation cannot enter Portable Workstation mode.
	A red exclamation mark means a non-required task failed, but the workstation can enter Portable Workstation mode. This is mostly used for back up tasks.
	A yellow exclamation mark means a task succeeded, but there is a note. Click the plus sign (**+**) next to the task to see what happened. This is mostly used when a task is not selected.
	A green exclamation mark means a non-required task succeeded, but there is a note. Click the plus sign (**+**) next to the task to see what happened. This is mostly used when a back up task has succeeded.
	A yellow question mark means the task is in an unknown state. This is mostly used when a task is not selected for synchronization.

If a task was successfully synchronized for a Detach, it will always succeed for subsequent Detaches.

Backup tasks that fail synchronization do not prevent the workstation from entering Portable Workstation mode.

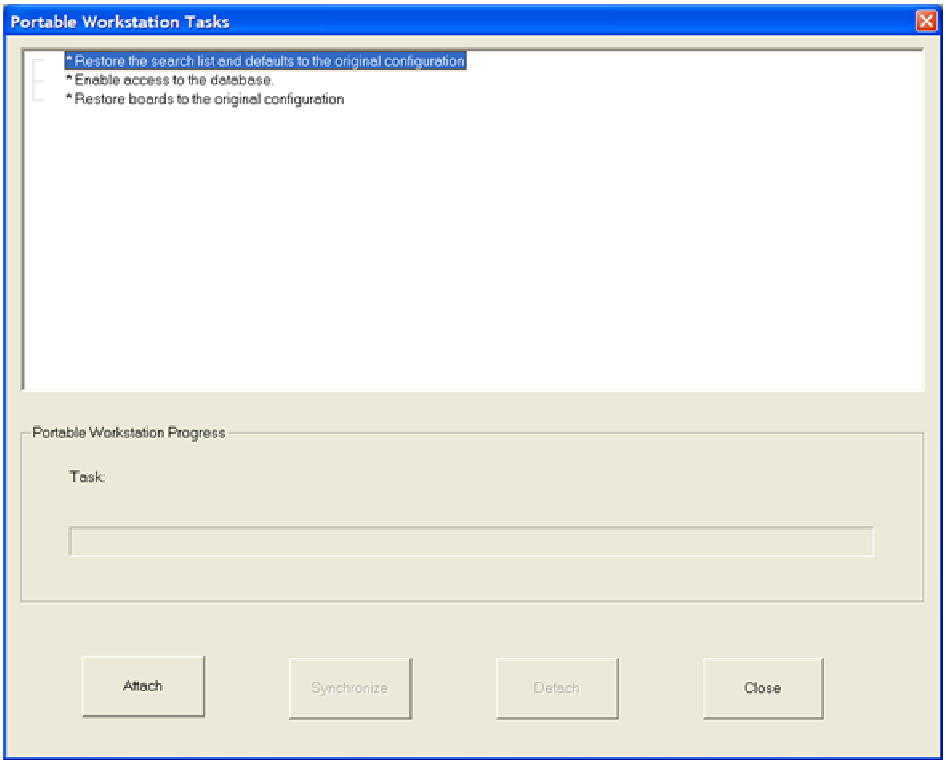
If a required task fails, the workstation can still enter Portable Workstation mode provided its corresponding backup task has been synchronized in a prior Detach.

If a task fails to synchronize, a future Detach will not allow it to Detach until it is successfully synchronized.

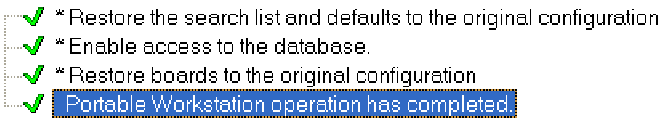
Attaching to the network

When you no longer need to use Portable Workstation mode and want to set ArtiosCAD back to its networked state, do the following.

1. Reconnect the physical network connection.
2. Start ArtiosCAD.
3. Click **Options** > **Portable Workstation** > **Attach to Network**.
4. The Portable Workstation Tasks dialog box appears similar to the one shown below.



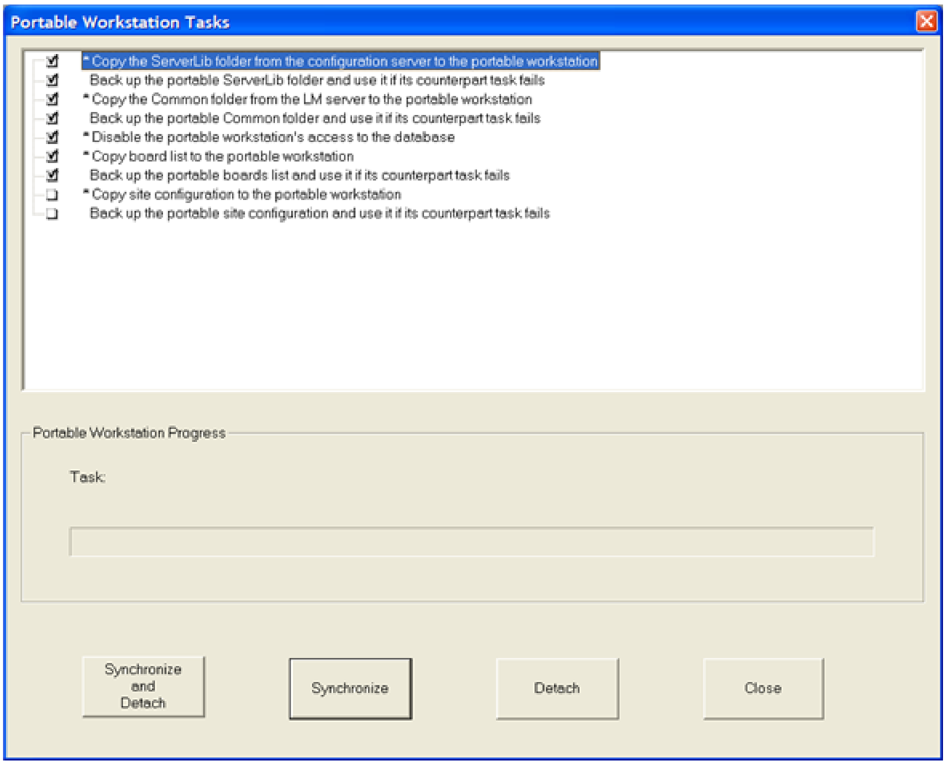
5. Click **Attach** to reconnect ArtiosCAD to its networked resources. The tasks will run and result indicators will appear next to the tasks similar to those shown below.



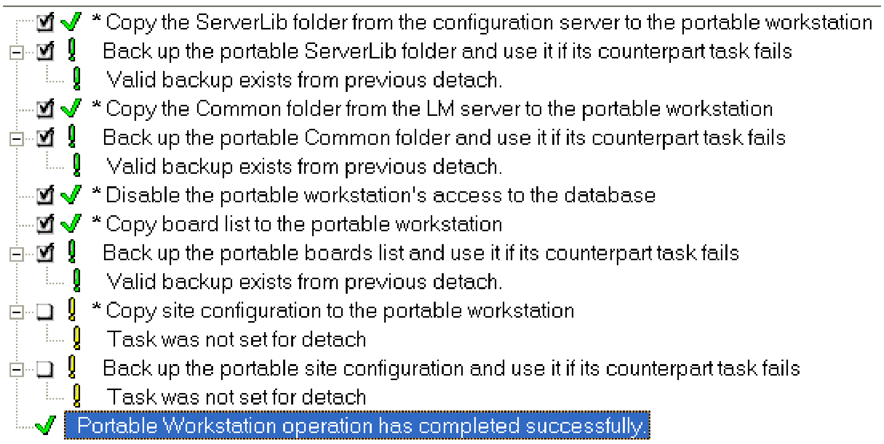
6. Click **Close** to close the Portable Workstation Tasks dialog box.
7. Open any workspaces you made while in Portable Workstation mode and resave them into resources, entering database information as necessary.
8. Resume working with ArtiosCAD as desired.

Subsequent uses of Portable Workstation mode

After you have used it once, on subsequent uses of Portable Workstation mode, you can choose the tasks to perform when detaching. Most of them are still selected by default. **Synchronize** and **Detach** are both independently available.



If you are confident that no network information has changed since you last Synchronized, you can Detach without performing a Synchronize first. Correspondingly, if you want to update the local copies of the network information, you can Synchronize without performing a Detach. Shown below are the results of Detach without a Synchronize with all the information shown.



Attaching to the network in subsequent uses of Portable Workstation mode is the same as it was the first time. To attach to the network, do the following:

1. Reconnect the physical network connection.
2. Start ArtiosCAD.
3. Click **Options** > **Portable Workstation** > **Attach to Network**.
4. In the Portable Workstation Tasks dialog box, click **Attach** to reconnect ArtiosCAD to its networked resources. The tasks will run and result indicators will appear next to the tasks.
5. In the Portable Workstation Tasks dialog box, click **Close**.

6. Open any workspaces you made while in Portable Workstation mode and resave them into resources, entering database information as necessary.
7. Resume working with ArtiosCAD as desired.

4. Builder

What is Builder?

Builder is the component of ArtiosCAD which allows you to create new designs based on standards. By itself, Builder does not allow you to modify the geometry of the created design by changing existing lines or creating new lines.

Running a standard

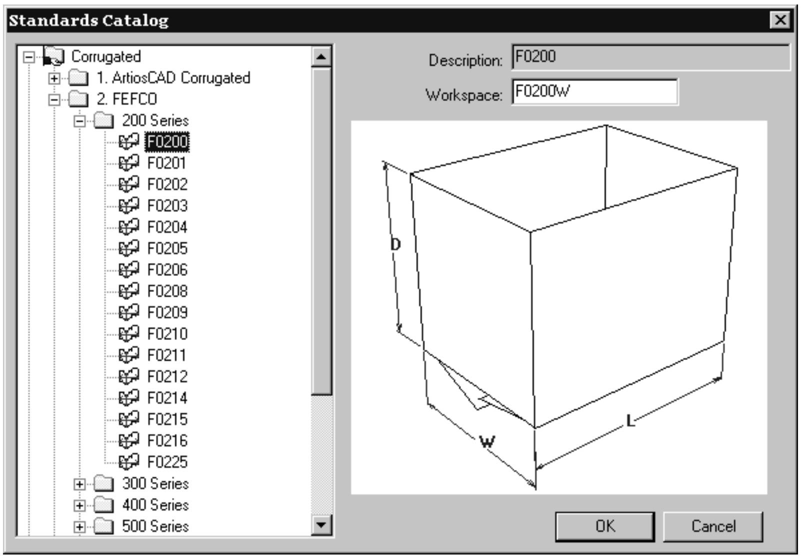
Use these steps to create a design from a standard style. A more detailed example follows.

1. Start ArtiosCAD.
2. Click **Run a Standard** on the **File** menu.
3. Choose a style from the Standards Catalog. Click **OK**.
4. Select a parameter set and board code. Click **OK**.
5. Enter the inside dimensions of the new design. Click **Next**.
6. At this point, each standard is different. Choose style options and enter values for variables if appropriate. A variable is a way to store a value so it can be changed in the future. Click the **Next** and **Previous** buttons to move between menus. If you know that the options and variables in future menus are set correctly, click **OK** to view the completed design.
7. Usually there is a diagram displayed to the left of the variable-entry area to show where the variables in the current menu are used in the design.
8. Position the mouse cursor over the name of a variable and wait a moment. A ToolTip will appear showing the description of the variable.
9. If you try to enter a value that does not meet a variable's check condition, you will be prompted to enter a more reasonable value. A check condition limits the values that can be assigned to a variable.
10. The **Next** button is unavailable when the current menu is the final menu of a style. Click **OK** to view the completed design.
11. The basic design is now structurally complete.

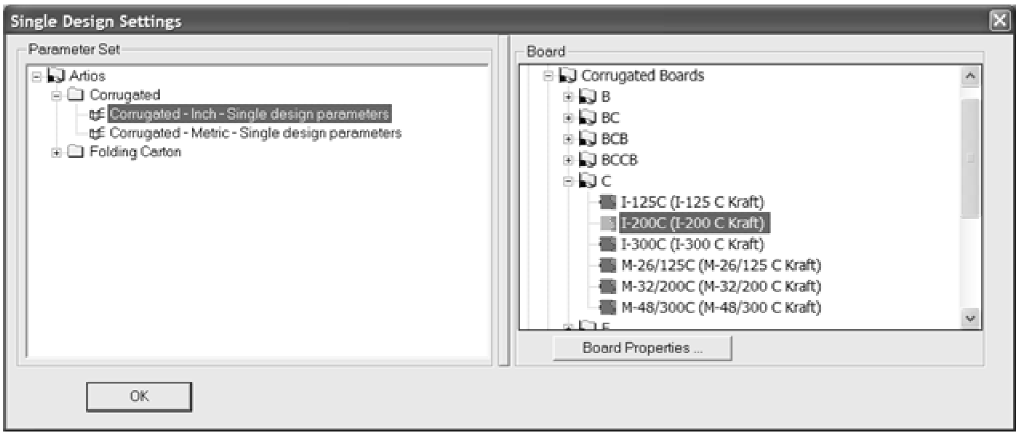
An example of using the Standards Catalog

Here is how to create a design using the FEFCO 200 standard style.

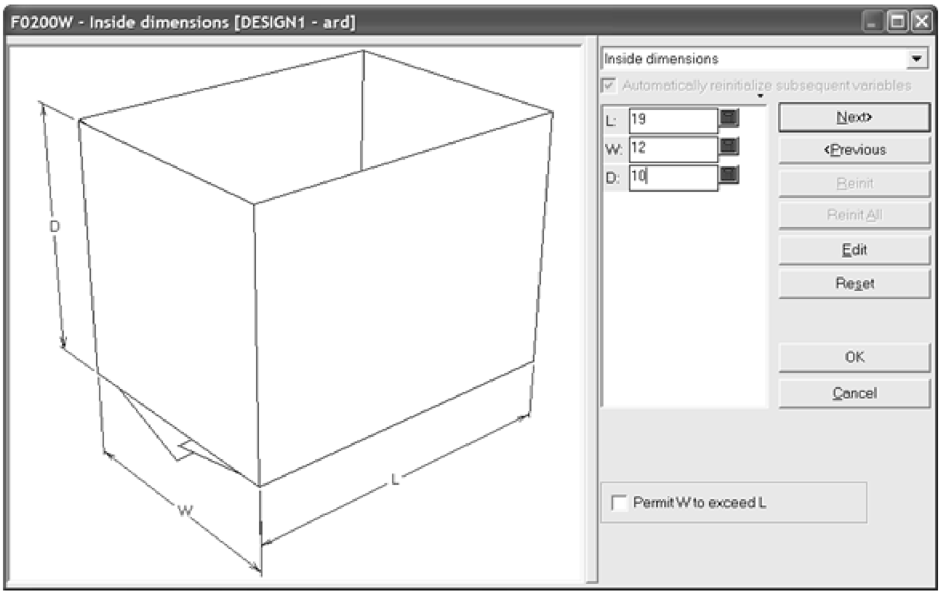
1. Start ArtiosCAD .
2. Click **Run a Standard** on the **File** menu. The Standards Catalog will appear.



3. Click the plus sign (**+**) next to the FEFCO Catalog to show its contents, then click the plus sign next to the 200 Series folder to show its contents. Click the **F0200** style and then click **OK**.
4. In the Single Design Settings dialog box, choose the parameter set to use and board which will be used to make the carton. Click **OK** to continue.



5. In the Inside Dimensions dialog box, enter the inside dimensions of the design. Click **OK** to complete the design using default values. Clicking **Next** takes you through menus of additional choices and values so you can review or alter the defaults of the standard. In the last menu for a standard, the **Next** button is unavailable and you must click **OK** to complete the standard.

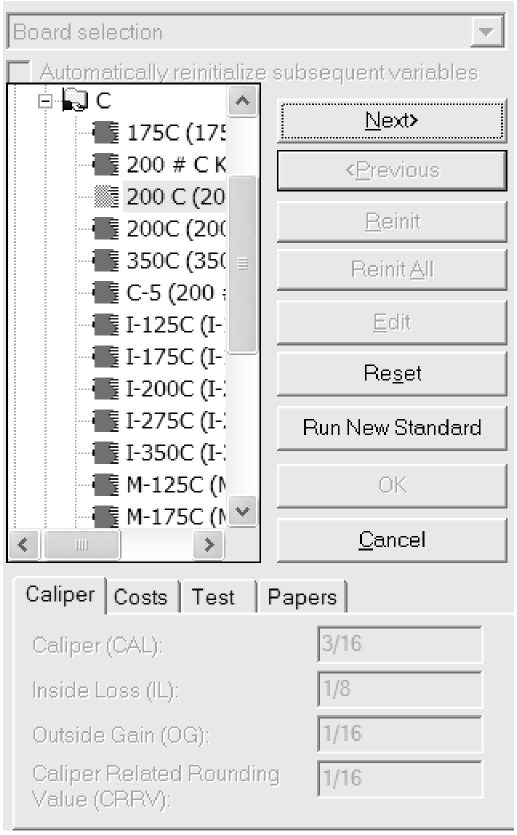


To change the display format of the variable labels, click the tiny triangle above the list of variables and select a different option. These options may be set in Defaults in **Options** > **Defaults** > **Design Defaults** > **Rebuild Options**.

Running a new standard

To run a new standard in the current workspace, click **Design** > **Run new standard**. You will be asked to confirm erasing all elements and variables in the current workspace. Click **OK** to run a new standard, or **Cancel** to return to the workspace without making any changes. After clicking **OK**, choose the new parameter set and board code, choose the standard from a catalog, set the dimensions, and so forth to create the new design. Values in the `L`, `W`, and `D` variables in the old design will be carried into the new design.

This option is also available on the Board selection menu when rebuilding a design based on a standard.



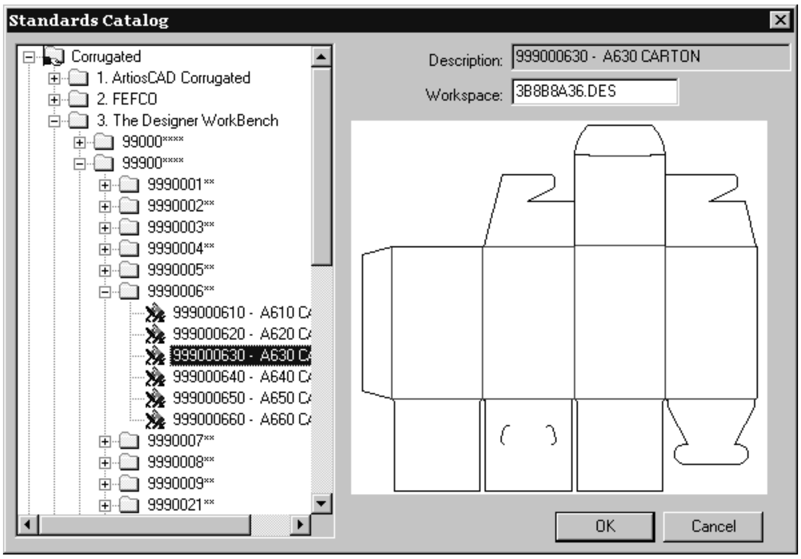
Running non-ArtiosCAD standards

ArtiosCAD can run standards from INTERACT, LASERPOINT, LASERPOINT IQ 1, LASERPOINT IQ 2-4, and The Designer WorkBench.

The standard libraries from INTERACT, LASERPOINT, LASERPOINT IQ (all versions), and The Designer WorkBench standards are optionally loaded when ArtiosCAD is installed.

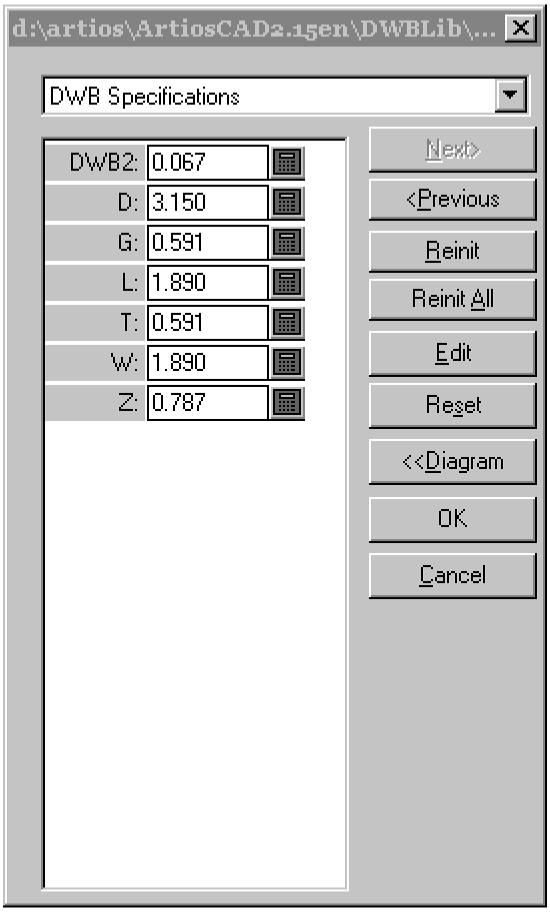
Designer WorkBench and LASERPOINT IQ 2-4 Standards

The Designer WorkBench and LASERPOINT IQ 2-4 standards run as if they were native ArtiosCAD standards.



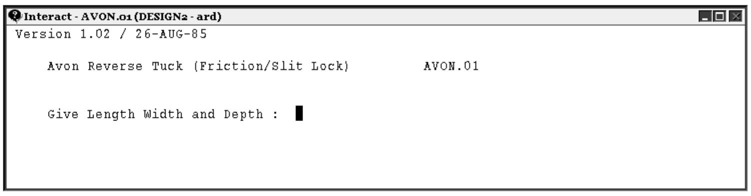
Variables and style choice menus work the same way for these standards as they do for native ArtiosCAD standards. Run them as you would normally run any other ArtiosCAD standard.

Designer WorkBench standards run with the same default values for variables that they had when they were run in Designer WorkBench. However, the #in the name of the variable in DWB is converted to **DWB** when the standard is run in ArtiosCAD.



INTERACT, LASERPOINT, and LASERPOINT IQ 1 standards

Standards made using INTERACT, LASERPOINT, and LASERPOINT IQ 1 are run using a separate data-entry window.

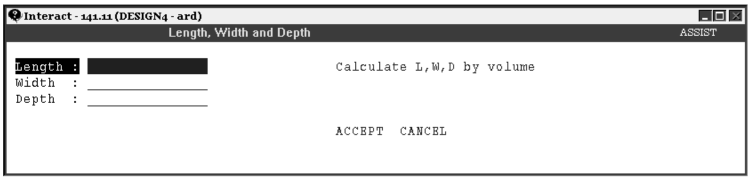


To run INTERACT standards, you must enter information in the same format as the prompt used to request it. In the picture shown above, you are prompted to `Give Length Width and Depth.` This means you must enter the dimensions on the same line with nothing but spaces between them – e.g. `5 2 5`. When you have finished entering data, press `Enter`.

Note:

When running an INTERACT standard, do not click inside the black-background window. This will be interpreted as an Enter keystroke and the window will close. ArtiosCAD will issue an error message and you will have to start over.

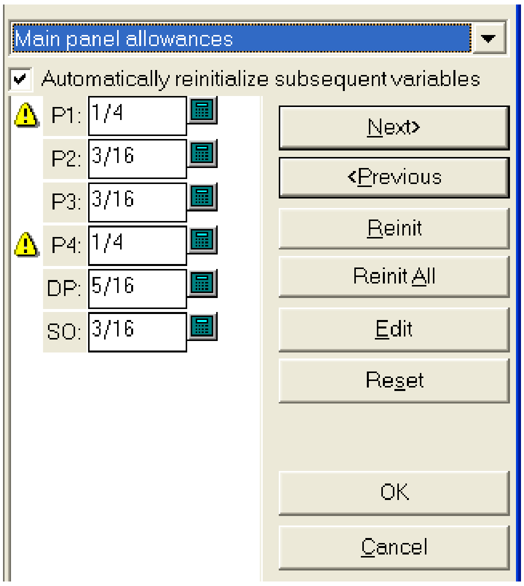
LASERPOINT and LASERPOINT IQ 1 standards also run using a black window for data input. Enter a value in each field underlined with a yellow line. Click **ACCEPT** to proceed through the menus. Click **OVERVIEW** to go to a menu of all menus. Click **REINIT** to reset the variables in the current menu to their default values.



Rebuilding any of these standards will bring up the appropriate black-background window.

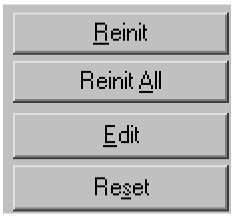
Rebuilding designs

 To rebuild a design, click the **Rebuild** button on the Toolbar. The same dialog boxes which appeared when you built the design will reappear. Change the values to change the dimensions of the final design.



Most variables you change from their default values are flagged with yellow triangles as soon as you click something else. Variables which have both a default formula and a default value that fails a check condition (such as `W` with a default formula of `0` and the check condition being `W>0`) are not flagged with yellow triangles. Style choice variables with default formulae are also not flagged.

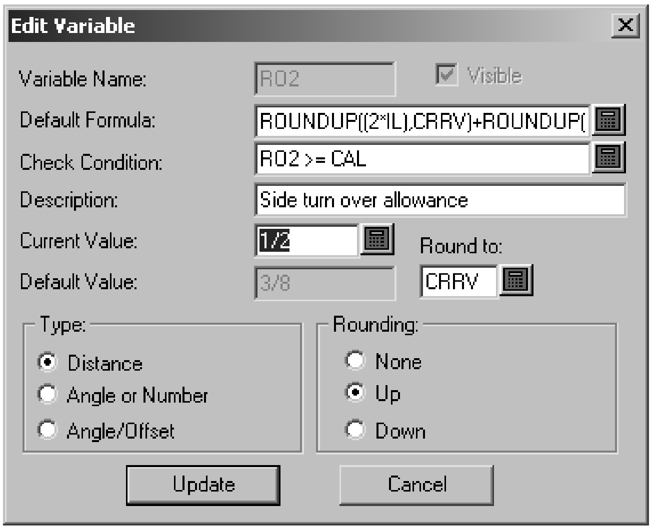
Four buttons and one option in the Rebuild dialog box change the values of the variables as you rebuild the design:



Reinit reinitializes the currently-selected variable to its default value as defined in StyleMaker.

Reinit all reinitializes all variables in the current menu to their default values as defined in StyleMaker.

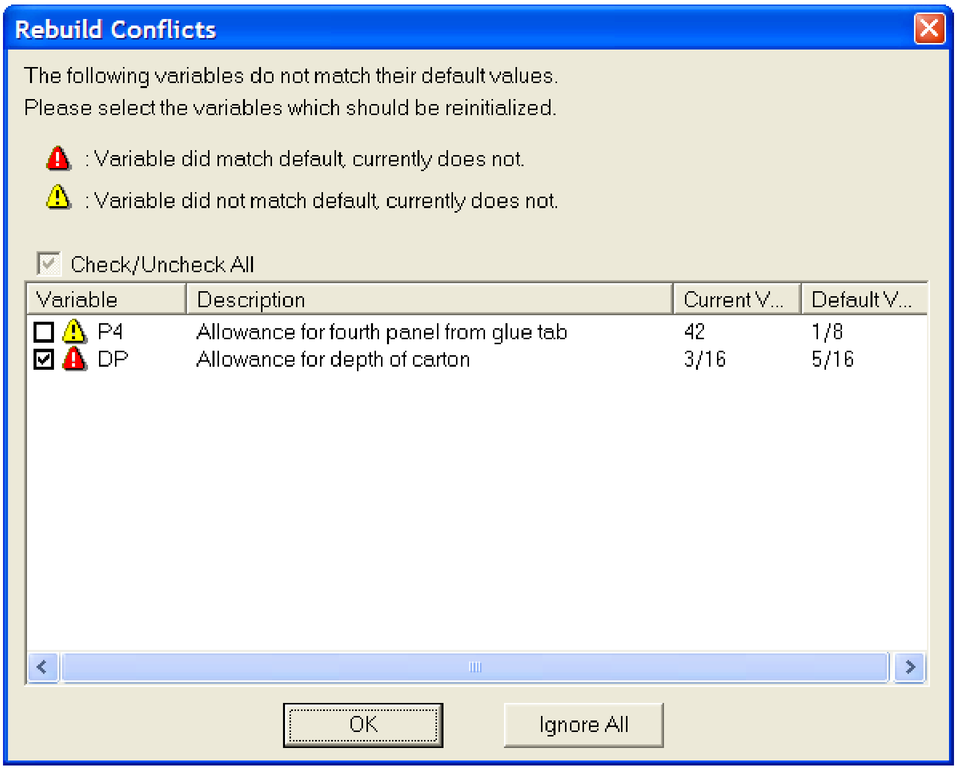
Edit opens the Edit Variable dialog box where you can edit the formula used by the variable, the variable's check condition, the description of the variable, the variable type, the current value of the variable, and the rounding method and amount.



Reset resets the variable to the value it had when the menu was accessed last. For example, if you change a variable from 5 mm to 10 mm, go to the next menu, realize you made a mistake and go back, change the variable to 7 mm and click **Reset**, the variable will change to 10 mm, which is what it was when you entered the menu. If you wanted to change it back to 5, click **Reinit** instead of **Reset**.

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.



The checkbox to the left of the variable name controls if the variable is reinitialized to its current default value when you click **OK**. Variables with yellow triangles are not automatically selected, while those with red triangles are automatically selected for reinitialization.

Variables with yellow triangles are those which did not initially match their default values at the start of the rebuild. Variables with red triangles did match their default values, but currently do not.

Click **OK** to reinitialize the checked variables and continue the rebuild. Clicking **Ignore All** ignores the conflicts and continues the rebuild process.

Note: This value-checking procedure only works for variables which have default values set. There is no checking for variables which have no default set but instead depend on other variables.

Annotations and Dimensions

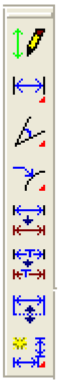
Builder allows you to add dimensions and annotations to your designs.

Dimensions are measurements showing the length of lines, the sweep of angles, the radii of arcs, and so forth. The tools on the Dimension toolbar control the creation, placement, and properties of dimensions.

Details are magnifications of a user-defined area of the design that show the construction in detail.

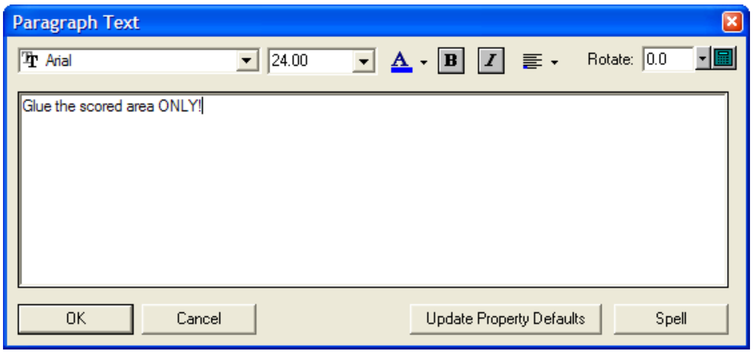
You can also add text and arrows using those buttons on the Annotations toolbar. Both toolbars are shown below.

Note: Some of the tools on the Annotation toolbar are only available if you have Designer in addition to Builder.



Paragraph Text tool

 The first button on the Annotation toolbar lets you add a paragraph of text. Click the tool, indicate the position for the text, and enter the desired text in the Paragraph Text dialog box.

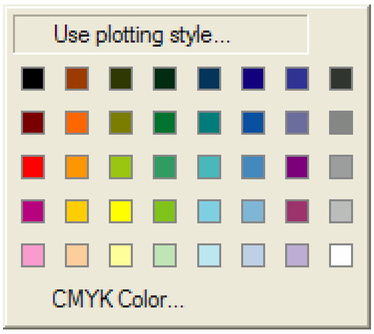


As you enter text in the dialog box, it appears as drag text at the put-down point in ArtiosCAD. Changes are immediately reflected in the drag in the design window, and they apply to all the text entered in the dialog box.

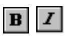
Set the font and font size using the drop-down list boxes at the top of the dialog box. You may also change the font size manually by typing the new size in the drop-down list box field.

Note: PCIQ fonts may appear incorrectly in drag text but should appear correctly when the text is placed.

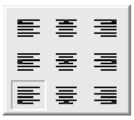
 The color under the **A** indicates the color of the text. A bright blue indicates that the plotting style color is being used. When you click the button, a control appears. Click the desired color. **Use plotting style** sets the text to the color defined by the plotting style. Below that are standard color swatches. **CMYK Color** lets you choose or define a color using its percentages of cyan, magenta, yellow, and black.



Note: If you assign a color to text, it is considered graphics and will not be output to a CAM device using an internal ArtiosCAD driver. If you need to output text to a CAM device, do not change its color. However, define a color when planning to convert the design to 3D and export it to VRML.

 The **B** and **I** buttons toggle **bold** and *italic*.

 The **Justification** button shows how the text is currently justified to the put-down point. When clicked, it shows 9 buttons corresponding to different placement options; click the desired button to change the justification.



The controls in the **Rotate:** field let you rotate the text as desired. Either select the desired degree of rotation from the drop-down list box, enter it in the field manually, or click the keypad icon and use the keypad to input the degree.

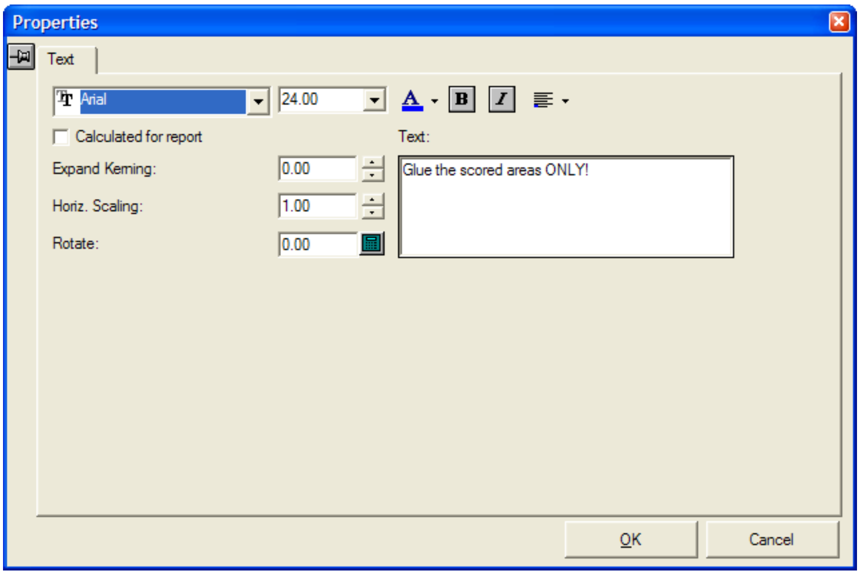
While the Paragraph Text dialog box is open, reposition the text as desired by clicking in different places.

Update Property Defaults sets the current font and size values in the dialog box for use in future instances of this dialog box in the same ArtiosCAD session.

Once you have entered the text in the Paragraph Text dialog box, click **OK** to place it.

After you have added text, double-click it with the **Select** tool to access its Properties dialog box. Use the commands in the Properties dialog box to change the attributes of the text, such as font , color, size, rotation, kerning, etc. Click **OK** to make the changes, or click **Cancel** to discard them.

Note: Note: ArtiosCAD supports up to 38 fonts in a workspace.

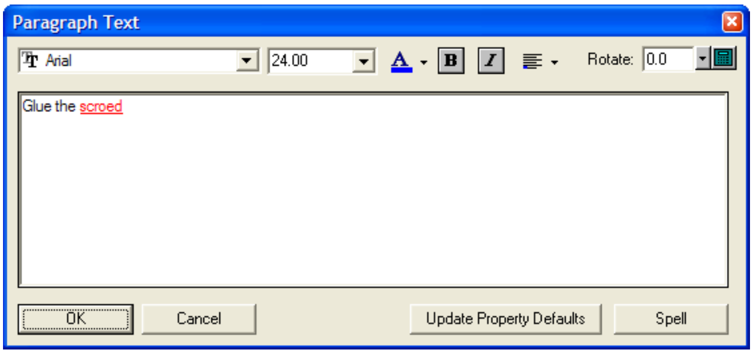


The controls in the Properties dialog box are the same as the controls in the Paragraph Text dialog box, except **Calculated for report** lets you modify the properties of calculated text on Reports.

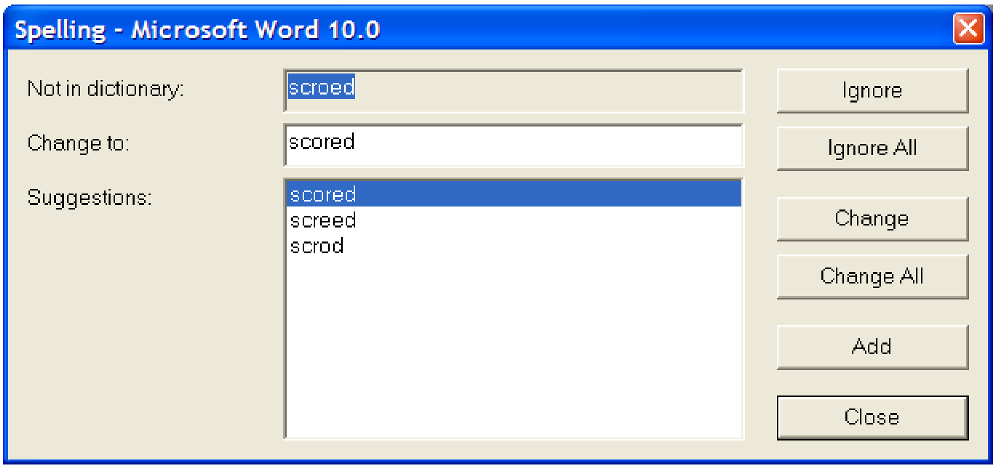
Spell checking in the Paragraph Text tool

If you have Microsoft Word installed on your system, ArtiosCAD can use its spell checking functionality if it is enabled in Defaults (**Options** > **Defaults** > **Shared Defaults** > **Startup defaults** > **Spelling**). It is enabled by default. The Paragraph Text dialog box may take a few moments to appear after the tool is clicked for the first time.

As you enter text in the **Paragraph Text** tool, ArtiosCAD communicates with the spell checker in Microsoft Word, and any unrecognized words are underlined and shown in red.



To spell check the unrecognized word(s), either right-click and click **Spell** on the context menu, or click **Spell**. The standard Microsoft Word Spelling dialog box appears similar to the one shown below.



The unrecognized word is shown in the **Not in dictionary:** field. The **Suggestions:** field contains the list of suggested words, and the top one is selected and seeded to the **Change to:** field. You may edit the word in the **Change to**: field as desired.

Ignore and **Ignore All** respectively ignore this instance and all subsequent instances of the unrecognized word for this invocation of the spell checker. The next unrecognized item is then displayed.

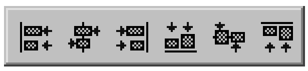
Change changes the current instance of the unrecognized word to the contents of the **Change to:** field. **Change All** changes all subsequent instances of the unrecognized word in the workspace to the contents of the **Change to:** field; previously-ignored instances are not changed. The next unrecognized item is then displayed.

Add adds the unrecognized word to the custom ArtiosCAD dictionary file so that it will be recognized in the future. The next unrecognized item is then displayed.

Close closes the Spelling dialog box and returns to the Paragraph Text tool.

Align Text tools

The second button on the Annotation toolbar activates the **Align Left** tool, and when held down, activates the Align Text tools flyout toolbar.



The **Align Left** tool aligns the selected items along their left edge.

The **Align Horizontal Centers** tool aligns the selected items along their horizontal centers.

The **Align Right** tool aligns the selected items along their right edge.

The **Align Bottom** tool aligns the selected items along their bottom edges.

The **Align Vertical Centers** tool aligns the selected items along their vertical centers.

The **Align Top** tool aligns the selected items along their top edges.

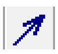
To use these tools, do the following:

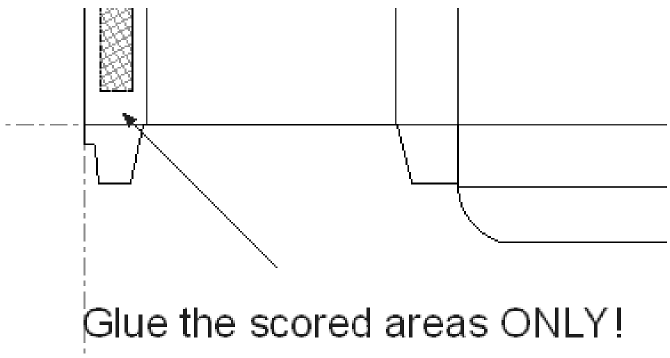
1. Select one or more items of text, and click the desired Align Text tool. The drag will show a common alignment line.

2. Indicate a pick-up point, and then use the drag to set the put-down point. Shown below is the drag for the **Align Left** tool.

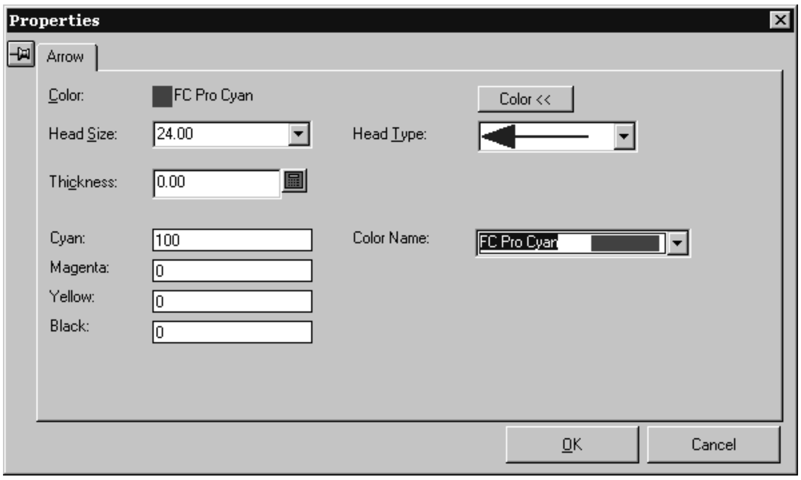
3. The text will be aligned in the manner consistent with the tool chosen.

Arrow tool

 The third button on the Annotation toolbar activates the **Arrow** tool. To make an arrow, click the Arrow tool, click the start point of the arrow, and then click the end point of the arrow. The head of the arrow will be drawn at the end point of the arrow.

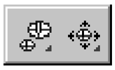


Once you have added an arrow, double-click it to change its properties.

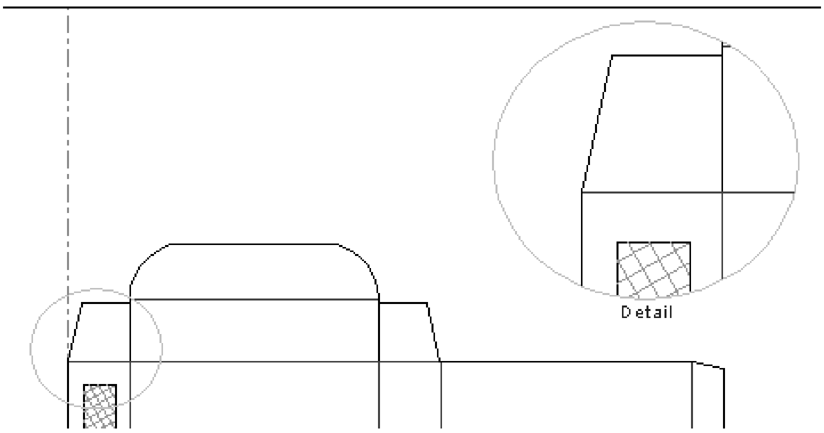


Detail tool

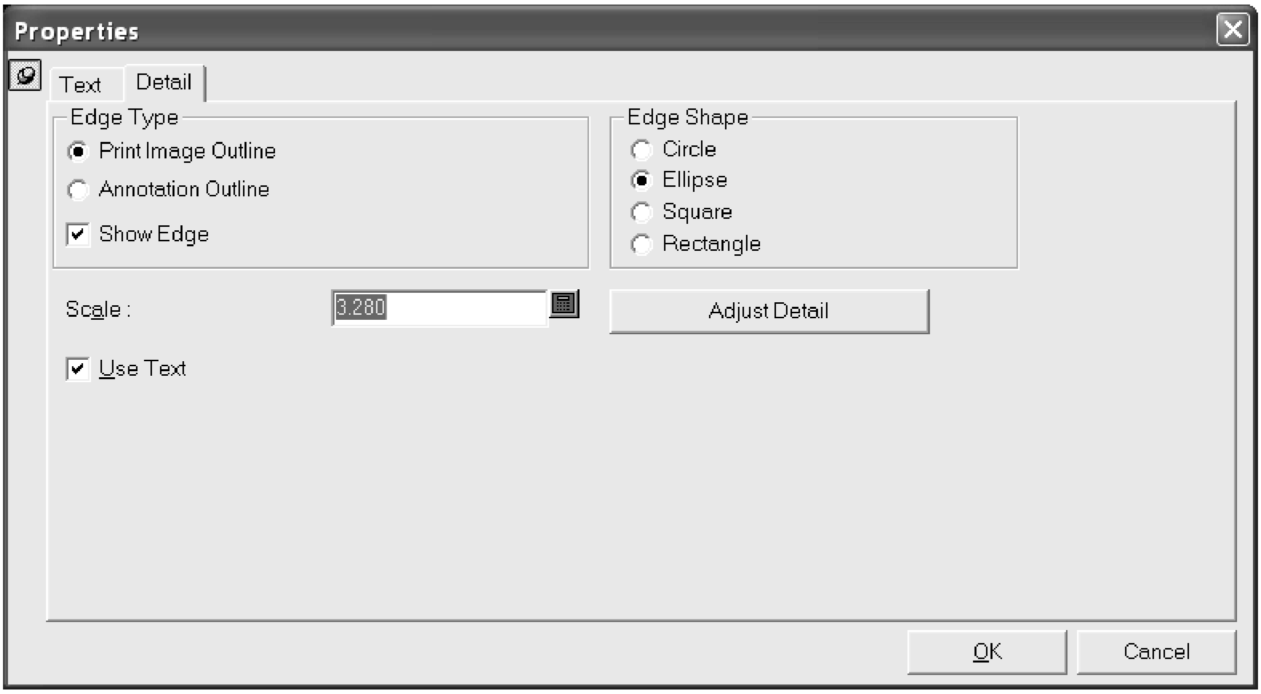
 The fourth button on the Annotation toolbar activates the **Details** flyout toolbar. Details are magnifications of a designated area of the design.



The first button on the Detail toolbar, and the default Detail tool shown on the Annotation toolbar, is the **Detail** tool. The Detail tool, when clicked, prompts for the first corner of the clipping window. The clipping window (window 1 below) defines the area of the design to magnify. Indicate the corner, keep depressing the mouse button, and indicate the second corner of the clipping window. ArtiosCAD then prompts for the first corner of the detail window. The detail window (window 2 below) is where the magnified view appears. Click the mouse button at the first corner, and drag the mouse to the diagonally opposite corner of the detail window.



Double-click any item inside the detail window to change the properties of that detail. Click the **Detail** tab in the Properties dialog box to view the properties of the detail.



The clipping and detail windows will update as their properties are changed. The **Adjust Detail** button closes the Properties dialog box and starts the **Adjust Detail** tool.

Adjust Detail tool

 The second button on the Detail flyout toolbar activates the **Adjust Detail** tool. The Adjust Detail tool modifies the size and position of the clipping window and the detail window.

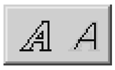
The Adjust Detail tool has three modes: **Move Clip**, **Move Detail**, and **Move Side**. **Move Clip** lets you move the clipping window to another location. Use this option by selecting the **Move Clip** option button, clicking inside the clipping window, and clicking at the new location of the clipping window.

Move Detail lets you move the detail window to another location. This option works the same way as **Move Clip**, except click in the detail window instead of the clipping window.

Move Side lets you drag a side of either the clipping window or the detail window to a new location, thereby changing the size of the window.

Outline text tools

 The **Outline Text** tool, which is the fifth button on the Annotation toolbar, activates the **Outline Text** tool when clicked, and when held down, activates the Outline Text Tools flyout toolbar.



This tool converts text into print image lines. The lines comprising letters or numbers are placed in separate groups. Shown below is text before and after being converted to outlines.

Use this tool by selecting the text to convert and then click the button on the toolbar.

Line text tool

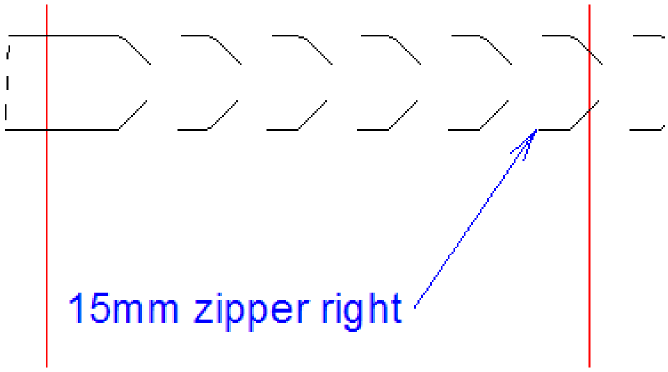
 The second button on the Outline Text tools flyout toolbar activates the **Line Text** tool.

This tool converts text into geometric lines which can be manufactured. To use this tool, click it, and then click the block of text to change. The text will change to lines with the closest possible appearance to the original font. Shown below is a block of text in the Georgia TrueType font before using the Line Text tool:

After the tool is used, the text appears like this:

Line type label tool

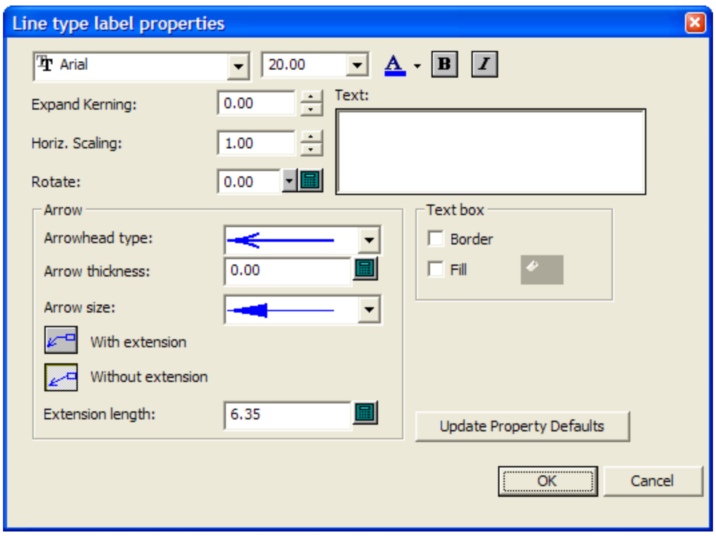
 The sixth button on the Annotation toolbar activates the **Line Type Label** tool. This tool creates a piece of text and an arrow pointing to a particular line.



The following controls appear on the Status bar when this tool is active:

The **Size**: drop-down list box changes the size of the line type label you are about to make.

More Options (**...**) opens the Line Type Label Properties dialog box as shown below.



Standard font controls are arranged along the top of the dialog box. **Expand Kerning:** increases or decreases the space between letters. **Horiz. Scaling**: increases or decreases the overall width of the text. **Rotate** allows you to place the text at an angle.

In the **Arrow** group, choose an **Arrowhead type:** and overall **Arrow size:** from their respective drop-down list boxes. **Arrow thickness:** adjusts the arrow thickness in points.

With extension creates a short horizontal line out from the label text using the value in the **Extension length:** field before the start of the arrow. **Without extension** places the end of the arrow directly next to the label text.

In the **Text box** group, **Border** and **Fill** each create their namesakes around the label text. The border has the same color as the label text, but the fill color is set using the button that becomes available when **Fill** is checked.

Update Property Defaults makes subsequent line type labels made in this session of ArtiosCAD use the same Properties as those set in the current dialog box.

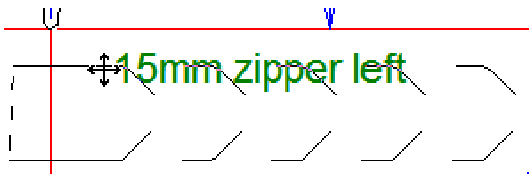
Returning to the Status bar, **Update Property Defaults** sets the size of the current line type label to be the default for the session.

Text: is the text that will become the line type label. If this field is blank, ArtiosCAD will automatically fill it in with the selection in the **Initial text:** field as you drag across lines.

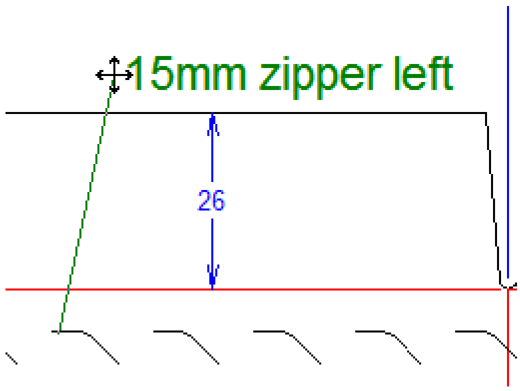
The **Initial text:** drop-down list box lets you choose what ArtiosCAD automatically inserts in the **Text:** field as you drag. You can choose the **Line type name**, the **Line type name and label**, and the **Line type label**. The **Line type label** only applies for special rules, and is defined in the **Rule label:** field on the Manufacturing tab of the Properties dialog box for a special rule; rule labels may also be defined for special rules in Defaults.

To make a line type label, do the following:

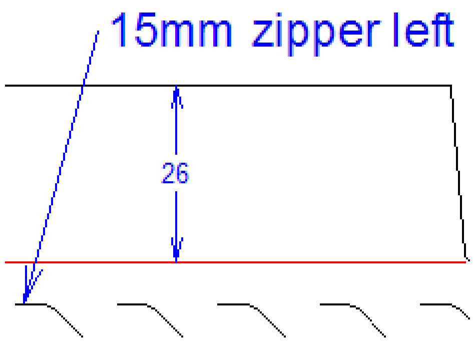
1.  Click the **Line Type Label** tool.
2. If the current layer is not appropriate for a line type label, ArtiosCAD prompts you to change to an appropriate layer or confirm using the current layer.
3. As you drag, the cursor will show the name of the nearest rule or special rule. If you do not click close enough to a rule, enter the text for the label in the **Text**: field on the Status bar; the input focus is set to that field if ArtiosCAD does not fill it in automatically and the drag will show the text you entered. Click at the point or coordinate which will be the end of the arrow.



4. Use the drag to set the position of the label.

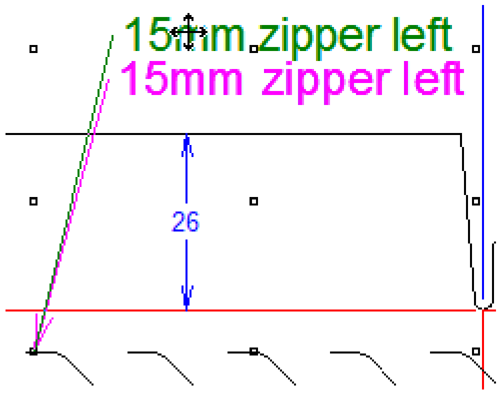


5. If desired, change the options on the Status bar. The drag will remain active.
6. Click to set the final position of the label.

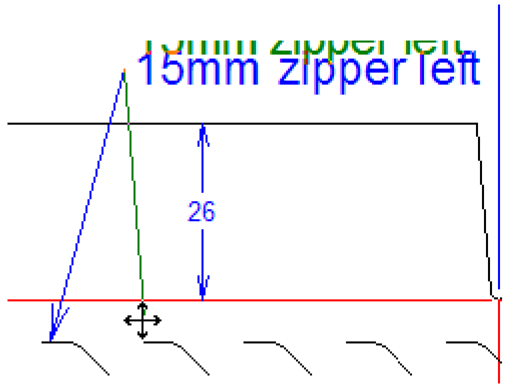


7. The tool remains active in order to add more line type labels. Either make more line type labels or activate another tool.

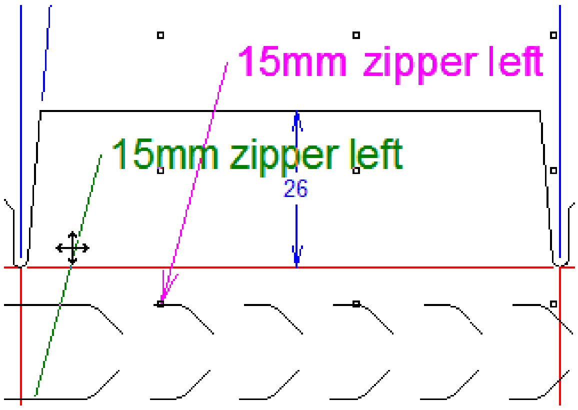
You can use drag to move the line type label (or elements of it) after it is created. If you use the Select tool and click near the text box, you can move the text box.



Click near the end of the arrow to move the end of the arrow.



To move the entire label, click somewhere else nearby.



The default initial text option can be changed in **Options** > **Defaults** > **Startup defaults** > **Line type label options**. Text defaults for this tool are in **Options** > **Defaults** > **Property defaults** > **Line type label**.

Note: Avoid using the Select tool in **Selection ignoring groups** mode to delete individual parts of a line type label, as this will lead to unpredictable results.

If you change the line type of a line which has a line type label associated with it, ArtiosCAD now automatically updates the line type label to match the new line type according to the table of conditions below.

Condition for the existing line type label	Result after changing the line type
Line type label has custom text entered in the **Text** field.	No result; the label is not changed.
Label initial text is set to **Line type label**.	Changed to match new line type.
Label initial text is set to **Line type label** (defined on the **Manufacturing** tab of the properties of a special rule type).	Changed to match the new Manufacturing name if one is defined for that special rule, else changed to match new line type.

Condition for the existing line type label	Result after changing the line type
Label initial text is set to **Line type name and label**.	Both are changed.
Line type label is in a turned-off layer.	The line type label is changed and the layer is turned on.
Line type label is in a locked layer.	The line type label is not updated and a warning appears.
Line type label is in a single design embedded in a manufacturing document.	The undo history is cleared and a warning appears that this action cannot be undone.

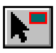
Hatch tool

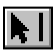
 The **Hatch** tool fills an area of the design with either a pattern of lines or a custom pattern you have designed and added to the Hatch Catalog.

- It uses a catalog of entries to use different hatches. ArtiosCAD displays a catalog control on the Status bar when the Hatch tool is active.
- You can design custom hatch patterns, including those that do not use lines, to form any fill pattern desired.
- You can Output a Hatch Legend, which shows up to 7 hatches used in the current single design.

Hatch Status bar

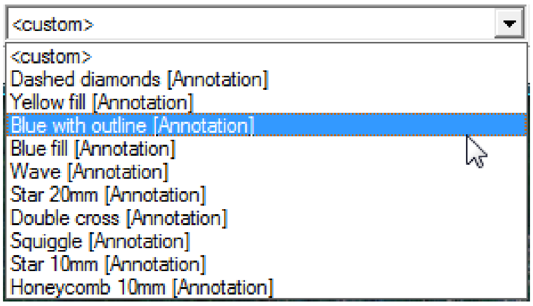
When you click the **Hatch** tool, the following controls appear on the Status bar:

 The first control on the Status bar is the toggle for **Select a Panel** mode. When the tool is in this mode, ArtiosCADit hatches the panel you click inside.

 The second control on the Status bar is the toggle for **Select Lines** mode. When the tool is in this mode, ArtiosCAD prompts you to select the lines forming a loop that, when complete, will be hatched.

Hatch Lines, the third control, is available only when a complete loop of lines is selected when the tool is in Select Lines mode. It hatches the area within the loop.

Use the fourth control on the Status bar to select an entry from the Hatch catalog. Click the drop-down list activator and click the desired entry.



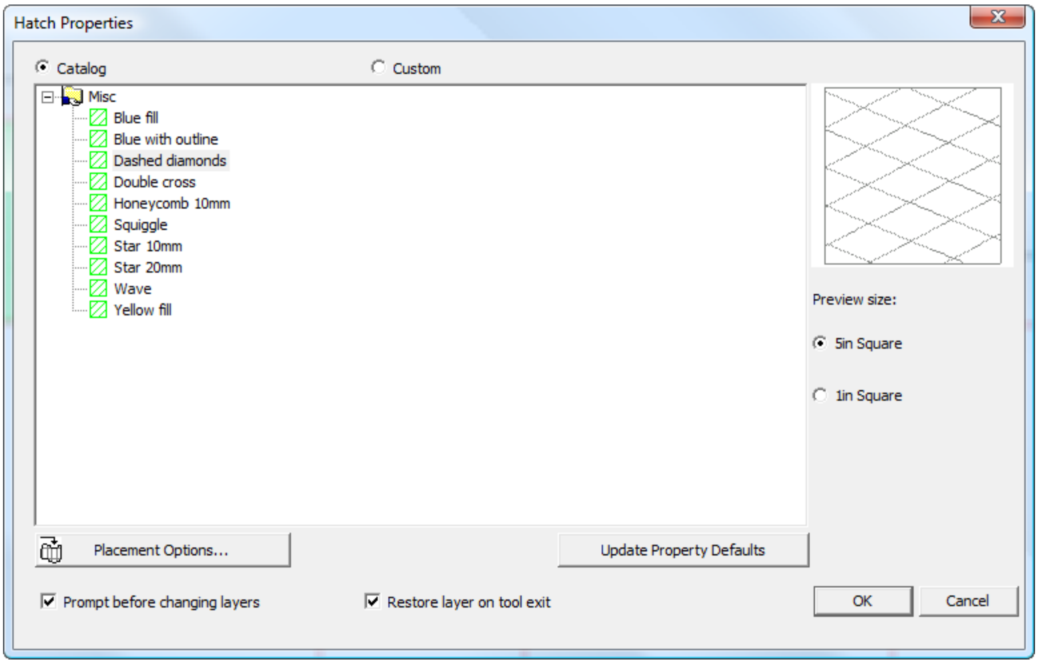
The fifth control on the status bar, ..., opens the **Hatch Properties** dialog box. If the catalog selection is set to a specific selection, the Hatch Properties dialog box for that selection opens showing the entry selected in the catalog. Click a different catalog entry to change the hatch style.

Clicking the **Custom** option button with a catalog entry selected changes the dialog box to show the properties for the selected entry. This same dialog box appears when **<custom>** is selected in the drop-down list box on the Status bar and you open the Hatch Properties dialog box, except the properties for the selected entry are shown, not the default non-cataloged hatch of green lines at 45 and 135 degrees. Refer to the next section for more information about this dialog box.

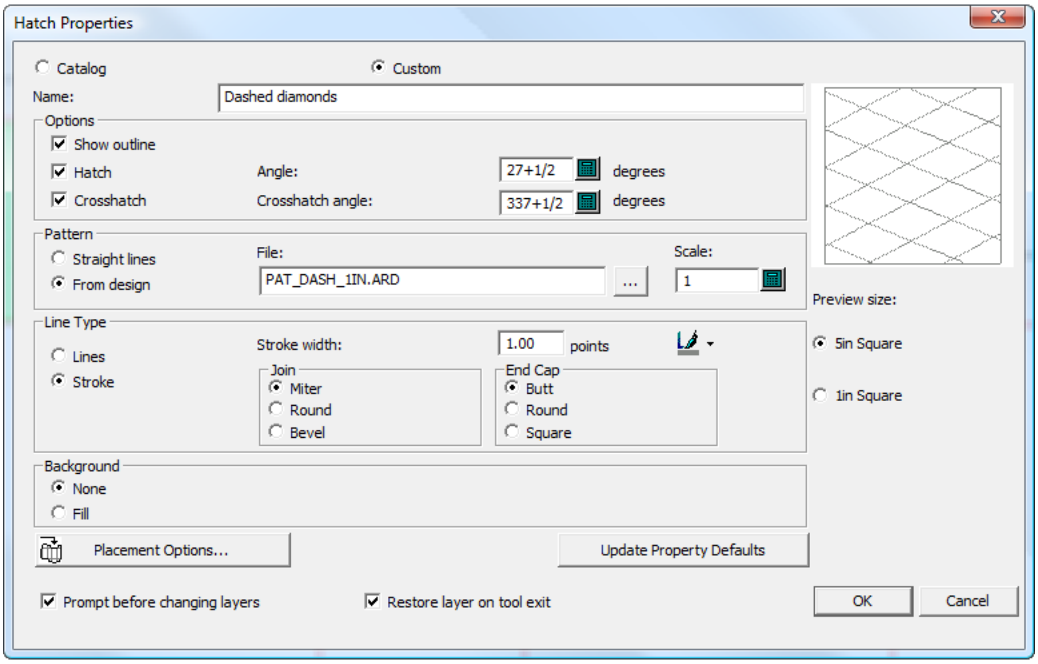
The values in **Offset from cuts** and **Offset from creases**, the sixth and seventh controls on the Status bar, both control how far away the hatch stays from the lines defining the panel or area being hatched.

Hatch Properties dialog box

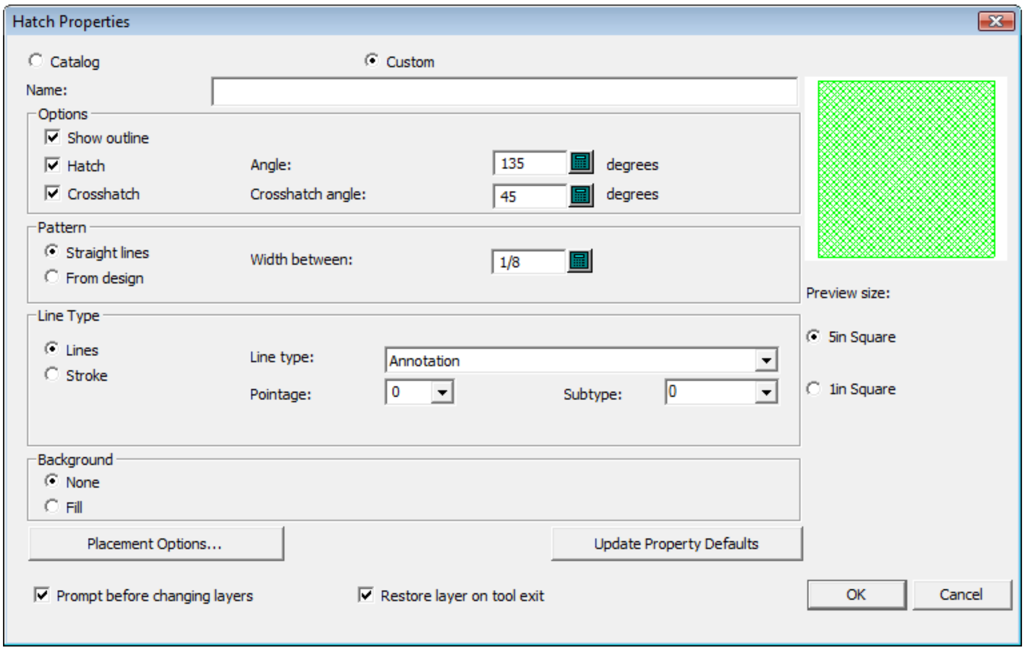
The **Hatch Properties** dialog box has two variants - **Catalog** mode and **Custom** mode. In Catalog mode, the currently-selected catalog entry is highlighted and you can select a different catalog entry if desired.



When you start in Catalog mode and click **Custom**, the properties of the catalog entry appear.



In Custom mode with no catalog entry selected, you can specify a pattern workspace or line style to use, along with other properties such as line type and background fill.



When you view the properties for an existing hatch by double-clicking its edge with a Select tool, the Properties dialog box does not display a preview of the hatch, because the hatch in the workspace is updated as you change values in the dialog box. In addition, the placement options and related fields are hidden when viewing the properties of an existing hatch.

Enter a name for these hatch properties, if desired, in the **Name** field.

The controls in the **Options** group define the construction of the hatch. If **Show outline** is checked, a border is drawn around the hatch that is a scaled-down copy of the lines defining the hatch area. **Hatch** controls the construction of the hatch lines made at the value specified in the **Angle** field. **Crosshatch** makes a second set of lines at the angle defined in **Crosshatch angle**. Deselecting **Hatch** turns off the crosshatch and makes its checkbox unavailable.

The **Pattern** group controls determine if the hatch is made of **Straight lines** or is based on a design file you specify by choosing **From design** and then browsing for the design in the **File** field. If you choose straight lines, set the distance between them using the **Width between** field. If you choose a design file, set the scale of the pattern using **Scale**.

In the **Line Type** group, to use just structural lines, choose **Lines** and set the **Line type**, **Pointage**, and **Subtype** fields appropriately. To use stroked structural lines, choose **Stroke** and set the **Stroke Width**, **Join**, and **End Cap** as desired.

 To set the color of the stroke, click the standard Windows color control (the underlined paintbrush) and then choose the desired color from the pop-up dialog box.

In the **Background** group, to have a fill behind the hatch, choose **Fill** and set the color with the standard Windows color control. To have no fill, choose **None**. Fill will cover the background regardless of the layer it is in.

Placement Options is described in the next section.

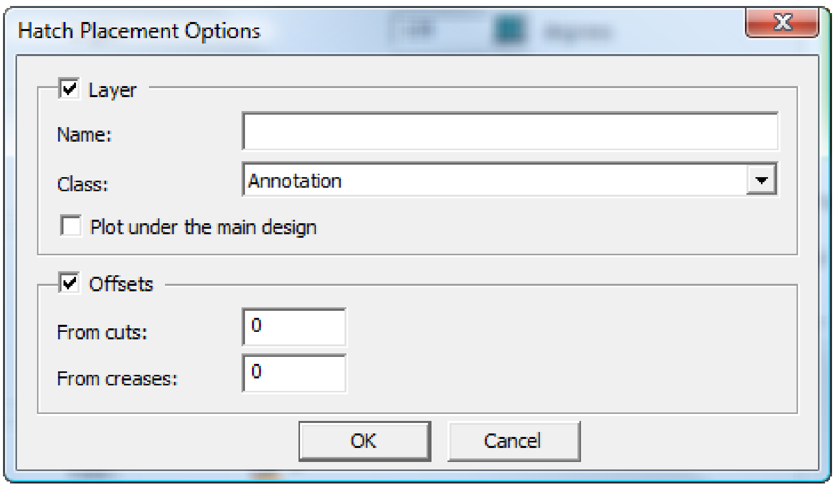
Update Property Defaults updates the current design's Defaults so that future hatches you make in this design will use these same properties. Clicking this button does not affect hatches you have already made.

Prompt before changing layers controls whether ArtiosCAD asks you to confirm changing layers if you are in a different layer than the one specified in the Placement Options dialog box. If you do change layers for hatch, **Restore layer on tool exit**, when checked, returns you to the layer that you were in before you started the Hatch tool.

Choose the larger **Preview size** option to see the preview from further away; choose the smaller option to see the preview more closely.

Hatch placement options

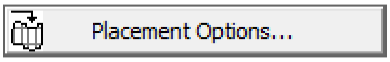
Clicking **Placement Options** in the **Hatch Properties** dialog box opens the **Hatch Placement Options** dialog box.



To create the hatch in a specific layer other than the current one, check the **Layer** checkbox, enter a name for the layer in the **Name** field, and choose the layer class in the **Class** drop-down list box. **Plot under the main design** draws the hatch and fill (if any) beneath the structure.

To create a gap between the hatch and any cuts or creases in the design, check **Offsets** and enter the offset values in the fields.

When either checkbox is checked, the **Placement Options** button in the Hatch Properties dialog box changes to include the icon as shown below.

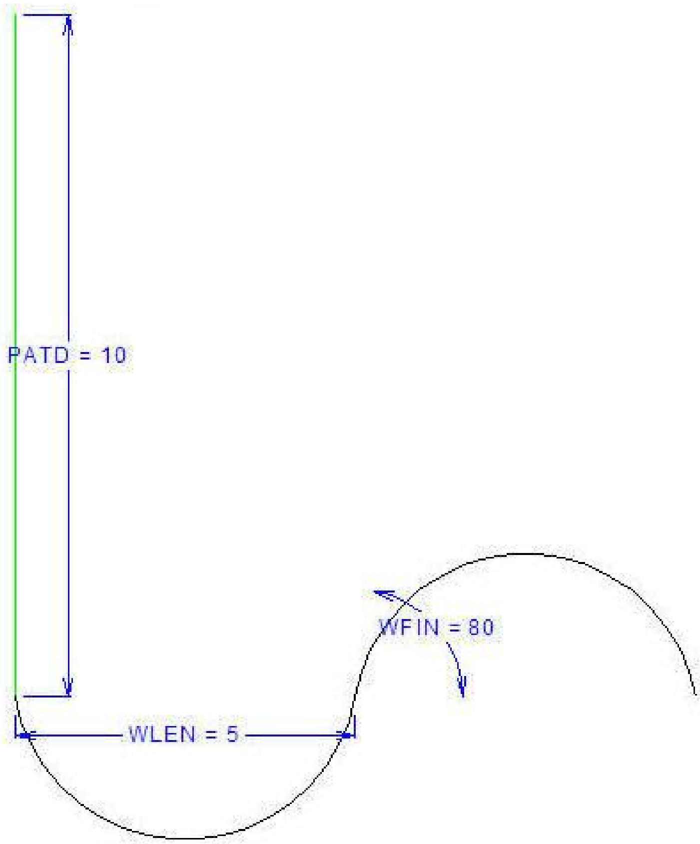


Using a hatch pattern file

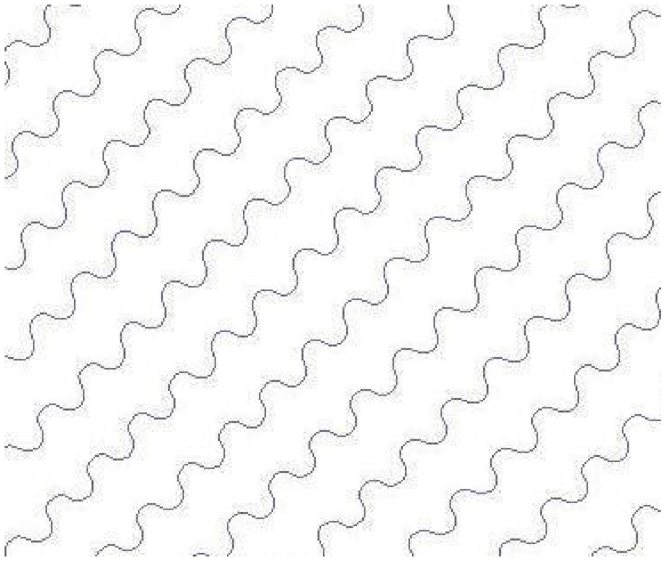
To design custom hatch patterns, make single design workspaces with the desired geometry following the guidelines in this section. Custom hatch patterns may function as both hatches and crosshatches.

If you have StyleMaker, you can make the workspaces resizable, but note that they will not resize when you resize the design - you must create different versions of the pattern workspace at the desired sizes, save those files in `ServerLib` or `ClientLib`, and add Hatch Catalog entries that use those workspaces.

Shown below is the `PAT_CIRCLEWAVE.ARD` pattern workspace.



The length of the vertical annotation line is defined by variable `PATD`. The annotation line sets the distance between the wavy lines when the pattern is stepped and repeated at 45 degrees as shown below.



In the example workspace, variable `WLEN` sets the horizontal distance for each arc and variable `WFIN` sets the start angle of the arcs.

To make a custom hatch pattern workspace, do the following:

1. Start ArtiosCAD and create a new single design.
2. To make the pattern resizable, you must have StyleMaker. If so, define the variables you will need to construct the geometry.
3. Draw the desired pattern at 0 degrees horizontally. Use an Annotation line connected to the start of the pattern to define the distance between the pattern repetitions.
4. Save the workspace to `ServerLib` or `ClientLib`.
5. Add an entry to the Hatch Catalog in Defaults that uses this pattern workspace.
6. Use the catalog entry as desired.

To make different versions of the same resizable pattern, do the following:

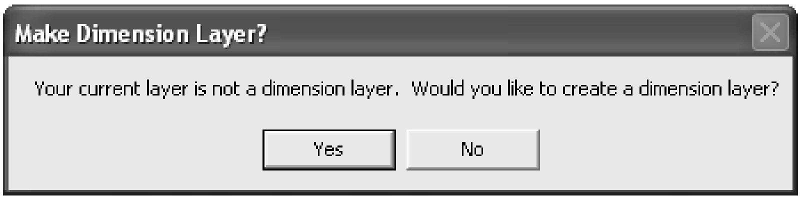
1. Resize the pattern workspace as desired.
2. Save each version to `ServerLIb` or `ClientLib` with a unique name.
3. in Defaults, create a Hatch Catalog entry for each iteration of the pattern workspace.
4. Use the new Hatch Catalog entries as desired.

Dimensions

Dimensions are measurements added to the design so that someone looking at the design knows the length, angle, and radii of the geometric elements of the design. Shown below is the Dimensions toolbar.

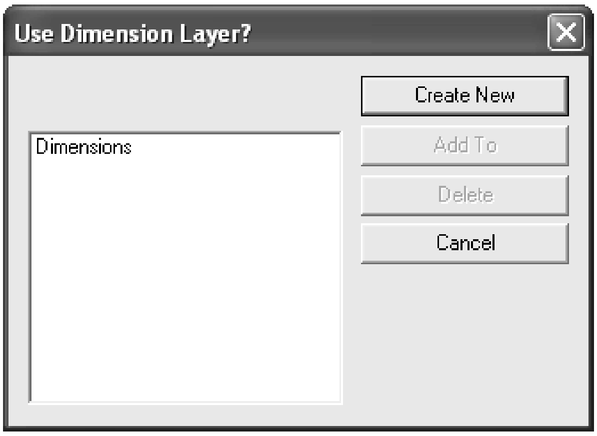
Before creating dimensions, ensure there are design lines in the **Main Design**, **Windows and Cutouts**, or **Manufacturing** layer of the design. If there are no design lines in any of those layers, you will get a warning dialog box when attempting to add dimensions. To work around this warning, move design lines to the appropriate layers and either change the dimension size or toggle **Scale size upon rebuild** in **Edit** > **Property Defaults** for Dimensions.

If you are not in a layer with class **Dimensions** or **Overall Dimensions,** the first time you click a dimension-creation tool you are asked if you want to create such a layer.

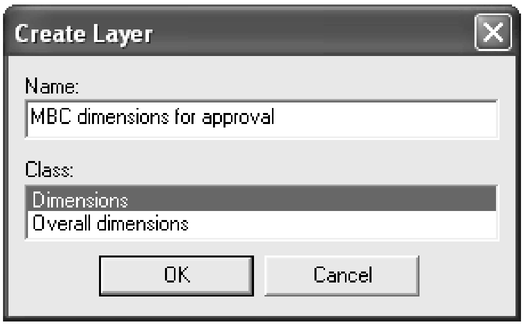


Click **Yes** to create a Dimension layer or **No** to keep working in the current layer. If you click **No**, and then proceed to make dimensions, you are not asked this question again. If you click **No** and do not make any dimensions, you are asked the question again the next time a dimension-creating tool is clicked.

If you click **Yes**, and there is no layer of class **Dimensions** or **Overall Dimensions** already in the design, the Use Dimension Layer dialog box appears.



To create a layer with a custom name, click **Create New**, enter the layer name, choose its class, and click **OK**.



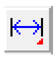
Or, to use a **Dimensions** layer, select it and click **Add To**. You can also use **Add To** to add to selected already-existing layers shown in the list.

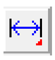
Use **Delete** to delete the selected layer. Clicking **Cancel** completely stops the tool.

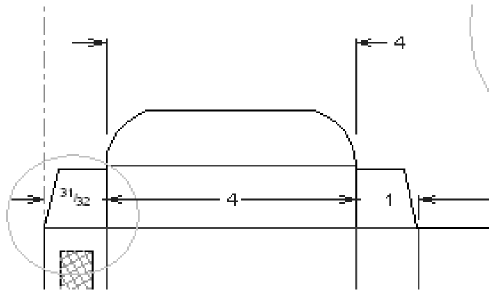
 The first button on the Dimensions toolbar activates **Temporary Dimensions** mode. Dimensions made in this mode:

- Are not subject to any layer creation issues.
- Are made with light green extension lines to distinguish them from regular dimensions.
- Are not selectable, and thus may not be changed or deleted.
- Are not affected by a **Select** tool's Nudge by Stretch mode or by the **Stretch Point** tool.
- May be moved or stretched by the **Stretch by Polygon** tool. If only one of the end-points of a temporary dimension is within the selection polygon, the dimension is stretched instead of moved.
- Are shown only on the screen, and are not printed, Output, or exported to other file formats.
- Are deleted on Rebuild.
- Are in existence only until **Temporary Dimensions** mode is turned off by clicking the **Temporary Dimensions** button again.
- Are not saved when the single design is saved.

If **Temporary Dimensions** mode is active while in a **Dimensions** or **Overall Dimensions** class layer, when the Layers dialog box is opened, the pencil icon is dimmed to indicate **Temporary Dimensions** mode. It returns to normal when **Temporary Dimensions** mode is turned off.

 The second button on the Dimension toolbar, and the default tool on the Distance Dimension flyout toolbar, activates the Distance Dimension tool. This tool measures the distance between two lines or points. To use this tool, do the following:

1. Click  to activate the tool.
2. To adjust the text position of the dimension after setting the line or point to measure to, select **Adjust text position** on the Status bar.
3. Click the line or point to measure from.
4. Click the line or point to measure to.
5. Click the extend position (where the measurement is to be placed).

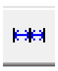


6. If you selected **Adjust text position** on the Status bar previously, click to set the desired position of the text.

If, when making dimensions next to each other, the arrows overlap the text, the overlapping arrows are replaced by a diamond.

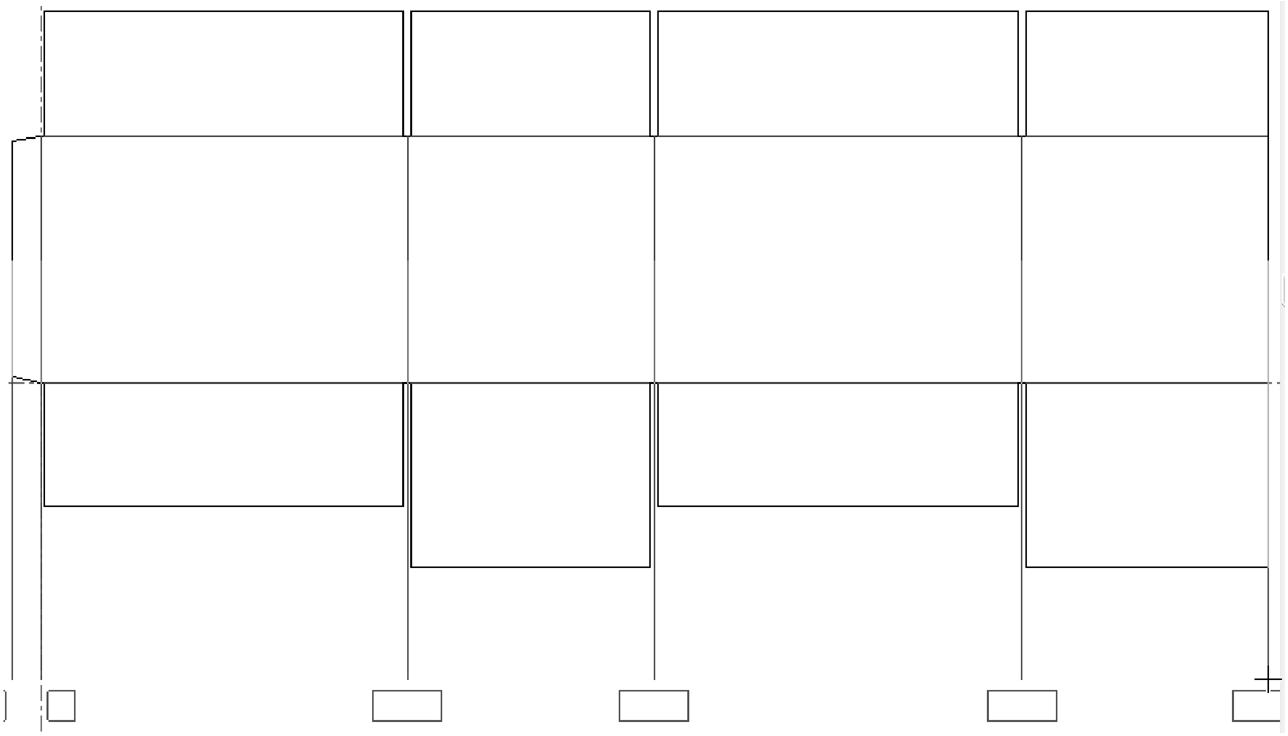
 The second tool on the Distance Dimension flyout toolbar is the **Auto-Aligned Distance Dimension** tool. This tool constructs a series of distance dimensions between consecutive lines and then prompts for a common extension point. To construct auto-aligned distance dimensions, do the following.

1. Click the tool.
2. Click the line to measure from.
3. Drag across the line(s) to include in the aligned dimensions.
4. When you have made as many measurements as desired, indicate the extension point for the dimensions by using the drag and clicking.
5. All the dimensions will then have the same extension point.

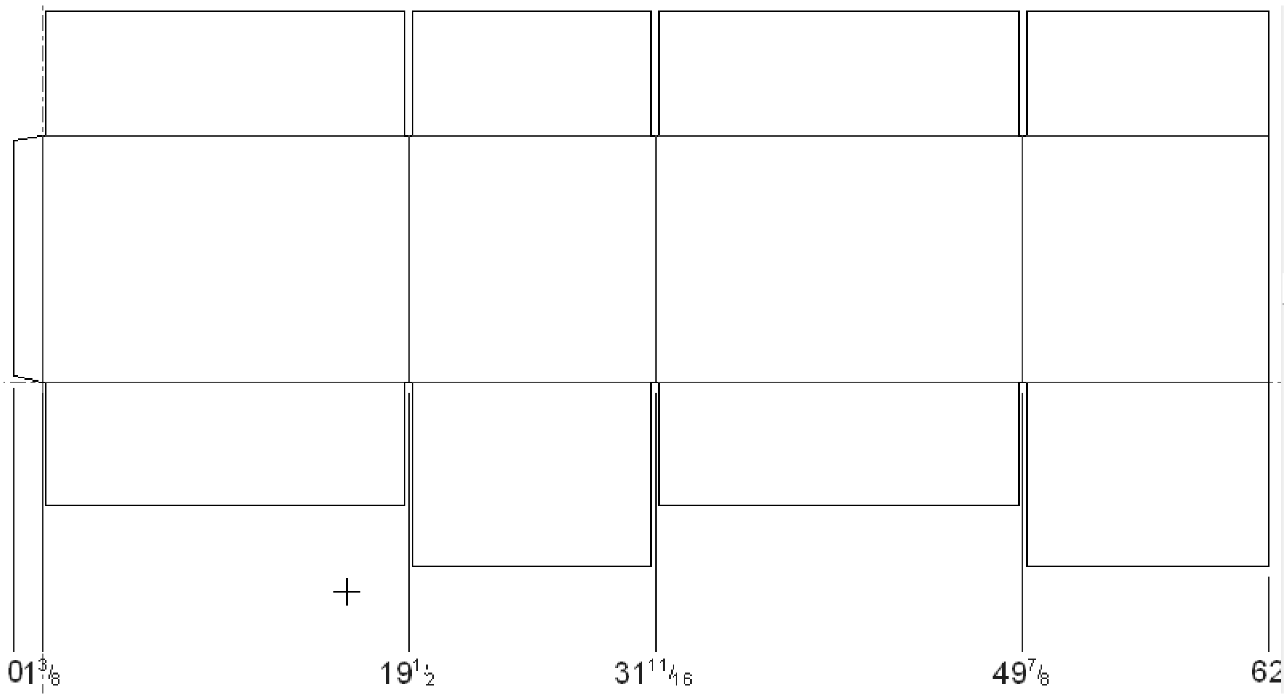
 The third tool on the Distance Dimension flyout toolbar is the **Aligned Distance Dimension** tool. Use this tool to construct consecutive dimensions whose extend positions are collinear. Click the first line to measure, then the second line to measure, and so forth, and then click to set the extension point for all the dimensions.

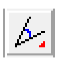
 The fourth tool on the Distance Dimension flyout toolbar is the **Cumulative Distance Dimension** tool. This tool creates cumulative dimensions from a start point along a straight line, measuring the distance from the start point at each intersection. To use it, do the following:

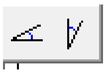
1. Click the tool.
2. Click the point or line to start measuring from.
3. Set the angle at which to measure.
4. Use the drag to intersect all the lines and points to measure.



5. Click to set the number of dimensions; the dimensions are inserted. You may need to adjust their positions to aid clarity.



 The third tool on the Dimensions toolbar, and the default button on the Angle Dimension flyout toolbar, is the **Angle Dimension** tool. This tool measures the angle between two lines. Click this tool, indicate the first leg of the angle, then indicate the second leg of the angle, and then indicate the position of the measurement. The format of the measurement depends on where you position it.

 The other tools on the Angle Dimension flyout toolbar are the **Angle From Horizontal** tooland **Angle From Vertical** tool .

The **Angle From Horizontal** tool measures the angle at which a line is offset from the horizontal axis. To use this tool, click it, indicate the line to measure, indicate the point on the line to measure, and then set the position of the measurement.

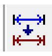
The **Angle From Vertical** tool measures the angle at which a line is offset from the vertical axis. This tool works the same way as the Angle From Horizontal tool except that it measures from the vertical axis.

 The fourth button on the Dimensions toolbar, and the default button on the Radius Dimension flyout toolbar, activates the **Radius Dimension** tool . This tool measures the radii of arcs and circles. To use this tool, click it, indicate the arc or circle to measure, and then indicate the location of the measurement.

 The second tool on the Radius Dimension flyout toolbar is the **Radius To Circle Dimension** tool . This tool measures the distance from an arc or circle to its center. To use this tool, click it, indicate the arc or circle to measure, and then indicate the location of the measurement.

 The third tool on the Radius Dimension flyout toolbar is the **Diameter Dimension** tool. This tool measures the diameter of a circle or arc. To use it, click it, then click the circle or arc to measure, and then click the extension point for the measurement.

 The fourth tool on the Radius Dimension flyout toolbar is the **Diameter to Circle Dimension** tool. This tool measures the diameter or a circle or arc through the center so that the arrows on the extension lines are on the circumference of the arc or circle. To use this tool, click it, then click the circle or arc to measure, and then click the extension point for the measurement.

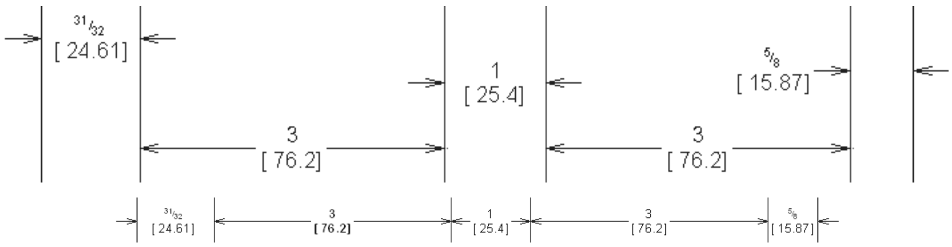
 The fifth button on the Dimensions toolbar activates the **Change Dimension Alignment** tool . This tool lets you align dimensions with each other. This tool is available only when one or more dimensions are selected. To use this tool, select the dimensions to align, click the tool, and indicate the new alignment point. A drag image of the selected dimensions will appear to guide you in selecting the new alignment point.

You can align parallel dimensions by selecting them with the **Select** tool. When you use the drag to move them, they will all change to have the same extension point. You cannot use freehand drag to align non-parallel dimensions. When freehand dragging a single dimension, press the `SHIFT` key to keep the text position in line with its original position.

You can align the selected dimensions with an existing dimension by clicking the existing dimension to snap to.

A snap circle will appear around the dimension text used to align the dimensions.

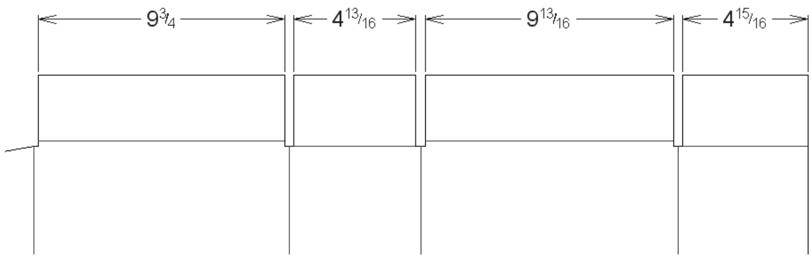
Shown below are dimensions before they are aligned and after they are aligned.



 The sixth button on the Dimensions toolbar activates the **Change Text Position** tool. This tool lets you move the position of text in a dimension. To use it, select the dimension whose text position you want to change, click the tool, and indicate the new text position. If you try to use this tool with more than one dimension selected, all the text will be moved together and the results will not be what you had anticipated.

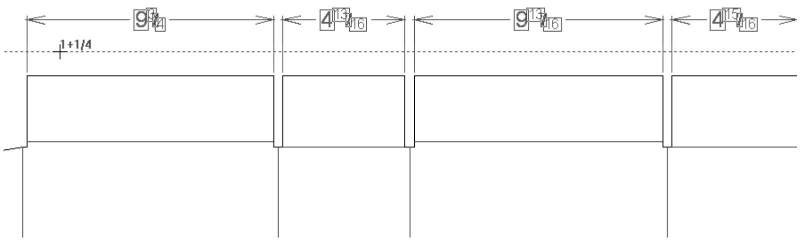
 The seventh button on the Dimensions toolbar activates the **Change Extension Line Length** tool. Use this tool to move the end of the selected extension lines for a dimension, or for a series of parallel dimensions.

Shown below is a series of parallel dimensions whose extension lines all come close to touching the panels of the box.

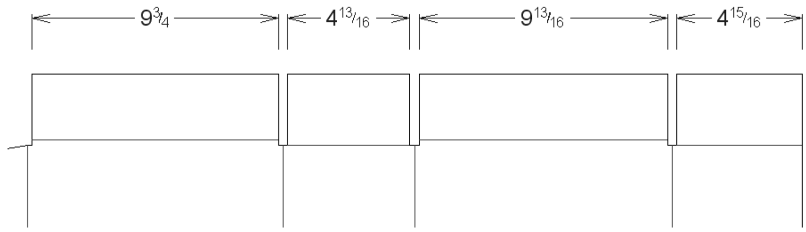


To move them away from the panels, do the following.

1. Select the dimensions with the **Select** tool, holding down `SHIFT` to select more than one item at once.
2.  Click the **Change Extension Line Length** tool. Drag will appear, enabling you to change the length of the extension lines.



3. Release the mouse button and the lines' length will be changed to meet the point where you set the drag.



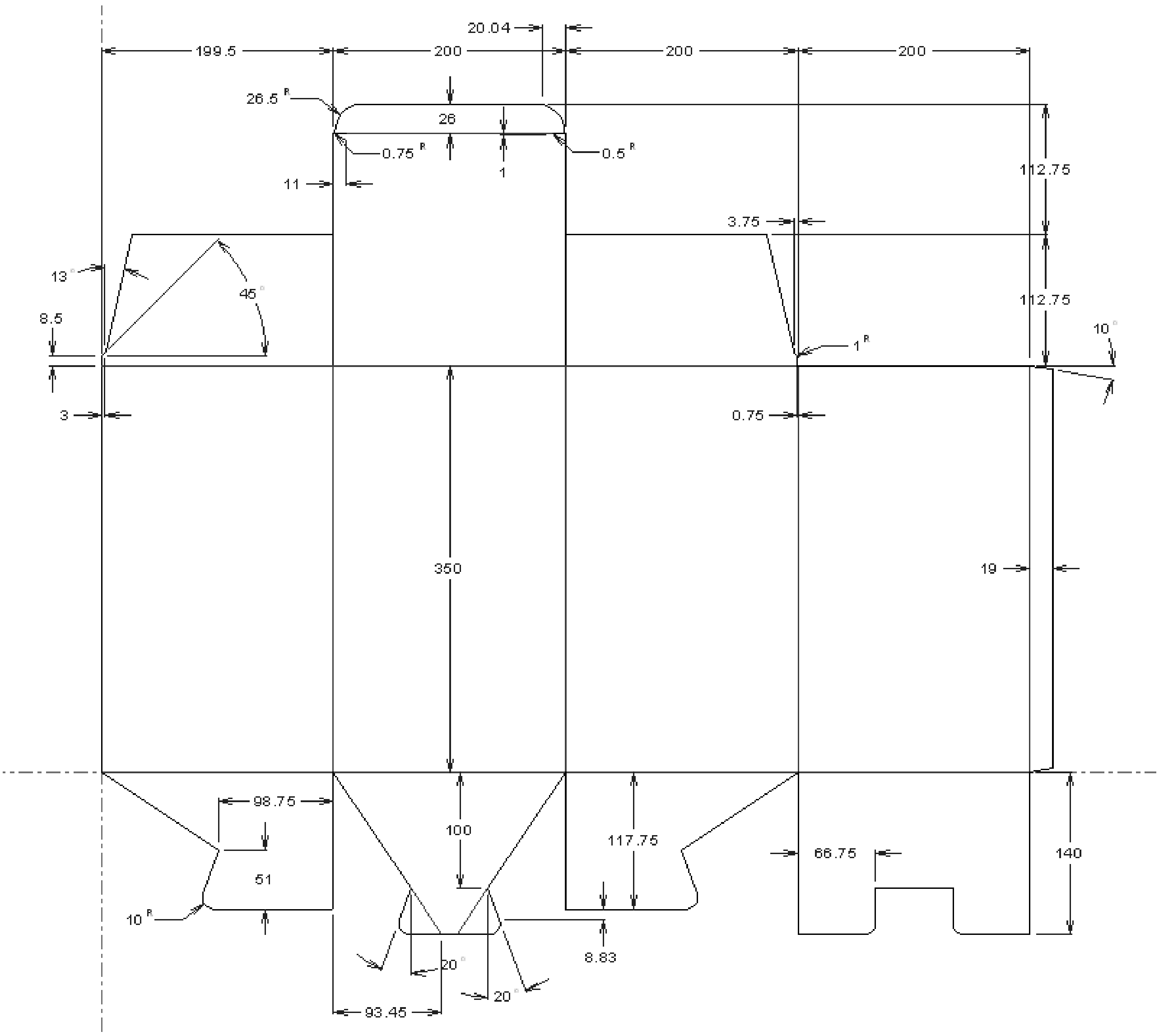
To reset extension lines to their default positions, drag the extend point back toward the design, or set the **Extension length**: field in the Properties dialog box to a large value. The extension lines will stop short of intersecting any design lines.

The eighth button on the Dimensions toolbar, and the default button on the Automatic Dimensions flyout toolbar, is the **Auto Dimension** tool. As the name implies, this tool creates automatic dimensions. It finds the longest horizontal and vertical lines and uses those to define the main panels. Dimensions for the main panels are aligned with the extension point you specify. Other dimensions are then added for the rest of the geometry, avoiding dimensioning identical flaps. The tool avoids making overlapping dimensions as much as possible, but in some cases dimensions may require manual adjustment.

Note: If the dimension property default for text position is set so the number is at the end of the line, the Automatic Dimensioning tools will put the numbers in the middle of the lines instead.

To use this tool, do the following:

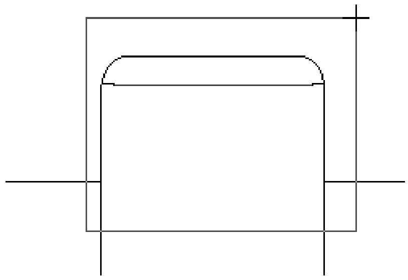
1. Construct the desired geometry.
2. Click **Auto Dimension**. If you are not in a dimension layer, ArtiosCAD prompts you to create one or stay in the current layer.
3. Indicate the extension point to use for aligning the dimensions.
4. The dimensions are drawn. To produce the example below, the extension point is above and to the right of the design.



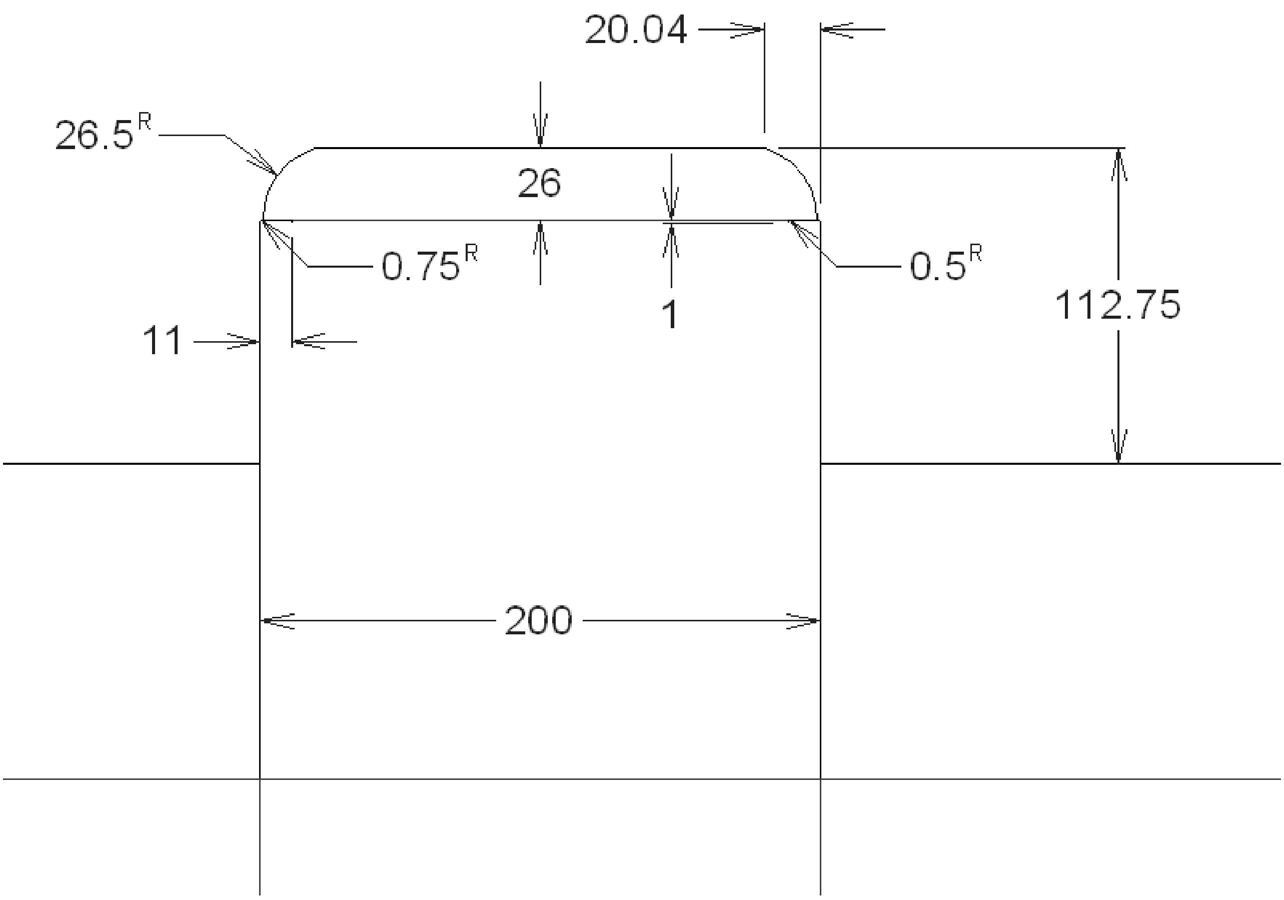
The second tool on the Automatic Dimensions flyout toolbar is the **Auto Dimension Window** tool. It works similarly as the Auto Dimension tool, but only on the portion of a design you indicate.

To use this tool, do the following:

1. Construct the desired geometry.
2. Click **Auto Dimension Window**. If you are not in a dimension layer, ArtiosCAD prompts you to create one or stay in the current layer.
3. Click and drag to indicate the portion of the design to auto dimension.



4. Dimensions are created for every point in the window and every line crossing the window boundary.



Notes and warnings about automatic dimensioning tools

Diagonal lines at a whole number of degrees are dimensioned with an angle and a horizontal or vertical dimension, whichever is longer.

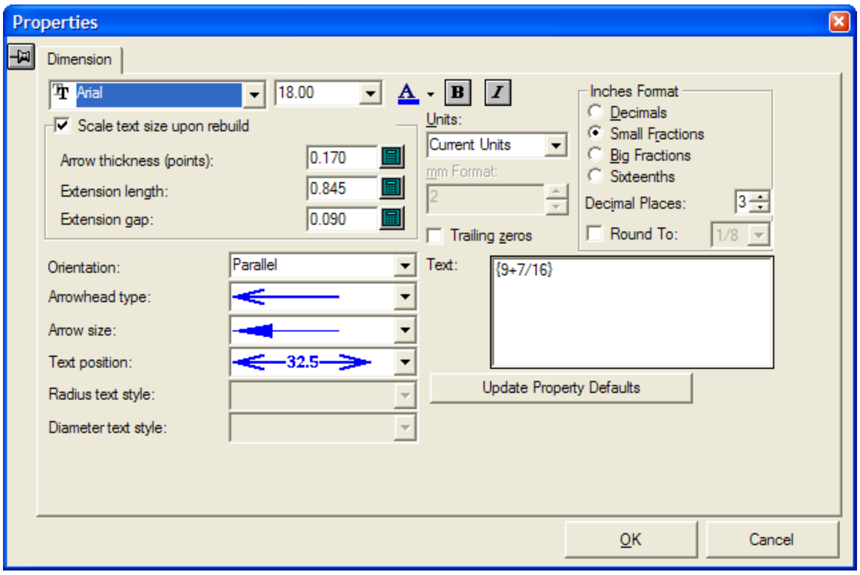
Blends are dimensioned from their corners, not from the ends of the arcs.

The automatic dimensioning tools measure all structural lines except for zero-length lines. Avoid using them with Auto-Trace or digitized shapes with many short lines.

The automatic dimensioning tools are unavailable in Manufacturing.

Dimension properties

You can change the properties of dimensions, as with other ArtiosCAD entities, by double-clicking them.



In addition to being able to change the standard text attributes (text size, color, bold, and italic), you can also set these attributes:

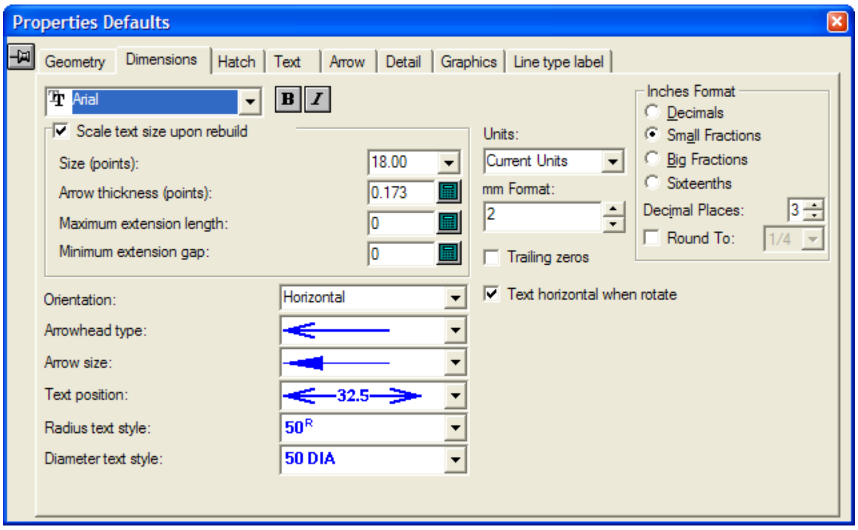
- The kind of units displayed (imperial, metric, or a combination) and the format in which they are displayed.
- The number of decimal places.
- The inclusion of trailing zeros.
- The orientation of the text relative to the extension lines.
- The type of the arrowhead and size of the arrow.
- The format for the radius and diameter text styles (if a radius or diameter dimension is selected).
- The length of the extension lines (if a dimension with extension lines is selected).
- The length of the gap between the extension lines and the design edge. If the extension line ends at an angle or close to an arc, ArtiosCAD measures the gap from the nearest perpendicular line.
- The position of the dimension text between the extension lines.
- The text actually shown as the dimension in the **Text:** field. To create a dimension including the distance that changes automatically as the design is rebuilt, enter **{V} desired text**, such as `{V} typ. 4 places`.
- The value by which to round the dimension.

Update Property Defaults sets all dimensions created after this one in the current ArtiosCAD session to use the values you specify in this dialog box before clicking it. **Extension Length** is included in this setting only if you changed the entry in the field manually.

When the **Scale text size upon rebuild** checkbox is checked, the dimensions you selected when you opened the Properties dialog box will change size relative to the overall size of the design when it is rebuilt. Their position relative to the edge of the design will also be maintained. Turn this option off to create dimensions and avoid warnings when there are no design lines in the **Main Design**, **Windows and Cutouts**, and **Manufacturing** layers.

The **Extension length**: field allows you to set the length of the extension lines for all selected dimensions at once. The length is measured from the dimension toward the design.

All these properties can be set for the current ArtiosCAD session by clicking **Edit** > **Property Defaults** and then clicking the **Dimensions** tab.



Text horizontal when rotate is an additional property for Dimensions in the Properties Defaults dialog box that is not available in the usual Properties dialog box when a dimension is selected. When enabled, this option keeps the text of the dimensions horizontal when the design is rotated using the rotate and mirror tools in Designer and Manufacturing.

This option is enabled by default. To set the default, click **Options** > **Defaults** > **Property Defaults** > **Dimensions** and change it there. You may also change the other defaults as desired. When in Defaults, you can set the size of the design on which dimension size is calculated in the **Scale text size upon rebuild** group; the default design size used for this calculation is 40 inches or 1000 mm.

Note: Dimensions created by the **Copy Times** and **Copy Times Rotate** tools are not affected by this option.

Note: If you change the color of a dimension from the default, it is considered graphics when changing the View Mode.

Rounded dimensions

Use the **Round to** checkbox in the **Inches format**: group with caution. It will cause the selected dimension to round up or round down as appropriate and not match its actual value.

If there are rounded dimensions in the design, or the dimensions are rounded by a dimension format override, and there are smaller dimensions that add up to larger dimensions, the values of the larger dimensions will be adjusted to match the sum of the rounded values of the smaller dimensions.

The default for rounding is in **Defaults** > **Shared Defaults** > **Startup defaults** > **Dimension Format Override**.

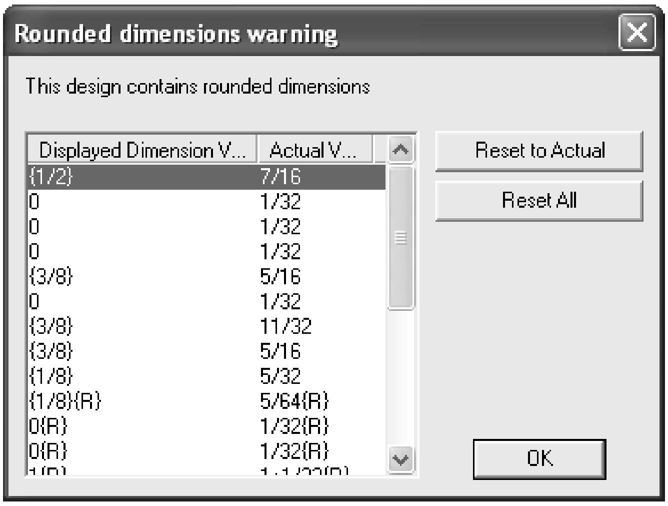
If you use overall dimensions, they may not match the sum of individual rounded dimensions. You can ensure that they match for the current design by checking the **Consistent overall rounded dimensions** checkbox in **Options Dimension Formatting**. This can also be set in Defaults in **Defaults** > **Shared Defaults** > **Startup defaults** > **Dimension Format Override**. It affects horizontal and vertical dimensions only; it does not affect diagonal dimensions.

If you set **Consistent overall rounded dimensions**, it affects all aspects of ArtiosCAD, not just the viewing of dimensions in Builder, Designer, and Manufacturing. It affects dimensions in outputs (regardless if it uses a Report), blank size on Reports, and the Blank Size dialog box on the Info menu.

If you want to use rounded dimensions with a report made in a pre-5.**x** version of ArtiosCAD, and have blank sizes defined on the report, delete and re-add the blank size variables from the Calculated Expressions Catalog.

You can check for rounded dimensions when a single design or an embedded single design is opened by checking the **Warning if design contains rounded dimensions** checkbox in **Defaults** > **Shared Defaults** > **Startup defaults** > **Dimension Format Override**.

When that checkbox is checked, the Rounded Dimensions Warning dialog box appears as shown below.



Reset to Actual un-rounds the selected dimensions. Select more than one dimension in the list by holding down ctrl and clicking the desired entries.

Reset All un-rounds them all.

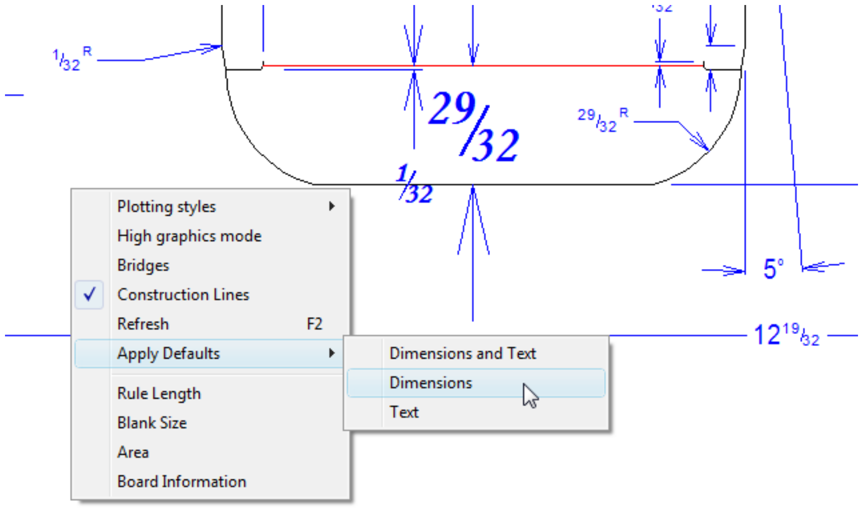
Easily resetting and applying Property Defaults

The commands on the **Apply Defaults** sub-menu on the Edit menu and context menu (the menu that appears when you right-click the mouse) change the format of dimensions, text, or both dimensions and text with one click.

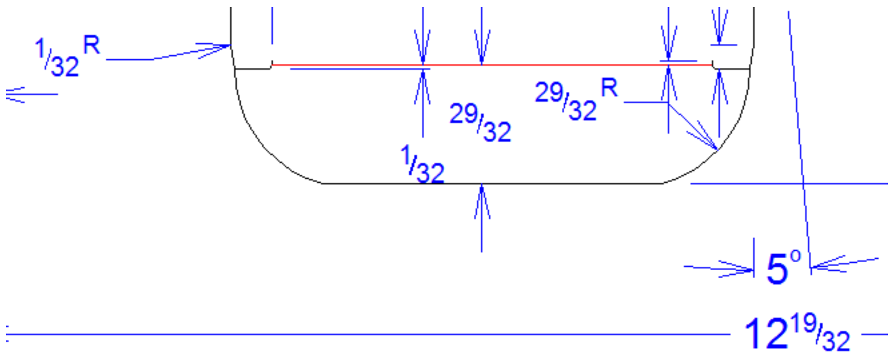
These commands change the current Property Defaults of the workspace to the Property Defaults specified in Defaults, and then apply the new Property Defaults to all the dimensions and/or text in visible, unlocked layers of the workspace depending on the menu item you clicked. Elements in turned-off layers and/or locked layers are not changed by this command.

To use these tools, do the following:

1. Click **Edit** > **Apply Defaults** or right-click and click **Apply Defaults** (as shown below).



2. Click the desired option for those elements to change: **Dimensions and Text**, **Dimensions**, or **Text**.
3. ArtiosCAD may prompt you to confirm the change. Click **Yes** to confirm.
4. ArtiosCAD applies the settings in Defaults to the current Property Defaults and changes the appropriate workspace elements to match those settings.



Note:

If you Undo any of these commands, only the workspace elements are changed back to their prior formatting. The Property Defaults of the workspace remain at their current settings.

Palletization in Builder

If you use CAPE or TOPS palletization software, you can export data from them and have ArtiosCAD create a package from that data.

You can also create secondary packaging in them from 3D and view pallet loads in 3D from them. See the *3D* chapter for more information.

For answers to some frequently asked questions about palletization, see *Palletization FAQ* in the *Outputs* chapter.

Note:

Before using these features for the first time, make sure you have mapped CAPE or TOPS styles and materials to ArtiosCAD standards and boards as described in the *Builder* chapter. You may also want to set CAPE or TOPS defaults as described in the *Defaults* chapter of the Administrator Guide.

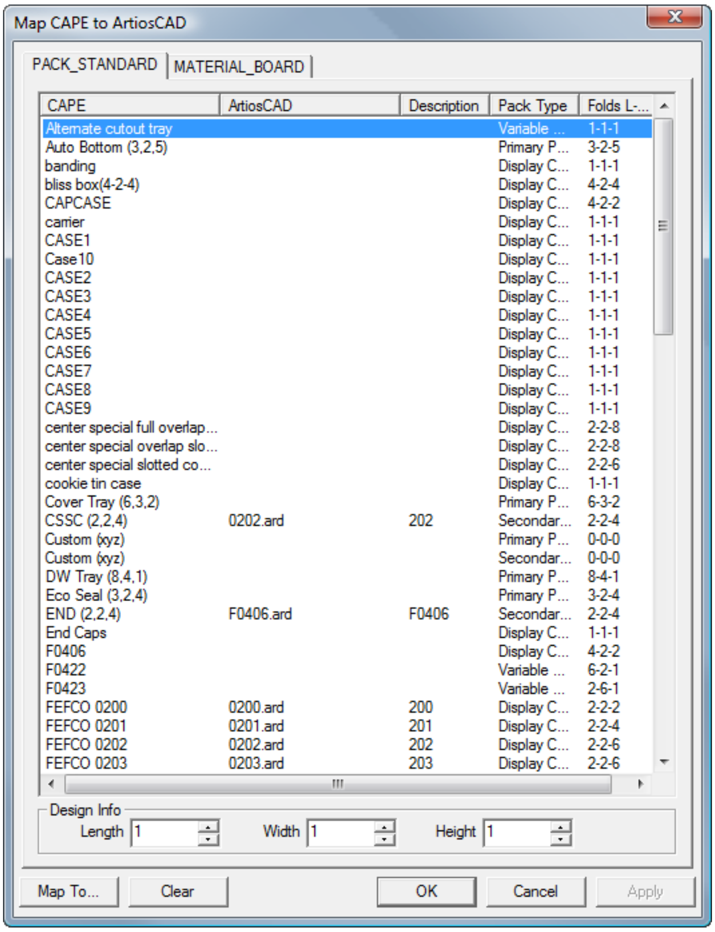
The workflow for palletization in ArtiosCAD is the same regardless of which palletization application you are using.

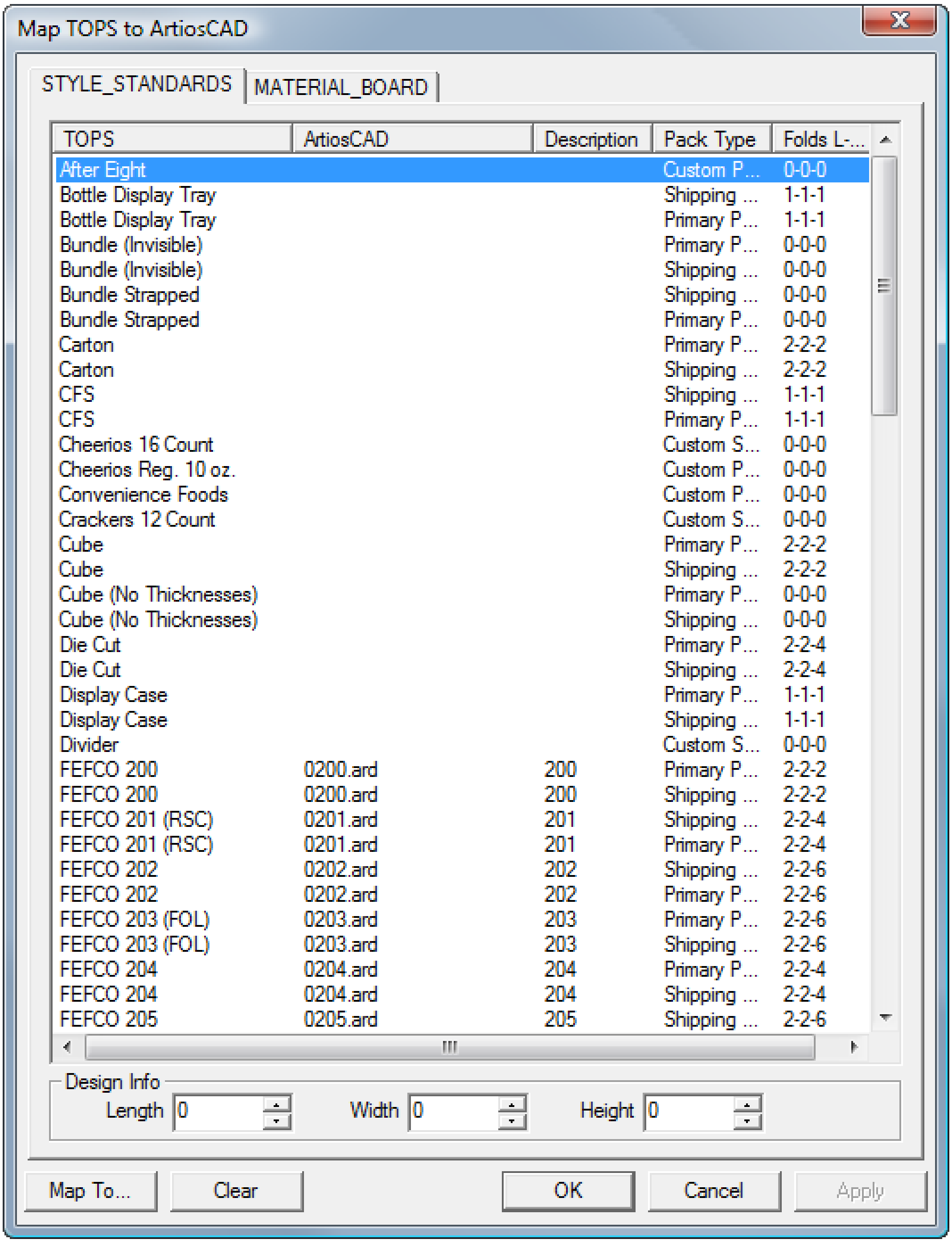
Mapping styles

To map the styles between CAPE/TOPS and ArtiosCAD, do the following:

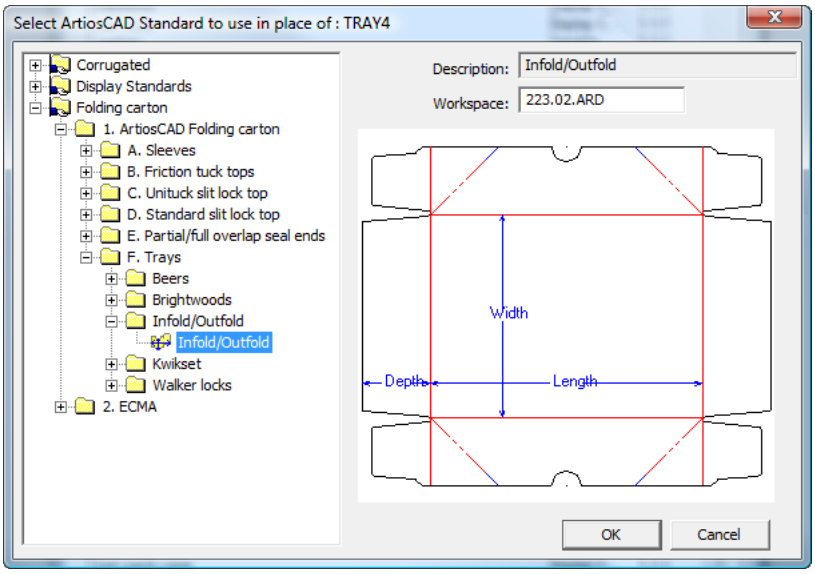
1. Launch ArtiosCAD.
2. Click **Options** > **CAPE/TOPS to ArtiosCAD** .

   The **Map CAPE/TOPS to ArtiosCAD** dialog box appears.


Map CAPE to ArtiosCAD
PACK_STANDARD
MATERIAL_BOARD
CAPE
ArtiosCAD
Description
Pack Type
Folds L-...
Alternate cutout tray Variable ... 1-1-1
Auto Bottom (3,2,5) Primary P... 3-2-5
banding Display C... 1-1-1
bliss box(4-2-4) Display C... 4-2-4
CAPCASE Display C... 4-2-2
carrier Display C... 1-1-1
CASE1 Display C... 1-1-1
Case10 Display C... 1-1-1
CASE2 Display C... 1-1-1
CASE3 Display C... 1-1-1
CASE4 Display C... 1-1-1
CASE5 Display C... 1-1-1
CASE6 Display C... 1-1-1
CASE7 Display C... 1-1-1
CASE8 Display C... 1-1-1
CASE9 Display C... 1-1-1
center special full overlap... Display C... 2-2-8
center special overlap slo... Display C... 2-2-8
center special slotted co... Display C... 2-2-6
cookie tin case Display C... 1-1-1
Cover Tray (6,3,2) Primary P... 6-3-2
CSSC (2,2,4) 0202.ard 202 Secondar... 2-2-4
Custom (xyz) Primary P... 0-0-0
Custom (xyz) Secondar... 0-0-0
DW Tray (8,4,1) Primary P... 8-4-1
Eco Seal (3,2,4) Primary P... 3-2-4
END (2,2,4) F0406.ard F0406 Secondar... 2-2-4
End Caps Display C... 1-1-1
F0406 Display C... 4-2-2
F0422 Variable ... 6-2-1
F0423 Variable ... 2-6-1
FEFCO 0200 0200.ard 200 Display C... 2-2-2
FEFCO 0201 0201.ard 201 Display C... 2-2-4
FEFCO 0202 0202.ard 202 Display C... 2-2-6
FEFCO 0203 0203.ard 203 Display C... 2-2-6
Design Info
Length 1
Width 1
Height 1
Map To...
Clear
OK
Cancel
Apply



3. Double-click a style to associate with an ArtiosCAD standard.
4. In the **Select ArtiosCAD Standard to use in place of** dialog box, choose the desired ArtiosCAD standard and click **OK**. You may only select ArtiosCAD-format standards.



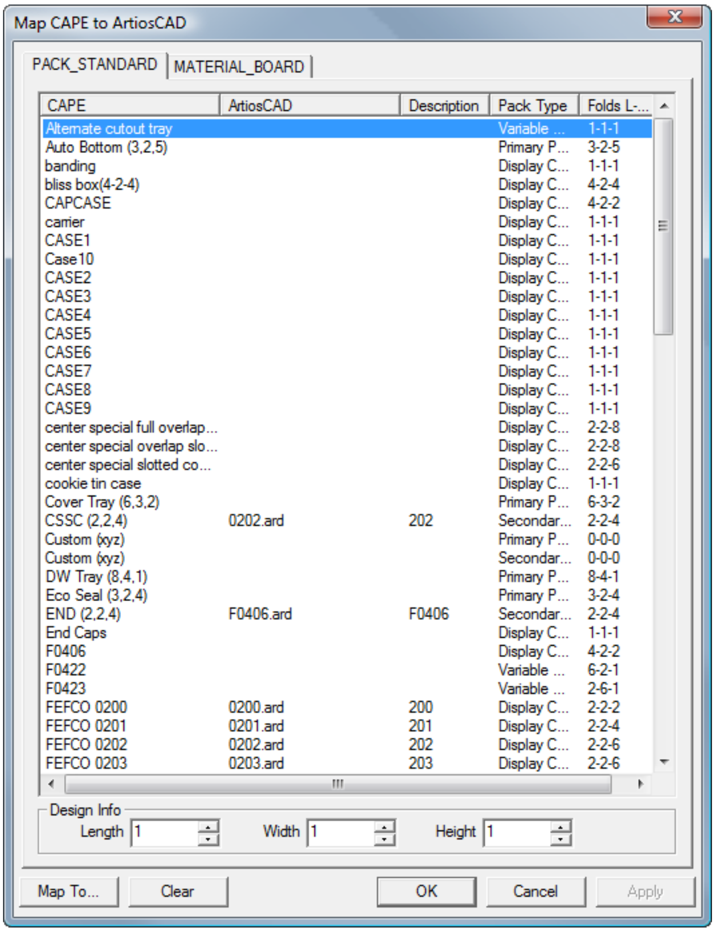
5. In the **Length**, **Width**, and **Height** fields, set the number of folds in each direction.
6. Repeat as desired by double-clicking another style to map.
7. Click **OK** when done.

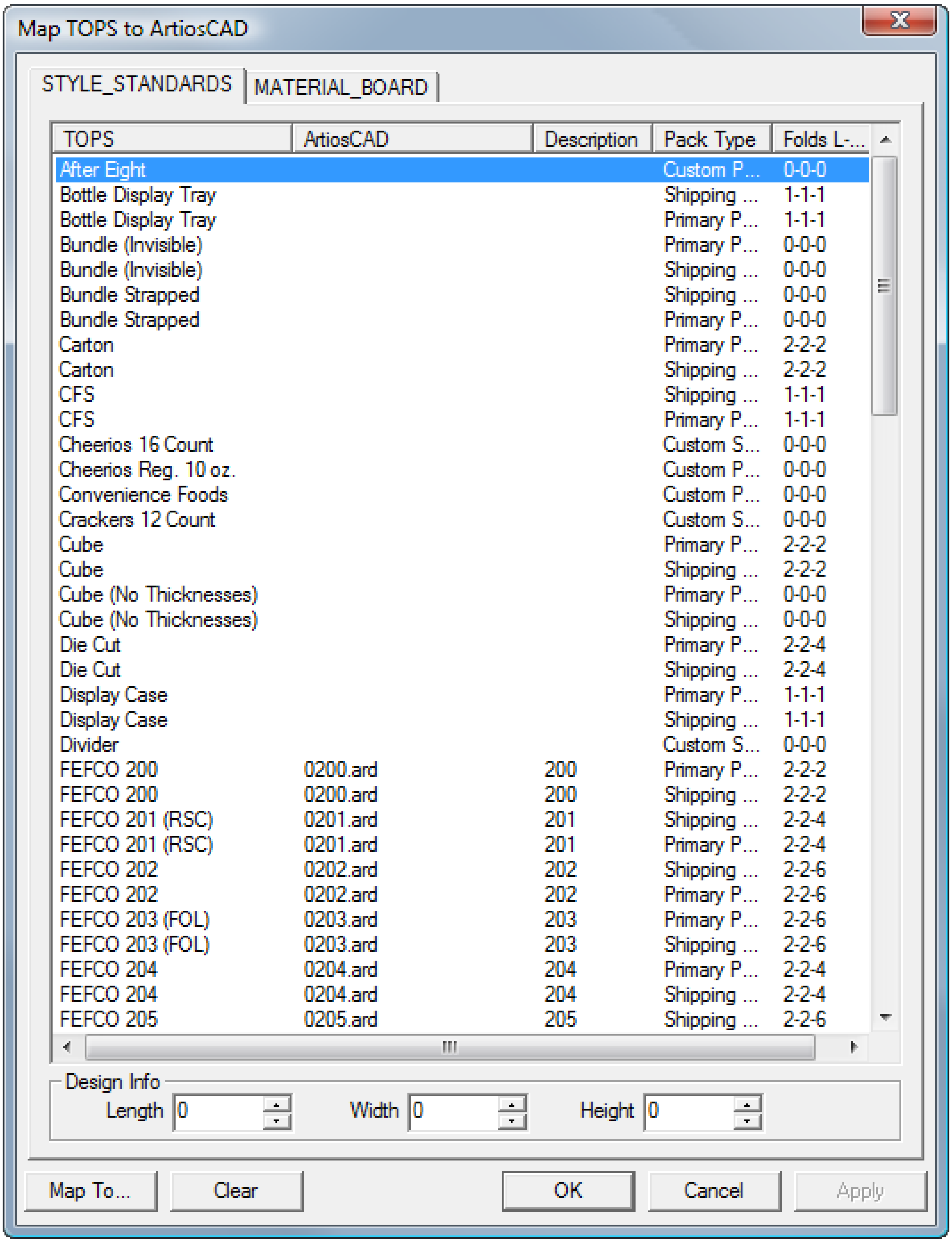
Mapping boards

To map the boards between CAPE/TOPS and ArtiosCAD, do the following:

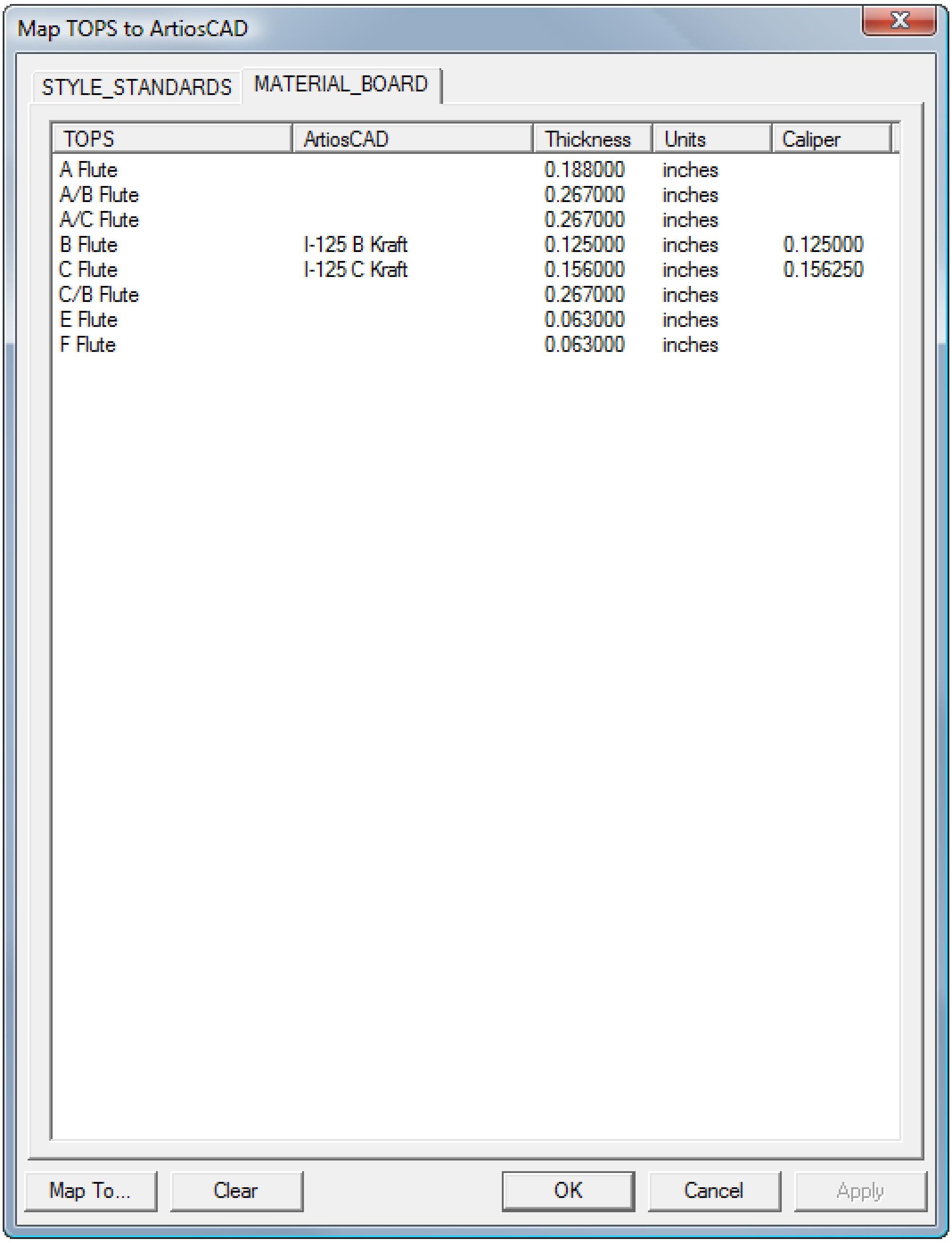
1. Launch ArtiosCAD.
2. Click **Options** > **CAPE/TOPS to ArtiosCAD** .

   The **Map CAPE/TOPS to ArtiosCAD** dialog box appears.


Map CAPE to ArtiosCAD
PACK_STANDARD
MATERIAL_BOARD
CAPE
ArtiosCAD
Description
Pack Type
Folds L-...
Alternate cutout tray Variable ... 1-1-1
Auto Bottom (3,2,5) Primary P... 3-2-5
banding Display C... 1-1-1
bliss box(4-2-4) Display C... 4-2-4
CAPCASE Display C... 4-2-2
carrier Display C... 1-1-1
CASE1 Display C... 1-1-1
Case10 Display C... 1-1-1
CASE2 Display C... 1-1-1
CASE3 Display C... 1-1-1
CASE4 Display C... 1-1-1
CASE5 Display C... 1-1-1
CASE6 Display C... 1-1-1
CASE7 Display C... 1-1-1
CASE8 Display C... 1-1-1
CASE9 Display C... 1-1-1
center special full overlap... Display C... 2-2-8
center special overlap slo... Display C... 2-2-8
center special slotted co... Display C... 2-2-6
cookie tin case Display C... 1-1-1
Cover Tray (6,3,2) Primary P... 6-3-2
CSSC (2,2,4) 0202.ard 202 Secondar... 2-2-4
Custom (xyz) Primary P... 0-0-0
Custom (xyz) Secondar... 0-0-0
DW Tray (8,4,1) Primary P... 8-4-1
Eco Seal (3,2,4) Primary P... 3-2-4
END (2,2,4) F0406.ard F0406 Secondar... 2-2-4
End Caps Display C... 1-1-1
F0406 Display C... 4-2-2
F0422 Variable ... 6-2-1
F0423 Variable ... 2-6-1
FEFCO 0200 0200.ard 200 Display C... 2-2-2
FEFCO 0201 0201.ard 201 Display C... 2-2-4
FEFCO 0202 0202.ard 202 Display C... 2-2-6
FEFCO 0203 0203.ard 203 Display C... 2-2-6
Design Info
Length 1
Width 1
Height 1
Map To...
Clear
OK
Cancel
Apply

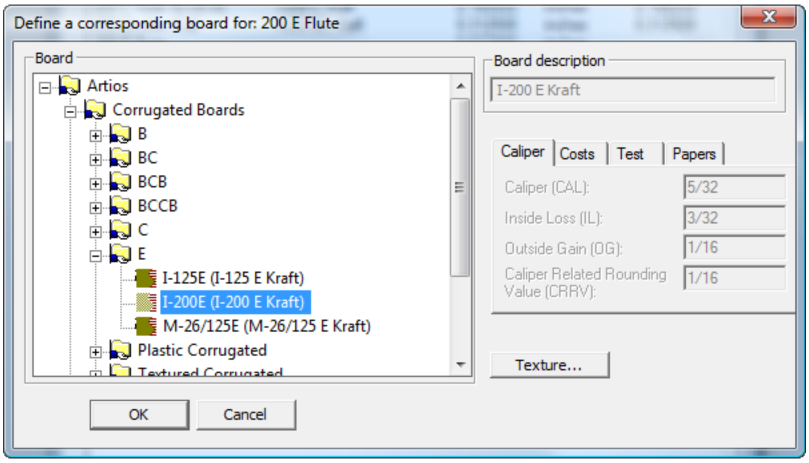


3. In the **Map CAPE/TOPS to ArtiosCAD Standard** dialog box, click the MATERIAL_BOARD tab.



4. Double-click a CAPE/TOPS board.

   The **Define a corresponding board for** dialog box appears.

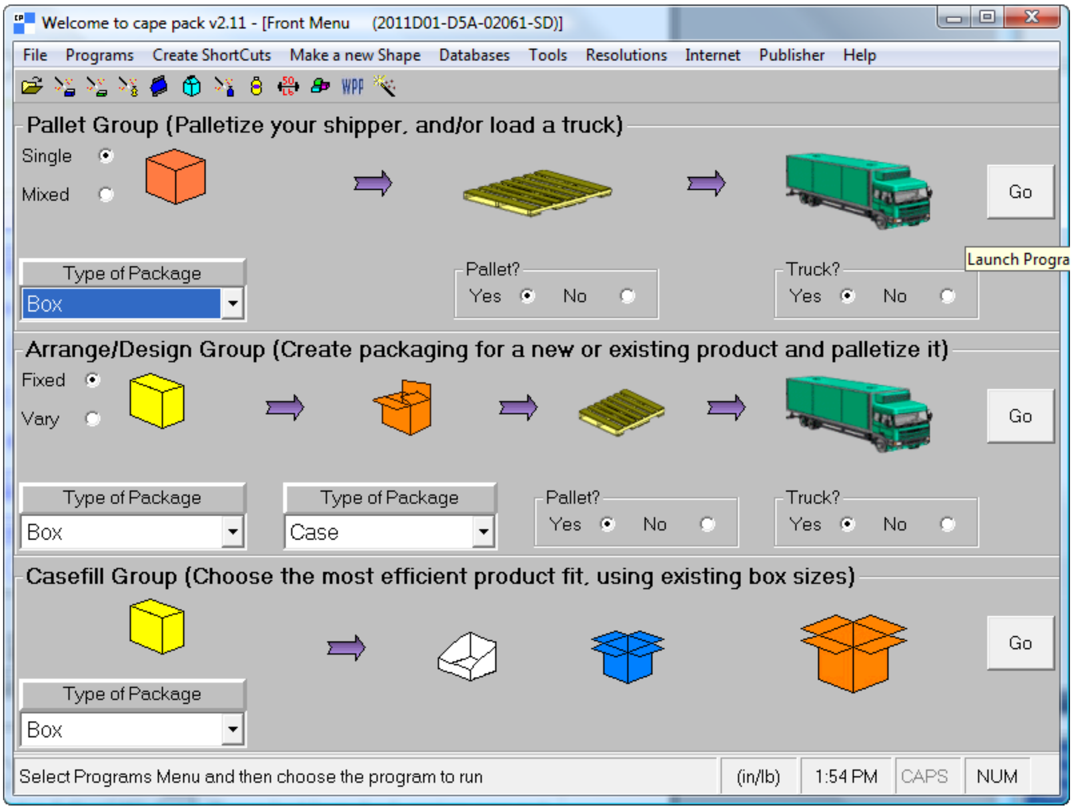


5. Open the **Artios** catalog and select the desired board. Click **OK** when you have selected the desired board. If you cannot find a corresponding definition, open DataCenter Admin and create one, close this dialog box and its parent, and resume at step 2.
6. Repeat as desired by double-clicking another board to map.
7. Click **OK** when done.

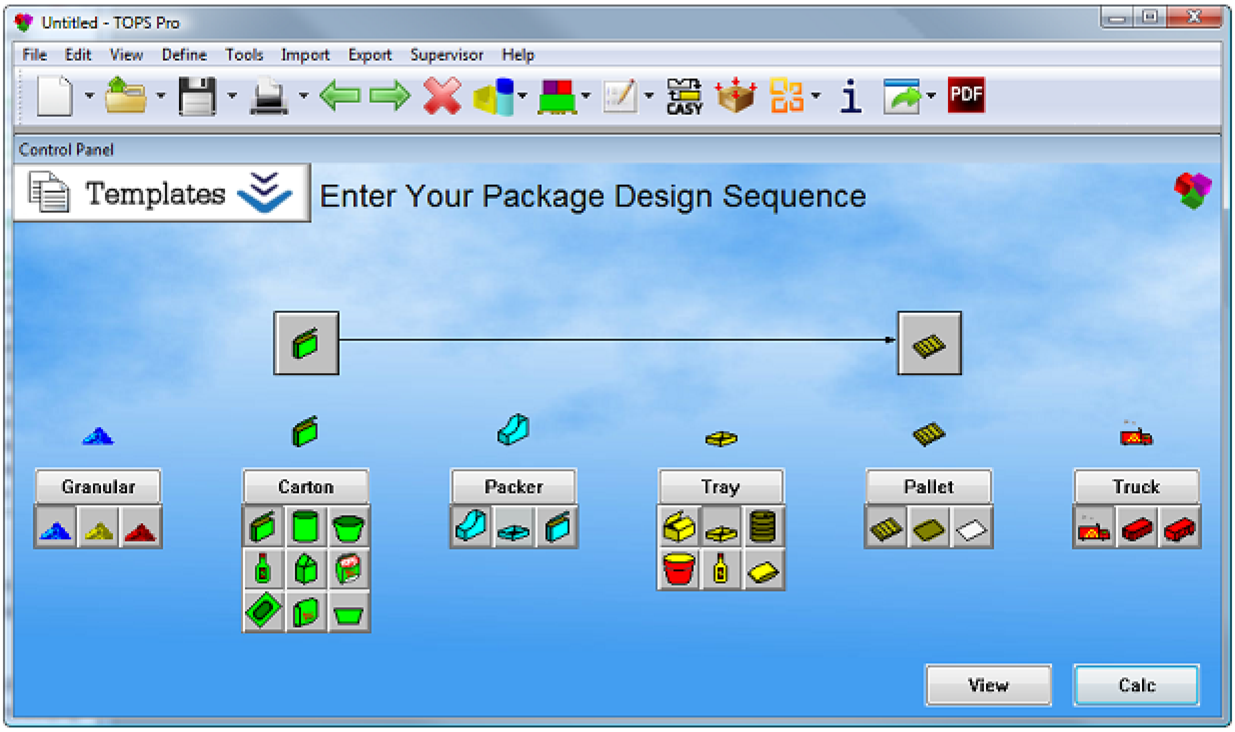
Creating a pallet group

To create a pallet group using a combination of CAPE/TOPS and ArtiosCAD, do the following:

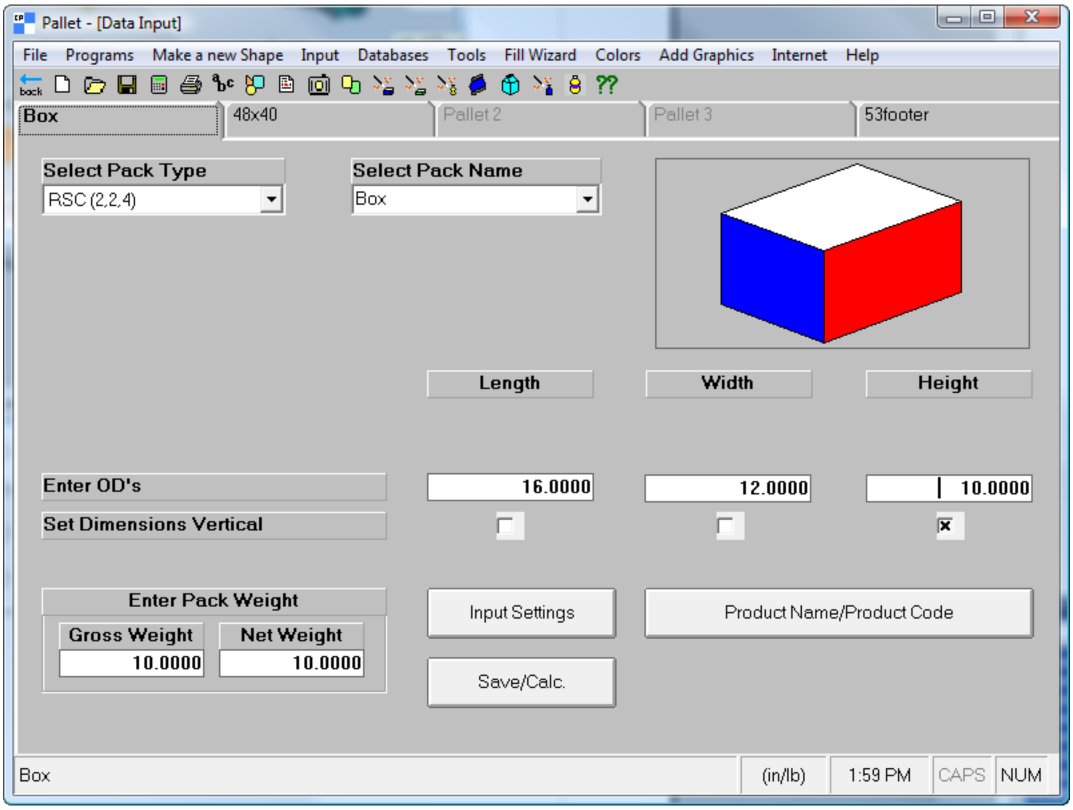
1. CAPE/TOPS will use the last-launched version of ArtiosCAD. If your last-launched version was not 12 or later, launch it and leave it open.
2. Launch CAPE/TOPS.
3. In CAPE, in the **Pallet Group**, make your selections as desired and click **Go**.

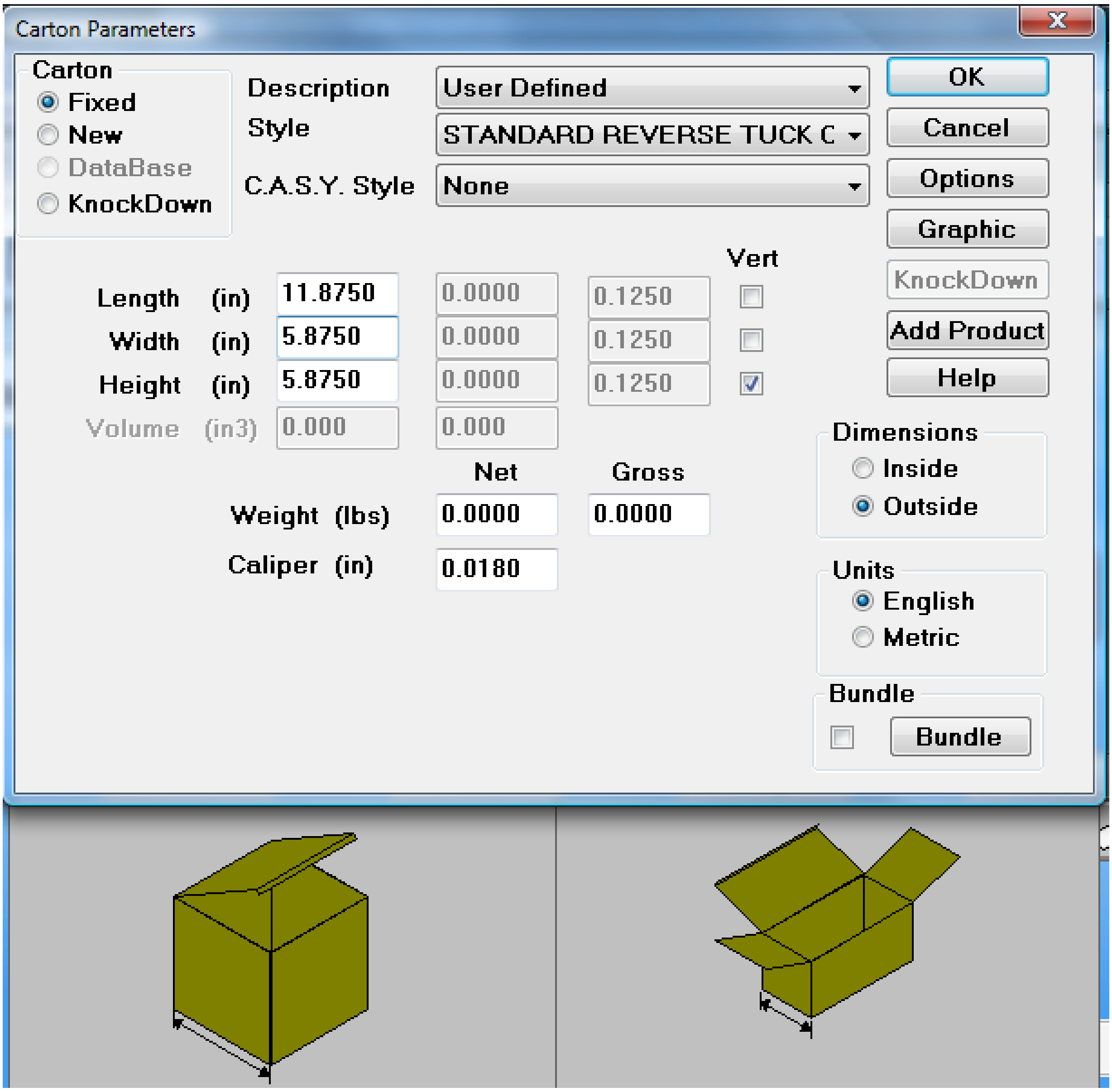


In TOPS, enter your package design sequence and click **Calc**.

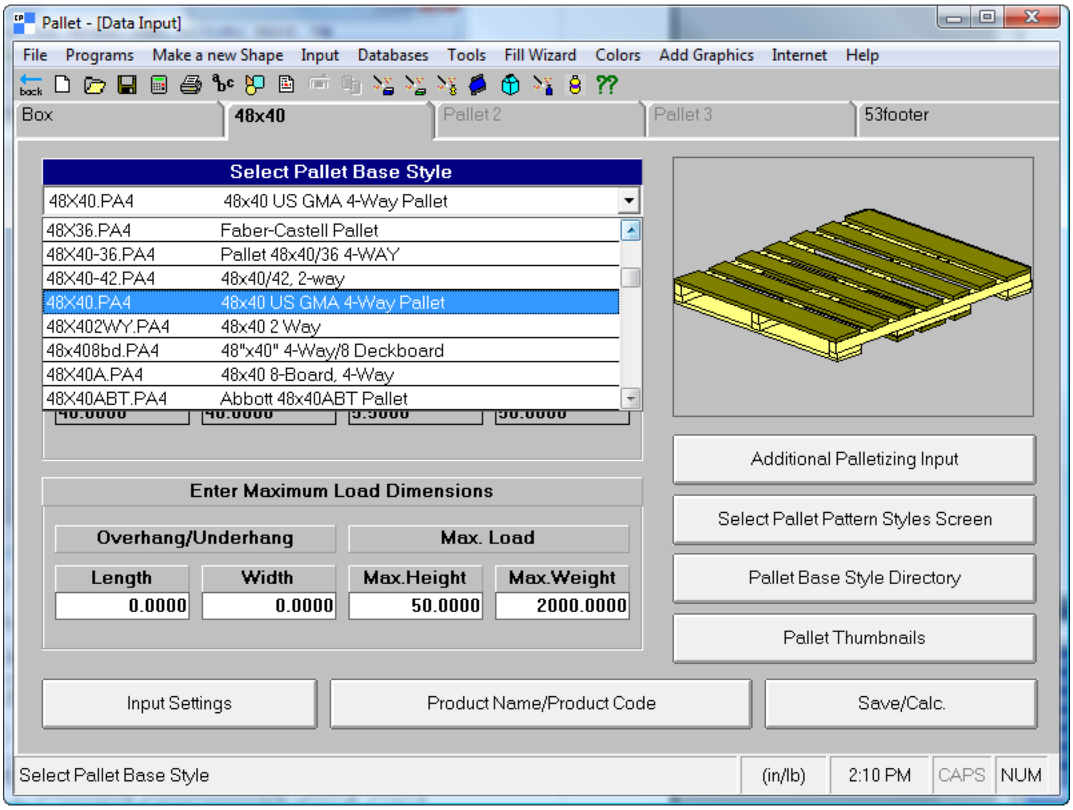


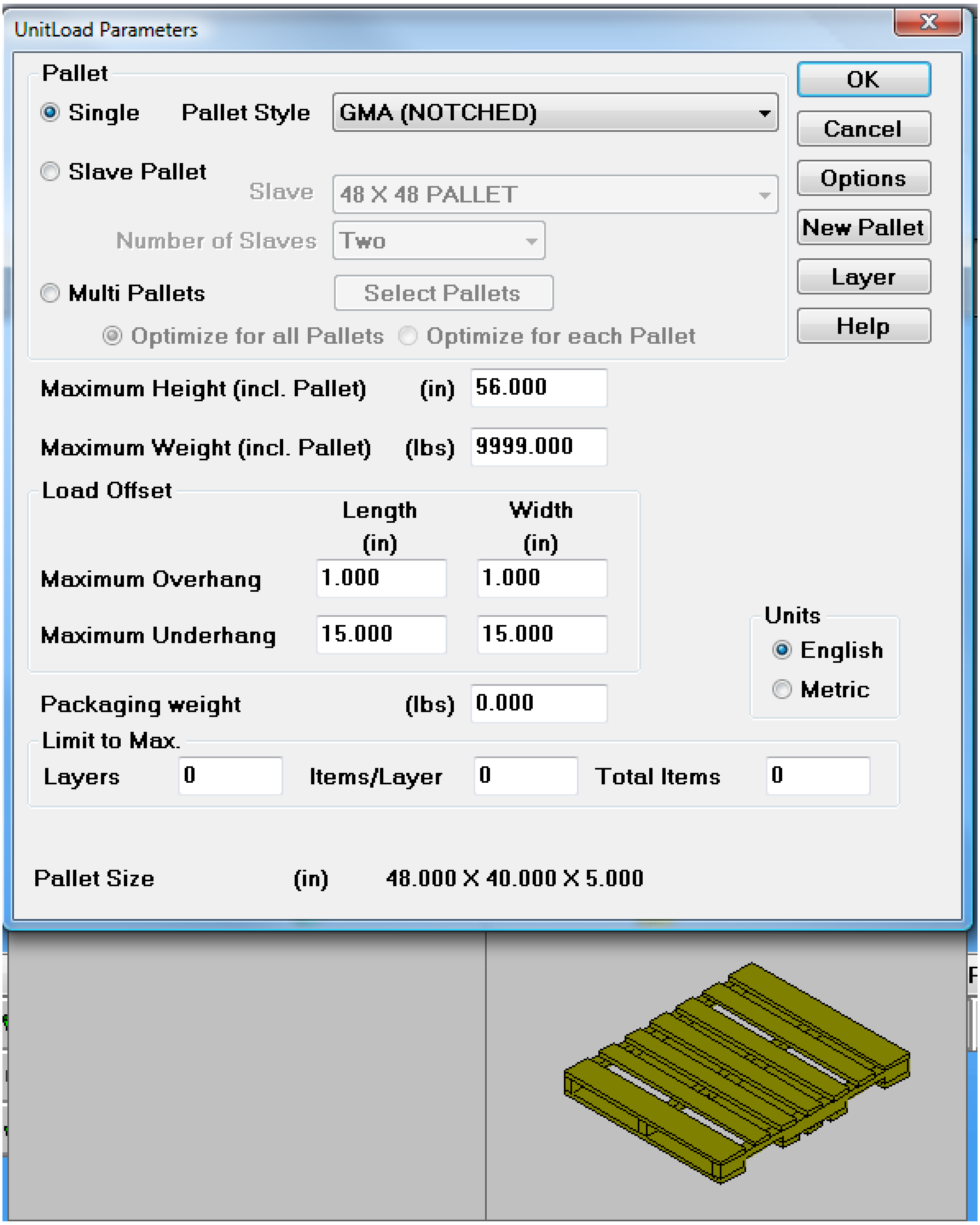
4. Choose the type of package to make and enter the appropriate values for your package.


Pallet - [Data Input]
File Programs Make a new Shape Input Databases Tools Fill Wizard Colors Add Graphics Internet Help
Box
48x40
Pallet 2
Pallet 3
53footer
Select Pack Type
RSC (2,2,4)
Select Pack Name
Box
Length
Width
Height
Enter OD's
16.0000
12.0000
10.0000
Set Dimensions Vertical
Enter Pack Weight
Gross Weight
10.0000
Net Weight
10.0000
Input Settings
Product Name/Product Code
Save/Calc.
Box
(in/lb)
1:59 PM
CAPS
NUM



5. CAPE/TOPS will use its default pallet. To change it, click the next tab on the screen (in the example above, it is labelled **48x40**) and choose a new one. To not change the pallet, skip to the next step. You can also change the truck size on the last tab but ArtiosCAD does not use that information.

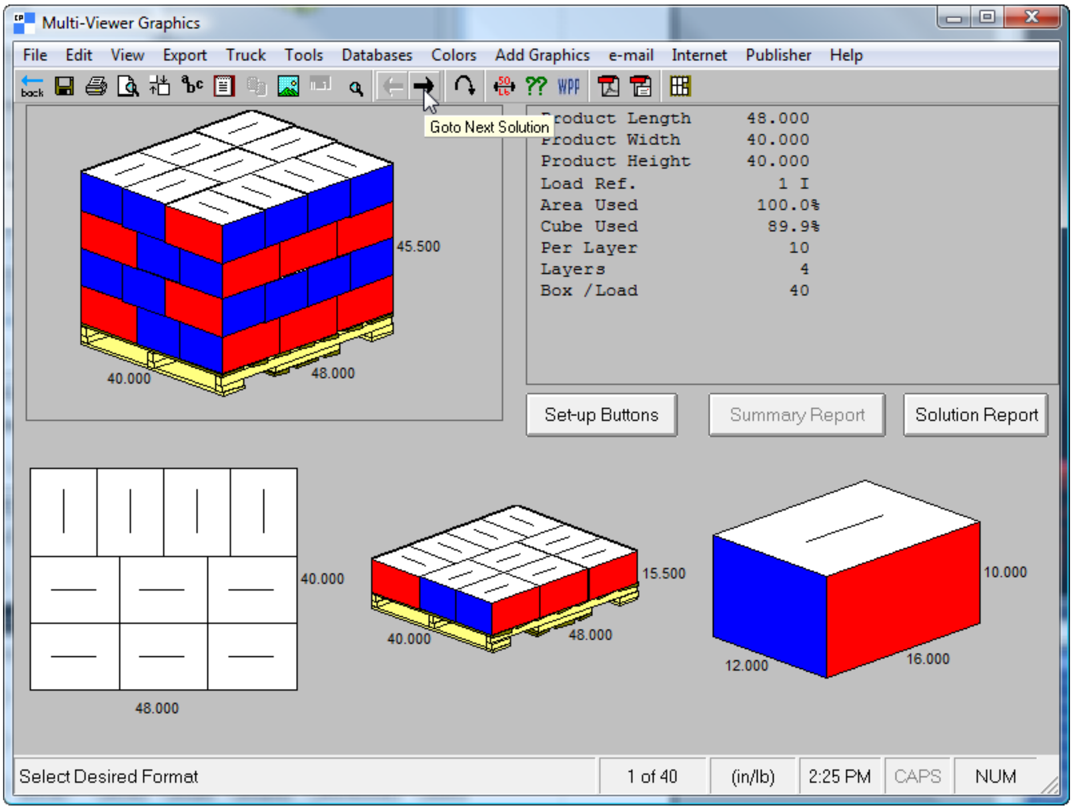

Pallet - [Data Input]
File Programs Make a new Shape Input Databases Tools Fill Wizard Colors Add Graphics Internet Help
Box
48x40
Pallet 2
Pallet 3
53footer
Select Pallet Base Style
48X40.PA4 48x40 US GMA 4-Way Pallet
48X36.PA4 Faber-Castell Pallet
48X40-36.PA4 Pallet 48x40/36 4-WAY
48X40-42.PA4 48x40/42, 2-way
48X40.PA4 48x40 US GMA 4-Way Pallet
48X402WY.PA4 48x40 2 Way
48x408bd.PA4 48"x40" 4-Way/8 Deckboard
48X40A.PA4 48x40 8-Board, 4-Way
48X40ABT.PA4 Abbott 48x40ABT Pallet
Enter Maximum Load Dimensions
Overhang/Underhang
Max. Load
Length
Width
Max.Height
Max.Weight
0.0000
0.0000
50.0000
2000.0000
Additional Palletizing Input
Select Pallet Pattern Styles Screen
Pallet Base Style Directory
Pallet Thumbnails
Input Settings
Product Name/Product Code
Save/Calc.
Select Pallet Base Style
(in/lb)
2:10 PM
CAPS
NUM



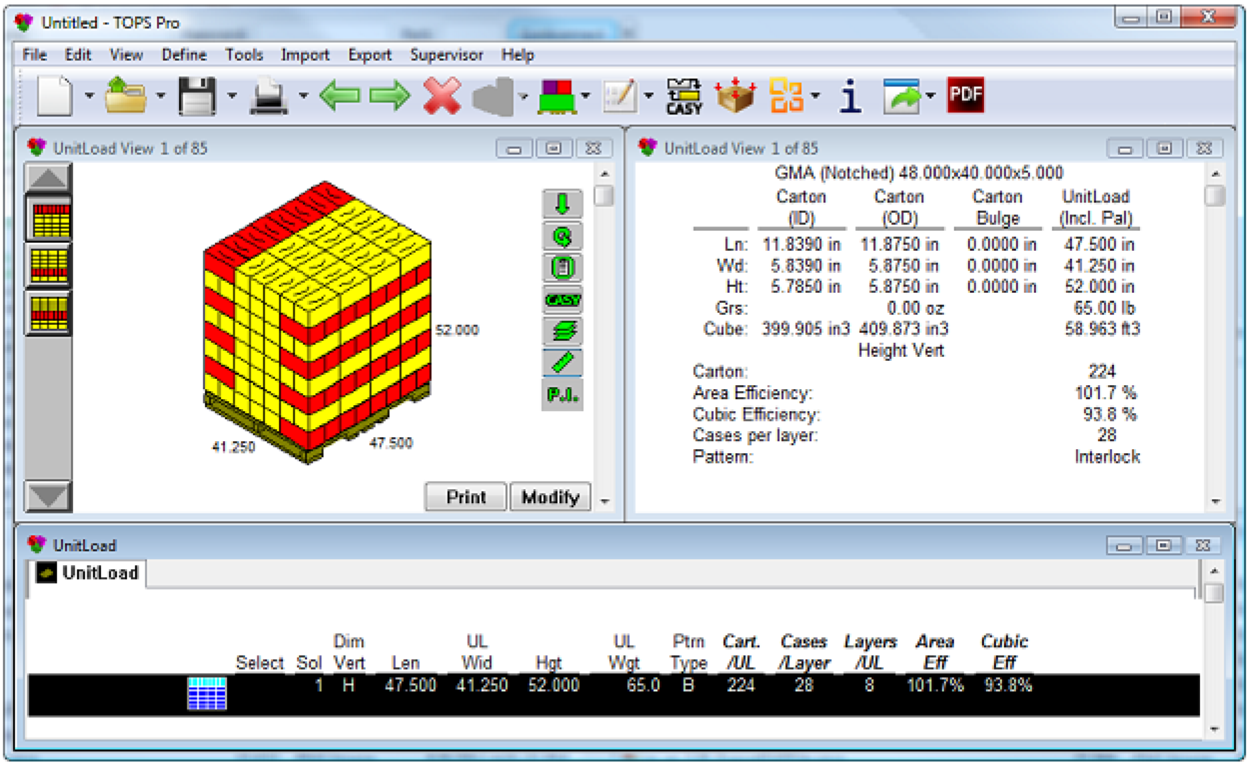
6. For CAPE, click **File** > **Calculate Only** . For TOPS, click **OK**.

   Click **Yes** or **No** to any alerts as desired.

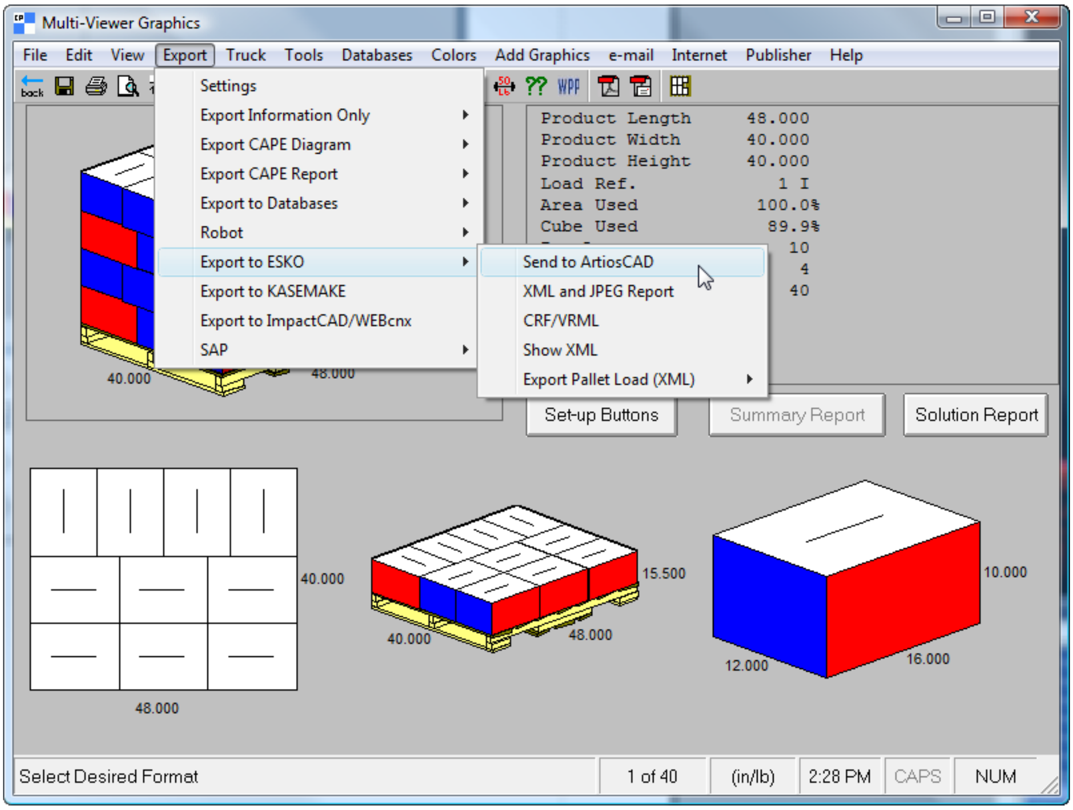
7. For CAPE, use the **Goto Next Solution** and **Goto Previous Solution** arrows to find the best solution.



For TOPS, scroll through the list of solutions in the UnitLoad window and click a solution to view it.



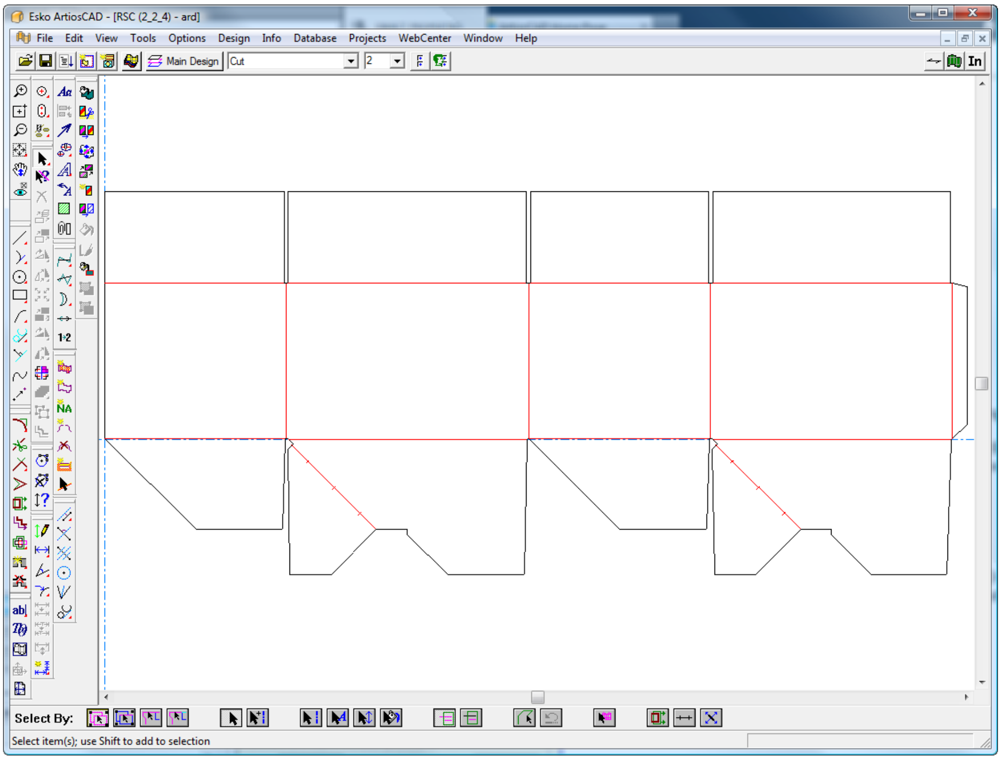
8. For CAPE, when you have found the best solution, click **Export** > **Export to ESKO** > **Send to ArtiosCAD** .



For TOPS, when you have found the best solution, click **Export** > **Send to ArtiosCAD** . Choose which elements to export and click **Export**, and click **Cancel** in the Analysis Save As dialog box.

If you do not have either the style or board mapped as described previously, ArtiosCAD will prompt you to choose the missing elements.

9. ArtiosCAD displays the standard it constructed using the CAPE/TOPS data. Work with the single design as you would any other workspace.

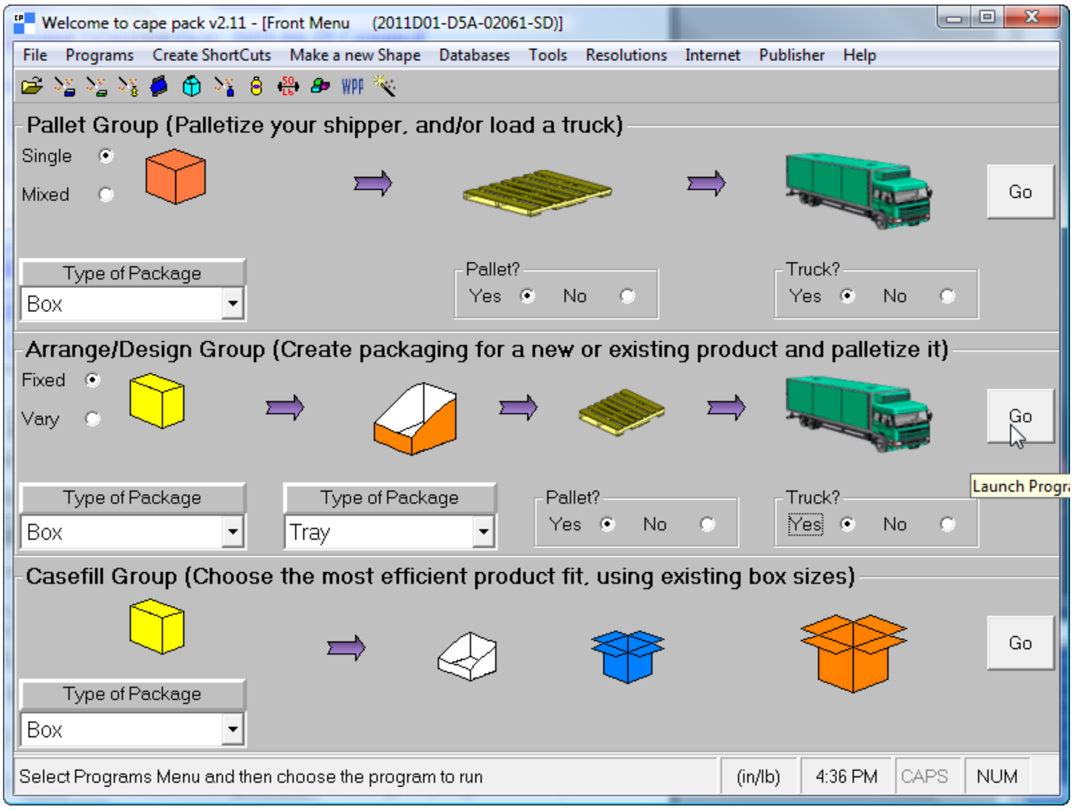


Arranging/Designing a group

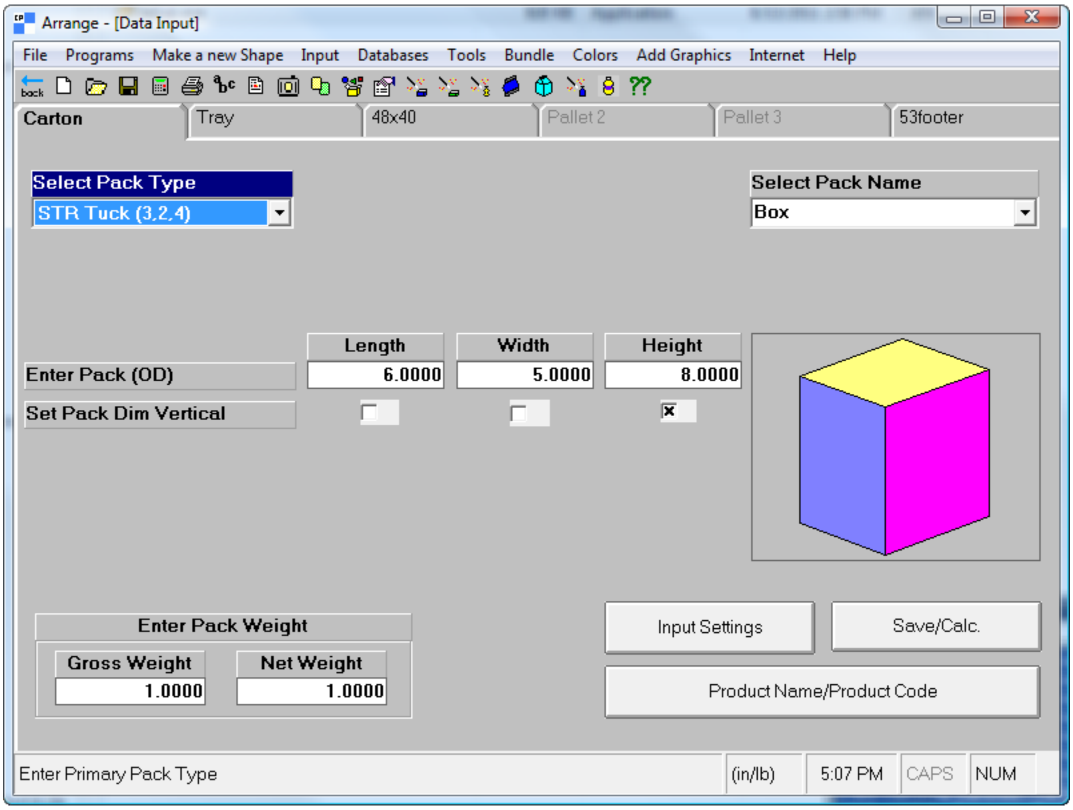
The Arranging/Designing a Group workflow between CAPE/TOPS and ArtiosCAD is similar to that of creating a shipper for use on a pallet, but at the end there are two designs in ArtiosCAD.

If you are using CAPE, follow the instructions below. For TOPS, the workflow is similar in that you select more than one element to create, create them, and then use the **Send to ArtiosCAD** command on the **Export** menu.

1. CAPE will use the last-launched version of ArtiosCAD. If your last-launched version was not 12 or later, launch it and leave it open.
2. Launch CAPE.
3. In CAPE, in the **Arrange/Design Group**, make your selections as desired and click **Go**.



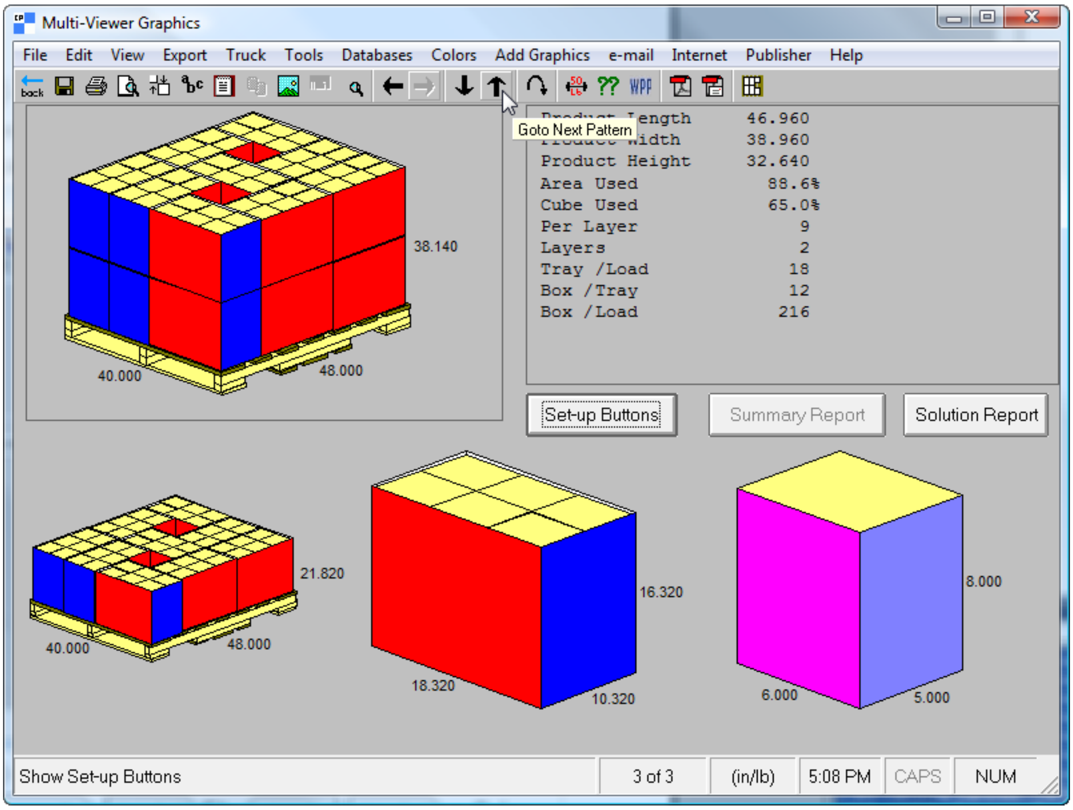
4. In the Data Input window, click the three tabs to choose the packages and pallet and enter values in the fields as appropriate.



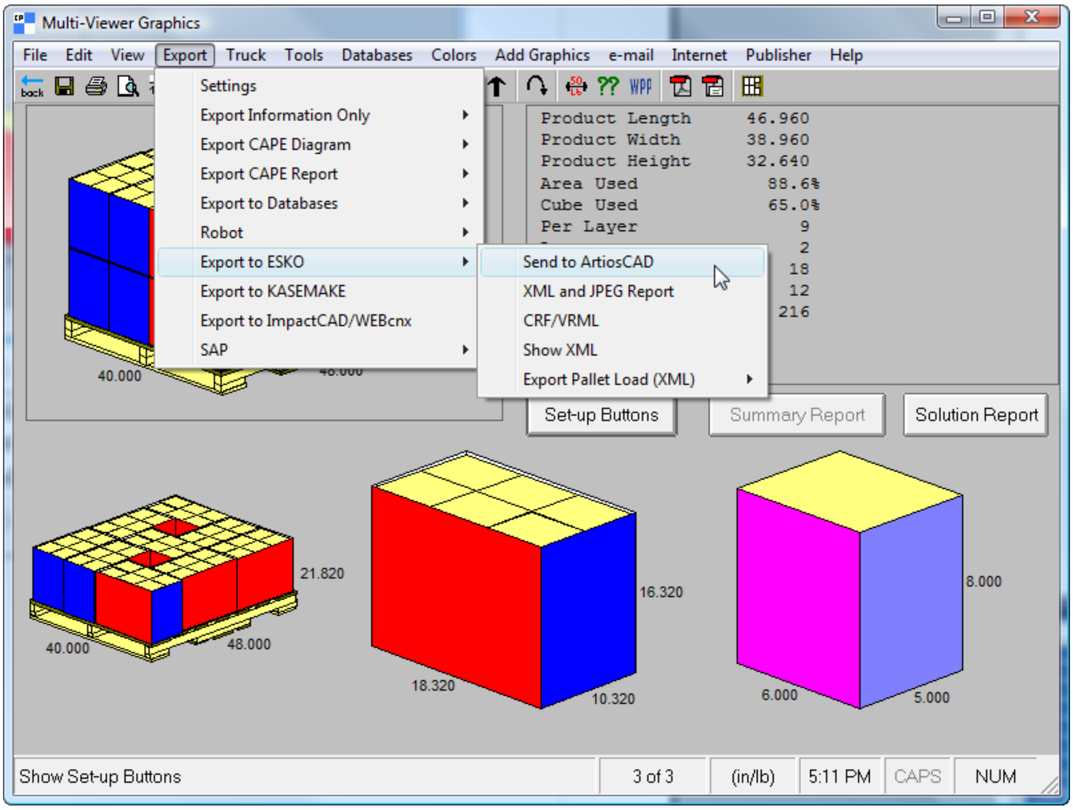
5. Click **File** > **Calculate Only**.

   Click **Yes** or **No** to any alerts as desired.

6. Use the **Goto Next Solution**, **Goto Previous Solution**, **Goto Next Pattern**, and **Goto Previous Pattern** arrows to find the best solution.

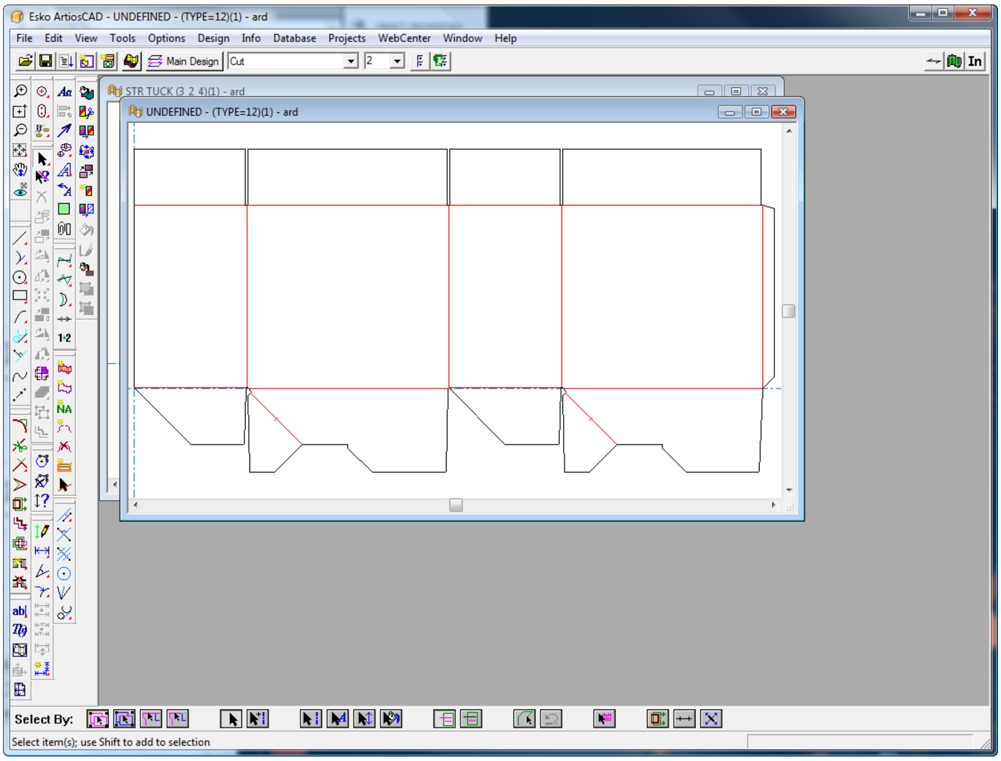


7. When you have found the best solution, click **Export** > **Export to ESKO** > **Send to ArtiosCAD**.



If you do not have either the styles or boards mapped as described previously, ArtiosCAD will prompt you to choose the missing elements.

8. ArtiosCAD displays the standards it constructed using the CAPE data in cascaded windows. Work with the single designs as you would any other workspaces.



If pallet information exists in the ArtiosCAD session, a pallet appears in the 3D workspace when you convert the design to 3D.

5. Designer

Introduction to Designer

There will be times when you need to create a design from scratch instead of using an entry from a Standards Catalog. ArtiosCAD handles those situations by providing sophisticated design tools that empower you to quickly and easily turn your ideas into reality.

These tools are available on systems with the Designer option. Certain other modules provide access to some of the tools. Systems with only the Builder option cannot use these tools.

Overview of Designer

All the tools work the same way. Click the tool and satisfy the drag prompts by moving the pointer and clicking the mouse button. The geometry is created when enough prompts are satisfied.

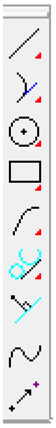
Examples of tasks completed using Designer are:

- Creating original designs and modifying existing designs.
- Creating and deleting lines, arcs, circles, rectangles, curves, and beziers.
- Adjusting and transforming the linear and physical properties of elements in the design such as line type, number of bridges, or physical placement in the design.

Creating geometry

What is geometry?

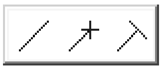
In Designer, *geometry* is any item that is on a die press manufacturing tool. Dimensions, annotations, and graphics are not considered geometry. Create geometry with the tools on the Geometry toolbar.



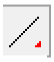
Lines

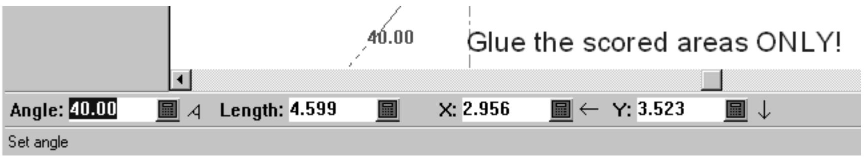
Lines are one of the basic building blocks of ArtiosCAD. Any shape can be made out of lines and arcs. There are three Designer tools to create lines: the Line Angle/Offset tool, the Line Horiz/Vert tool, and the Line Angle/Length tool.

The first button on the Geometry toolbar activates the **Line** tools. The Line tools are used to make straight lines. When you click and hold a Line tool button on the toolbar, the Line flyout toolbar appears.



Lines at an angle

 To make a line at an angle, click the **Line Angle/Offset** tool. The line will begin at the current position. The Status bar will prompt for the angle of the line and the offset of the end-point. Set the prompts using drag and the line will be drawn. Use the arrow keys to move among the fields on the Status bar – the highlighted field is the one being set by the drag.



If you click an established point when prompted for the offset, the line being drawn will end at the point you clicked if possible. If the end-point and the point you clicked are not collinear, the endpoint will be aligned with the point you clicked.

You may use any Line tool to construct a line at an angle. Simply change the Status bar prompts being set by the drag. The toolbar buttons only specify which prompts are selected when the tool is activated.

When using this tool, holding down `CTRL` while clicking performs a **Move By**.

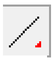
Lines with a known X, Y offset of the end-point

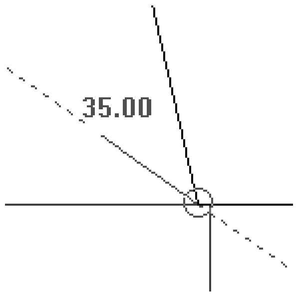
 To make a line from the current position with a known X,Y offset of the end-point, click the **Line Horiz/Vert** tool. Set the prompts. If the end-point is a known point, simply click that point, and the line will be created.

Lines of a specific length

 To make a line at a specific length, use the **Line Angle/Length** tool. Set the angle and length fields using drag, or if you prefer, type the desired values.

Line to a known point

 To make a line from the current position to a known point, click the **Line Angle/Offset** tool, then click the point that is the end-point of the line. The angle and length will be set automatically. A small temporary circle will appear around the point that was snapped to.



Arcs and circles

Arcs and circles are one of the basic building blocks of ArtiosCAD. Any shape can be made out of lines and arcs. There are six Designer tools to create arcs and circles: the **Arc Start Angle** tool, the **Arc End Point** tool, the **Arc Center** tool, the **Arc Through Point** tool, the **Circle** tool, and the **Ellipse** tool.

The second button on the Geometry toolbar activates the Arc tools, and when held down, activates the Arc flyout toolbar. The third button on the Geometry toolbar activates the Circle tools, and when held down activates the Circle flyout toolbar.

ArtiosCAD prompts for five pieces of information when creating an arc:

- **Start angle**. This is the angle at which the arc projects from the start-point. This is shown as **A1** on the Status bar.
- **Arc radius**. This is the distance from the arc to the center of the circle of which the arc is a piece.

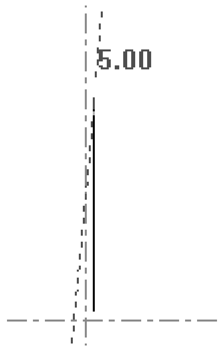
- **X** and **Y** offsets. These define the position of the arc's end-point in relation to its start-point.
- **End angle**. This is the angle at which the arc ends. This is shown as **A2** on the Status bar.

ArtiosCAD constructs an arc as soon as it knows any 3 of these 5 pieces of information.

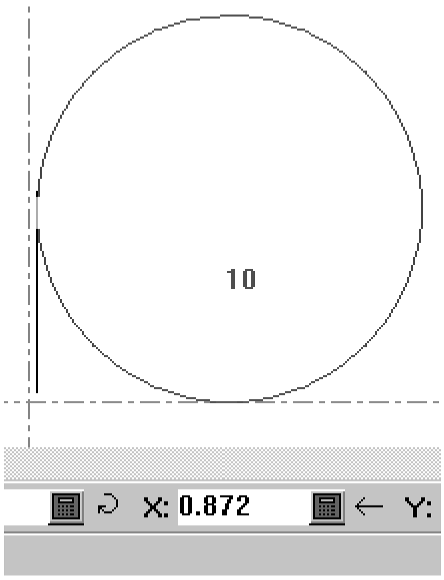
Arcs using a start angle, radius, end-point offset

If you know the angle, radius, and location of the end-point of the arc you want to make, use the **Arc Angle Offset** tool to construct the arc.

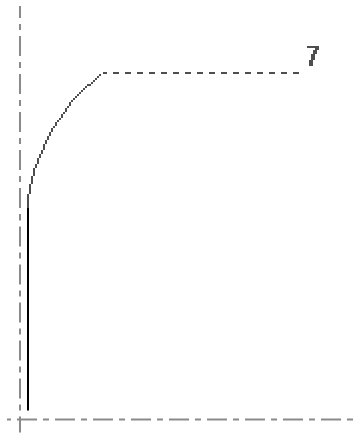
1. Click the **Arc Angle Offset** tool on the Geometry toolbar.
2. Use the mouse to set the drag for the start angle.



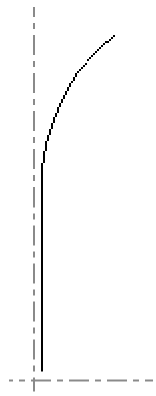
3. Once the start angle is set, use the drag to set the radius of the arc.



4. Set the Y offset of the end-point using the drag. To set the X offset instead, use the arrow keys to move the highlight to the X offset field.



5. As ArtiosCAD now has the pieces of information it needs to construct the arc, the arc is made.



Arcs using an X, Y offset, radius

Use the **Arc End Point** tool to make an arc when you know the radius and the X,Y offsets of the end-point.

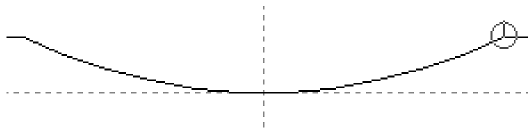
1. Click the **Arc End Point** tool and use the drag to set the X offset of the arc's end-point.
2. ArtiosCAD will then prompt for the Y offset of the arc's end-point.
3. After the Y offset of the arc's end point has been set, ArtiosCAD will prompt for the radius of the arc.
4. The arc is now created and the tool remains active in order to make another arc.

Arc Center

The third tool on the Arc flyout toolbar is the **Arc Center** tool. This tool makes an arc when any 3 of its 5 data fields are set: the radius, the X value for the center of the arc, the Y value for the center of the arc, the X value for the end-point, or the Y value for the end-point.

Arc Through a Point

The fourth tool on the Arc flyout toolbar is the **Arc Through Point** tool. The arc starts at the current position. To use this tool, click it, indicate the point the arc will go through, and then click the end-point of the arc.



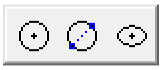
In the picture above, the end-point of the arc has a small snap circle around it.

If the start point is on a construction circle or arc, the drag is constrained to follow the arc.

Note: This tool can only be made using drag. There are no status bar prompts.

Circle

The **Circle** tool is the third button on the Geometry toolbar, and when held down, activates the Circle flyout toolbar.



To make a circle, click the **Circle** tool and then set the radius of the circle either by using drag or by typing a value in the prompt on the Status bar.

Once you set the radius, ArtiosCAD constructs the circle using the current position as the center of the circle. As with other tools, you can snap the radius drag to a point; a temporary circle will appear around the snapped-to point.

Circle Diameter

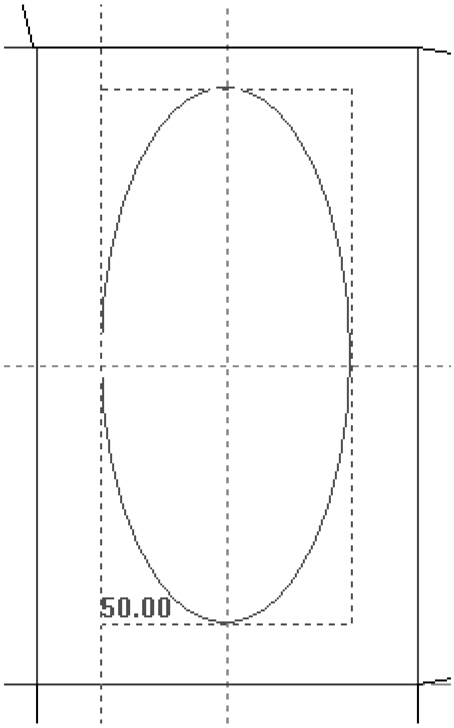
 The **Circle Diameter** tool is the second tool on the Circle flyout toolbar. To make a circle using this tool, do the following:

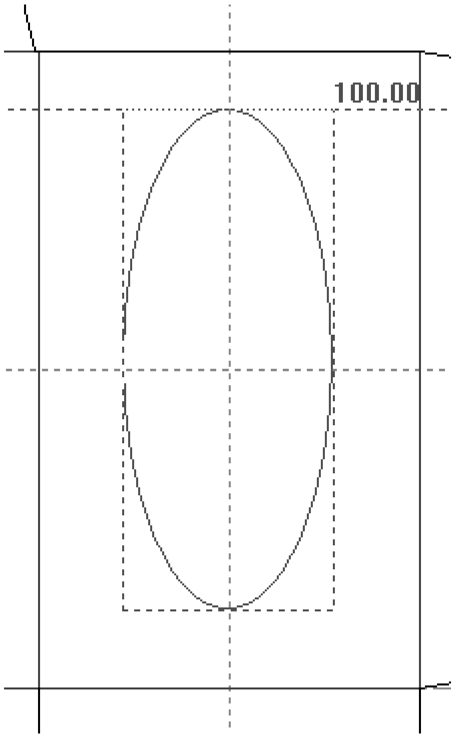
1.  Click the **Circle Diameter** tool.
2. Drag and click to set the angle from the current position of the line forming the diameter of the circle.
3. Drag and click to set the diameter of the circle. Alternately, press `Tab` on the keyboard to choose a different field on the Status bar to set with the drag - either the X offset or the Y offset of the end-point of the diameter.
4. Once you have set the end-point of the diameter, ArtiosCAD constructs the circle.

The tool remains active and the current position does not move so that you may continue drawing circles.

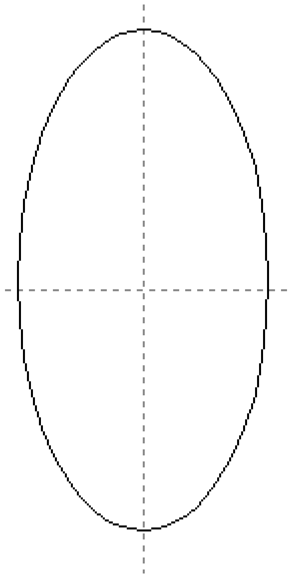
Ellipse

The third tool on the Circle flyout toolbar is the **Ellipse** tool .To make an ellipse, click the **Ellipse** tool and use drag to set the X and Y offsets of the edge of the ellipse from the center point. *Angle* refers to the angle at which the ellipse is viewed. It does not affect the construction of the ellipse. Setting the X and Y offsets are shown below.





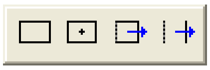
When both the X and Y offsets are set, ArtiosCAD constructs the ellipse.



Rectangles

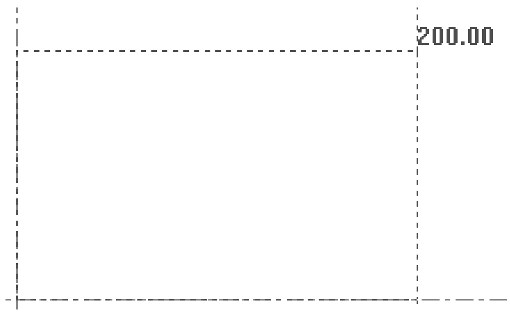
Instead of making you draw four lines to construct a rectangle, ArtiosCAD makes the construction of rectangles easy by providing a tool for that purpose. You can make a rectangle from scratch, or you can expand a line into a rectangle.

The fourth button on the Geometry toolbar activates the **Rectangle** tools, and when held down, activates the Rectangle flyout toolbar. The Rectangle tools are used to create rectangles.



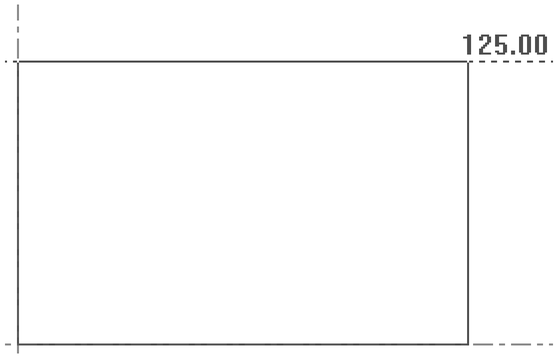
Rectangle

Use the **Rectangle** tool to create rectangles in two easy steps. To make a rectangle, click the **Rectangle** tool and indicate the X offset of the point diagonally opposite from the current position as shown in the picture below.

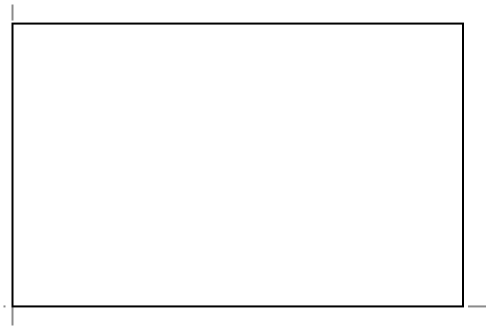


To construct the rectangle with blended corners, enter a value in the **Blend:** field on the Status bar before setting the X offset. Press `Enter` once you have entered the value to lock it in.

Once the X offset of the corner point is set, set the Y offset using the drag as shown in the picture below.



When the X and Y offsets have been set, ArtiosCAD constructs the rectangle and the current position moves to the corner of the rectangle whose offsets were set. Shown below is the completed rectangle.



The **Rectangle** tool remains active so that another rectangle may be made.

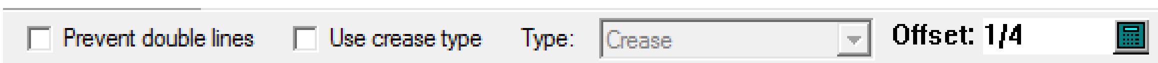
Rectangle from Line

The second button on the Rectangle flyout toolbar activates the **Rectangle from Center** tool. This tool creates a rectangle from its center with an optional blend on the corners. To use it, do the following:

1. Move the current position to the center of the desired rectangle by using `CTRL-W`, `CTRL-Q`, or **Tools** > **Current Point**.
2.  Click **Rectangle from Center**.
3. On the Status bar, enter an optional value in the **Blend:** field and press enter to lock the value in the field.
4. Use drag to set the X and Y coordinates of a corner of the rectangle. The values in the fields on the Status bar are for the entire width and length of the rectangle, not half of it.

The third button on the Rectangle flyout toolbar is the **Rectangle From Line** tool. This tool prompts you to select a line, and then uses the selected line as the base of a rectangle created using drag.

When you use this tool, the following controls appear on the Status bar:



To use this tool, click it and then select a line. Use the drag to set the size of the rectangle, or type the desired value in the Status bar. **Prevent double lines** deletes or trims any of the rectangle lines that overlap with other design lines. Use crease type, when checked, causes the connecting line to be changed to the crease type you specify in the **Type** drop-down list box. If the opposite line is within the perimeter of the design, it is also changed to a crease; if it is not, the other sides of the rectangle are changed to creases. Be careful of this when using this feature in rectangular holes, as the specified line and the opposite line will both be inside the and thus will both change to creases.

Once the offset of the opposite edge of the rectangle is set, ArtiosCAD constructs the rectangle. The tool remains active to create another rectangle from the one just created.

Note:

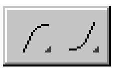
The change of the line type depends on whether or not the line is inside the design at the time when the rectangle was created; it will not change depending on the geometry when you rebuild. For example, if you make a flap with offset 4 and the line is changed to crease, if you then rebuild with -4 or 0 the line will still be a crease even though it is no longer supposed to be.

Prevent double lines prevents double lines against all line types in all layers, even those that are turned off.

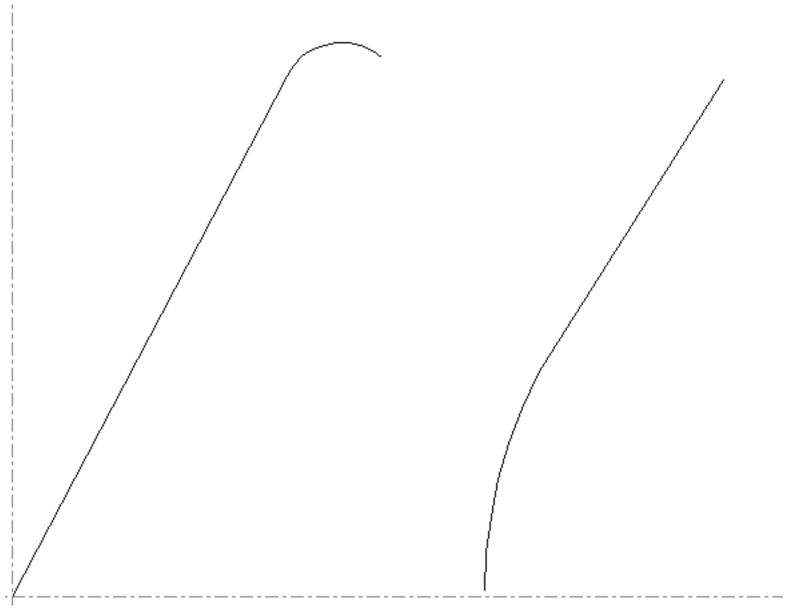
The fourth button on the Rectangle flyout toolbar activates the **Offset Line** tool. Use this tool to offset a parallel copy of a line from another. To use this tool, click it, click the line to copy, and then indicate the offset using drag.

Straight-curve and Curve-straight

The fifth button on the Geometry toolbar activates the Straight Curve tool, and when held down, activates the Straight Curve flyout toolbar. A straight curve is a straight line with an arc attached at the end. A curve straight is an arc with a straight line attached at the end.



In the picture below, item A is a straight curve. Item 1 indicates the start of the straight curve. Item 2 shows the drawing direction. Item B is a curve straight, and item 3 is the start of the curve straight.

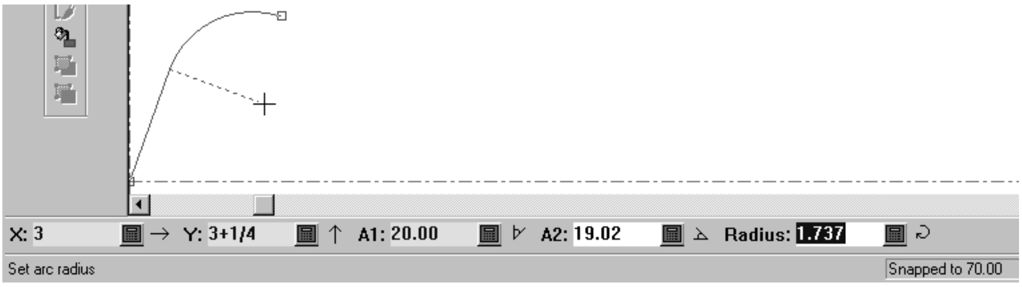


The first button on the Straight-Curve flyout toolbar, and the default Straight Curve button on the Geometry toolbar, is the Straight-Curve tool. To make a Straight-Curve, do the following:

1. Click the **Straight-Curve** tool on the Geometry toolbar.
2. Set the X and Y offsets of the end-point of the Straight-Curve.

3. Set the Start Angle of the straight portion of the Straight-Curve.

4. Set the radius of the Curve portion of the Straight-Curve with the drag.



5. Once ArtiosCAD knows 4 of the 5 pieces of information in the Status bar, it constructs the Straight-Curve.

The second button the Straight-Curve flyout toolbar is the **Curve-Straight** tool. It works the same way as the Straight-Curve tool, but the arc is made at the beginning of the line, not the end.

Line Join tools

 The sixth button on the Geometry toolbar activates the **Line Join** tool, and when held down, activates the Line Join tools flyout toolbar.

This tool joins two points, two circles, or a point and a circle with a straight line. It also works with construction line circles.

To use this tool, click it, indicate the first point, arc, or circle, and then indicate the second point, arc, or circle. When one of the objects being joined is a circle, the line is made tangent to the circle.

If you click an arc or a circle as the first object, to make a line tangent to that arc or circle, use the drag to set the angle and then either set the length or set the X and Y prompts for the end-point of the line. If you click close enough to a line or arc, it will snap to that line or arc.

 The second button on the Line Join tools flyout toolbar activates the **Arc Join** tool. As its name implies, it joins a point, line, arc, or circle to another point, line, arc, or circle with an arc. If two lines are being joined, they cannot be parallel. To use this tool, click it, indicate the first object to join, then the second object to join, and then finally the radius of the arc used to join the two objects. Depending on the points chosen, it is possible that the arc joining the two objects will intersect them.

Depending on how you set the drag, the points at which the arc joins the two objects could be different. The same happens when two circles are joined together with an arc – the points of tangency switch depending on the radius of the arc.

 The third button on the Line Join tools flyout toolbar is the **Tangent Arc** tool. This tool makes an arc tangent to three lines. To use this tool, click it, and then indicate the three lines to which the arc is to be tangent.

Line at Angle to a Line tool

 The seventh button on the Geometry toolbar activates the **Line at Angle to a Line** tool. This tool draws a line at the desired angle from the current position. To use it. click it, select a line or arc to measure the angle from, and then use the drag to make a line at the desired angle.

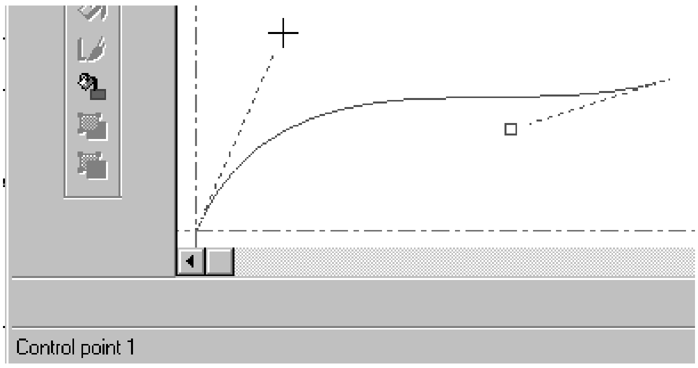
Beziers

The eighth button on the Geometry toolbar activates the **Bezier** tool. A bezier, also called a *French curve*, is a curve defined by four points: the start point, the endpoint, and two control points.

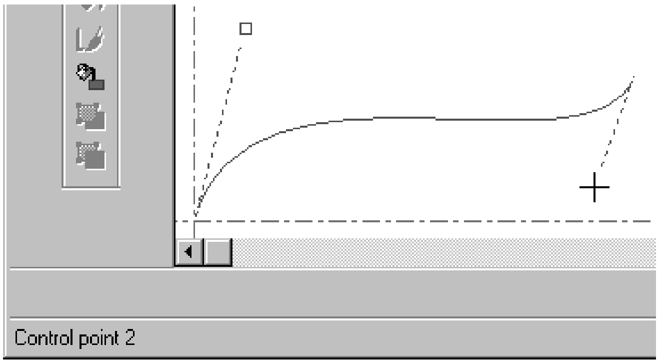
Freehand coordinates in the Snap Options dialog box must be checked for this tool to function. Click **Options** > **Snap** and then look in the **Snap to Point** group. If you see an error message in the Status bar `Cannot select coord` when trying to use this tool, Freehand coordinates is turned off and you must turn it on.

To make a bezier, do the following:

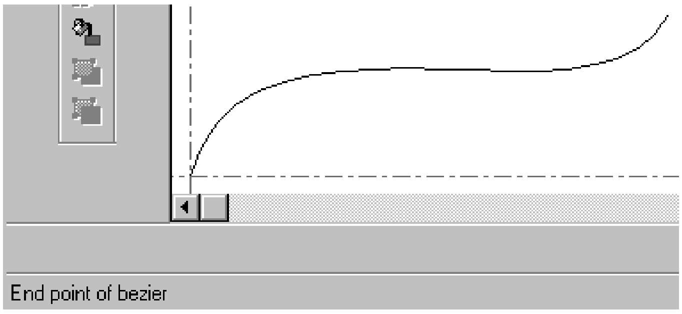
1. Click the **Bezier** tool on the Geometry toolbar.
2. The bezier will begin at the current position. Click the desired position for the endpoint.
3. Use the drag to set the position of the first control point. In the example below, the first control point is at the top of the left dashed line.



4. Use drag to set the position of the second control point. In the example below, the second control point is at the bottom of the right dashed line.



5. ArtiosCAD constructs the bezier once the second control point is set.



Drafting aids

ArtiosCAD provides two sets of tools to help create geometry – construction lines and extensions. Construction lines create non-manufacturable geometry. Extensions extend existing lines into infinity and existing arcs into circles.

Construction lines

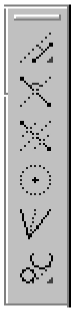
Construction lines are special lines used as drafting aids. They are not considered geometry, but they are used to make geometry.

How and when to use construction lines

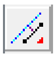
Use construction lines when you want to reference points that are not part of the geometry of a design. For example, if you want to go to a point that is a third of the way down a line, an easy way to do this is to divide the line in thirds using the **Conline Division/Midpoint** tool. Once the construction lines intersect with the actual line, you can move the current position to one of those intersections.

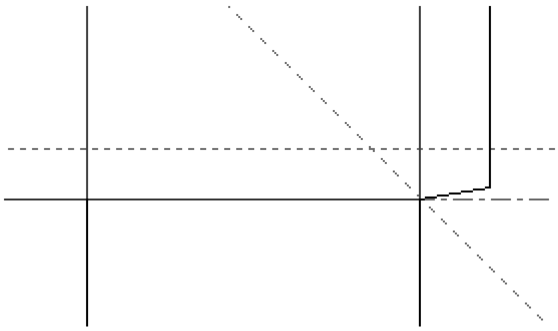
Construction lines also help when constructing rebuildable designs by providing points of reference. For instance, suppose you have imported a DDES file and want to make it rebuildable. When building a flap, the angle at which the flap is cut back varies with the width of the flap. You can use a construction line drawn over the side of the flap to “remember” the angle of the flap line. Then when you move the corner point, you can place it directly on the construction line, and the original angle is duplicated perfectly.

Creating construction lines is accomplished by using the tools on the Conlines toolbar, or the command on the Conlines submenu of the Tools menu.

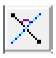


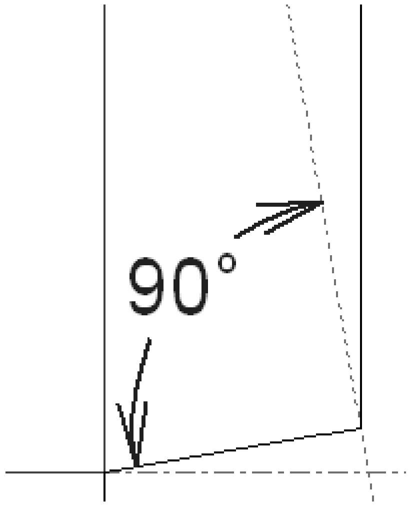
Conline Offset/Angle tool

 The first button on the Conlines toolbar is the **Conline Offset/Angle** tool. When activated, this tool prompts you to select a point or a line. If you select a line, you are then prompted for the offset between the line and the new construction line. If you select a point, you are prompted for the angle at which the construction line will be created.



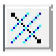
Conline At Angle From Line tool

 The second button on the Conlines toolbar activates the **Conline At Angle From Line** tool. When activated, this tool prompts for a line to measure from and a point that the construction line will pass through.



In the picture shown above, the line used to measure from is the line at the bottom of the glue flap, and the point passed through is the intersection of the bottom and side of the glue flap.

Conline Division tool

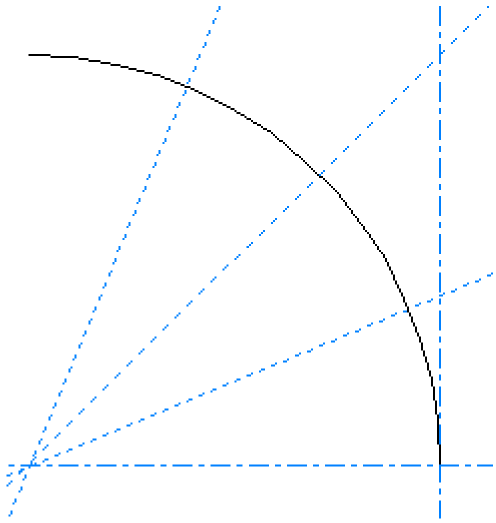
 The third button on the Conlines toolbar activates the **Conline Division/Midpoint** tool. When activated, this tool prompts for the number of divisions and a line or arc to divide. Click the line, first point of the line, or arc to divide, enter the number of divisions to make in the field on the Status bar, and press enter.

Note: To divide an arc, click the arc itself, not its endpoints, else the arc will be treated as a line and the dividing lines will be parallel instead of meeting at the center of the arc.

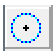
The default number of divisions is two. If you activate the tool and click a line or arc, it will automatically be divided in two by a construction line. Note that the object being divided is not split; for that, you must use the **Split Line** tool and click the points created by the construction lines.

The number of construction lines created by this tool will be one less than the number specified in the **Number of Divisions** field.

Shown below is an arc divided into four segments by three divisions.

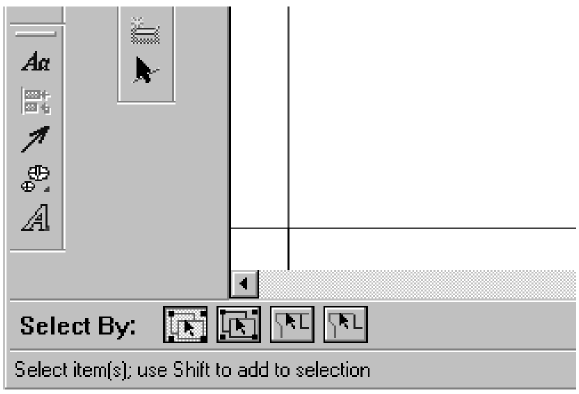


Conline Circle tool

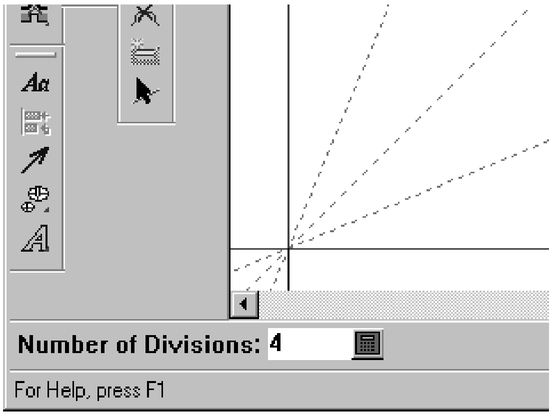
 The fourth button on the Conlines toolbar activates the **Conline Circle** tool. When activated, this tool prompts you to indicate the center of the circle. As long as **Freehand Coordinates** is checked in the Snap options dialog box, the center can be anywhere; it does not have to be on a point. Once you have indicated the center, ArtiosCAD prompts for the radius, which you can set either with drag or by typing in the Status bar. ArtiosCAD constructs a construction line circle once the radius has been entered.

Conline Angle Divide tool

 The fifth button on the Conlines toolbar activates the **Conline Angle Divide** tool. This tool works similarly to the **Conline Division/Midpoint** tool, but instead of dividing a line into line segments, it divides an angle into smaller angles.

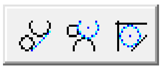


To use this tool, click it, enter the desired number of divisions, indicate one leg of the angle, and then indicate the other leg. The tool will remain active once it has made the divisions.

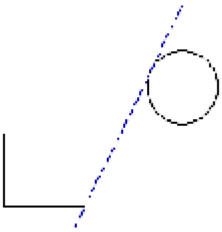


Conline Join tools

 The sixth button on the Conlines toolbar activates the Conline Join flyout toolbar.

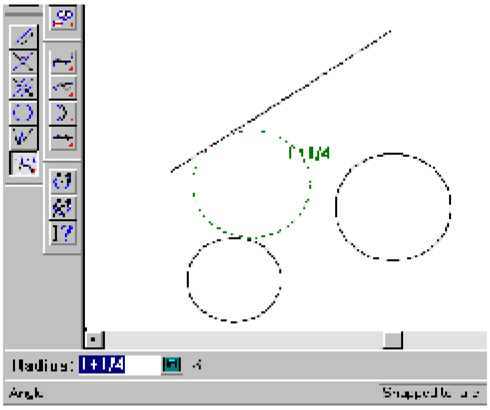


 The first button on the Conline Join flyout toolbar, and the default Conline Join button on the Geometry toolbar, is the **Conline Join** tool. This tool joins two points, two circles, or a point and a circle with a straight construction line. To use this tool, click it, indicate the first point, arc, or circle, and then indicate the second point, arc, or circle

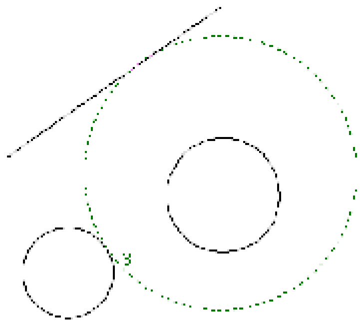


Note: When one of the objects being joined is a circle, the construction line is made tangent to the circle.

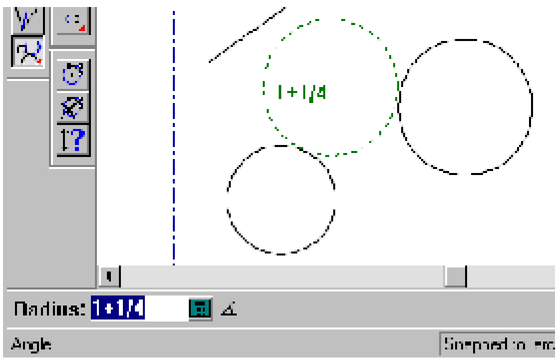
 The second button on the Conline Join flyout toolbar activates the **Conline Join With Concircle** tool. As its name implies, it joins a point, line, arc, or circle to another point, line, arc, or circle with a construction line circle. If two lines are being joined, they cannot be parallel. To use this tool, click it, indicate the first object to join, then the second object to join, and then finally the radius of the circle used to join the two objects. Depending on the points chosen, it is possible that the construction line circle joining the two objects will intersect them.



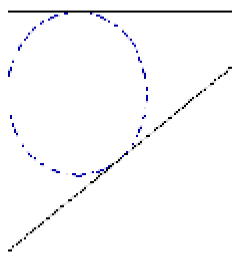
Depending on how you set the drag, the points at which the construction line circle joins the two objects could be different.



The same happens when two circles are joined together with a construction line circle – the points of tangency switch depending on the radius of the construction line circle.



 The third button on the Conline Join flyout toolbar is the **Conline Tangent Concircle** tool. This tool makes a circle tangent to three lines. To use this tool, click it, and then indicate the three lines to which the circle is to be tangent.

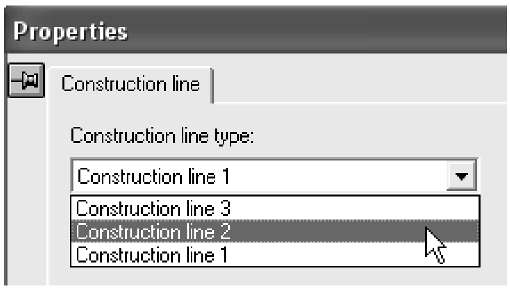


Changing construction lines

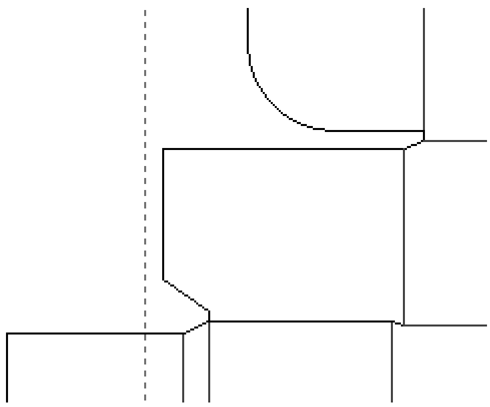
You may select construction lines with the **Select Conlines** tool  on the Select tools flyout toolbar, or by using the normal **Select** tool with either of the construction line options on the Status bar.

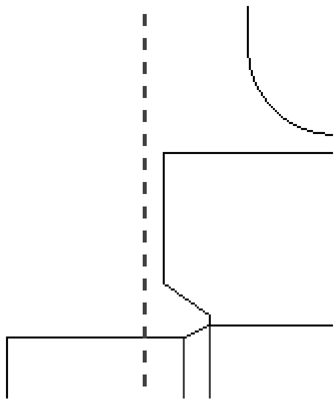
Each plotting style contains three construction line definitions.

Double-click a construction line with the **Select Conlines** tool to change its type in its Properties dialog box. To change the type of many construction lines at once, select them all and then press `ALT-Enter` to open the Properties dialog box.



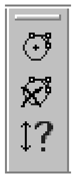
Shown below is a construction line before its type is changed and after its type is changed.

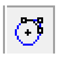


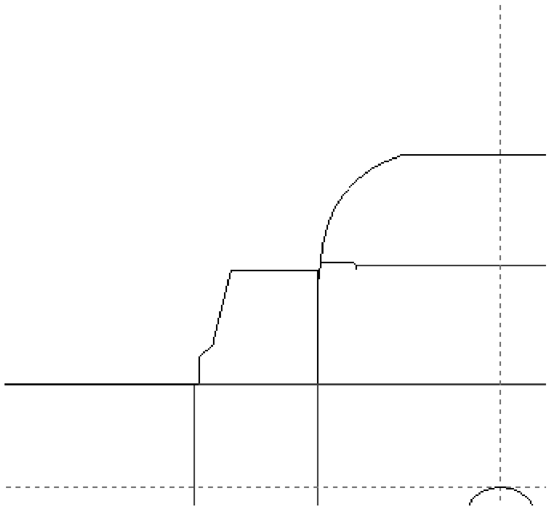


Extend/Measure tools

The tools on the Extend/Measure toolbar work in conjunction with construction lines by making geometry that is not manufactured.

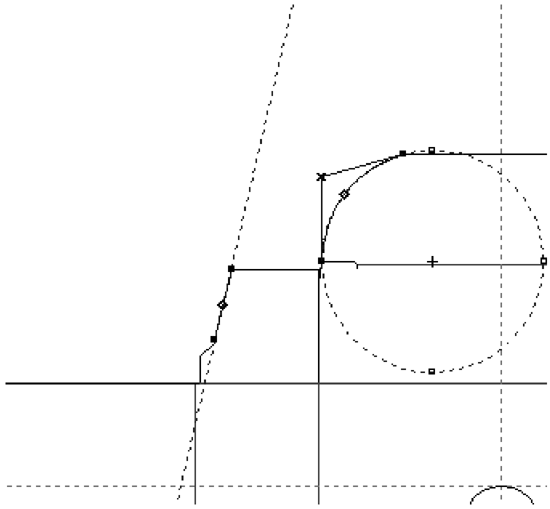


 The first button on the Extend/Measure toolbar activates the **Extend** tool. The Extend tool extends lines into infinity and arcs into circles. This is useful when you want to align objects with a line segment or an arc because the extended geometry provides extra points.



To use the Extend tool, click it, and then indicate a line or an arc to extend. Normally the Extend tool is a one-use tool and does not remain active after it extends the line or arc, but if you hold down `SHIFT` as you click the first item, the tool remains active and you can click other items to extend. Make sure to continue holding down `SHIFT` for the tool to continue. Hold down `CTRL` to use the Extend tool ignoring groups.

The points displayed using the Extend tool can be used as snap points and measure points.



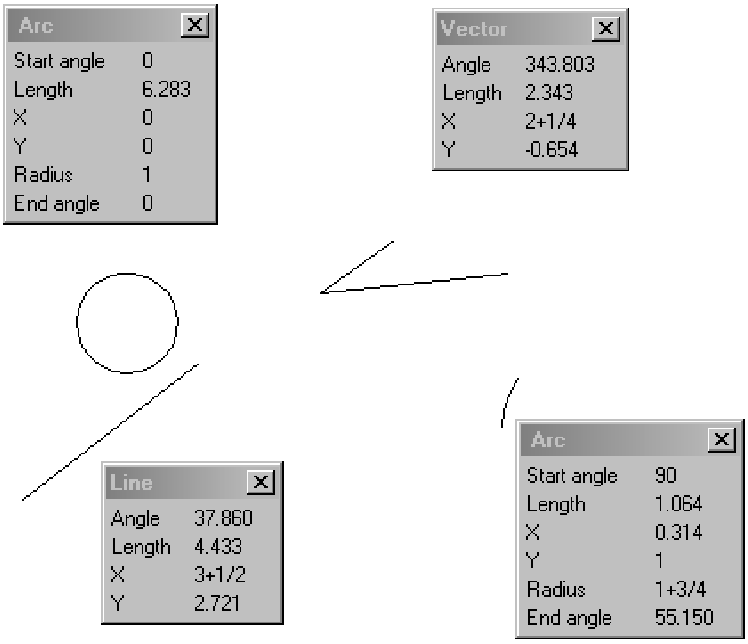
If the cursor is over an item when the Extend tool is started using a keyboard shortcut, the item is extended automatically.

When you use an extended point geometrically, such as creating a line to it or moving it, the extend points are cleared after the tool completes.

 The second button on the Extend/Measure toolbar activates the **Clear Extend** tool. The Clear Extend tool deletes all extensions. This is the only way to delete extensions.

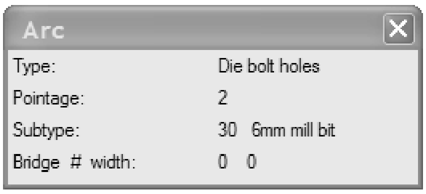
 The third button on the Extend/Measure toolbar activates the **Measure** tool. In Measure mode, the Measure tool displays pertinent measurements of arcs, lines, and the relationship between two points. In Attributes mode, the Measure tool displays information about the object under the mouse cursor. Change modes by selecting the appropriate button on the status bar, or by pressing `Tab`, or by using the left and right arrow keys.

To use the Measure tool in measure mode, click it, and then click an arc, line, circle, or set of two points to measure. Shown below are different types of measure tool information dialog boxes. To view just pop-ups, hover over any desired object.



Hold down CTRL while using this tool to measure between pixels. Move the mouse after selecting the second pixel to see the snap. If you measure between pixels, you might have to refresh the view when using high graphics mode.

Attributes mode shows various bits of information about the object under the mouse cursor such as length, line type, pointage, the number and width of bridges, etc. Only geometric elements can be viewed using the Measure tool.



Click the mouse button to make the measurements remain on the screen. To close a single measurement dialog box, click the **Close** button in its title bar. To delete all the measurement dialog boxes, click **Delete all measures** on the status bar.

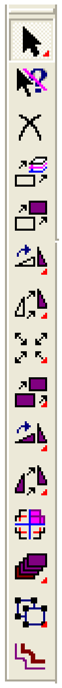
The Measure tool also functions as a tape measure. Click a point, and drag to another point; the measurement is displayed. Click the second point to make the measurement stay on the screen.

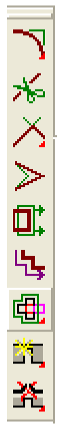
To exit the Measure tool, click **Exit Measure Tool** on the status bar.

Changing geometry

You can change geometry in many ways once it is created. For instance, you can change the type or thickness of rule used on the die, the location of a piece of geometry, and so forth. There are two ways that geometry can be changed: linearly or physically. To change a piece of geometry's linear properties means you are changing an aspect of the geometry without moving it on the die. Types of linear changes include the line type and pointage. Types of physical changes include moving geometry, copying it, stretching it, rotating it, and mirroring it.

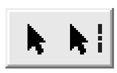
Use the commands on the Edit and Adjust toolbars shown below to change geometry.





Select tools

The first button on the Edit toolbar activates the Select tool, and when held down, activates the Select flyout toolbar.



The Select tools are used to select objects so that they may be changed.

 The first button on the Select flyout toolbar, and the default Select button on the Edit toolbar, activates the **Select** tool. Use this tool to select items in the drawing area – lines, text, dimensions, graphics, and anything else.

You can use the Select tool to move items by selecting them and dragging them to the desired position as long as the checkboxes for the objects you select are turned on in the Snap Options dialog box (click **Options** > **Snap**).

Hold down the `CTRL` key while using the Select tool to make a copy of the item(s) selected. Release the mouse button to put the copies down.

Hold down the `SHIFT` key to restrict the Select tool to vertical and/or horizontal movement only.

The **Select** tool has 17 modes of operation. The mode is selected by clicking the appropriate button on the Status bar.

 The first button indicates normal selection mode. This mode is the default mode and is how the Select tool has worked in the past. When you select an item which is a member of a group, the entire group is selected.

 The second button indicates that the Select tool will ignore groups. In this mode, if you select a member of a group, only it is selected, not all members of the group. Do not use this mode to select fills and strokes.

 The third button indicates that the Select tool will select connected lines, even if they pass through intersections.

 The fourth button indicates that the Select tool will select connected lines, stopping at the first intersection.

Whichever Select mode is selected in the first group of four buttons is remembered for this document in the current ArtiosCAD session.

 The fifth button indicates that the Select tool will select only design lines.

 The sixth button indicates that the Select tool will select only design and construction lines.

 The seventh button indicates that the Select tool will select only construction lines.

 The eighth button indicates that the Select tool will select only text items.

 The ninth button indicates that the Select tool will select only dimensions. Note that if you change the color of a dimension, it is considered graphics, not a dimension.

 The tenth button indicates that the Select tool will select only graphics.

For any of the above selection modes, if the objects being affected by the mode are turned off in the current view, activating the Select tool in that mode turns the appropriate objects on in the View Mode. For example, if dimensions are turned off in the View Mode, choosing the dimension-selecting mode turns dimensions on in the View Mode.

The eleventh button selects all items that are contained within or cross the selection rectangle or polygon. This is the default mode.

The twelfth button selects only those items with bounding boxes or clipping paths that are completely contained within the selection rectangle or polygon.

The thirteenth button lets you construct an irregularly-shaped polygon to select items. Click to set the start point, and then drag and click to make sides of the polygon. To delete the last side of the polygon, click

Undo next to the tool. To complete the polygon, join the end of the last side to the start point.

The fourteenth button selects all visible objects that would be selected using the current Select mode set by buttons five through ten.

The fifteenth button turns on **Nudge with stretch** mode. More information about using nudge mode appears later in this section.

The sixteenth button turns on **Nudge by a fixed value** mode.

The seventeenth button turns on **Nudge in a direction specified by a line** mode.

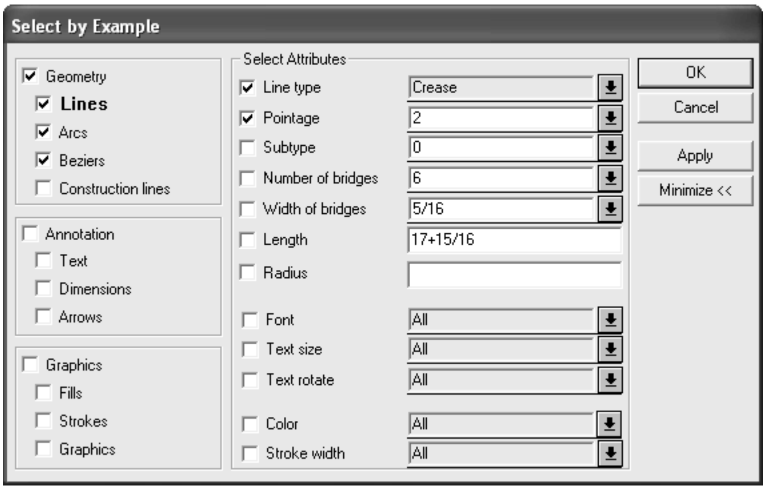
You can activate more than one Nudge mode at a time by clicking the mode buttons as desired.

Select Conlines tool

The second button on the Select flyout toolbar activates the **Select Conlines** tool. Use this tool to select construction lines.

Select by Example tool

The second button on the Edit toolbar activates the **Select by Example** tool. Use this tool to select items with the properties you define in the Select by Example dialog box.

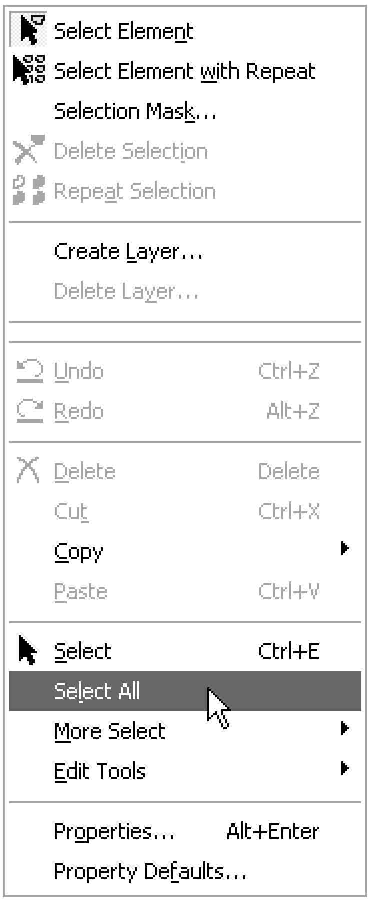


Once you have selected the attributes of the items to be selected, click **Apply** to see the results of your selection, or click **OK** to select the objects and dismiss the dialog box. When the items are selected, double-click them while still holding down `SHIFT` or `CTRL` to change their properties (or press `ALT-Enter`). You can also use any of the Edit tools.

Select All tool

The **Select All** tool is on the Edit menu. It selects everything in all visible layers; the items turn magenta to show they are selected. The **Select All** tool does not select items in layers that are not turned on.

To use this tool, click **Edit** > **Select All**.



Nudge mode in the Select tool

To *nudge* is to move by a very small distance. Nudging is available in the **Select**, **Select by Example**, **Select Conlines**, **Select Oneups** and **Select Periphery Lines** tools.

There are 3 nudge modes - **Nudge with Stretch**, **Nudge by a Fixed Value**, and **Nudge in Direction Specified by a Line**. These modes allow you to nudge or stretch the selected geometry or text by very small increments using the arrow keys along with **CTRL** and **SHIFT**. ArtiosCAD shows the distance last stretched or nudged as feedback on the Status bar.

The modes may be combined as desired - they are not mutually exclusive. Each mode is explained in further detail in the following sections.

Note:

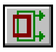
Regarding Undo and Redo, nudges are consolidated into one action. If you nudge too far, nudge back instead of using Undo. When using Nudge in Direction Specified by a Line, nudges in a direction are grouped together into one Undo action.

Default nudge distances

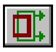
The table below shows how to move different distances in nudge mode. These are the default distances and can be customized as desired either system-wide in Startup Defaults or on a per-session basis by clicking **Options** > **Nudge**.

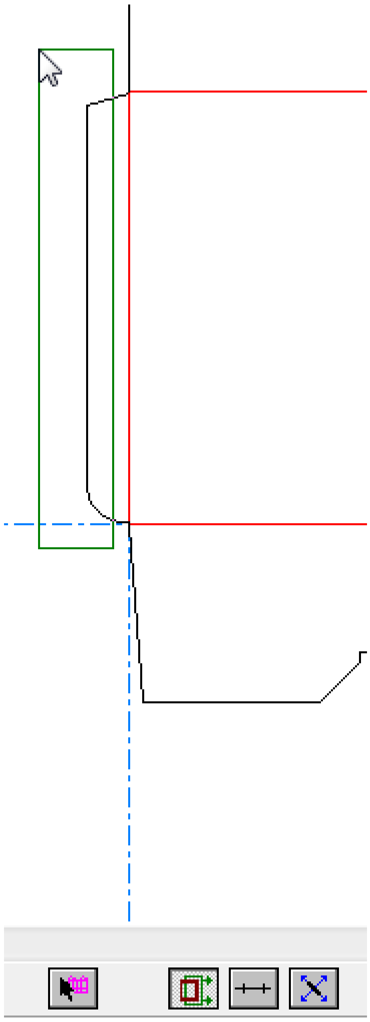
Press	To move in mm	To move in inches
Arrow key	1	1/16
CTRL-arrow key	5	1/2
SHIFT-arrow key	1/2	1/32
CTRL-SHIFT-arrow key	10	1

Nudge with Stretch

 The **Nudge by Stretch** mode works with a rectangle selection window. Items fully inside the selection rectangle are moved, while items crossing the border of the selection rectangle are stretched. Note that other Select tool controls affect what is selected when using a window to select items.

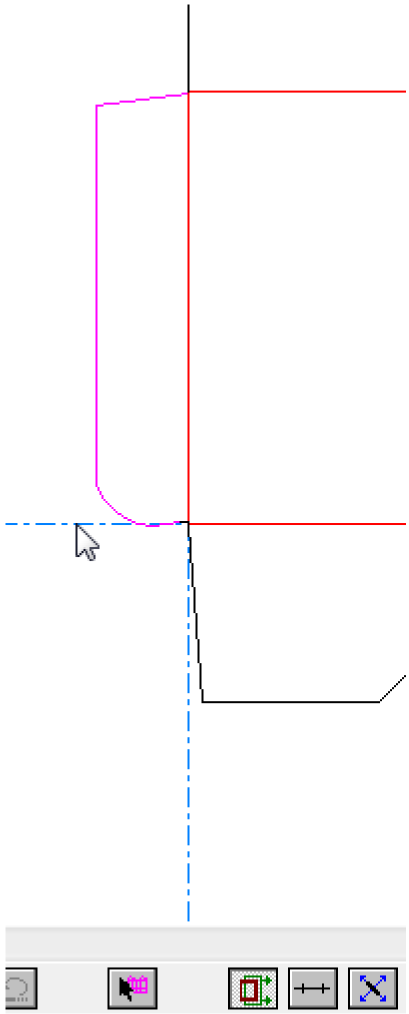
To use this mode, do the following:

1.  Start the **Select** tool in an open single design or manufacturing file.
2.  Click the **Nudge with stretch** button on the Status bar.
3. Click and drag a rectangle selection window around the items to nudge.



4. Use a combination of CTRL, SHIFT, and the arrow keys to nudge the selection in the desired direction. Arcs and beziers are always stretched using the distance option (as opposed to angle or radius).
5. The items remain selected for you to continue to adjust.

Shown below is the nudged glue flap with the nudge exaggerated for effect.



Note:

Nudge with Stretch will not move or stretch images, and will only move (not stretch) text items.

Note:

Nudge with Stretch cannot be used immediately after an Undo or Redo, as Undo and Redo clear the current selection and restart the Select tool.

Note:

 When Nudge with Stretch cannot be used, such as when the Select tool is restarted while there is a current selection, a do-not-use symbol (circle with a slash through it) appears on its button and the nudge will move the selection rather than stretching it.

Note:

This mode is not available in the **Select Conline**, **Select by Example**, and **Select Oneup** tools.

Nudge by a Fixed Value

The **Nudge by a Fixed Value** mode works with the current selection and moves the selected items by a fixed distance depending on the keys you press. It is similar to the nudge functionality in previous versions of ArtiosCAD. Refer to the *Default nudge distances* section for more information on distances and modifier keys.

To use this mode, do the following:

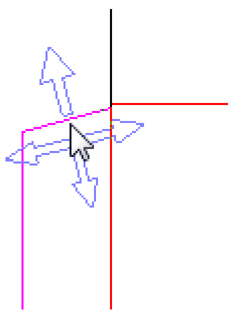
1. Start the **Select** tool in an open single design or manufacturing file.
2. Select the items to nudge.
3. Click the **Nudge by a Fixed Value** mode selector on the Status bar. If you hover over the button, ArtiosCAD shows the current nudge distances in a message on the Status bar.
4. Use a combination of CTRL, SHIFT, and the arrow keys to nudge the selection in the desired direction. Arcs and beziers are always stretched using the distance option (as opposed to angle or radius).
5. The items remain selected for you to continue to adjust.

Nudge in Direction Specified by a Line

The **Nudge in Direction Specified by a Line** mode nudges the current selection in a direction at 90 degrees (or a multiplier of 90 degrees) to a line. When you hover over a line in this mode, the drag changes to a four-direction display of arrows with a large arrow indicating the nudge direction of the up arrow key.

To use this mode, do the following:

1. Start the **Select** tool in an open single design or manufacturing file.
2. Select the items to nudge.
3. Click the **Nudge in Direction Specified by a Line** mode selector on the Status bar.
4. Hover over the line to use to set the direction of the nudge. The drag will change to 4 arrows as shown below.



5. Use a combination of CTRL, SHIFT, and the arrow keys to nudge the selection in the desired direction. Arcs and beziers are always stretched using the distance option (as opposed to angle or radius).
6. The items remain selected for you to continue to adjust.

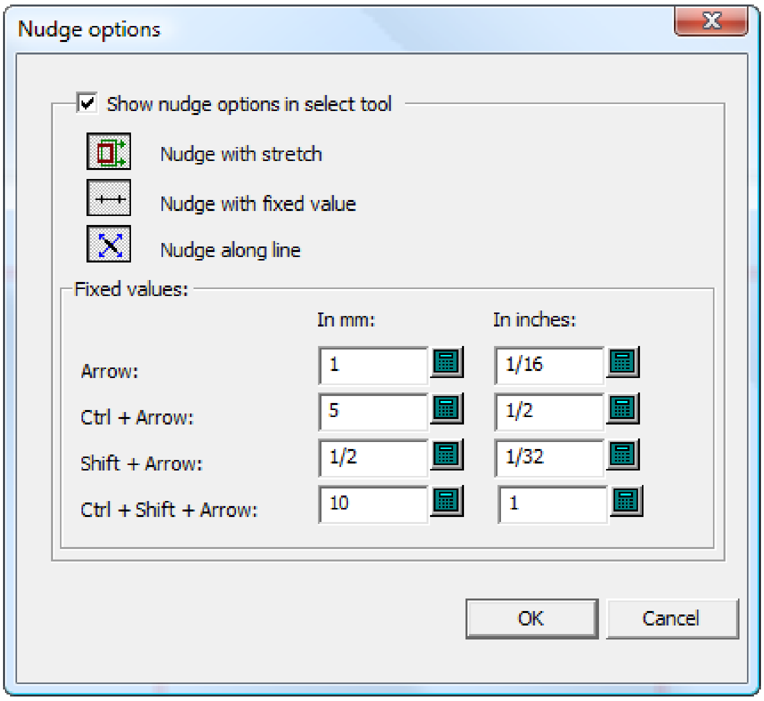
Note:

This nudge mode always uses the current angle of a line. Unanticipated results might occur when you use this tool on a parametric line in a resizable design.

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:

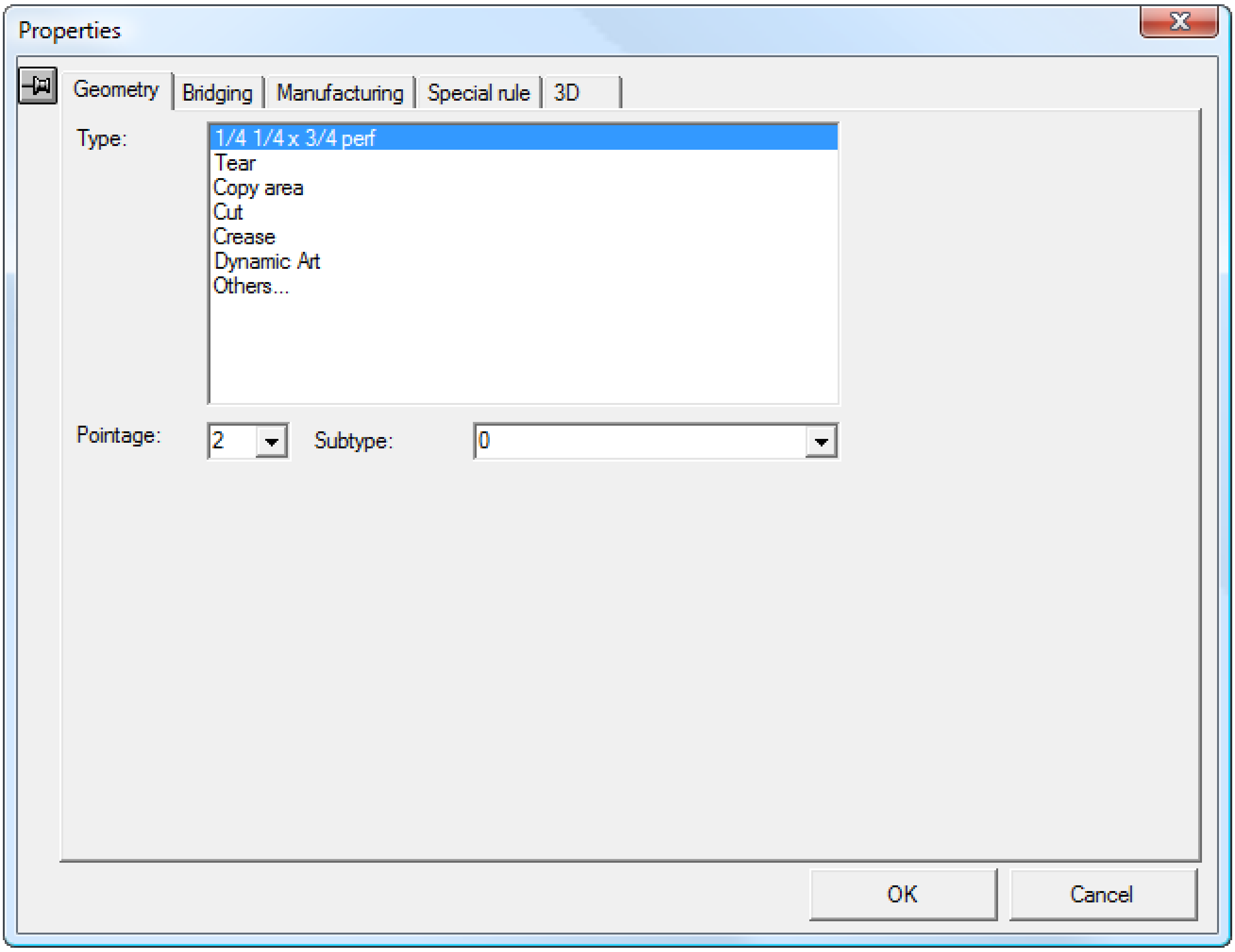
If you change the nudge options or Defaults while a Select tool is active, the tool will be restarted to update it and the current selection will be lost.

Changing linear properties

Linear properties are those properties of objects that are not related to their physical placement.

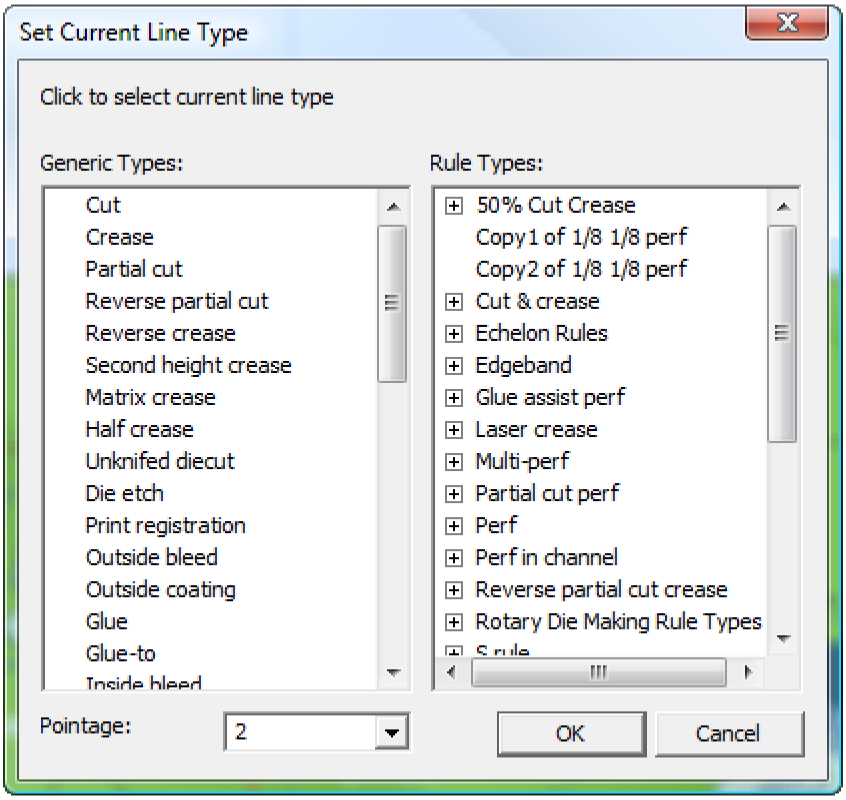
Changing line type and pointage

To change the line type and pointage, activate the **Select** tool and double-click the object to change. To select more than one object at a time, hold down `SHIFT` while selecting objects, and double-click the last object. The Properties dialog box will appear.



On the Geometry tab of the Properties dialog box, you can change the type of a line, its pointage, and its subtype. You can change other properties on additional tabs if they are available.

Lines which are already used in the workspace are listed in the **Type**: group. Clicking **Others...** in the **Type**: group opens the Set Current Line Type dialog box, which lets you change to a line type which is not listed in the **Type**: group.



If the name of the line type contains **Inside** or **Outside**, use it in a corresponding **Inside** or **Outside** layer class. **Inside** types and classes are used on the unprinted side of a design, while **Outside** types and classes are used on the printed side.

Once you have chosen a new line type in this dialog box, click **OK** to set it, or **Cancel** to ignore your selection. Either click will return to the Properties dialog box.

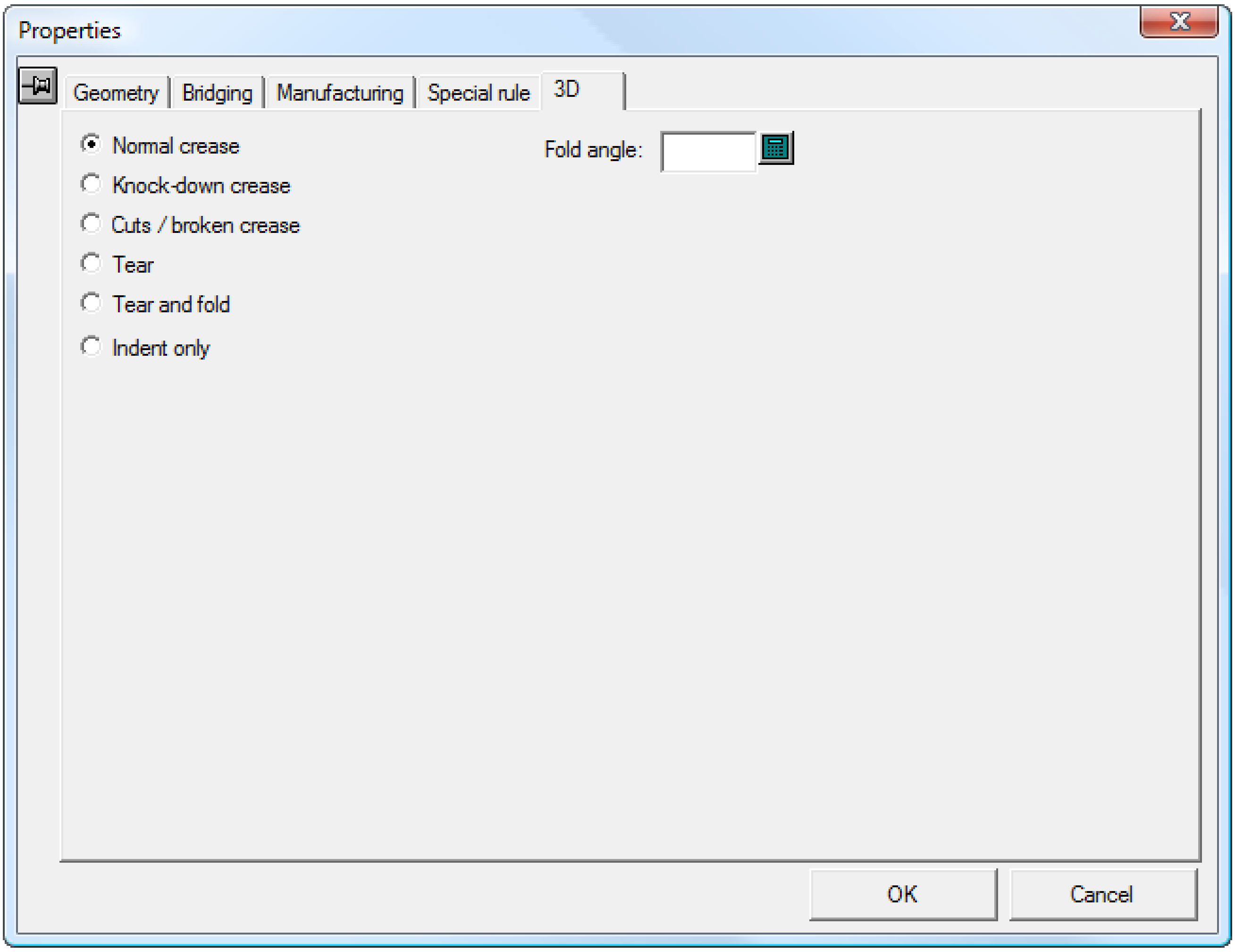
After making the desired property changes, click **OK** to apply the changes or **Cancel** to discard them.

Foldable line type properties in Single Design

The 3D tab in the Properties dialog box in Single Design lets you choose how the current selection behaves in 3D. Foldable lines are:

- Crease
- Second-height crease
- Matrix crease
- Reverse crease
- Partial cut
- Reverse partial cut
- Perf
- Cut-crease
- Partial cut-crease.

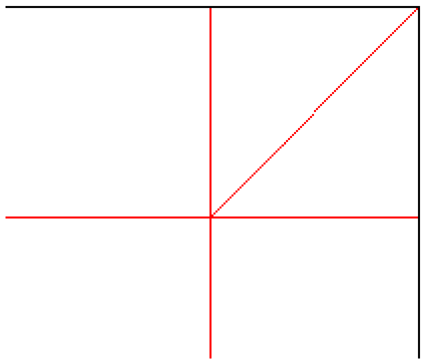
When you double-click one of these types of lines, set the how the crease will behave in 3D on the **3D** tab of the **Properties** dialog box.



Normal creases fold or bend in 3D as they should.

Knock-down creases are creases that are only used for shipping the carton, and are not part of the structural integrity of the carton. They are shown as indentations in 3D.

Cuts / broken creases appear and function as cuts. These are used where creases meet at a point and ArtiosCAD does not know how to set all the fold angles correctly so that they meet. In the example below, set one of the three creases at the corner to be a broken crease.



Tears are creases or special rules that tear.

Tear and fold special rules both tear and fold. Note that curved creases can either tear or fold, but not both.

Indent only perfs make an indentation on the panel but do not fold or tear.

You may also set or change the **Fold angle** of this line in 3D in this dialog box.

Click **OK** to change the properties of the line and return to ArtiosCAD.

Notes about special rules

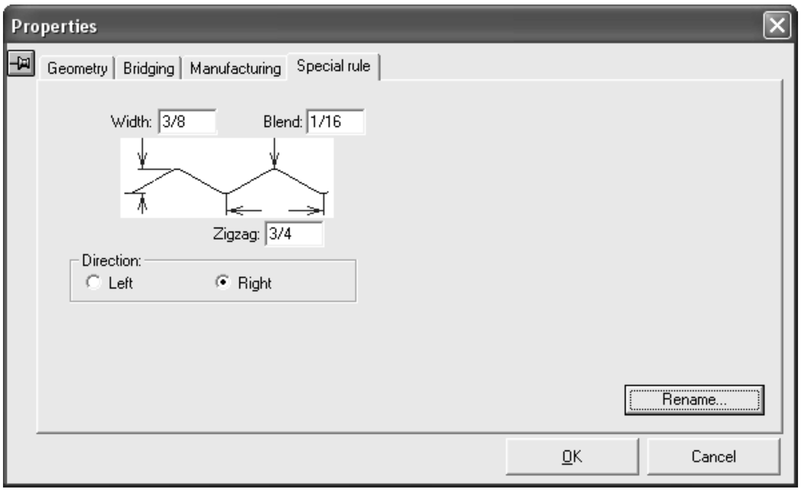
Special rules are rules which have specific manufacturing attributes, such as 5mm Zig Zag Left rules or ½ inch Echelon Right rules. Their definition is explained in more detail in the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

Different special rules are treated as one rule on output if they are end to end, and there are no end points of other lines within 1/32 inch of the join, and the lines are within 18 degrees of tangential at the join.

To treat such special rules as separate lines, split any lines that intersect them at the intersection point.

The **Expand Special Rules** command on the **Tools** > **Adjust** menu separates connected special rules into individual lines and arcs.

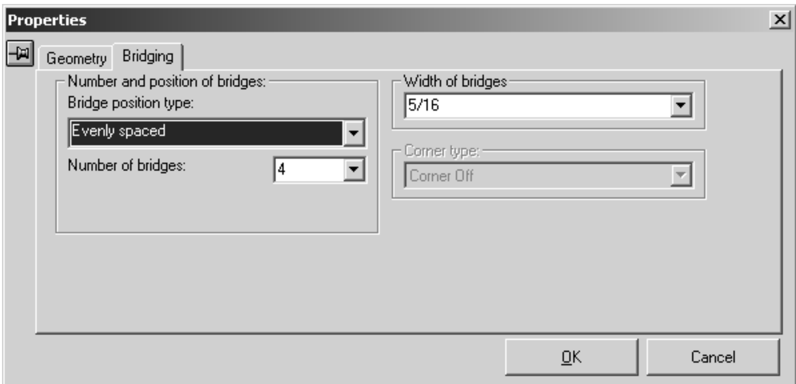
You can rename a special rule by clicking **Rename** on the Special rules tab of the Properties dialog box.



When a design containing the same name for differently-defined special rules is opened, the duplicate special rule name will be changed to COPY<number> of <special rule name>, such as **Copy1 of 3mm x 3mm perf.**

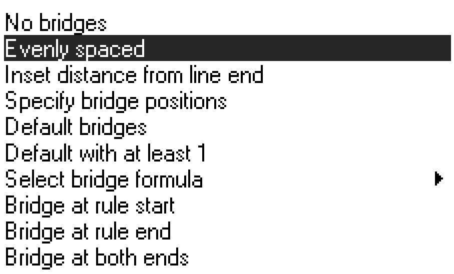
Changing Bridging properties

Shown below is the **Bridging** tab in the Properties dialog box when geometry is selected in Single Design.



Number and position of bridges

The **Bridge position type** drop down list box displays the following choices **for automatic bridging**:

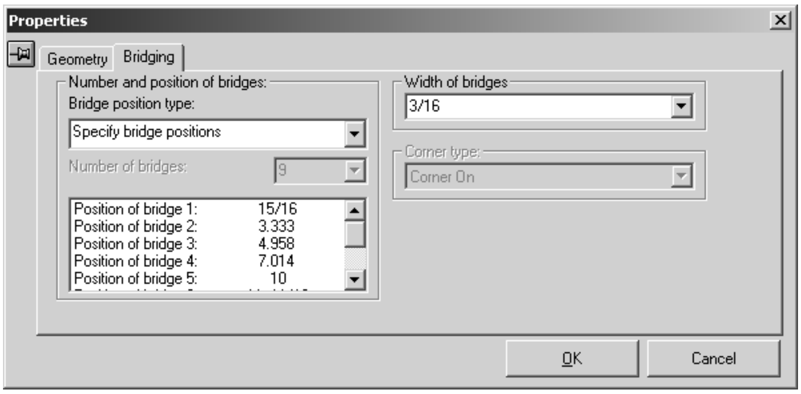


No bridges removes all automatically-placed bridges from the selected geometry.

Evenly spaced positions up to 100 bridges evenly along a line.

Inset distance from line end specifies the distance of a pair of bridges from each end of the line. If the line requires more than two bridges, they are evenly spaced between the two inset bridges.

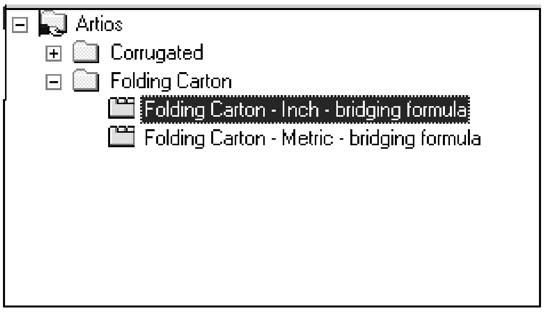
Specify bridge positions shows the position of up to 100 pre-existing bridges on a line.



Default bridges regenerates the bridges of the selected geometry based on the current default bridging formula.

Default with at least 1 is used in the construction of a standard when the size of the line in the final design is not known but requires at least one bridge.

Select bridge formula lets you choose the bridging formula to apply to the selected geometry. The current bridging formula is selected by default.



Bridge at rule start adds a bridge at the start of each selected line and removes all other bridges in the selected geometry.

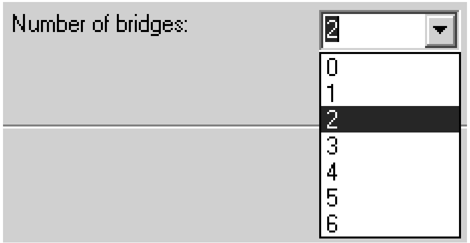
Bridge at rule end adds a bridge at the end of each selected piece of geometry and deletes all the other bridges in the selected geometry.

Bridge at both ends places a bridge at both ends of each line in the selected geometry.

Bridge at rule start, **Bridge at rule end**, and **Bridge at both ends** places one or two bridges at the half bridge with from either end of the line. That is, ArtiosCAD indents a half bridge width from the end of the line and then places the bridge.

Changing the number of bridges

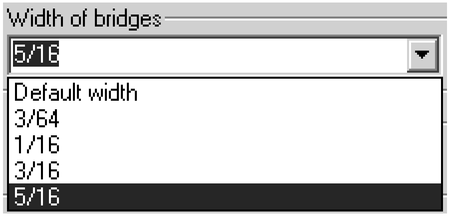
The **Number of bridges**: field and drop-down list box let you manually specify the number of bridges to assign to the selected geometry.



You can also type an integer in the field if the desired value is not shown.

Changing the width of bridges

The **Width of bridges**: field and drop-down list box shows the width of bridges in the currently-selected geometry and allows the selection of a new width.

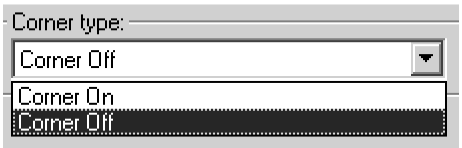


The **Default width** is the width defined by the current bridging formula. The other choices on the list are the widths of other bridges in the design. You may type a new bridge width in the field.

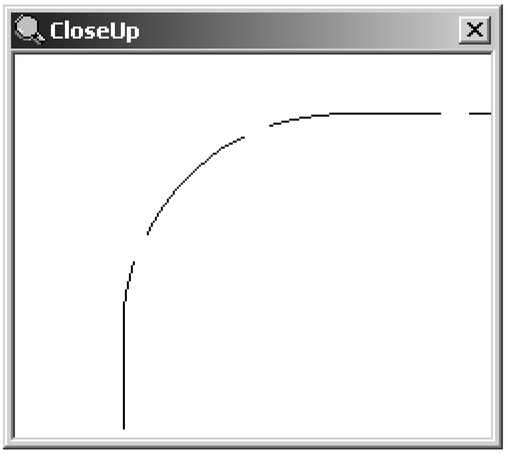
If a line needs to have bridges of different widths on it, that line must be split and the bridge widths applied separately. The **Add Bridge** tool can split lines as needed.

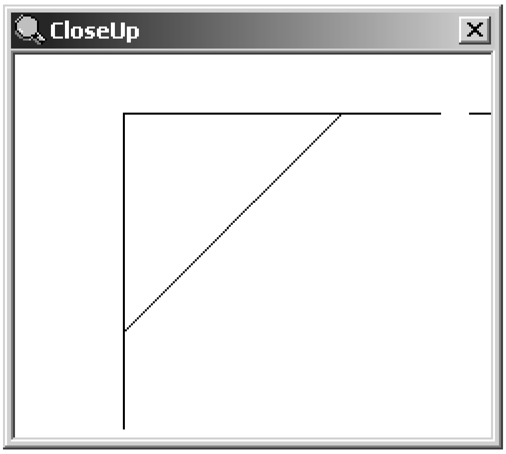
Changing the bridging for arc corners

The **Corner type** drop-down list box controls the behavior of bridging in corners made by an arc. **Corner Off** specifies that the arc is unaffected. **Corner On** specifies that the corner will be squared and cut by an additional line.



Shown below is an arc with **Corner Off**, and below that is **Corner On**.

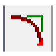




Changing the current bridging formula

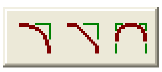
To change the current bridging formula, click **Options** > **Change Bridge Formula**.

Blend tool

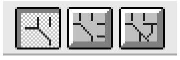
 A *blend* is an arc connecting two lines. Blends are used to round corners. They can be used on any non-parallel lines - if the lines don't share an endpoint, they will both be extended to a point where they can be blended. The **Blend** tool is the first button on the Adjust toolbar. It prompts for the radius of the blend, and then prompts you to click the two lines forming the corner to be rounded. Corners can be rounded as long as this tool is active.

Chamfer tool

 The **Chamfer** tool is the second button on the Blend tools flyout toolbar. **Chamfers** are bevelled corners. The Blend tools flyout toolbar is shown below.

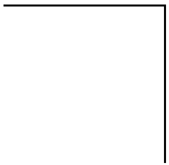


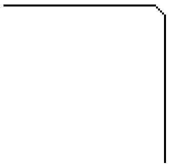
The Chamfer tool has three modes of operation, each controlled by a button on the Status bar. The availability of the fields on the Status bar is controlled by the current mode selection button.



 The first button on the Status bar turns on **Chamfer Equal Distances** mode. This causes the Chamfer tool to cut each line of the corner back the same distance and then connect them to form the chamfer. To use the Chamfer tool in this mode, do the following:

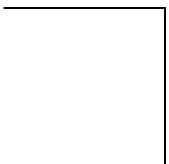
1.  Click the **Chamfer** tool. The Chamfer Equal Distance mode selection button is selected by default.
2. Enter the length of the cutback in the **Length along first line**: field:, for example ¼ inch or 6 mm.
3. Click the first line that forms the corner to be chamfered. It will turn magenta to indicate it is selected.
4. Click the second line that forms the corner to be chamfered. The chamfer will be made. Shown below is a corner before creating a ¼ inch chamfer and then the same corner after chamfering.

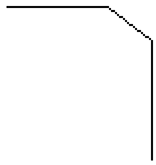




 The second button on the Status bar turns on **Chamfer 2 Distances** mode. This causes the Chamfer tool to cut each line of the corner back a different distance and then connect them to form the chamfer. To use the Chamfer tool in this mode, do the following:

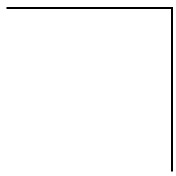
1.   Click the **Chamfer** tool, and then click the **Chamfer 2 Distances** mode selection button on the Status bar
2. Enter the length of the first line's cutback in the **Length along first line**: field, and enter the amount of cutback for the second line in the **Length along second line**: field, for example 12 mm for the first and 9 mm for the second.
3. Click the first line that forms the corner to be chamfered. It will turn magenta to indicate it is selected.
4. Click the second line forming the corner to be chamfered. The chamfer will be made. Shown below is a chamfer 12 mm cut back on the first line and cut back 9 mm on the second.

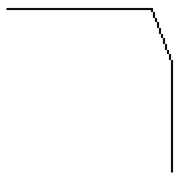




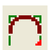
 The third button on the Status bar turns on **Chamfer Distance/Angle** mode. This causes the Chamfer tool to cut back a set distance along the first line of a corner and then make a line at the specified angle to meet the second line of the corner, thereby creating the bevelled edge. To use the Chamfer tool in this mode, do the following:

1.   Click the **Chamfer** tool, and then click the **Chamfer Distance/Angle** mode selection button on the Status bar.
2. Enter the length of the first line's cutback in the **Length along first line**: field, and enter the angle for the chamfer in the **Angle to first line**: field, for example `12` mm for the cutback and `70` for the angle.
3. Click the first line that forms the corner to be chamfered. It will turn magenta to show it is selected.
4. Click the second line forming the corner to be chamfered. The chamfer will be made. Shown below is a chamfer 12 mm cut back and at 70 degrees.

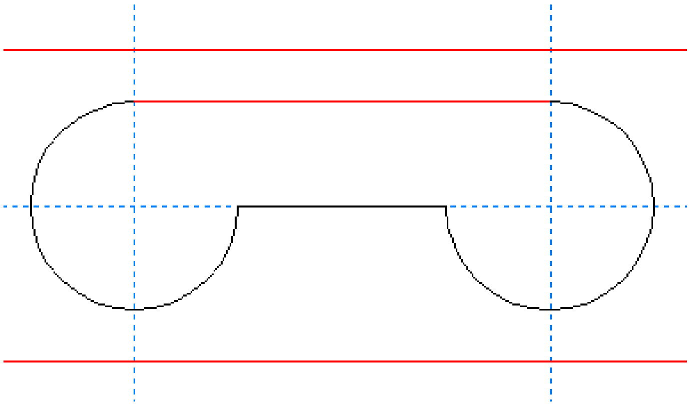




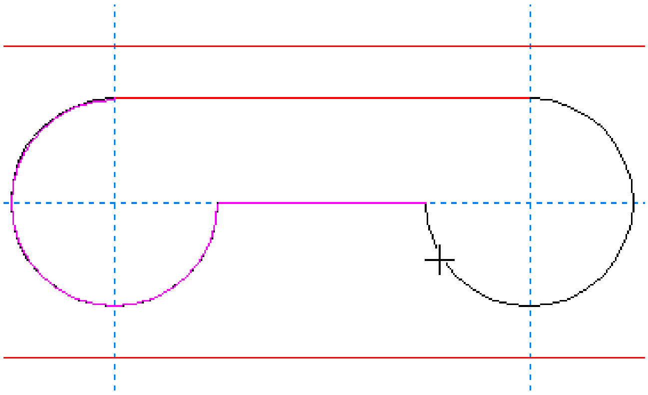
Blend 3 Lines tool

 The **Blend 3 Lines** tool is the third button on the Blend tools flyout toolbar. Use it to blend 3 lines or arcs together; the geometry does not have to touch. The middle line or arc is changed to a blend and the end lines or arcs are shortened to meet the blend. ArtiosCAD determines the needed radii settings automatically. If the lines or arcs are not able to be blended, ArtiosCAD displays a message that line construction is not possible.

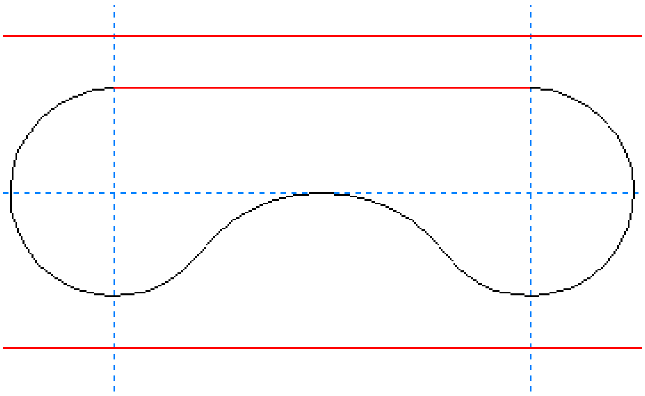
Shown below is an example of some geometry before it is blended:



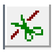
Using the tool to select the arcs and lines to be blended:



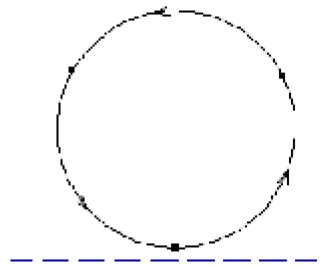
The third click creates the blend:



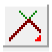
Split line at point

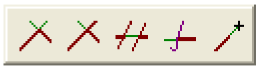
 The second button on the Adjust toolbar activates the **Split Line** tool. This tool divides a line into two lines anywhere along the line where you click the mouse. Activate the tool, click the line to split, and then click the split point. The **Direction** view mode is automatically turned on when you select this tool, and automatically turned off when you select a different tool. This tool behaves as if the **Freehand Coordinates** checkbox in the Snap Options dialog box is turned on.

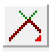
The picture below shows a circle after being split into 3 arcs. There are now 3 points and 3 direction indicators.



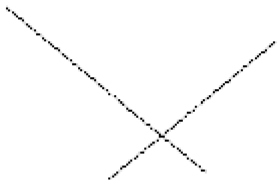
Trim/Extend tools

 The third button on the Adjust toolbar activates the Trim/Extend Two Lines tool, and when held down, activates the Trim/Extend flyout toolbar.

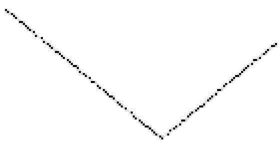


 The first tool on the Trim/Extend flyout toolbar, and the default Trim/Extend tools button on the Adjust toolbar, is the **Trim/Extend Two Lines** tool. It makes two lines that are not parallel extend to meet each other. It also makes one line that intersects another end at the intersection. When clicked, this tool prompts for the first line to extend, and then prompts for the second line to extend. Arcs may be trimmed against as well.

An example of trimming two lines:

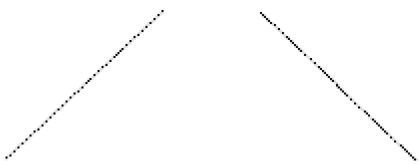


After using the Trim/Extend Two Lines tool:

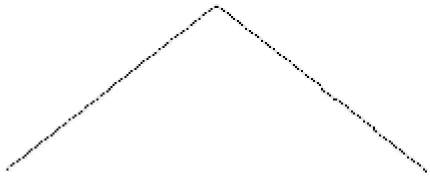


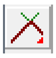
When trimming two lines against each other, select the portions of the lines to keep.

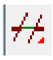
An example of extending two lines:



After using the Trim/Extend Two Lines tool:

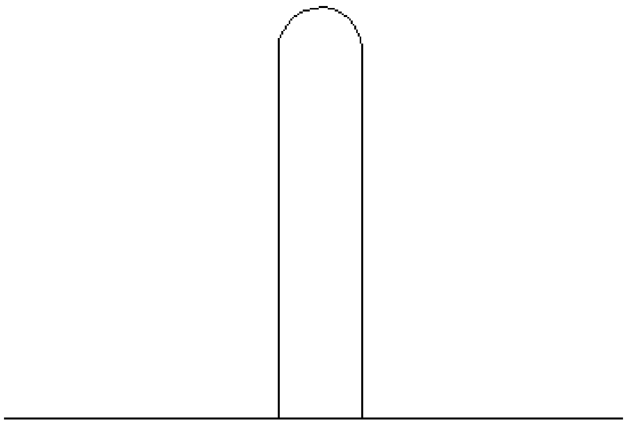


 The second tool on the Trim/Extend flyout toolbar activates the **Trim/Extend One Line** tool. This tool works the same way as the Trim/Extend Two Lines tool, except it only trims or extends one line.

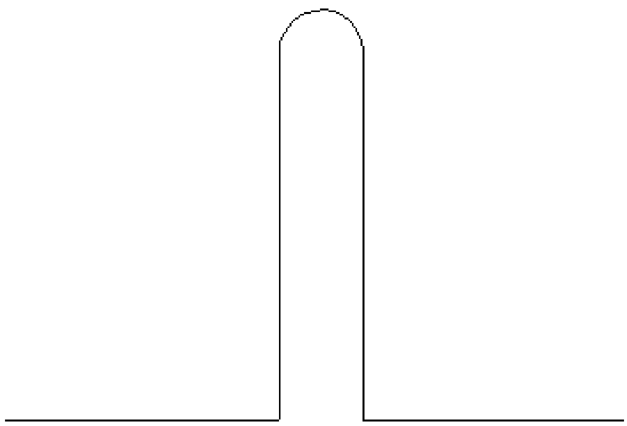
 The third tool on the Trim/Extend flyout toolbar activates the **Trim Interior** tool. This tool trims a segment of a line, arc, or bezier which intersects with at least one other item. It trims along the item you click in both directions. If only one intersection is found, the item is trimmed back to that intersection. If an intersection is found in both directions, the segment between the intersections is deleted. The resulting lines are rebridged using the default bridging formula. This tool works for lines, arcs, and beziers.

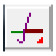
To use this tool, click it, and then click the segment to delete.

Shown below is a construction before using the Trim Interior tool.



The Trim Interior tool is used to remove the segment at the slot opening in the picture shown below.



 The fourth tool on the Trim/Extend flyout toolbar is the **Trim Against Selection** tool. This tool trims or extends multiple lines against the current selection at the same time. You must select at least one line, arc, or bezier before starting this tool. This selection is called the *trim against* selection.

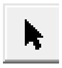
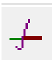
This tool now has two modes: *Limited* mode and *Full* mode. The mode is determined by how the lines trimmed against or extended to are connected. If the lines are connected so that there is a two-sided path, the tool can operate in Full mode, and all commands on the Status bar are available:

If the trim-against lines are not connected, or are connected or intersect in such a way that ArtiosCAD cannot follow them and clearly identify which side is which, the tool operates in Limited mode, and no commands on the Status bar are available. This functionality is equivalent to how the tool worked in previous versions.

There are a few considerations to keep in mind when using this tool:

- Only lines and arcs are extended. Beziers cannot be extended because it takes more information than is available to define their curves precisely.
- Items are trimmed to meet the *trim against* selection.
- The **Reverse trim** checkbox is available only when an item to be trimmed or extended crosses the *trim against* selection.
- If an item cannot be trimmed or extended to meet the *trim against* selection, it is not deleted.
- Right-clicking the mouse stops the tool and resets the current selection.

To use the **Trim Against Selection** tool in Limited mode, do the following:

1.  Use the **Select** tool to select the lines, arcs, or beziers to trim against.
2.  Click **Trim Against Selection**. The message in the Status bar should indicate that the *trim against* lines do not form one continuous path.
3. Click the object to trim on the part that you want to keep.
4. ArtiosCAD trims the object against the trim selection.

If you have selected a two-sided path of lines, arcs, or beziers, when you start the **Trim Against Selection** tool, it starts in Full mode with the controls on the Status bar enabled.

The **Offset**: field lets you specify a distance from the *trim against* lines that the lines will be trimmed back or extended past instead of being trimmed or extended flush with the *trim against* lines. When you specify an Offset, a red path appears at the designated distance next to the trim-against lines indicating where items will be trimmed or extended.

Reverse trim toggles the side of the selected the items to trim or extend. The parts to be kept are highlighted magenta, while the parts to be trimmed are turned grey.

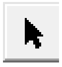
Note:

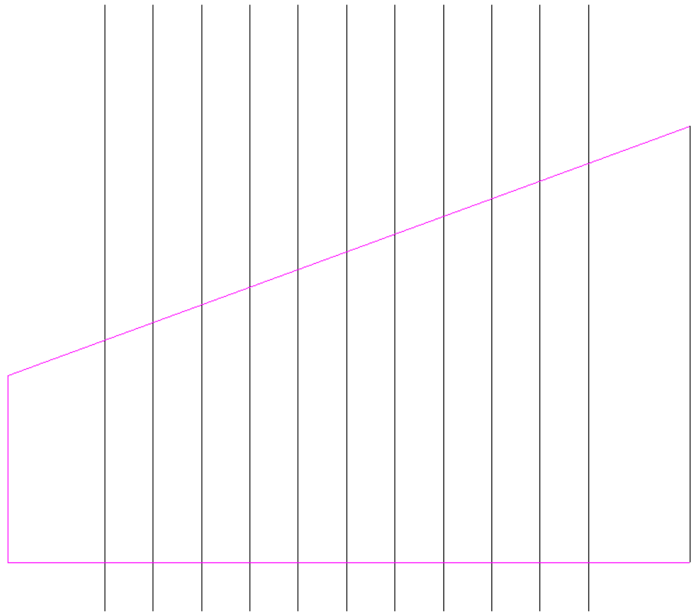
Reversing the trim does NOT change the offset, however! The selected items still meet the same offset line. You must change the side on which you are selecting the items in order to change the offset line they meet.

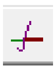
Trim performs the trim or extend.

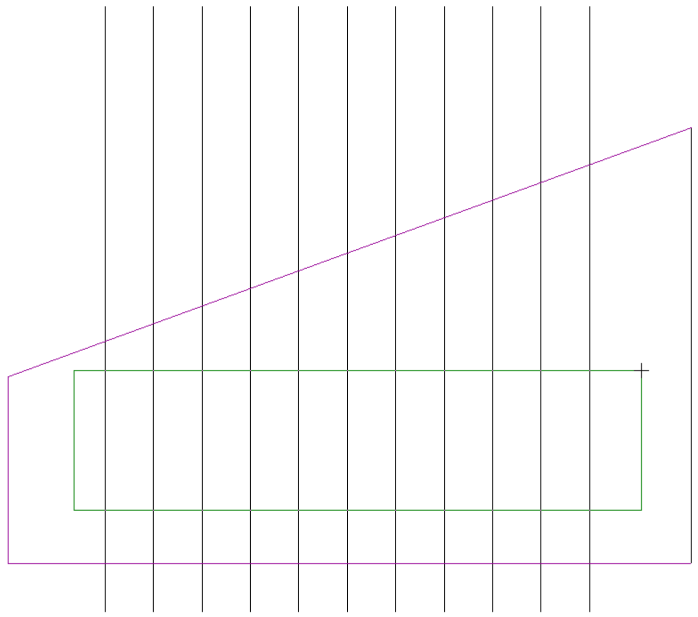
When you start the tool, select the items to be trimmed or extended. Select multiple items by holding down `SHIFT` while clicking, or you can make a window selection by clicking in empty space and dragging the opposite corner of the rectangle; everything touching a border of the rectangle is selected. Where you select objects determines which parts are kept. Where you release the mouse button determines the position for all the selected items.

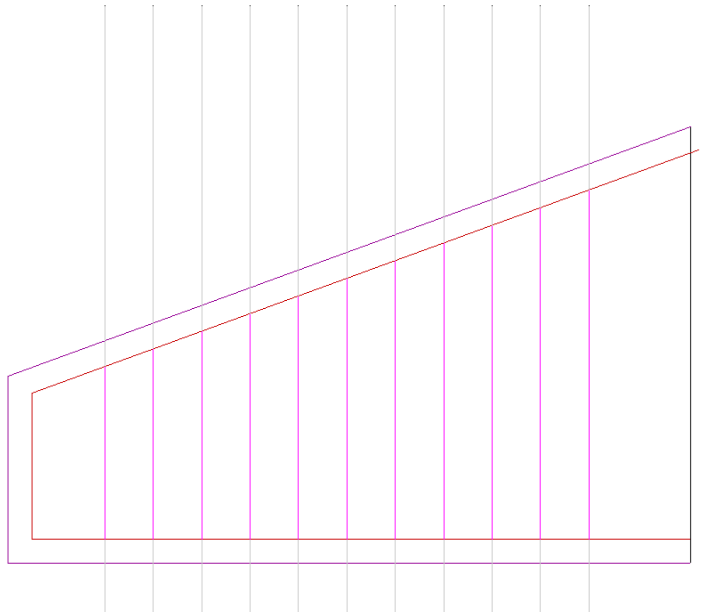
Shown below is an example of trimming lines inside a polygon with an offset.

1.  Use the **Select** tool to select the trim-against lines. These lines are connected and form a two-sided path, so the **Trim Against Selection** tool can use Full mode.

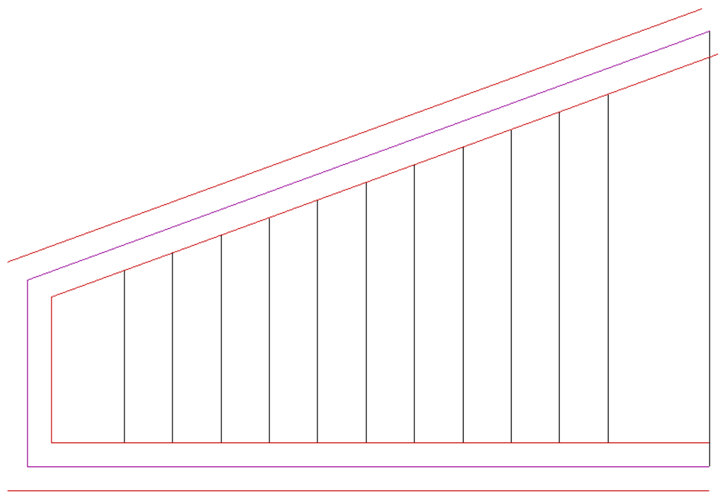


2.  Click **Trim Against Selection**.
3. Set an offset of 5 mm in the **Offset**: field on the Status bar.
4. Use a window selection to select the lines inside the polygon. Once the lines are selected, the red offset border appears, the lines to be kept turn magenta, and the lines to be trimmed turn grey.

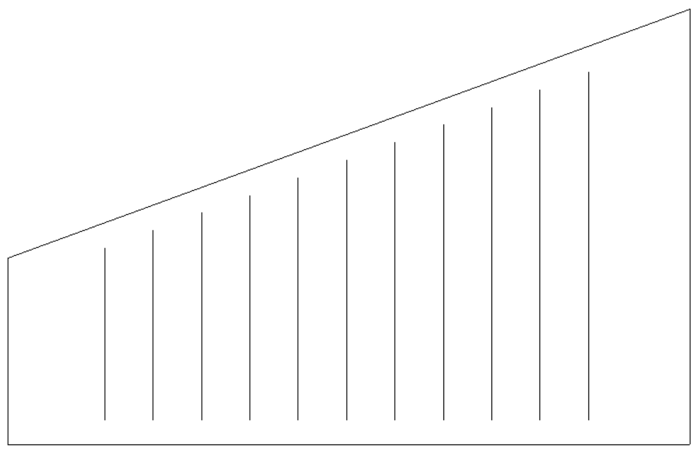




5. Click **Trim** to trim the lines.

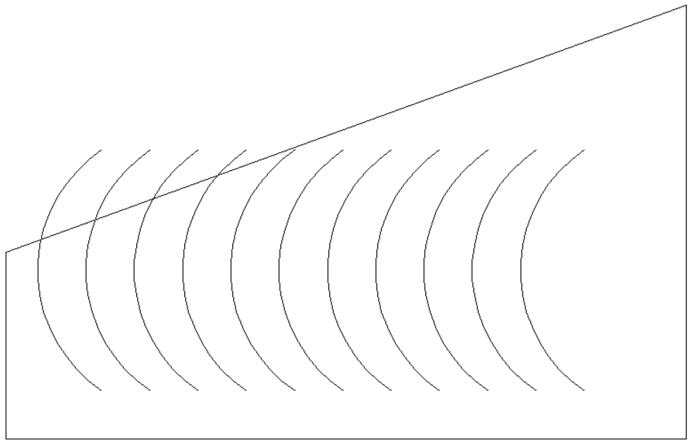


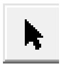
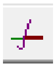
6. The tool remains active. Right-click the mouse to end it and see the result of trimming the lines.

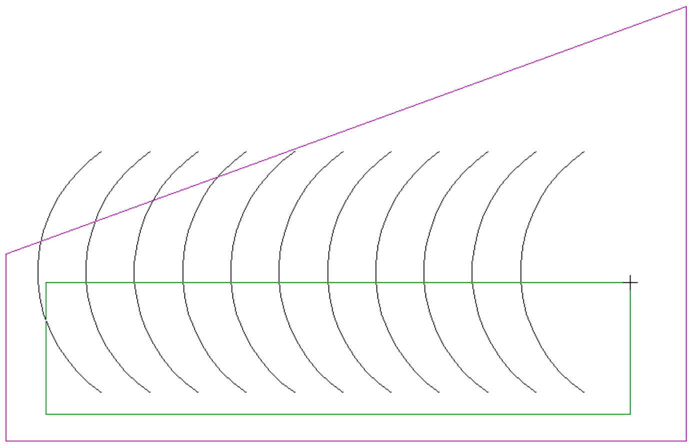


Trim Against Selection, in Full mode, can also extend lines and arcs as well as trimming lines, arcs, and beziers. Shown below is an example of how it can trim and extend arcs at the same time.

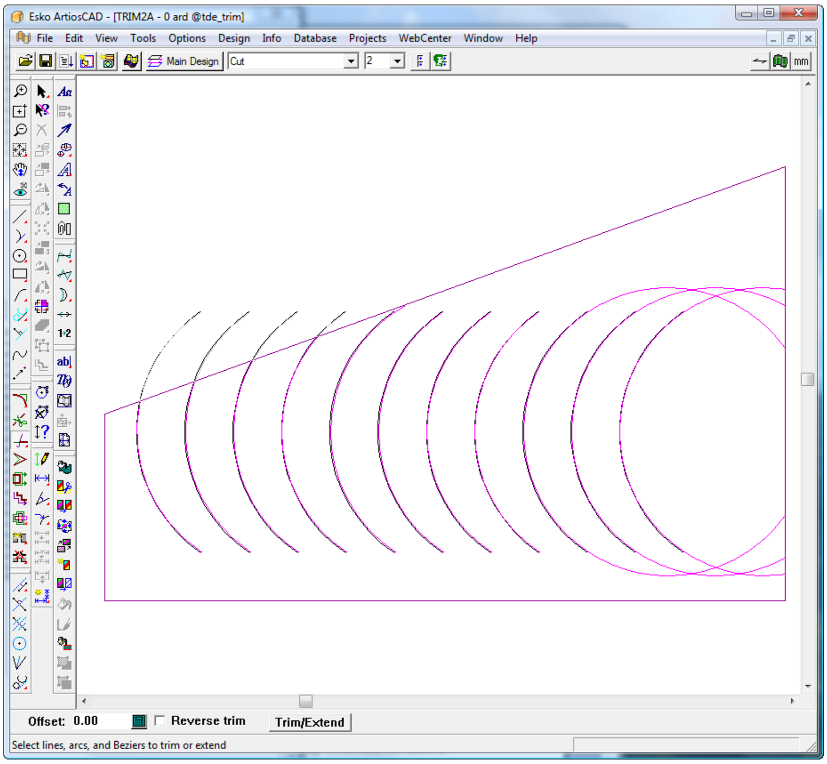
1. Shown below is the geometry before it is changed.



2.  Use the **Select** tool to select the lines forming the polygon to be the *trim-against* lines.
3.  Click **Trim Against Selection**.
4. Use a window to select all the arcs inside the polygon.

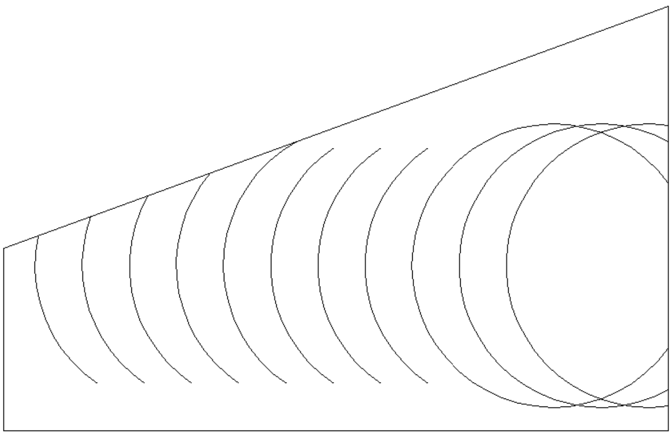


5. The parts to be trimmed turn grey, and the parts to be extended turn magenta. Note how some arcs stay the same; they are neither trimmed nor extended because before trimming, they do not touch one of the *trim-against* lines, and after extending, they still would not touch one of the *trim-against* lines.



6. Click **Trim/Extend** to trim and extend the arcs.

7. Right-click the mouse to end the tool and see the results.



 The fifth tool on the Trim/Extend flyout toolbar is the **Extend Line** tool. This tool extends the current line or arc by a given distance. Specify a negative length in order to trim the line or arc.

The following options are on the Status bar when this tool is active:

The **X|Y** option button extends the line or arc in either the X or Y axis depending on whether the geometry is closer to being horizontal or closer to being vertical.

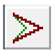
Length extends the object by the length in the **Extend distance:** field.

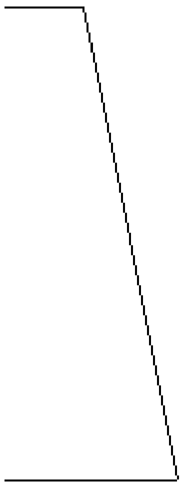
To use this tool, do the following:

1.  Click the **Extend Line** tool.
2. On the Status bar, choose either the **X|Y** option or the **Length** option.
3. Enter a value in the **Extend distance:** field. (Enter a negative value to perform a trim.)
4. Click near the end of the item to change.
5. The item is extended or trimmed.

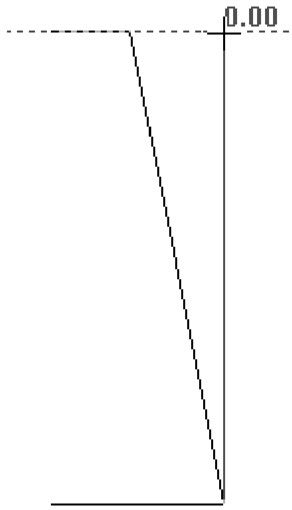
Note: ArtiosCAD reduces the number of bridges on the object as needed if they do not fit. If you specify a larger negative value than the length or radius of the line or arc, ArtiosCAD creates a zero-length line or arc. You cannot extend a full circle by a positive value. Extending an arc with by a large positive value only extends it enough to form a full circle.

Stretch point

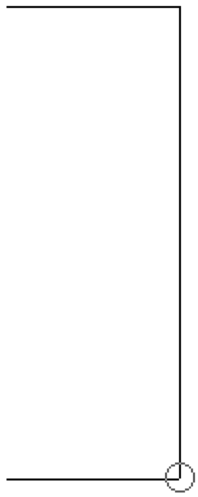
 The **Stretch Point** tool on the Adjust toolbar moves a point and any lines, arcs, or beziers that use that point as an endpoint. It can also adjust the end of a line when you activate the tool and click close to the end of the line. Turning on End points in the View Mode dialog box makes this tool easier to use.



After selecting a point to stretch, ArtiosCAD prompts for the angle at which to move the point.

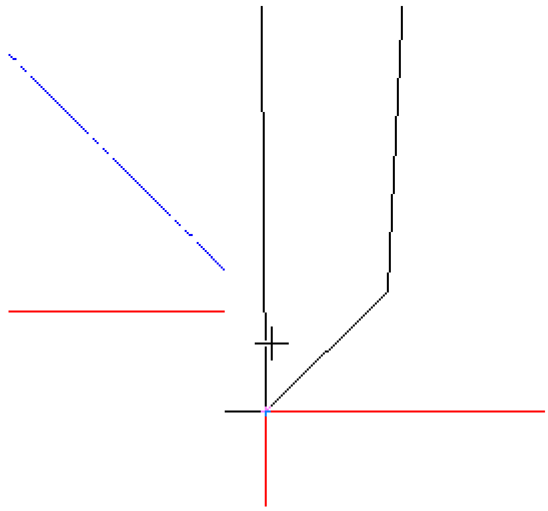


Then ArtiosCAD prompts for the distance to move the point. Use the pointer to align the drag with the bottom point. The completed view of the moved point is shown below.

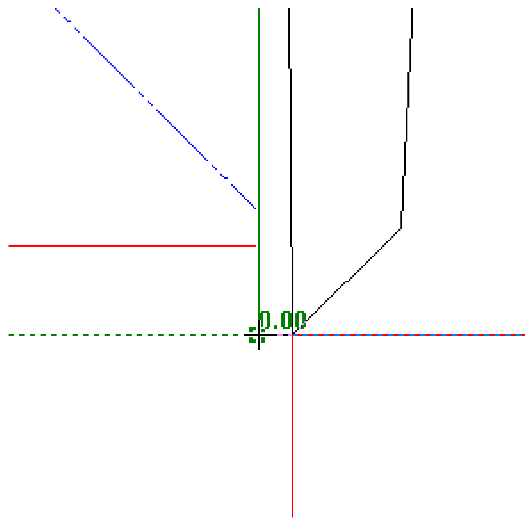


To move the end of a line, use the following workflow:

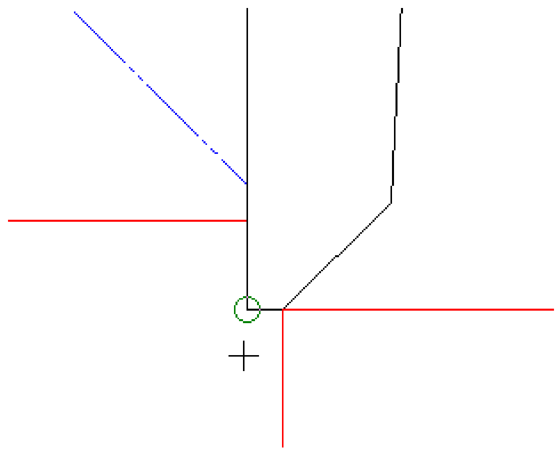
1. Activate the **Stretch Point** tool and click close to the end of a line, but outside the snap tolerance and not the endpoint itself (else the point will be stretched).



2. Drag the end of the line to its new location.



3. Click to set the put-down point.



Stretch by Polygon

 The fifth button on the Adjust toolbar activates the **Stretch by Polygon** tool, which moves lines while keeping them attached to connected lines, and is also used to move connected endpoints. Stretch by Polygon works with lines, arcs, beziers, arrows, and distance dimensions (both temporary and permanent). Use this tool by activating it, defining an area to stretch with a polygon, selecting the point to stretch from (the reference point), selecting the point to use as a handle while stretching, and then indicating the put-down point. The tool works by moving the lines or connected endpoints in the direction you set, given the requirements to preserve angles or radii and the requirement to keep common endpoints.

The stretch behaves differently depending on the option chosen in the drop-down list box on the Status bar. The reference point and the pick-up point can be the same.

If one end of a line, arc, bezier, arrow, or dimension is selected, ArtiosCAD moves the selected end and not the other. If both ends of an object are fully contained within the selection window, but its endpoints are not connected to any other selection, the object is moved in the direction you specify.

The **Distance**, **Angle** and **Radius** options in the drop-down list box on the Status bar modify the way ArtiosCAD stretches lines, arcs and arrows that cross the selection window or polygon. **Distance** preserves the distance between objects that are fully within the selection window or polygon. **Angle** preserves the angle of lines and arcs that cross the selection window. **Radius** preserves the angles of lines and the radii of arcs that cross the selection window or polygon. ArtiosCAD updates other lines fully inside the selection window or polygon that connect to these as needed.

Note: ArtiosCAD does not maintain angles when one end of a line is selected and this line is perpendicular to the stretch direction, because preserving the angle is not possible in this situation.

Note:

ArtiosCAD does not update radius and angle dimensions changed by this tool.

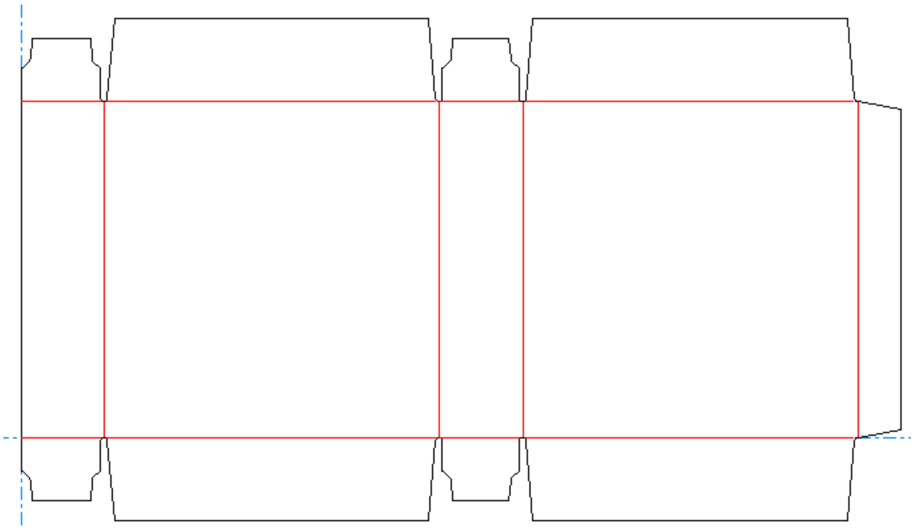
After the tool is activated, click in many locations to make a custom-shaped polygon to define the area to stretch, or hold down the mouse button and drag a rectangular polygon.

ArtiosCAD stretches dimensions if one end or an extension line is within the selection window or polygon and updates their values. Text is moved but not stretched.

This tool does not affect construction lines.

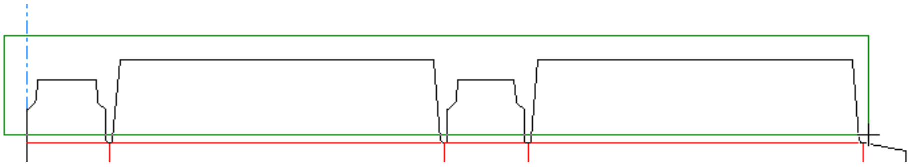
The default preservation option (Distance, Angle, or Radius) is set in **Options** > **Defaults** > **Startup defaults** > **Stretch by polygon options**.

Shown below is a seal-end container that is not resizable. This tool can change the size of the flaps while preserving their angles.

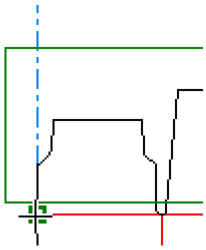


To change the size of the flaps, do the following:

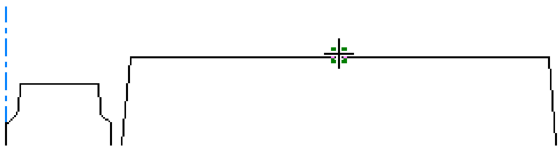
1.  Click **Stretch by Polygon**.
2. Click at the upper left of the design and drag a window to the lower right corner of the flap at the right end, making sure to not select any of the creases.



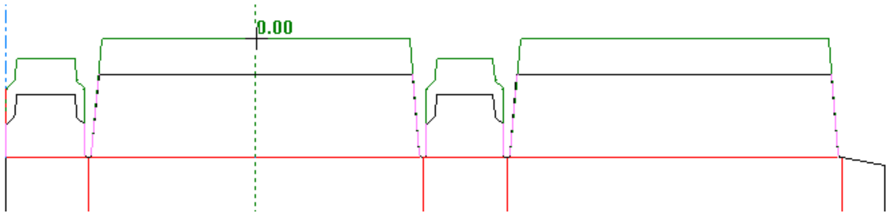
3. Select **Angle** in the drop-down list box on the Status bar.
4. Click the reference point. This is the point used as the anchor for the stretch. In this example, the bottom left corner of the leftmost flap is the reference point.



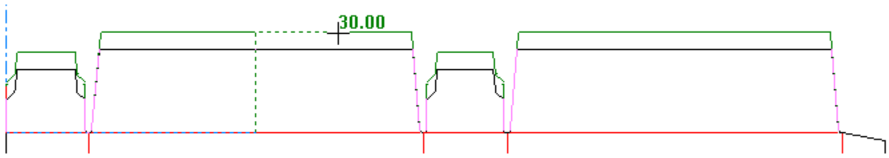
5. Click the handle point. This is the point that you pick up to stretch.



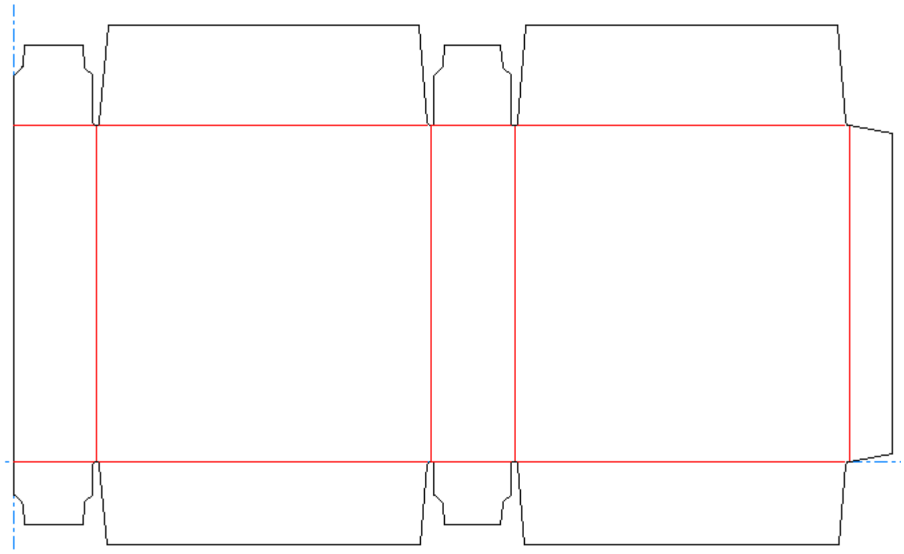
6. Click to set the angle of the stretch.



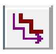
7. Click to set the distance of the stretch.



8. ArtiosCAD stretches the lines and preserves their angles. Note how the top flaps are larger than the bottom flaps.

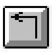


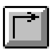
Follow

 The sixth button on the Adjust toolbar activates the **Follow** tool. This tool is used to trace existing geometry and copy its outline. It is useful when making bleed outlines or when something needs to be reduced or enlarged by a specific distance rather than a scale.

The **Follow** tool works by prompting for a start point. It then prompts for more points along the path to be followed. The points do not need to be on the same line; the Follow tool will turn corners to reach the point you indicate.

The first four buttons on the Status bar control how **Follow** reaches points.

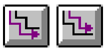
 The first button, **Follow Left**, has the tool turn left at every intersection to reach the point indicated.

 The second button, **Follow Right**, has the tool turn right at every intersection.

 The third button, **Direct Path to Point**, sends the tool directly to the point indicated

 The fourth button, **Arc Path to Point**, makes an arc from the current position through the through point you indicate to the end point you indicate.

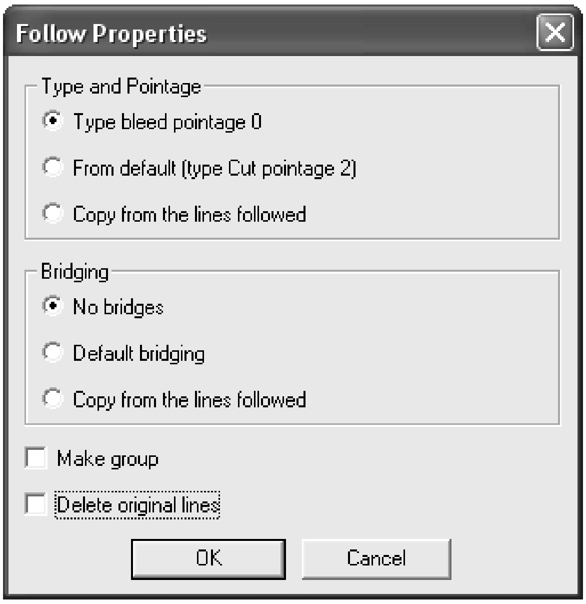
The **Offset**: field is used to specify the distance away from the original lines the path will be placed when complete.

 The fifth and sixth buttons control the offset of the new lines from the lines being followed. If you are following in a counter-clockwise (anti-clockwise) direction, the first button offsets them to the left or outside of the original lines (a negative offset) and the second button offsets them to the right or inside of the original lines (a positive offset). If you are following in a clockwise direction, the first button does a positive offset and the second button does a negative offset.

When selected, the **Round Corner** checkbox uses blends to connect lines instead of them meeting end-to-end.

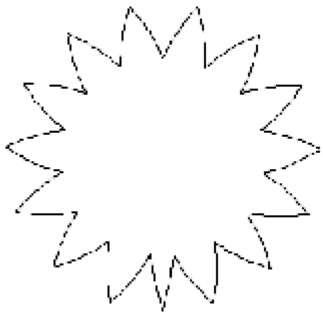
 The sixth button deletes the last path made.

OK accepts the path and leads to the Follow Properties dialog box where the properties for the new lines are set. Make the desired selections and click **OK** to make the new lines.



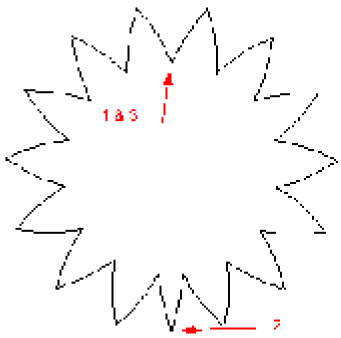
Example of using the Follow tool

The Follow tool concepts may be difficult for new users to grasp. This handy example is meant to show a particular situation in which the Follow tool makes life easier. The shape shown below needs to be made larger by 12 millimeters all around.

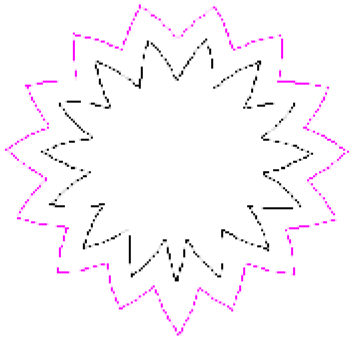


To make the star cutout larger, do the following:

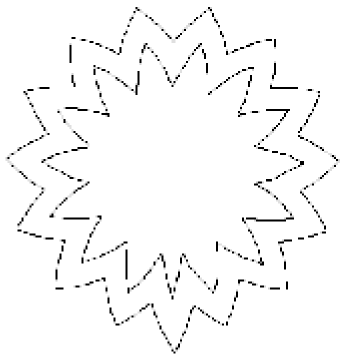
1. Click the **Follow** tool on the Adjust toolbar. The **Follow Left** button is on by default.
2. Indicate the start point (Point 1), the next point along the path (Point 2), and the end point for the path (Point 3).



3. Make sure the system is in metric mode by checking the state of the **Units** button on the View bar at the top of the window. Enter `12` in the **Offset** field and click the **Negative Offset** button. A magenta outline will appear of the followed shape.

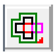


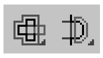
4. In the Follow Properties dialog box, set the **Type** and **Pointage** and **Bridging** options to **Copy from the lines followed**. Click **OK**. The cutout should now look like the picture below.



5. Delete the inner lines and the operation is complete.

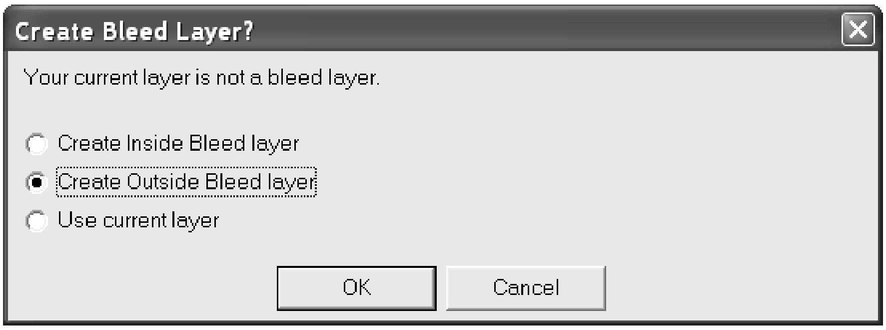
Bleed and Coating tools

 The seventh button on the Adjust toolbar activates the **Bleed** tool, and when held down, activates the Bleed/Coating flyout toolbar

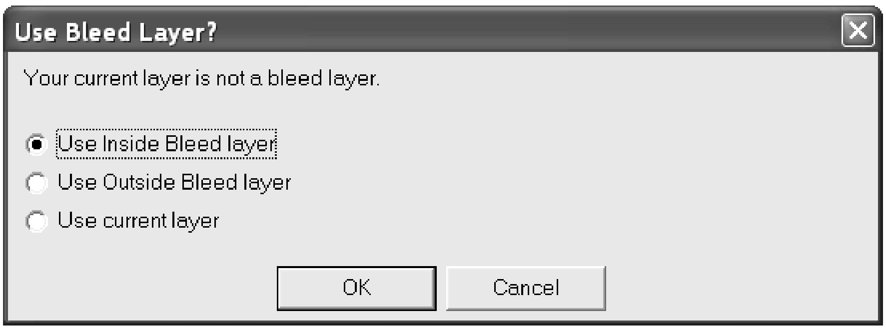


Bleed tool

Bleeds are used to define the area of ink bleed when the sheet moves through the printing press. To use the Bleed tool, create the design that will have a Bleed outline, and click the **Bleed** tool. If you are not in a Bleed Outline layer and there are none already created, you will be asked if you wish to create one.

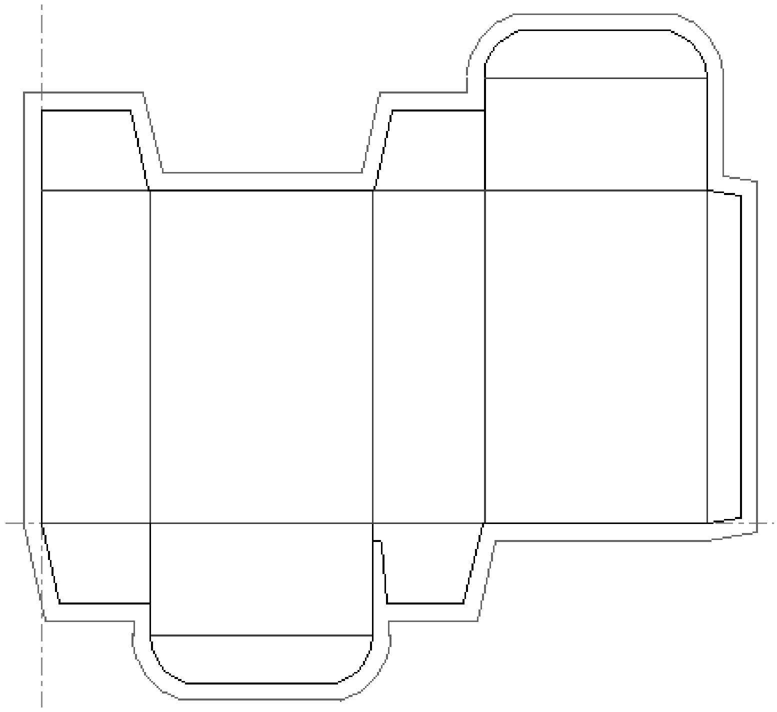


If you are not in a bleed layer and one already exists, you will be asked which layer to use:

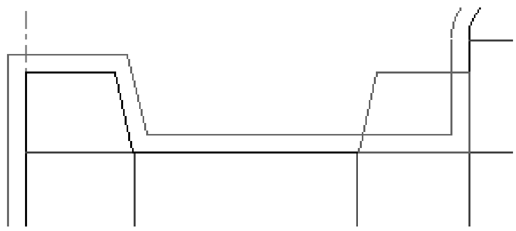


Choose the layer to create, or choose the current layer, and click **OK**. The inside or outside layer type option button is initially set according to the current side of the design.

A default bleed is automatically created, but you can modify it as desired as described below. The final bleed is not created until you click **OK** on the Status bar.

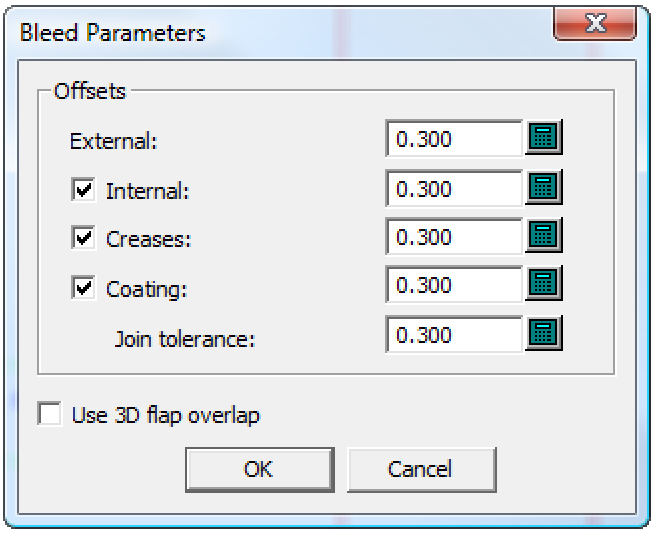


Click inside any panel to exclude it from the bleed. The right flap below is excluded from the bleed.



To change the distance between the bleed and the perimeter of the design, change the value in the **Offset** field on the Status bar.

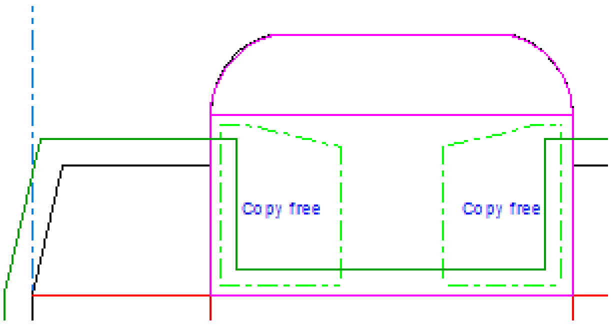
Clicking More Options (**...**) opens the Bleed Parameters dialog box.



The checkboxes to the left of the optional offsets enable or disable them. If you enable the internal offset, the Status bar changes to have **External** and **Internal** fields.

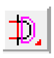
If you enable a separate coating offset, you can also set the **Join tolerance** to control the smoothness of how the coating lines join the bleed lines.

Use 3D flap overlap uses the flap information from **Flap Priority** and **Update 2D** in the 3D module to determine the bleed outline.

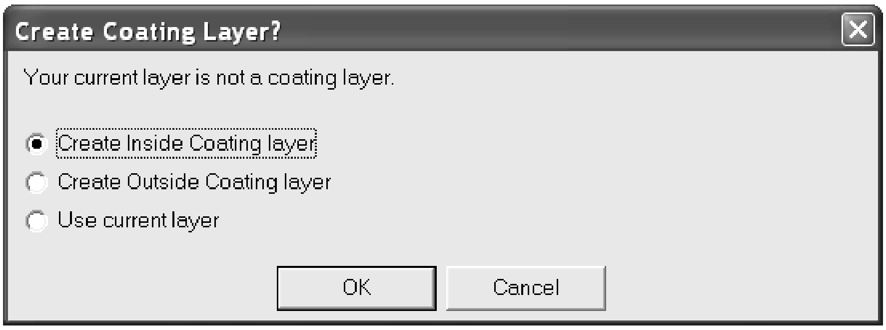


 Click the **Undo** button to undo a bleed action. Click **OK** to finish using the tool and create the final bleed.

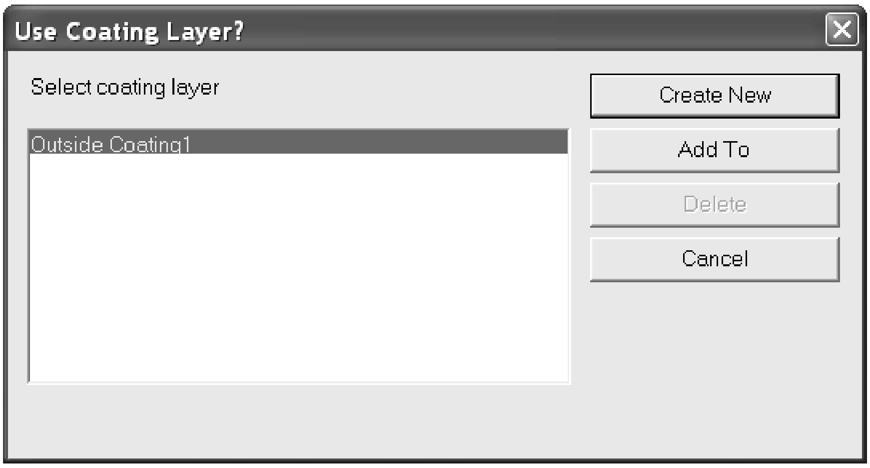
Coating Free Area tool

 The second tool on the Bleed/Coating flyout toolbar is the **Coating Free Area** tool. The **Coating Free Area** tool defines panels on the box that will not receive a coating as the sheet passes through the printing press.

As with the **Bleed** tool, when you click the **Coating Free Area** tool, if you are not in a layer of class Coating, ArtiosCAD asks the kind to create, or to use the current one.



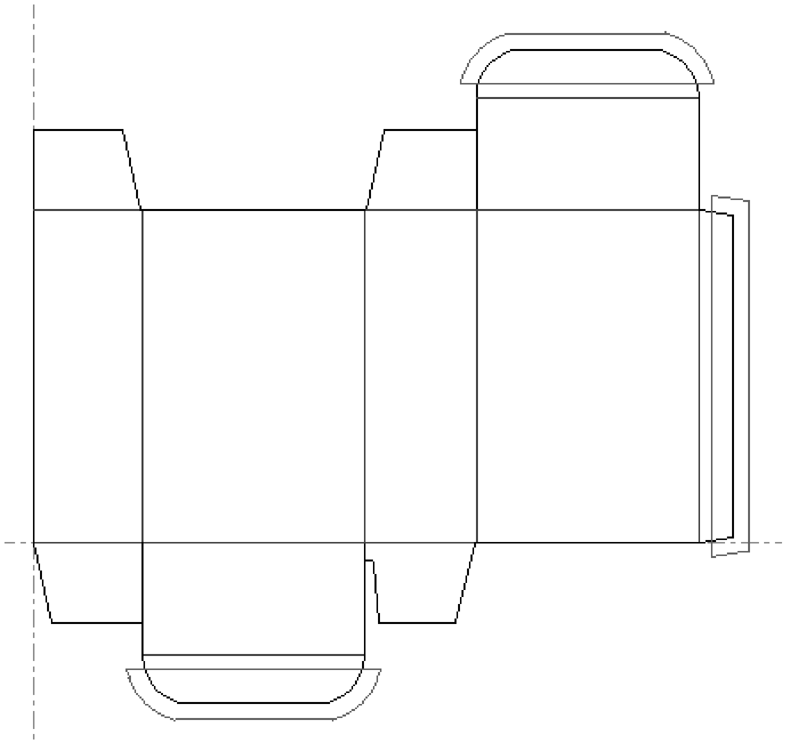
If a layer of class Coating already exists, the same layer-creation questions and dialog boxes appear as for the Dimension tools. You are asked to add to an existing layer, or to create a new one. Select an already-existing layer and click **Add To**, or click **Create New** to create a new coating layer.



Once the layer management is decided, the tool activates. Click the panels of the box to be coating-free. The coating free area outlines will appear using the tolerances set in the fields on the Status bar. The **External** field controls how far the outline extends from the perimeter of the design, while the **Internal** field controls the distance from the outline to any internal lines in the panel.

 Click the **Undo** button to undo a varnish action. Click **OK** to finish using the tool.

Shown below is a picture of panels which are coating free areas.



Notes on the Bleed and Coating tools

You must click **OK** on the status bar when done indicating what to create. If you change the window focus or select another tool, the tools are cancelled.

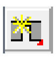
The tools only account for manufacturing line types in the Main Design, Manufacturing, and Windows and Cutouts layers. Line types such as print image and annotation are ignored when using these tools.

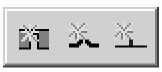
If you make a mistakenly select a panel, click the **Undo** button on the Status bar or restart the tool.

Defaults for these tools are contained in the single design parameter sets.

When you are in an inappropriate layer and activate the Coating Free Area tool, ArtiosCAD only displays option buttons for the first instance of each class of coating layer. For example, if you have two **Outside Coating1** layers in a print item, and activate the Coating Free Area tool in the Main Design layer, ArtiosCAD prompts you to choose a coating layer. It displays only the first coating layer in each class, so if you chose the **Outside Coating1** layer, it would distribute items to the first instance of the layer. To work in the second **Outside Coating1** layer, change to that layer before activating the tool so that you are not prompted to choose a coating layer.

Add Bridge tool

 The eighth tool on the Adjust toolbar activates the **Add Bridge** tool, and when held down, activates the Add Bridge Tools flyout toolbar.

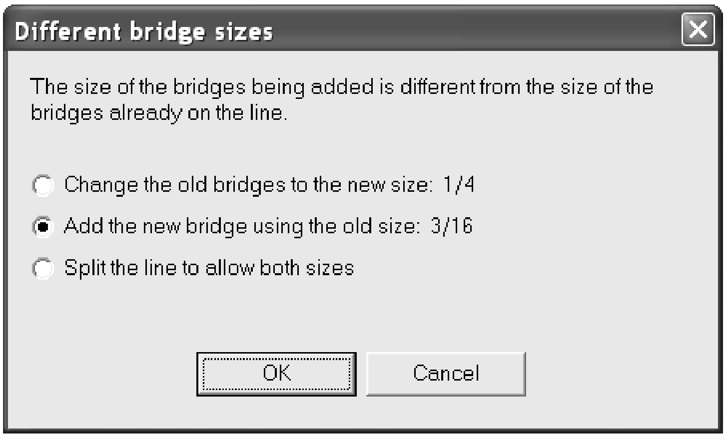


ArtiosCAD automatically turns on the display of bridges in the View Mode when the **Add Bridges** tool is activated. Bridges are also on the right-click context menu in Single Design.

To add a bridge to a line, activate the **Add Bridge** tool, set the width of the bridge in the **Bridge Width**: field in the Status bar, and click the location of the center point of the new bridge.

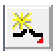
The value entered in the **Bridge Width**: field will be the default value for the current session of ArtiosCAD.

If you try to add a bridge that is a different size than other bridges on the line, the Different Bridge Sizes dialog box appears. In it, you can choose to change the old bridges to the new size, to add a new bridge using the old size, or to split the line to allow both sizes. Choose the desired option and click **OK** to add the bridge, or **Cancel** to return to Designer without adding the bridge.



Note: If you change the nick size to a bigger size, some bridges may be removed to avoid overlapping.

Add Tack Bridge tool

 The second button on the Add Bridge Tools flyout toolbar activates the **Add Tack Bridge** tool.

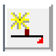
To use this tool to add a tack bridge to a wood edge line type, simply activate it and click the position for the new tack bridge.

If size of the tack bridge differs from tack bridges already on the line, a warning dialog box will appear. Choose the desired option and click **OK** to change the size of the tack bridges.

To change the tack bridge formula, click **Options** > **Change Tack Bridge formula** and select the new formula.

When a tack bridge is viewed in a Preview for an output that cuts or mills, the allowance for the tool diameter will make it appear larger than it would for a laser output.

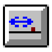
Add Nick tool

 The third button on the Add Bridge Tools flyout toolbar activates the **Add Nick** tool. This tool places a nick on the line that you select after activating the tool. By default, nicks appear as short lines perpendicular to the selected line, but this can be configured in Defaults.

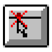
When active, this tool has configuration options on the Status bar as shown below.

The **Between nicks**: field appears only when the value in the **Number**: field is 2 or greater.

The five buttons on the status bar control the placement of the nick.

 The first button offsets the nick by the distance specified in the **Offset** field from the start or end of a line depending on which is closer to where you click.

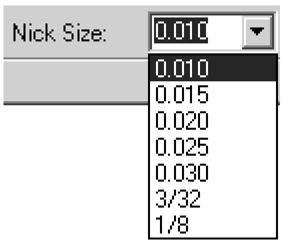
 The second button places the nick at the center of the line clicked.

 The third button places the nick anywhere on the line you indicate. If you click within the snap tolerance of a point such as an intersection with a construction line, the nick snaps to the intersection.

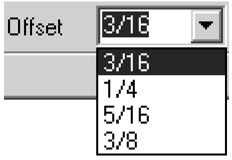
 The fourth button fills the line with nicks. Each nick is separated from the others by the distance in the **Spacing**: field.

 The fifth button places the nick at the end of a line.

The **Nick size** control sets the size of the nick. Either specify a value in the field or select a value from the drop-down list box.

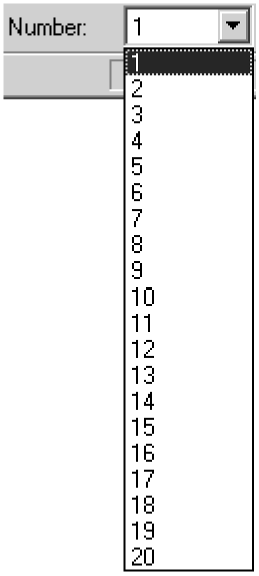


The **Offset** control sets the distance used in conjunction with the Offset placement method.

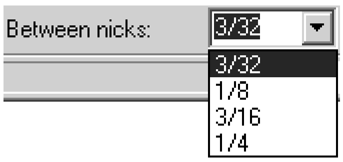


This control changes to the **Spacing** control when the Fill Line placement method is selected.

The **Number** control sets the number of nicks added when you click. A line may have up to 20 nicks. Either type the desired number of nicks in the field or select the value from the drop-down list box.



When the Number control is set to more than 1, the **Between nicks** control appears. This control sets the equidistant spacing between nicks. Either type the desired distance in the field or select the value from the drop-down list box.

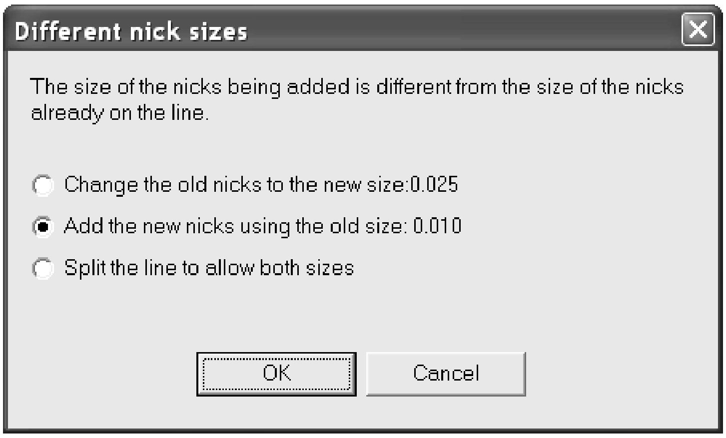


Nicks may move if they are added before removing double knives.

When making a multiple selection, the Properties page for that selection will now show nick controls if any of the items selected have nicks. If you change nicking options, only those items already having nicks change.

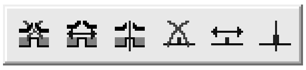
Note: When using the default nick size, if you change the board in the Board Information dialog box, and thus change the caliper, actual nick sizes are not updated unless the design is rebuilt.

As with bridges, if you try to add nicks to a line of a different size than nicks already along the line, the Different Nick Sizes dialog box appears. Choose the desired option and click **OK** to add the nicks, or click **Cancel** to return to Designer.



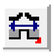
Delete Bridge tool

 The ninth button on the Adjust toolbar activates the **Delete Bridge** tool when clicked, and when held down, activates the Change Bridging and Nicking Tools flyout toolbar. When this tool is clicked, bridges are turned on in the View Mode if they are not on already.

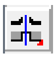


To delete a bridge, click this tool, and then click the bridge to delete.

Move Bridge tool

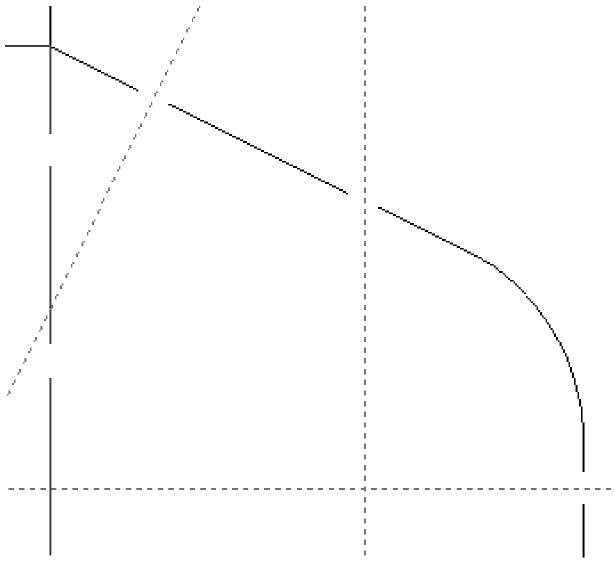
 The second tool on the Change Bridging And Nicking Tools flyout toolbar activates the **Move Bridge** tool. To move a bridge, click this tool, click the bridge to move, and click its new position on the line. The bridge will be moved so that it is centered on the position you select, and it will snap to a point if the click is within the snap tolerance.

Conline Through Bridge tool

 The third button on the Change Bridging And Nicking Tools flyout toolbar activates the **Conline Through Bridge** tool.This tool creates a construction line which passes through the center of a bridge. To use this tool, click it, choose one of the placement options on the Status bar, and then click the bridge.

The placement options are:

- **Perpendicular to the line**, which creates a construction line at 90 degrees to the line, regardless of the orientation of the line;
- **Vertical**, which creates a vertical construction line (parallel to the Y axis of the design);
- **Horizontal**, which creates a horizontal construction line (parallel to the X axis of the design).



Delete Nick tool

The fourth button on the Change Bridging And Nicking Tools flyout toolbar activates the **Delete Nicks** tool. To use this tool, click it, and the click the nick(s) to delete. This tool remains active until another tool is selected.

Delete all nicks on a line at once by clicking **Delete All Nicks** on the Bridging tab of that line's Properties dialog box.

Move Nick tool

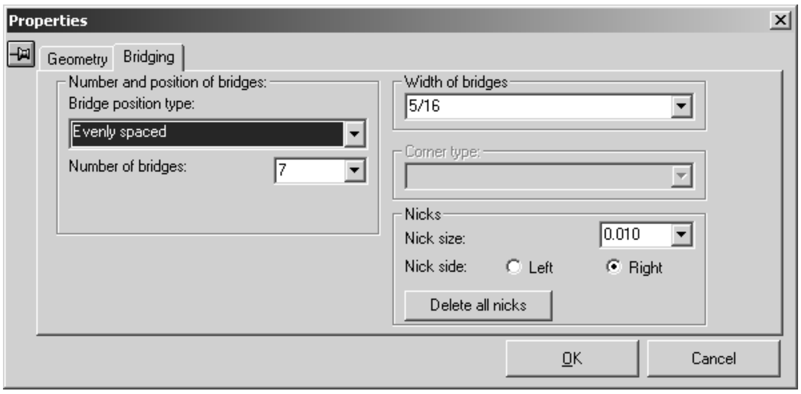
The fifth button on the Change Bridging and Nicking Tools flyout toolbar activates the **Move Nick** tool. To move a nick, click this tool, click the nick to move, and then click the new position along the line for the nick. The nick will snap to its new position if you click within the snap tolerance of a point or an intersection.

Conline Through Nick tool

The **Conline Through Nick** tool is the sixth button on the Change Bridging and Nicking tools flyout toolbar. It places a construction line through a nick either perpendicular to the line containing the nick, vertical through the nick, or horizontal through the nick depending on which option is selected on the Status bar.

Changing nick properties

Once nicks have been added to a line, their size and the side of the line they are on can be modified by accessing the Bridging tab of the Properties dialog box for that line. The **Nicks** group on the Bridging tab appears only when a line contains nicks.

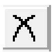


The **Nick size**: drop-down list box lets you choose a new size for all the nicks on the line. The **Left** and **Right** option buttons in the **Nick side**: field set the side of the line the nicks are on. (Check the **Direction** checkbox in the View Mode dialog box, and follow the direction of the line to determine which is the left side and which is the right side.)

Changing physical properties

Changing physical properties of objects means copying, rotating, moving, mirroring them, or making any other change that affects their physical placement. The Edit toolbar and the Adjust toolbar contain most of the commands used to change physical properties. Most tools on the Edit toolbar are disabled until at least one item is selected.

Delete tool

 The **Delete** tool is the third tool on the Edit toolbar. Use this tool to remove unwanted elements from your design. There must be a current selection for this tool to be available.

When possible, use **Undo** rather than **Delete**.

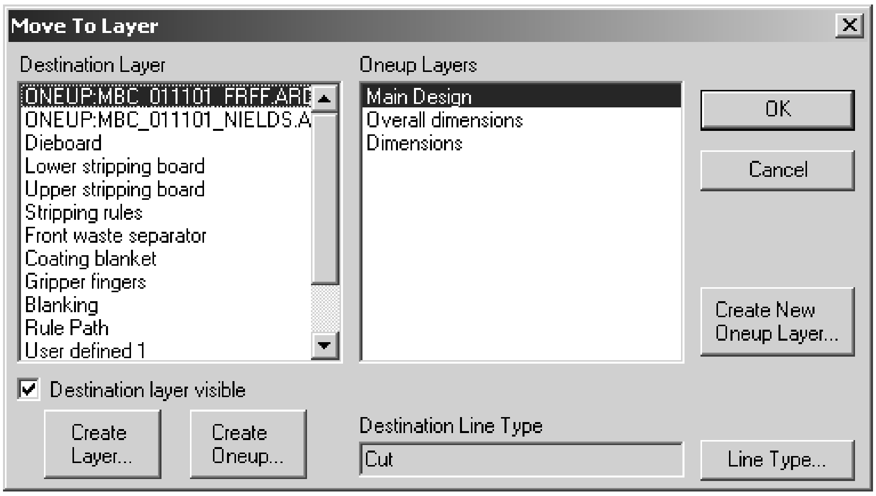
If you delete something that you should not have deleted, use **Undo** to reverse the deletion.

Move To Layer tool

 The fourth button on the Edit toolbar activates the **Move To Layer** tool. At least one object must be selected for this tool to be available.

This tool has two modes of operation - one for Single Design, and another for Manufacturing. For the Manufacturing version of this tool, see the *Manufacturing* chapter.

To use this tool in Single Design, select at least one object, then click the tool. The Move To Layer dialog box will appear.

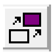


The existing unlocked layers of the design are shown in the Destination Layer list. If the desired layer is locked, cancel the tool, unlock the layer, and then reactive the tool. If the layer into which you want to move the item does not yet exist, click **Create**, and the Create Layer dialog box will appear. Enter the name of the new layer, choose its class, and click **OK** to return to the Move To Layer dialog box.

To change the line type of the object(s) being moved, click **Line Type** and choose the new line type. You may select any line type, but it will be checked for layer appropriateness upon clicking **OK** to return to the Move to Layer dialog box. Click **OK** to return to the Move To Layer dialog box.

Once you have chosen the destination layer and have optionally changed the line type, click **OK** to move the object(s) into the destination layer, or click **Cancel** to cancel the operation. When you click **OK**, ArtiosCAD checks the layer appropriateness and warns if necessary, performs the move, and makes the new layer the active layer.

Move tools

 The fifth button on the Edit toolbar activates the **Move** tool.

Use this tool to move the current selection to a different location. There must be a current selection for this tool to be available.

When clicked, the Move tool prompts for a pick-up point. This is the handle you want to use to move the current selection – everything in the current selection is moved in reference to the pick-up point. Once a pick-up point is selected, ArtiosCAD prompts for the put-down point. This is the point around which the current selection is arranged identically to the way it was around the pick-up point. Use the fields on the Status bar to choose the drag method.

Rotate tools

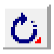
 The sixth button on the Edit toolbar activates the **Rotate Right 90** tool, and when held down, activates the Rotate flyout toolbar. If you click a point in any item while a Rotate tool is active, you can move it using standard Move tool functionality.

 The first button on the Rotate flyout toolbar, and the default Rotate button on the Edit toolbar, activates the **Rotate Right 90** tool. This tool rotates the selected object 90 degrees clockwise around its center. The Rotate Right 90 tool has no prompts; it works instantly.

 The second button on the Rotate flyout toolbar activates the **Rotate Left 90 tool**, which rotates the current selection 90 degrees counterclockwise around its center.

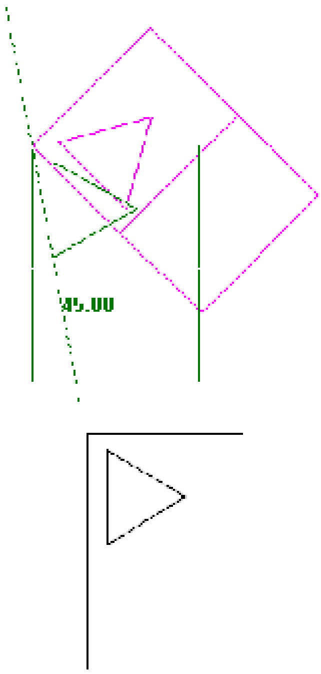
Rotate Left 90 has no prompts; it is instantaneous.

 The third button on the Rotate Tools flyout toolbar activates the **Rotate 180** tool. This tool rotates the selected objects 180 degrees around the horizontal axis.

 The fourth button on the Rotate flyout toolbar activates the **General Rotate** tool. When activated, this tool prompts for a fixed point around which to rotate the current selection and a handle point (where the push is applied to perform the rotation).

Note: The points you choose do not have to be in the object being rotated - they can be any points in any part of the design.

The drag extend line goes through the fixed point and the handle point.



Mirror tools

 The seventh button on the Edit toolbar activates the default **Mirror** tool, and when held down, activates the Mirror flyout toolbar.

There are two types of tools on this flyout toolbar. Those with arrows in the icons perform a move on the selection after mirroring it. Those without the arrows do not perform a move; rather, they mirror the selection about its center without moving it.

To use a tool which does not perform a move, select the item(s) to mirror, and then click one of the tools. The item will be mirrored in-place.

 The first button on the Mirror flyout toolbar activates the **Mirror About Vertical** tool; this tool mirrors the selection about the Y axis and then performs a move.

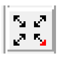
 The second button on the Mirror flyout toolbar activates the **Mirror about Horizontal** tool. This tool mirrors the selection about the X axis and then performs a move.

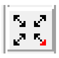
 The third button on the Mirror flyout toolbar activates the **Vertical Mirror About Center** tool. This tool flips the current selection around on the Y axis.

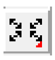
 The fourth button on the Mirror flyout toolbar activates the **Horizontal Mirror About Center** tool. This tool flips the current selection around on the X axis.

 The fifth button on the Mirror flyout toolbar activates the **Mirror About Line** tool. This tool flips the current selection around any line or construction line you indicate.

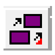
Scale tools

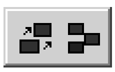
 The eighth button on the Edit toolbar activates the Scale Tools flyout toolbar.

 The first button on the Scale flyout toolbar, and the default Scale tool on the Edit toolbar, activates the **Scale** tool. The Scale tool prompts for a fixed point, prompts for a handle point, and then changes the size of the current selection proportionally in all directions.

 The second button on the Scale flyout toolbar activates the **Differential Scale** tool. The Differential Scale tool prompts for a fixed point and a handle point, but then lets you set the X scale and the Y scale independently.

Copy tools

 The ninth button on the Edit toolbar activates the **Copy** tool, and when held down, activates the Copy Tools flyout toolbar.



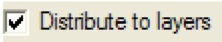
To use the Copy tool, do the following:

1. Select at least one object to copy with a **Select** tool.

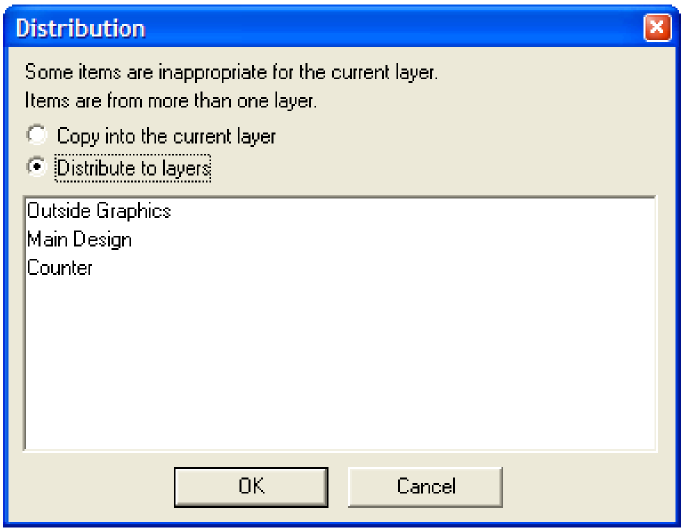
2. Click the **Copy** tool.
3. Click the pick-up point of the copy. This can be any point; it does not have to be part of the object(s) being copied.
4. Click to set the put-down point, or set the values in the fields in the Status bar to the desired angle, offsets, or distances. The copied objects will be oriented the same way in relationship to the put-down point as they were to the pick-up point.

Distribute to Layers

The **Distribute to Layers** checkbox on the Status bar of most of the Copy tools controls whether the copies of the items selected go into the same layers as the originals automatically or if they are placed in the current layer.

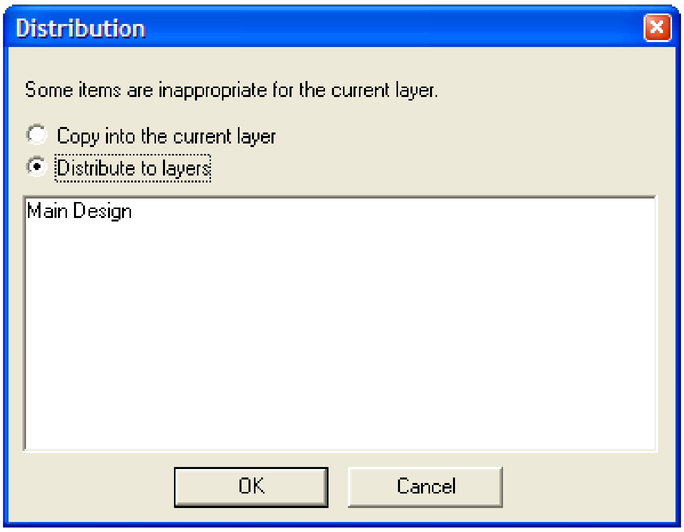


If **Distribute to Layers** is deselected and some of the selected items have inappropriate line types for the current layer, ArtiosCAD prompts you to select the desired action. Make the choice as desired and click **OK** to perform the copy or **Cancel** to cancel it.



If an appropriate layer for the copy does not exist, a new appropriate layer is shown in green with **New** next to its name, and it will be created when you click **OK**. If the appropriate layer exists but is locked, its name is shown in red with **must unlock first** next to its name. You must cancel the copy and unlock the layer in order to use it for the copy.

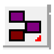
This same dialog box appears when you use **Copy** and **Paste** on the Edit menu.

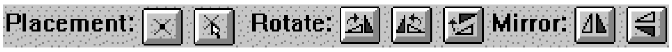


Layer distribution is available only with the tools in Single Design; it is not available with the same tools in Manufacturing.

To set defaults for Distribute to Layers, click **Options** > **Defaults** > **Design defaults** > **Distribute to Layers**. You may set whether the checkbox is selected or deselected by default, and to prompt if there is an inappropriate layer or switch to an appropriate layer automatically. If you choose to automatically switch, ArtiosCAD will use the Annotation layer class if no other is appropriate. If there is more than one appropriate layer, ArtiosCAD will use the one that was created first.

Repeated Copy tool

 The second button on the Copy tools flyout toolbar activates the **Repeated Copy** tool when clicked. This tool lets you make copies of selected objects repeatedly until you activate another tool. This tool is available only when there is a current selection. When this tool is active, seven buttons appear on the Status bar.



The two **Placement** buttons control the put-down points. The first button allows only a point (an intersection between two lines, an endpoint of another line, and so forth) to be used as a put-down point. The second button allows any coordinate to be used as a put-down point.

The first two **Rotate** buttons rotate the copy by 90 degrees - the first 90 degrees to the right; the second, ninety degrees to the left. The third button rotates the copy by 180 degrees.

The two **Mirror** buttons mirror the copy vertically and horizontally, respectively.

There are no drag prompts in the Status bar to aid placing the copied objects. The copied objects can be placed using the mouse only.

Copy Rotate tools

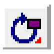
 The tenth tool on the Edit toolbar activates the **Copy Rotate Right 90** tool, and when held down, activates the Copy Rotate tools flyout toolbar.

As with the other Copy tools, there must be a current selection for this tool to be available.

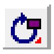
To use this tool, select the item(s) to be copied, and click the tool. The selected items will be copied and rotated 90 degrees to the right using the center of the selected item(s) as the handle point. Clicking the tool again makes another copy, but adds 90 degrees to the angle of the copy. Click the tool once, and one copy is made at 90 degrees; click the tool again, and another copy is made, but this time the copy is at 180 degrees from the original. Clicking the tool a third time results in another copy at 270 degrees, and clicking the tool a fourth time makes a copy at 360 (or 0) degrees.

 The second button on the Copy Rotate tools flyout toolbar activates the **Copy Rotate Left 90** tool. This tool works similarly to the Copy Rotate Right 90 tool, but it rotates left instead of right. As with the Copy Rotate Right 90 tool, successive clicks are cumulative - each copy is rotated by 90 degrees more than the last copy.

 The third button on the Copy Rotate tools flyout toolbar activates the **Copy Rotate 180** tool. This tool works similarly to the Copy Rotate Left 90 and Copy Rotate Right 90 tools, but it rotates 180 degrees.

 The fourth button on the Copy Rotate tools flyout toolbar activates the **Copy General Rotate** tool. As with the other Copy Rotate tools, there must be a current selection for this tool to be available.

To use this tool, do the following:

1. Select the item(s) to copy.
2.  Click the **Copy General Rotate** tool.
3. Click the fixed point around which the item(s) will rotate.
4. Click the handle point. This is where the rotational force is applied.
5. Set the angle for the copy by using drag or by entering a value in the **Angle**: field in the Status bar.

6. Once the angle is set, the copy is made. The copy becomes the current selection.

Copy Mirror tools

 The eleventh button on the Edit toolbar activates the **Copy Mirror about Vertical** tool, and when held down, activates the Copy Mirror tools flyout toolbar. As with the other Copy tools, these tools are available only when there is a current selection.

The **Copy Mirror about Vertical** tool copies the current selection and mirrors the copy about the vertical (Y) axis. It then activates the Move tool with the copy selected so that you can easily move the copy. To use this tool, do the following:

1. Select the item(s) to be copied and mirrored.

2. Click the **Copy Mirror about Vertical** tool. The item(s) will be copied and mirrored but not offset from the original - the mirrored copy will appear on top of the current selection.
3. Click the pick-up point for the copy.
4. Click the put-down point to place the copy.

 The second button on the Copy Mirror tools flyout toolbar activates the **Copy Mirror about Horizontal** tool. This tool works similarly to the Copy Mirror about Vertical tool described previously but mirrors about the horizontal (X) axis instead of the vertical (Y) axis.

 The third button on the Copy Mirror tools flyout toolbar activates the **Vertical Copy Mirror About Center** tool. This tool copy-mirrors the current selection about the vertical (Y) axis passing through its center. It does not perform a move. Distribute to Layers is unavailable when using this tool.

 The fourth button on the Copy Mirror tools flyout toolbar activates the **Horizontal Copy Mirror About Center** tool. This tool copy-mirrors the current selection about the horizontal (X) axis passing through its center. It does not perform a move. Distribute to Layers is unavailable when using this tool.

 The fifth button on the Copy Mirror tools flyout toolbar activates the **Copy Mirror about Line** tool. This tool makes a mirrored copy of the selected items around the line or construction line you select. It does not activate the Move tool afterward. As with the other Copy tools, this tool is available only when there is a current selection. Distribute to Layers is unavailable when using this tool.

To use this tool, do the following:

1. Select the item(s) to be copied and mirrored.
2.  Click the **Copy Mirror About Line** tool.
3. Click the line or construction line to use as the axis around which the copy will be mirrored. The copy is made when you click.
4. The copy becomes the current selection so that other tools may be used on it.

Complete design from half/quarter tool

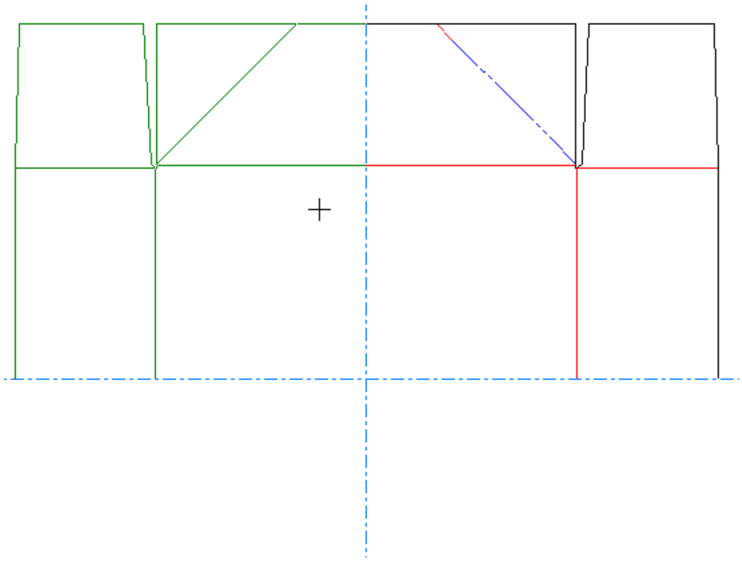
 The twelfth button on the Edit toolbar is the **Complete design from half/quarter** tool. Use this tool to quickly construct a symmetrical design from just a quarter or half of the design. This tool copies the lines and joins them appropriately so that the proper bridging is made

If there is no current selection, the tool automatically selects all the lines in the layer; to only copy selected lines, select them before activating this tool.

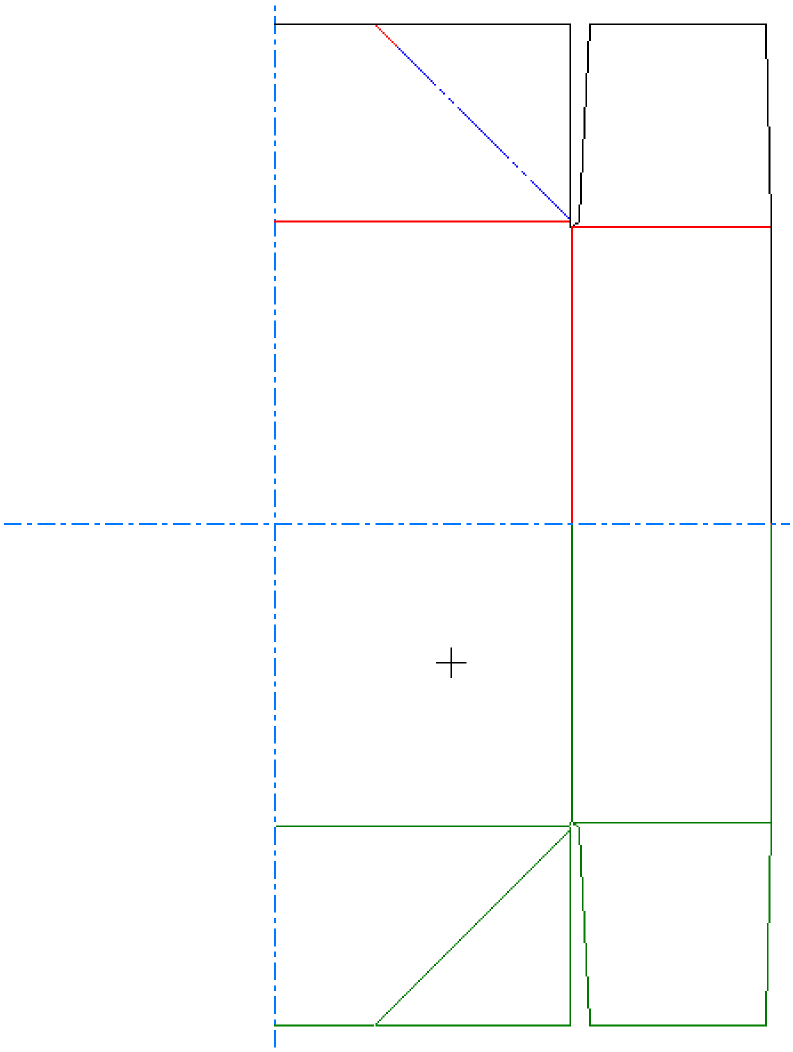
Note: Lines are merged only if they join and can be replaced by a single line. Beziers are not merged. Lines on the mirror axis are not copied.

When this tool is active, three mode selectors are on the Status bar as well as an OK button.

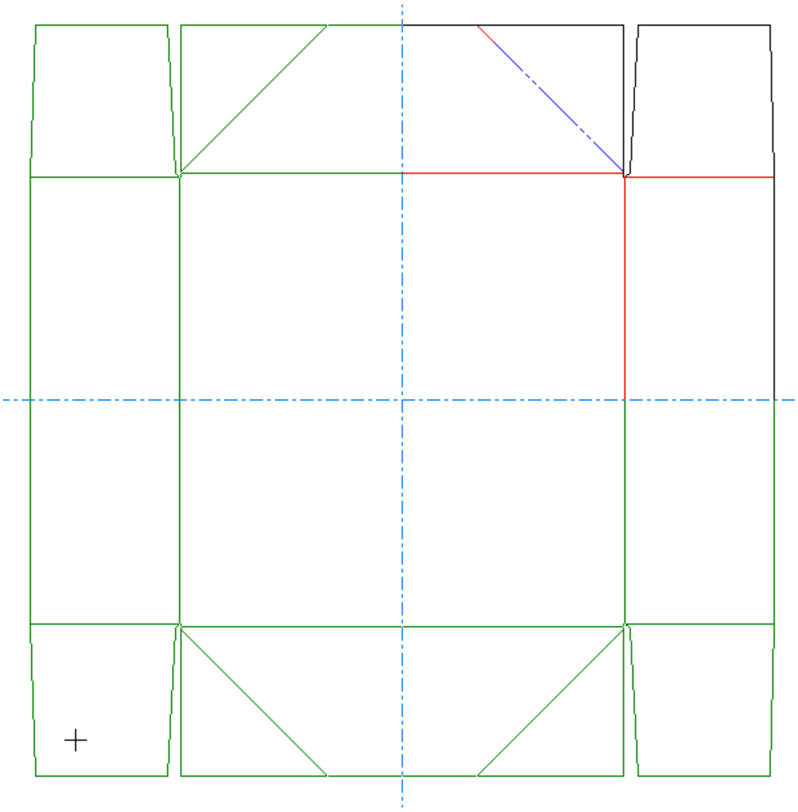
The first mode selector copies the design or selection horizontally.



The second mode selector copies the design or selection vertically.



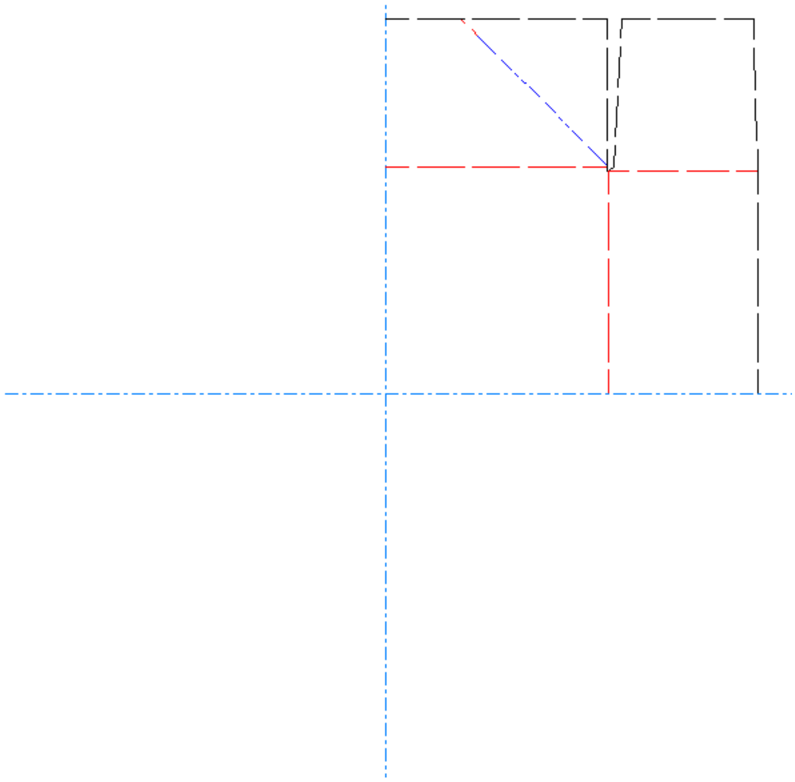
The third mode selector copies the design or selection both horizontally and vertically.



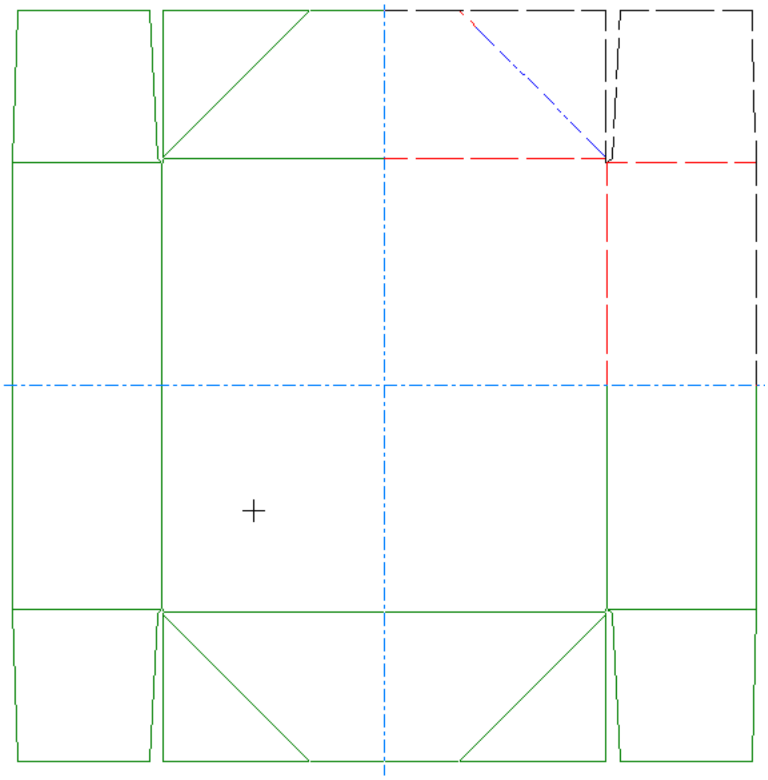
To use this tool, do the following:

1. Design the quarter or half design as you would normally.
2. If you only want to replicate part of the design, use the Select tool to select those lines.
3. Click the **Complete design from half/quarter** tool.
4. The third mode selector is the default, so if you want only a horizontal copy or a vertical copy, choose one of those modes.
5. If the point about which the copy is to be made is not the origin, click that point.
6. The drag shows what the result will be if you click OK.
7. Click **OK** to make the copy and join the lines. The copied lines are distributed into layers as appropriate.

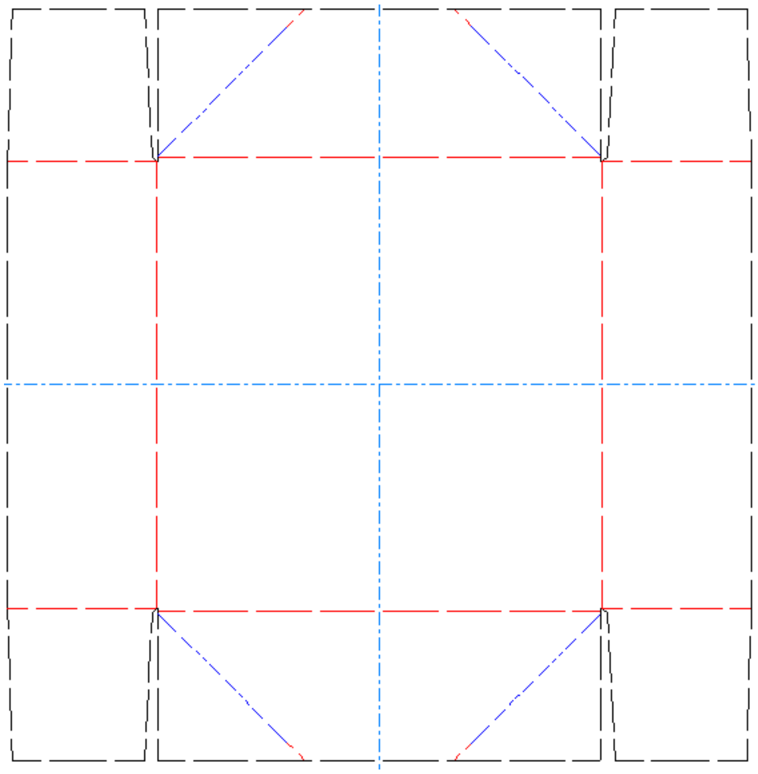
Shown below is an example of using the tool to complete a quarter design. The first step is to design the quarter to be duplicated.



Next, choose the mode selector and drag in the direction of the duplication.

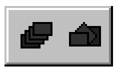


Finally, click **OK** to accept the duplication and join the lines. Note how the bridges span the axes instead of the lines butting against them as they would when using the Copy Mirror tools.



Copy Times tools

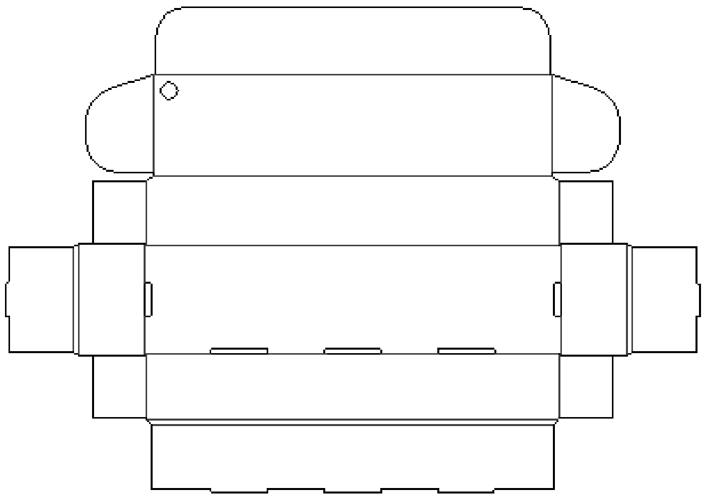
 The thirteenth button on the Edit toolbar activates Copy Times Tools flyout toolbar. The Copy Times tools are used to make multiple copies of the current selection and either offset them from the original by a regular distance or rotate the copies around a common point.



 The first button on the Copy Times flyout toolbar, and the default Copy Times tool on the Edit toolbar, activates the **Copy Times Offset** tool. There must be a current selection in order for this tool to be available.

Example of using the Copy Times Offset tool

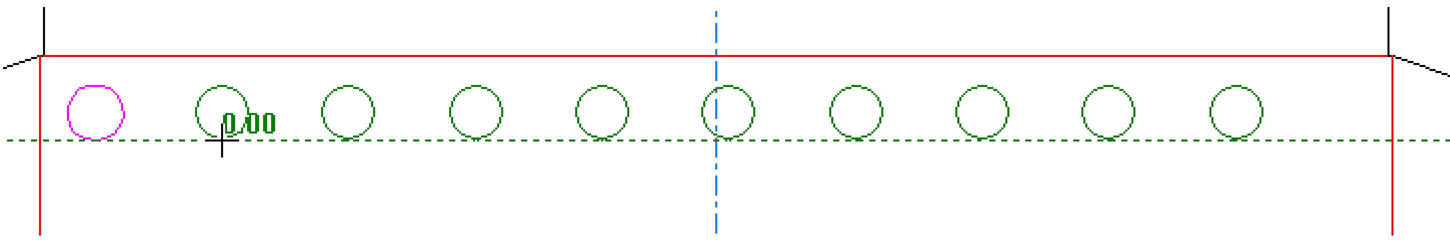
In the one-piece container shown below, the hole in the lid needs to be repeated throughout the entire lid. Copy Times Offset is the perfect tool for the job.



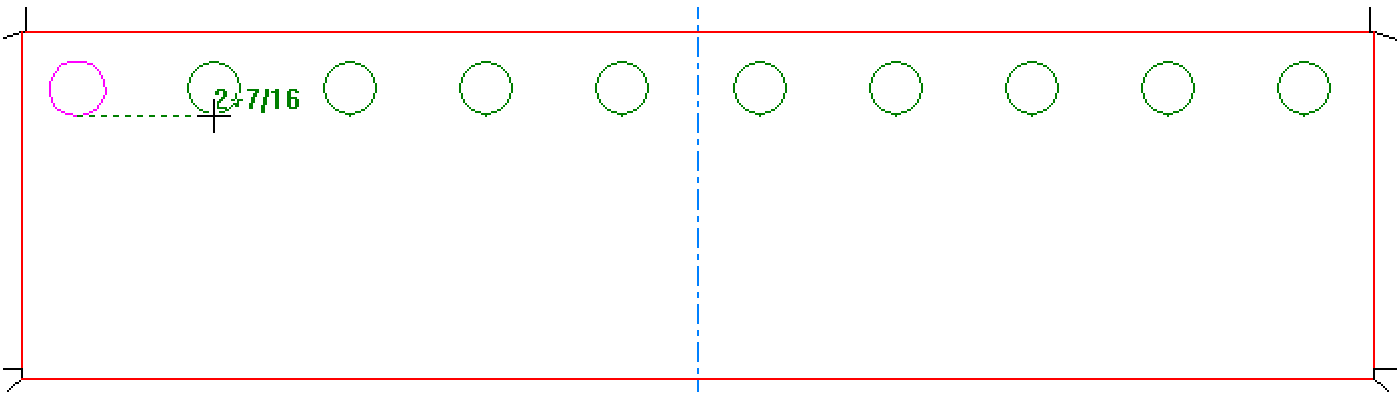
To make the copies, do the following:

1. Use the **Select** tool to select the hole.
2.  Click **Copy Times Offset** and enter the first and second numbers of copies desired in the fields on the Status bar. Press `Enter`. In this example, 9 copies and then 2 copies will be made, for a total of 30 holes.

3. Choose the point at the bottom of the hole as the pickup point. Set the angle to 0.

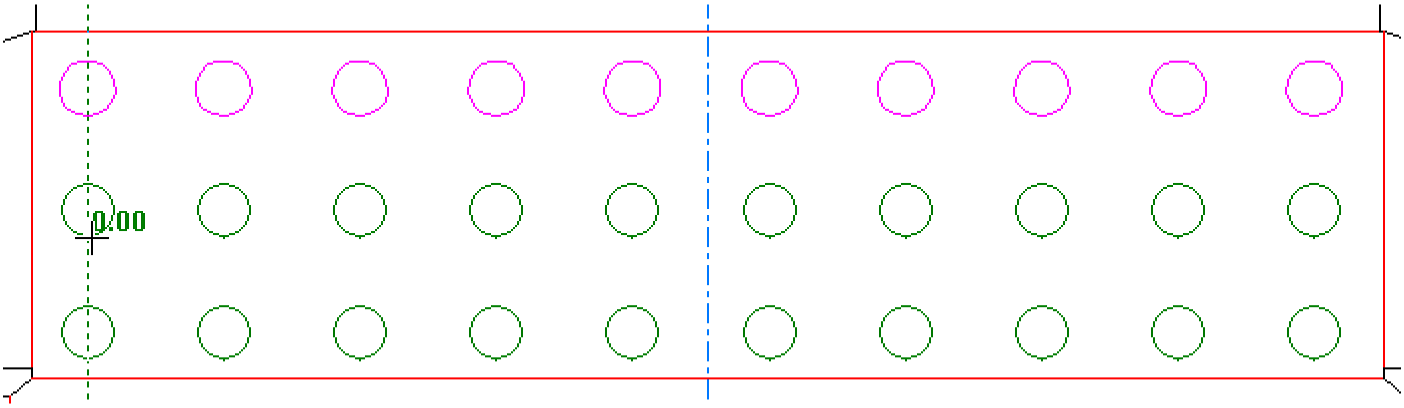


4. Choose the offset distance between the pick-up point and the put-down point.

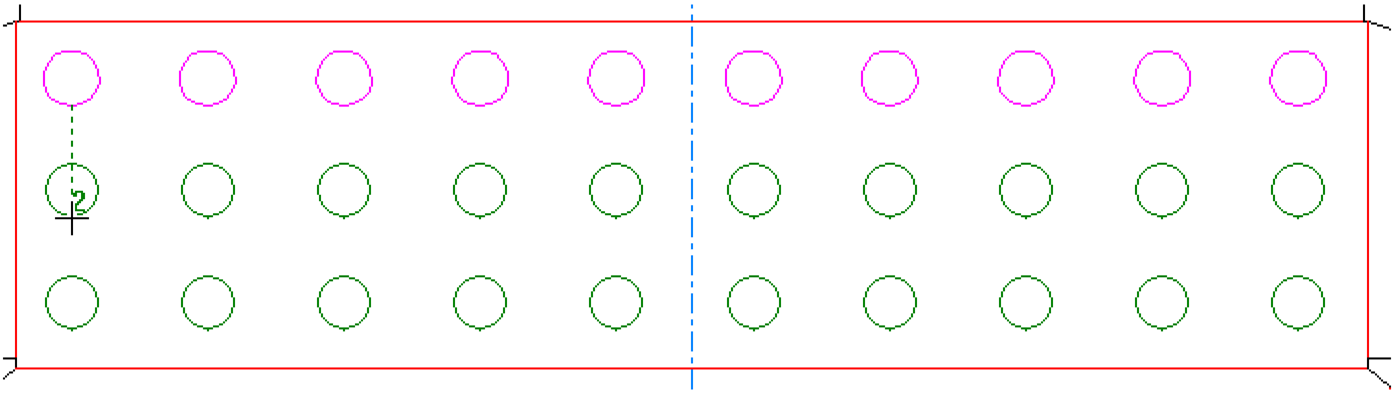


5. Press `Enter` to confirm the number of second copies on the Status bar.

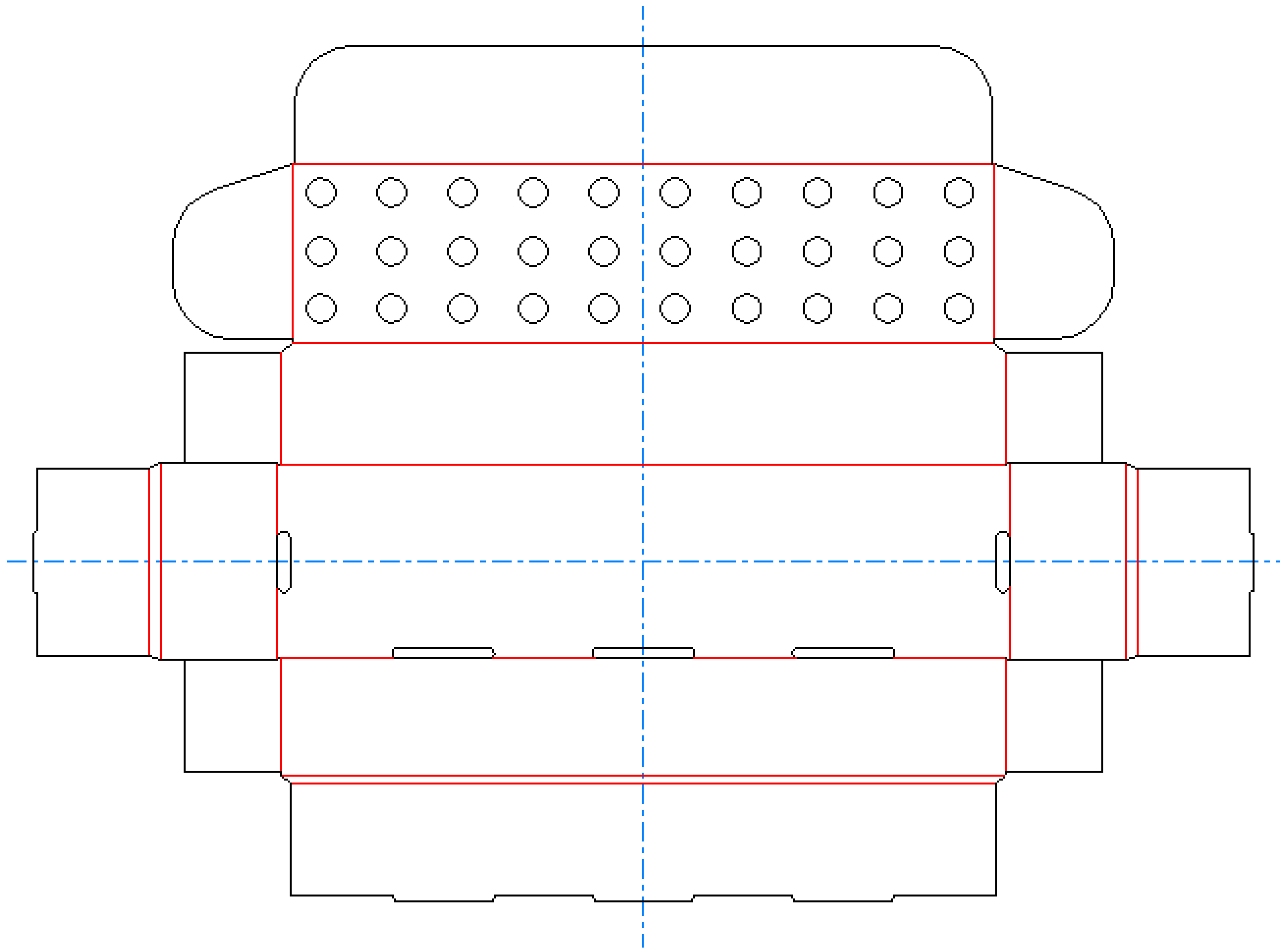
6. Set the angle for the second copy.



7. Set the offset for the second copy and click to set the put-down point.



The end product should look like the picture below.

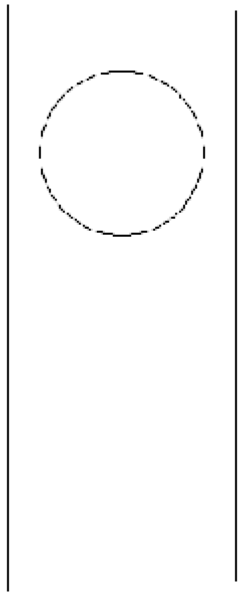


 The second button on the Copy Times flyout toolbar is the **Copy Times Rotate** tool. There must be a current selection for this tool to be available. This tool prompts for the number of copies, the fixed point (the point around which the selection is rotated), and the handle point (the point from which rotation is initiated).

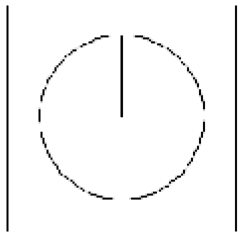
Example of using the Copy Times Rotate tool

The following instructions can be used to make a starburst design in a bottle hanger in ArtiosCAD.

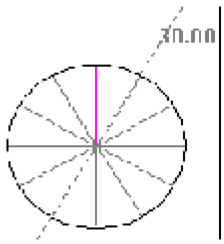
1. Use the **Rectangle** tool to make a rectangle, and then use the **Circle** tool to put a circle in it.



2. Use **Move to Point** to move the current position to the center of the circle.

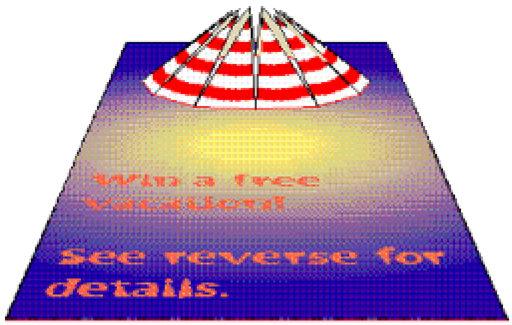


3. Draw a line from the center of the circle to the edge of the circle.
4. Use the **Select** tool to select the line created in Step 3.
5.  Click the **Copy Times Rotate** tool on the Copy Times flyout toolbar.
6. Enter `11` in the **Number of copies**: field.
7. Click the point at the center of the circle as the fixed point.
8. Click the point on the edge of the circle as the handle point.
9. Set the angle to `30` in Quadrant I and click to set the drag angle.

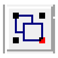


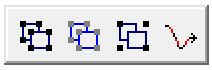
10. Delete the circle and draw a series of straight lines connecting the ends of the radial lines. Change these lines to creases.
11. The starburst opening is now complete.

An example of the design after it has had graphics added and brought into 3D:

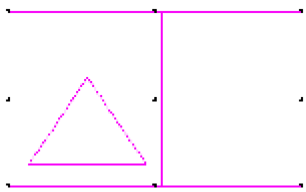


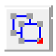
Group tools

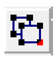
 The fourteenth button on the Edit toolbar activates the Group Tools flyout toolbar.



The first button on the Group Tools flyout toolbar, and the default Group tool on the Edit toolbar, activates the **Group** tool. The Group tool makes ArtiosCAD treat multiple objects as one object. A group shows nine handle points when it is selected Multiple groups may be nested together to form one large group.



 The second button on the Group Tools flyout toolbar activates the **Ungroup All** tool. This tool ungroups all items in the current selection, including nested groups, so that the individual components at every level are available.

 The third button on the Group Tools flyout toolbar is the **Ungroup** tool. When clicked, the Ungroup tool removes a level of grouping, making available the individual components or nested groups that comprise the current group.

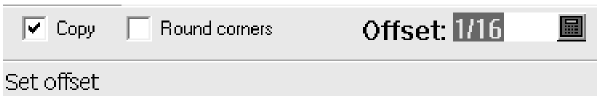
 The fourth button on the Group Tools flyout toolbar is the **Group Sequence** tool. You must have a current selection for this tool to be available, and the selected lines should be the same line type. When clicked, this tool groups the selected sequential lines together. To use this tool, select the lines to group, click **Group Sequence**, and optionally check **Reverse** on the Status bar to reverse the line direction.

If your current selection includes more than one path, ArtiosCAD groups each path separately.

If your current selection contains mirrored special rules, ArtiosCAD converts them to the same line type.

Offset Lines tool

The fifteenth button on the Edit toolbar activates the **Offset Lines** tool. This tool moves or copies offset lines from the current selection, which much be a selection of end-to-end lines. Shown below are its controls on the Status bar.



Copy copies the current selection. When it is off, the current selection is moved by offset.

Round corners inserts blends on corners where the offset is outward from the corner. No blends are added when the offset is inward.

Offset sets the amount of the offset.

To use this tool, select the lines to offset, activate the tool, check **Copy** or **Round Corners** as desired, and set the drag either using the mouse or by entering the offset directly in the field. The current selection will be moved or copied accordingly.

Horizontal/Vertical tools

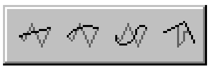
The first button on the Adjust Outline toolbar activates the Horizontal/Vertical flyout toolbar.

The first button on the Horizontal/Vertical flyout toolbar, and the default Horizontal/Vertical tool on the Adjust Outline toolbar, activates the **Make Horizontal/Vertical** tool. The Make Horizontal/Vertical tool makes the line you select either horizontal or vertical depending on which axis forms an acute angle in conjunction with the selected line.

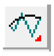
The second button on the Horizontal/Vertical flyout toolbar activates the **Align Horizontal/ Vertical** tool. Using this tool, you can align a series of points and the lines between them. Click the tool, drag a window that encompasses all the points to align, and then indicate the alignment point. Once the window is defined, all the lines and points in the window turn magenta to indicate they are selected.

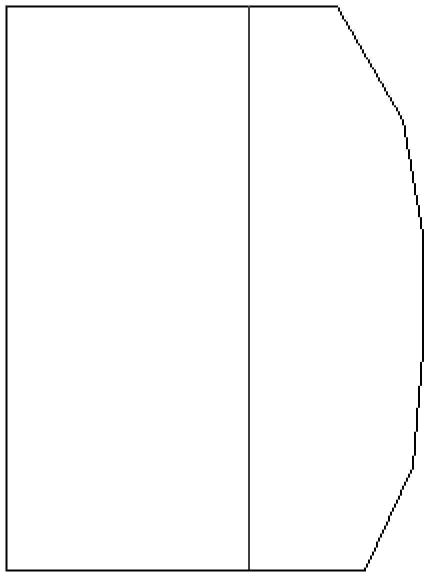
Merge Lines tools

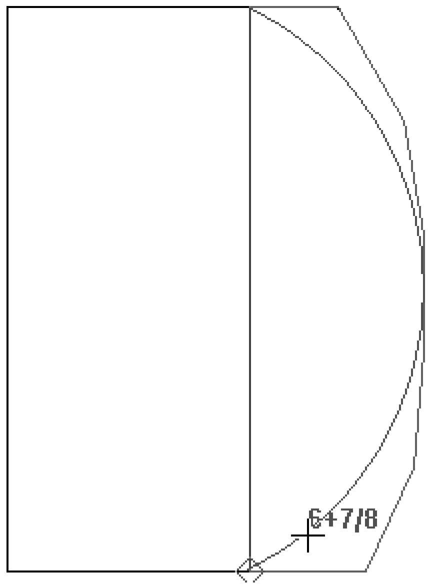
The second button on the Adjust Outline toolbar activates the Merge Lines flyout toolbar.

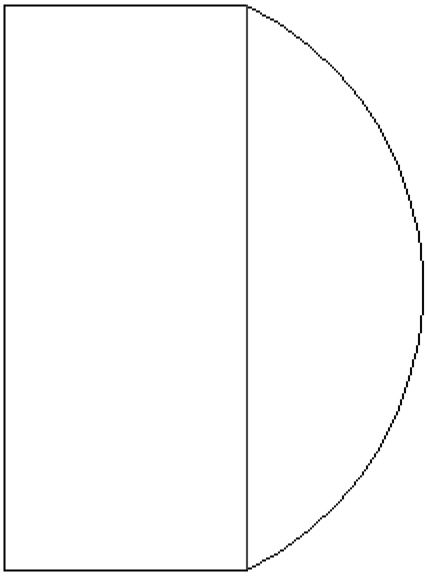


The first button on the Merge Lines flyout toolbar, and the default Merge Lines tool on the Adjust Outline toolbar, activates the **Merge Lines Straight** tool. This tool merges a series of connected lines, arcs, or beziers into one straight line. Click the tool, indicate the first line in the series, and then indicate the last line in the series. The various lines will merge into one line.

 The second button on the Merge Lines flyout toolbar activates the **Merge Lines Into Arc** tool. This tool merges a series of connected lines, arcs, or beziers into an arc. Click the tool, indicate the first line in the series, indicate the last line in the series, and set the radius of the arc. The various lines will merge into an arc.





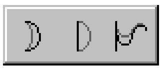


 The third button on the Merge Lines flyout toolbar activates the **Merge Lines Into Bezier** tool. This tool merges a series of connected lines, arcs, or beziers into a bezier. Click the tool, indicate the first line in the series, indicate the last line in the series, and then set the control points for the bezier. The various lines will merge into a bezier.

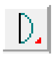
 The fourth button on the Merge Lines flyout toolbar activates the **Merge Lines to Intersection** tool. This tool merges a series of selected lines, arcs, or beziers into two lines that intersect. Click the tool, indicate the first line in the series, and then indicate the last line in the series. The first and last lines will intersect and any intermediate lines will be deleted.

Adjust Arc tools

 The third button on the Adjust Outline toolbar activates the Adjust Arc flyout toolbar.

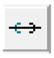


 The first button on the Adjust Arc flyout toolbar, and the default **Adjust Arc** tool on the Adjust Outline toolbar, activates the Adjust Arc tool. Use the Adjust Arc tool to adjust the radius of any arc. Straight lines may be converted into arcs using this tool.

 The second tool on the Adjust Arc flyout toolbar activates the **Straighten Arc** tool. This tool prompts for the selection of an arc or bezier and then replaces it with a straight line.

 The third tool on the Adjust Arc flyout toolbar activates the **Adjust Bezier** tool. This tool converts a line or arc into a bezier, or lets you adjust the control points of an existing bezier. The control point of the bezier you adjust using this tool is determined by which end of the object you clicked closer to when selecting the object to adjust.

Direction tools

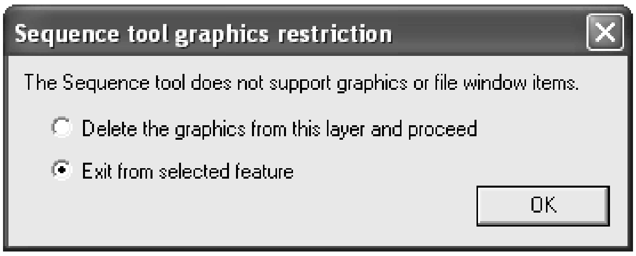
 The fourth tool on the Adjust Outline toolbar activates the **Reverse Direction** tool . This tool reverses the cutting direction of objects selected after activating this tool. Activate the tool and then indicate the line, arc, or bezier whose direction is to be reversed.

 The fifth button on the Adjust Outline toolbar activates the **Sequence** tool. This tool:

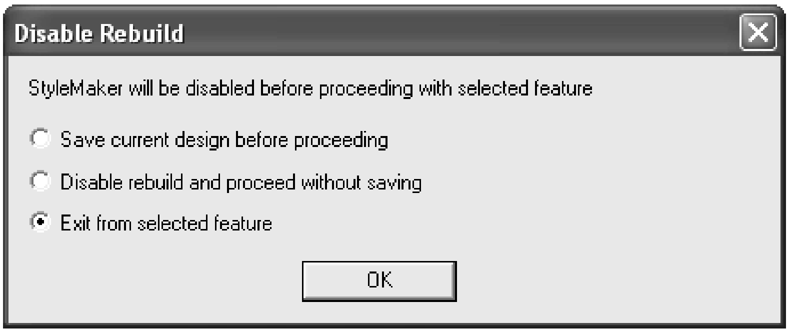
- Lets you manually specify the cutting order of lines and groups in the design.
- Works with lines, arcs, beziers, text, dimensions, and arrows.
- Works in any non-empty layer in Designer, but only works in user-defined or output layers in Manufacturing.
- Does not work with graphics or file windows, or with counter layouts.
- Disables the ability of a design to be rebuilt when used in a layer other than an output layer.

Note: There are important notes and warnings regarding using this tool and Outputs in the next section. Please read them carefully before using this tool.

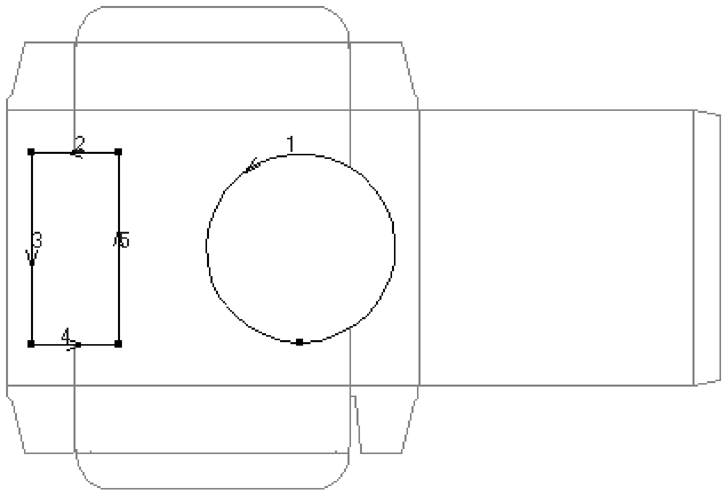
If there are either graphics or file windows in the current layer, a dialog box asks you to either delete the graphics or exit the feature. Click **OK** after making your choice.



The standard Disable Rebuild dialog box appears if the design is rebuildable.



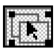
Once all the warning dialog boxes have been dismissed, the tool is active. All items not in the current layer turn gray, and line direction arrows and line numbers are turned on.

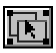


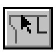
The following buttons controlling the current selection and the sequencing method appear on the Status bar:

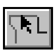
This tool works on the concept of there being different levels of items to resequence. Initially, everything in the current layer is thought of being at the same level - that there is one item number for each line in the layer. You can create subgroups of items that have a different cutting order than the items would if they were not grouped together. Groups and subgroups can be nested together in a hierarchy of levels. For example, you could resequence a complicated assembly of arcs and beziers based on the tolerances and abilities of your samplemaker or laser.

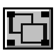
The first group of buttons on the Status bar controls the selection of items to group, ungroup, or reverse. They work the same way as the normal Select tool, except only items in the current group can be selected. Hold down `CTRL` to select more than one item.

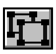
 The first button selects normally, including groups.

 The second button selects ignoring groups.

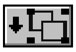
 The third button selects connected lines and continues the selection through intersections.

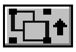
 The fourth button selects connected lines and stops the selection at intersections.

 The fifth button makes a subgroup out of the current selection and enters the subgroup to facilitate the sequencing of the lines within the group. The subgroup can then be sequenced as a single entity with other lines in the design. Item numbers for groups under the current selection level or group are shown in bold.

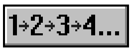
 The sixth button ungroups the currently-selected group and moves its items up a level. It is available only when a group is currently selected.

The second group of buttons on the Status bar controls the current level of groups.

 The seventh button enters a group to let you resequence the items it contains. It is available only when a group is currently selected.

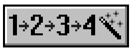
 The eighth button leaves a subgroup and resumes working in the level above it.

The third group of buttons on the Status bar controls the sequencing in the current level.



The ninth button sequences all the items and subgroups in the current group. Click the first item to sequence and click the rest of the items in the desired sequence.

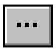
The tenth button sequences a range of items. Click the item which immediately precedes the first item to resequence, and then click the rest of the items in the desired sequence.



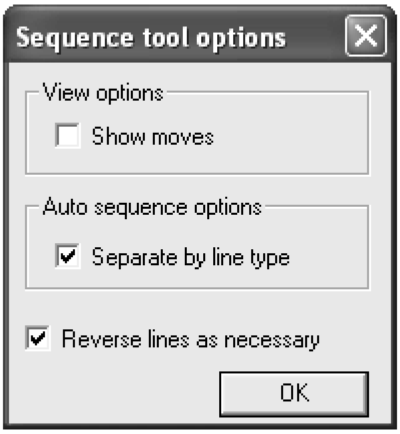
The eleventh button automatically sequences all the items and subgroups in the current group, and also makes subgroups out of connected items.

The third group of buttons on the Status bar controls the direction of lines and options for the tool.

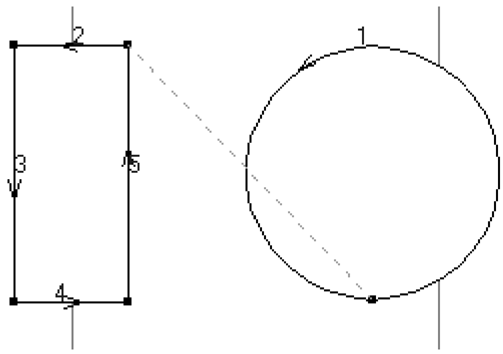
 The twelfth button reverses the lines in the current selection, or all the items in the group if there is no current selection.



The thirteenth button opens the Sequence Tool Options dialog box as shown below.



Show moves, when checked, shows a green dotted line between the end of one item and the start of the next. If the items are in a subgroup, it draws light blue dotted lines.



Separate by line type, when checked, causes the Auto Sequence button to separate the lines by type and pointage into groups, and then sequence each group. If text, arrows, or dimensions are present, they are moved into one group, and numbered in light green. When the option is not checked, lines are sequenced together regardless of type and pointage.

Reverse lines as necessary, when checked, reverses lines as needed to yield the best sequence. For the Sequence All and Sequence Range buttons, the first item selected is not reversed. When the second item is selected, both the first and second items may be reversed depending on the position of their start and end points. When the third and subsequent items are selected, they may be reversed, but the preceding items will not be reversed. For the Auto Sequence button, a new subgroup is created for each move, replacing any subgroups. When this option is not checked, the direction of lines does not change. However, the order of existing lines and subgroups changes, but the items within the subgroups do not change order.

Using the Sequence tool with Outputs

As most Outputs are configured to automatically resequence, it is important that you follow these procedures when outputting resequenced lines.

Sample output. In Defaults, copy an existing samplemaker output, rename it, and configure it using **Workspace Layer** for the **Driver Type** on the Device tab. Name the layer something appropriate such as **Resequenced**. If you plan to use this tool for sample layouts as well, clear the **Sequence one-ups in layout** checkbox on the Sample Sequencing tab. This has to be done only once.

Once the Output has been set up, use it, and then change to the layer just created. The line types will have been changed to sample line types. Use the **Sequence** tool as desired, and then output that layer to the samplemaker normally. The cutting order for lines set as sample line types will not be changed.

Plot output. In Defaults, copy an existing plotter output, rename it, and select **No optimization** in the **Optimization options** group on the Processing tab. If you plan to use this tool before plotting sample layouts as well, clear the **Sequence one-ups in layout** checkbox as well. This has to be done only once.

CAM output maintaining exact sequencing of all lines. In Defaults, copy an existing CAM output and rename it. On the Tool Selection tab of the new entry, set all the different tools to belong to group one, and then on the Optimization tab, clear the **Optimization** tab for group one. This entry will then maintain manual sequencing.

CAM output with optimized grouping but maintaining exact sequencing of lines within the optimized groups. In Defaults, copy an existing CAM output and rename it. On the Optimization tab of the new entry, clear the **Optimize** checkbox for each group in which manual sequencing is desired. Arrange the groups and sequencing as appropriate to achieve the desired results. When used, this Output will move the sequenced lines into the groups as defined, but will not reorder the lines in a group if the checkbox is not checked for that group.

Counter output. Counter outputs do not support manual sequencing.

Working with non-geometric elements

Non-geometric elements are elements of an ArtiosCAD design that are not burned on the die. They usually serve to organize parts of the design or to provide more information about the geometric elements.

Layers

ArtiosCAD uses the concept of layers to build a complete design. Think of a piece of paper with a drawing of a basic container. Then on top of that, add transparencies. One transparency could contain dimensions, another could contain art, and yet another could contain additional design geometry such as a window or an optional flap. A design may have up to 100 layers. All designs have one layer called Main Design. Designs made using standards may have many layers depending on the complexity of the standard.

Use layers to keep different parts of the design from interfering with each other. For example, if you are doing intricate design work, dimensions might clutter the workspace. Putting the geometry in one layer and the dimensions in another allows you to turn the dimensions off when desired.

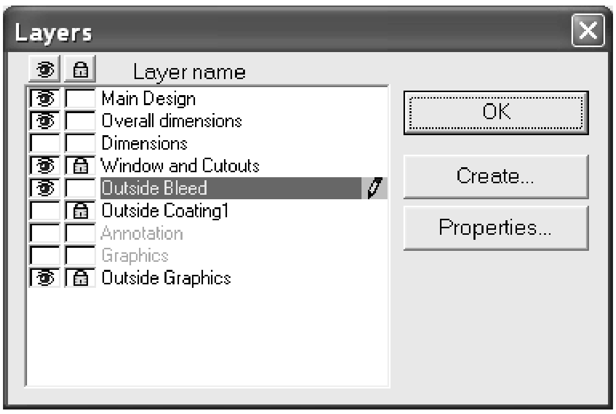
Another reason to use layers is to control the flow of information. When exporting data from ArtiosCAD into a different format, only the layers that are turned on are exported. This way you can put sensitive data in one layer, turn it off, and then export the geometry to a file for your diemaker.

ArtiosCAD works with the layers that are turned on. Only layers that are visible are printed or exported to a file. However, all layers are saved when the design is saved regardless of their viewing status.

The layers that are displayed when a design is opened depends on the setting in **Options** > **Defaults** > **Shared Defaults** > **Design Defaults** > **Default View Mode**. You can choose to display the layers as they were when the design was saved, or you can choose to always display the same set of layers.

Layers are controlled by the Layer button on the View bar. The Layer button shows the name of the current layer - it is not labelled "Layer."

Clicking the Layer button activates the Layers dialog box. Shown below is a sample Layer dialog box.



The current layer is highlighted and has a pencil icon at the right end of the highlight. Empty layers have dimmed names. Click the name of a layer to switch to it.

By default, the Layers dialog box in Single Design lists layers in the order in which they were created. To change the list order, change the default in **Options** > **Defaults** > **Design defaults** > **Layers Dialog Display Order** between **Show layers in the order they were created** and **Show layers in the order they are drawn.** The Layers Properties dialog box shows layers in the order in which they are drawn.

At the far left of the dialog box is the **Visibility** column. An eye icon in the box indicates the layer is turned on. Click the box to turn the layer on or off. The eye button at the top of the column turns all layers off except the current layer.

Between the Visibility column and the layer name is the **Lock** column. A padlock icon indicates the layer is locked and its contents may not be selected or changed. Click the box to lock or unlock the layer. The padlock button at the top of the column locks or unlocks all layers.

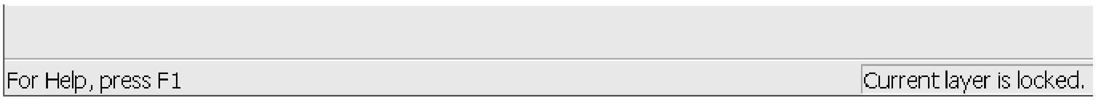
If **Temporary Dimensions** mode is active while in a **Dimensions** or **Overall Dimensions** class layer, when the Layers dialog box is opened, the pencil icon for the layer is dimmed to indicate **Temporary Dimensions** mode. It returns to normal when **Temporary Dimensions** mode is turned off.

If you have items selected when turning a layer on or off, if those items reside in the layer whose status is changing, the selection is cleared; otherwise, the items remain selected.

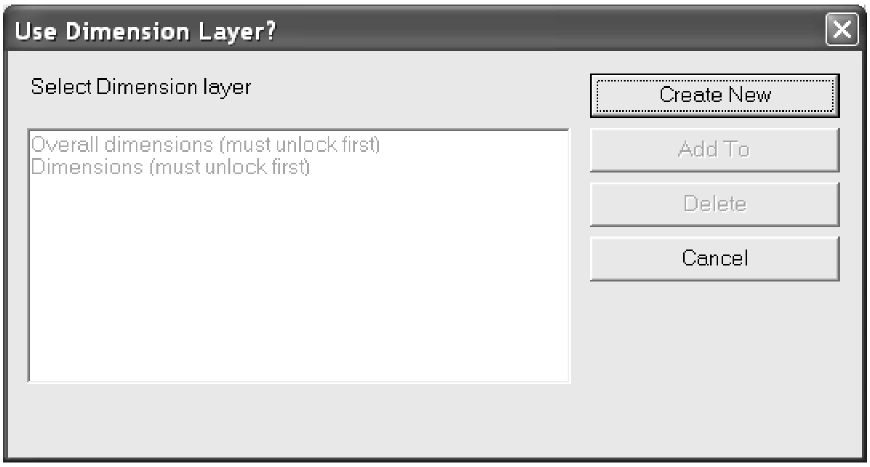
Locked layers

If you try to modify a locked layer with a tool that uses drag, the cursor changes to a pen with a red line through it as shown below, and no tools work. You must unlock the layer before attempting to modify it.

Attempting to use tools without drag in a locked layer results in **Current layer is locked** in the Status bar.



When you click a tool which requires a layer selection, ArtiosCAD indicates in various ways that a layer is locked and must be unlocked before it can be selected for use with the tool. Several examples of the different types of dialog boxes are shown below.

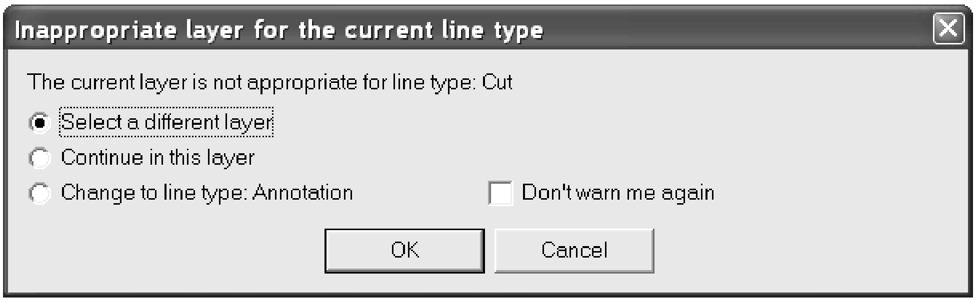


If you have items selected when locking or unlocking a layer, if those items reside in the layer whose status is changing, the selection is cleared; otherwise, the items remain selected.

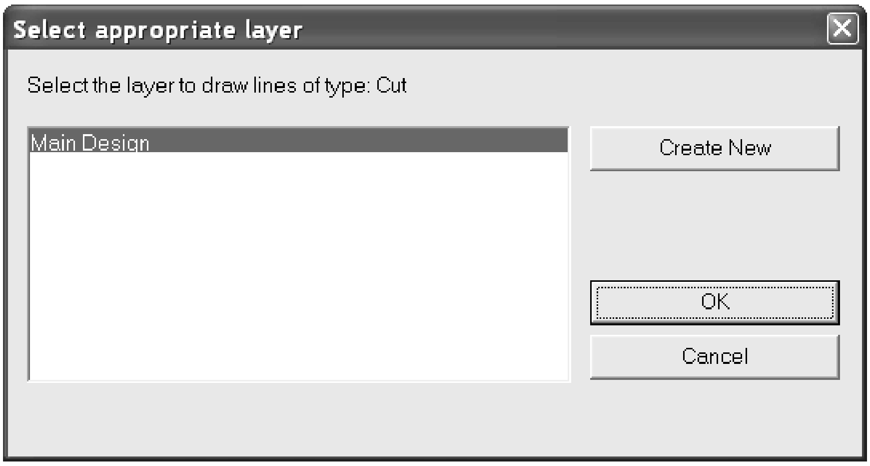
Appropriate layers

When changing layers, ArtiosCAD checks that the current line type is appropriate for the layer class. For example, when switching to a Dimensions layer, ArtiosCAD checks the current line type, and if it is inappropriate (such as Cut or Perf), ArtiosCAD changes it to a type appropriate for the layer class (such as Annotation).

After changing layers, if you try to change the current line type to an inappropriate type for the layer and then activate any of the line-drawing tools on the Geometry toolbar, ArtiosCAD warns you as shown below.



Select a different layer lets you choose a different layer in which to work, or to create a new layer. The list of layers shown contains only layers of those classes appropriate for the chosen line type. Choose a layer from the list and click **OK**, or click **Create New** to create a new layer.



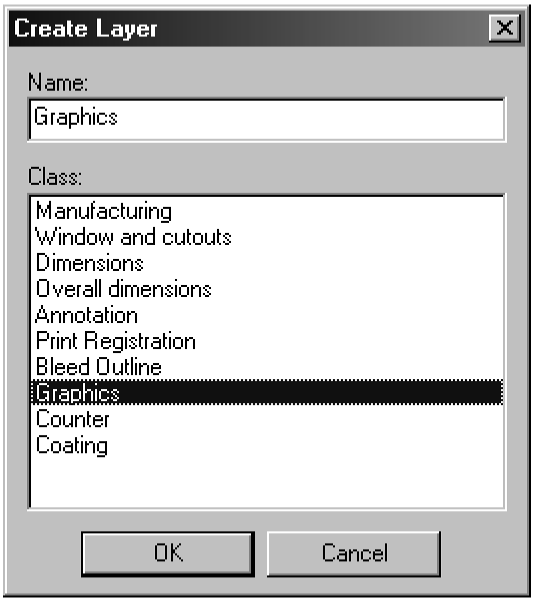
Continue in this layer keeps the line type you selected. Once one formerly inappropriate line is placed into a layer, ArtiosCAD will not warn again for that type in that layer.

Change to line type: line type changes to the line type ArtiosCAD suggests for this layer class.

Don't warn me again turns off all inappropriate layer warnings for the current ArtiosCAD session.

Adding layers

To add a layer, click the Layer button on the View bar. Then click the **Create** button to activate the Create Layer dialog box. Click the class for the new layer. The class name will be used as the name of the new layer by default. If you want to name the layer something else, erase the contents of the **Name**: field and enter the desired name. Click **OK** to create the layer or **Cancel** to cancel the creation. When you create a new layer, you are placed in it automatically. The Layers dialog box will remain open after you create a layer.



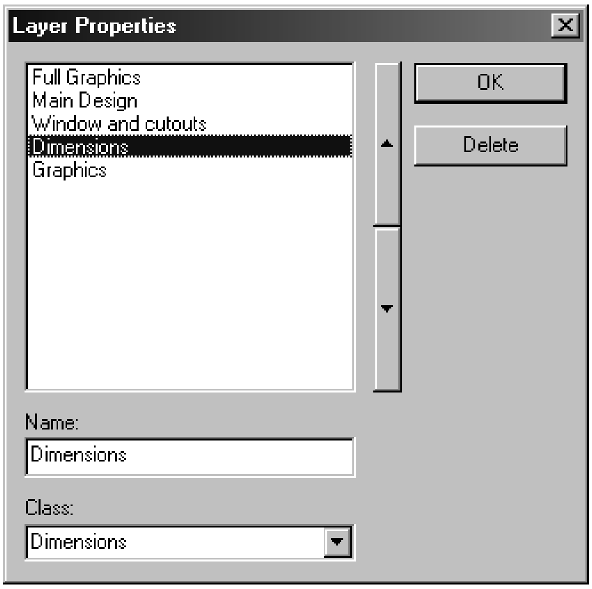
Manipulating layers

To turn a layer on, check the checkbox next to its name in the Layers dialog box.

To turn a layer off, clear the checkbox next to its name in the Layers dialog box.

The current layer is the one showing a pencil to the right of its name. You cannot turn off the current layer. To change the current layer, click the name of a different layer.

To change the name or class of a layer, select it in the Layer dialog box and click **Properties**. The Layer Properties dialog box will appear.

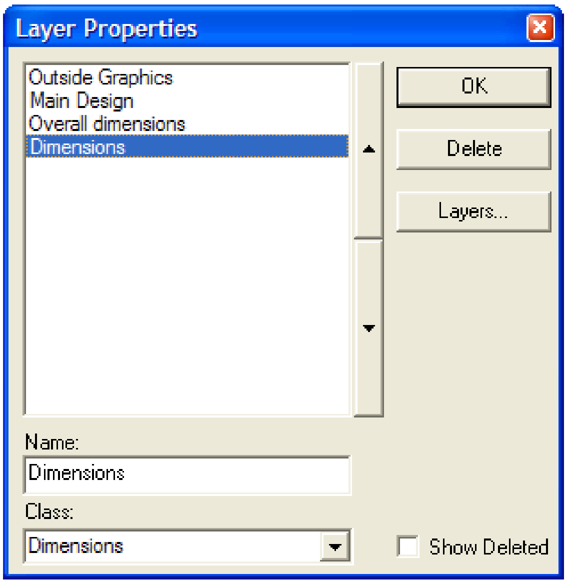


The names of the layers correspond to their position in the design. The layer at the top of the list is drawn first; the layer at the bottom of the list is drawn last. Layers toward the bottom of the list mask those toward the top of the list. Click the two arrows at the side of the list to re-order the selected layer.

Deleting layers

To delete a layer, do the following:

1. Click the Layer button on the View bar.
2. Click **Properties** in the Layers dialog box.
3. Select the layer to delete.



4. Click **Delete**. The name of the layer disappears from the list.

When you delete a layer, the current selection is reset, any extended items are reset, and the current tool is restarted after the layer is deleted. If you delete a layer while using a tool that requires a specific layer such as a bleed layer or coating layer, that tool will be finished and the Select tool will be activated.

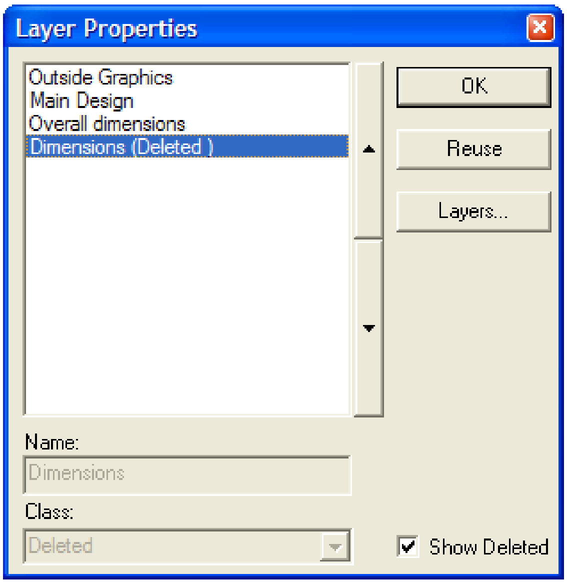
Note: Deleting most layer types removes them from the layers listed in the Layers and Layer Properties dialog boxes and also removes the contents of the layer. However, any variables associated with the layer remain.**DocPlot** layers (made with Advanced StyleMaker) and **Output to Layer** layers (made with an Output) are physically deleted from the workspace. Opening a workspace containing deleted layers in an earlier version of ArtiosCAD will warn about unknown commands, but you may safely ignore that warning.

Reusing a deleted layer

ArtiosCAD workspaces have a limit of 99 layers. If you delete many layers, you may encounter this restriction unexpectedly when creating a new layer. To avoid this limit, you can reuse a deleted layer by removing its deleted marker.

To reuse a layer, do the following:

1. Click the Layer button on the View bar.
2. Click **Properties** in the Layers dialog box.
3. Check **Show Deleted**. (This needs to be done only once per ArtiosCAD session.)
4. Select the layer to reuse. It will have **(Deleted)** after its name.



5. Click **Reuse**.
6. Set the **Class:** field appropriately for the content of the layer.
7. Click **OK**.
8. Change the contents of the layer as desired.

Moving items between layers

There are two ways to move items between layers. The first is to cut and paste, and the second is to use the **Move to Layer** tool. For the first method, see below; for the second*#id1052159UER23*

To move items between layers, do the following:

1. Select the items to move and cut them as enhanced metafiles using **Edit** > **Copy** > **Copy as Enhanced Metafile,** or press `CTRL-C`. This requires the Designer module.
2. Click the Layer button on the View bar and set the current layer to the one where the objects are to be placed.
3. Paste the objects into the new current layer.

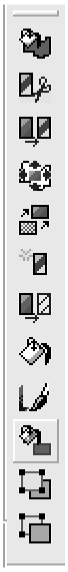
Note: Items copied and pasted between layers are not checked for appropriate line types.

Annotations and Dimensions

See the *Builder* chapter for complete instructions on using Annotation and Dimension tools.

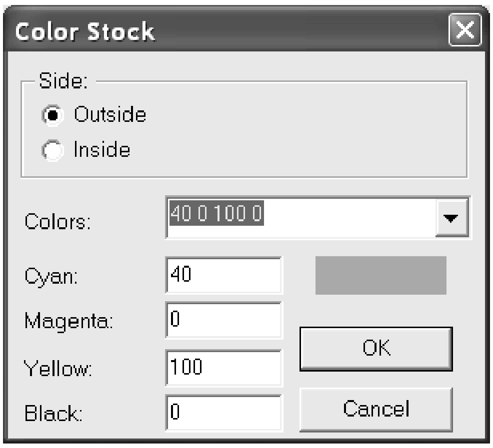
Graphics

Graphics add color and excitement to your designs. Anyone can view ArtiosCAD designs containing graphics, but you must have the ArtMaker module in order to add graphics to a design and manipulate them. Graphics are manipulated using the tools on the Graphics toolbar.



Adding a stock color

 The first button on the Graphics toolbar activates the **Color Stock** tool. This tool adds a uniform color to all panels of the design. Choose the side of the design to color, and then choose a color from the **Colors** drop-down list box, or enter the CMYK values for the desired color. The numbers in the name of the colors are the respective CMYK values that comprise the colors.



You may have only one stock color per side per print item. To change the current stock color, click the Color Stock tool and choose the new color.

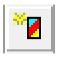
Deleting a stock color

To delete a stock color, use **Undo** if possible. If that is not possible, select the edge of the design and double-click it (or press `ALT-Enter`), then click **Remove Fill** on the Graphics tab of the Properties dialog box.

Adding graphics

One way to add a graphic to your design is to open it in your drawing application (such as Adobe Illustrator or Microsoft Paint), copy it to the Windows clipboard, and use the Paste command in ArtiosCAD.

Another way is to use the **Import File** command on the File menu and choose the file.

 Another way to add a graphic to your design is to use the **Add Graphics** tool on the Graphics toolbar. This prompts you for the filename of the bitmap image you want to add.

Whichever method you use, ArtiosCAD adds the graphic to the current side of the design. Use the **Side** control on the View bar to change the side of the design to the desired side before adding a graphic.

When you click **Add Graphics**, ArtiosCAD opens the last directory accessed. To add graphics from a file not in this directory, navigate to the desired directory, select the file, and click **Open**.

ArtiosCAD can import TIFF, BMP, JPEG and PNG files in 24-bit color, 8 bits each for Red, Green, Blue.

For JPEG files, ArtiosCAD can import JPEG files up to 6000 by 6000 pixels. Likewise, ArtiosCAD can import PNG files up to 6000 by 6000 pixels.

TIFF has several different compression tags, but ArtiosCAD can only import uncompressed color data. When ArtiosCAD imports a CMYK TIFF, it converts it internally to RGB. The maximum image size that can be plotted at a reasonable speed is about 28MB less than the memory in the computer. The uncompressed image size is calculated as width multiplied by height (in pixels) multiplied by 3 bytes per pixel.

When saving the TIFF file for use in ArtiosCAD, use an integer for the number of pixels per unit of measurement, for example, 72 dots per inch, not 72.5 dpi.

Note: If the desired graphic file has more than one period in the name, rename it so there is only one period in the name.

Regardless of which method you use, once the image is in your design, ArtiosCAD automatically selects it and starts the **Move** tool. Click the pick-up point, set the drag, and then click the put-down point.

The Edit tools work on graphics – use them to fine-tune the appearance of the graphic. When using the Select tool to select a graphic, click the graphic's edge. The edge of the graphic should turn magenta and handle points should appear.

Any graphics are compressed when the design is saved. If you will be sending files to other people using versions of ArtiosCAD earlier than 6.5**x**, you must export them using the appropriate entries in the **File** > **Export** menu.

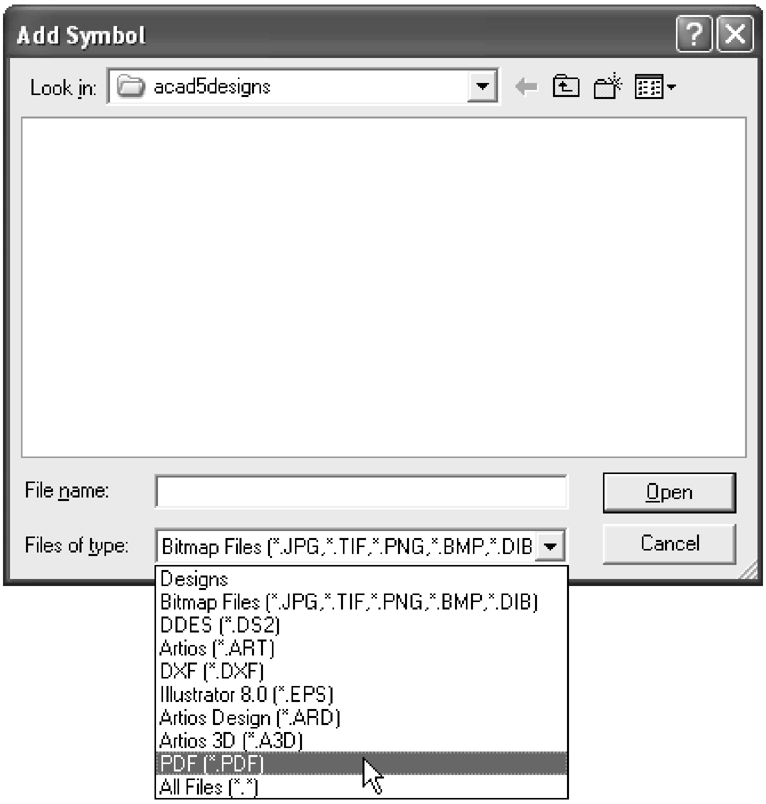
ArtiosCAD can import Adobe Acrobat PDF files if you have purchased the PDF option.

Adding a PDF file

To import a PDF file, do the following:

1. Make a normal single design.

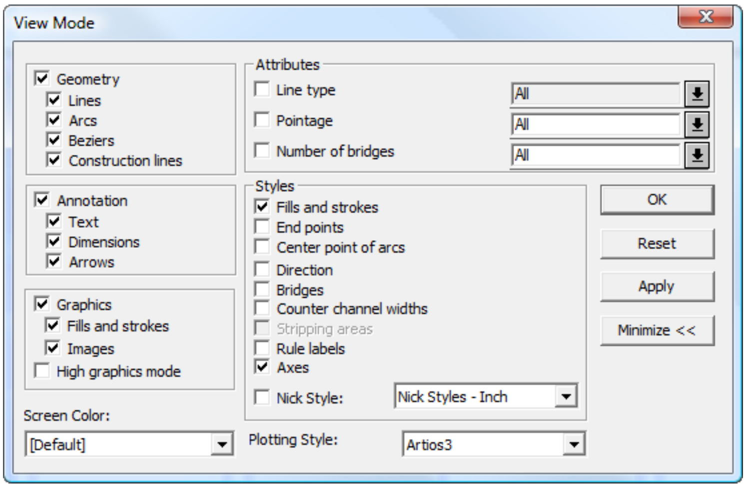
2.  Click **Add graphics** on the Graphics toolbar.
3. Navigate to the directory where the PDF file is located and change the **Files of Type**: drop-down list box to PDF.

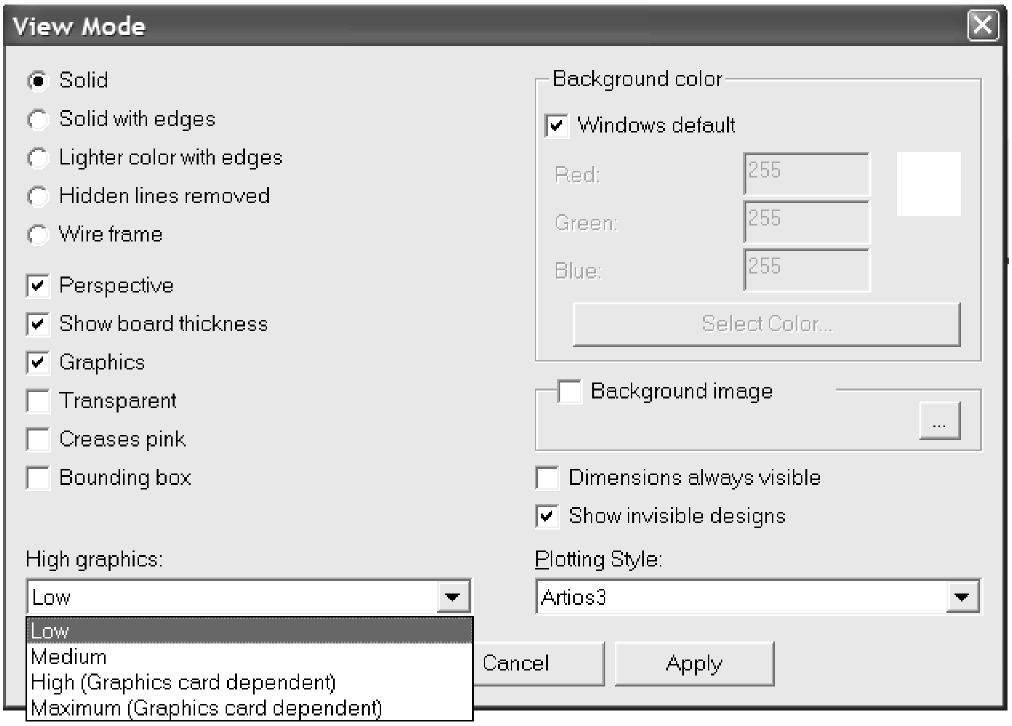


4. Select the file and click **OK**.
5. **Rendering Preview** will appear on the Status bar and then the image will appear in ArtiosCAD, ready to manipulate just like any other graphic image.

Working with Adobe Acrobat PDF files

If you have purchased the PDF option, ArtiosCAD displays more detail as you zoom in on a PDF graphic. This greater detail is called **high graphics mode**, and can be turned on either in the View Mode dialog box, and also on the context menu accessed with the right mouse button. Shown below are the View Mode dialog boxes in Single Design and 3D.





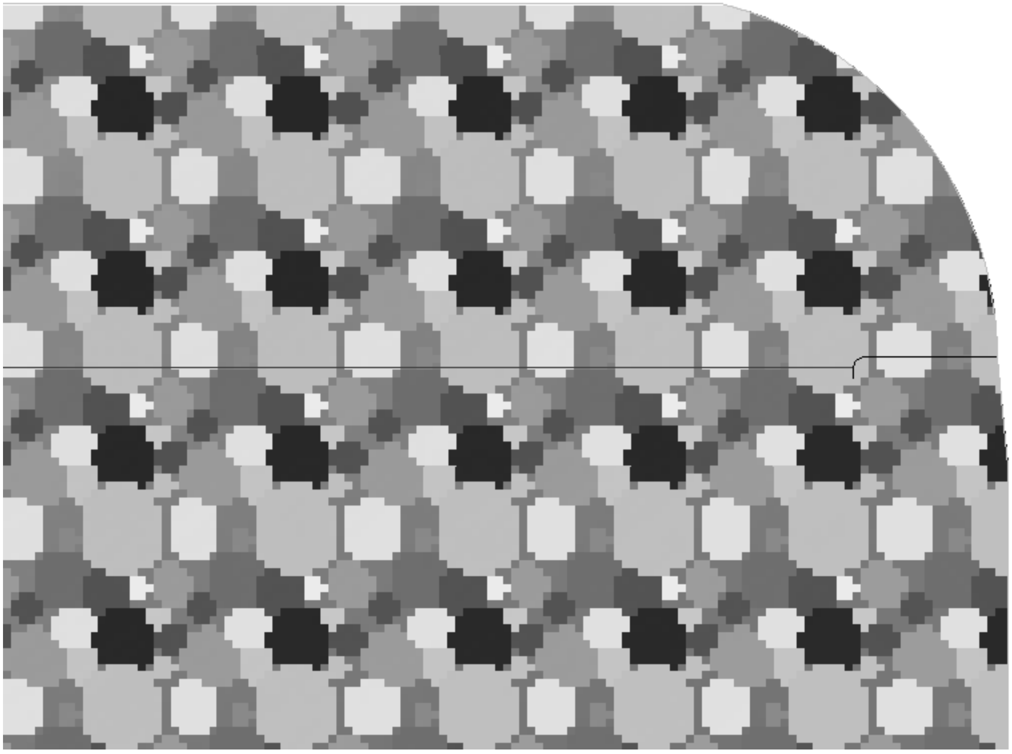
When high graphics mode is off or set to Low, ArtiosCAD shows the PDF graphic as a preview-quality bitmap. However, when high graphics mode is on, ArtiosCAD re-renders the image each time you zoom in or out for optimal viewing quality.

Use of this feature is very dependent on the performance and amount of memory on your system-specific display adapter.

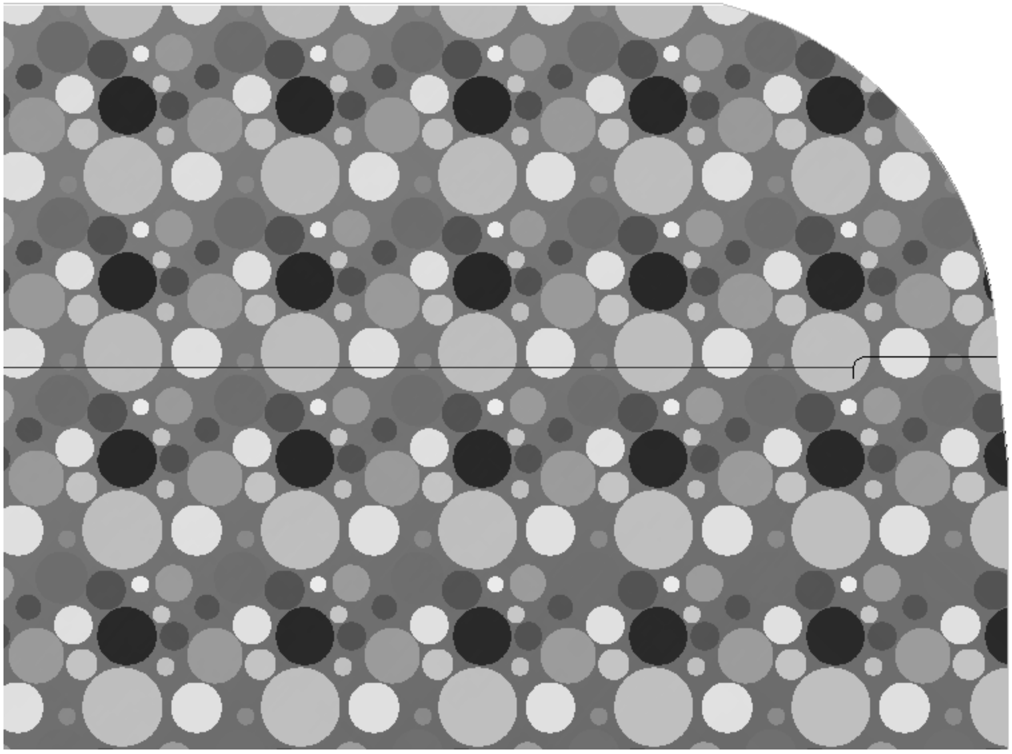
Low, **Medium**, **High**, and **Maximum** select the resolution used to display the graphics. Lower resolution results in faster performance; higher resolution results in slower performance. PDF graphics can usually be rendered at higher resolutions.

Most systems can display Low and Medium resolution with no difficulty. Some will display correctly in High mode. Only display adapters with 256 MB of memory or more will render Maximum correctly.

Shown below is a design with a PDF graphic at preview resolution when high graphics mode is off. Notice the pixelation and jagged edges of the graphic.



Shown below is the same graphic with high graphics mode turned on. Notice the smoothness of the curves.

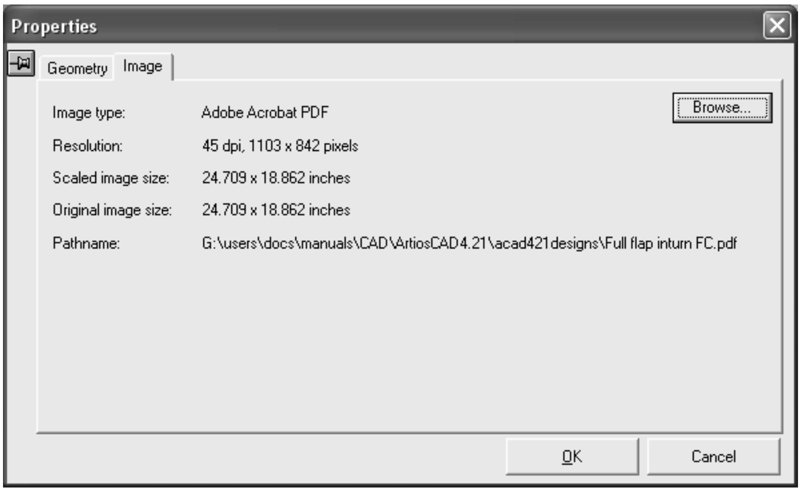


Note: When scrolling, you may have to refresh the screen using F2 or the **Refresh** command for high graphics mode to re-render the bitmap.

Note: The quality of the PDF file directly influences the quality of the zoom in ArtiosCAD. If the PDF file is downsampled or compressed, the effect of high graphics mode is limited compared with importing the graphic in another format such as TIFF or BMP.

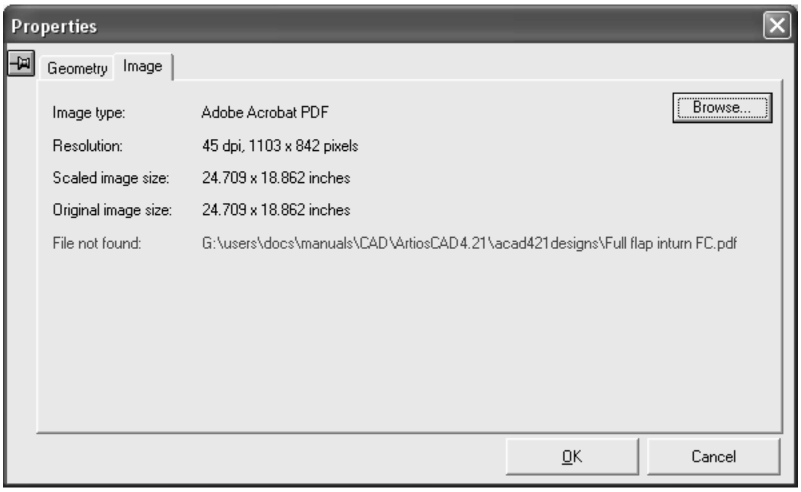
Changing the reference to the PDF file

If you need to change the reference to the PDF file, for example if you rename the PDF file, or move it to another directory, double-click the edge of the PDF graphic in ArtiosCAD to access its Properties dialog box. Next, click **Browse** and reselect the file in its new location.



Troubleshooting

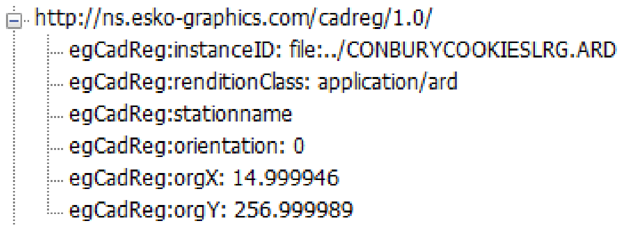
After adding the PDF file to the design, if the PDF file is not found, no error message is displayed, and the preview image is used instead. When the file is not found, the **Pathname** field in the Properties dialog box will change to **File not found** and will be in red. Click **Browse** to find and reselect the file.



Automatic registration of Esko normalized PDF files in a single design

ArtiosCAD automatically registers, or precisely aligns, a PDF file to a single design if the PDF file was created either with the Structural Design Import and Esko PDF Export plug-ins for Adobe® Illustrator® or with the most recent version of BackStage.

Specifically, the PDF file must contain embedded XMP CAD-related data. You can check for this data in either a full version of Acrobat (not Acrobat Reader) or Illustrator. In Acrobat, click **File** > **Properties** > **Description** > **Additional metadata** > **Advanced**, or in Illustrator, click **File** > **File Info** > **Advanced**. Then expand the `http://ns.esko-graphics.com/ cadreg/1.0` catalog entry and look for six `egCadReg` entries similar to those shown below.



Note:

Auto-registering PDF files cannot be added to a 3D workspace using **Add a Design**. Instead, open the auto-registering PDF and then use **Add to Open 3D**.

Registering a normalized PDF file

There are two ways to register a normalized PDF file to the single design workspace:

- Use ArtiosCAD to open the normalized PDF file directly, interpret the embedded XMP data, and open the referenced workspace.
- Use ArtiosCAD to open the single design workspace, then click **File** > **Open** and select the normalized PDF file.

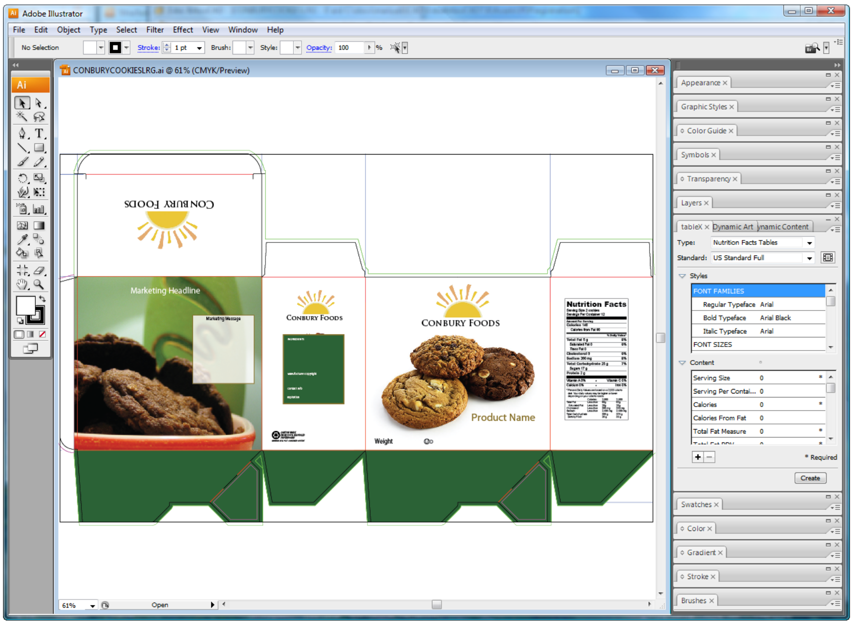
In the first method, ArtiosCAD searches for the single design workspace in the following locations:

1. The location specified in the embedded XMP data of the PDF file;
2. The same folder as the PDF file;
3. A relative folder from the folder containing the PDF file, specified in the embedded XMP data, such as `\data` or `..\graphics`.

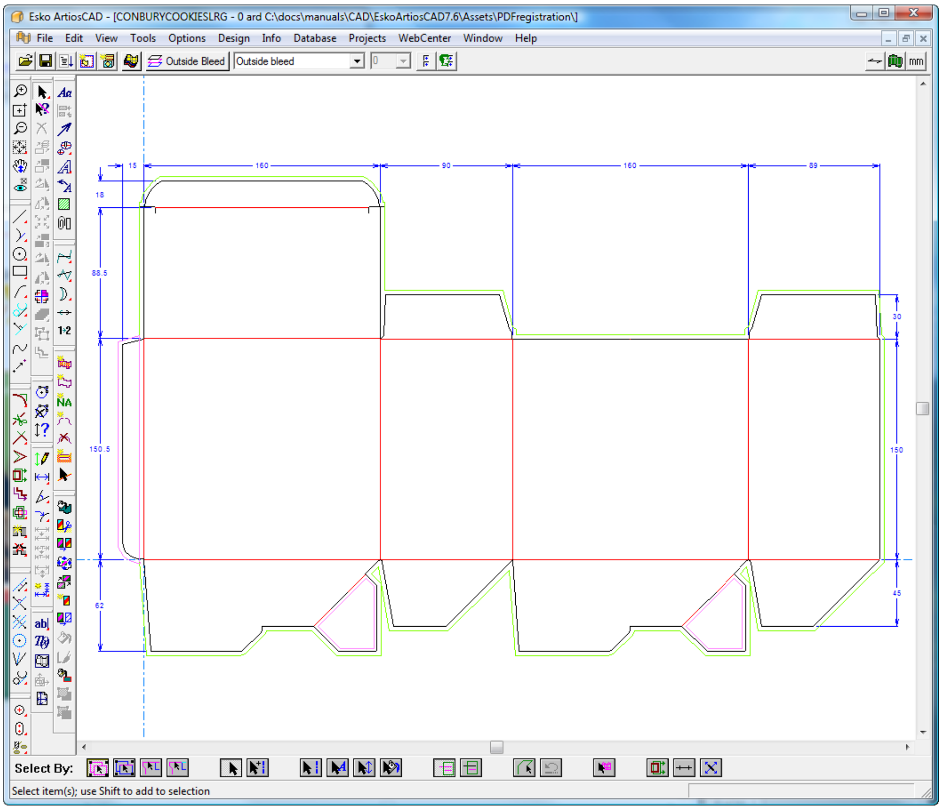
If ArtiosCAD cannot find the single design workspace to which to register the PDF file, it will open the PDF file using the vector import workflow.

In the second method, ArtiosCAD always places the PDF file in the Outside Graphics layer. If there is an existing Outside Graphics layer containing a PDF file with the same name as the one being opened, the layer is reused, else ArtiosCAD creates a new Outside Graphics layer.

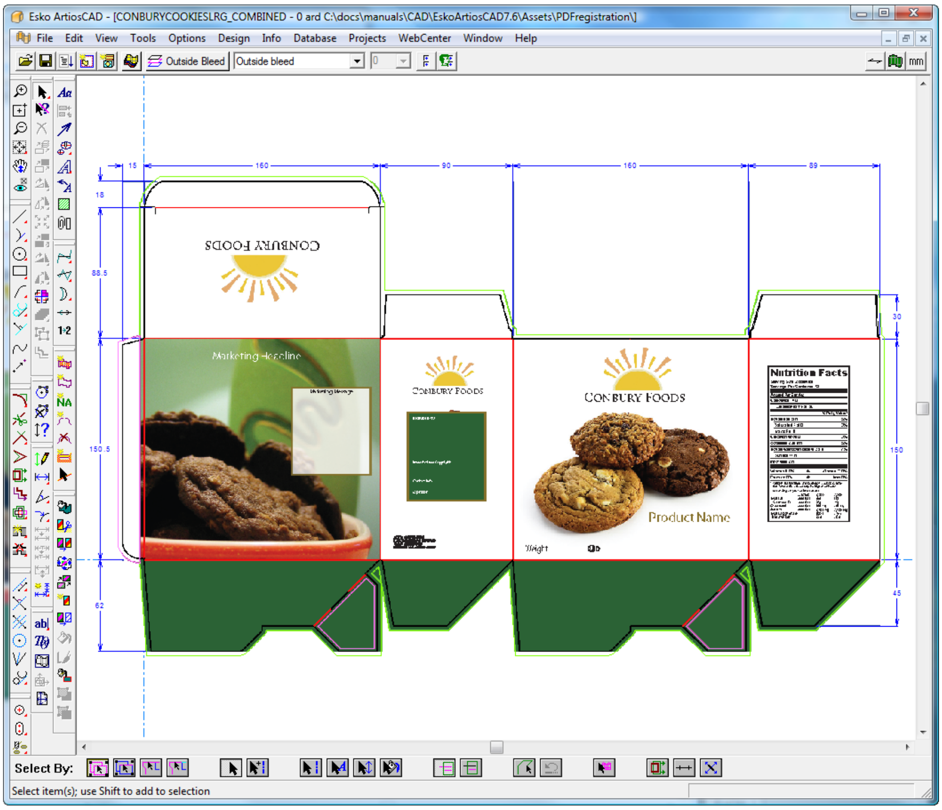
Shown below is an example of an Illustrator workspace that has had a single design imported into it using the Structural Design Import plug-in and then has had graphics applied:



The single design workspace in ArtiosCAD:



The Illustrator document exported as a normalized PDF file and combined with the single design workspace:



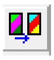
Other auto-registration formats

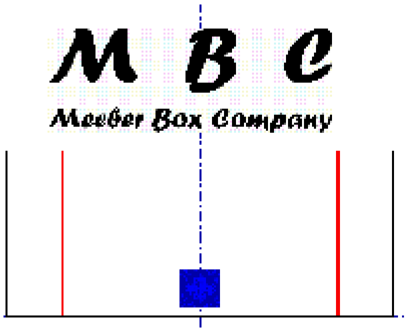
PDF files may also contain links to Collada or bag files. When ArtiosCAD opens such a PDF file, it will look for the referenced linked file, and if found, will automatically register the PDF graphics in 3D as the first printable area on the Collada or bag file.

Clip graphics

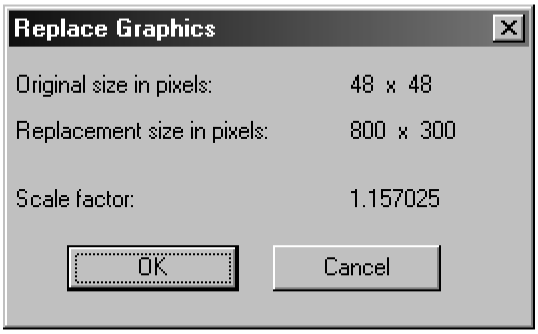
 The second button on the Graphics toolbar activates the **Clip Graphics** tool. Use this tool to trim the edges of a graphic against a loop of lines in the design (called a *clipping path*). Click this tool, indicate the graphic to clip, indicate the clipping path, and click **OK** on the Status bar.

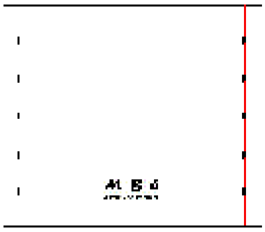
Replace graphics

 The third button on the Graphics toolbar activates the **Replace Graphics** tool. This tool replaces one graphic in the design with another graphic in the design. The replacement graphic must already be in the design before activating this tool. Click the tool, indicate the graphic to be replaced, and then indicate the graphic doing the replacing. If the replacement graphic is a different size than the original, a dialog box will appear showing the difference in size and the necessary scale factor for the replacement graphic to be the same size as the original.



The MBC logo is to replace the existing graphic. After you activate the **Replace Graphics** tool and indicate the two graphics, ArtiosCAD prompts for confirmation of the replacement.



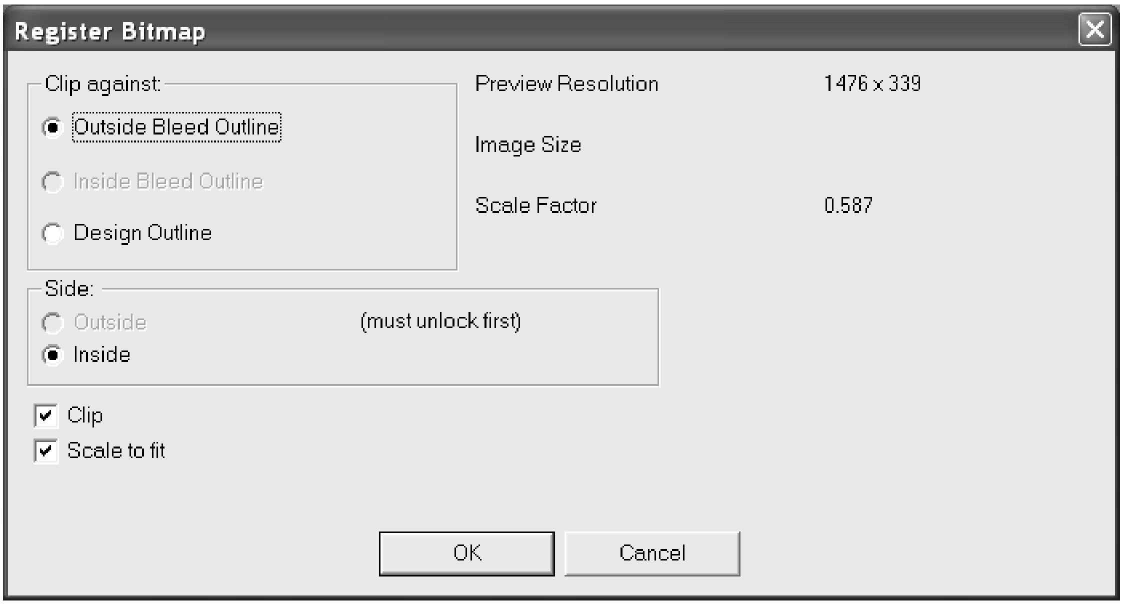


Register bitmap

 The **Register Bitmap** tool, the fourth button on the Graphics toolbar, is used to align a bitmap precisely with the design. Use this tool when you get artwork for the entire design from an outside source. The bitmap will be associated with the current print item; change to another print item if desired before using this tool.

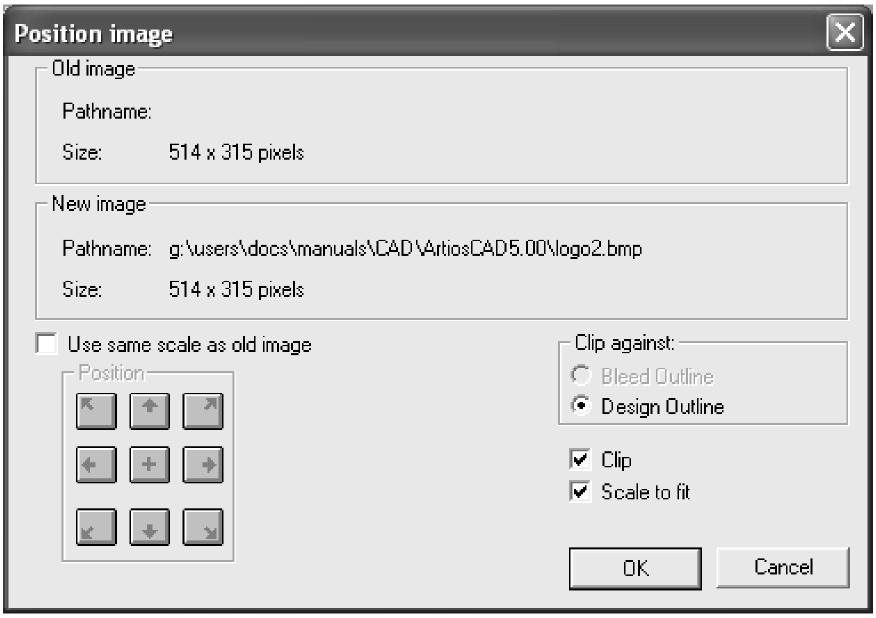
To use **Register Bitmap**, do the following:

1. Add the graphic, then click **Register Bitmap**.
2. Click the edge of the bitmap to register.
3. A dialog box appears asking what to clip against, the side of the design the bitmap will be on, whether to clip, and whether to scale to fit. If you set the **Side** to **Outside**, the graphic is added to the Outside Graphics layer; if you set the **Side** to **Inside**, the bitmap is added to the Inside Graphics layer. If a layer is locked, you may not choose it until you unlock it.



4. Click **OK** and the bitmap will be scaled to fit within the edges of the design or outline selected in the **Clip against** group.

To replace a registered bitmap and change the clipping and scale options, double-click the graphic to access its Properties. In the Properties dialog box, click **Browse** and select the new graphic. The Position Image dialog box appears as shown below. **Use same scale as old image** disables all options except the Position group and replaces the graphic at the same size. Set the options as desired and click **OK**.



When registering a bitmap to a multi-piece single design, use this workflow: To register to the design, add the image to the design. Use the **Select** tool to select the manufacturing lines for a piece (without the bleed). To register to the bleed, select the bleed lines, without the manufacturing lines, selecting a connected loop of lines. Click the **Register Bitmap** tool. In the **Register Bitmap** dialog box, the **Clip Against** option button will seed to **Outside Bleed Outline**, **Inside Bleed Outline**, or **Design Outline** depending on the selection. Click **OK** to register the bitmap.

Move Graphics

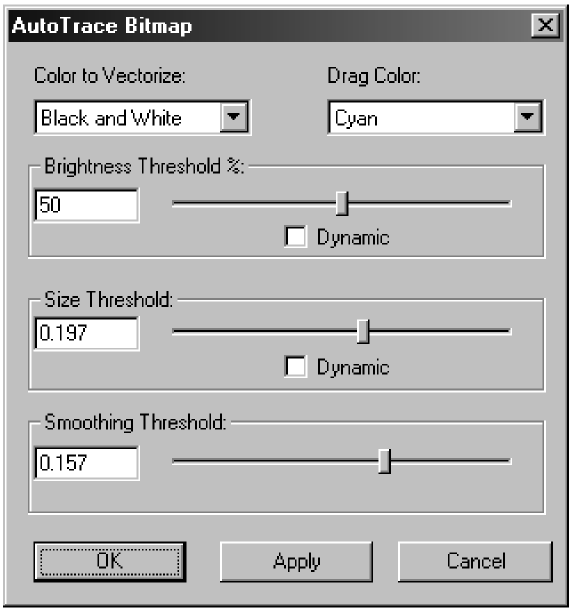
 The **Move Graphics** tool moves a graphic from one place to another.

To use this tool, ensure there is a graphic in the workspace, and then do the following:

1.  Click the **Move Graphics** tool.
2. Click the location to use as the pick-up point. It can either be on the edge of the graphic or within the boundaries of the graphic.
3. Move the mouse cursor to the desired put-down location and click the mouse button. Unlike the pick-up point, the put-down location will snap to points as well as a plain coordinate. The graphic will move appropriately.

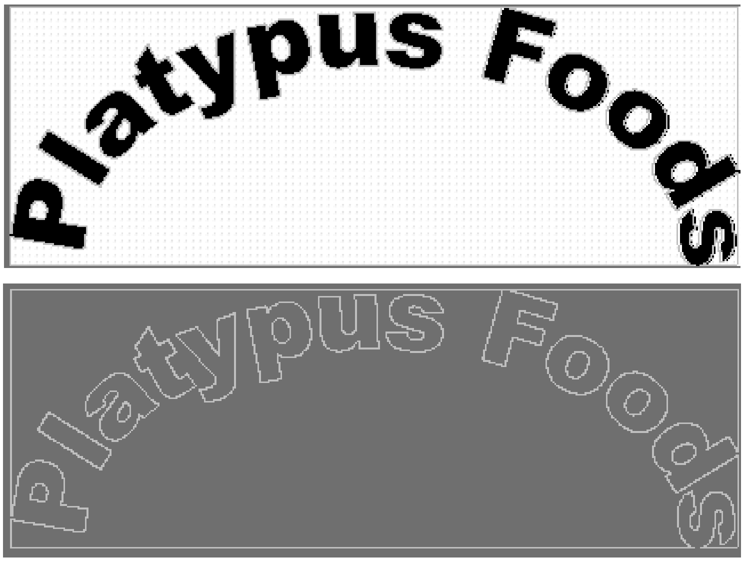
AutoTrace bitmap

 The **AutoTrace Bitmap** tool on the Graphics toolbar converts a bitmap image (such as a TIFF or BMP file) into lines and arcs that can be manipulated in ArtiosCAD. AutoTrace works best on line art drawings.



The **Color to Vectorize** option lets you choose the color that ArtiosCADwill convert to lines and arcs. **Drag Color** is the color of the lines used to show the lines that will be created. **Brightness Threshold %** controls how much will be vectorized in relation to the brightness of the color selected to vectorize. **Size Threshold** sets the size limit of items to vectorize. **Smoothing Threshold** controls the jaggedness of the new lines created. The **Dynamic** checkboxes, when checked, tell ArtiosCAD to update the screen as you drag the slider.

Below is shown a bitmap imported from another program which has been AutoTraced. The bitmap can be deleted and the remaining lines can be manipulated by all ArtiosCAD tools.



Fill

 The Fill tool on the Graphics toolbar lets you fill an area with color that is formed by a closed loop of design lines. To use this tool, select the lines that form a closed loop, and then click the **Fill** tool. The default fill color (FC Orange 021) will fill the area. To change the fill color, double-click one of the lines that forms the loop and choose the new color from the **Graphics** tab of the Properties dialog box.

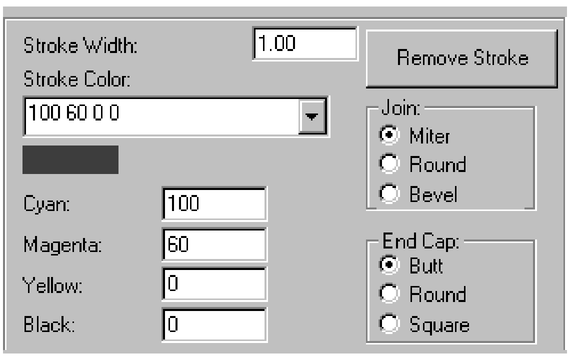
You can change the default fill color on the Graphics tab of the Property Defaults control in Defaults. When an item is filled, the lines forming the closed loop are changed to type print image, and the lines are grouped together.

Stroke

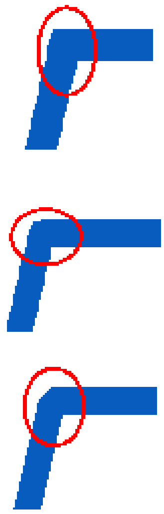
 The **Stroke** tool on the Graphics toolbar adds a ribbon of color around the edge of a line or group of lines you select.

The stroke evenly straddles the selected lines.

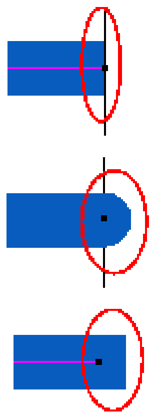
When applying a stroke, there are several options for how strokes meet and how they turn corners.



The options in the **Join** group control how strokes that meet are connected. **Miter** makes the inside and outside angles of the intersection the same. **Round** blends the outside corner of the stroke. **Bevel** chamfers the outside edge. Shown below is the junction of two stroked lines, with a mitered corner, a rounded corner, and a bevelled corner.



The options in the **End Cap** group control the ends of the stroke. **Flat** ends the stroke at the same point as the end of the object being stroked. **Round** makes a semicircle at the end of the stroke that projects beyond the end of the object being stroked. **Square** extends the end of the stroke half a stroke-thickness beyond the end of the object being stroked. Shown below is a flat stroke, a rounded stroke, and a squared stroke end. In the examples, the line being stroked is shown selected to clarify the stroke option.



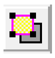
Fill panel

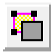
 The **Fill Panel** tool on the Graphics toolbar fills the panel you indicate with the default fill color.

To use this tool, click it. The cursor will change to a paint bucket pouring paint. Click inside the panel you want to fill. ArtiosCADwill automatically exclude any cutouts that exist inside the panel.

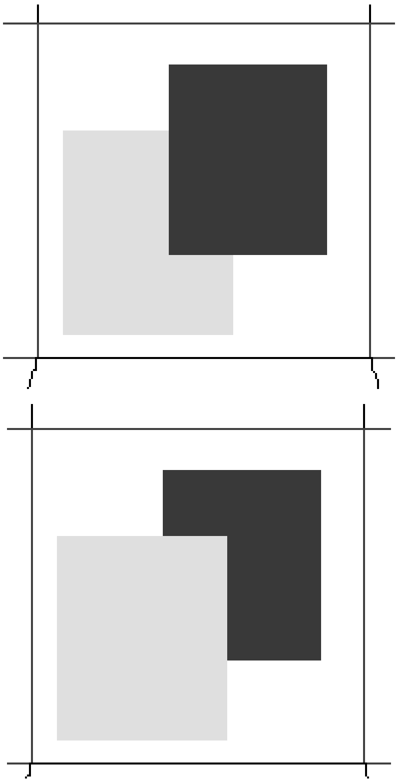
Send to front and Send to back

Graphics in ArtiosCADare applied in a layer one on top of another like a stack of paper on a desk. Layers are also stacked; to change layer order, and not the order of graphics within a specific layer, use the arrow controls in the Layers dialog box to change the layer order.

 **Send to front** moves the selected graphic to the top of the stack so that it is visible in front of all other graphics occupying the same area in the design.

 **Send to back** moves the graphic to the bottom of the stack.

To use these tools, first select the object(s) you want to send to the front or back, and then click either Send to button. Shown below are two rectangles - one in front and one in back, and then reversed.

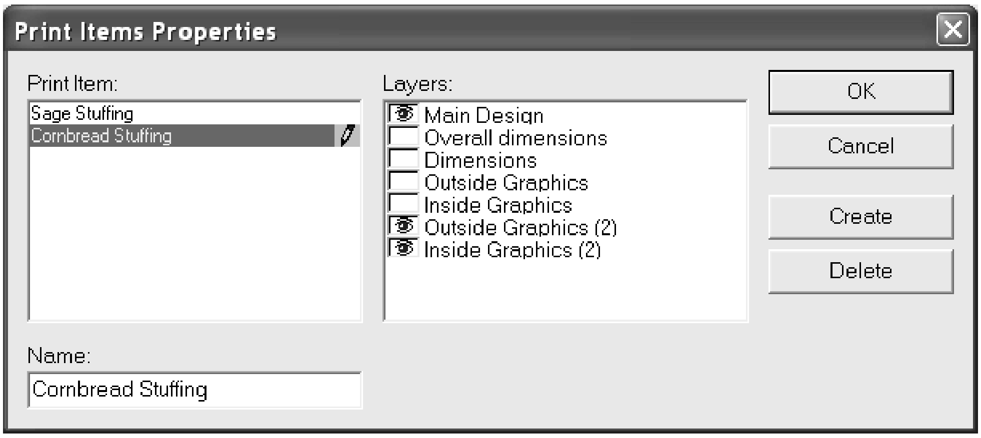


Print items

Print items are groupings of layers into logical units. For example, if the same carton is to be printed in green and in red, instead of making two copies of the same design, you can create two print items and associate different overlays with each item. Then you can design a set of manufacturing tools that uses both print items. There may be up to 100 print items in a design.

 The **Print Item** button on the View bar controls print items. The name of the current print item, if any, appears next to the icon.

To create a print item, click the **Print Item** button. If this is the first print item in the design, enter the name for the print item when prompted. If this is not the first print item in the design, click **Create** in the Print Item Properties dialog box and enter the name of the new print item.



The current print item is highlighted in the **Print Item**: pane. The layers associated with the selected print item have eye icons in the **Layers**: pane. In the example above, the Main Design, Outside Graphics (2), and Inside Graphics layers are part of the Cornbread Stuffing print item.

The Main Design layer is always associated with every print item; its checkbox cannot be cleared.

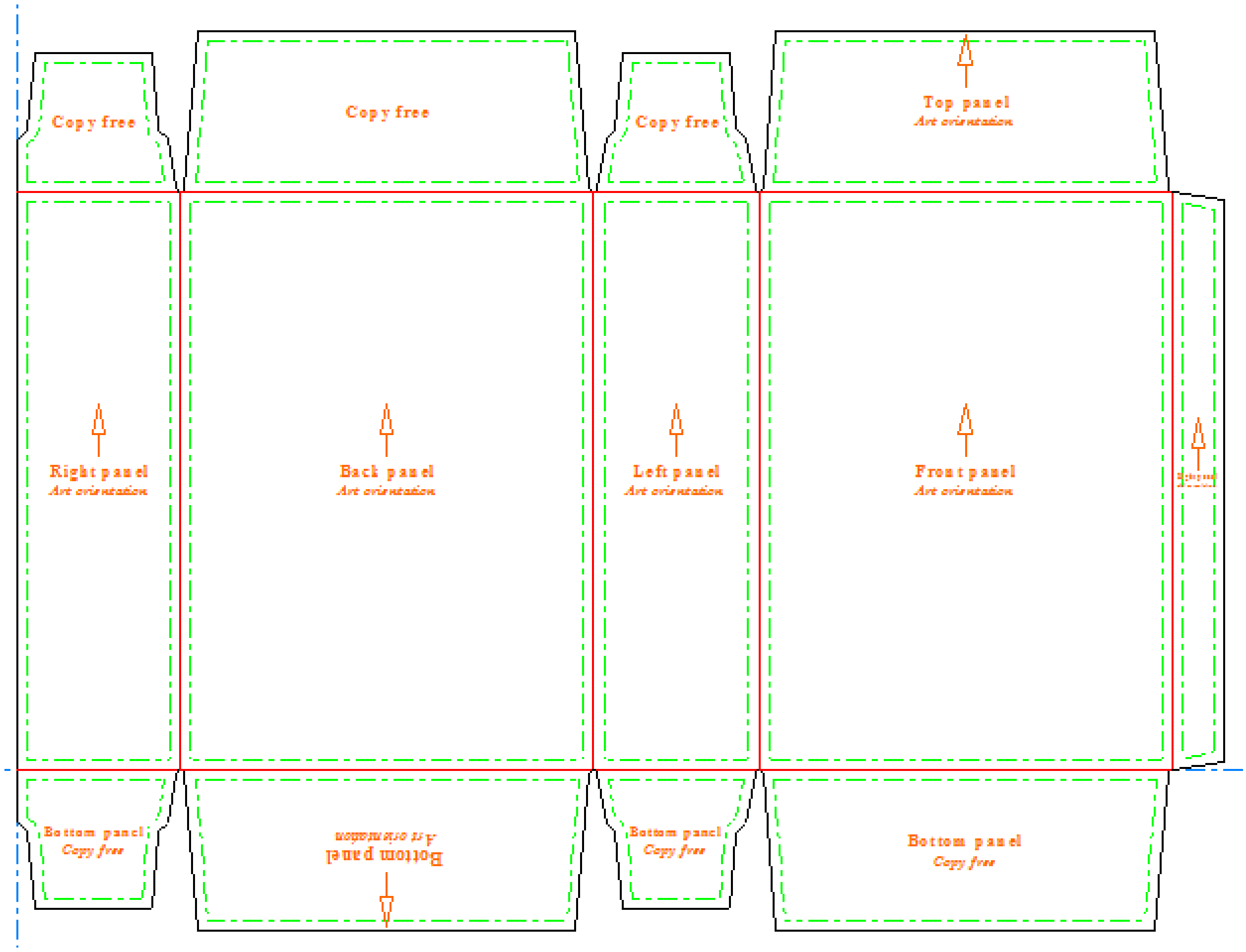
To delete a print item, click **Delete** in the Print Items Properties dialog box.

Artwork Panels

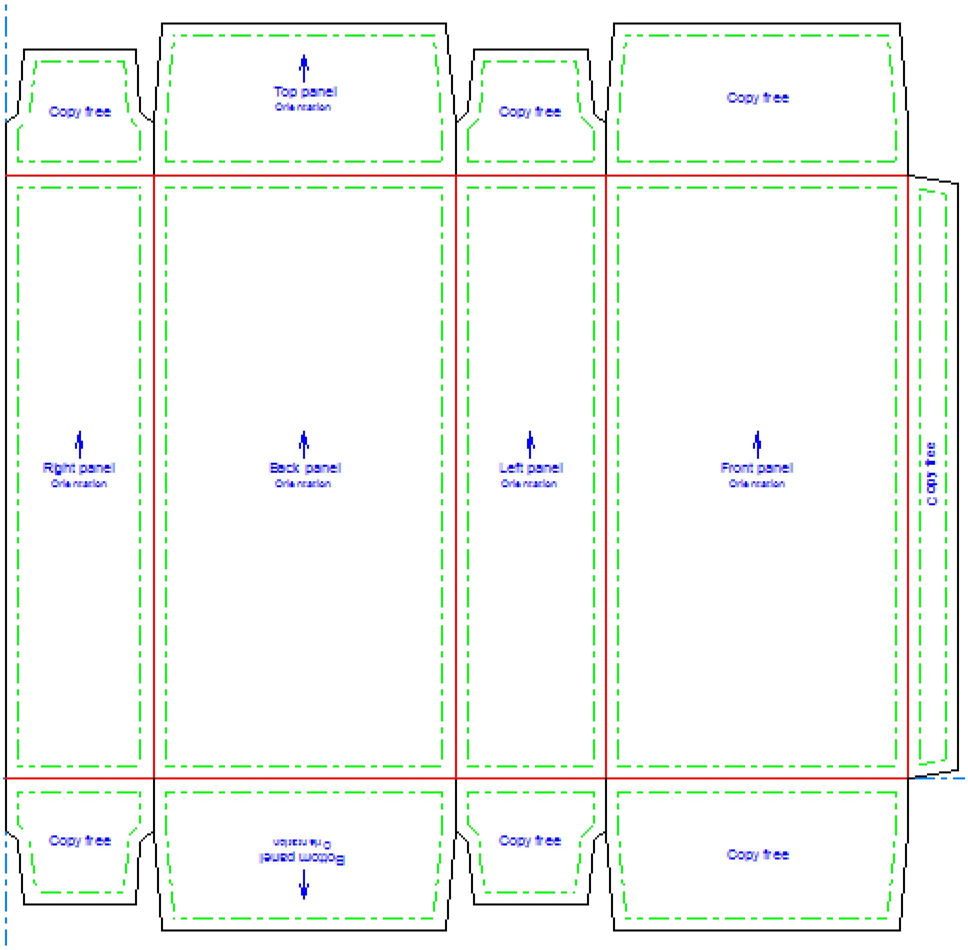
 *Artwork panels* are a way for you, the structural designer, to label which panels are visible and should therefore contain artwork to others in the supply chain such as a graphic desginer. The panels are normally labeled **Front**, **Back**, **Left**, **Right**, **Bottom**, and **Top**. You can also indicate the **Principal** panel.

You can define artwork panels in three ways:

- In 3D, you can fold the design, position it as desired, export the information back to 2D, and have Artwork Panels automatically create the labels.



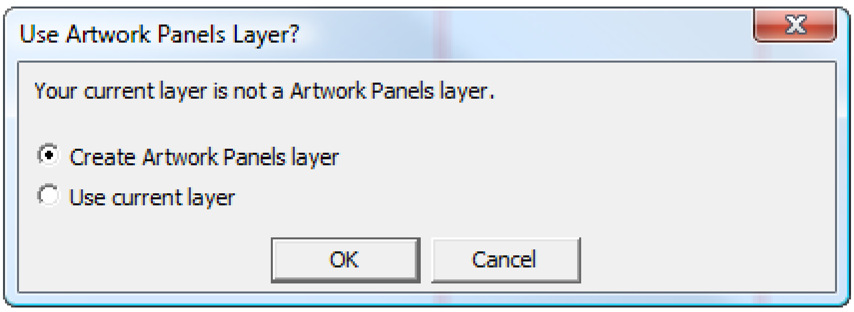
- If you do not have 3D, you can easily label the panels manually within **Artwork Panels**.



- You can also create a custom panel based on lines that form an outline.

Using the Artwork Panels Tool

 When you click **Artwork Panels**, ArtiosCAD first prompts you to create an Artwork Panels layer if one does not already exist. Click **OK** to create the layer.



It then labels the panels of the design with their function if you have used the **Update 2D** tool in 3D. It gets information about panel positioning from 3D, or you can designate the panels manually.

Artwork Panels has the following controls on the Status bar:

Offset controls how far the panel label edges are from the design lines.

The drop-down list box showing **Copy free** sets the type of label created by your next click. If a panel is defined already, changing the selected item in the list changes a panel to that type when you click inside it. The default panel types and uses are:

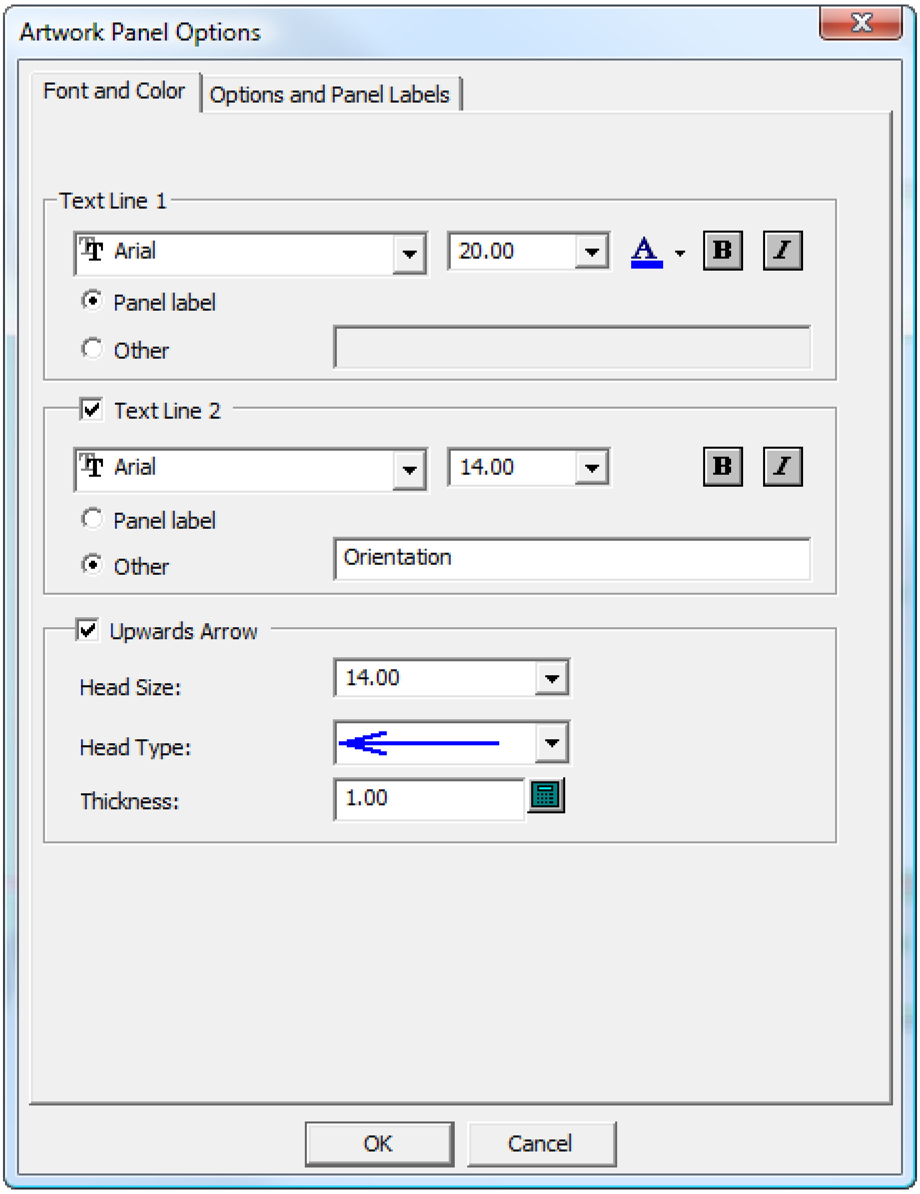
Label	Purpose
Copy area	Can have copy text, but is not a front, back, left, right, top, or bottom panel
Rotate front	Rotates the front panel label between the front, right, back, and left panels so that a different panel is front. If you have 3D, you do this by changing the view angle.
Front	Make this a front panel
Back	Make this a back panel
Left	Make this a left panel
Right	Make this a right panel
Top	Make this a top panel
Bottom	Make this a bottom panel
Principal Display Panel	Make this the Principal Display Panel independent of the front, back, left, right, top, or bottom panel
Partial copy area	Make a copy outline from the outline of the visible part of the panel
Copy free	Make this a copy free panel
Partial copy free	Make a copy free area from the outline of the invisible part of the panel
Ink free	Make this an ink free panel, such as a glue flap
Clear	Deletes the label in the panel

A **copy area** is where the graphic artist may place copy text that is visible to the customer.

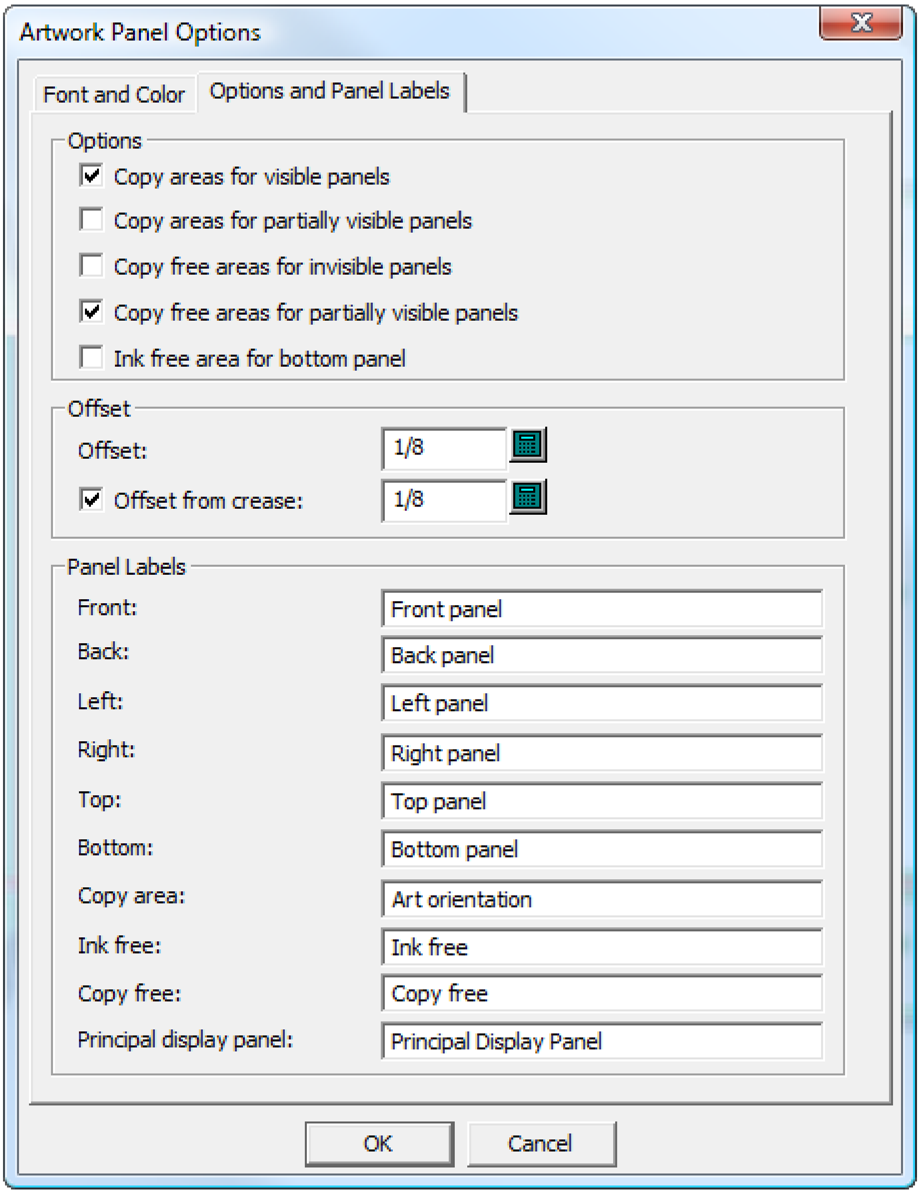
A **copy free area** is where the graphic artist should not place copy text because it cannot be seen or is hidden by a flap or the product. However, graphics may go in copy free areas.

An **ink free area** may not be printed.

 More options opens the Properties dialog box.



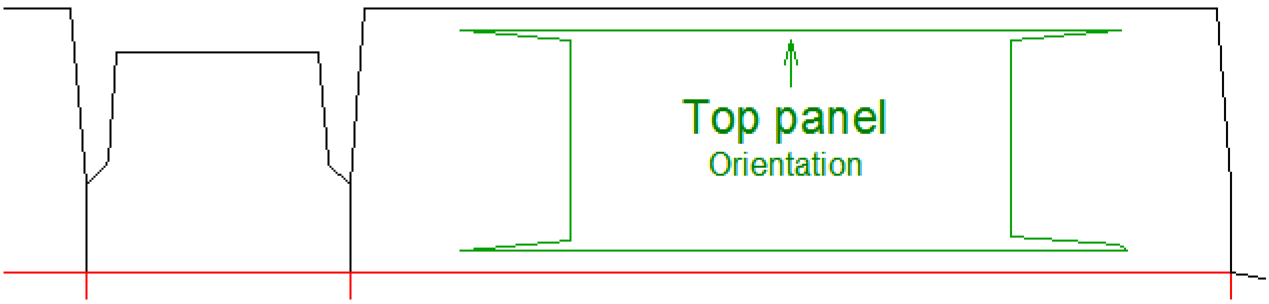
On the Font and Color tab are the standard font and color controls for the label, as well as controls to control the creation and appearance of an arrow. Set these options as desired.



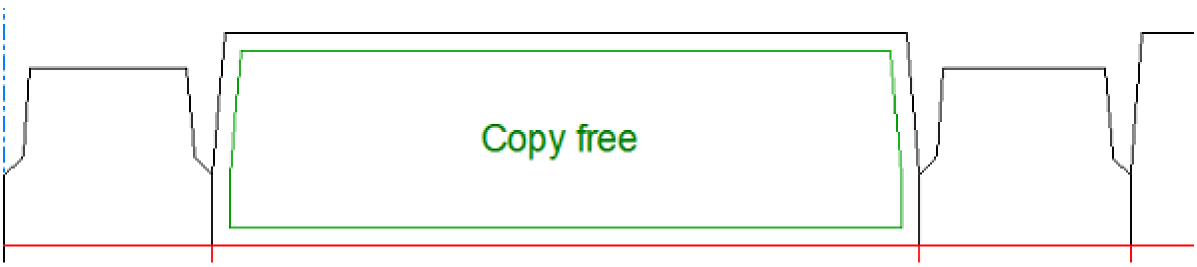
The options on the Options and Panel Labels tab control the types of panels **Artwork Panels** creates, the offsets for the panel edges, and the labels of the panels.

Copy areas for visible panels creates copy areas on visible panels. This option is enabled by default.

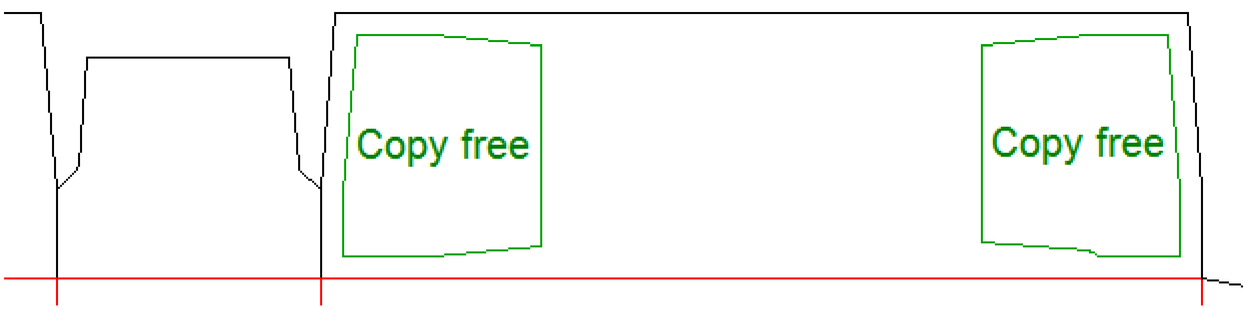
Copy areas for partially visible panels creates copy areas on those parts of panels that aren't hidden by other panels (such as flaps). If you have a flap arrangement like the one shown below, with the secondary flaps on top of the primary flaps, having this option selected would create an artwork panel as shown beneath that.



Copy free areas for invisible panels labels those panels customers will not see as copy free.



Copy free areas for partially visible panels labels the invisible parts of partially visible panels as copy free areas.



Ink free area for bottom panel labels all the bottom panels as ink free.

The options in the **Offset** group control the distance between the panel edges and the design lines. If desired, you can have a larger or smaller offset from creases by changing the value in **Offset from crease**.

In the **Panel Labels** group, change the labels of the panels as desired

Note: All of these options have defaults in the Artwork Panels section of Startup Defaults.

Click **OK** to close the More Options dialog box and return to the tool.

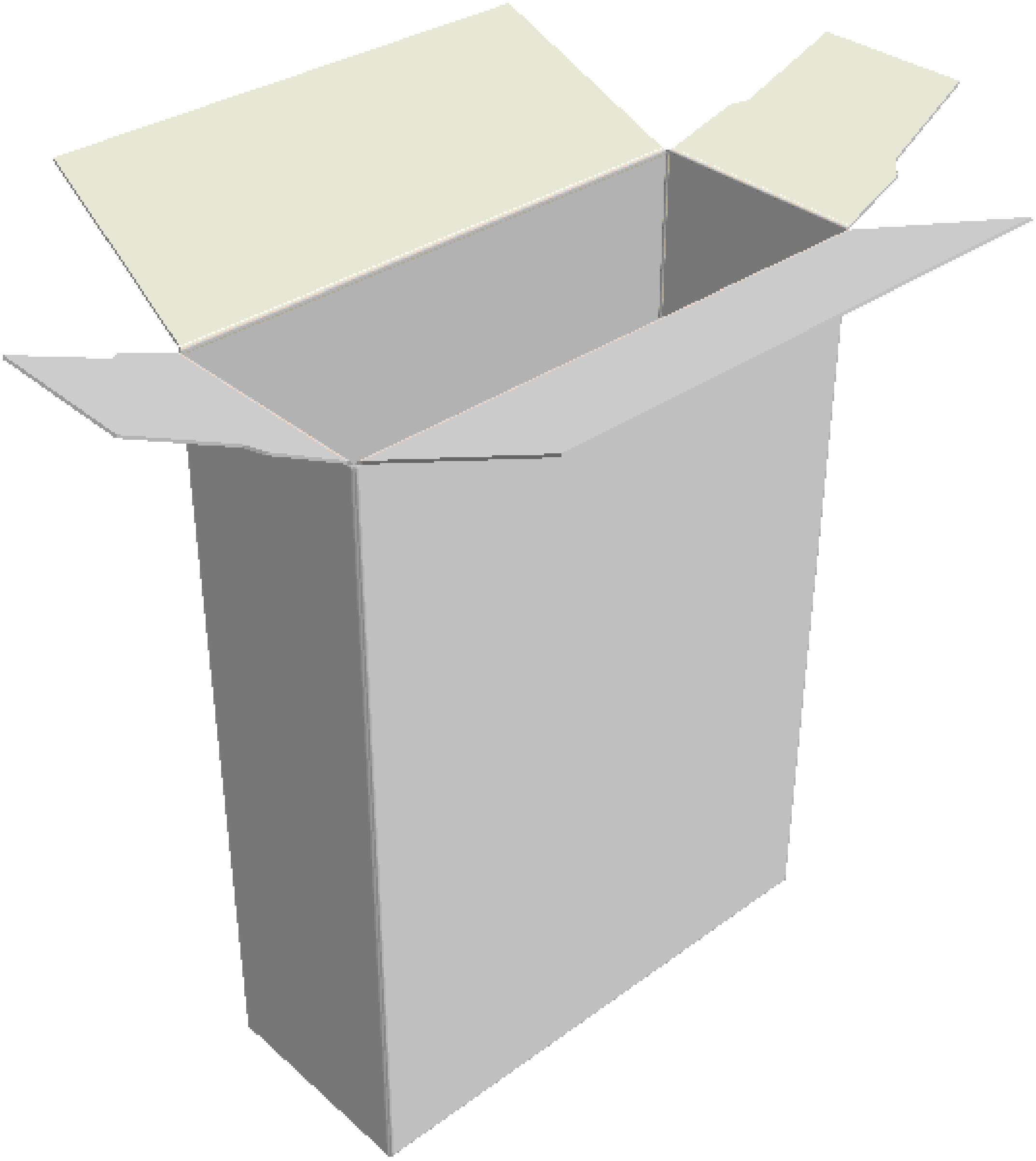
Once you have created the artwork panels as desired, click **OK** on the status bar to finalize them. ArtiosCAD changes all the panel edges to Copy area lines.

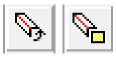
Note: **Artwork Panels** is limited to 20 panels per use, which should be enough for most designs. If you need more, use the tool again to manually add the extra panels.

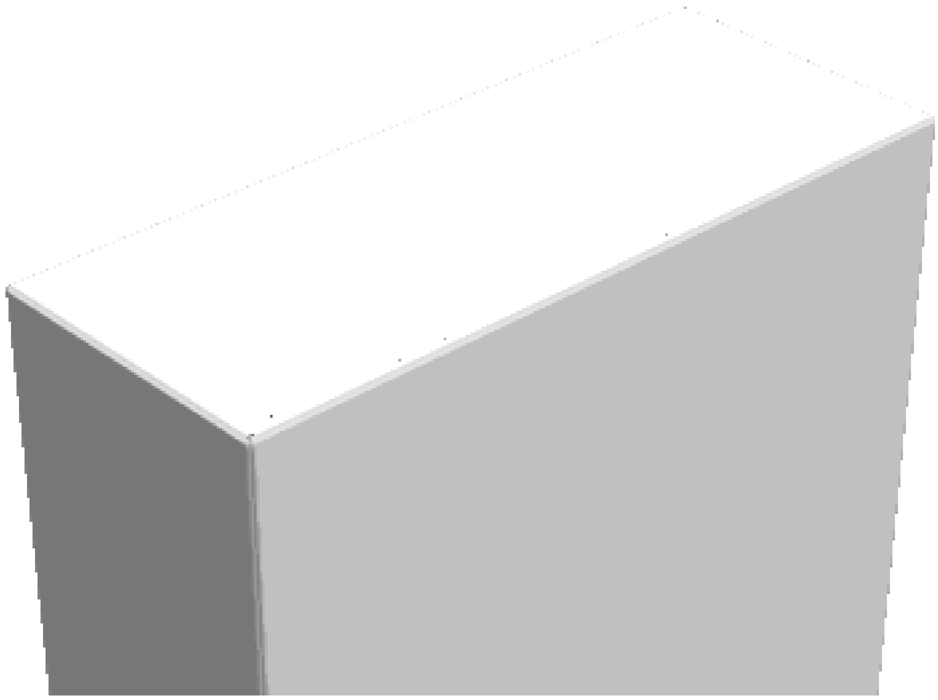
Creating Artwork Panels from 3D Information

The quickest way to create artwork panels is to fold the design in 3D, set the flap priority and orientation, and export the 3D information to the 2D workspace.

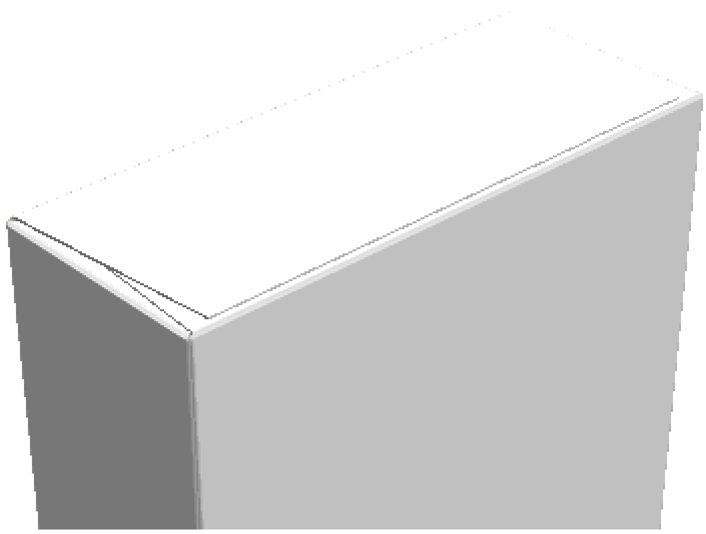
1. Create or restore a design and make sure that for any non-folding creases or perfs, their 3D property is set to **Tear** or **Indent only**. Otherwise, **Artwork Panels** will later create extra panels.
2.  Convert the design to 3D. With the box open like this, ArtiosCAD



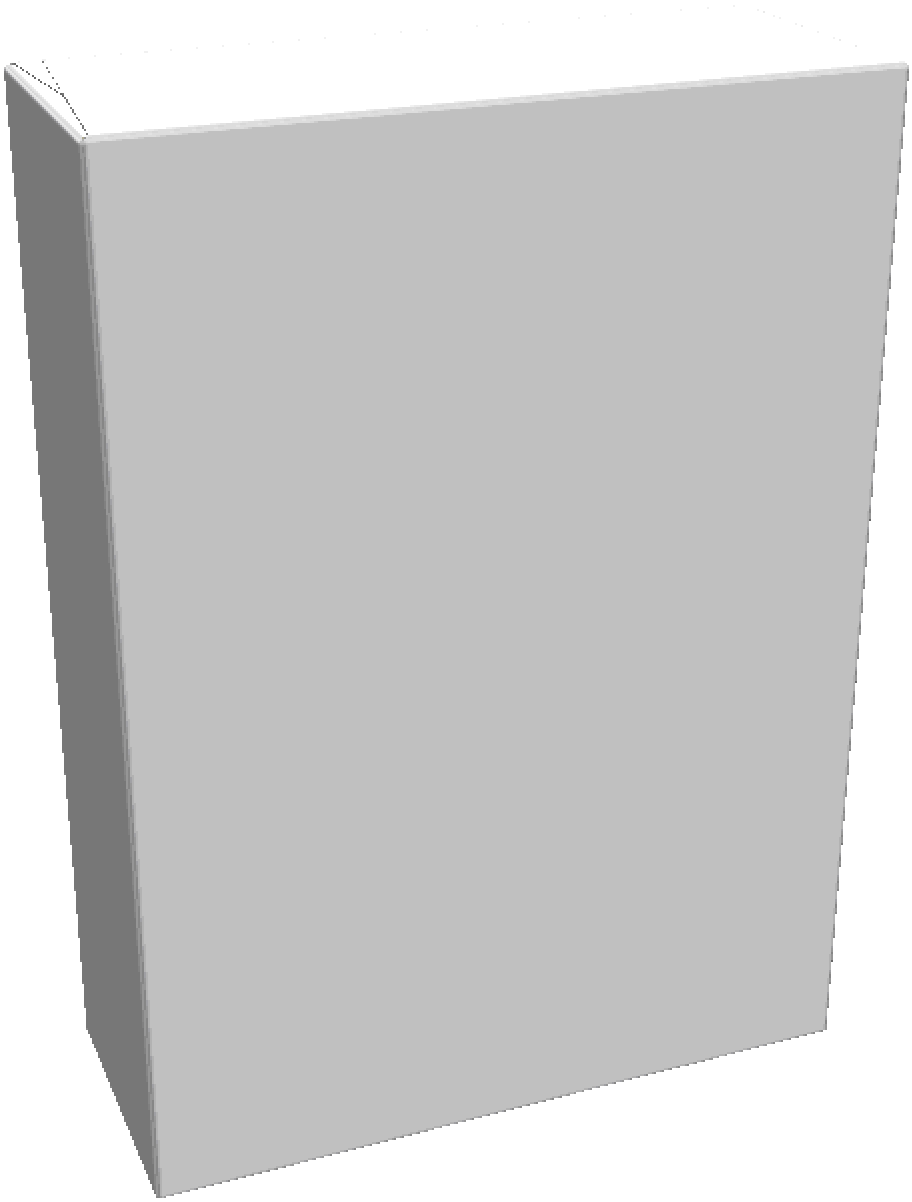
3.  Use one of the **Fold** tools to close the design.



4. Use **Flap Priority** to set the order of the flaps that are visible. However, at this view angle, ArtiosCAD

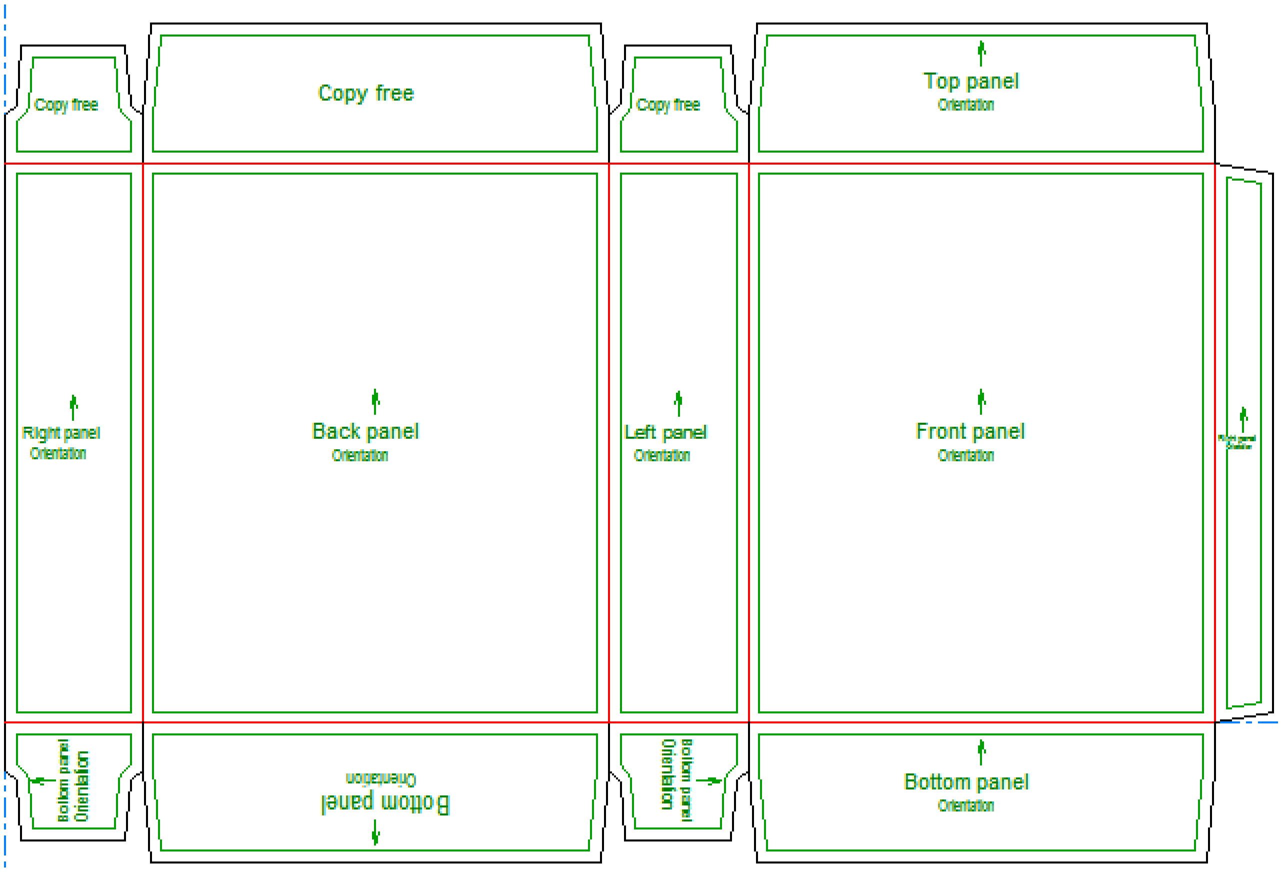


5. Use **View Angle** to position the design so that the order of panels is not ambiguous. There is no doubt about which is the front panel in the example shown below.



6.  Click **Update 2D** to send the 3D panel information back to the single design workspace.

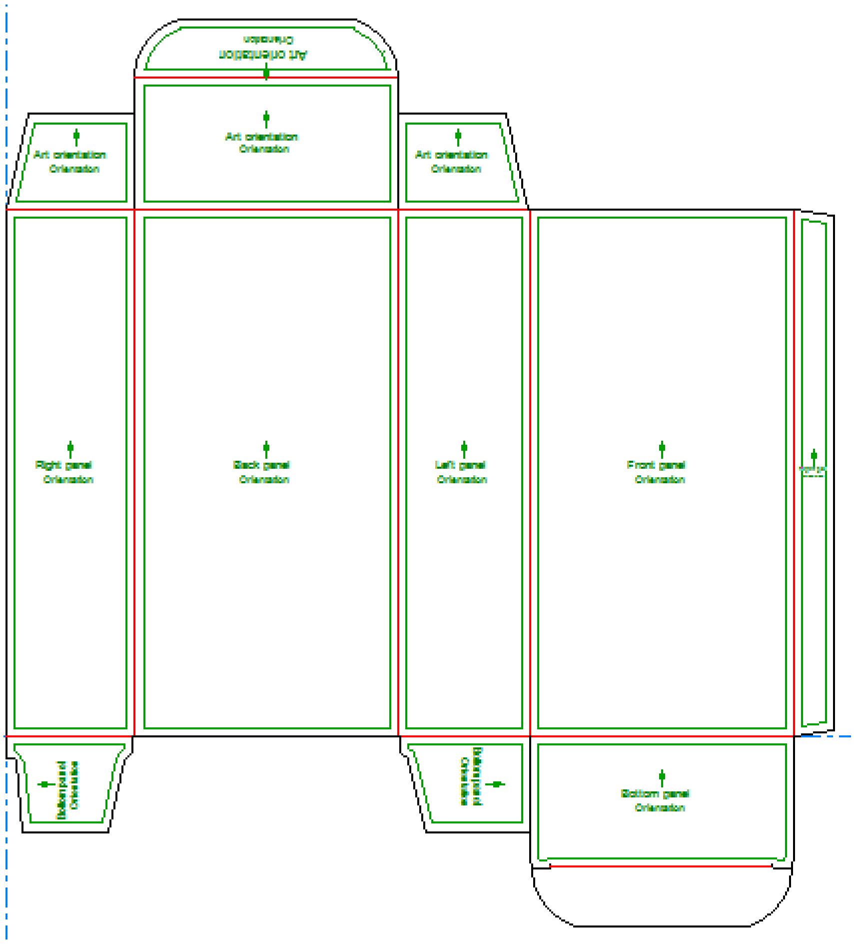
7.  Switch back to the single design and click **Artwork Panels** to start the process of labeling the panels as described previously. ArtiosCAD puts temporary labels in the panels for you to finalize before clicking **OK**.



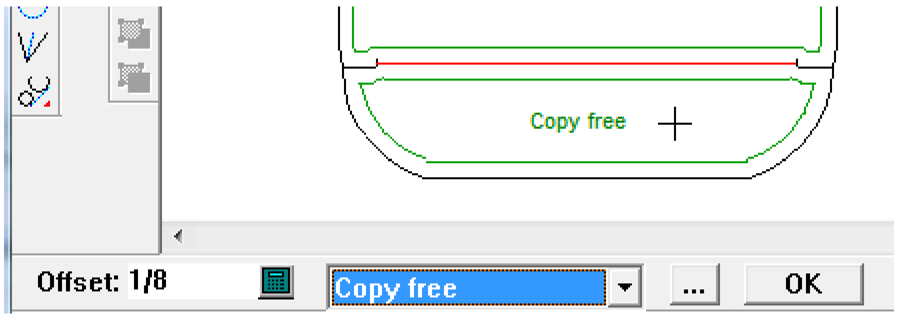
Creating Artwork Panels Manually

To create artwork panels manually, do the following:

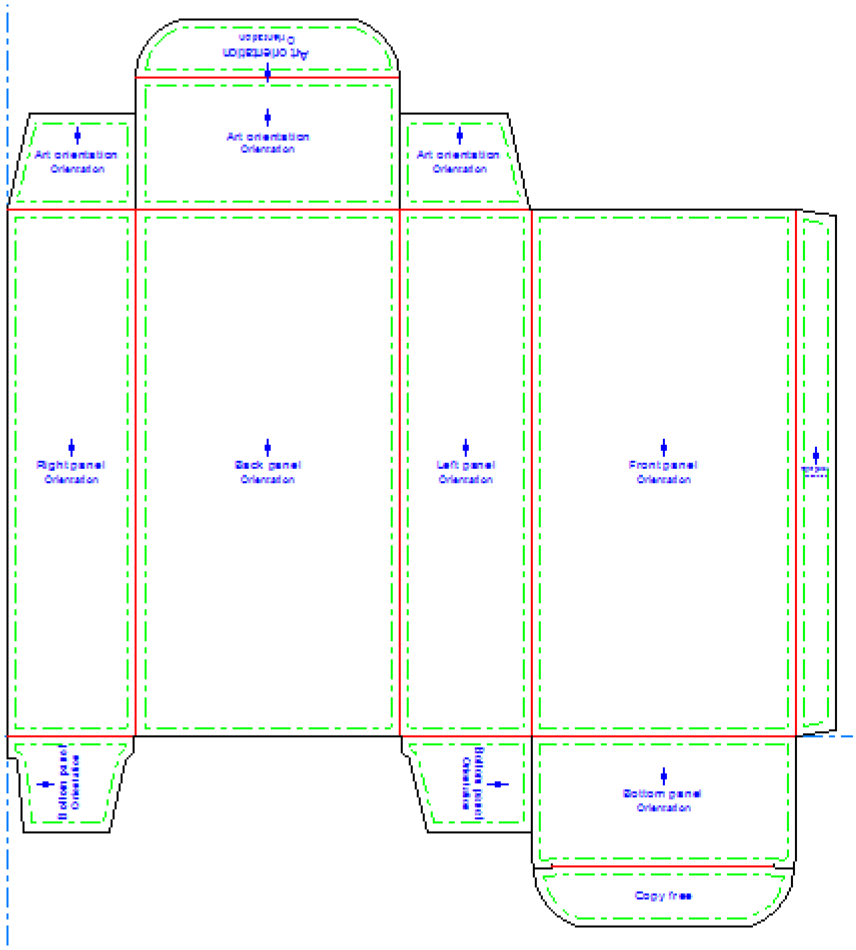
1. Create a new design or restore an existing design.
2.  Click **Artwork Panels**. ArtiosCAD labels the panels with its best guesses as to their function.



3. To change a panel, set the controls on the Status bar as desired and then click inside the panel.



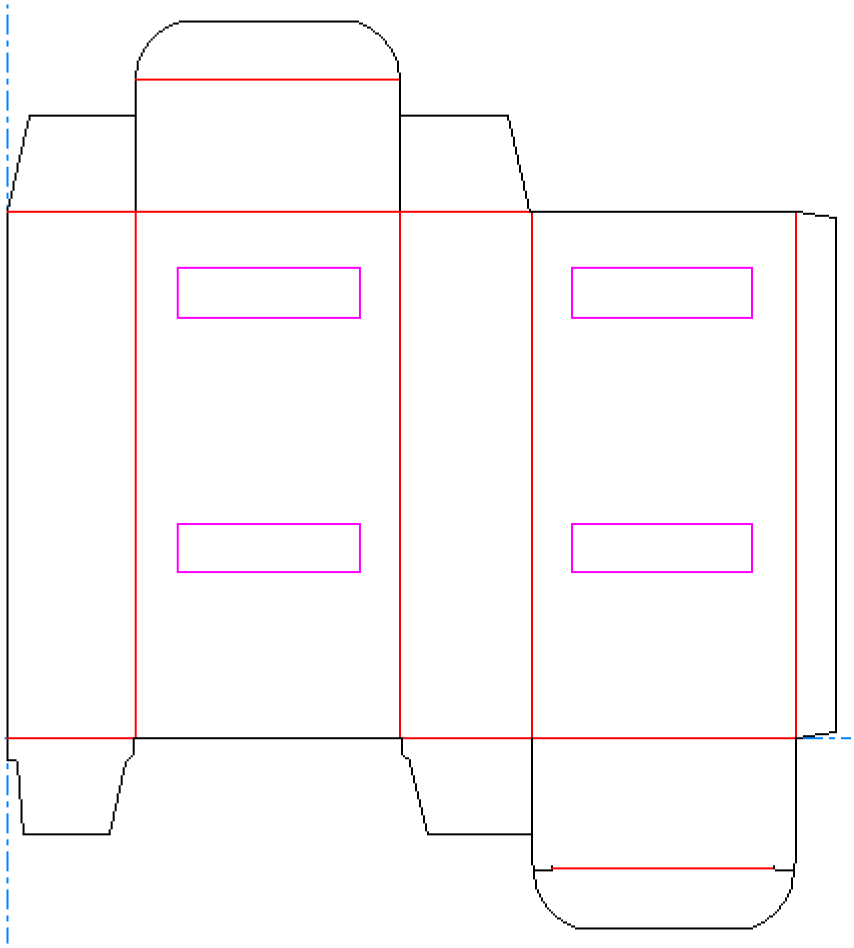
4. Click **OK** when done to finalize the panel labels.



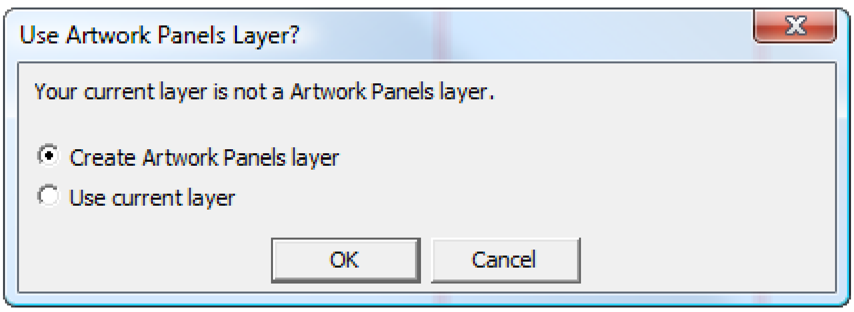
Creating Artwork Panels from Custom Outlines

To create artwork panels from custom outlines, do the following:

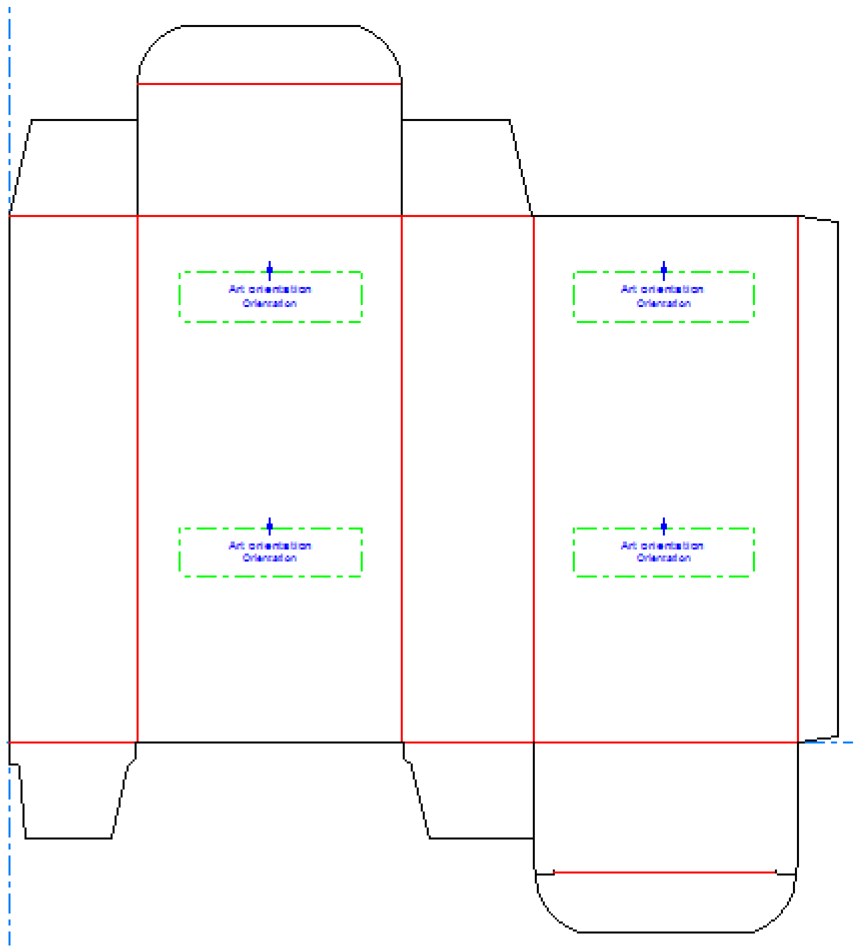
1. Create or restore a design and design the custom outlines. The outlines must form a loop.
2. Select the outlines.



3.  Click **Artwork Panels**.
4. Click **OK** to create an Artwork Panels layer.



5. Once you click **OK**, ArtiosCAD creates the panels.



6. If you want to modify the panels, click **Artwork Panels** again.
7. Click **No** when prompted to clear the current Artwork Panels layer.
8. Use the tool as desired to modify the panels, clicking **OK** on the Status bar when done to finalize the changes.

Dynamic Art

Dynamic art is database-driven artwork that graphic designers add to designs using other Esko software. The **Dynamic Art** tool in ArtiosCAD lets you designate placeholder rectangles for that artwork.

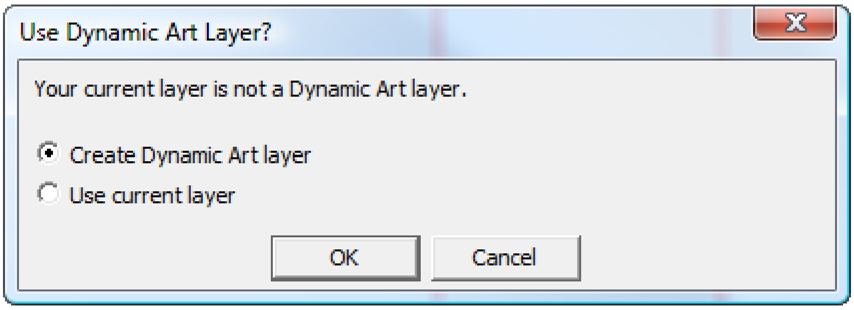
In ArtiosCAD, Dynamic Art rectangles have a copy type and sometimes a preview image. The four types of Dynamic Art are:

1. **Copy**. Used for text such as Product Name or Net Weight.
2. **Barcode**. Used for one- or two-dimensional bar codes.
3. **Symbol**. Used for symbols and images such as recycle or hazard symbols, or product logos.
4. **Table**. Used for tabular data such as Nutrition Facts tables.

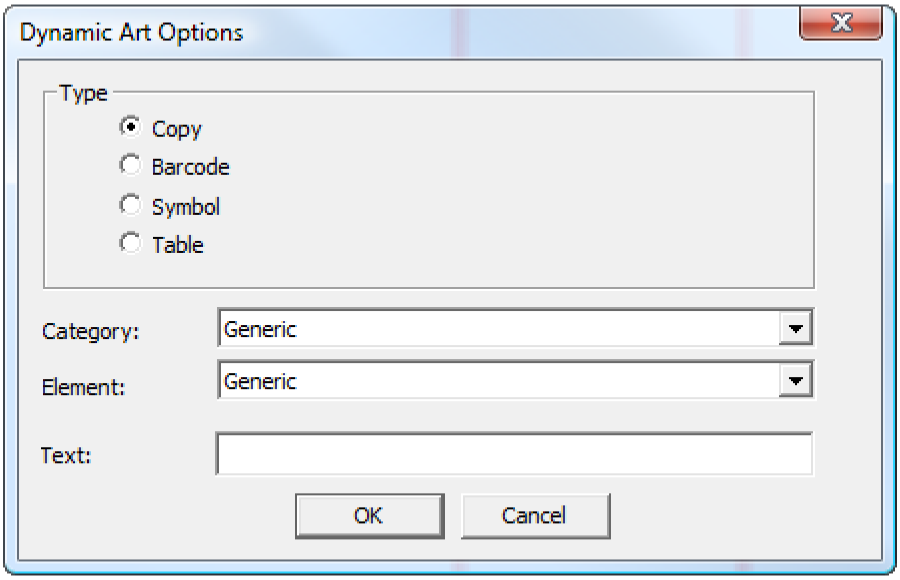
Dynamic Art definitions are derived from the GS1 global business standard.

Using the Dynamic Art Tool

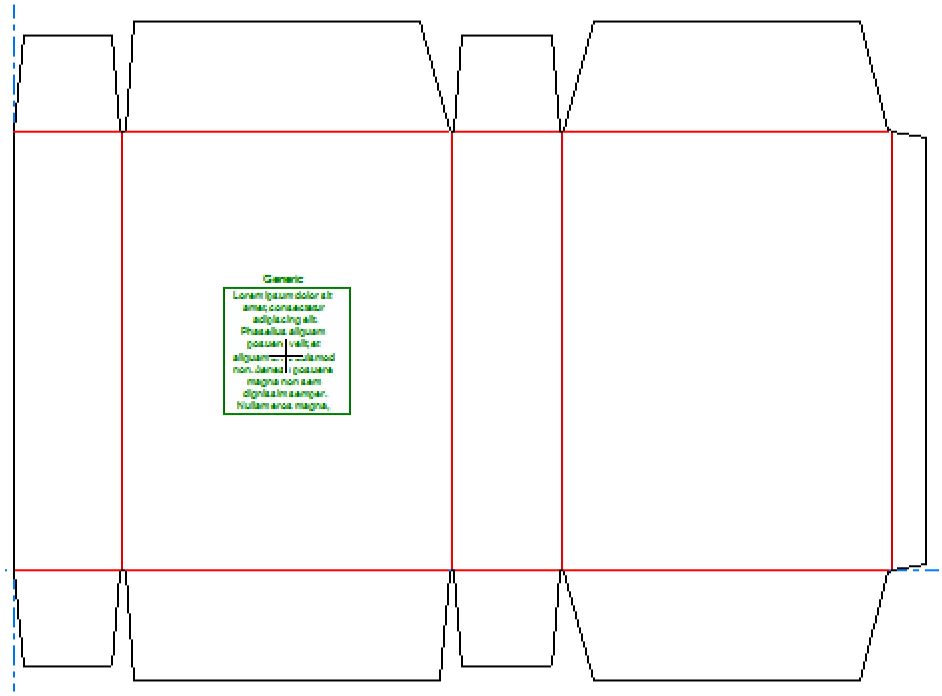
When you click **Dynamic Art**, ArtiosCAD first prompts you to create an Dynamic Art layer if one does not already exist. Click **OK** to create the layer.



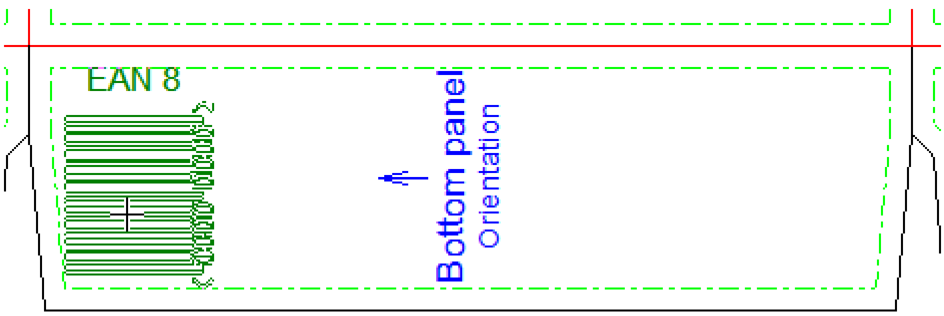
The Dynamic Art Options dialog box appears in which you select the type of Dynamic Art placeholder to create and set its options. Change the **Type**, **Category** (available only for Copy), **Element**, and **Text** as desired and click **OK**.



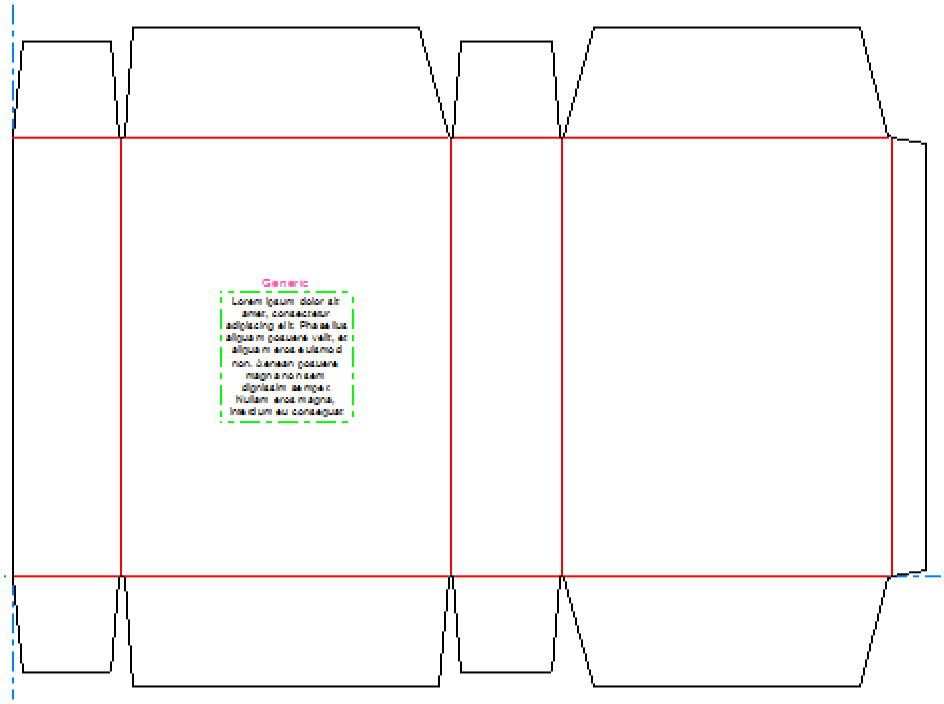
Drag appears for the placeholder. Click inside a panel to set the put-down point. It can be in a corner, the middle of a side, or the center.



If you have an Artwork Panel defined, the orientation of the Dynamic Art placeholder follows the Artwork Panel definition.



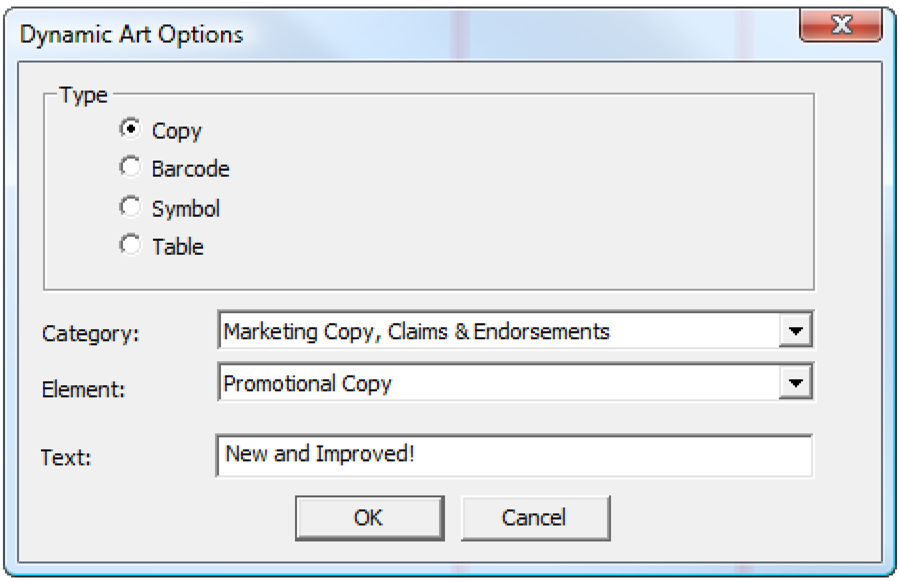
Set the options on the Status bar and click **OK** to finalize the placeholder. When you click inside the **Width** or **Height** field, the size automatically expands to the size of the panel minus the panel offset.



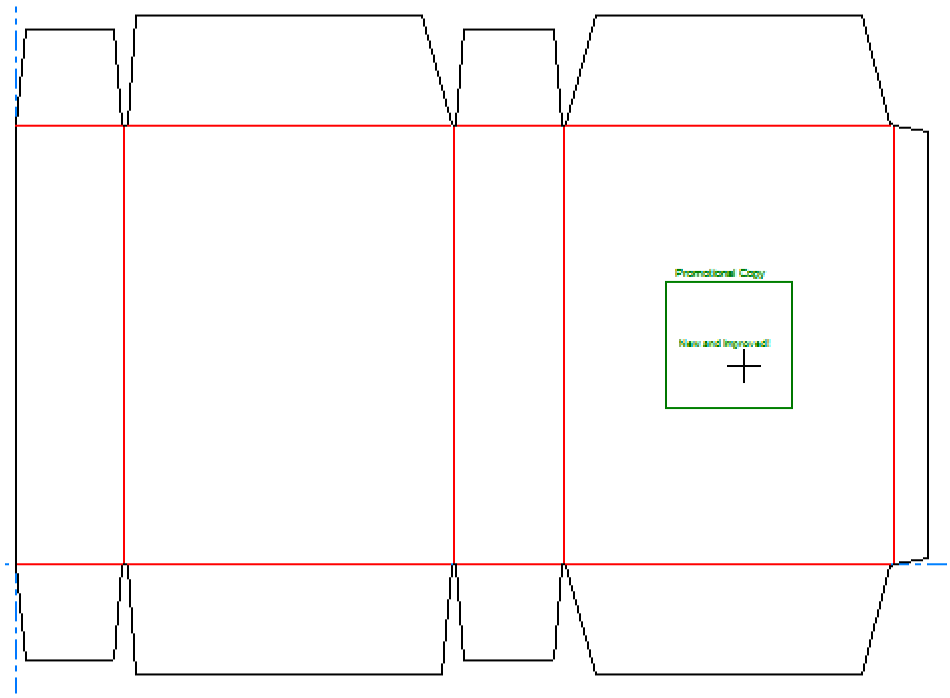
Adding a Copy Placeholder

To add a copy placeholder, do the following:

1.  Click **Dynamic Art** and click **Yes** if ArtiosCAD prompts you to create a Dynamic Art layer.
2. In the Dynamic Art Options dialog box, select **Copy** for the **Type** and choose the desired **Category** and **Element** from the drop-down lists. Enter any desired text in the **Text** field and click **OK**.

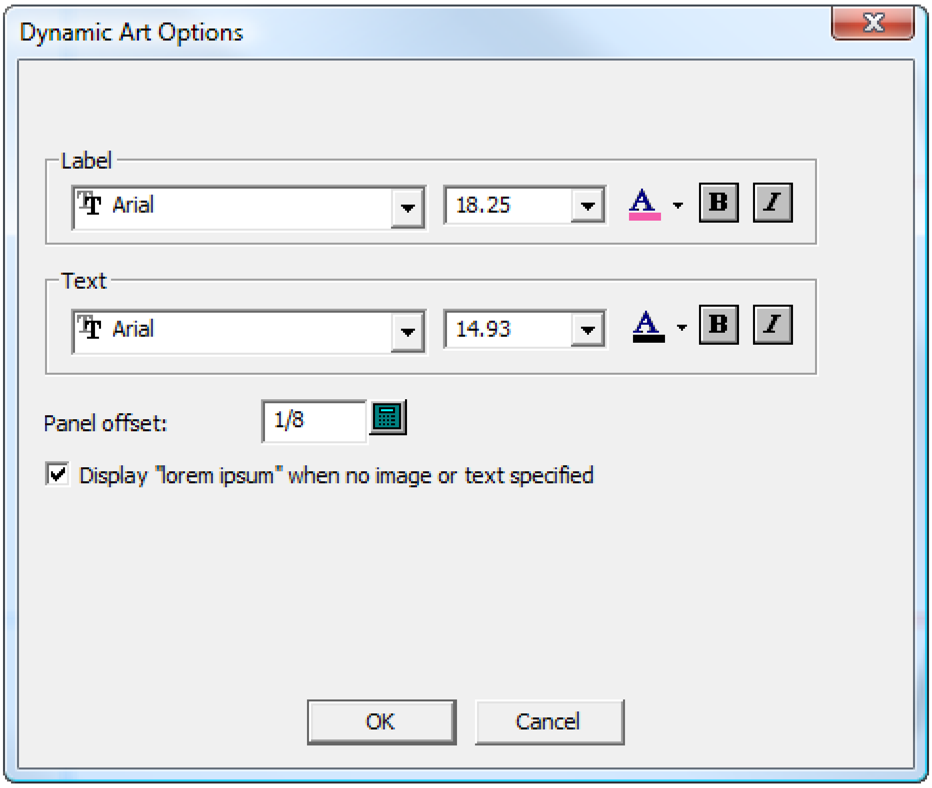


3. Use drag to set the put-down point of the placeholder.

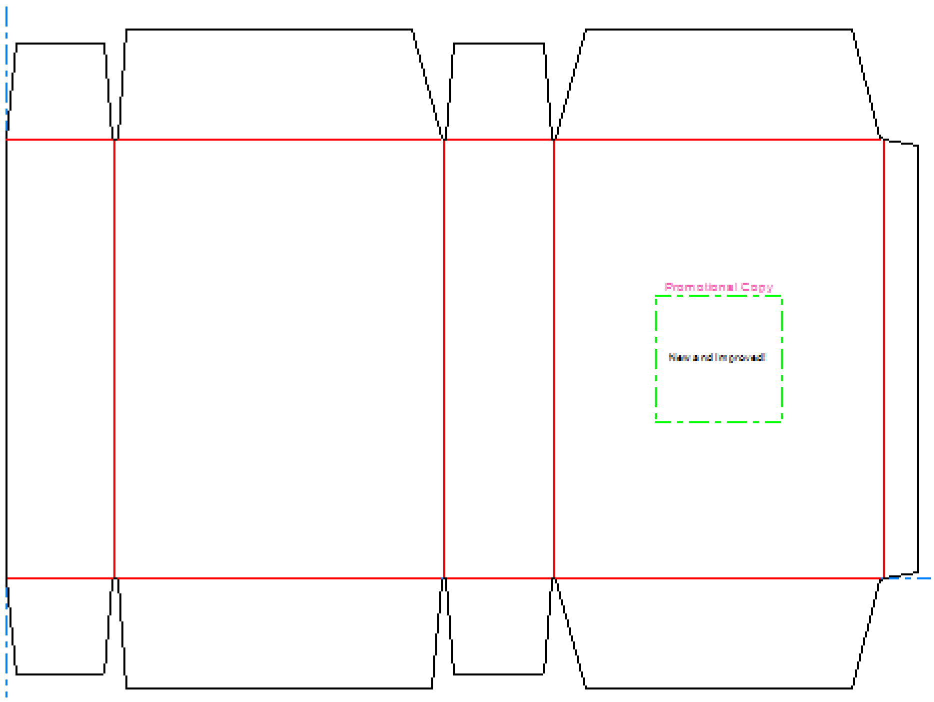


4. Set the fields on the Status bar as desired. **Width** and **Height** control the width and height of the placeholder. **Text** is the same text you entered in the Dynamic Art Options dialog box in the previous step. The directional label controls let you change the position of the placeholder's label.

Clicking More Options (**...**) opens the Dynamic Art Options dialog box. There are standard controls for font, color, and appearance. **Panel offset** controls the gutter between the placeholder and the panel edge. **Display "lorem ipsum" when no image or text specified** supplies contents if you don't enter any. When you are done setting options, click **OK**.



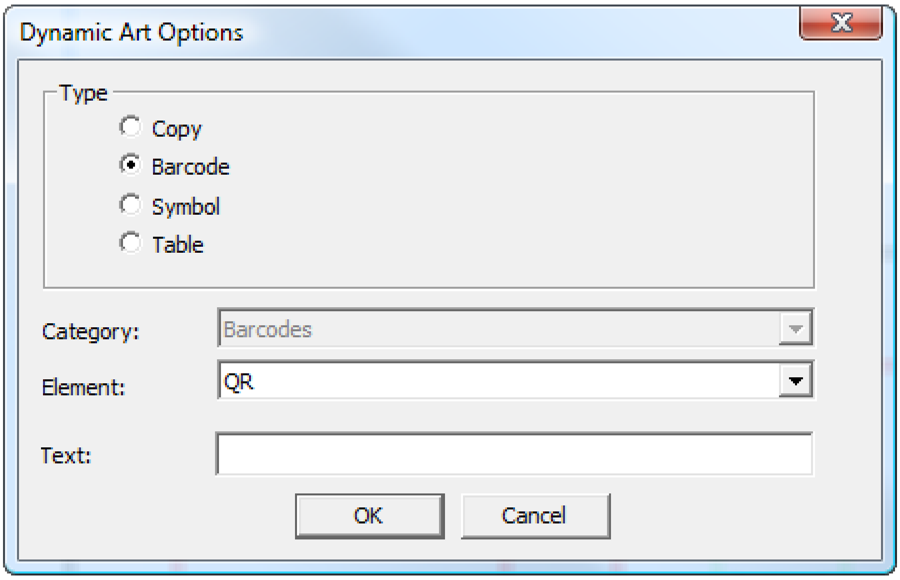
5. Click **OK** on the Status bar to finalize the copy placeholder.



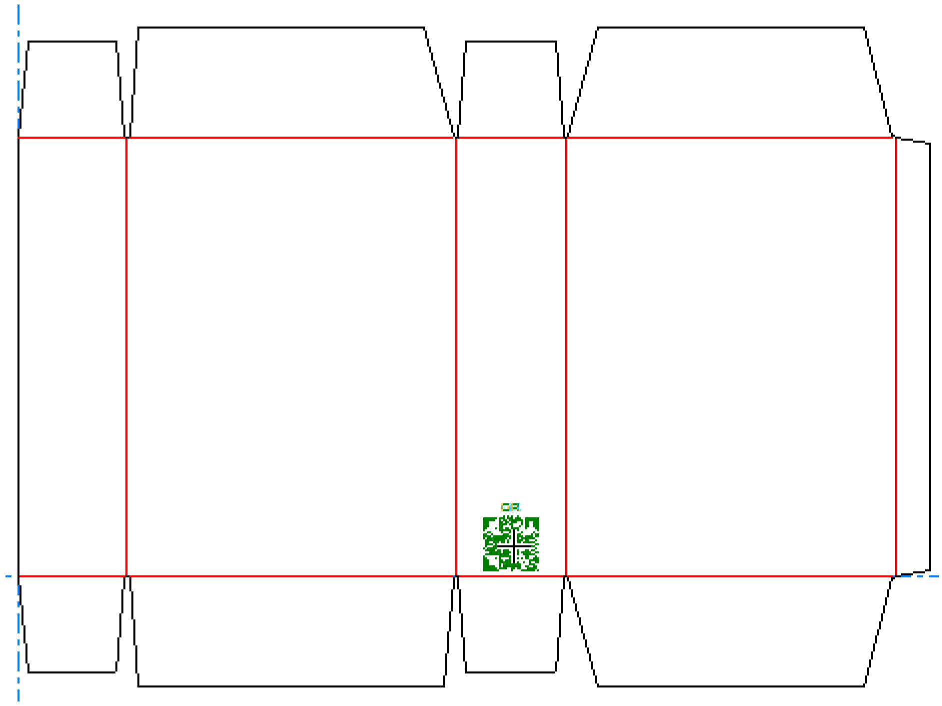
Adding a Barcode Placeholder

To add a barcode placeholder, do the following:

1.  Click **Dynamic Art** and click **Yes** if ArtiosCAD prompts you to create a Dynamic Art layer.
2. In the Dynamic Art Options dialog box, select **Barcode** for the **Type** and choose the desired **Element** from the drop-down list. Enter any desired text in the **Text** field and click **OK**.

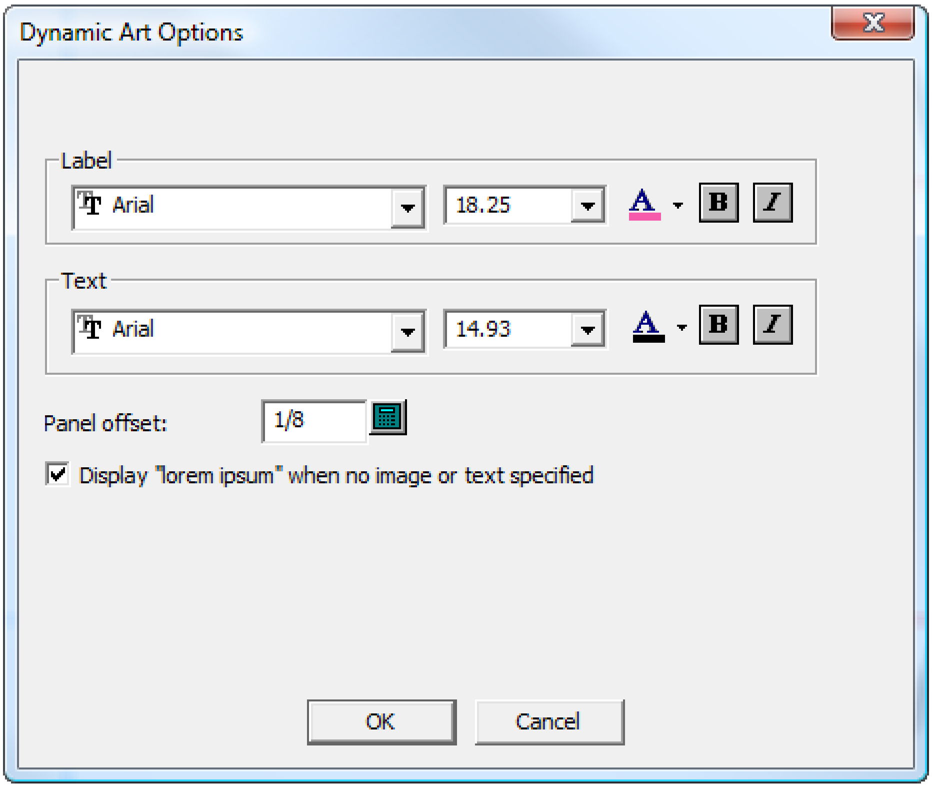


3. Use drag to place the put-down point of the placeholder.



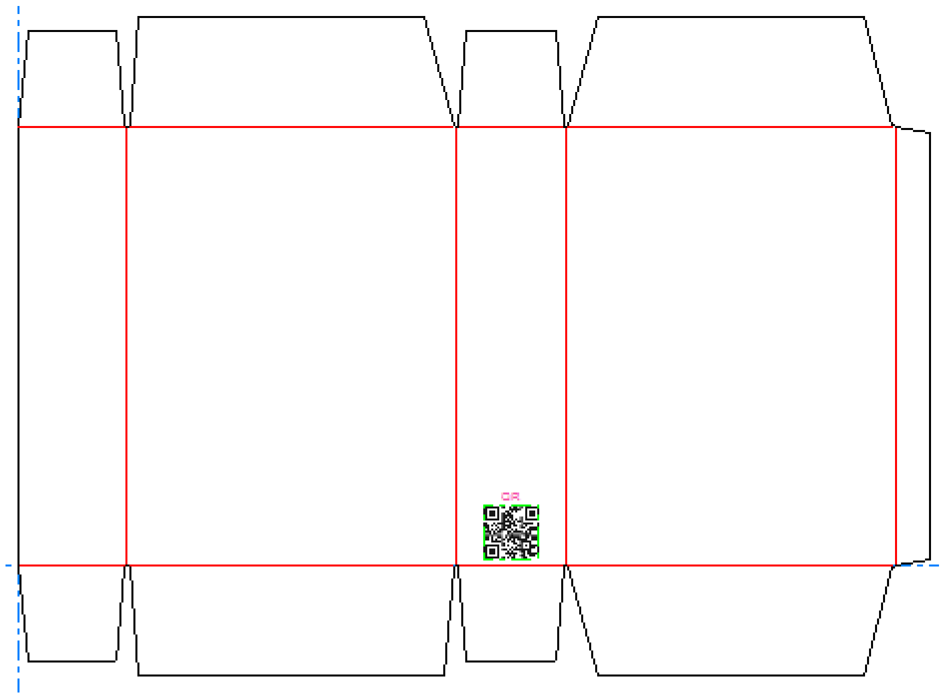
4. Set the fields on the Status bar as desired. **Scale** lets you set the scale of the placeholder. **Width** and **Height** control the width and height of the placeholder. **Text** is the same text you entered in the Dynamic Art Options dialog box in the previous step. The directional label controls let you change the position of the placeholder's label.

Clicking More Options (**...**) opens the Dynamic Art Options dialog box. There are standard controls for font, color, and appearance. **Panel offset** controls the gutter between the placeholder and the panel edge. **Display "lorem ipsum" when no image or text specified** supplies contents if you don't enter any. When you are done setting options, click **OK**.



The Dynamic Barcodes plug-in will draw a barcode at its default size in Adobe Illustrator regardless of the scale or size specified for the placeholder in ArtiosCAD.

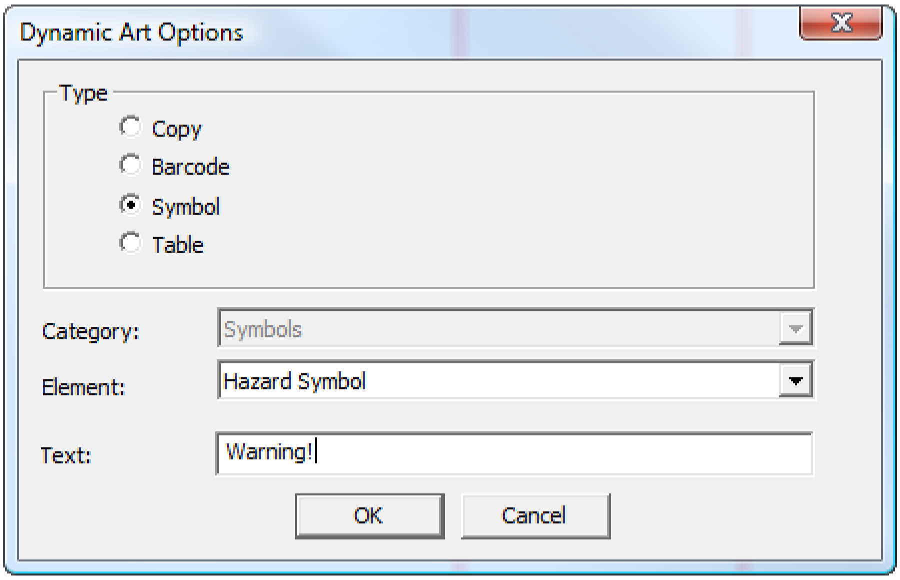
5. Click **OK** on the Status bar to finalize the barcode placeholder.



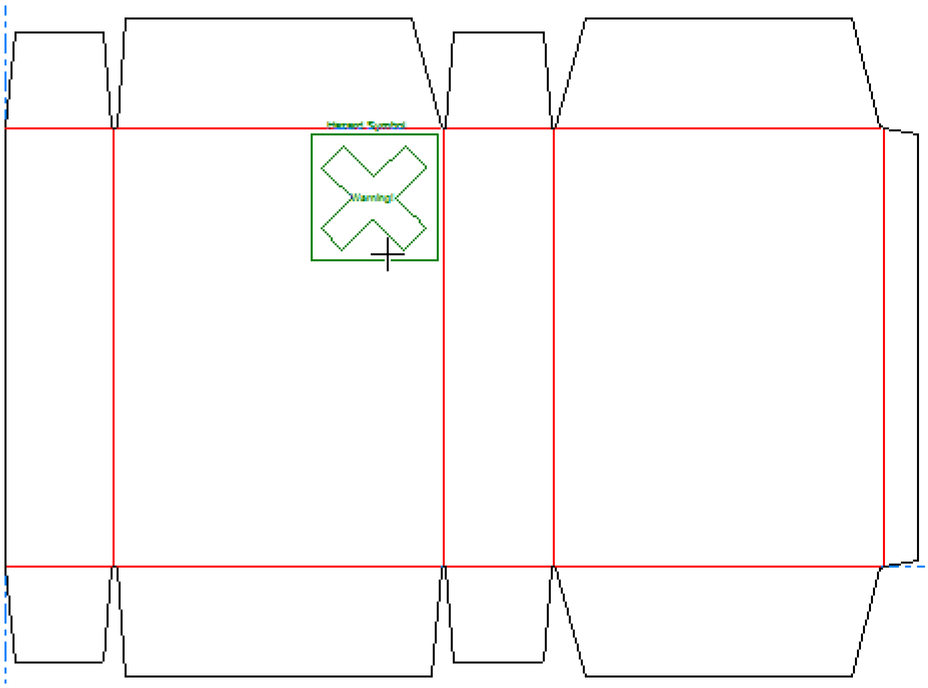
Adding a Symbol Placeholder

To add a symbol placeholder, do the following:

1. Click **Dynamic Art** and click **Yes** if ArtiosCAD prompts you to create a Dynamic Art layer.
2. In the Dynamic Art Options dialog box, select **Symbol** for the **Type** and choose the desired **Element** from the drop-down list. Enter any desired text in the **Text** field and click **OK**.

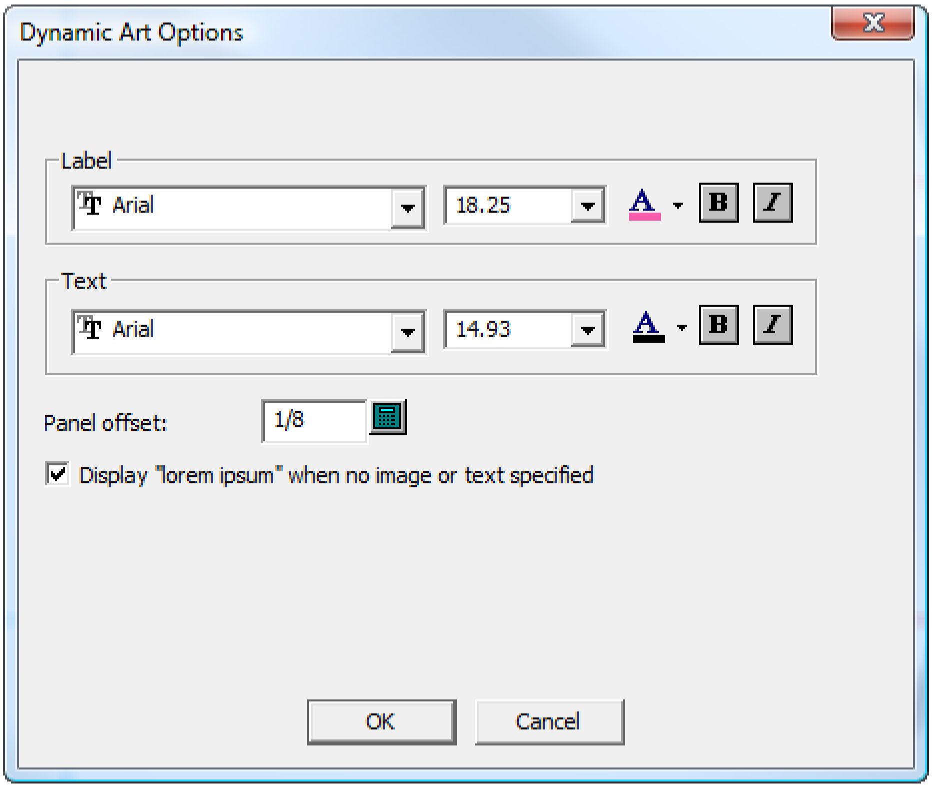


3. Use drag to place the put-down point of the placeholder.

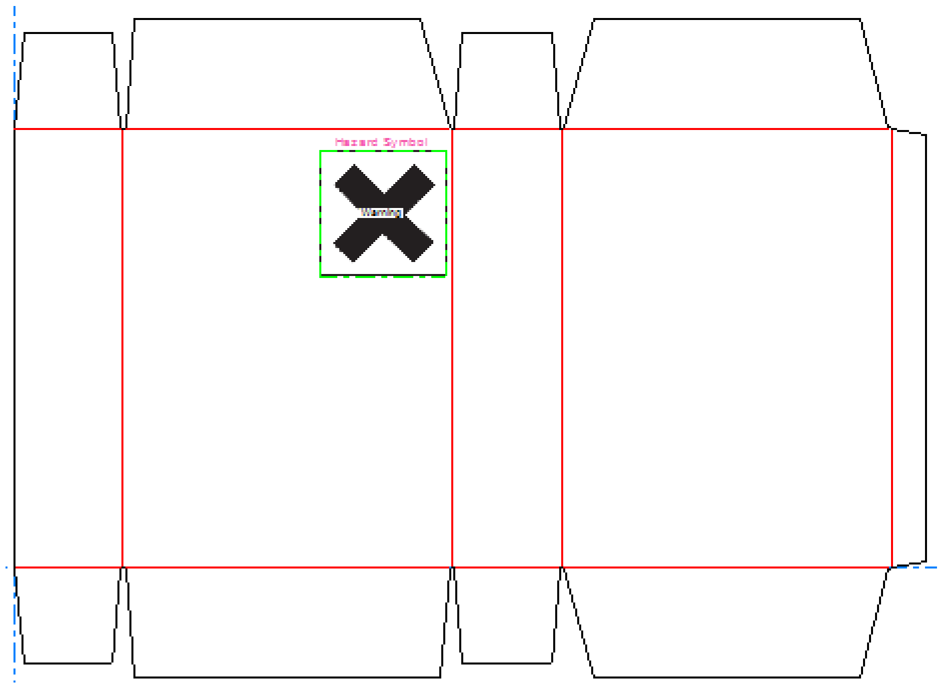


4. Set the fields on the Status bar as desired. **Width** and **Height** control the width and height of the placeholder. **Text** is the same text you entered in the Dynamic Art Options dialog box in the previous step. The directional label controls let you change the position of the placeholder's label.

Clicking More Options (**...**) opens the Dynamic Art Options dialog box. There are standard controls for font, color, and appearance. **Panel offset** controls the gutter between the placeholder and the panel edge. **Display "lorem ipsum" when no image or text specified** supplies contents if you don't enter any. When you are done setting options, click **OK**.



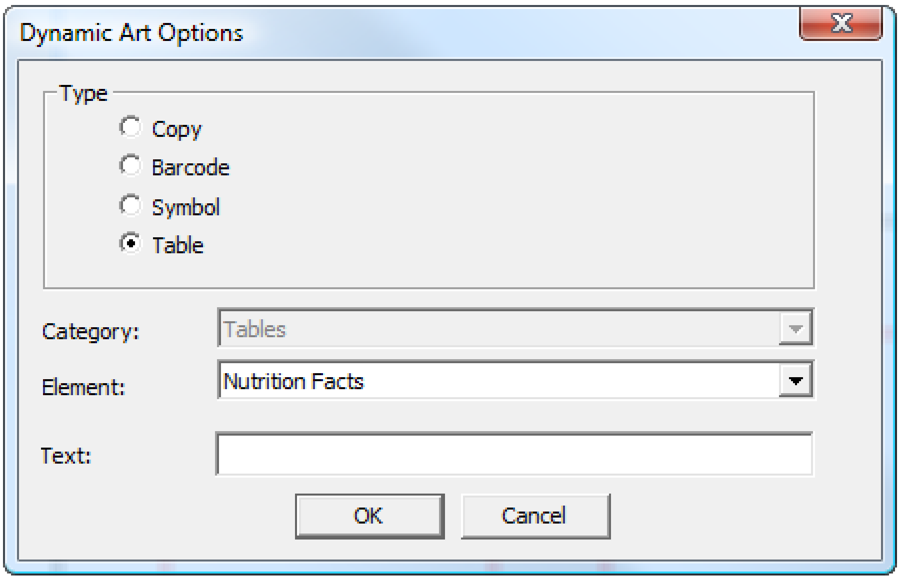
5. Click **OK** on the Status bar to finalize the symbol placeholder.



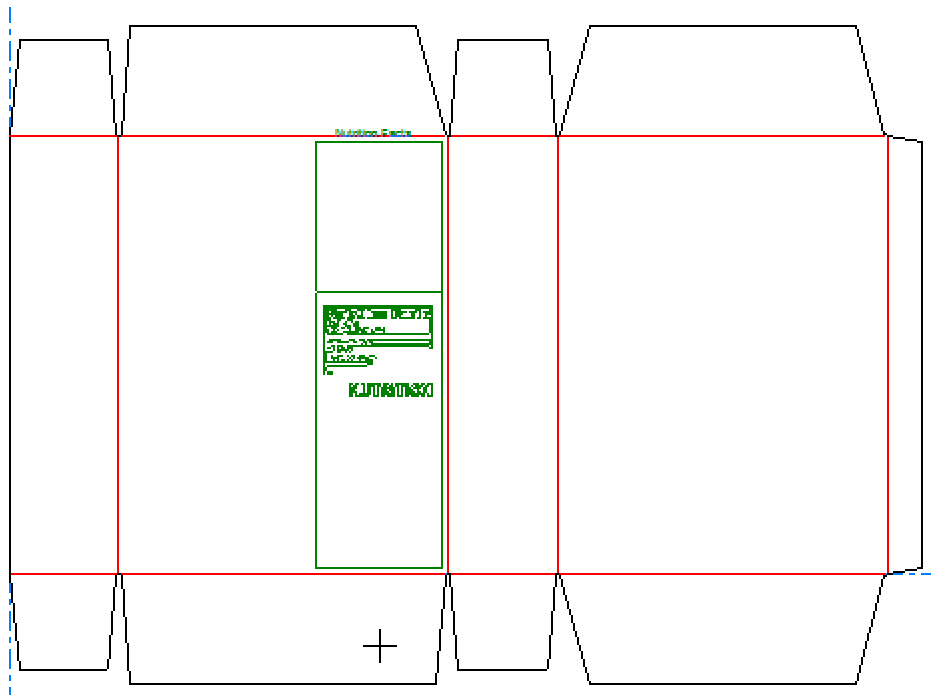
Adding a Table Placeholder

To add a table placeholder, do the following:

1. Click **Dynamic Art** and click **Yes** if ArtiosCAD prompts you to create a Dynamic Art layer.
2. In the Dynamic Art Options dialog box, select **Table** for the **Type** and choose the desired **Element** from the drop-down list. Enter any desired text in the **Text** field and click **OK**.

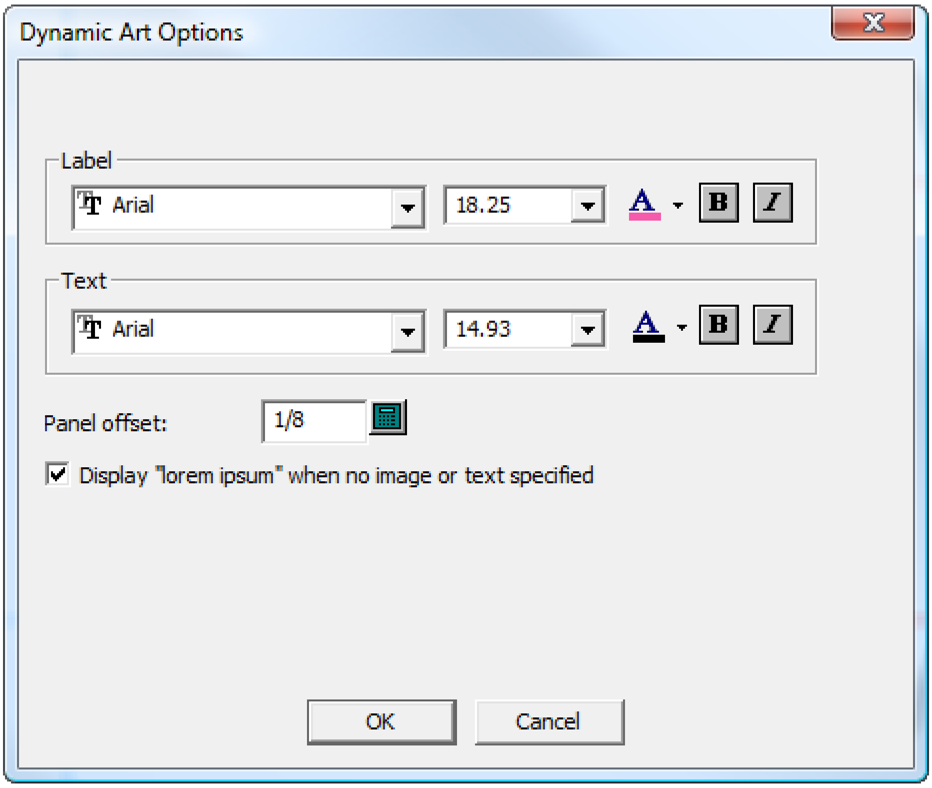


3. Use drag to place the put-down point of the placeholder.

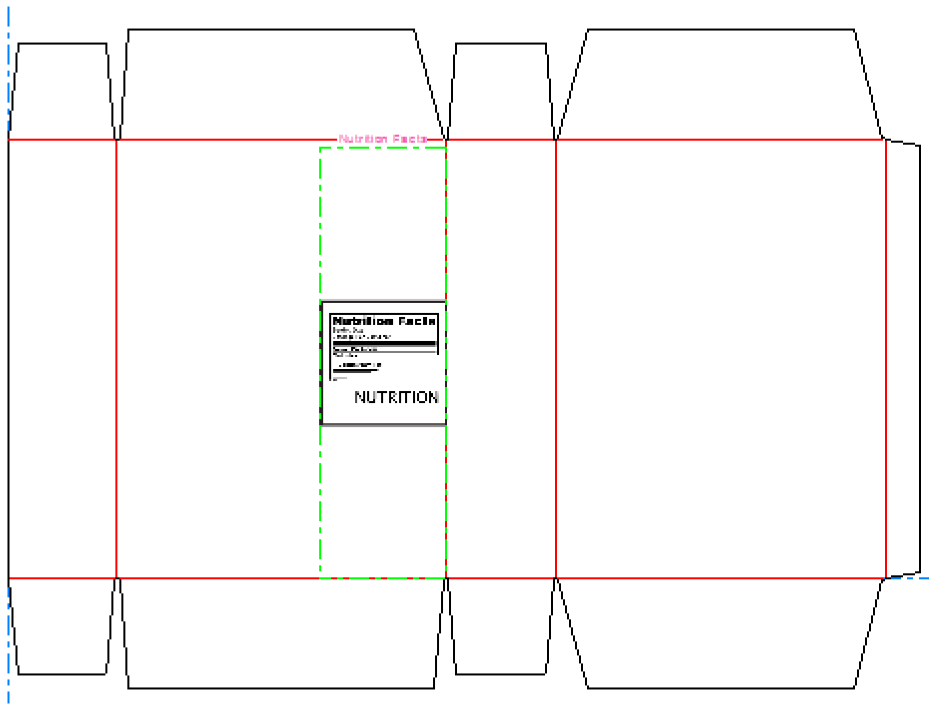


4. Set the fields on the Status bar as desired. **Width** and **Height** control the width and height of the placeholder. **Text** is the same text you entered in the Dynamic Art Options dialog box in the previous step. The directional label controls let you change the position of the placeholder's label.

Clicking More Options (**...**) opens the Dynamic Art Options dialog box. There are standard controls for font, color, and appearance. **Panel offset** controls the gutter between the placeholder and the panel edge. **Display "lorem ipsum" when no image or text specified** supplies contents if you don't enter any. When you are done setting options, click **OK**.

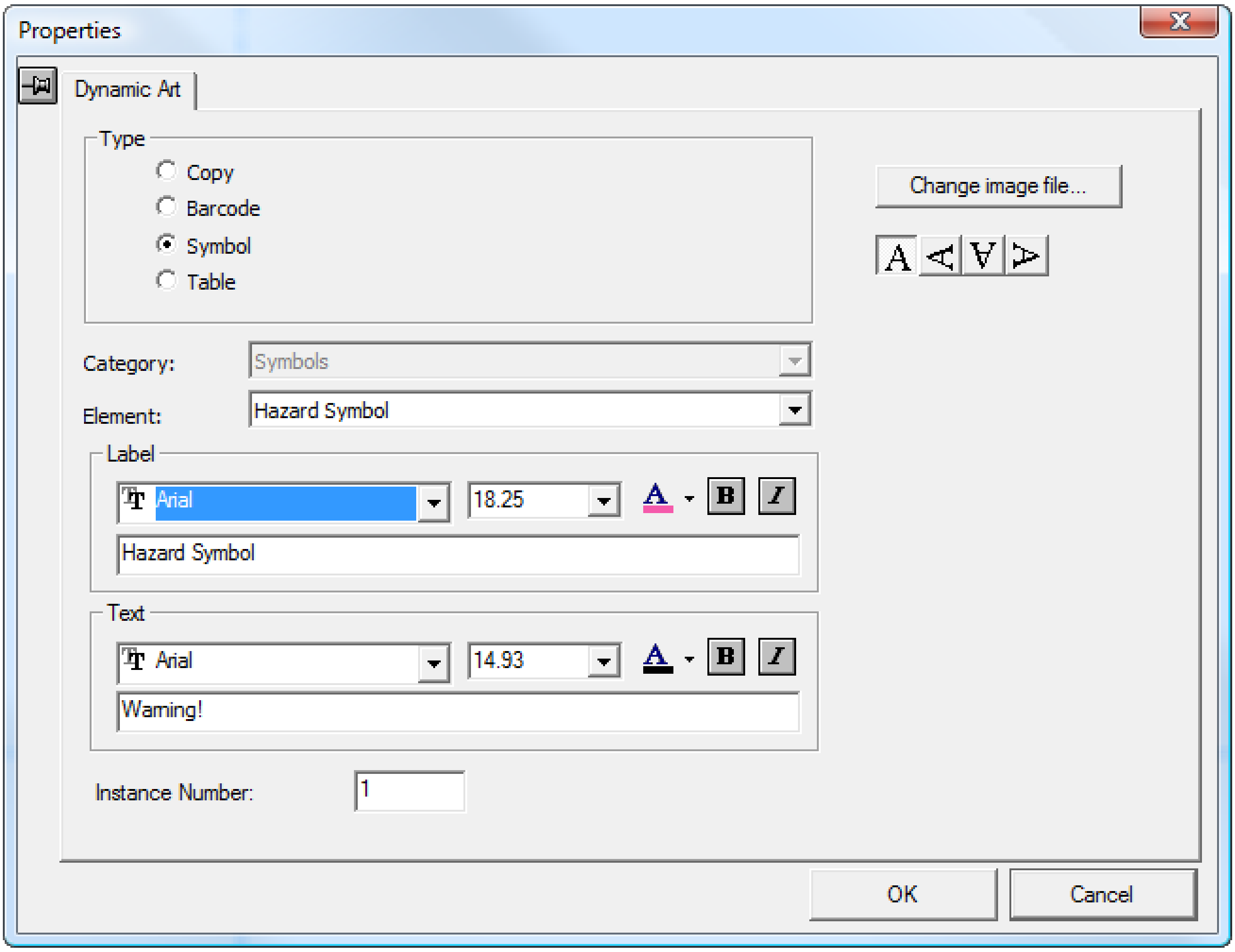


5. Click **OK** on the Status bar to finalize the table placeholder.



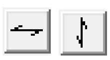
Changing Placeholder Properties

To change the properties of a Dynamic Art placeholder after you have created it, double-click it with the **Select** tool. In the Properties dialog box, you can change all its properties, and change the preview image file (if there is one).

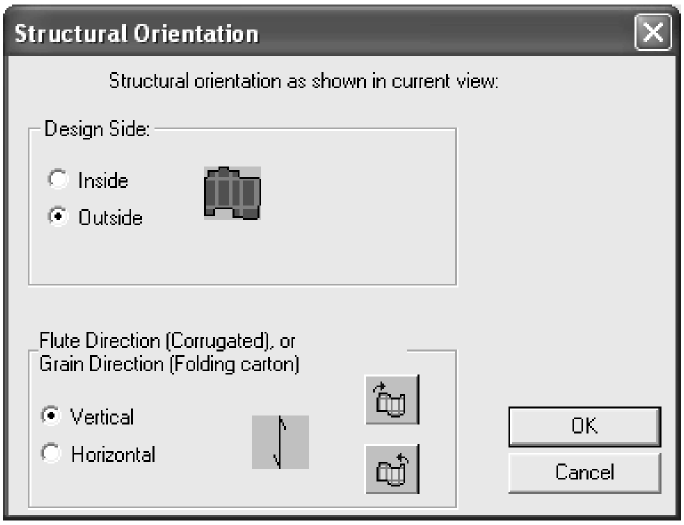


Other functions in Designer

Setting the Grain/Corrugation direction

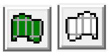
 The **Grain/Corrugation** button on the View bar controls the grain/corrugation direction of the design. A vertical symbol indicates a vertical direction; a horizontal symbol indicates a horizontal direction. Clicking the button opens the Structural Orientation dialog box, where you can set the side of the design and the grain/flute direction. The corresponding buttons on the Toolbar will reflect any

changes you make in this dialog box. Changes made in this dialog box alter the way the current view is interpreted.



Setting the current side of the design

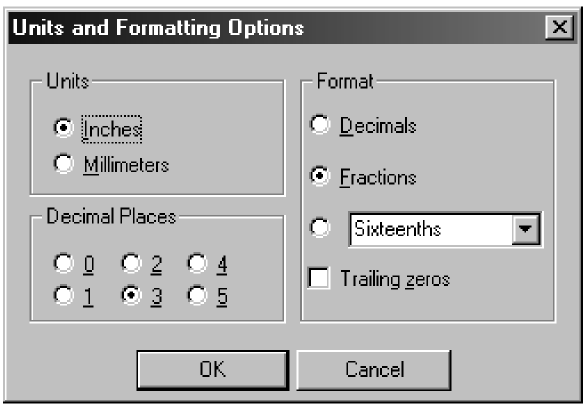
A design has two sides – the printed side and the unprinted side.

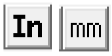
The **Side** button  on the View bar controls which side is up. Green indicates printed side up; white indicates unprinted side up. This button flips the design when clicked. You may have graphics on both sides of the design.

Units and formatting

The **Units and Formatting** command on the **Options** menu controls the calculation, input, and display of numerals. Change the options as desired. In the **Format** group, you can choose between sixteenths and twentieths.

Note: You cannot change between sixteenths and twentieths in the Dimension Properties dialog box.



The Units button 

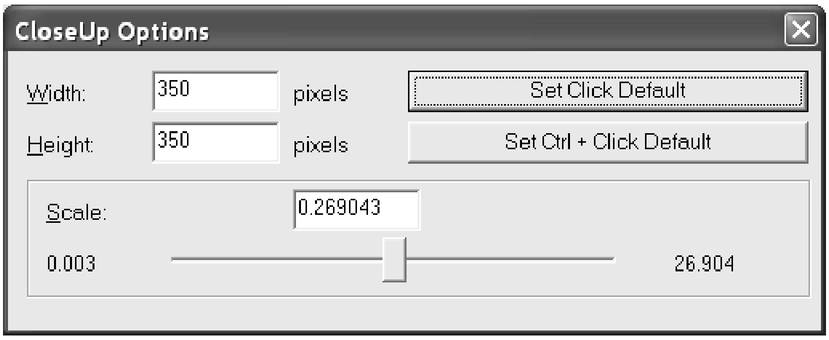
on the View bar changes the units when clicked.

CloseUp windows

Click the middle mouse button to create a CloseUp window. If you don't have a middle mouse button, click **CloseUp** on the View menu or press `CTRL-Space`. Holding down `CTRL` while opening a CloseUp window opens it at a different scale.

A CloseUp window shows a zoomed-in view of the area surrounding the point you clicked. A CloseUp window will change if **Show window contents while dragging** is enabled in the Display Properties of the operating system.

Click the magnifying glass icon on the title bar of the CloseUp window to access its menu. Dynamic controls the real-time updating of the window as you drag it. **Options** opens the CloseUp Options dialog box.



Width and **Height** change the size of the CloseUp window.

Scale and the slider beneath it specify the scale for the CloseUp window.

Set Default makes the values you set the values that are used for the remainder of the current ArtiosCAD session for regular CloseUp windows. **Set Ctrl + Click Default** sets the session default for CloseUp windows opened using the `CTRL` key.

Clicking the middle mouse button while in a CloseUp window closes the current window. `SHIFT-middle mouse button` closes all CloseUp windows, as do `SHIFT-Spacebar` and **Close All CloseUps** on the View menu.

Press `CTRL-A` to zoom out in CloseUp windows.

CloseUp window defaults are set in **Options** > **Defaults** > **Shared Defaults** > **Startup defaults** > **View tools options**.

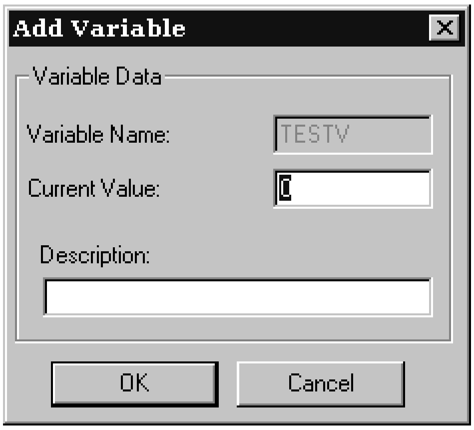
StyleMaker

StyleMaker is an optional feature of ArtiosCAD that allows you to make your designs once and rebuild them with different measurements an infinite number of times.

StyleMaker works by using variables instead of absolute values. *Variables* are ways to store pieces of data that can change in the future. In ArtiosCAD variable names must start with a letter and can be up to 6 characters long. Only use uppercase letters and numbers for the variable name.

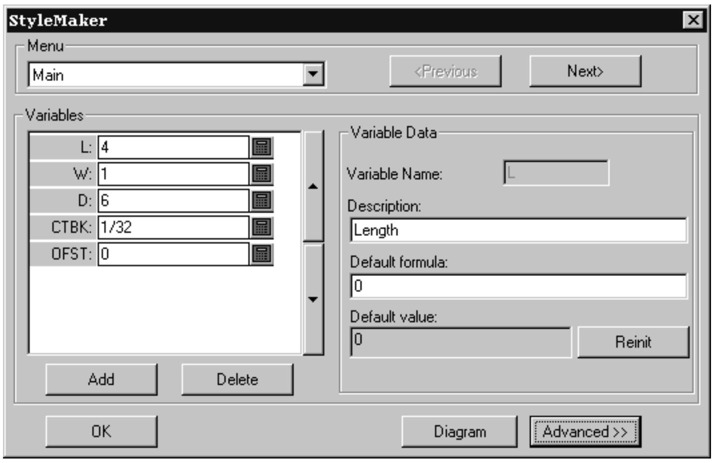
Manipulating variables

There are two ways to define variables – by entering them on the fly in Status bar prompts, or by clicking **StyleMaker** on the Design menu. If you enter a variable on the fly, and have not defined the variable whose name you typed, ArtiosCAD asks you if you want to define the variable now. If you do, you are taken to the **Add Variable** dialog box.



Enter a value to use for the variable in the **Current Value** field and enter the name of the variable in the **Description** field.

When you click **StyleMaker** on the **Design** menu, the StyleMaker dialog box opens.



The **Variable Name**: field contains the name of the variable. It is unavailable if you typed the name in the Status bar. The **Description** is what appears in the menu as a ToolTip flyout when you rebuild the design. In the picture shown above, variable L is set to value of 4 with the description of Length. This means that when you are constructing geometry that refers to the length of the design, you can type L to answer prompts instead of 5.

To add a variable, click **Add** in the StyleMaker dialog box, or type its name in a Status bar prompt. All variables you define are shown in the Variable Keypad.

To delete a variable, click **Delete** in the StyleMaker dialog box.

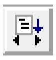
The **Advanced** button in the StyleMaker dialog box leads to the optional Advanced StyleMaker feature.

Rebuilding a design

Rebuilding a design lets you change the values of the variables you have defined. The process is similar to running a standard. Click **Rebuild** on the **Design** menu and enter new values in the fields next to the variable names. When you are done entering new values, click **OK**.

Designs based on a pre-LASERPOINT IQ 2 version of LASERPOINT will be rebuilt using a black text window as they were in LASERPOINT.

Rebuilding a design with Rebuild Playback

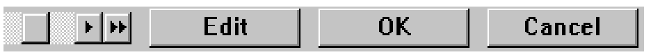


The **Rebuild Playback** tool on the **Undo/Redo** toolbar and the **Design** menu lets you rebuild a design step by step while looking at the design as it is rebuilt. This is a powerful feature. Always save your design before using this command.

Note: Any commands in the Undo/Redo buffer are lost when you enter the Rebuild Playback tool. Perform all desired Undos and Redos before starting this tool.

When you make a rebuildable design, ArtiosCAD records each mouse click, menu command, and tool selection you make. These commands and procedures are stored in the design file itself. ArtiosCAD rebuilds the design by executing these stored commands with the updated variable values that you specify. When you rebuild a design normally, ArtiosCAD takes the new values and completes the new geometry automatically. The **Rebuild Playback** tool provides more control over the rebuild process.

To use this tool, click **Rebuild Playback**. The following tools will appear in the Status bar.



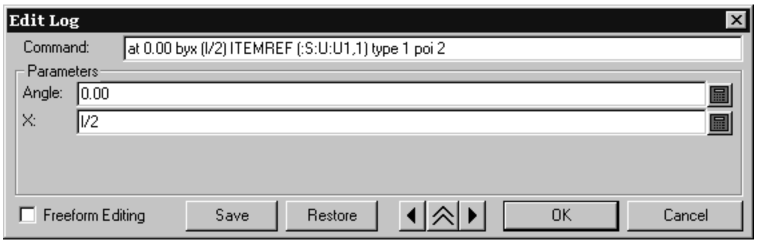
The **Position** scroll bar lets you step through each step of design construction, with the left most end being the unbuilt design, and the right most end being the built design. The single arrow buttons move the progress indicator by one step, while the double arrow buttons move to either the unbuilt design (the arrow pointing to the left) or the built design (the arrow pointing to the right). As you click the arrows, you will see the design being built. You may also drag the scroll bar button to rebuild the design.

The **Edit** button is described in depth in the following section.

Clicking **OK** accepts the changes you made. If the scroll button on the Position slider is not fully to the right, you are asked if you want to finish the design with the rest of the commands as they are, or if you want to ignore them. Choose appropriately and then click **OK**.

Editing the command history

When the **Rebuild Playback** tool is active, the **Edit** button on the Status bar brings up the **Edit Log** dialog box where you can change the commands used to rebuild the design.



The current command is shown in the **Command** field. You can modify the command directly, or you can just change its parameters by entering new values in the fields in the **Parameters** group. Each command has different parameters.

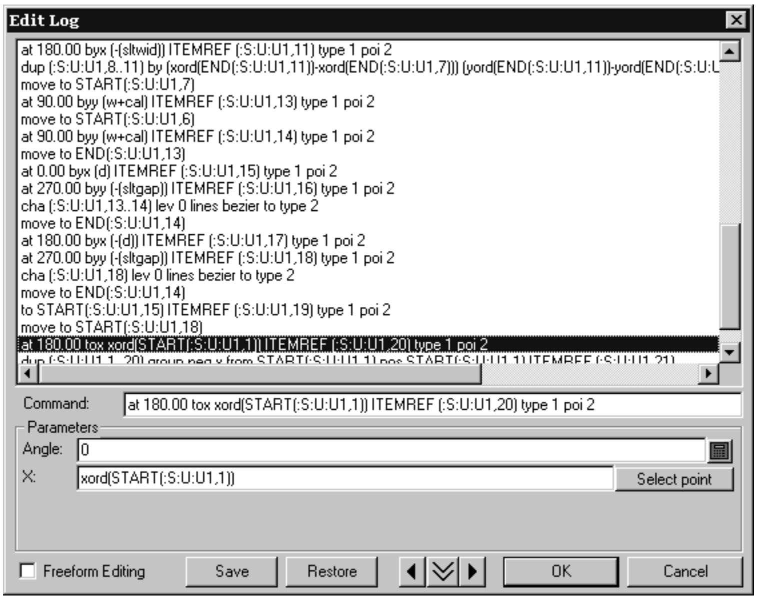
There are three parameter types. The parameter type affects the button at the end of the parameter input field.

 In the example above, the button is the keypad since both parameters are numeric. To change a numeric parameter, enter a new value, enter a variable or expression (you will be prompted to define the variable if it does not exist yet), or use the buttons on the keypad to indicate distances and angles.

When the parameter refers to a line, the button is **Select line**. If you click it, you can then choose a different line to be made with the command. Use this with caution!

When the parameter in the Edit Log dialog box refers to a point, the button is **Select point**, and the point is highlighted in the design area of the ArtiosCAD window. If you click it, you can then choose a different point to be acted on by the command. Use this with caution!

The arrow buttons step through the construction commands just as they do on the Position scrollbar. The double chevron button expands the dialog box to show the actual commands.



To select a command to modify, click it with the mouse, or move to it using the arrow keys on the keyboard.

The **Freeform editing** checkbox toggles the display of the **Command** and **Parameter** groups. Make changes to the commands by selecting them and then typing over them as you would in a word processor. There is a limit of 4096 characters per line. When in Freeform editing mode, the **Update** button updates the design.

The **Save** and **Restore** buttons let you save the commands to a text file and then restore them if desired. This would be convenient in case you made changes you did not mean to make.

Rebuild Playback notes and warnings

The dimension, **Fill**, and **Stroke** tools are not affected by Rebuild Playback.

The `SET`, `DSET`, `IF`, `TRY`, and `LABEL` commands cannot be used in the **Rebuild Playback** tool.

The **Rebuild Playback** tool works only with the commands saved in the design file and commands stored in the **Undo** command buffer. These are set up when the design is first rebuilt. If a restored design has never been rebuilt, starting the **Rebuild Playback** tool will start the rebuilding process first.

Define all desired variables in StyleMaker before running **Rebuild Playback**. Do not define variables in **Rebuild Playback** as this may lead to unpredictable results.

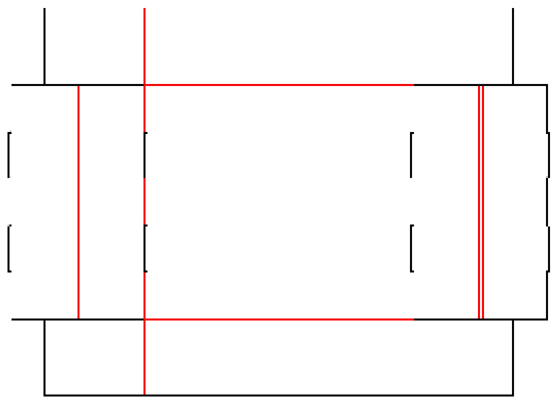
Using Advanced StyleMaker techniques to construct a standard

Creating standards in ArtiosCAD is easy. StyleMaker lets you construct standards with built-in design intelligence. For example, you can create a design in which the number of slots in the base is determined by the width of the design.

You must have purchased the Advanced StyleMaker option in order to use Advanced StyleMaker. If you do not have Advanced StyleMaker, you cannot complete the instructions in this section.

Variables are used to create rebuildable designs. The value of a variable can either be a formula that tells ArtiosCAD how to calculate the value, or it can be a constant value you specify each time the design is rebuilt. To make rebuildable geometry, you specify a variable when asked for a distance, angle, radius, or other measurement.

These instructions assume you already know how to construct geometry and use the tools on the Geometry and Edit toolbars. In this chapter, you will learn how to make a variable number of slots and tabs in a design so that you can make a tray similar to the one shown below.



There are four steps to making a standard:

1. Plan the design.
2. Create the design by creating menus, adding variables, and constructing geometry.
3. Test the design.
4. Add the design to a Style Catalog.

Step 1 – Planning the design

The first step in designing a standard is to plan your design. Is there geometry that will always be the same, such as an angle setting or the radius of an arc? Which parts of the design need to be made with variables, and are their measurements dependent on other parts of the design? Be as thorough as possible.

The number of slots and tabs will depend on the width of the design. The standard will also give you the option of specifying the number of slots and tabs desired.

What do you know about the slots? You know that:

1. The slots should all be the same size.
2. The gap(s) between the slots and the slots should be the same length.
3. There should not be a slot touching either end of the line it is on.
4. There will be one more gap than the number of slots.
5. The length of each slot should be based on a fraction of the width of the design.
6. The size of the gap between the slots should be the length of the line that the gaps are next to, minus the distance all the slots take up, divided by the total number of slots plus 1.
7. The width of the slots should be twice the caliper plus the outside gain.
8. There will be times when you will want to specify the number of slots regardless of any formula.

These same bits of knowledge also apply to the number of tabs.

Let's assign variables and convert these concepts into expressions that can be used as formulae in ArtiosCAD.

Concept	Value
Length, Width, Depth	`L, W, D`
Caliper of the board	`CAL`
Method of creating slots (automatically or manually)	`SLTMET`
Number of slots and tabs	`SLTNUM`
Slot length	`SLTLEN=W/(2*SLTNUM+1)`
Slot width	`SLTWID=2*CAL`
Gap between slots	`SLTGAP=((W+CAL)- SLTNUM*SLTLEN) /(SLTNUM +1)`

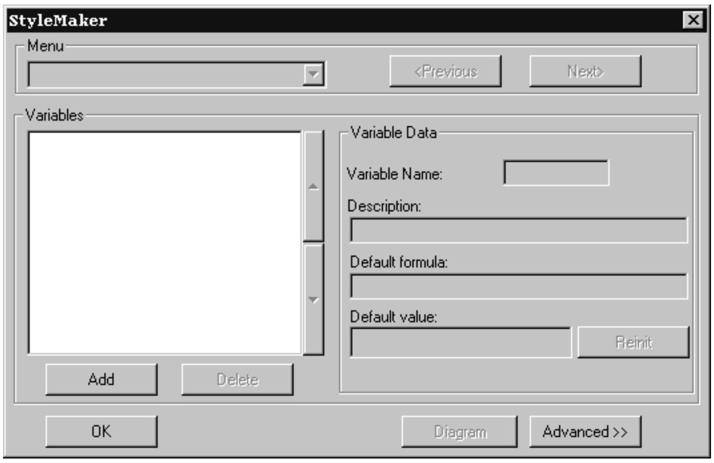
Step 2 – Creating the design

Once you have planned as much as possible, it is time to begin using ArtiosCAD to transform your creative ideas into a design.

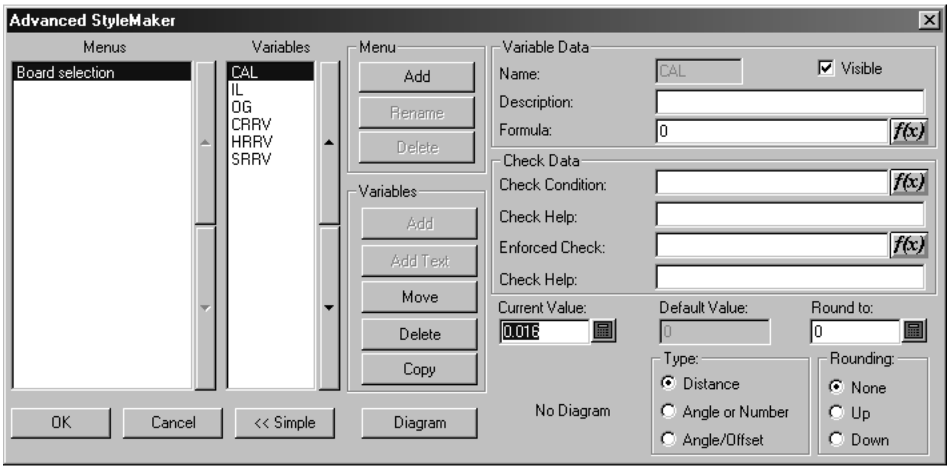
Create a blank design by starting ArtiosCAD and clicking **New Design** on the **File** menu. Choose an appropriate parameter set and board code.

Enter StyleMaker by clicking **StyleMaker** on the **Design** menu.

Click **Advanced** to enter Advanced StyleMaker.



Shown below is the Advanced StyleMaker dialog box.



Making menus and creating variables

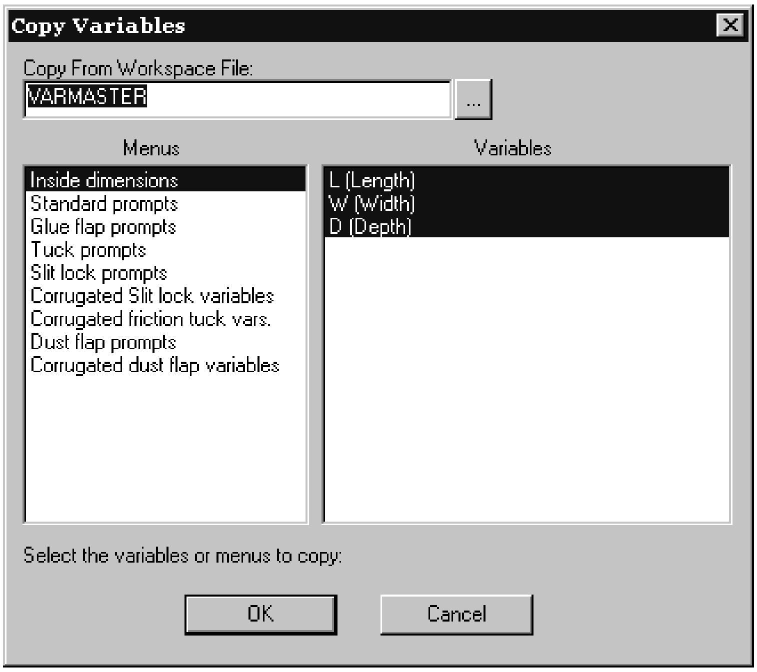
When you create a new blank design, ArtiosCAD automatically inserts a Board Selection menu so that you can change the board specifications when rebuilding the design.

Even though variables may use variables previously defined in the same menu as elements of their default values, the variable values are only saved when the menu is exited, and that variables within a menu may not reinitialize as expected. You could combine the variables into fewer menus, but that could affect the robustness of the rebuild process. `SLTLEN` and `SLTGAP` should be in separate menus after `SLTNUM` is defined. Thus, you need to add five menus.

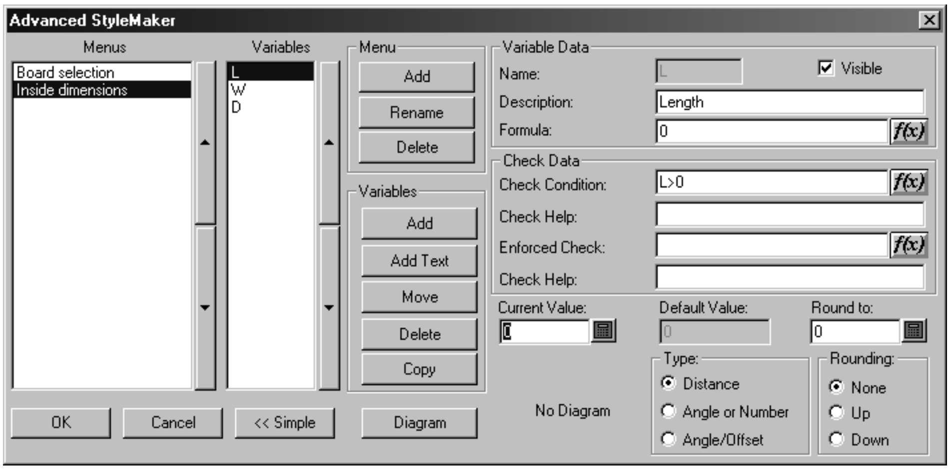
The first menu should contain the basic dimensions of the design. The second menu should contain slot style choices – how ArtiosCAD will determine the number of slots in the design. The third menu should have the variables needed to calculate the number of slots. The fourth menu should have the slot length variable, and the last menu should have the slot gap variable.

Adding the Inside Dimensions menu

ArtiosCAD comes with a set of predefined menus and variables to aid in the construction of standards. Click **Copy** in the **Variables** group of the Advanced StyleMaker dialog box.



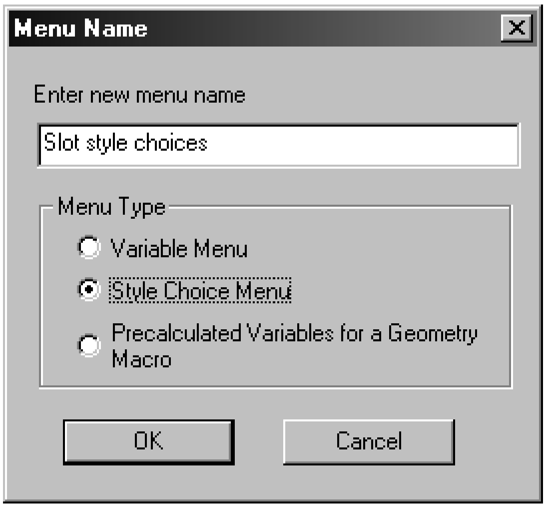
The **Inside Dimensions** menu and the variables within it are selected by default. Click **OK** to copy them into your design. The Advanced StyleMaker dialog box should look similar to the one shown below.



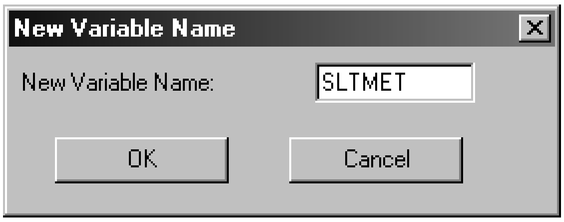
Select each variable and enter a value in the **Current Value** field. Suggested values are 24 inches or 600 millimeters for the length and width, and 12 inches or 300 millimeters for the depth.

Adding the Slot style choices menu

Click **Add** in the **Menu** group to add the menu for the slot style choices. In the Menu Name dialog box, enter `Slot style choices` as the title of the menu. Set the menu type option button to **Style Choice Menu** as shown in the figure below. Click `OK`.

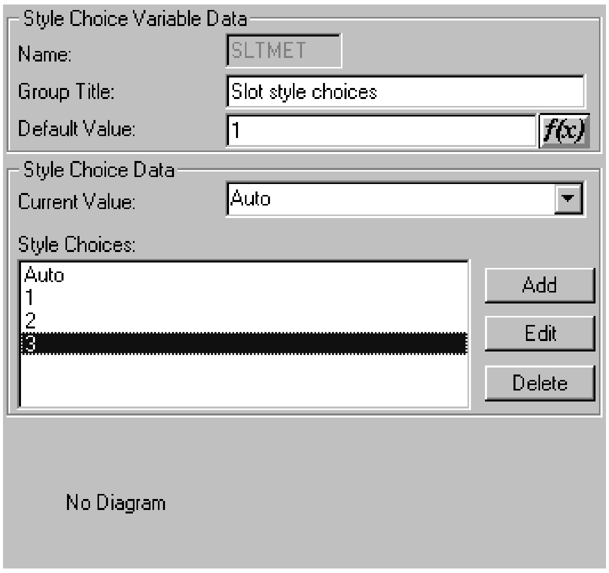


Click the **Add** button in the **Variables** group of the Advanced StyleMaker dialog box. Enter `SLTMET` in the **Variable Name** field and click **OK**.



Once you click **OK**, the variable will appear in the list of variables. In the **Group title** field, enter `Slot style choices`. This is what will appear as the title of the group of choices presented when rebuilding the design. The way style choice variables work is that they set the variable to a value from a series of values that correspond to the entries you create in the **Style Choices** group. For example, the first entry you add sets the variable to 1, the second to 2, the third to 3, and so on. You then build geometry with the logic of “If X=1 then do A, if X=2 then do B, if X=3 then do C” and so on.

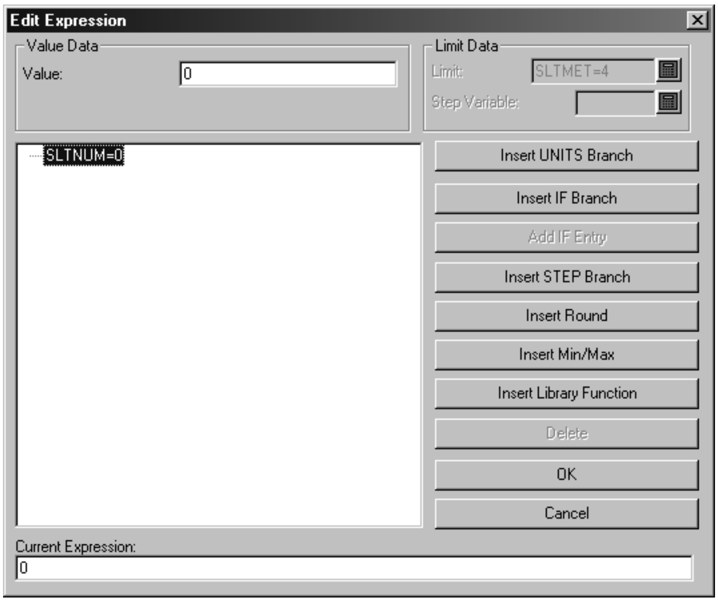
Click **Add** in the **Style Choice Variable Data** group and enter `Auto` as the caption (name) of the first choice. Click **Add** again and enter `1` as the caption. Repeat and enter 2 as the caption. Repeat and enter 3 as the caption. You should have a dialog box that looks similar to the one shown below.



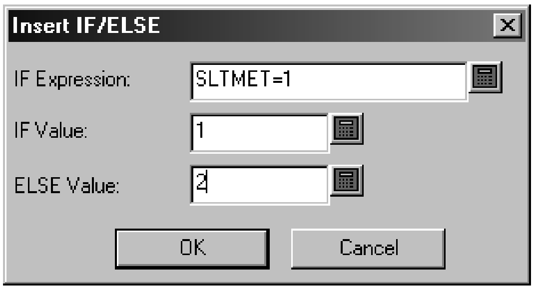
In this example, when rebuilding the design, if you pick choice **Auto**, SLTMET will be 1. If you pick choice **1**, SLTMET will be 2, and so forth.

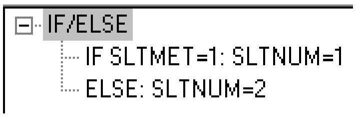
Adding the Number of slots menu

Now comes the part where we build intelligence into the design. Click **Add** in the **Menu** group to add a new menu. Name it `Number of slots` and set the menu type option button to **Variable Menu** and click **OK**. Click **Add** in the **Variable** group and enter `SLTNUM`. Enter `Number of slots` in the **Description** field. Set the type to **Angle or Number** in the **Type** group. Click the formula button at the end of the **Default formula** field to open the Edit Expression dialog box.



The logic to build into the design is an IF…THEN…ELSE structure. Click **Insert IF Branch**. In the **IF Expression**: field, enter `SLTMET=1`. In the **IF Value**: field, enter `1`. In the **ELSE Value**: field, enter 2. This means that if SLTMET=1 (if the Slot style choice is set to 1), then set the value of SLTNUM to 1, else set it to 2. Click **OK**.



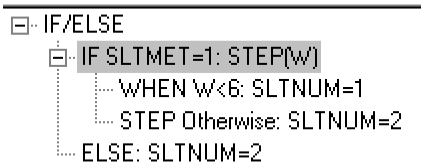


But when the first Slot style choice is chosen (the Auto choice), ArtiosCAD is to automatically decide how many slots are needed. Therefore, a STEP function needs to be inserted in the IF statement.

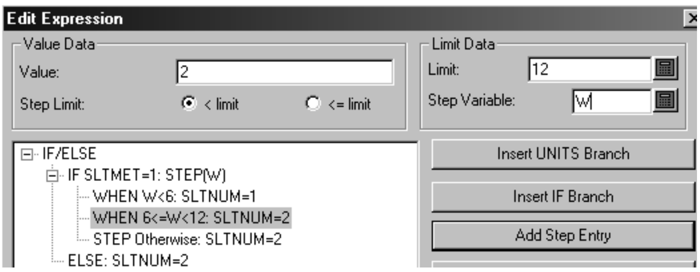
A STEP function is an expression that uses this logic: When A is less than constant 1, set B to value X. When A is greater than (or equal to) constant 1 but less than constant 2, set B to value Y. When A is greater than (or equal to) constant 2 but less than constant 3, set B to value Z, and so on.

Therefore, select the **IF SLTMET=1: SLTNUM=1** entry and click **Insert STEP Branch**.

The number of slots should be based on the width of the design. Therefore, set the **STEP** variable to `W`. When the width of the design is less than 6, only 1 slot should be made; set the **Step Limit** to 6 and the **Step Value** to `1`. Set the **Otherwise** value to 2. Click **OK**. The Edit Expression dialog box should look similar to the figure below.



Now we want to add a STEP branch for when the Width is greater than 6 but less than or equal to 12. Click the **STEP Otherwise** line and then click **Add Step Entry**. Enter `12` for the **Step Limit** and 2 for the **Step Value**.



Repeat this process of adding STEP entries 4 times, increasing the **Step Limit** by 6 each time and the **Step Value** by 1 each time. The final **IF SLTMET=1** entry of the IF statement should look like the one in the next picture.

```
IF/ELSE
  IF SLTMET=1: STEP(W)
    WHEN W<6: SLTNUM=1
    WHEN 6<=W<12: SLTNUM=2
    WHEN 12<=W<18: SLTNUM=3
    WHEN 18<=W<24: SLTNUM=4
    WHEN 24<=W<30: SLTNUM=5
    WHEN 30<=W<36: SLTNUM=6
    STEP Otherwise: SLTNUM=2
  ELSE: SLTNUM=2
```

Now ArtiosCAD knows the number of slots to make if the **Auto** choice is chosen in the **Slot Style Choices** menu. However, IF statements must still be added for the other style choices.

Click the **ELSE: SLTNUM=2** entry and then click **Add If Entry**. Enter `SLTMET=2` in the **If Expression**: field and `1` in the **If Value**: field. Click **OK**.

Add two more **IF** entries with the first having the **If Expression**: of `SLTMET=3` and the **If Value**: of `2`, and the second having the **If Expression**: of `SLTMET=4` and the **If Value**: of `3`. When you have finished, the Edit Expression dialog box should look like the following picture. Click **OK** to return to the Advanced StyleMaker dialog box.

```
IF/ELSE
  IF SLTMET=1: STEP(W)
    WHEN W<6: SLTNUM=1
    WHEN 6<=W<12: SLTNUM=2
    WHEN 12<=W<18: SLTNUM=3
    WHEN 18<=W<24: SLTNUM=4
    WHEN 24<=W<30: SLTNUM=5
    WHEN 30<=W<36: SLTNUM=6
    STEP Otherwise: SLTNUM=2
  IF SLTMET=2: SLTNUM=1
  IF SLTMET=3: SLTNUM=2
  IF SLTMET=4: SLTNUM=3
  ELSE: SLTNUM=2
```

Now that the number of slots has been determined, you can set up the variables for the slot length and the gap between slots.

Adding the Slot dimensions menu

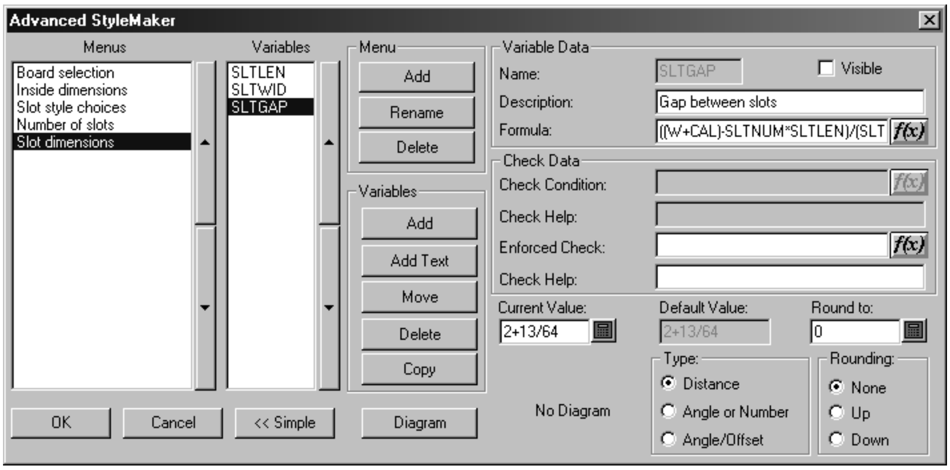
Click **Add** in the **Menu** group. Enter `Slot dimensions` for the menu name and set the menu type option button to **Variable Menu**. Click **OK**.

Click **Add** in the **Variable** group. Enter `SLTLEN` as the name of the new variable and click **OK**. Enter `Slot length` in the **Description**: field. Going back to what you know about the slots, enter `W/(2*SLTNUM+1)` in the **Default formula** field.

Click **Add** in the **Variable** group. Enter `SLTWID` as the name of the new variable and click **OK**. Enter `Slot width` in the **Description**: field. Going back to what you know about the slots, enter `CAL*2` in the **Default formula** field.

Now that the slot dimensions have been added, the slot gap variable can be added. This variable can be in the same menu as the Slot length and Slot width variables since it will be a hidden variable as it should not be modified when rebuilding the design.

Click **Add** in the **Variable** group. Enter `SLTGAP` as the name of the new variable. Clear the **Visible** checkbox so that no check appears in it. Enter `Gap between slots` in the **Description**: field. Going back to what you know about the slots, enter `((W+CAL)-SLTNUM*SLTLEN)/(SLTNUM+1)` in the **Default formula** field. You may have to put in a temporary default formula in order to clear the **Visible** checkbox.



Now that all the menus and variables are defined, design construction can begin. Click **OK** and then save the design before proceeding.

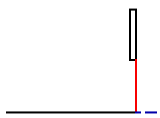
Constructing the design

We will draw only half the design as it can be copied using the **Copy Vertical Mirror** tool to make a complete design.

Using the **Line** tool, draw a line at 0 degrees horizontally right from the origin with an X offset of L/2.

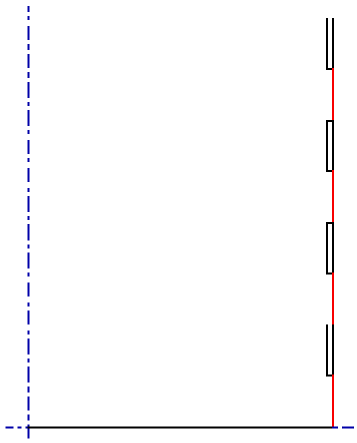
Draw a line from the end of line 1 up by SLTGAP. Change this line to a crease by selecting the **Select** tool, then double-clicking the line and selecting **Crease** in the Properties dialog box and clicking **OK**.

Change to the **Rectangle** tool and move the cursor in an up and left direction. Set the X offset to SLTWID and the Y offset to SLTLEN. You should have something that looks like the following picture.



Click the **Select** tool and select the crease and the slot.

Click the **Copy Times Offset** tool. The status bar will prompt for the number of copies to be made. Enter `SLTNUM-1` and press `Enter`. Use the bottom point of the crease as the pick-up point. Set the angle to 0 vertically. Use the top right corner of the slot as the put-down point. You should have a construction similar to the figure shown below.



Use the **Move to Point** tool to move to the intersection of the bottom length line and the crease with the slots.

Make a horizontal line to the right using D for the X offset.

Make another horizontal line to the right using CAL*2 as the X offset.

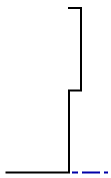
Make yet another horizontal line to the right using D for the X offset.

Make a vertical line going up using SLTGAP as the Y offset.

Make a horizontal line going right using SLTWID as the X offset.

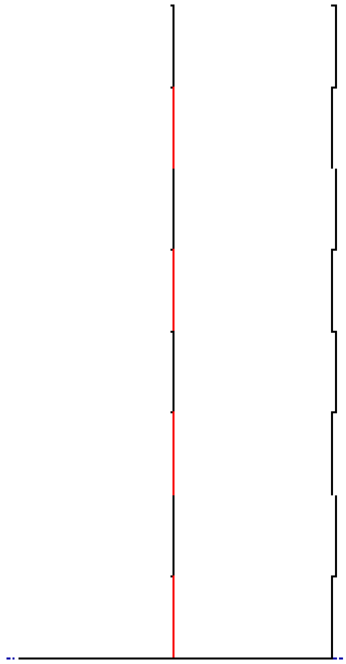
Make a vertical line going up using SLTLEN as the Y offset.

Make a horizontal line going left using SLTWID as the X offset. You should now have a construction similar to the one shown below.



Use the **Select** tool to select the tab construction.

Use the **Copy Times Offset** tool to copy the tabs the same way as you copied the slots. Use SLTNUM-1 as the number of copies, use the intersection of the horizontal cut line and the vertical cut line as the pick-up point, set the angle to 0 vertically, and use the top left point of the tab as the put-down point. The result should be similar to the construction shown below.



Click the **View Mode** button on the Zoom toolbar and turn on **Endpoints** in the **Styles** group.

Use the **Move to Point** tool to move to the right most point of the group of two points on the bottom cut line. You may need to zoom in to see the two points.

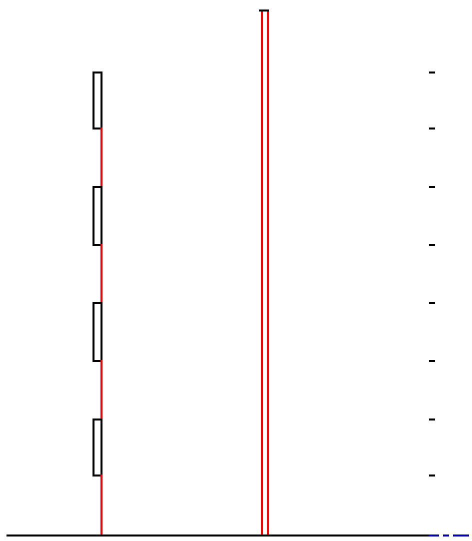
Use the **Line** tool to create a vertical line going up using W+CAL as the Y offset.

Use the **Move to Point** tool to move to the left most point of the group of two points on the bottom line.

Use the **Line** tool to create a vertical line going up using W+CAL as the Y offset.

Connect the tops of the two lines with a cut line.

Change both vertical lines just created to creases and turn off the viewing of endpoints. You should have a construction similar to the one in the picture below.



Use the **Move to Point** tool to move to the top of the right crease.

Draw a horizontal line to the right using D as the Y offset.

Draw a vertical line down using SLTGAP as the Y offset. IMPORTANT: DO NOT SNAP TO THE TOP OF THE TAB!

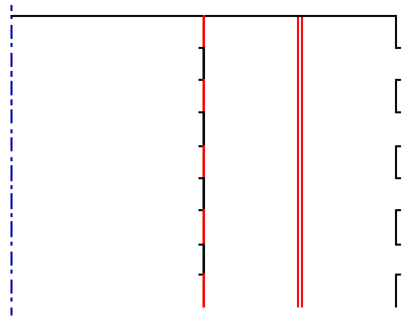
Use the **Move to Point** tool to move to the top of the left crease.

Draw a horizontal line to the left using D as the Y offset.

Draw a vertical line down using SLTGAP as the Y offset. IMPORTANT: DO NOT SNAP TO THE TOP OF THE SLOT!

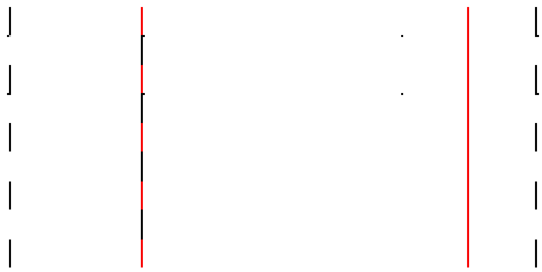
Change this vertical line to a crease.

Draw a horizontal line from the top of the crease just created to the vertical axis. You should have a construction similar to the one shown below.



Use the **Select** tool to select all the lines in the design.

Click the **Copy** button on the Edit toolbar, then click the **Mirror About Line** button. Click to the immediate left of the vertical axis. You should have a construction similar to the one shown below. Save the design!

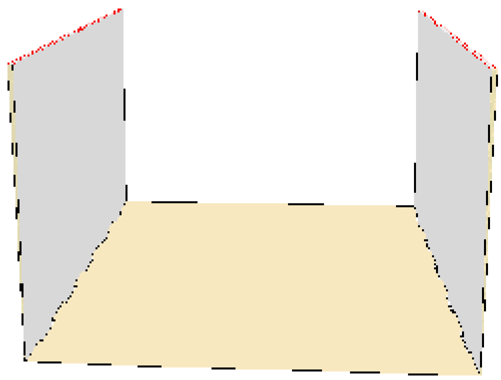


Construct the rest of the tray using the techniques learned so far.

Step 3 – Testing the design

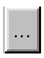
There are two things to test in this design – its rebuildability and how it actually will work if produced. Rebuild the design using the **Rebuild design** command on the Design menu. Use different combinations of values for the length, width, and depth. Set the slot style choice to something different every time. You will have to REINIT the Slot Length variable each time you rebuild.

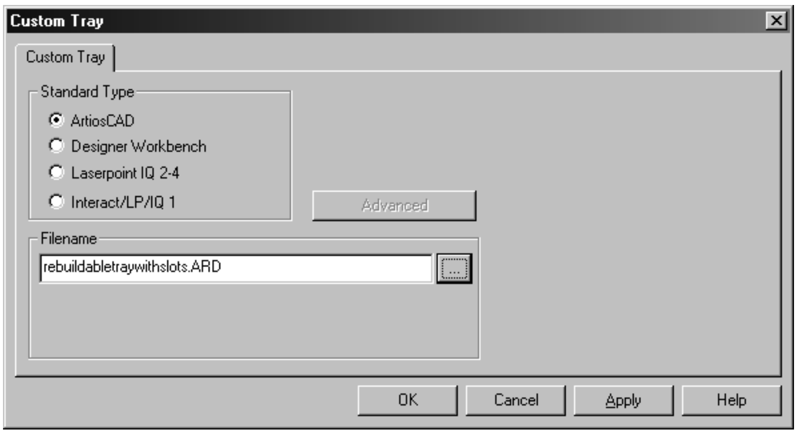
Use 3D to test its worthiness for becoming a production design. Make sure the tabs fit in the slots and that the rollover works correctly.



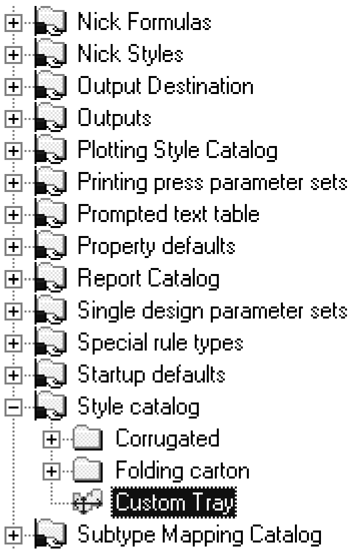
Step 4 – Adding the design to a Style Catalog

To give everyone using ArtiosCAD access to this new standard, add it to a Style Catalog. You should also put a copy of the design file in `\Artios\ArtiosCADnn.nnll\ServerLib.`

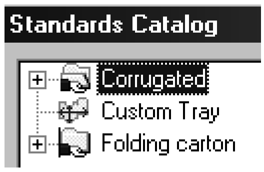
1. Exit from any open design so that the only commands on the menu bar are File, Options, and Help.
2. Click **Defaults** on the **Options** menu.
3. Right-click Style Catalog in the left-hand pane and pick **New**, then **Data** from the context menu.
4. Enter a name such as `Custom Tray`. Do not press `enter` when done; rather, double-click the icon that precedes the name.
5. Set the **Standard Type** option button to **ArtiosCAD**.
6. Click  and double-click the resizable design you want to add to the catalog. The dialog box should have the same title as the name you entered, and should look similar to the following picture.



7. Click **OK** to add the design to the Style Catalog. The Shared Defaults pane of the Defaults dialog box should look similar to the following picture.



8. Click **File**, then **Save** to save the shared defaults. Next time you run Builder, you will see your custom design in the catalog listing as shown in the following picture.

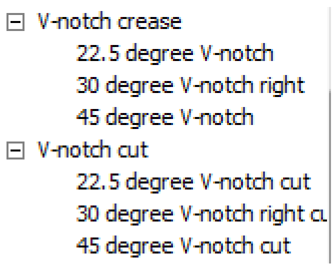


Your rebuildable design is now complete and can be used by everyone who uses this installation of ArtiosCAD.

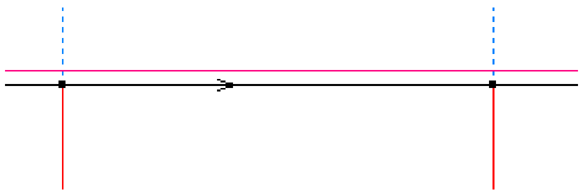
Designing for Re-board®

Re-board® is a thick board made from recycled paper. Using it requires some different design techniques in ArtiosCAD from working with regular boards.

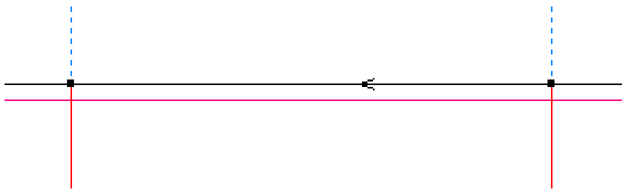
When working with Re-board®, you must choose the proper V-notch line types when designing a single design. ArtiosCAD comes with three crease types and three cut types in the special rule catalog as shown below, but you can add others of your own specification to the board table in DataCenter Admin.



The V-notch cut lines draw double lines but act as generic cuts when determining area, making a layout, and so forth. The principal black line shows the bottom of the angled cut, and the additional red line shows the top edge of the angled cut. Cuts can slant to either the outside or the inside. Shown below is a V-notch cut slanting to the outside:



Shown below is a V-notch cut slanting to the inside:



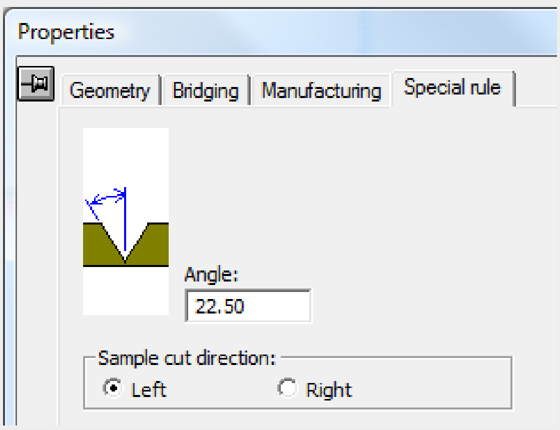
Reverse V-notch cut lines as necessary using **Reverse Direction** before doing a sample Output. When you reverse a V-notch cut, the red line flips from one side to the other. Sample output sequences normal cut lines but does not change the direction of V-notch cuts; ArtiosCAD outputs in the direction you drew them.

Note:

Curved lines are not supported for V-notch lines and Re-board® does not bend.

Note:

If your samplemaker's V-notch tool is not angled to the left, change the **Special rule** > **Sample cut direction** from left to right on the Special rule tab of the Properties dialog box for the rule in the workspace or in **Options** > **Defaults** > **Special rule types** > **<Rule type>**.



Designing Retail-Ready Containers

More and more, retailers are asking for retail-ready packaging in which part of the shipping container is torn off, the remainder is used as a display for the contents, and the whole thing is put on the shelf ready for customer selection and purchase.

In Single Design, there are special ways to design these features which you can then easily show in 3D. This section explains how to do the underlying design work.

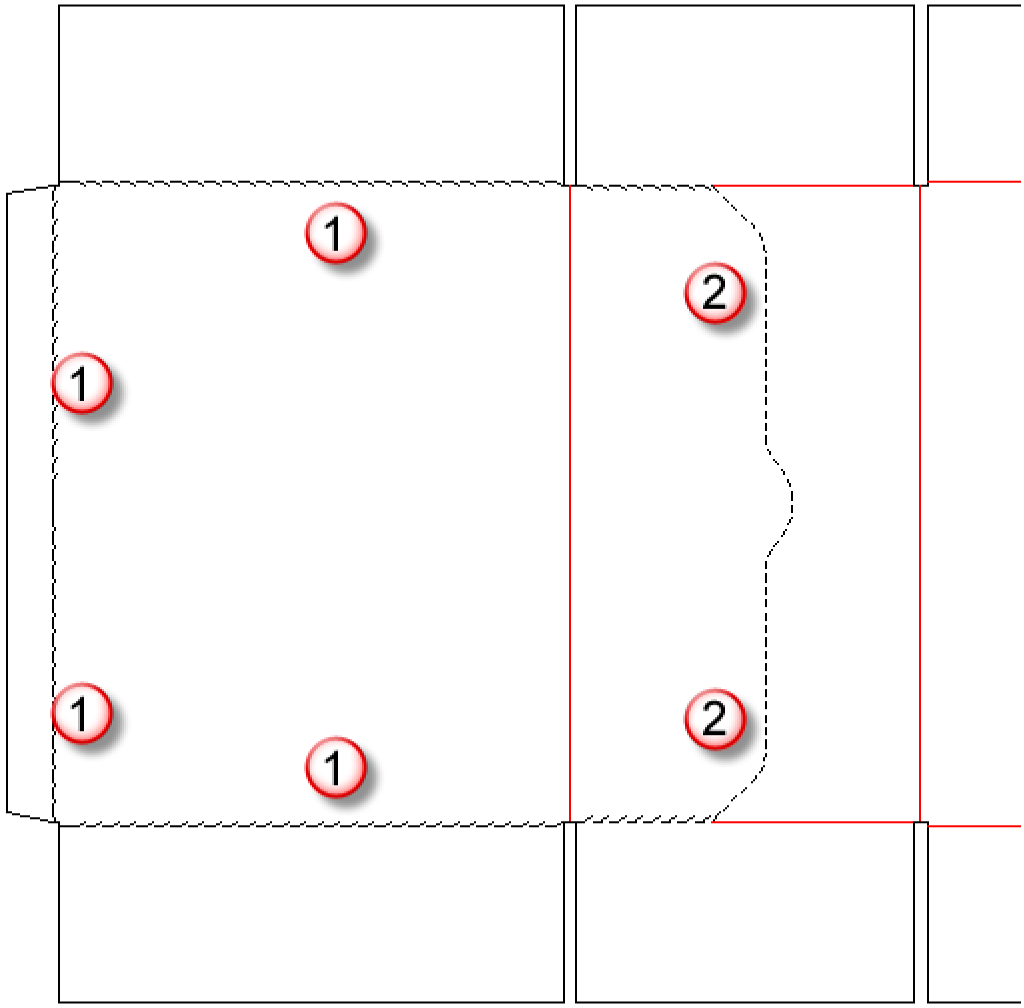
Defining Tear-Away Parts

To define the tear-away parts:

- Use perfs or zippers
- Use cuts with gaps between them
- Use a combination of the above methods

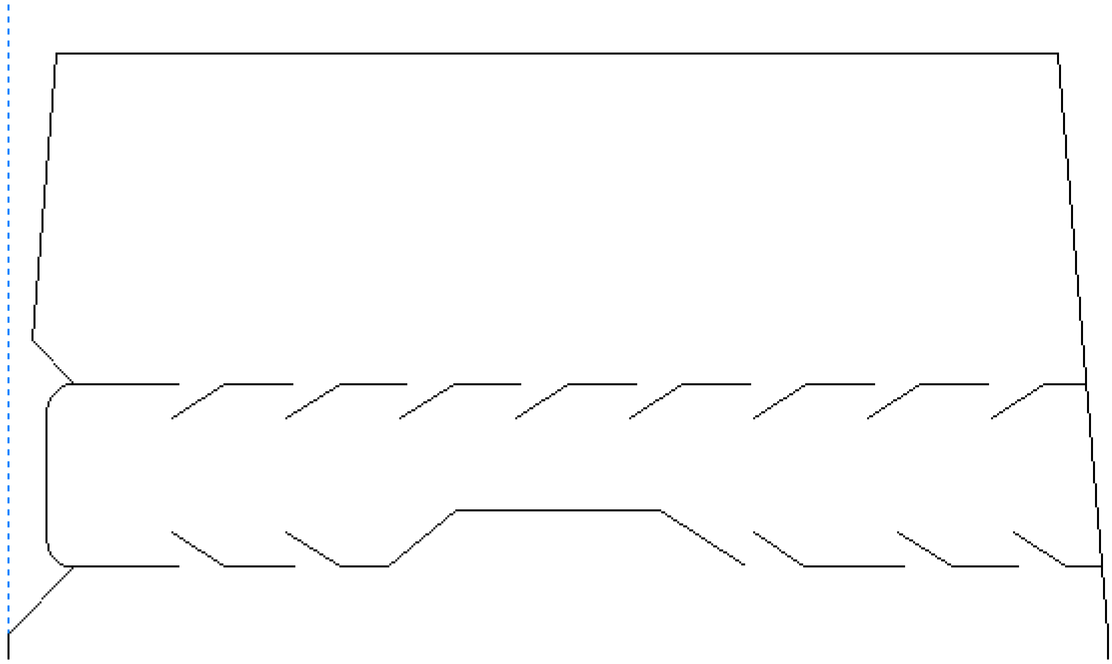
Note: Curved creases can tear or fold but not both.

1. Design the parts of the container that will tear away with perfs and zippers. In the example below, lines labeled 1 are zippers set to tear and fold and lines labeled 2 are perfs set to tear.

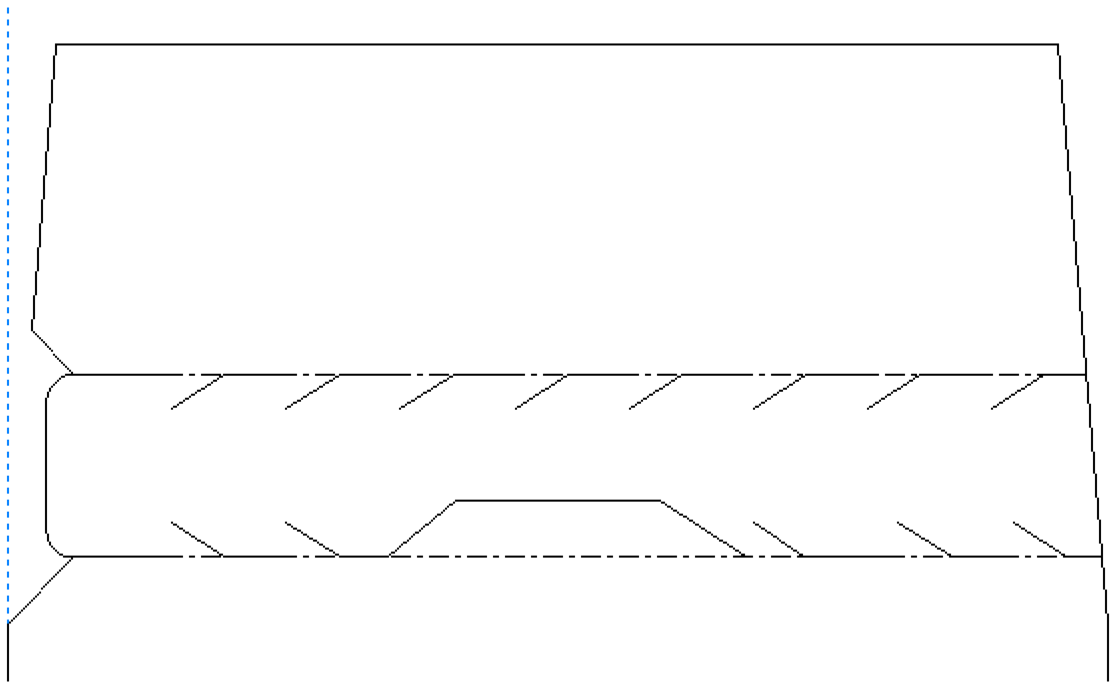


When a special rule has a sequence of lines like the example shown above, use the **Group Sequence** tool to group and align them.

2. Alternately, use a series of cuts and gaps to define a tear-away part, such as the custom zipper shown below.



To define the outline of the tear-away part, add tear lines in a 3D Assist layer. The tear lines can either be individual lines that join the cut segments, or they can be two continuous lines over the zipper rules as shown below.

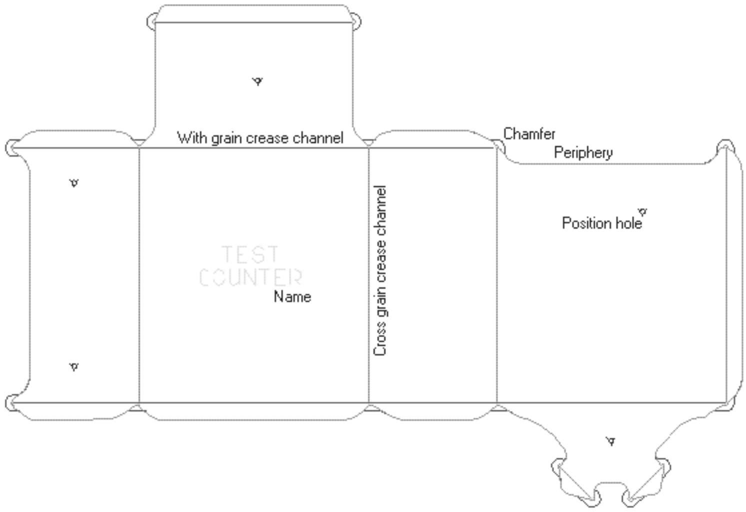


3. As the last design task before you convert the design to 3D, double-click the perfs or zippers with the **Select** tool and set them to either **Tear** or **Tear and fold** on the 3D tab of the Properties dialog box. If you cut and paste any lines with these attributes, you need to reapply them to the pasted copies, as that attribute is not copied.

Intelligent Counters

Counters are special templates made out of phenolic resin which are used in folding carton manufacturing. They are part of the die press tooling and are positioned between the folding carton stock and the bottom plate of the press. They aid in the production of creases by providing a place for the material to flex into when struck with creasing rule.

The Intelligent Counters module of ArtiosCAD automatically creates counters with just a few mouse clicks for most designs. Shown below is a typical counter.



The edge of the counter, the **periphery**, is shaped as it is because it has to go outside the creases but stay inside the cuts. It also needs to go far enough inside the cuts to avoid making marks on the board. Thus, a compromise has to be reached at the places where cuts and creases meet. This compromise is the subject of most of Intelligent Counters' setup parameters.

There are twelve line types used for making counters:

Table: Counter line types

Line type	Description
With grain channel	Crease channels for with grain creases (those that are parallel to the grain of the board.
Cross grain channel	Crease channels for cross grain creases (those that are perpendicular to the grain of the board). Usually 0.004 inch wider than with grain creases.
Crease end tool	Cuts off the end of the crease channel where it meets a cut.
Periphery tool	Cuts out the counter with a bevelled edge.
Chamfer tool	Trims the edge of the counter with a gentler slope (except near creases) than the periphery tool.

Line type	Description
With grain slot	Cuts a slot for a with grain cut-crease.
Cross grain slot	Cuts a slot for a cross grain cut-crease.
Name tool	Etches an identifying label into the counter.
Position holes tool	Cuts the outline of the position hole, usually a circle.
Position holes tool 2	Second position holes tool, if using two tools.
Counter mill outline	Outline of area to be milled, for reverse creases, reverse partial cuts, or for embossing.
Milling tool	Mills out an area with a series of parallel lines.

These line types are mapped to physical tools by the plotting style, and they are offset or multiple-passed as necessary to allow for the tool widths.

Note: When using the milling tool, areas to be milled that are exactly the same width as the milling tool may sometimes be overlooked. In these situations, adjust the size of the area by 0.05 mm to allow the area to be milled.

Configuring Intelligent Counters

Intelligent Counters uses parameter sets to organize its parameters into easily-selectable groups. You must choose a parameter set when creating a counter. A default parameter set is installed with ArtiosCAD. See the *Esko ArtiosCAD Administrator Guide* for more information.

Intelligent Counters workflow

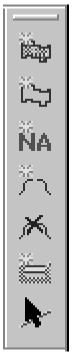
The general steps in making a counter are as follows:

1. Construct your design making sure the perimeter of the design has no gaps and is composed entirely of cut lines.
2. Click the **Create or Rebuild Counter** tool on the Counter toolbar. Choose a counter parameter set and make appropriate selections in the prompts and options on the tabs.
3. Click **OK** to create the counter.
4. Add a name, tack bridges, and straps as appropriate.
5. Modify counter lines if needed.

6. Save the design and output it to the countercutter.

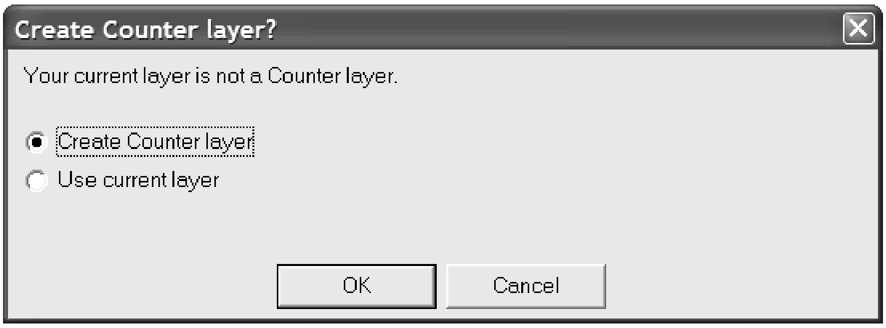
Creating and modifying a counter

The tools on the Counter toolbar create and modify counters. Turn on the Counter toolbar by checking its box in the **View** > **Toolbars** dialog box. Alternately, you may turn it on via its button on the ToolRack, and you may select the tools directly from the **Tools** > **Counter** menu.

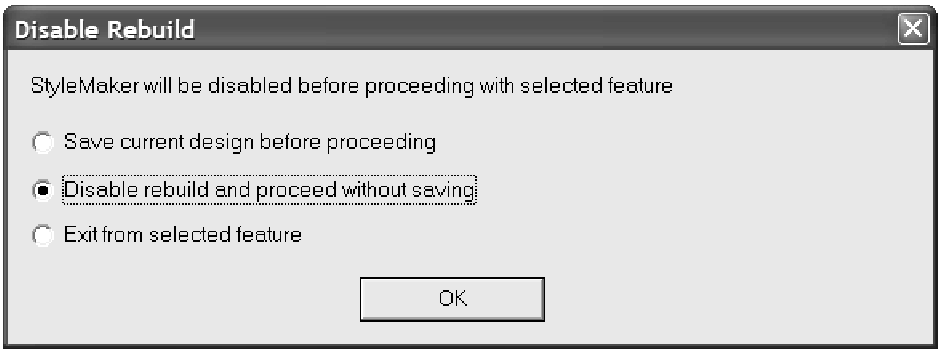


 The first tool on the Counters toolbar is **Create or Rebuild Counter**. When clicked for the first time, this tool prompts you to select a counter parameter set. Select the set to use and click **OK**.

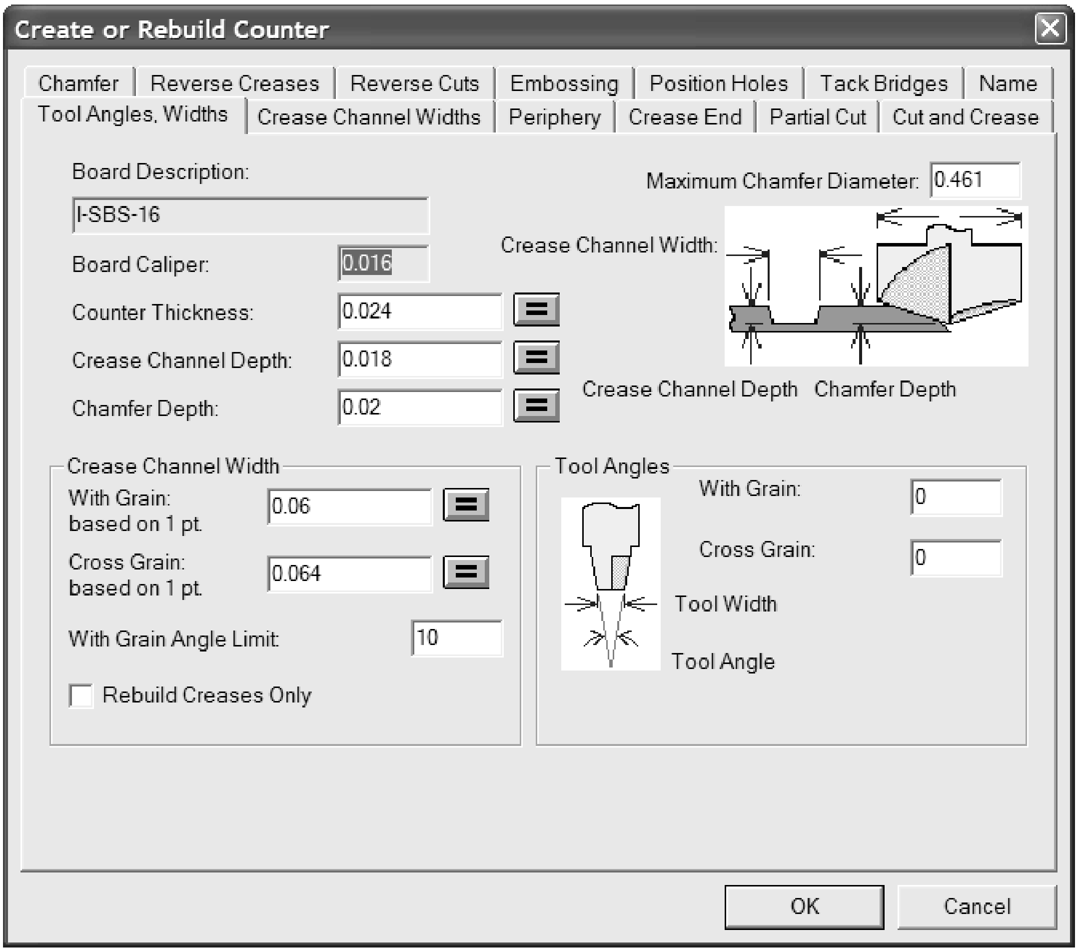
If your current layer is not a counter layer, you are prompted to create one. Click either option as desired and then click **OK**.



Creating a counter disables the rebuildability of a design. You can either save the design before proceeding, disable rebuild and go on without saving, or exit the counter creation process. Select the desired option and click **OK**.



Once you have chosen a parameter set, this tool opens the Create or Rebuild Counter dialog box which is the heart of Intelligent Counters.



Clicking the equal-sign button at the end of a field recalculates the value in that field according to the formula in the counter parameter set. You can change the value itself in the field, but to change the expression used to calculate the value, you must modify the parameter set in Defaults.

The **Rebuild Creases Only** checkbox appears only when rebuilding a counter, and when clicked, causes this tool to only recalculate the crease channels.

Crease channel widths tab

The **Crease Channel Widths** tab of the Create or Rebuild Counter dialog box lets you specify different crease channel widths for with-grain creases and cross-grain creases after you have assigned them different subtypes in the Counter layer. You may define the subtypes and channel widths before creating or rebuilding the counter; the channel widths are set appropriately on Output.

In the **Crease Channel Width** group, **With Grain:** and **Cross Grain:** are the initial values for crease channel widths of creases with no subtype assigned, or with a subtype defined that is not defined in the table of additional widths. These are initially set by the parameter set, and are repeated from the **Crease Channel Width** group on the Tool Angles, Widths tab. The equal-sign buttons at the end of each field reset the field to the value calculated by the formula for that field in the counter parameter set.

In the **Additional crease channel widths** group, you can define up to nine subtypes with different widths for with grain creases and cross grain creases in the appropriate fields.

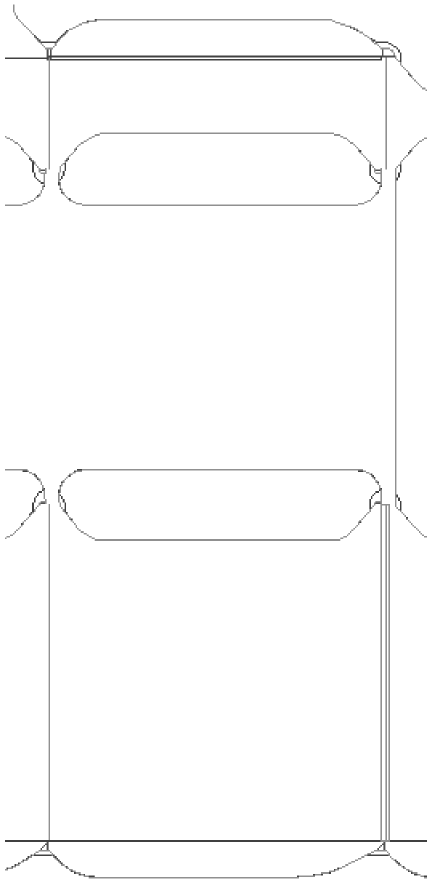
You may also set the crease channel widths without rebuilding the counter by clicking **Tools** > **Counter** > **Crease Channel Width**.

Note: Rebuilding the counter removes any subtype assignments previously made. You must reassign the crease subtypes to lines in the Counter layer after rebuilding the counter. The subtype definitions are not cached; you will have to re-enter all the information.

If there are creases of the same subtype but different pointages, the channel widths from the table are used for the crease with the minimum pointage of that subtype, and creases with higher pointages get larger channel widths offset by the difference in pointage.

For any crease with a subtype defined in the **Additional crease channel widths** group, each subtype defines a specific channel width regardless of crease pointage. A 5-point with grain crease, for example, will have the same channel width as a 2-point with grain crease if both with grain creases have the same subtype applied.

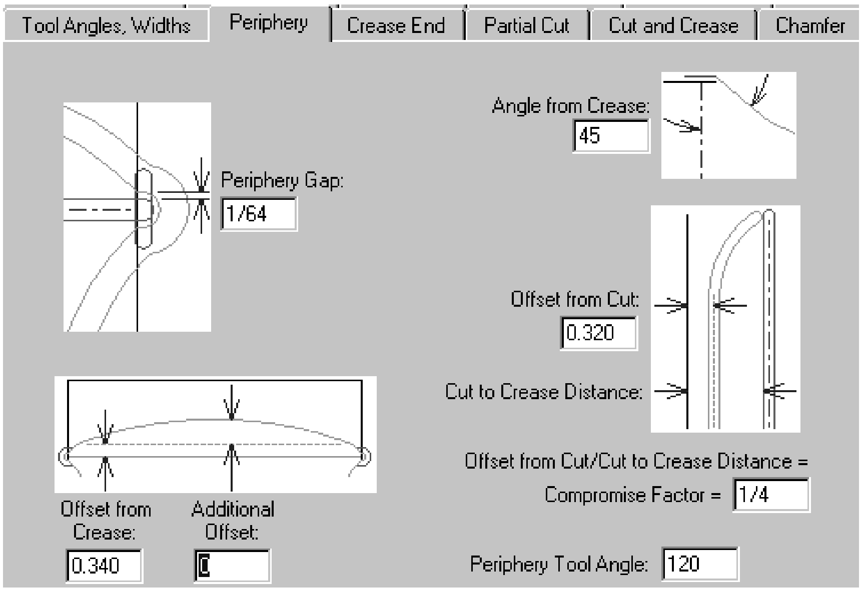
Shown below is an example of wider channel widths in an Output made to a workspace layer. The top horizontal crease and the bottom right vertical crease are both wider than the default.



Note: The channel width adjustment using subtypes is meant to be small. The periphery tool path is **not** adjusted to compensate for the different channel widths.

Periphery tab

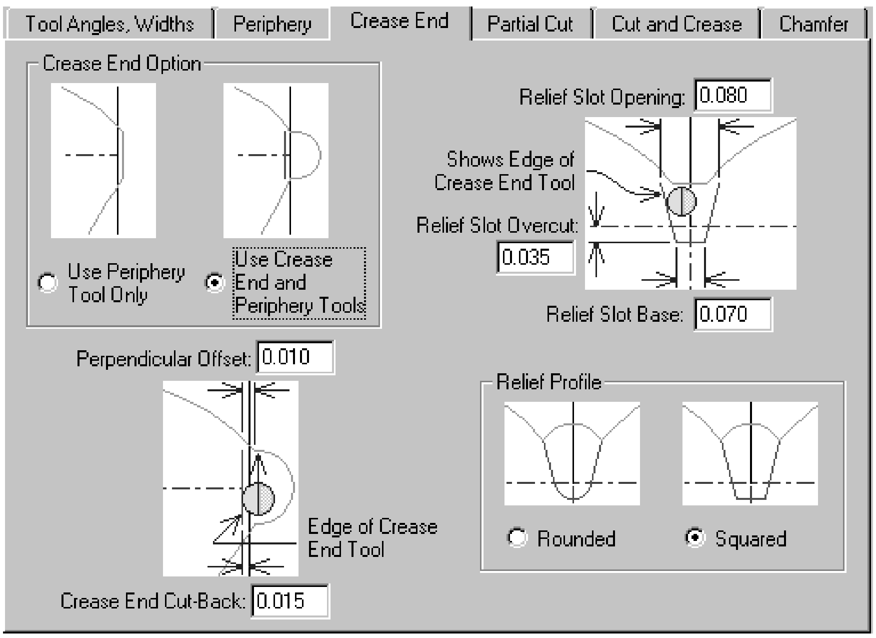
The **Periphery** tab of the Create or Rebuild Counter dialog box contains settings which affect the periphery tool.



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

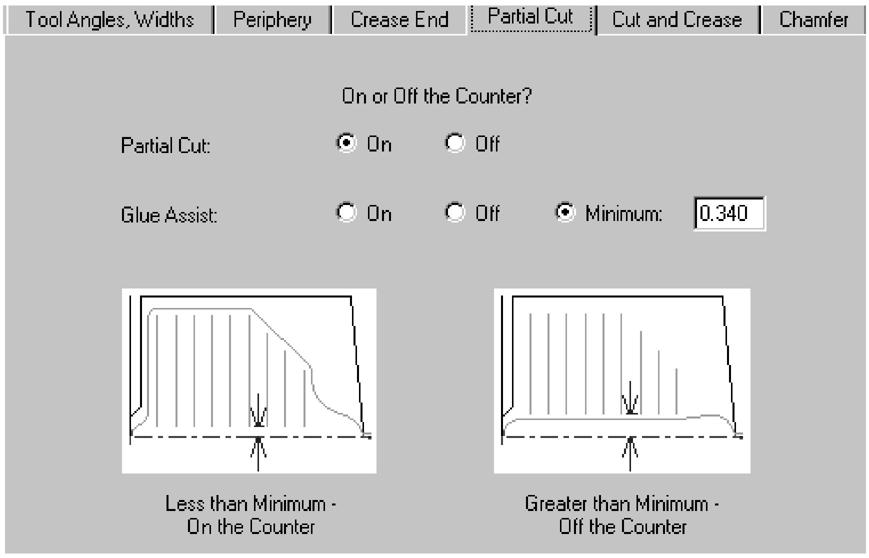
The **Crease End** tab of the counter parameter set dialog box controls how the crease ends are machined.



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.

Partial cut tab

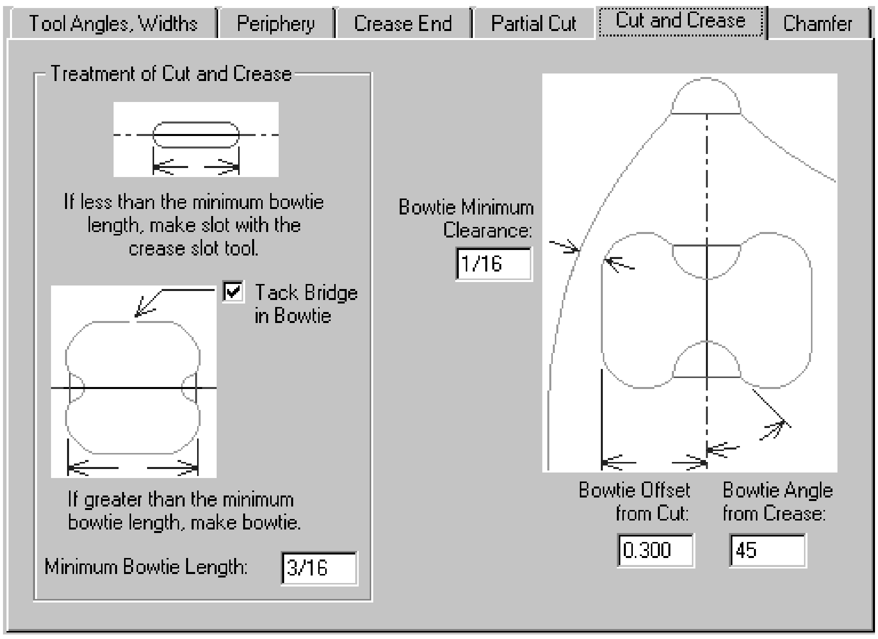
Partial cuts and glue assist lines can have the counter extend underneath them if you so desire.



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** for this counter.

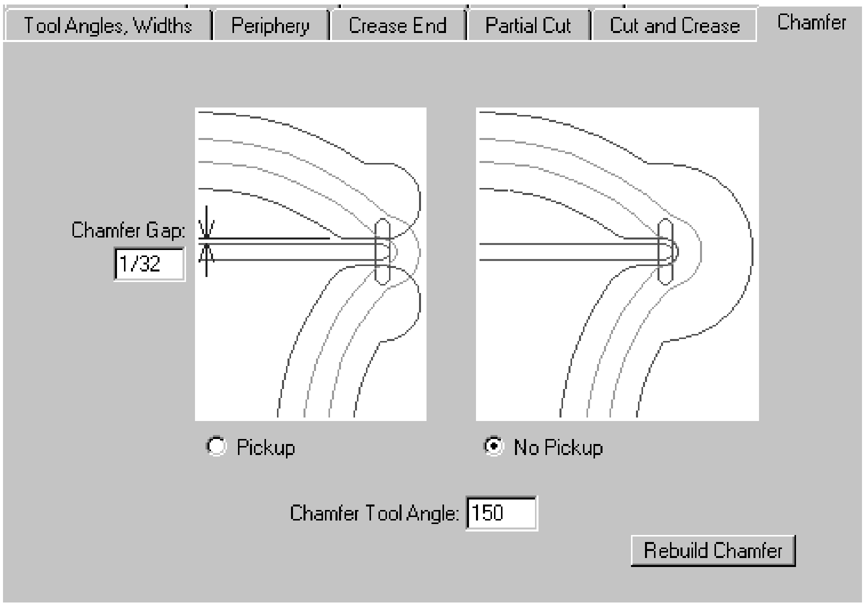
Cut and Crease tab

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.

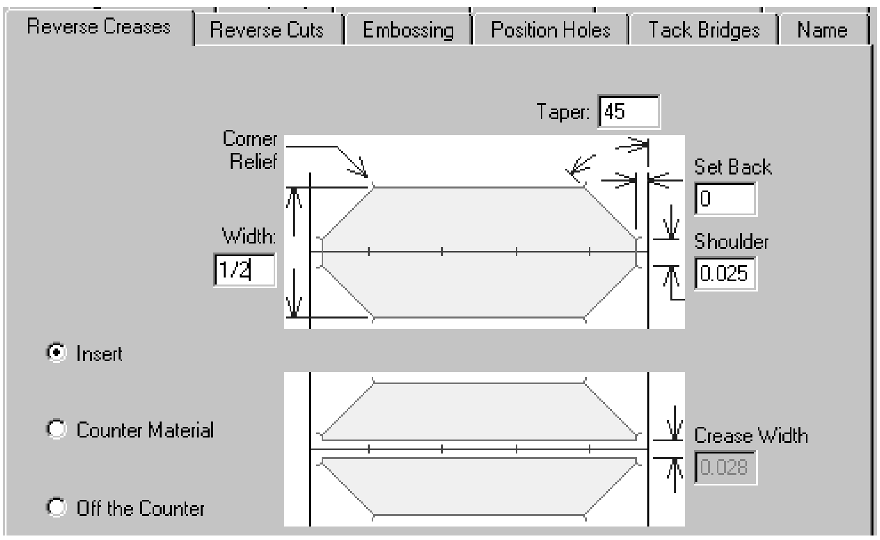


The **Chamfer Gap** is the distance between the inside edge of a chamfer and the crease channel. The **Pickup** and **No Pickup** options control whether the chamfer is continuous around the end of the crease channel. The **Chamfer Tool Angle** is the angle of the tool, which, together with the thickness of the counter material, determines the width of the chamfer channel.

The **Rebuild Chamfer** button does what it says.

Reverse Creases tab

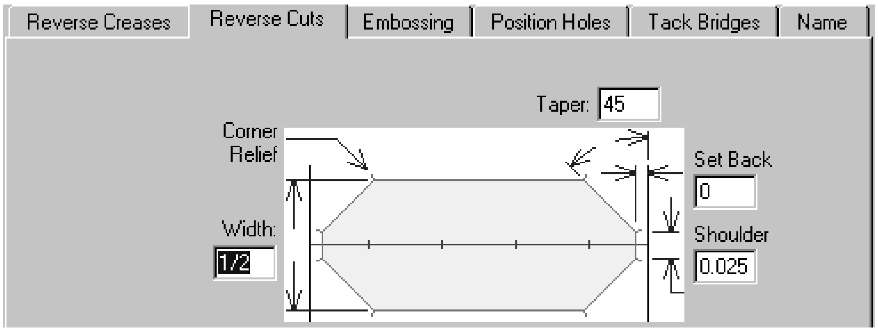
Reverse creases are used to make a panel fold outward instead of inward.



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.

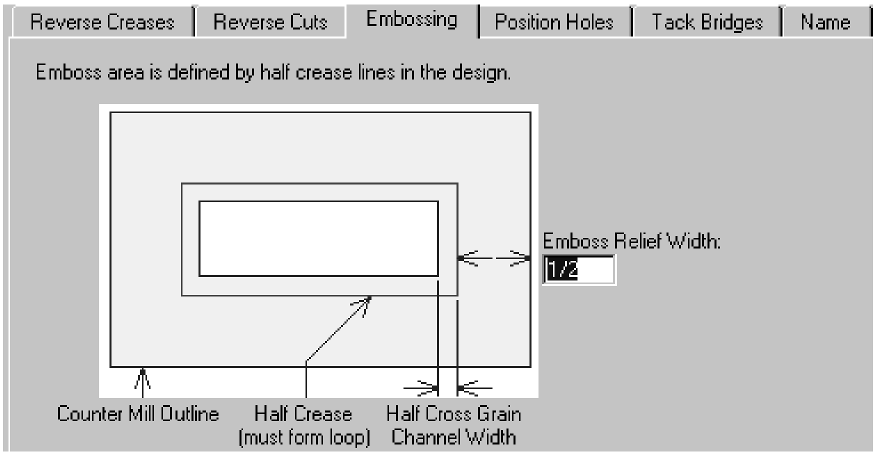
Reverse Cuts tab

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.



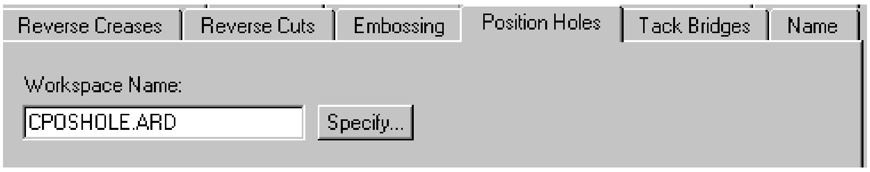
Embossing tab

An area of the board can be embossed by milling out the part of the counter surrounding the embossed area. An area is milled out in the counter half-crease channel width inside the half-crease lines, forming one side of a crease, thus making a raised emboss area. A relief is milled out on the outside of the half-crease lines to avoid making the other half of the crease channel. As with other tabs, the parameters on the **Embossing** tab are explained in the picture.



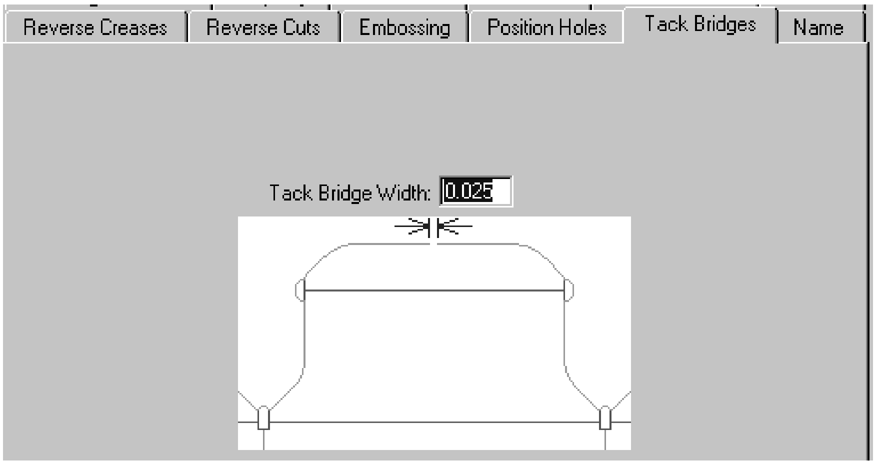
Position Holes tab

The **Position Holes** tab contains one parameter - the workspace name for the position hole. Click **Specify** to browse for a new workspace. In order to use the **Allow for tool diameter option** on the Counter Tool Widths tab in Outputs, you must make the outline of the hole in this workspace using the positioning hole tool line type. You should put any workspace you create in `ServerLib`.



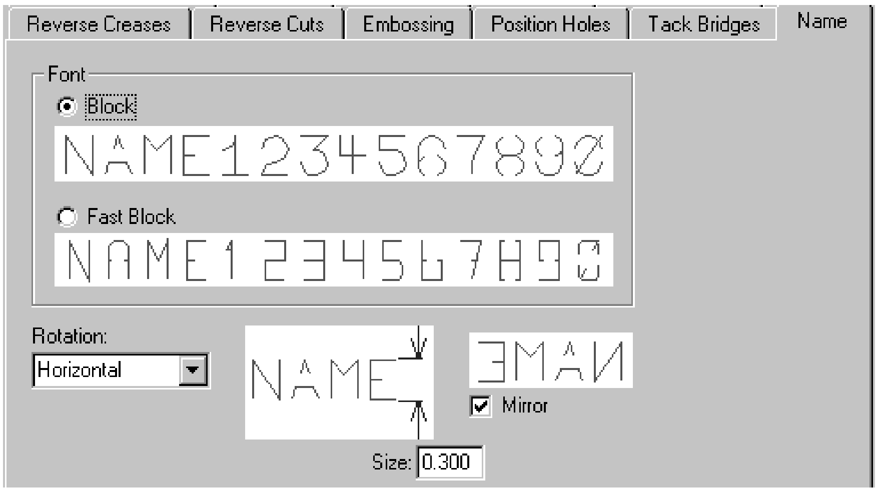
Tack Bridges tab

The **Tack Bridge Width** field on the **Tack Bridges** tab sets the width of tack bridges in the periphery of the counter for the next time the tack bridge tool is used, and it also sets the width of the tack bridges created automatically for bowties.



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.

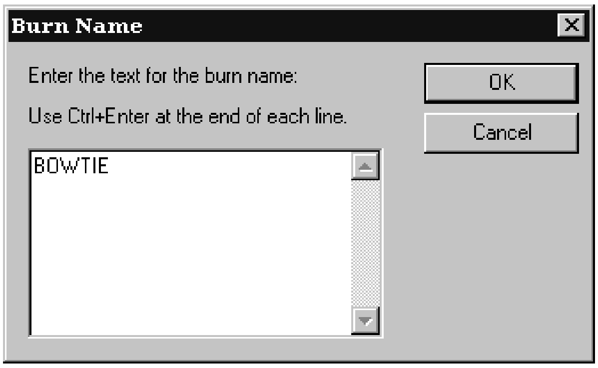


Once you have finished reviewing the options on the tabs, click **OK** to make or remake the counter and dismiss the Create or Rebuild Counter dialog box. Click **Cancel** to cancel the creation or modification of the counter. Click **Apply** to apply the changes you made to the options on the tabs while keeping the dialog box open.

The second button on the Counter toolbar activates the **Rebuild Chamfer** tool. Clicking this tool rebuilds the chamfer in the current counter layer. If you have made any manual changes to the chamfer, they will be lost. This tool also resequences the periphery lines into loops; if you have made any changes to the order, the changes will be lost.

The third tool on the Counter toolbar activates the **Add Counter Name** tool. This tool uses the options on the Name tab of the Create or Rebuild Counter dialog box to place the name of the workspace on the counter. Fields appear in the Status bar containing the attributes of the name.

The only thing different about the fields on the Status bar compared with those on the **Name** tab is the **...** button after the **Text** field. This button leads to the Burn Name dialog box, and by clicking it, you can enter a counter name spanning multiple lines of text.



Enter the text for the first line, press `CTRL-Enter`, enter the second line of text, and so forth, and then click **OK**. A ghost drag image of the text will appear under the cursor. Position it and click to place it on the counter. This tool remains active after placing the name; click another tool to deactivate it.

The **Add Counter Tack Bridge** tool is the fourth tool on the Counter toolbar and does just what it says. Click this tool, then indicate the position in the periphery where a tack bridge is to be placed. A tack bridge will be placed at the point you indicate with the width shown in the **Tack Bridge Width** field on the Status bar.

The fifth tool on the Counter toolbar is the **Delete Counter Tack Bridge** tool and it, too, does just what it says. Click this tool and then click the tack bridge(s) to be deleted.

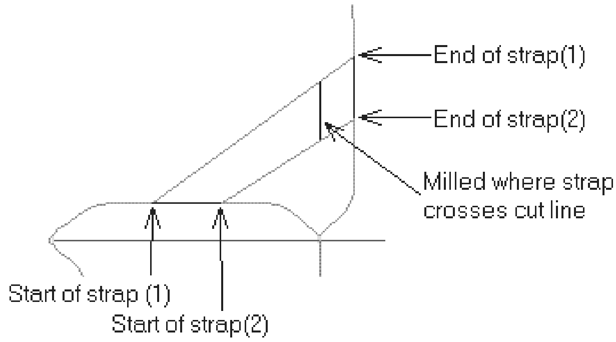
The sixth tool on the Counter toolbar is the **Add Counter Strap** tool. **Straps** are thin pieces of counter material that hold delicate counters together. Straps are made using the periphery tool and are milled where they cross cut lines.

You should use this tool after designing the rest of the counter. To use this tool, do the following:

1. Make sure your counter is complete except for the straps.
2. Click the **Add Counter Strap** tool on the Counter toolbar.
3. Click the point in the periphery where you want the first side of the strap to start.

4. Click the ending point of the first side of the strap in the periphery.
5. Click the start point in the periphery for the second side of the strap.
6. Click the ending point in the periphery for the second side of the strap.

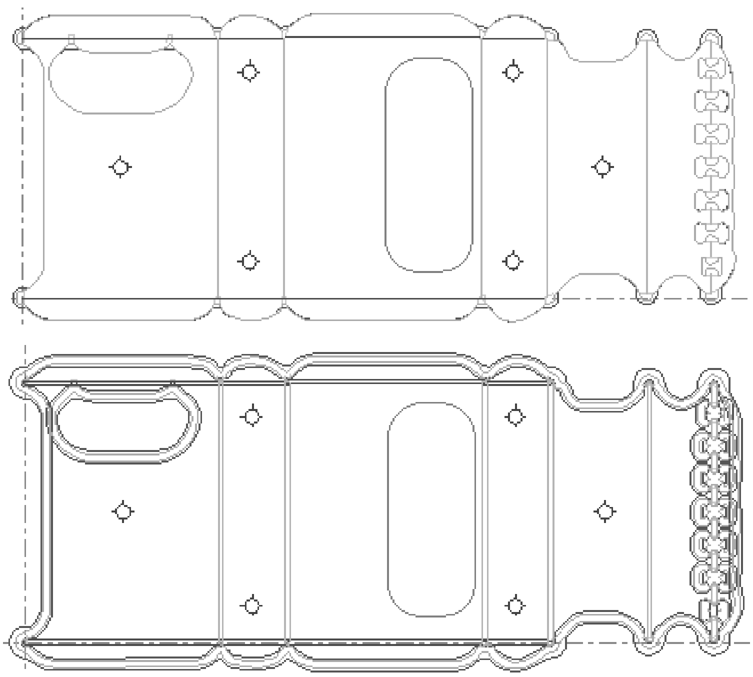
You should have a construction that looks similar to the one shown below.



 The sixth tool on the Counter toolbar is the **Select Periphery Lines** tool. As periphery lines are often close to other lines, selecting them alone can be imprecise using a normal Select tool. Use this tool to select only lines made with the periphery tool. This tool has the same selection options on the Status bar as the Select tool in Designer. Once the periphery lines are selected, you can then move them, delete them, change them, and so forth.

Viewing the counter

To view the actual paths made by the counter tools, check the **Counter channel widths** checkbox in the View Mode dialog box. Shown below is a counter with the **Counter channel widths** option off and then on.



Note: Milled areas are not shown filled in with mill tool lines when the **Counter channel widths** checkbox is selected. They are only shown in the Preview dialog box of the Output.

Counter Position Holes

Counter position holes are made by inserting Laser Positioning Holes into the single design before creating a counter. Add Laser Position Holes via **Tools** > **Geometry Macros** > **Utilities** > **Laser Position Holes**.

Construct the Laser Positioning Holes design file with the origin at the center of the hole using symmetrical geometry and consecutive lines. Use only the **Positioning Hole Tool** line type and the **Positioning Hole Tool 2** line type in the Laser Positioning Holes design file.

Switching counter parameter sets

To switch to a different counter parameter set after choosing one initially, click **Change Counter Parameter Set** on the **Options** menu. You should rebuild the counter after changing the parameter set.

Outputting imported INTERACT and LASERPOINT IQ counters

INTERACT counters use different line types than those used in ArtiosCAD. You must change all the counter lines to ArtiosCAD line types by hand.

When outputting imported LASERPOINT IQ counters, use a **Plot** output type instead of a **Counter** output type when configuring the Output.

Digitizing

Digitizing is the process of using a digitizer to input lines and arcs into ArtiosCAD. A **digitizer** is a flat tablet which uses electromagnetic energy to track movements of the digitizing puck over the surface of the digitizer. The geometric information derived from these movements is then sent to ArtiosCAD which translates it into lines and arcs. Digitizing is useful when you have a physical sample of a design but no CAD file to accompany it.

The digitizer must be set up before you can use it in ArtiosCAD. For information on setting up a digitizer, see the *Peripherals* chapter of the *ArtiosCAD Administrator Guide*.

There are three physical locations used when digitizing.

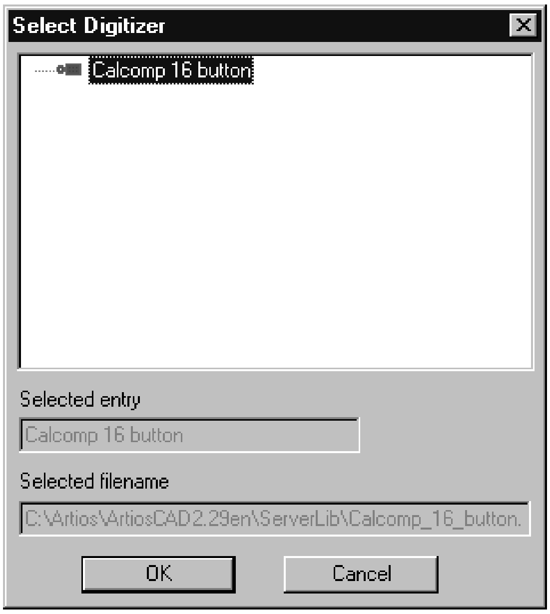
1. Any coordinate on the digitizer, referred to as <c>.
2. An existing point, referred to as <p>. A point, when digitizing, is an end point or intersection point of existing lines in the current design. These lines may be pre-existing or just-digitized. To select a point, digitize a point **on the digitizer** within the snap distance of one of these points.
3. An existing line, referred to as <l>. To select a line, digitize a point **on the digitizer** within the snap distance of an existing line in the design.

The workflow for digitizing is straightforward.

1. Create a new ArtiosCAD design or open an existing one.
2. Navigate to the desired layer into which you wish to place the new geometry.
3. Click **Tools**, **Digitize**, and then **Begin Digitizing**.
4. Select the digitizer to use.
5. Digitize as desired by issuing digitizing commands.
6. Activate any other non-Zoom tool in ArtiosCAD to exit Digitizing.

Starting to digitize

Once you have gotten to the point where you are ready to digitize (steps 1 and 2 in the workflow above), click **Tools** > **Digitize** > **Begin Digitizing**. You will be prompted to select a digitizer:



Select the digitizer to use and click **OK**. The status bar will change to look similar to the picture below.

The **Set Radius** field is used by some of the commands. There is no need to align the digitizer or place the menu at this time.

Secure the image to be digitized to the digitizer and start digitizing by issuing digitizing commands.

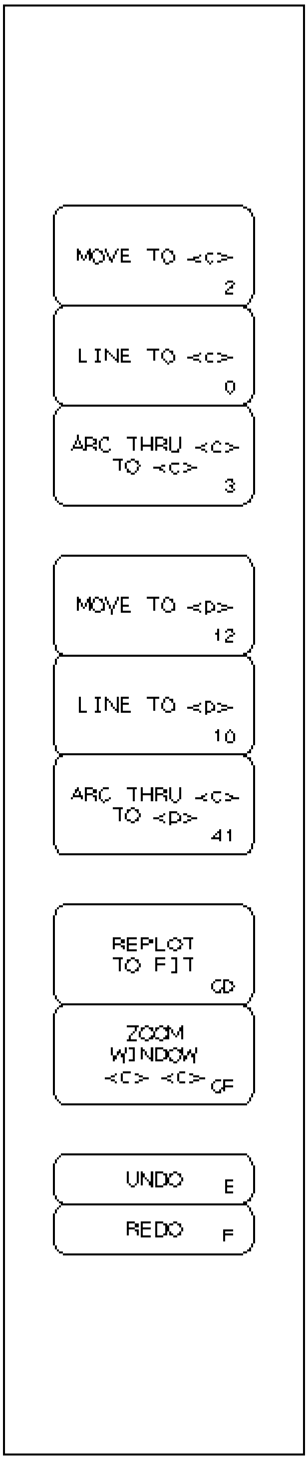
Issuing digitizing commands

There are two ways to issue digitizing commands. The first is by pressing the buttons on the digitizing puck. Press the button(s) that are listed in the table below to perform the command. For example, to create an arc through a coordinate, press 3 to indicate the coordinate through which the arc passes, and then 3 again to indicate the end coordinate of the arc.

Key sequence	Command
2	MOVE TO <c>
12	MOVE TO <p>
0	LINE TO <c>
10	LINE TO <p>
3	ARC THROUGH <c> TO <c>
41	ARC THROUGH <c> TO <p>
14	ARC THROUGH <p> TO <c>
44	ARC THROUGH <p> TO <p>
77	CENTER <c> RADIUS <c>
76	CENTER <c> RADIUS <p>
7A	CENTER <c> RADIUS radius
66	CENTER <p> RADIUS <c>
67	CENTER <p> RADIUS <p>
6A	CENTER <p> RADIUS radius
52	INTERSECT <l> <l>
51	INTERSECT <l> <l> KEEP 1st
BBB	BLEND <l> <l> THROUGH <c>
BB1	BLEND <l> <l> THROUGH <p>
BBA	BLEND <l> <l> RADIUS radius
D	DELETE <l>
E	UNDO
F	REDO
CD	REPLOT - SCALE TO FIT
CC	REPLOT - CURRENT SCALE
CA	ZOOM BACK
CF	ZOOM WINDOW <c> <c>

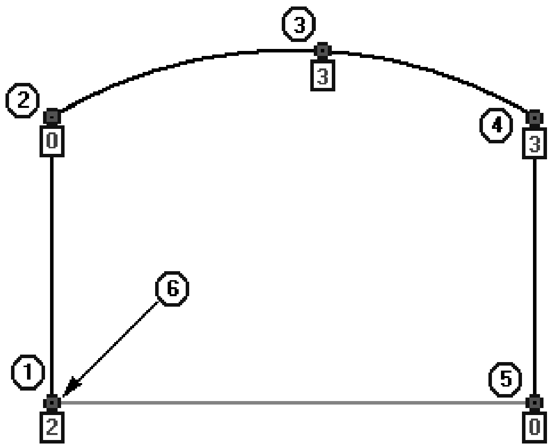
If a command requires more keystrokes to identify it than there are data elements required for the command, the most recent digitized points are used.

The second way to issue commands is to use the digitizing menu configured when you set up the digitizer. Simply place the cross-hair over the desired menu command and press any button on the digitizing puck to activate the command. You can also use any button on the digitizing puck to indicate coordinates, points, and lines since the command is already known. Shown at left is the default digitizing menu. Note that it does not contain all the commands available. If you are a gasket maker, you may wish to add the circle-creation tools to it using the procedure in the *Peripherals* chapter of the *ArtiosCAD Administrator Guide*.



Example - Digitizing a simple shape

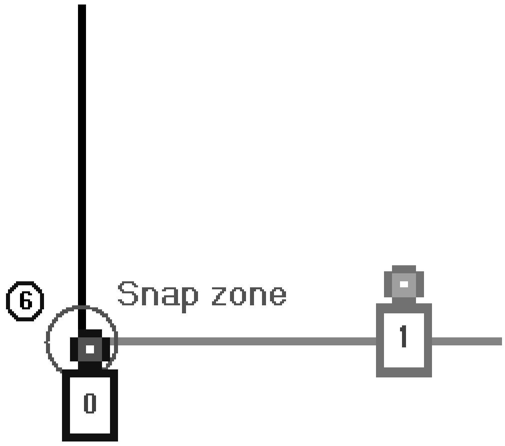
This example shows how to digitize the shape below using the commands that follow it. The numbers in the circles are the step numbers, while the numbers inside the digitizing puck are the buttons pressed on the digitizing puck.



Key sequence	Menu command
0	LINE TO <c>
10	LINE TO <p>
2	MOVE TO <c>
12	MOVE TO <p>
3	ARC THROUGH <c> TO <c>

Start ArtiosCAD and open a new design. Click **Tools** > **Digitize** > **Begin Digitizing**. Choose the digitizer to use and click **OK** to enter Digitizing.

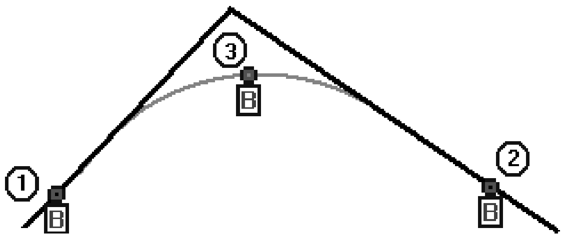
1. Digitize a MOVE to the beginning of the first line using the **2** button.
2. Digitize the end of the first LINE by moving the digitizing puck up a few inches and pressing the **0** button.
3. Digitize a coordinate the arc will go THROUGH by moving the puck up and to the right and pressing the **3** button.
4. Digitize the end-point of the arc by moving the puck down and to the right and by pressing any button. Any button will do because ArtiosCAD knows the ARC THROUGH <c> TO <c> command is being used.
5. Digitize the end of the next LINE by moving the puck straight down and pressing the **0** button.
6. Digitize close to the snap point at the start of the first line. The command is made of two button-presses. The first button pressed is **1**, and can be pressed with the digitizing puck anywhere. The second button pressed is **0**, and must be within the snap tolerance of the start of the first line.



The shape is now complete.

Example - Blending a digitized corner

This example demonstrates how to blend a corner while digitizing. As in the previous example, the number inside the circle is the step number and the character inside the digitizing puck is the button pressed.



Start ArtiosCAD and open a new design. Click **Tools** > **Digitize** > **Begin Digitizing**. Choose the digitizer to use and click **OK** to enter Digitizing.

1. Digitize within the snap tolerance of the first line by pressing the **B** button.
2. Digitize within the snap tolerance of the second line by pressing the **B** button.
3. Digitize a THROUGH coordinate of the blend radius by pressing the **B** button again to complete the command.

Aligning the image on the digitizer

It can be difficult to place the image on the digitizer so that it is completely aligned with the digitizer's axes. To compensate for this, the digitizer can be aligned with known points on the image and known points in the ArtiosCAD Design window. For example, the item to digitize may be a cutout on an existing carton. The corners of the panel will therefore make good points for aligning the image to the design on the screen.

Between 0 and 3 points can be used to align the digitizer with the ArtiosCAD design window.

If 0 points are used, the origin of the digitizer matches the origin in ArtiosCAD.

If 1 point is used, the point selected on the screen corresponds to the digitized point. No scaling or rotation is performed.

If 2 points are used, the first point selected on the screen corresponds to the first point digitized. Scaling and rotation will be performed so that the second points align.

If 3 points are used, a best-fit is performed on the three points on the screen compared to the three points on the digitizer. The image is transformed, scaled, and rotated appropriately.

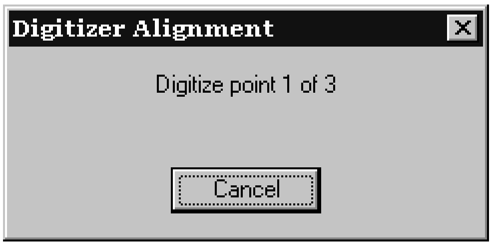
When aligning the digitizer, the screen points are selected first followed by the points on the digitizer.

To align the digitizer, do the following:

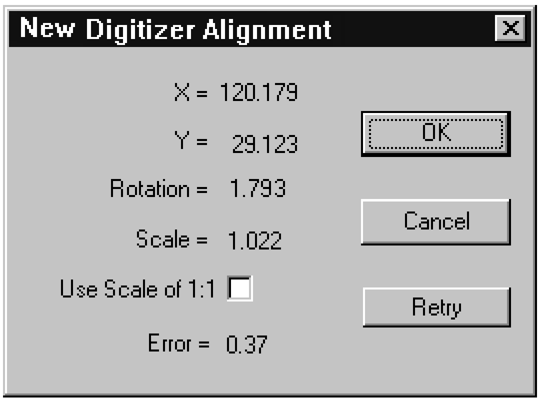
1. Start ArtiosCAD and open a new design. Click **Tools** > **Digitize** > **Begin Digitizing**. Choose the digitizer to use and click **OK** to enter Digitizing.
2. 

Click **Align Digitizer** on the Status bar.
3. The Status bar will change:

4. Click the mouse on-screen at the desired point. The X and Y fields will have values in them and the point number will increment. If you make a mistake, click the **Redo** button.
5. Repeat step 4 as desired depending on how you want to align the digitizer.
6. Click **OK** when done. If you have selected three points, you do not need to click OK.
7. Now that the screen points are selected, you must indicate the digitizer points.



8. Move the digitizing puck to the first point on the digitizer and press any button. Repeat as prompted, moving to the second and third points if necessary.
9. Once enough points have been digitized, the New Digitizer Alignment dialog box appears.



The new alignment is displayed. If two or more points were used, the scale might not be 1.000 and the Error will show how accurate a fit was made.

Checking the **Use Scale of 1:1** checkbox overrides the scale derived from the digitized points.This is useful when the scale factor isn't quite 1.000 but you want to use the rotation derived from the digitizer.

Click **Retry** to perform the alignment procedure again.

Clicking **OK** accepts the new alignment; clicking **Cancel** cancels it. The new alignment is effective until ArtiosCAD is restarted.

Using the digitizing menu

The digitizing menu provides an alternate way to use the digitizer. Selecting a point on a menu button (also called a **Menu Item Area**) with any button on the digitizer puck activates that command.

Using the menu means pressing more buttons on the digitizing puck than just by using the puck alone. For example, if you have a menu button which activates the **CENTER <p> RADIUS <c>** tool, two extra button-presses are required after selecting the button — one to satisfy the <p> and the next to satisfy the <c>. Any button can be pressed to make the data input.

Placing the menu is described in the *Peripherals* chapter of the *ArtiosCAD Administrator Guide*.

Projects

ArtiosCAD provides two ways to work with Projects. The first way is using the Projects menu, which is the method used in this chapter. The second way is using the Projects browser in DataCenter Admin, which is described more in the *DataCenter* chapter.

Getting started with Projects

Projects help to track and identify the relationships between designs. Use Projects to group logical single designs together, such as a display base and the items it will hold, or a shipping box and its dividers.

Projects are also used for generating Bill of Materials (BOM) Reports. See the *Outputs* chapter for more information about BOM Reports.

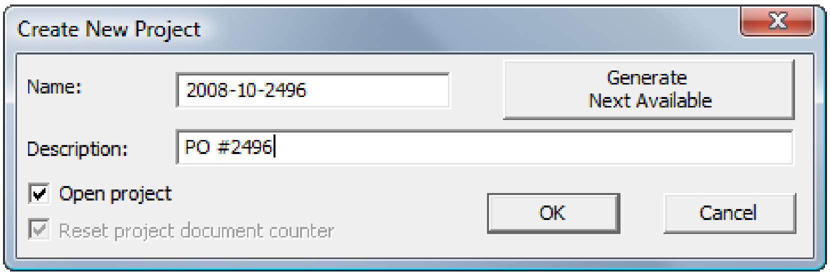
As with single designs, ArtiosCAD can assign names automatically to Projects and the documents within them if the appropriate configuration is performed in DataCenter Admin. See the *DataCenter* chapter for more information.

Creating a new Project

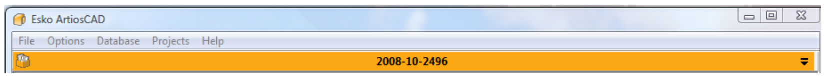
To create a new Project within ArtiosCAD, do the following:

1. Click **Projects** > **New Project**. The **Create New Project** dialog box appears.
2. Enter the appropriate information in the **Name:** and **Description:** fields, or, if available, click **Generate Next Available** to have ArtiosCAD assign the next available number as the Project

name. (This requires the proper configuration in DataCenter Admin.) The Project name must be unique and less than 80 characters.



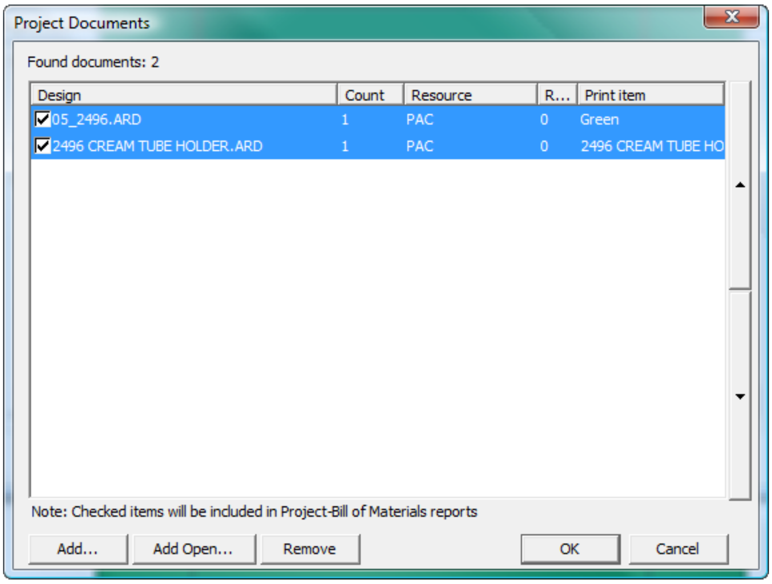
3. Click **OK** to create the Project and open it. To simply create the Project without opening it, deselect **Open project** before clicking **OK**.
4. ArtiosCAD automatically opens the **Project Information** dialog box so you can add more detailed information about the Project.
   - If you set a description when you created the Project, you can change it by editing the **Description**: field.
   - Set the **Customer**:, **Owner**:, **Salesperson**:, and **Manager**: fields by clicking the drop-down list box arrows and choosing an entry from the list shown for each field. To access the standard database searching dialog box for each type of entry, click **...** at the end of each field.
   - To set characteristics or userfields for the Project, click the corresponding **Characteristics** or **Userfields** buttons. Projects have their own type of Userfields which must be configured in DataCenter Admin before they become available for use in ArtiosCAD. For more information about configuring userfields, see the *DataCenter* chapter.
5. If you chose to open it, the empty new Project is shown on the document bar.



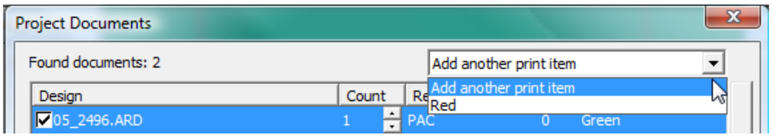
Adding documents to a Project

To add documents (single design workspaces) to a newly-created Project, do the following:

1. With the newly-created Project open, click **Projects** > **Project Documents**.
2. Click **Add** in the **Project Documents** dialog box.
3. In the **Add document(s) to project** dialog box, in the **Add from**: drop-down list box, select the Resource containing the desired documents.
4. Select the document to add by clicking its name. To select more than one document, hold down `CTRL` on the keyboard and select the desired documents.
5. Click **Add**. ArtiosCAD adds the selected document(s) to the Project and lists the results in the **Project Documents** dialog box as shown below.



6. If a selected document has more than one print item, you can add another print item to the Project by selecting it from the drop-down list box that appears above the list of documents.

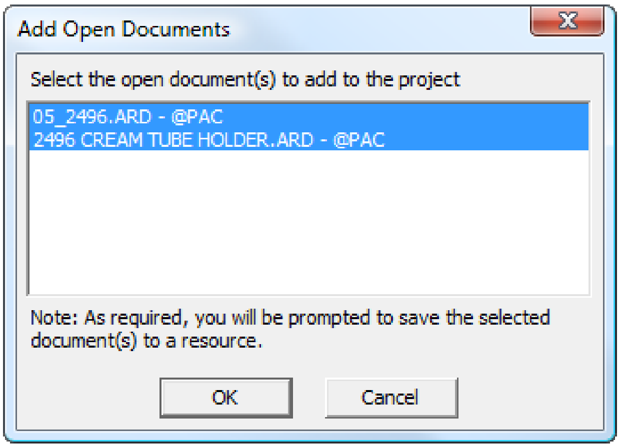


7. Click **OK** to return to ArtiosCAD. ArtiosCAD does not automatically open the documents you add to a Project.

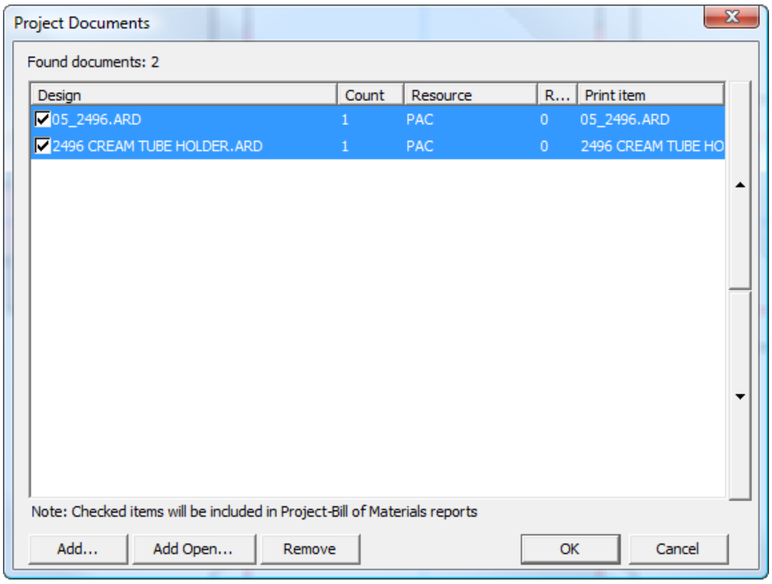
Adding open documents to a Project

To add open documents to a Project, do the following:

1. Open the documents and the Project to which they will be added.
2. Click **Projects** > **Project documents**.
3. Click **Add Open**.
4. In the **Add Open Documents** dialog box, all the open documents are preselected. Click any document to deselect it from being added to the Project. If you make a mistake, hold down `CTRL` and select the documents to add to the Project.



5. Click **OK** to add the documents to the Project. If any of the designs are not in a Resource, ArtiosCAD prompts you to save it/them in a Resource.
6. The documents appear in the **Project Documents** dialog box as shown below. Click **OK** to return to ArtiosCAD.



Opening a Project

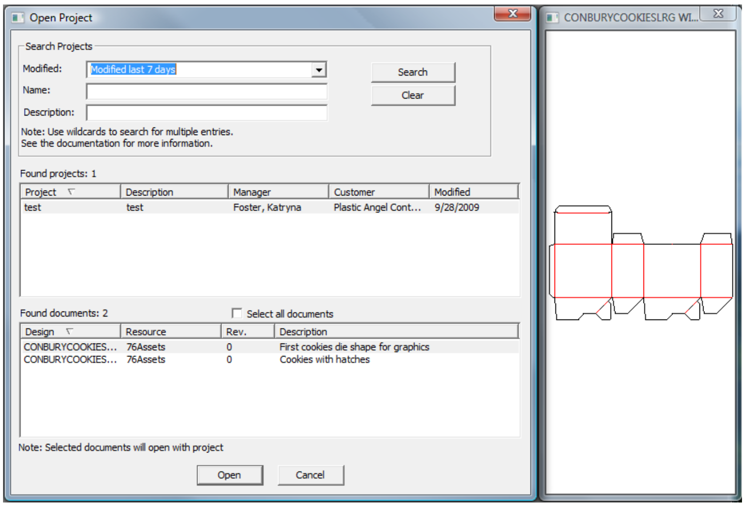
ArtiosCAD keeps a list of the most-recently used Projects on the Projects menu. To open one of those most-recently used Projects, simply click it; ArtiosCAD also opens the documents that were open when the Project was last closed.

Note:

The number of saved Projects is configured in **Options** > **Defaults** > **Startup defaults** > **Most Recently Used Project List**.

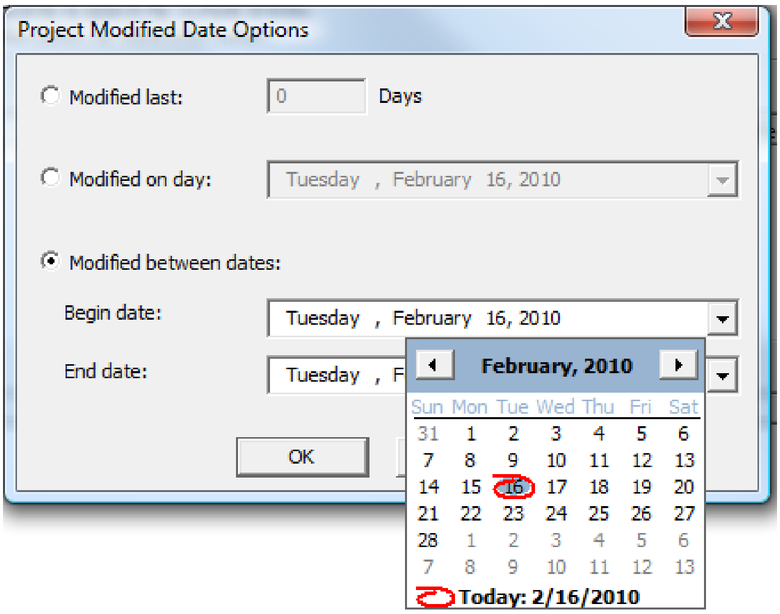
To open a Project that is not on the most-recently used Project list, do the following:

1. Click either **File** > **Open Project** or **Projects** > **Open Project** to open the **Open Project** dialog box.



2. To open a Project using the **Open Project** dialog box, perform a search and then select the desired Project from the search results. You can search by the date the Project was last modified, by the name of the Project, the description of the Project, or by a combination of the three.

   - The **Modified:** drop-down list box lets you search for Projects modified during the last day, last 7 days, last 30 days, or a range of your choice by choosing **More options**. When you choose **More options**, you can specify the number of passed days within which to search for the Project, or you can set the beginning and ending dates of the range within which to search for the Project. Clicking either the **Begin date:** or **End date:** drop-down list box arrow when **Modified between dates** is selected activates a pop-up calendar that you use to choose the date to begin or end the range of date-searching. Click **OK** to return to the **Open Project** dialog box.



Note:

To not search by time the Project was last modified, set the **Modified** drop-down list box to **----**.

Note:

When using the drop-down calendar in any of the fields, you can click the month and the year in the date and enter a different month or year, or use the up and down arrow buttons to change years.

- To form a search phrase in either the **Name:** or **Description:** fields beginning with a certain combination of letters, enter that combination of letters and an asterisk (*), such as 2496*. To form a search phrase containing a certain combination of letters, enter that combination preceded with and followed by asterisks, such as *2496*. To form a search phrase equaling an exact combination of letters, enter that combination of letters. To use an wildcard single-character match, use ? in place of the character, such as *24?6*, which will find all Projects containing 2406, 2416, 2426, and so forth.

Note:

% may be substituted for * in “match all” searches and _ may be substituted for ? in “match only one” searches. For more information about searching, see the Projects section in the *DataCenter* chapter.

3. Once you have chosen the method by which to search - **Modified:**, **Name:**, or **Description:**, and have entered the appropriate information, click **Search**. ArtiosCAD displays all matching projects in the **Found projects** pane, with the first Project selected. The documents in the first Project are listed in the **Found documents** pane, with the first one selected and shown in the Preview window.
4. To open the Project and the selected document, click **Open**. To open the Project and all its documents, check **Select all documents** and click **Open**. Clicking **Cancel** closes the dialog box.

 You can also open a Project by selecting it in the Project Browser and clicking **Open Project**.

A Project’s Modified date gets updated when you change any of the following:

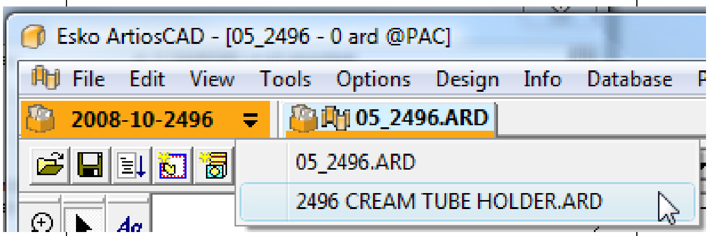
- Any database information, such as the description, customer, owner, and so forth
- Any characteristics
- Any userfields or their values
- Any documents (add or remove documents, change the BOM count, and so forth).

Opening documents in a Project

ArtiosCAD provides two ways to open Project documents. To open documents in a Project, do the following:

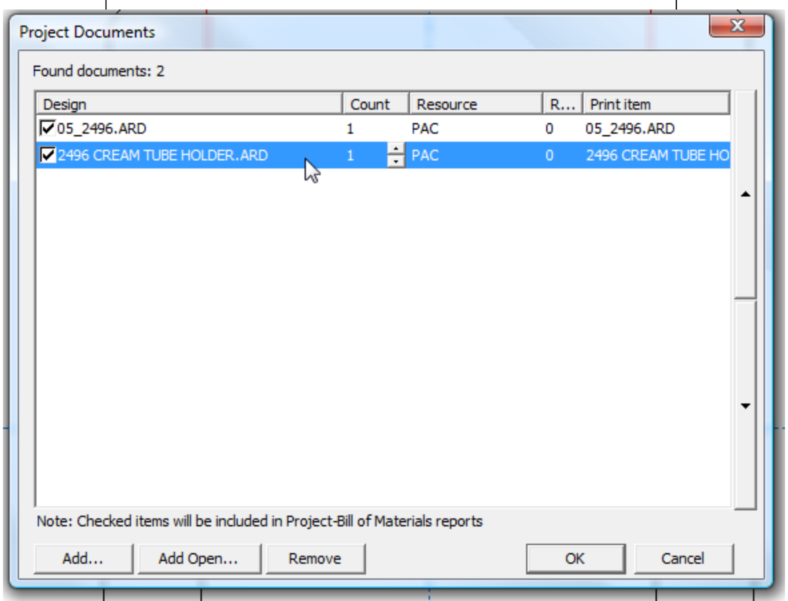
1. Open the Project containing the desired documents. If you use the Most-Recently Used Project list on the Project menu to open the Project, ArtiosCAD opens the documents that were open when the Project was last closed.
2. Decide how you want to open the documents, and then proceed using the choices in the table below.

<table>
<tr><th>To open documents by...</th><th>Do this...</th></tr>
<tr><td>Using the document bar</td><td>
Click the Project button on the document bar and then click the name of the document to open.



ArtiosCAD opens the selected document and shows the Project icon in its tab.

Defaults for the document bar are set in Options > Defaults > Startup defaults > Document Bar options. Here you control when the Document Bar is shown and the color of the Project button.
</td></tr>
<tr><td>Using a dialog box</td><td>
Click Projects > Project documents, and in the Project Documents dialog box, double-click the document to open.

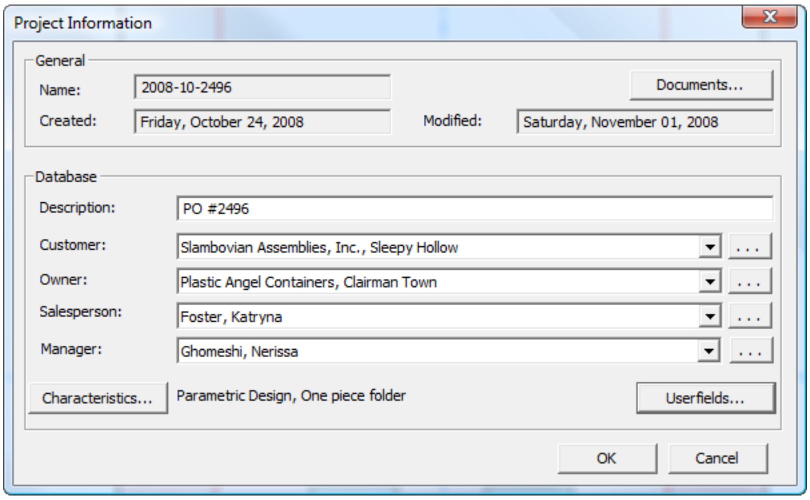


</td></tr>
</table>

To open documents by...	Do this...
	ArtiosCAD closes the **Project Documents** dialog box, opens the selected document, and shows the Project icon in its tab.

Setting Project information

Each Project may have a description, a customer, an owner, a salesperson, a manager, characteristics, and Project userfields assigned to it using the **Project Information** dialog box.ArtiosCAD prompts for this information when you create a new Project, but you can change this information at any time. Shown below is the default **Project Information** dialog box. You may design a custom Project Information dialog box in Defaults.



To set Project information after the Project has been created, do the following:

1. Open an existing Project.
2. Click **Projects** > **Project Information**.
3. If you set a description when you created the Project, you can change it by editing the **Description**: field.
4. Set the **Customer**:, **Owner**:, **Salesperson**:, and **Manager**: fields by clicking the drop-down list box arrows and choosing an entry from the list shown. To access the standard database searching dialog box for each type of entry, click **...** at the end of each field.
5. To set characteristics or userfields for the Project, click the corresponding buttons **Characteristics** or **Userfields**. Projects have their own type of Userfields which must be configured in DataCenter Admin before they become available for use in ArtiosCAD. For more information about configuring userfields, see the *DataCenter* chapter.
6. To add or remove documents to or from the Project, click **Documents**, which opens the **Project Documents** dialog box as described earlier.

7. Click **OK** to return to save the changes and return to ArtiosCAD. Any changes made to the information are saved to the database once you click **OK**.

Closing a Project

To close the active Project, do the following:

1. Click **Projects** > **Close Project**.
2. If any Project documents are open, ArtiosCAD will ask if you want to close them. Click **Yes**.
3. If any Project documents being closed have unsaved changes, ArtiosCAD will ask if you want to save changes. Click **Yes** or **No** as desired.

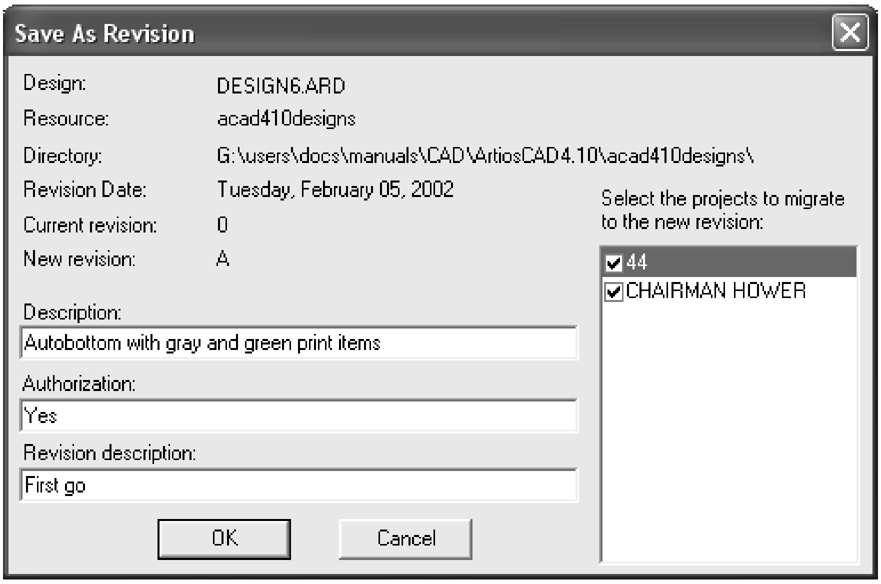
Notes and warnings for Projects

A **Projects** item has been added to the General information category of the calculated text table. When used on a Report, this item generates a comma-separated list of Project codes.

Revisions

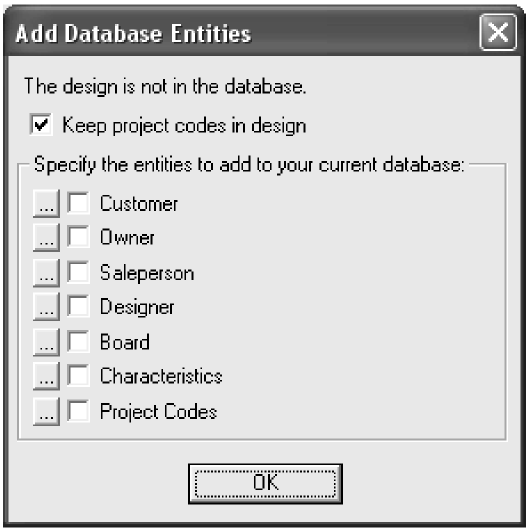
A Project may only contain one revision of a single design.

When you click **Save As Revision** in a single design that is part of a Project, the Save As Revision dialog box will show a list of the Projects of which the single design is a member. Migrate the Projects information to the new revision by selecting the checkbox next to the Project name. To keep the Project information in the old revision, clear the checkbox next to the Project name.



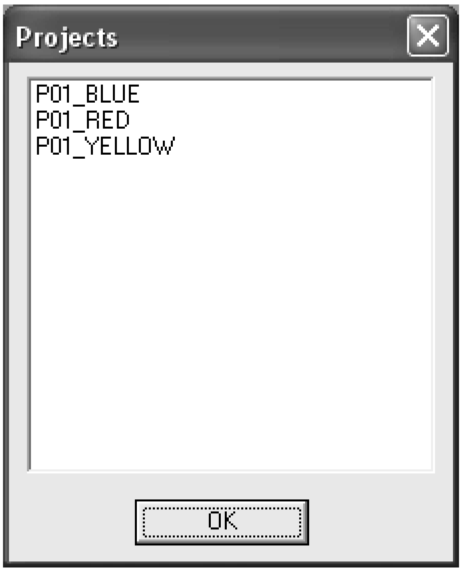
Opening files from another database

When you open a single design containing records from another database, the Add Database Entities dialog box appears. In this dialog box, choose which entities to add to your database by checking the checkbox next to each available category. The enabled categories depend on which entities are set in the single design.



The **Keep project codes in design** checkbox controls whether or not ArtiosCAD leaves the Project codes from the foreign database in the single design or if it strips them out.

To see the specific entities for each category of database entities, click **...** next to the checkbox. All entities will be added if you check the checkbox; there is no way to choose which specific entities are added to the database. Specific Project entities are shown below; click **OK** to return to the Add Database Entities dialog box.



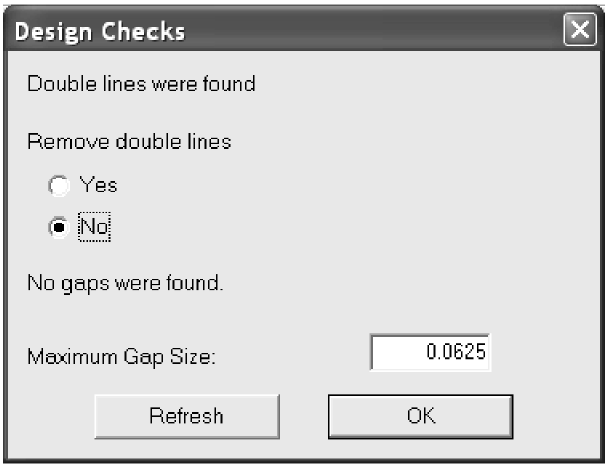
To add a design containing Project codes from another database to your database and additionally have the Project codes in that design added to your database, do the following:

1. Put the design into a resource (whether by saving it from an email message, copying it off a floppy disc, or whatever method you use).
2. Start ArtiosCAD and open the design.
3. In the Add Database Entities dialog box, check both the **Keep project codes in the design** and the **Project Codes** checkboxes.

4. Save the design, making sure to not change the name or resource.

Design Checks

Use **Design Checks** on the Design menu to check the design for double lines and gaps in the perimeter.



To remove double lines, click **Yes** and then click **OK**. To leave them unchanged, click **No**. 50 double lines are reported initially when the check is run.

Maximum Gap Size sets the size of the largest gap that will be found by Design Checks. The tool finds gaps smaller than the value set for this field. If it finds any gaps, they are circled with magenta circles.

To search for other gaps, change the value in the **Maximum Gap Size** field and click **Refresh**.

If **Design check before Save, Save as** is checked in **Options** > **Defaults** > **Shared Defaults** > **Design Defaults** > **Save Options**, the design is checked for double lines and gaps in the perimeter of the design when it is saved. This option is off by default.

6. Manufacturing

Introduction to Manufacturing

Manufacturing is the module of ArtiosCAD that makes production tools from single designs. These tools are mounted in presses. As the sheets of board move through the press, the tools cut the design out of the sheet and remove interior and exterior waste.

Within Manufacturing are submodules that perform different tasks. Each module is purchased separately according to your individual needs.

Layout manually arranges single designs on sheets of board.

DieMaker creates flat dies.

Rotary DieMaker creates rotary dies.

Stripping removes interior and exterior waste.

Intelligent Layout automatically makes many different layouts, compares their waste percentages, and lets you choose the best layout to make.

Cost/Estimating calculates how much a layout will cost to make and run across multiple cost centers.

Intelligent Counters, in the context of Manufacturing, manually arranges counters on a sheet of counter material for milling on a countermaker.

Designing manufacturing tools usually follows this general workflow:

1. Create a layout.
   - Create a new blank layout and add single designs to it, **or**
   - Convert a single design to a layout, **or**
   - Convert a single design to a Standard Sheet layout.
2. Create tooling for that layout.
   - Create a die wood edge.
   - Add stripping rules, carrier rules, and a mounting bar. The stripping rules cut the waste into smaller pieces for easier removal, and the carrier rules prevent the sheet from sagging as it moves through the press.
   - If using a rotary dieboard, create rule paths and add splits if necessary.
   - Add stripping components, rules, pins, and blocks to rid the sheet of waste. The upper and lower stripping boards (also known as top and bottom, male and female) are created automatically.

Within this two-part workflow there are additional decisions to make. Will you make your own nest or use the automatic nesting tools? Is Intelligent Layout or Costing/Estimating available? If you are a folding carton user and have Intelligent Layout, do your designs have print items defined? If you are a corrugated user and have Costing/Estimating, do you have an idea of possible order quantities desired?

When you open saved Manufacturing files, the layers displayed are set according to the defaults in **Options** > **Defaults** > **Shared Defaults** > **Design Defaults** > **Default View Mode**.

Manufacturing prerequisites

Before designing a layout and creating tooling, you should make sure of the following:

- Machines are set up in Defaults.
- Cost centers are set up in Defaults (if you plan to use Intelligent Layout and/or Costing and Estimating).
- Board weights and costs are set correctly in DataCenter Admin.
- If intending to use Standard Sheet Layout, that standard sheet sizes are set up in Defaults. See the *Defaults* chapter of the *ArtiosCAD Administrator Guide* for more information.

In addition, the Manufacturing modules have the following limits:

- For manual layout, the maximum number of repeated designs is 300.
- For Intelligent Layout, the maximum number of repeated designs is 255.
- The maximum number of different designs in a layout is 100.
- The maximum number of design/print item combinations in Intelligent Layout is 100.
- A single design may have up to 100 print items.
- Each single design included must have a unique filename.

Standard Sheet Layout

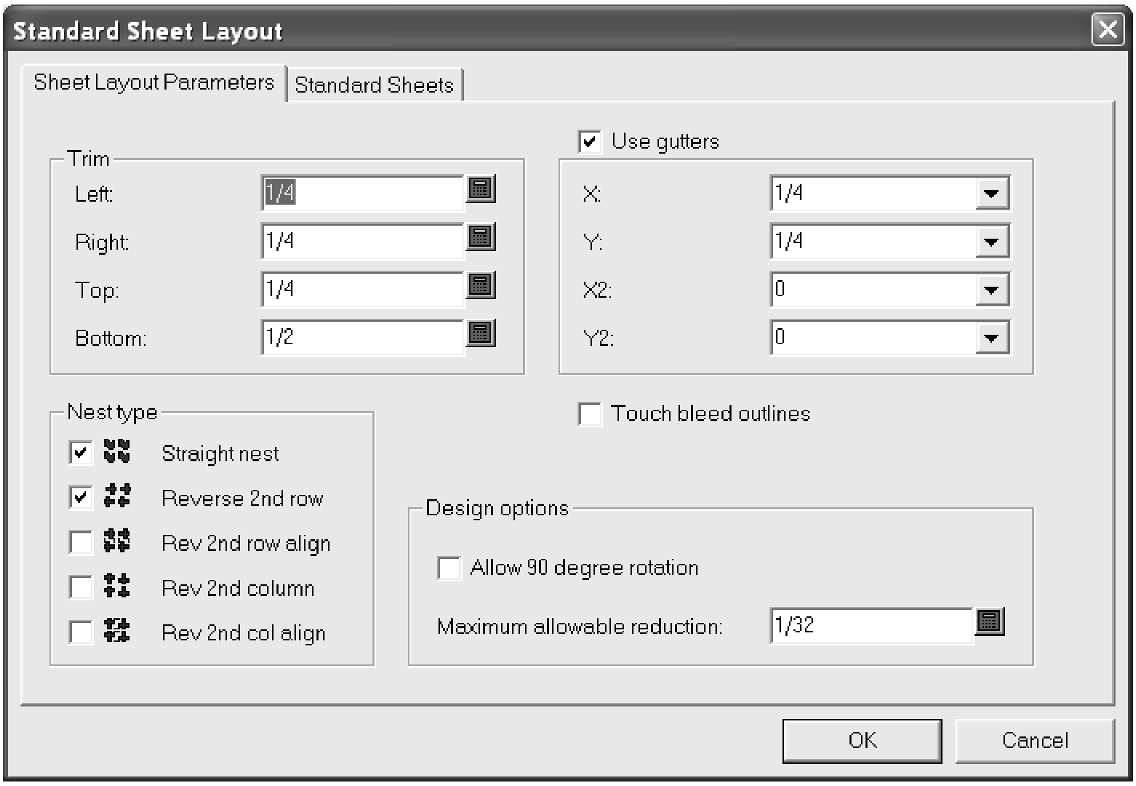
Standard Sheet Layout quickly nests a single design on a sheet and lets you choose the layout to make from a list of automatically generated layouts. It works with one single design workspace; to make a combination layout with more than one single design, either use manual layout or Intelligent Layout. Standard Sheet Layout requires Intelligent Layout.

To use it, do the following:

1. Open the single design to turn into a layout.
2. Click **File** > **Standard Sheet Layout**. The Standard Sheet Layout dialog box opens.

   Sheet Layout Parameters tab

   The Sheet Layout Parameters tab is the first tab in the Standard Sheet Layout dialog box 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** drop-down list boxes 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.

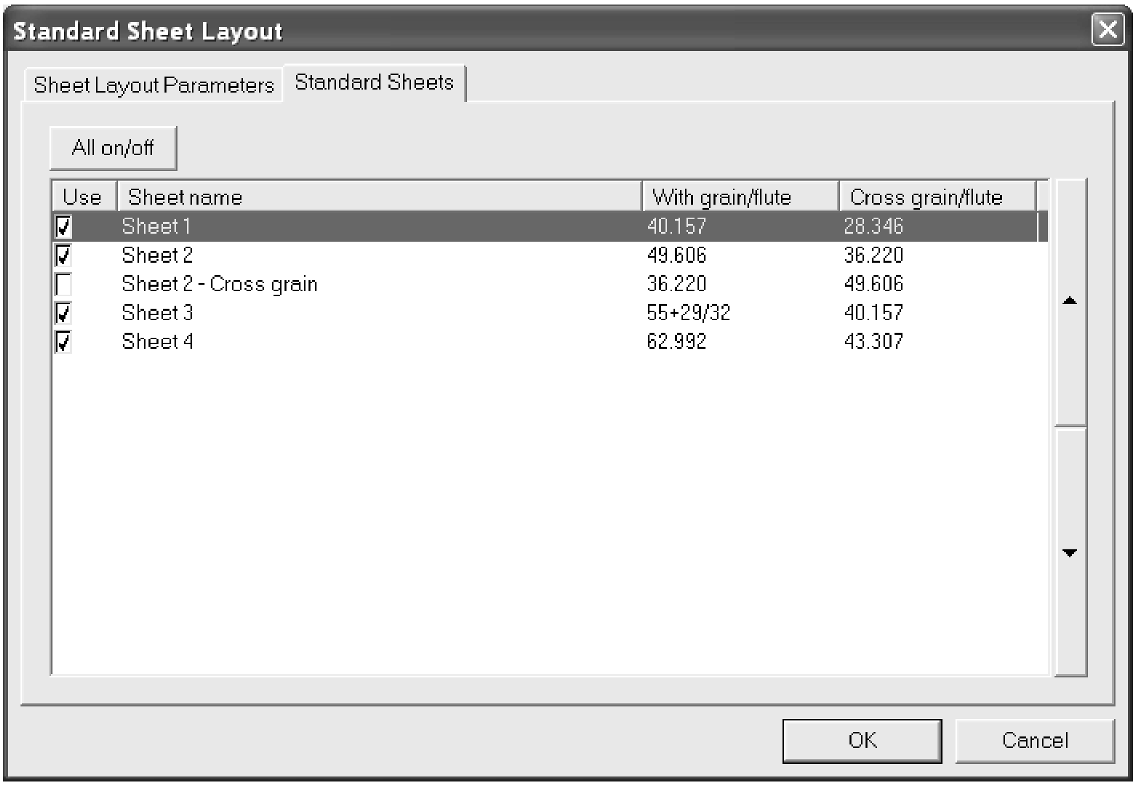
In the **Nest type** group, the checkboxes control the kinds of nests made automatically. Select the checkboxes next to the type of nests to use; clear those next to undesired nests.

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.

Standard Sheets tab

1. Click **Standard Sheets** to view or modify the values on the Standard Sheets tab as shown below.

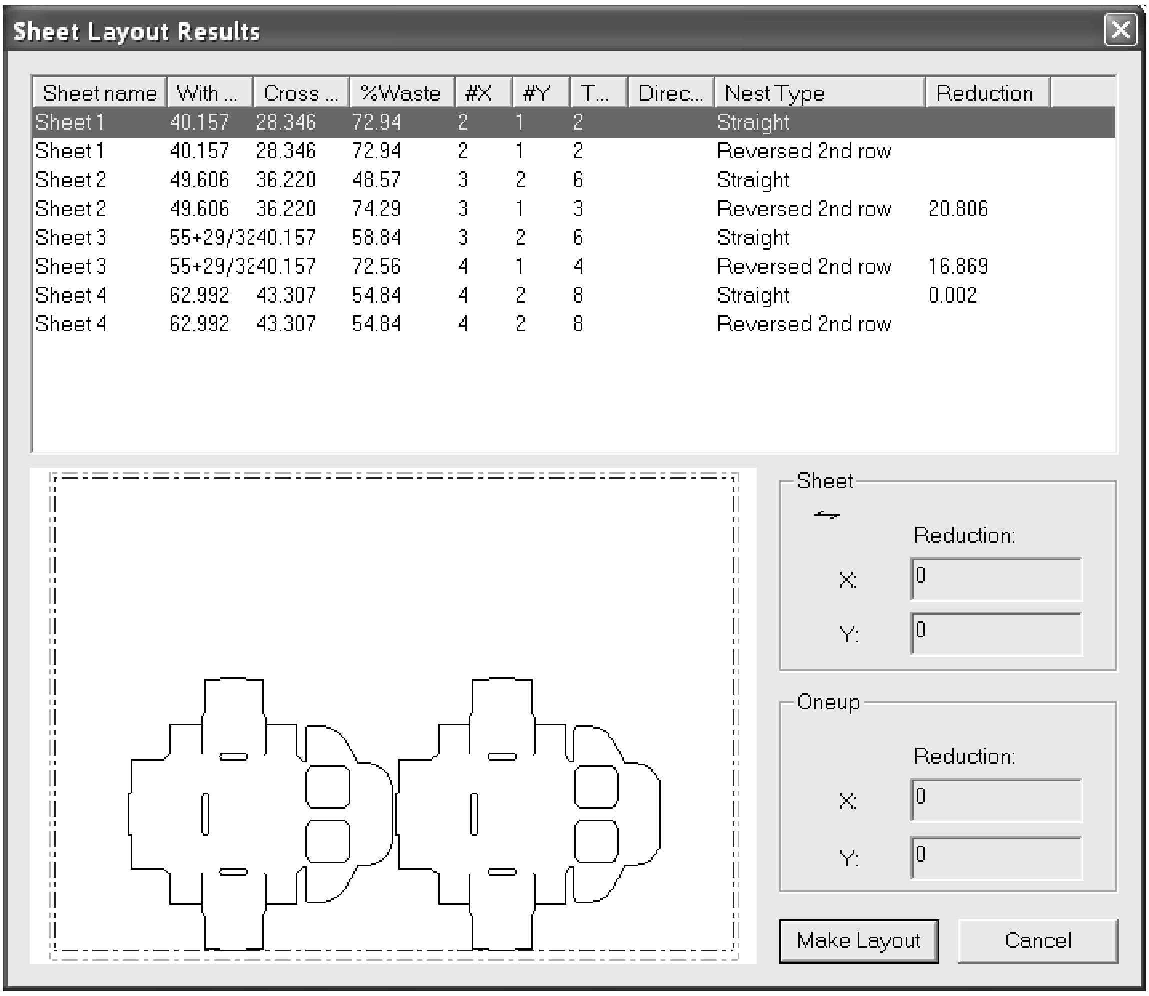


Listed on the tab are the standard sheets that are defined in Defaults. To use a sheet for possible layout solutions, check its box in the **Use** column. To disable its use, clear the checkbox. To toggle the usage of all sheets at once, click **All on/off**.

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.

Viewing results

1. Click **OK** to generate prospective layouts. The Sheet Layout Results dialog box opens as shown below.



The list of prospective layouts is initially sorted first by sheet used and then by nest type. Click any column heading to sort by that column. The first three columns repeat the entries in the sheet table, while the remaining columns display information about the prospective layout. **%Waste** is the percentage of waste on the sheet ignoring holes in the single designs. **#X** is the number of single designs across the sheet and **#Y** is number back. **Total** displays the number of single designs on the sheet. The **Direction** column is blank if the all single designs are with-grain, or displays **Rotated** if that option was enabled and used to construct the prospective layout.

The selected prospective layout is shown in the preview window. Click another layout to see its preview.

The **Sheet** group shows the grain/corrugation direction and the amount of total reduction needed for another row or column of single designs to fit on the sheet.

The **Oneup** group shows the reduction needed in each single design for another row or column of single designs to fit on the sheet. If the single design was rotated, the word **Rotated** appears in red.

Click **Cancel** to return to the Standard Sheet Layout dialog box to adjust parameters.

1. Click **Make Layout** to make the selected prospective layout.
2. Choose the parameter set and presses for the layout and click **OK**.

3. The layout is created and can be modified as desired. To create tooling, *#id1287874UNG28*

Notes and warnings

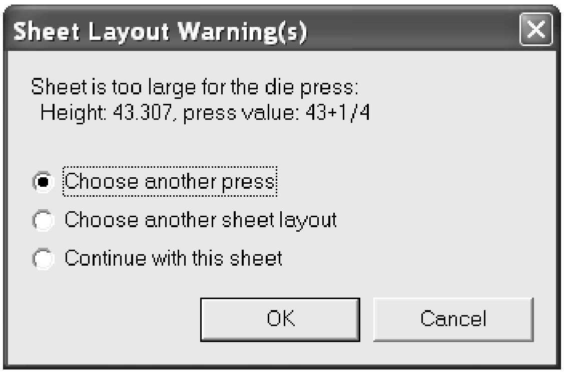
Sheet sizes have the with-grain direction first and the cross-grain direction second.

A single design is initially placed on the sheet so that its grain direction matches that of the sheet. If your single design is being rotated inexplicably and you did not check **Allow 90 degree rotation**, check its grain direction in Designer to see if it matches that of the sheet. The **Reverse** nest types and the gutters are implemented relative to the laid-out orientation of the single design.

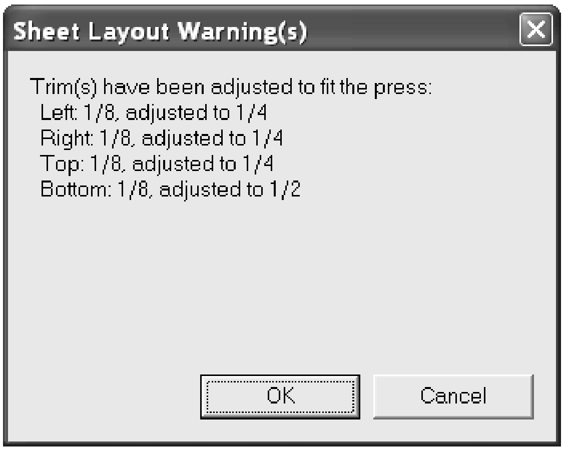
After using Standard Sheet Layout, the original single design workspace is still open and unsaved behind the layout. When you save the workspace, the manufacturing parameter set information used to construct the layout is saved in it. If you later convert it to manufacturing or another Standard Sheet Layout, the presses you selected the first time are selected by default and the **Use design parameter set** and **Use design press** checkboxes are checked. To choose a different parameter set or press, deselect those checkboxes.

To create a workspace that does not have this manufacturing information in it, create an Information Filter output that does not include Manufacturing parameter sets, and then output the workspace through that Output.

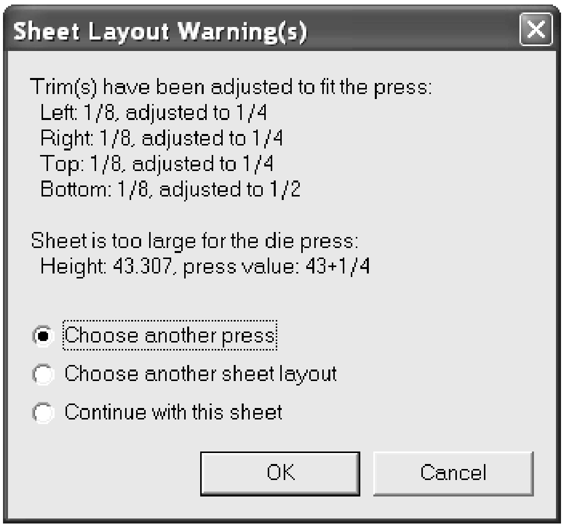
If the sheet size of the selected prospective layout conflicts with the one in the manufacturing parameter press, a warning dialog box appears as shown below after **Make Layout** is clicked. Choose the appropriate choice and click **OK,** or click **Cancel** to return to the single design.



If the trim is too small, the following dialog box is displayed. Click **OK** to make the layout or click **Cancel** to return to the single design.

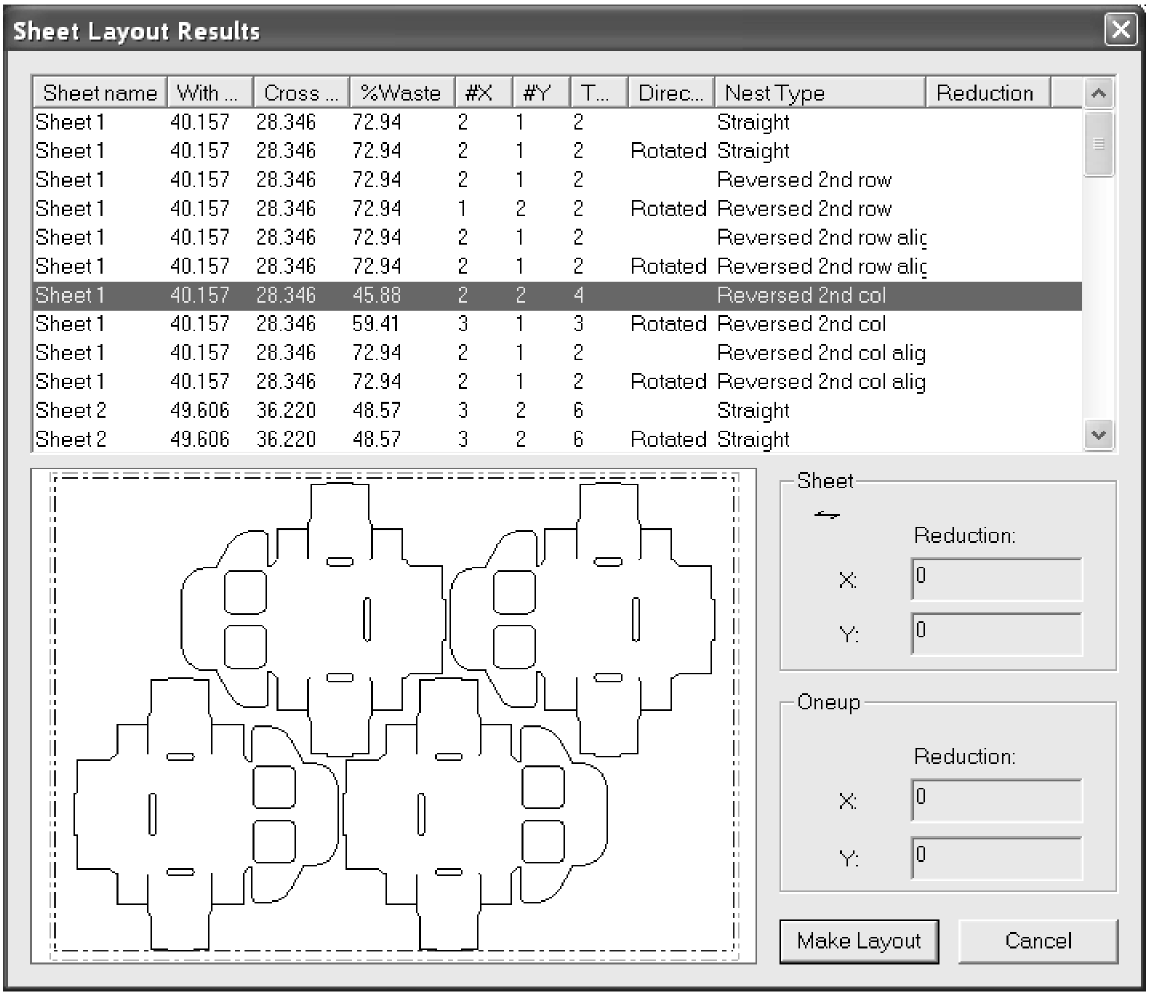


If the trim is too small and the sheet is too big, the following dialog box appears. Choose the appropriate choice and click **OK** to make the layout, or click **Cancel** to return to the single design.

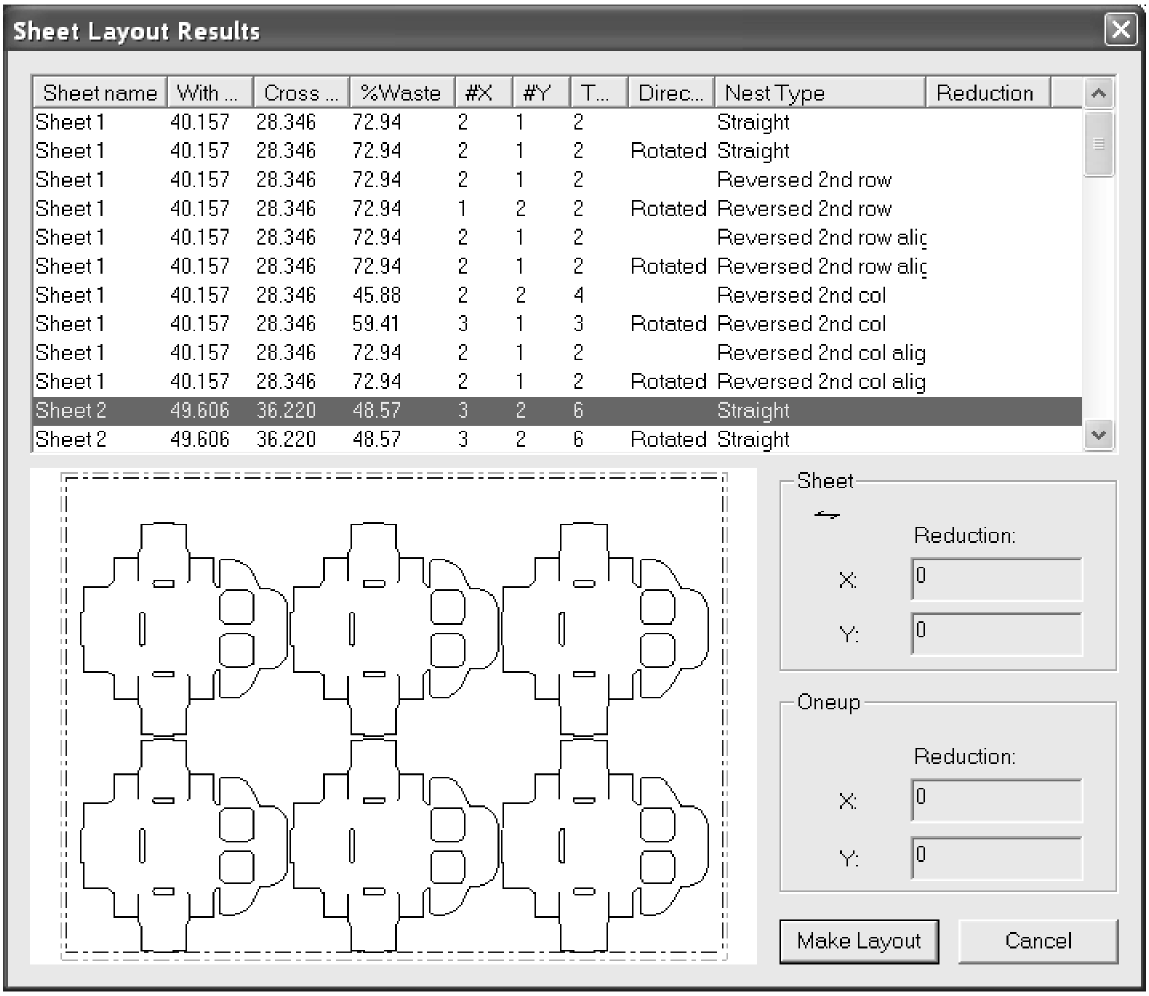


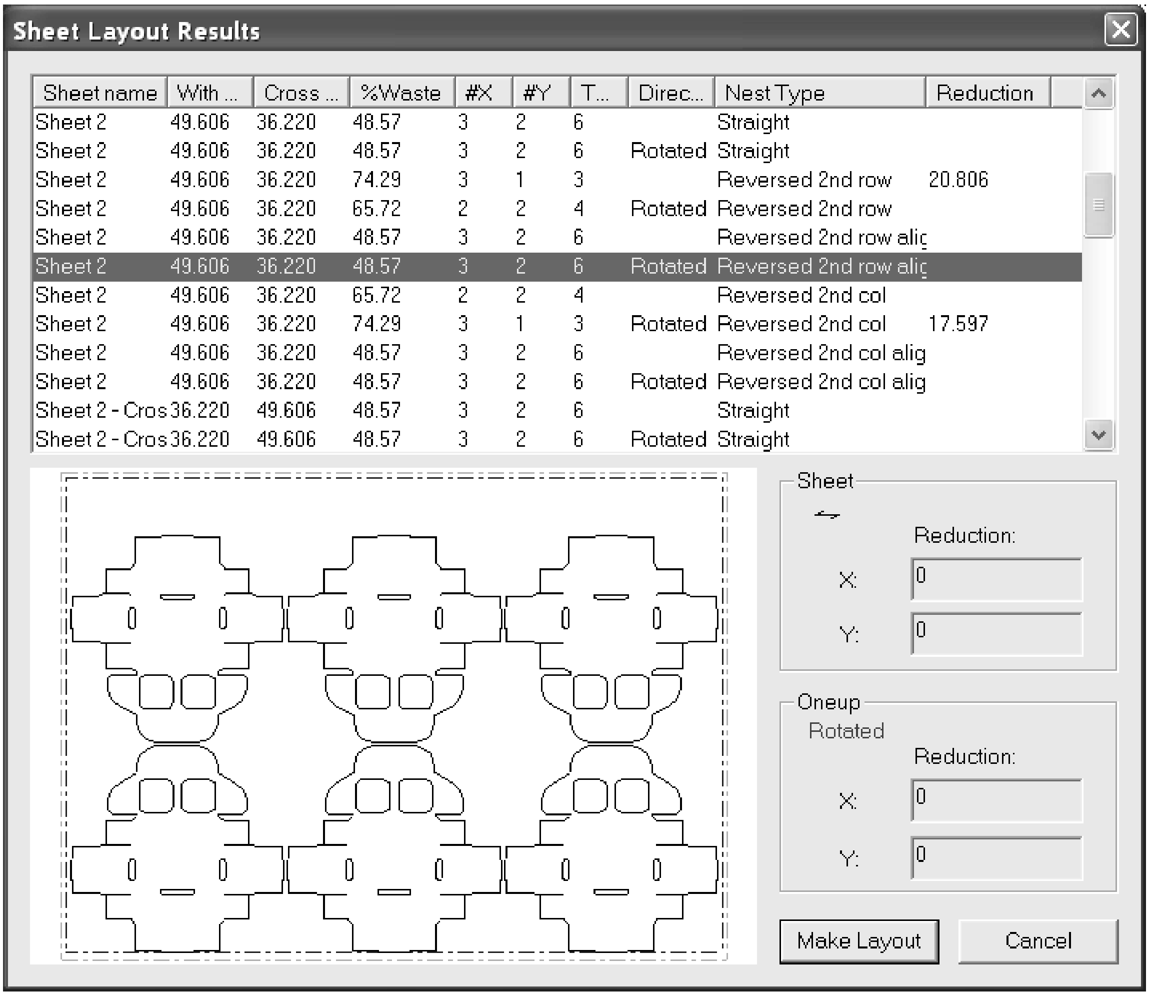
Further examples

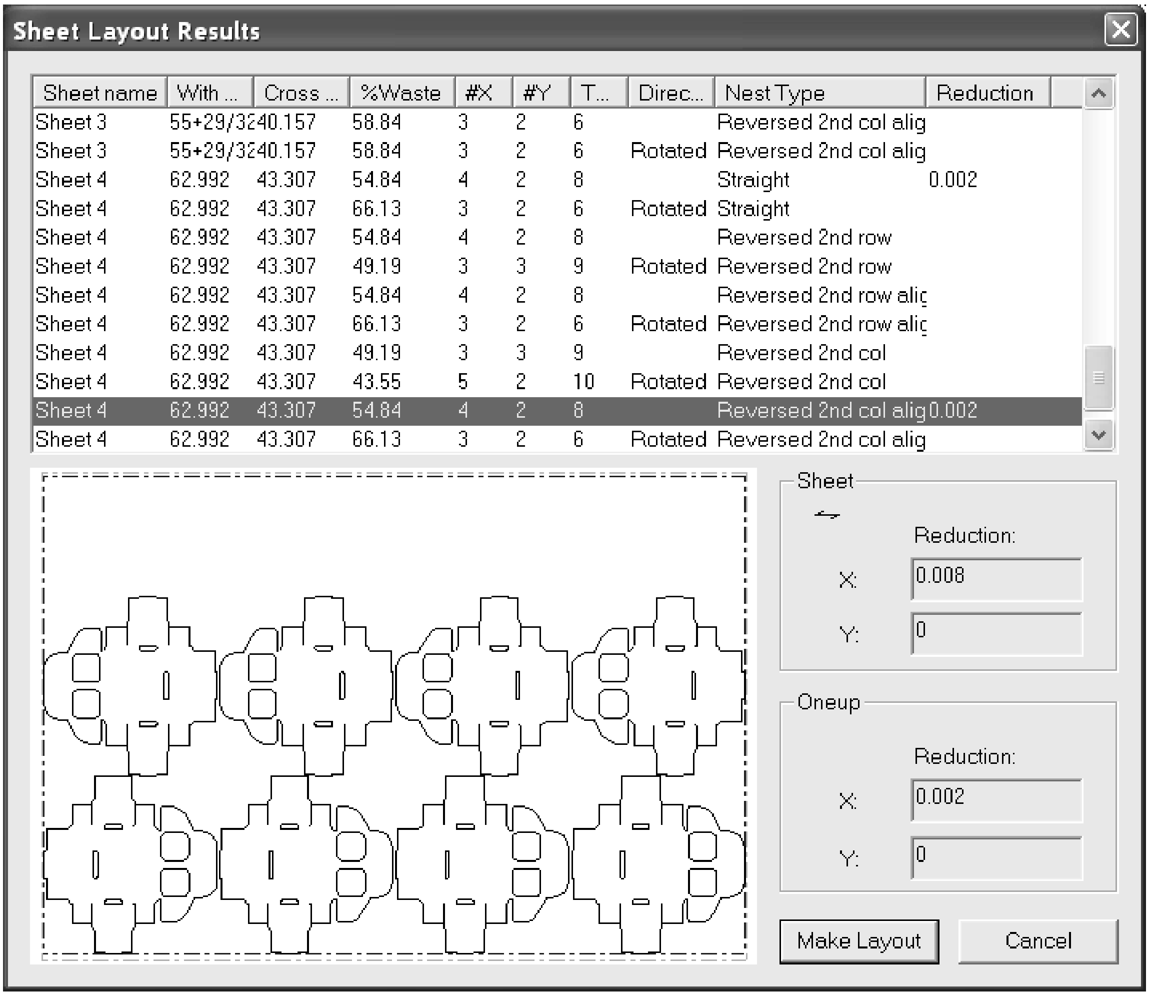
Shown below are further examples of the Sheet Layout Results dialog box with all sheets and nest types selected, as well as **Allow 90 degree rotation** enabled.



Sheet name	With ...	Cross ...	%Waste	#X	#Y	T...	Direc...	Nest Type	Reduction
Sheet 1	40.157	28.346	72.94	2	1	2		Straight	
Sheet 1	40.157	28.346	72.94	2	1	2	Rotated	Straight	
Sheet 1	40.157	28.346	72.94	2	1	2		Reversed 2nd row	
Sheet 1	40.157	28.346	72.94	1	2	2	Rotated	Reversed 2nd row	
Sheet 1	40.157	28.346	72.94	2	1	2		Reversed 2nd row alic	
Sheet 1	40.157	28.346	72.94	2	1	2	Rotated	Reversed 2nd row alic	
Sheet 1	40.157	28.346	45.88	2	2	4		Reversed 2nd col	
Sheet 1	40.157	28.346	59.41	3	1	3	Rotated	Reversed 2nd col	
Sheet 1	40.157	28.346	72.94	2	1	2		Reversed 2nd col alig	
Sheet 1	40.157	28.346	72.94	2	1	2	Rotated	Reversed 2nd col alig	
Sheet 2	49.606	36.220	48.57	3	2	6		Straight	
Sheet 2	49.606	36.220	48.57	3	2	6	Rotated	Straight	


Sheet Layout Results
Sheet name With ... Cross ... %Waste #X #Y T... Direc... Nest Type Reduction
Sheet 1 40.157 28.346 72.94 2 1 2 Straight
Sheet 1 40.157 28.346 72.94 2 1 2 Rotated Straight
Sheet 1 40.157 28.346 72.94 2 1 2 Reversed 2nd row
Sheet 1 40.157 28.346 72.94 1 2 2 Rotated Reversed 2nd row
Sheet 1 40.157 28.346 72.94 2 1 2 Reversed 2nd row alic
Sheet 1 40.157 28.346 72.94 2 1 2 Rotated Reversed 2nd row alic
Sheet 1 40.157 28.346 45.88 2 2 4 Reversed 2nd col
Sheet 1 40.157 28.346 59.41 3 1 3 Rotated Reversed 2nd col
Sheet 1 40.157 28.346 72.94 2 1 2 Reversed 2nd col alig
Sheet 1 40.157 28.346 72.94 2 1 2 Rotated Reversed 2nd col alig
Sheet 2 49.606 36.220 48.57 3 2 6 Straight
Sheet 2 49.606 36.220 48.57 3 2 6 Rotated Straight
Sheet
Reduction:
X: 0
Y: 0
Oneup
Reduction:
X: 0
Y: 0
Make Layout
Cancel


Sheet Layout Results
Sheet name | With ... | Cross ... | %Waste | #X | #Y | T... | Direc... | Nest Type | Reduction
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | | Straight |
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | Rotated | Straight |
Sheet 2 | 49.606 | 36.220 | 74.29 | 3 | 1 | 3 | | Reversed 2nd row | 20.806
Sheet 2 | 49.606 | 36.220 | 65.72 | 2 | 2 | 4 | Rotated | Reversed 2nd row |
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | | Reversed 2nd row alic |
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | Rotated | Reversed 2nd row alic |
Sheet 2 | 49.606 | 36.220 | 65.72 | 2 | 2 | 4 | | Reversed 2nd col |
Sheet 2 | 49.606 | 36.220 | 74.29 | 3 | 1 | 3 | Rotated | Reversed 2nd col | 17.597
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | | Reversed 2nd col alig |
Sheet 2 | 49.606 | 36.220 | 48.57 | 3 | 2 | 6 | Rotated | Reversed 2nd col alig |
Sheet 2 - Cros | 36.220 | 49.606 | 48.57 | 3 | 2 | 6 | | Straight |
Sheet 2 - Cros | 36.220 | 49.606 | 48.57 | 3 | 2 | 6 | Rotated | Straight |
Sheet
Reduction:
X: 0
Y: 0
Oneup
Rotated
Reduction:
X: 0
Y: 0
Make Layout
Cancel



Intelligent Layout and Costing/Estimating

Intelligent Layout performs automatic sheet layout calculation. Costing/Estimating provides detailed production costing information for your estimators.

There are two workflows when using Costing/Estimating. One is if you have Intelligent Layout as well, and the other is if you do not have Intelligent Layout. Costing/Estimating without Intelligent Layout is designed to be particularly useful in the following scenarios:

1. If you already know what the layout will be (e.g. if you have pre-made dies, or if there will be one design per sheet).
2. If you will have a long run of a layout with a straight nest using the maximum sheet size.

If you have Intelligent Layout without Estimating, you can automatically make layouts based on minimizing printing and diecutting costs. Note that the board cost, die cost, and press cost information need to be approximately correct (within a factor of two), even if you are not using the cost results, because these costs affect how layouts are selected.

Costing/Estimating enables the following:

1. Item cost centers such as folder-gluers

2. The Ink coverage, Prompted variables, Cost breakdown, and Cost/item tabs in the Quantities and Costs dialog box.
3. The Quantities Mode option to have different layouts for different quantities

If you have Estimating but not Intelligent Layout you have to make your layouts manually before you do your estimating. This is perfect for corrugated users who usually only have one or two designs on a sheet.

Overview: How to design a layout using Intelligent Layout

1. Create designs to use in the layout.
2. Create a new layout and add the new designs to it.
3.  Click **Quantities and costs**.
4. Check the board weight and cost to make sure they are correct.
5. Enter the ordered quantities and click **Rebuild**.
6. Examine the information on the Total Cost tab, decide which layout best meets your qualifications and click **OK**.
7. Create tooling.

Refer to the upcoming example for more detailed instructions.

Overview: How to use Costing/Estimating when creating a layout

1. Create designs to use in the manufacturing file.
2. Create a new manufacturing file and add the new designs to it.
3. Use the nesting and copying tools to build a layout.
4.  Click **Quantities and costs**.
5. Check the board weight and cost to make sure they are correct.
6. Set ink coverage, if any.
7. Set the values for prompted variables, if any.
8. Examine the information on the Total Cost tab, decide which layout best meets your qualifications and click **OK**.
9. Create tooling.

Refer to the example which follows for more detailed instructions.

Example - How to use Intelligent Layout with Costing/Estimating to produce a layout with multiple items

This example makes a layout with three different designs and uses Intelligent Layout and Costing/ Estimating.

Create three designs

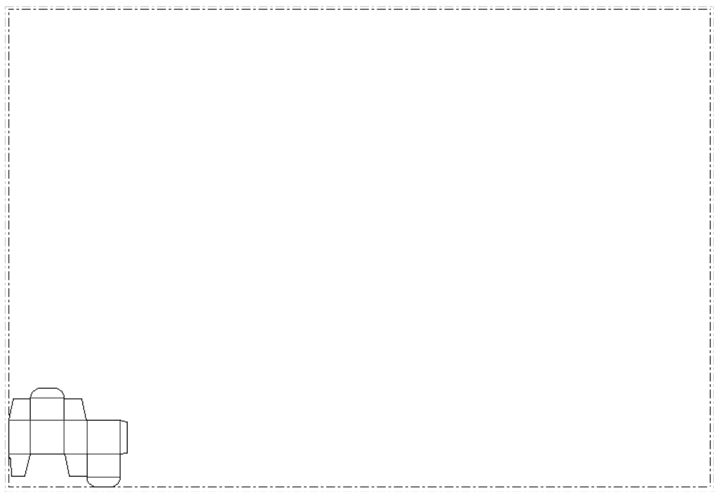
For the purposes of this example, create three designs, all style RT Top Tuck on 2nd (in the Folding Carton Standards Catalog).

DESIGNA	4 x 3 x 5 inches or 100 x 75 x 125mm
DESIGNB	4 x 2½ x 4 inches or 100 x 60 x 100mm
DESIGNC	3 x 2 x 3 inches or 75 x 50 x 75mm

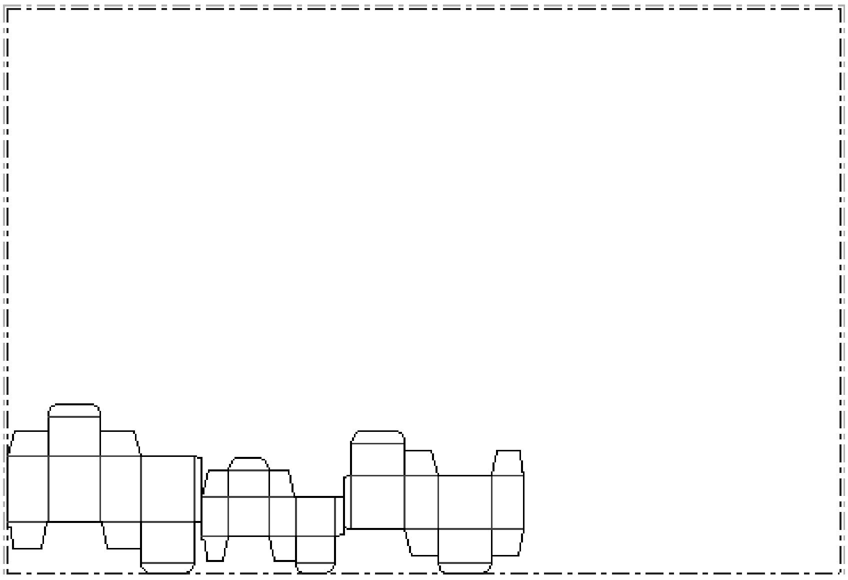
If you use different sizes, you will get a different yield and a different number up.

Create a new layout and add the three designs

1. Click **File**, and then click **New Manufacturing**.
2. Click **OK** to accept default parameter sets and machines, or choose others if desired. A blank manufacturing file will be created.
3.  Click **Add oneup**.
4. Navigate to the directory or resource containing the designs you made, highlight `DESIGNA`, and click **Open**. DESGINA appears in the bottom left of the sheet.

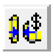


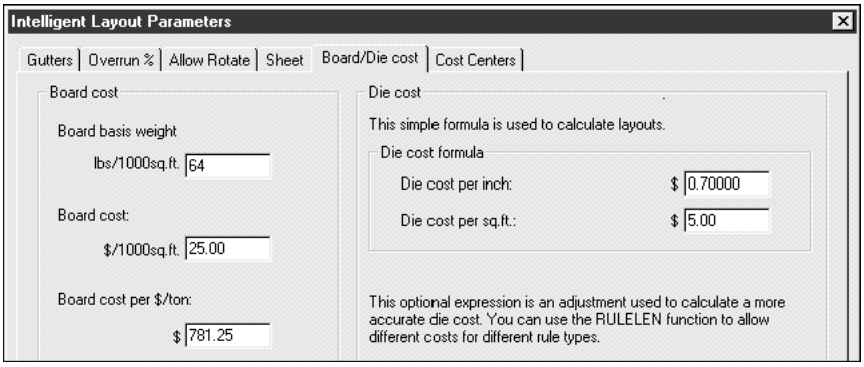
5. Repeat steps three and four for `DESIGNB` and `DESIGNC`.



6. Click **Change gutter distance**.
7. Select **¼ inch** or **5mm** from the drop down list.
8. Click **OK**.

Check the board weight and cost

1. Make sure the Intelligent Layout toolbar is turned on using the Toolbars master control.
2.  Click **Intelligent layout parameters**, and then click the Board/Die cost tab. If the designs were not made in ArtiosCAD2.10 or later, they likely have incorrect board basis weight and cost. The board basis weight and cost should be reasonable, for example:

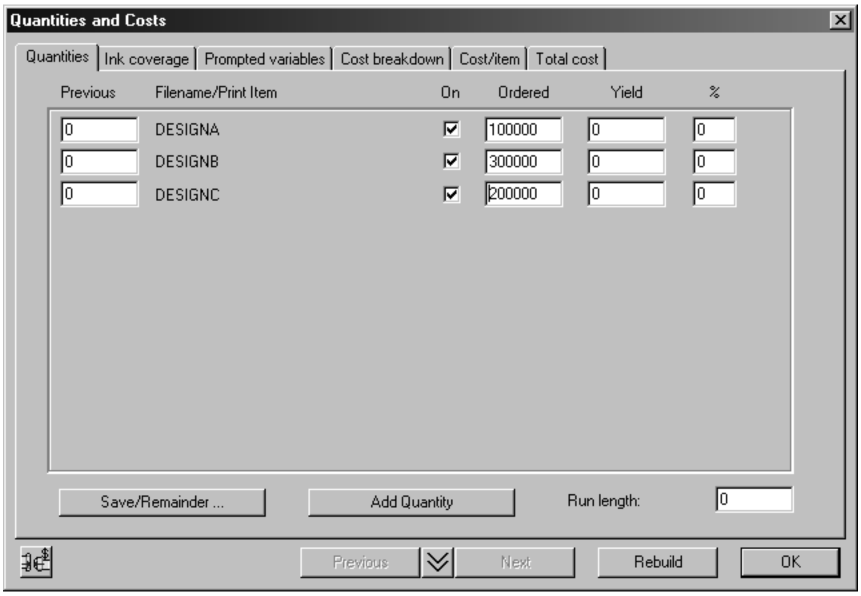


Correct the basis weight and cost if necessary. The board cost needs to be approximately correct within a factor of two for Intelligent Layout to give meaningful results.
3. Click **OK**.

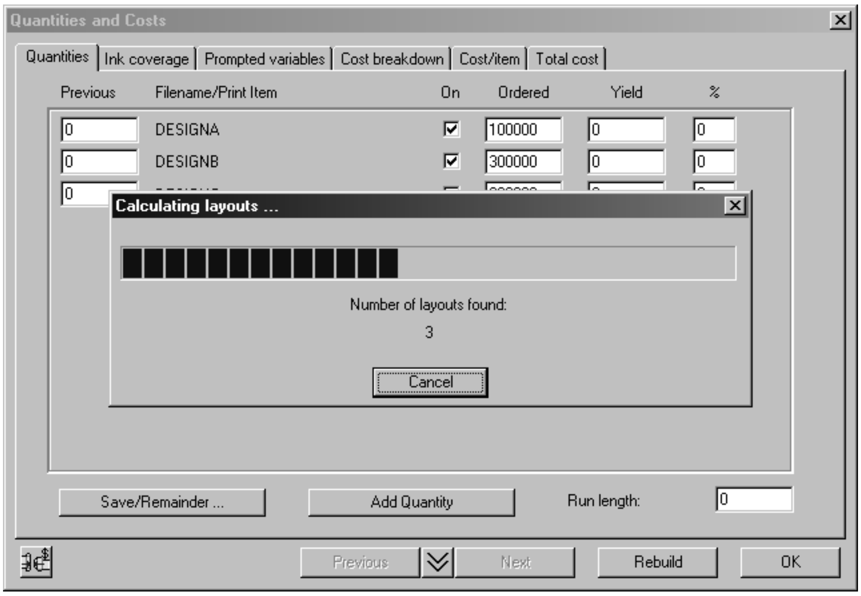
Enter the ordered quantities and Rebuild

1.  Click **Quantities and costs**.

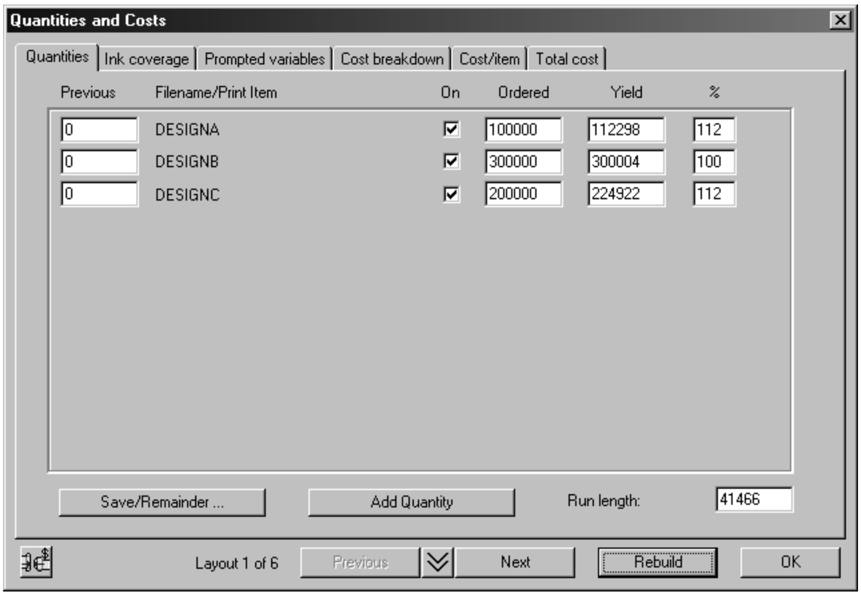
2. Type ordered quantities such as `100000` for DESIGNA, `300000` for DESIGNB, and `200000` for DESIGNC as shown in the following dialog box:



3. Click **Rebuild** to calculate a layout. You will see a progress indicator for a few seconds while it calculates layouts:



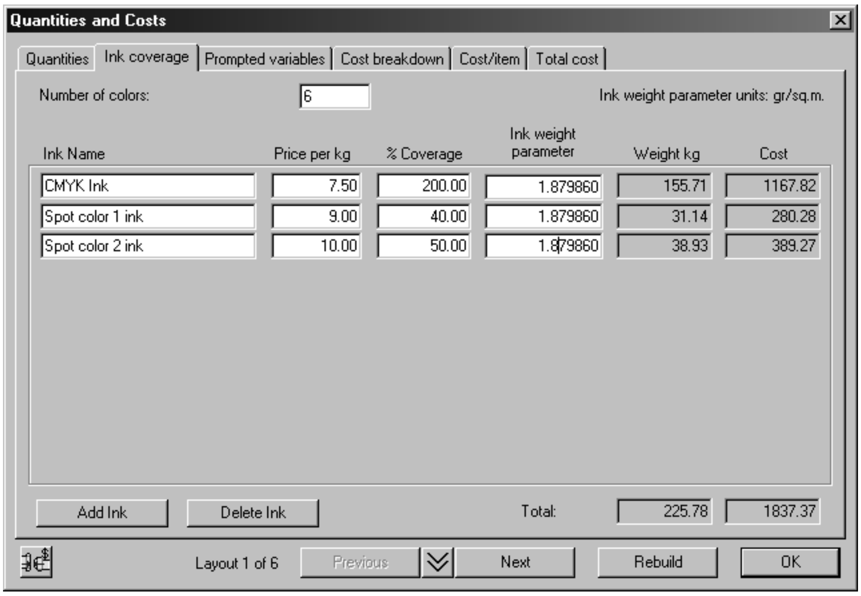
4. When it has finished calculating layouts, the progress control disappears, the run length and yields are calculated, and the Quantities tab is updated:



The text at the bottom of the dialog Layout 1 of 6 indicates that it calculated six layouts; cycle through these layouts using the **Next** and **Previous** buttons. Note that layouts are ordered by ascending sheet cost.

Set the ink coverage

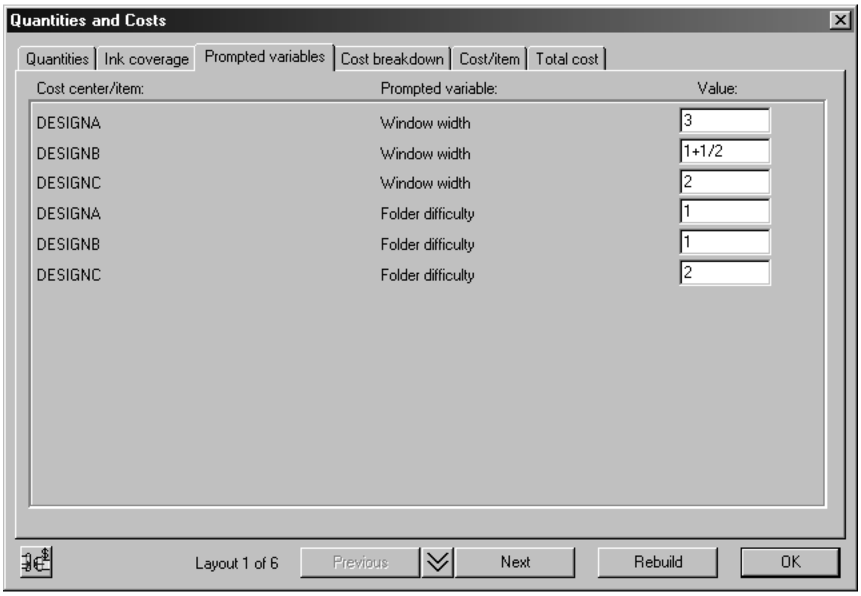
1. Click the Ink coverage tab and enter the number of colors and the inks used. The number of colors is used to determine the make ready for the printing press and the ink information is used to calculate ink cost. The number of inks does not have to be the same as the number of colors. For each ink you need to estimate the coverage and enter the cost per pound/kilo. As an example you might enter an ink called `CMYK` at `200%` coverage, followed by any spot colors used.



2. After changing the number of colors click **Rebuild**, as specifying ink affects the selection of the layout.

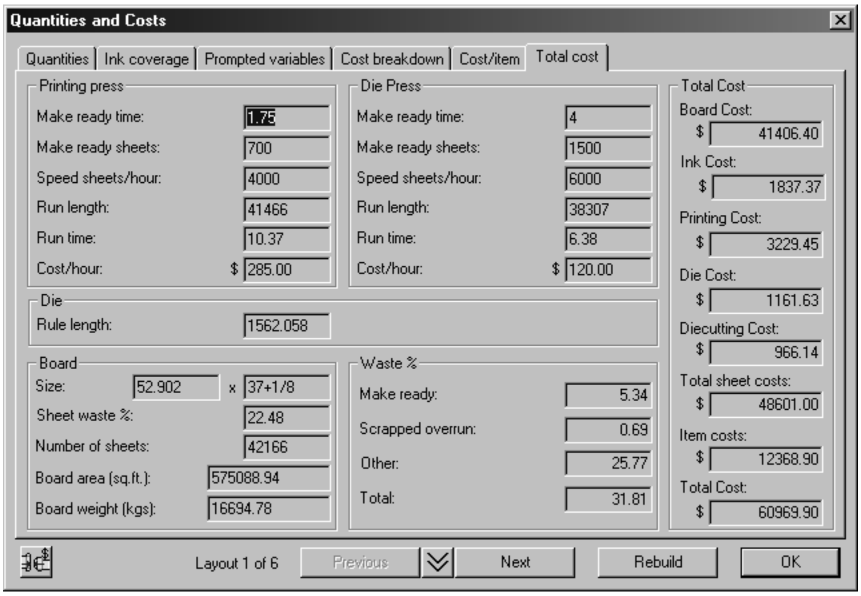
Set the prompted variables

1. The default cost centers for Intelligent Layout include a window width parameter for the windowing machine, and a Folder difficulty for the folder-gluer. Set these parameters for each design as shown in the following dialog box.



Examine the total cost

1. Click the Total cost tab in the Quantities and Costs dialog box to see information similar to that shown in the following dialog box:



Here is an explanation of the values in this dialog:

Table: Quantities and Costs

Term	Meaning
Make ready time	The time to set up the machine in hours.
Make ready sheets	The number of sheets used while setting up the machine.
Speed sheets/hour	Average speed of the machine while running.
Run length	Number of sheets used while running, separate from make ready. The run length of the die press is shorter than the run length of the printing press due to run waste on the printing press and make ready on the die press.
Run time	Time the machine is running, hours.
Cost/hour	Cost of using the machine per hour.
Rule length	Approximate rule length in the die. Does not allow for double line removal or stripping rules. Used in the calculation of the die cost.
Rule length warning	A warning is shown if the rule length exceeds the maximum rule length for the die press.
Sheet size	The size of the sheet in X and Y.
Number of sheets	Total number of sheets used, including make ready.
Board area	Total area of board used, including make ready.
Board weight	Total weight of board used, including make ready.
Layout waste	(area of waste in a single sheet) / (area of sheet) * 100%.
Make ready waste	(Amount of board used in make ready and run time waste for all cost centers, including folder-gluers and so on) / (Total board used) * 100%.
Scrapped overrun	(Area of board scrapped due to excess overruns) / (Total board used) * 100%.
Total waste	(Area of board in the cartons delivered) / (Total board used) * 100%. Note: This is not a sum of all waste.
Ink Cost	Cost of inks used as specified in the ink coverage page.
Printing Cost	Cost of using the printing press, including make ready.
Die cost	There is a formula for the die cost based on the sheet area and rule lengths. Note that since there is no separate formula for stripping boards, varnish blankets and other tooling, the die cost formula should be set up to include these as well.
Diecutting cost	Cost of using the die press including make ready.

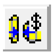
Term	Meaning
Total sheet costs	Subtotal of board cost + Ink cost + Printing cost + Die cost + Diecutting cost + any other sheet cost centers.
Item costs	Cost of the item cost centers (the ones downstream from the die press) such as folder-gluer, windowing and shipping.
Total cost	Total sheet costs + Item costs.

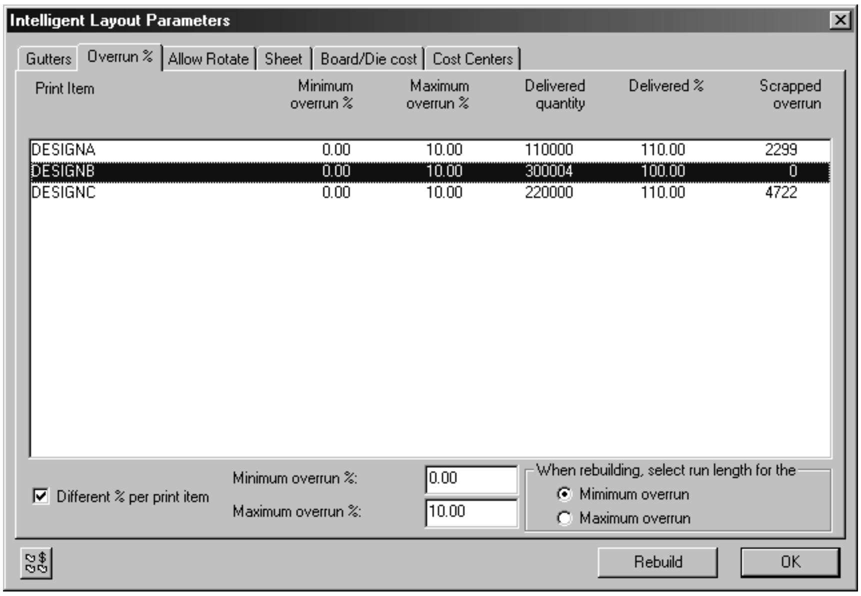
The formula for calculating the cost of a cost center is:

(make ready cost) + ((run time) * (cost/hour)) + ((# sheets) * (cost/sheet))

Setting the overrun percentage

This section illustrates how the overrun percentage can be controlled by the Overruns tab.

1.  Click the icon in the lower left to change from the Quantities and Costs dialog box to the Intelligent Layout Parameters dialog box, and then click the Overruns tab. You will see the following dialog box:



The table below explains the Overruns dialog box.

Table: Overruns

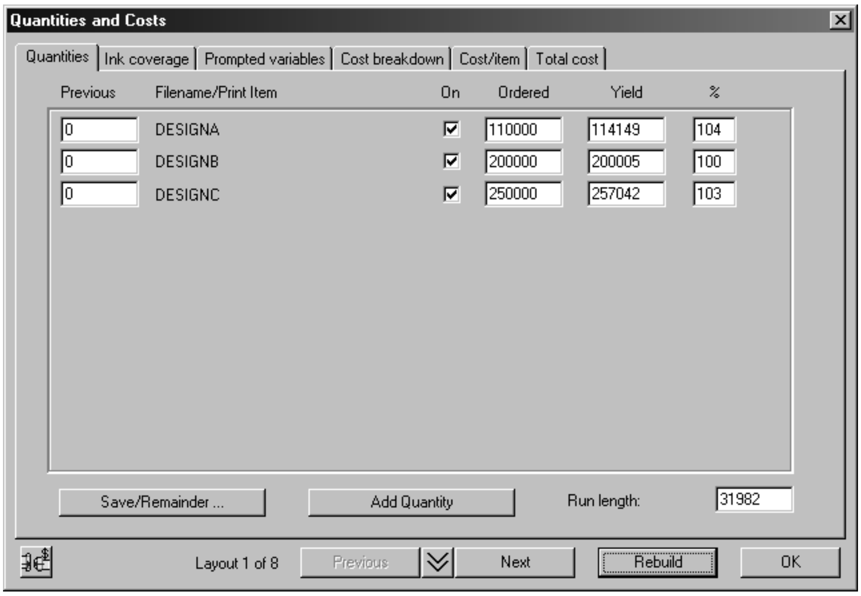
Term	Meaning
Minimum overrun %	The minimum overrun acceptable. 0 means make at least 100% of the ordered quantity.
Maximum overrun %	The maximum overrun acceptable. 10 means that the customer will accept up to 110% of the quantity ordered. Anything over 110% is assumed to be scrap.

Term	Meaning
Delivered Quantity	Quantity delivered to the customer (does not include the scrapped overrun). Note that the Yield column on the Ordered Quantities page does include the scrapped overrun.
Delivered %	Quantity delivered to the customer as a percentage of the ordered quantity. Should be in the range 100+minimum% and 100+maximum%.
Scrapped Overrun	The number of excess cartons above the maximum overrun. It is assumed that these will be scrapped (e.g. thrown out or recycled) and not processed by downstream machines such as the folder-gluer.
Different % per print item	When on, you can select multiple rows by `SHIFT`- or `CTRL`-clicking. Use this to set the overrun % to be different for different items. When off, all the items are selected, and you change the overrun % for all items at the same time.
Minimum overrun option button	Calculate the run length as the smallest that gives yields >= minimum overrun for all items on the sheet.
Maximum overrun option button	Calculate the run length as the largest that gives yields <= maximum overrun and yields >= minimum overrun for all items on the sheet.

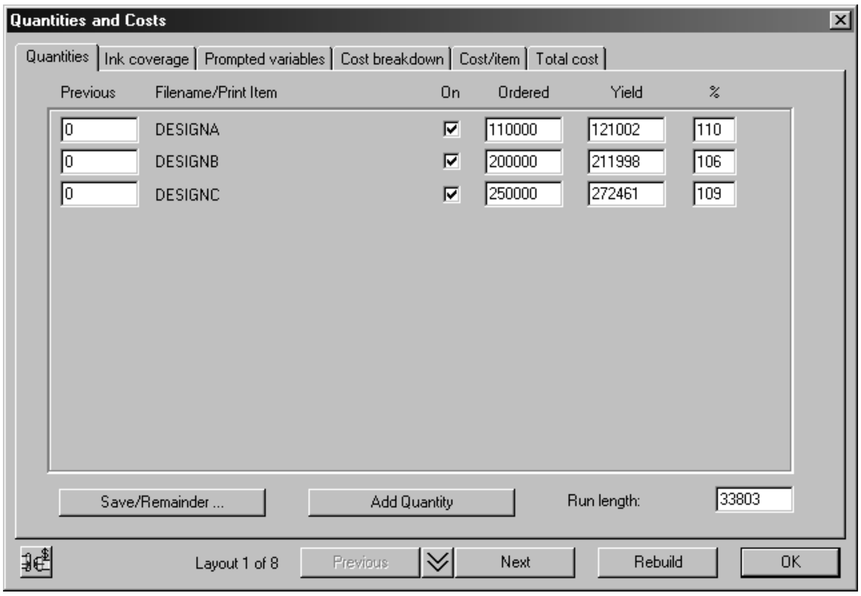
Effect of minimum and maximum overrun

The following example illustrates the difference between the Minimum and Maximum option button selections.

1.  Set the option button to **Minimum overrun**, switch back to the Quantities and Costs page by clicking the icon, enter new ordered quantities as follows, and rebuild:

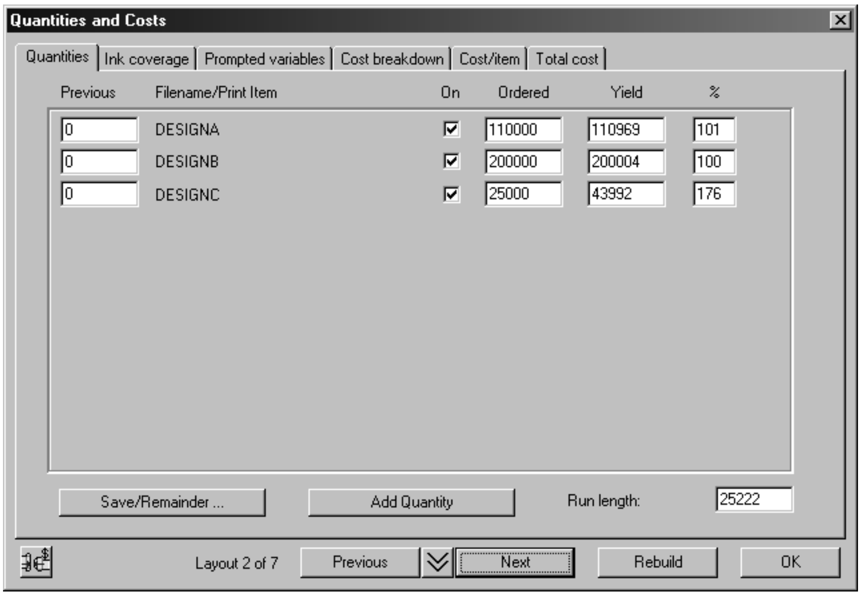


2. You will get a new layout and different yields. Note that you will often get a layout that does not give the exact yield quantity required, because you can only put a whole number of each design on the sheet. In this case DESIGNB is 100%, whereas DESIGNA and DESIGNC are slightly over, at 104% and 103%.
3. Switch to the Overruns tab, change the option button to **Maximum overrun**, and switch back to the Ordered Quantities page. Do not click Rebuild! Now the run length has increased to fit the maximum overrun which is 10%:

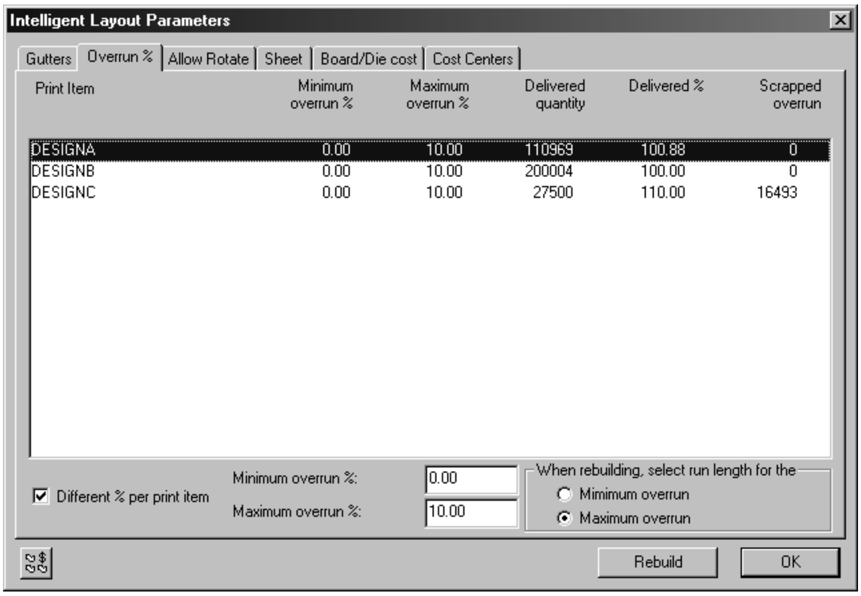


Scrapped overrun

Sometimes the overrun is larger than the maximum and you get scrapped overrun, particularly when there are only 1 or 2 of that design on the sheet. For example, change the ordered quantity of DESIGNC from `250000` to `25000`, and rebuild. The new layout has only 2 of DESIGNC, giving a yield of 176%:



Since the customer will not accept over 110%, the remainder is shown as scrapped overrun on the Overrun % page:



Note that when you have scrapped overrun, it makes no difference which overrun method button is selected.

How to use print items with Intelligent Layout and Costing/Estimating

ArtiosCAD supports the concept of print items. Several print items may have the same structure, but different print on each structure. All the print items are stored in the same ArtiosCAD design file.

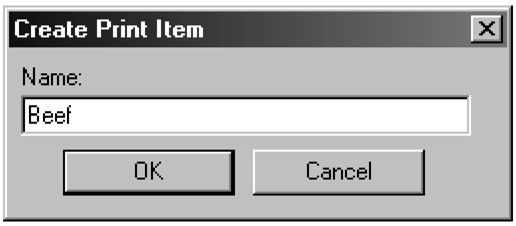
Example: Using print items with Intelligent Layout and Costing/ Estimating

The following is an example of how print items work in Intelligent Layout.

Creating Print Items

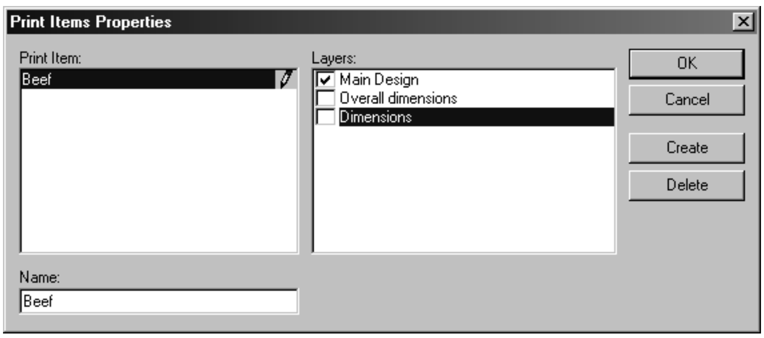
Start from the layout you made in the previous example and add some print items to the design DESIGNC.

1.  Click **Select oneups** and select one of the small designs.
2.  Click **Open embedded design**. A new window appears, titled `layoutname:DESIGNC`. This is the copy of design DESIGNC that is embedded in the manufacturing document.
3.  Click **Print item**.
4. Enter the name of the first print item, such as Beef.
5. Click **OK**.



The button  changes to  showing that the current print item is Beef.

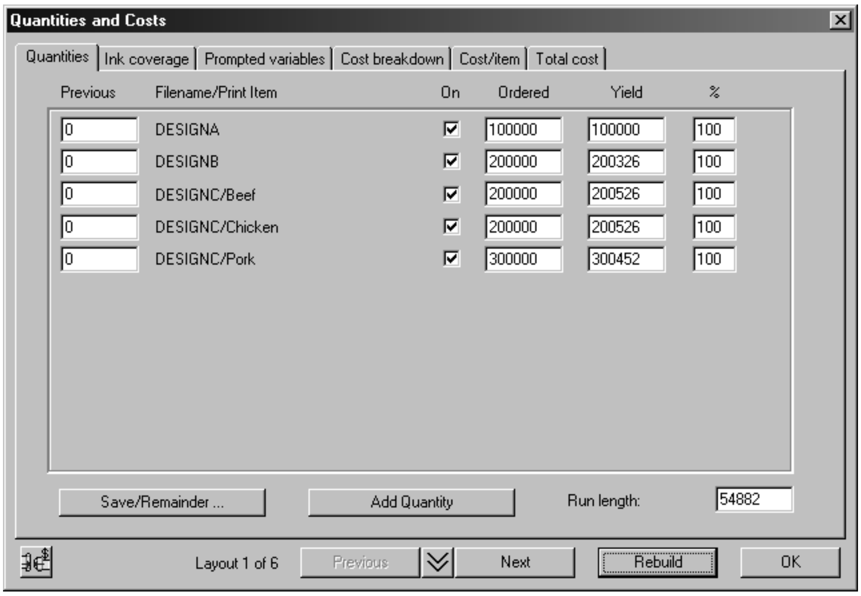
6. Click  again.



7. Click **Create**.
8. Type the name of another print item, such as `Chicken` and click **OK**.
9. Repeat to add another print item, such as `Pork` and click **OK**.

Return to Manufacturing

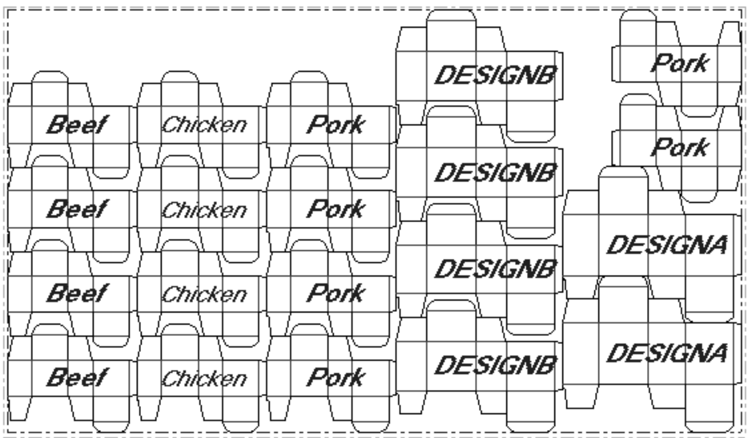
1. Click **File**, and then click **Return to Manufacturing** to close the embedded design.
2.  Click **Quantities and costs**.
3. Enter new ordered quantities, e.g. 200000 beef, 200000 chicken, and 300000 pork.
4. Click **Rebuild**.



5. Click **OK** to close the dialog to see the layout better.

Check the Print Items in the Layout

1. Click **View** and then click **Item labels**.
2. Select **Use print item names**.
3. Click **OK**.

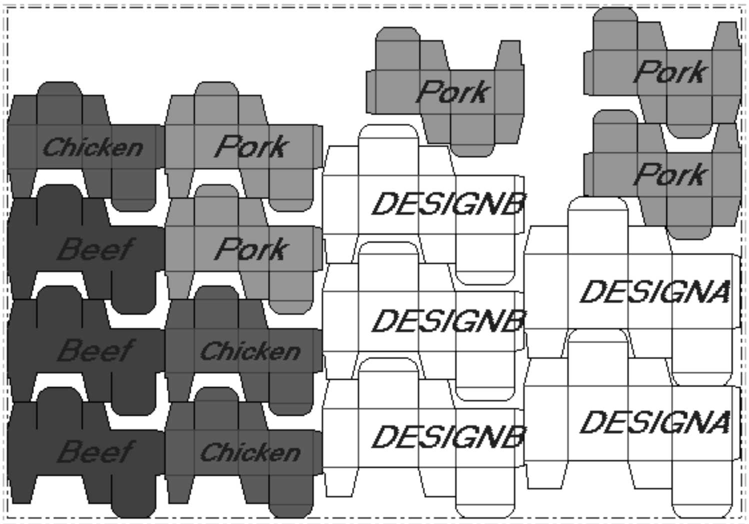


If you have ArtMaker, you can add graphics to the designs and view them by turning on the **Graphics** checkbox in the Oneup Layers Class dialog box on the **View** menu.

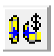
Ghosting constraints

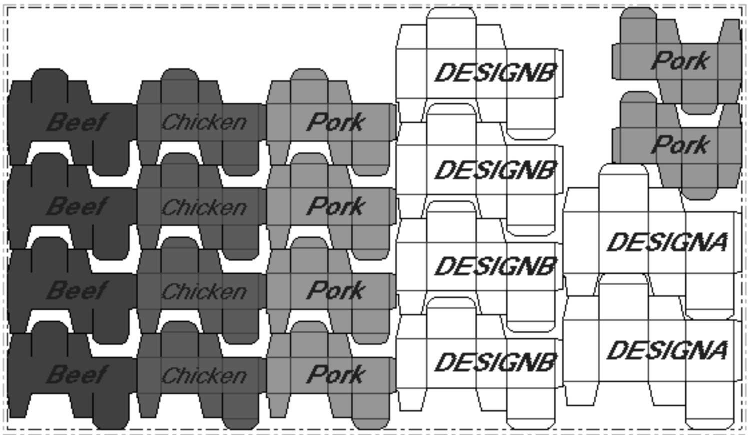
This is an illustration of how ghosting constraints can affect a layout (for the pictures shown here, the maximum sheet size was 60 x 42 inches).

With no ghosting constraints, the layout looks like this:



There is a light red item behind a dark green item and this may not be acceptable due to printing considerations (ghosting constraints). The green ink may bleed into the red ink while the cartons move through the press. To change the ghosting constraints:

1.  Click **Intelligent Layout Parameters**, then click **Gutters**.
2. Click **Beef** on the left and then clear the **Chicken** checkbox on the right. This means that the Chicken item cannot be placed behind the Beef item.
3. Click **Chicken** on the left and then clear the **Beef** checkbox on the right. This means that the Beef item cannot be placed behind the Chicken item.
4. Click **Rebuild**. The new layout is built with the constraint that Beef and Chicken items cannot be placed in the same column:

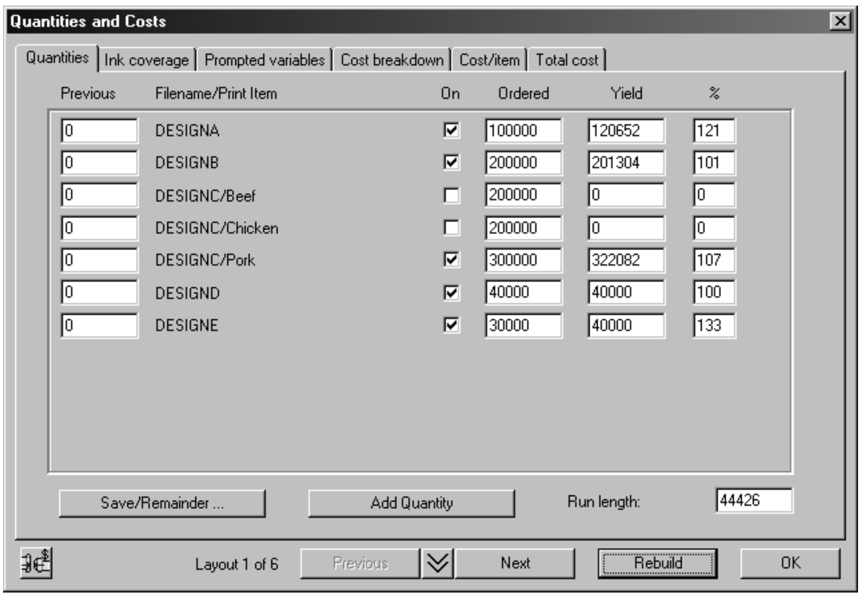


Multiple Sheets

To illustrate the need for more than one sheet, suppose you had more designs in an order. From the earlier layout, add two more designs DESIGND and DESIGNE, so the complete list is:

DESIGNA	**4 x 3 x 5 inches or 100 x 75 x 125mm**
DESIGNB	4 x 2½ x 4 inches or 100 x 60 x 100mm
DESIGNC	3 x 2 x 3 inches or 75 x 50 x 75mm
DESIGND	3 x 2 x 4 inches or 75 x 50 x 100mm
DESIGNE	5 x 4 x 6 inches or 125 x 100 x 150mm

Enter new ordered quantities for DESIGND of 40000 and DESIGNE of 30000, with a maximum sheet size of 60 x 40 inches or 1500 x 1000 mm, and then click **Rebuild**. Get results as follows:

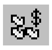


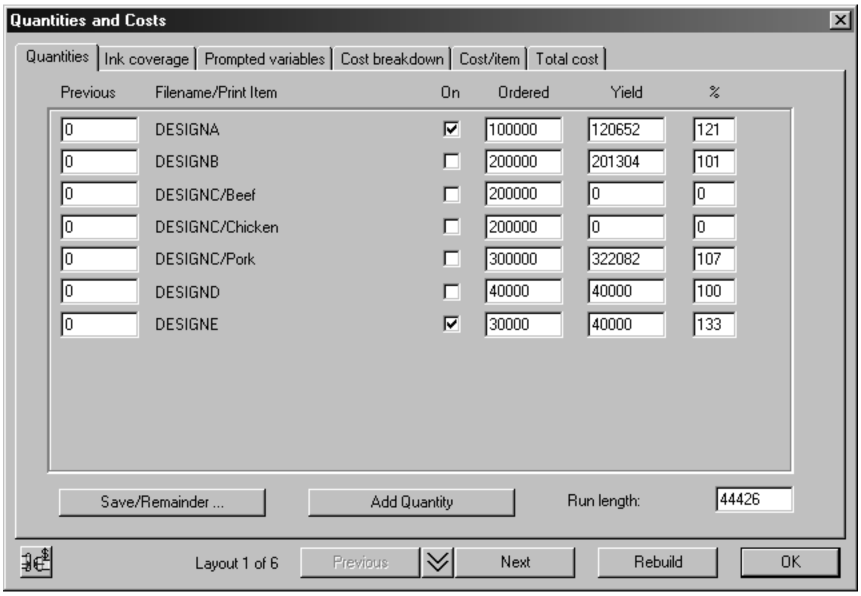
When there are many designs, the proportions might not fit particularly well on the sheet, and there may be overruns of one or more items. In this case there is 133% of DESIGNE. Because the numbers do not work out well, we can try fulfilling the order with two sheets.

Intelligent Layout does not handle multiple sheets automatically but it provides some tools for assistance in this case.

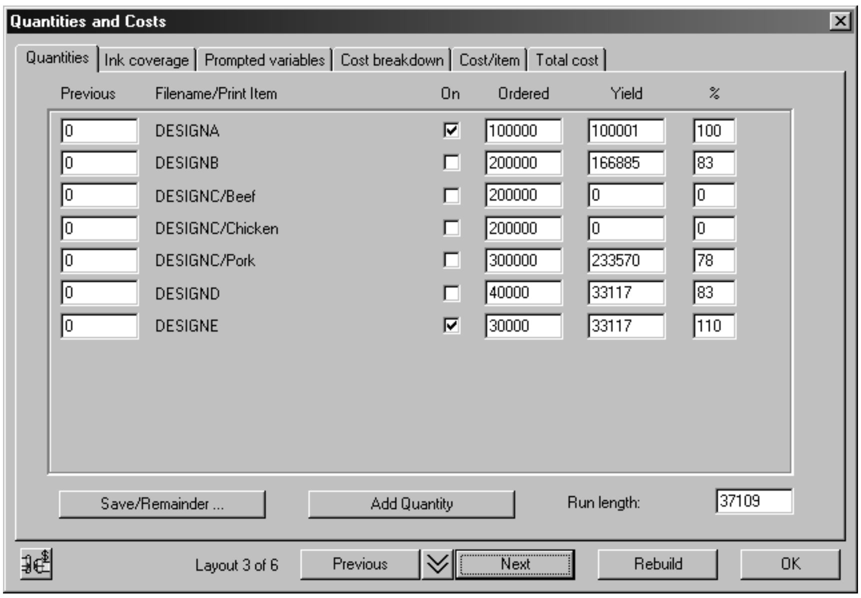
Using the On Checkboxes and Save/Remainder

You could try putting the larger designs on a separate sheet. The two larger designs are DESIGNA and DESIGNE. To make a sheet with just those, do the following:

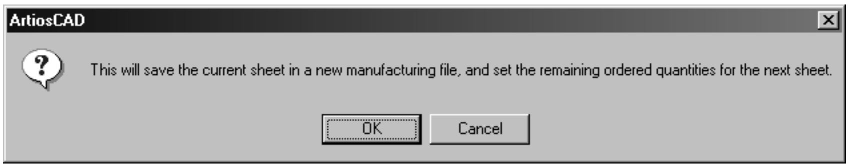
1.  Click **Quantities and costs.**
2. Turn off the checkboxes for designs DESIGNB, DESIGNC, DESIGND, and click **Rebuild**:



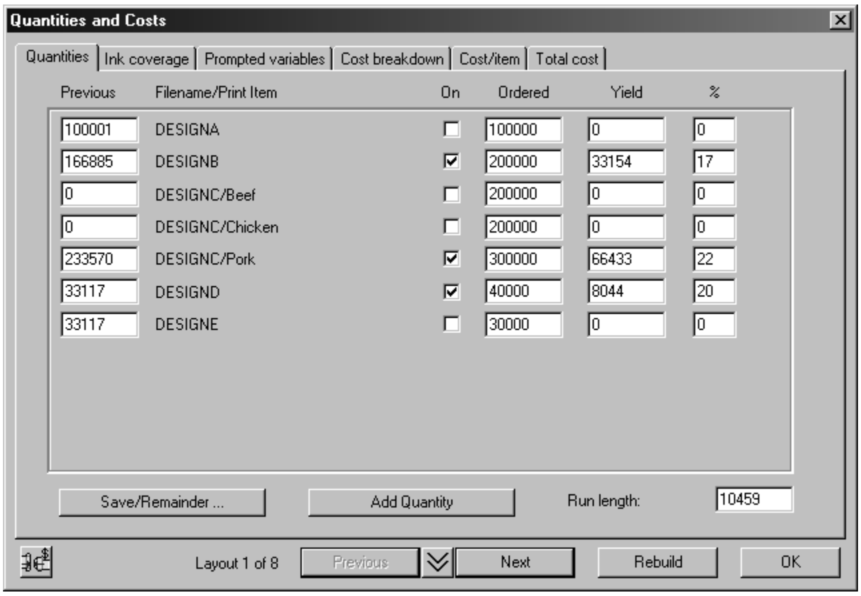
3. Use **Previous** and **Next** to find a layout with the least overrun.



4. Click **Save/Remainder**. A confirmation dialog box will appear.



5. Click **OK** to save this layout with a new name.
6. The yield quantities from the layout you just saved are now in the Previous column of the new layout, the checkboxes for the designs in the layout just saved are turned off, and the checkboxes for the remaining designs are turned on. Click **Rebuild** to make a layout for the remaining items:



How to adjust layouts made using Intelligent Layout

To adjust layouts made using Intelligent Layout, use the tools described in this section. Any manual changes you make to a layout will be lost if you do a Rebuild.

Specify Nest tool

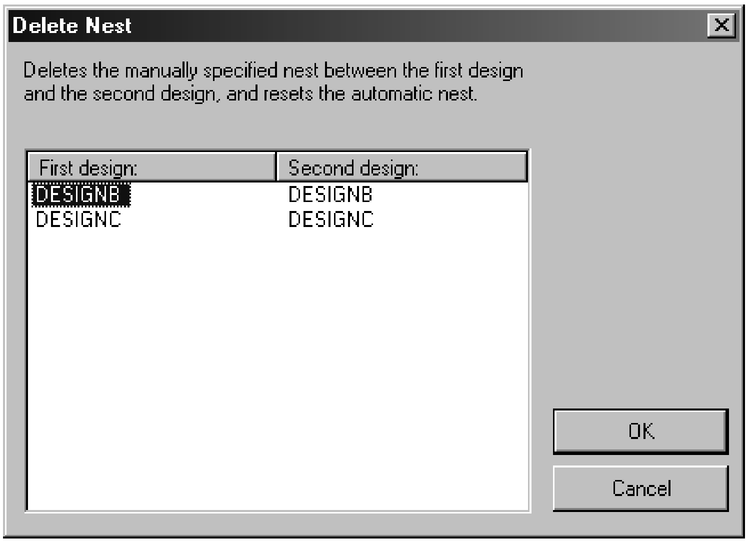
 The **Specify Nest** tool overrides the automatically calculated nest between designs. To use this tool:

1. Identify a pair of designs one above the other, for which you want to change the nest.
2.  Click the **Specify Nest** tool.
3. Click a pickup point in the higher of the two designs.
4. Drag this design horizontally, and place it using a put-down point in the required position relative to the design underneath it.

The column containing these two designs is rebuilt using the new nest you have defined. The new nest will be used in the next rebuild, and if the two designs are the same, in the next use of the manual nest tools.

Delete Nest tool

This tool allows nests created by the **Specify Nest** tool can be deleted. In the Delete Nest dialog box, hold down `CTRL` to select more than one nest at a time. Click `OK` to delete the selected nests.



Move Design or Column tool

The **Move Design or Column** tool makes quick, simple adjustments to a layout created by Intelligent Layout.

To move a design up or down in a column:

1. Click the **Move Design or Column** tool.
2. Select the design to move.
3. Click near the new position in the same column

To move a column:

1. Click the **Move Design or Column** tool.
2. Click a design in the column to move.
3. Click near the position (in a different column) where you want to move the column.

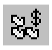
Manual Adjustments

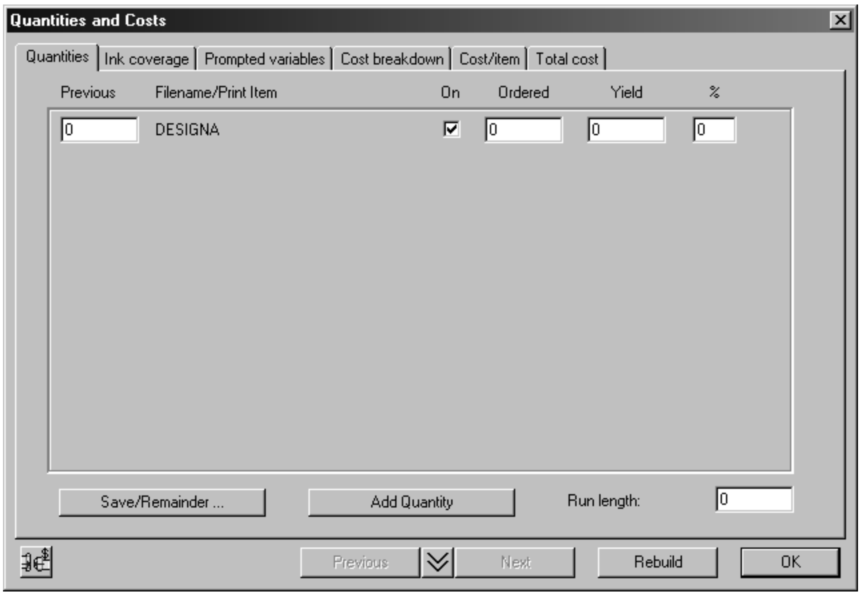
If you make manual adjustments to a layout created by Intelligent Layout, or create the whole layout manually, the value in the Quantities and Costs dialog will be automatically updated.

Multiple Quantities in Costing/Estimating

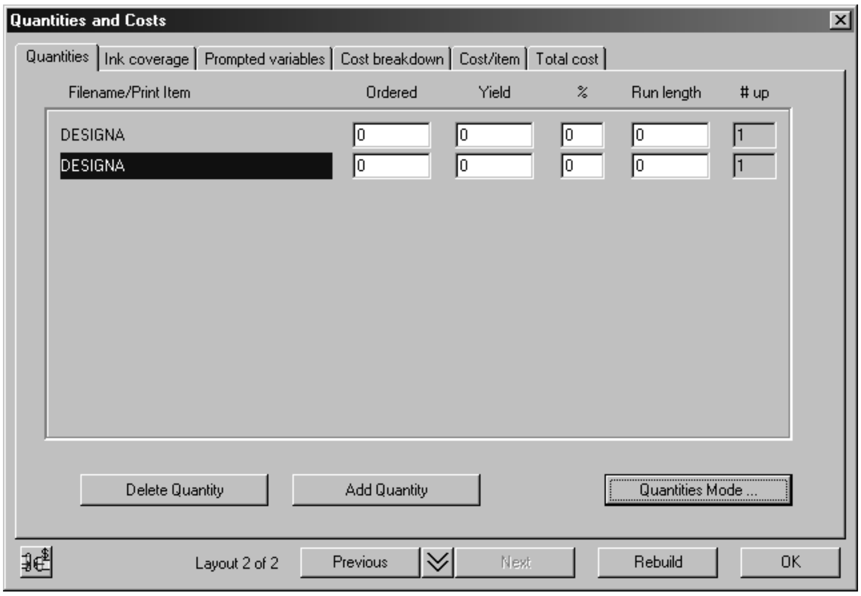
The other mode of using Costing/Estimating is when you have only one design, but you want to quote several estimates depending on the quantity ordered by the customer. The following example shows how this works.

Adding to the list of quantities

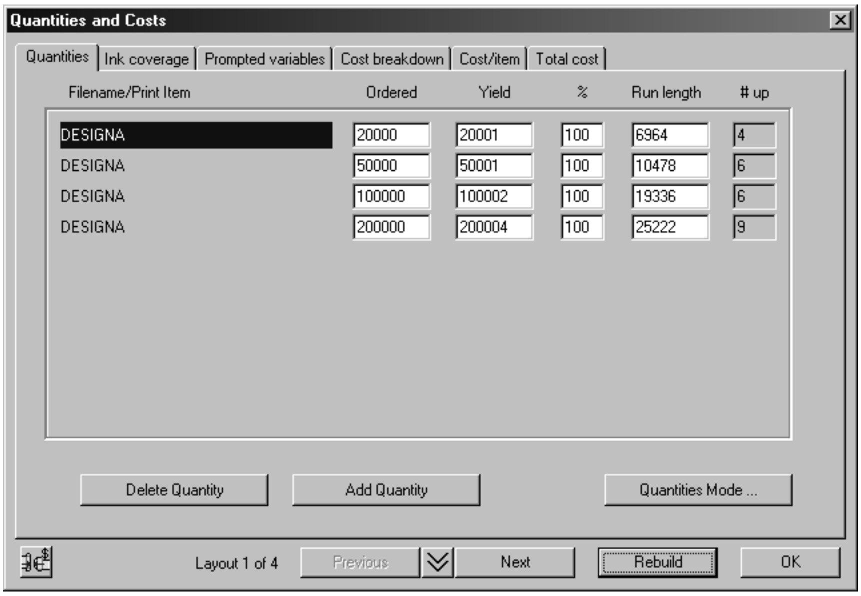
1. Open one design in ArtiosCAD.
2.  Click **Convert to manufacturing** and choose a parameter set and presses to use.
3.  Click **Quantities and costs**. The Quantities and Costs dialog box appears showing the single design.



4. Click **Add Quantity**. The format of the Quantities tab changes, showing multiple quantities instead of multiple items:



5. Click **Add Quantity** twice more.
6. Enter ordered quantities `20000`, `50000`, `100000`, `200000`.
7. Click **Rebuild**.



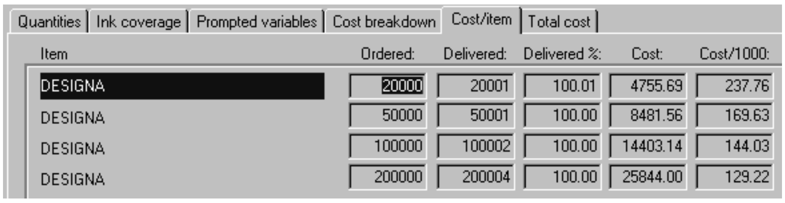
Different Layout for Each Quantity

Note that there is a different number up, hence a different layout, for each quantity. An important principle in estimating is that the best layout depends on the quantity. For a long run, the biggest cost is the press time, which is optimized with a large sheet. For a short run, the biggest cost is the make ready, which is optimized with a smaller sheet. For intermediate run lengths, there is a trade-off between make ready and the press time. Intelligent Layout does this automatically, but it is important to understand what is happening.

Clicking on the first column changes between layouts in the same way as the **Next** and **Previous** buttons.

Cost/1000 for Different Quantities

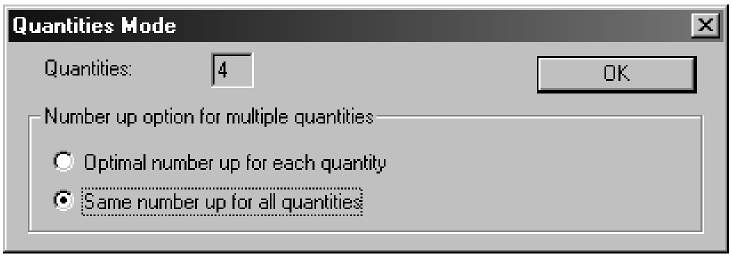
The Cost/item tab of the Quantities and Costs dialog box shows costs per thousand items for each ordered quantity.



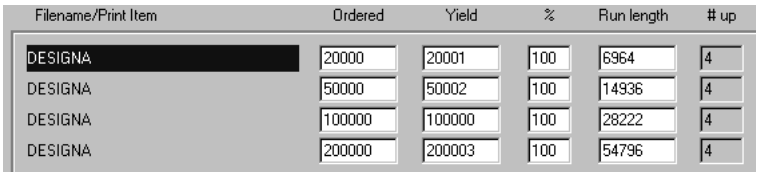
Item	Ordered:	Delivered:	Delivered %:	Cost:	Cost/1000:
DESIGNA	20000	20001	100.01	4755.69	237.76
DESIGNA	50000	50001	100.00	8481.56	169.63
DESIGNA	100000	100002	100.00	14403.14	144.03
DESIGNA	200000	200004	100.00	25844.00	129.22

Whether there is a different layout or the same layout for each Quantity is controlled by the Quantities Mode. If you want to have the same layout for each quantity:

1. Go to the Quantities tab in the Quantities and Costs dialog box.
2. Select the desired layout.
3. Click **Quantities Mode**.
4. Select **Same number up for all quantities**.



5. Click **OK**.



Filename/Print Item	Ordered	Yield	%	Run length	# up
DESIGNA	20000	20001	100	6964	4
DESIGNA	50000	50002	100	14936	4
DESIGNA	100000	100000	100	28222	4
DESIGNA	200000	200003	100	54796	4

If you only purchased Estimating and not the Intelligent Layout option, the **Quantities Mode** is **Same number up for all quantities** always.

How to use manual layout tools to design a layout

This section describes how to create a new set of manufacturing tools completely from scratch. A set of manufacturing tools includes a die wood, into which the cutting rule is inserted; a female stripping board, through which waste material is pushed; and a male stripping board, which pushes the waste through the female board.

ArtiosCAD can create many of these elements automatically, but you have the flexibility to design your own tools as well.

You must create and save the designs to be used in the manufacturing tools before creating a new layout.

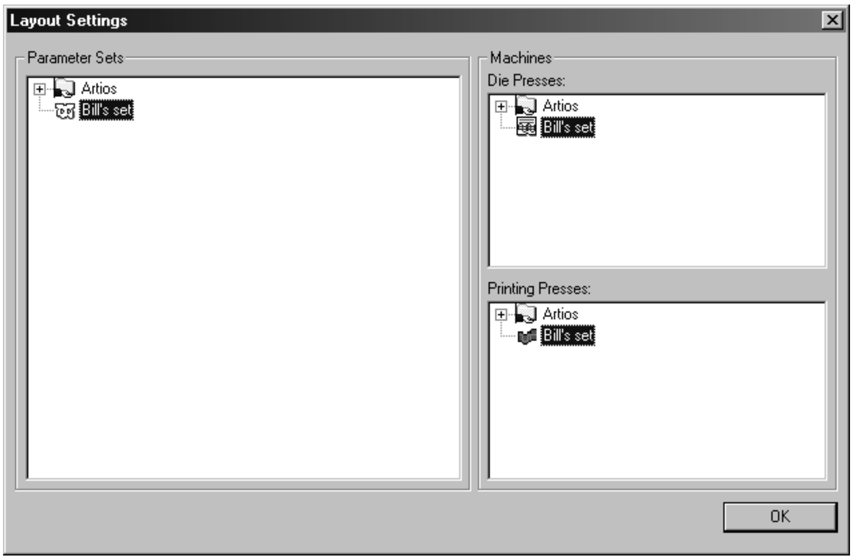
Overview: How to design a layout manually

1. Open one of the designs to be used in the layout and convert it to a manufacturing file. Choose a parameter set and machines. Add more designs if desired.
2. Use nesting or copy tools to duplicate the design(s) across the sheet.
3. Create tooling.

Creating the workspace and defining machines

The first step in creating a new set of manufacturing tools is to create the workspace that stores them and to define which machines will use the tools.

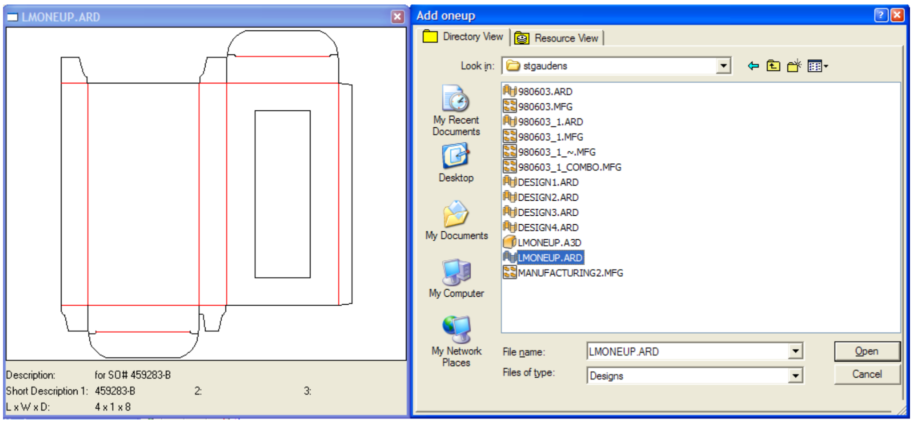
To create a new set of manufacturing tools, start ArtiosCAD and click **New Manufacturing File** on the **File** menu. This will create the workspace and prompt you to select a manufacturing parameter set, a die press, and a printing press. Click **OK** when you have made your selections.



Once you click **OK**, an empty sheet will be created that is sized according to the values in the parameter set.

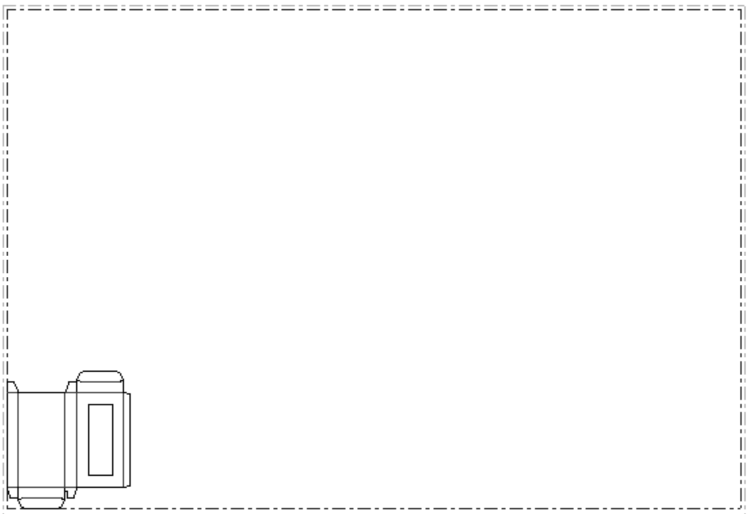
Adding designs to the sheet and arranging them

 The next step is to add a design to the layout. Click the **Add Oneup** tool. The Add oneup dialog box will appear along with a window that previews each design clicked.



Double-click the name of the design to add to the layout. Repeat this process to add different designs. Do not do this to add more copies of the same design.

Note: Adding more than one design to a layout creates a combination layout, which is covered later in this chapter.



 If the design has print items, click the **Change Print Items** tool to select the print item to use.

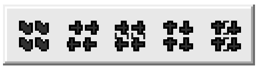
Arranging designs using the Nest tools

After placing the design on the sheet, the next step is to arrange as many copies as possible on the sheet. This process is called nesting.

Note: There is a limit of 300 designs per manufacturing sheet.

The distance, or **gutter**, between designs is set to zero by default. You can change it before using a Nest tool. Alternately, you can change it while using a Nest tool by specifying different step values between the rows and columns in the edit boxes on the Status bar.

1.  Click the **Select Design** tool and then click a line in the design. Handle points will appear at its perimeter and it will turn magenta to indicate it is selected.
2.  Click the **Straight Nest** tool, or choose one of the other Nest tools from the Nest Tools Flyout toolbar shown below.

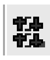
   

   The first tool on the flyout toolbar is the Straight Nest tool, which performs no rotation or alignment on the single designs as it arranges them.

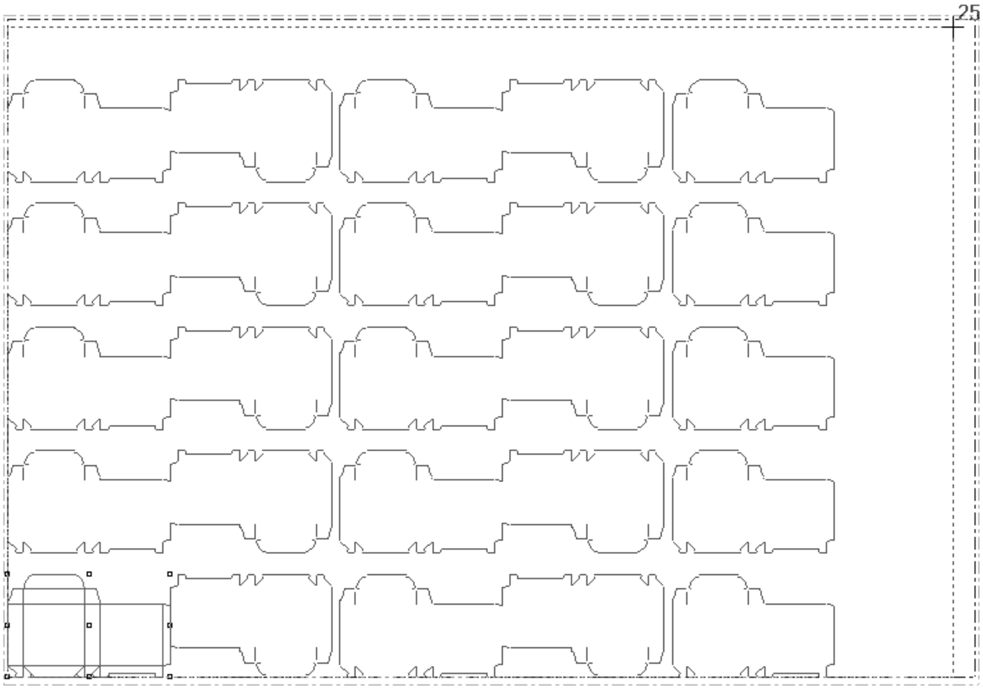
    The second tool on the flyout toolbar is the **Reverse 2nd Row Nest** tool, which flips the design horizontally every other row.

    The third tool on the flyout toolbar is the **Reverse 2nd Row Align Nest** tool, which works similarly to the **Reverse 2nd Row Nest** tool, except that it butts each row of designs against the sheet edge instead of nesting them as close together as possible.

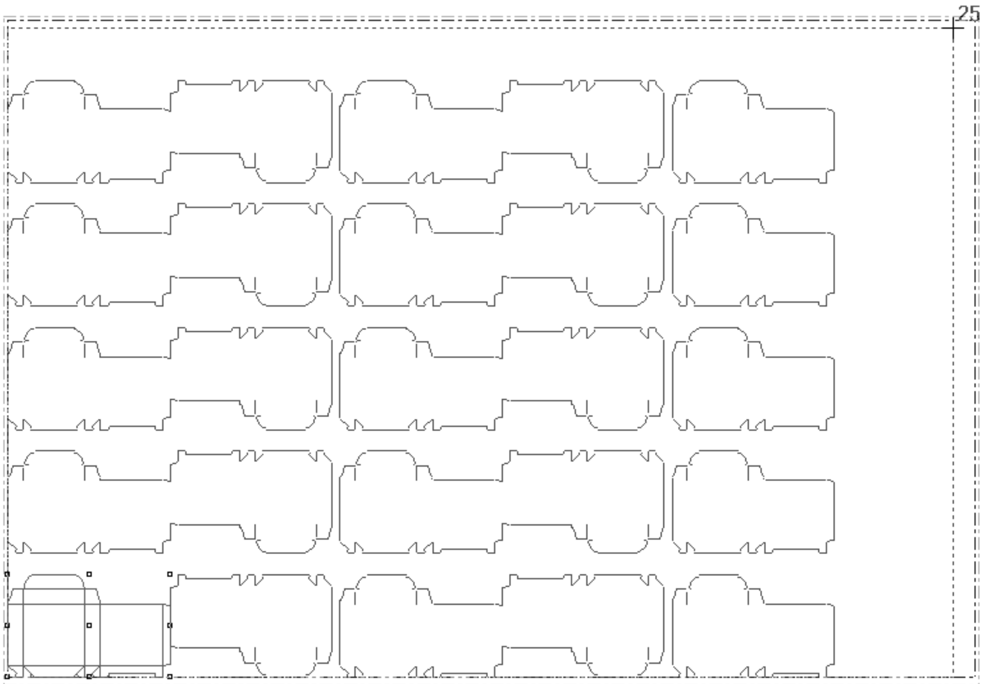
    The fourth tool on the flyout toolbar is the **Reverse 2nd Column Nest** tool, which flips the design vertically every other column.

    The fifth tool on the flyout toolbar is the **Reverse 2nd Column Align Nest** tool, which works similarly to the **Reverse 2nd Column Nest** tool, except that it butts each column of designs against the sheet edge instead of nesting them as close together as possible.
3. Once you select a Nest tool, as you move the drag with the mouse, ghost images of the design appear.

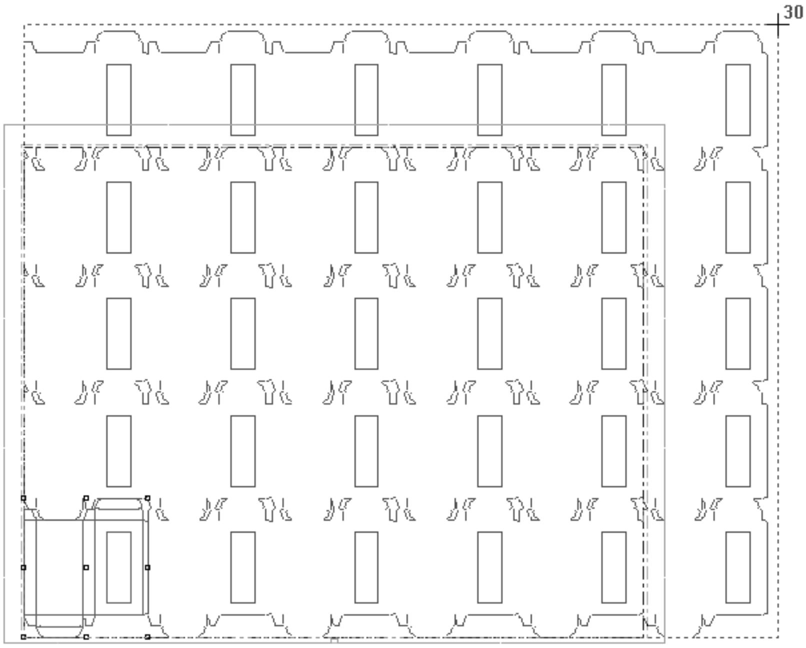
   Press `tab` while the drag is active to change the number of designs and step values between designs as desired. **X1** is the horizontal distance between the left edges of each single design's bounding box. **Y1** is the vertical distance between the bottom edges of each single design's bounding box. **X2** is the horizontal distance between the left edges of the second and third columns of single designs, and then the fifth and sixth. **Y2** is the vertical distance between the bottom edges of the second and third rows of single designs, and then the fifth and sixth. Shown below is an example of different **X1**, **X2**, and **Y2** values.



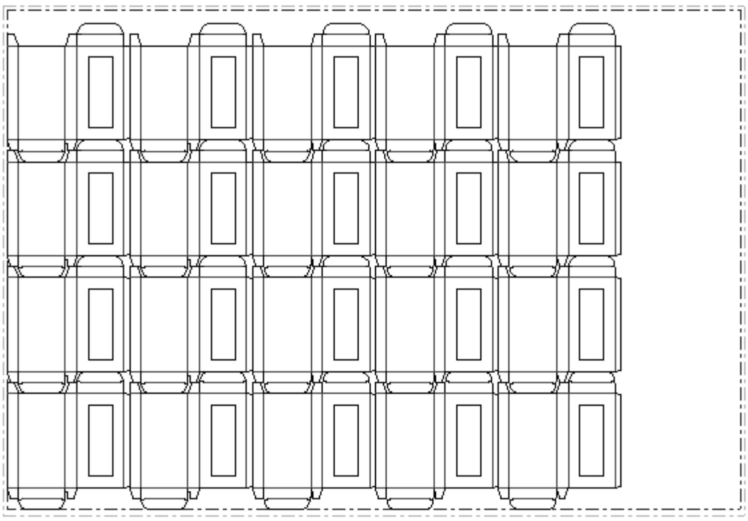
4. Click the mouse button when the ghost images fit inside the sheet without going over an edge. The other tools on the Nest flyout toolbar may allow more designs on a sheet; try them all. Shown below is an acceptable nest. Note that the total number of designs in the nest is shown next to the cursor.



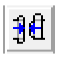
Shown below is an unacceptable nest as the single designs extend beyond the edge of the sheet.

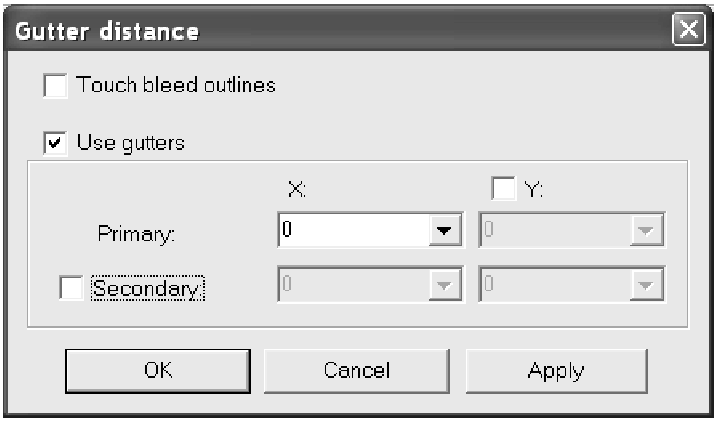


ArtiosCAD allows nests to overflow by one design so that you can see the distance by which you miss nesting an extra row. If you miss by only a small distance, this allows you to adjust your sheet size or gutter as needed so that the extra row fits. If you miss by a large distance, this shows that you have to accept less designs on the sheet.



Changing the gutter between designs

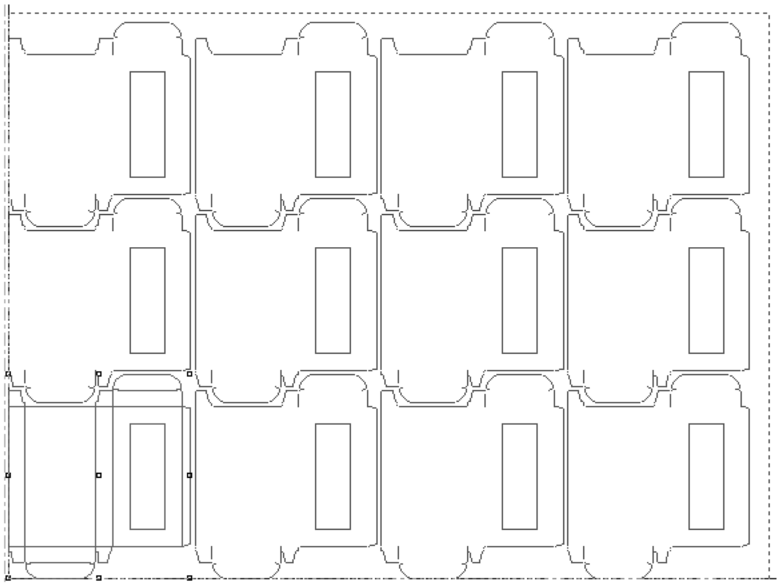
 The *gutter* is the distance between designs. By default, the gutter between the nested copies is set to zero. If you want a gutter between the copies, use the **Change Gutter Distance** tool and set the options in the Gutter Distance dialog box to the desired values.



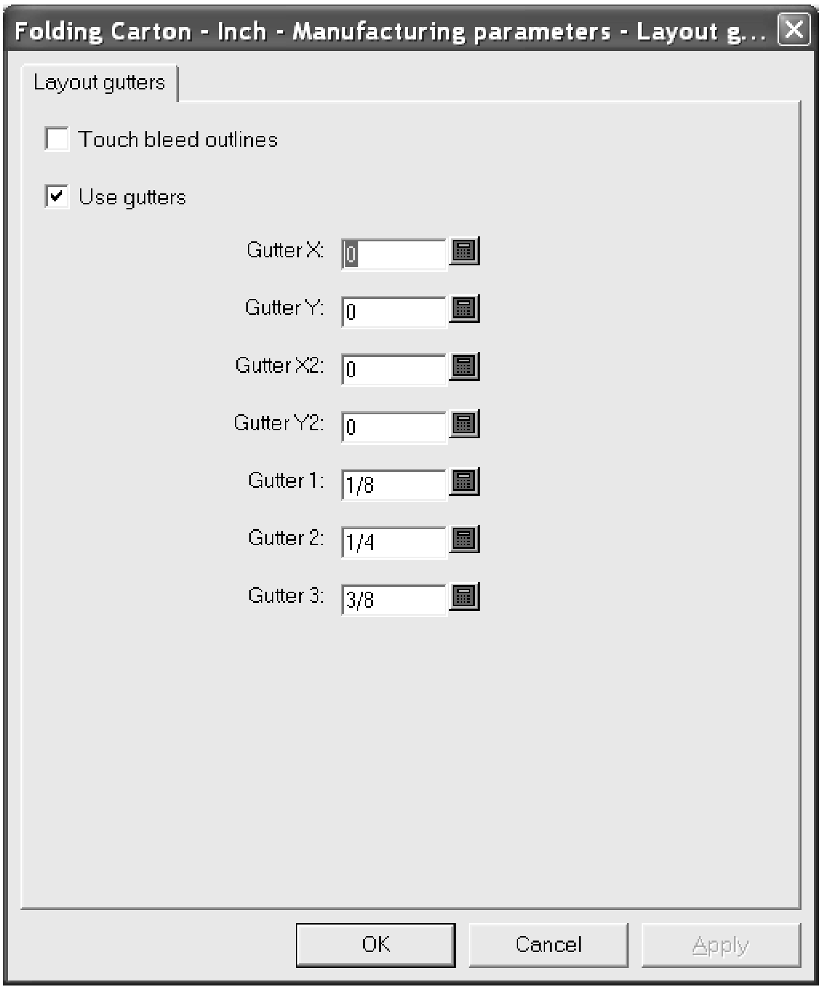
Touch bleed outlines uses the outer-most lines in the Inside Bleed or Outside Bleed layers of the single design(s) as the basis for the nest.

Use gutters enables gutters between the design edges. The checkboxes next to **Y** and **Secondary** enable different gutters as described in step 3 of the previous procedure. X and Y in the **Primary** row are X1 and Y1, while X and Y in the **Secondary** row are X2 and Y2.

The gutter will appear in the Nest tool drag.



Gutter defaults are contained in **Defaults** > **Manufacturing parameter sets** > **Parameter set** > **Startup defaults** > **Layout gutters**.



Arranging designs manually

If you don't want to use a Nest tool to arrange the designs on the sheet, you can arrange them manually using the following tools:

- the **Copy Designs** tool
- the **Copy Times Designs** tool
- the **Move Designs** tool
- the **Rotate Design** tools
- the **Mirror Design** tools.

To arrange the designs, you must first add at least one design to the layout using the **Add oneup** tool. Then select the design using the **Select Design** tool and use the tools listed above to arrange the designs as you wish.

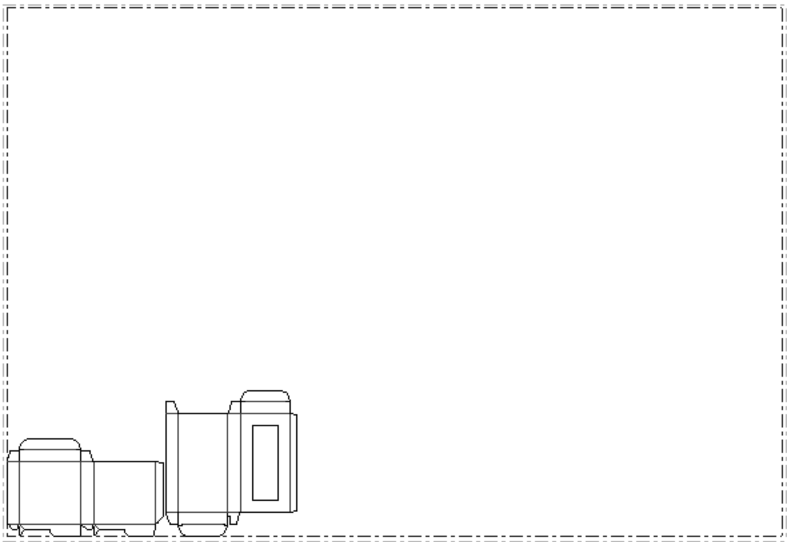
When arranging designs, hold down the SHIFT key to constrain the angle to 0/90/180/270 degrees. Clicking will only set the angle of the drag – it will not snap to a point if SHIFT is held down.

Making a combination layout

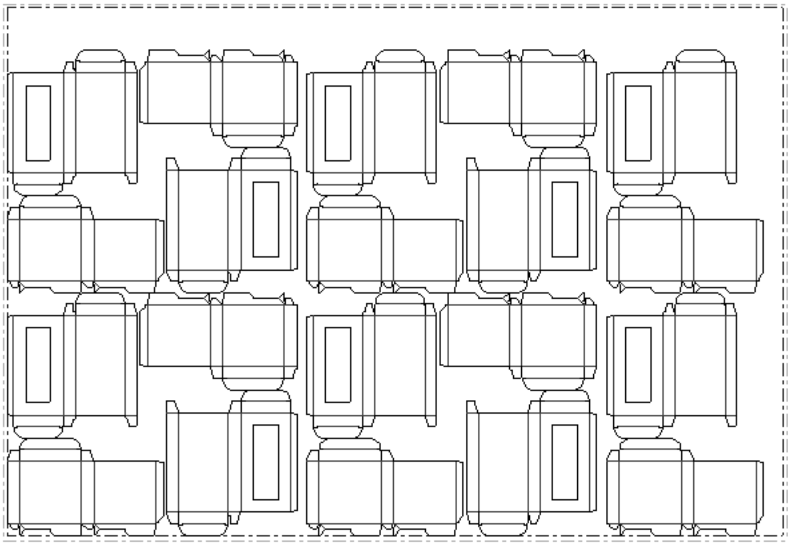
A combination layout is a manufacturing file that contains different designs.

To create a combination layout, do the following:

1.  Add the different designs to the sheet using **Add oneup** tool. Use the adjustment tools to change their position if desired.

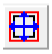


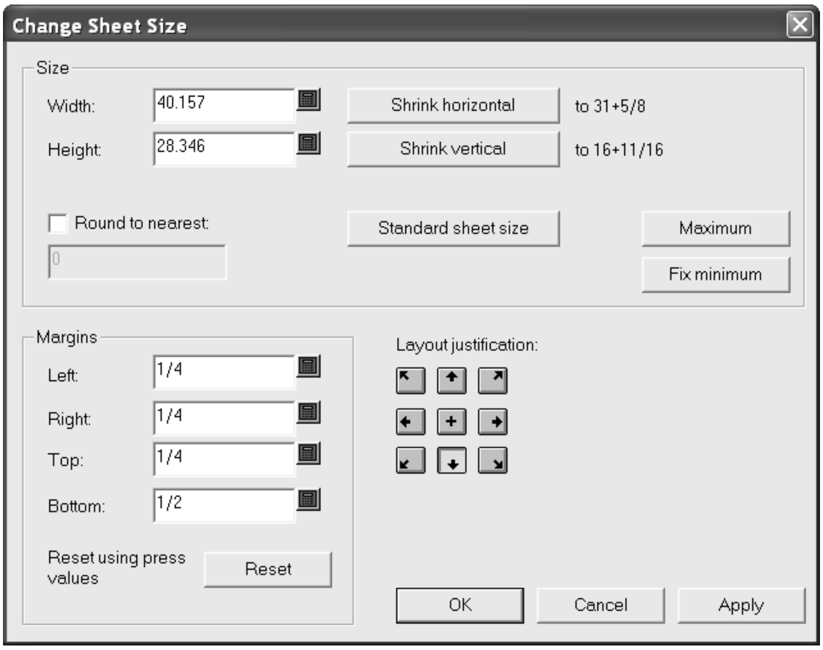
2.  Select all the designs with the **Select Designs** tool.
3. Use the tools on the Edit Layout toolbar to copy and move the designs.



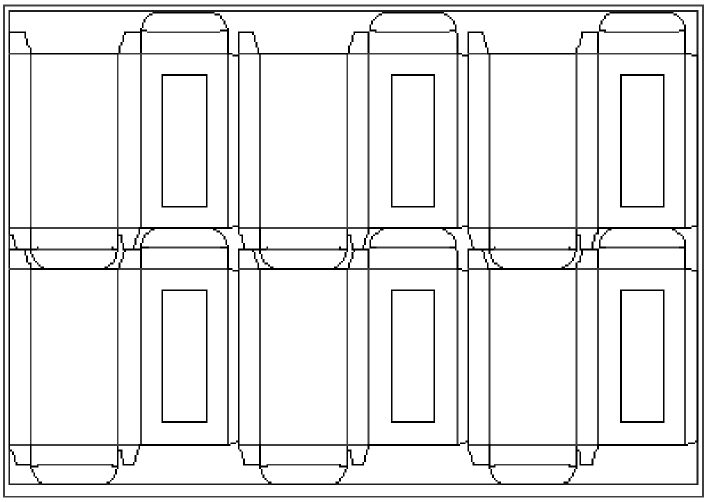
Changing the sheet size and justification of the designs

If the nested designs don't completely fill the sheet, you may want to adjust the size of the sheet or the position of the designs on the sheet.

 Click the **Change Sheet Size** tool to do this.



You must choose a layout justification method before you can change the fields in the **Size** and **Margins** groups. The buttons in the **Layout Justification** group move the group of designs so that the bottom center of the designs is aligned with the bottom center of the sheet (as shown above); the other buttons in the group move and justify the oneups in the direction indicated. Once a layout justification method is chosen, all the other fields become available and you can adjust the size of the sheet and its margins. **Standard Sheet Layout** lets you choose from the list of standard sheets defined for Standard Sheet Layout. Shown below is a sheet shrunk to the perimeter of the single designs.



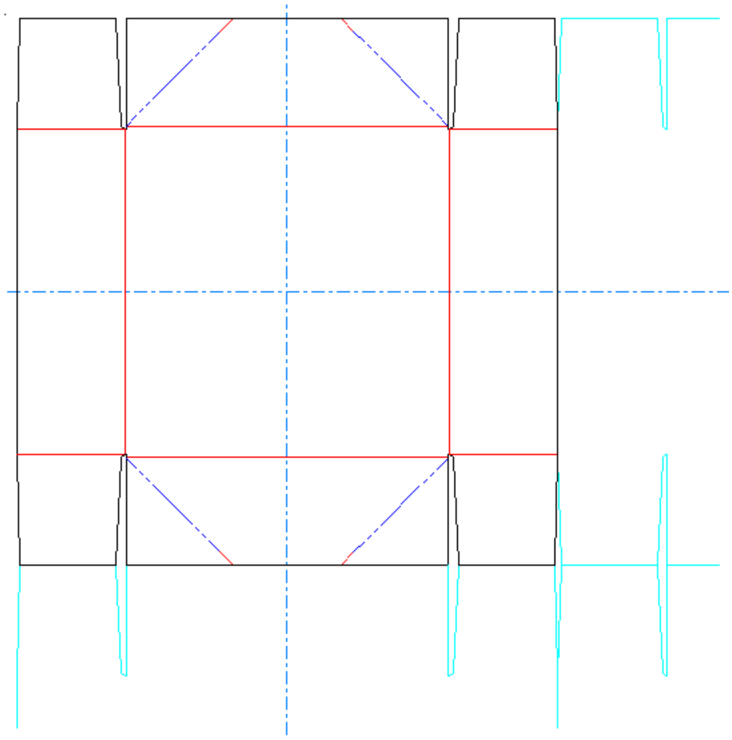
How to modify a single design embedded in a manufacturing file

Single designs that you add to a manufacturing file are copied and embedded in the manufacturing file.

Note: You cannot rebuild an embedded design. Also, do not change both the side up (using the **Side Up** tool on the View bar) and the structural orientation (using the **Structural Orientation** tool on the View bar); change one or the other but not both. ArtiosCAD stores the creation time of the embedded design and compares the internal timestamp to the one of the original workspace on the hard disc to check if the original design has changed since it was embedded. If the layout workspace came from another site, ArtiosCAD cannot compare the timestamps and it will take a long time to open the workspace. To disable this check and have the workspace open faster, deselect the **Check embedded designs upon opening MFG file** checkbox in **Options** > **Defaults** > **Shared Defaults** > **Startup defaults** > **Embedded Design Check**.

To change a single design embedded in a manufacturing file, do the following:

1.  Click the **Open Embedded Design** tool and select the oneup to open. If desired, deselect the **Add layout lines** checkbox. If you leave the checkbox selected, annotation lines representing the designs touching the selected design are shown in Single Design for reference while editing as shown below.



The initial state of the checkbox and the amount of annotation lines shown are set in **Options** > **Defaults** > **Startup defaults** > **Embedded Design Window Size**.

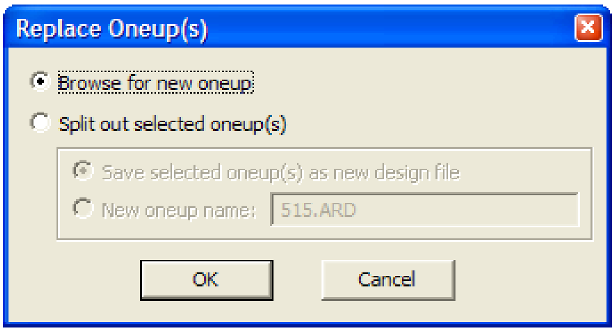
2. Modify the design as desired.
3. To save the embedded copy as the original file, click **Save** and then click **File** > **Return to Manufacturing**.
4. To keep the original copy as it was and only modify the embedded version, click **File** > **Return to Manufacturing**.

How to replace designs in a layout

To replace a design in a layout, use the Replace Selected Oneups tool. The Replace Selected Oneups tool can also rename designs in a layout, or save them as entirely new single design workspaces with new names.

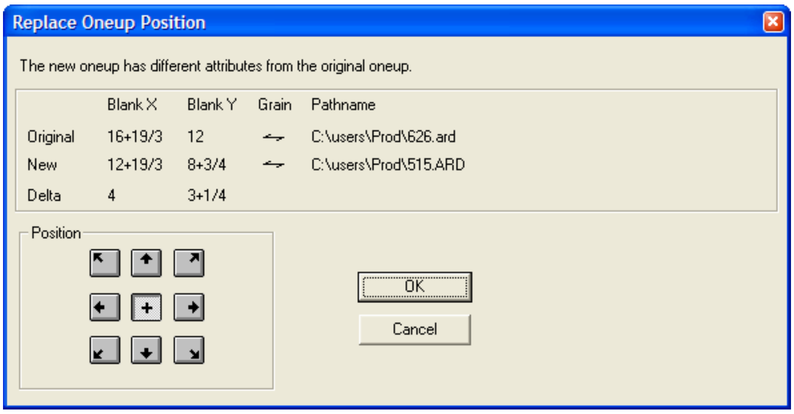
To replace a design in a layout with another, do the following:

1.  Select the instances of the design to replace with the **Select Oneups** tool. Hold down `CTRL` to select more than one instance.
2.  Click **Replace Selected Oneups**. The Replace Oneup(s) dialog box appears.



3. There are two main option buttons in the Replace Oneup(s) dialog box. **Browse for new oneup**, when clicked, prompts you to select a new file for replacing the one currently in the layout. Selecting **Split out selected oneup(s)** lets you either save the selected design(s) as a new single design workspace (Save selected oneup(s) as new design file) or just rename the selected design(s) in the layout (New oneup name). Choosing **New oneup name** does not create a new file.
4. Click **OK** to either browse for the new design, save a new workspace, or rename the design in the layout.

If the new design is a different size than the original, or has a different grain/corrugation direction, or a different position, the Replace Oneup Position dialog box appears as shown below.



To change the position of the new design in reference to the position of the bounding box of the old design, click the appropriate button in the **Position** group.

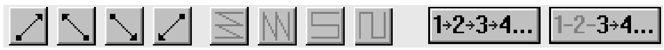
How to perform manual sequencing of designs

 ArtiosCAD usually performs automatic sequencing of single designs within a layout when it is output. If desired, you can turn off the fully automatic sequencing in the Output definition and control the automatic sequencing of the single designs yourself using the **Layout Manual Sequencing** tool.

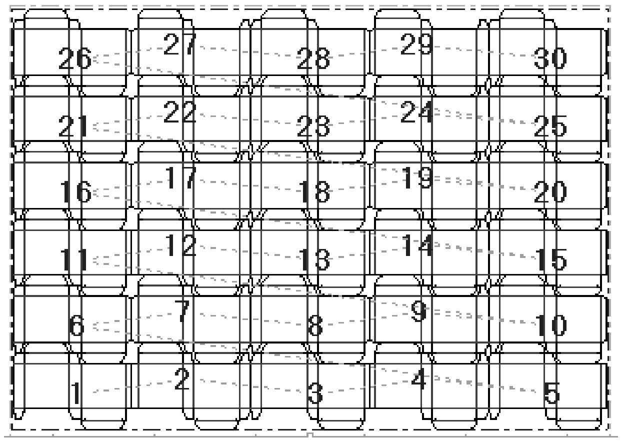
This tool is available only in the Dieboard layer of manufacturing files and also in counter layouts. This tool does not have Undo functionality.

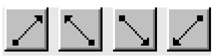
Note: Any sequencing done with this tool will be lost on Output if sequencing is enabled in the Output definition in Defaults. To check for a CAM output, open the Output definition in Defaults and look on the Processing tab. If the **Sequence one-ups in layout** option is checked, ArtiosCAD will discard the manual sequencing you specified when the layout is output. To deselect the checkbox, check the **Enable subroutined output** checkbox and clear the **Sequence one-ups in layout** checkbox. To check for a Sample output, look on the Sample Sequencing tab and clear the **Sequence one-ups in layout** checkbox.

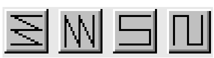
When activated, this tool places option buttons on the Status bar as shown below and performs a default nest from the bottom left by row.



Sequence order numbers appear in the center of each single design indicating their cutting order. Green dashed lines connect the designs. The picture shown below is the default sequence.

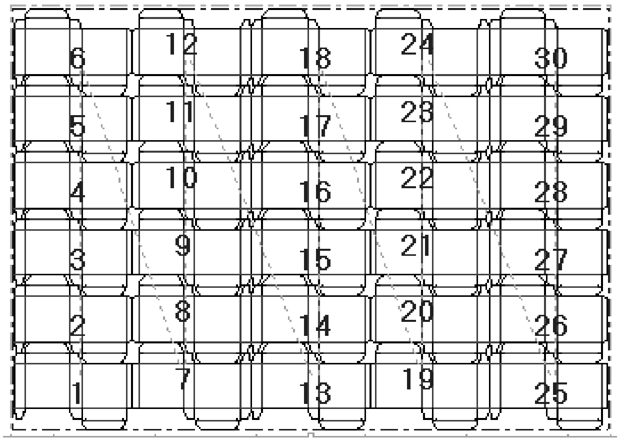


 The first group of four buttons sets the starting point for the sequencing. The first button starts the sequencing at the bottom left of the layout. The second button starts the sequencing at the bottom right of the layout. The third button starts the sequencing at the top left of the layout. The fourth button starts the sequencing at the top right of the layout.

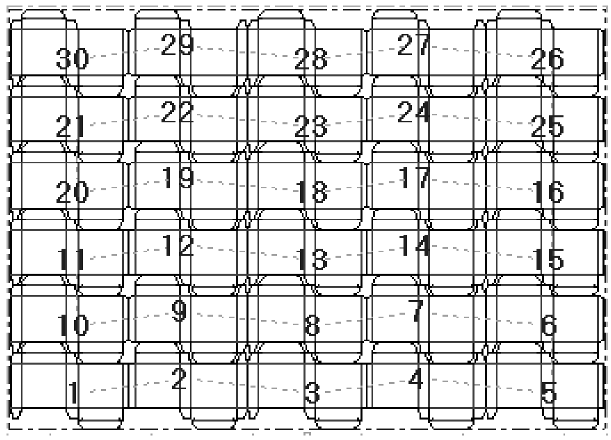
 The second group of four buttons controls the sequencing direction and is not available until a button in the first group is pressed.

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.

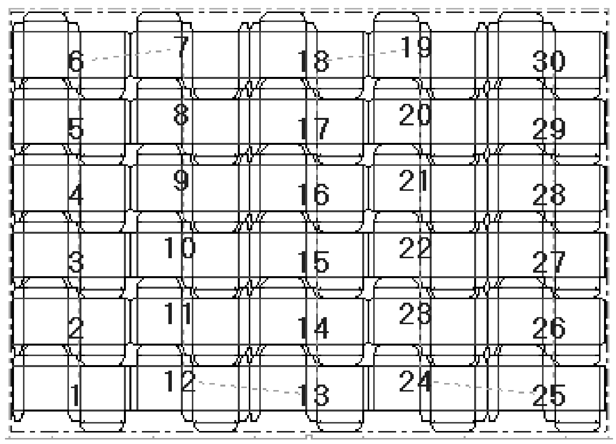
The second button sequences each vertical column, starting each column on the same side of the layout so that each column is sequenced in the same direction. The picture shown below starts from the bottom left.



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 picture shown below starts from the bottom left.



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. The picture shown below starts from the bottom left.



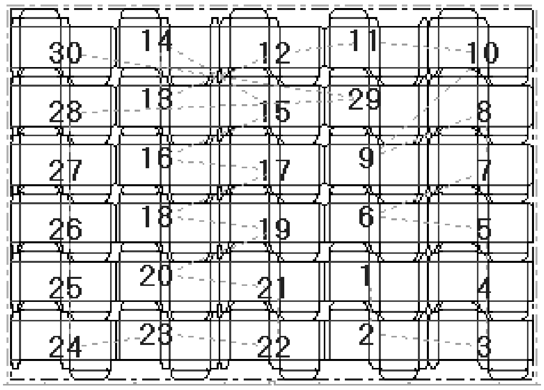
The **Sequence All** button lets you manually sequence the single designs in the layout.

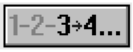
To use this tool, do the following:

1. 

Click **Sequence All**.
2. Click the first design in the sequence. It will become number one, and all the others will increment up by one to reflect the change.
3. Continue clicking single designs. As you click them, they will be incremented up by one - the first design clicked is one, the second two, the third three, and so forth, with the remaining designs incrementing up by one conforming to the pattern you selected on the Status bar.

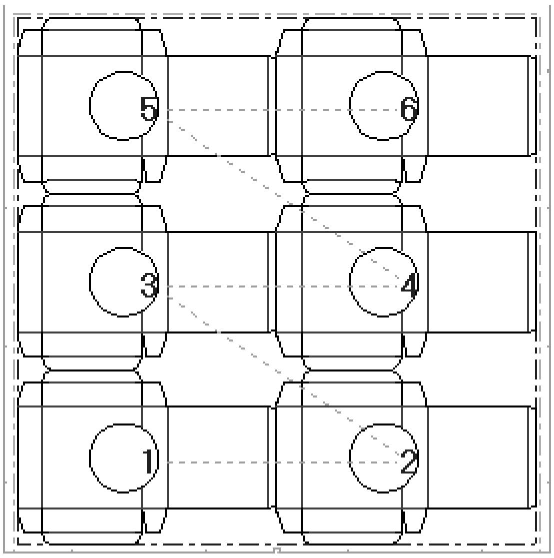
The sequence shown below is exaggerated for effect.



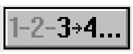


The **Sequence Range** button lets you resequence a specific range of one-ups, instead of sequencing all of them.

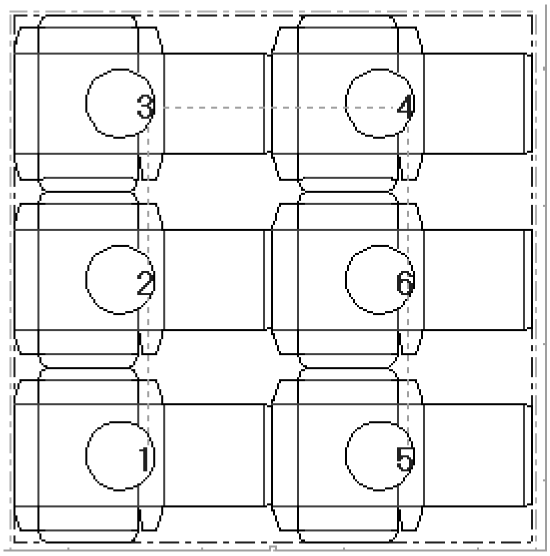
For example, in the layout shown below, it is sequenced by row, but perhaps it should be a vertical snake instead.



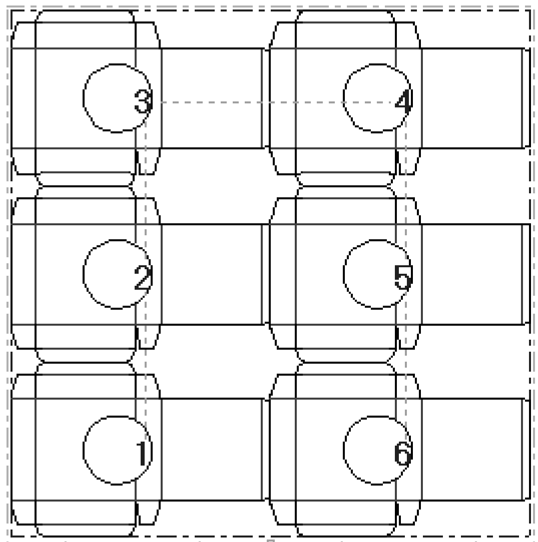
To resequence the single designs without using the automatic sequencing tools, do the following.

1. 

Click **Sequence Range** and then click design three. It will not change.
2. **Indicate next item** will appear on the status bar. Click design six. It will change to design four.



3. Click design six to change it to design five. What was design five changes to design six at the same time, and the resequencing is complete.

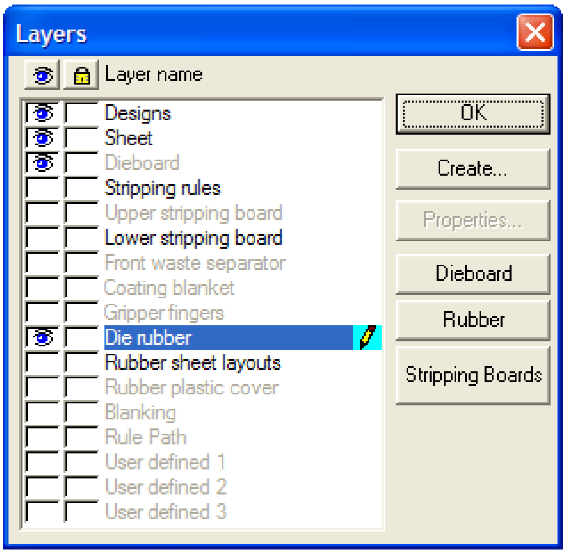


This tool works in the same way for resequencing counter layouts.

Note: Any sequencing done with this tool will be lost on Output if sequencing is enabled in the Output definition in Defaults. To check, open the Output definition in Defaults and look on the Processing tab. If the **Sequence one-ups in layout** option is checked, ArtiosCAD will discard the manual sequencing you specified when the layout is output. To deselect the checkbox, check the **Enable subroutined output** checkbox and clear the **Sequence one-ups in layout** checkbox. To check for a Sample output, look on the Sample Sequencing tab and clear the **Sequence one-ups in layout** checkbox.

How to create tooling

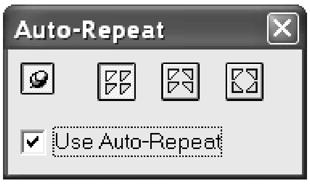
Tooling refers to the manufacturing elements in the presses that actually make the containers. Die boards and upper and lower stripping boards are tooling. You can easily switch which tooling layers are turned on by clicking **Dieboard**, **Rubber**, or **Stripping Boards** in the Layers dialog box.



A note on Auto-repeat

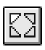
 The **Auto-repeat** tool on the View bar automatically repeats elements such as stripping rules, holes, and pins throughout congruent areas of the layout. Clicking the tool turns it on and off.

Auto-Repeat has three different modes. The mode is set in the Auto-Repeat dialog box opened by clicking the drop-down arrow next to the Auto-Repeat button on the View bar.



 **No Mirror** auto-repeats those elements, areas, or constructions that look exactly like the object of the current tool's action.

 **Mirror about Vertical** auto-repeats elements, areas, or constructions that match the object of the current tool's action either exactly or mirrored about the vertical axis (in effect horizontally).

 **Horizontal and Vertical Mirror** auto-repeats elements, areas, or constructions that match the object of the current tool's action exactly, mirrored about the vertical axis (horizontally), or mirrored about the horizontal axis (vertically).

 If the pushpin is up, the dialog box closes automatically on the next mouse click. If the pushpin is down, the dialog box remains open until it is explicitly closed or the manufacturing file is closed.

The **Use Auto-Repeat** checkbox turns Auto-Repeat on and off. If Auto-Repeat is off, the tool on the View bar appears disabled. Click the tool on the View bar to enable it.

Table: Tools using Auto-Repeat

Add Bridge	Add Interference	Add Nick	Strip Area
Add Tack	Add Upper Block	Add Upper Pin	
Add Upper Rule	Create Carrier Rule	Create Stripping Rule	
Delete (with manufacturing selection)	Delete Bridge	Delete Nick	
Delete Strip Area	Make Air Hole	Move Bridge	
Move Nick	Repeat	Repeat Strip Area	

Auto-Repeat works by comparing what items an object touches or crosses.

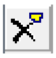
Table: Auto-Repeat Dependencies

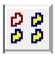
Element	Auto-Repeated depending on
Stripping rules Air holes	Which designs they touch or cross
Lower holes	Waste areas
Pins Upper rules Blocks	Lower hole or edge region they are in
Lower carrier rules	Which hole or edge region they cross
Block pins	Congruency of blocks contained in
Rule paths	Congruency of potential Rule paths

How to manipulate elements and designs

In order to geometrically manipulate elements (the non-design parts of a manufacturing file) and designs, you first have to select them. Selected items turn magenta. You can use the tools on the Adjust Outline toolbar to manipulate elements as well as the tools listed below.

 The Select Element tool selects individual elements. Use this tool to select pins, pieces of rule, mounting holes, bars, etc. This tool cannot be used to select any part of a design. Hold down CTRL to make multiple selections.

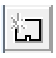
 The **Delete Element** tool deletes the selected element(s). This tool is available only when there is a current selection. This tool uses the settings in the Auto-Repeat dialog box.

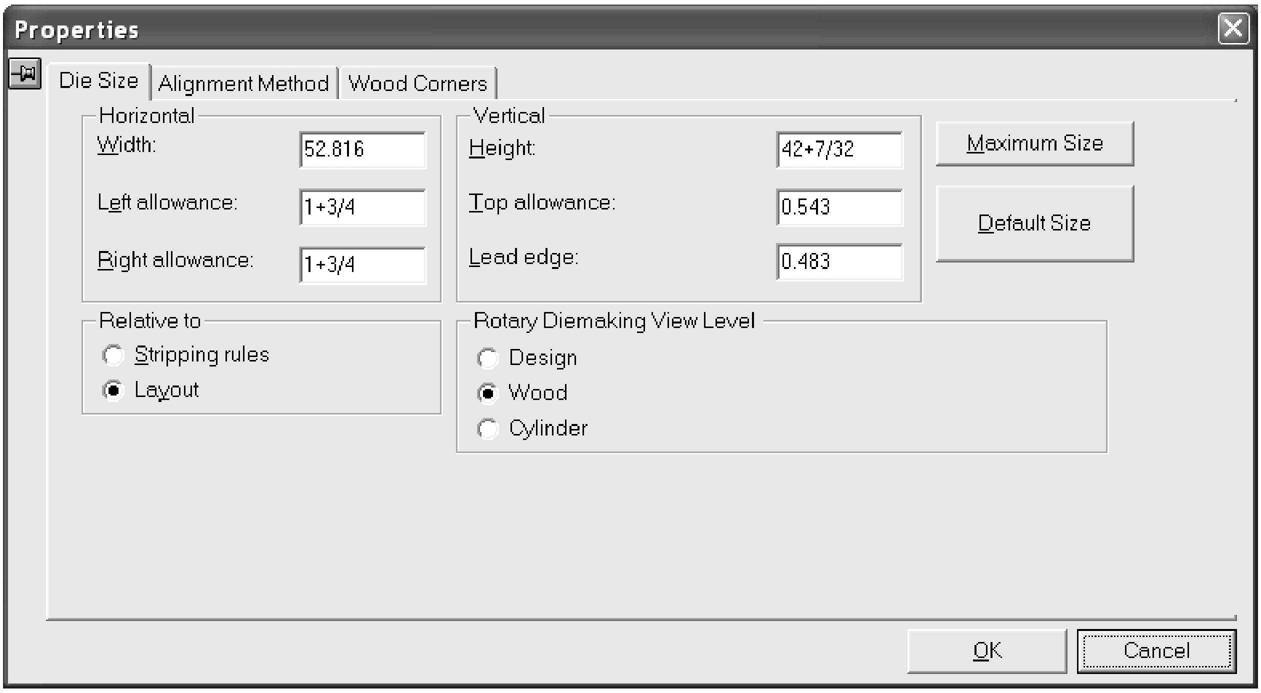
 The **Repeat Element** tool takes a single element and auto-repeats it throughout the manufacturing file. This tool is available only when there is a current selection. It mirrors according to the settings in the Auto-Repeat dialog box.

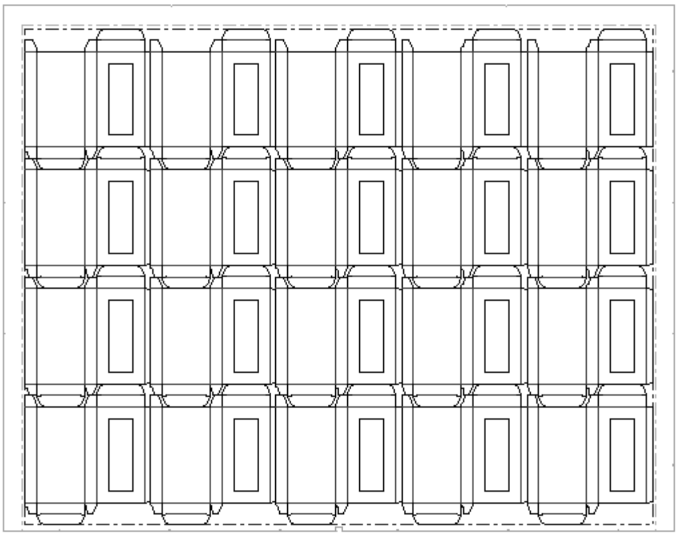
 The **Select Design** tool selects a design. Hold down CTRL to make multiple selections.

 The **Delete Design** tool deletes the selected design(s). This tool is available only when there is a current selection.

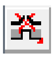
Creating the dieboard edge

 Once the designs are nested, create the dieboard edge. If necessary, switch to the Dieboard layer using the layer control on the View bar. Click the **Create Wood Edge** tool to automatically create a dieboard edge. If the die wood is not in the position you would like, select it and double-click it to access the Properties dialog box. You can then change the dieboard size, how it is aligned, the method for making the corners, and if it is a rotary die, the level at which the layout is viewed. Click **OK** to return to the layout.



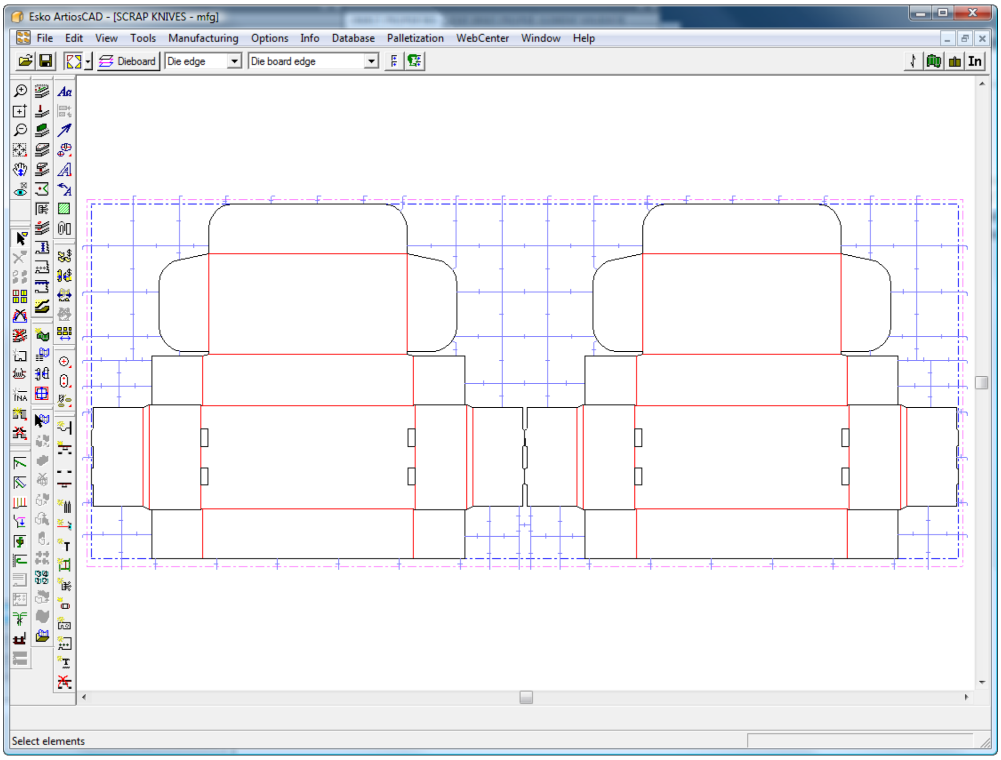


Once the dieboard edge is created:

-  use the **Create Hand Hold** tool to add hand holds.
-  use the **Create Burn Name** tool to add a burn name, or use a Geometry Macro to add a more verbose burn name.
-   use the **Add Tack Bridge** tool and the **Delete Bridge tool** to add and remove tack bridges. Click the **Add Tack Bridge** tool and click the point on the die wood edge where you want the tack bridge. Click the **Delete Bridge** tool and indicate the tack bridge to delete.
- Double-click the dieboard edge with a **Select** tool to change its properties.

How to create scrap knives

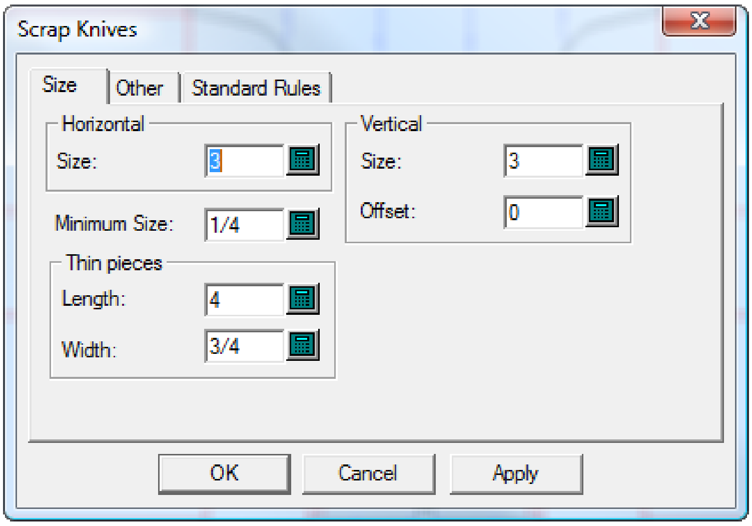
Scrap knives chop the waste into evenly sized rectangles. Where the waste meets the design, the waste follows the contour of the design but fits into the grid of scrap knives on its other edges.



Scrap knives use the same stripping edge and hook parameters as **Create Stripping Rule**.

Scrap Knives dialog box

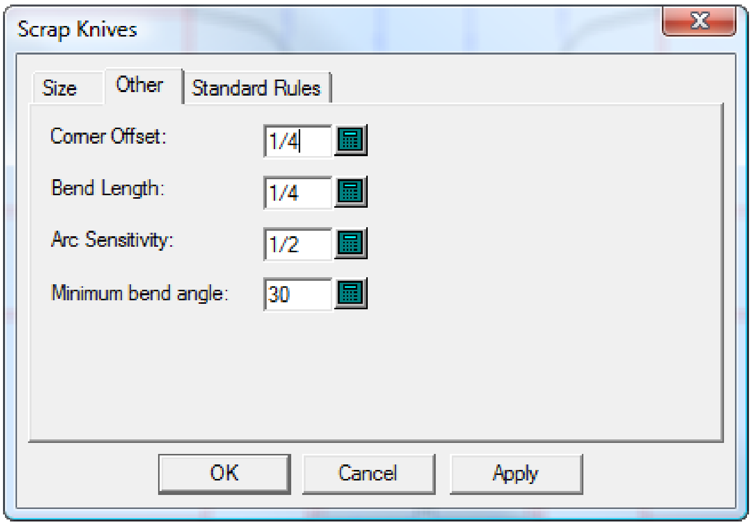
When you create scrap knives, the Scrap Knives tool first opens the Scrap Knives dialog box so you can set how ArtiosCAD will create them.



The values in **Horizontal** and **Vertical** set the size of the pieces. If **Offset** is set to a non-zero value, the vertical lines zig-zag by that amount. There can be slight exceptions to these sizes to compensate for corner offsets and bend lengths.

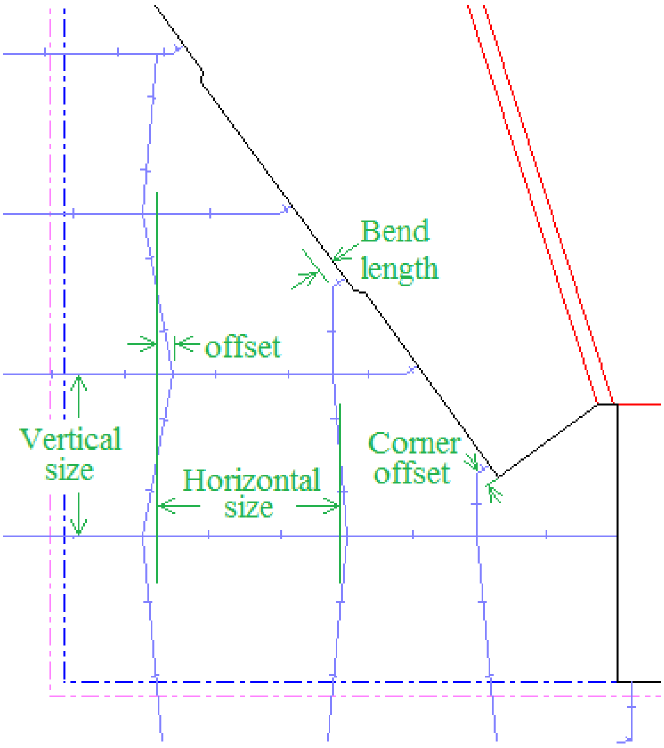
Thin pieces of waste can be a little longer than the nominal width or height of normal scrap pieces. In the **Thin Pieces** group, set the size limits for such pieces. ArtiosCAD chops pieces of scrap whose width is less than or equal to the thin width into pieces whose length is less than or equal to the thin length.

The Other tab of the Scrap Knives dialog box is shown below.

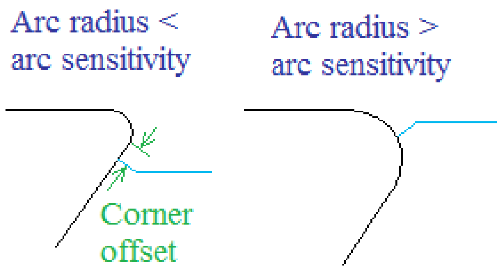


Corner Offset sets how far scrap knives must be from a corner. For optimal results, the corner offset should be less than 1/5 the horizontal and vertical size.

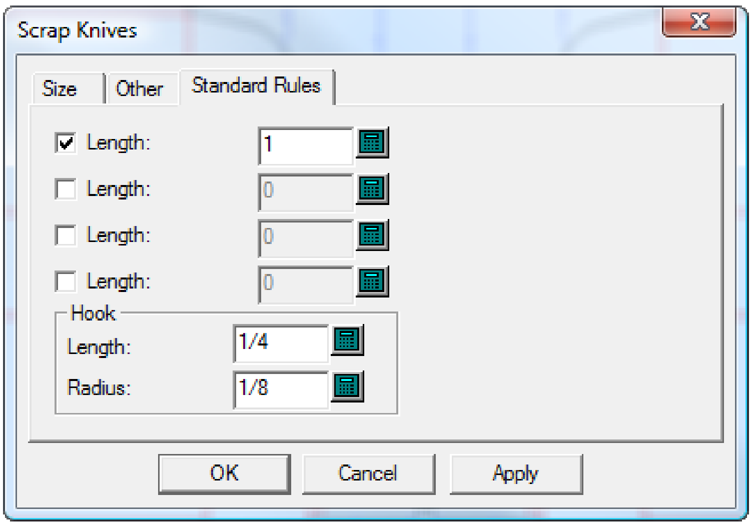
ArtiosCAD will bend a scrap knife to meet the design perpendicularly if the angle at which it meets the design is more than the **Minimum bend angle**. The value in **Bend Length** sets the length of the segment that is perpendicular to the design.



ArtiosCAD considers arcs larger than the value in **Arc Sensitivity** to be blends. Scrap knives generally do not meet arcs and are positioned away from them by the amount of the corner offset. But if an arc is large enough, a scrap knife can meet it.



Some diemakers use standard-sized pieces of rule with an optional hook on the end when a scrap knife extends off the sheet. Standard rules may be slightly shorter than the scrap knives they replace so that they do not go outside the stripping edge. These standard rules are defined on the Standard Rules tab of the Scrap Knives dialog box.



You can define four such pieces of rule in Defaults (**Options** > **Defaults** > **Die Press parameter sets** > **Your press** > **Scrap knives** > **Standard Rules**). Check the checkboxes for the standard pieces of rule to use and change the **Length** if desired. Change the **Length** and **Radius** for the hook if desired.

ArtiosCAD follows these rules when placing standard rules:

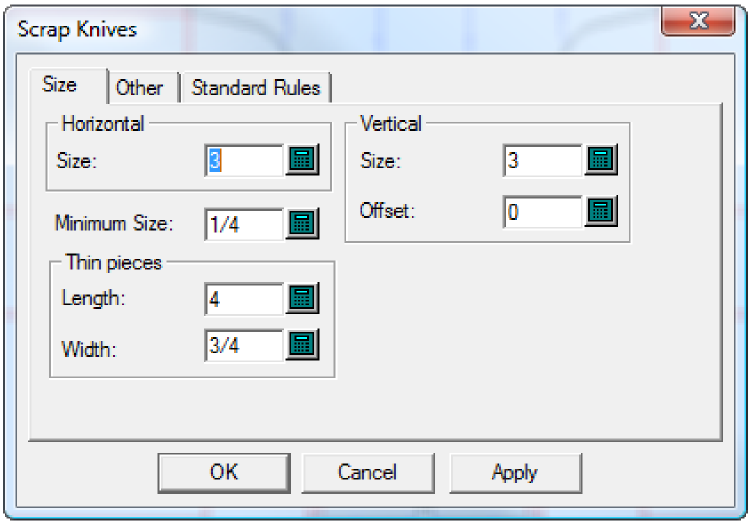
- If the length of the rule being replaced is between the lengths of two standard rules, it will use the shorter one.
- The length of the standard rule must be less than or equal to the length of the rule it replaces.
- The amount the standard rule passes the edge of the sheet minus the hook must be at least half as much as the rule it replaces.
- Standard rules on the leading edge do not get hooks, because there is usually no room to fit the hook in the die.

Adding scrap knives to a layout

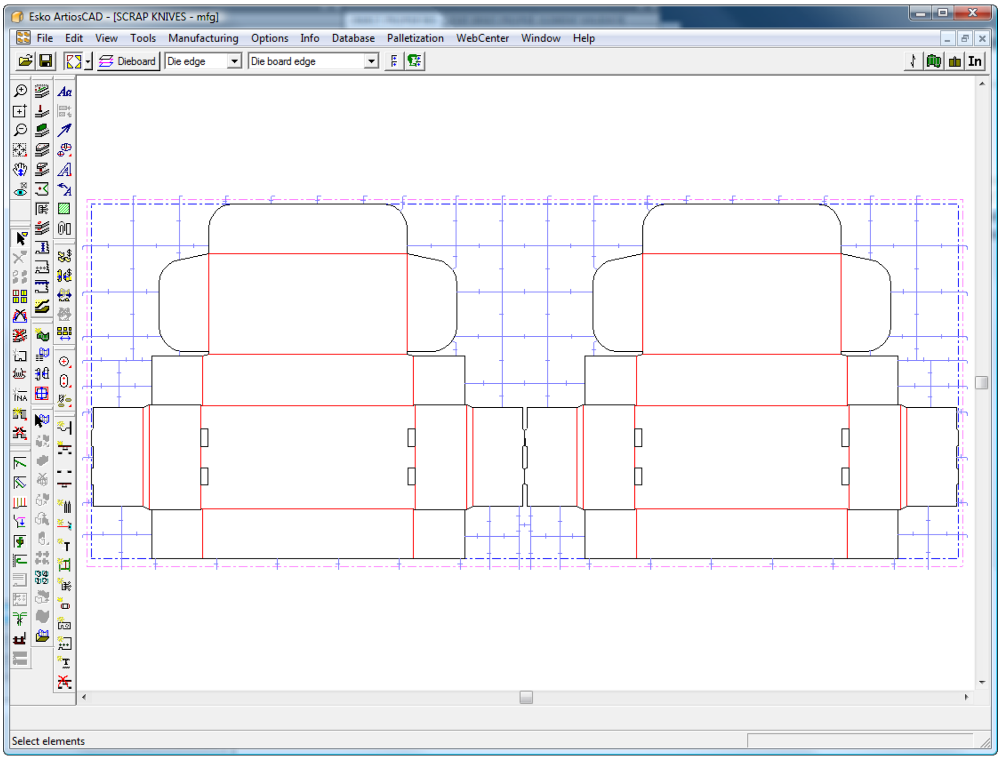
To add scrap knives to a layout, do the following:

1. Create a layout in ArtiosCAD.
2. Click **Tools** > **Scrap Knives**.

   The Scrap Knives dialog box appears.



3. When you have finished setting the options on all the tabs of this dialog box as described previously, click **OK** to create the scrap knives.



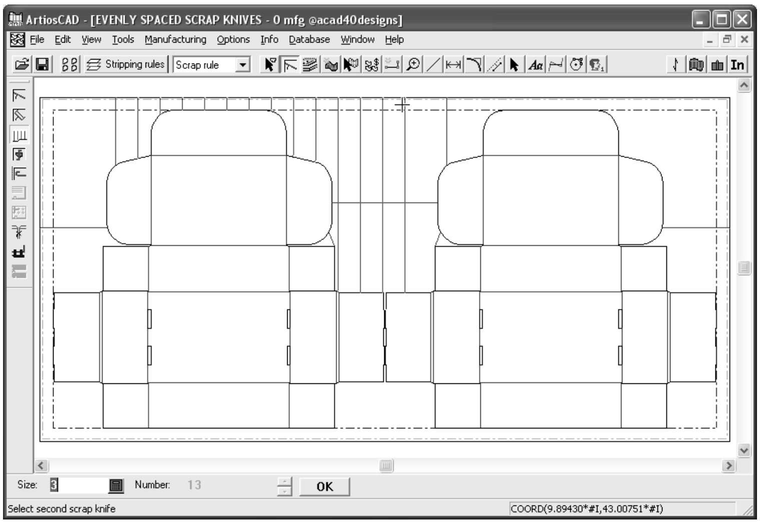
To make a corner be one continuous rule, add stripping rules at the corner before adding scrap knives.

How to create evenly-spaced scrap knives

 The **Evenly Spaced Scrap Knives** tool creates scrap knives in the waste to the side of an existing line, typically a stripping rule.

The **Evenly Spaced Scrap Knives** tool has two modes. The first mode is when a stripping rule is clicked as the first scrap knife and a freehand coordinate is clicked as the second point. The second mode is when a stripping rule is clicked as the first scrap knife, and then another stripping rule is clicked as the last scrap knife.

Shown below are the scrap knives created when the stripping rule on the left dust flap of the lid of the left design is chosen as the first scrap knife and the drag is moved to the right with the cursor being kept near the top of the sheet. The knives are evenly spaced until the drag is set, after which the **Number**: field becomes available for changing as desired. Once the **Size**: and the **Number**: fields are set, click **OK** to create the scrap knives.

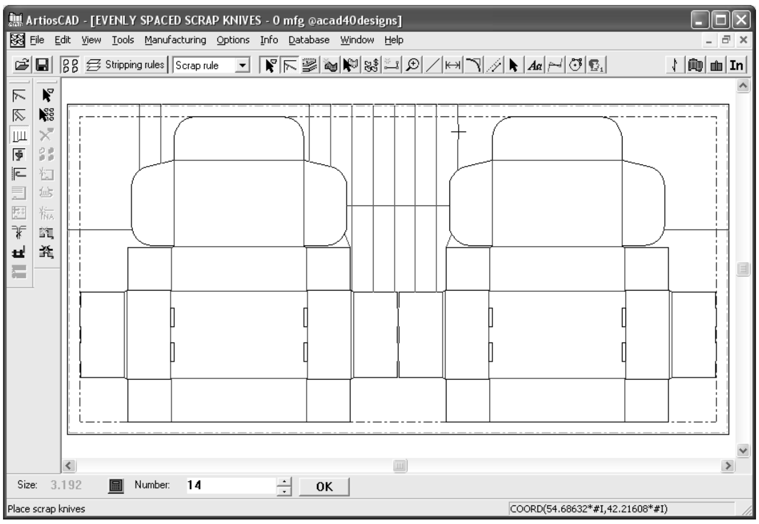


To use the tool in its first mode, do the following:

1. Create a layout and add stripping rules to it.
2.  Click the **Evenly Spaced Scrap Knives** tool.
3. Click a stripping rule from which the scrap knives will be offset.
4. Use the drag to set the number of scrap knives and click the mouse button somewhere that is not on a line.
5. Review the values in the **Size**: and **Number**: fields, changing them if desired.
6. Click **OK** to create the scrap knives.

To use the tool in its second mode, do the following:

1. Create a layout and add stripping rules to it.
2.  Click the **Evenly Spaced Scrap Knives** tool.
3. Click a stripping rule from which the scrap knives will be offset.
4. Next, click another parallel stripping rule as the final scrap knife.



Note that the **Size**: field has changed so that a whole number of scrap knives can fit between the first stripping rule and the second stripping rule.

5. Adjust the value in the **Number**: field as desired and click **OK** to create the scrap knives. Note that the distance between each knife changes as the number of knives changes.

The drag for this tool is very sensitive. The tool can operate differently just by clicking a few pixels apart.

If you click another line as the last scrap knife, that has the effect of accepting the currently-shown scrap knives and restarting the tool with the selected line being the first scrap knife. This allows the quick and easy creation of several sets of chop knives.

Properties of evenly spaced scrap knives

Scrap knives created with the Evenly Spaced Scrap Knives tool are made with line type **scrap knives**.

They use default bridging and their default pointage is set in **Options** > **Stripping rules**.

Whether they have hooks on the end is set in **Options** > **Hook Defaults**.

For rotary dies, the bend length (or perpendicular offset) is set in **Tools** > **Scrap Knives** > **Other** > **Bend Length**.

If a scrap knife is made from more than one line, a group is created so it can be selected as one item.

The initial value for the **Size**: field is set in **Options** > **Defaults** > **Die press parameter sets** > **Parameter set** > **Scrap Knives** > **Size** > **Horizontal Size**. To change it on the fly, click **Tools** > **Scrap Knives** and set the **Size**: field in the **Horizontal** group.

How to move scrap knives

 Use the **Move Scrap Knife** tool to move a scrap knife or stripping rule that is not in the desired location.

When this tool is active, it has settings on the Status bar as shown below.

The first button sets the offset of the moved or copied scrap knife to be from the selected rule, while the second button sets the offset to be from the nearest end of a line.

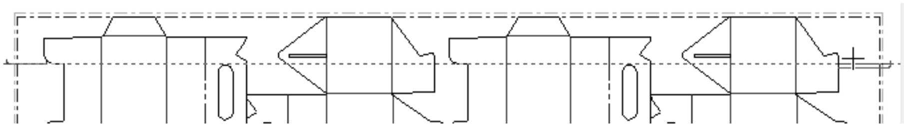
Parallel controls whether or not the drag will only snap to positions parallel to the original scrap knife or if the drag will snap to positions perpendicular to the original alignment as well.

Copy makes a copy of the selected scrap knife instead of moving it.

Offset sets the offset distance from either the selected rule or the nearest end of a line according to which button is pressed.

To use this tool, do the following:

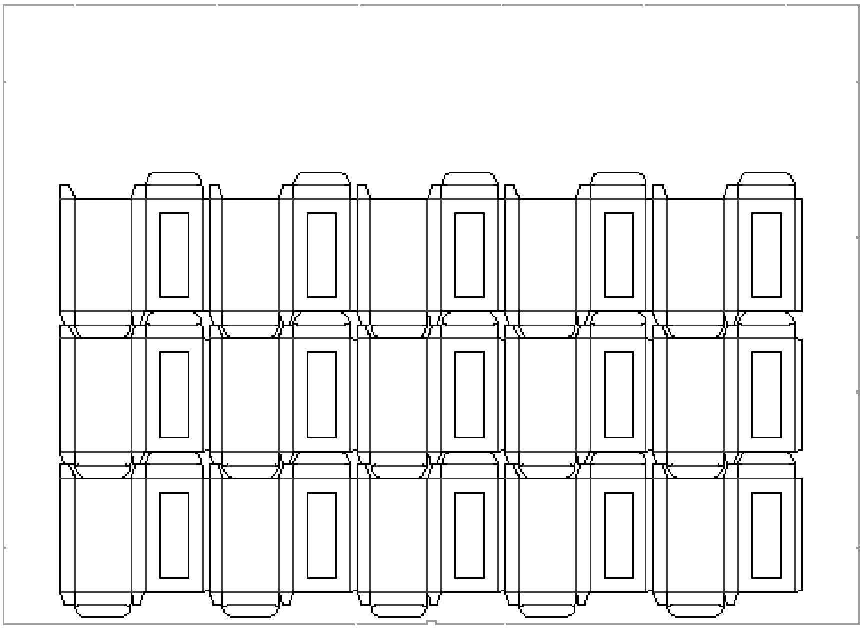
1.  Click the **Move Scrap Knife** tool and then click the scrap knife to move.
2. Drag the knife to the new position. If there is a bend on the start of the rule, it will be recalculated as needed. If you align the new scrap knife with an existing scrap knife, a dotted line will be drawn between the two to indicate alignment.



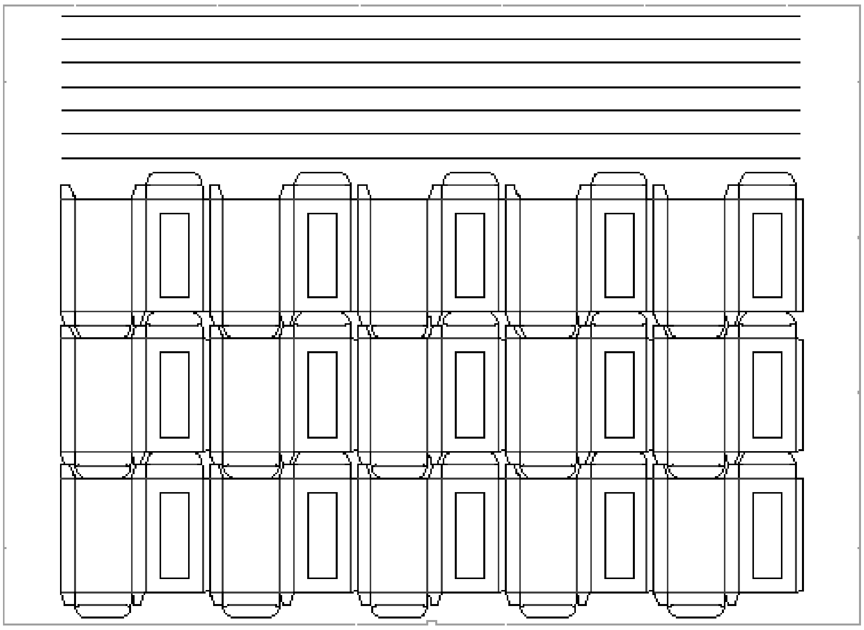
3. Click the put-down point for the new scrap knife.

How to add balance knives

 Balance knives are used to balance the die wood when the die wood has a lot of empty space on it. To use this tool, click it. Balance knives will be added according to the values in the fields in the Status bar. Shown below is a layout before adding balance knives.



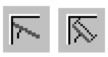
Shown below is the same layout after balance knives have been added.



Edit the properties of balance knives by using the fields on the Status bar. **Distance between**: is shown only for reference and may not be changed.

Balance knife operation is defined in the Die Press parameter set in Defaults. The length of the rule can be calculated according to a rule-saver formula, which minimizes the rule needed, or the rule can be set to extend to the design blank edge or the stripping blank edge. Balance knives are computed in the Rule Length dialog box as **Balance rule**. The **#BRLEN** expression calculates the length of the balance knives.

How to create stripping rules

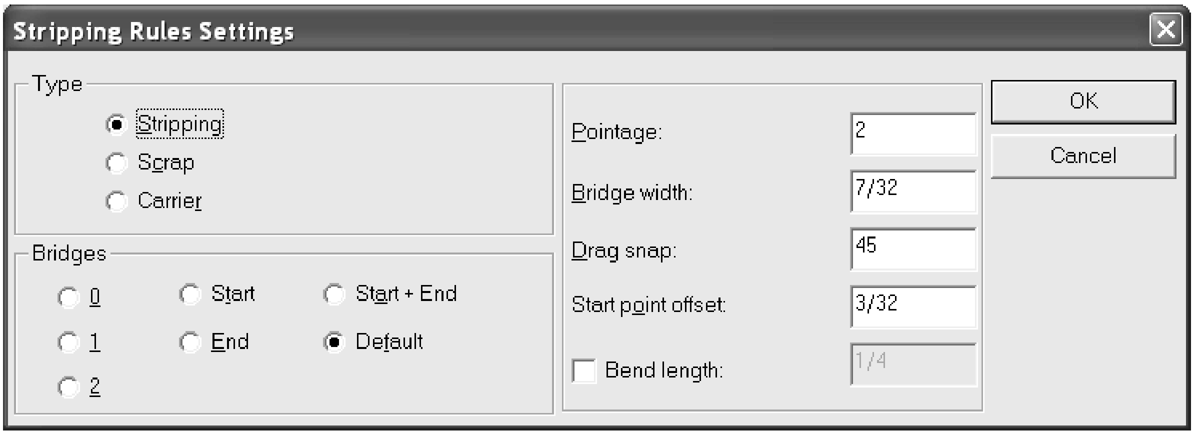
 Two tools create stripping rules: the **Line/Edge** tool and the **Angle/Offset** tool. The **Line/Edge** tool creates a stripping rule from the point you indicate in the direction that you set until it intersects with another line or an edge. The **Angle/Offset** tool creates a stripping rule with the angle and offset end-point that you specify. Hooks are automatically added to the stripping rules on the trailing edge. The stripping rules are automatically repeated throughout the layout unless Auto Repeat is turned off.

Note: You cannot create exterior stripping if there is a full lower board.

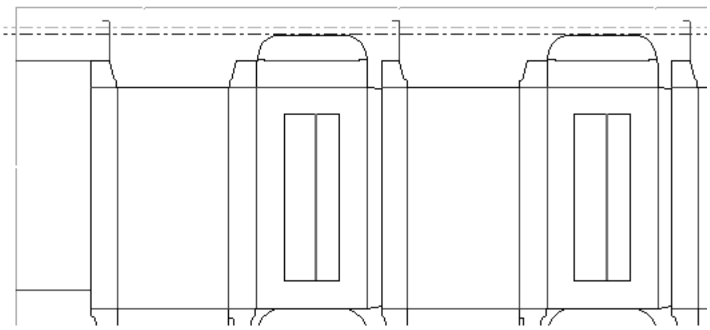
When either of these tools are active, options which control their use appear on the Status bar.

The three buttons on the left control the starting point of the stripping rule. The left most button is **Offset** from the end of a line, the middle button is **Center** of a line, and the rightmost button is **Anywhere**.

More options lets you change the stripping rule options in the Stripping Rules Settings dialog box, also accessible by clicking **Options** > **Stripping Rules**, as shown below.



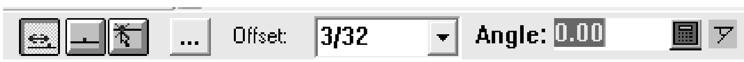
In the picture below, stripping rules extend from some of the dust flaps.



How to create carrier rules

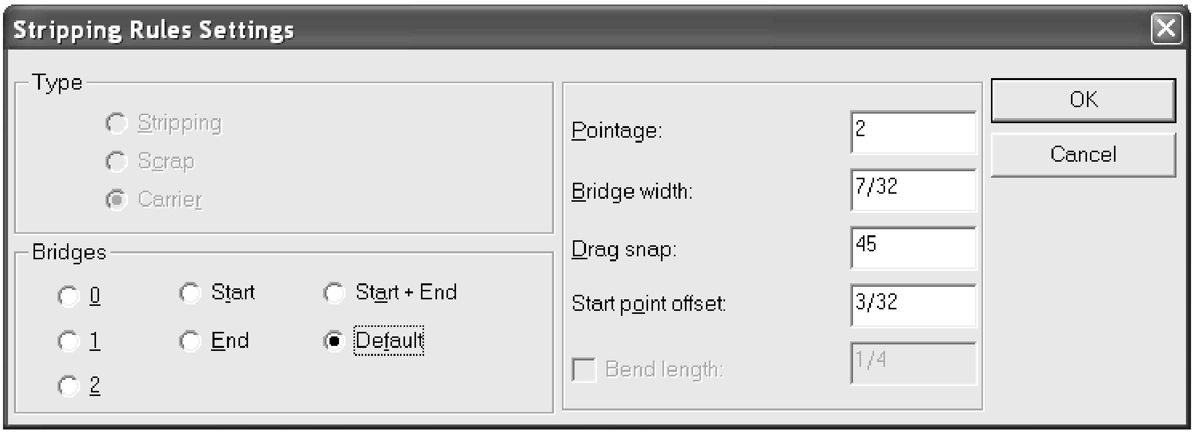
Use the **Create Carrier Rule** tool to create carrier rules. Carrier rules support gaps in the boxes through the press so that they do not snag and jam the press. To use this tool, click it, indicate the point at which the carrier rule starts, and then indicate the point where it ends. The flaps are added to carrier rules when you strip the carried area.

To add carrier rules, click the **Create Carrier Rule** tool and indicate the start and end-points for the carrier rule. The Status bar shows the same start options for carrier rules as it does for stripping rules.



The three buttons on the left control the starting point of the carrier rule. The left most button is **Offset** from the end of a line, the middle button is **Center** of a line, and the rightmost button is **Anywhere**.

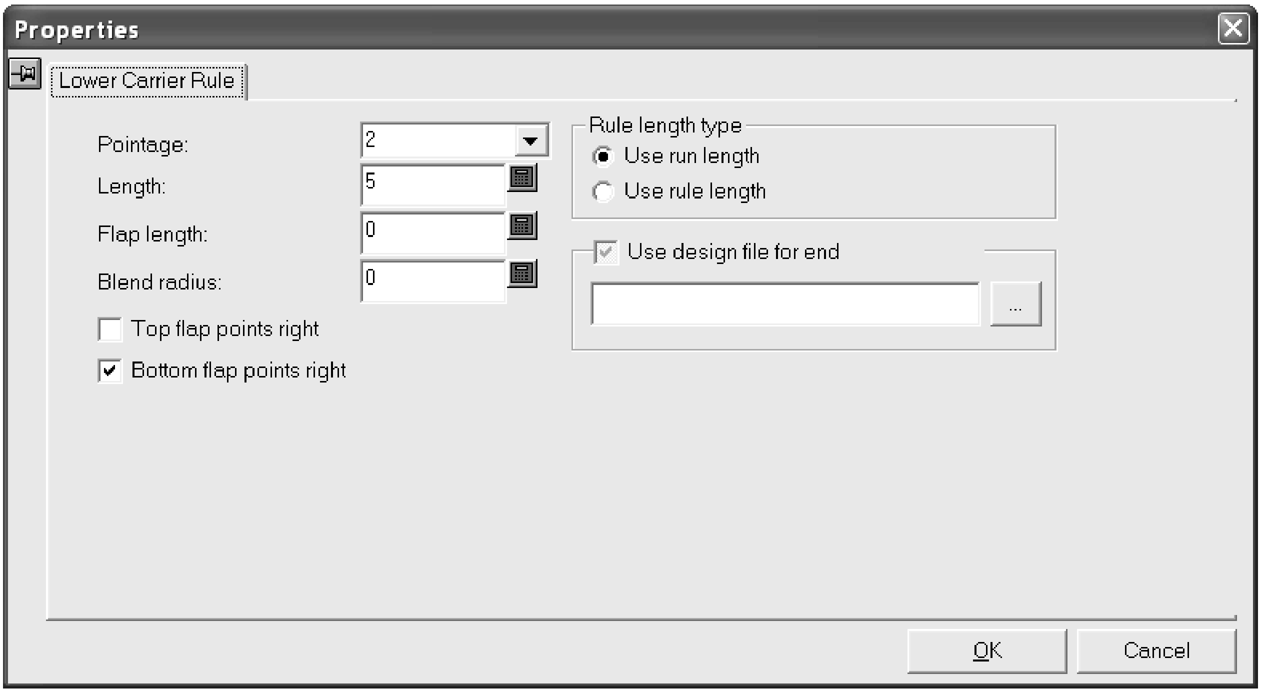
More options opens the Stripping Rules Settings dialog box. Since these are carrier rules, you cannot change the type of rule being created. Set the number of bridges, pointage, bridge width, the snap angle for the drag, and the start point offset as desired. Click **OK** when done to return to the tool.



Once the carried areas are stripped, there are two ways to change the properties of a carrier rule.

The first way to set carrier rule properties is to use the **Select Element** tool in the Stripping Rules layer to double-click the carrier rule. This changes how the stripping rule that is a carrier rule is displayed in the Stripping Rules layer. A Properties dialog box appears with rule properties as shown below.

The second way to set carrier rule properties is by changing to the Lower Stripping Board layer and using the **Select Element** tool to double-click the rule. This changes how the rules are manufactured in the stripping board itself. The Lower Rule Properties dialog box opens as shown below.



Change the pointage of the carrier rule by choosing a new value from the **Pointage** drop-down list box.

The length of the carrier rule shown in the **Length**: field depends on the option button chosen in the **Rule length type** group. **Use run length** shows the length of the rule without the flaps. **Use rule length** shows the length of all the rule.

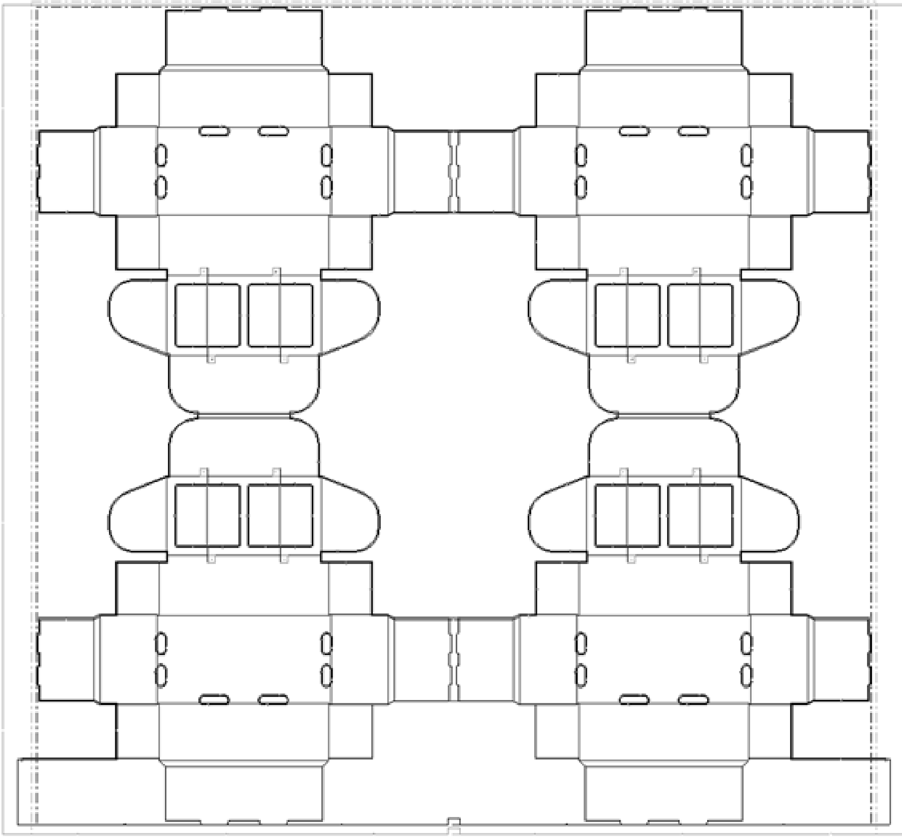
Use design file for end lets you specify a custom flap end.

You can design any end as desired. In Designer, start at the origin and work to the upper right using line type **Unknifed lower** for rule and line type **Lower registration hole** for any holes. The origin will be the end point of the end of the carrier rule. Do not use line type Annotation5. Save the workspace to `ServerLib` and specify it in Defaults in the Carrier Rules entry in Automatic Stripping in a manufacturing parameter set.

The **Top flap points right** and **Bottom flap points right** checkboxes change the direction of the top and bottom flaps, respectively. The flaps are applied when the layout is stripped.

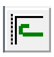
Change the properties as desired and click **OK**.

In the example below, one carrier rule was created manually. The rest of the designs have the carrier rule Auto-Repeated in them.



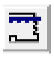
When an area containing a carrier rule is stripped, the rule between the edges of the hole is converted to an Annotation5 line type. This prevents the laser from burning a rule path in the material under the hole when the stripping board is created.

How to add hooks to stripping rules

 ArtiosCAD creates hooks on the ends of stripping rules by default. This can be changed, however, in the Hook Defaults dialog box accessed via the **Options** menu. If you have added stripping rules without hooks and later decide that you want hooks, use the **Make Hook** tool. To use this tool,

click it, and then indicate the stripping rule that you want to have a hook. If Auto-Repeat is on, the hook will be created on all congruent stripping rules.

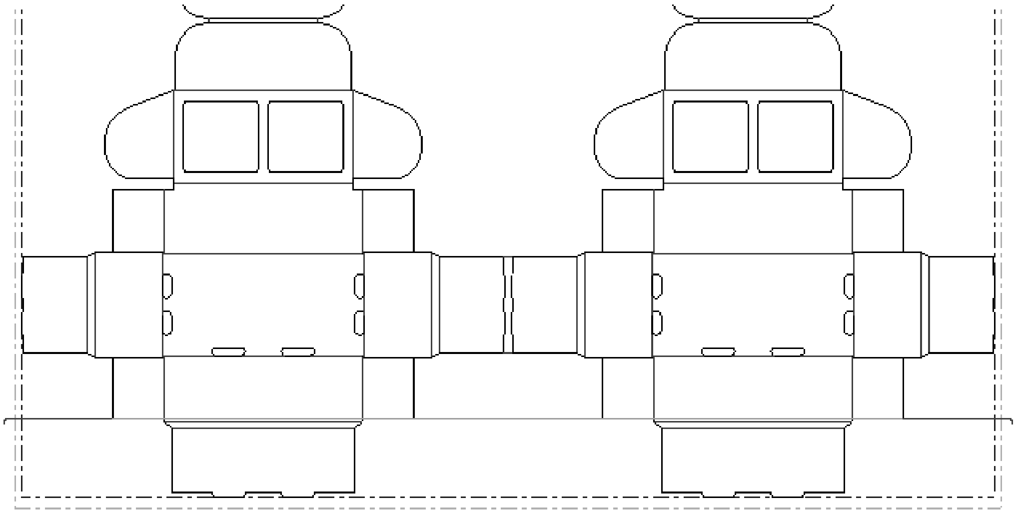
How to create a lower board mounting bar

 When making a stripping board that follows the layout edge, you must create a mounting bar for the lower stripping board. The mounting bar defines where to start stripping the edges. Use the **Add Lower Board Mounting Bar** tool to add the bar.

When this tool is clicked, options for it appear on the Status bar as shown below.

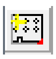
Strip between designs inserts stripping rules between single designs under the mounting bar. (They are only visible when the Lower stripping board layer is turned off.) **Offset** lets you specify the distance from the board edge to the mounting bar instead of setting it with drag.

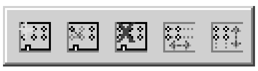
To use this tool, click it, set the options on the Status bar as desired, and either specify an offset for the bar location or use the drag to set it. A sample mounting bar is shown below.



This tool uses the stripping rules settings and is consistent with **Move Scrap Knives**. To control if there are hooks on the ends, click **Options** > **Stripping Rules** > **Bend Length**.

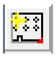
How to add and work with mounting holes

 The **Add MHP Holes** tool adds Mounting Hole Pattern holes to the manufacturing workspace. When held down, it activates the MHP Tools flyout toolbar.



These tools are available only after the die wood edge has been created.

Add MHP tool

 The **Add MHP Hole** tool copies the mounting hole pattern workspace associated with the selected die press into the manufacturing file. Mounting holes can be either **good** or **bad** - bad holes are those that are too close to other manufacturing elements as defined by the T-nut Distance default.

There are two options in the Status bar: **Show all holes** and **Show only good holes**. Good holes are green and bad holes are red. Choose what kind of holes to see, and then hold down either `SHIFT` or `CTRL` and click each mounting hole to use. Click **OK** when all desired holes have been selected, and the selected holes will be created.

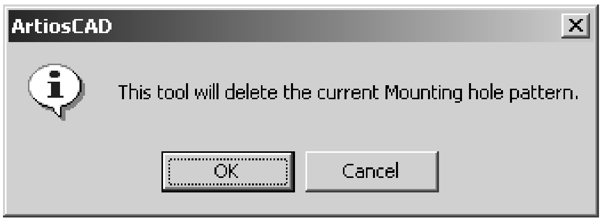
The side of the layout the pattern is applied to is defined in Defaults on the **Hole Patterns** page of the **Press Defaults** in the **Die press parameter sets** folder.

Select MHP tool

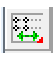
 The second button on the MHP Tools flyout toolbar activates the **Select MHP** tool. This tool redisplays the mounting hole pattern and lets you select more holes to add using the same procedure as the **Add MHP** tool. Pre-existing holes are automatically selected.

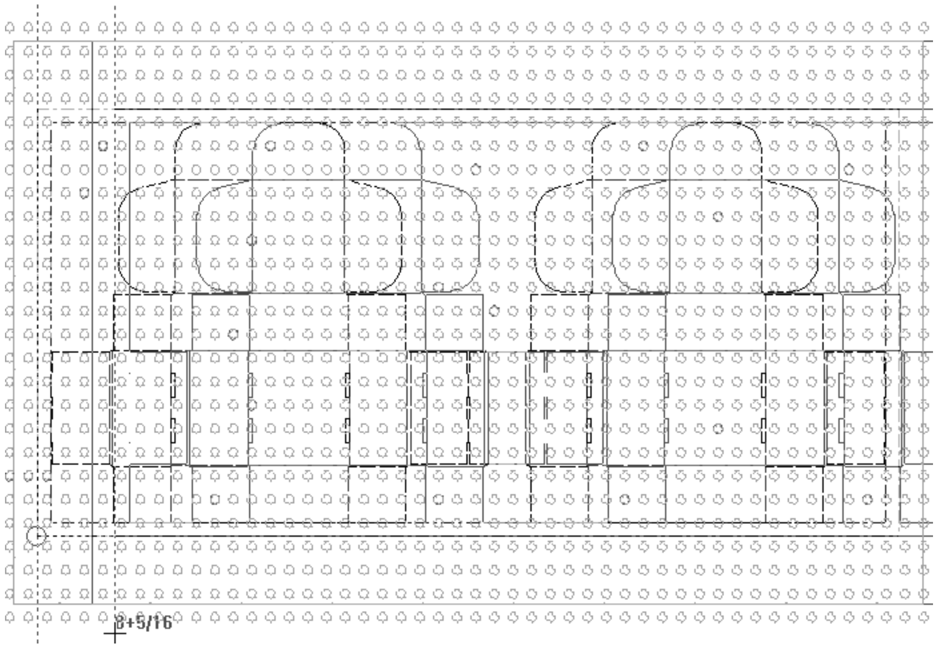
Delete All MHP tool

 The third button on the MHP Tools flyout toolbar activates the **Delete All MHP** tool. This tool deletes the current mounting holes. When clicked, ArtiosCAD prompts for confirmation; to delete the holes, click **OK**; to keep the holes, click **Cancel**.

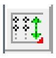


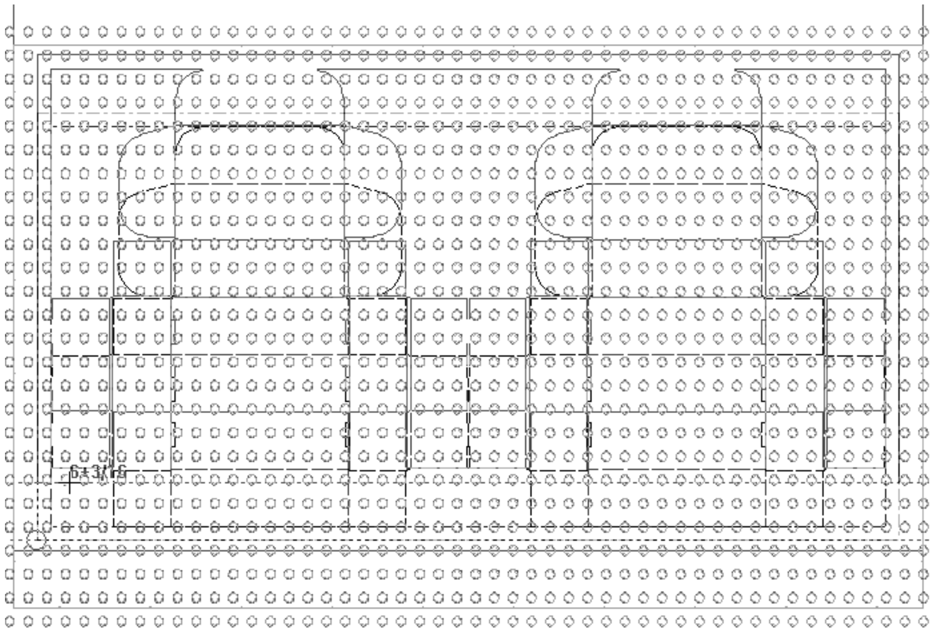
Adjust MHP Horizontally tool

 The fourth button on the MHP Holes flyout toolbar activates the **Adjust MHP Horizontally** tool. This tool shifts the layout horizontally in reference to the MHP to allow for more flexible die placement on the press. To use this tool, click it, click the pick-up point, drag the ghosted layout to its new position, and set the put-down point. Annotation lines for this tool appear only for rotary dies. The measurement indicates the offset from the original MHP placement point (either the Machine Reference Point or the bottom left of the dieboard).



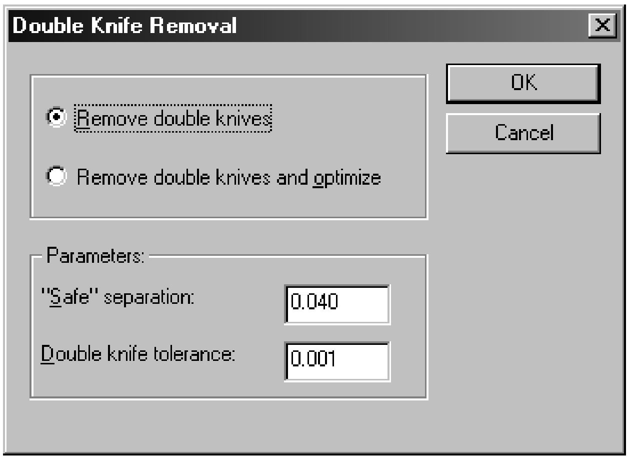
Adjust MHP Vertically tool

 The fifth button on the MHP Tools flyout toolbar activates the **Adjust MHP Vertically** tool. This tool shifts the layout vertically with respect to the MHP to allow for more flexible positioning of the die. To use this tool, click it, select a pickup point, drag the ghosted layout to its new position, and set the put-down point. Annotation lines for this tool appear only for rotary dies. The measurement indicates the offset from the original MHP reference point (either the Machine Reference Point or the bottom left of the dieboard).

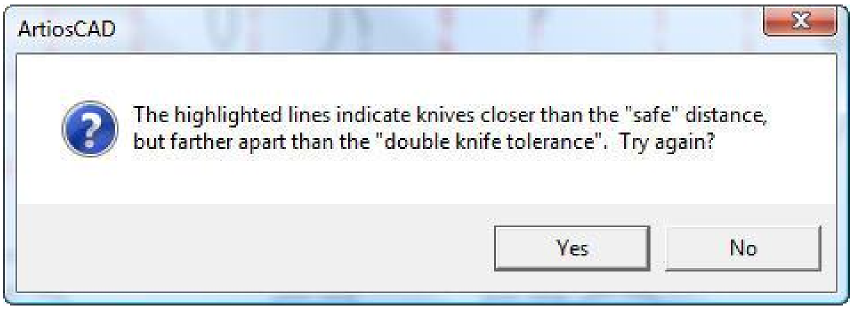


How to remove double knives

 Double knives are created when cartons abut each other. To remove them, click the **Double Knife Removal** tool. In the Double Knife Removal dialog box, choose the option you'd like to perform.



In the **Parameters** group, **Double knife tolerance** is the distance in current units that rules must be apart from each other to not be considered double knives. **"Safe" separation** is the minimum distance two lines must be from each other to be ruled individually. If you have lines that are closer than the "safe" separation distance but further than the Double Knife Tolerance, the following dialog box appears when you attempt double knife removal:

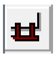


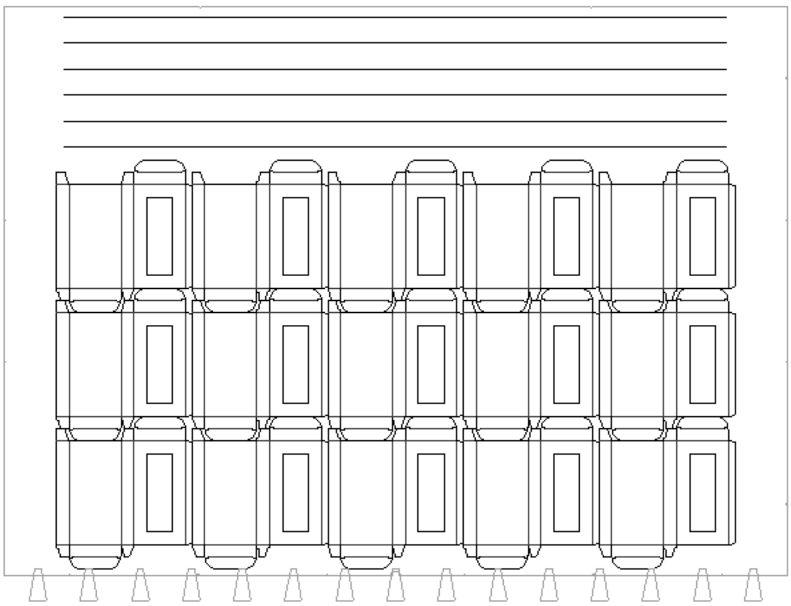
Click **Yes** to return to the Double Knife Removal dialog box or click **No** to return to ArtiosCAD to fix the error.

Nicks may move if double knives are removed after adding nicks.

Double knife tolerance defaults are stored in **Options** > **Defaults** > **Shared Defaults** > **Startup defaults** > **Double Knife Removal**.

How to show gripper fingers

 Click the **Add Gripper** tool to show gripper fingers for the die press.



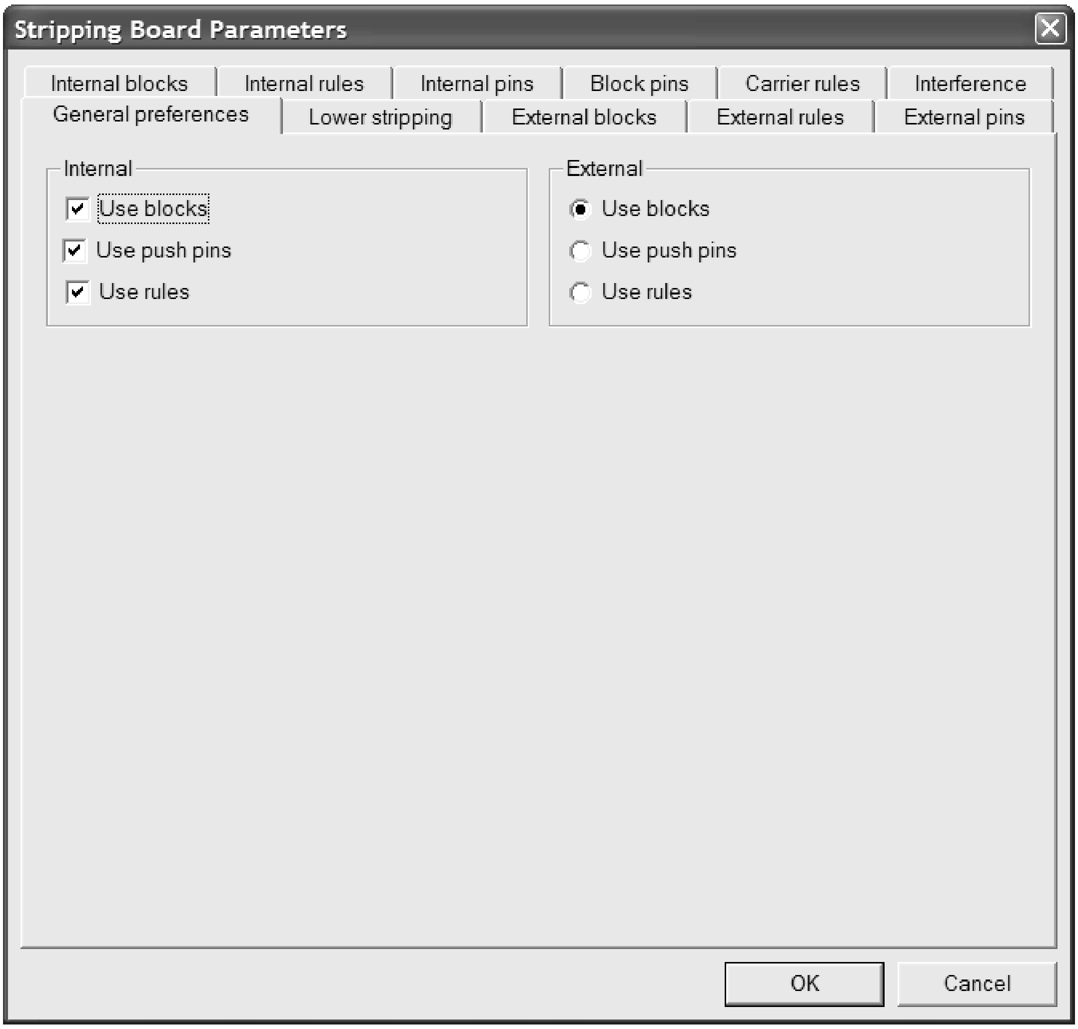
How to work with bridges, tack bridges, and nicks

Bridges, tack bridges, and nicks in Manufacturing use the same tools as in Single Design; see the *Designer* chapter for more information.

Nicks may move if they are added before removing double knives.

How to create internal and external stripping

Once a mounting bar or stripping rules have been added to the layout, you can add interior and exterior stripping just by clicking inside the area to strip. To ensure that the settings are correct, check the Stripping Board options by clicking **Stripping Boards** on the **Options** menu and reviewing the settings on the tabs. The default settings are set in the Automatic Stripping catalog of a manufacturing parameter set in Defaults.



 If the settings on the tabs in the Stripping Board Parameters dialog box are acceptable, begin stripping the sheet using the **Strip Area** tool. This tool can create holes in the upper and lower stripping boards, add blocks, pins, and rules, make a guillotine, and add support pins. Blocks, pins, and rules are added in that order.

The option buttons on the Status bar determine what stripping tools are automatically created. The picture below is of the option buttons on the Status bar.

Lower, **Upper**, **Internal**, and **External** control which areas of stripping are created; select them as desired.

The  button (More Options) opens the Stripping Board Parameters dialog box as shown previously. **Recalculate all** deletes and recreates the existing stripping board edges and internal and external stripping.

Note: On the Lower Stripping tab of the Stripping Board Parameters dialog box, if you make the value in the **Offset below wing:** field different from the **Offset:** field with Auto-Repeat on, the repeated block has the wrong offset. The strip areas are considered to be congruent because they have the same shape, even though the blocks have different offsets.

Click inside an area of waste to strip it. Any congruent areas will be stripped according to current Auto-Repeat settings. You cannot strip areas that are part of a design unless they are windows or cutouts. Or, to perform all the stripping at once, activate the tool and click **Recalculate All**.

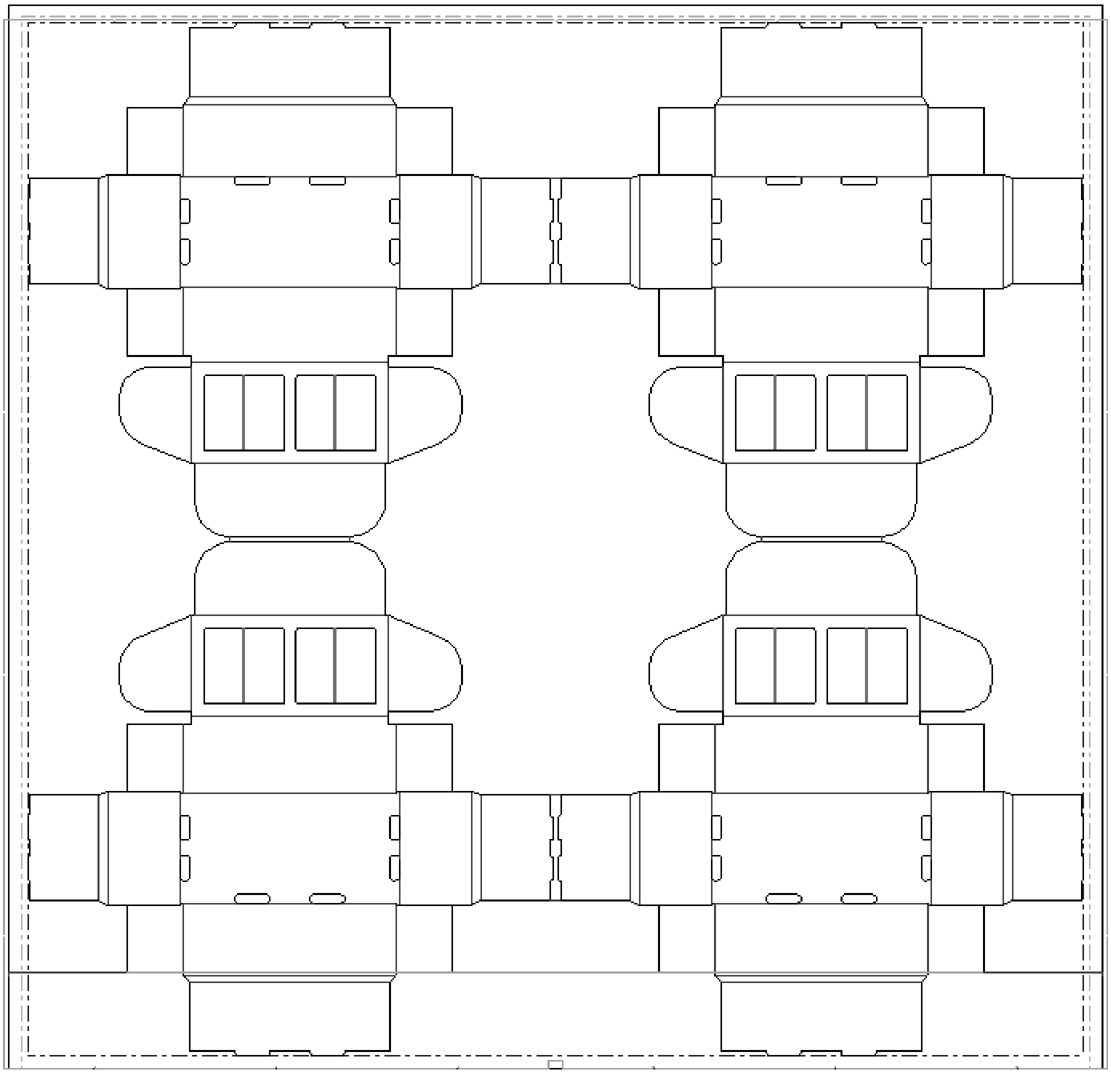
Click in the waste outside the design edge between stripping rules to strip the external waste. If slots in the external waste are divided from the exterior waste by scrap knives, carrier rules, stripping rules, or are narrower than the slot width parameter, they are stripped as if they were interior holes. Exterior blocks made this way follow stripping rules but ignore scrap knives.

Note: When a stripped area such as a window is split with a stripping rule or carrier rule, **Recalculate All** creates upper stripping components in both halves. When creating tooling by clicking in each individual area, particularly when the area is both symmetrical and symmetrically split, it is possible that the upper components are created only in the area in which the cursor was clicked and are not Auto-Repeated.To add upper components to the other side, deselect the **Lower** checkbox on the Status bar and click in the other side. This accommodates stripping of areas containing mixtures of stripping and scrap knives on dies with large waste areas towards the front of a die.

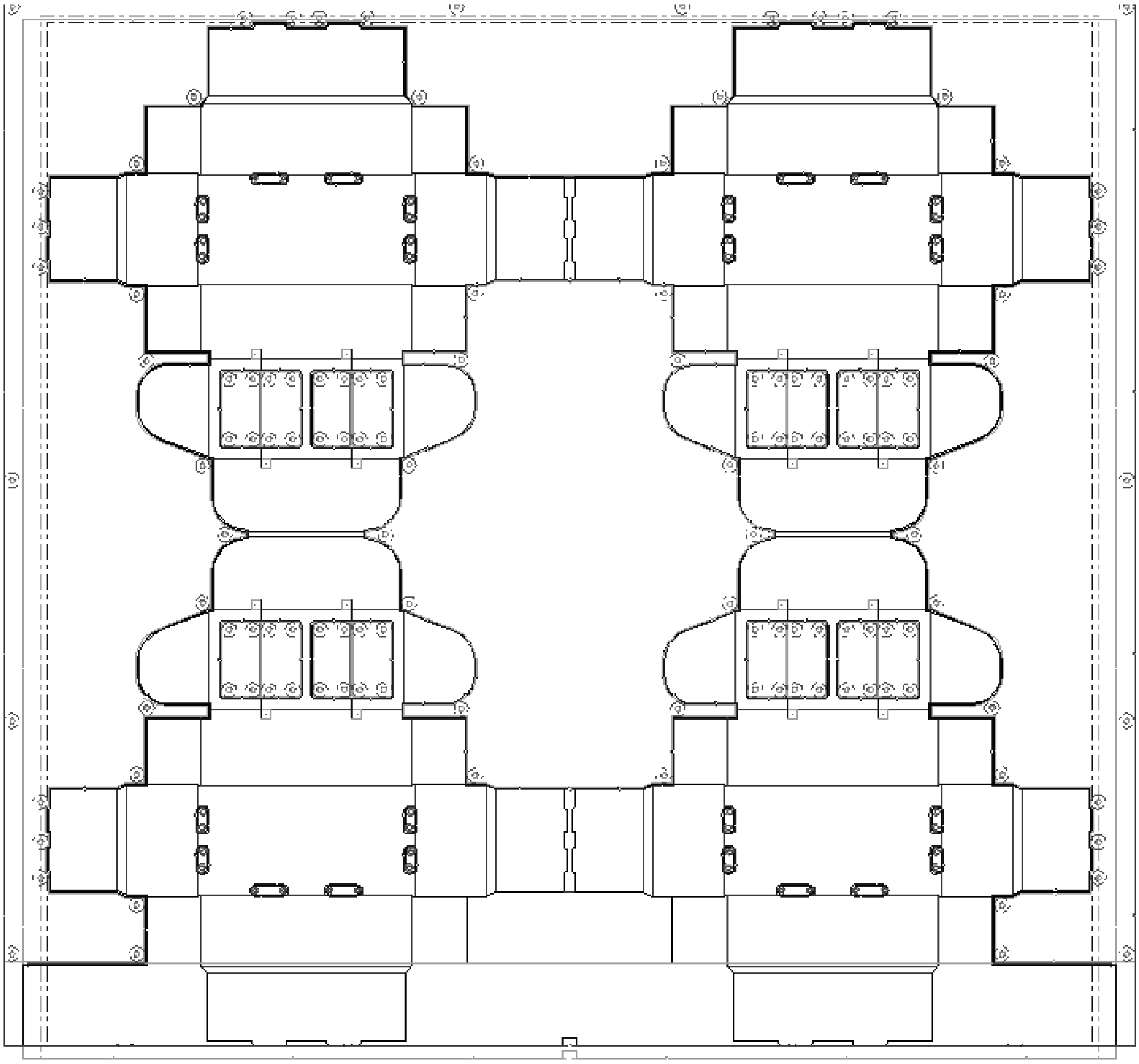
Note:

There is a similar procedure to add different upper parts to a hole split by multiple carrier rules. Strip the hole with **Strip Area**, making sure that **Lower** is checked on the Status bar. Turn **Lower** off after you have stripped the hole. Change the stripping parameters as desired by clicking Properties (**...**) on the Status bar and clicking **OK** when done. Then click inside the areas of the hole to strip with those parameters. This will add upper parts to this part of the hole and leave the existing parts in other parts of the hole. Normally ArtiosCAD will not put small, thin blocks in thin areas where there is no room for support pins. To include small, thin blocks, change the minimum block area and slot width on the Internal blocks tab of the Stripping Board Parameters dialog box.

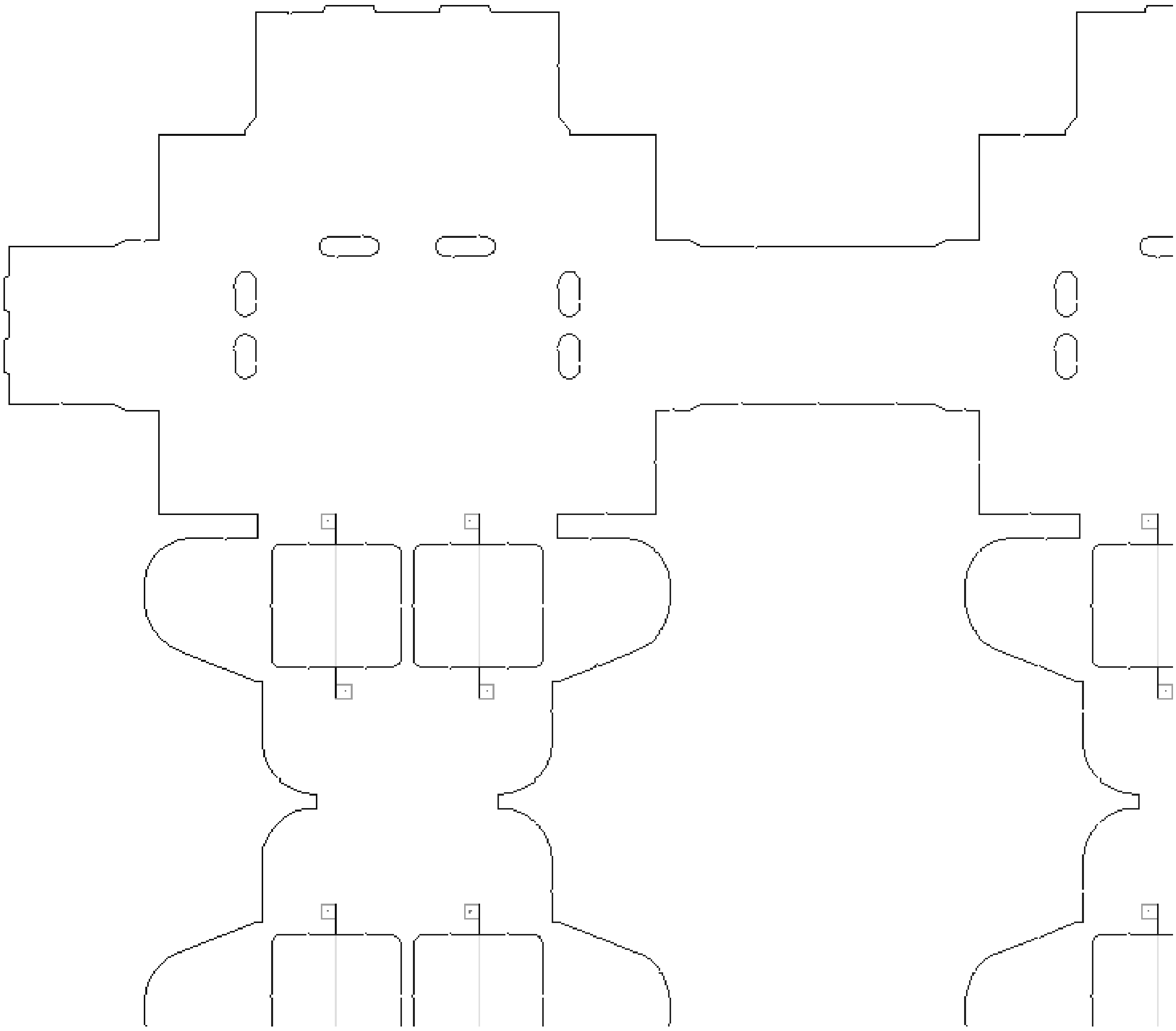
Shown below is a layout before using **Strip Area**.



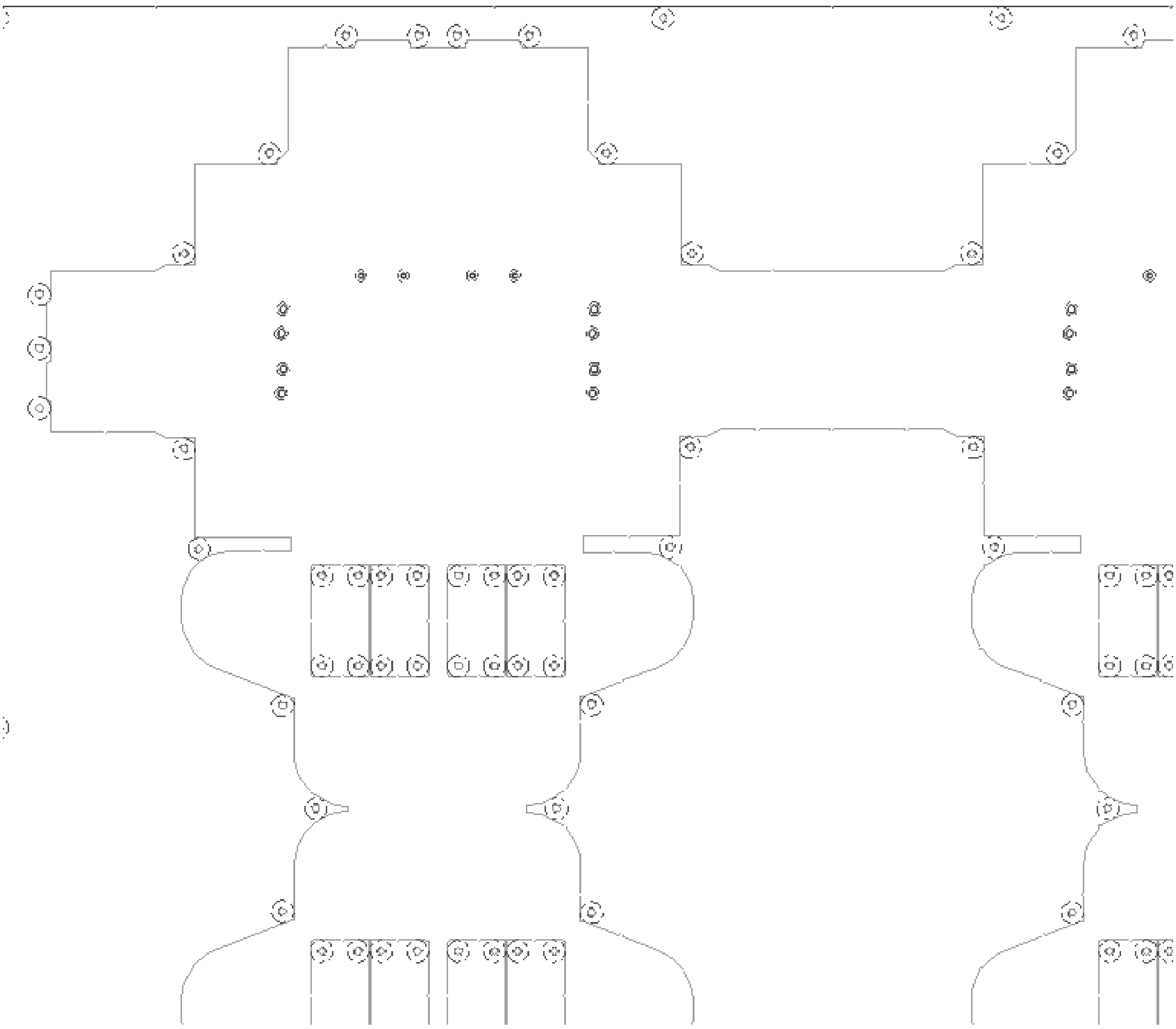
Shown below is the same layout after clicking the tool and then clicking **Recalculate All**.



Shown below is a close-up of the lower stripping board.



Shown below is a close-up of the upper stripping board.



How to repeat changes in stripping to other areas

Use the **Repeat Strip Area** tool on the Manufacturing toolbar to repeat manual changes to a stripping area to other congruent stripping areas. A stripping area can be a hole, a block, or the external stripping area. For instance, if you add a rule to a stripping area using the Line tool, that change is not repeated because the Line tool does not use Auto-Repeat. The Repeat Strip Area tool supports Auto-Repeat; if it is on, stripping areas congruent to the destination stripping area are automatically changed as well.

To use the Repeat Strip Area tool, do the following:

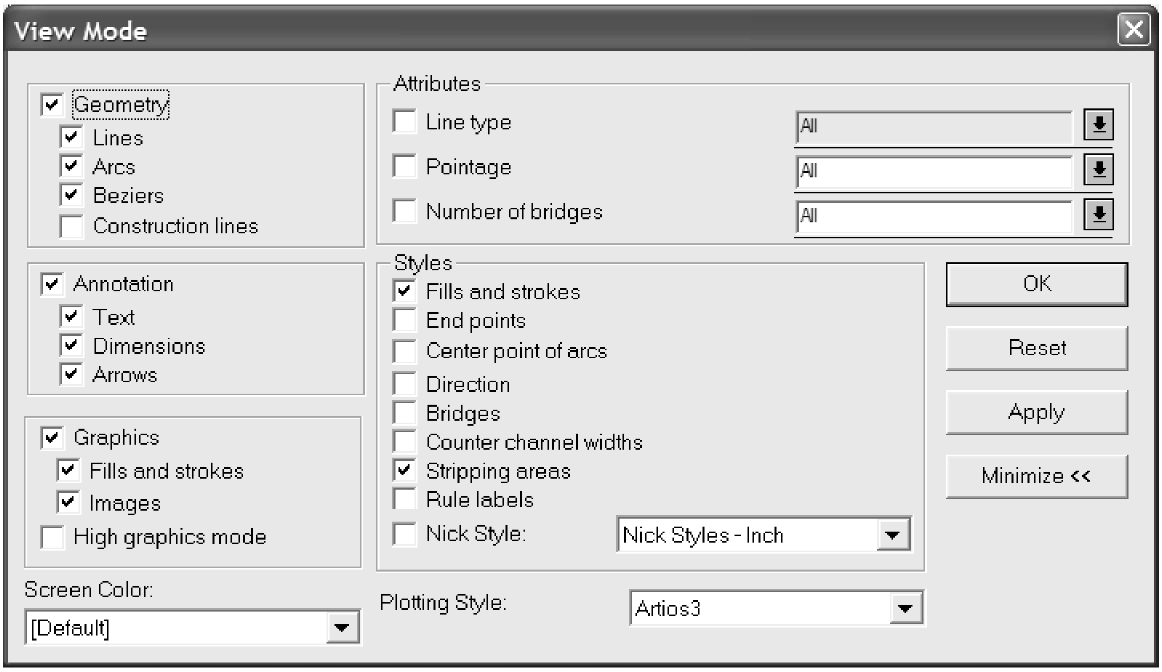
1. Change the stripping area as desired.
2. Click **Repeat Strip Area**.
3. On the status bar, choose the boards in which to repeat the stripping area.
4. Click inside the stripping area to repeat (the source stripping area).
5. Click inside the stripping area to be changed (the destination stripping area).

6. Repeat step 5 as desired in other destination stripping areas. The tool remains active until another tool is selected.

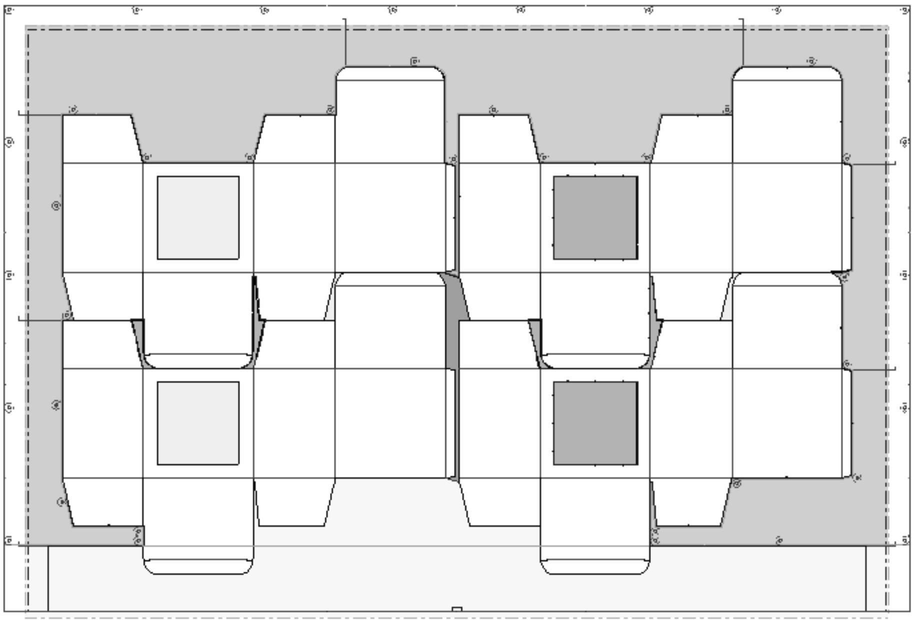
Stripping components must have horizontal mirror symmetry in their alignment lines for them to use mirror repeating.

Color-coded stripping areas

If you turn on the **Stripping areas** checkbox in the View Mode dialog box in Manufacturing, different colors are shown to differentiate between the different stripping areas in the layout.



Shown below is a manufacturing file that has color-coded stripping turned on. (It will look much better on your screen as opposed to this grayscale print.)



Green indicates a waste area with a lower hole and upper components. **Blue** indicates a waste area with a lower hole only. **Light blue** indicates a waste area with no lower hole. **Yellow** indicates the waste is removed after the stripping station. **Pink** indicates ArtiosCAD could not find a consistent pattern to the waste and therefore could not decide its type.

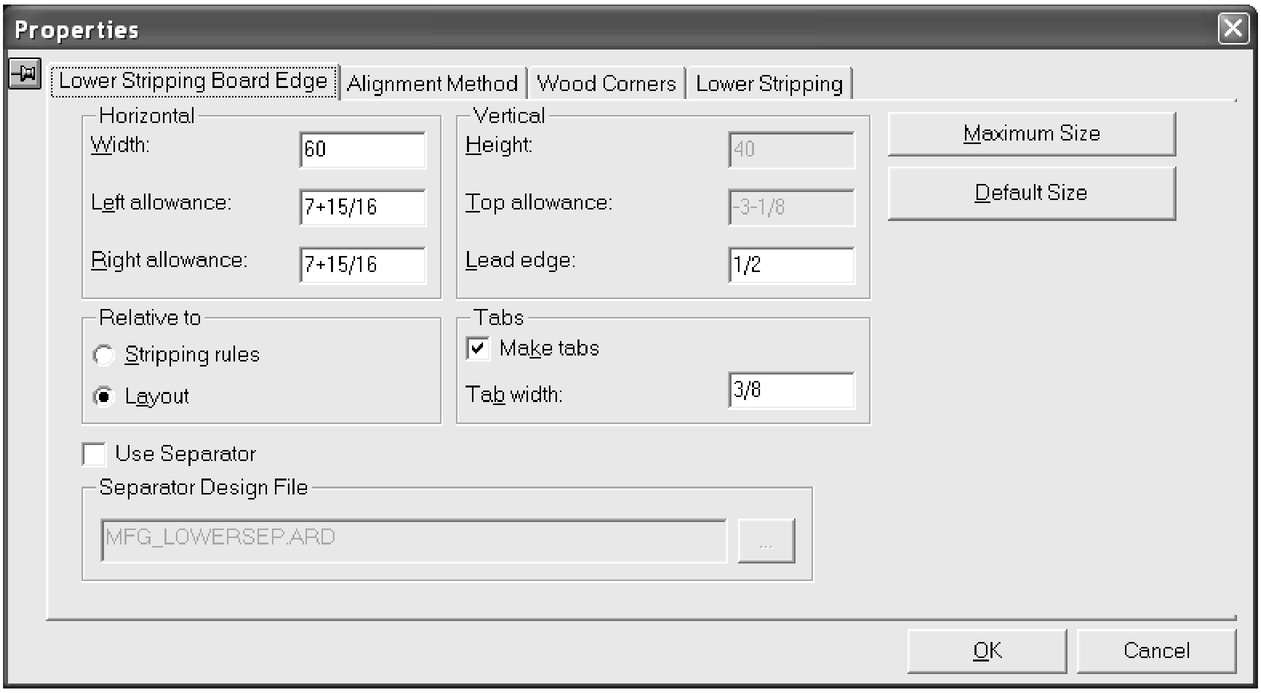
Changing stripping boards

Use the tools on the Stripping Board toolbar to modify stripping boards.

You can also use **stripping components** to alter stripping boards.

How to change the edge of the lower stripping board

To change the edge of the stripping board, double-click it with the **Select Element** tool, adjust the fields on the different tabs as desired, and click **OK**.

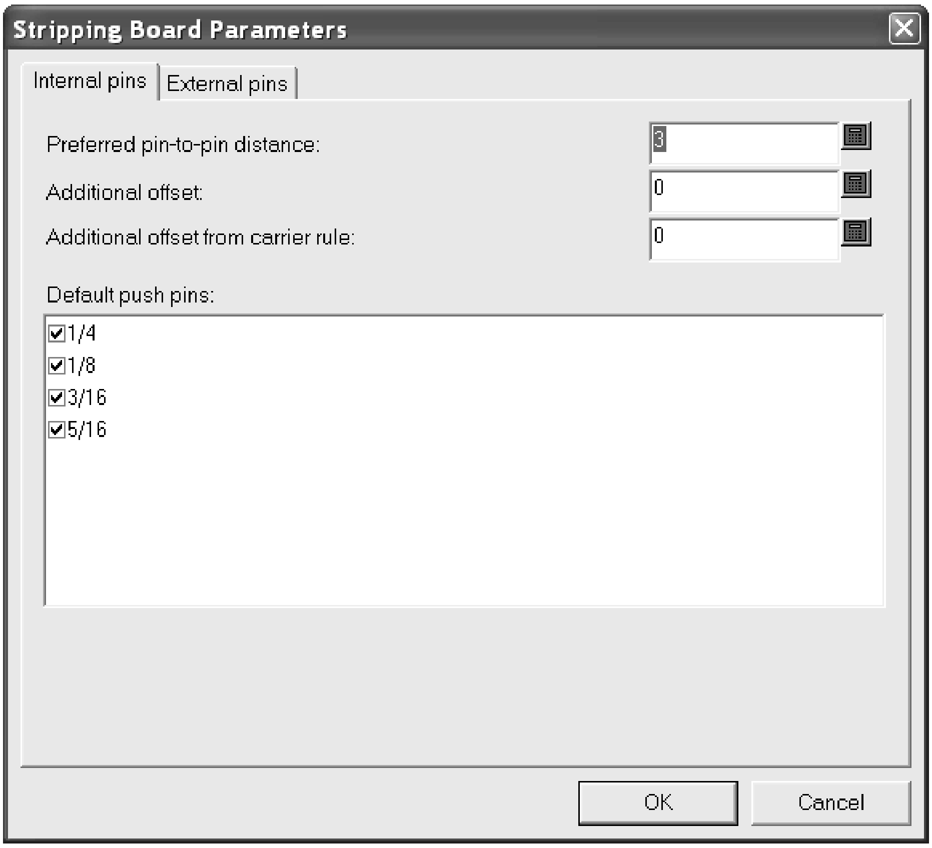


How to add an upper push pin

When you use the **Strip Area** tool to create both the upper and lower stripping boards, ArtiosCAD will automatically add pins, rules, and blocks. However, if you use the **Strip Area** tool to create only the lower board, you can manually add pins, rules, and blocks. The lower stripping board must contain holes before you can add upper push pins.

 The **Upper Push Pin** tool adds a pin to the upper stripping board. When you click the tool, a Pin Size drop-down list box and a **...** button (More Options) appear on the Status bar.

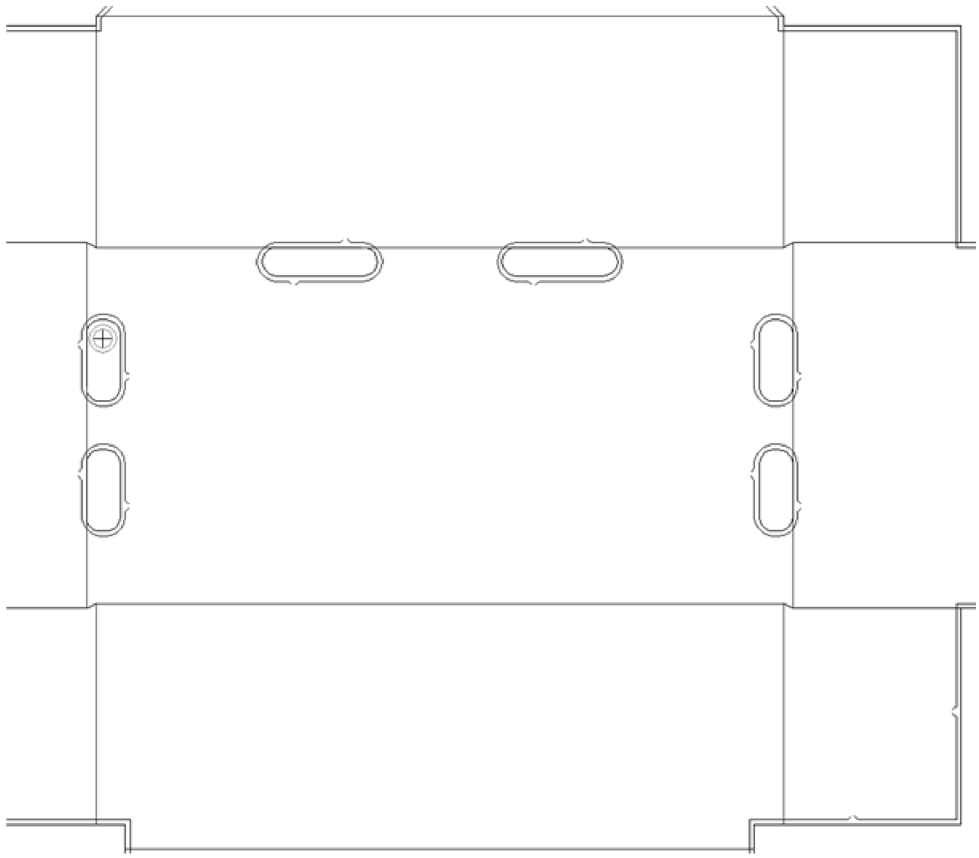
Clicking **More Options** opens a Stripping Boards Parameters dialog box with just the tabs for internal pins and external pins. These options are based on the settings in Automatic Stripping catalog for the manufacturing parameter set in Defaults. Change them as desired and click **OK**.



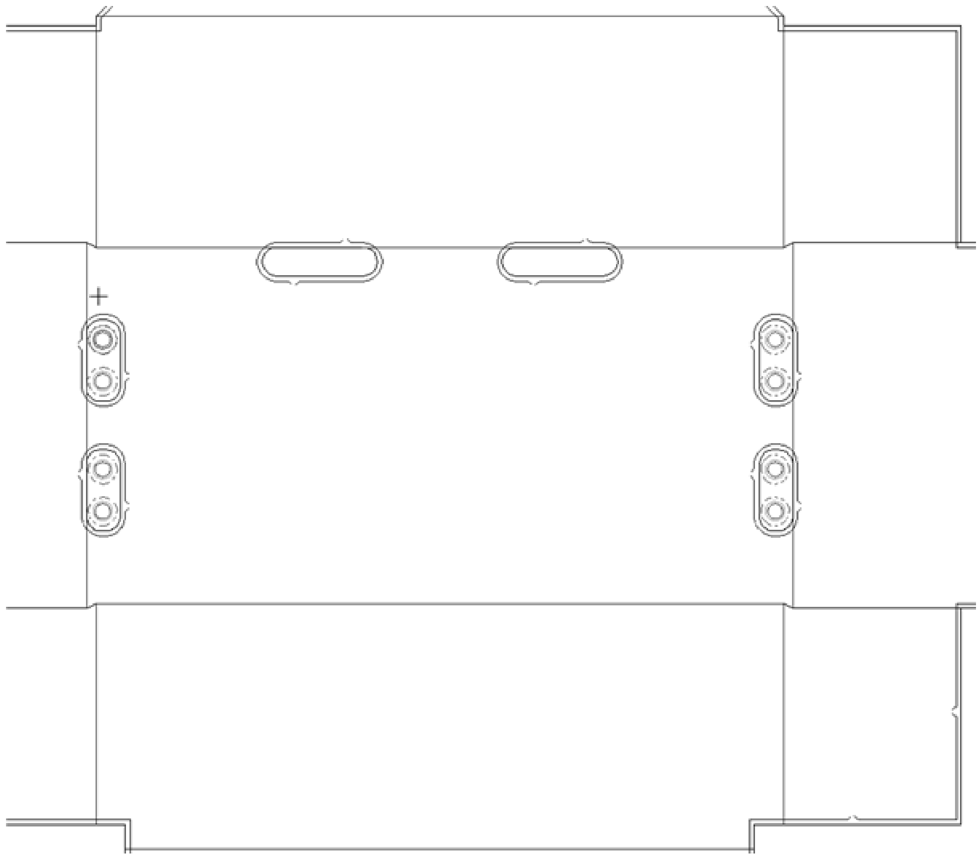
To add an upper push pin, do the following:

1. Make sure there are holes in the lower stripping board. Use the **Strip Area** tool if necessary.
2. Click **Upper Push Pin**.
3. Select the desired pin size from the **Pin size**: drop-down list box on the Status bar.
4. Use the drag to set the location for the pin and click to place the pin. Green drag indicates the pin is snapped to the outline of the hole; blue drag indicates that it is freehand, and red dashed drag represents an unavailable area for placement. If Auto-Repeat is on, the pin will be repeated accordingly. Additional offsets may be applied according to the options set in either Defaults or the Stripping Boards Parameters dialog box.

   Shown below is adding a pin at the upper left corner of a hole.



Shown below is the added pin which was Auto-Repeated.

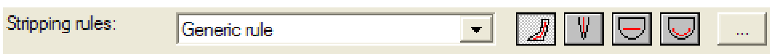


5. Repeat as desired to add more pins.

You can design custom push pins as geometry macros of type **Push pin component**. They must be circular.

How to add an upper rule

 The Upper Rule tool adds a piece of rule to the upper stripping board. As with the Upper Push Pin tool, you must click inside a hole in the lower board to add the rule. When you click the tool, a **Stripping Rules** drop-down list box, four snap positioning option buttons, and a **...** button (More Options) appear on the Status bar.

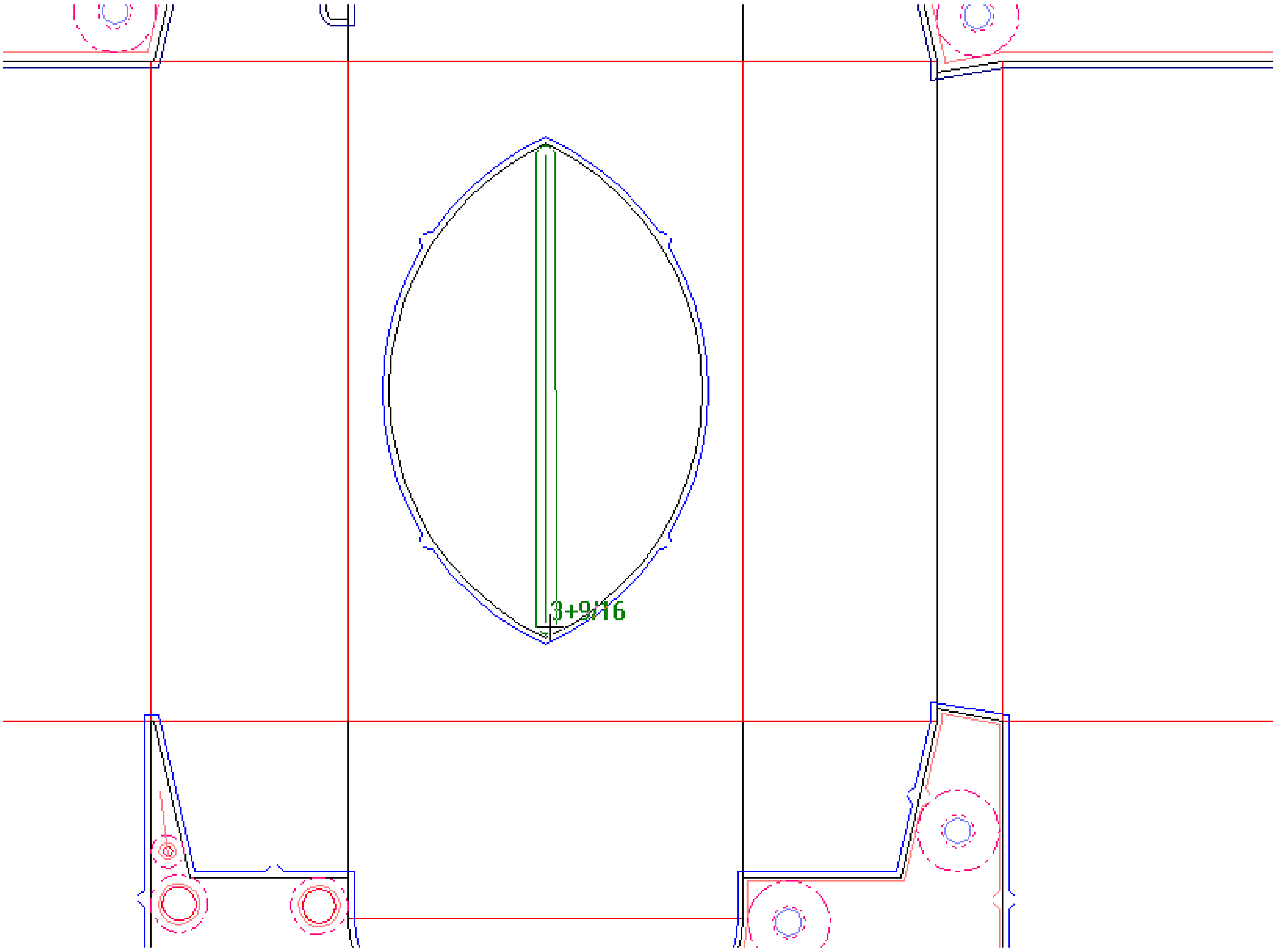


On the **Stripping rules**: drop down list-box are Generic rule, which is just plain straight rule, and three geometry macros that offer special constructions: 20mm wave rule, 8mm Right rule (a stripping component), and Variable rule that has a 90-degree angle at its end. Your system may have different geometry macros available depending on how the system has been customized.

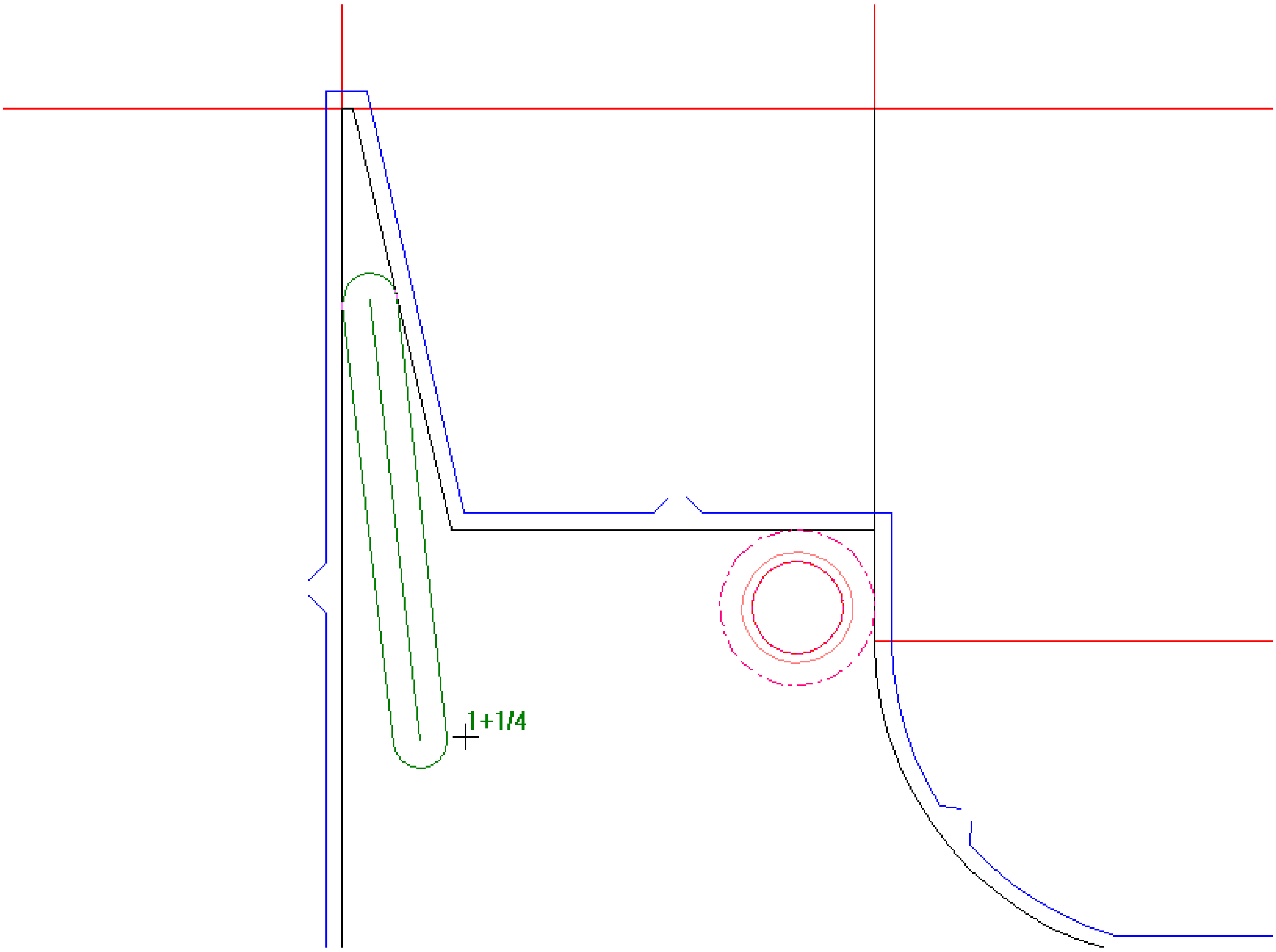
The four snap options on the Status bar after the Stripping rules: drop-down list box are listed below.

 **Dynamic Rule** lets you place rule and use the color of the drag for feedback. The drag for the guide lines changes color based on the intended position of the rule. Green drag indicates the item is snapped to the edge of a hole; blue drag indicates it is being placed freehand; red drag indicates it cannot be placed at that position.

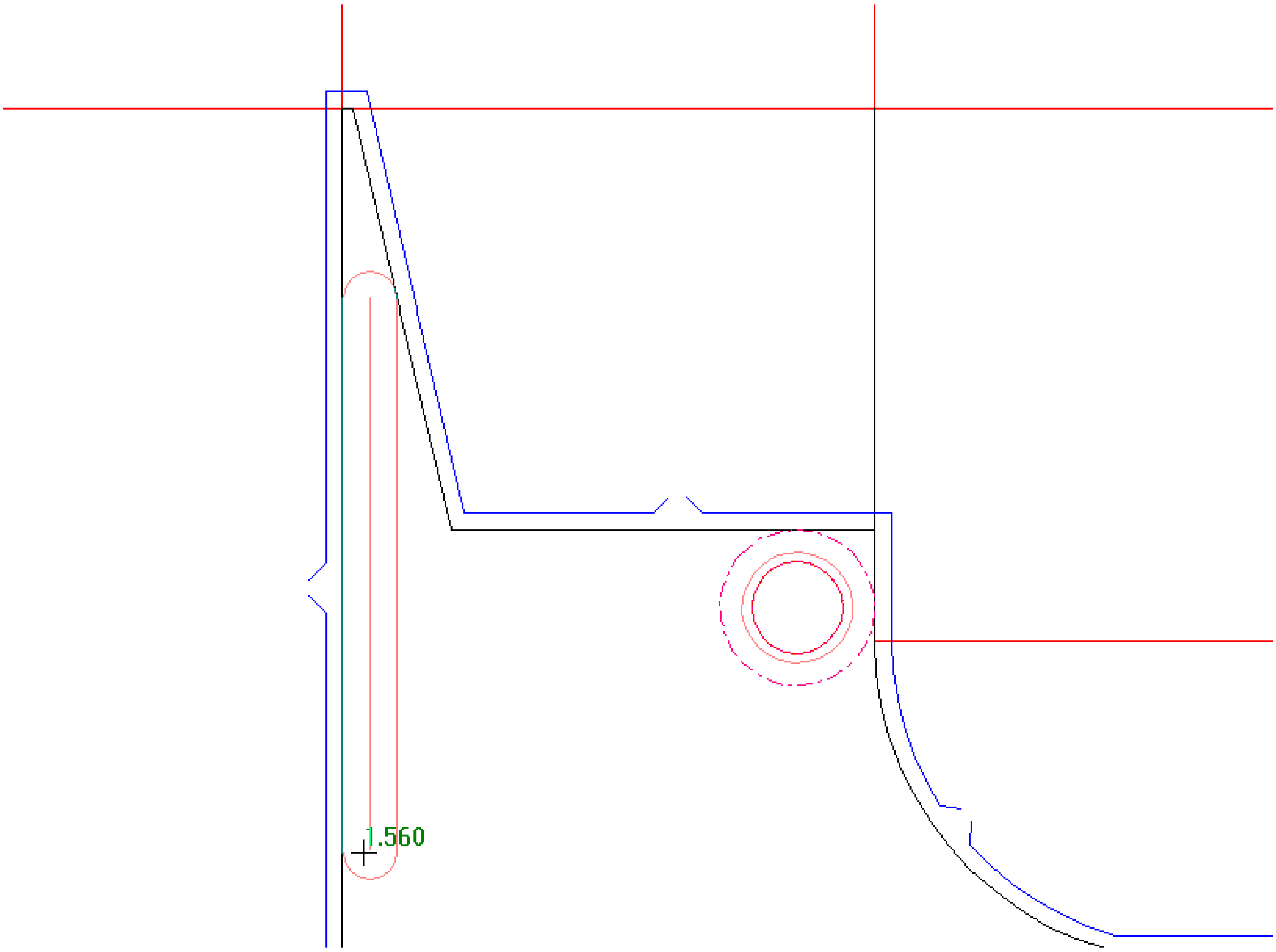
This option works with all the stripping rule types available in the drop-down list box.



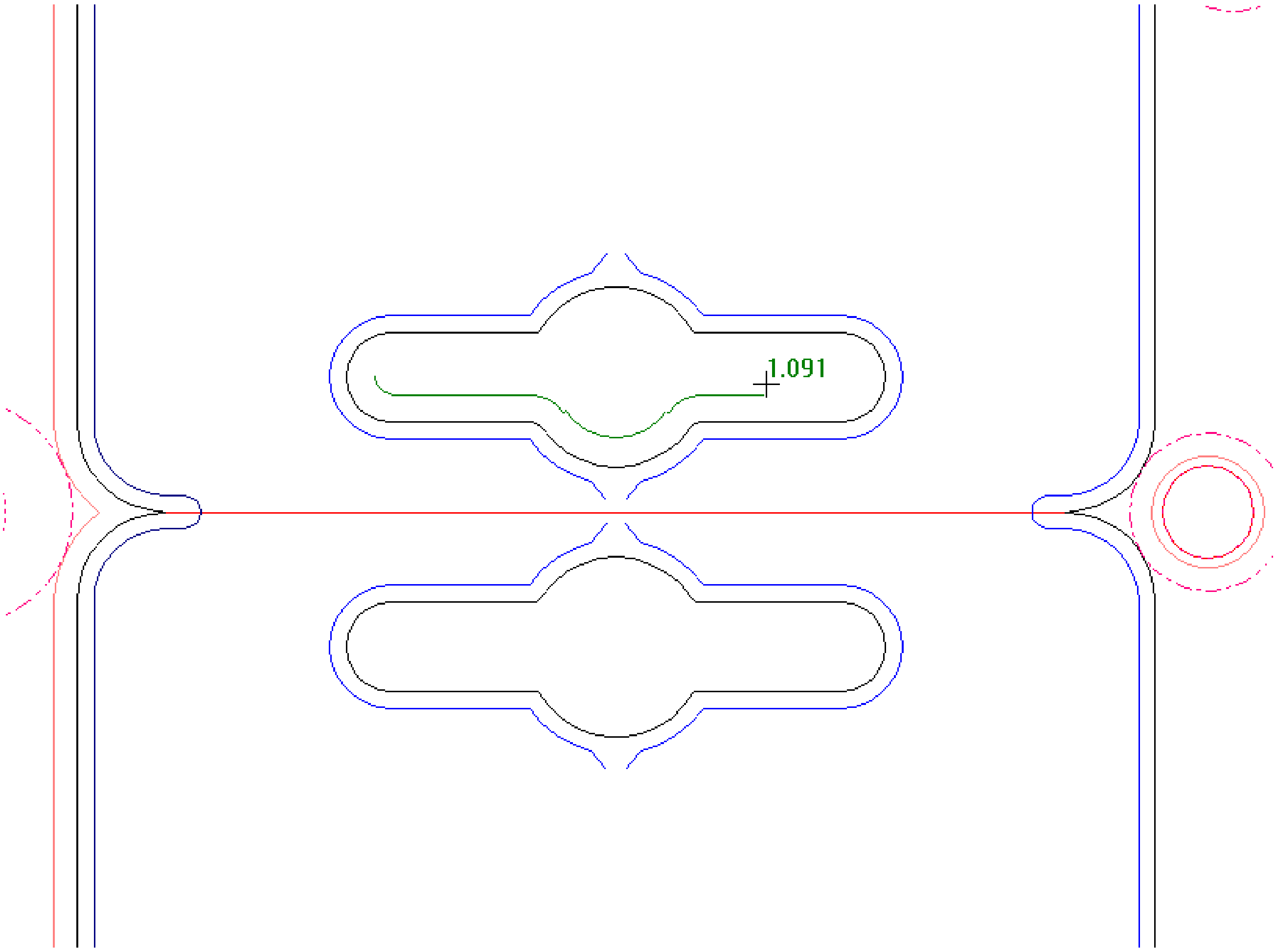
Snap to Middle Rule automatically snaps the rule in the middle of a slot. This option works with all the stripping rule types available in the drop-down list box.



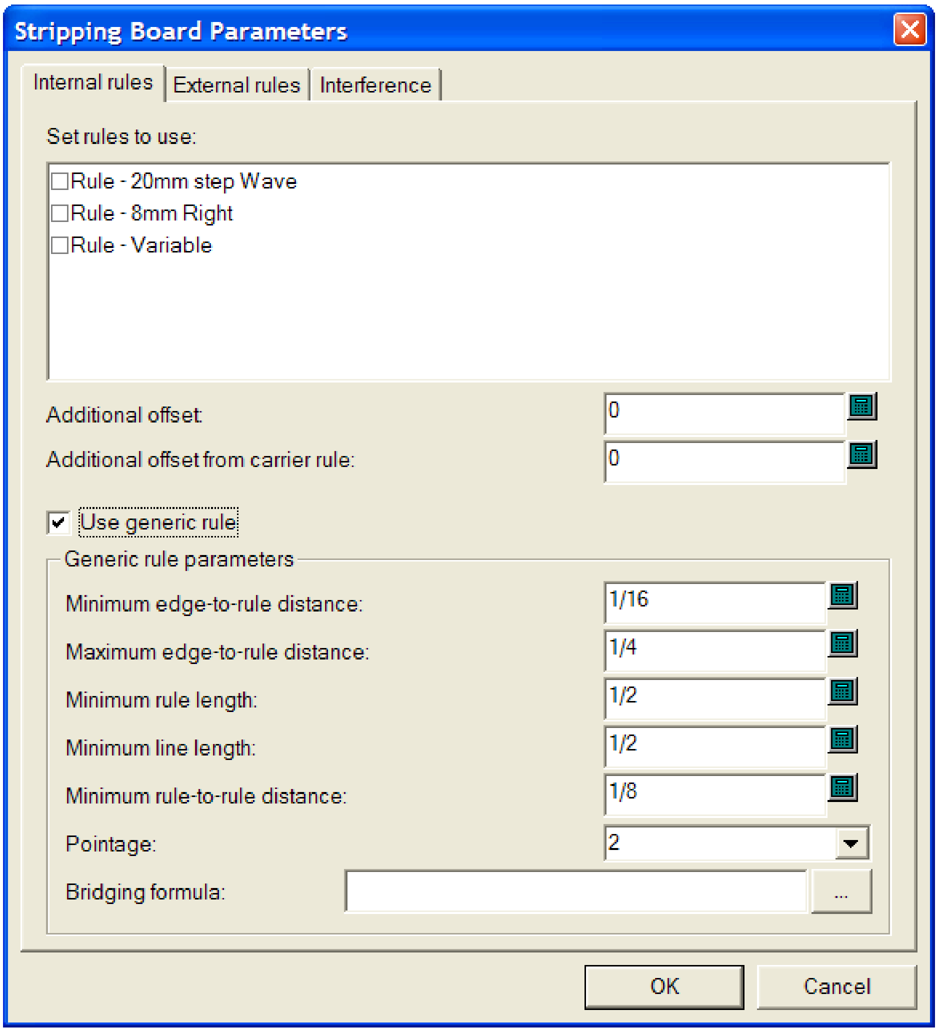
 **Generic Straight Rule** changes the type of rule in the drop-down list box to **Generic rule** and prompts you to indicate the start and end points of a straight piece of rule.



 **Generic Follow Rule** changes the type of rule in the drop-down list box to **Generic rule** and prompts you to indicate the start and end points of a piece of rule that follows the shape of the lower hole.



Clicking  **More Options** opens the Stripping Boards Parameters dialog box with the tabs for internal rules, external rules, and interference. These options are based on the settings in the Automatic Stripping catalog for the manufacturing parameter set in Defaults. Change them as desired and click **OK**.



To add a piece of upper rule, do the following:

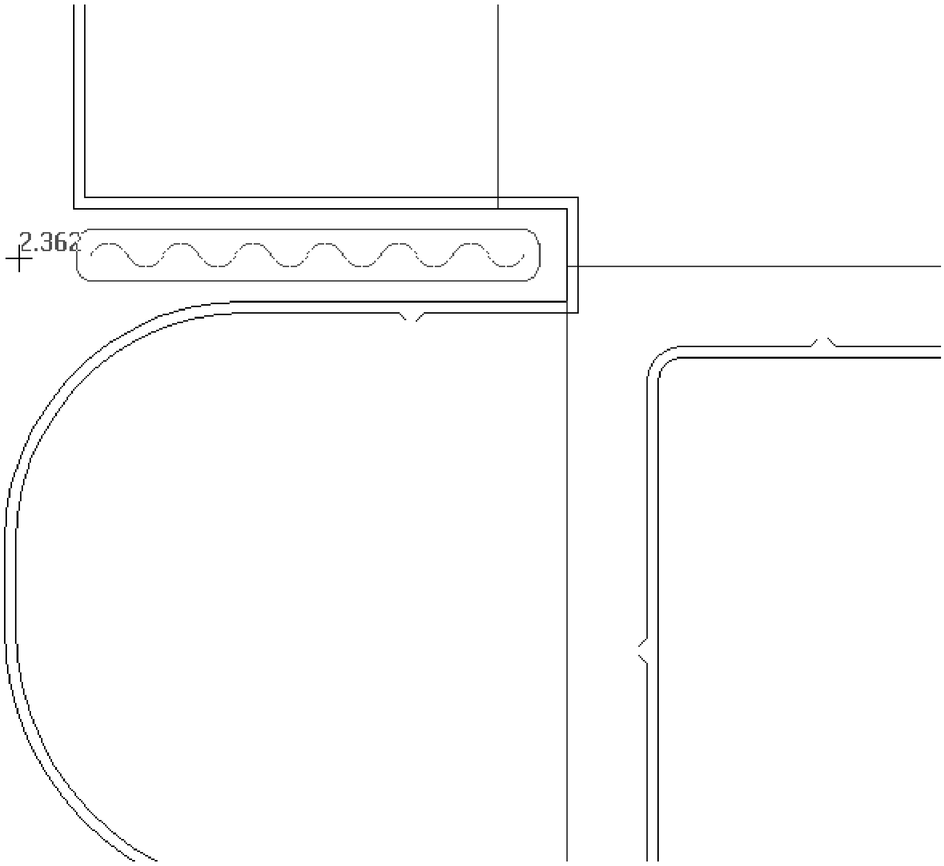
1. Make sure there are holes in the lower stripping board. Use the Strip Area tool if necessary.
2. Click the **Upper Rule** tool.
3. Select the type of rule to add from the **Stripping Rules**: drop down list-box.
4. Click the appropriate snap placement button. **Dynamic Rule** is selected by default.
5. Place the rule. If the rule selected on the drop-down list box is of variable length, click to set the start point, drag to extend it, and then click to set the end point.

   If the rule selected on the drop-down list box is not variable, place it as desired.

   Green drag indicates the item is snapped to the edge of a hole; blue drag indicates it is being placed freehand; red drag indicates it cannot be placed at that position.

   When **Snap to Middle Rule** is the current placement method and the end point of the rule being added is a piece of existing generic rule, the new rule is intersected with the piece of existing rule.

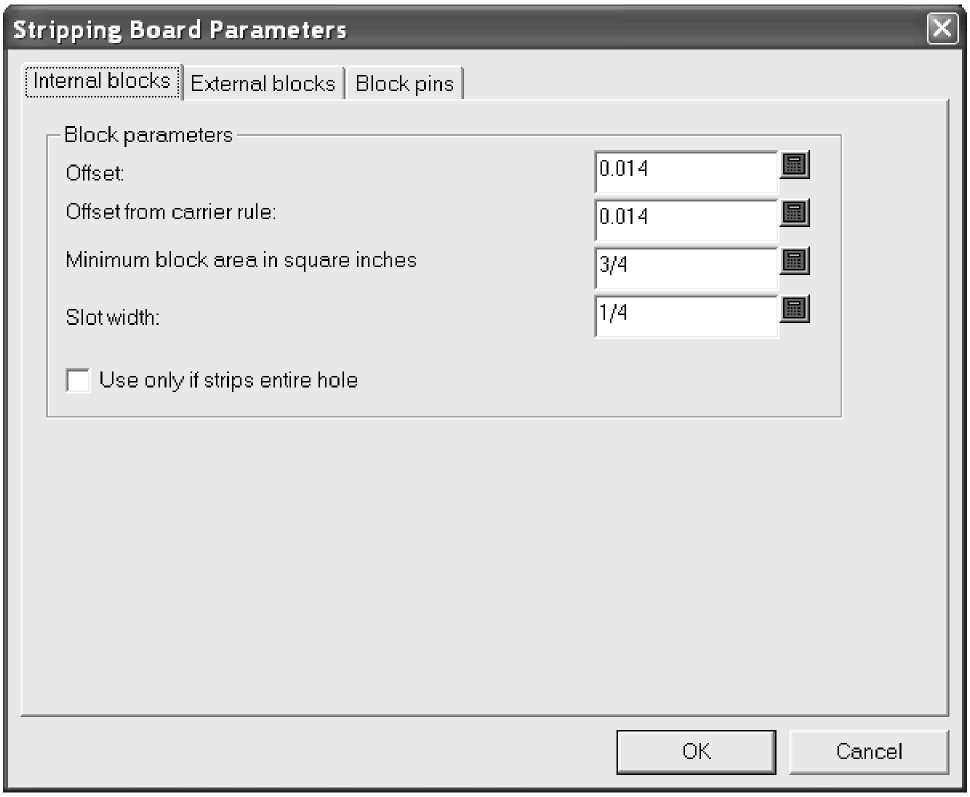
   Shown below is an example of setting the end point for a piece of 20mm step Wave rule with the clearance lines shown around it and **Snap to Middle Rule** selected as the placement method. The start point of the rule is at the right end of the slot and the drag is toward the left.



How to add an upper block

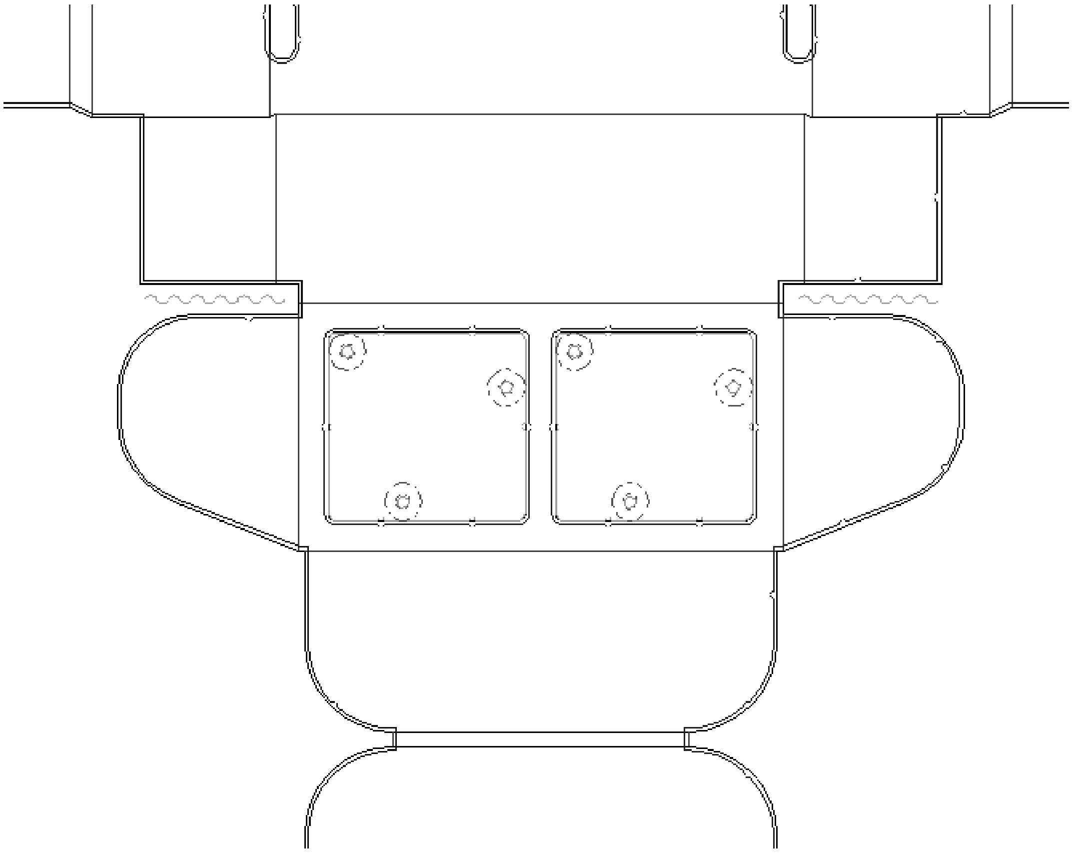
 The **Upper Block** tool adds a block to the upper stripping board in the hole you indicate. As with the Upper Push Pin tool and the Upper Rule tool, you must click inside a hole in the lower board to add the block. When you click the tool, an **Add Block Pins** checkbox and a **...** button (More Options) appear on the Status bar.

Clicking **More Options** opens a Stripping Boards Parameters dialog box with tabs for internal blocks, external blocks, and block pins. Modify the fields as desired and click **OK**.



To add an upper block, do the following:

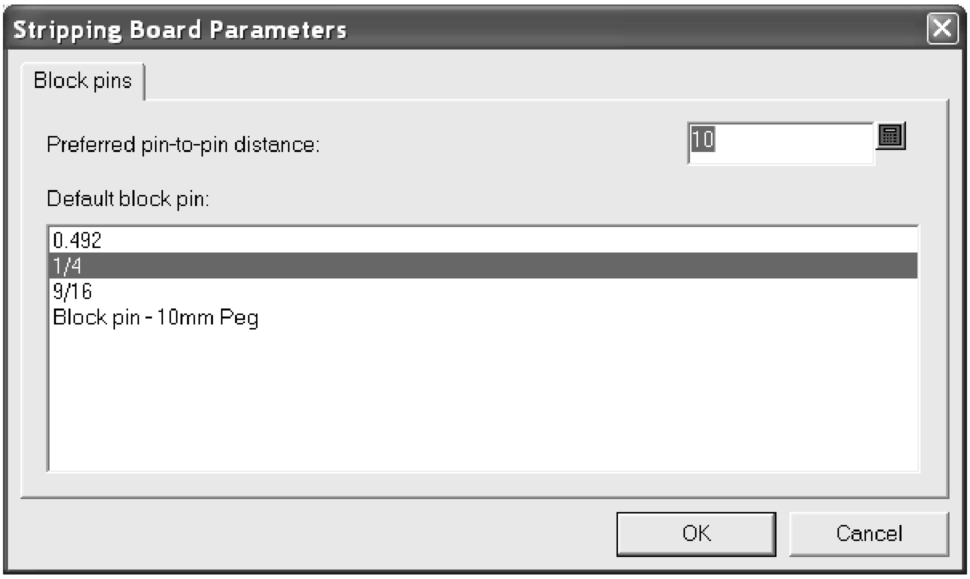
1. Make sure there are holes in the lower stripping board. Use the Strip Area tool if necessary.
2. Click the **Upper Block** tool.
3. Select or deselect the **Add block pins** checkbox on the Status bar as desired.
4. Click inside the hole for the new block. The block is created with or without pins and repeated according to the Auto-Repeat status as shown in the example below.



How to add an upper block pin

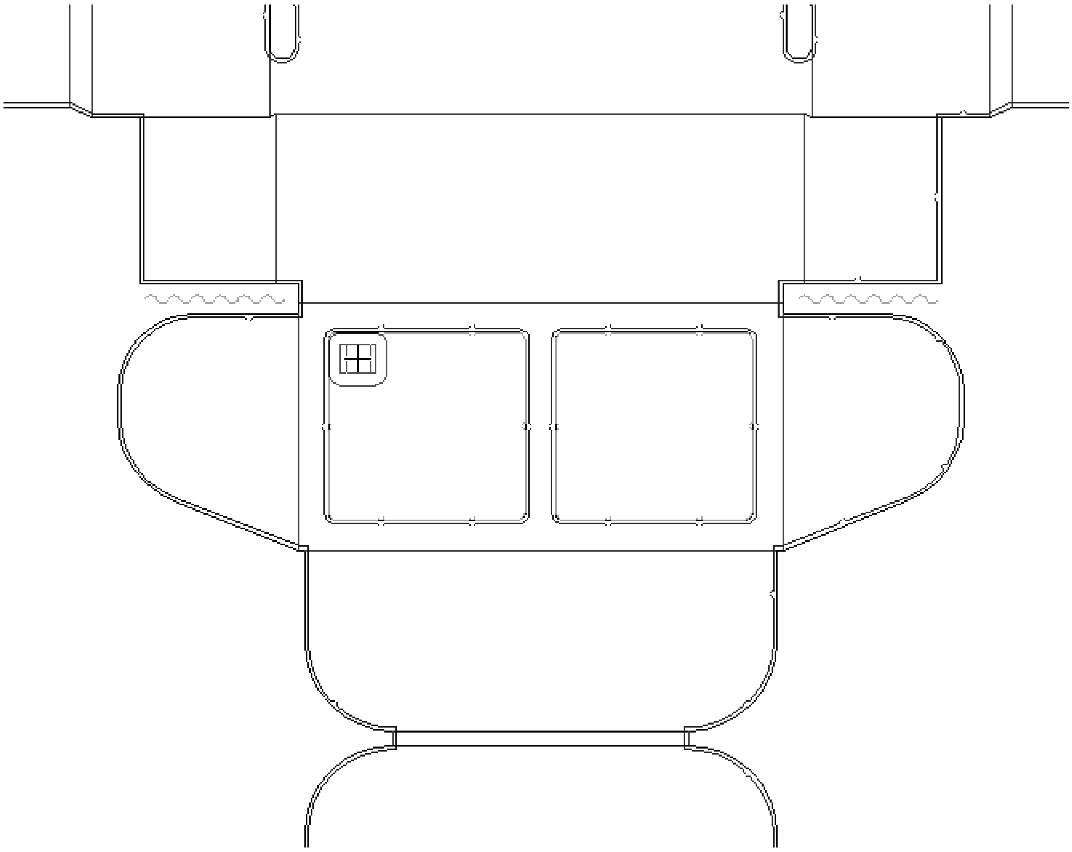
The **Upper Block Pin** tool lets you add upper block pins manually. When you click this tool, a **Pin size**: drop-down list box and a **...** (More Options) button appear on the Status bar.

Clicking **More Options** opens a Stripping Boards Parameters dialog box with a tab for block pins. Change the distance between pins or the default pin as desired and click **OK**.

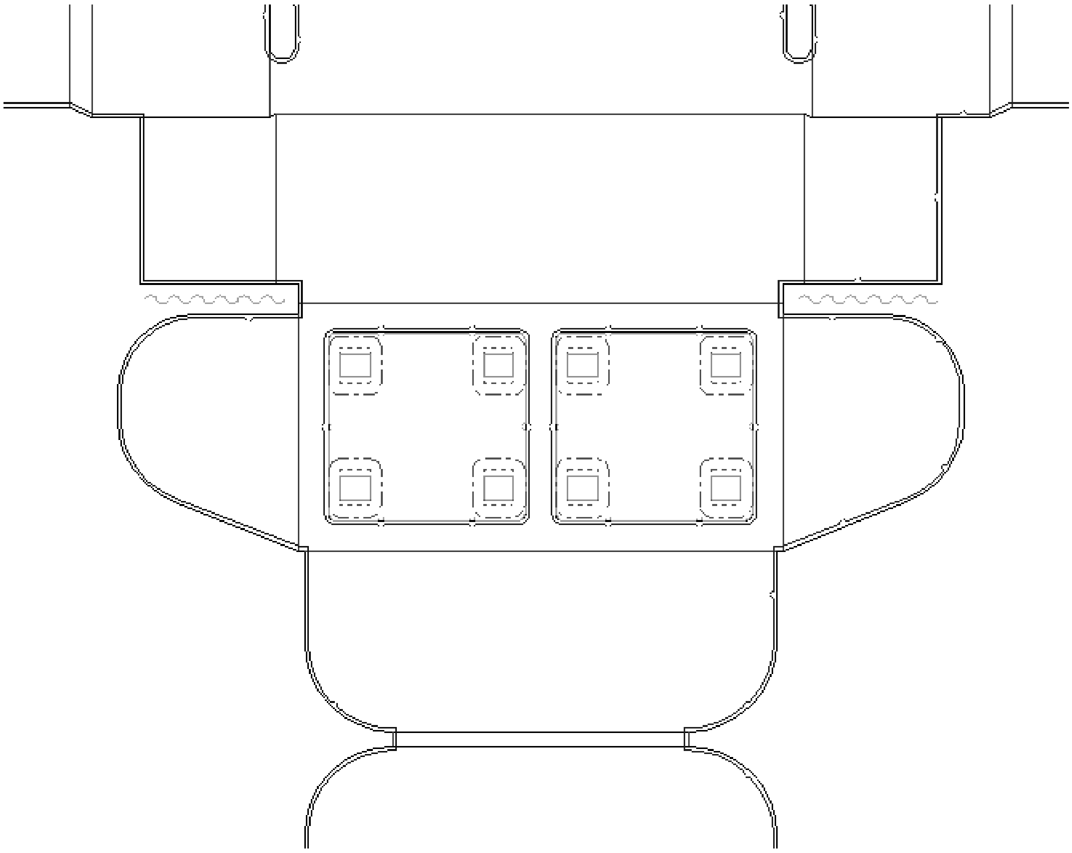


To use this tool, do the following:

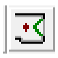
1. Make sure you have an upper block created.
2. Click the **Upper Block Pin** tool.
3. Choose the upper block pin to use from the **Pin size**: drop-down list box.
4. Use the drag to place the pin as desired. Green drag indicates the pin is snapped to a hole edge; blue drag indicates it is being placed freehand; red drag indicates the desired position is unavailable. The example shown below uses the 10mm block peg.



5. The pin is Auto-Repeated according to the current Auto-Repeat setting.



How to add interference

 Sometimes when a piece of waste is small (and therefore light), the partial vacuum in the press can suck the waste back up through the holes in the bottom stripping board. *Interference*, also known as power stripping, is the process of adding a little bump into the edge of a hole using inside modifier lines so that the waste stays where it is supposed to be and doesn't come back into the press. The **Add Interference** tool is one way to add interference. You can optionally add a pin in the top stripping board to force the waste through the now-smaller hole.

Options for this tool appear on the Status bar when the tool is clicked.

Round, **square**, and **chamfer** control the shape of the interference. **Width** sets the size of the interference. **Add pin** controls whether a pin is added to the upper board to force the waste through the modified hole. Interference added without a pin adds modifier lines to the lower board. Interference added with a pin adds the pin and modifier lines to the upper board. **Pin offset** sets the distance to the pin from the edge of the hole, and is available only when **Add pin** is selected. **Pin size** sets the size of the pin and is available only when **Add pin** is selected.

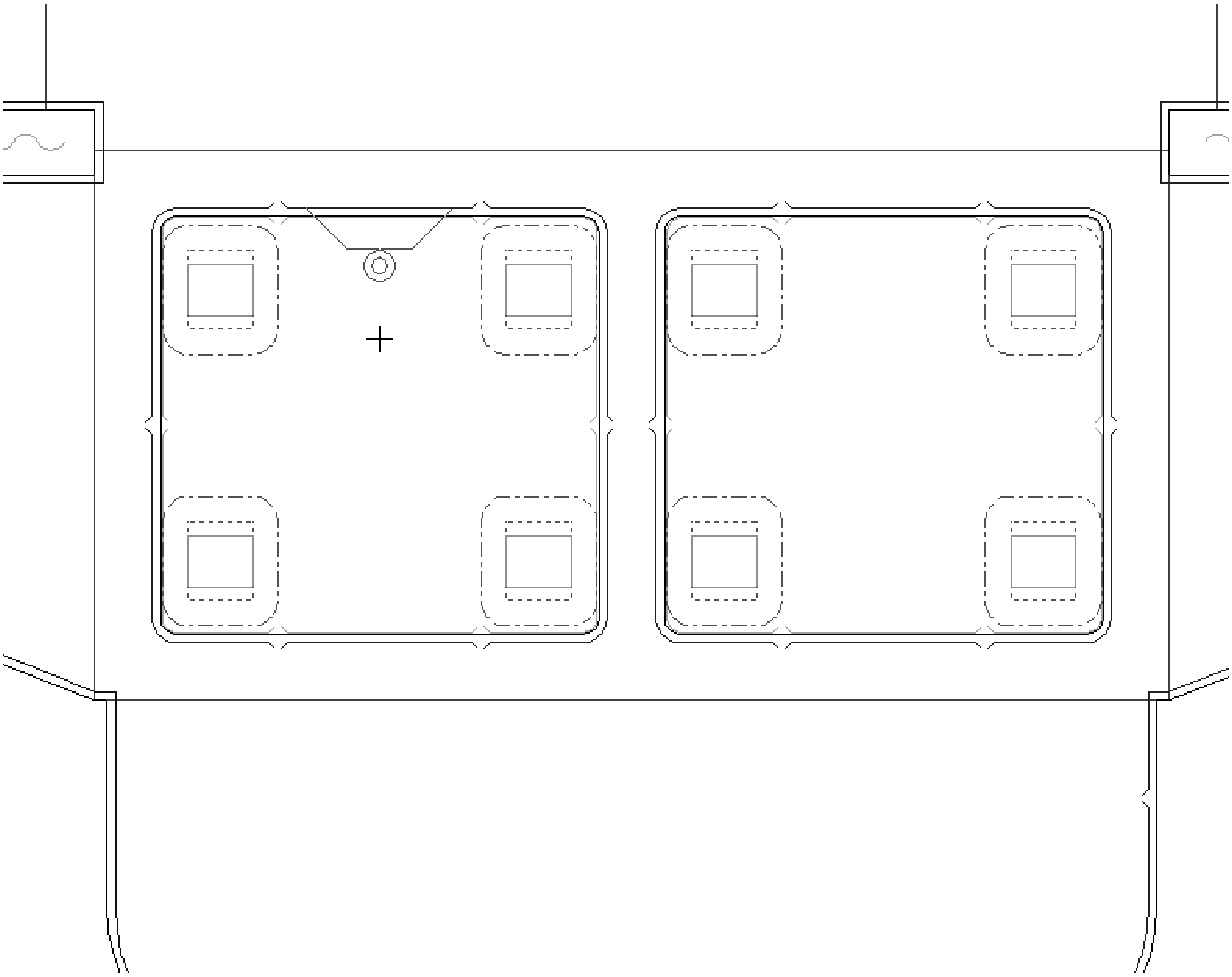
When interference with a pin is added, it goes into the upper board. When the interference is added without a pin it goes into the lower board. When a block is added to a hole with interference, if the interference is in the lower board the block goes around the interference. If the interference is in the upper board it is deleted.

The **Repeat Strip Area** tool repeats the interference whether it is in the upper or lower board, according to the **Upper** and **Lower** checkboxes. It replaces any existing interference in the repeated holes.

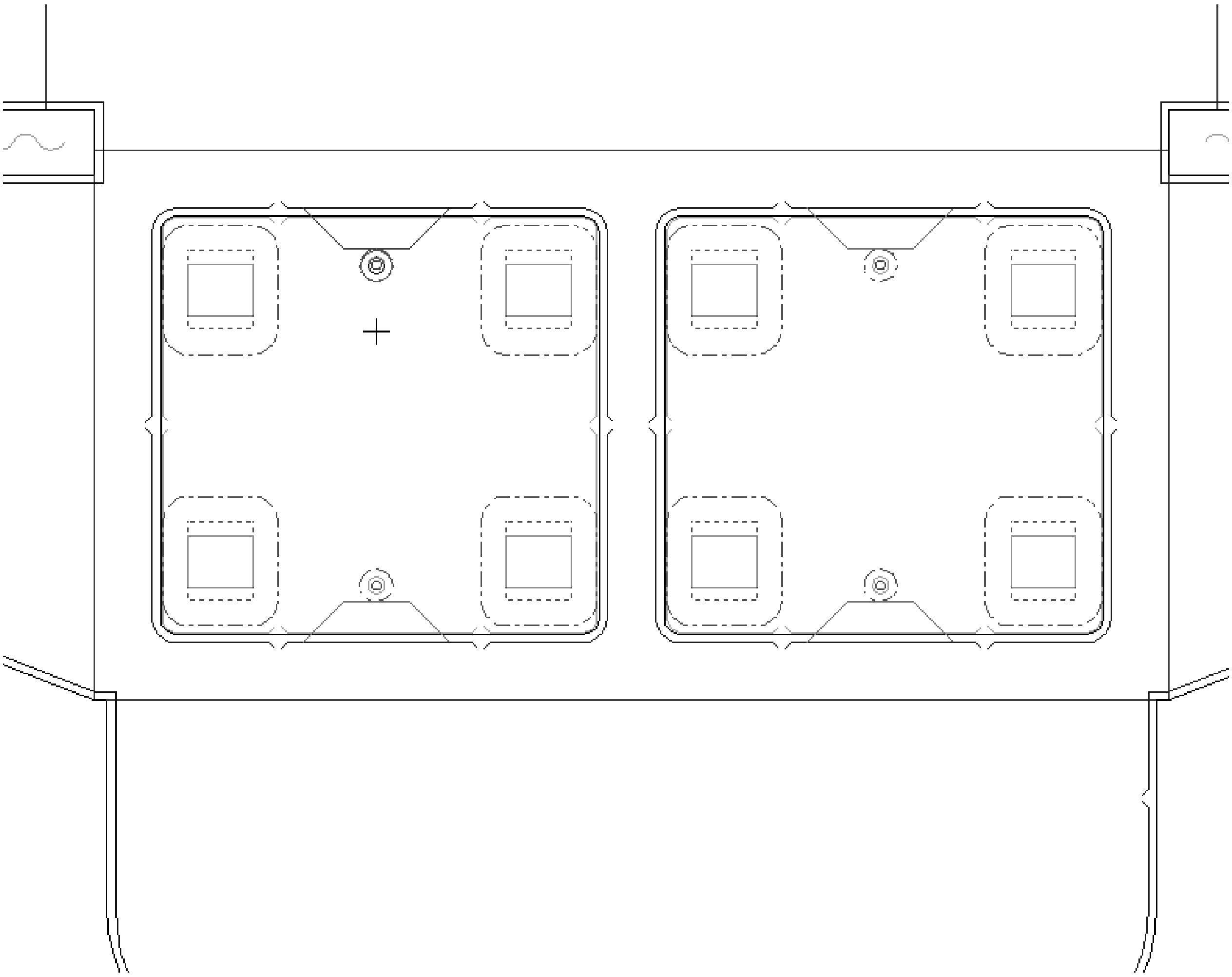
You can specify additional offsets for interference on the Interference tab of the Stripping Board Parameters dialog box.

To use the tool, do the following:

1. Make sure there are holes in the lower board.
2. Click **Add Interference** and set the options on the Status bar as desired.
3. Use the drag to set the position along a lower hole edge for the interference. Green drag indicates the pin is snapped to a hole edge; blue drag indicates it is being placed freehand; red drag indicates the desired position is unavailable.



4. Click to place the interference. The interference is Auto-Repeated according to the Auto-Repeat settings.

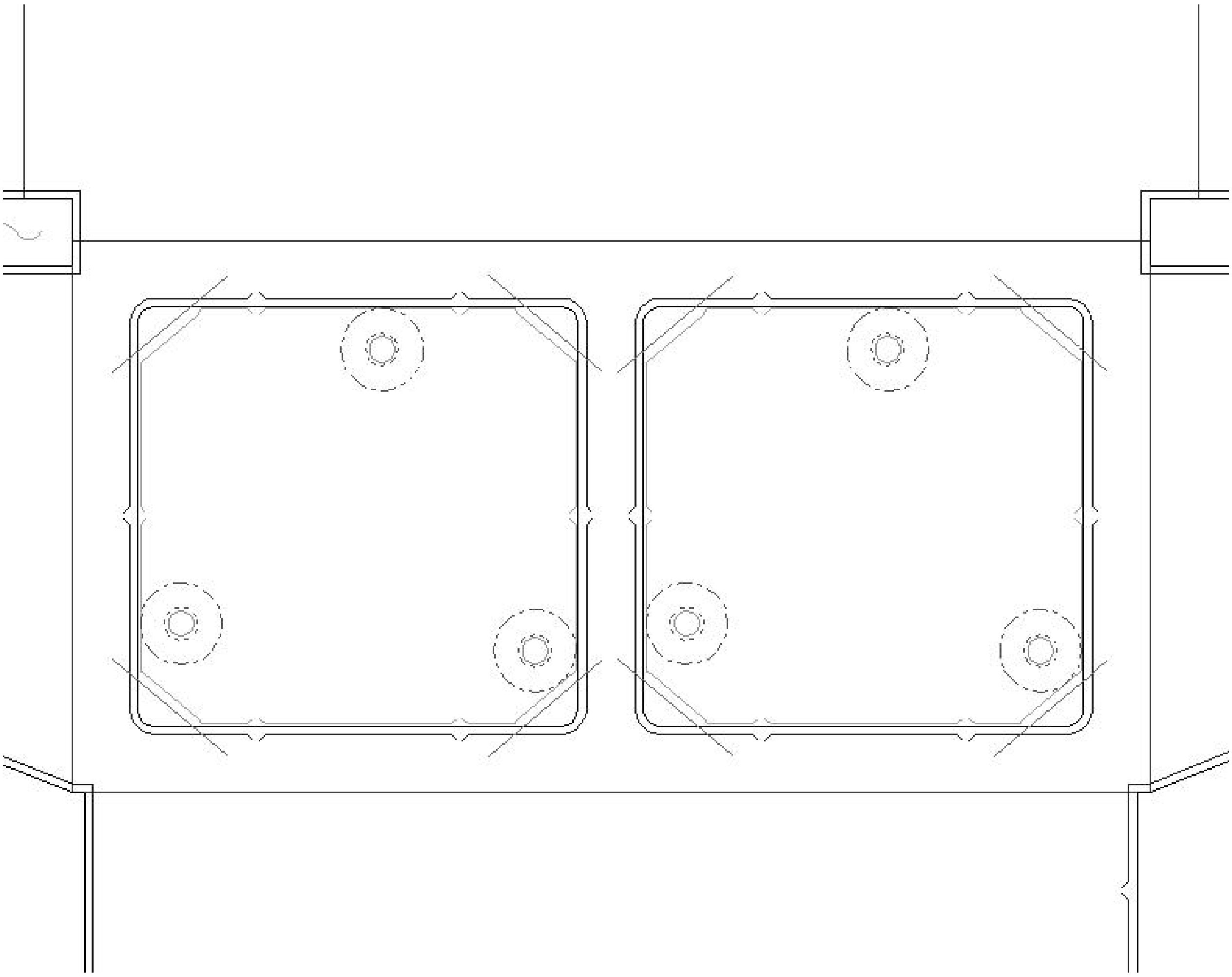


Set defaults for this tool in **Options** > **Defaults** > **Shared defaults** > **Manufacturing parameter sets** > **parameter set catalog** > **parameter set name** > **Upper stripping board** > **Interference Stripping**.

Another way to add interference is to create manual interference by adding lines of type **Hole inside modifier** that cross the edge of the hole to the **Inside modifier** sublayer of the **Lower stripping board** layer. Any pins, rules, or blocks added will follow the inside modifier lines. Select the inside modifier lines with **Select Element** and repeat them using **Repeat Element**. The shape of the lower hole will be modified on Output.

Manual interference is deleted if you use the **Strip Area** tool with the **Lower** checkbox selected after adding manual interference.

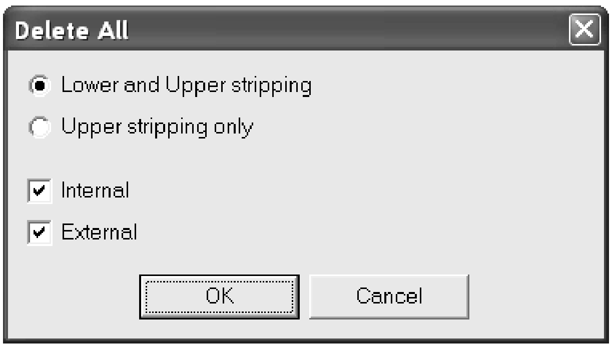
Shown below is an example of manual interference. A hole modifier line was drawn across the corner of a lower hole and was repeated, and then an upper block was added with pins and repeated. The edge of the upper block follows the lower hole inside modifier lines.



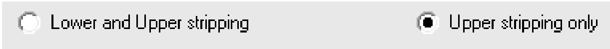
How to remove stripping

Two tools on the Manufacturing toolbar remove stripping.

 **Delete All** removes all stripping from the layout. When clicked, the Delete All dialog box appears. Choose the options as desired and click **OK** to remove the stripping.



 **Delete Strip Area** removes stripping from individual stripping areas. When clicked, **Lower and Upper stripping** and **Upper stripping only** option buttons appear on the Status bar as shown below.



To use this tool, do the following:

1. Click **Delete Strip Area**.
2. Choose the appropriate option button on the Status bar.
3. Click inside the stripping area to delete. Deletions will be Auto-Repeated according to current Auto-Repeat settings.
4. Repeat in additional stripping areas as desired.

How to add an air hole

 Modern presses operate so quickly that partial vacuums are sometimes created from the sheets rushing through them. These vacuums can ruin a press run by causing the machine to jam. Air holes made in the upper stripping board are one way to combat this problem.

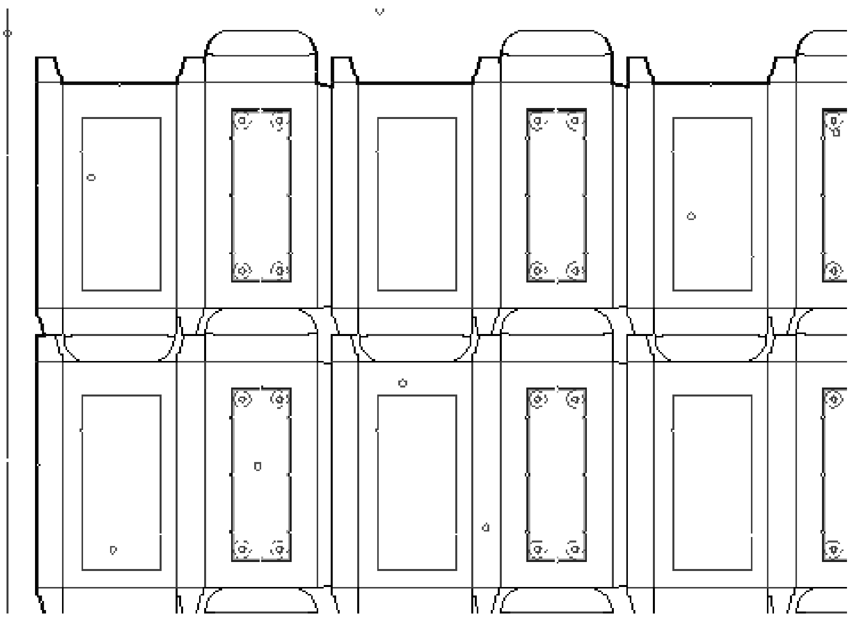
Three options for the shape of the air hole appear on the Status bar when the tool is active.

The **Radius** field appears only when the shape is set to **Circle**.

To use this tool, do the following:

1. Click **Make Air Hole**.
2. On the Status bar, choose the shape of air hole to create.
3. Click at the start point of the air hole and use the drag to set the end point.
4. The air hole is created and Auto-Repeated according to the current Auto-Repeat settings.

Shown below are examples of air holes.



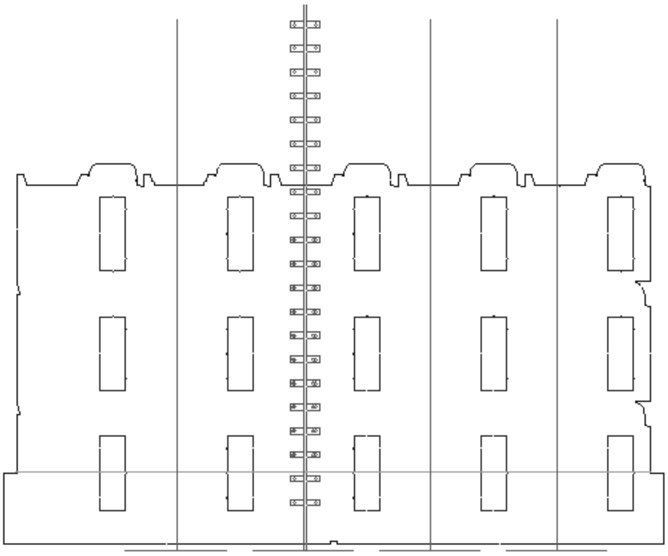
How to make an alignment hole

Alignment holes are used to quickly and easily align a set of manufacturing tools without resorting to measuring tapes. To make an alignment hole, click the **Make Alignment Hole** tool and click the location for the alignment hole.

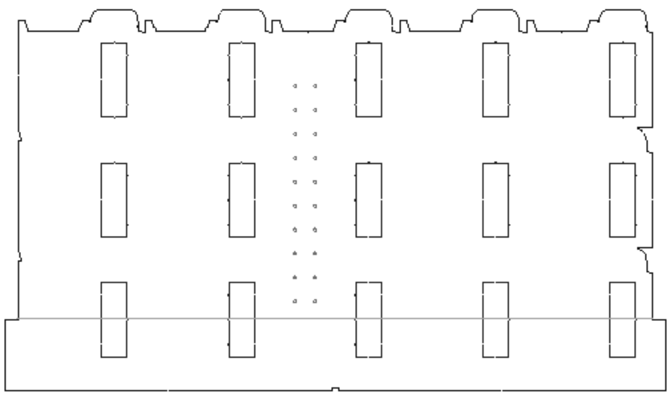
How to add a support bar

The **Add Support Bar** tool adds support bar holes to the lower stripping board. The support bar holes always go into the lower stripper.

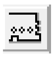
To use this tool, click it. Construction lines will appear in the recommended locations for the support bar. Use the drag to set the location of the support bar. Once the location is set, use the mouse to select the holes to be made for the support bar. When you have finished selecting holes, click **OK** on the status bar. Shown below is the tool drag prompting you to set the location.

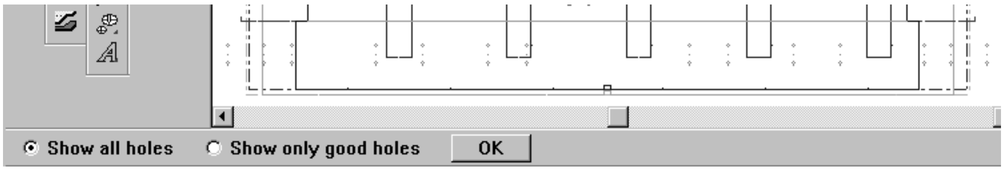


Shown below are the completed holes for the support bar.



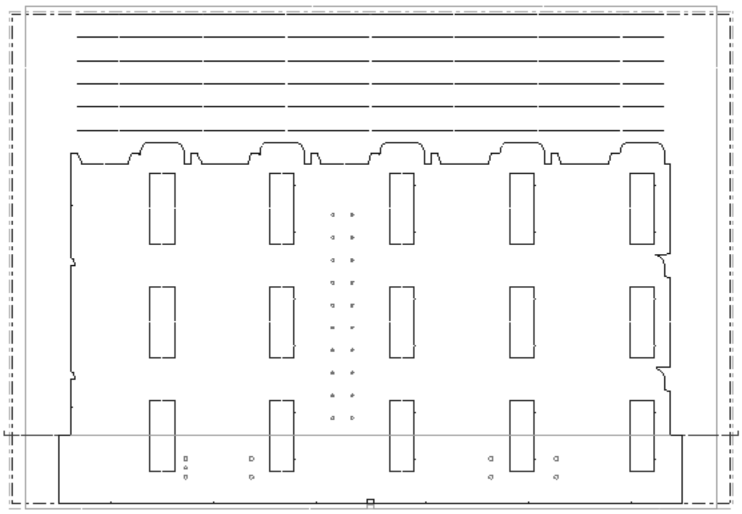
How to add registration holes

 To add registration holes across the lower edge of the board, click the **Add Registration Holes** tool. Holes will appear across the bottom of the board, and two options in the Status bar allow you to select from all holes or from only good holes.



Select the holes to add and click **OK**. The holes will be added.

Shown below are the registration holes.



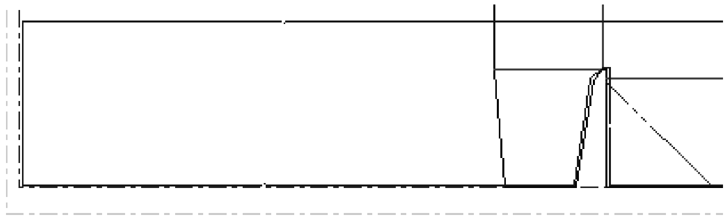
How to add a Front Waste Separator

A **Front Waste Separator** strips the waste from the lead edge of the layout. The separator is made of top and bottom stripping boards which follow the edges of the designs and any stripping rules.

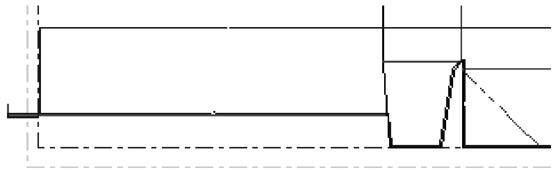
The vertical edges of the Front Waste Separator are found by using the following points: the points at which the layout designs meet the usable edge of the sheet; where they meet the bottom-most stripping rules which intersect the vertical usable-sheet edge; or an existing bottom stripping board mounting bar. If none of these points are found, the separator edge just extends horizontally to the sheet edge.

The Front Waste Separator is placed into the Front Waste Separator layer when it is created.

Shown below is the edge when none of the points are found.

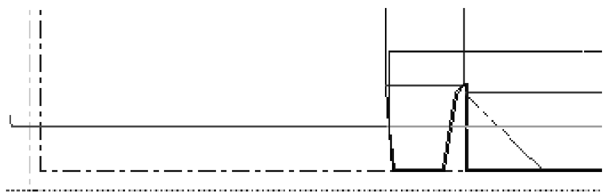


When there is a stripping rule:



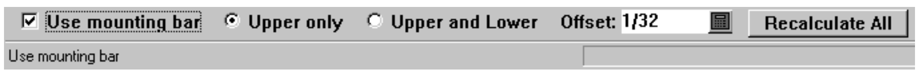
Notice how the stripping edge goes up along the edge of the tab and then runs along the stripping rule to the edge of the sheet.

When there's a bottom mounting bar:



To use this tool, do the following:

1. Create a layout.
2.  Click the **Create Separator** button on the Stripping Board toolbar.
3. The Status bar will change:



4. Set the options on the Status bar.

   The **Use Mounting Bar** checkbox controls if the Front Waste Separator is positioned based on the mounting bar. If there is no mounting bar, the separator is based on the lead edge.

   Upper only creates the Front Waste Separator out of the upper stripping board. **Upper and Lower** creates the separator using both upper and lower stripping boards.

   Offset controls the gap between the lead edge and the edge of the Front Waste Separator.
5. Click **Recalculate All**. The Front Waste Separator is created.

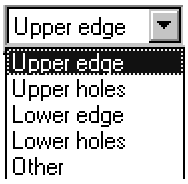
Make sure to click **Recalculate All** after each change you make to the options on the Status bar while the tool is active. Deactivate the tool by clicking another tool.

How to modify a Front Waste Separator

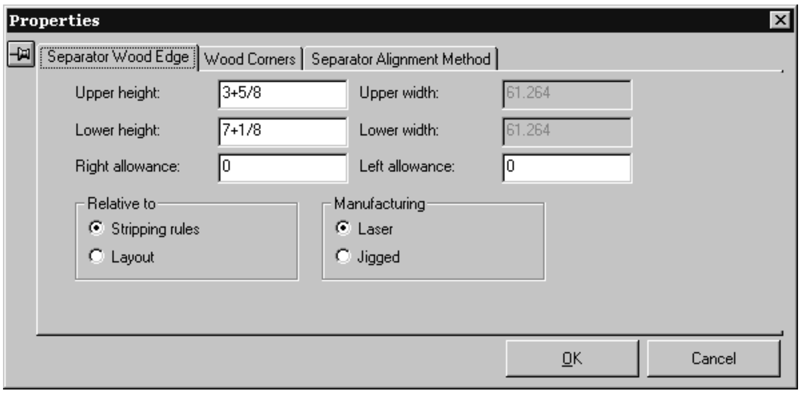
To modify a Front Waste Separator, activate the tool and choose different options on the Status bar, and then click **Recalculate All**.

You can also modify its properties by doing the following:

1.  Click the **Select Element** tool.
2. Choose the element to modify from the drop-down list bow on the View bar.



3. Double-click the element to modify, or select it and press `ALT-Enter`. The Properties dialog box for that element will appear.



4. Set the options on the tabs as desired and click **OK** to make the changes or **Cancel** to ignore them.

Front Waste Separator defaults are stored in the Front Waste Separator section of the **Die press parameter sets** folder in Defaults.

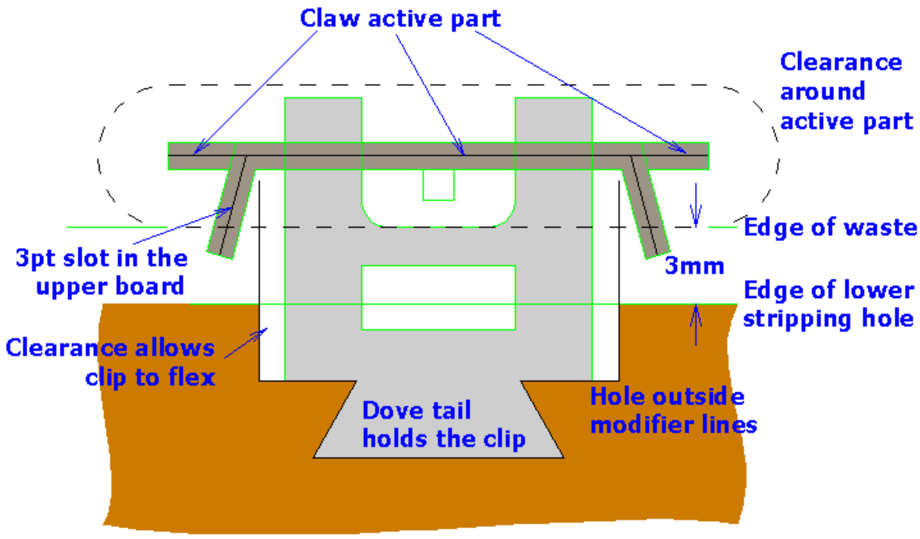
The workspace containing the mounting hole pattern is `...\InstLib\FWS_BOLT-HOLES.ARD.` The name of this workspace is set in the **Hole Patterns** page of the **Press Defaults** in the **Die press parameter sets** folder in Defaults. On that page you can also set the side of the die the patterns are placed on to die knife side, die back side, or undefined.

Using stripping components

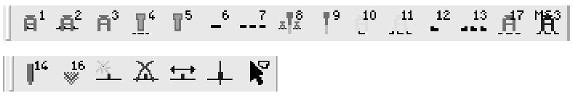
Stripping components are geometry macros which create space for a claw in the upper stripping board and a clip in the lower stripping board. During stripping, the board material is pressed by the claw against the clip, which flexes and traps the waste, thereby eliminating the need for lower stripping board pins.

Several predefined stripping component geometry macros come with ArtiosCAD, but you can make your own in Designer.

Shown below is a sample stripping clip.



The predefined stripping components are on two customized toolbars as shown below. Use the Customized Toolbars Master Control to turn them on and off.



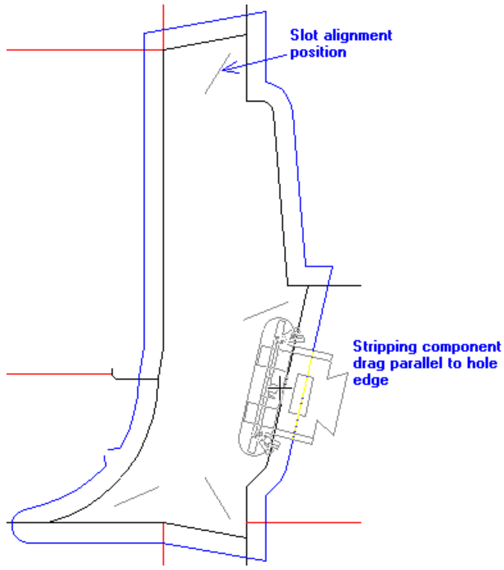
Adding a stripping component

Perform any stripping using the **Strip Area** tool before adding stripping components. Using the **Strip Area** tool will overwrite stripping components.

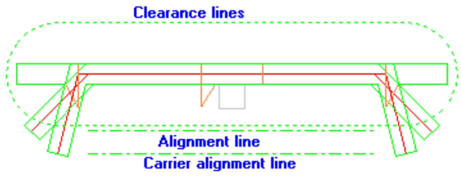
To add a predefined stripping component, do the following.

1. Make sure the customized toolbars are turned on and click the button for the component to add. The upper and lower boards will be turned on if they were not on already.
2. Drag for the component will appear under the cursor. Position it as desired, keeping the following placement options in mind when dragging:

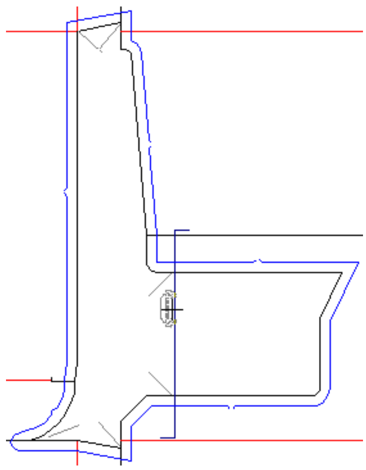
   - **Close to the edge of a hole**. The stripping component is aligned with the edge of the waste (the cut lines) in the stripping hole or the lower board edge so that the alignment lines touch the inside of the hole. If the clearance lines do not fit inside the hole, the drag turns red.



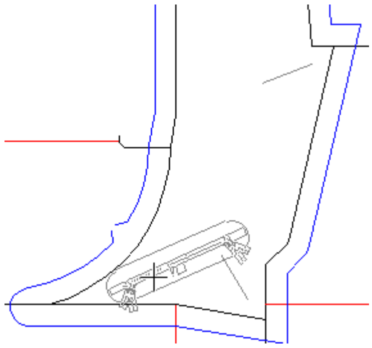
- **Close to a carrier rule.** This is the same as being close to the edge of a hole but a different offset is used - that of the distance between the regular alignment line and the carrier alignment line.



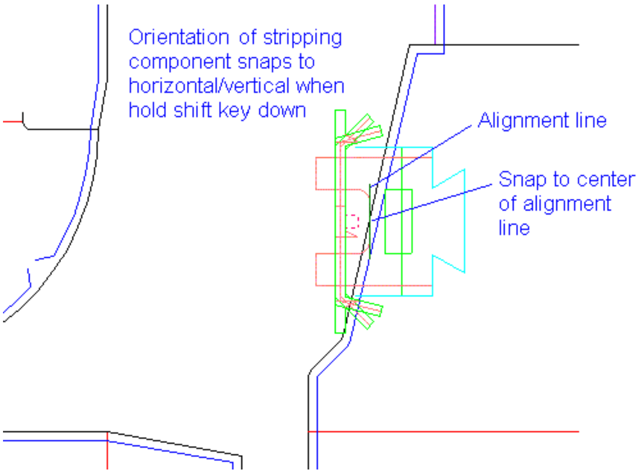
The stripping component is aligned with the carrier rule so that the carrier alignment lines touch the carrier rule. If there are no carrier alignment lines, the alignment lines are used instead. If the clearance lines do not fit, the drag turns red.



- **Close to a slot.** Select a slot by moving the mouse within snap tolerance of the slot alignment line. The stripping component is aligned so one end of the clearance outline touches the end of the slot depending on the side of the line closest to the mouse cursor.



- **Not in a hole**. If the mouse is not in a stripping hole, or if the cut lines in the stripping hole or board edge do not form a loop, the Status bar prompt changes from **Place stripping component** to **Could not find the stripping hole** and the drag turns to dotted red. If you click outside a hole, the component will be placed wherever you click.
- **Holding down** `SHIFT`. When `SHIFT` is held down, the stripping component snaps to horizontal or vertical (whichever is closer, based on rotating around the center of the alignment line).



Modifying stripping components

To modify a stripping component, first select it with the **Select** or **Select Element** tool. Next, right click it to access its context menu.

Note: If you select more than one component by holding down `SHIFT` as you select, only **Delete**, **Replace**, **Replace All**, and **Properties** are available.

Note: If Auto-Repeat is active, changes made to one stripping component are repeated for similar components.

Note: Selecting multiple stripping components in the same hole with the **Select Element** tool will group them together. If you plan to change the positions of multiple components in the same hole, use only the **Select** tool to select them.

Select subgroup lets you select a subgroup of the component such as a clip or claw.

Move parallel moves the selected component parallel to the edge of the hole. However, you cannot move a component from one hole to another. To do this, delete the original component and add a new one.

Copy parallel does the same thing as Move parallel except it copies instead of moving.

Move perpendicular moves the selected component perpendicular to the edge of the hole. If the perpendicular offset from the edge of the hole has been changed, it will be maintained. If the component is rotated or aligned on a slot, the perpendicular offset is reset to zero.

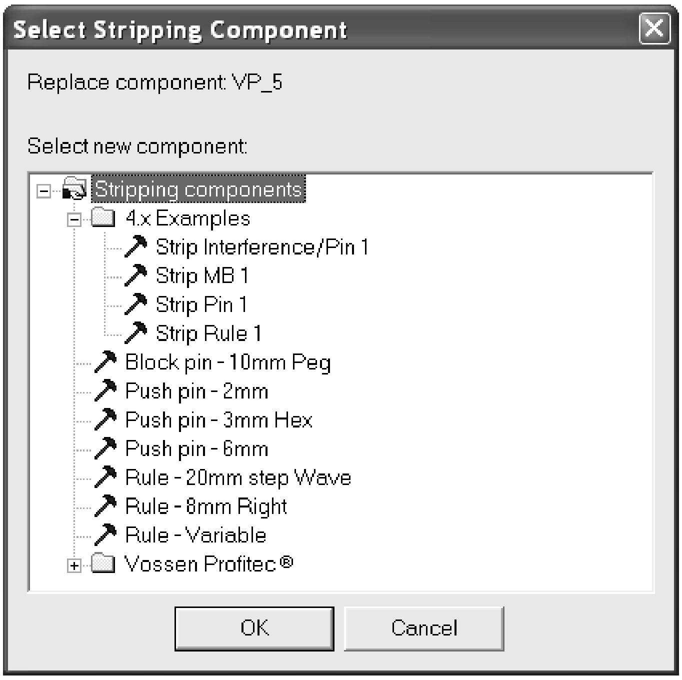
Move freehand moves the selected component anywhere within the same hole.

Rotate 180 rotates the selected component 180 degrees about the center of the clearance lines.

Rotate prompts for an anchor point and rotates the selected component about the anchor point.

Delete deletes the selected component(s).

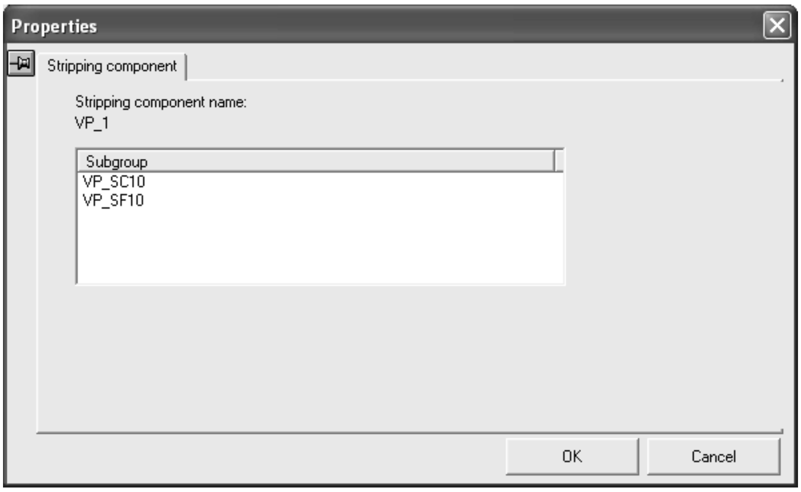
Replace lets you replace the selected stripping component with a new stripping component chosen from the Stripping Components catalog. The new stripping component is repeated accorded to the current Auto-Repeat setting. Shown below is the Stripping Components catalog.



Select the new stripping component and click **OK** to use it as the replacement.

Replace All replaces all instances of the selected stripping component with a new stripping component.

Properties shows the workspace name and any subgroups for the component.



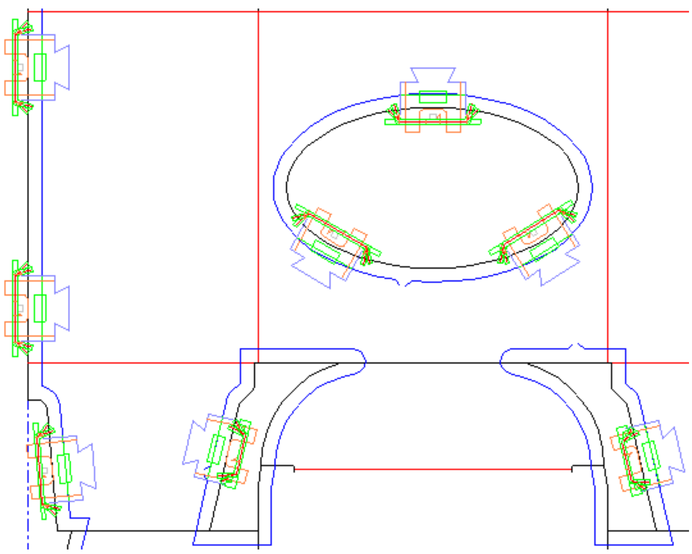
Outputting boards containing stripping components

In a stripping component, the hole outside modifier lines change the shape of the hole in the lower board when performing an Output.

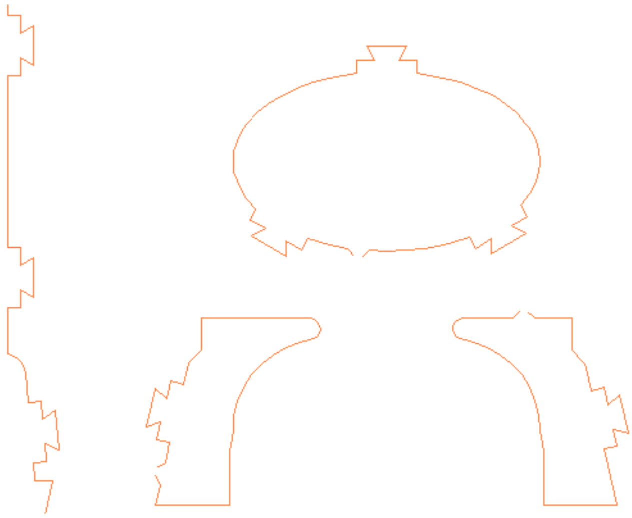
The upper stripping rule and pusher base lines are burned in a CAM output that selects the **Upper stripping board.** The hole inside modifier and hole outside modifier lines are applied to the outline of the lower stripping board in a CAM output that selects **Lower stripping board with upper blocks**. CAM outputs ignore the other lines types such as **Pusher active part**, **Pusher clearance**, and **Stripping component alignment**.

Any tack bridges under a stripping component get moved on Output.

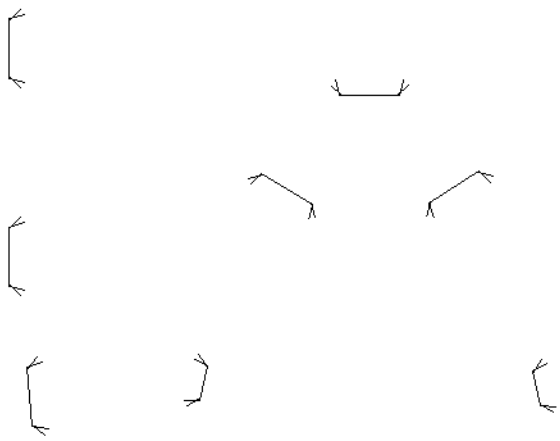
Show below is a portion of a layout containing several stripping components.



During an Output of the lower board, only the hole modifier lines are considered. Shown below is the lower stripping board for the same section of the layout as shown above.



Shown below is the upper stripping board for the same section of the layout showing the stripping rule lines.



Creating your own stripping components

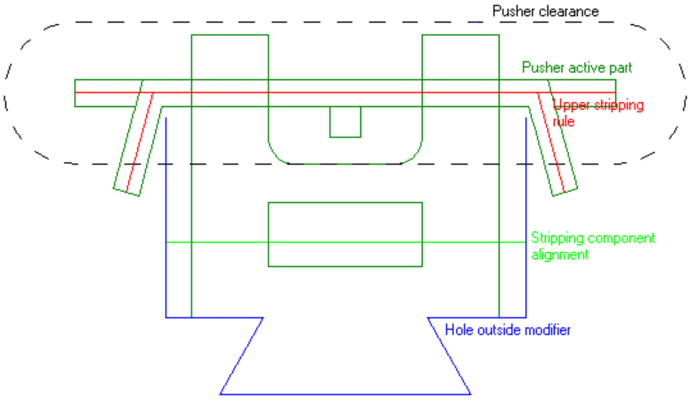
Use Designer to create your own stripping components. Save them as normal workspaces and copy them to **ServerLib**. Then add them to the Stripping Components Catalog in the Geometry Macro Catalog in Defaults, making sure to use the **Stripping hole component** placement type. Save the Defaults. If desired, create icons for them and put them on a customized toolbar.

Refer to the following table of line types when designing a stripping component. The line types are split between the Lower Board and Upper Board line type catalogs.

Table: Line types for stripping components

Line type name	Line type number	Category	Description
Annotation	0	Annotation	A line type used for drawing the component that does not burn.
Pusher active part	188	Annotation	A drawing of the active part, such as the clip or the claw. Does not burn.
Upper stripping rule	186	Rule	The base of a stripping rule that is burned as a slot in the upper board.
Pusher base	187	Inside edge	The outline of the base of the pusher element that is burned in the upper board.
Pusher shoulder	189	Annotation	The shoulder that stops the element from being pushed through the upper board.
Pusher clearance	190	Annotation	The effective size of the element. The stripping component is considered to fit inside the stripping hole if the clearance lines are inside the waste area defined by cut lines inside the stripping hole. Also defines slot placement.
Hole inside modifier	180	Inside edge	The inside edge of an interference extending partly into the lower stripping hole. This loop is combined with the lower stripping board to make the hole smaller.
Hole outside modifier	181	Inside edge	A slot in the edge of a lower stripping hole which holds the outside edge of a stripping hole element. The loop is combined with the lower stripping board to make the hole larger.
Stripping component alignment	203	Annotation	These lines touch the edge of the stripping hole to align the stripping hole component.
Stripping component carrier alignment	204	Annotation	These lines touch the carrier rule when the component is aligned near a carrier rule.

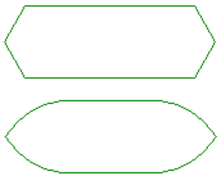
Shown below is an example of how these line types should be used.



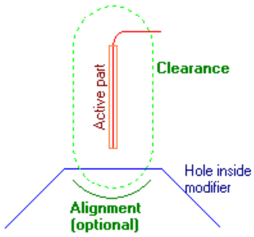
Clearances

For a stripping component that is wider than it is tall, the clearance perimeter should have both horizontal and vertical symmetry. The left or right end of the clearance is centered in a slot, depending on the position of the mouse cursor in relation to the side of the slot. Design clearances using the **Pusher clearance** line type in the Upper board line type catalog.

Shown below are alternate clearance shapes to show possibilities beyond an oval.



For a stripping component that is taller than it is wide, the clearance perimeter should have vertical symmetry. The bottom end of the clearance is centered in a slot.



For a pin, the clearance should be circular. The alignment lines should be omitted as they would usually be the same offset as the clearance.

Clearance lines are shown in the drag but are invisible after the stripping component is placed.

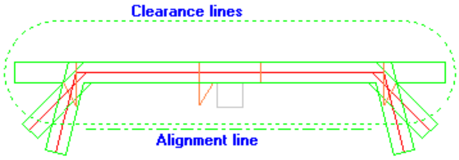
If there are no clearance lines, or if they do not form a loop, no slots are defined.

Alignment lines

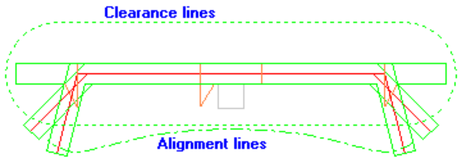
Alignment lines determine how the stripping components are aligned with the edge of the waste. Design alignment lines using the **Stripping component alignment** line type.

Alignment lines:

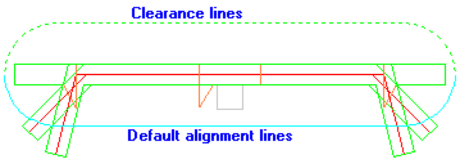
- Should be a single set of connected line segments (but not a loop) below the lower edge of the clearance.



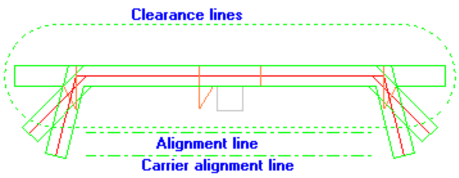
- Should be horizontal, they do not need to be straight - they can be arcs for use when the waste edge is curved.



- Can be along the bottom edge of the clearance lines, but should not be within them.
- Are optional if the clearance lines are a loop, in which case the lower half of the clearance lines form alignment lines.



When designing a stripping component for use on carrier rules, add a horizontal carrier alignment line below the alignment lines, or below the clearance lines if using the default alignment.



Listing stripping components on a Report

You can create a Report that lists the stripping components used in a stripping board by adding items of calculated text in the Stripping Component Information catalog in the Manufacturing / Layout catalog of calculated expressions.

When you first add a stripping component to a layout that does not have any, the stripping component gets assigned a number called its **index**. The index of the first stripping component is set to 1. If the stripping component is composed of subgroups, those subgroups also get assigned an index number of their own. For example, if you have stripping component MBC Pin 1 which has three subgroups, the pin as a whole is set to index 1, and the three subgroups are set to indices 2, 3, and 4. The next pin would therefore have indices 5, 6, 7, and 8.

The calculated expressions work in terms of the index of an item or its name. Enter each as appropriate. Each calculated expression only lists information for the item corresponding with the index number you entered when prompted. If you only add one item of calculated text based on index 2 but have four stripping components, only information about the second index is included on the Report. Add multiple calculated expressions to result in multiple pieces of information about the stripping components.

Notes and warnings about stripping components

Hole modifier lines can be added manually to the upper and lower stripping boards, but they affect only the lower board on Output. If there are inside or outside modifier lines which do not cross a lower hole or the lower board edge, a warning dialog box is displayed.

If inside or outside modifier lines cross more than one hole, only the first hole will be affected. If modifier lines do not cross any holes, a warning will be displayed on Output.

When creating a composite stripping component out of smaller components, delete excess clearance and alignment lines so only one set remains.

Notes and warnings on manual stripping board modification

When ArtiosCAD makes stripping board holes, inside modifier lines are connected into a loop following the shortest route around the hole. ArtiosCAD then subtracts inside modifier loops, or adds outside modifier loops as needed. If the result has more than one piece, ArtiosCAD keeps only the largest piece.

A geometry tool like the line, rectangle, arc tool can put lines in any layer, so there is a potential for having lines in the wrong layers, or lines in the wrong order not forming loops.

When you switch to a lower/upper board tool, or open an manufacturing file, ArtiosCAD activates a fix stripping board function to move the lines to the correct layers, and fix the upper/lower board exteriors so they are counter-clockwise loops.

The stripping board functions, for example dragging a pin along the board edge, only work after the layers and loop direction have been fixed.

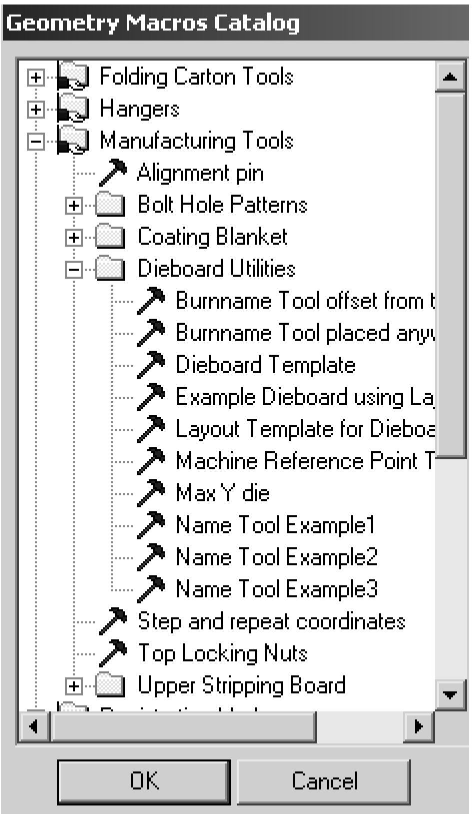
The fix stripping board function does the following:

- Forms loops out of the lower board edge and lower hole lines.
- Lower board edge and lower holes lines that do not form loops are moved into sublayer **Lower Other**
- Upper board edge lines that do not form loops are moved into sublayer **Upper Other**
- Moves the inside modifier lines to the lower board layer and makes loops if possible.
- Moves stripping components, pins, rules, blocks into an internal or external upper sublayer depending on if they are in holes or on the board edge.

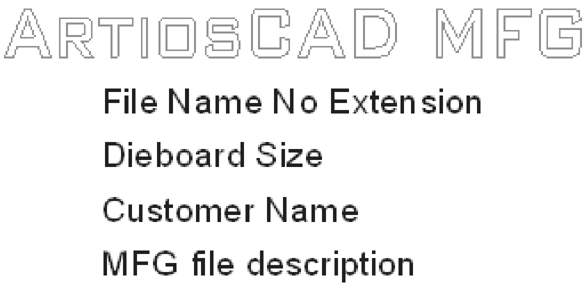
- Makes loops out of air hole lines and moves them into the air holes sublayer

Using a Geometry Macro to add a burn name

There are three Name Tools in **Tools** > **Geometry macros** > **Manufacturing tools** > **Dieboard utilities**. These tools place information on the dieboard itself.



Name tool Example1 inserts a geometry macro containing the file name with no extension, the dieboard size, the customer name, and the manufacturing file description.



Name tool Example2 inserts only the file name with no extension.

Name tool Example3 inserts the filename with no extension in the INTERACT font.

Counter layouts

Counter layouts are used to make many counters at once on a countercutter.

A few notes about counter layouts:

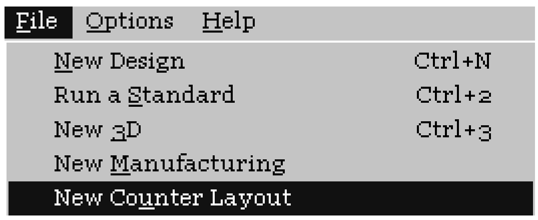
- Only the counter layer from each embedded design is shown. Manufacturing layers and design layers cannot be seen.
- Only manual layout tools are available.
- Counter layouts are not tracked in the database.
- Side and grain are not applicable in counter layouts. The top of the counter is always shown. However, the grain direction for embedded designs is set to match that of the original designs.
- Counter layout workspaces use the **.CTL** filename extension.

Creating a counter layout

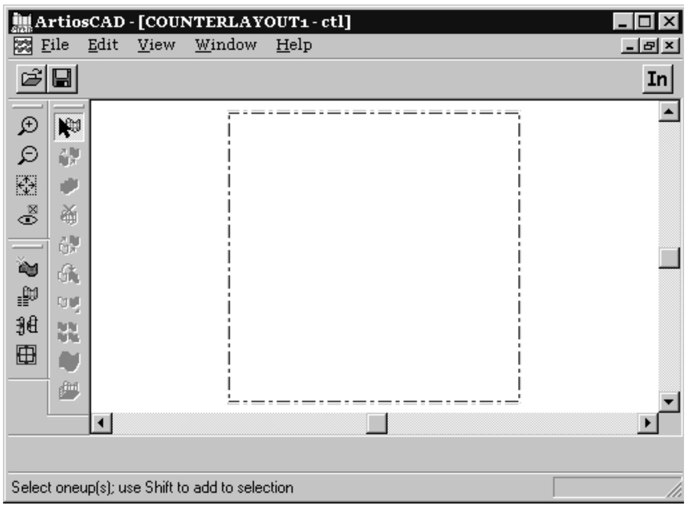
Before creating a counter layout, you must have at least one design containing a counter for inclusion in the layout.

To create a counter layout, do the following:

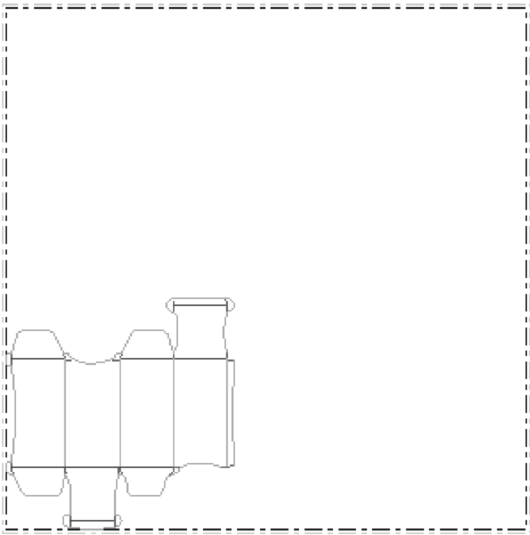
1. Start ArtiosCAD.
2. Click **File**, and then click **New Counter Layout**.



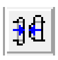
3. A new counter layout will be created containing an empty sheet.



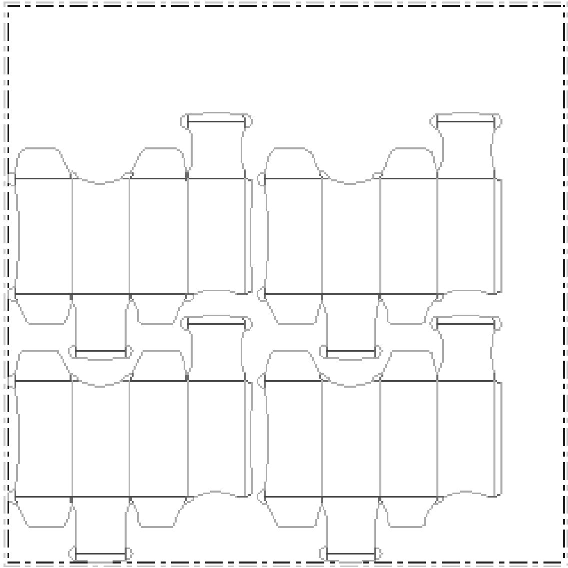
4.  Click the **Add Oneup** tool and then select the design(s) to add. Click **OK**.



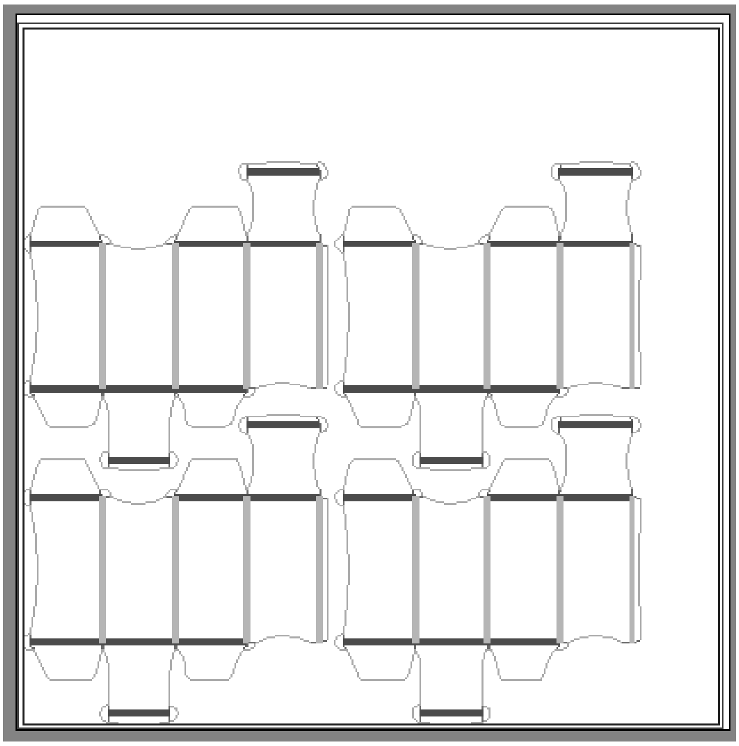
5.  Click the **Select Oneups** tool and select the design you just added.
6. Use any of other Manual Layout tools to arrange the design on the sheet. The quickest and easiest way to do this is to set the gutter and then use one of the Nest tools to step the counters out on the sheet.

    To change the gutter between the counters when using one of the Nest tools, click the **Change Gutter** tool and enter the new gutter distance.

    Use one of the tools on the Nest tools flyout toolbar. Shown below is a picture of an example counter layout using the **Straight Nest** tool.



7. Output the counter layout using a counter Output on the **File** menu. A preview of the output to a Kongsberg PentaCut is shown below.



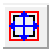
8. Complete the Output and save the layout.

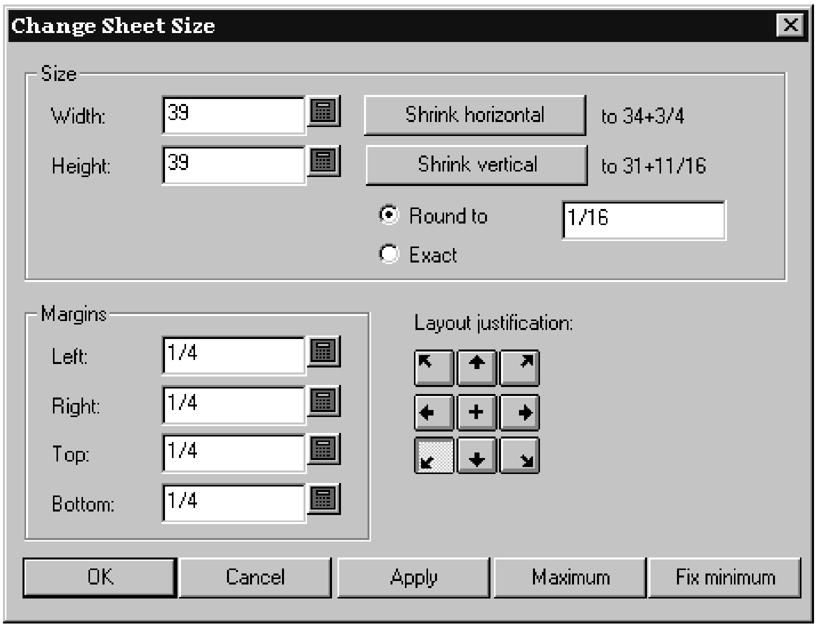
The counter layout is now complete.

Modifying a counter layout

To modify the position of the design on the counter layout, use the tools on the Edit Layout toolbar.

If the nested counters don't completely fill the sheet, you may want to adjust the size of the sheet or the position of the counters on the sheet.

 Use the **Change Sheet Size** tool to do this.



You must choose a layout justification method first. The buttons in the **Layout Justification** group move the group of designs so that the bottom center of the designs is aligned with the bottom center of the sheet, and so forth. Once a layout justification method is chosen, all the other fields become available and you can adjust the size of the sheet and its margins.

Coating blankets

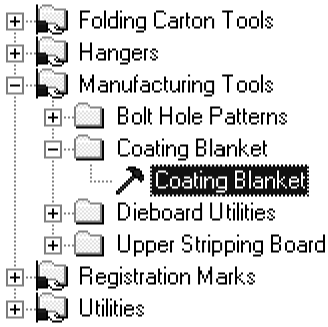
Coating blankets, also called varnish blankets, are used to protect those areas of the sheet from having coating applied as the sheet moves through the printing press.

Coating blankets use the bleed or varnish layers in the single designs comprising the layout. If the designs do not have one of these layers, a coating blanket can not be made.

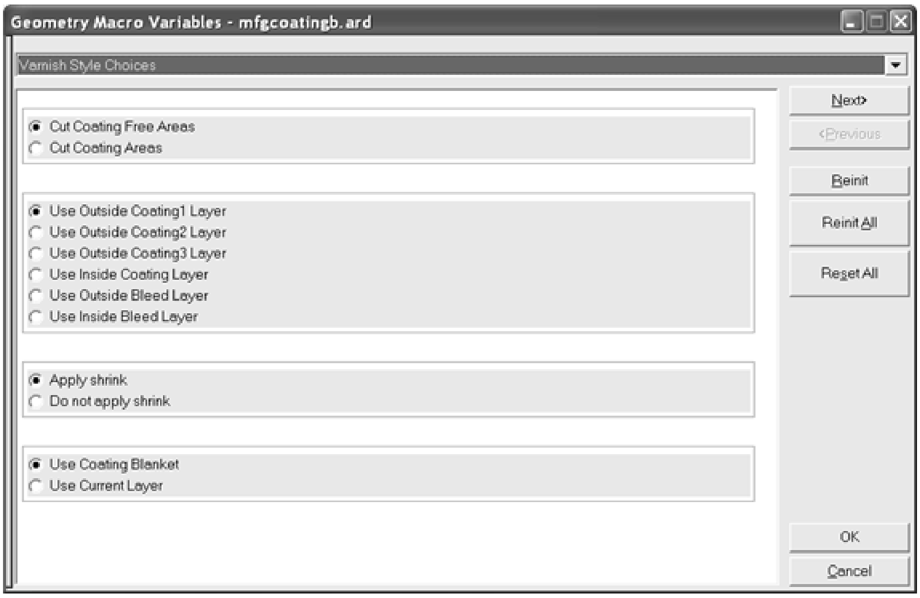
Making a coating blanket

To make a coating blanket, construct a layout of single designs containing bleed or varnish layers. Then, do the following:

1. Click **Tools**, and then click **Geometry Macros**.
2. Open the **Manufacturing Tools** folder and then the **Coating Blanket** folder by clicking the plus signs next to them.
3. Select the **Coating Blanket** geometry macro.



4. Click **OK**. As this is a geometry tool, it runs like a style in a Standards Catalog.



Cut Coating Free Areas and **Cut Coating Areas** control what is actually cut - those areas not being coated or those areas being coated. Use the former choice to make strips in the unused portion of the blanket.

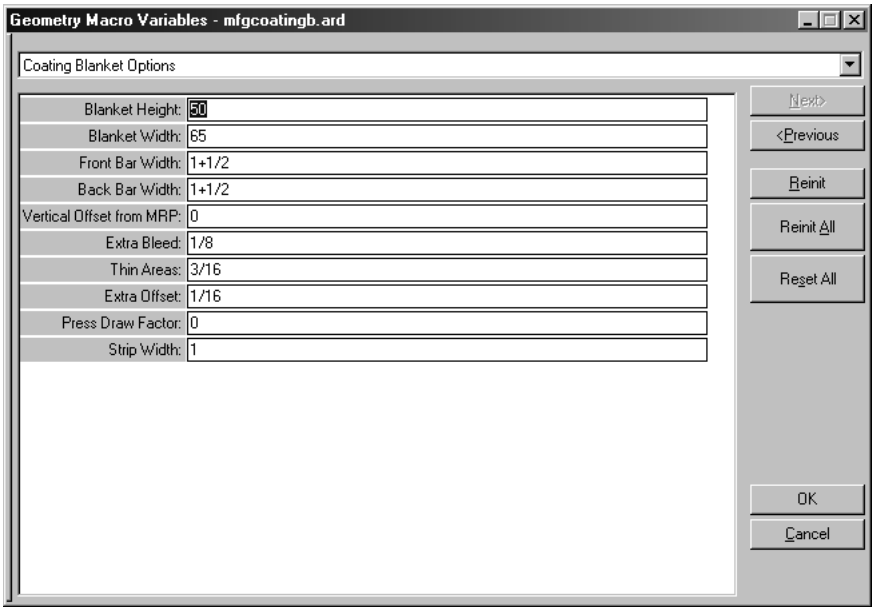
The layer choices control which single design layer is used to make the blanket. There can be six separate coating layers.

Apply shrink and **Do not apply shrink** control if the tool automatically shrinks the blanket upon creation or upon output.

Use Coating Blanket and **Use Current Layer** control where the coating blanket is created. The former choice uses the predefined layer you chose earlier in the menu. The latter uses the current layer.

Select the desired options and click **Next**.

5. The Coating Blanket Options menu is where the coating blanket settings are configured.



The variables in this menu are explained in the table below.

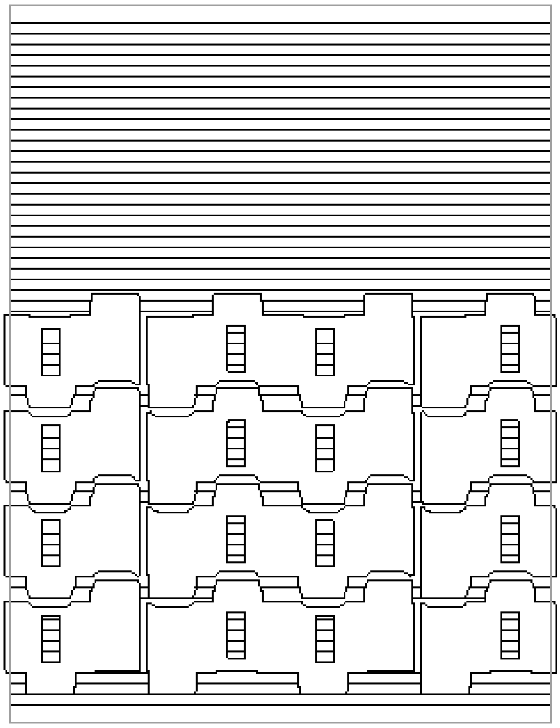
Table: Variables In The Coating Blanket Options Menu

Variable	Meaning
Blanket Height	Height of coating blanket.
Blanket Width	Width of coating blanket.
Front Bar Width	Height of the front bar.
Back Bar Width	Height of the back bar.
Vertical Offset from MRP	Offset of the coating blanket with respect to the front bar. A negative value moves the coating blanket down; a positive value moves the coating blanket up.
Extra Bleed	Extra bleed allowance to use when creating the blanket using coating-free areas.
Thin Areas	If a strip of coating material would be smaller than the width specified by this variable, it is discarded.
Extra Offset	Extra offset to be applied to the coating blanket outlines

Variable	Meaning
Draw Press Factor	Value that the interior of the blanket is shrunk in the Y direction to compensate for the blanket being mounted on a cylinder. This does not affect the bar widths or overall size of the coating blanket.
Strip Width	Width of the strips created.

Click **OK** once you have set these variables as desired.

6. The coating blanket is created. See the following picture for an example of a coating blanket with all layers turned off except for the coating blanket. You may have to turn on the **Coating Blanket** layer in the Layers dialog box to see it.



Modifying a coating blanket

Use any of the Designer tools to modify lines in the coating blanket.

To remake the coating blanket, turn off all the other overlays, select all the lines in the Coating blanket overlay, and delete them. Then rerun the geometry tool.

Configuring alternate coating blankets

The Coating Blanket geometry tool is contained in the workspace `..\InstLib\MFGCOATINGB.ARD`. To make alternate coating blankets with different default values, copy this workspace, modify the workspace variables of the new file, and then add the new workspace into the Geometry Tool Catalog in Defaults. Be very careful to not draw any lines in the workspace.

Rotary Diemaking

Instead of cutting containers out of material by using a cookie-cutter action on a stationary sheet, rotary diemaking uses rotating cylindrical dies on moving sheets. These dies can be made out of more than one piece of wood and are bolted to the machine cylinder.

The technology of rotary diemaking, while sharing many elements of flat diemaking, has special features, such as curved wood and notched rule, which are not applicable to flat diemaking. For that reason, it is a separate option from regular DieMaker.

In rotary diemaking:

- Tooling can be designed at three levels: the cylinder level, the wood level, and the rule level.
- Elements which are designed at one level are transformed (into a new size, shape, or both) when processed on another layer to compensate for the curve of the dieboard or rule.
- Tooling is made from single or multiple pieces of wood which contain mounting holes, steel rule paths, and other geometry which can be manufactured or drawn on the surface of the wood. Multiple sections can be output individually from ArtiosCAD.

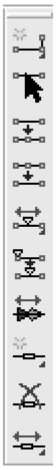
The options on the **View Options** submenu of the View menu make it easier to work with rule paths. You can toggle the display of bridges, line direction, rule path ends, and construction lines. You can also assign these menu options to Shortcut keys for faster access.

Tools for rotary dies / DieSaw output

The **Rule Path Tools** toolbar, the **DieSaw** toolbar, and the **Registration Hole Tools** toolbar contain the tools for making rotary dies and/or outputting to the Kongsberg DieSaw.

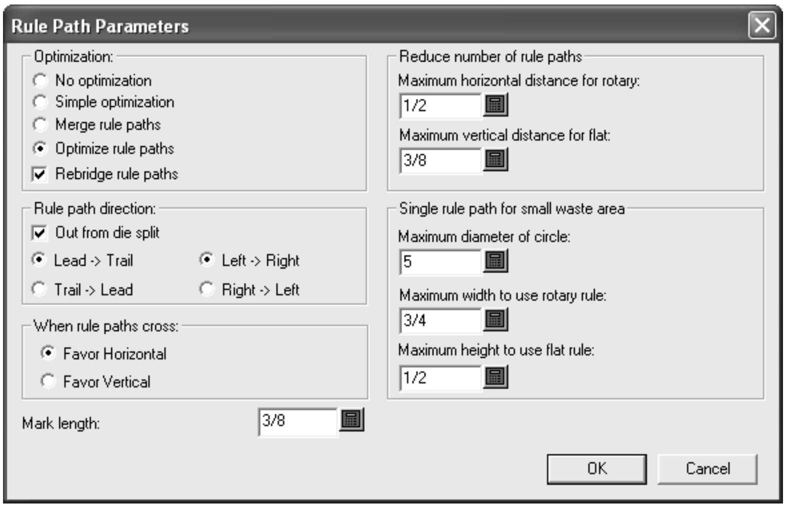
Rule Path tools

A **rule path** is the outline of the rule which is inserted into the dieboard. Diemakers strive for the most cost-efficient and structurally sound rule paths. Shown below is the Rule Path toolbar.

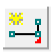


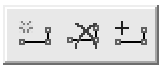
Other than the **Build Rule Path** tool, all tools on this toolbar are unavailable outside of the **Other** sublayer of the Rule Path layer. Also, while in the Rule Path layer, the tools on all other toolbars, except for those on the View toolbar and the Move tool, are unavailable so that the rule paths are not modified accidentally. You must change to another layer to use other tools.

The parameters by which rule paths are built are in the Rule Path Parameters dialog box on the Options menu in Manufacturing.

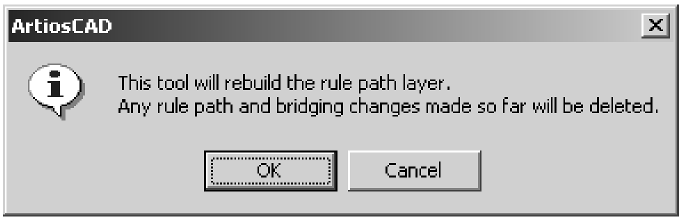


Build Rule Path tool

 The first button on the Rule Path toolbar activates the **Build Rule Path** tool. When held down, it displays the Build/Delete/Add Rule Path flyout toolbar as shown below.

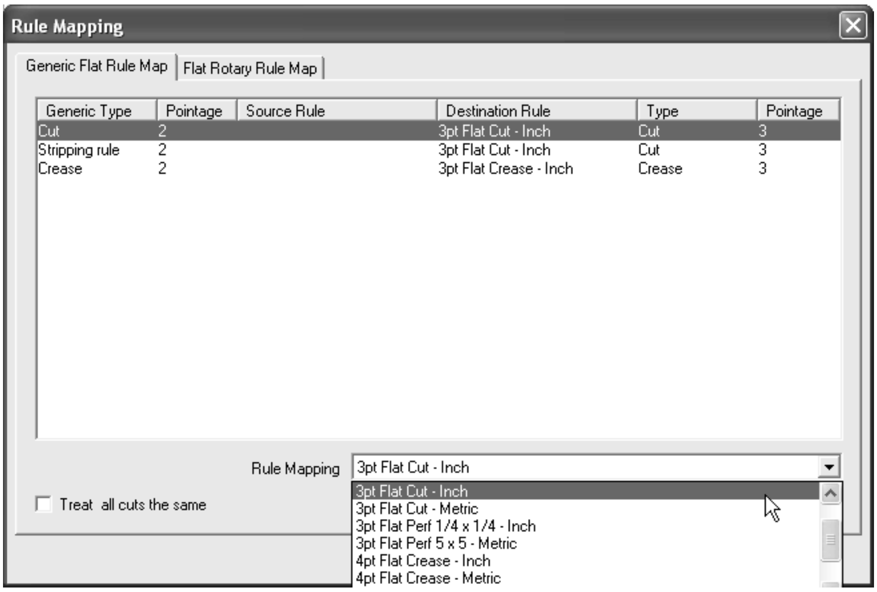


When activated, it first checks to see if double knives are present; if so, it prompts for their removal. Once the double knives issue has been resolved, ArtiosCAD asks for confirmation to rebuild the rule path layer, as any existing rule paths and bridging changes will be deleted:



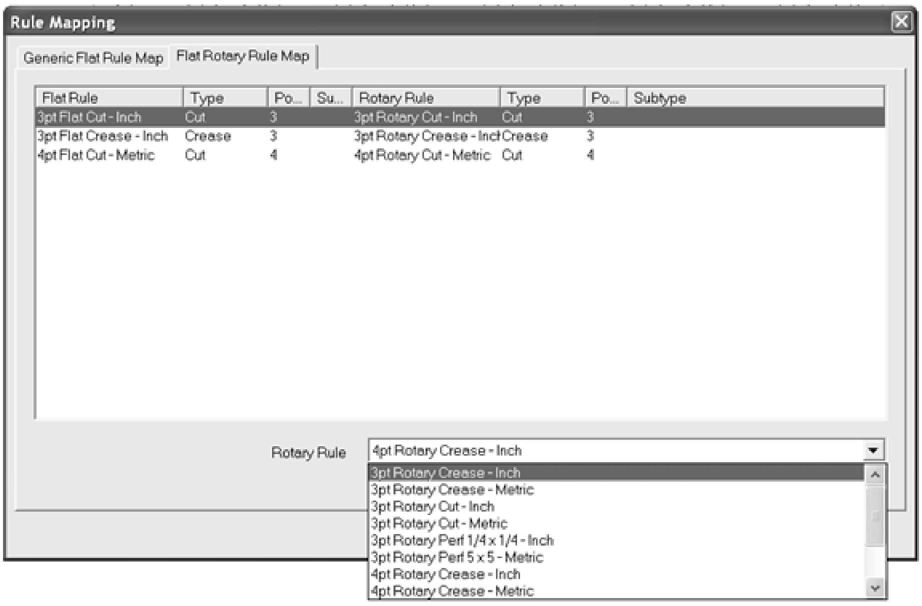
Click **OK** to continue the process of automatic rule path generation, or click **Cancel** to deactivate the tool. Once the rule mapping process is completed, the current layer will be changed to the **Other** sublayer of the **Rule Path** layer.

ArtiosCAD will prompt to map generic rules to specific special rules the first time it builds the rule paths. To do so, select the generic rule to map, click the drop-down list activator arrow at the end of the **Rule Mapping**: field, and choose the appropriate special rule type from the list displayed. If the **Treat all cuts the same** checkbox is checked, all cuts are processed with the same rule. If it is clear, you can choose different rules for each type of cut.

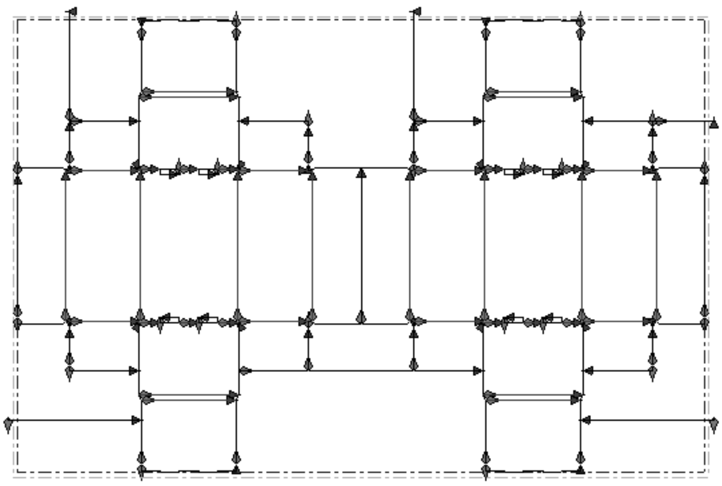


Any rule in the Special Rules Catalog which has a valid bridging formula is listed in the drop-down list box. If this is a flat die, click **OK** to proceed or **Cancel** to cancel building rule paths.

If this is a rotary die, flat rule must be mapped to rotary rule. Click the **Flat Rotary Rule Map** tab (which will only appear if this is a rotary die) as shown and verify that the mapping is correct. If the mapping is not correct, select the flat rule to change and choose a new rotary rule from the drop down list box in the **Rotary Rule**: field. If you choose a rotary rule with a different pointage from the flat rule, you will be warned and will have to click **OK** to proceed.



When you are done mapping rule, click **OK** to proceed with building the rule paths or **Cancel** to cancel. The Rule Paths layer is created and the rule paths are created in it. Rule path start points are shown as green teardrops and stop points are shown as red triangles; both point in the direction of the rule path.



Notes on building rule paths

Rule paths are grouped by rule type and pointage.

A rule path follows the contiguous outline of a design instead of branching off onto a dead-end piece of rule.

Lines sharing a piece of rule (those made by Double Knife Removal) share a rule path as well.

Outlines of the same rule type which form a contiguous waste area are grouped into one rule path.

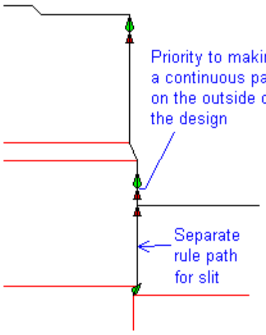
Defaults for rule paths are contained in the rule's entry in the Special Rule Types catalog and in the **Rule Path Params** entry in the selected Manufacturing parameter set.

When rule paths are created, the **Bridge from center** option is used for each rulepath which has a slot, overriding the option in the **Bridge from** group in the bridging formula for that special rule type.

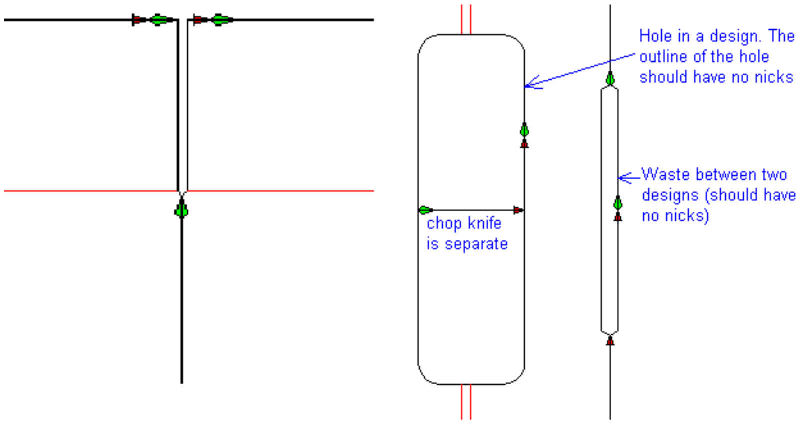
If rule paths do not wrap as expected, make sure that **Optimize Rule Paths** is selected in the **Optimization** group of the Rule Path Parameters dialog box. Be especially wary of this situation when converting a flat die to a rotary die by changing the press definition after creating the layout.

When three or more rule paths meet, the following rules are used:

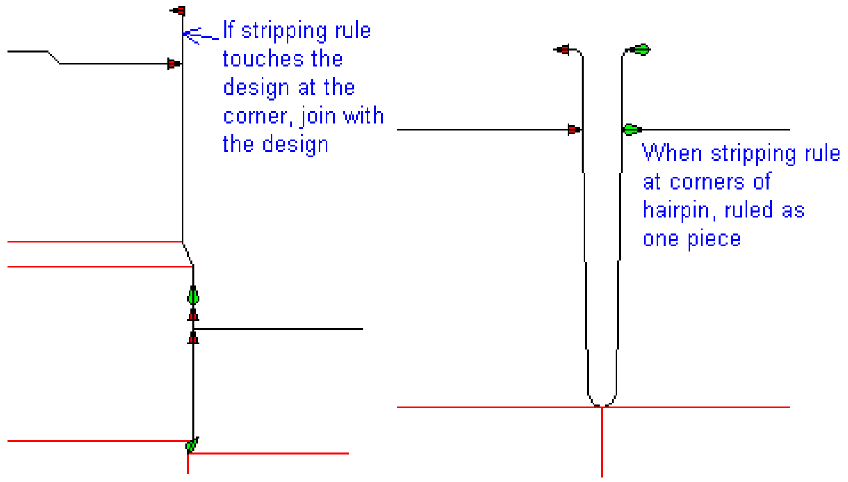
- **Separate rule paths for scrap knives**. Rule paths for horizontal and vertical scrap knives are separate. They do not join with each other or with cut lines in the design, except for a stripping rule at a corner.
- **Avoid joints at corners in the design outline**. A **joint** is a point where rules meet. With use over time, rules can bend at joints, leaving small gaps at the joints. ArtiosCAD places joints on horizontal or vertical pieces of rule by wrapping them around corners to minimize this problem.



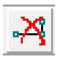
- **Avoid joints at corners in waste areas.** When two designs have no gap between them, there can not be a continuous outline. ArtiosCAD avoids joints at corners in the waste between designs.



- **Stripping rules/scrap knives at a corner**. When a stripping rule or scrap knife touches a design at the corner, it is joined with the design. When there are stripping rules at the corner of a hairpin, they are joined to form one rule path.

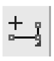


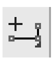
Delete Rule Path tool

 The second button on the Build/Delete/Add Rule Path flyout toolbar activates the **Delete Rule Path** tool. When clicked, you are asked to confirm the deletion of the layer and its contents:

Click **OK** to delete the Rule Path layer and its contents, or click **Cancel** to keep the layer and its contents.

Add Rule Path tool

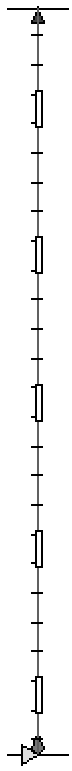
 The third tool on the Build/Delete/Add Rule Path flyout toolbar is the **Add Rule Path** tool. Use this tool when you have added lines to the layout after building the rule path layer and do not want to rebuild the entire layer. To use this tool, do the following:

1.  Use the **Select** tool on the Single Design Edit toolbar to select the lines to make into rule paths.
2.  Click **Add Rule Path**.
3. In the Rule Mapping dialog box, map the line type conversion for these new rule paths in the same way as when the layer is first built and click **OK**.
4. The Rule Path layer will be entered and the new rule paths created.

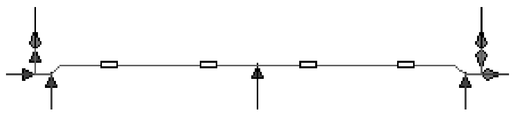
Select Rule Path tool

 The second button on the Rule Path toolbar activates the **Select Rule Path** tool. As its name implies, when the tool is active, click the mouse button to select a rule path. To select more than one rule path, hold down the `CTRL` or `SHIFT` key while clicking.

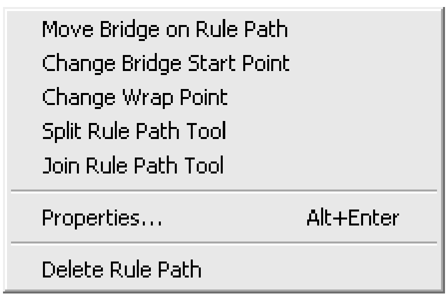
When you select a rotary rule path, ArtiosCAD shows the teeth, bridges, bridge start indicator, and end points of the rule path as shown in the following diagram.



A flat rule path only shows bridges and the start and end of the rule path.



Right-clicking a rule path once stops the current tool and activates the **Select Rule Path** tool. Right-clicking a rule patch twice activates a context menu as shown below:



The tools on this menu are also on the Rule Path toolbar, and are explained as the chapter progresses.

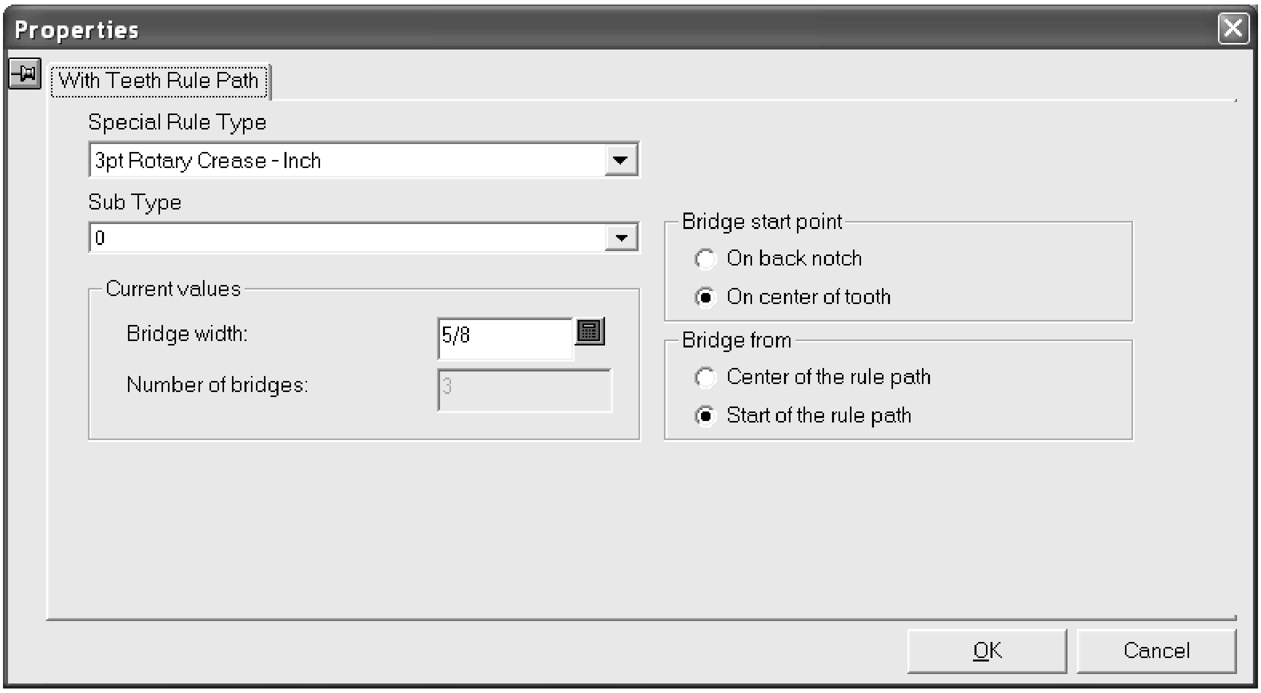
Changing rule path properties

To change the special rule type a rule path is based upon, or to rebridge a rule path, do one of the following:

- Activate the **Select Rule Path** tool and double-click the rule path(s) to change;
- Activate the **Select Rule Path** tool and select the rule path(s) to change and press `ALT-Enter`;
- Activate the **Select Rule Path** tool, select the rule path(s) to change, and click **Edit** > **Properties**.

Note: Do not change the properties of a rule path by using the **Select** tool to make a multiple selection of it along with other types of manufacturing elements. The Properties dialog box will show **Can not display properties for mixed selection of rule path and other items** instead of properties for the selected items.

The Properties dialog box will appear. Rules with teeth have different properties than rules without teeth.



Clicking the drop-down list activator arrow in the **Special Rule Type** field displays a list of available special rule types. Click the new special rule type for the rule path. If you select a special rule type which does not have a teeth bridging formula defined, the dialog box will change to the Without Teeth Rule Path Properties dialog box shown on the next page.

Clicking the drop-down list activator arrow in the **Sub Type** field displays a list of available subtypes. If desired, choose a subtype for the rule.

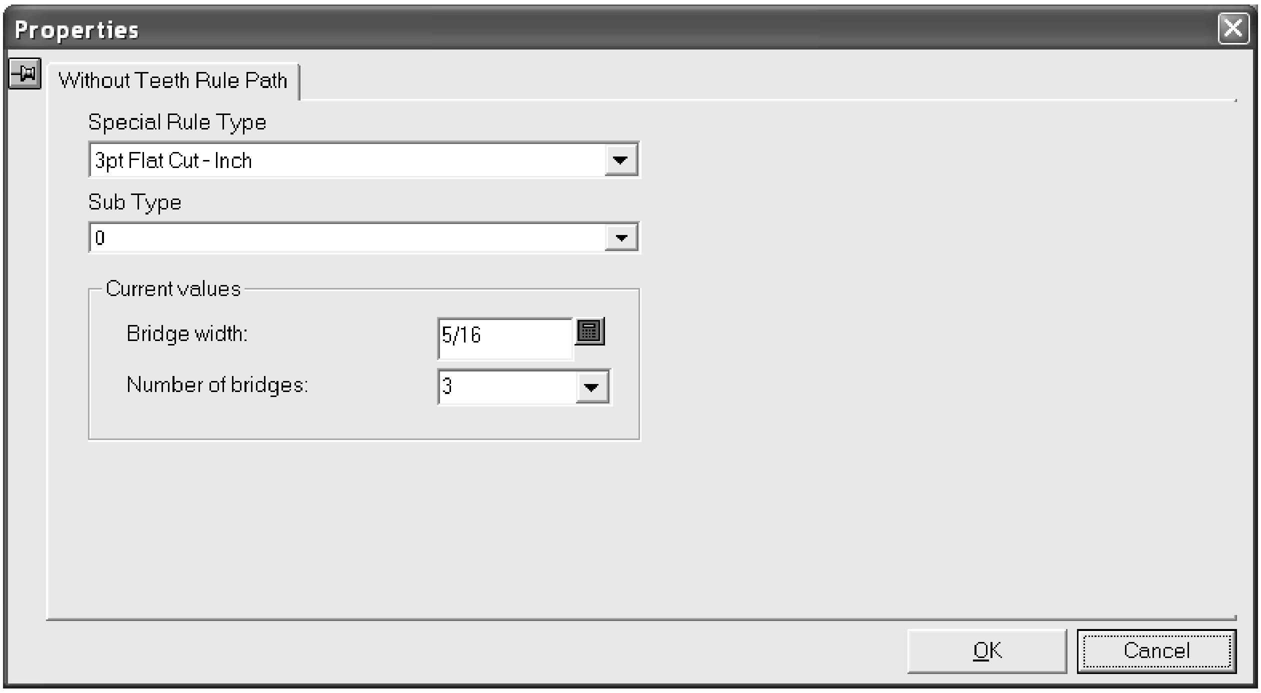
The options in the **Bridge start point** group control the position of the start of the bridge in the direction of the rule. When this option is changed, the bridges on the rule path are moved by half a tooth.

The options in the **Bridge from** group controls the position at which bridging starts. When this option is changed, the bridge start position is moved and the rule path is rebridged. This option applies only to rule paths without slots.

To change the bridge width on the selected rule(s), enter a new value in the **Bridge width**: field. If the bridge width is changed to zero and then another value, the rule path is rebridged and the bridge start point will be set according to the default bridging formula for that special rule type.

The number of bridges on a rule with teeth cannot be changed manually.

When a special rule type without teeth is selected, the Without Teeth Rule Path Properties dialog box is shown:



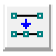
Clicking the drop-down list activator arrow in the **Special Rule Type** field displays a list of available special rule types. Click the new special rule type for the rule path. If you choose a rule which has a teeth bridging formula defined, the dialog box will change to the With Teeth Rule Path Properties dialog box shown previously.

Clicking the drop-down list activator arrow in the **Sub Type** field displays a list of available subtypes. If desired, choose a subtype for the rule.

Change the bridge width and number of bridges by changing the values in the **Current values** group. If the number of bridges is set to zero, the bridge width is also set to zero. If you change the number of bridges to zero and then to some other number, the bridge width is calculated from the default formula for that special rule type, and the new bridge width is displayed in the **Bridge width**: field. If the bridge width is changed to zero and then to some other number, the number of bridges is set to one and the new bridge will be placed in the middle of the longest line in the rule path.

Click **OK** to make the changes indicated in the Properties dialog box, or click **Cancel** to discard them.

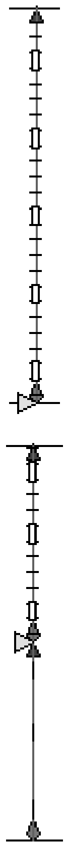
Split Rule Path tool

 The third button on the Rule Path toolbar activates the **Split Rule Path** tool. It is also on the Select Rule Path tool context menu. When this tool is activated, it displays the following controls on the Status bar.

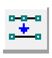
The first button, **Offset**, splits the rule at the nearest start/end point of the rule path with the offset specified in the **Offset**: field. **Center**, the second button, splits the selected rule path at its center. The third button, **Anywhere**, splits the rule at the nearest coordinate to the mouse click.

Choose a placement method and then click the rule to split.

Shown below is a rule path before and after being split at its center.

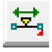


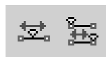
Join Rule Path tool

 The fourth button on the Rule Path toolbar activates the **Join Rule Path** tool. It is also available on the context menu of the Select Rule Path tool. To use it, click it, indicate the first rule path to join, and then indicate the second rule path to join. The lines on which the rule paths are based must share an endpoint. The resulting rule path will be the same special rule type as the first path selected.

Only rule paths in the same section of the dieboard can be joined. Rule paths can not be joined across dieboard splits.

Move Bridge Start Point tool

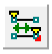
 The fifth button on the Rule Path toolbar activates the **Move Bridge Start Point** tool, and when held down, activates a flyout toolbar containing this tool and the **Reverse Rule Path** tool.



The **Move Bridge Start Point** tool moves the start point of the bridging on the rule path.To use it, click it, select the rule path to modify, and then drag the start marker to its new position.

Changing the bridge start point will automatically cause ArtiosCAD to rebridge the rule path using the rule path's bridging formula defined in Defaults.

Reverse Rule Path tool

 The second button on the flyout toolbar activates the **Reverse Rule Path** tool. This tool exchanges the start of the rule path with the end of the rule path. This will also change the current bridge start point from one end of the rule path to the other if the special rule path is defined to not start bridging at the center of the rule path.

Only rotary rule paths can be reversed.

Move Bridge Start Point to Center tool

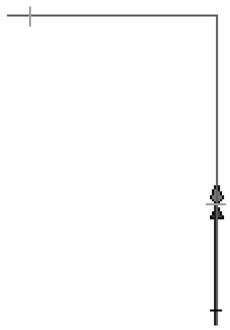
 The sixth button on the Rule Path toolbar activates the **Move Bridge Start Point to Center** tool. To use this tool, click it, and then click the rule path to change. It is intended to be used on rules with teeth; if you select a rule without teeth, **Bridge start point is not defined** appears in the Status bar and the tool continues to operate until you choose a rule with teeth or another tool.

Change Wrap Point tool

 The seventh button on the Rule Path toolbar activates the **Change Wrap Point** tool. It is also available on the context menu for the Select Rule Path tool. This tool moves the wrap point, which is the junction between flat rule and rotary rule.

To use this tool, do the following:

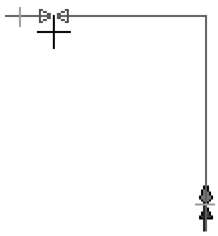
1.  Click the **Change Wrap Point** tool.
2. Click the wrap point to move. When three or more rule paths meet, you can select the wrap point either by a single click on the wrap point or by clicking each rule path separately. Shown below is a selected wrap point centered on the suggested wrap margin, with another suggested point on the other side of the corner.



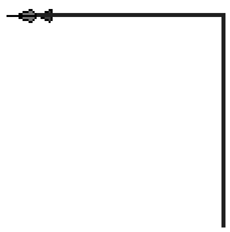
3. There will be drag markers of new snap positions shown at the wrap distance before and after each bend in the rule.

   If the bridge start point is the same as the current wrap position, the bridge start point will move with the wrap point. If the bridge start point is not at the wrap point, suggested snap positions are shown for each back notch position measured from the bridge start position.

   Move the drag of the wrap point to the new position.

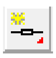


4. Click the mouse to set the position of the new wrap point.



You can also use this tool to move the start point of a looping rule path without changing the bridge starting point.

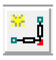
Add Bridge on Rule Path tool

 The eighth button on the Rule Path toolbar activates the **Add Bridge on Rule Path** tool, and when held down, activates a flyout toolbar containing it, the **Re-bridge Rule Path Layer** tool, and the **Add Mark to Rule Path** tool.

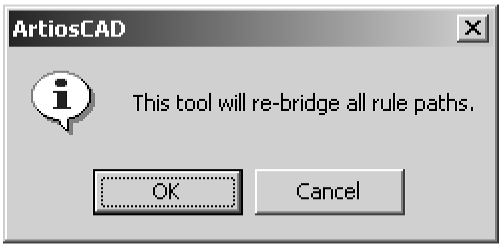
To use this tool, click it and then click the position for the new bridge. The rule path will be selected and the new bridge will appear. Click another position along the rule path to add another bridge.

If a rule path needs to have bridges of different widths on it, that rule path must be split first, and bridge width changed on each piece of rule separately.

Re-bridge Rule Path Layer tool

 The second button on the flyout toolbar activates the **Re-bridge Rule Path Layer** tool.

When you click this tool, ArtiosCAD displays a warning dialog box:



Clicking **OK** deletes all the bridges and re-bridges the rules according to their bridging formulae configured in Defaults. Clicking **Cancel** returns you to Manufacturing.

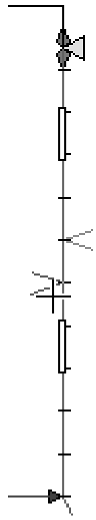
Add Mark to Rule Path tool

 The third button on the flyout toolbar activates the **Add Mark to Rule Path** tool. This tool adds a mark to a rule path and snaps to any of these points: the rule path start position, the bridge start position, and back notch positions. Marks are added as grouped lines and can be selected by the **Select** tool.

To use this tool, do the following:

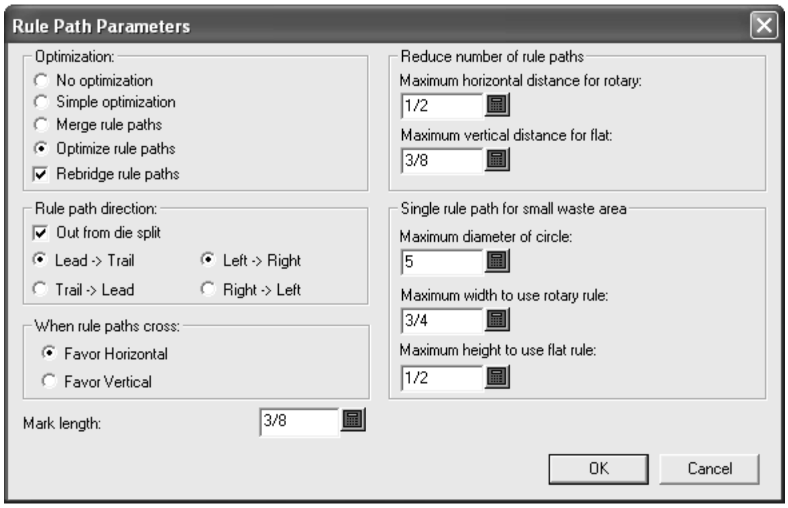
1.  Click the **Add Mark to Rule Path** tool.
2. Select the shape of the rule mark to add by clicking its button on the Status bar. You can change the size of the mark about to be added in the **Mark length**: field.

3. Click the rule to which the mark will be added.
4. Drag the mouse to position the mark as desired. If the marker is not symmetrical, it is drawn on the same side of the line as the cursor. Click the point at which to add the marker.



5. To continue adding marks, repeat steps 3 and 4.

To change the mark length for new marks when not using the tool, change the **Mark length**: field in the Rule Path Parameters dialog box on the Options menu in Manufacturing.



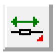
To change the default mark length, change the **Mark length**: field in the Rule Path Params section of the Manufacturing parameter set.

Delete Bridge on Rule Path tool

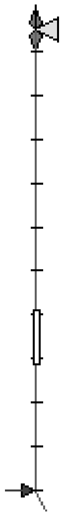
 The ninth button on the Rule Path toolbar activates the **Delete Bridge on Rule Path** tool. The viewing of bridges is automatically turned on when this tool is activated.

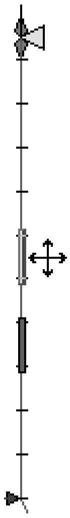
To delete a bridge, select it with the Select Rule Path tool, then click the **Delete Bridge on Rule Path** tool and click the bridge to delete. Click the **Select Rule Path** tool again, select the rule, and note that the bridge is gone.

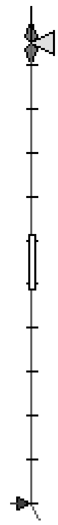
Move Bridge on Rule Path tool

 The tenth button on the Rule Path toolbar activates the **Move Bridge on Rule Path** tool, and when held down, activates a flyout toolbar containing it and the Align Bridge on Rule Path tool.

The viewing of bridges is automatically turned on when this tool is activated. To use the **Move Bridge on Rule Path** tool, click it, then click and drag the desired bridge to its new position.







This tool can also add and delete bridges. To add a bridge, click a bridge and hold down `CTRL` while dragging it. To delete a bridge, click it and press `Delete` on the keyboard.

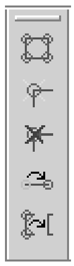
Align Bridge on Rule Path tool

The second button on the Bridge Position flyout toolbar activates the **Align Bridge on Rule Path** tool. This tool aligns bridges either horizontally or vertically with bridges in other rule paths. Aligning bridges is made much easier by turning bridges on in the View Mode dialog box

To use this tool, click it, indicate the bridge to align others with (called the **anchor bridge**), and then indicate the other bridges.

DieSaw / Leader Hole Tools toolbar

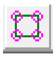
The DieSaw / Leader Holes Toolbar contains tools for making leader holes. Leader holes are used by the DieSaw when it is sawing a dieboard, upper stripping board, or lower stripping board. The toolbar is shown below.

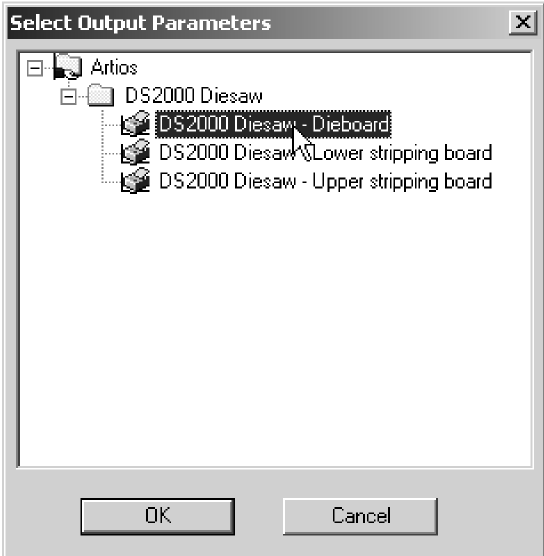


Leader hole creation is configured in the Kongsberg DieSaw group on the Processing page in an Output definition. The physical tool they are created with is configured in the CAM Tooling Setup Catalog.

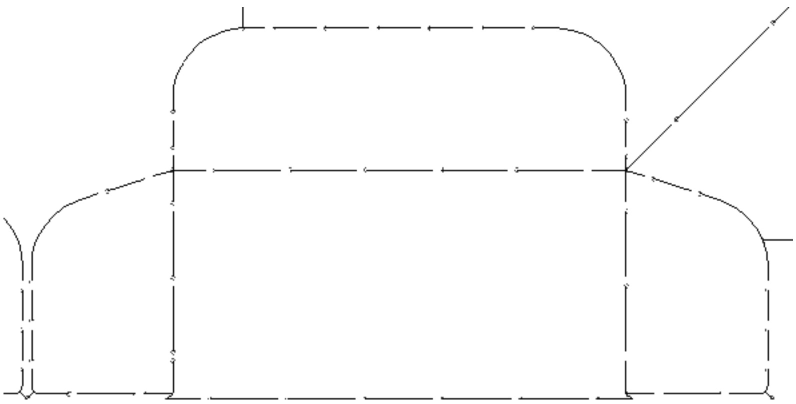
Note: If you are outputting to a DieSaw and you have not made any leader holes yourself (for example, if you know you will not need to adjust them), they are created automatically during the output procedure and deleted when the output is done.

Generate Leader Holes tool

 The first tool on the DieSaw / Leader Hole Tools toolbar activates the **Generate Leader Holes** tool. This tool automatically generates leader holes. After you click **Generate Leader Holes,** ArtiosCAD will prompt you to select an Output so that it can read the leader hole parameters stored in the Output definition. Only those Outputs configured to generate leader holes are listed. Select an Output and click **OK** to proceed, or click **Cancel** to cancel.



Once you click **OK**, ArtiosCAD changes to the layer specified on the View tab of the Output definition (Dieboard, Upper stripping board, or Lower stripping board), creates a DieSaw sublayer, copies the contents of the appropriate layer to the DieSaw sublayer and changes them to the appropriate Manufacturing-class line types, and creates leader holes in that sublayer. The lines created are in the Manufacturing category of the plotting style. Shown below is a magnified view of a dieboard with leader holes.

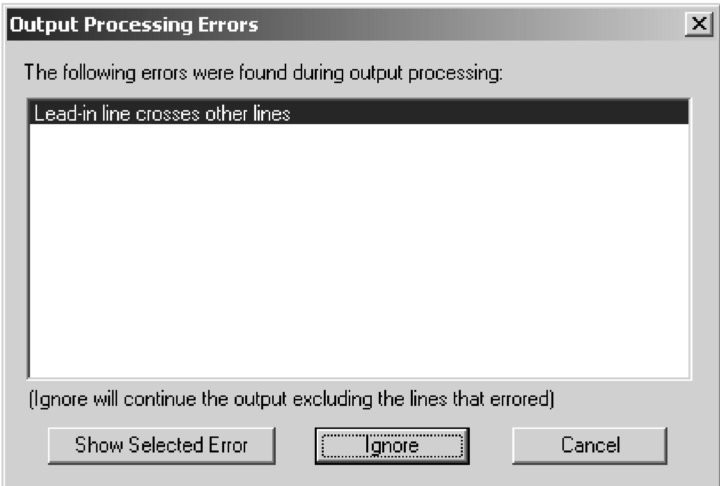


Leader Hole error checking

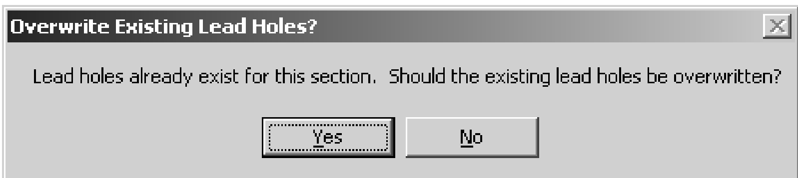
ArtiosCAD performs a series of checks on the structure of the manufacturing file before generating the leader holes. If there are problems, an error dialog box appears listing the error(s). Possible errors are:

- Drill hole is not a circle;
- Inside edge lines do not form a loop;
- Outside edge lines do not form a loop;
- Mill outline does not form a loop;
- Cannot offset inside edge lines;
- Cannot offset outside edge lines;
- Cannot offset mill outline;
- Cannot offset side bevel lines;
- DieSaw output with no SAW tools defined;
- Die split lines do not connect to the dieboard edge;
- Saw lines have no leader hole;
- Enveloped corner with saw not recommended;
- Lines enveloped with saw.

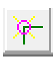
In the Output Processing Errors dialog box, you can choose to view the problem and not generate the leader holes by clicking **Show Selected Error**, or ignore the problems and generate the leader holes anyway. Clicking **Ignore** causes the offending lines to not be included in the layer, so use this option carefully.



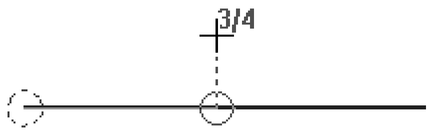
If you have already generated leader holes in this manufacturing file, you will be prompted to overwrite the existing ones. To overwrite the existing holes, click **Yes**; any manual changes you made to the leader holes will be lost if you click **Yes**. Clicking **No** initiates no changes.



Add Leader Hole tool

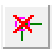
 The second button on the DieSaw / Leader Hole Tools toolbar activates the **Add Leader Hole** tool. To add a leader hole, click the tool, and then click the location of the desired leader hole.

If the location clicked is within the snap distance of an end point of a rule, the leader hole snaps to the end point; otherwise, the cursor changes to a cross-hair and you have to drag along the rule to set the position of the leader hole as shown in the picture below. Drag away from the rule to create a lead-in line.



Note: No checks are performed when adding leader holes manually to ensure that they do not interfere with each other.

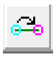
Delete Leader Hole tool

 The third button on the DieSaw / Leader Hole Tools toolbar activates the **Delete Leader Hole** tool. To delete a leader hole, click the tool, and then select the leader hole to delete.

All rules that used the deleted leader hole will be reversed and a leader hole will be added at the other end if needed.

If the deleted leader hole had a lead-in line, the lead-in line will be deleted as well.

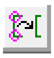
Reverse Cut tool

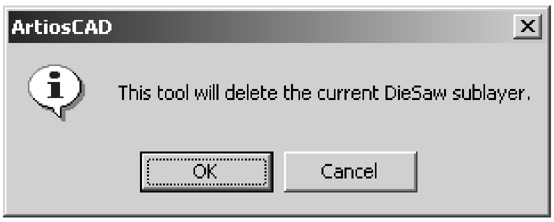
 The fourth button on the DieSaw / Leader Hole Tools toolbar activates the **Reverse Cut** tool. This tool reverses the sawing direction of the rule and moves the leader hole to the other end of the rule.

If the rule segment shares a leader hole with another rule segment, the leader hole will move to the other end of the selected rule segment and the shared leader hole will be kept.

If the leader hole for the segment uses a lead-in line, the rule segment will reverse and add a leader hole at the opposite end from the current leader hole and not include a lead-in line.

Delete DieSaw Layer tool

 The fifth button on the DieSaw / Leader Hole Tools toolbar activates the **Delete DieSaw Layer** tool. When clicked, this tool will ask for confirmation, and if you click **OK**, it will delete the contents of the DieSaw layer. If you click **Cancel**, nothing will happen.



Notes on Leader Holes

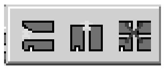
If there is a milling area at the intersection of lines which would otherwise share a leader hole, the rules are reversed so that each line has an individual leader hole and cuts into the area that is milled.

Dieboard Split tools

The Dieboard Split tools are on the Dieboard toolbar. They are active only when the **Dieboard** layer is the current layer.

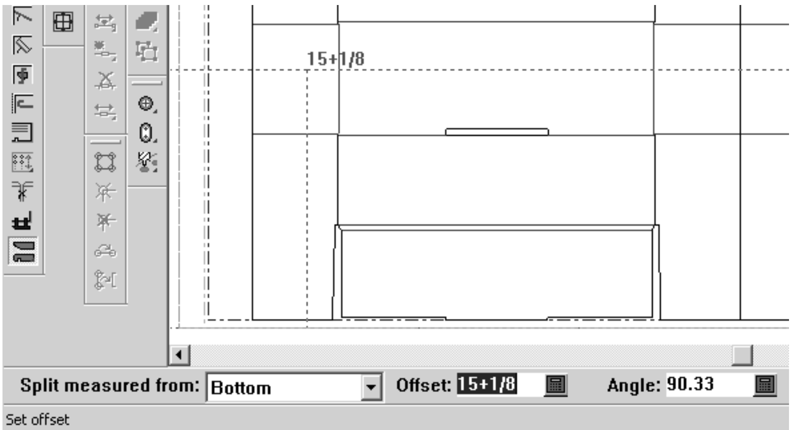
Horizontal Split tool

 The ninth button on the Dieboard toolbar activates the **Horizontal Split** tool, and when held down activates the Dieboard Split Tools flyout toolbar.



To add a horizontal split, click the tool, drag the ghost split to the desired position, and click to set it in place. Tack bridges are created on the lines forming the split.

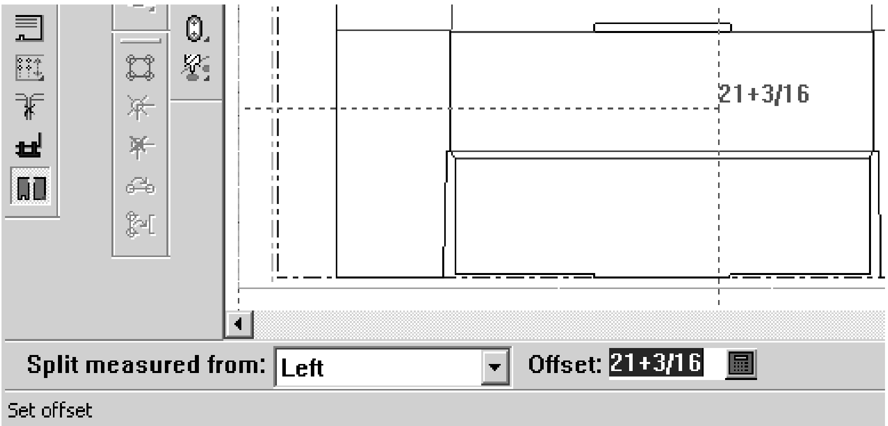
The options on the Status bar will let you choose either the Top or the Bottom edge of the die to measure the **Offset** from. The **Angle** refers to the angle **through the center of the cylinder** that this split creates.



Vertical Split tool

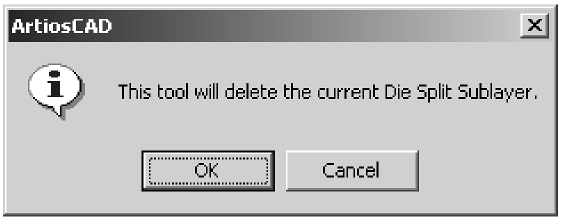
 The second button on the Dieboard Split Tools flyout toolbar activates the **Vertical Split** tool. This tool works similarly to the Horizontal Split tool except the split is measured from the left or right side of the die depending on the selection in the Status bar. Tack bridges are created on the lines forming the splits.

Vertical splits are not at an angle, so no angle is displayed.



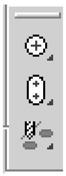
Delete Split tool

 The third button on the Dieboard Split Tools flyout toolbar activates the **Delete Split** tool. When clicked, this tool asks for confirmation, and if OK is clicked, it deletes the contents of the Die Splits sublayer. Click **Cancel** to keep any dieboard splits.



Registration Hole Tools toolbar

The **Registration Hole Tools** toolbar contains the tools necessary for adding holes to the workspace.

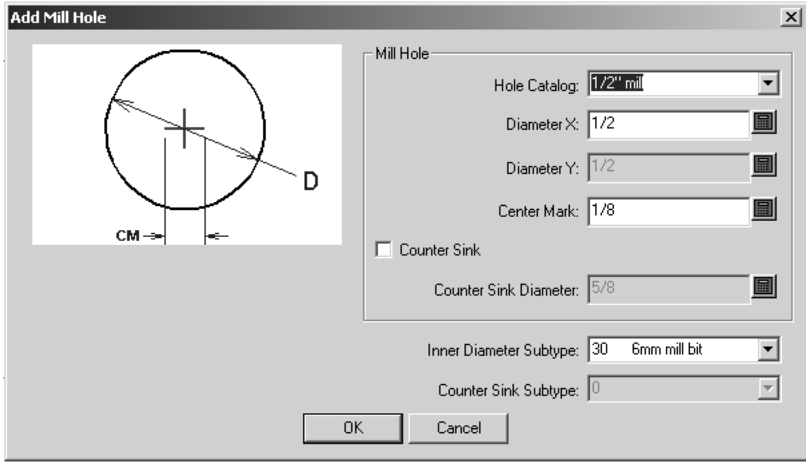


Add Hole tool

 The first button on the Registration Holes toolbar activates the **Add Hole** tool, and when held down, activates the Add Holes flyout toolbar.

To add a hole, click **Add Hole**. The Status bar will change to show hole creation options.

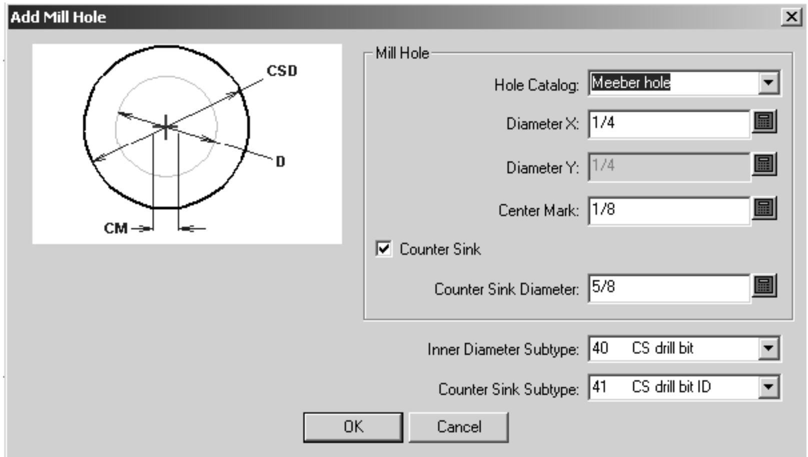
Choose the type of general hole to add from the drop-down list box. Click More options (**...**) to invoke the Properties dialog box for the hole, where you can modify the hole parameters before adding it.



When you are done modifying the properties for the hole, click **OK** to save the changes and return to Manufacturing, or click **Cancel** to discard the changes and return to Manufacturing. To set the position of the hole, click at the desired location. The tool repeats until you click **OK** or select another tool.

Add Countersunk Hole tool

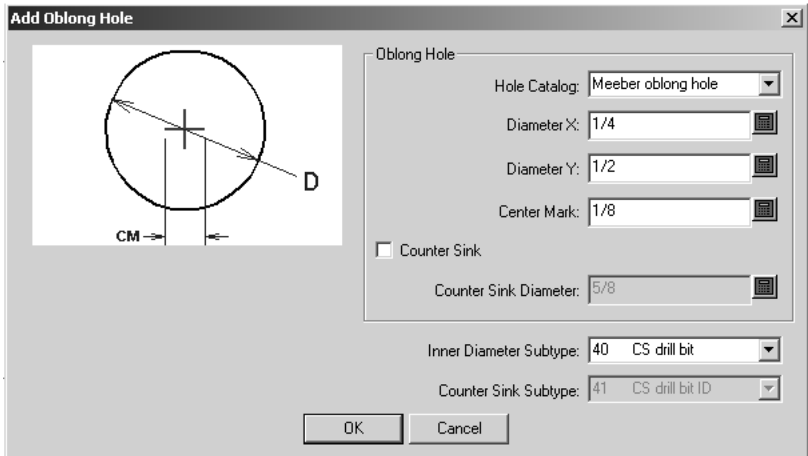
 The second button on the Add Holes flyout toolbar activates the **Add Countersunk Hole** tool. This tool works in the same fashion as the Add Holes tool except it adds holes which are defined as countersunk. Select the type of hole to add from the drop-down list box on the Status bar, modify its properties if necessary by clicking the More options (...) button, click the locations at which to place a hole, and click **OK** to end the tool. Shown below is the Properties dialog box for an example countersunk hole.



Add Oblong Hole tool

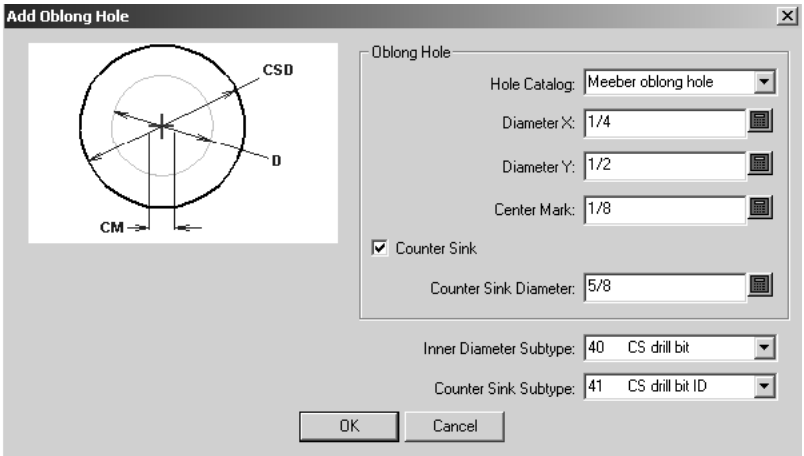
 The second button on the Registration Hole Tools toolbar activates the **Add Oblong** Hole tool, and when held down, activates the Add Oblong Holes flyout toolbar.

This tool adds a non-circular hole to the workspace. It works the same way as the other hole tools except you can specify different X and Y diameters for the hole diameter.



Add Countersunk Oblong Hole tool

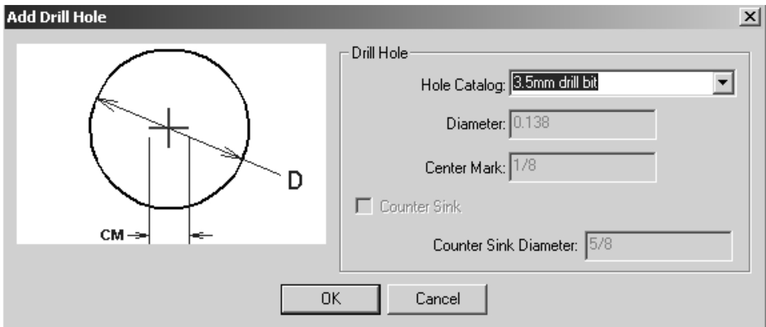
 The second button on the Add Oblong Holes flyout toolbar activates the **Add Countersunk Oblong Hole** tool. Its method of operation is similar to the other Add Hole tools, except that it is countersunk.



Add Drill Hole tool

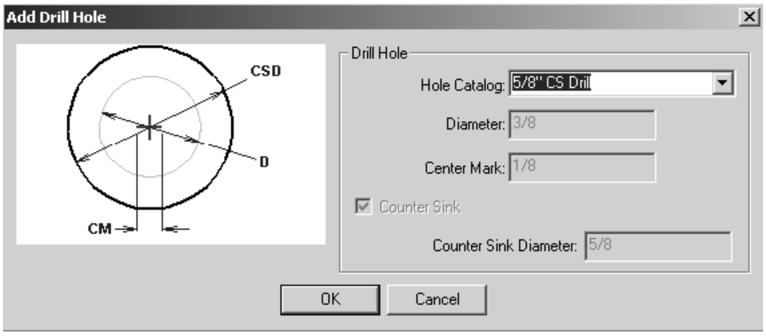
 The third button on the Registration Hole Tools toolbar activates the **Add Drill Hole** tool, and when held down activates the Add Drill Hole Tools flyout toolbar.

This tool works the same as the other hole tools, except on its Properties page, you can only change the type of hole being made. No other configuration options are available when adding the hole.



Add Countersunk Drill Hole tool

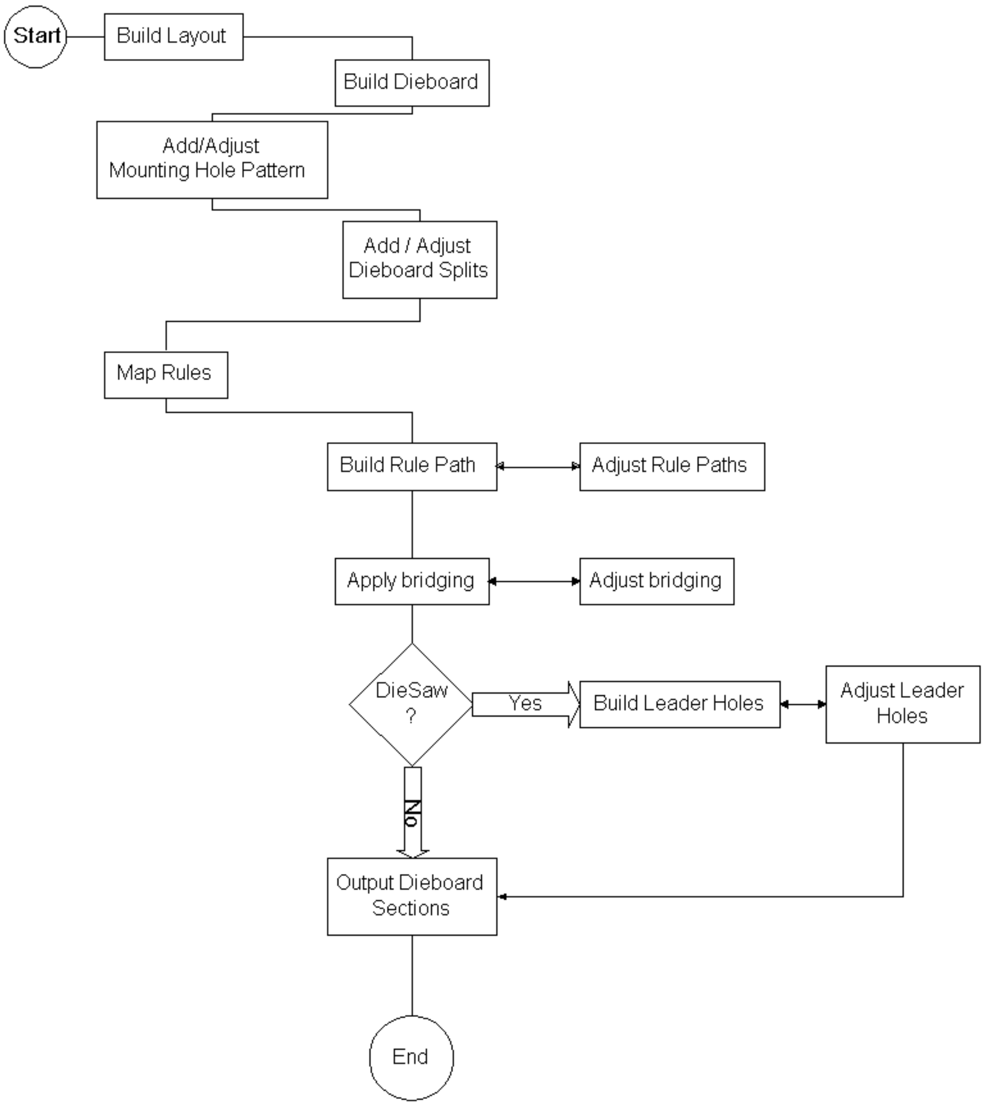
 The second button on the Add Drill Hole tools flyout toolbar activates the **Add Countersunk Drill Hole** tool. Its mode of operation is the same as for the Add Drill Hole tool. Using the **Hole Catalog** drop-down list box, only the type of hole being made can be changed on its Properties page; the **Diameter**, **Center Mark**, **Countersink checkbox**, and **Counter Sink Diameter** fields are unavailable.



Rotary Diemaking Workflow

This is a flowchart of a possible workflow, followed by a step-by-step explanation using the tools described earlier in the chapter.

Rotary Diemaking Workflow



1. Design and save the single design(s) to include in the manufacturing file.
2. Create the manufacturing file and arrange the designs as desired.
3. Create the dieboard and stripping rules.
4. Add the mounting holes and adjust if necessary.
5. Add the dieboard splits and adjust if necessary.
6. Map the flat rules to rotary rules.
7. Build the rule path and adjust if necessary.
8. Apply the bridging and adjust if necessary.
9. Is the dieboard being made with a DieSaw? If so, build the leader holes and adjust them if necessary.

10. Output the dieboard sections.

Prerequisites

To follow the sample workflow, ArtiosCAD must have the following Defaults defined:

- A Bridging On Teeth formula set up and saved;
- Rotary rules created and saved in the Special Rules Catalog;
- A rotary press defined;
- A dieboard which uses an alignment method other than notch;
- Sheet allowances on side-to-side, but 0 on front-to-back;
- An Output defined for rotary dieboards, which on the View tab has the Rule Path layer being output at the Wood level;
- A Mounting Hole Pattern (MHP) defined.

Most of these defaults have example entries. Copy the examples and modify them with your specific information.

Using Layers

When you change layers, the active tool stops and no new tool starts. This prevents putting elements in the wrong layer.

The layers that are displayed when a manufacturing file is opened depends on the setting in **Options** > **Defaults** > **Shared Defaults** > **Design Defaults** > **Default View Mode**. You can choose to display the layers as they were when the manufacturing file was saved, or you can choose to always display the same set of layers.

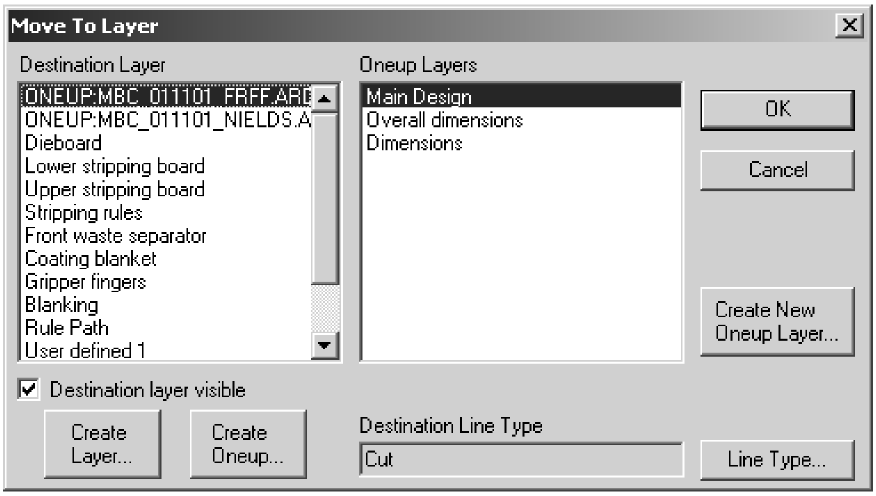
Default layers in Manufacturing

You can specify layers that will be created in new manufacturing files and in existing manufacturing files when they are opened. See the *Defaults* chapter of the *ArtiosCAD Administrator Guide* for more information.

Move to Layer tool

The **Move to Layer** tool in Manufacturing works similarly to its namesake in Single Design, but it offers greater functionality as objects can be moved into layers within included designs as well as layers in the manufacturing file.

To use this tool, select the object(s) to move to another layer using the Select tool on the Single Design Edit toolbar, and click the tool, located on the Move tools flyout toolbar on the Single Design Edit toolbar. The enhanced Move To Layer dialog box will appear similar to the one shown below.



Only unlocked layers are listed.

To move the objects into a different layer of the manufacturing file, pick the layer in the **Destination Layer** pane, and then pick the sub-layer in the right pane. To create a new layer in the manufacturing file, click **Create Layer**, enter the name for the layer in the resultant field, and click **OK**.

To move the object(s) into an existing single design embedded in the manufacturing file, select the single design in the left pane (its name will be preceded by the word ONEUP), and then select the desired layer of the selected single design in the right pane. Click **OK** to perform the move.

To create a new embedded single design in the manufacturing file, click **Create Oneup**, enter its name, and click **OK**. The new single design will appear in the list of destination layers. You can then select it and create new layers in it if desired.

To create a new layer in the single design, click **Create New Oneup Layer**, name the layer, select its class, and click **OK**.

The **Destination layer visible** checkbox turns the selected layer in the ArtiosCAD window on or off while the Move To Layer dialog box is open.

The line type that the selected object(s) will be changed to upon moving is shown in the **Destination Line Type**: field. Clicking **Line Type** lets you change the destination line type. If more than one item is selected, and they are all the same type, that type will be shown in this field. If they are of different types, UNCHANGED is shown.

When the selected Destination Layer is a manufacturing element, the Destination Line Type shown is the default line type for that element, and the Line Type button is unavailable. For example, when the Destination Layer is the **Dieboard** and the Manufacturing Element is **Hand hold**, the **Destination Line Type** is **Die registration hole** and the Line Type button is unavailable.

When moving items, if an entire group is being moved, it will remain a group after the move. If only part of a group is moved, those items will be removed from the original group and ungrouped when moved. The rest of the original group will remain a group.

Clicking **Cancel** dismisses the dialog box and does not move the selected items. However, any layers or single designs that have been created using **Create Layer**, **Create Oneup**, or **Create New Oneup Layer** will remain.

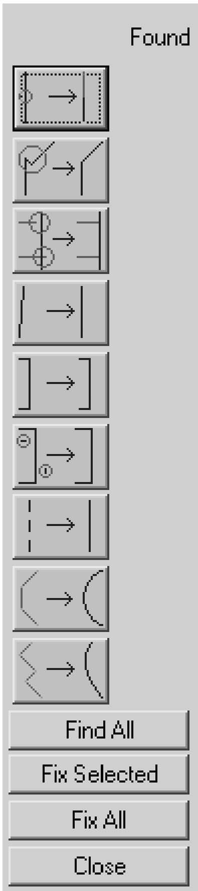
Designer's Fix-It

Designer's Fix-It is a tool on the **Manufacturing** menu in Manufacturing and the **Design** menu in Single Design. It cleans up and reorganizes geometric data imported into ArtiosCAD from other sources.

Designs may look fine on the surface, but errors called *anomalies* can lurk below and ruin a resultant tooling set and production run.

The general workflow is to import the data, use the **Move To Layer** tool to create single designs out of the oneup data, open those embedded designs and run Designer's Fix-It, and then run Designer's Fix-It in Manufacturing to clean up the non-embedded-design data.

When you start Designer's Fix-It, a toolbar appears to the right of the drawing area.



Clicking an icon on the toolbar invokes the particular tool. If the tool finds no anomalies, a dash is displayed in the **Found** column. If the tool finds anomalies, the total is shown in the **Found** column, and the anomalies are highlighted in the workspace.

To find all anomalies at once, click **Find All**.

Anomalies which can be fixed are dark blue when the screen background is light, and light blue when the screen background is dark. Anomalies which cannot be automatically fixed are light blue when the screen background is light and dark blue when the screen background is dark.

To select an anomaly, click it; to select more than one, hold down `SHIFT` and click the individual anomalies, or hold down `CTRL` and use a selection window. Selected anomalies will turn magenta.

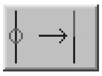
To fix the selected anomalies, click **Fix Selected**. To fix all the anomalies, click **Fix All**.

Some of the tools in Designer's Fix-It have parameters which appear in the Status bar. These parameters let you adjust what is an anomaly and what is not.

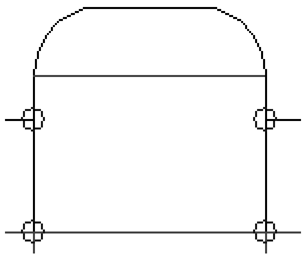
Note: The tools in Designer’s Fix-It do not search for anomalies in any of the Graphics layers. Also, the tools work on lines and arcs only; if there are beziers in the workspace, select them all and convert beziers to arcs as desired using **Tools** > **Adjust Outline** > **Change Beziers to Arcs** in Single Design. Using Designer's Fix-It in an existing ArtiosCAD workspace disables StyleMaker. You will be prompted to save the design before proceeding, disable rebuild and proceed without saving, or to exit from the selected feature.

Tool descriptions and uses

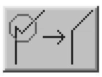
Split Lines tool

 **Split lines.** These are lines that intersect other lines at points not at the end of the line. When you click the icon, the tool intersects each line in the workspace with every other line, and searches for intersections which are not at the end-points of lines. ArtiosCAD performs the intersections invisibly.

Shown below are intersections that do not occur at two end-points. However, these are acceptable anomalies based on the basic structure of the carton.

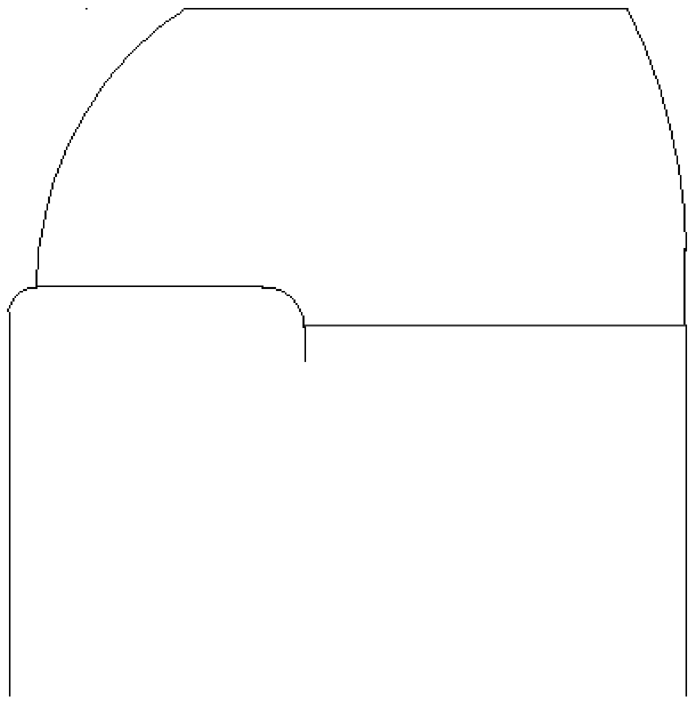


Merge Points tool

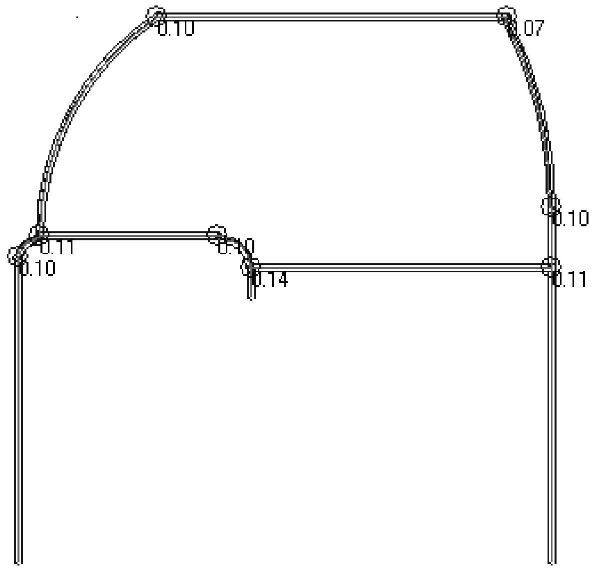
 **Merge Points.** When multiple lines have end-points which are very close to each other, they should be combined into a common end-point. The direction and amount of moving of the multiple end-points into one end-point depends on the type of lines involved and the angle of the lines forming this anomaly. Horizontal and vertical lines take preference over non-horizontal and non-vertical lines, and creases take preference over cuts.

The **Merge Points** tool has one parameter: **Size**. The value in the **Size**: field on the Status bar sets the maximum gap between points for two points to be considered an anomaly.

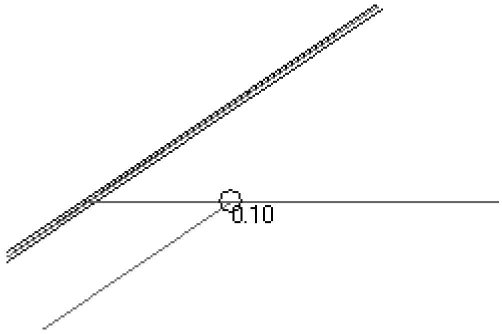
The design below may look normal:



However, when the **Merge Points** tool is run, it finds these anomalies:

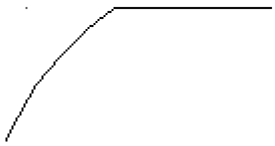


Every intersection in the design shown above is wrong. The numbers in the picture above are in millimeters; the gap at the top left intersection is one-tenth of a millimeter. When the point is zoomed in upon, it looks like this:

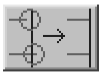


To fix these intersections, select them and click **Fix Selected**.

The anomaly shown above is shown fixed below.

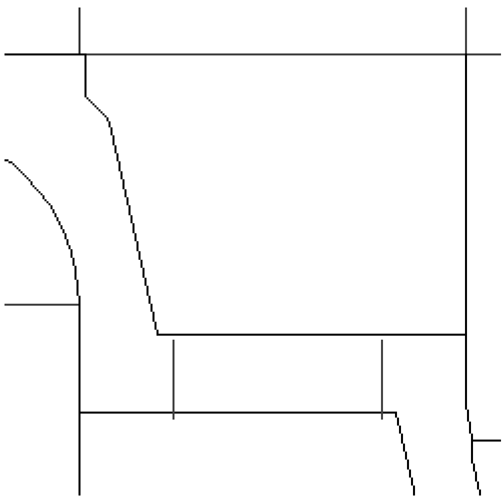


Overruns – Underruns tool

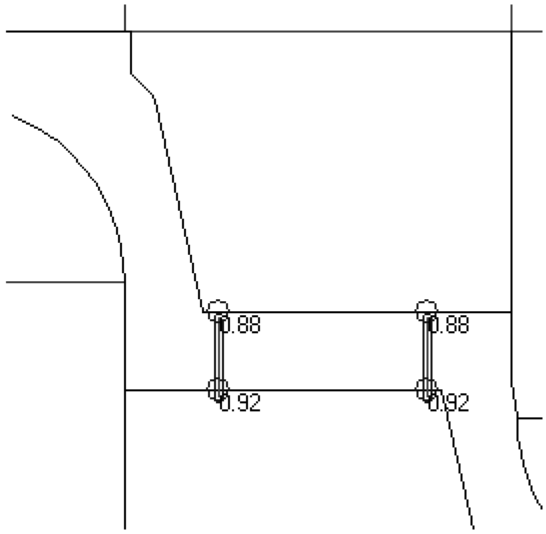
 **Overruns – Underruns**. An **overrun** is when a line extends slightly beyond its intersection with another line. An **underrun** is when the endpoint of a line falls slightly short of causing its line to intersect with another line. Anomalies of this type are displayed with their size next to them.

This tool has two parameters - **Size** and **Minimum Angle**. **Size** sets the maximum distance between the end-point of a line and another line that is considered an anomaly; overruns or underruns larger than this value are ignored. **Minimum Angle** sets the minimum angle at which the two lines must intersect in order for them to be checked for overruns and underruns; if they intersect at an angle smaller than the value specified, any overrun or underrun is ignored.

The layout below, while appearing normal, has Overrun – Underrun anomalies in the stripping rules:



Clicking the **Overrun – Underrun** tool results in these anomalies being displayed:

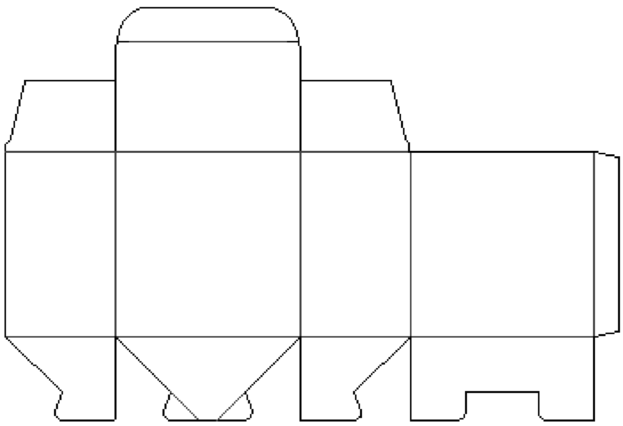


Nearly Horizontal or Vertical tool

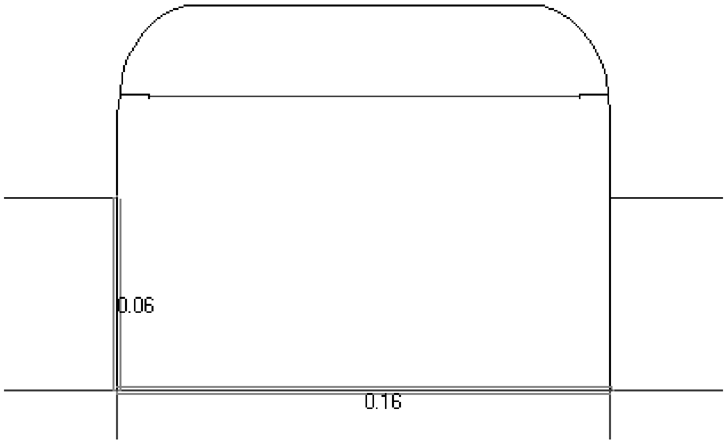
Nearly Horizontal or Vertical. This tool searches for lines which are nearly horizontal or vertical. The **Size**: parameter prompts for the minimum length of the line, and the **Maximum Angle**: parameter sets the largest angle of deviance from horizontal and vertical that the tool will find.

This tool only highlights anomalies - you must fix them manually by leaving Designer's Fix-It and using the Edit tools.

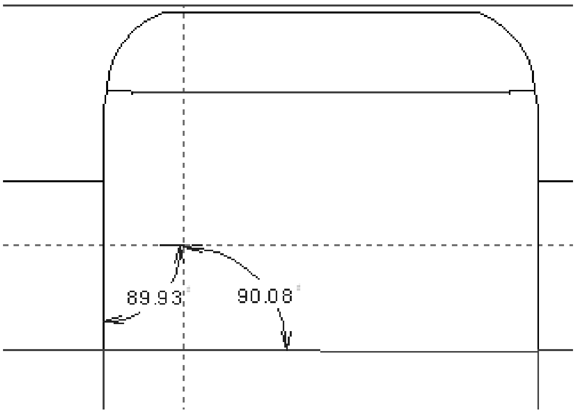
Shown below is what looks to be perfectly orthogonal geometry.



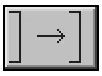
When the **Nearly Horizontal or Vertical** tool is run, however, anomalies are found:



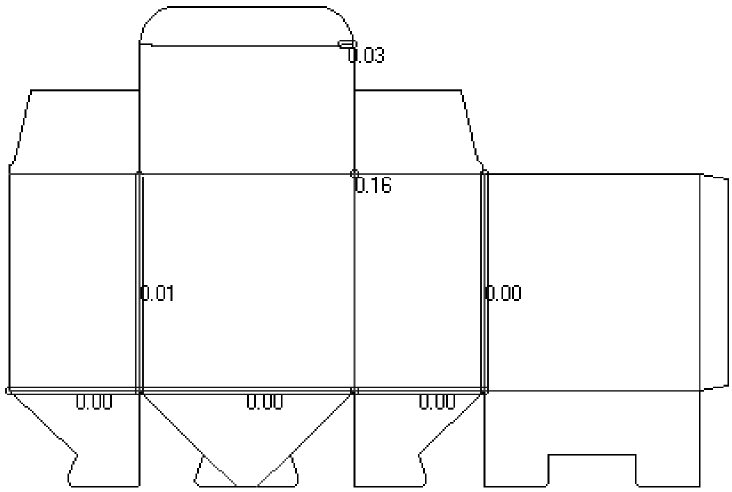
The anomalies are shown below with the aid of perfectly horizontal and vertical construction lines and the Measure tools. Use the Edit tools to fix them.



Double Knives tool

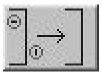
 **Double knives.** This tool searches for lines that are either on top of each other or are separated by a tiny gap and nearly parallel. The **Size**: field sets the minimum length of lines to be checked for duality and the **Maximum Angle**: field sets the maximum angle between lines to check against.

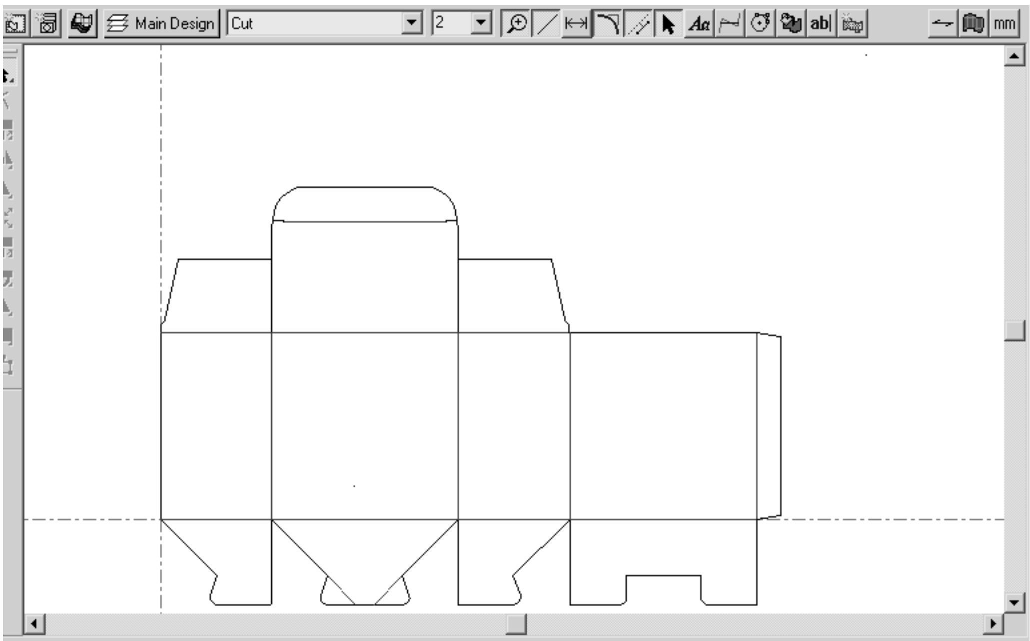
Shown below is a design with some Double Knife anomalies. The number indicates the distance between the double lines.



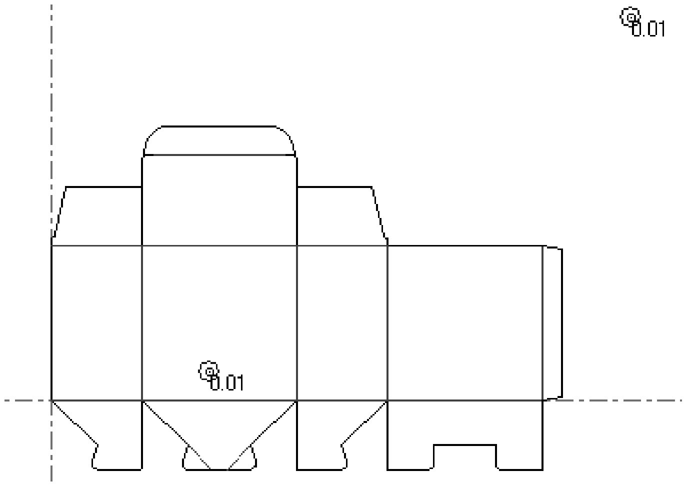
Fixing the double lines is an invisible process.

Short Lines tool

 **Short Lines.** A Short Line anomaly occurs when a single line has both endpoints within the distance set by the **Size:** parameter. To be considered an anomaly, the short line cannot be connected to another line. Shown below is a design plotted using **Scale To Fit**.

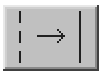


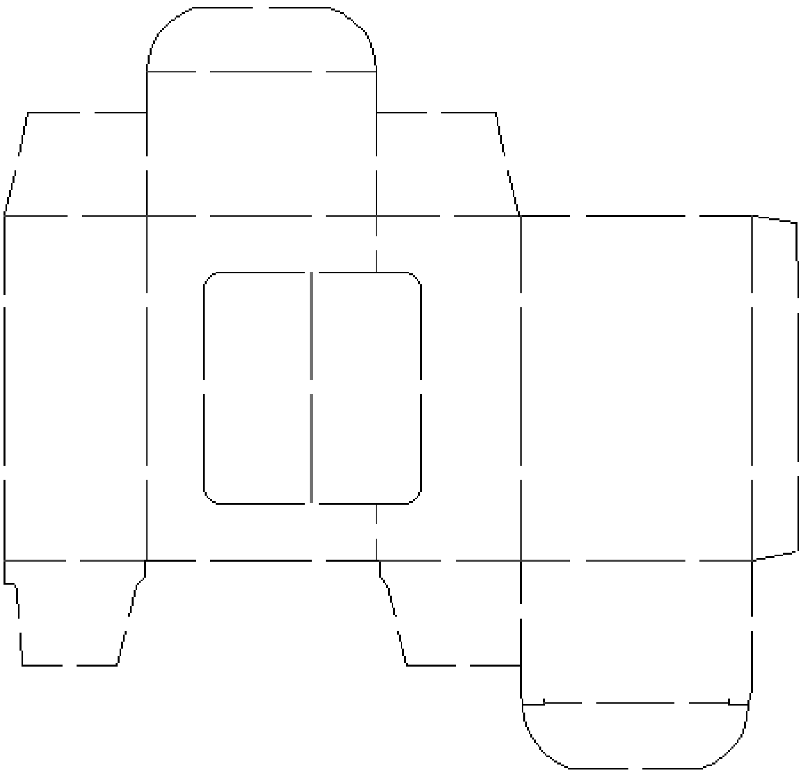
Note that even though Scale To Fit was used, the design does not fill the drawing area. Using the **Short Lines** tool reveals that there are two short lines.



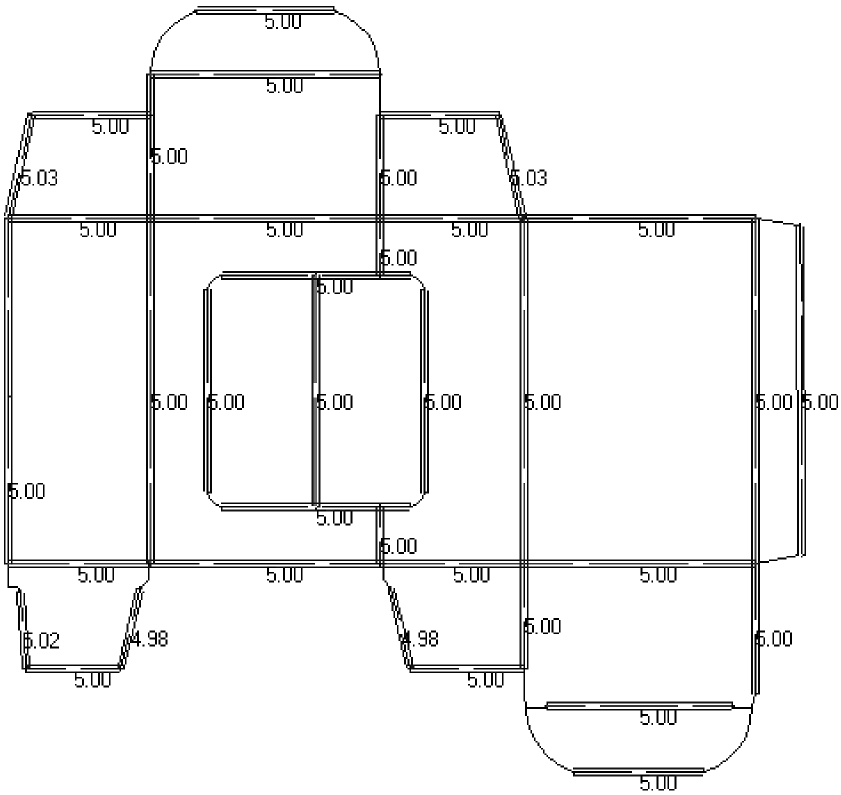
The number next to the anomaly indicates the size of the short line.

Line – Gap Sequence tool

 **Line – Gap Sequence**. This tool finds collinear sequences of lines and gaps that should be turned into one line with bridges. Shown below is a design with what looks like normal bridging.



Using the **Line – Gap Sequence** tool reveals that nearly all of the lines and gaps could be converted into lines with bridges:



Magnifying the left most line of the sleeve body, one can see that there are actually two lines that can be combined into one if the **Width tolerance** and **Shift Tolerance** are adjusted accordingly. Also, the fact that the bridge width indicator is not centered on the line is a sure giveaway that there is more than one line present.

Three parameters on the Status bar control the size of gaps that are considered anomalies, and how those gaps can be manipulated to make one line.

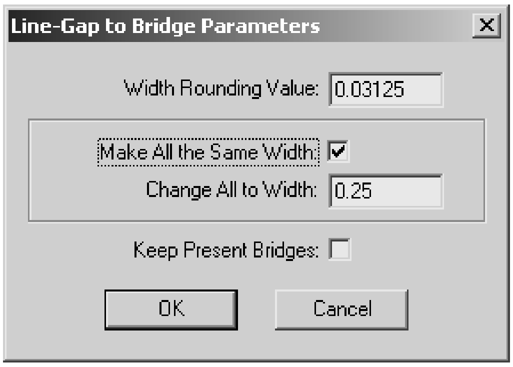
The value in the **Size**: field sets the maximum size of individual lines and gaps to consider as anomalies.

The **Width Tolerance** lets the tool compare gap sizes. If the gap sizes are the same, plus or minus the Width Tolerance, then the gaps can be considered bridges in the new line.

The **Shift Tolerance** lets the tool move the gaps by up to the specified amount along the line to try to distribute them evenly so that they can be considered bridges.

The two parameters work together to let the tool make a bridged line out of a series of lines and gaps when there is a slight discrepancy in the size of the gaps and their positions relative to the line segments.

Clicking **Parameters** opens the Line-Gap to Bridge Parameters dialog box as shown below:



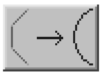
The **Width Rounding Value** rounds the size of the variously-sized gaps so they are exact fractions of the unit system being used.

The **Make All the Same Width** checkbox and the **Change All to Width** field instruct the tool to change all the gaps into bridges of the size specified.

The **Keep Present Bridges** checkbox instructs the tool to keep any defined bridges in the lines contained in the line-gap sequence.

The lines and gaps must be all in the same layer and the lines must be of the same line type for this tool to consider them anomalies.

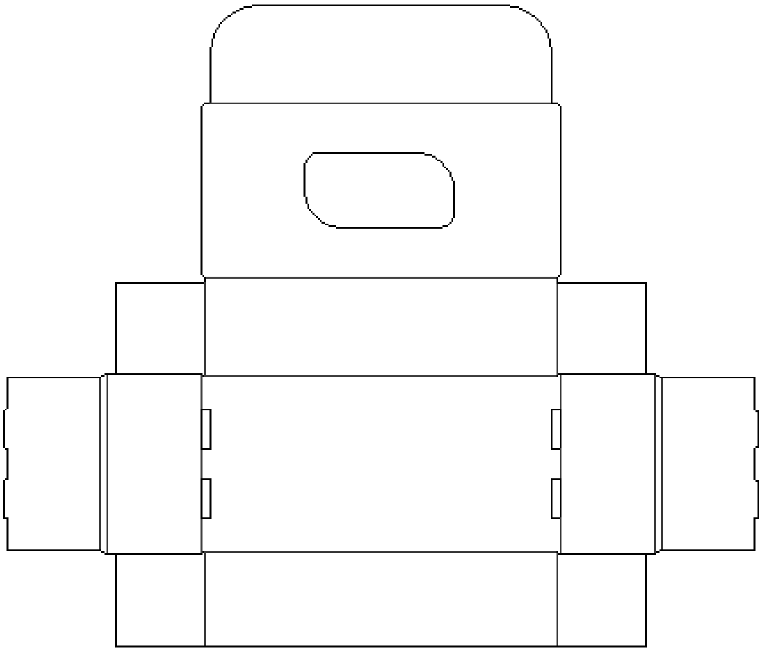
Arc from Lines tool

 The **Arc from Lines** tool converts a chain of line segments approximating an arc into an arc. The replacement arc's end-points are the same as those of the beginning and ending line segments.

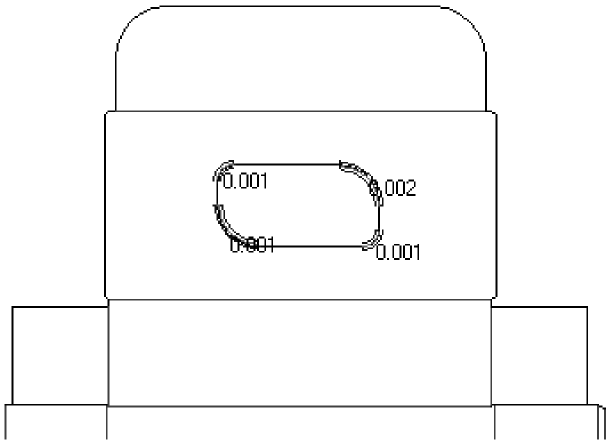
The **Size**: parameter sets the maximum size of the line segments to be considered anomalies.

Note: This tool works with lines and arcs only. If the design contains beziers, you will be prompted to convert them to arcs or to cancel the tool. If the beziers are contained within an embedded design, they cannot be converted to arcs, and the **Smooth Lines** and **Arc from Lines** tools will be disabled.

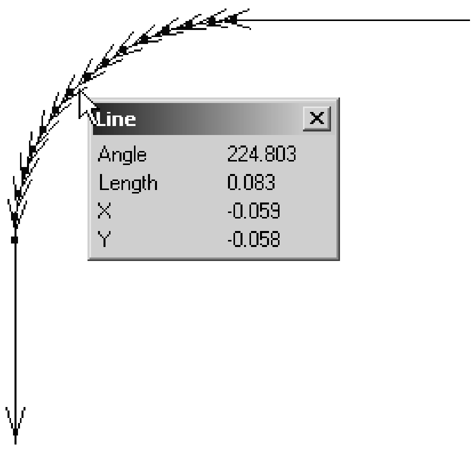
Shown below is a corrugated box with a cutout in its lid.



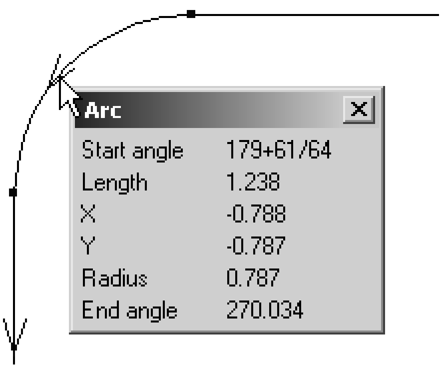
Using the **Arc from Lines** tool reveals that the cutout in the lid is not what it appears to be:



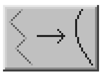
Magnifying an anomaly and using the Measure tool reveals a series of small lines:



Fixing the anomaly results in a single arc.



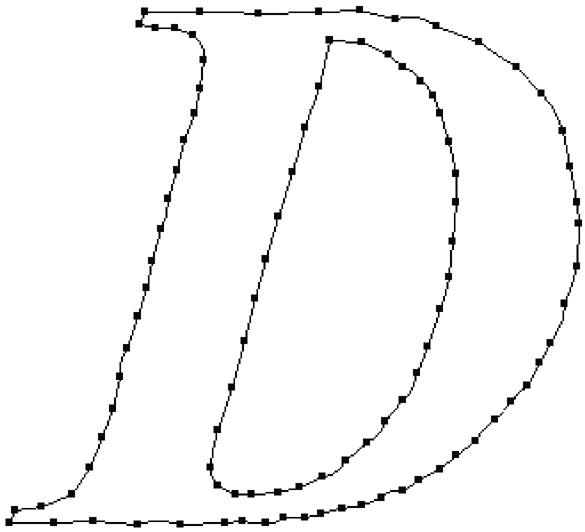
Smooth Lines tool

 The **Smooth Lines** tool converts a chain of small connected line segments into a line or arc that best approximates the outline of the original lines. The chain of line segments must not branch or intersect with any other lines, must be in the same layer, and must be of the same line type.

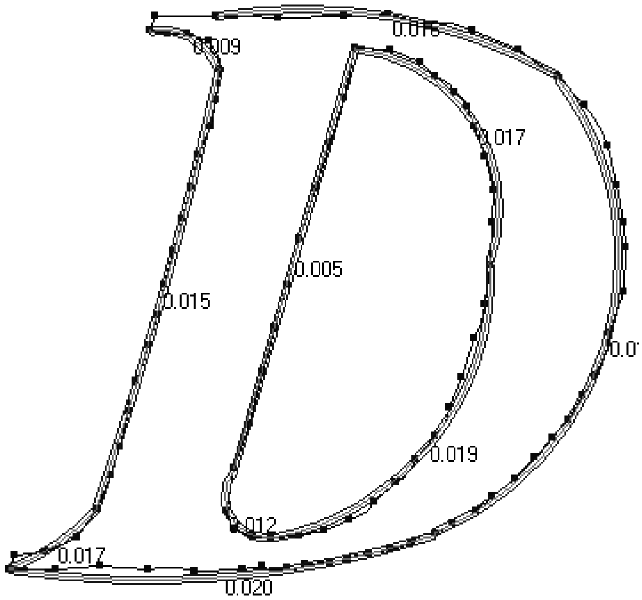
The **Size**: parameter controls the size of the line segments considered anomalies. The **Smoothing Limit**: controls how far the new geometry will stray from the path of the original geometry.

Note: This tool works with lines and arcs only. If the design contains beziers, you will be prompted to convert them to arcs or to cancel the tool. If the beziers are contained within an embedded design, they cannot be converted to arcs, and the **Smooth Lines** and **Arc from Lines** tools will be disabled.

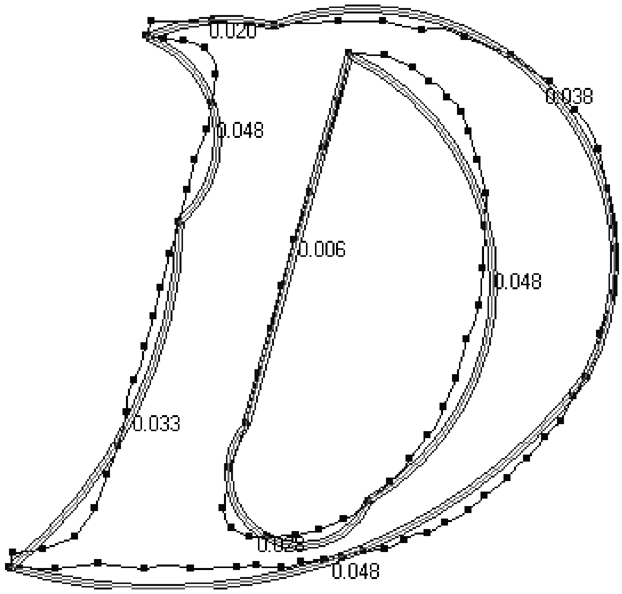
Shown below with endpoints on in View Mode is a piece of text that was converted to lines before being imported into ArtiosCAD:



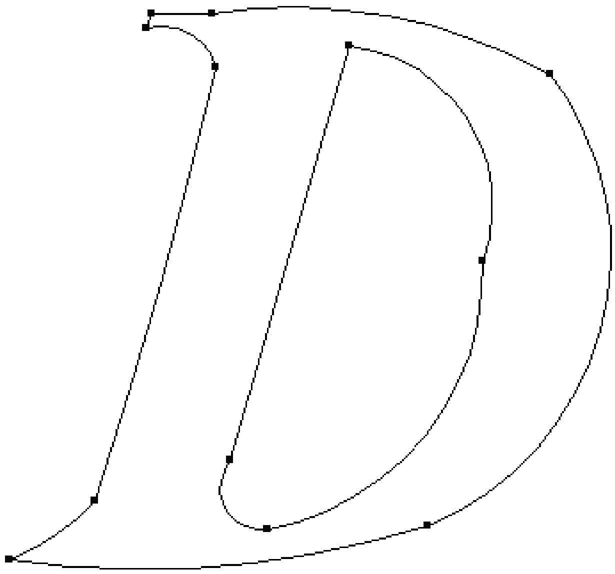
Clicking the **Smooth Lines** tool results in the following anomalies being shown:



Making both the **Size:** and the **Smoothing Limit:** larger results in a more circular proposed fix:

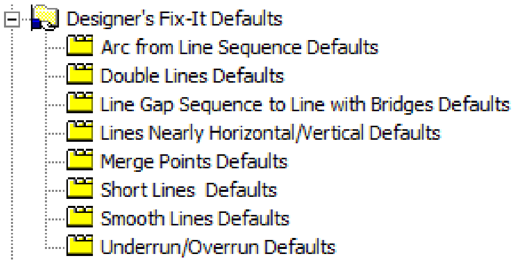


Finally, having reset the parameters to their initial values, selecting all the anomalies and fixing them results in:



Designer's Fix-It defaults

Designer's Fix-It uses the Designer's Fix-It Defaults catalog. In it, you can set the defaults for all the tool parameters.



As with other defaults, double-click the entry to modify and change the values accordingly.

Rubbering

Introduction to rubbering

Some diemakers put strips and pieces of rubber on the dieboard. The **Rubber Design** module provides tools to create rubber pieces that follow the contours of the design lines in the layout. The **Rubber Design and Layout** module enhances that further by efficiently nesting the rubber pieces on rubber sheets, which can then be output to a rubber-cutting device. The Rubber Design and Layout module requires an additional USB security key.

Different types of rubber are used for different purposes on the dieboard. Profile/strip rubber comes pre-cut in strips to be placed by hand so it is generally not included in Outputs. Other types of rubber are cut out of larger sheets. These modules are designed to work with the types of rubber cut out of the larger sheets, but you can also place profile/strip rubber on the dieboard so that the places for profile/strip rubber are indicated when the dieboard is plotted.

Both of these modules are options which must be purchased.

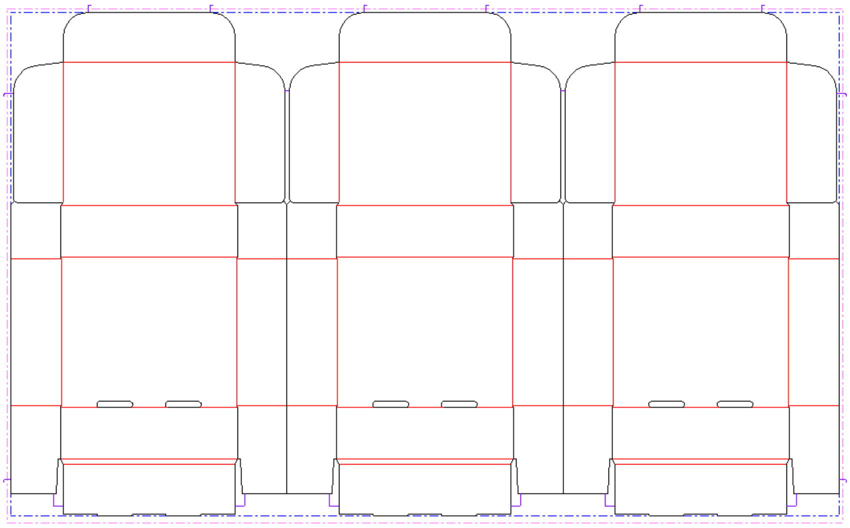
Generic special rule and zero pointage rule are ignored; generic cut/crease is treated as a crease. Change generic special rule to specific special rule and set a pointage for all rule before using any rubbering tools.

ArtiosCAD includes a sensible example table of rubber types, example rubber parameter sets, and example rubber Outputs, but you should review them and modify them to fit your needs before using the Rubbering features for production. See the *Defaults* chapter in the *ArtiosCAD Administrator Guide* for more information.

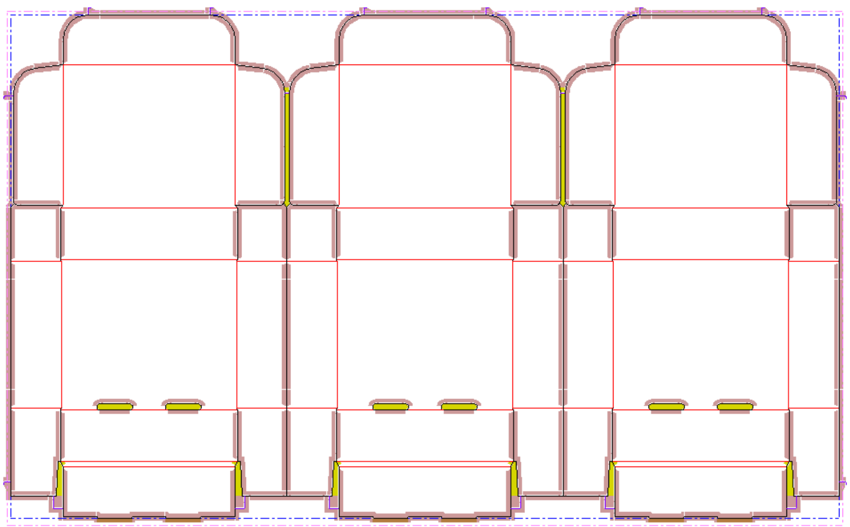
Rubber workflow overview

A general Rubbering workflow could be as follows:

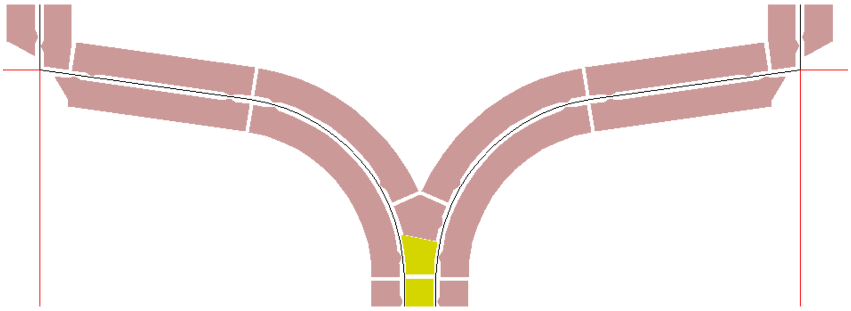
1. Create a manufacturing workspace, perform double knife removal, and save the workspace.
2. Add stripping rules (also called *breaker knives*) where appropriate, both on the exterior and interior of the designs. If waste areas are large and **Make Splits** is checked in the parameter set, ArtiosCAD creates pseudo temporary stripping rules to break up large pieces of rubber for easier manufacture.



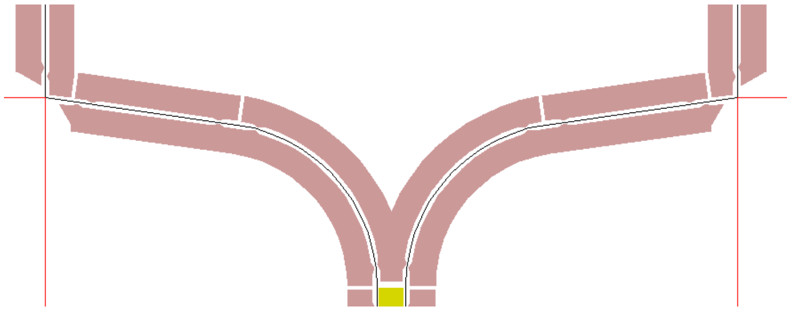
3.  Click **Rubber area** on the Rubbering toolbar.
4. Click **Recalculate all** on the Status bar. ArtiosCAD adds rubber to the layout according to the values in the Rubber parameter set selected in the Manufacturing parameter set. Different types of rubber have different colors.



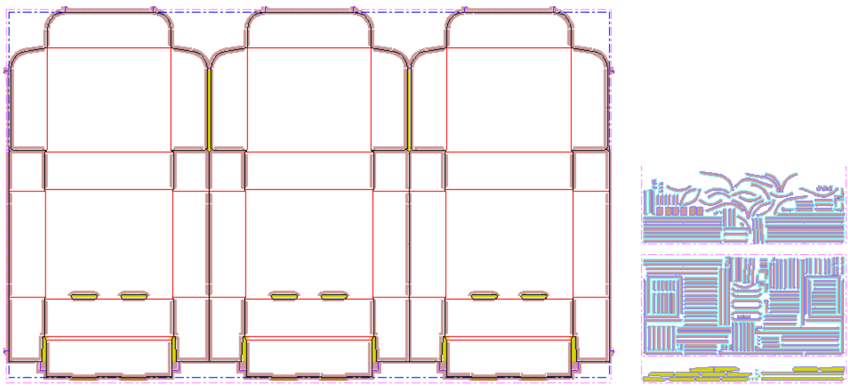
5. Use the manual edit tools to refine the rubber layout, adding joins, splits, and fills where necessary. Shown below is the result of a Recalculate All before editing.



The same area after editing:



6.  Once you have adjusted the rubber as desired, use the **Layout Rubber** tool to nest it together on rubber sheets. (This requires the optional Rubber Design and Layout module.)



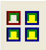
7. Next, make an Output. There are many example Reports in **File** > **Outputs** > **Artios** > **Rubber**. To output the rubber sheets themselves to HPGL files for use by a rubber cutter, use **File** > **Outputs** > **Artios** > **Rubber** > **Rubber sheet output**. Each tile of this Output is an individual rubber sheet.
8. Finally, save and close the workspace, and move on to the next task.

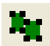
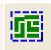
Rubber tools

The Rubber tools are on two toolbars, the **Rubbering** toolbar and the **Rubber view options** toolbar. The tools on the Rubbering toolbar require the Rubber Design license, while the tools on the Rubber view options toolbar can be used by anyone with a Connection license.

Shown below are the Rubbering toolbar with its fly-outs expanded, and the Rubber view options toolbar beneath it, along with a description of each tool's function. Tools with more complex workflows are further described in their own sections after the overview.

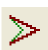
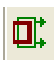
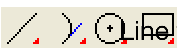
Table: Rubbering tools overview

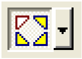
Tool	Name	Description
	Rubber Area	Automatically applies rubber to the lines in the area where you click, or rubbers the entire layout by clicking **Recalculate all** on the Status bar.
	Repeat Rubber Element	Repeats the last rubber element made in the area where you click.
	Delete All Rubber	Deletes the die rubber, rubber sheet layouts, and top plastic cover depending on the items you select in the Delete All Rubber dialog box.
	Delete Rubber Area	Deletes the rubber in the area where you click.
	Select Rubber	Selects the piece of rubber under the cursor. Hold down `CTRL` while clicking to select more than one piece at a time.
	Split Rubber	Splits a piece of rubber where you click.
	Join Rubber	Joins two pieces of rubber when you click near a split. If the two pieces of rubber are different types, click the piece of rubber whose type you want to keep.
	Move Rubber Split	Move a rubber split to the point you click.
	Trim Rubber	Trims a piece of rubber against another or against a counter.
	Add Rubber	Creates or adds to existing rubber using a loop of design lines.
	Subtract Rubber	Subtracts an area of looped design lines from existing rubber.
	Fill Rubber Area	Fills a small area formed by looped design lines with rubber. Not for use with large areas; use **Rubber Area** for those.
	Add Separator	Adds a separator between the edge of the rubber and the rule.
	Remove Separator	Removes a separator from rubber where you click.
	Move Separator (on fly-out toolbar)	Moves a separator to where you click.
	Add Open-out	Adds a cut in the rubber so that it can be spread flat on the sheet for easier cutting.
	Delete Open-Out (on fly-out toolbar)	Deletes the open-out cut where you click.
	Subtract Bolt Holes	Removes areas of rubber covering bolt holes in the dieboard.

Tool	Name	Description
	Update Plastic Cover	Updates the plastic cover to match changes in the slot rubber. Use this tool if you change the slot rubber after already having created the plastic cover with the **Rubber Area** tool. Click **...** on the Status bar to adjust the cover parameters or click **Recalculate All** to completely regenerate the cover.
	Group Rubber	Groups selected adjacent rubber or plastic cover pieces together so that they are nested on the sheet as a group. This allows easier finding of pieces that go near each other on the dieboard.
	Ungroup Rubber (on fly-out toolbar)	Ungroups rubber or plastic cover pieces.
	Layout Rubber	Creates nested sheets for some or all rubber types. Requires optional Rubber Design and Layout module and additional USB security key.
	Highlight Rubber with Repeat	Shows matching pieces of rubber on the dieboard and on the sheet for easy identification and placement assistance.
	List Rubber Elements	Opens a dialog box listing the rubber elements in which you can click entries and have them highlighted in the layout and sheet (if available).
	Rubber View Options	Sets options for the view such as fill, color by type or by number, show element numbers, and so forth.

In addition to the tools listed above, other tools are particularly useful for working with rubber.

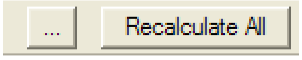
Table: Other tools useful for rubbering

Tool	Name	Purpose
	Delete Element (or the del key on the keyboard)	Deletes pieces of rubber selected with a Select tool.
	Merge Lines Straight	Adjusts the rubber shapes as needed and auto-repeats.
	Merge Lines Arc	Adjusts the rubber shapes as needed and auto-repeats.
	Merge Lines to Intersection	Adjusts the rubber shapes as needed and auto-repeats.
	Stretch Point	Adjusts the rubber shapes as needed and auto-repeats.
	Stretch by Polygon	Adjusts the rubber shapes as needed and auto-repeats.
	Line tools	Creates custom-shaped looped design lines that then can be used with **Add Rubber** or **Subtract Rubber**.

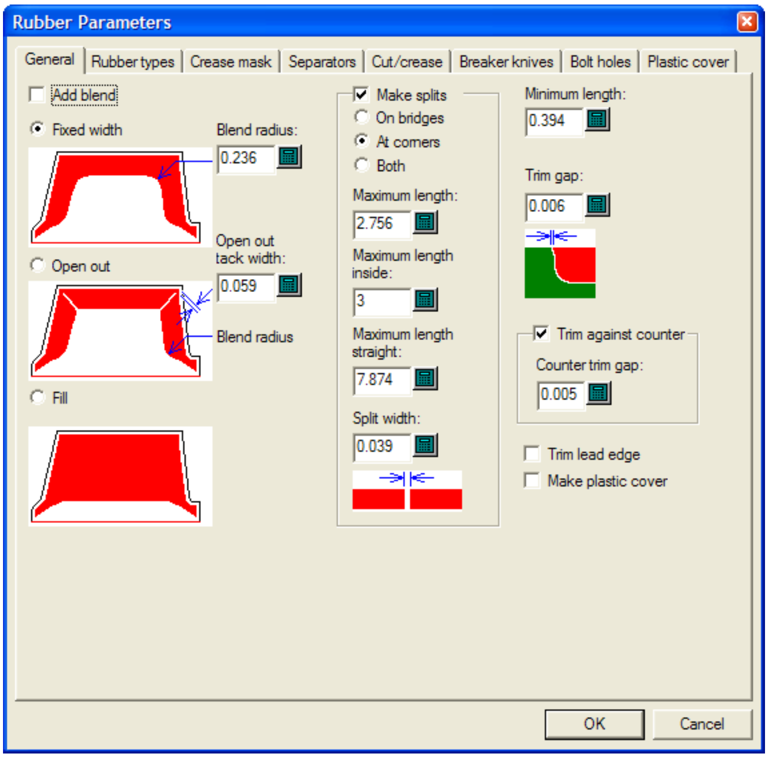
Tool	Name	Purpose
	Auto-Repeat	Controls if manual changes are repeated in congruent geometry.

Rubber Area tool

 The **Rubber Area** tool automatically adds or updates rubber in the area in which you click after activating the tool. If the **Rule Path** layer has rules in it, the rubber is applied to those rules; otherwise, the rubber is applied to lines in the **Designs**, **Dieboard**, and **Stripping Rules** layers. When the tool starts, it turns on the **Rubber**, **Designs**, **Dieboard**, and **Stripping Rules** layers, and the Status bar includes the following controls:



The **...** button leads to the Rubber Parameters dialog box, which lets you set or change all the values in the Rubber parameter set associated with the parameter set for this layout.

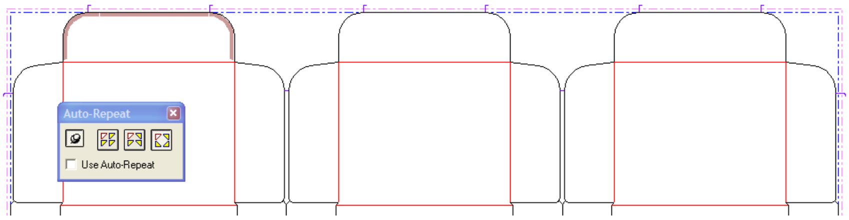


For more information about setting these values, refer to the *Defaults* chapter in the *Esko ArtiosCAD Administrator Guide*.

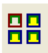
Repeat Rubber Element tool

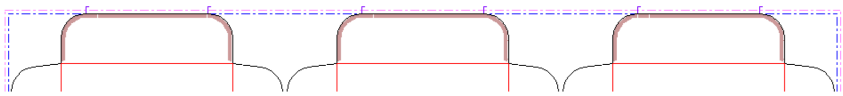
 The **Repeat Rubber Element** tool repeats the selected pieces of rubber in congruent locations when Auto-Repeat is turned off.

Shown below is a layout with rubber created using the **Rubber Area** tool and Auto-Repeat turned off.



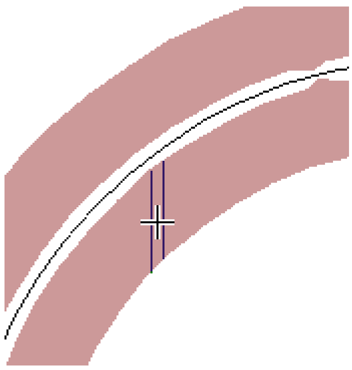
To use the tool, do the following:

1.  Select the pieces of rubber to repeat, either individually or by a marquee selection.
2.  Click **Repeat Rubber Element**.
3. ArtiosCAD creates rubber in congruent areas.

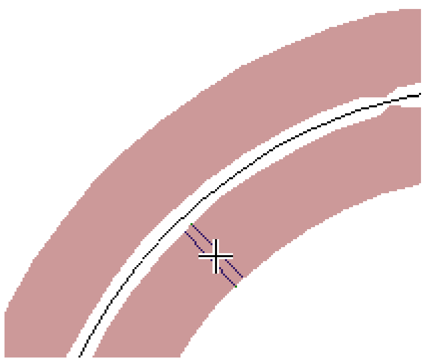


Split Rubber tool

 The **Split Rubber** tool splits a piece of rubber perpendicularly if its sides are parallel; otherwise, the split is roughly perpendicular, snapping horizontally or vertically if the **Horizontal/Vertical** checkbox is checked on the Status bar. The width of the split is set by the value in the **Split Width**: field on the Status bar. Shown below is when the **Horizontal/Vertical** checkbox is selected.



When the **Horizontal/Vertical** checkbox is not selected, the direction of the split is perpendicular to the side of the rubber closest to the cursor:

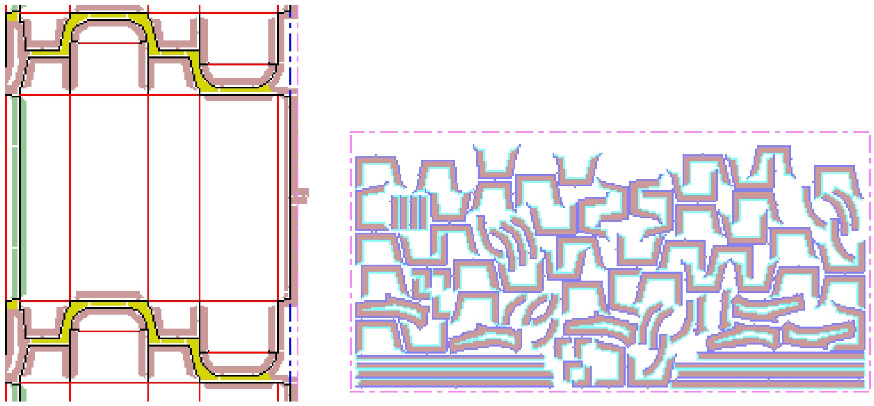


The split position snaps to corners, midpoints, and bridges, and also to construction lines in the **Die rubber** layer.

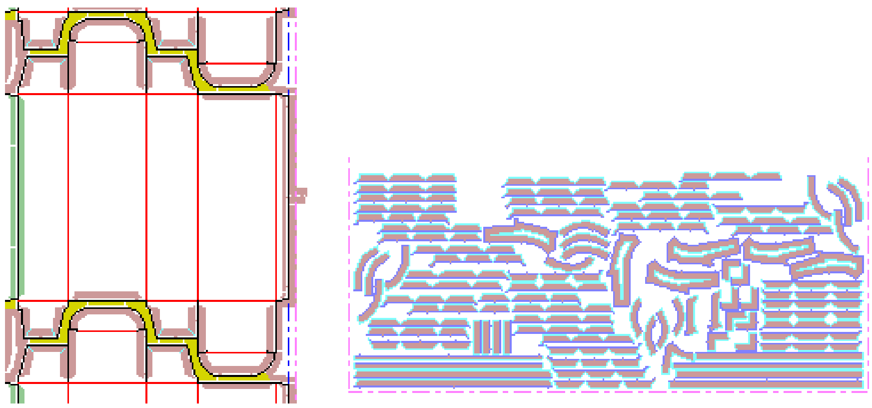
After splitting a piece of rubber, if appropriate, the smaller piece is changed to the type of rubber type used in narrow areas.

Add Open-out tool

 The **Add Open-out** tool creates a cut in a corner of a piece of rubber so that it may be unfolded flat on the rubber sheet for less waste, and to accommodate rubber-cutting machines that can make a bevel in one direction only. To use this tool, activate it and then click in the corner of a piece of rubber. Shown below is a rubber layout with no open-outs and its corresponding rubber sheet.



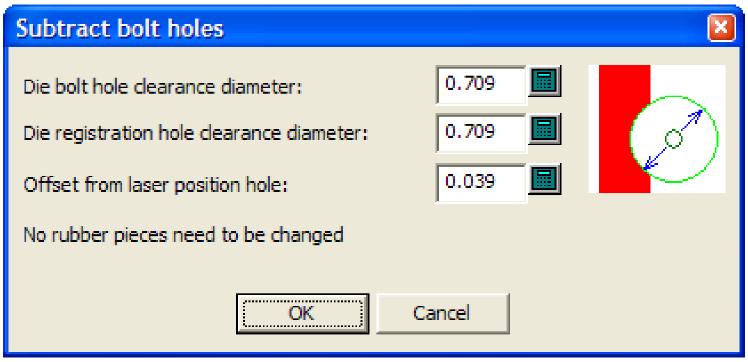
After adding open-outs to the corners of the U-shaped pieces of rubber in the four flaps, the regenerated rubber sheet looks like this. Note the open-outs in the layout.



Subtract Bolt Holes tool

 The **Subtract Bolt Holes** tool removes the areas of rubber that cover die bolt holes, die registration holes, and laser positioning holes in the die. The parameter set may be set to remove these areas automatically (but this affects auto-repeat); it is when that option is not selected that this tool becomes necessary.

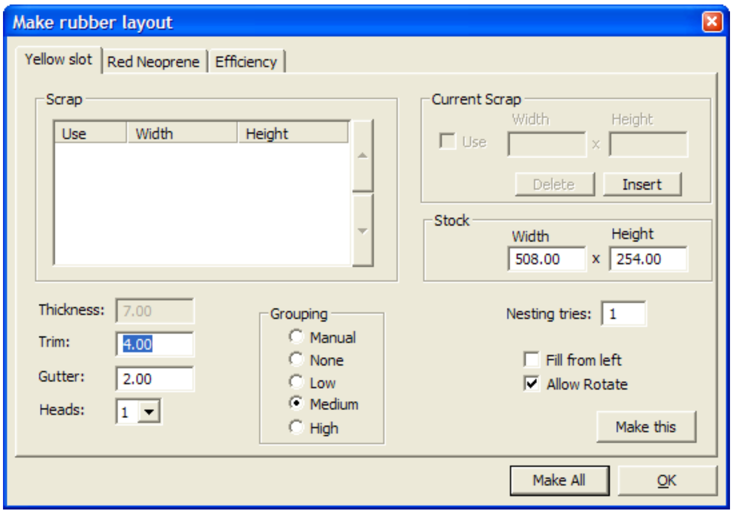
To use this tool, activate it, enter the desired offsets in the Subtract Bolt Holes dialog box, and click **OK**. The Subtract Bolt Holes dialog box will show if rubber pieces need changing or not as shown below.



Often the bolt holes are far away enough from the rubber so that this tool does nothing.

Layout Rubber tool

 The **Layout Rubber tool** efficiently nests the rubber elements together on sheets that can then be output to files for a rubber-cutting machine. Note that this tool requires a license which must be purchased, and also has an additional USB security key. When activated, the tool opens the Make Rubber Layout dialog box, shown below in its initial state before a layout has been made.



Each type of rubber used in the layout (except for premade strip/profile rubber) has its own sheet created, and correspondingly has its own tab in the Make Rubber Layout dialog box.

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.

Some rubber cutters have more than one cutting head so that more than one sheet of rubber can be cut at a time. In this case, specify the number of heads in the **Heads** drop-down list box. If you specify two or more heads when making rubber sheet layouts, this reduces the number of copies output of each piece. For example, if there are 11 copies of a piece, ArtiosCAD might place five copies on a sheet cut twice and one on a sheet cut once. The sheets that should be cut twice are labeled **x2** both in the rubber sheets layer (at the top left of the sheet) and in the Output.

ArtiosCAD automatically groups pieces of rubber on the sheets based on their shapes. 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.

You can define up to six sheets of scrap rubber to use instead of fresh sheets. To add a scrap sheet, click **Insert** in the **Current Scrap** group, enter its dimensions, and check the **Use** checkbox. Click **Delete** to delete the selected piece of scrap from the list.

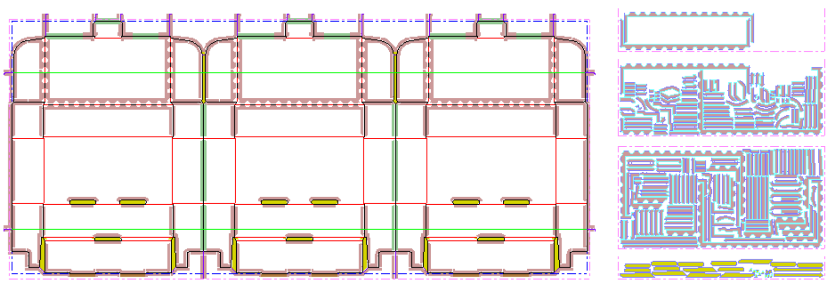
Increasing the value in the **Nesting tries:** field causes ArtiosCAD to try more nests, but this takes more time.

Fill from left tells ArtiosCAD to fill the sheet in a left-to-right sequence. When this is off (the default), the sheet is filled from the bottom to the top.

Allow Rotate allows the nesting process to use angles other than 180 when trying potential nests.

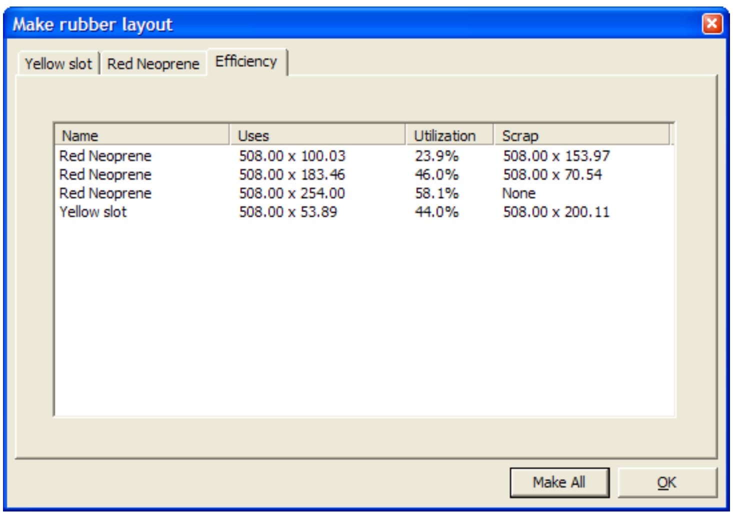
Make This generates a sheet of rubber for the current tab.

Make All generates sheets of nested pieces of rubber according to their types. More than one sheet per type is generated if necessary.



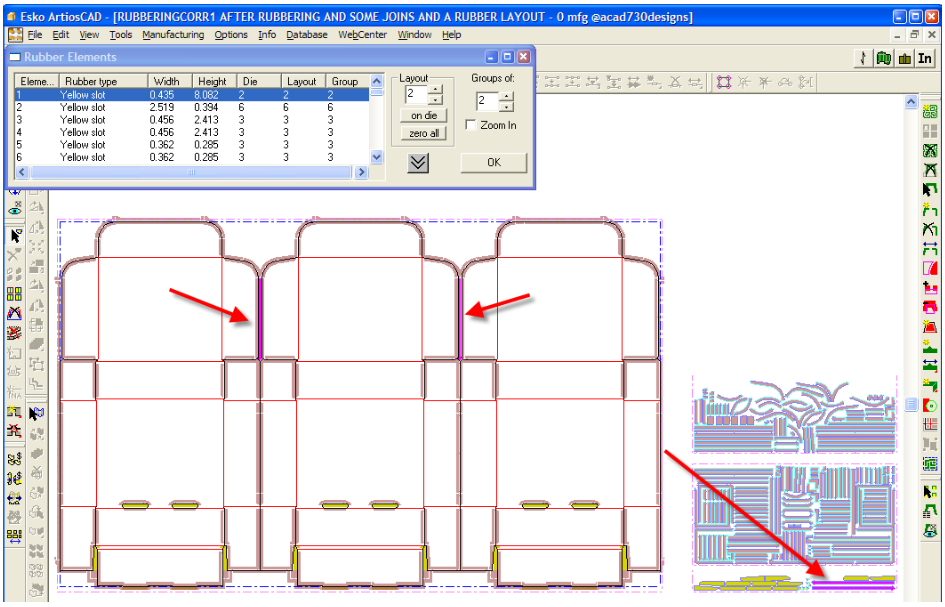
Clicking **OK** makes nothing and returns to ArtiosCAD.

Once you have made a rubber sheet layout, ArtiosCAD populates the Efficiency tab of the Make Rubber Layout dialog box as shown below. For each sheet of rubber, ArtiosCAD lists the sheet's rubber type name, the dimensions of a rectangle containing all nested pieces (always including one full sheet dimension), the percentage of the area in the rectangle that is actually used, and the size of the rectangular sheet that is remaining when the nested rectangle is removed.

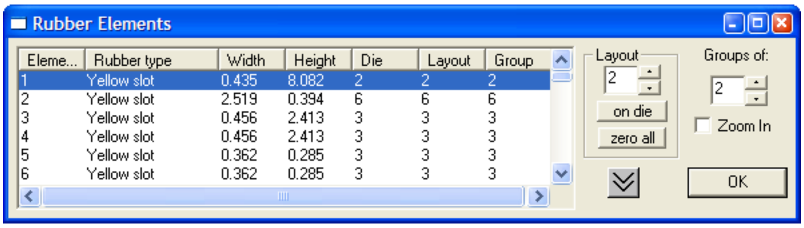


List Rubber Elements tool

The **List Rubber Elements** tool opens a dialog box in which you can select rubber elements and see them highlighted on the dieboard and rubber sheets (if they have been generated). Some elements of the dialog box are unavailable without the Rubber Design and Layout option.



Shown below is the Rubber Elements dialog box in greater detail.

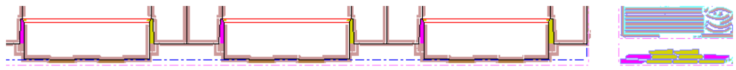


The left side of the dialog box shows the list of all the rubber elements, ordered by rubber type and then element size. The **Die** column lists the number of pieces on the dieboard, while the **Layout** column lists the number of pieces in the rubber sheet layout (these numbers are the same initially by default). The **Group** column indicates the maximum number of groups containing this element in the sheet layout (with 1 being no groups).

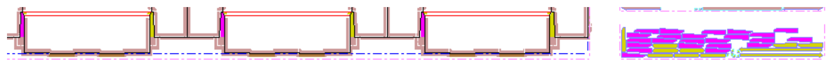
The down-arrow chevron expands the dialog box to the bottom of the monitor, and the up-arrow chevron it changes to when clicked shrinks the dialog back to its previous size.

In the **Layout** group, the value listed shows how many instances of the current selection are in the sheet layout. You can change this number to make more or less of the current selection if you regenerate the layout. This is so that if you need to produce a few more pieces of a delicate item, you can do so easily. **On die** resets the number of elements in the layout to the number of elements on the dieboard, while **Zero all** resets all the elements on the sheet to zero. (You would then select the few pieces to make again, set their numbers in the Layout field to the desired numbers, click **OK**, and regenerate the layout with the **Layout Rubber** tool.)

For example, suppose you need 20 more of these, of which you currently have 3:



You would increase the number in the **Layout** field to 23, regenerate the Yellow Slot rubber sheet, and arrive at the layout shown below:



The **Groups of:** group controls how many instances of the same rubber piece are placed near each other on the sheet. For example, if there are 12 instances of the same element, the **Group** column is initially set to 12 so that they are all together on the sheet, but by changing the **Groups of**: value to 2 and regenerating the sheet, you would have up to 6 groups of 2 scattered throughout the sheet. However, more than 2 pieces may still be grouped near each other if that results in greater efficiency.

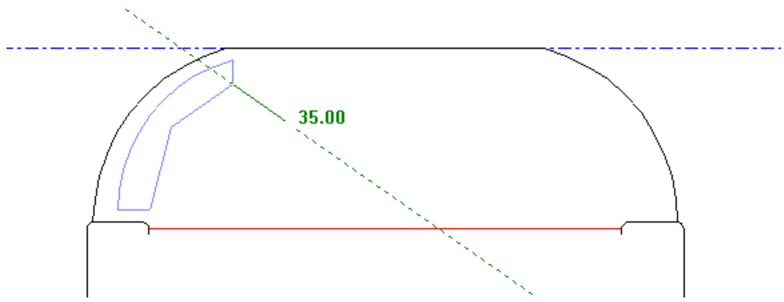
When you change the number in the **Groups of:** field, the **Grouping** option button in the Make Rubber Layout dialog box changes to **Manual** so that your change to the layout method is preserved if you regenerate the rubber sheet layout.

Zoom in zooms in on the selected rubber element so that it fills the window. Deselecting it performs a Scale to Fit to show the entire workspace.

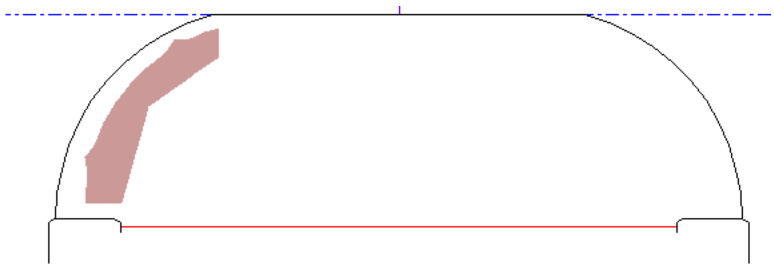
Making manual changes to a rubber layout

To design your own rubber shapes, draw them in the **Die Rubber** layer and then use the Add Rubber tool as shown in the steps below.

1. Change to the **Die Rubber** layer of the layout, turn on the appropriate toolbars, and draw the desired geometry. Make sure to draw a loop of lines.

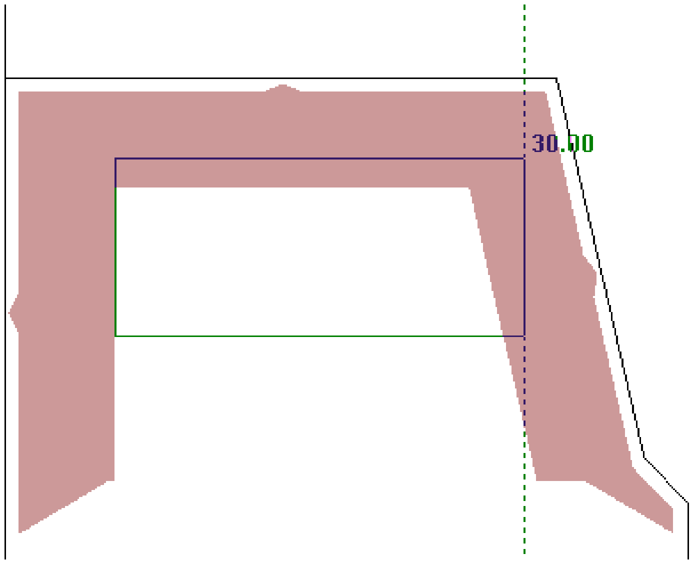


2.  Click **Add Rubber**, select the just-drawn geometry either by clicking it or by using a marquee selection, and set the Status bar options as desired. If the lines do not form a loop, OK remains unavailable. Click **OK** when it is available.
3. The geometry is changed to a piece of rubber and is auto-repeated in congruent areas if Auto-Repeat is enabled.

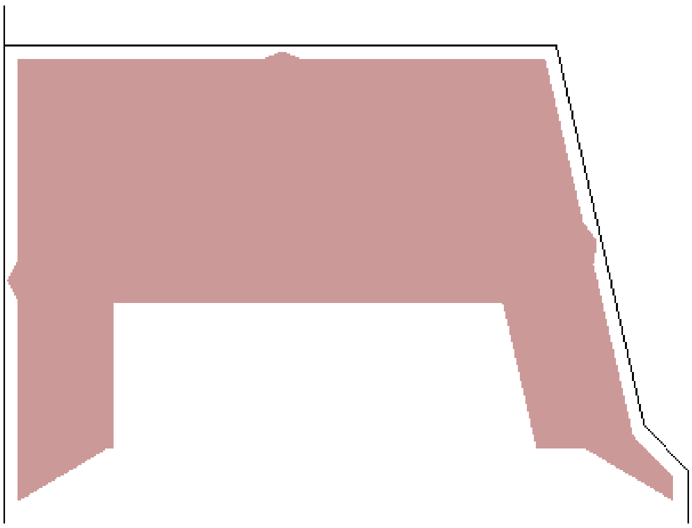


To add to an already-existing piece of rubber, do the following:

1. Change to the **Die Rubber** layer of the layout, turn on the appropriate toolbars, and create a loop of lines that overlaps the desired piece of rubber.

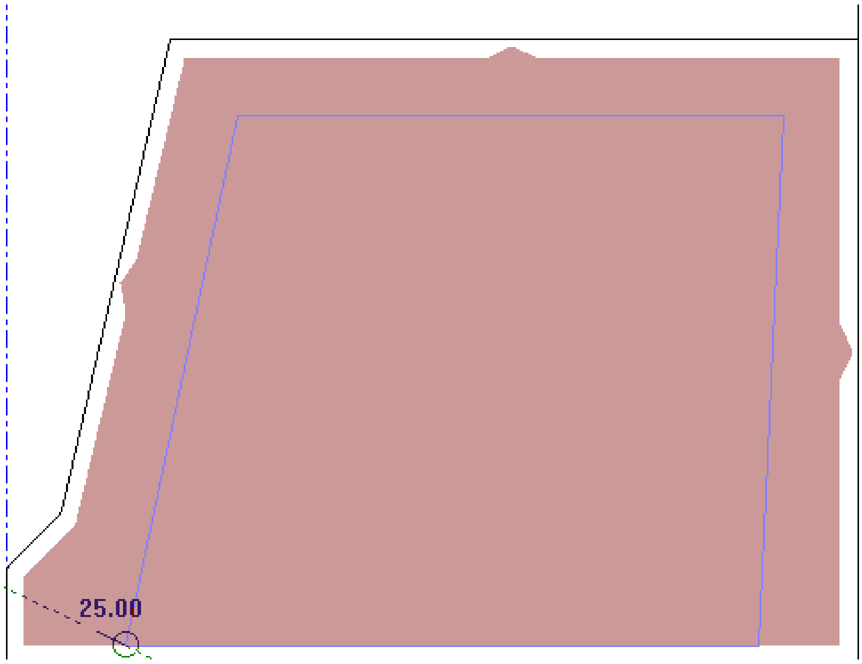


2. Click **Add Rubber**, select the geometry either by clicking it or by using a marquee selection, and set the Status bar options as desired. Click **OK**.
3. ArtiosCAD adds rubber to the area within the looped lines that did not have rubber before, and these changes are auto-repeated to congruent areas if Auto-Repeat is enabled.

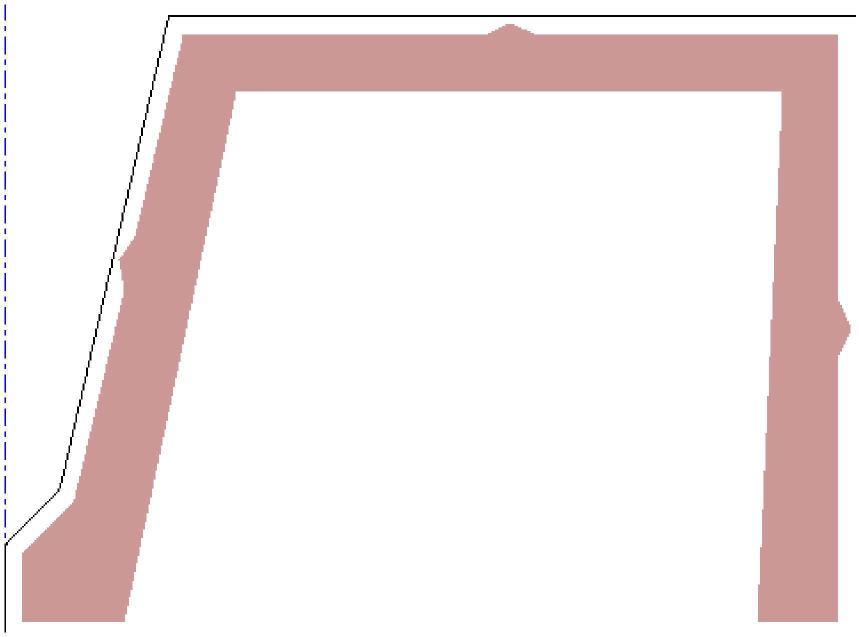


To remove rubber, use the **Subtract Rubber** tool in the same way as the Add Rubber tool.

1. Change to the **Die Rubber** layer of the layout, turn on the appropriate toolbars, and create a loop of lines in the shape of the rubber to remove.



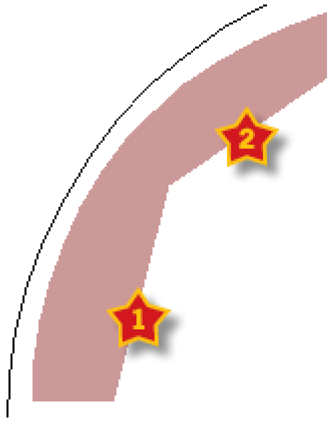
2.  Click **Subtract Rubber**, select the geometry either by clicking it or by using a marquee selection, and set the Status bar options as desired. Click **OK**.
3. ArtiosCAD removes rubber from the area within the looped lines that had rubber before, and these changes are auto-repeated to congruent areas if Auto-Repeat is enabled.

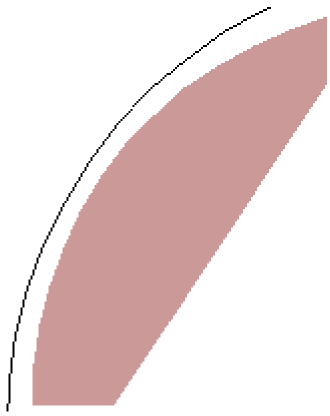


To make small changes to the rubber layout, use the **Select**, **Move**, and **Rotate** tools. However, avoid using the **Copy** tools with rubber elements as this could lead to unpredictable results.

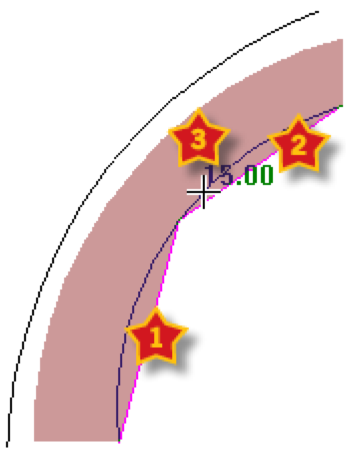
Use the tools on the Adjust and Adjust Outlines toolbars to adjust the outline of rubber pieces. **Merge Lines Straight**, **Merge Lines into Arc**, and **Merge Lines to Intersection** can all adjust the outlines of rubber pieces in two to three clicks. **Stretch Point** and **Stretch by Polygon** are also handy. These tools work with Auto-Repeat if it is enabled.

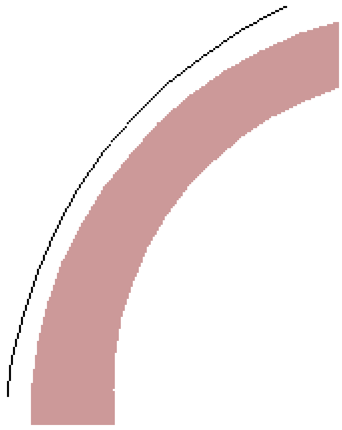
 **Merge Lines Straight** straightens a series of lines, and when used with rubber, changes the shape of a piece of rubber. Activate the tool and click lines 1 and 2 to produce the second picture.





 **Merge Lines Arc** works similarly; activate the tool, indicate the lines to merge, and then set the radius of the arc:



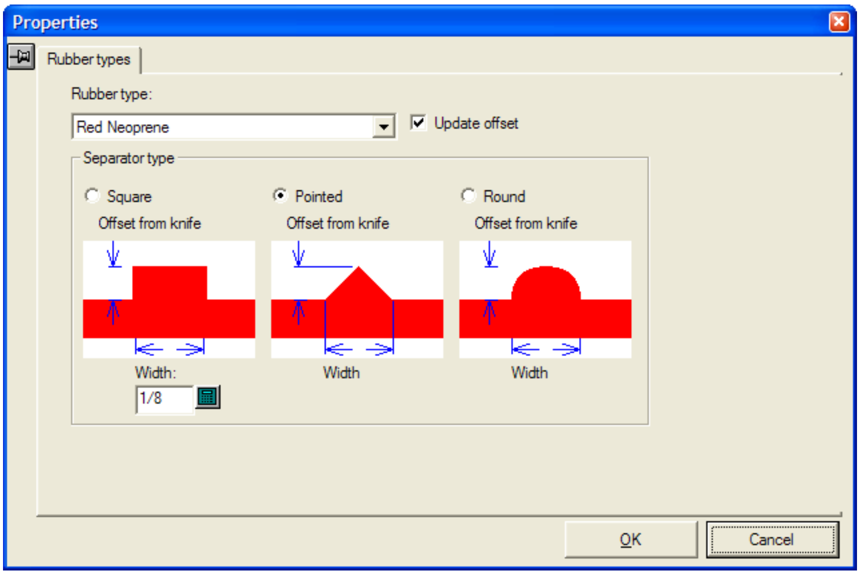


The rest of the previously-mentioned tools work the same way and all reapply separators.

Changing rubber properties

To change the type of a piece of rubber or its separator type, do the following:

1. Click the **Select Rubber** tool and double-click the piece of rubber to change; or, to change more than one piece of rubber at once, hold down `CTRL` while selecting pieces and then press `ALT-Enter`.
2. The Properties dialog box appears as shown below.



3. In the dialog box, use the **Rubber type**: drop-down list box to change the type of the rubber. **Update offset** is selected by default so that ArtiosCAD updates the offsets since they may differ between types of rubber. In the **Separator type** group, change the shape and size of the separators as desired. Slot and strip/profile rubber pieces do not have separators made for them. Changes made in this dialog box are immediately applied to the layout.
4. Click **OK** to resume working with the layout.

Blanking

Blanking is the process of removing the cartons/boxes (called blanks) from the sheet as it travels through the die press. Blocks and pins on the upper board push the blanks through holes in the lower board into stacks on the pallet. The rest of the sheet moves out of the press as waste. Having the press separate the blanks from the waste reduces the time needed to complete jobs and also reduces the risk of error.

Before using the tools described in this section, you should have a thorough understanding of your company's best practices for blanking. If you are unfamiliar with blanking, good places to start learning are in the documentation for your press and at your press manufacturer's website.

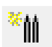
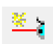
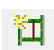
Blanking defaults are contained in the Blanking parameter sets. To review or change them, click **Options** > **Defaults** > **Blanking machine parameter sets** and **Blanking parameter sets**.

Blanking tools list

Turning on the Blanking Toolbar

Before you can use the blanking tools for the first time, use the Toolbar Master Control for Manufacturing (**View** > **ArtiosCAD Toolbars** to turn on the Blanking toolbar. It will appear as a floating toolbar the first time you turn it on; after that, dock it or leave it floating as desired.

Tools on the Blanking Toolbar

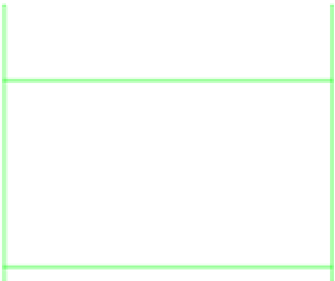
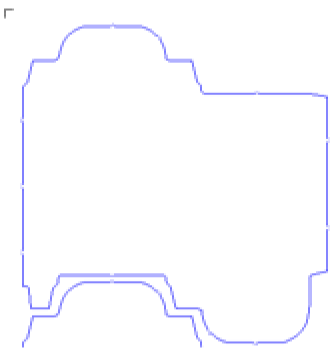
Button	Function
	Add Grid Bar
	Generate Blanking Tools
	Adjust Blank Holes
	Adjust Pushers
	Add Non-stop Swords
	Add Photocells
	Add Jogger Guides
	Add Support Bars
	Add Air Holes
	Add Blanker Pusher Pins
	Add Hardware

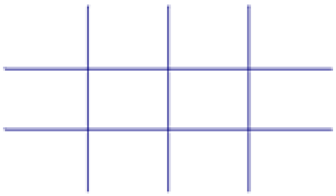
Button	Function
	Add Tie-bolts
	Add Pressers
	Delete Blanking

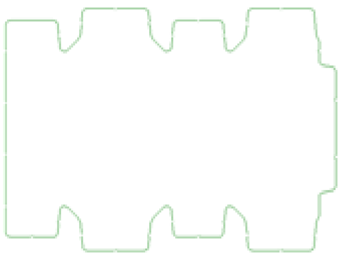
Blanking Elements

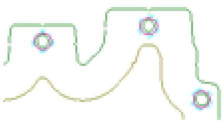
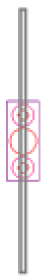
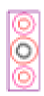
Use the blanking tools in ArtiosCAD to create these blanking elements. If no blanking layers already exist in a manufacturing workspace, using these tools creates them.

The context menu commands are available when you right-click an element. The validation describes what ArtiosCAD checks when placing an element. Some blanking elements have multiple shapes that may differ from those shown here.

Blanking Element	Purpose	Context menu commands	Validation
Frame 	Holds the lower tool in the press	**Delete**	None
Blank holes 	Separates the blanks from the sheet.	**Delete** **Smooth** (works on point blends) **Unsmooth** (works on corner and slot bends) **Retack** **Smooth** and **Unsmooth** use the values from the Blanking Parameter Sets dialog box in the	None

Blanking Element	Purpose	Context menu commands	Validation
		Generate Blanking Tools tool	
Grid bars 	Separates the blanks from the sheet	**Delete** **Repeat** **Trim/Extend** (uses values from the Blanking Parameter Sets dialog box in the Add Grid Bar tool)	Must not overlap the center line of any other grid bar **Note:** This validation means that grid bars do not repeat when **Trim to grid** is turned off in the Blanking Parameter Sets dialog box.
Non-stop swords 	Supports the blanks as the stack underneath is removed from the press	**Delete**	Must not overlap any joggers or other non-stop swords
Photocells 	Detects when the stack of blanks is too high and needs to be removed from the press	**Delete**	Optionally must be within press limits Must not overlap any joggers or other photocells
Joggers 	Guides the blanks to the correct position on the pallet	**Delete**	Must not overlap any other joggers Must not overlap any lines in the lower board

Blanking Element	Purpose	Context menu commands	Validation
			Component active part lines must not overlap photocells, support bars, or non-stop swords
Support bars	Reinforces the lower tool	**Delete**	Must not overlap joggers Should not overlap any other support bars
Pusher edge 	The outer edge of a pusher	**Delete** **Smooth** (works on point blends) **Unsmooth** (works on corner and slot bends) **Retack** **Smooth** and **Unsmooth** use the values from the Blanking Parameter Set dialog box for the Generate Blanking Tools tool.	None
Blanker pusher pins, also called stand-off pins (SOP)	Attaches the pushers to the upper board	**Delete**	Must not overlap other blanker pusher pins, pusher ID text items, pusher edges, tie-bolts, or

Blanking Element	**Purpose**	**Context menu commands**	**Validation**
			pusher air holes
Pusher ID text	Labels each pusher for easier assembly	**Delete**	Must not overlap any blanker pusher pins or pusher edges
Pusher board air hole and Upper blanking board air hole	Reduces suction caused by board movement in the press	**Delete**	Must not overlap any blanker pusher pins, pusher ID text items, or any pusher edges
Pressers  	Holds the sheet waste as the the blanks are separated	**Delete**	Must not be in optional machine-restricted zones, or overlap other pressers or upper board air holes
Hardware	Varies. Grid clamps attach the grid bars to the frame. Plastic film acts as joggers.	**Delete**	None
Tie-bolt	Attaches the upper board to the press	**Delete**	Must be within the upper board Must not overlap any other tie-bolt

Blanking Element	Purpose	Context menu commands	Validation
			hole, upper board air hole, or blanker pusher pin in the upper board
Lower blanker board	Guides the blanks into the proper position	**Delete** Properties dialog box with Alignment Method and Wood Corners tabs	None
Upper blanker board	Pushes the blanks through the lower board	**Delete** Properties dialog box with Alignment Method and Wood Corners tabs	None

Blanking Congruency

Congruency is the property of two or more elements being identical except for physical placement. For blanking, ArtiosCAD uses contour congruency and layout congruency to determine where to put elements when Auto-Repeat is enabled.

Note:

Many factors affect congruency in unanticipated ways. Use caution when Auto-Repeat is enabled and always check that elements are placed correctly before performing an Output. Each element created with Auto-Repeat is checked for validity (except for those created with the Add Hardware tool) but in some cases the checks are not as effective as your experienced eye.

ArtiosCAD determines layout congruency based on the position of an element relative to the layout. If an element is contained within a blank, that element is repeated within all similar blanks. If an element crosses blank boundaries, it is repeated where that same group of blanks appears. For those imported workspaces where manufacturing components are actually subprograms - treated as if they were part of the manufacturing workspace but truly are not - blanking Auto-Repeat will not work correctly.

Two or more elements have contour congruency if they can be transformed into each other by mirroring or rotation. When Auto-Repeat is enabled, if a change is made to an element, ArtiosCAD applies that change to all other elements having contour congruency.

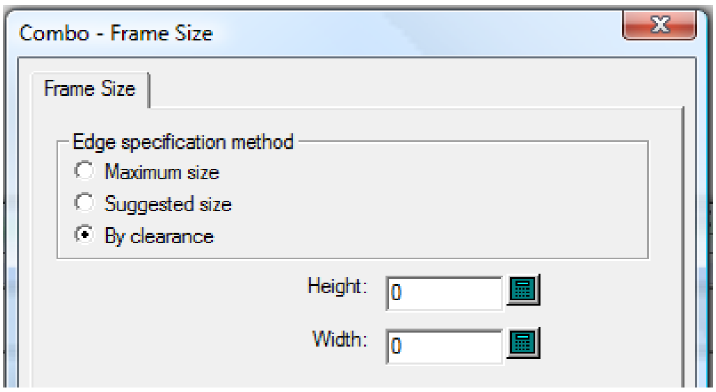
The table below lists the blanking tools and their congruency method.

Layout Congruency	Contour Congruency
Add Grid Bar	Generate Blanking Tools
Add Non-stop Swords*	Adjust Blank Holes
Add Support Bars*	Adjust Pusher
Add Jogger Guides	Add Blanker Pusher Pins
Add Air Hole	Smooth/Blend Contour
Add Hardware	Edit Contour
Add Presser	
*does not support mirrored blanks	

Frame Creation

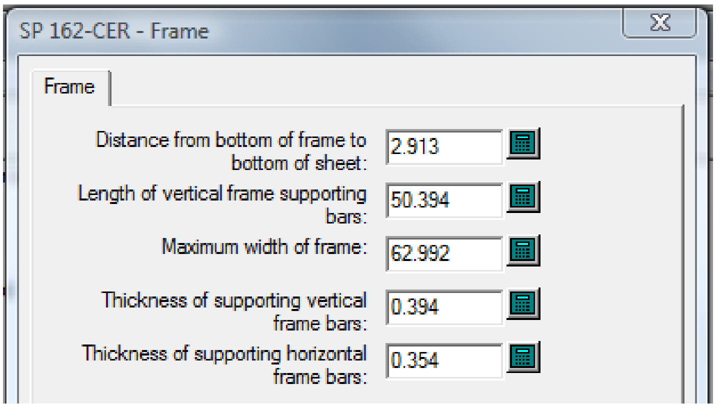
Most of the blanking tools require the frame to exist, so if the frame does not exist, ArtiosCAD prompts you to create it before starting the tool you clicked.

ArtiosCAD bases the frame size on the **Frame Size** parameter in the Frame catalog of a blanking parameter set.

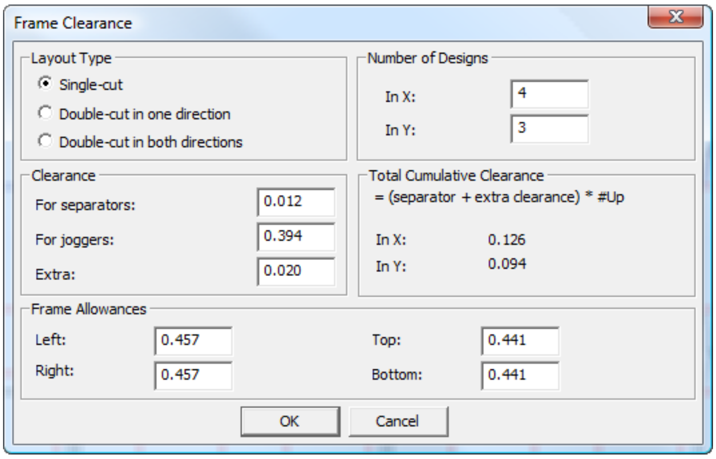


It can be:

- **Maximum size**, which takes the frame size from the size defined in the Frame catalog of the blanking machine parameter set.



- **Suggested size**, which uses the values you specify on the Frame Size tab.
- **By clearance**, which calculates the proper size from the clearances you supply in the Frame Clearance dialog box that appears when you create the frame.



When using **By clearance**, follow the recommendations of your press manufacturer for the values to supply in the **Clearances** group. One leading press manufacturer recommends adding a half millimeter plus the thickness of the separator for every blank for single-cut layouts. For double-cut layouts, the layout is treated as one large blank since the blanks are not connected.

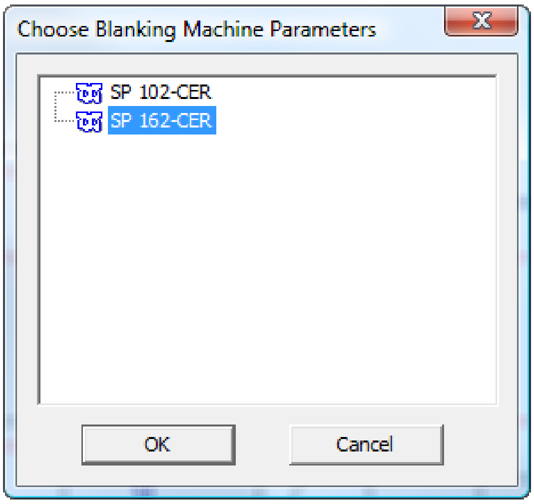
ArtiosCAD calculates the clearances based on the information you supply and creates the blanking frame accordingly.

Blanking Example Workflow: Lower Grid

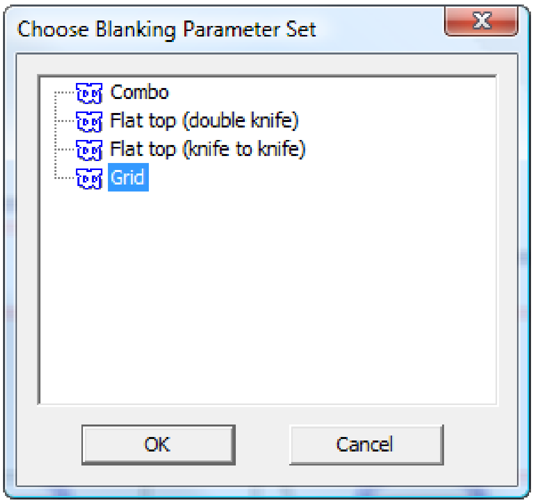
A common blanking method is using a grid as the lower blanking board. Here is one way to make a grid.

1. Design a layout in ArtiosCAD.
2. If desired, create stripping tools for the layout.

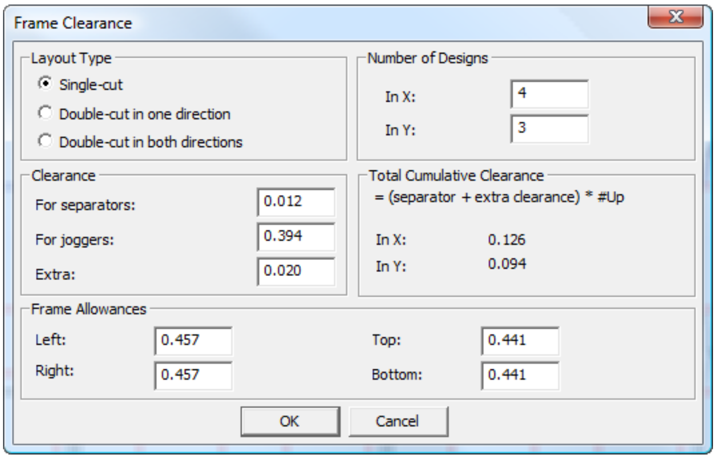
3. Click **Add Grid Bar** on the Blanking toolbar.
4. Choose a blanking machine parameter set in the Choose Blanking Machine Parameters dialog box and click **OK**.



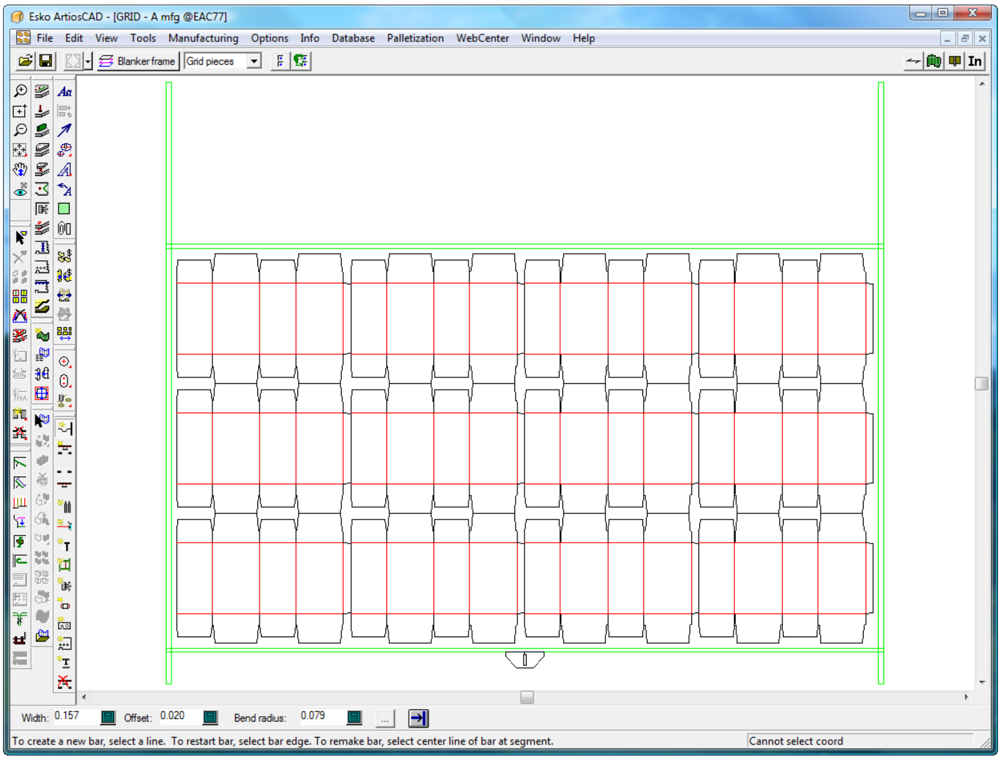
5. Choose a blanking parameter set in the Choose Blanking Parameter Set dialog box and click **OK**. For this example, choose **Grid**.



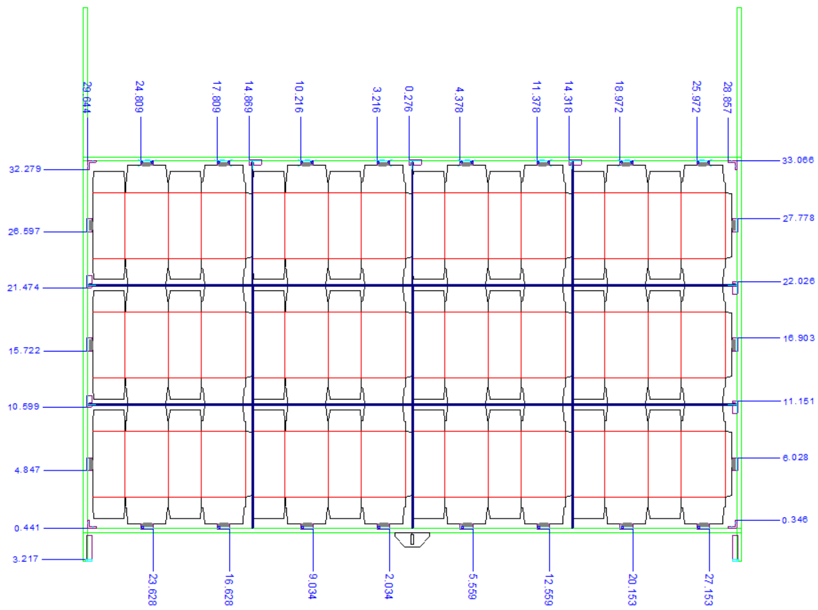
6. In the Frame Clearance dialog box, change any values as desired and click **OK**.



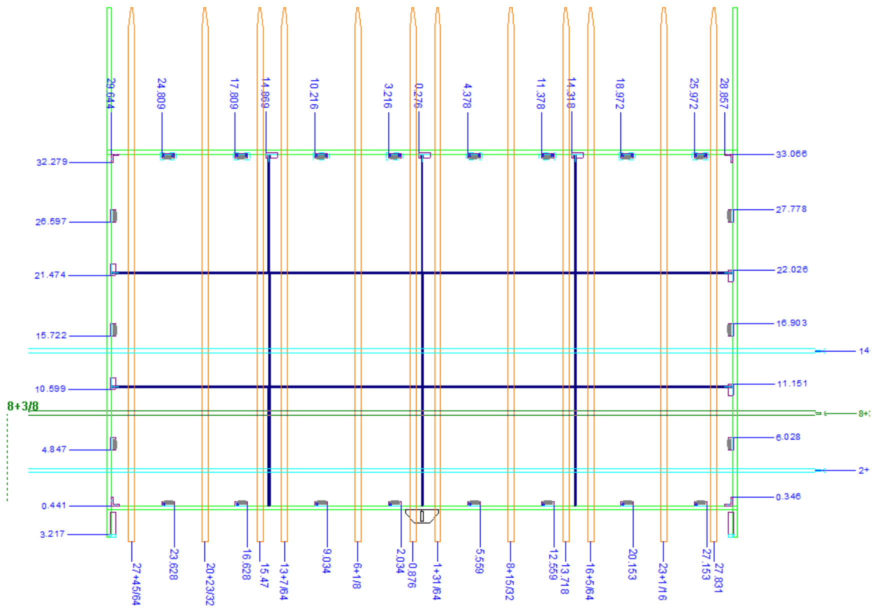
ArtiosCAD creates the grid frame, which is represented as double green lines around the layout.



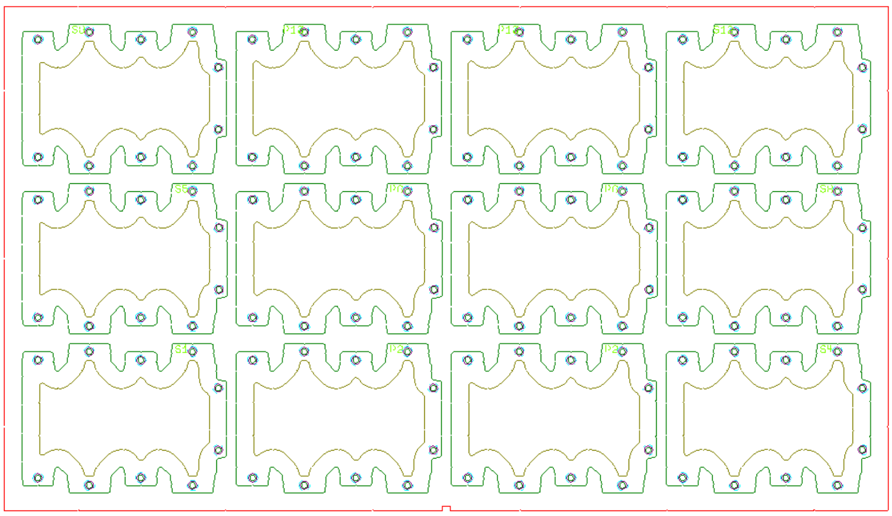
7.  Use **Add Grid Bar**, **Add Jogger Guide**, and **Add Hardware** to construct the grid. More information about how to use them is later in this guide.



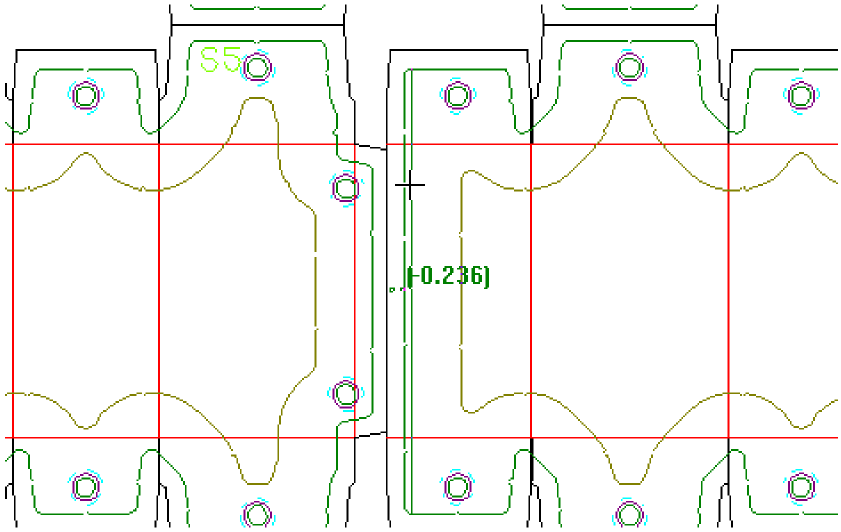
8.  Use **Add Non-Stop Swords** and **Add Photocells** to add those parts.



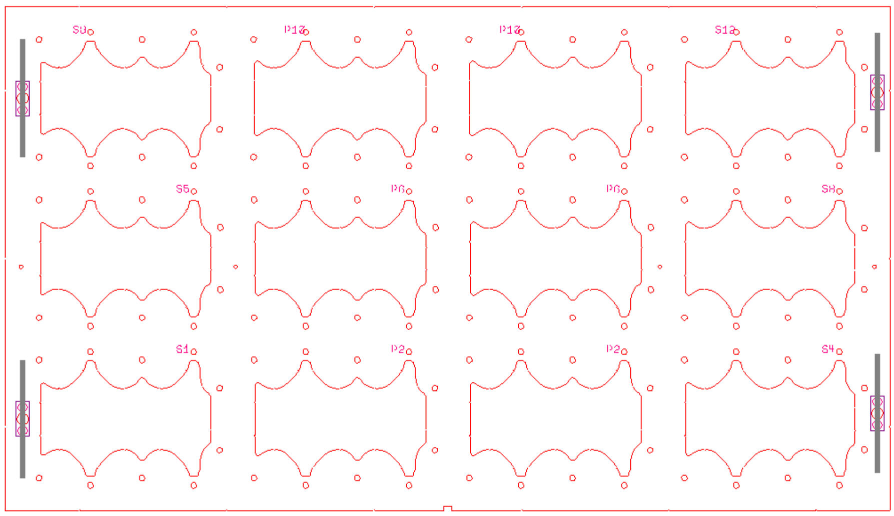
9. Use **Generate Blanking Tools** to create the pushers and the upper board.



10. Use **Adjust Pushers** to adjust the offsets from the pusher edge to the grid and/or design edge.



11. Finish the upper board by using **Add Tie-bolts** and **Add Pressers** to add the pressers.



12. Save the workspace and run a blanking Output such as the Blanking Parts Report shown below, or one of the grid bar reports.

Blanking Parts Report

File : GRID.MFG

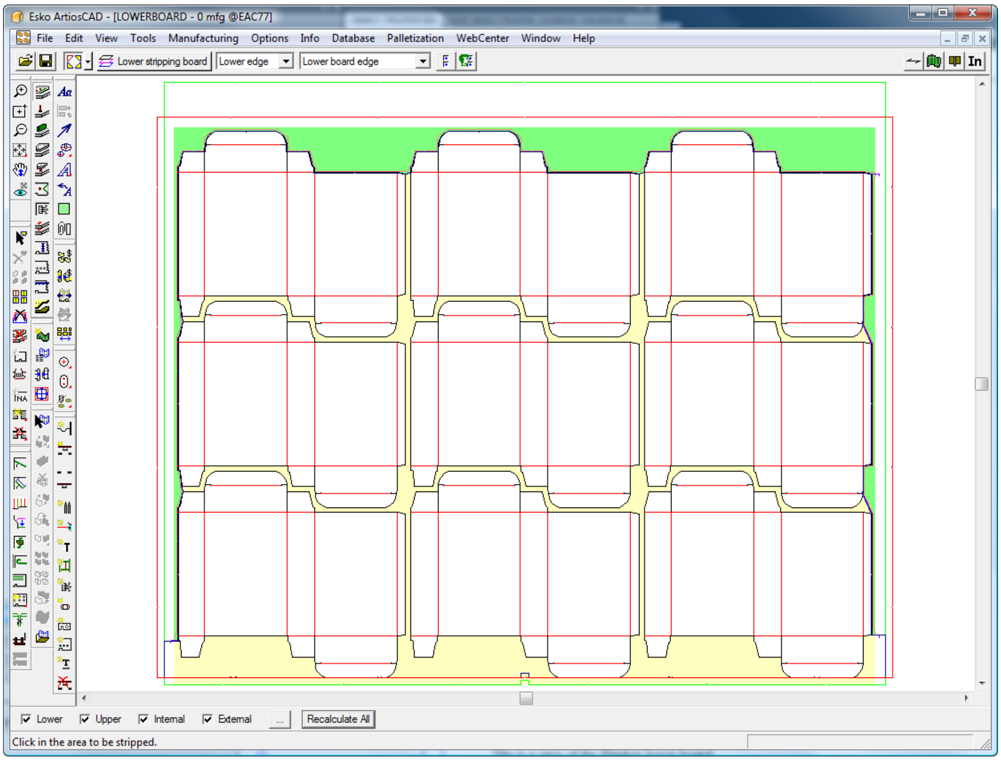
Date : 05/23/2011

8 - Front jogger 10mm
8 - Rear jogger 10mm
6 - Side jogger 10mm
7 - Grid Clamp for frame tool
4 - M6
4 - Long presser 6mm
120 - SOP 10mm rnd

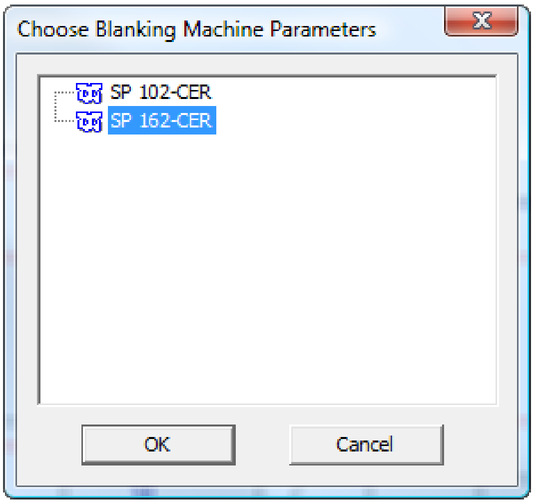
Blanking Example Workflow: Lower Board

Another common blanking method is using a lower board with holes in it for the blanks to be pushed through by the upper board. Here is one way to make a lower board blanking tooling set.

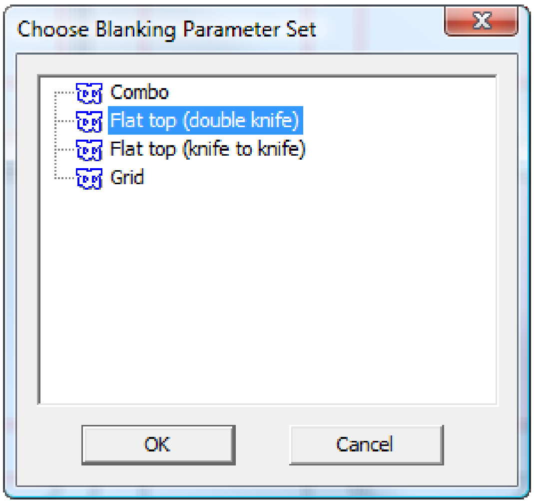
1. Design a layout in ArtiosCAD.
2. If desired, create stripping tools for the layout.



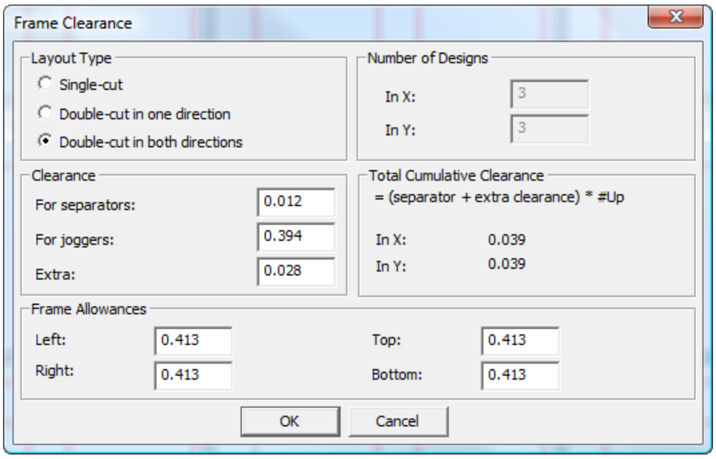
3.  Click **Generate Blanking Tools** on the Blanking toolbar.
4. Choose a blanking machine parameter set and click **OK**.



5. In the Choose Blanking Parameter Set dialog box, depending on if your layout is double-cut (the designs have a gutter between them) or single cut (the designs share common knives on their exteriors), choose either **Flat top (double knife)** or **Flat top (knife to knife)** and click **OK**.

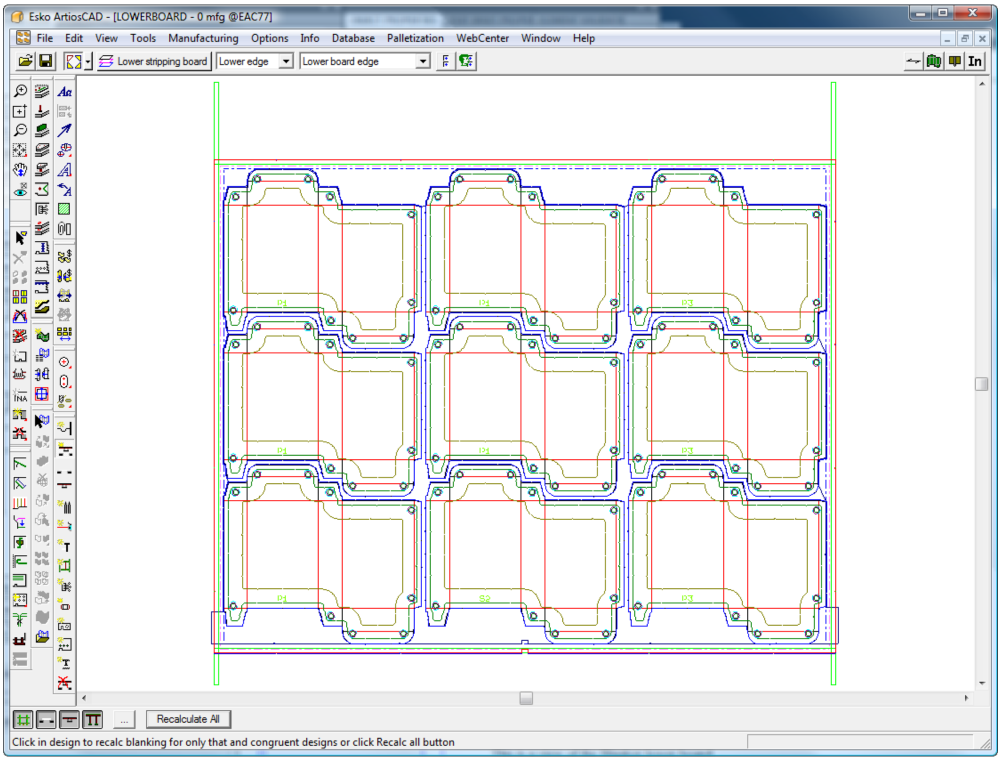


6. In the Frame Clearance dialog box, change any values as desired and click **OK**.

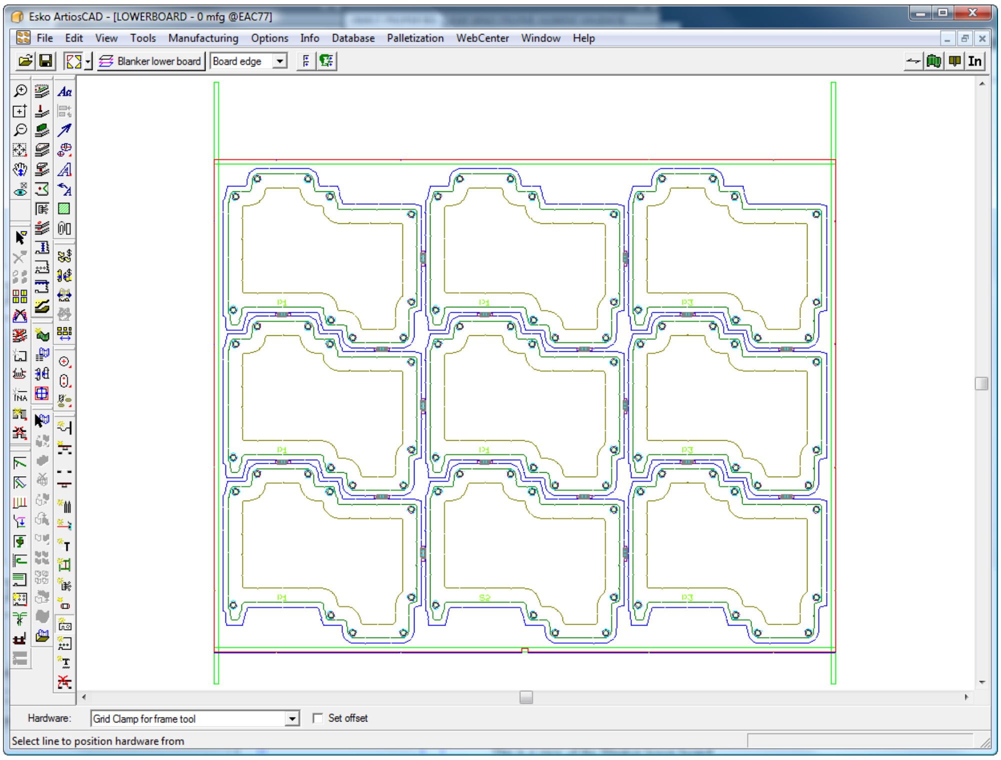


7. Make sure all four mode selection buttons are selected and click **Recalculate All** on the Status bar.

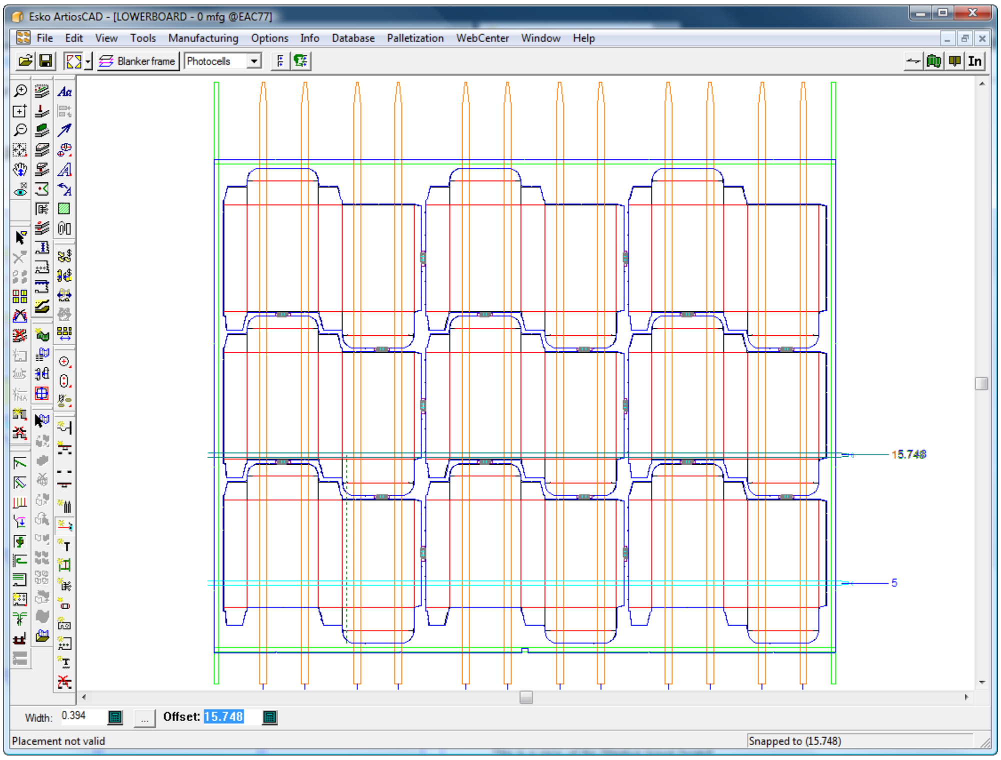
   ArtiosCAD automatically generates the frame, holes, and pushers for the layout.



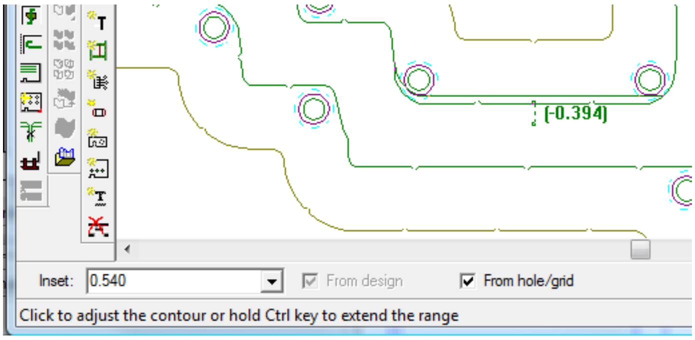
8.   Use **Add Jogger Guide** and **Add Hardware** to add more hardware. More information about how to use them is later in this guide.



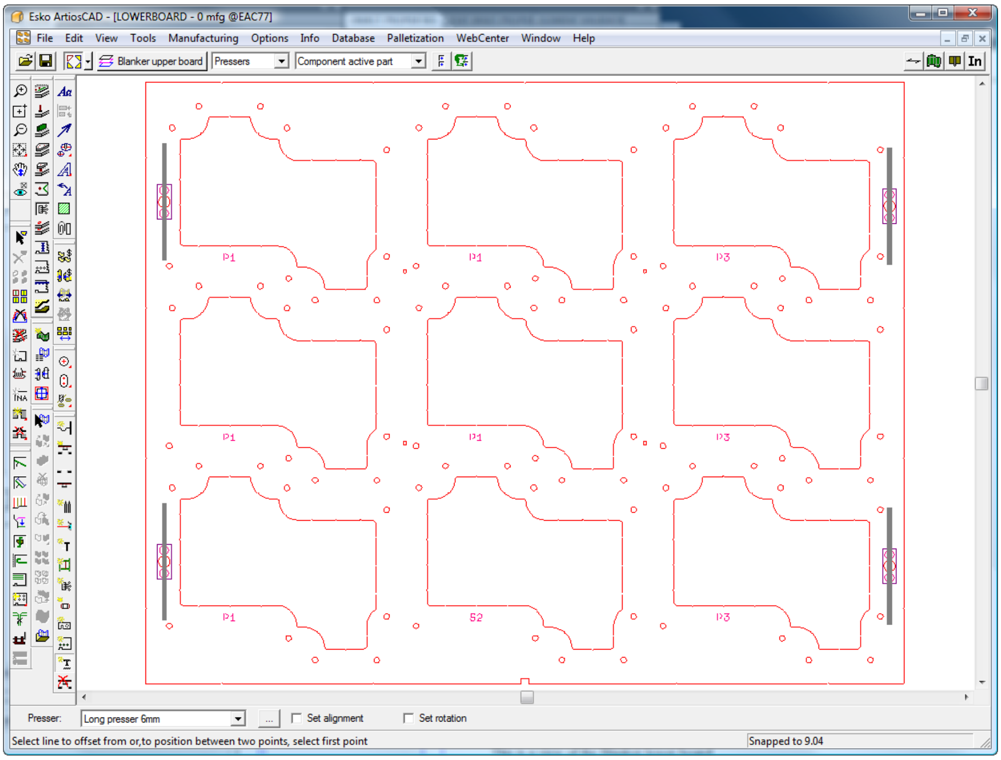
9. Use **Add Non-Stop Swords** and **Add Photocells** to add those parts.



10. If desired, use **Adjust Pushers** to adjust the offsets from the pusher edge to the hole and/ or design edge.



11. Finish the upper board by using **Add Tie-bolts** and **Add Pressers** to add the pressers.



12. Save the workspace and run a blanking Output such as a Blanking Parts Report.

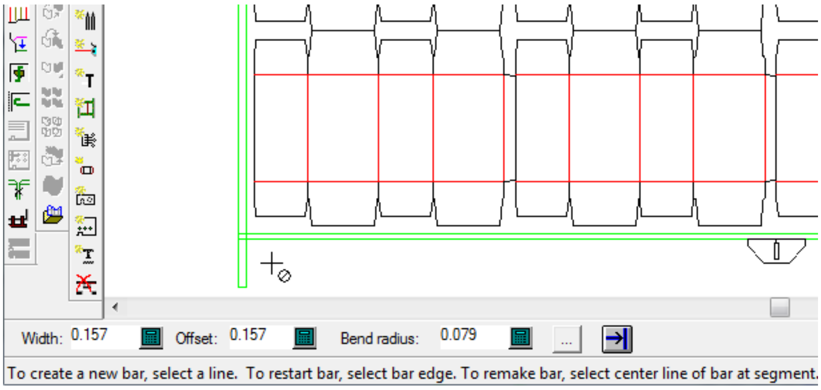
Blanking Tools Reference

This section explains how to use each tool on the Blanking toolbar.

Add Grid Bar tool

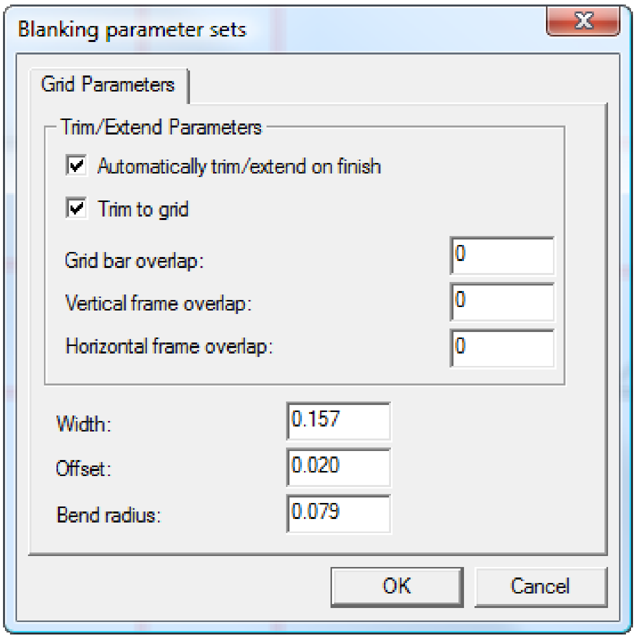
 The **Add Grid Bar** tool lets you add grid bars when constructing a grid-type lower blanker board. Grid bars separate the blanks as they are pushed through the grid. Use this tool to follow the contours of a blank and ArtiosCAD will construct the grid bar to match.

When this tool is active, the Status bar looks as follows.



The **Width**, **Offset**, and **Bend radius** fields all control the dimensions of the grid bar.

The **Properties** (...) button opens the Blanking parameter sets dialog box.

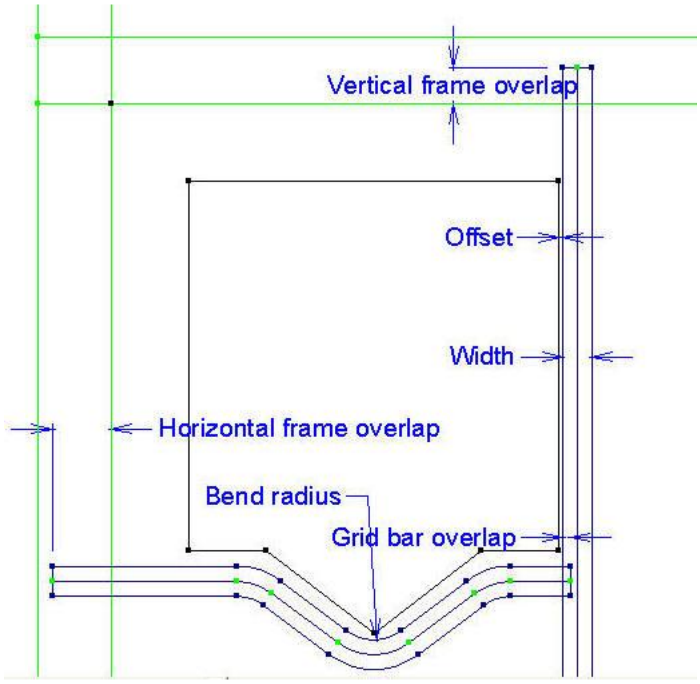


These parameters are explained by the following diagram, with the green lines being the frame edge and the triple black lines being a blanking grid bar (with **Fills and strokes** in the Styles group of the View mode turned off and **End points** turned on). The rectangle with the point is the blank.

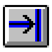
Note:

These parameters are saved with the manufacturing workspace.

Automatically trim/extend on finish trims or extends the grid bar so that it meets the frame when you click **Finish** on the Status bar, and **Trim to grid** trims the grid bars against each other.



In the diagram above, the grid bars are not filled and the end points are visible. To see just the edges of the grid bars, turn off **Fills and strokes** in the Graphics section of the View Mode; to see the edges and middle of the grid bars, turn off **Fills and strokes** in the Styles section of the View Mode but leave it on in the Graphics section.

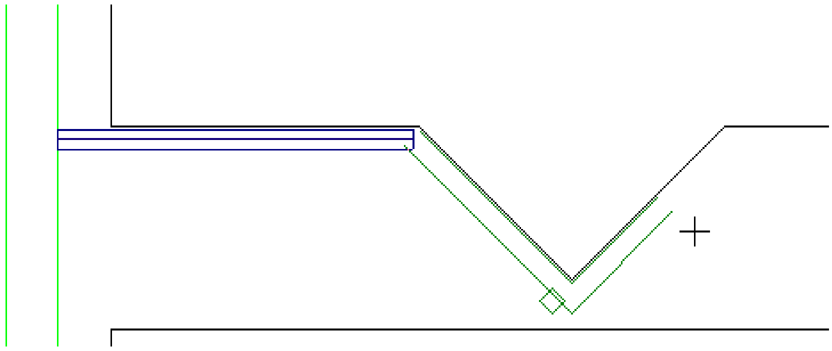
 **Finish** extends the grid bar to the closest frame edge if **Automatically trim/extend on finish** is enabled in the Grid Parameters dialog box, and if Auto-Repeat is enabled, repeats the grid bar to congruent sections of the layout. ArtiosCAD restarts the tool so you can create a new grid bar.

Copy Pickup/Putdown mode lets you copy a grid bar as you create it and optionally mirror it horizontally or vertically as desired.

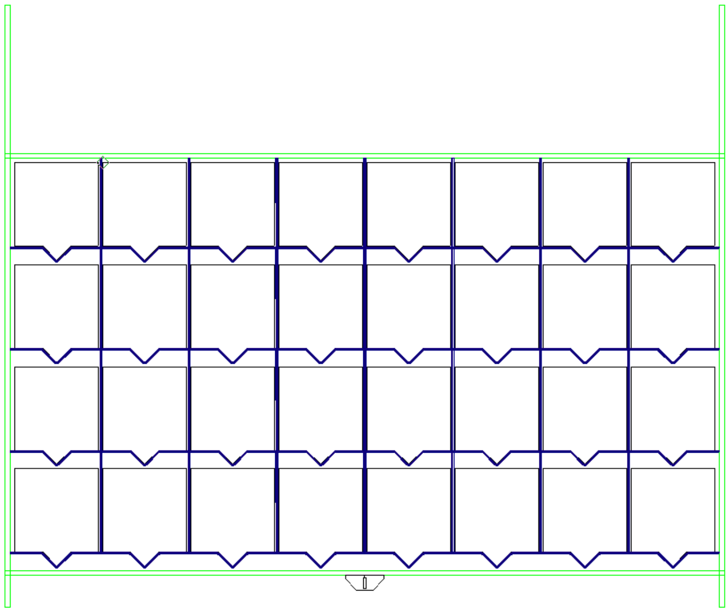
To create a new grid bar, do the following:

1.  In a layout, click **Add Grid Bar** on the Blanking toolbar. If no blanking has been done so far, ArtiosCAD prompts you to select a machine and a parameter set, and creates the blanking frame.
2. Click the line in the blank closest to the grid frame against which the grid bar is offset. Drag will appear from the grid frame allowing you to position the grid bar either on top of the line or offset from it depending on where you position the cursor - either to the left or right of the line. Hold down **CTRL** to set the offset halfway between the line and the cursor as indicated by the drag. To center a grid bar between two points (instead of offsetting it from a line), click those two points. The cursor has a circle with a slash through it when you are not close enough to a line.

3. Click the next lines in the blank to follow, using the position of the cursor to determine the offset, or click a point to offset the segment halfway between the point and the line. Click the line itself to center the grid bar on the line. Use **Copy Pickup/Putdown** mode to aid construction where possible. Copy Pickup/Putdown mode is available as you are constructing grid bars. First click the button on the Status bar, then select the grid segment to copy, and then click the pick-up point and the put-down point.



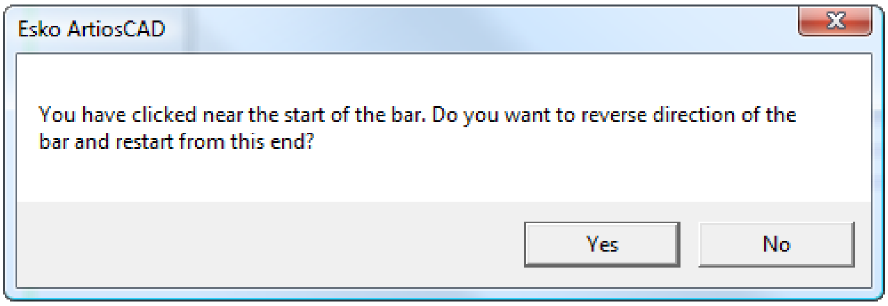
4. Continue constructing grid segments, using **Undo** and **Redo** to correct mistakes.
5.  When you have rounded the last bend before the frame, click **Finish** to extend the grid bar to the edge and Auto-Repeat it to congruent areas if Auto-Repeat is enabled.
6. Create the other grid bars in the same way. The example below was created by following the blank lines horizontally but using the two points on adjoining blanks to construct the vertical segments.



To see which end of a grid bar is the start and which is the end, turn on **Direction** in the View Mode.

To restart a grid bar, click the edge of the end of the bar.

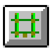
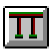
To reverse the direction of a bar and recreate it, click the edge of it near its start and click **Yes**.



Generate Blanking Tools tool

 **Generate Blanking Tools** is a one-click method to create blanking tools. It can create the frame, generate blank holes in the lower board (if used), generate pushers, and so on depending on the parameter set you choose. It does not create grid bars.

This tool has the following controls on the Status bar:

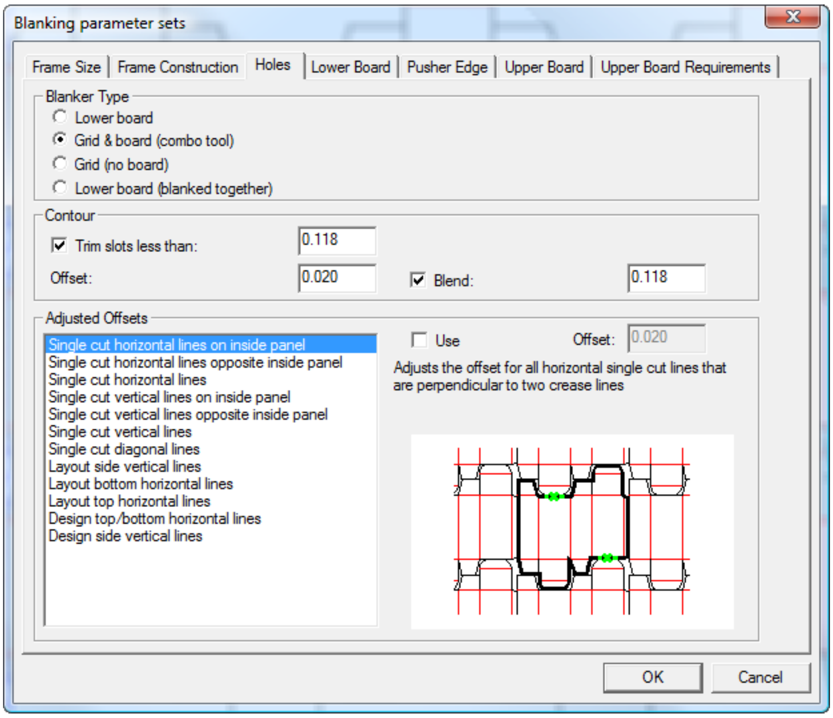
Control	Generates
	The blank frame
	The blank holes - creates blank holes and the lower board wood edge
	The pusher contours - creates pushers and upper board wood edge
	The pusher details

The initial states of these controls are determined by which layers contain anything or are turned on. These controls are independent from each other; click the ones corresponding to the blanking elements for ArtiosCAD to generate. Hold down `CTRL` to toggle the state of the other controls as you click one.

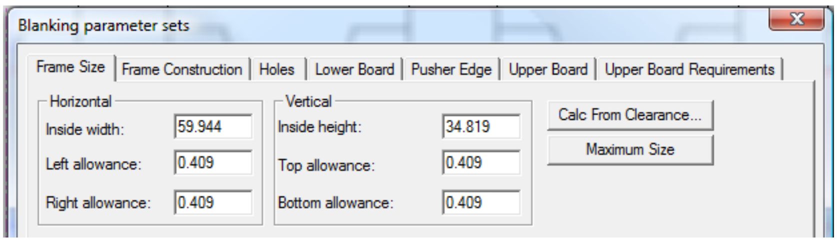
Clicking the Properties button (**...**) opens the Blanking Parameter Sets dialog box whose contents are based on the controls selected for generation on the Status bar.

Clicking **Recalculate All** generates the selected tools with the parameters from the Blanking Parameter Sets dialog box. To generate the tools for just one blank, click inside the single design for that blank.

Your Blanking Parameter Sets dialog box may not have the same tabs as the example shown below.



Set the allowances and clearances for the frame on the **Frame Size** tab:

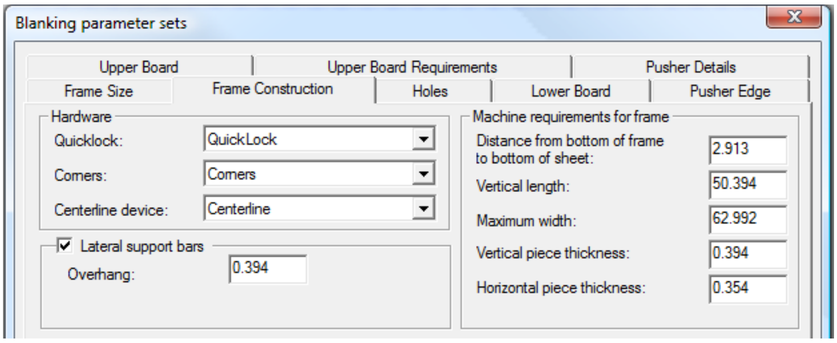


The frame allowances are relative to the layout size. **Calc From Clearance** opens the Frame Clearances dialog box discussed in the Frame Creation section. When you close the Frame Clearances dialog box, the values on the Frame Size tab are updated accordingly.

Clicking **Maximum Size**:

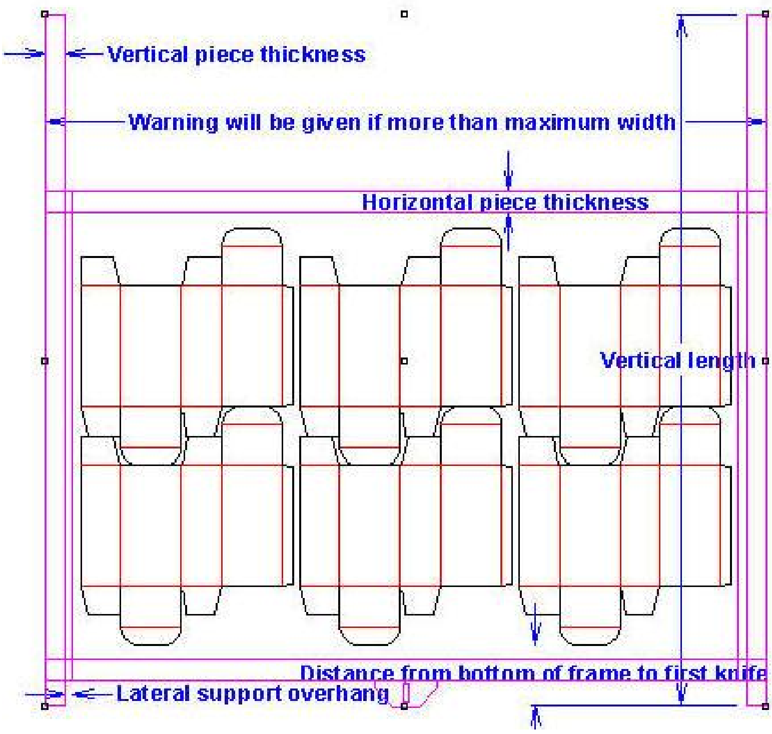
- sets **Inside width** and **Inside height** to their maximum values allowed by the machine
- leaves **Bottom allowance** unchanged
- resets the **Top allowance** to (Maximum height - layout height - Bottom allowance)
- recalculates the **Left allowance** and **Right allowance** to (Maximum width - layout width/2).

Set the parameters for the frame's construction on the **Frame Construction** tab.

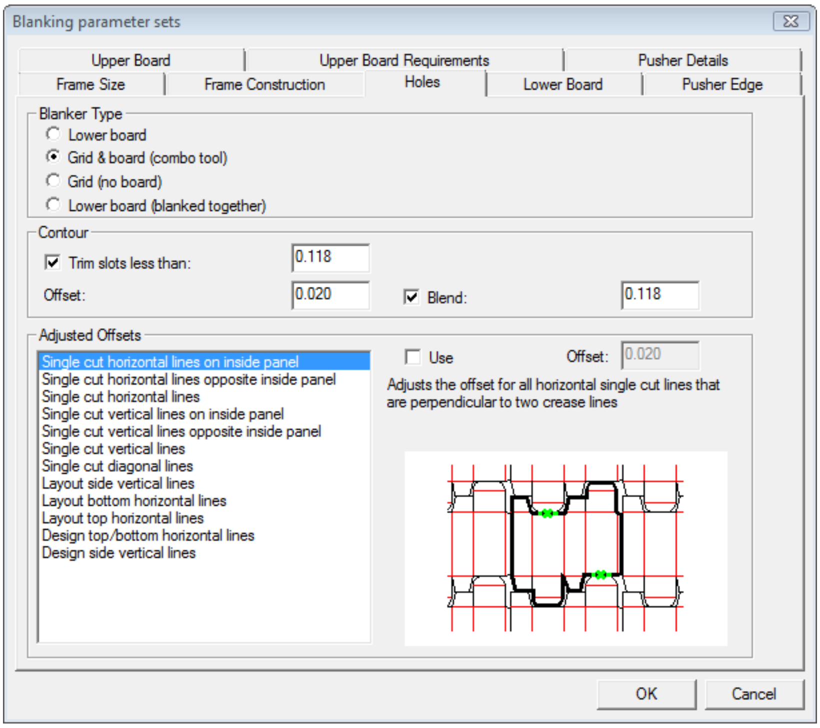


Choose the type of hardware to use on the frame from the drop-down list boxes in the **Hardware** group. You can make your own custom hardware as geometry macros and add them to the Blanking catalog in the Geometry Macro catalog in Defaults.

Refer to the diagram below to set the other fields on this tab.



The fields on the **Holes** tab control how the holes are made in the lower board (if applicable).

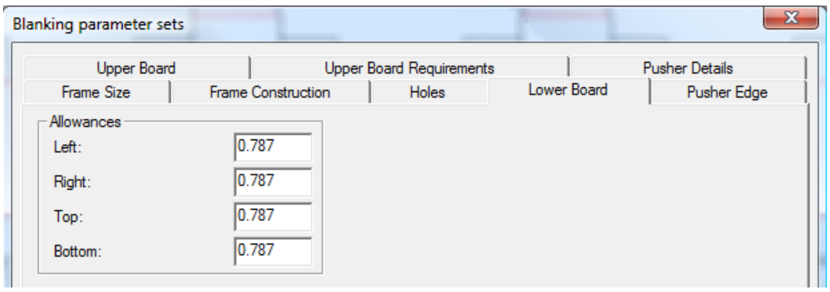


In the **Blanker Type** group, choose the type of blanking for the tool to perform. This should be pre-selected from the parameter set.

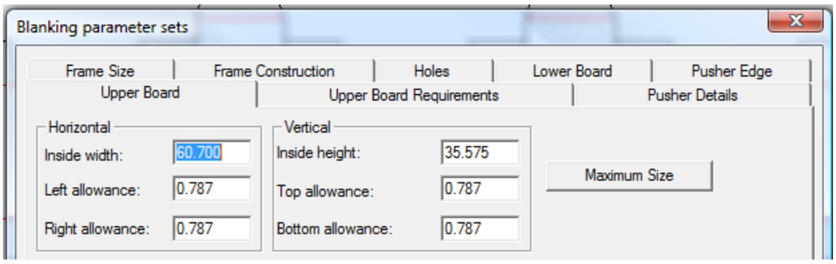
In the **Contour** group, set the values for the paths in the layout which ArtiosCAD will follow to create holes in the lower board.

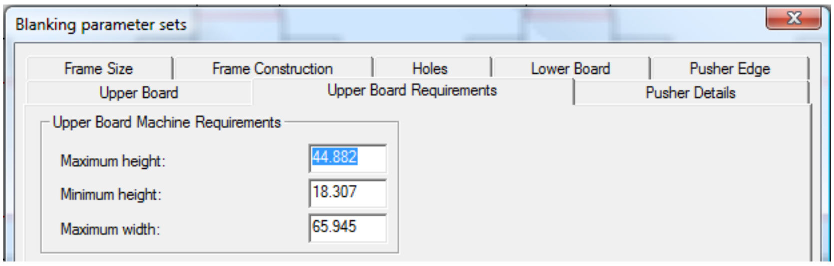
In the **Adjusted Offsets** group, choose the type of lines that require adjusted offsets for their corresponding hole contours. The preview picture shows each type of offset. Make sure to check **Use** for each desired offset while it is selected, and adjust the size of the offset as needed in the **Offset** field. ArtiosCAD marks each in-use offset with an asterisk in its name.

As the size of the lower board is determined from the frame size, not the layout, the only fields on the **Lower Board** tab are for the allowances. To adjust the lower board's alignment method and corner shape, right-click its edge after it is created and click **Properties** on the context menu.



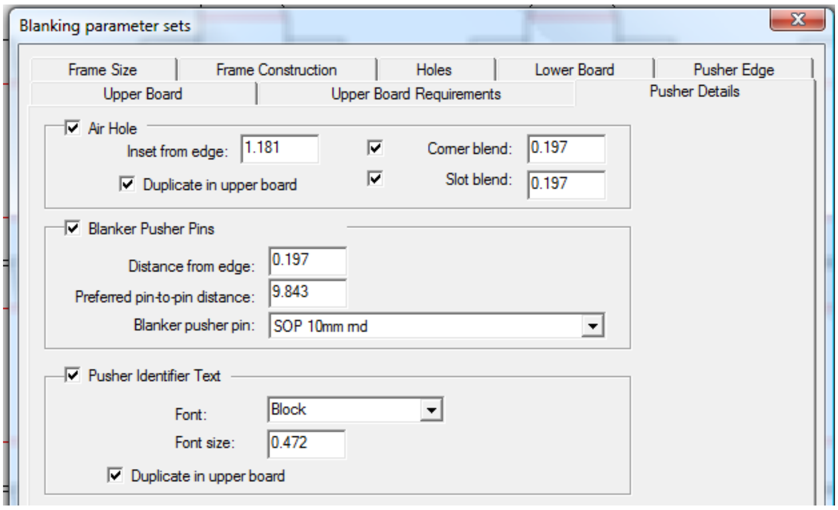
The size of the upper board is based on the layout size, not the frame size. To adjust the upper board's alignment method and corner shape, right-click its edge after it is created and click **Properties** on the context menu. On the Upper Board tab, adjust the size and allowances as desired. Alternately, click **Maxiumum size** to make it as large as possible using the parameters from the **Upper Board Requirements** tab.

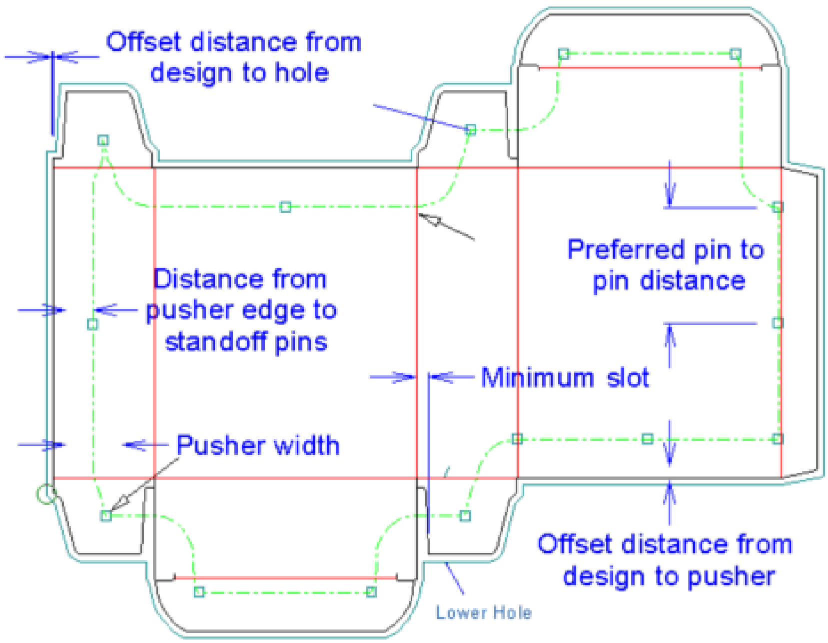




The **Pusher Details** and **Pusher Edge** tabs control the creation of the pushers on the upper board.

Settings for the air hole, blanker pusher pins, and pusher identification text are on the Pusher Details tab as shown below.

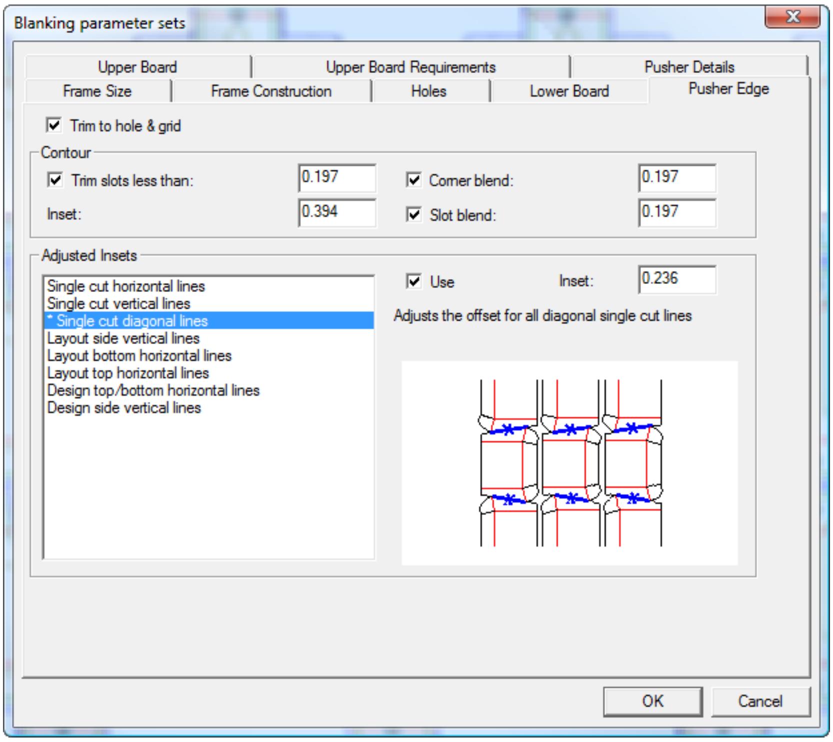




As with the frame, you can create your own blanker pusher pins as geometry macros and add them to the **Stand off pins** sub-catalog in the **Blanking** catalog of geometry macros in Defaults.

The pusher ID text is either an S# or a P# where # is a number, S stands for station, and P stands for pusher. The P# form indicates that there may be one or more other identical pushers. The S# form uniquely identifies the pusher in the layout. For example, if you have a double-knife layout of one design with Auto-Repeat enabled, then when you generate the pushers, all the pushers should have the same P# ID text. If you then disable Auto-Repeat and make an adjustment to a pusher, then that pusher will use the S# form.

The **Pusher Edge** tab controls how ArtiosCAD shapes the pushers. The options in **Adjusted Offsets** work the same as they do on the **Holes** tab - select an inset, check **Use**, and set the inset in the field. Only insets with asterisks are used when ArtiosCAD generates the blanking tooling.



Once you have set the parameters on all the tabs as desired, click **OK** to return to using the tool.

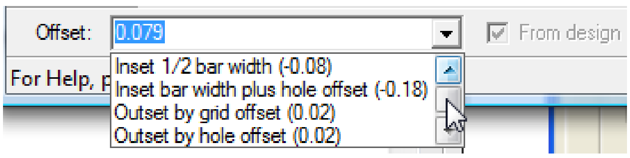
To use this tool, do the following:

1. Create a layout in ArtiosCAD.
2.  Click **Generate Blanking Tools** on the blanking toolbar.
3. Choose the machine and parameter set.
4. Set the frame parameters in the Frame Clearance dialog box and click **OK**.
5. On the Status bar, choose the items to generate.
6. Click the Properties button (**...**) to open the Blanking parameter sets dialog box. In it, review the settings on each tab ArtiosCAD will use to generate the tools. Click **OK** when done.
7. Click **Recalculate All** on the Status bar.

Adjust Blank Hole tool

 The **Adjust Blank Hole** tool on the Blanking toolbar allows you to adjust the shape of a blank hole in the lower board. If Auto-Repeat is enabled, ArtiosCAD modifies all congruent blank holes.

When you click this tool, ArtiosCAD turns on the Blanker lower board layer, turns off the other blanking layers, and puts an **Offset** drop-down list box on the Status bar:

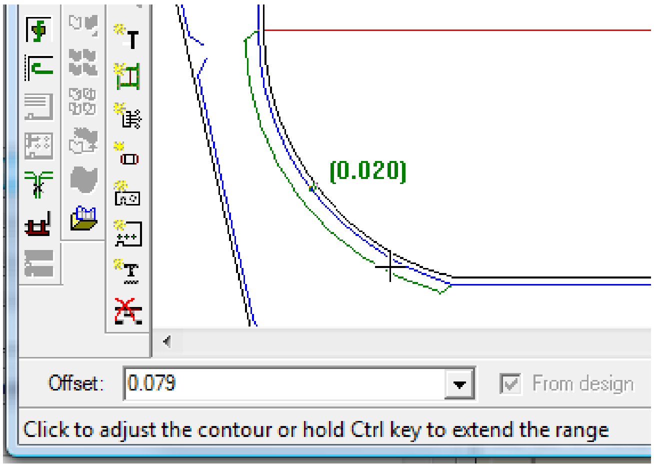


You can either type a value or expression in the **Offset** field or choose a value from the drop-down list. A positive number offsets the hole outside the blank; a negative number insets it inside the blank.

From design is unavailable, but when it is checked, it indicates that the current offset of the line or arc is the distance perpendicularly from the midpoint of the line to the design.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the lower blanking board.
2. Click **Adjust Blank Holes**.
3. On the Status bar, adjust the offset as desired.
4. Move the cursor close to the line to change. The drag will indicate the adjusted offset while the number in the drag is the current offset. To change more than one line, hold down CTRL and drag to the next connected line.



Note:

Only change one or two lines at a time as the adjustment's start angle is set by the first line you choose. Choosing more than two lines at a time may result in incorrect adjustments.

5. Click the line to change. To change more than one line at once, click the last line in the sequence.
6. ArtiosCAD adjusts the hole edge.

Adjust Pushers tool

 The **Adjust Pushers** tool changes the shape of the pushers on the upper board. When you click this tool, ArtiosCAD makes the Blanker pushers layer active and turns off the other blanking layers.

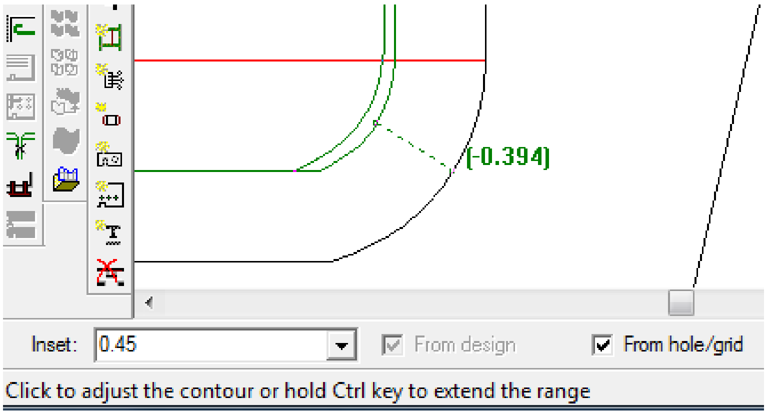
This tool has the following controls on the Status bar:

Inset is the distance the current pusher edge is from the hole edge or grid bar. A positive value moves the pusher edge further from the hole or grid; a negative value brings it closer. Any values you type in this field will appear on the drop-down list for later use in this session.

From design is unavailable, but when it is checked, it indicates that the current offset of the line or arc is the distance perpendicularly from the midpoint of the line to the design.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2. Click **Adjust Pusher**.
3. On the Status bar, adjust the inset as desired.
4. Move the cursor close to the line to change. The drag will indicate the adjusted inset while the number in the drag is the current inset. To change more than one line, hold down `CTRL` and drag to the next connected line.



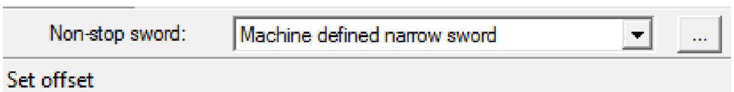
> **Note:**
>
> Only change one or two lines at a time as the adjustment's start angle is set by the first line you choose. Choosing more than two lines at a time may result in incorrect adjustments.

5. Click the line to change. To change more than one line at once, click the last line in the sequence.
6. ArtiosCAD adjusts the pusher edge.

Add Non-stop Swords tool

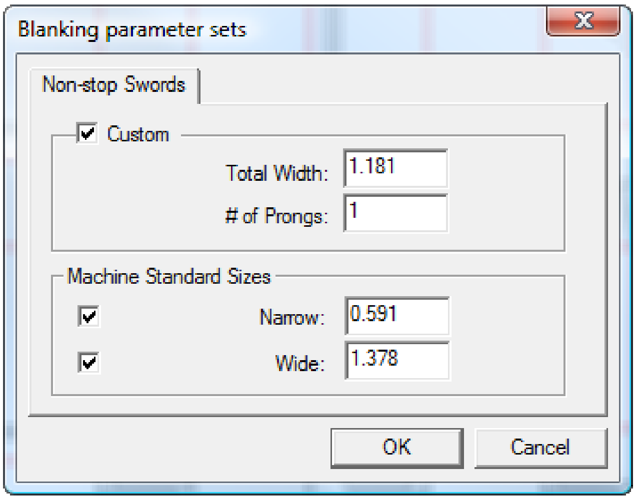
 The **Add Non-stop Swords** tool on the Blanking toolbar lets you indicate where the non-stop swords will go on the press. They may not overlap other swords or joggers. When you click this tool, ArtiosCAD turns on the Blanker frame layer and turns off the other blanking layers.

This tool has the following controls on the Status bar:



In the **Non-stop sword** drop-down list box, you can choose to add a machine-defined narrow sword, a machine-defined wide sword, or a custom sword.

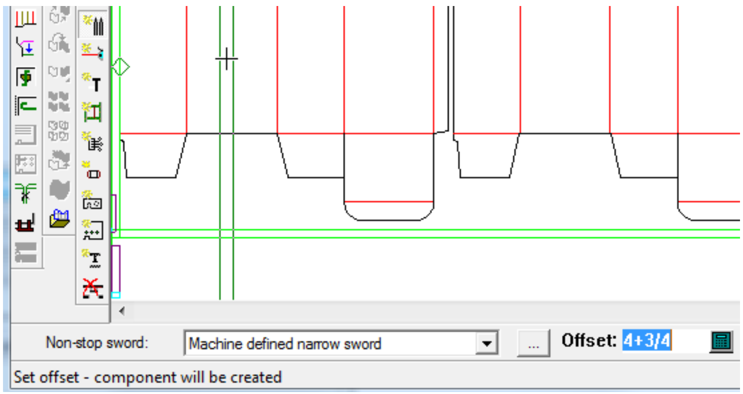
Clicking Properties (**...**) opens the Blanking parameter sets dialog box where you can set the availability, number of prongs, and width of the swords.



Click **OK** when you have set the values as desired.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Non-stop Swords** on the Blanking toolbar.
3. Select a line or point from which to offset the leftmost sword. Cut lines display their points when you place the cursor near them for easy placement, but you can select any design line. If you select a line, ArtiosCAD prompts you for the offset for the sword, which you can set with drag or by entering a value in the **Offset** field. If you select a point, ArtiosCAD prompts you for another point or freehand coordinate (by holding down CTRL) and places the sword halfway between that and the first point.



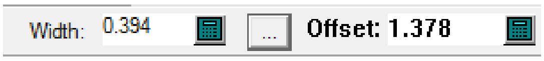
4. If Auto-Repeat is enabled, ArtiosCAD repeats the sword to congruent areas.
5. Continue adding swords as desired. The drag is green for valid positions and red for invalid positions.

Add Photocells tool

 The **Add Photocells** tool lets you add photocells to the blanking tooling. Photocells signal the press when the stack of blanks needs to be removed from the press to make room for new blanks

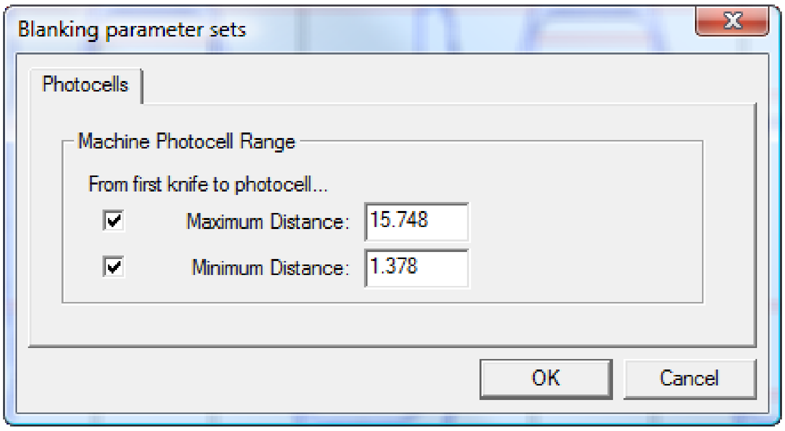
to accumulate. Photocells must not overlap joggers or other photocells. When you click this tool, ArtiosCAD turns on the Blanker frame layer and turns off the other blanking layers.

This tool has the following controls on the Status bar:



In the **Width** field, set the width of the photocell beam. This field is preset from the values in the Blanking parameter sets dialog box opened by the Properties (**...**) button. Use the drag to set the **Offset** or enter a value in the field manually. The drag is green for valid positions and red for invalid positions.

Shown below is the Blanking parameter sets dialog box.

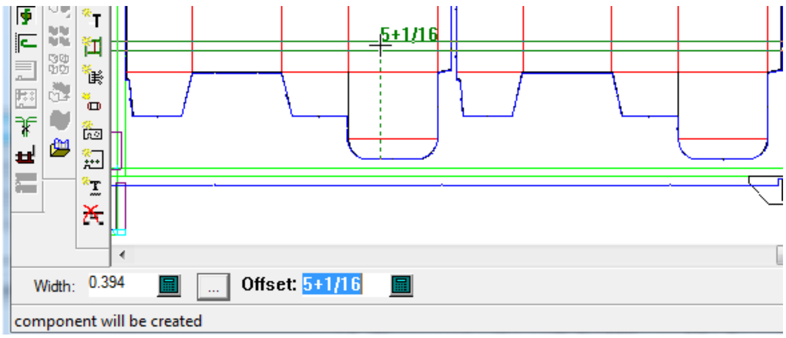


The **Maximum Distance** and **Minimum Distance** fields control the range of placement for the photocells. The checkboxes before them enable their limits; for example, to be able to set the photocell anywhere in the frame after the minimum distance, deselect **Maximum Distance**.

Click **OK** when you have set the values as desired.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Photocells** on the blanking toolbar.
3. If desired, change the width of the photocell in the **Width** field.
4. Use the drag to set the offset for the photocell's position, or enter it manually in the **Offset** field.



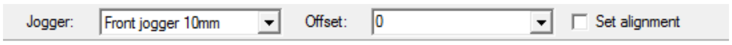
5. Continue adding photocells as desired. The drag is green for valid positions and red for invalid positions.

Add Jogger Guides tool

 The **Add Jogger Guides** tool lets you position joggers, the hardware that helps guide the blanks as they are separated from the sheet. When you click this tool, ArtiosCAD turns on the Blanker frame layer and turns off the other blanking layers.

Joggers must not overlap other joggers, photocells, or non-stop swords. ArtiosCAD checks three parts of the jogger for interference: below the board (the active part), at the board level, and above the board.

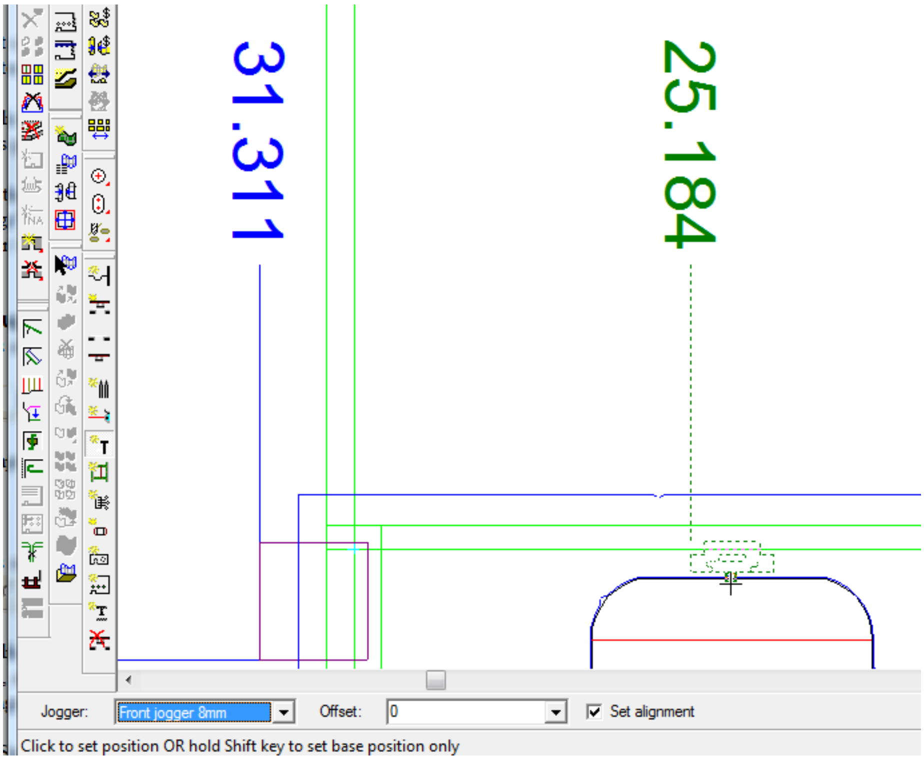
This tool has the following controls on the Status bar:



Select the jogger to use from the **Jogger** drop-down list box. In the **Offset** drop-down list box, select **0**, **Midway between 2pts**, or enter a custom value. Check **Set alignment** so that when you click to add a jogger, you first place the jogger on a line and then click a point with which to align it.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Jogger Guides** on the Blanking toolbar.
3. Choose the jogger and offset from the drop-down list boxes on the Status bar.
4. Place the jogger in the desired position in the layout by clicking where you want it along a line in the frame or a design edge. ArtiosCAD will measure the offset of the jogger edge from the center of the blanking frame for joggers placed along the frame. You can achieve different positioning by holding down `SHIFT` or `CTRL` as you click; follow the prompts on the Status bar. The drag will be green for valid positions and red for invalid positions. If you want to align the jogger with a point, check **Set Alignment** on the Status bar before clicking its position. If Auto-Repeat is enabled, ArtiosCAD repeats the jogger to congruent areas.



5. Continue adding joggers as desired.

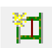
Add Support Bars tool

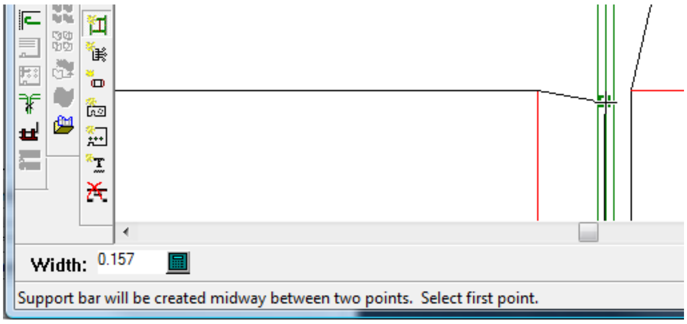
 The **Add Support Bars** tool lets you add support bars to the lower blanking board for reinforcement. When you click this tool, ArtiosCAD turns on the Blanker frame layer and turns off the other blanking layers.

This tool has the following control on the Status bar:

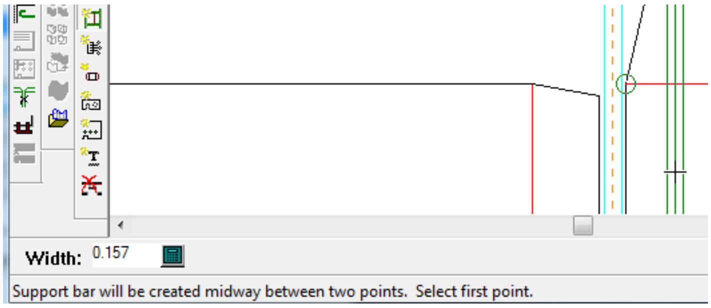
Set the width of the support bar in the **Width** field. The initial value is part of the blanking parameter set.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Support Bars** on the Blanking toolbar.
3. ArtiosCAD positions support bars midway between two points. Click the first point.



4. Click the second point or hold down CTRL and click a freehand point.

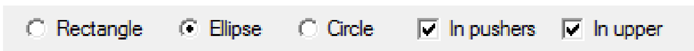


5. Continue adding support bars as desired.

Add Air Holes tool

 The **Add Air Holes** tool lets you add air holes in the pushers, the upper blanking board, or both. Air holes reduce the suction created by the movement of the tooling. When you click this tool, ArtiosCAD turns on the Blanker upper board layer and turns off any other blanking layers.

This tool has the following controls on the Status bar:

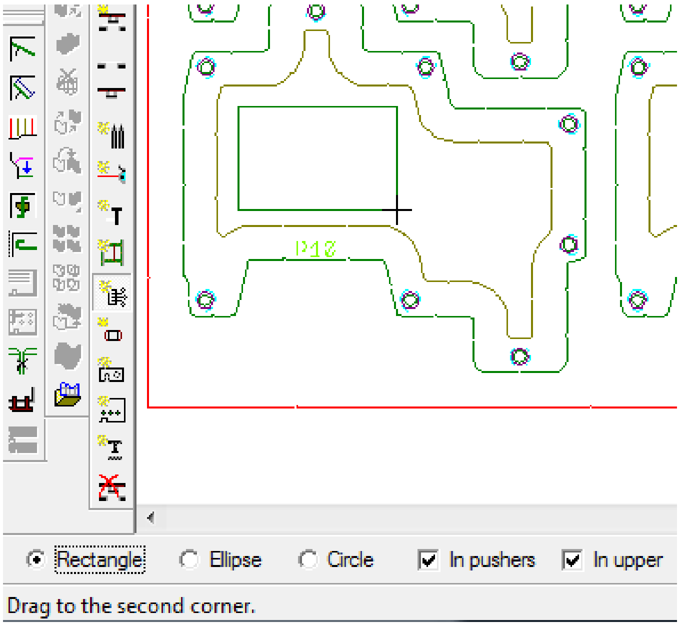


Choose the shape of the air holes from the option buttons and choose the objects in which to create the air holes from the checkboxes.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Air Holes** on the Blanking toolbar.
3. Choose the shape of air holes to add and the objects in which to create them.
4. Follow the prompts on the Status bar to create a hole. If Auto-Repeat is enabled, ArtiosCAD will use layout congruency to repeat it.

> **Note:**
>
> There is no validation to ensure that hole edges do not cross each other or pusher edges. Be careful with your construction.

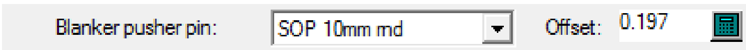


5. Create more holes as desired.

Add Blanker Pusher Pins tool

 The **Add Blanker Pusher Pins** tool adds the pins that attach the pushers to the upper blanking board. These are also called stand-off pins (SOP). When you click this tool, ArtiosCAD turns on the Blanker upper board and Blanker pusher layers and turns off the other blanking layers.

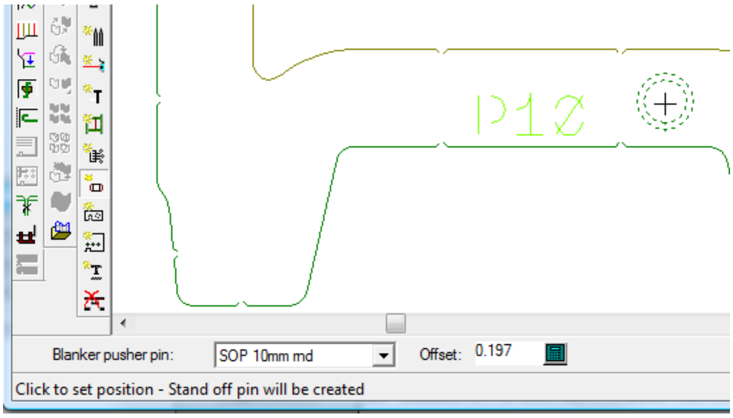
This tool has the following controls on the Status bar:



Choose the pin to add from the **Blanker pusher pin** drop-down list box. The **Offset** field sets the distance from the pusher edge to the pin. The initial value is part of the blanking parameter set.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Blanker Pusher Pins** on the Blanking toolbar.
3. Choose the type of pin to add from the list in the **Blanker pusher pin** drop-down list box.
4. If desired, change the offset for the pin in the **Offset** field.
5. Click the position inside the pusher edge where you want to place the pin.



6. Continue adding pins as desired.

Add Hardware tool

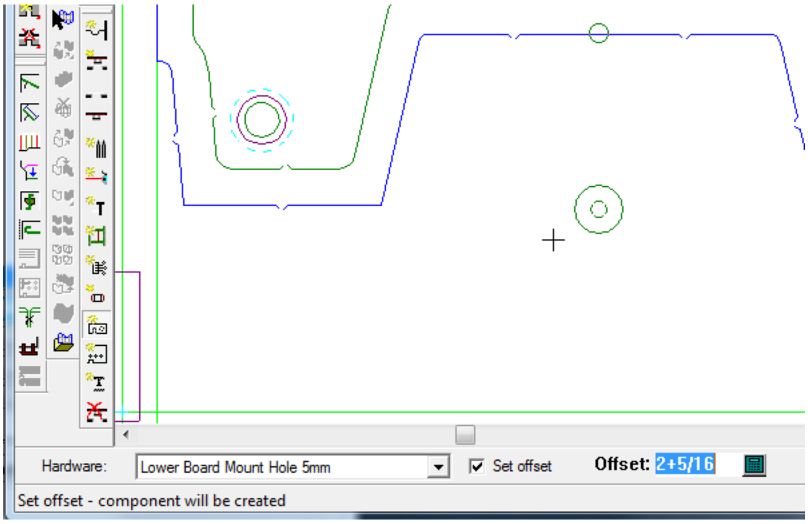
 The **Add Hardware** tool lets you add pieces of additional hardware to any of the blanking layers. You can design your own pieces of hardware, add them to the appropriate geometry macro catalog, and use this tool to add them to the blanking tooling.

This tool has the following controls on the Status bar:

Choose the hardware to add from the **Hardware** drop-down list box. To be prompted for an offset after selecting the hardware's alignment point, check **Set offset**. Lines of lanker lower board types are copied to that layer, lines of blanker upper board types are copied to that layer, lines of blanker pusher types are copied to that layer, and the rest remain in the Blanker frame layer.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Hardware** on the Blanking toolbar.
3. Choose the hardware to add from the **Hardware** drop-down list box and click **Set offset** to be prompted to set an offset after aligning the hardware.
4. Place the hardware by clicking a line to place it against using drag and click to set its alignment point.
5. If you selected **Set offset** before placing the hardware, set the offset either with drag or by typing a value in the **Offset** field.

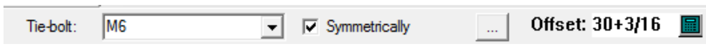


6. Continue adding hardware as desired.

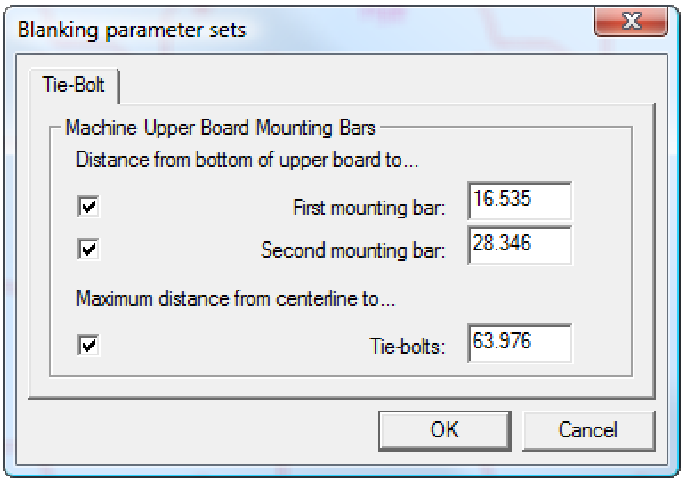
Add Tie-bolts tool

 The **Add Tie-bolts** tool lets you add holes for the bolts that attach the boards to the press. When you click this tool, ArtiosCAD makes the Blanker upper board layer active and turns off the other blanking layers.

This tool has the following controls on the Status bar:



Choose the tie-bolt to use from the **Tie-bolt** drop-down list box. To add a corresponding bolt on the other side of the center line, leave **Symmetrically** checked. Set the offset of the tie-bolt either with drag or by entering a value in the **Offset** field. Clicking Properties (**...**) opens the Blanking parameter sets dialog box as shown below.

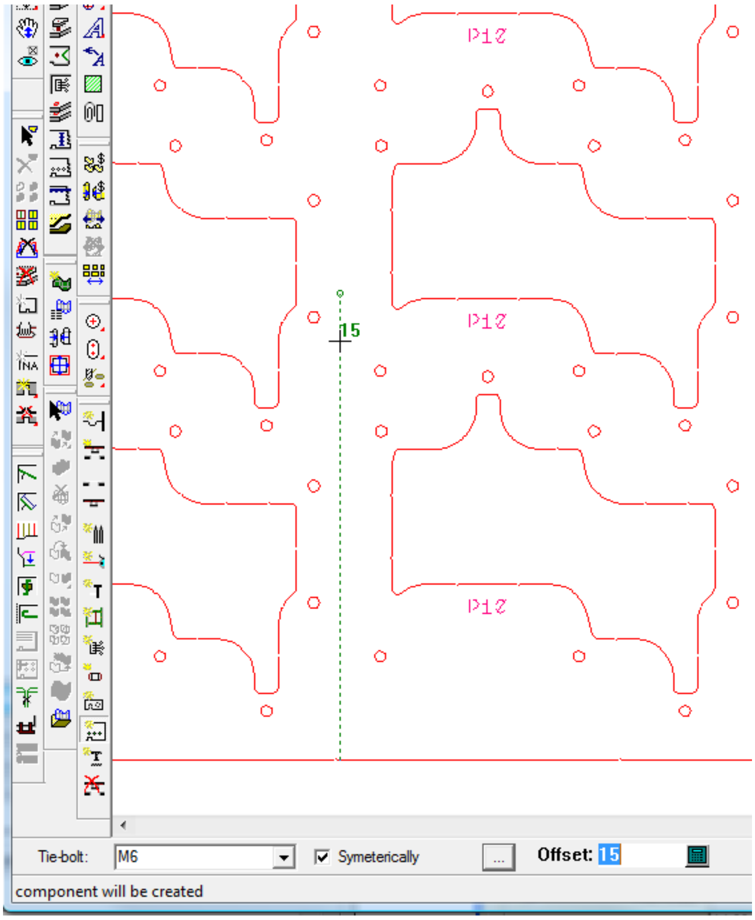


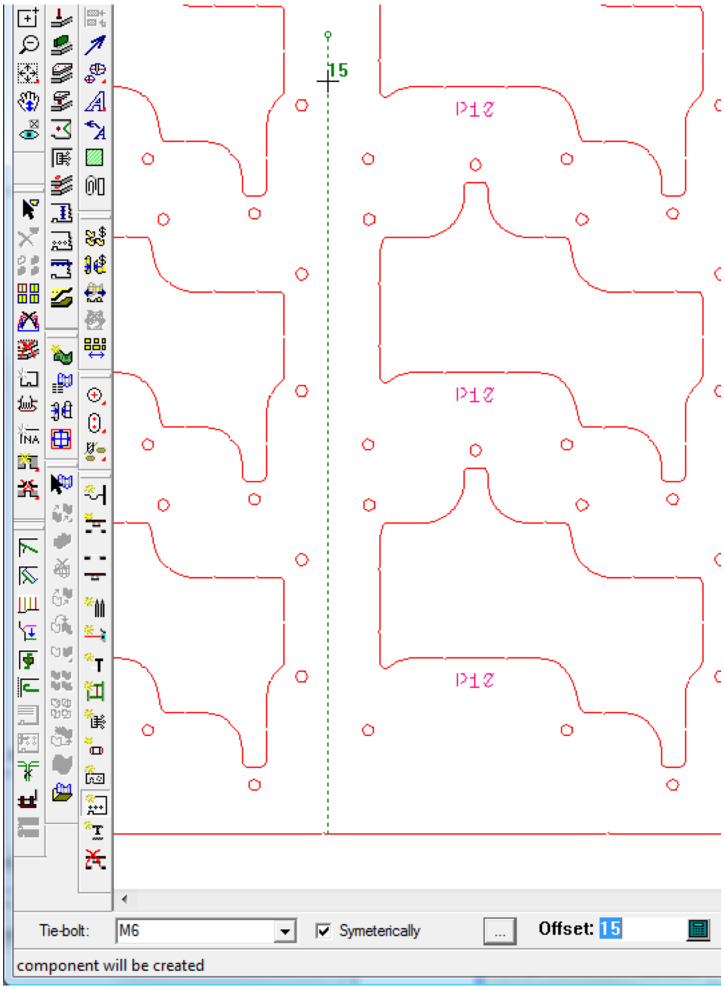
Set the distances as desired. The checkbox for a distance controls if you can add a mounting hole to that support bar, while the checkbox for the maximum distance enforces a limit on the offset distance from the centerline.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Tie-bolts** on the Blanking toolbar.
3. Choose the tie-bolt to use from the **Tie-bolt** drop-down list box.
4. Use the drag to set the **Offset** field. Depending on which mounting bars are enabled in the Blanking parameter sets dialog box, the drag snaps to show the bar for which you are adding a hole; you must use drag to choose the bar for the hole. The drag is green for valid positions and red for invalid positions. If **Symmetrical** is checked on the Status bar, then a corresponding tie-bolt hole is shown in the drag mirrored around the center line.

   Shown below is drag for the first bar and then for the second bar. Once you have set the offset, click to add the hole.





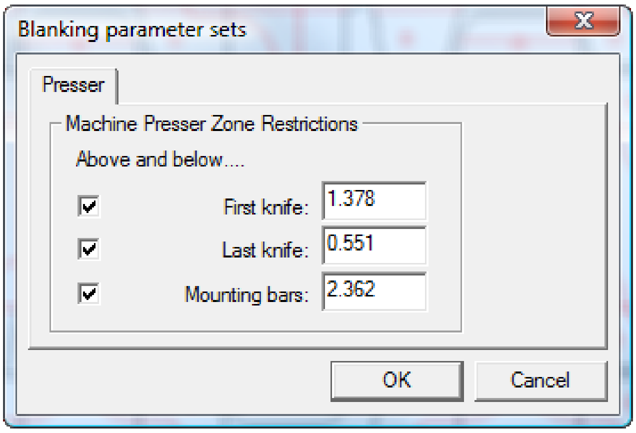
5. Continue adding tie-bolt holes as desired.

Add Pressers tool

 The **Add Pressers** tool adds pressers to the upper board. Pressers are essential for holding the waste while the blanks are separated from the sheet. When you click this tool, ArtiosCAD turns on the Blanker upper board layer and turns off the other blanking layers.

This tool has the following controls on the Status bar:

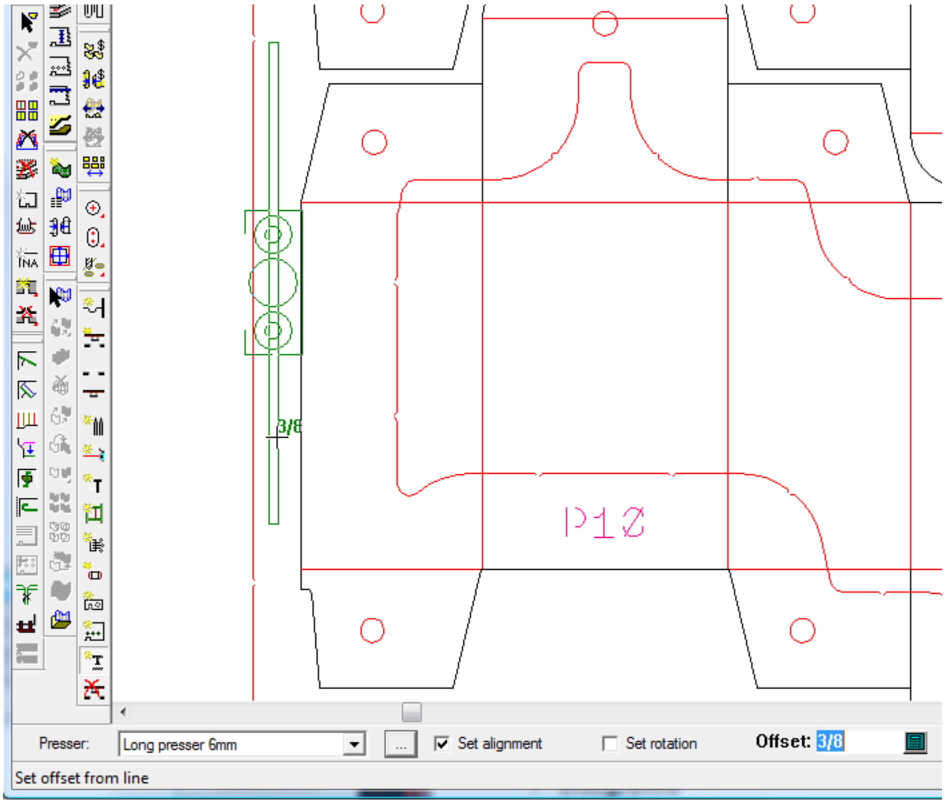
Choose the presser to use from the **Presser** drop-down list box. **Set alignment** and **Set rotation**, when checked, allow you to set an alignment point after choosing the placement and to set a rotation angle on the presser before clicking the final position. The Properties (**...**) button opens the Blanking parameter sets dialog box as shown below.



The checkboxes control enforcement of the restrictions for placement of the pressers relative to the first knife, last knife, and mounting bars. The values in the fields come from the blanking parameter set; change them if desired.

To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Add Pressers** on the Blanking toolbar.
3. Choose the type of presser to add from the **Pressers** drop-down list box.
4. Check **Set alignment** and **Set rotation** as desired.
5. Place the presser by clicking the design line from which to offset the presser, or click two points to position it between. Use drag to position it, or enter a value in the **Offset** field on the Status bar. The drag will be green for valid positions and red for invalid positions. Hold down CTRL to place the presser freehand.



If you checked **Set alignment** or **Set rotation**, ArtiosCAD prompts you to set the alignment and rotation before creating the presser. If Auto-Repeat is enabled, ArtiosCAD repeats the presser using layout congruency.

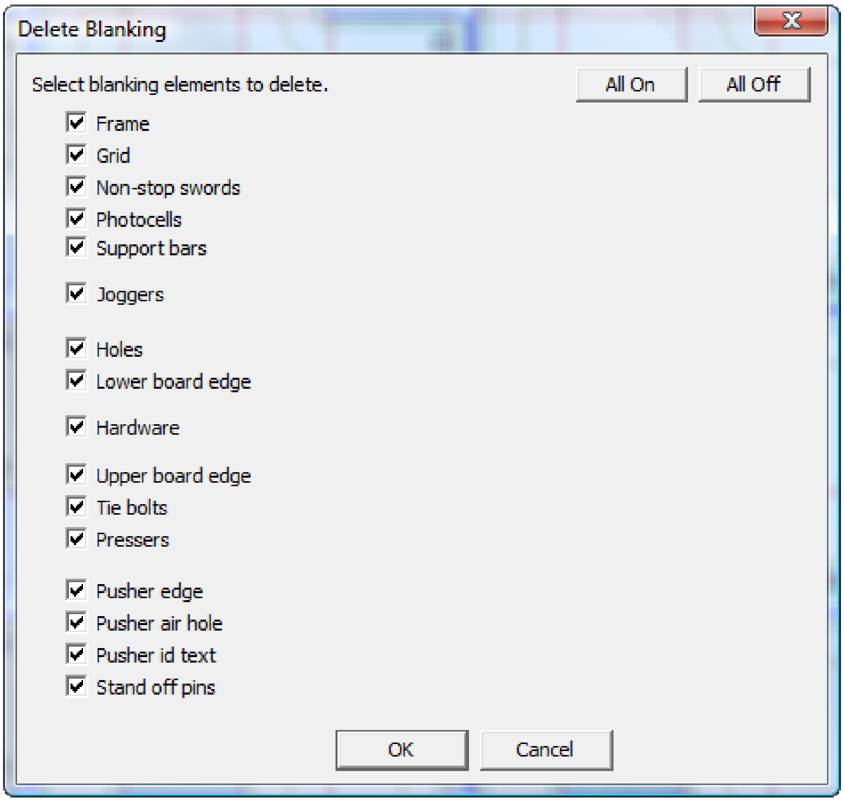
6. Continue adding pressers as desired.

Delete Blanking tool

 The **Delete Blanking** tool lets you quickly delete entire categories of blanking elements. When you click this tool, ArtiosCAD turns on all the blanking layers.

To use this tool, do the following:

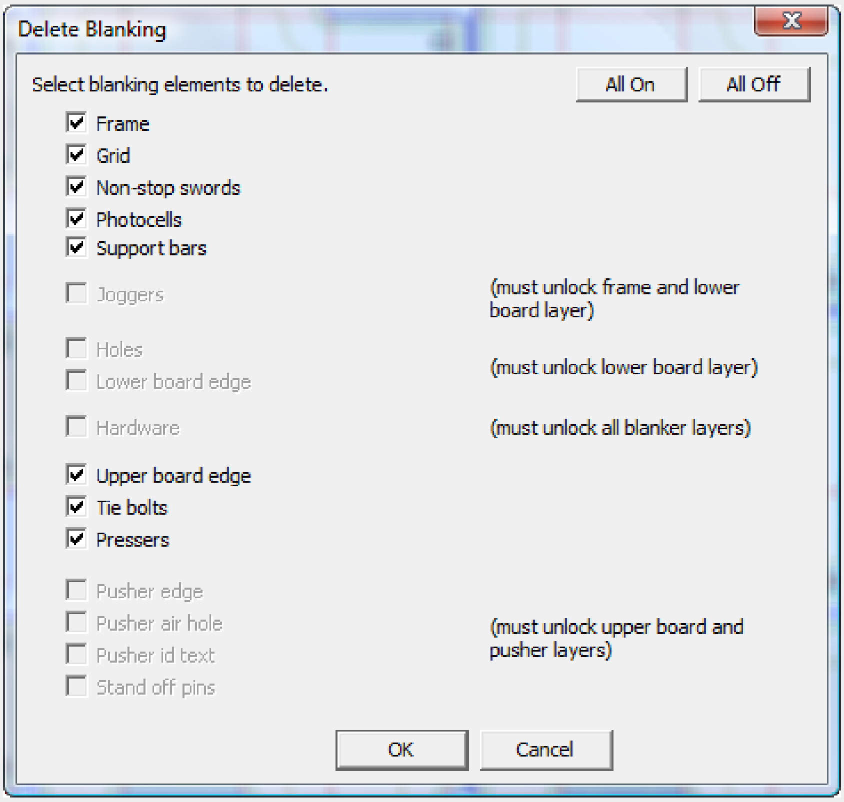
1. Create a layout in ArtiosCAD and generate the blanking tools.
2.  Click **Delete Blanking** on the Blanking toolbar. The Delete Blanking dialog box opens as shown below.



3. Select the checkboxes in front of the blanking elements to delete. All checkboxes are turned on by default. To toggle their states simultaneously, click **All On** or **All Off**.
4. Once you have clicked the checkboxes for the elements to delete, click **OK**.
5. ArtiosCAD deletes the selected blanking elements.

Note:

If you have locked any of the blanking layers, you cannot delete elements that have components in those layers as their checkboxes will be unavailable. Unlock those layers to enable deleting elements in them.

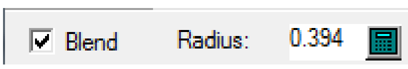


 To delete individual blanking elements, use the **Select Element** tool and the `DEL` key on the keyboard.

Smooth/Blend Contour tool

The **Smooth/Blend Contour** tool is on the context menu when you are adjusting the contour of a lower hole, pusher edge, or pusher air hole. It lets you blend contour corners or smooth the entire contour.

This tool has the following controls on the status bar:

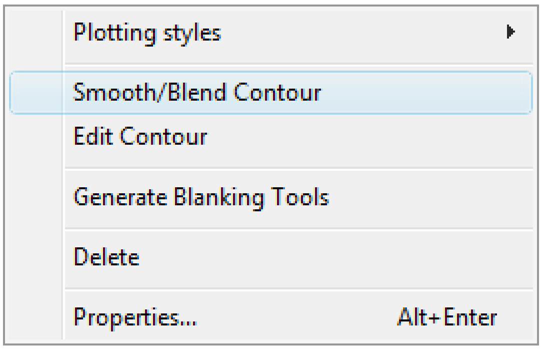


Blend blends the corner between two segments by the value specified in the **Radius** field.

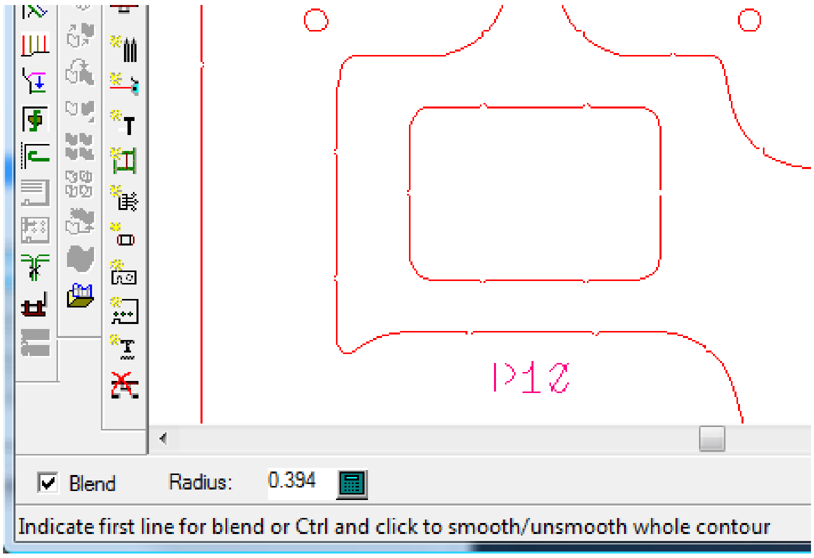
To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2. Use a tool to add or adjust a lower hole, pusher, or pusher air hole.

3. While one of those tools is active, right-click the contour and click **Smooth/Blend Contour** on the context menu.



4. On the Status bar, check **Blend** to blend two segments together by the amount in the **Radius** field.
5. Either click the first and second lines to blend, or hold down `CTRL` to blend the whole contour at once. If Auto-Repeat is enabled, ArtiosCAD uses contour congruency to repeat the changes to matching elements. Shown below is a rectangular air hole that had all its corners blended at once by using this tool while holding down `CTRL`.



Note:

This tool smooths only the inside corners of lower holes.

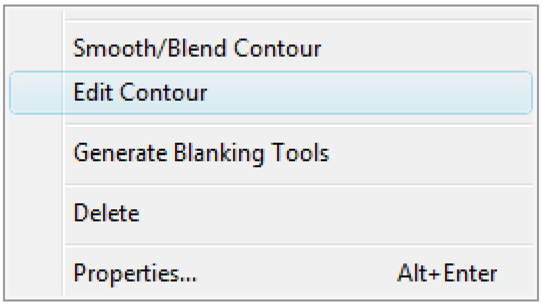
Edit Contour tool

The **Edit Contour** tool is on the context menu when you are adjusting the contour of a lower hole, pusher edge, or pusher air hole. It lets you change the shape of the contour.

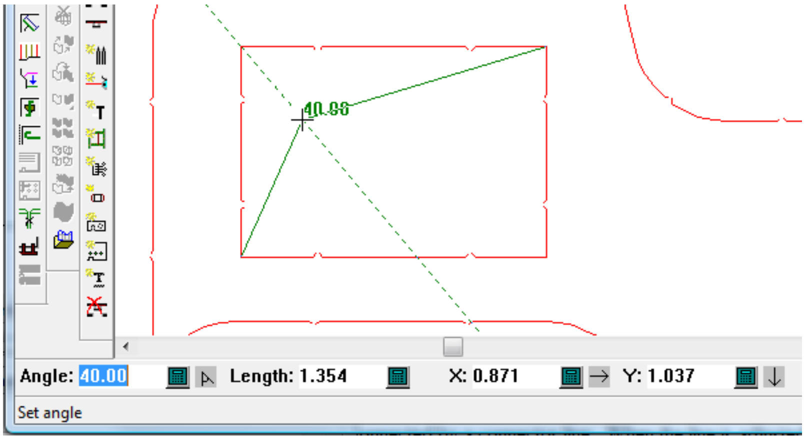
To use this tool, do the following:

1. Create a layout in ArtiosCAD and generate the blanking tools.
2. Use a tool to add or adjust a lower hole, pusher, or pusher air hole.

3. While one of those tools is active, right-click the contour and click **Edit Contour** on the context menu.



4. Click the point to move or the line to split. Use the Stretch Point technique if you click a point or the Split Line technique if you click a line. If Auto-Repeat is enabled, ArtiosCAD uses contour congruency to repeat the changes to other elements. Shown below is a point being dragged.

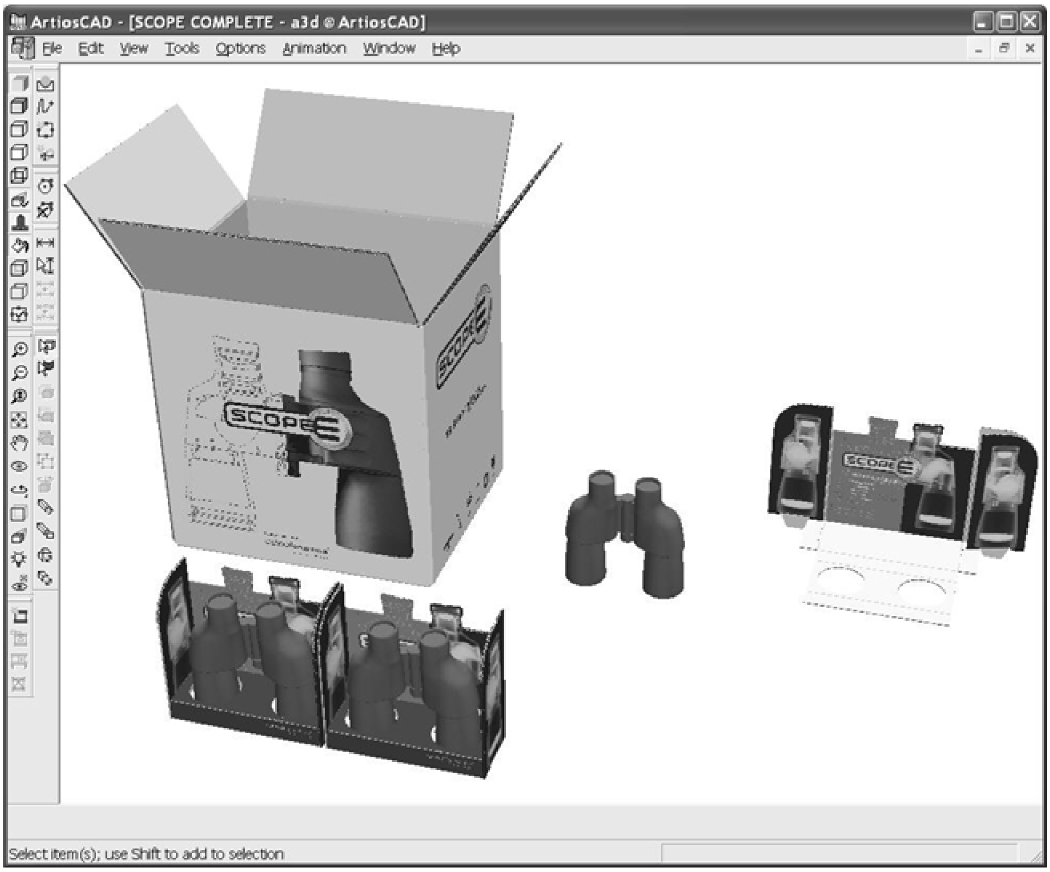


5. Continue editing the contour as desired.

7. 3D

Introduction to 3D

3D has three modules that provide different abilities for viewing how the design will look in the real world. **3D** allows you to fold flat single designs in three dimensions. With the **3D Animation** module, you can create a movie of how the design folds. With **3D Designer**, you can import a model of the object being packaged, make intersecting pieces of board through any cross section, and then run a standard style of box to contain the assembly. The examples and procedures in this chapter may reference options and features that are not part of your system.



Concepts and Ideas in 3D

3D is simple to use, but very powerful. Before you use it, you should understand the basics.

In 3D:

- You work in three dimensions using the X, Y, and Z axes. Those axes can optionally be displayed in the lower left corner of the design window.
- You import flat designs and then fold panels along straight creases (or similar lines such as perfs) so that the edges of the panels meet to form an image of the box as it looks in real life.
- You pick which panel of the box will remain fixed. All the other panels will fold around this one, called the base face.
- Lines in a design that are invisible in real life can be made invisible in 3D.
- You can see board thickness and corrugation if your system has the Graphics option.
- Multiple designs can be in one workspace, and multiple workspaces can be open at once. For example, in one workspace, boxes can be placed in trays that rest upon display bases.
- The viewing angle of the workspace can be changed at any time so that you can see your workspace from any side.
- Individual designs can be grouped together for easier manipulation.
- You can change the position of the light source to change the intensity of any graphics.
- You can animate a fold sequence and export it as a VRML file, viewable in any Web browser with a VRML plug-in. (This ability must be purchased.)
- You can make a solid of revolution by defining its vertical cross section and horizontal cross sections in the single design workspace, adding labels and shrink-wrap graphics if desired, to create a 3D model.
- You can import 3D models from other applications depending on the ArtiosCAD options purchased.
- You can copy a workspace to the Windows Clipboard with graphics and lines for use in Single Design or other applications, or export it as a PNG or JPG file for greater resolution and transparency support (PNG only).

Workflow in 3D

The core of using 3D for a folding container is shown below.

1. Make a single design in ArtiosCAD.
2. Start 3D by clicking the Convert to 3D button  on the View bar.
3. Select a base face for the flat design to fold around.
4. Fold the panels as desired, capturing animation frames as desired.
5. Add more designs to the workspace as needed.
6. Adjust the viewing angle, elevation, light sources, and perspective as desired.
7. Add a background image as desired and align the designs with the background image.
8. Turn on shadows as desired and adjust the light sources as needed.
9. Save the workspace.
10. Print or export the workspace.

Snap

The availability of pick-up and put-down points for objects in 3D are controlled by the Snap Options dialog box on the Options menu. In addition to the same options as in Single Design, **Snap to solids** enables snapping to any point in a solid (of which there are many because of the number of polygons used to make the solid); this checkbox is off by default. **Snap to cartons** enables snapping to midpoints and endpoints in corrugated or folding carton containers; this option is on by default.

Tools in 3D

There are two types of tools in 3D. The first requires one or more designs to be selected before that tool can be used. The second does not require a design to be selected. Any tool that manipulates the physical position of a design requires the design to be selected first.

Table: Tools and Selection Status

Tools requiring design to be selected	Tools not requiring design to be selected
Move Point to Point	Zoom Rectangle
Move Designs	Zoom Out
Move Designs in X, Y, Z	Center-Point Zoom
Duplicate Designs	Pan/Zoom
Duplicate Designs in X, Y, Z	Next View
Rotate Designs	Previous View
Group	Scale to Fit
Ungroup	View Mode tools
	Select Designs
	Fold Angle
	Fold All
	Fold 1 to Meet
	Fold 2 to Meet
	View Angle
	Perspective
	Light Source
	Dimension
	Select Dimensions
	All of the Animation tools
	Extend
	Clear Extend
	Convert to 2D

For example, if you want to move a design, it must be selected before the Move Design tool can be activated. The same applies to the **Move Point to Point**, **Duplicate Designs**, and **Rotate Design** tools.

Notes and warnings in 3D

3D workspaces created in this version of ArtiosCAD may not be opened in pre-6.5*x* versions of ArtiosCAD. In 6.5*x* there will be no board textures, no background image, and animations will not Scale to Fit.

The **3D View** toolbar must be turned on for flyout toolbars to work.

The amount of graphics in a workspace, the number of designs in a workspace, the smoothness selected for solids of revolution, and the number of workspaces open all affect the performance of your system and 3D. To keep your system running at the fastest possible speed, turn Graphics off in the View Mode dialog box (if that option is available) and keep the number of open workspaces to a minimum.

Multi-piece single designs without creases (such as words made of cut lines) are limited to having a maximum of 300 pieces when converted to 3D.

There is a limit of 29,900 graphic elements on a face in 3D. Each panel of a container is a face. Each fill or stroke is a separate graphic element; grouping them does not merge them for the purposes of circumventing this limit. Text that is converted to an outline has one or two fills per letter.

Performance will be slow with more than 100 holes per face.

Likewise, ArtiosCAD will not render more than 100 perf segments per panel. It will still tear or fold as required, however.

Curves in board face edges are converted to straight lines in order to draw their edges.

Faces may be divided by gaps up of up to 1.5 mm. Designs that have intentional gaps larger than this are not supported and may have unexpected faces.

When a design with no creases is opened in 3D, the Select Base Face dialog box is skipped as there is only one face.

If nothing happens when you turn on board thickness, there might not be a board defined in the workspace. The best way to turn on board thickness for 3D workspaces created in earlier versions of ArtiosCAD is to choose a new board for that workspace in the Properties dialog box.

When a crease is folded, it folds in the middle of the board as ArtiosCAD assumes the inside loss is half a caliper. It is often more accurate to fold in this manner because the inside loss in the board table is often rounded.

The VRML Export Options dialog box in Defaults (**Options** > **Defaults** > **Outputs-3D** > **Artios** > **VRML**) has a checkbox to control whether or not corrugations are exported to VRML from 3D.

Do not make a handle with a gap between the edge of the handle and the main body of the design on single designs you intend to convert to solids of revolution. Solid handles attached to the designs with no gap are constructible as long as the cross sections are defined correctly, or you could make a handle as a separate design with a large board thickness and add it to the 3D workspace.

When working with the High or Maximum settings in high graphics mode, only have one 3D workspace open at a time. There is no error message when the video adapter runs out of memory, so if parts of the graphics are missing, try High or Medium.

When folding a design, all edges of coplanar flaps may not be shown. Change the View Mode to **Solid with edges** to see the flap edges. Alternately, change the fold angle(s) by a degree or two.

A PS/2 laptop mouse may not have correct middle mouse button functionality. If you have such a mouse and the middle mouse button functionality does not work properly, try an external USB mouse.

Solid with Edges View mode in 3D now responds faster than it did in previous versions. It does this by drawing all the fills without graphics as a single bitmap and grouping all the lines together. In previous versions, the panels were independent of each other and the fills without graphics were drawn as separate pieces.

If you frequently use **Copy** and **Paste** between 3D and Single Design, you may notice this changed way of working more than users who do not frequently copy and paste between the two modules.

OpenGL provides sharper results when copying and pasting between modules because of its larger limit on bitmap size than Direct3D. Try using OpenGL if Direct3D does not meet your needs.

If you need your creases to be black when printing, create a plotting style with black crease lines and use it as needed.

A 3D workspace may have up to 30,000 individual designs in it.

Most special rules can fold and tear. Cruved rules can tear or fold per the table below, but cannot both tear and fold.

Special rule type	**Fold**	**Tear**
Crease, partial cut, reverse partial cut	Yes	Yes
Perf, cut/crease, partial cut/crease, all perf types	Yes	Yes
Multi-perf	Yes	Yes
Zipper rule	Yes	Yes
Wave or zig zag with non-zero nick size	Yes	Yes
Wave or zig zag with zero nick size	No	No
Scallop rule (has no nicks)	No	No

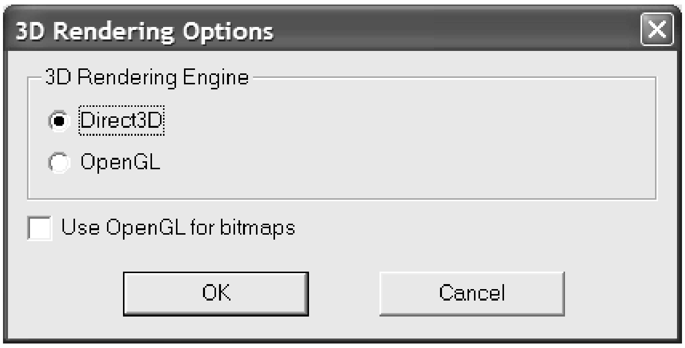
Special rule type	Fold	Tear
Staggered perf, S-rule, echelon perf	Yes	Yes

3D Rendering Options

3D 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.
- 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 wire frame view mode, OpenGL seems to be able to display more lines.
- When 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. Closing ArtiosCAD may also help Cortona to work more predictably.

To switch between rendering languages, click **Options** > **3D Rendering Options**, choose the desired language, and click **OK**.



Use OpenGL for bitmaps enables ArtiosCAD to output bitmaps using OpenGL regardless of the screen rendering method chosen. OpenGL supports the output of larger bitmaps than Direct3D.

Defaults are updated automatically with the selections made in this dialog box.

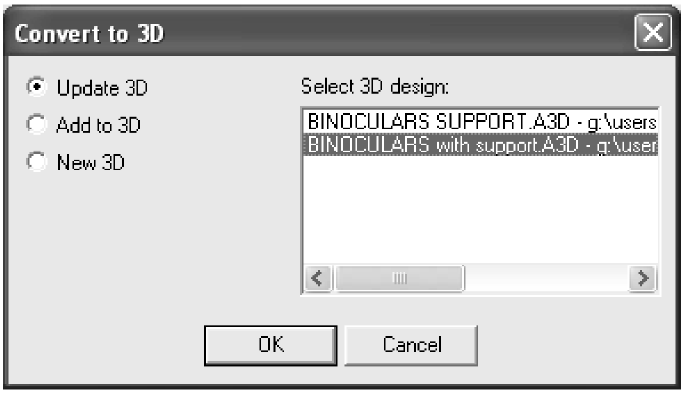
Understanding the Basics of 3D

Creating a new 3D workspace

To create a new 3D workspace from an existing flat design, do the following:

1. Open the design in ArtiosCAD.

2.  Click **Convert to 3D** on the View bar. If there is a 3D workspace open already, the Convert to 3D dialog box appears as shown below. If there is no 3D workspace open, skip to the next step.



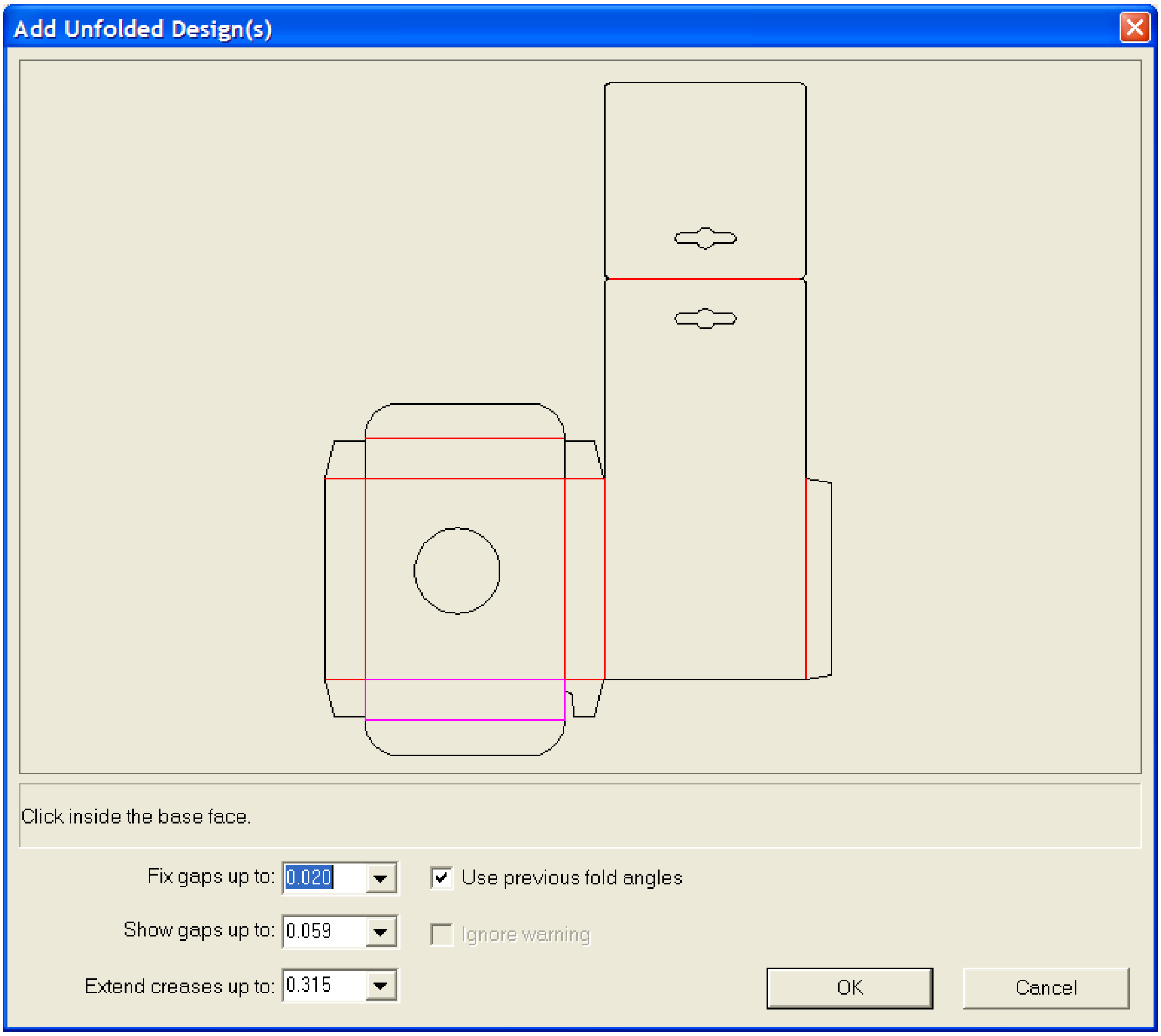
Update 3D is enabled if you are modifying a design contained in one of the open 3D workspaces. Choose the 3D workspace to update and click **OK**.

Add to 3D adds the design being converted to the selected 3D workspace. Click **OK**.

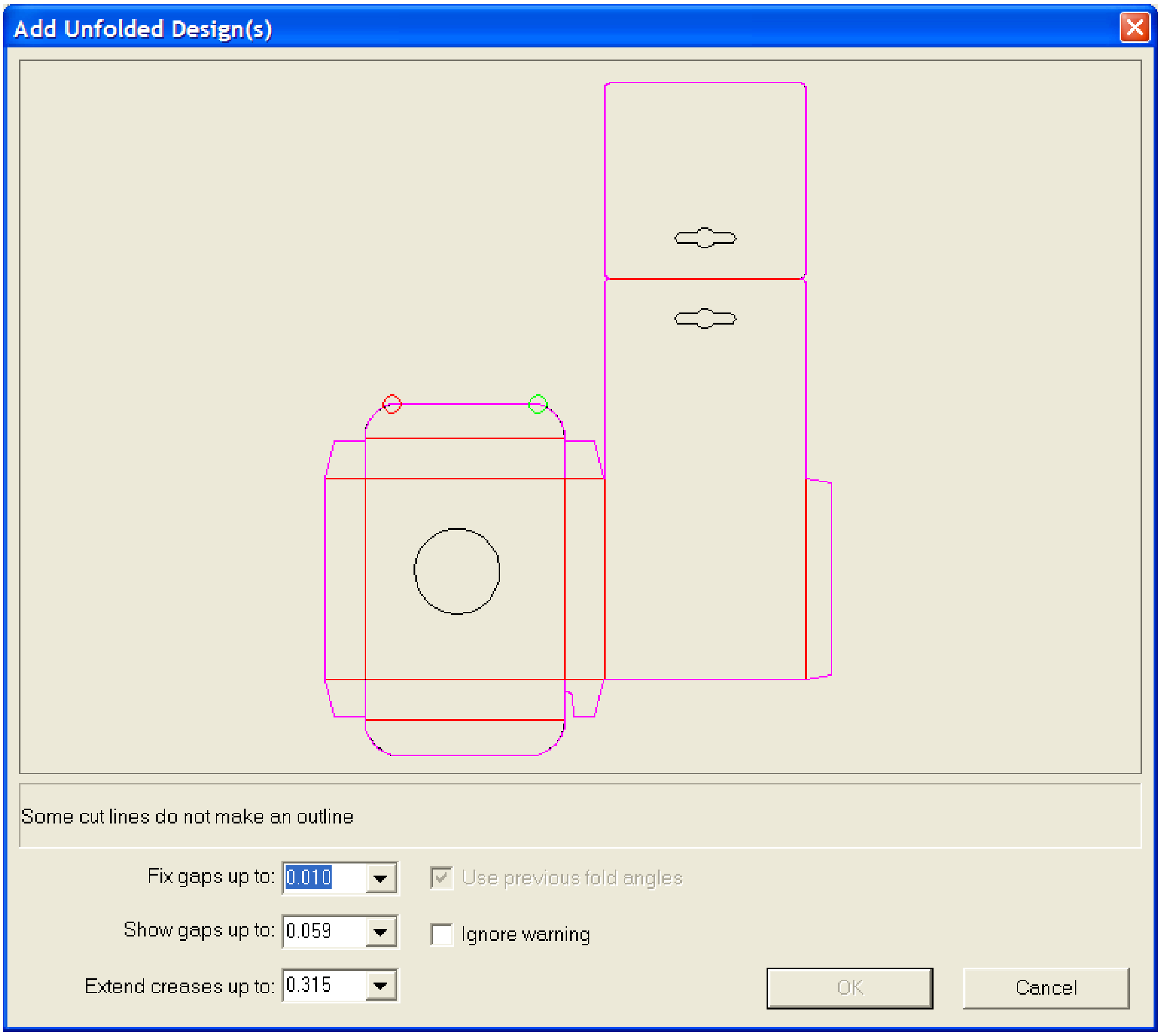
New 3D creates a new 3D workspace. Click **OK**.

Click **Cancel** to cancel the conversion to 3D.

3. The Add Unfolded Design(s) dialog box appears. ArtiosCAD checks the design for gaps in the perimeter as well as for creases that do not meet other cuts or creases to form panels. Any error messages or instructions are shown in the box beneath the picture of the design.

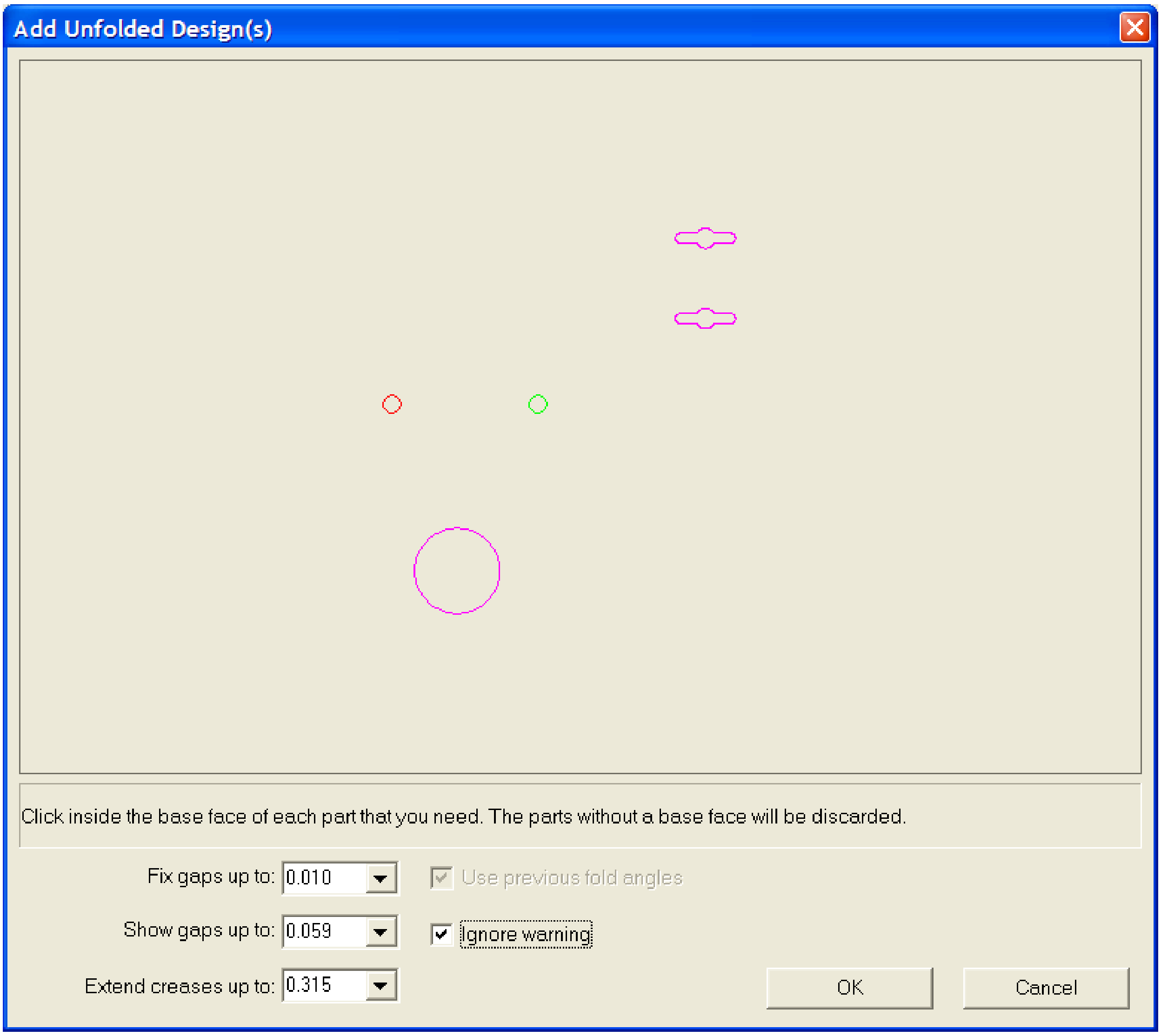


Any perimeter gaps smaller than the value in the **Fix gaps up to:** field are automatically fixed and are indicated by green circles. Gaps between the size to be shown and the size to be fixed are indicated by red circles. Perimeters with gaps are shown in purple.

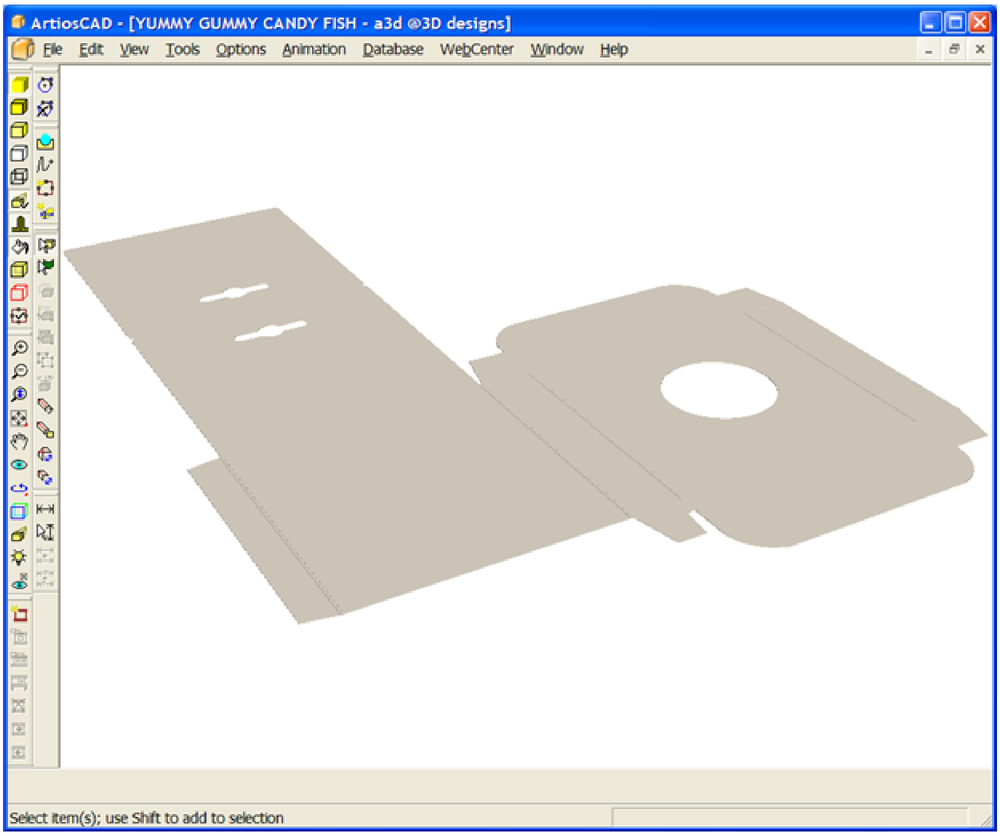


ArtiosCAD extends cut-back creases up to the amount selected in the **Extend creases up to:** drop-down list box so that they meet other cuts and creases. Creases will be extended at least 3mm. If the crease reductions were intentional, decrease the value in this field to prevent ArtiosCAD from extending the creases. Creases which do not completely form panels are shown as perfs.

Ignore warning is available when creases cross more than one panel or when cut lines do not form an outline. Checking this checkbox enables the **OK** button. Ignoring warnings may lead to unintended consequences as shown below.



4. Once any issues are resolved, click inside the base face when prompted. This is the panel that all other panels will fold around. If the single design contained fold angles, **Use previous fold angles** is available; check it if desired.
5. Click **OK**.
6. The design will be opened in 3D and you can start to fold it. If any graphics layers for the print item were turned on in Single Design before converting the design, they are all brought into 3D.



Note: If it will take longer than a minute to render the edges when creating a new 3D workspace, edges will be turned off if they are on in the current view. You may turn them back on manually by changing the View Mode.

Defaults for the gap tolerances and the crease extension allowance are set in **Options** > **Defaults** > **Shared Defaults** > **Startup defaults** > **3D Tools Defaults**.

To create a completely new 3D workspace, click **File** > **New 3D**. You can then use **File** > **Add a Design** to add designs or solids to the new workspace.

When converting a flat design to 3D, line types are processed as shown in the table below.

Table: Line types when converting to 3D

Line type	Resulting action
Cut	Form the outside edge of the design or holes in the interior.
Crease, Second height crease, Reverse crease	Straight creases between panels can be folded. Curved creases will cause an error message. Creases that do not connect cut lines are treated as perfs.
Perf, Reverse crease, Partial cut, Reverse partial cut	Straight lines between panels can be folded. Lines that do not connect cuts are shown as thin slots. The number of such slots able to be processed is limited; do not use with text outlines.

Line type	Resulting action
Print image, Annotation	Ignored. Use graphics such as strokes or fills instead.

Opening an existing 3D workspace

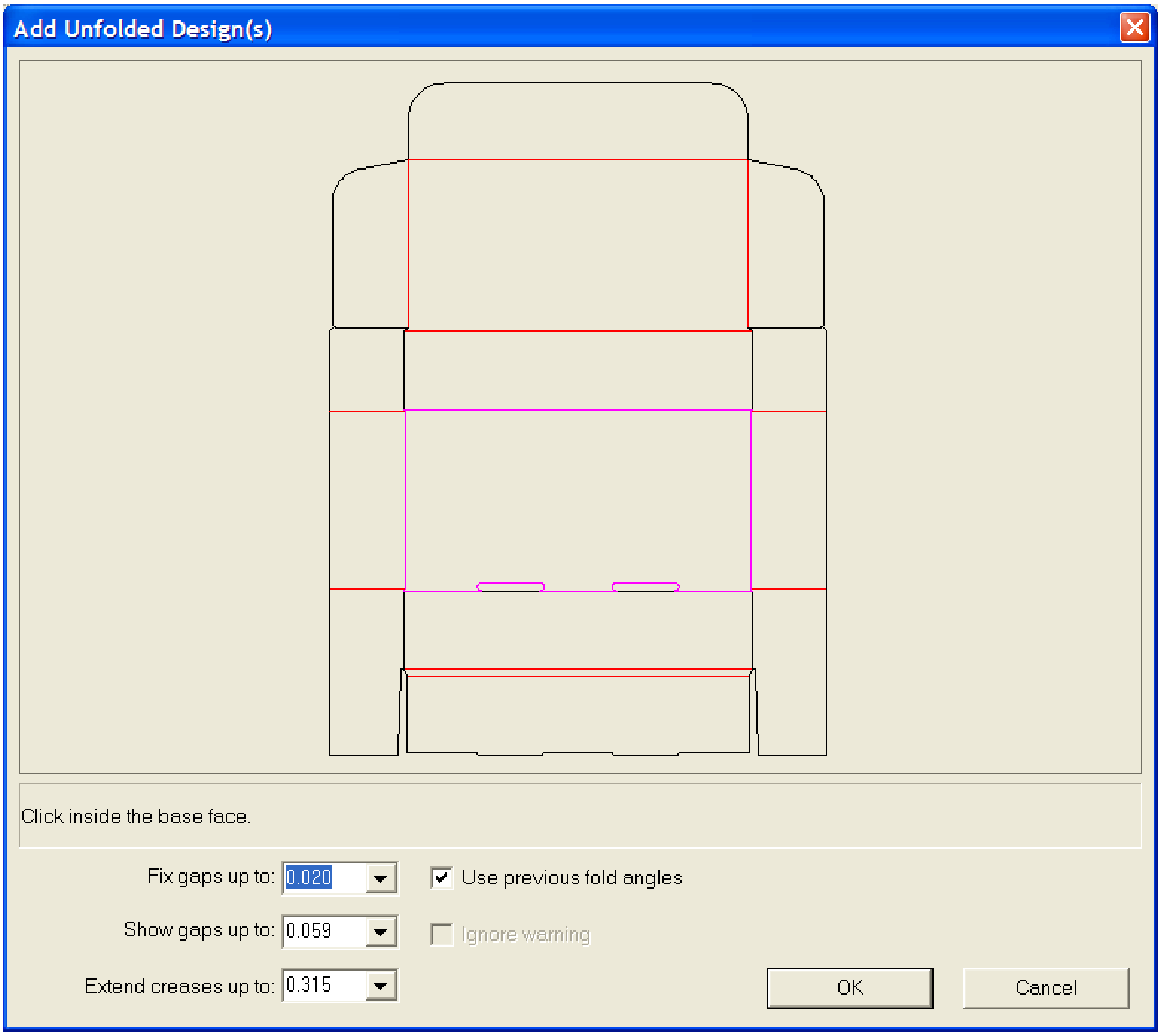
To open an existing 3D workspace, do the following:

1. Click **File** > **Open**.
2. Navigate to the desired resource or directory, select the file from those listed, and click OK. If your 3D design does not have the .A3D extension, enter the full name of the workspace in the filename box, or change the listed file type to All Files and then select the file.
3. The workspace will be opened.

Note: If it will take longer than a minute to render the edges when opening a design, edges will be turned off if they are on in the current view. You may turn them back on manually by changing the View Mode.

Remembered fold angles and base face

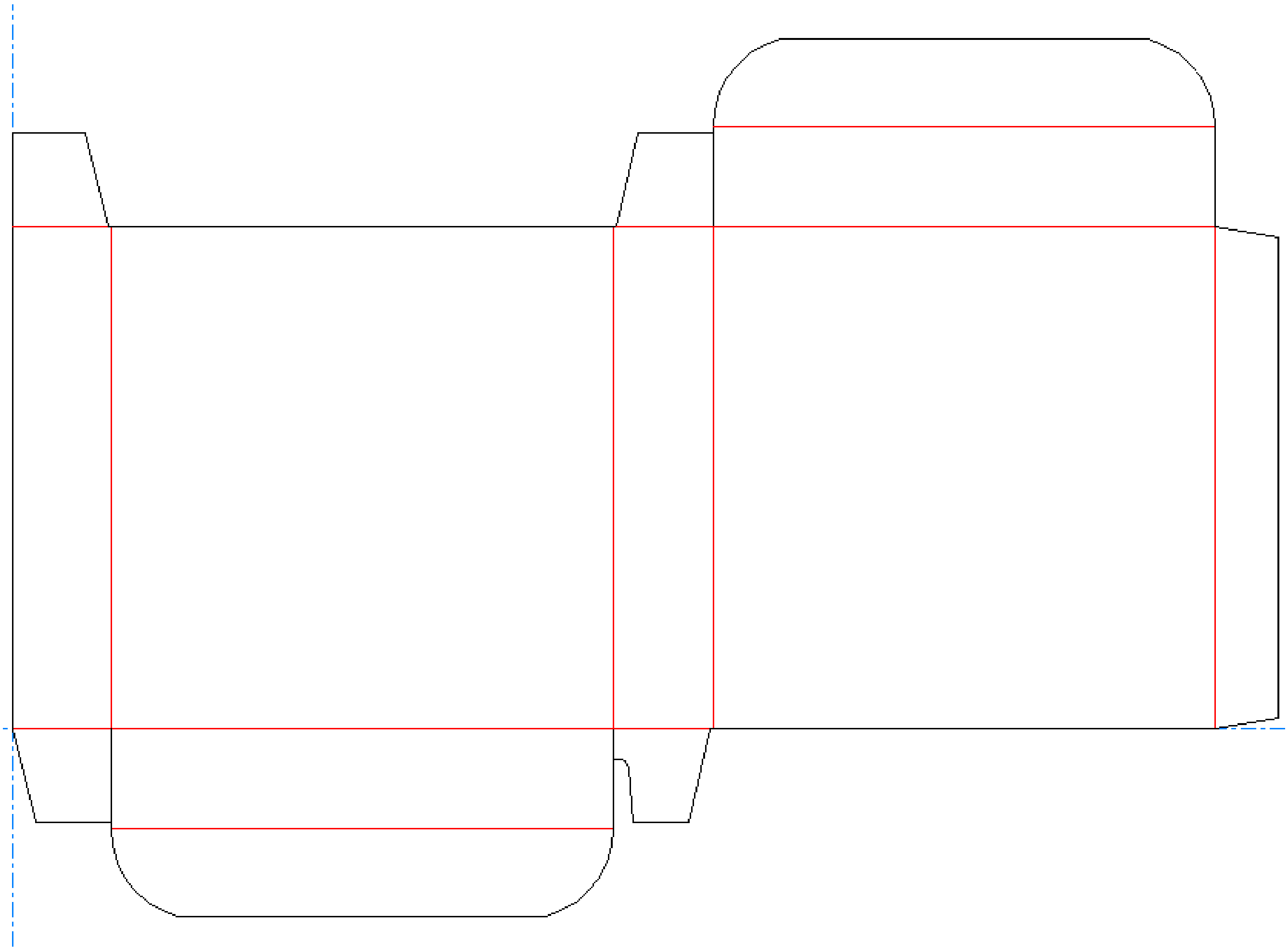
3D saves fold angle and base face information. If you convert a design to 3D, fold it, save it, and then make changes to the original flat design, when you convert the changed flat design to 3D, 3D gives you the option of reusing the folded angles and the base face from the previous version of the folded design. If you want to choose a new base face, click inside it, and clear the Use previous fold angles checkbox to select new fold angles. This feature only works on designs that have been saved as it uses the design name to link the 3D workspace to the flat ArtiosCAD design.



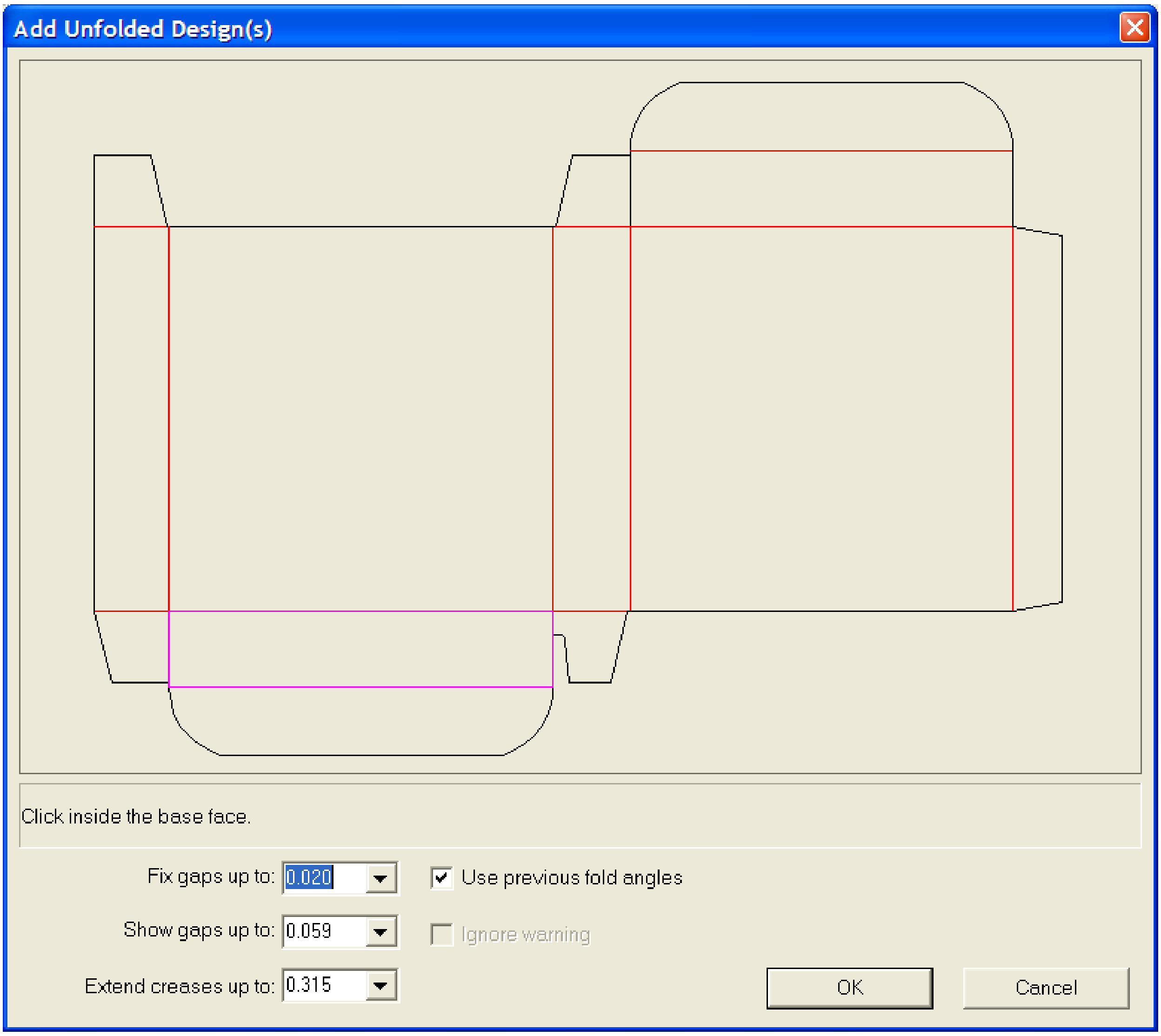
Pre-defined fold angles in designs created from standards

When you create a standard and fold it, the fold angles are stored in the standard workspace when it is saved. When a single design is then based on that standard, the fold angles and base face are remembered when the new single design is converted to 3D.

Shown below is a single design based on a standard.

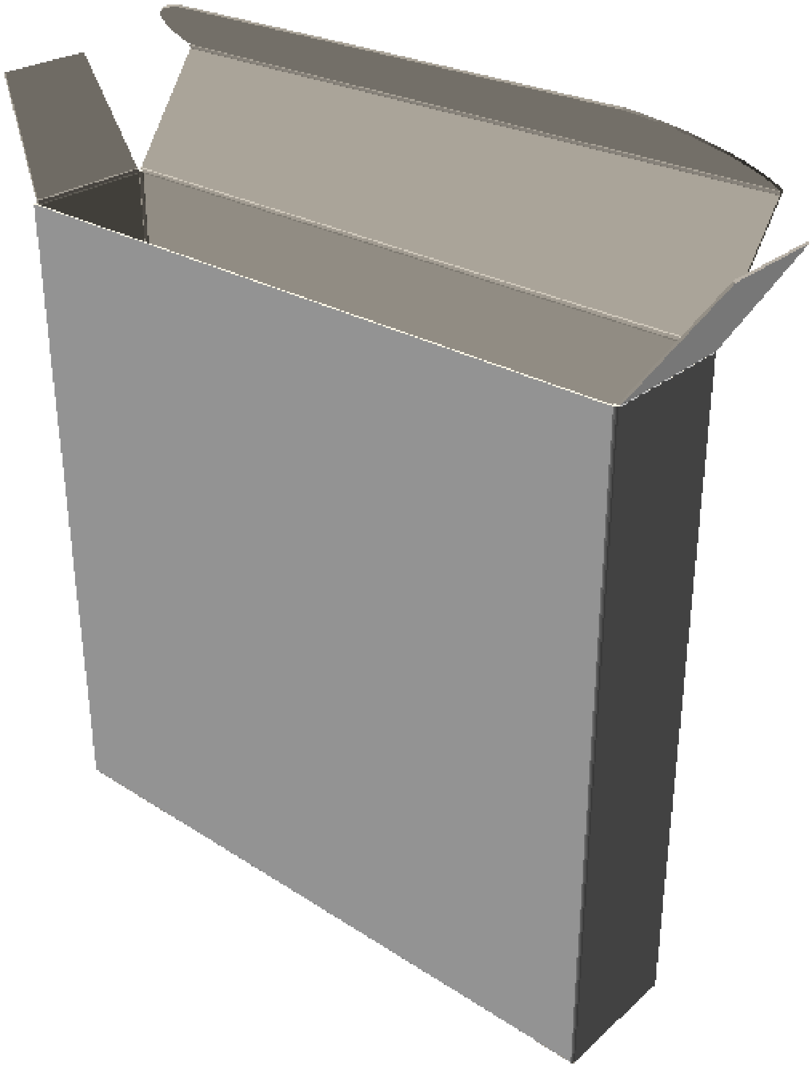


When the **Convert to 3D** button is clicked on the tool bar, the Add Unfolded Design(s) dialog box opens. The base face is preselected and the **Use Previous Fold Angles** checkbox is selected.



Note: When using multi-part designs, fold angles are remembered for only one of the parts.

When you click **OK** to accept the default base face and use the previous fold angles, the design will appear fully folded.



Folding all creases in a design at 90 degrees

To fold all creases at 90 degrees, do the following:

1.  Click the **Fold All** button.
2. Select a crease in the design you want to fold. If you want to fold more than one design, hold down the `SHIFT` key and select a crease in each design.
3. Select **90** from the list box for the **Angle** field.
4. All the creases in the selected designs will be folded at 90 degrees.

You may need to click **Scale to Fit** to reset the view of the design.

Folding individual angles

To fold angles individually, do the following:

1.  Click the **Fold Angle** button.
2. Select the crease or other foldable line to fold. To fold more than one crease at once, hold down the `SHIFT` key and select additional creases, or hold down `CTRL` while dragging to make a window selection.
3. Set the angle using the slider or the list box.
4. The angles will fold.

You may need to click **Scale to Fit** to reset the view of the design.

Adding another design to a 3D workspace

To add another design or import another file to your workspace, do this:

1. Click **File** > **Add a Design**.
2. Make sure the drive, directory, and file type settings are correct.
3. Double-click the name of the desired file.
4. Repeat steps 1 through 3 as desired.

Note: The ability to import certain 3D file types is controlled by the licenses purchased for the system. Your system will not be able to import 3D files of certain types without corresponding licenses. Contact your Esko salesperson for more information on purchasing licenses for importing 3D files. If it will take longer than a minute to render the edges when adding another design, edges will be turned off if they are on in the current view. You may turn them back on manually by changing the View Mode.

Add to Open 3D tool

The **Add to Open 3D** tool on the **Tools** menu lets you add one 3D workspace to another, avoiding the intermediate step of saving the first workspace and then using **Add a Design** in the second workspace.

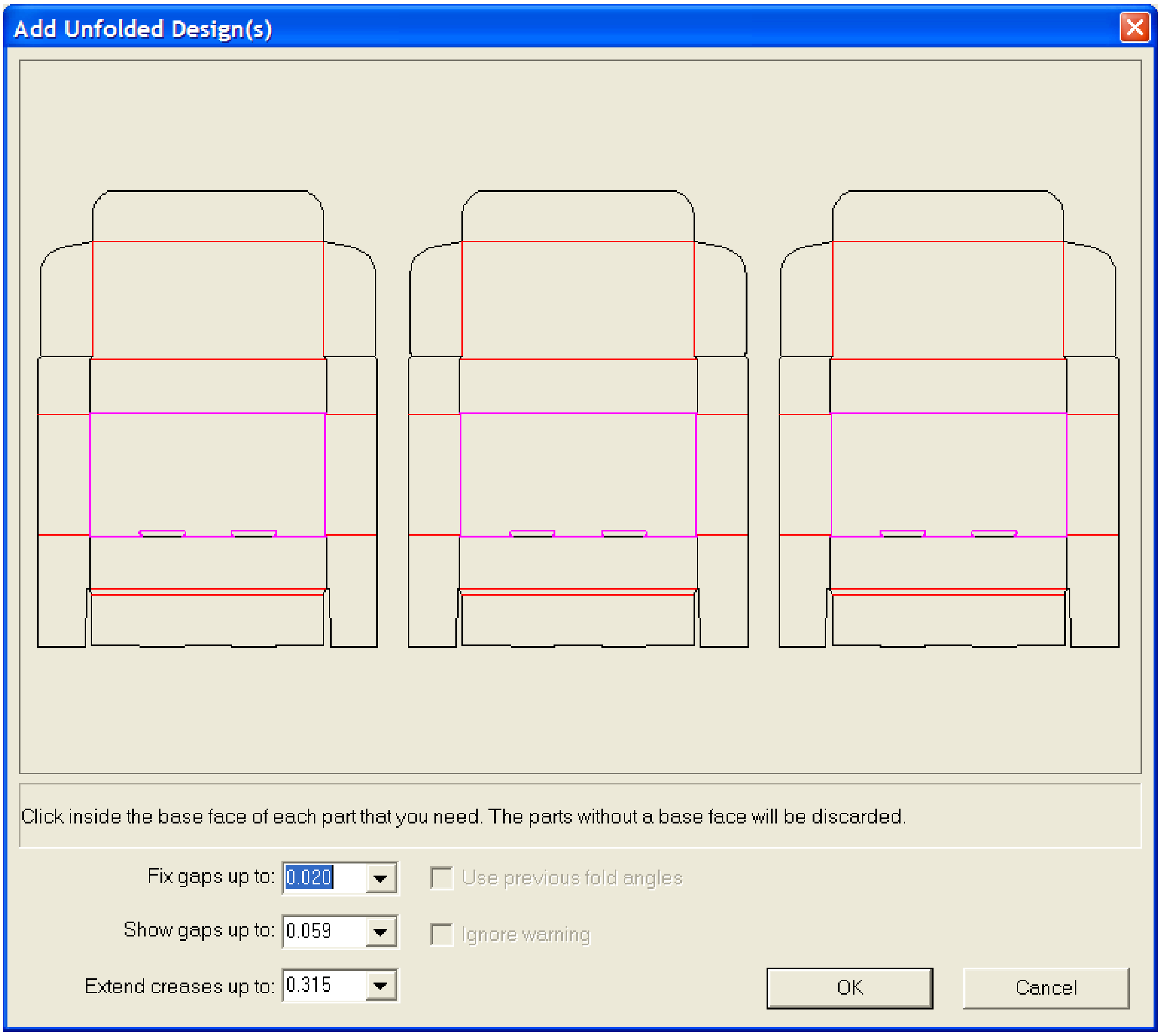
To use this tool, do the following:

1. Start ArtiosCAD and create, open, or convert at least two 3D workspaces.
2. On the **Window** menu, select the source workspace (the one to add to another workspace).
3. Click **Tools** > **Add to Open 3D**.
4. If you have more than two 3D workspaces open, select the destination workspace in the dialog box and click **OK**. If there are only two workspaces open, ArtiosCAD automatically selects the destination workspace.
5. ArtiosCAD adds the source workspace to the destination workspace in front of the contents of the destination workspace.

Converting a multi-part design to a 3D workspace

To convert a multi-part design to 3D, do the following:

1. Put all the parts together in one Single Design workspace, and then convert it to 3D. Up to 50 parts can be touching; if there are more than 50 touching parts, separate them into arrangements of 50.
2. Click inside the base face for each part to convert to 3D. Any part with creases that does not have a base face selected will be discarded. When there are many parts touching, it is not clear which parts are useful and which parts are waste. ArtiosCAD discards pieces thinner than 20 mm as waste; specifically deselect larger waste pieces.



3. Click **OK** to convert the multiple parts to 3D.

Note: Multi-piece single designs without creases (such as words made of cut lines) are limited to having a maximum of 300 pieces when converted to 3D.

Changing the view of a workspace in 3D

To change how a workspace is viewed, do the following:

1.  Click the **View Angle** tool.
2. Drag the sliders for **Angle**, **Elevation**, and **Roll Angle** to the desired positions, or type in the values in their boxes, or select predefined values from the list boxes. You may also click and drag inside the drawing window to change all three angles simultaneously. To change only the Roll Angle, hold down `CTRL` while dragging from to the left or right. The VIew angle will change according to the settings in the View Mode dialog box.
3. The view changes appropriately.

You may need to click **Scale to Fit** to reset the view of the design.

Changing the base face

The base face is the panel of the design around which all other panels fold. Imagine the container resting on a table. The base face is the panel on the table. To change the base face, do the following:

1. Select a design using the **Select Designs** tool.
2. Click **Change Base Face** on the **Tools** menu.
3. Click inside the new base face and click on **OK**.
4. The base face will be changed. You will not see a difference until you change fold angles.

Printing a folded workspace

To print a workspace, do the following:

1. Click **Print** on the **File** menu.
2. Change any values as appropriate (for example, to make two copies, set the number of copies to 2).
3. By default, one copy of the workspace will be printed in portrait mode at the largest scale that can fit on one page.
4. Click **OK**.

For more information about performing other Outputs from 3D, see the section about 3D Outputs.

Saving a workspace in 3D

Before saving the design, ensure High Graphics mode is not set to **Maximum** in the View Mode dialog box, as this may cause the preview to not be generated correctly.

To save a workspace, click **Save** on the **File** menu. The workspace will be saved with the name and database resource (if applicable) shown in the title bar of the workspace window. If the 3D workspace was created from a flat workspace which is still open, the fold angles from the 3D workspace are saved to it.

If this is the first time you have saved the workspace, you will be brought to the Save As dialog box, where you will be prompted to enter a name for the workspace.

If you have already saved this design, a current copy of the workspace will overwrite the saved file.

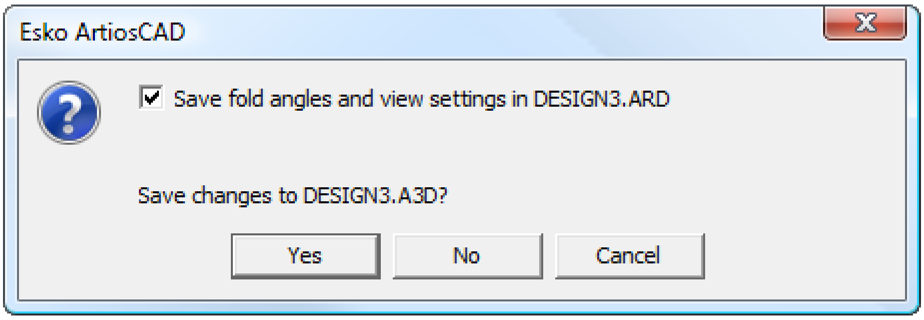
If you want to save the design under a different name and keep working on the original file, click **Save Copy As** and enter a name for the copy. If you want to save the workspace under a different name and not under the current name, click **Save As** on the File menu and enter the new name. If you did not save the workspace with the old name, it will be lost.

The full pathname of the source file for each design in the workspace is retained. When using tools that act upon the individual designs and require filenames, the full paths are shown in the **Filename**: field.

For more information about performing file-based Outputs from 3D, see the section on 3D Outputs.

Saving fold angles changes

When you close a 3D workspace and its associated single design is still open, you can choose whether or not to save the fold angles and view settings back to the single design.



To not save the fold angles and view settings back to the single design, clear the **Save fold angles and view settings in *design name*** checkbox.

Then click **Yes** or **No** to save the changes to the 3D workspace.

Note:

These are two separate actions. Clearing the checkbox only applies to the saving of the fold angles, not to the saving of the changes you made in the 3D workspace.

Copying a 3D workspace to the Windows Clipboard

To copy a low-resolution picture of the workspace onto the Windows clipboard, click **Edit** > **Copy as Bitmap**. Transparency is not supported. A better workflow is to export the workspace as a JPEG or PNG file using **File** > **Outputs-3D** and then import the JPEG or PNG file as desired into other applications.

To copy line information for use in Single Design, click **Copy** > **Copy as Enhanced Metafile**. This copies the workspace to the clipboard as both a low-resolution bitmap and as lines (also called *vectors*). The colors of the lines are set by the plotting style.

Using **Copy as Enhanced Metafile** is a way to make assembly drawings - copy views in 3D and then paste them into a Single Design workspace as desired, fixing any hidden line anomalies with the Select or Trim/Extend tools.

Elements included when performing a **Copy as Enhanced Metafile** include cartons and solids, board thickness, inside and outside graphics and board textures, the background image (if defined), and dimensions (which are converted to text and arrows for scalability). Bounding boxes are not included.

Notes and warnings

In complex workspaces, hidden line removal may not perform optimally. Any stray lines are easily removed in Single Design.

Depending on the capabilities of your system, this command may take several minutes to finish when there are more than 10 cartons in a workspace or for solids with more than 10,000 polygons. To show the number of polygons, click **Help** > **Diagnostics** > **List Embedded Designs**.

If there are too many polygons to render in less than a minute, a warning dialog box will appear asking you to choose to copy the design as a bitmap only, or continue and render as vectors. Choose the desired option and click **OK**, or click **Cancel** to cancel the copy to clipboard.

Copy as Enhanced Metafile includes edges even if they are not turned on in the current view.

Exiting 3D

To exit 3D, click **Exit** on the **File** menu. If you have changed your design since last saving it, you will be asked if you want to save it again. If the 3D workspace was created from a flat workspace which is still open, the fold angles from the 3D workspace are saved to it. Click **Yes** to save the 3D workspace, or click **No** to exit without saving. **Cancel** returns you to the workspace without exiting 3D.

This also exits ArtiosCAD. To exit the 3D module alone, close any open 3D workspaces.

Moving, duplicating, and rotating objects in 3D

The tools in this section affect the physical placement of objects in the workspace.

Working with different simultaneous view angles

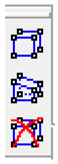
Sometimes different simultaneous view angles make moving or copying elements of the workspace easier. Using a CloseUp window allows you to work with different angles and avoid tool restarts caused by clicking between design windows. To create different viewing angles using a CloseUp window, do the following:

1. Set the view angle as desired in preparation for the first view angle.
2. Click the middle mouse button at the location of the center of the first view angle, or press `CTRL-Space` and click the left mouse button in that location. This creates the first CloseUp window.
3. Click the magnifying glass icon in the title bar of the CloseUp window and deselect **Dynamic**. Move the CloseUp window as desired.

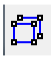
4. Change the view in the main ArtiosCAD window as desired to create the second view angle.
5. Use a Select tool to select the object(s) to move or copy.
6. Use a move or copy tool to pick-up in the CloseUp window and put-down in the ArtiosCAD window.

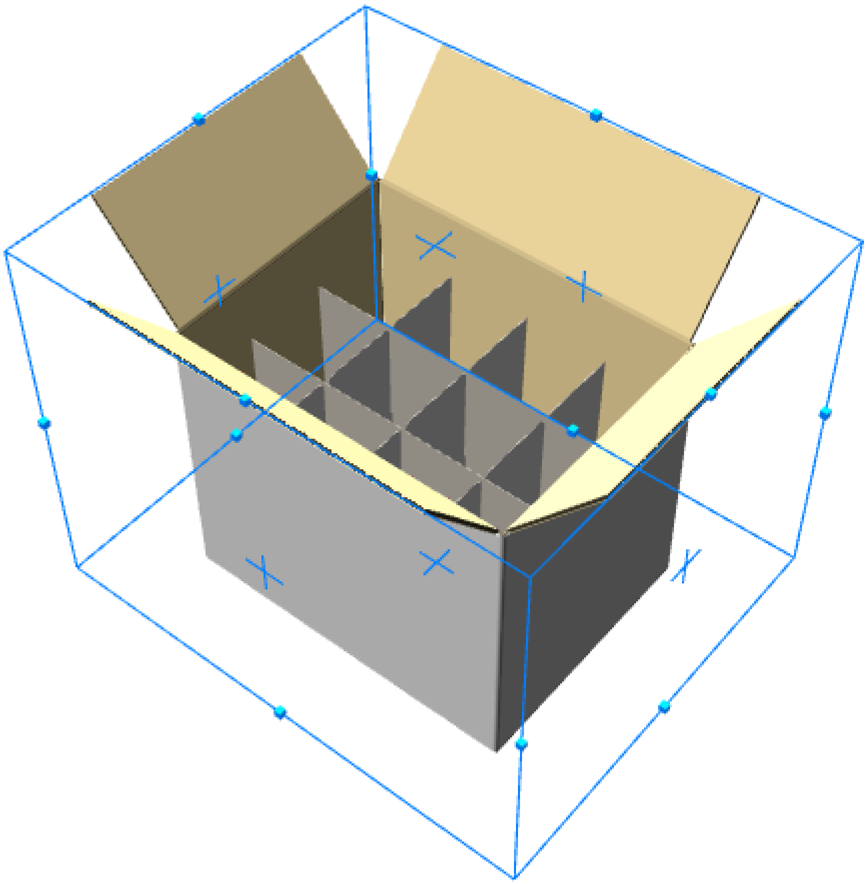
3D Extend tools

The **Extend** and **Clear Extend** tools on the 3D Extend Tools toolbar create and remove bounding boxes around objects in 3D. Turn on the toolbar by clicking **View** > **Toolbars** , clicking the button next to **3D Extend**, and clicking **OK**.

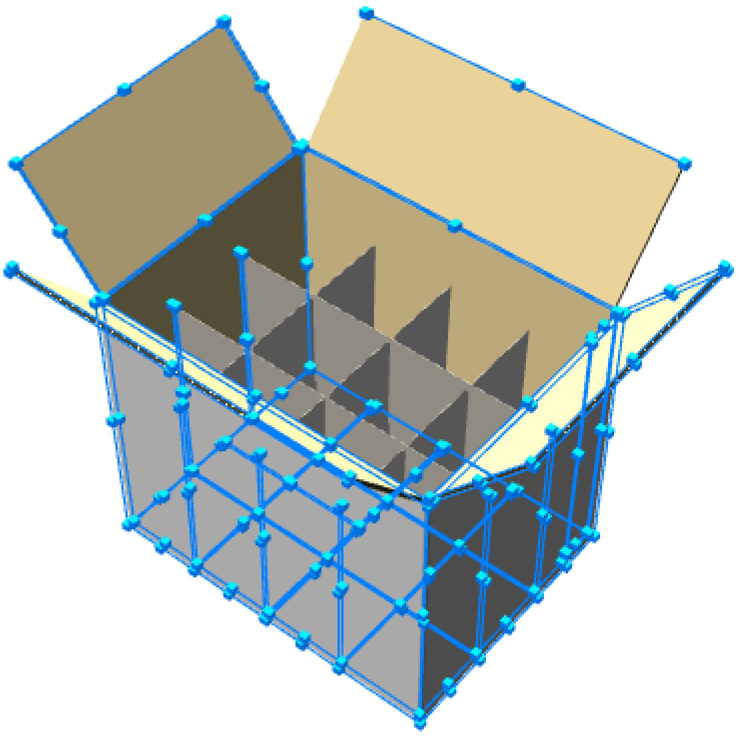


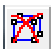
The bounding boxes create 20 points than can be used as pick-up and put-down points. They make it easier to move, copy, and place objects.

 The **Extend** tool creates a bounding box. To use it, click the tool, and the click the object to extend. The bounding box will appear, with selectable points at the corners and midpoints of each line. More than one object can have a bounding box. Bounding boxes remain visible until they are removed with the **Clear Extend** tool. Shown below is a bounding box for the outer box.



The **Extend Designs** tool shows extend points instead of a bounding box.



 The **Clear Extend** tool removes all bounding boxes and extend points. To use it, click it, and all bounding boxes and extend points disappear.

Select Designs tool

 The **Select Designs** tool is used to select one or more objects. Selected objects can then be moved, rotated, copied, or deleted. Delete objects by selecting them and pressing `Delete` on the keyboard.

Click the **Select Designs** tool, then click on any line in a design. To select more than one design, hold down the `SHIFT` key as you select the designs. Selected objects will turn magenta to indicate they are selected.

Move Point to Point tool

 The **Move Point to Point** tool moves two designs together so that the pick-up point of the first design is aligned exactly with the put-down point of the second design.

1. Select a design using the **Select Designs** tool.
2. Click the **Move Point to Point** tool.
3. Select the pick-up point in the design you just selected.

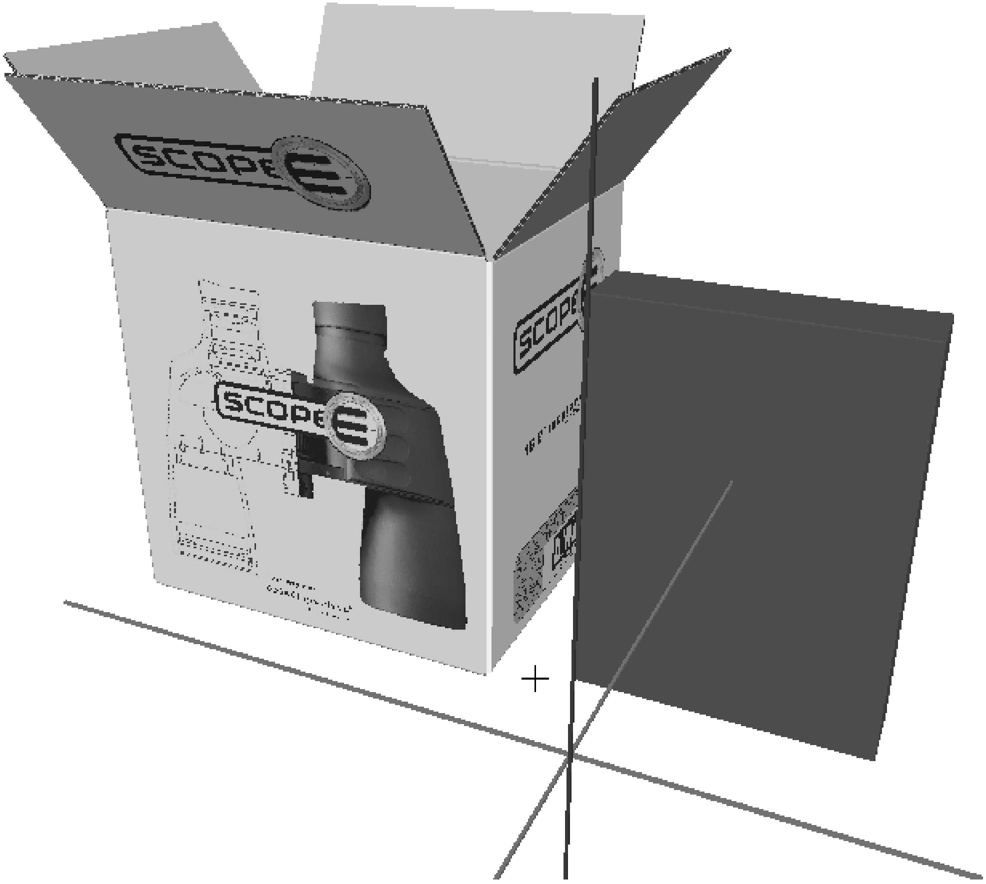
4. Select a put-down point in another design. The first design will move so that the pick-up and put-down points are aligned.

Move Designs tool

The **Move Designs** tool is the first tool on the Move Designs flyout toolbar. This tool moves one or more designs in one direction.

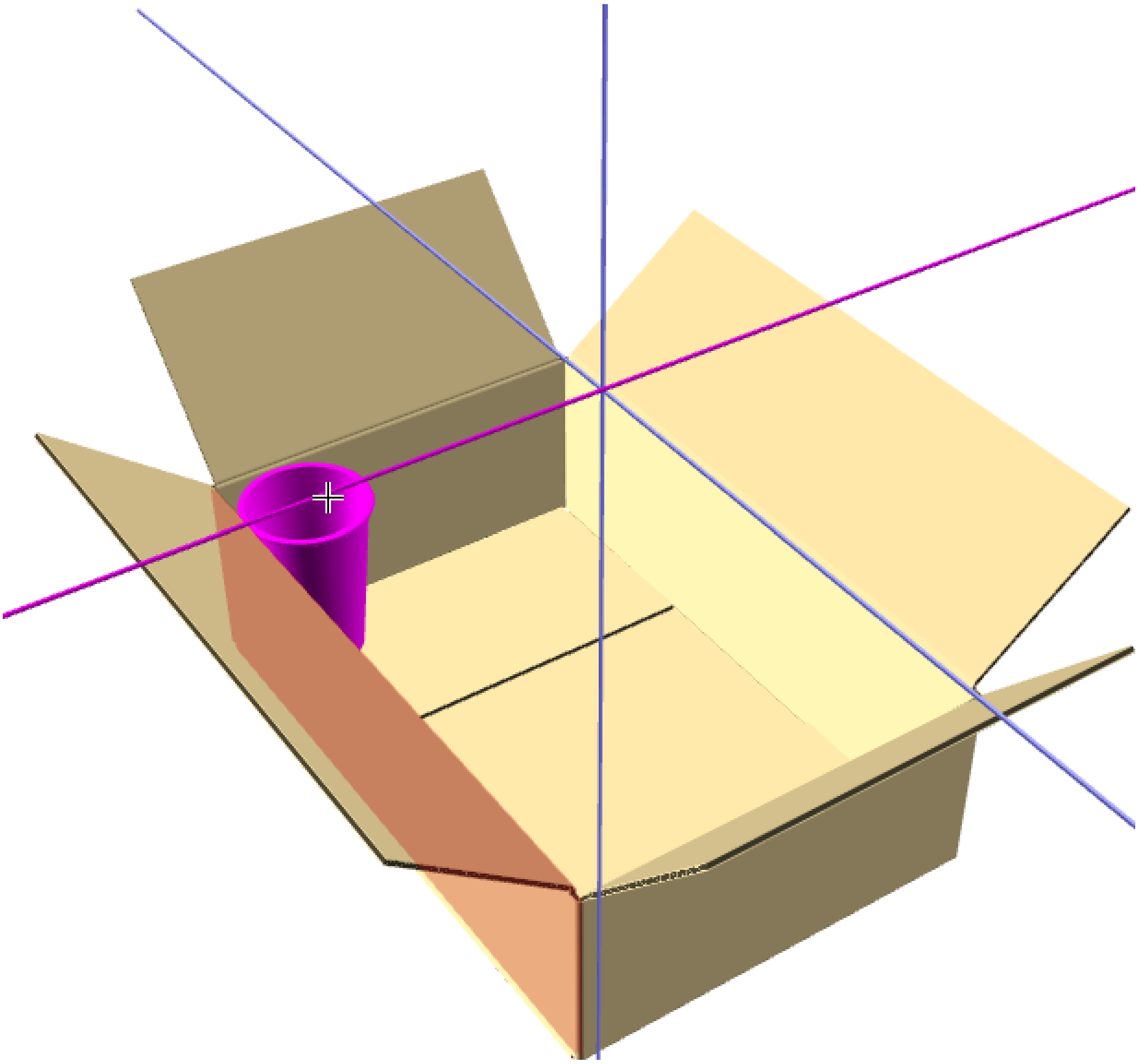
1. Select one or more designs using the **Select Designs** tool. It will turn magenta to show it is selected.
2.  Click the **Move Designs** tool.
3. Select a pick-up point. There are pick-up points on the inside and outside of lines. The three directional axes will appear centered on the pick-up point.

4. Click an axis or line to set the direction in which to move the selected designs.
5. Move the mouse to move the selected designs in the direction indicated in step 4, or enter a distance on the Status bar.

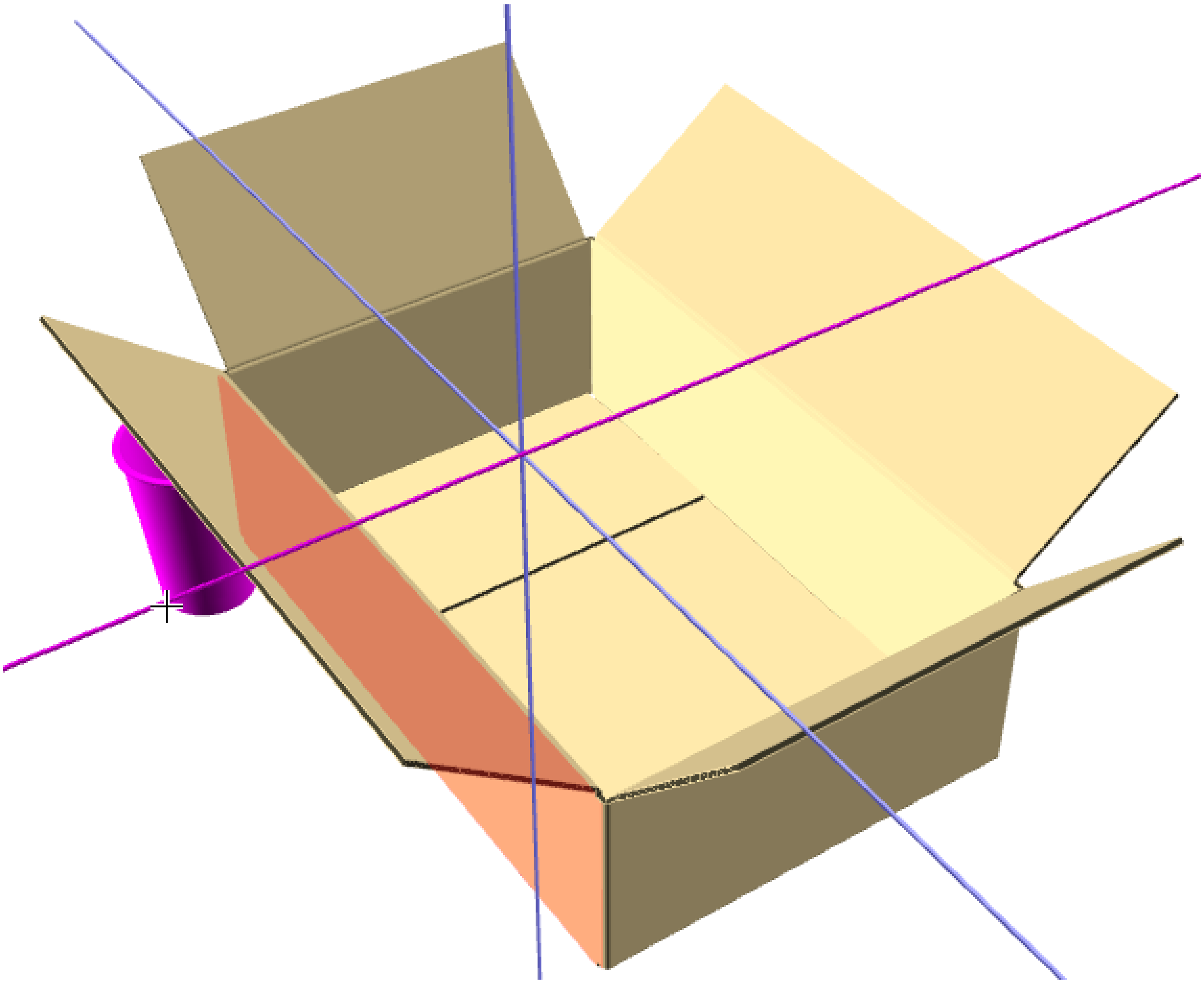


6. Click to set the put-down point of the selected design, or click an existing point to align the put-down point with it. Hold down CTRL while clicking to add another offset before setting the final put-down point. The Move tool remains active with the current selection so that another move may be performed without having to reselect the tool and pick-up point. Shown below is the moved design.

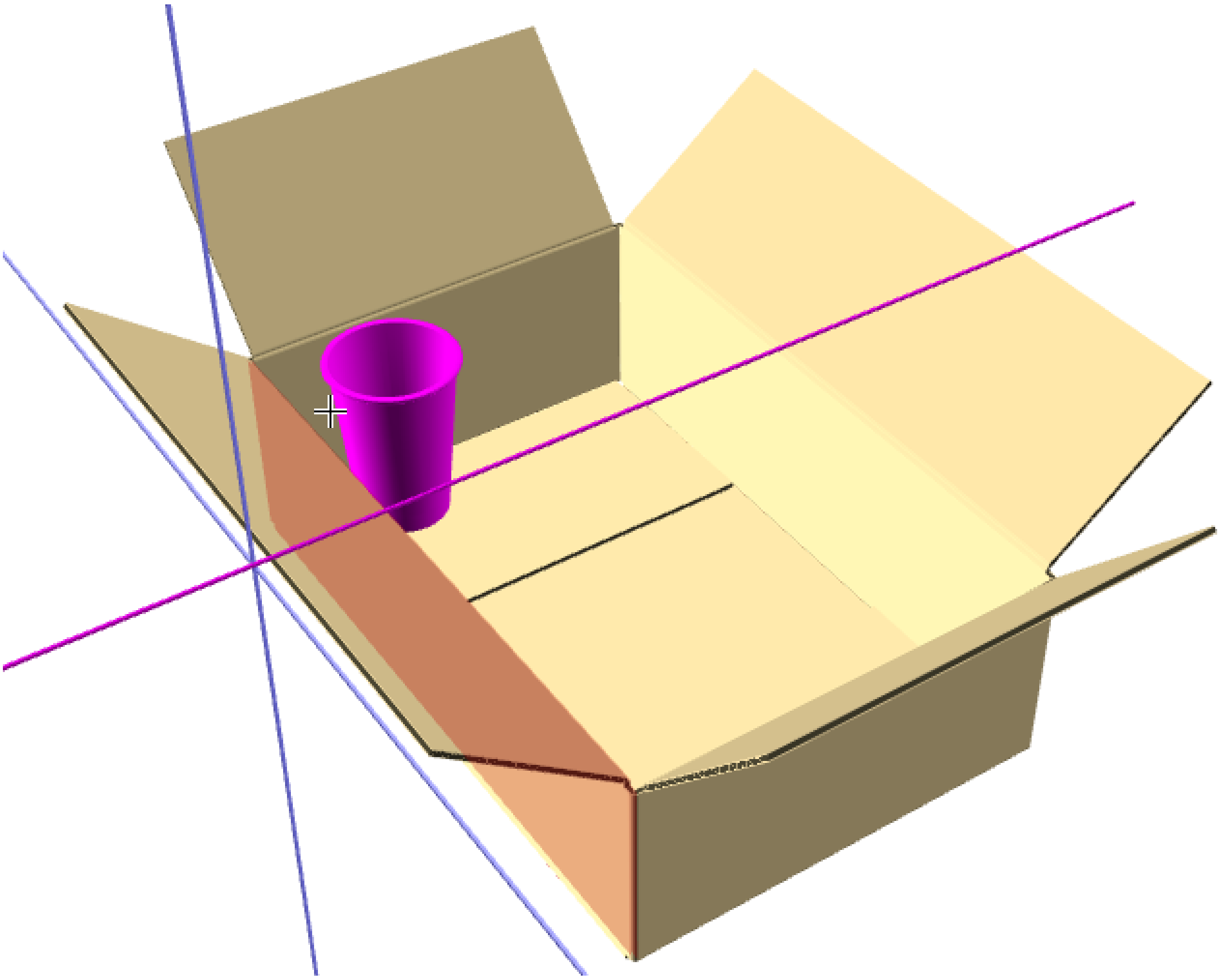
If you check **Check collisions** on the Status bar, the object you are moving stops when it hits another object, and the object it hits turns red.



If you continue dragging, the object you are moving moves through the interfering object once it is far enough away to not collide with it.



Clicking **Gutter** and entering a value in the field sets a gutter between the edge of the moving object and the interfering object. Shown below is the same example with a gutter of one inch.



Move Designs X, Y, Z tool

The **Move Designs X, Y, Z** tool is the second tool on the Move Designs flyout toolbar.

It moves the current selection in any or all of the three dimensions at once.

To use this tool, do the following:

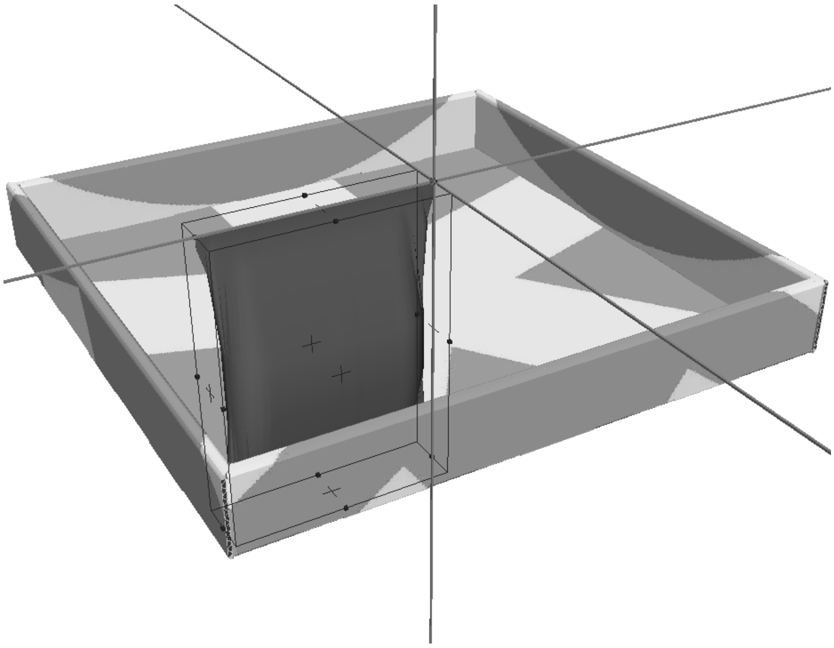
1. Click the **Select Designs** tool and select the design(s) to move or copy.
2. Click the **Move Design in X, Y, Z** tool.
3. Click the pick-up point. There are pick-up points on the inside and outside of lines.
4. Use the drag to set the X offset, or enter a value in the **X**: field on the Status bar.
5. Once you have set the drag for the X offset or entered a value, the Y offset becomes active. Set it using drag or by entering a value in the **Y**: field on the Status bar.

6. Once you have set the drag for the Y offset or entered a value, the Z offset becomes active. Set it using drag or by entering a value in the **Z**: field on the Status bar. If you set the Z offset using drag, the design will move when you click the mouse button. If you entered the Z offset on the Status bar, click **OK** and the design will move.

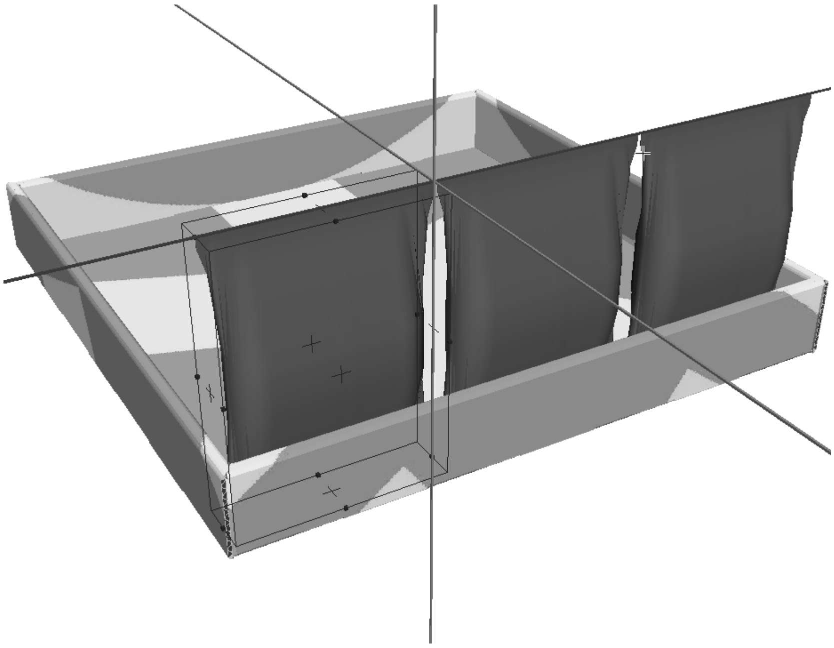
Duplicate Designs tool

 The **Duplicate Designs** tool makes one or more copies of the currently selected designs. It is the first tool on the Duplicate Designs tools flyout toolbar.

1. Select the design or designs to copy.
2. Click the **Duplicate Designs** tool.
3. On the Status bar, enter the number of copies to make in the **Number of copies:** field.
4. Select a pick-up point. There are pick-up points on the inside and outside of lines, and on the bounding box if it is turned on using the **Extend** tool.
5. Select an axis or line that indicates the direction to move the duplicates.



6. Indicate the distance to move the duplicates either using drag or by entering a number in the **Distance**: field on the Status bar. The copies will move with the mouse cursor.



7. Click the mouse button to set the put-down point of the outlined designs. Hold down `CTRL` while clicking to add another offset before setting the final put-down point. The designs are duplicated. Repeat as desired.

Shown below is the finished product, with the row of three bags duplicated 10 times.

Note: When an object in 3D is duplicated, all duplicates are instances of the original object for more efficient memory usage. The duplicates may have different fold angles than the original, but the material properties of the duplicates are the same as those for the original. If you select fewer than all the copies and try to change them, ArtiosCAD prompts you to change all instances of the first copy or to make a separate copy to which the new properties would then be applied. If you choose to make copies, the amount of memory used and disc space required will increase accordingly.

Duplicate Designs X, Y, Z tool

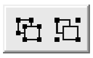
 The second tool on the Duplicate Designs tools flyout toolbar is the **Duplicate Designs X, Y, Z** tool. It works the same way as the Move Designs X, Y, Z tool, except it acts on a duplicate of the current selection.

To use this tool, do the following:

1.  Click the **Select Designs** tool and select the design(s) to copy.
2.  Click the **Duplicate Design in X, Y, Z** tool.
3. Click the pick-up point. There are pick-up points on the inside and outside of lines, and on the bounding box if it is turned on using the **Extend** tool.
4. Use the drag to set the X offset, or enter a value in the **X**: field on the Status bar.
5. Once you have set the drag for the X offset or entered a value, the Y offset becomes active. Set it using drag or by entering a value in the **Y**: field on the Status bar.
6. Once you have set the drag for the Y offset or entered a value, the Z offset becomes active. Set it using drag or by entering a value in the **Z**: field on the Status bar. If you set the Z offset using

drag, the design will move when you click the mouse button. If you entered the Z offset on the Status bar, click **OK** and the duplicated design(s) will move.

Group tools

 The Group tools flyout toolbar contains the **Group Designs** and **Ungroup Designs** tools. They work the same way as they do in other modules.

 To group items together, select them, and click the **Group Designs** tool. The items will be grouped together and the bounding box for the group will be turned on.

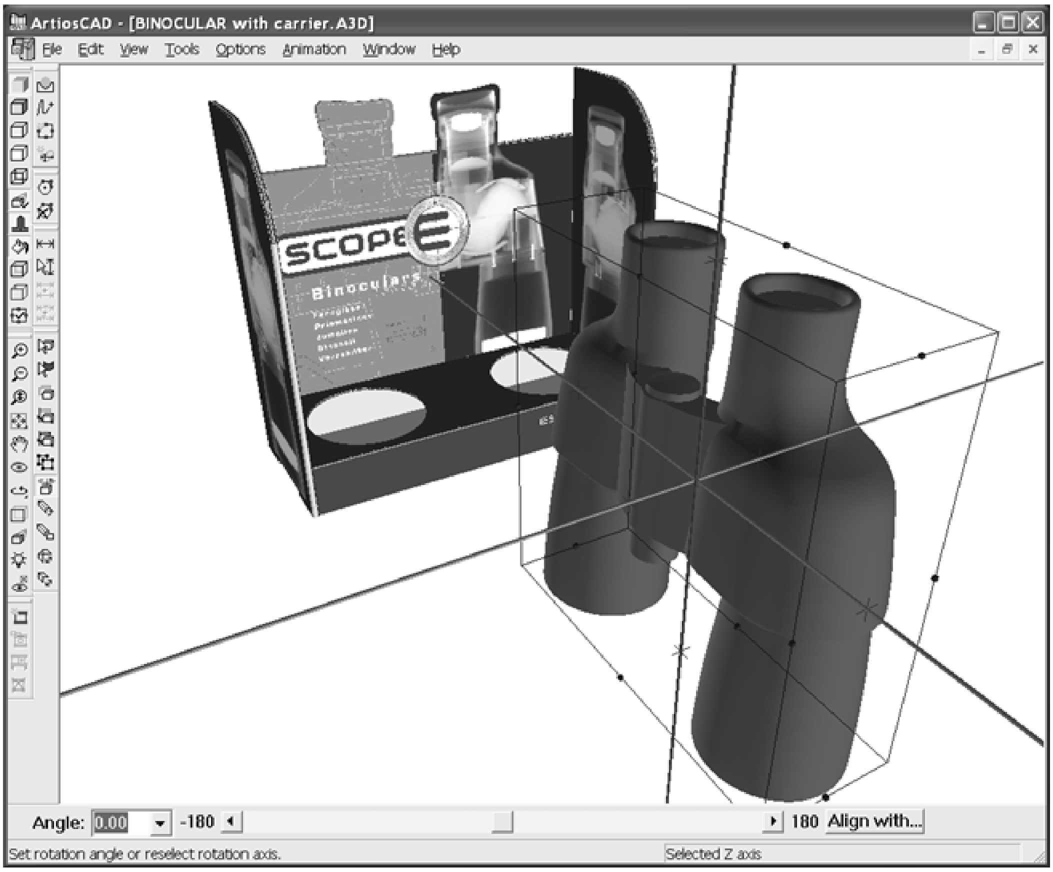
Whenever any grouped design is selected, the entire group is selected. You can nest groups within groups, but only the last-created group can be selected. When a grouped design is copied, the copy has the same group structure as the original.

 To ungroup a group, select it, and click the **Ungroup Designs** tool. The group will be ungrouped into its elements.

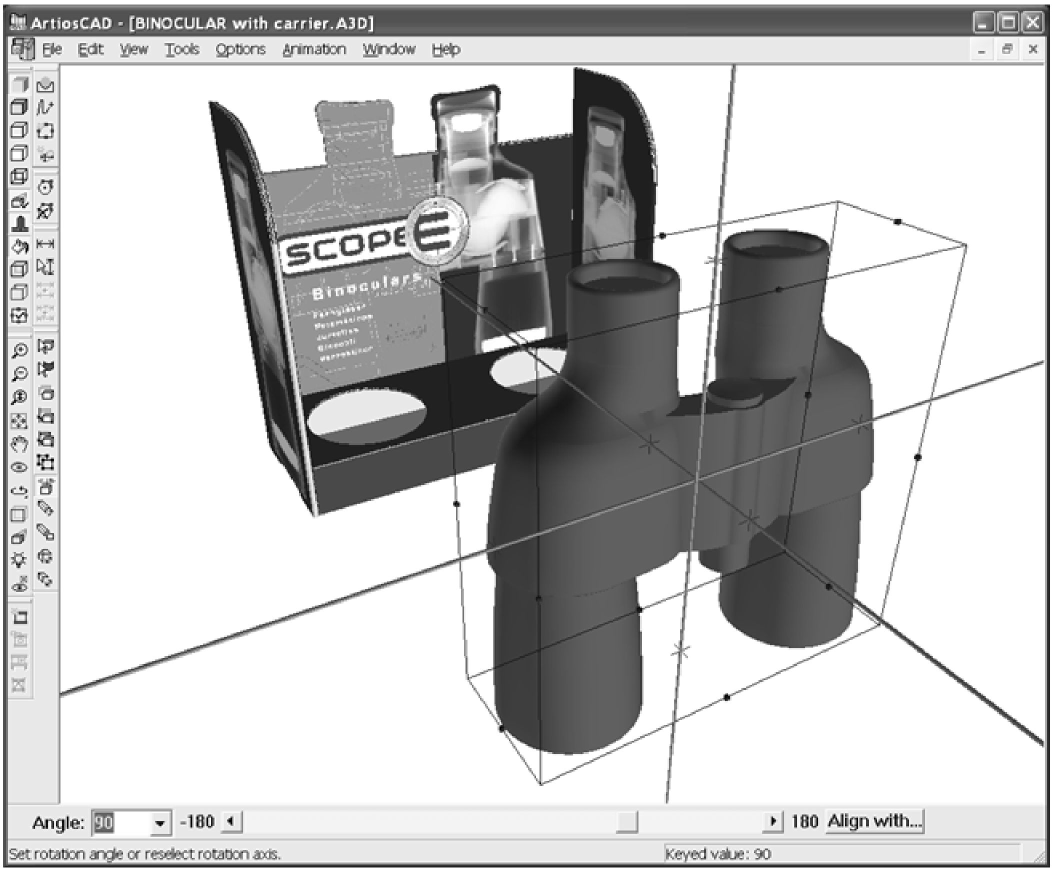
Rotate Designs tool

 The Rotate Designs tool rotates the selected designs.

1. Select the designs to rotate.
2. Click the **Rotate Designs** tool.
3. Click the line or axis to rotate the design around:



4. Select a rotation method:
   - Enter an angle in the **Angle** field or select one from the drop-down list box;
   - Use the slider to set the rotation angle;
   - Double-click an axis to rotate by 90 degrees, or to the next increment of 90 degrees if you have used the slider;
   - Click the **Align with** button. Select a line in the design(s) you are rotating, and then select either an axis or a line in another design to align with.

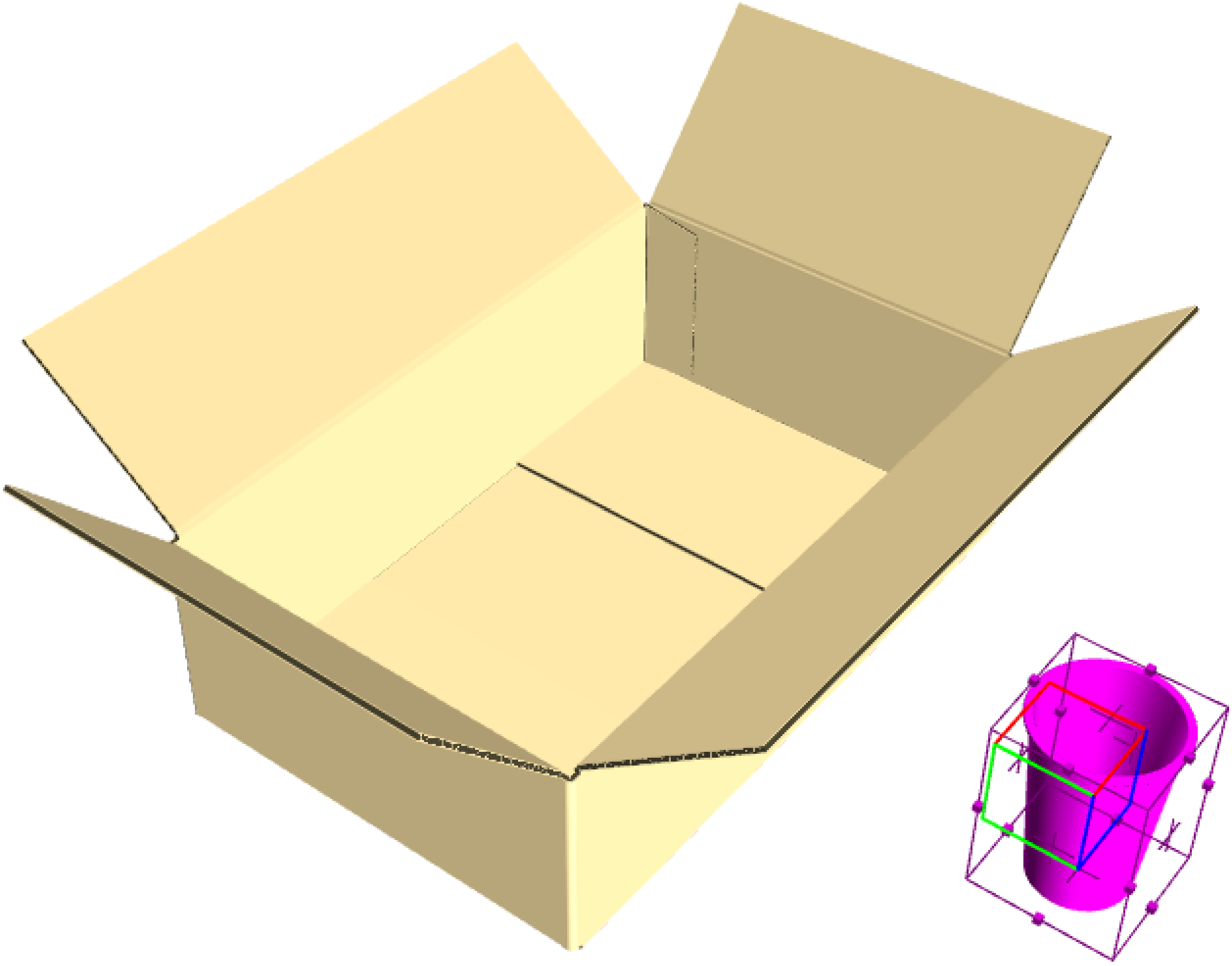


If you use Orthogonal view, the axis coming toward you is represented as a circle.

Drag on Plane tool

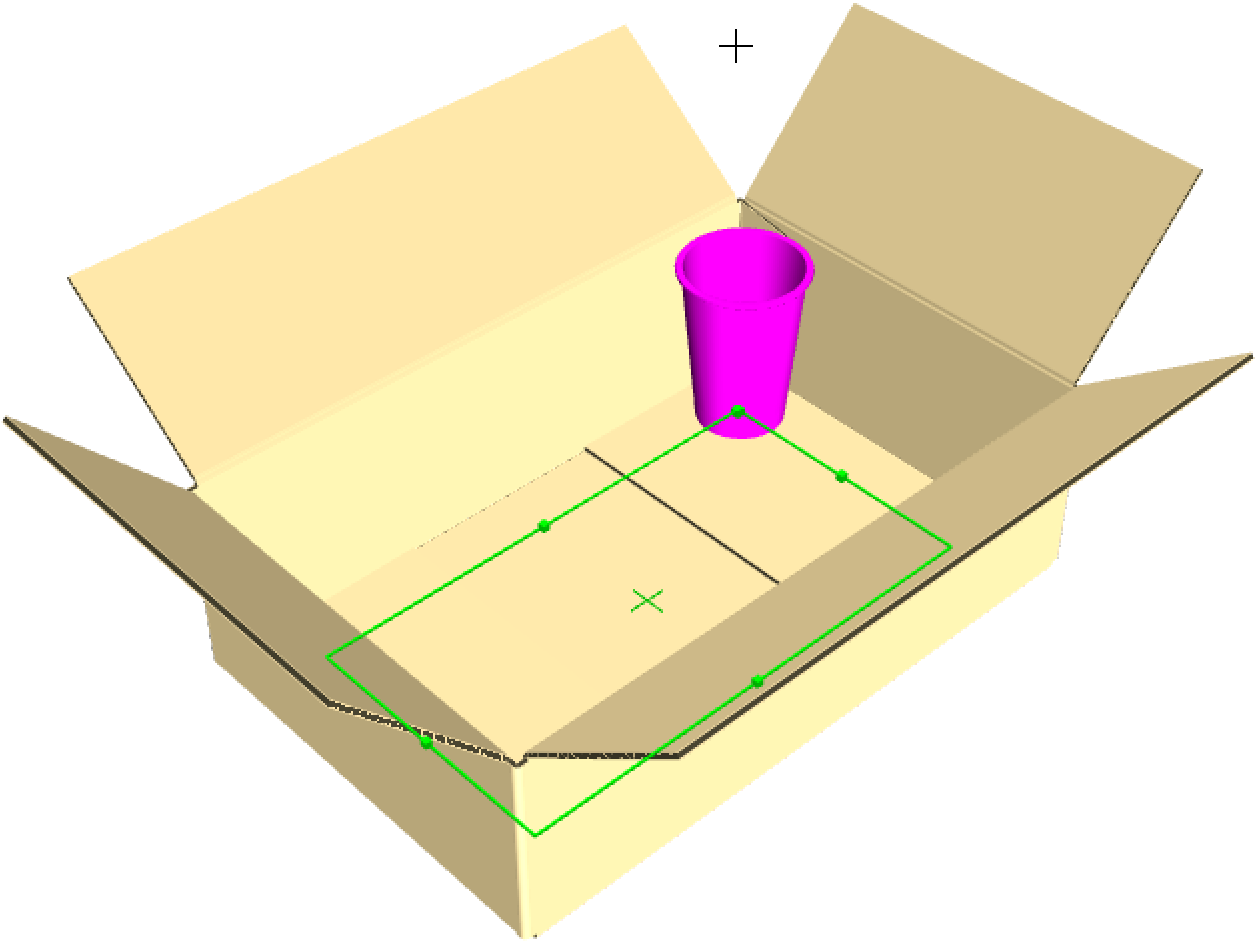
Drag on Plane lets you easily move an object onto a surface. Use it also to drag the current selection around the scene freehand.

1. Create or restore a 3D workspace containing the object(s) to move and the surface onto which to move it.
2.  Use **Select Design** to select the object(s) to move.
3.  Click **Drag on Plane**.

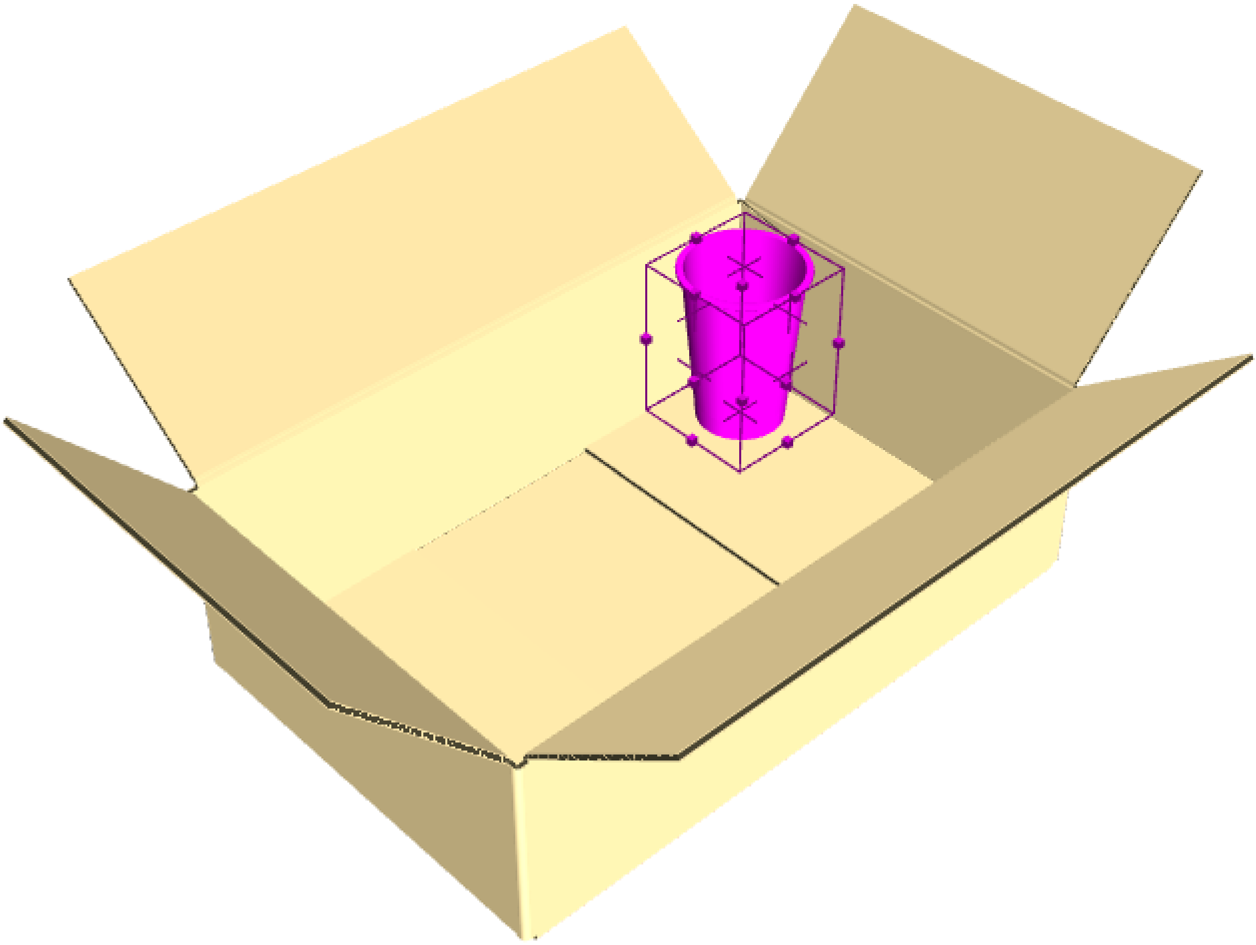


4. On the Status bar, check **Check collisions** if it is not already checked. If desired, check **Gutter** and enter a value to set a gutter between the surfaces that might collide with the object(s).

5. Click the surface onto which to move the selection. If **Check collisions** is checked, a green rectangle defines the area inside which you can move the selection. Note that the cursor is outside the box, yet the cup remains inside it.



6. Click inside the green rectangle to set the put-down point of the selection. The selection remains selected so that you can easily do something else to it, like using **Array Copy**.



If you do not check **Check collisions**, you can drag anywhere in the plane of the surface after clicking it.

Check collisions impacts performance; use it before making many copies. Also, it is designed to work with flat sides and angled sides; curved sides may cause it to behave unpredictably.

The surface onto which you move the selection must be one piece. If you have trouble placing the selection, group the pieces of the surface together.

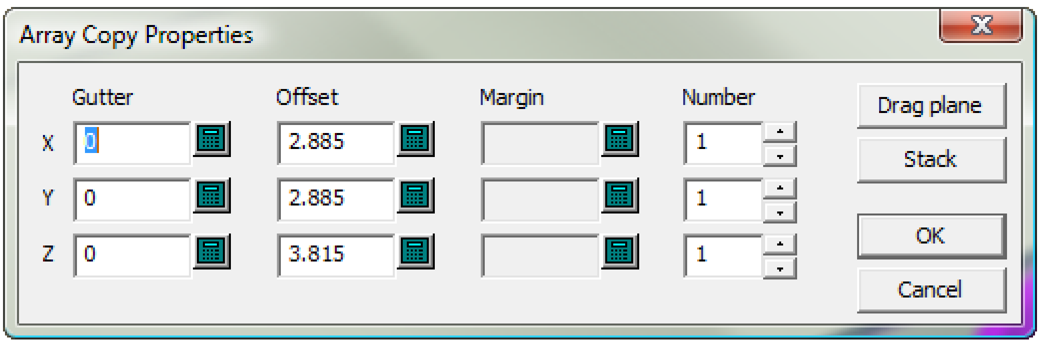
Array Copy Tool for Filling

Use **Array Copy** to copy the same design (or group of designs) multiple times in X, Y, and/or Z. It can also fill flat containers with objects. The example below took fewer than 10 clicks to make.

Before using **Array Copy** for filling, make sure the objects are aligned using the **Rotate Designs**' tool **Align with** option. Also before using it for filling, group any different surfaces together (such as planks on a pallet top). If filling a box, if the flaps are closed within one caliper, the tool considers it a continuous surface. Lastly, the tool does not fill curved surfaces, only flat ones.

When you activate this tool, the Status bar appears as follows:

Enter any desired gutter in the Gutter fields. Clicking More Options (**...**) opens the Array Copy Properties dialog box.

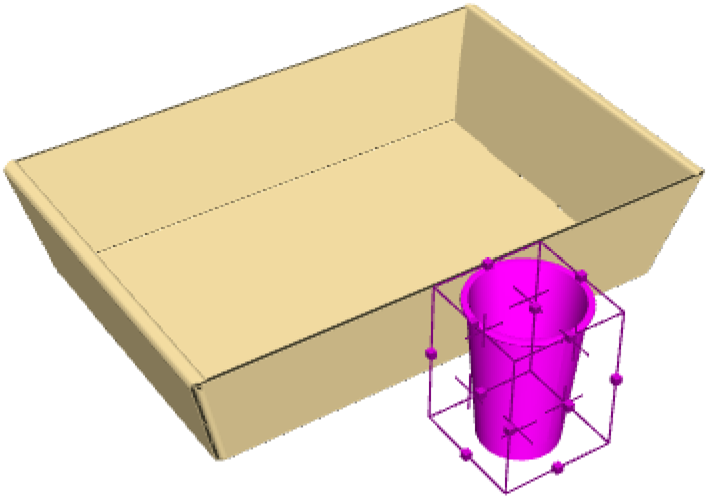


Any gutter you entered on the Status bar seeds the **Gutter** fields in this dialog box. Each **Offset** is the size of the bounding box for the object(s) you are copying. **Margin** lets you set a margin between the object(s) you are copying and the object you are filling. **Number** lets you set the number of copies and is limited to 100 to prevent inadvertently making too many copies. **Drag plane** lets you immediately drag out the number of copies in X and Y in the design window without being constrained by the object you are filling. **Stack** lets you drag out the number of copies in the Z axis without being constrained by the object you are filling.

Clicking **OK** on the Status bar finalizes the copy.

To use this tool for filling, do the following:

1. Create or restore a 3D workspace containing the object(s) to copy and an object to fill.
2.  Use the **Align With** mode of the **Rotate Designs** tool to align the object you are copying with the object you are filling.
3.  Use **Select Design** to select the object(s) to copy.



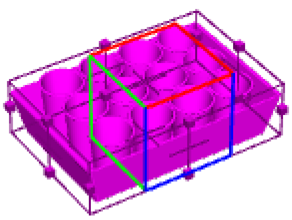
4.  Click **Array Copy**.
5. Click the surface to fill.

6. Click **OK** on the Status bar to finalize the copy.

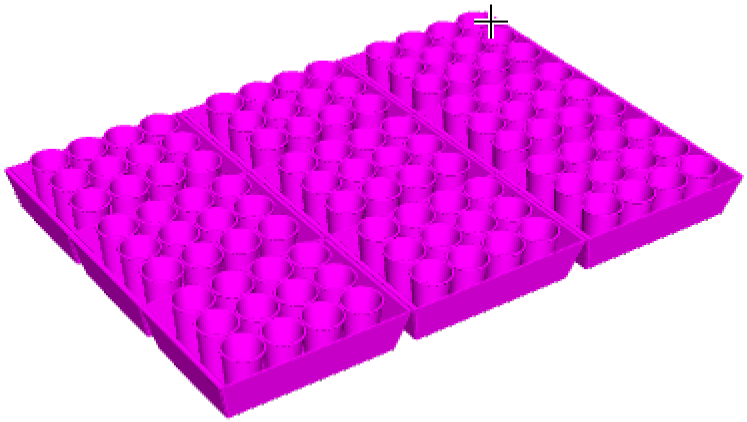
Array Copy Tool for Stacking

To use the Array Copy tool for stacking, do the following:

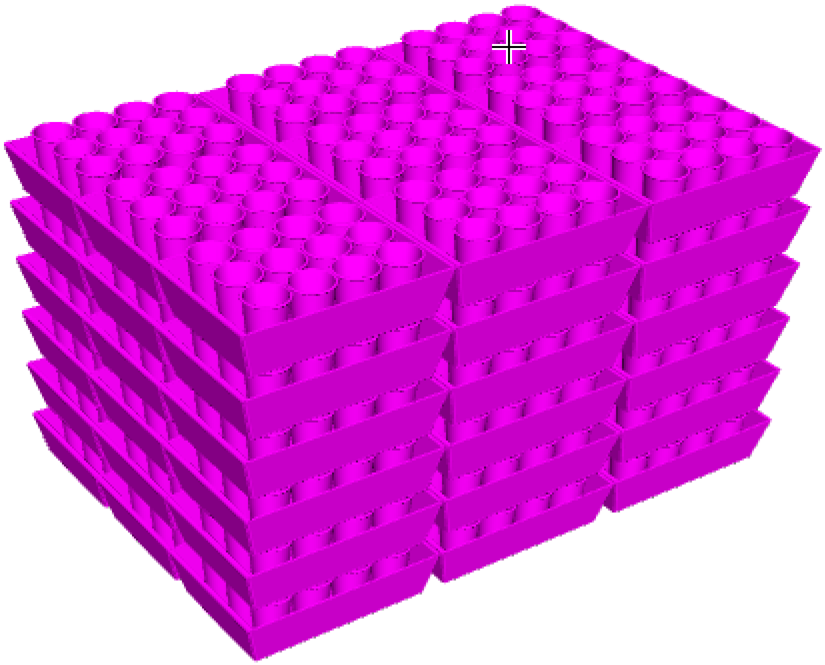
1. Create or restore a 3D workspace containing the object(s) to copy.
2. Use **Zoom out** to make room for the drag you will use with this tool.
3. Use **Select Design** to select the object(s) to copy.
4. Click **Array Copy**.
5. Click inside the red top of the cube to copy horizontally.



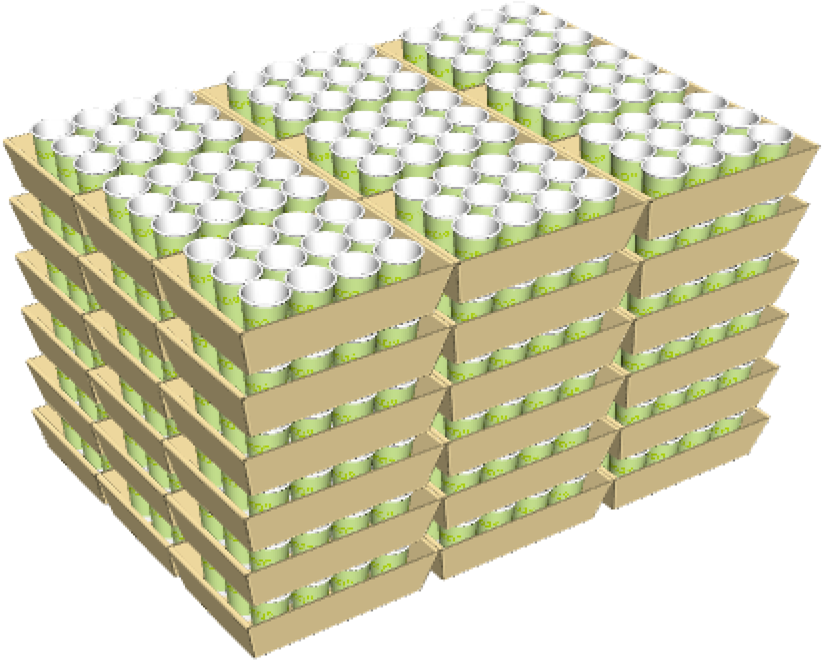
6. Drag the desired number of copies in X and Y and click to set the number. The drag is limited to 20 copies in each direction; to make more, click **More Options** and set the number of copies in the Array Copy Properties dialog box.



7. Click **Stack** on the Status bar and drag the height of the stack.



8. Click **OK** on the Status bar to finalize the copies.



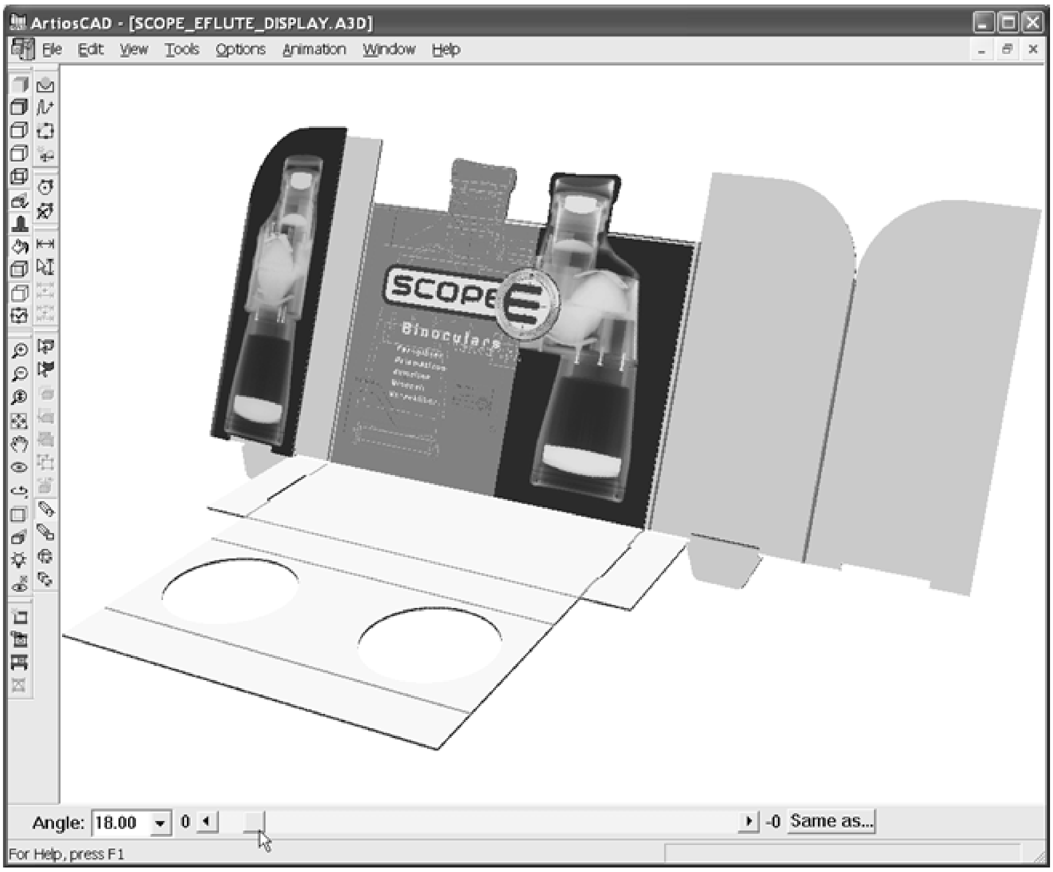
Folding designs

Use the tools in this section to fold a design in 3D.

Fold Angle tool

 The **Fold Angle** tool allows you to fold designs along selected creases (or other line types).

1. Click the **Fold Angle** button.
2. Select the crease, bend, perf, cut & crease, reverse crease, 50% cut, or hidden crease lines to fold. If you want to select more than one line at a time, hold down `SHIFT` as you select lines, or hold down `CTRL` and make a window selection.
3. Now select one of three methods to specify the fold angle:
   - Enter an angle in the **Angle** field, or select a predefined angle from the list box.
   - Drag the slider to set the angle. The view updates as you drag the slider; performance is directly related to the capabilities of the display adapter.
   - Click **Same As** and indicate a crease that already is folded at the desired angle.



You may need to click **Scale to Fit** to reset the view of the design.

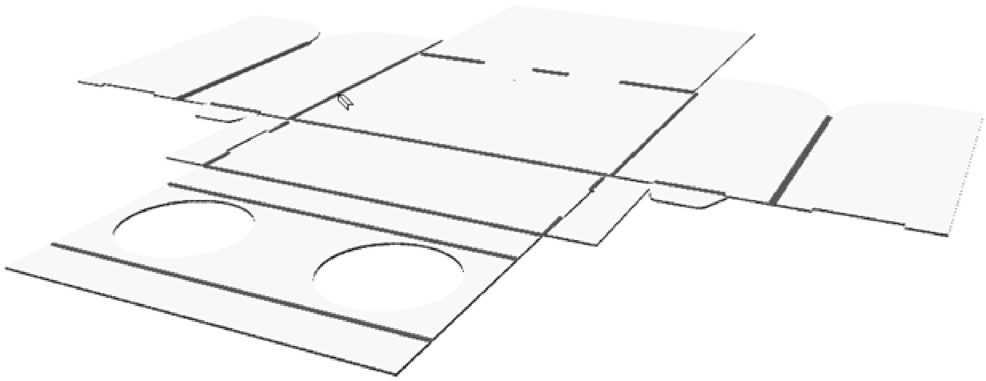
Fold All tool

 The Fold All tool folds all creases (or other foldable lines except bends) in a design at once.

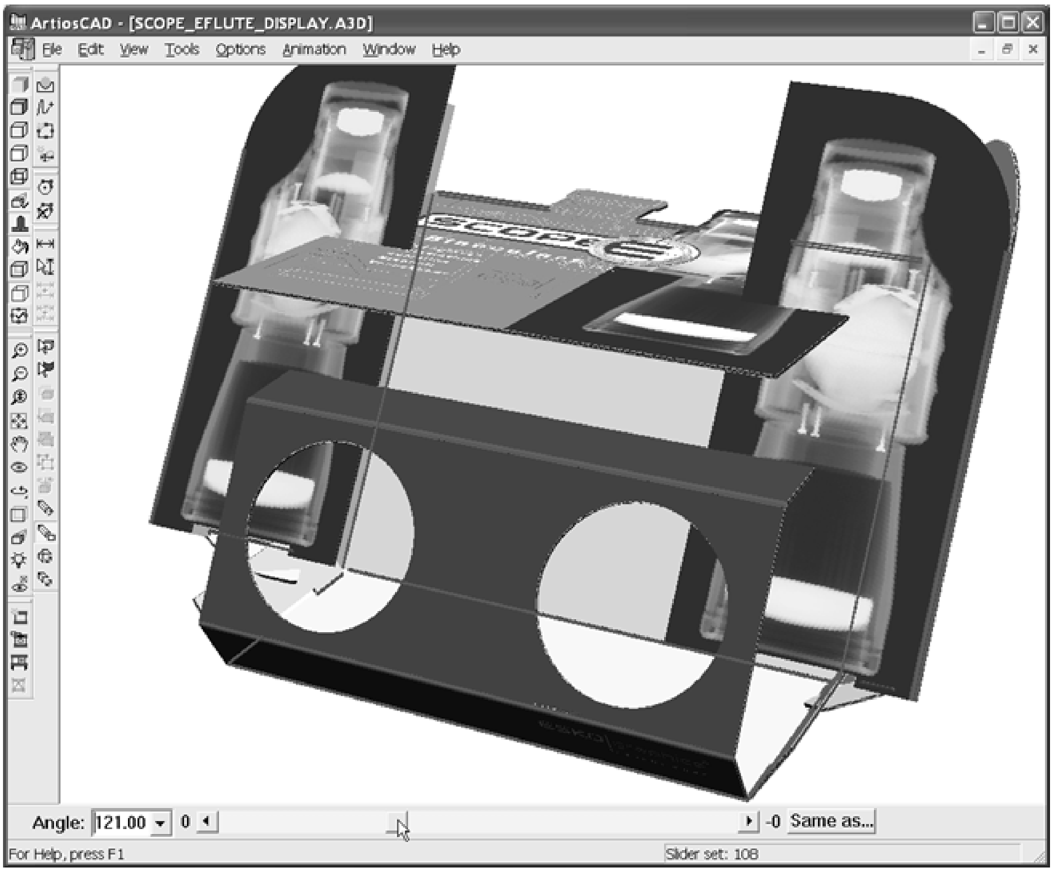
1. Click the **Fold All** button.
2. Select one foldable line in the design you wish to fold. All the foldable lines that are not bends will turn magenta to indicate they are selected. To select additional designs, hold down the `SHIFT` key and select the desired designs.
3. Select one of three methods to specify the fold angle:

   - Enter an angle in the Angle field, or select a predefined angle from the list box next to the Angle field.
   - Drag the slider to set the angle. You will see an outline of what is being folded.
   - Click the **Same As** button next to the slider and indicate a line that already is folded at the desired angle.

Shown below is a flat design.



Next is moving the slider to adjust all the fold angles.



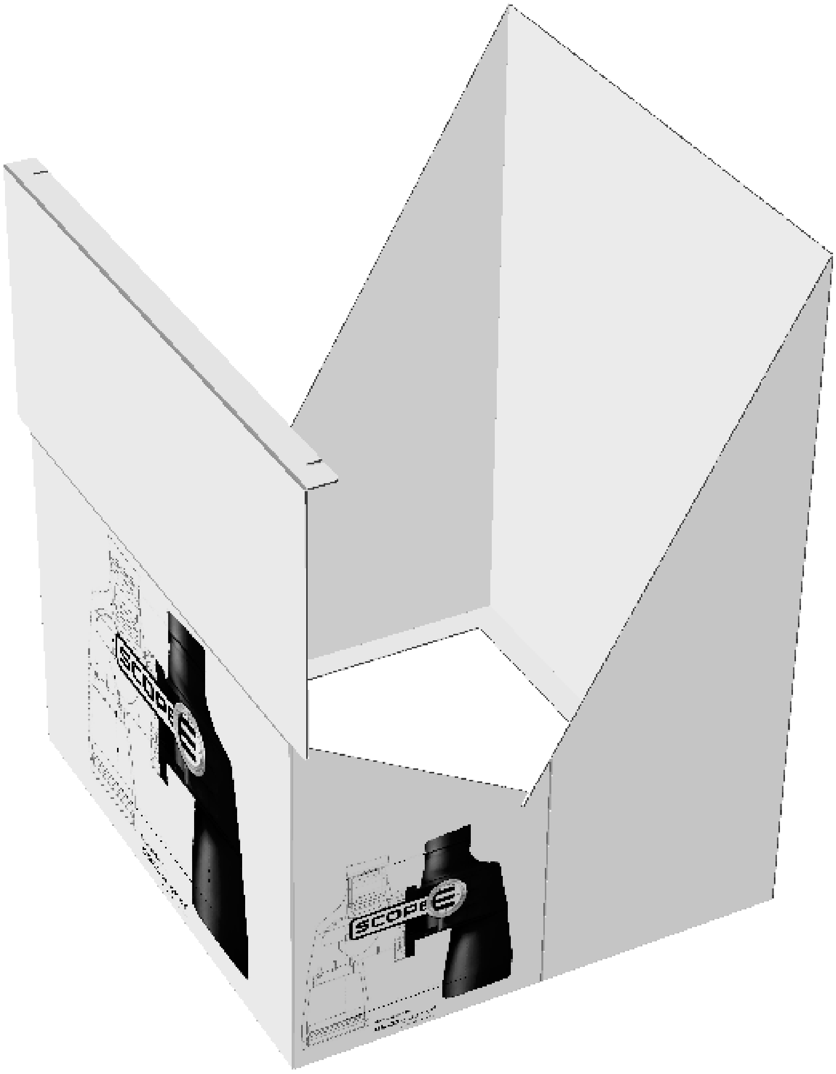
Finally, all creases are folded at 90 degrees.

You may need to click **Scale to Fit** to reset the view of the design.

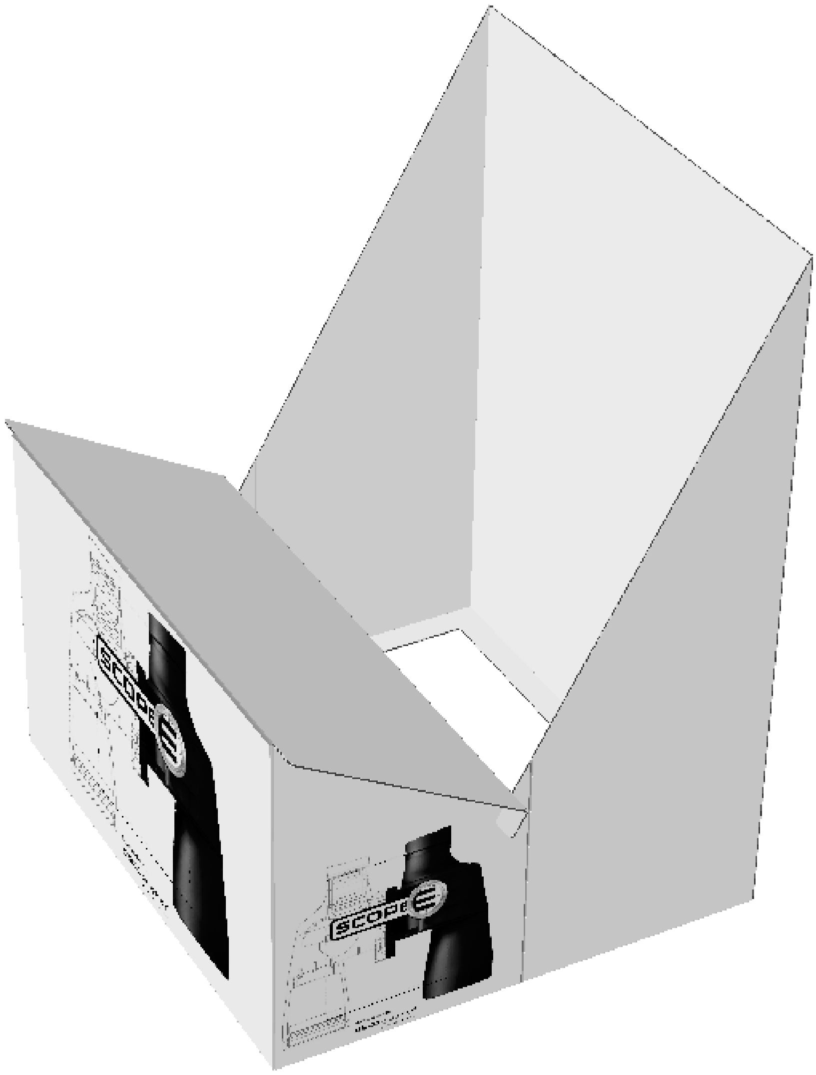
Fold 1 to Meet tool

 The **Fold 1 to Meet** tool allows you to change angles of folds to make two lines meet. This feature is used for designs that do not fold at 90 degrees, such as trapezoids.

1. Decide which line to move that will meet another.
2.  Click **Fold 1 to Meet**.
3. Click the line that will meet the second line.
4. Click the foldable line whose angle will change so that the lines will meet (the front crease below).
5. Click the line that the first line will meet.



Shown below is the display after using the tool.



You may need to click **Scale to Fit** to reset the view of the design.

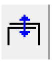
Fold 2 to Meet tool

 The **Fold 2 to Meet** tool works the same way as Fold 1 to Meet, except that it changes two foldable lines at once.

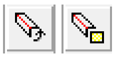
1. Decide which lines should meet after folding.
2.  Click the Fold 2 to Meet tool.
3. Click the first line that will move.
4. Click the foldable line whose angle will change in order to move the first line.
5. Click the foldable line whose angle will change in order to move the second line.
6. Click the second line that will move.

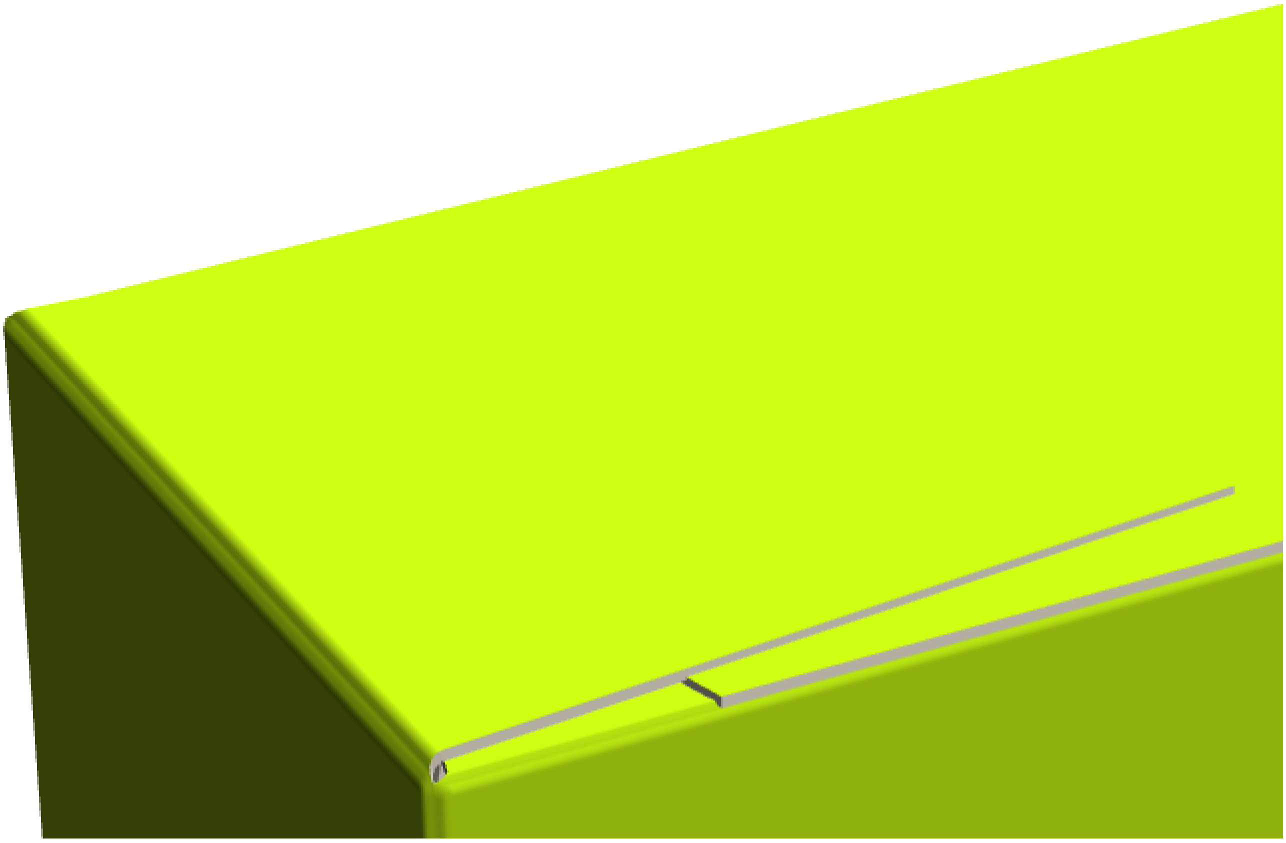
You may need to click **Scale to Fit** to reset the view of the design.

Flap Priority tool

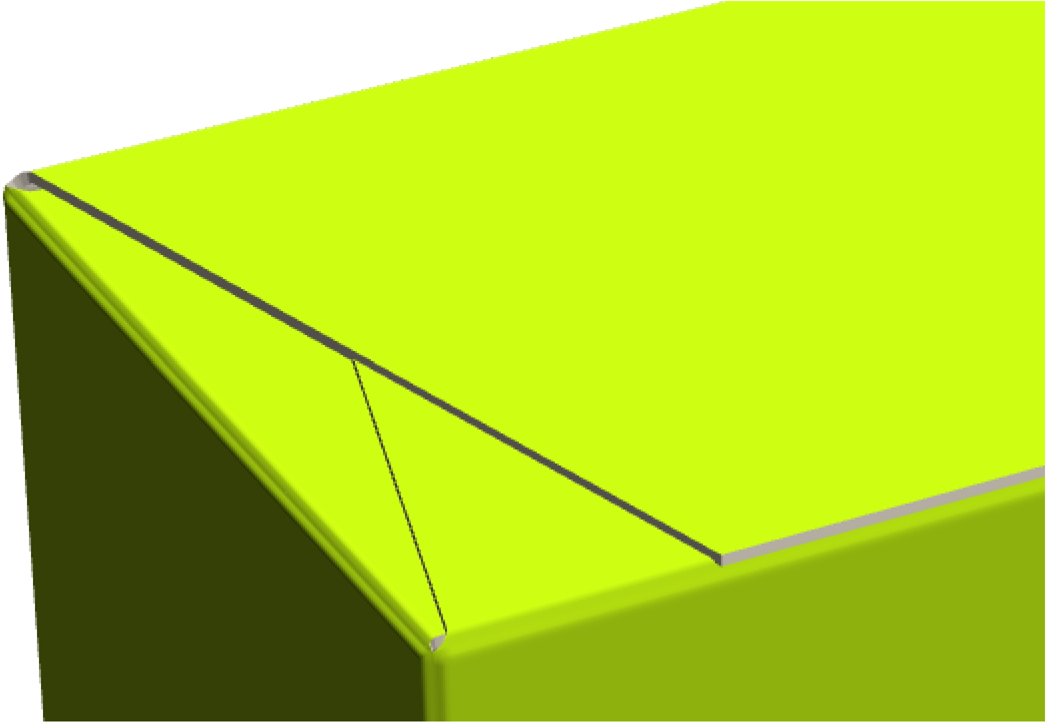
 **Flap Priority** lets you shuffle flaps on a closed box to let ArtiosCAD know which flap is visible on top. It does this by stretching the flap's crease by half a caliper. ArtiosCAD removes the stretch when you change the fold angle.

To use this tool, do the following:

1. Create or restore a 3D workspace.
2.  Use one of the **Fold** tools to close the design. The flaps may not be in the correct order.



3.  Click **Flap Priority** on a flap to move it to the top. Repeat as desired.



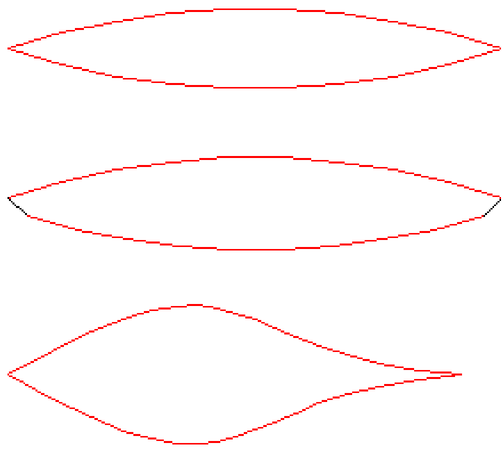
Folding designs with curved creases

ArtiosCAD can fold designs with reasonable curved creases using patent-pending technology.

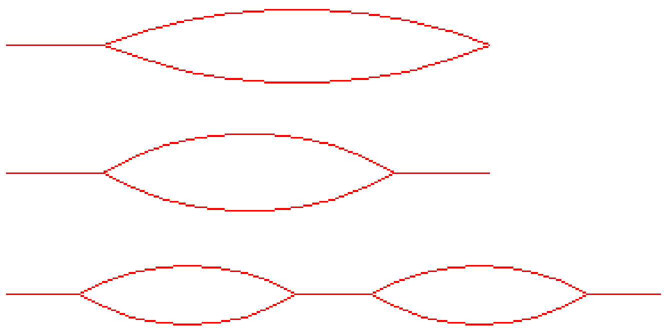
Linked folds

ArtiosCAD recognizes different types of folds as being connected in that you cannot fold one crease without the other creases folding at the same time.

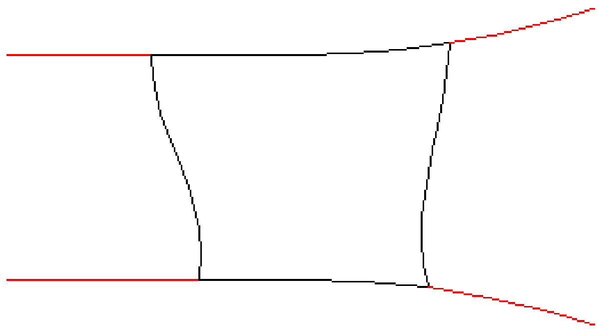
In the examples below, the arcs meet or almost meet and have a similar radius. Both will fold at the same angle.



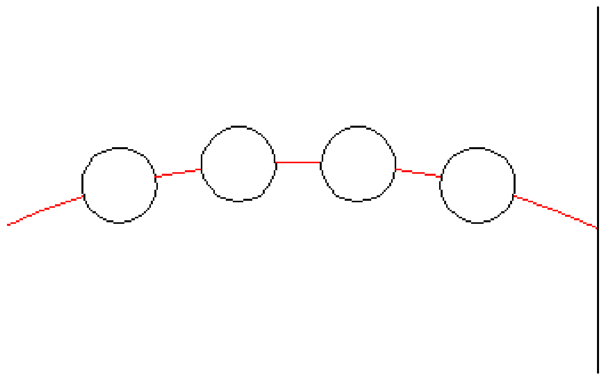
In the examples below, there are pairs of arcs in-line with straight lines. The angles of the arcs follow the angle of the straight fold.



In the example below, the creases are parallel between two faces and fold together.

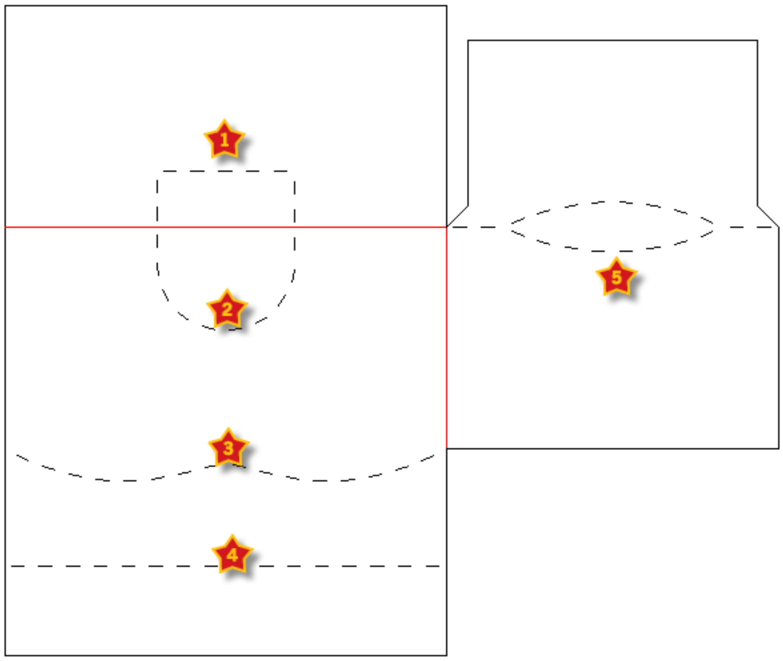


In the example below, the parallel folds between the same two faces are treated as one fold.



Curved perf behavior in 3D

ArtiosCAD tries to tell the difference between perfs that fold and perfs that tear, but as there is no property to identify this behavior, it has to guess. The figure below shows how ArtiosCAD may behave in some situations.

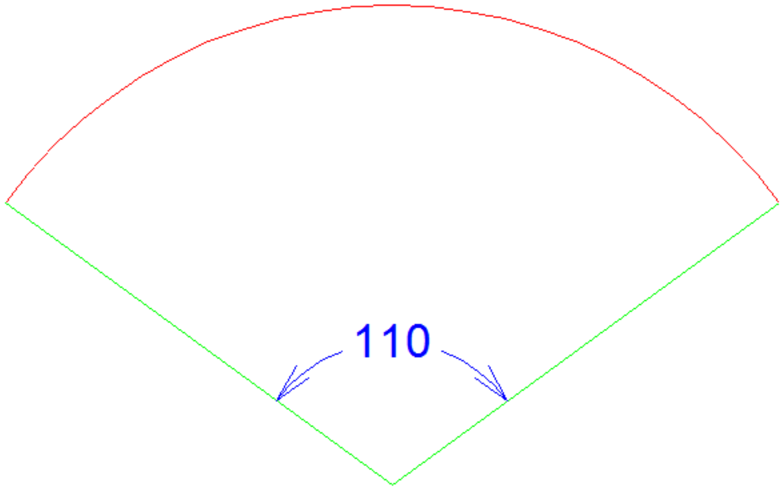


Number	Result
1	Perf with sharp angles treated as separate lines. One of the perf folds; the others are broken.
2	The arc has too large an angle to fold so it is treated as a cut.
3	Smooth curved perf that folds properly.
4	Straight perf that folds properly.
5	Diamond fold made from perf lines that folds properly.

Unsupported 3D designs and workarounds

Designs with curved creases that do not convert to 3D usually rely on features that are not supported, such as:

- Folds in a place where there is no crease, such as across the face of a panel
- Relies on twist as an essential part of construction.
- Has an arc angle of more than 110 degrees. If the fold angle is bigger than this, the distortion is too great and the fold is ignored.

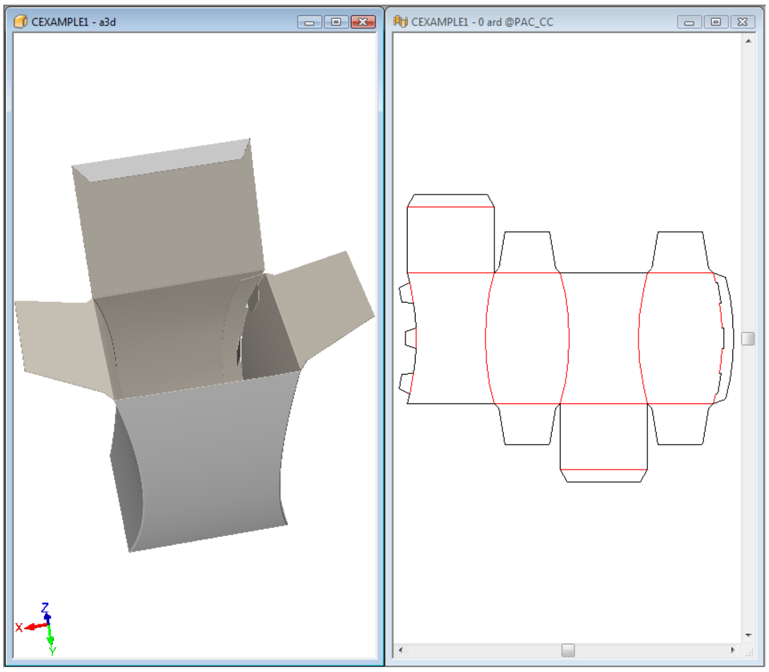


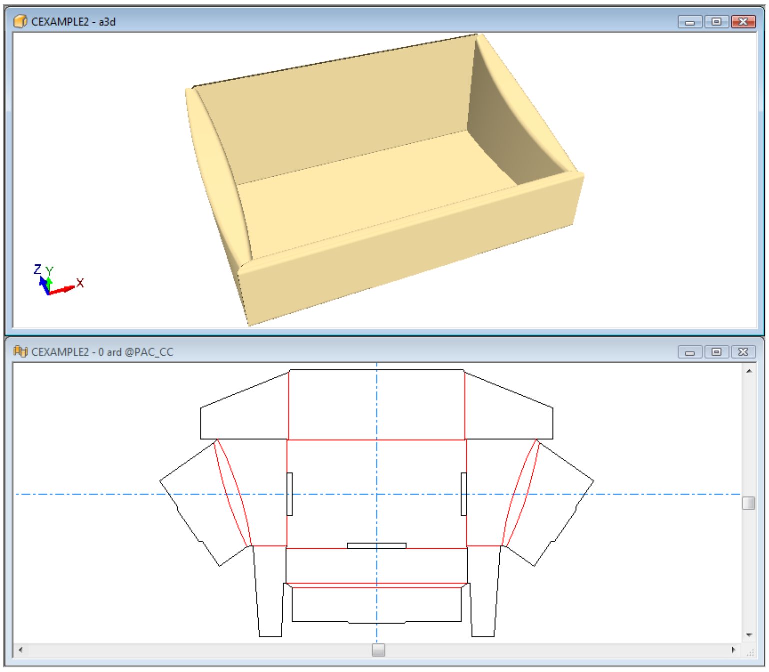
Sometimes these restrictions can be overcome by:

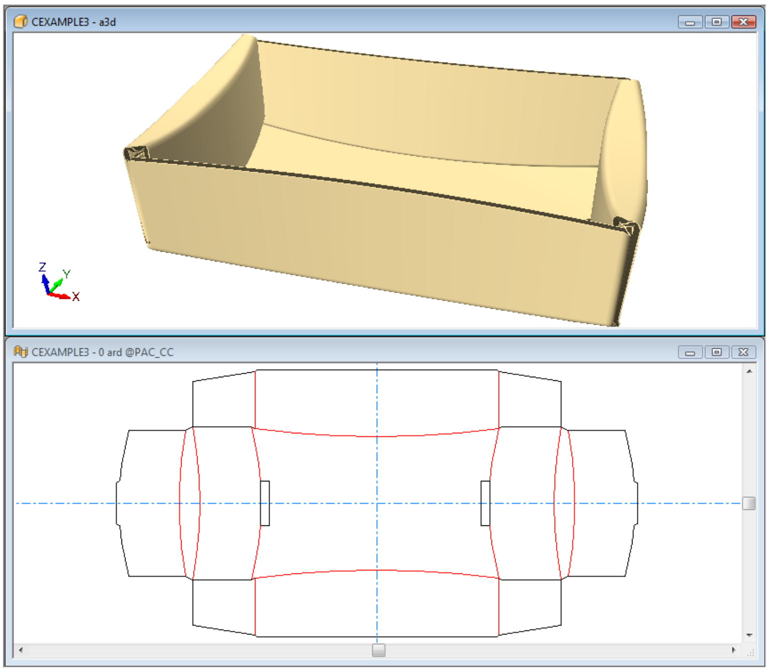
- Adding a crease or a bend line where there isn't one presently
- Breaking creases automatically when there are more than five connected creases
- Adding bend lines to simulate a twist.

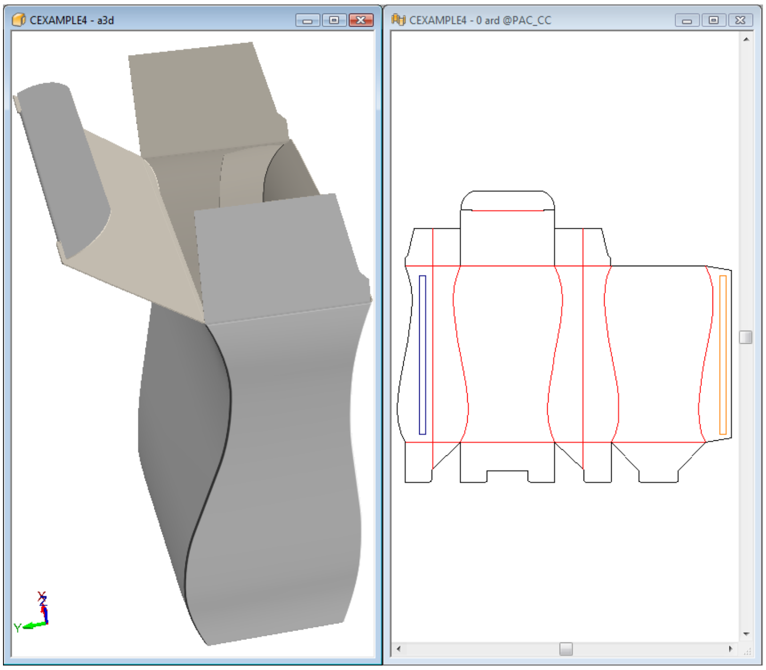
Examples of supported curved crease designs

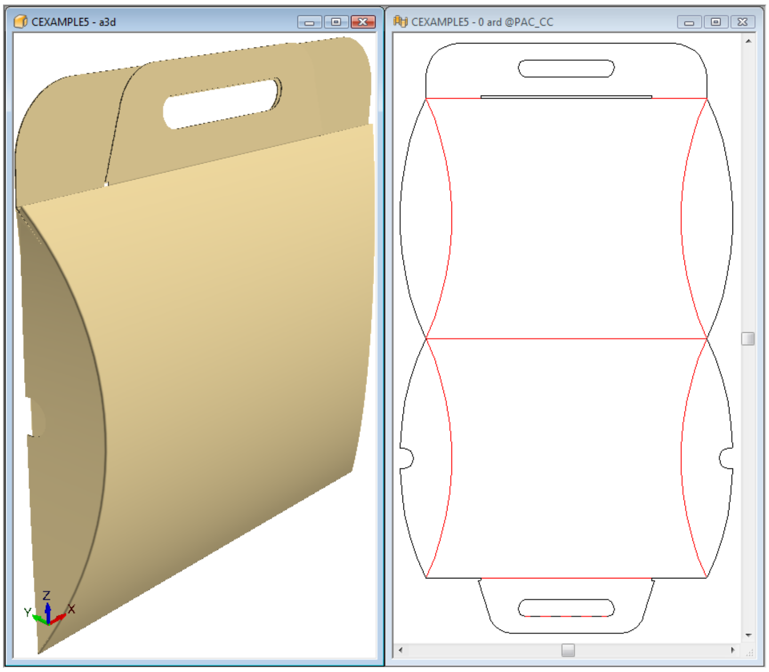
Shown below are unfolded and folded examples of supported types of curved crease designs. Graphics are turned off to emphasize the structure of the designs.

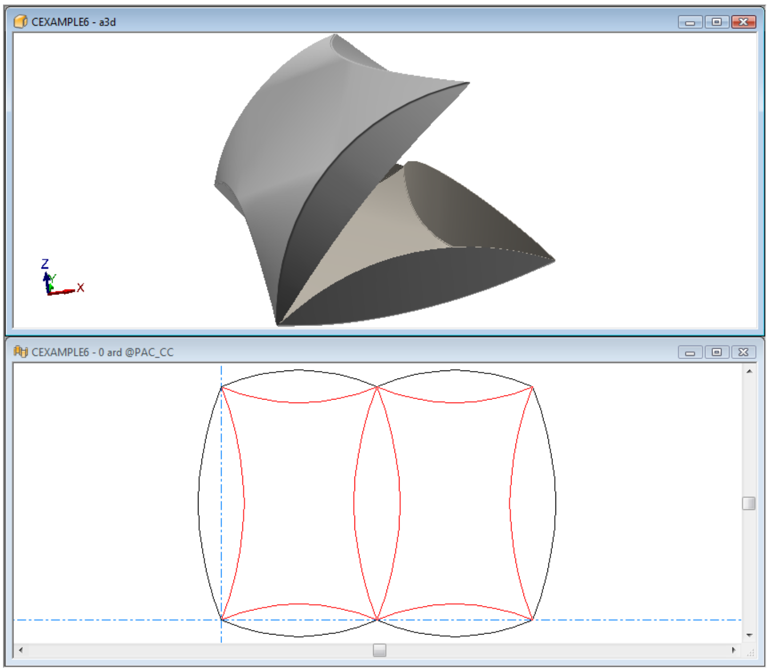


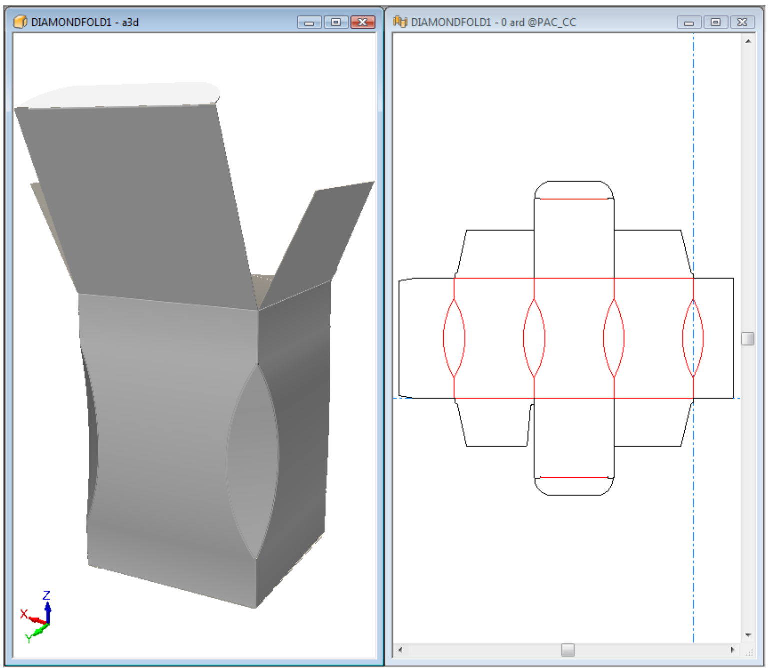

CEXAMPLE2 - a3d
CEXAMPLE2 - 0 ard @PAC_CC

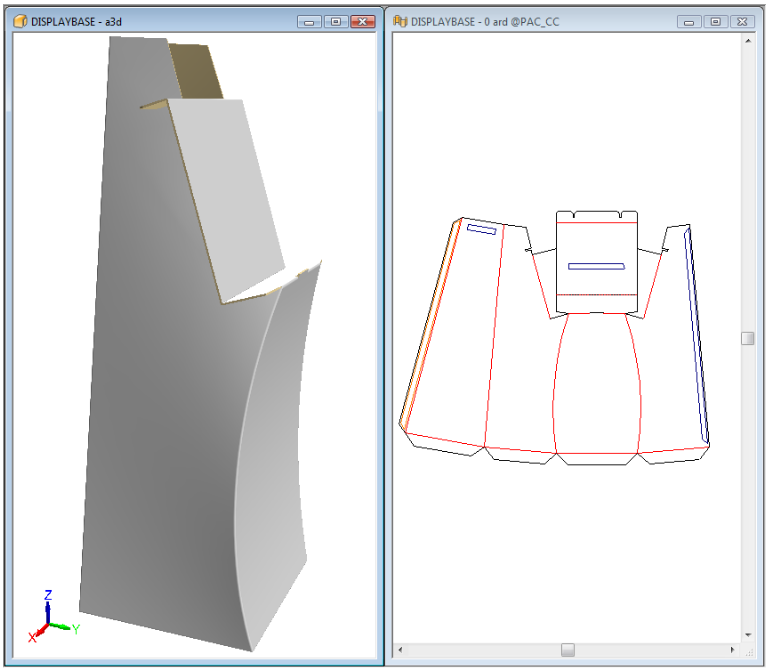

CEXAMPLE3 - a3d
CEXAMPLE3 - 0 ard @PAC_CC

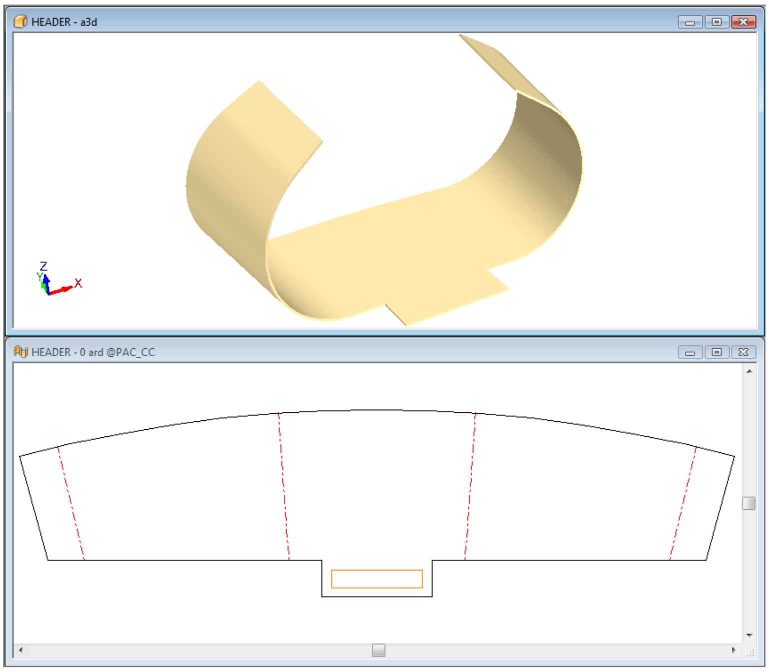

CEXAMPLE4 - a3d
CEXAMPLE4 - 0 ard @PAC_CC


CEXAMPLE5 - a3d
CEXAMPLE5 - 0 ard @PAC_CC


CEXAMPLE6 - a3d
CEXAMPLE6 - 0 ard @PAC_CC

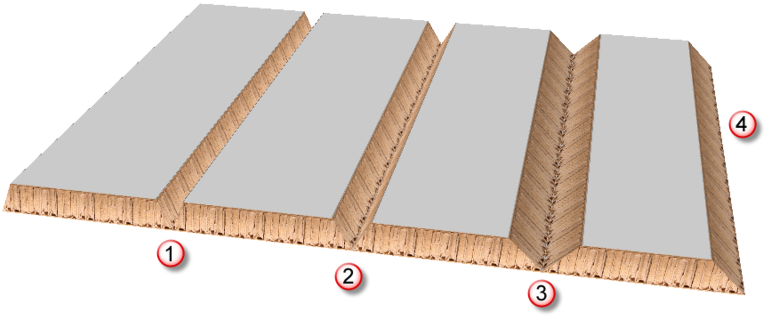

DIAMONDFOLD1 - a3d
DIAMONDFOLD1 - 0 ard @PAC_CC





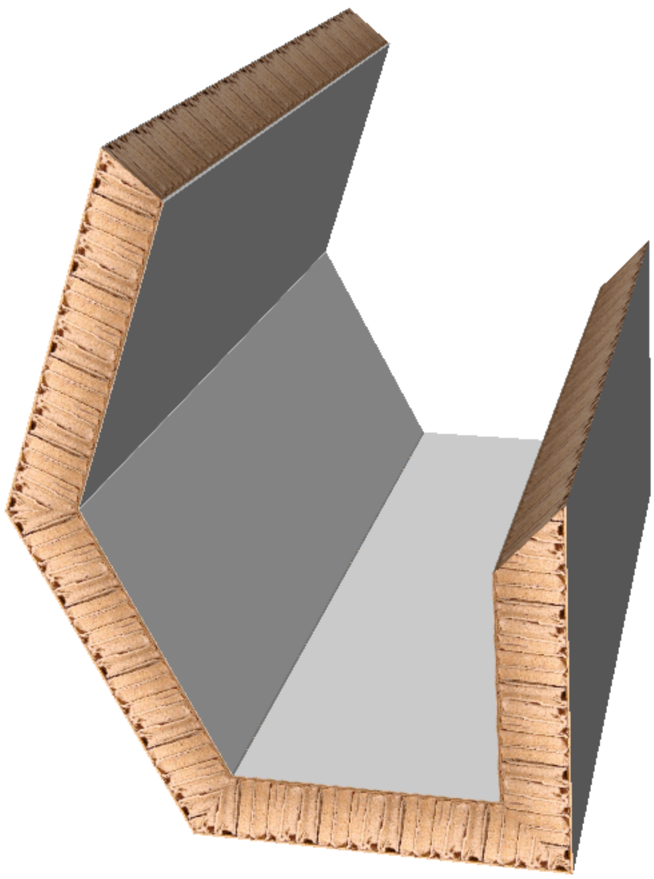
Folding designs with V-notch lines for Re-board®

ArtiosCAD accurately displays V-notch lines with angles from 5 to 60 degrees in 3D.



Line	Type
1	22.5 degree V-notch cut
2	30 degree V-notch cut
3	45 degree V-notch cut
4	45 degree V-notch cut

When folded, the above example looks like this:

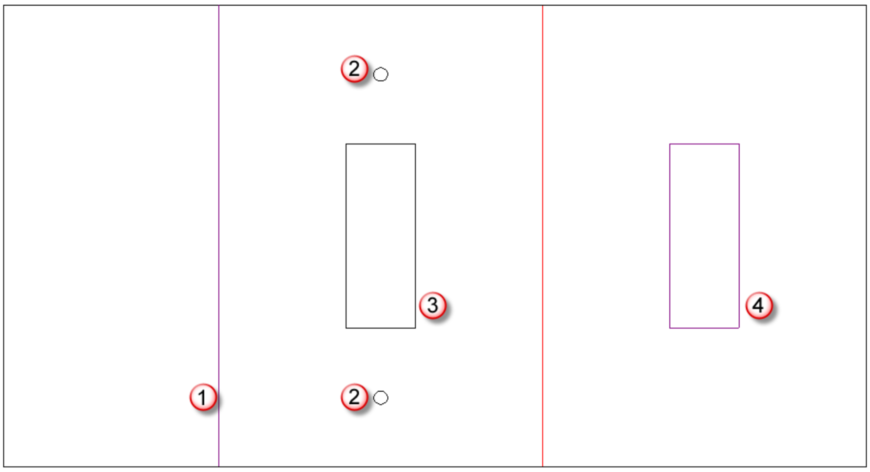


Note:

ArtiosCAD ignores the inside loss of the board when folding V-notch creases and partial cuts.

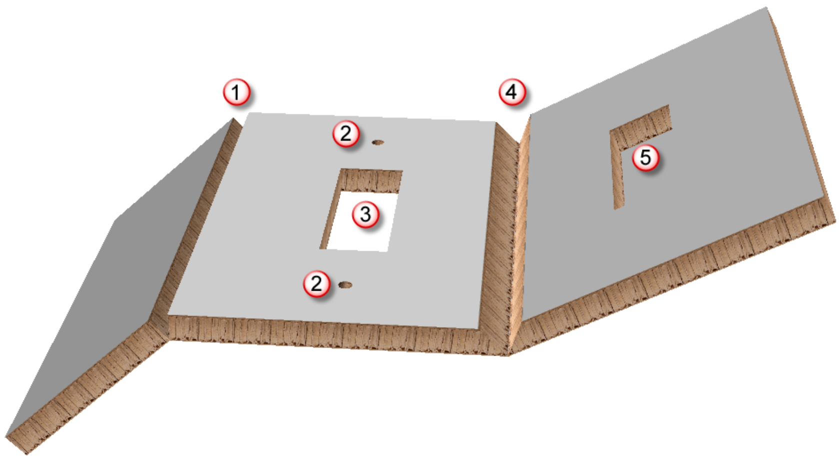
Partial cuts for Re-board®

When using Re-board®, a partial cut or reverse partial cut across a panel folds in the same way as a partial cut folds when using corrugated or foam. A partially cut hole is formed when the partial cut lines form a loop. The examples below show how partial cuts look.



Line	Type
1	Partial cut
2	Drill hole
3	Full-depth hole made with cut lines
4	Partial-depth hole made with partial cut lines

The flat design looks like this in 3D:

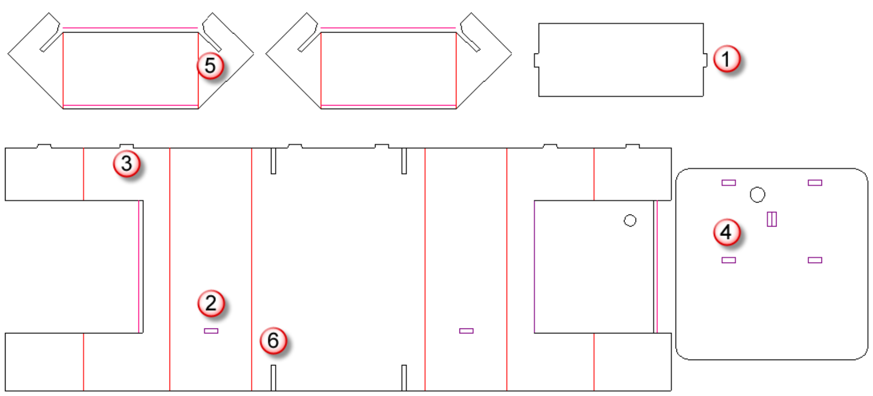


Object	Construction
1	Partial cut
2	Drill holes
3	Full cut hole
4	V-notch cut
5	Partial-depth hole

Folding Re-board® designs using the Snap Tab/Slot tool

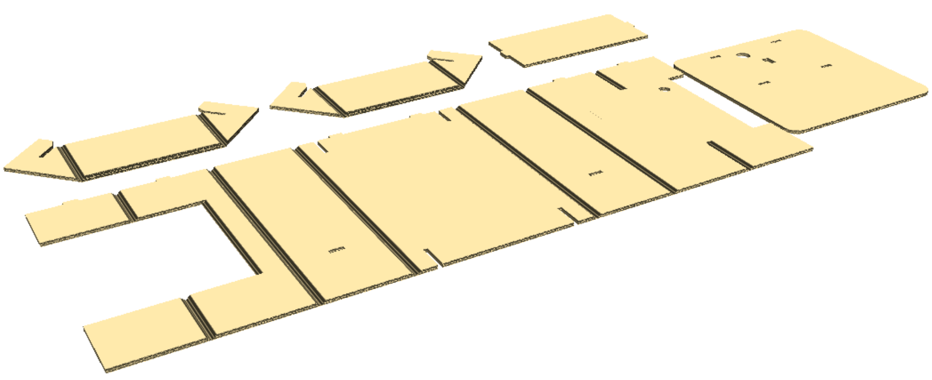
Designs made with Re-board® typically use slots and tabs for assembly. The **Snap Tab/Slot** tool eases this assembly in 3D.

Consider the five-piece single design below. Tab 1 fits into slot 2, tab 3 fits into slot 4, slot 5 fits into slot 6, and so on.

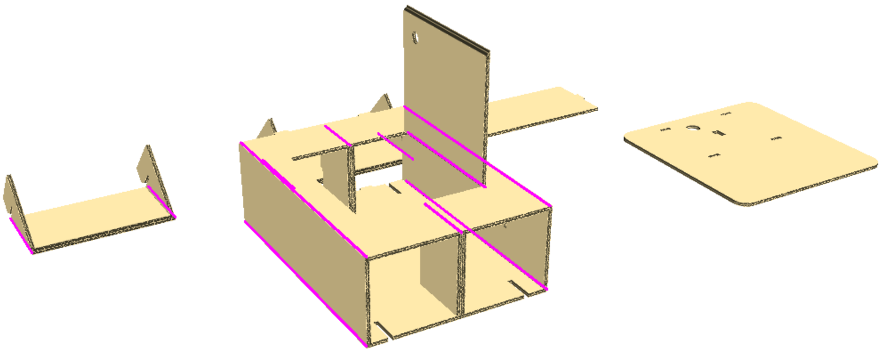


One way to assemble this design is as follows:

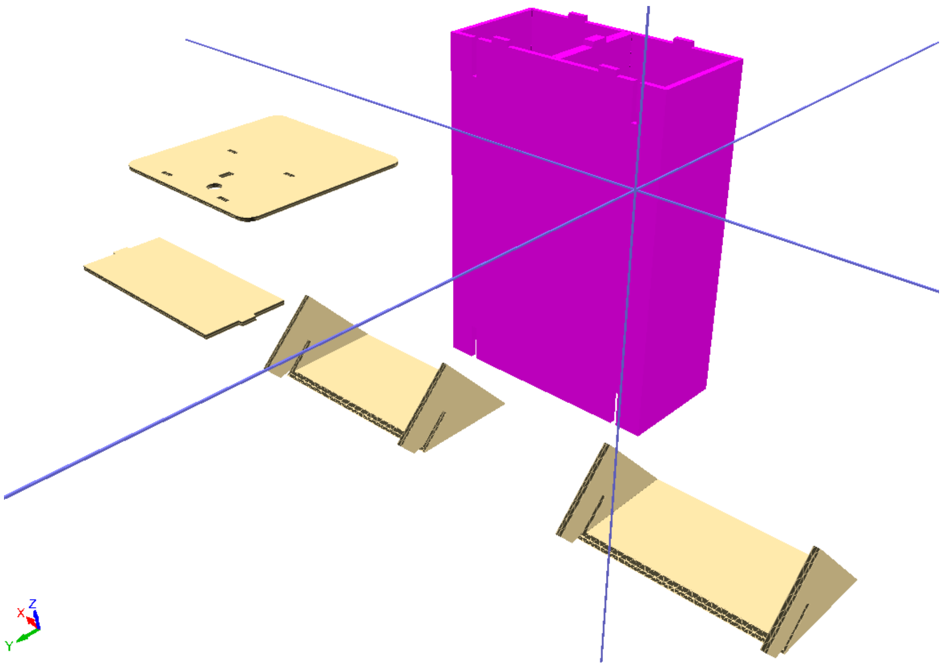
1. Convert the design to 3D.



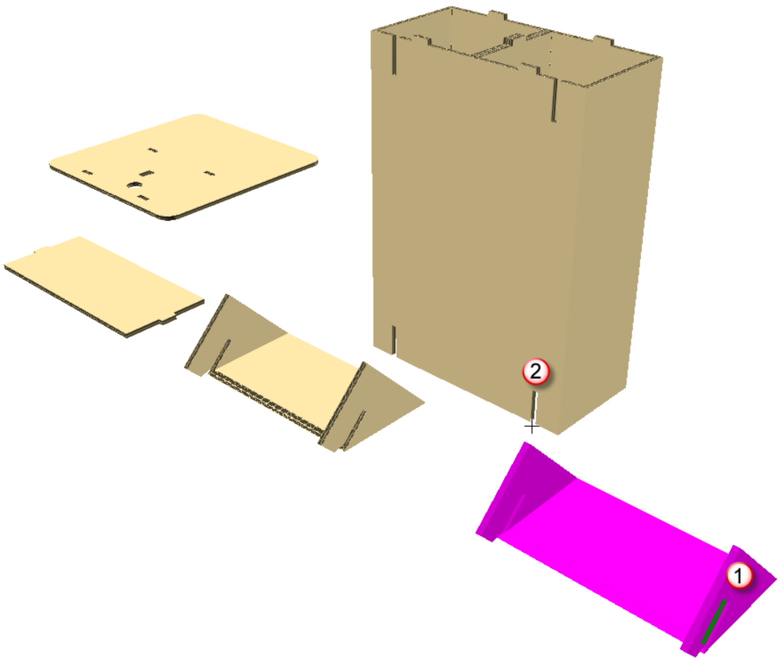
2. Click **Fold All**, select all the V-notch creases with a marquee selection, and set the fold angle to 90 degrees.



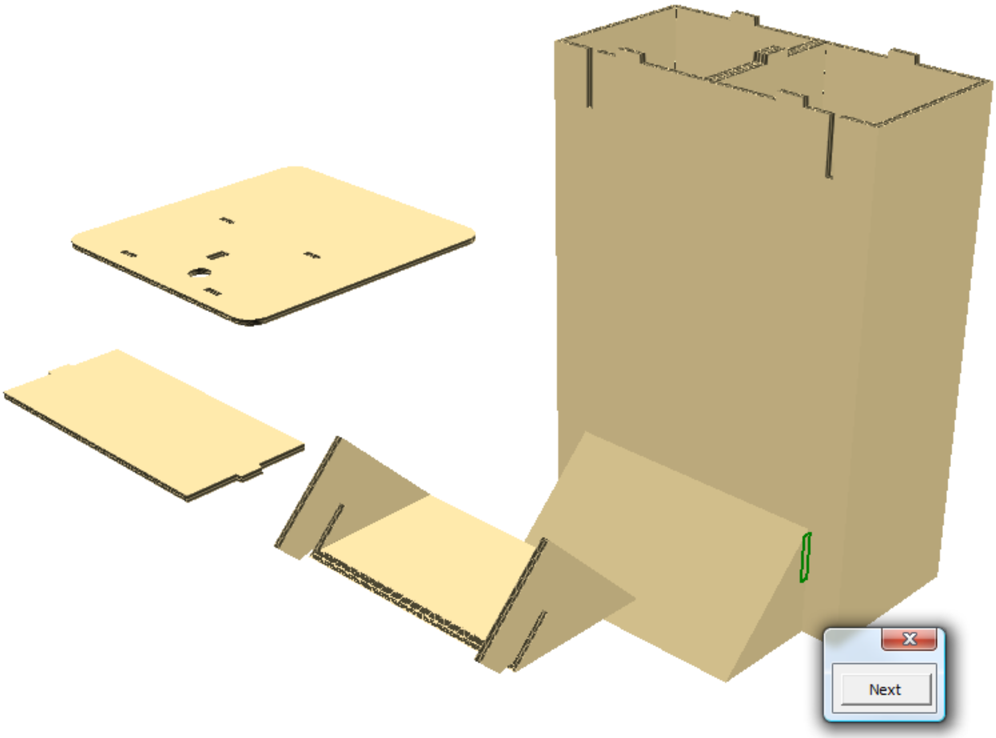
3. Use **View Angle** to change the view to the other side.
4. Use **Select Design** to select the base.
5. Use **Rotate Design** to rotate the base upward.



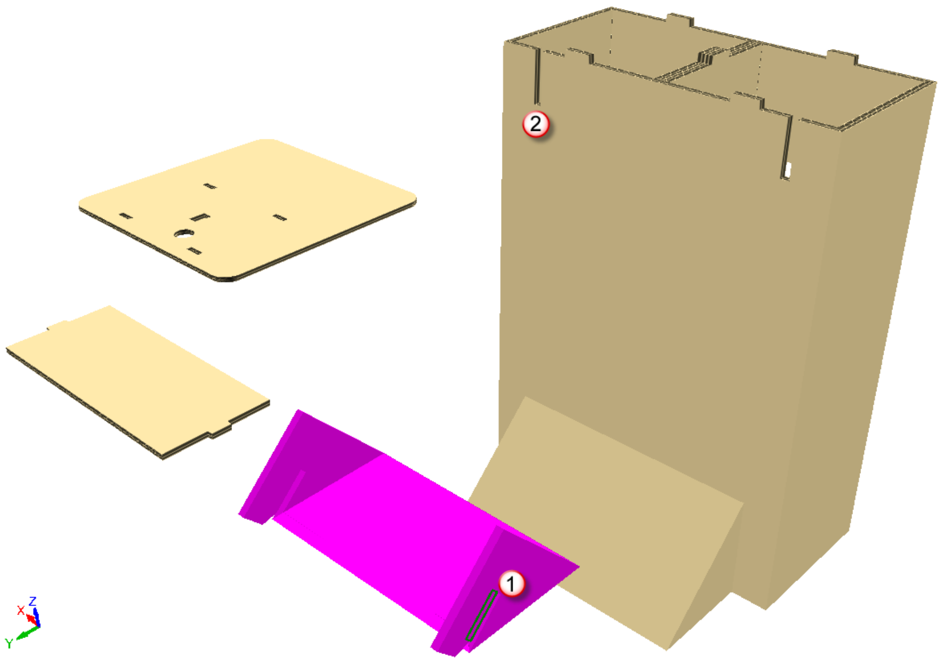
6.  Click **Snap Tab/Slot**, click slot 1, and then click slot 2 to join the bottom support to the base.



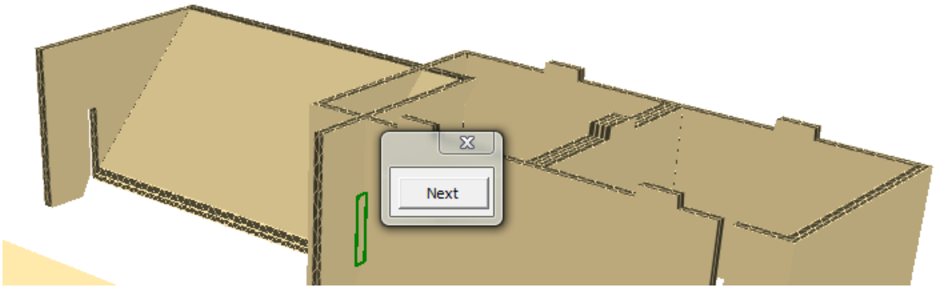
ArtiosCAD assembles the pieces.



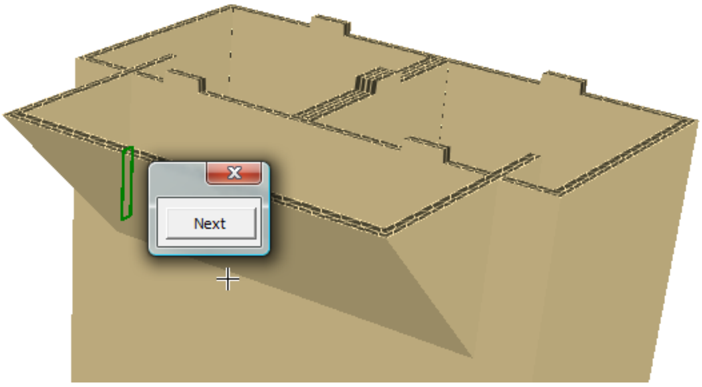
7. With **Snap Tab/Slot** still active, click slot 1, and then click slot 2 to join the top support to the base.



ArtiosCAD joins the top support to the base, but aligns it improperly.

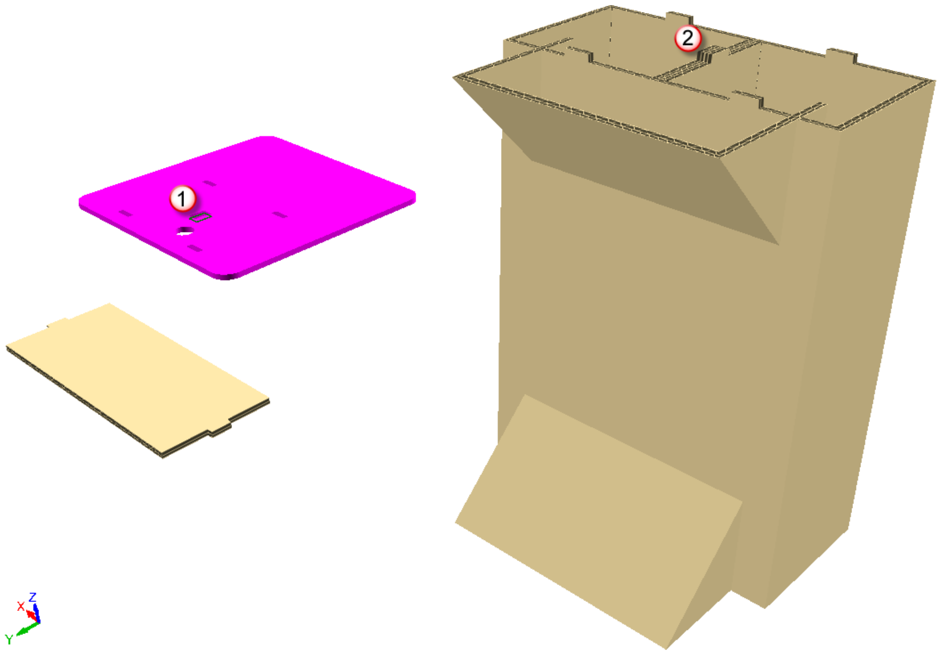


8. Click **Next** to choose the proper alignment, or use the left and right arrow keys on the keyboard.

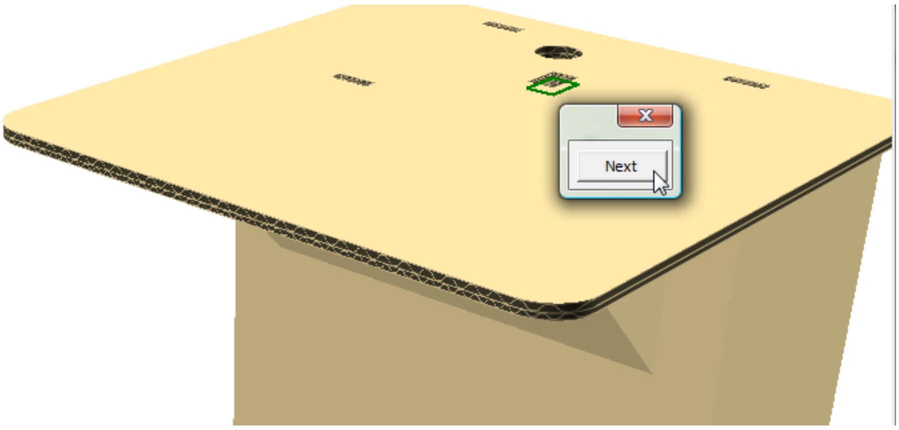


9. Click **X** to close the alignment selector.

10. With **Snap Tab/Slot** still active, click slot 1, and then click tab 2 to join the top to the base.



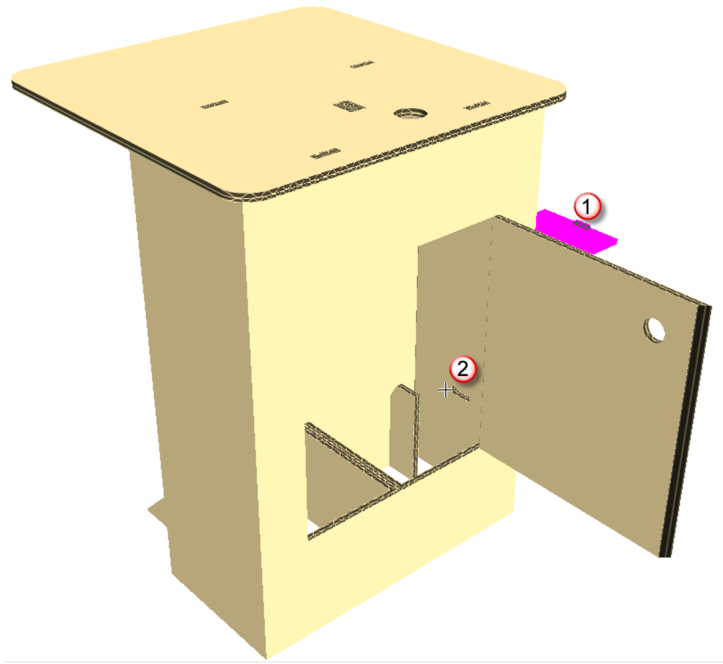
11. Click **Next** to choose the proper alignment, or use the left and right arrow keys on the keyboard.



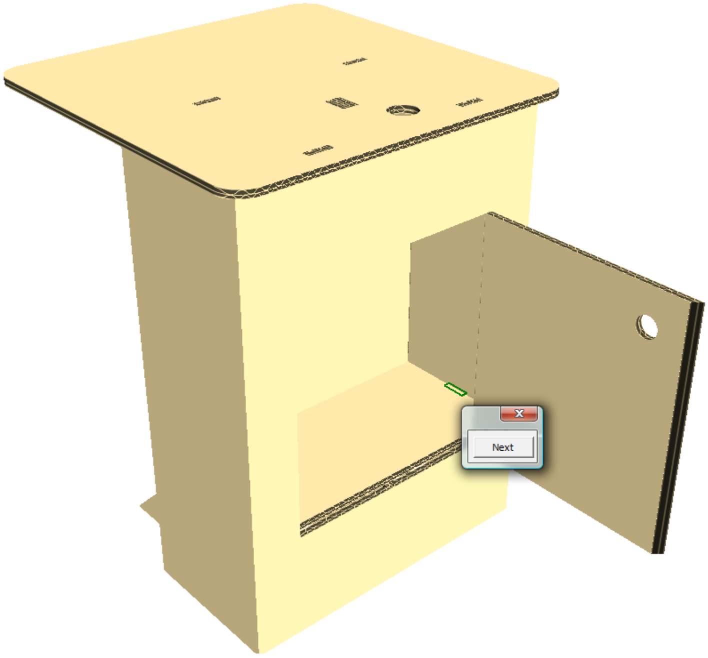
12. Click **X** to close the alignment selector.

13. Use **View Angle** to change the view to the other side.

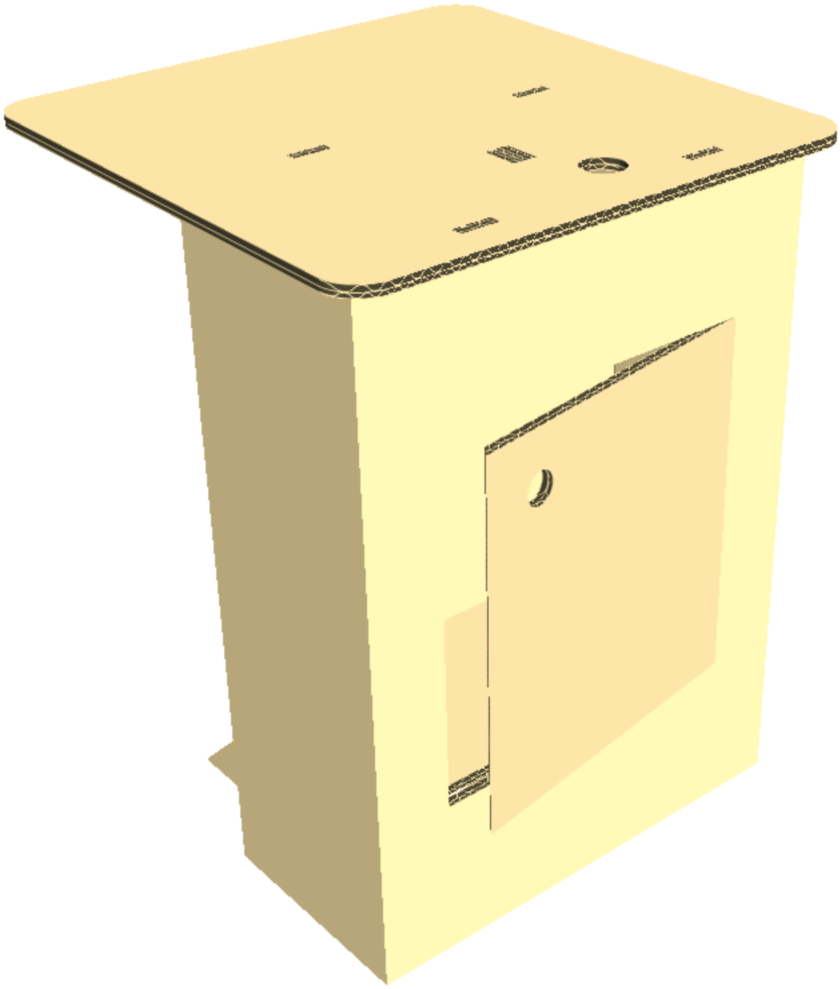
14. Click **Snap Tab/Slot**, click slot 1, and then click slot 2 to join the shelf to the base.



15. Click **X** to close the alignment selector.



16.  Use **Fold Angle** to shut the door on the base.

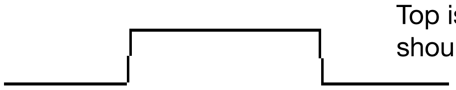
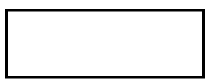


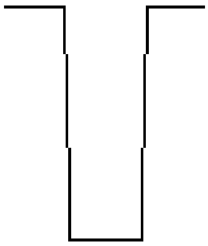
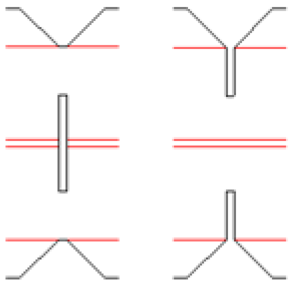
17. Save the completed assembly and Output it as desired.

Snap Tab/Slot tool notes

 The **Snap Tab/Slot** tool supports many shapes of tabs and slots.

The table below shows the types of tabs and slots ArtiosCAD supports.

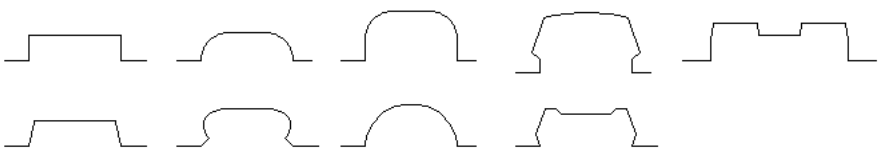
Tab /Slot	How it is recognized	How to use it
 Tab	Top is partially flat and shoulders are parallel	Fits in a hole slot or partial depth slot
 Hole slot	Rectangular hole of cut lines	Tab fits in either side

Tab /Slot	How it is recognized	How to use it
 Partial depth slot	Rectangular hole of partial cut lines	Tab fits on the inside
 Edge slot	Sides must be close to parallel	Fits in another edge slot or dual-edge slot
 Dual-edge slot	After folding angles at 90 degrees, two parallel edge slots act as one slot	Fits in another edge slot or dual-edge slot

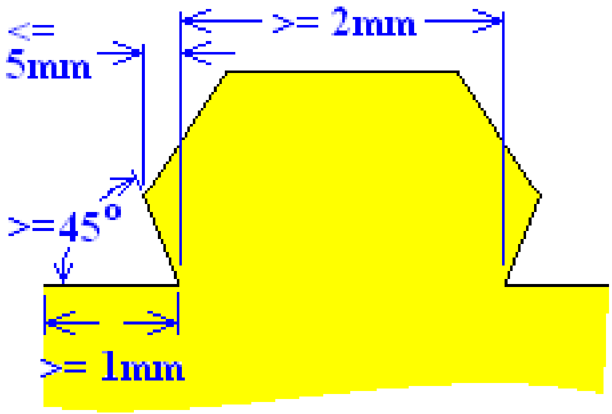
Note:

ArtiosCAD will warn you if the selected tabs and slots are substantially different sizes.

ArtiosCAD supports the following tab shapes:

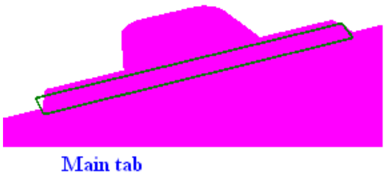


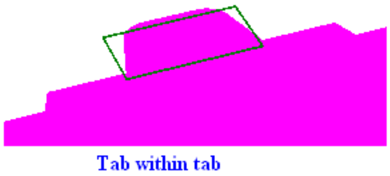
However, there are limits to the tab shape as shown:

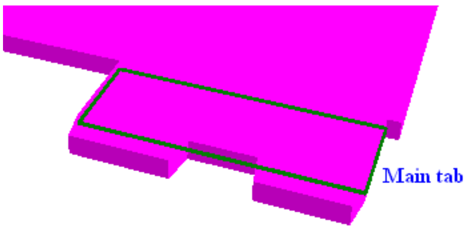


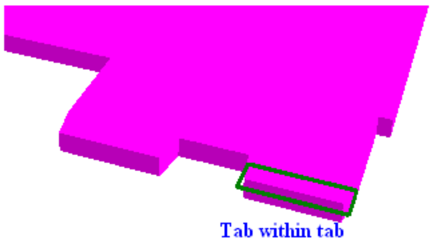
- The parallel lines before and after the tab must be at least 1mm long.
- The tab must be least 2mm wide.
- The start angle of the tab must be at least 45 degrees.
- The side of the tab cannot overhang the base by more than 5mm.

ArtiosCAD recognizes a tab within another tab in the following cases. You can choose either the main tab or the tab-within-a-tab.

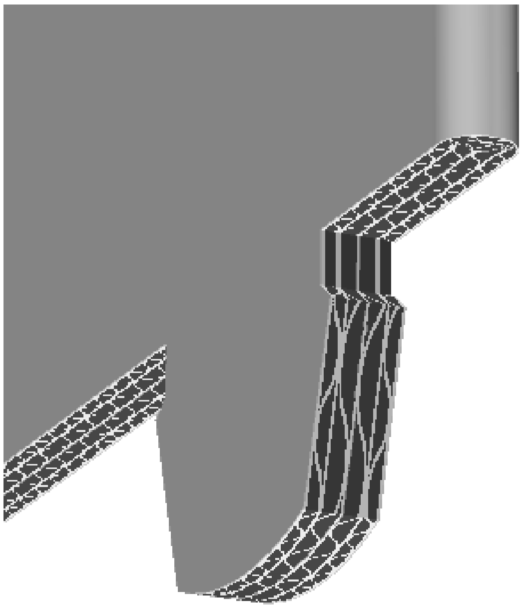




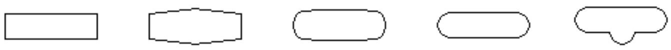




ArtiosCAD treats two tabs side by side as one tab.



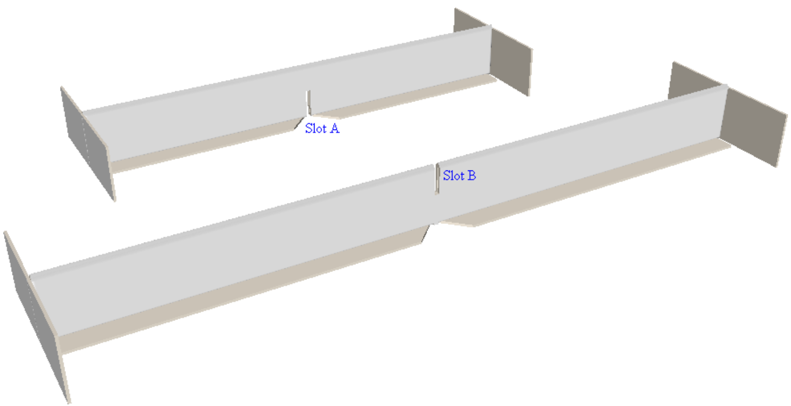
ArtiosCAD supports the following hole shapes for a tab to fit into:



Use the following considerations when designing a hole:

- Make the hole roughly rectangular.
- Make the hole with 4 to 12 lines.
- Make the hole longer than it is wide.
- Make the hole at least 2mm long.
- Make the sides of the hole approximately parallel.

The Snap Tab/Slot tool also supports paired slots in partitions:



Note:

The Snap Tab/Slot tool aligns slots and tabs in separate pieces of designs. It does not fold a tab into a slot in the same piece of a design.

Folding Retail-Ready Containers

Once you have designed your retail-ready package and made sure to set all the tearing lines' 3D properties correctly, convert it to 3D as usual.

Select Tear Away Part tool

 The **Select Tear Away Part** tool lets you select the part of a design that is separated from the main part of the design. Once it is selected, you can move or rotate it as desired.

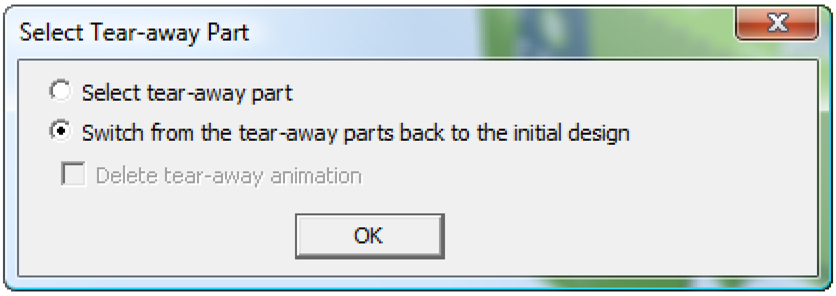
1.  Click **Select Tear Away Part** on the 3D Tools toolbar.
2. Click the tear-away part of the design. It turns magenta to show it is selected.

3. Change the position of the part as desired.

Rejoining the Parts

To rejoin the torn-away part to the base of the design as it was originally, do the following:

1.  Click **Select Tear Away Part**.
2. Click a part that was torn away.
3. In the Select Tear-away Part dialog box, choose **Switch from the tear-away parts back to the initial design**.



4. Click **OK**.

ArtiosCAD rejoins the parts.

Bend tool concepts

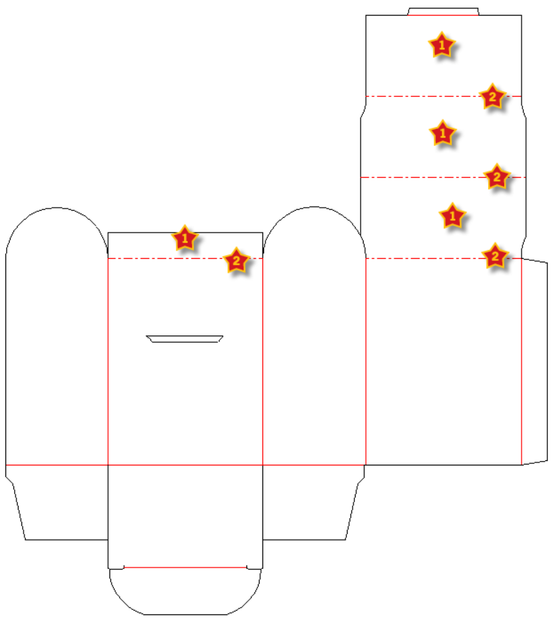
Bend panels differ from curved crease panels in that the panels themselves bend around something, not just the creases. Bend panels:

- Are defined by one or two bend lines, which in the default plotting style are red dot-dashed lines
- Have an angle changed by the **Bend** tool
- Do not have curved creases.

Bend lines divide the design into panels, and have fold angles in 3D the same as creases, but:

- They are not real creases
- Are not structural and can be hidden in the 3D Assist layer
- Do not have a crease bead
- Are not intended to be folded at a sharp angle.

In the design below, bend panels are marked 1 and bend lines are marked 2.



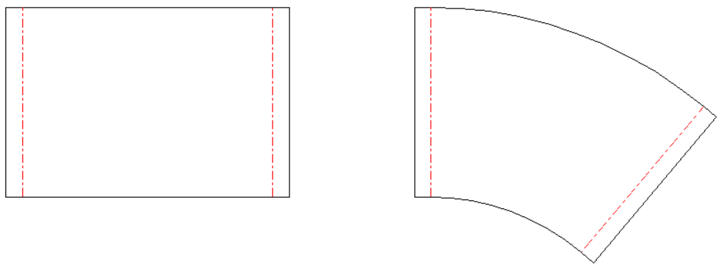
Types of bend panels

There are two types of bend panels: a cylindrical bend panel and a conical bend panel.

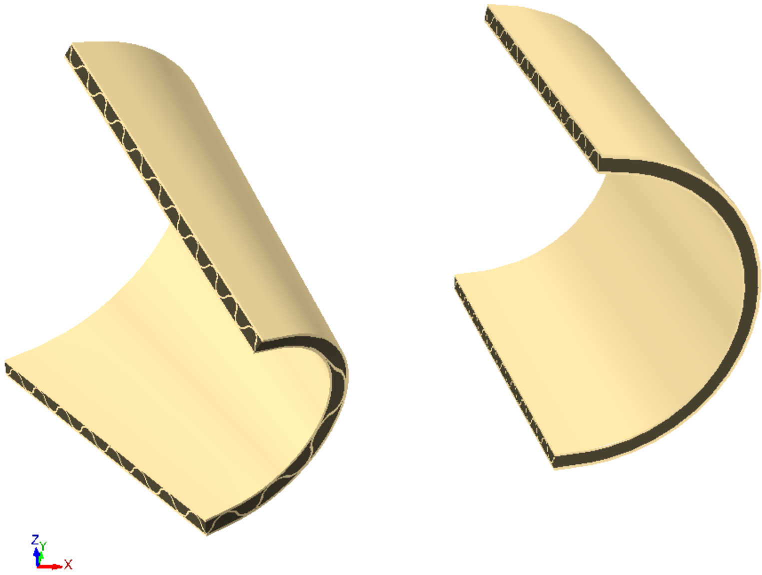
The bend panel is cylindrical if there is just one bend line or the bend lines are parallel.

The bend panel is conical if there are two bend lines that are not parallel. The bend lines should not meet.

In the example below, the left panel is a cylindrical bend, and the right panel is a conical bend.



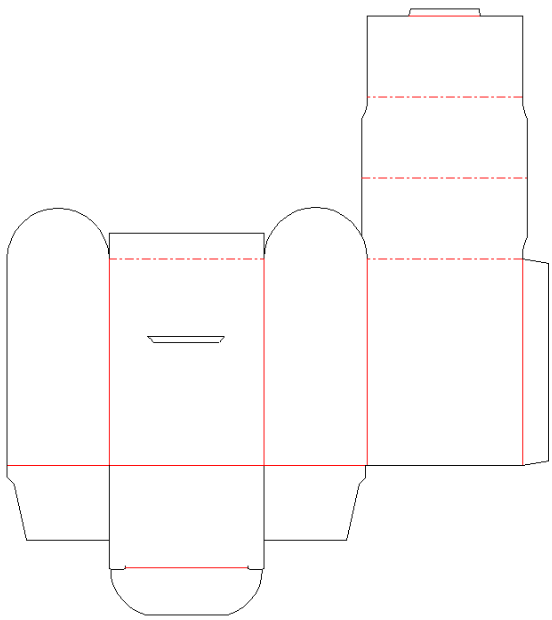
In 3D, they become:



Using the Bend tool in 3D

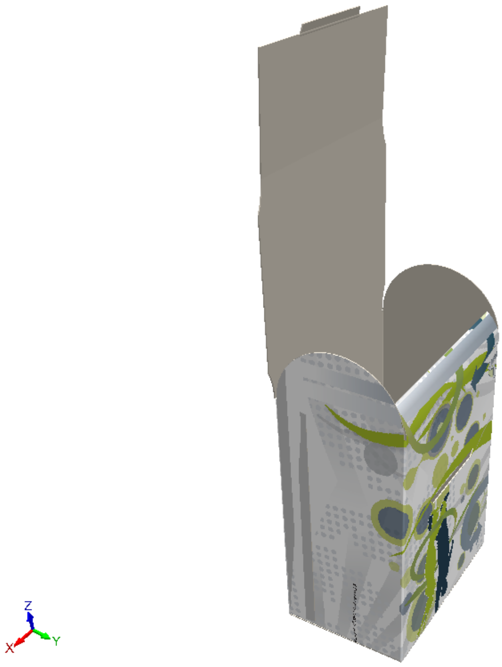
To use the Bend tool in 3D, do the following:

1. Create a single design or open an existing one with bend lines in its 3D Assist layer.

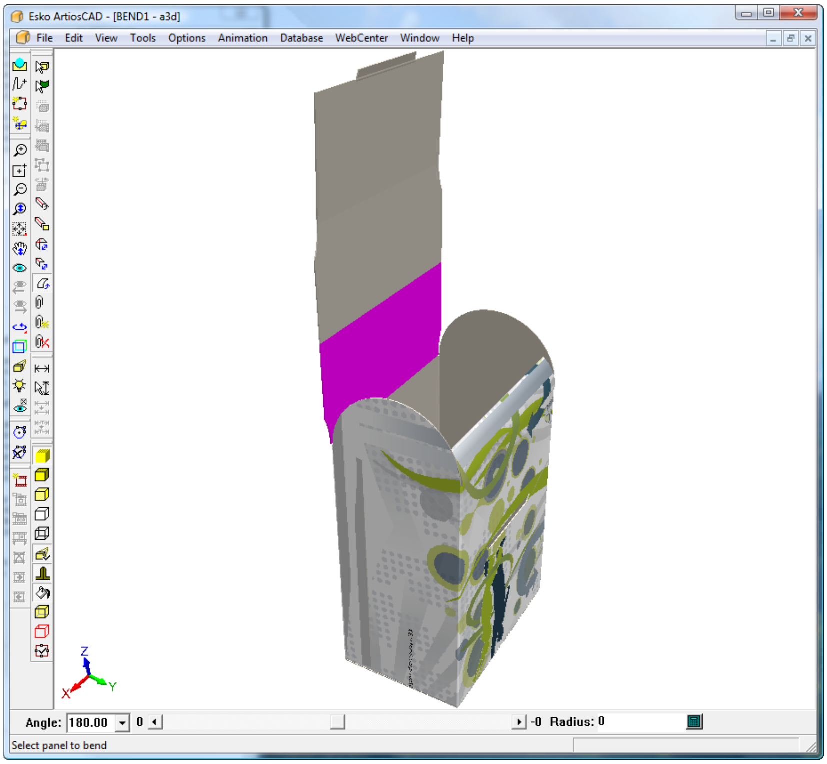


2.  Click **Convert to 3D** on the Tool Rack.

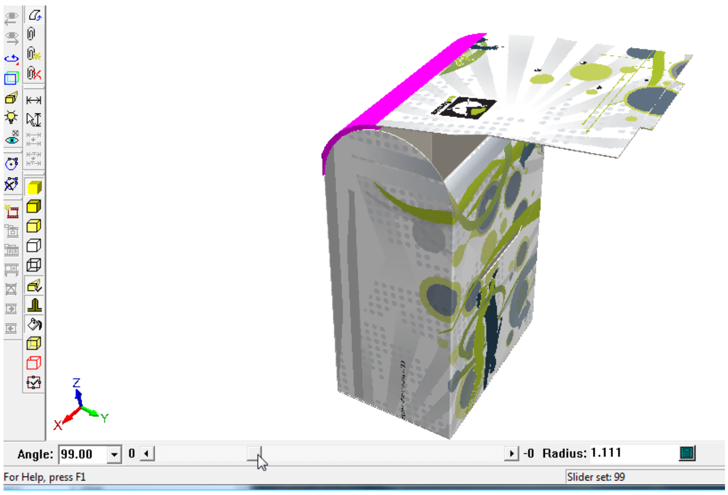
3. In the **Add Unfolded Design(s)** dialog box, click inside the base face and click **OK**.
4. Use the **Fold Angle** or **Fold All** tool to fold the creases in the design.



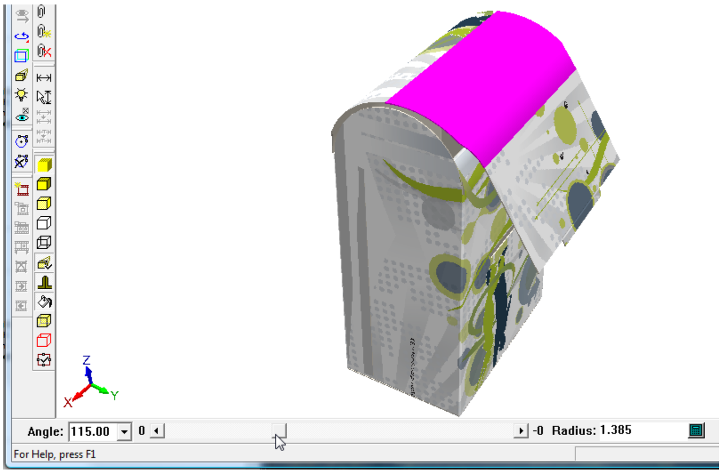
5. Click **Bend** and then click one of the bend panels.
6. The bend panel turns magenta to indicate it is selected and an **Angle** field and slider appear on the Status bar, along with a **Radius** field.



7. Change the **Angle** for the bend by either entering a value in the field, choosing a preset value from the drop-down list box, or by using the slider as shown below. You can also change the bend by entering a value in the **Radius** field.



8. When you are satisfied with the bend of the first panel, click the second panel and adjust its bend, and repeat for the remaining panel(s).



9. Shown below is the completed bend sequence.

Note:

A bend angle of 180 means it is flat. A positive bend angle has the print side on the outside of the bend, while a negative bend angle has the print side on the inside of the bend.

Glue/mate area concepts

The Glue/Mate area tools are designed to make it easier to connect designs together in 3D, and to indicate where glue will be applied to a carton as it is manufactured.

You define mate and target areas in Single Design and then when you convert to 3D, ArtiosCAD can automatically join the two areas, or you can join the areas manually. The rules for finding the matching area are:

- A mate/glue area mates with a target area. They must each be completely inside a panel.
- You can click either the mate or the target after starting the **Make Mate** tool.
- The mate/glue area must have the same name as the target, but the name may be blank.
- The mate/glue area and the target area must have identical lines, but they may be oriented differently or mirrored.

Rectangles are sufficient to define most mate areas, but for really specific needs, use an assymetric shape (such as a rectangle you moved a point on to have a diagonal side).

Mate types:

- *Dynamic mates* are on cartons based on 4-panel sleeves with flaps, or those on cartons made of two pieces that form 4-sided tubes. Once these types of mate areas are mated in 3D, the cartons behave as if the folds are connected, and adjusting fold angles on their creases affects the connected panels.
- *Static mates* glue two designs together without affecting fold angles. If you attach two designs outside to outside or inside to inside, the target should be a mirrored copy of the mate area. The smaller design moves to mate with the larger design.

For glue areas, you define them in Single Design the same way as you would a mate area (except specifying **Glue** as the **Type** in the **Mate Properties** dialog box) and then ArtiosCAD automatically shades that area in 3D when mate areas are turned on in the View Mode.

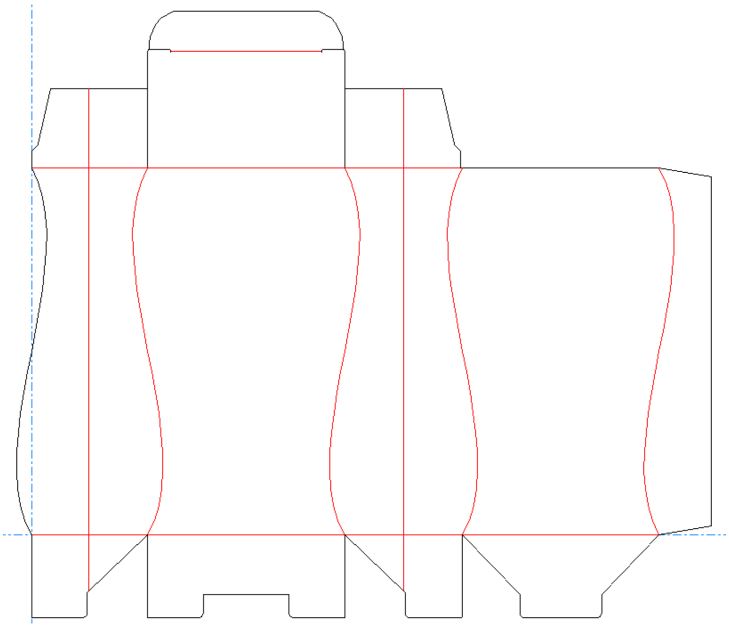
 If your 3D workspace has many pieces with mate and target areas defined, click **Mate All** to mate them all together with one click.

 To disconnect a mated area, click **Undo Mate**, and then click inside a green mated area. You may have to turn on mated areas in the View Mode beforehand.

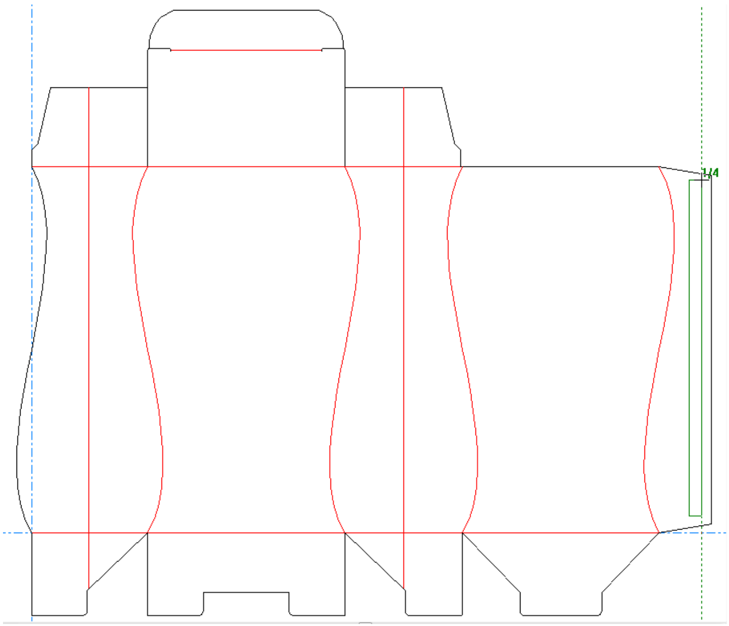
Defining Mate and Target areas

To define mate and target areas in a single design, follow this recommended workflow:

1. Create a new single design or open an existing one as desired.

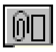


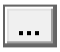
2. Use the **Rectangle** tool to draw the Mate area.

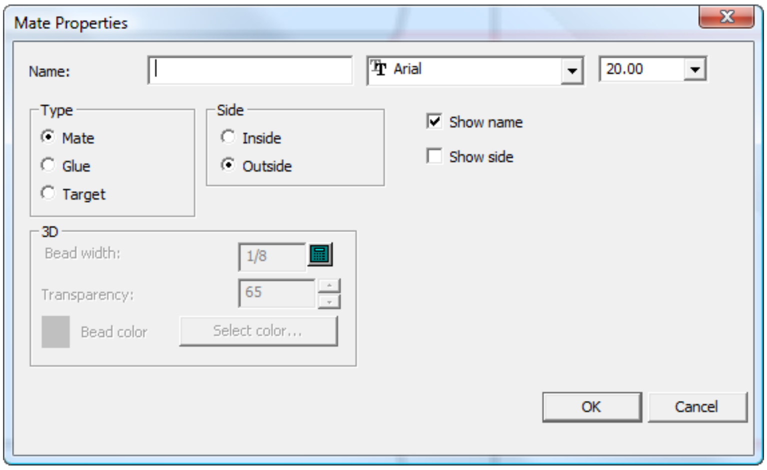


3. Use the **Select** tool to select the lines comprising the rectangle.

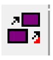
4. Click **Define Mate Area**. The tool starts and the following controls appear on the Status bar.

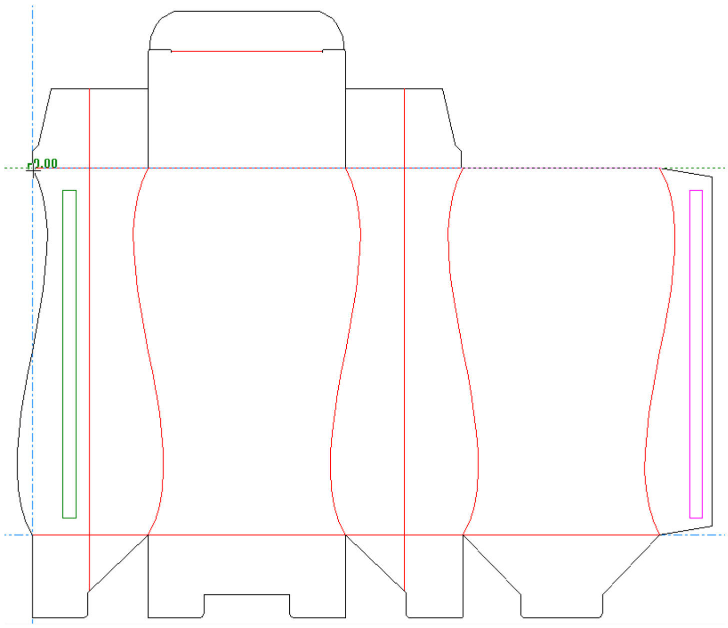
5.  Click **Make Mate** on the Status bar. The rectangle changes color and is moved to the 3D Assist layer automatically.

6.  Click **Properties**.

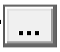


7. In the **Mate Properties** dialog box, assign a name such as `area1` in the **Name:** field. The name of each mate and its target must be the same, but the name can be blank. Set the font and size for the label as desired. In the **Type** group, select **Mate**. In the **Side** group, select **Outside**. Select **Show name** and **Show side** as desired. Click **OK** to close the **Mate Properties** dialog box.

8.  Use the **Copy** tool to copy the rectangle to the target panel.



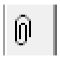
9.  Once the copy is placed, click **Define Mate Area**.

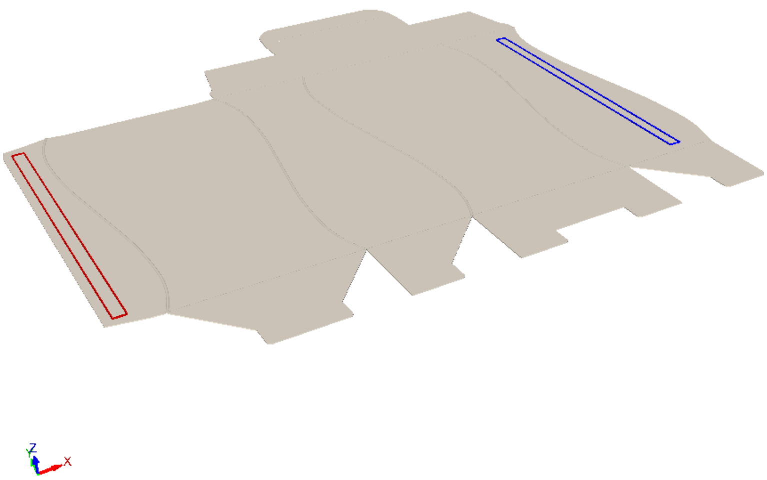
10.  Click **Mate Properties**.
11. In the **Mate Properties** dialog box, set the Type to be **Target**, set the Name to be the same as the name you specified for the Mate (such as `area1`), set the side to be the **Inside**, set the font and size fields as desired, and check or clear **Show name** and **Show side** as desired.
12. Click **OK** to close the Mate Properties dialog box.
13. Save the design.
14. The design now has mate and target areas defined and can be folded in 3D either automatically or manually.

Folding a design with a mate and target area

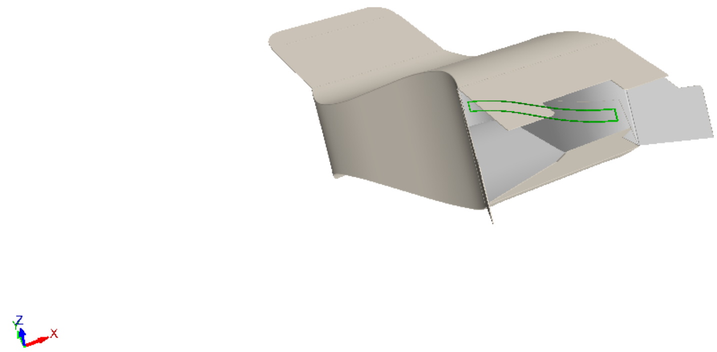
Once you have a design with a mate and target areas defined, convert it to 3D. For the purposes of these instructions, make sure to deselect **Use previous fold angles** if it is selected when converting to 3D.

Once the design is in 3D, do the following:

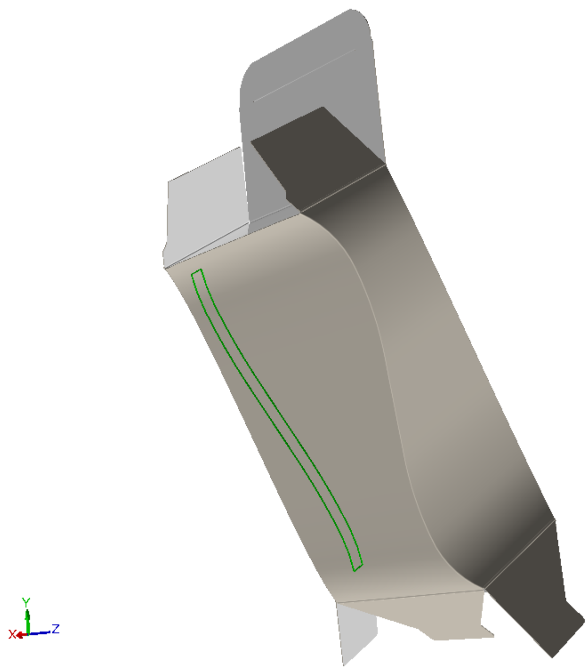
1.  Click **Mate**. The mate and the target areas on the design become visible; the blue area is the mate and the red area is the target.



2. Click inside either the mate (blue) or the target (red) area.
3. ArtiosCAD joins the mate and target areas and turns them green. Since this is a dynamic mate due to the carton being based on a 4-panel sleeve, the panels behave as if they are connected to each other. When you adjust fold angles affecting one of the joined panels, the other panels move as well, just as they do in real life. Note that there are no limits on this movement and you may unexpectedly encounter less than optimal results.



4. Change the View Angle and continue working on the design as desired.

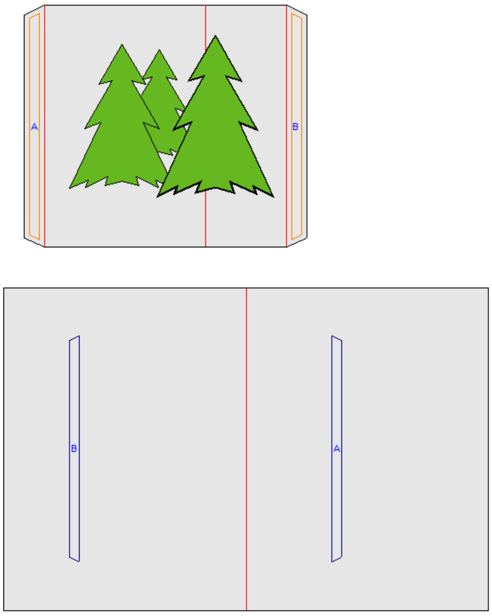


Attaching two designs in 3D using Mate and Target areas

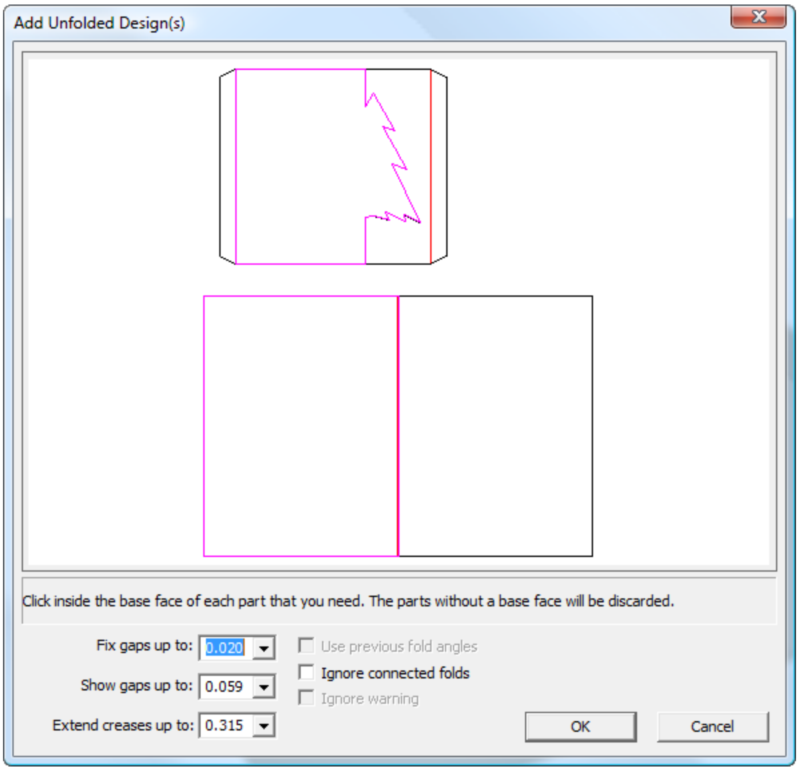
This is an example of another dynamic mate. Attaching these two pieces together forms a four-sided sleeve.

1. Create the two pieces in Single Design that will be mated together in 3D.

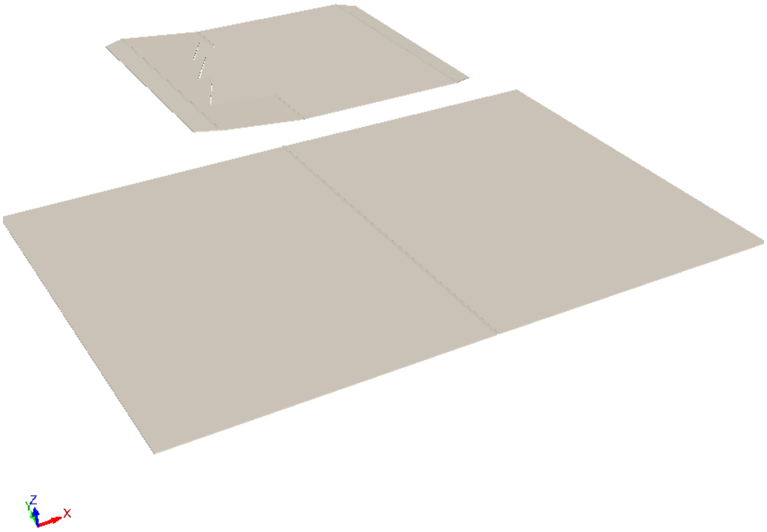
2. As there are two attachment points, create two mate areas and copy them to two target areas.



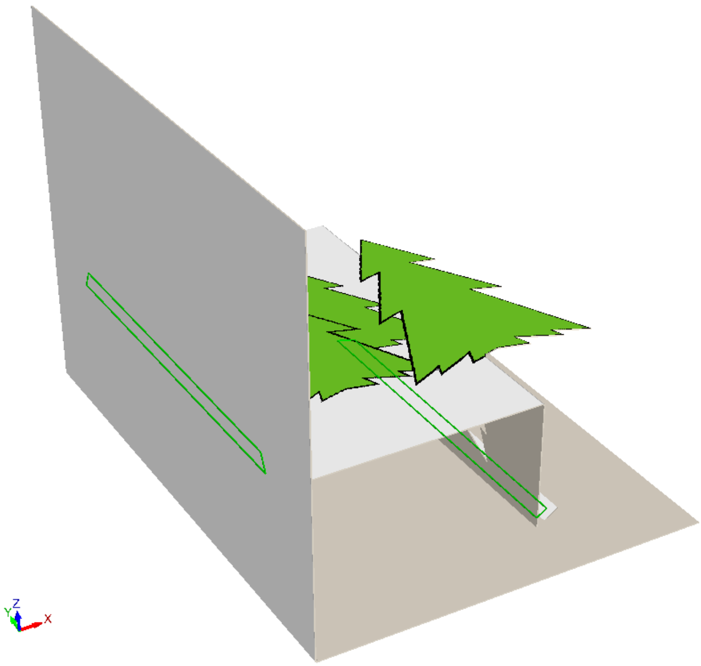
3. Click **Convert to 3D** on the Tool Rack.
4. In the **Add Unfolded Design(s)** dialog box, click inside each base face and click **OK**.



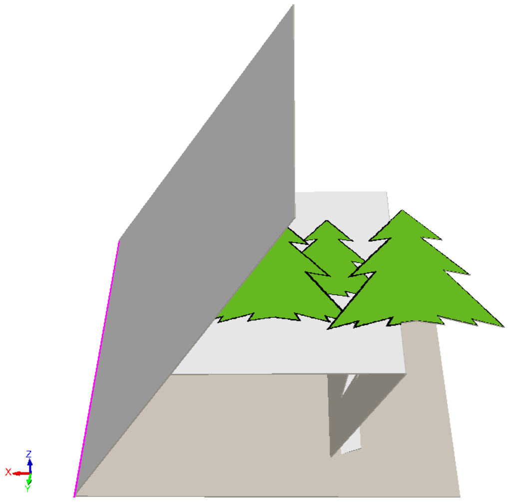
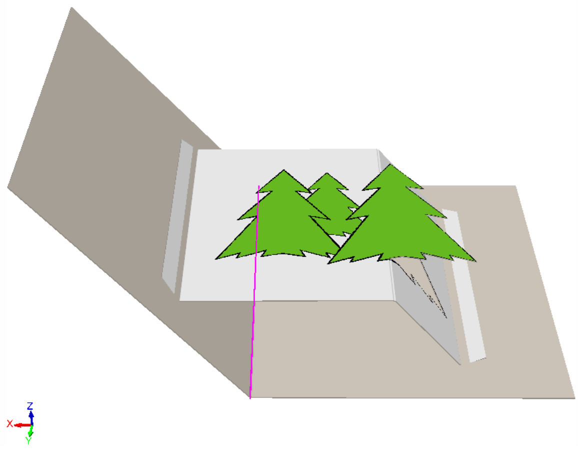
5. The two designs are brought into 3D.



6.  Click **Mate All** in 3D. The two pieces join together at their mate areas; turn on **Mate Areas** in the View Mode and they both will be green to indicate that the mates and targets joined successfully.



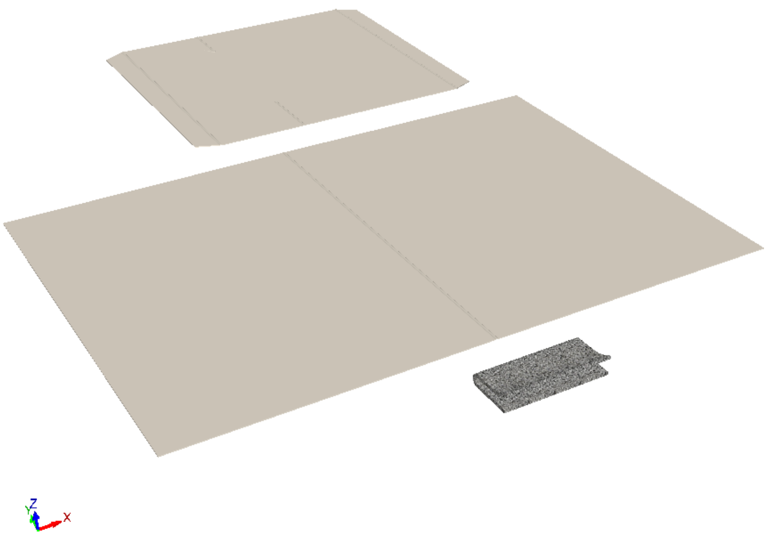
7. To see how the panels move together in this dynamic mate, change the fold angle of the crease folding the card in half.

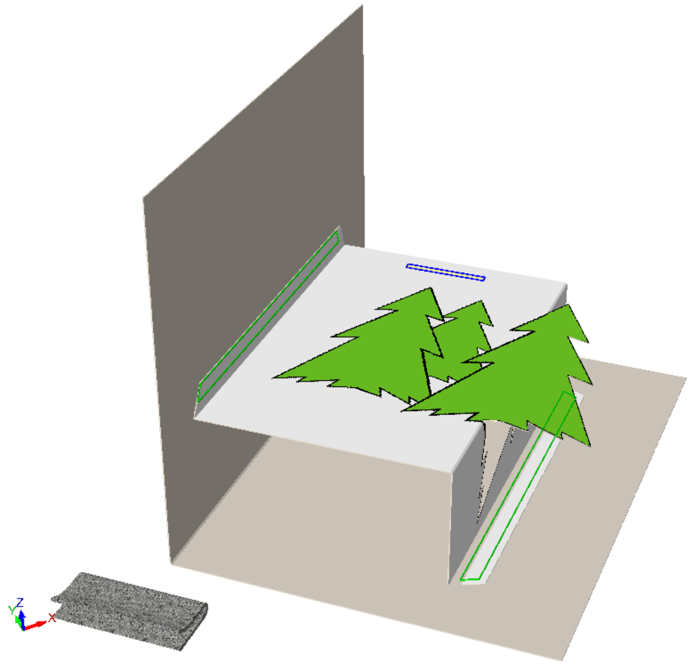
Attaching a solid to a carton in 3D

Solids cannot have mate or target areas defined, yet you can attach them to cartons that do by doing the following:

1. Bring all the pieces into 3D.



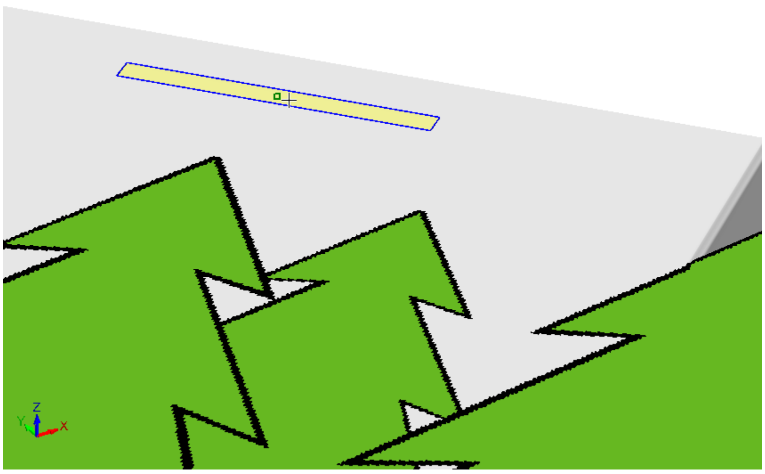
2. Mate the card pieces together, change the View Angle, and turn on **Show mate areas** in the VIew Mode.



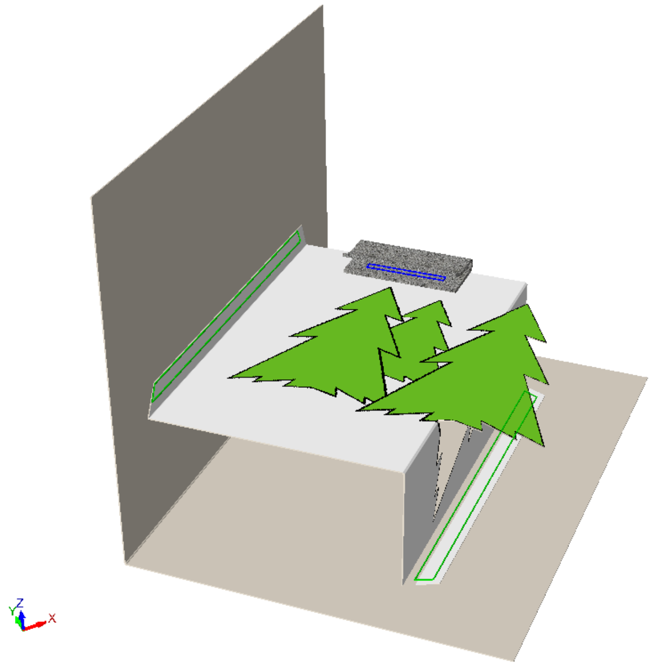
3. Click **Extend** and click the solid to turn on its extend points.

4. Click **Select Designs** and select the solid.

5. Click **Move Point to Point** and move the solid to within 5mm of the mate area. In this example, choose a pick-up point in the middle of the bottom of the clip. When you get close to the mate area, a put-down point appears in its center, as seen in the close-up below.



6. Click the put-down point. You may need to use the Rotate tools to get everything aligned to your satisfaction. The finished example of the card with a clip inside of it is shown below.



Undo and Redo

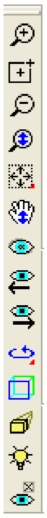
If you unintentionally move a design in a way you don't like, or fold a flap too much, use the **Undo** command on the Edit menu to reverse your incorrect action. Use **Redo** to redo the action undone. **Undo** and **Redo** work on the following tools:

Table: Tools affected by Undo and Redo

Affected	Not Affected
Moves and rotations	All tools on the 3D View and View Mode toolbars
Fold angles, Fold to Meet	Select tools
Copy, addition, deletion of designs	Light Source
Group, Ungroup	Material Properties
Dimension tools	Extend, Clear Extend
Intersect Design	Animation tools
Cross Section	Bounding Box
Run a Standard	

Changing how you view the workspace

Use the tools on the 3D View toolbar to change how you view the design in 3D.



Zoom Rectangle tool

 The Zoom Rectangle tool lets you view a portion of your workspace at an increased scale.

1. Click the **Zoom Rectangle** tool.
2. Move the mouse cursor to a corner of the area to zoom in upon, hold down the mouse button, and drag to the diagonally opposite corner.

3. Your screen changes to show a close-up view of the area you selected.

Center-Point Zoom tool

 The **Center-Point Zoom** tool, when clicked, prompts you to click the center of an area to be zoomed in upon, and then to drag to the corner of that same area. Release the mouse button to perform the zoom.

Zoom Out tool

 Click the **Zoom Out** tool to view your workspace at a smaller scale.

Zoom In/Out tool

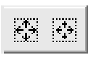
 Use the **Zoom In/Out** tool to change the zoom level by dragging the cursor in the workspace. To use this tool, do the following:

1.  Click the **Zoom In/Out** tool.
2. Click inside the drawing window of ArtiosCAD.
3. Click and drag the mouse up to zoom in or down to zoom out. Alternately, use the up and down arrow keys to incrementally zoom in and out.

This tool remains active until a different tool is clicked.

Scale to Fit tools

 Click the **Scale to Fit** tool to show the entire workspace at the largest scale possible within the borders of the window. When held down, it activates the Scale to Fit tools flyout toolbar.



 Click the **Scale to Fit with Border** tool to put a border around the scaled-to-fit view. The size of the border is configured in **Options** > **Defaults** > **Startup defaults** > **View tools options**.

Pan/Zoom tool

 The **Pan/Zoom** tool lets you grab a location on the screen and drag it as desired in Pan mode, or zooms in and out according to the direction you drag (up or down) when in Zoom mode. To use it in Pan mode, click it, click and hold a location on the screen, and drag to the new location. Click

the right mouse button or press ESC to exit the tool and return to the previous tool used. The cursor looks like a clenched hand while dragging.

Note: If you use the Pan/Zoom tool in high graphics mode, you must refresh the screen by pressing F2 to re-render the graphics.

To use the Pan/Zoom tool in Zoom mode, click it, click **Zoom** on the Status bar, position the cursor in the desired area, and then drag the cursor up to zoom in or down to zoom out. This tool does not interrupt the tool being used before it was clicked. Click the right mouse button or press ESC to exit the tool and return to the previous tool used.

Press TAB on the keyboard to change between Pan and Zoom modes without clicking their respective option buttons.

To exit the Pan/Zoom tool in either mode, either click the mouse button, press esc on the keyboard, or activate another tool. The previously-used tool will resume operation.

If you have a three-button USB mouse with a scroll wheel, using the scroll wheel at any time zooms in or zooms out as if you were using the Pan/Zoom tool in Zoom mode. Holding the middle mouse button down and dragging the cursor pans the view as if you were using the Pan/Zoom tool in Pan mode.

View Angle tool

Change the view angle, elevation angle, and roll angle at which you view this design by using the **View Angle** tool. You can set the values using four methods:

1. Enter a degree measurement in the **Angle**, **Elevation**, or **Roll Angle** field and press enter;
2. Select a predefined angle from a list box;
3. Drag a slider to set the value. As you drag the slider, the view changes accordingly.
4. Click and drag inside the drawing window. All three View Angle elements change accordingly. Use the arrow keys to rotate incrementally, or hold down CTRL to change the roll angle with the arrow keys.

Note that this tool does not change the physical location of the designs in the workspace. It only changes how you see the workspace.

Some of the sliders may not be visible if the ArtiosCAD window is not wide enough. Make the ArtiosCAD window wider to show them.

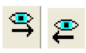
The View Angle tool can interrupt other tools. Click the tool and use it as desired, then either click the right mouse button or press ESC to return to the previously-used tool. This is especially useful when moving or aligning objects as you can change the view to reveal more pick-up or put-down points.

Shown below is a design before the view angle is changed.

Shown below is the design after the view angle has been changed.

You may need to click **Scale to Fit** to reset the view of the design.

Next and Previous View tools

 The **Next View** and **Previous View** tools let you switch between views once you have used the View Angle tool to change the view. ArtiosCAD remembers eight previous views by default, but you can configure ArtiosCAD to remember up to 100 views in **Options** > **Defaults** > **Startup defaults** > **View tools options**. Click **Previous View** to go back to previous views from the most recent view and click **Next View** to go forward to the most recent view from previous views. These two tools interrupt whatever tool was previously active; when the view changes, that tool becomes active again.

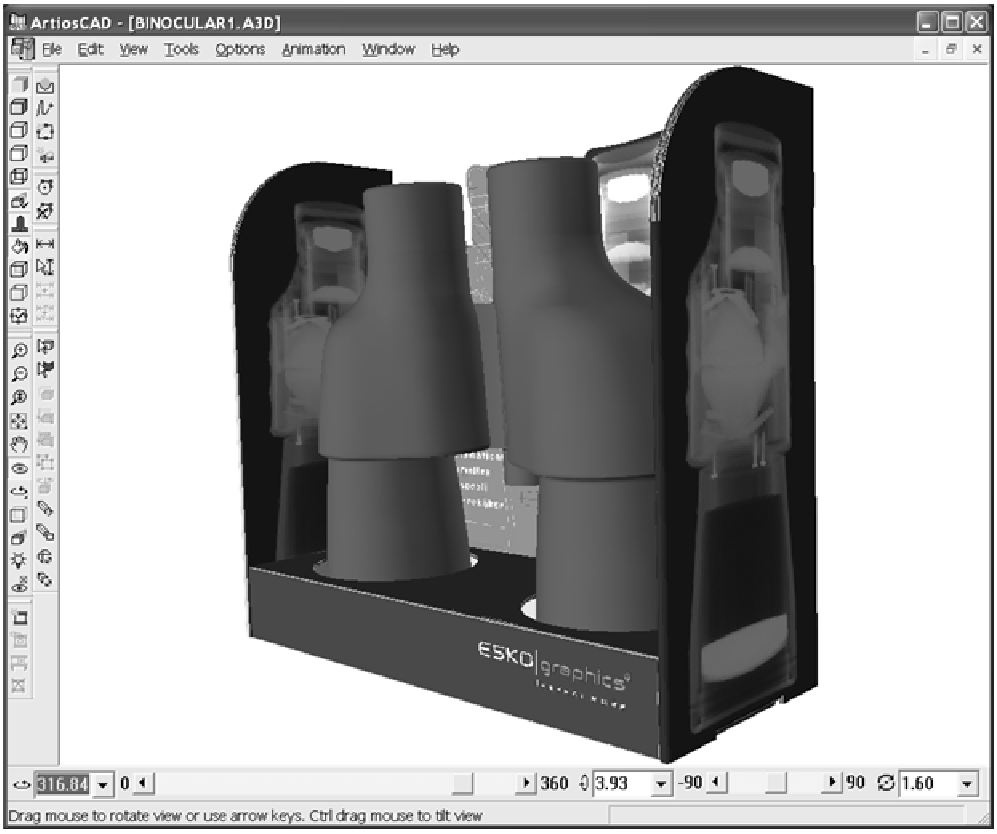
These tools are designed to work by allowing you to stack several different views, activate a tool such as the **Move Point to Point** tool, choose a pick-up point in a previous view (or the current view) and then choose a put-down point in the current view (or a previous view).

Rotate View tools

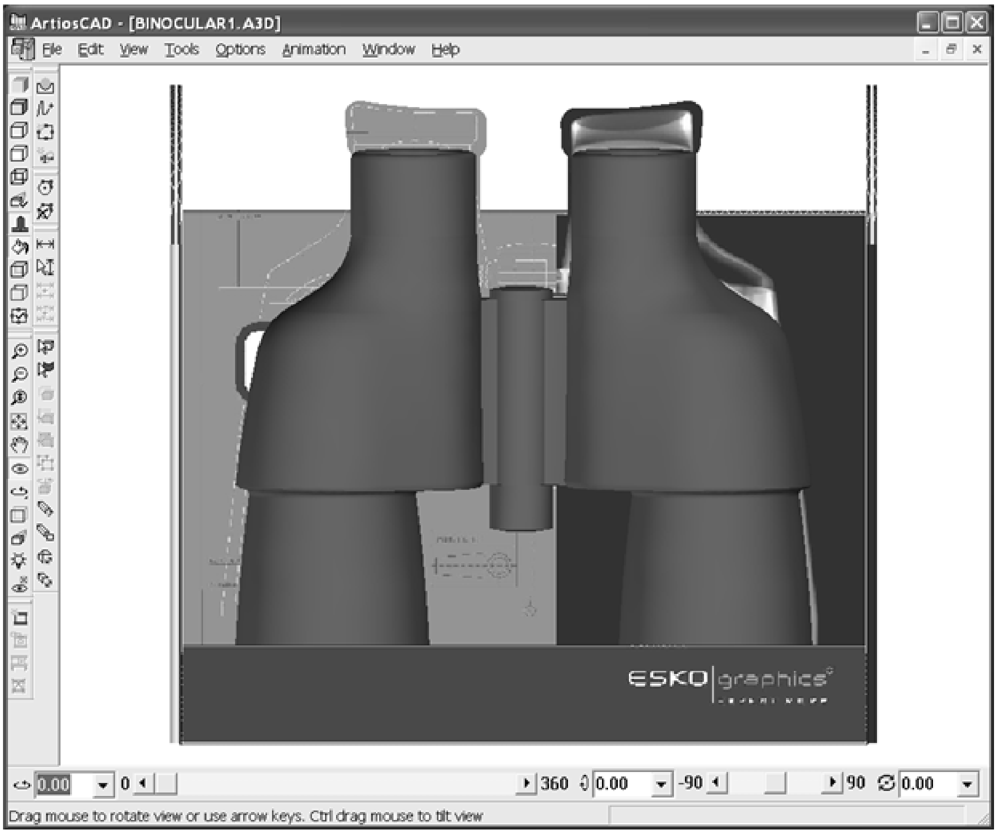
 The **Rotate Right** and **Rotate Down** tools on the Rotate View tools flyout toolbar rotate the view by 90 degrees in the specified direction.

Orthogonal View tool

 The **Orthogonal View** tool snaps the view to the nearest multiple of 90 degrees on all three view angles. This tool turns perspective off. The picture below shows the view before clicking the tool. Note the values in the view angle fields on the Status bar.



Shown below is the same workspace after clicking the **Orthogonal View** tool.



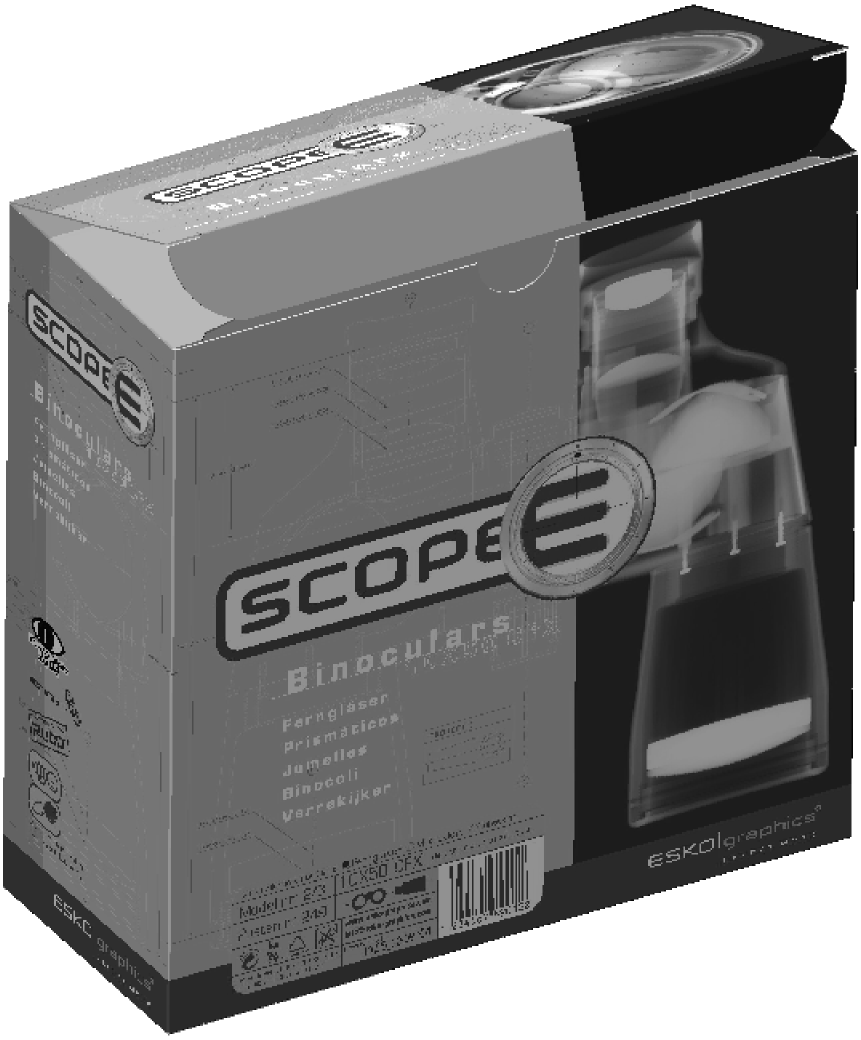
Note: Boards drawn thinner than 1 pixel may not appear in all orthogonal views when using OpenGL as the rendering method. Change the board thickness or view scale if necessary.

Perspective tool

 *Perspective* is the technique of making parallel lines meet at a point on the horizon in order to make the scene look more realistic. Change the amount or use of perspective using the **Perspective** tool.

1.  Click the **Perspective** tool.
2. To use less perspective, drag the slider to the left. To use more perspective, drag the slider to the right.
3. To toggle the use of perspective, select or clear the **Perspective** checkbox in the Status bar. You can also turn Perspective on or off using the icon on the View Mode toolbar or through the checkbox in the View Mode dialog box.

Shown below is a design using no perspective.

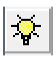


Shown below is the same design using normal perspective.

Shown below is the box with maximum perspective.

Lighting

Light Source tool

 The **Light Source** tool lets you add and remove light sources, and also change their positions. The light sources control the shading of the graphics and colors in the workspace. This tool is unavailable in Wireframe view mode. When the tool is active, the Status bar changes to look like the picture below.

If your ArtiosCAD screen is more than 70% of your screen width, the **Brightness** control appears. It affects all the lights. To change just one light, double-click it and use the vertical color brightness slider on the right.

When the tool is started for the first time, three lights appear in the workspace, one in front of the design, one in back of the design, and one above the design. The fill color shows their current color.

The active light has a magenta outline In the example above, the light bulb near the bottle is the current one. Click a light to select it.

 To add a light, click the **Add Light** tool on the Status bar and then click to set the position of the new light. The new light becomes the active light. Each workspace can have up to eight lights.

 To remove a light, click the desired light to select it and then click the **Delete Light** tool on the Status bar. You cannot delete the last light.

To change the position of a light, click it and drag it to its new position, or use the controls on the Status bar. When dragging a light with the mouse, there is a front position and a back position that correspond with the current cursor position. Drag the light to the left or right edge of the workspace to switch between front and back positions.

To change the color of a light, double-click it, and choose the new color from the standard Windows color selection dialog box. Before you change the color, consider adding the current color to the list of custom colors so that you may revert to it if desired. The light's custom color is saved with the workspace, but is not saved in the color palette.

Shown below is the same workspace, with the first picture showing a light source beneath and behind the designs, and the second picture showing two additional light sources above and in front of the designs.

Ambient light

Ambient light is the default light that suffuses a workspace that is not provided by a light source. To change it, click **View** > **Ambient light**, and choose the new color. Before you change the color, consider adding the current color to the list of custom colors so that you may revert to it if desired. The custom color is saved with the workspace, but is not saved in the color palette.

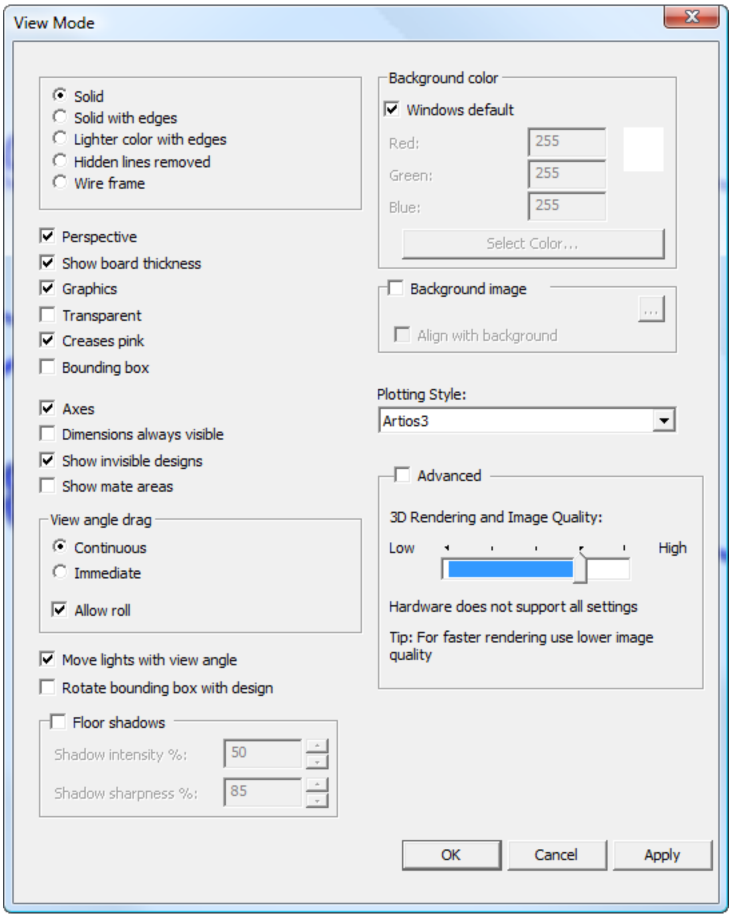
Ambient light has low contrast. The light bulbs of the **Light Source** tool have much more contrast.

Status bar

The status bar displays various messages and tool controls. When you position the mouse pointer over a tool button, for example, a short description of the tool appears at the left end of the status bar. The middle portion of the status bar returns a status message about the action just completed. For example, if you click the Move Designs or Rotate Designs tool, you will be prompted at the left end of the status bar to select a line and then an axis. After you select a line or axis, the right portion of the bar shows what was selected.

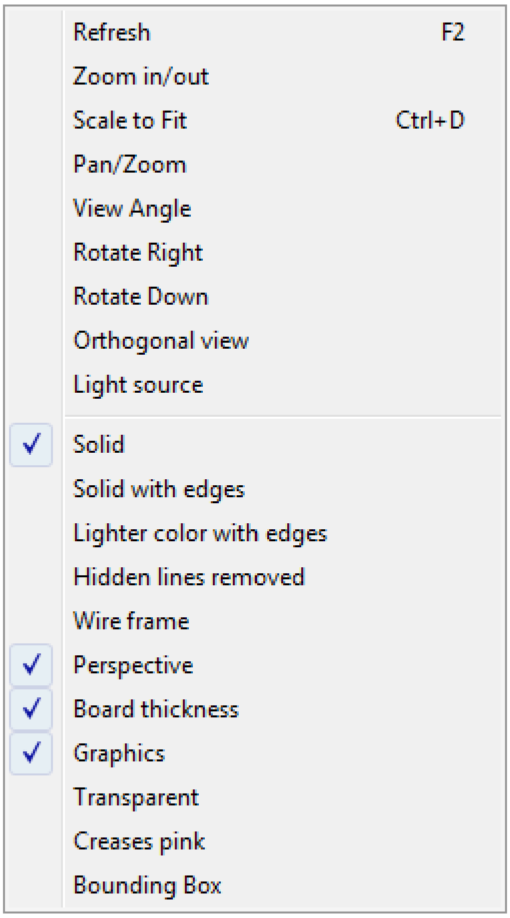
Setting the View mode

 The View mode controls many aspects of how designs are viewed in 3D. Click **View Mode** on the View menu or click the button on the 3D View toolbar.



Many of the option buttons and checkboxes on the left side of the dialog box are duplicated by tools on the View Mode toolbar. The topmost or leftmost tool is **Solid**, the next is **Solid with edges**, and so on.

Many of the same commands are also available on the context menu when the cursor is not over an object and the right mouse button is clicked:

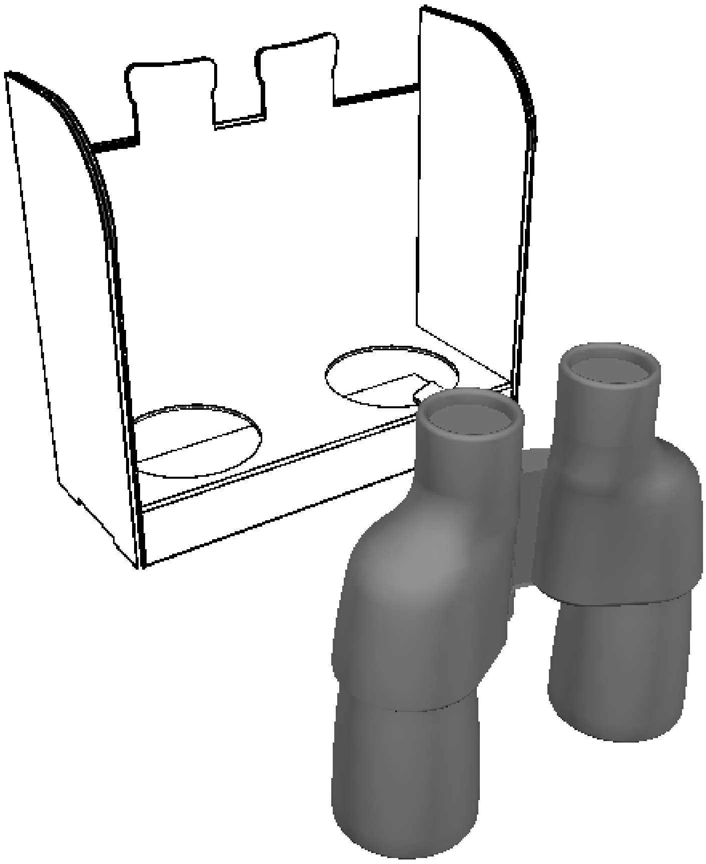


 **Solid** displays designs and solids as if they were photographed.

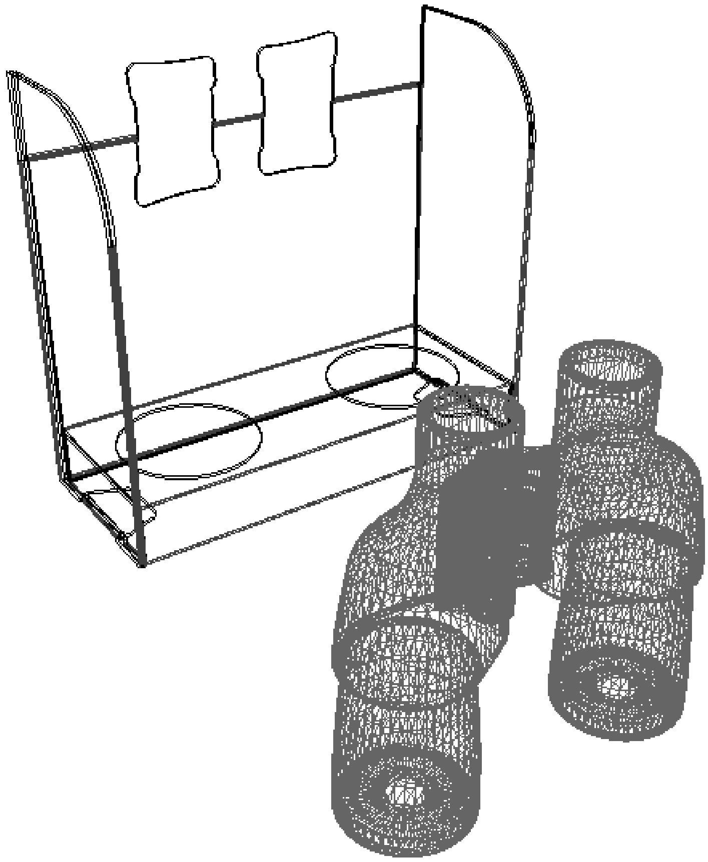
Solid with edges draws black lines on the edges of the designs. The colors of the lines are set by the plotting style. If the design is very complex, you may receive a warning about rendering the design, and ArtiosCAD will prompt for a confirmation.

Lighter color with edges enhances the contrast by lightening the color of the designs. If the design is very complex, you may receive a warning about rendering the design, and ArtiosCAD will prompt for a confirmation.

Hidden lines removed turns all designs white with black edges and no graphics. Solids in the workspace are displayed normally unless Graphics is turned off. If the design is very complex, you may receive a warning about rendering the design, and ArtiosCAD will prompt for a confirmation.



Wire frame shows outlines of the designs and solids in the workspace. Design lines are drawn according to the plotting style. The color used for the wire frame for a solid is defined by the color of the solid, except that if the solid is white, the wire frame is gray.



Perspective controls whether parallel lines meet at a point on an imaginary horizon.

Show board thickness shows the board thickness. **Graphics** must also be turned on to see corrugations.

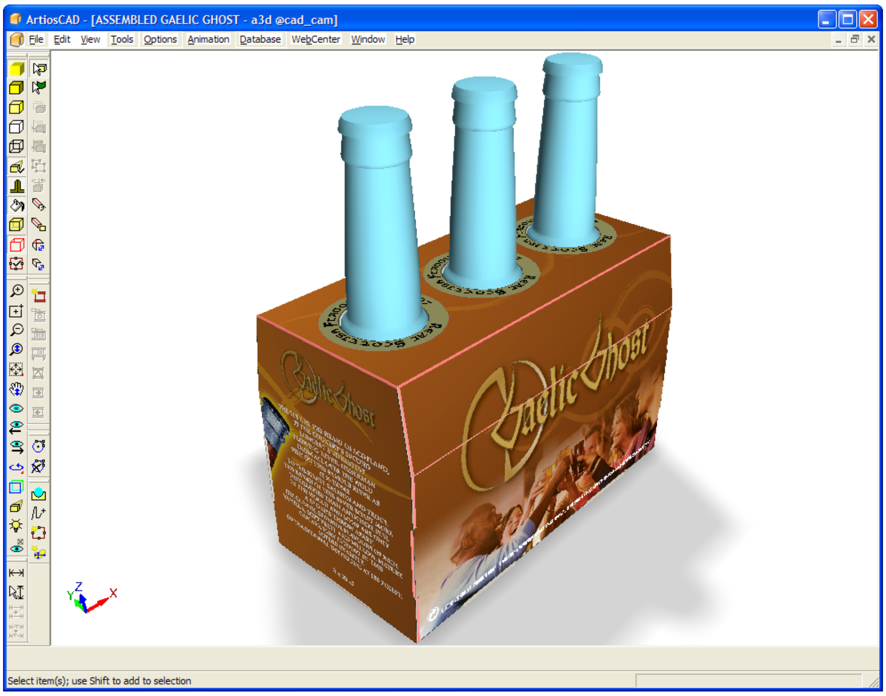
Graphics shows or hides graphics on designs and solids. It also controls the display of corrugation on board edges if **Show Board Thickness** is turned on.

Transparent turns the objects in the workspace partially transparent so that formerly hidden portions become visible. The appearance of transparent objects depends on the order in which ArtiosCAD draws them.

Creases pink shows creases in pink, perfs in blue, and partial cuts in purple for easier viewing. When this option is turned off, creases, perfs, and partial cuts are the same color as the board.

Bounding box turns the bounding box around all objects on or off. The bounding box provides points for picking up and putting down.

Axes controls whether or not ArtiosCAD displays the X, Y, and Z axes in the lower left corner of the design window. The X axis is red, the Y axis is green, and the Z axis is blue. The axes move appropriately as you change the view angle.



Dimensions always visible controls if dimensions hidden behind objects (such as boxes) are shown or not. The checkbox is off by default. JPG and PNG exports use this setting; VRML export does not as dimensions are not exported to VRML.

Show invisible designs controls whether or not objects set to be invisible are shown with 90% transparency or not shown at all.

Show mate areas turns on or off the rectangles that indicate areas that connect to each other automatically.

In the **View angle drag** group, **Continuous** describes the view angle continuing to change even when the mouse is not moving. **Immediate** stops changing the view angle when the mouse stops moving.

Allow roll turns rolling with the mouse on and off.

Move lights with view angle allows you to disconnect the lights from the view angle. It is checked by default - the lights move with the view angle. Clear this checkbox to not have the lights move with the view angle.

Rotate bounding box with design causes bounding boxes to rotate with a design when you rotate it. This makes moving and aligning boxes at an angle easier.

 **Floor shadows** turns the floor shadow on and off. **Shadow intensity %** controls how dark the shadow is; use a lower number for a lighter shadow and a higher number for a darker shadow. **Shadow sharpness %** controls the fuzziness at the edge of the shadow. Use a lower number for a more diffuse edge and a higher number for a sharper edge.

Note that the shadow uses the light sources defined in the workspace, not the light pictured in an optional background image.

When moving or rotating designs, the floor shadows remains static during the drag, and are updated when the tool finishes.

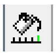
When using a floor shadow, the light source angles are limited to at least 60 degrees so that the shadows do not get too long. If the light source is at less than 0 degrees, and no other light sources are defined, no shadows are made.

Turning on the floor shadow turns on perspective. Turning off perspective turns off the floor shadow.

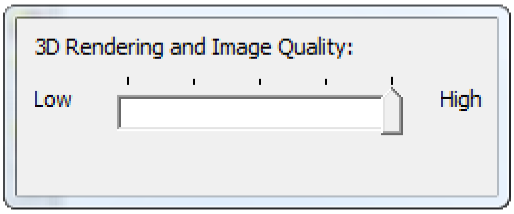
Windows default in the **Background color** group sets the background color of the workspace to the color defined by the Display Properties control panel. When it is deselected, the **Red**, **Green** and **Blue** fields become available, as does Select color. Enter values from 0 to 255 in the fields provided, or click **Select color** and choose the desired color from the Color dialog box.

Background image sets a background image for the workspace. **Align with background** lets you align the workspace with the background image. See the next section for more information about using this feature.

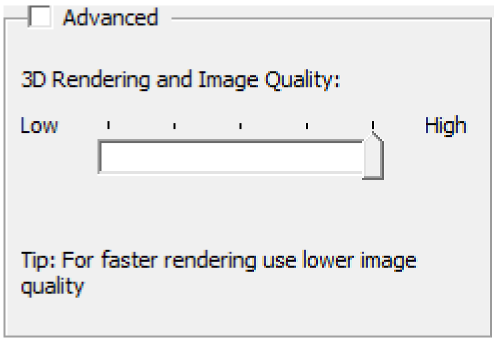
To change plotting styles, choose a new plotting style from the **Plotting Style** drop-down list box.

 The Performance slider, also known as the **3D Rendering and Image Quality** tool, exists both as a button on the View Mode toolbar and as a group within the **3D View Mode** dialog box.

Clicking the toolbar button opens a small dialog box next to the toolbar button:



In the **3D View Mode** dialog box:



The **Low** end of the slider represents faster performance at the expense of image rendering quality, while the **High** end represents better image quality at the expense of slower performance. If you try to snap to a setting that your computer cannot do, the slider will bounce back and a status message will appear on the Status bar. The toolbar icon indicates the current slider level as a green mark.

There may be a blue indicator in the slider field that shows the settings that your computer can accept.

Tick	Graphics resolution	Number of samples
1	Low	0

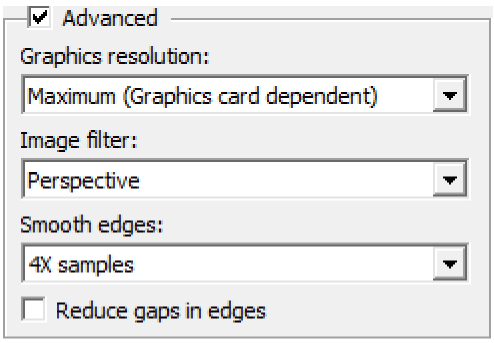
Tick	Graphics resolution	Number of samples
2	Medium	0
3	Medium	4x
4	High	8x
5	Maximum	16x

The settings between the two ends of the slider mix the advanced settings of graphics resolution and number of samples, which are discussed in the following topic. For example, if the design has no graphics on it and the display adapter supports 16x sampling, the slider can move to all the way to **High**. Conversely, if the design has many graphics and the display adapter can support large internal bitmaps, but with no support for smooth edges, the slider can still go to **High**. To determine the exact settings supported by your system, click **Advanced** and use the drop-down list boxes to list the available modes.

Set the default state of the items in the View Mode dialog box by clicking **Options** > **Defaults** > **Startup defaults** > **3D View mode**.

Advanced mode

In the 3D VIew Mode Dialog box, when you check the **Advanced** checkbox in the **3D Performance and Image Quality** section, the slider is replaced by the **Graphics resolution**, **Image filter**, and **Smooth Edges** drop-down list boxes, along with the **Reduce gaps in edges** checkbox.



Graphics resolution

Note:

The following discussion refers to the size of bitmaps that the display adapter can create in its internal memory. It does not refer to the size of bitmaps it can display on the monitor.

Note:

The quality of the graphics is limited by the quality of the original graphic files placed into the workspace. If the original graphics were low resolution or otherwise low quality, ArtiosCAD cannot improve them.

The **Graphics Resolution** drop-down list box has four settings which may or may not appear depending on the abilities of the display adapter.

Selected resolution	Overall bitmap pixel size
Low	1024 x 1024
Medium	2048 x 2048
High	4096 x 4096
Maximum	4096 x 4096 per panel or label

For designs, when you select the setting that is the highest your computer can render, the next higher setting causes ArtiosCAD to split the overall graphics into a bitmap per panel at the highest resolution possible. For solids, ArtiosCAD creates one bitmap per label or graphic.

For example, if the display adapter only supports 2048 x 2048 bitmaps, the best selection in the drop-down list box will be **High**. Because this is one level above what the display adapter supports, the graphics will be split into one graphic per panel for a design or label/graphic for a solid.

Note:

If present, a background image is unaffected by this setting and always uses the highest resolution possible.

Image filter

The **Image filter** drop-down list box has three values:

- *Nearest*. Nearest point filtering has been the standard image filter for 3D. It offers sharp edges but blocky results.
- *Linear*. This image filter blurs and blends the pixels together for a smooth look, but some sharpness is lost.
- *Perspective*. This image filter is similar to linear filtering but it compensates for the slope of the object upon which the graphics appear. This method requires more resources than the others so it may impact performance.

Note:

If a background image is present, these options affect its quality.

Smooth edges

The choices on the **Smooth edges** drop-down list box control how ArtiosCAD draws the edges of objects. ArtiosCAD reads each pixel on the edge of a polygon read several times in different places and combines the readings to give the best result, which is called Multi-Sample Anti-Aliasing.

The list in the drop-down list box shows the values your display adapter supports. The values range from **No samples** to **16X samples**, which means each edge pixel is sampled 16 times. The greater the number of samples selected, the longer the render will take.

Note:

Bitmap Outputs do not use smooth edges algorithms.

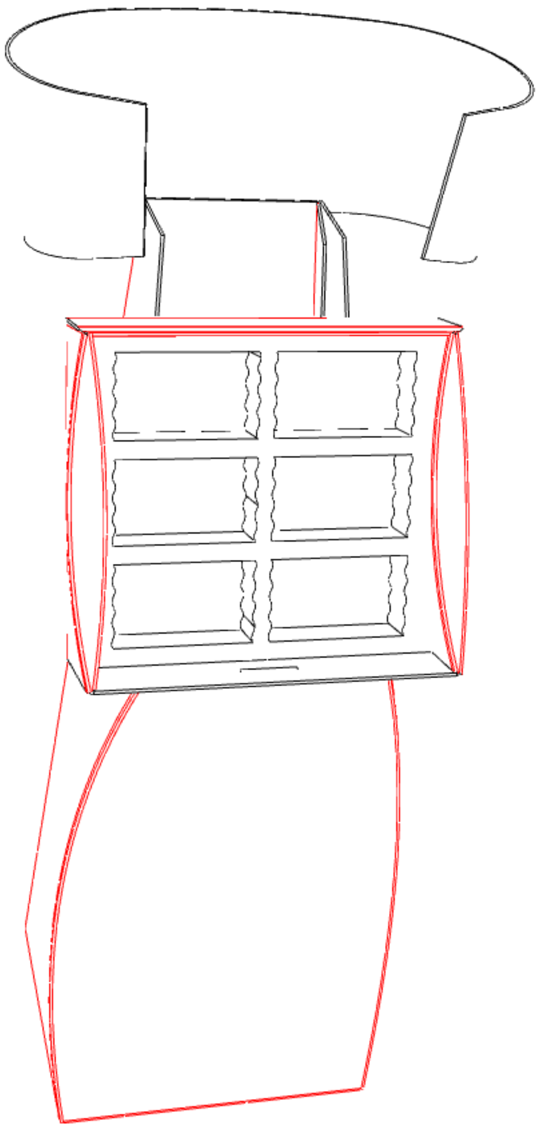
Reduce gaps in edges

The **Reduce Gaps in Edges** checkbox in the **Advanced** group of the 3D View Mode dialog box controls how ArtiosCAD displays visible vectors in 3D. When this checkbox is checked, **Hidden Line Removal**, **Solid With Edges**, and **Lighter Color With Edges** view modes all look smoother and better. This option also affects Outputs of type **Bitmap**.

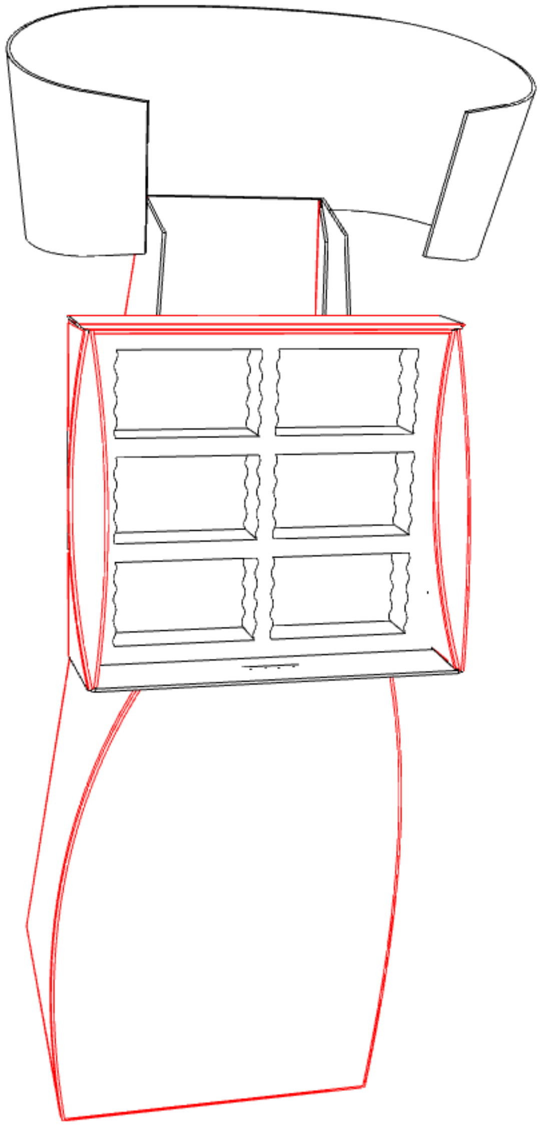
Note:

Reduce Gaps in Edges is enabled by default but is not used if there is a solid in the 3D workspace.

Without **Reduce Gaps in Edges** checked:



With **Reduce Gaps in Edges** checked:



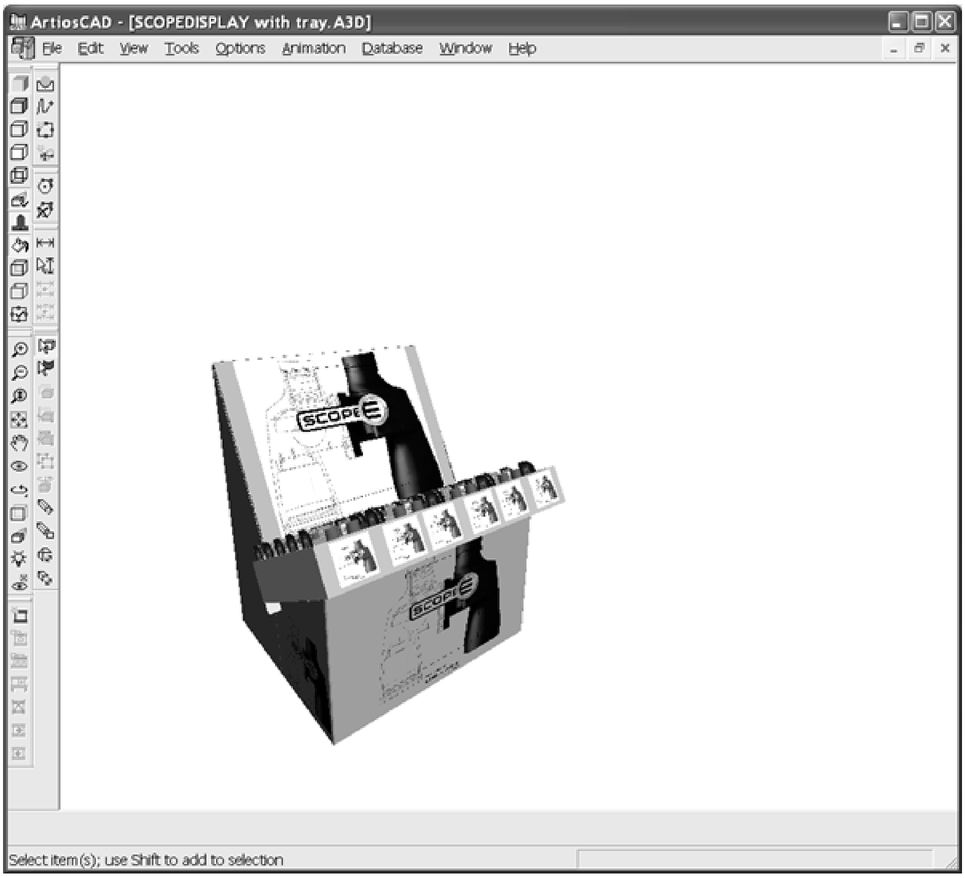
Using a freehand background image

The **Background image** group in the View Mode dialog box lets you set a background image for the 3D workspace. You can either position the design against the background freehand, or you can use the **Align with background** command on the View menu, which is documented in the next section.

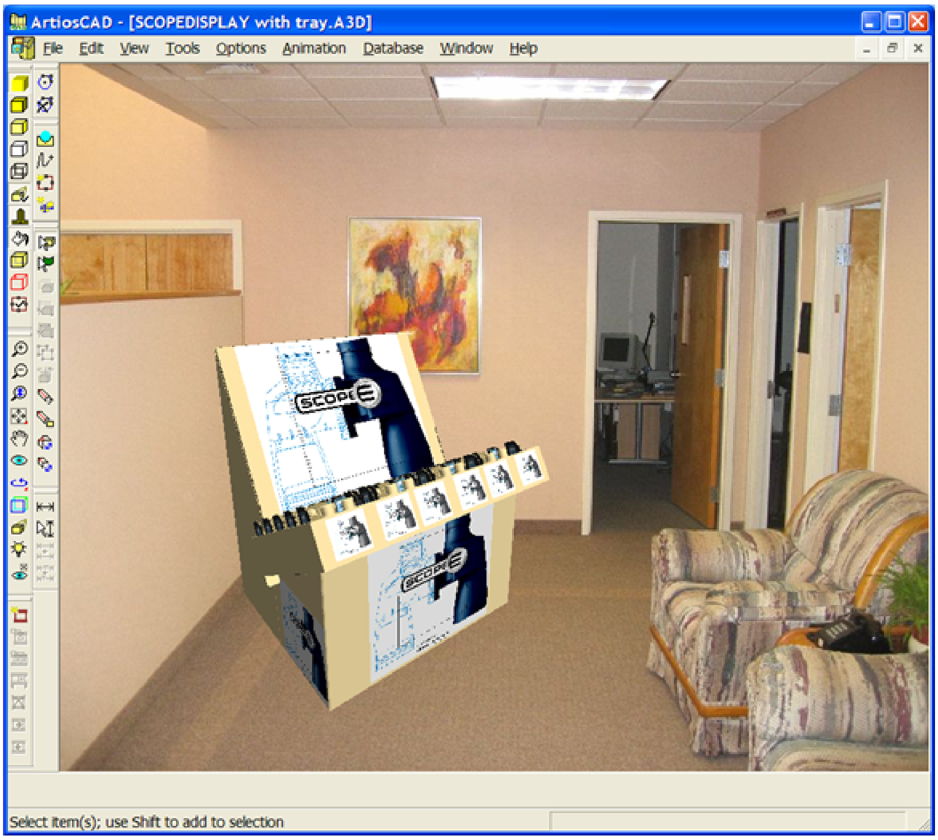
Using a background image significantly increases the time it takes to display the workspace. Therefore, position the scene as close as possible to the desired result before adding or turning on a background image. It helps to view the background image first in an image-viewing application to determine the scene before executing it in ArtiosCAD. Also, using Direct3D instead of OpenGL as the rendering method may lead to more robust performance.

To use this feature in freehand mode, do the following:

1. Position the objects in 3D as close as possible to the end result.



2.  Click **View Mode**, and then click **...** (Browse) in the **Background image** group.
3. Navigate to the folder containing the desired image file, select it, and click **OK**. PDF and Adobe Illustrator .AI files may not be used as background images.
4. Click **OK** in the View Mode dialog box to turn on the background image. The background image is centered to match the height of the image with the size of the window. If the image is wider than the window, it is clipped on both sides; if the image is narrower than the window, white space appears on either side of the image.
5. Use the **View Angle**, **Zoom In/Out**, **Pan**, and **Perspective** tools to adjust the position and rotation of the workspace against the background image. You may also use the **Move** and **Copy** tools but their performance will be impacted by having the background image turned on. The background image itself may not be moved. Shown below is a 3D workspace with background image.



Note: The complexity of the workspace, size of the background image, and 3D rendering method all affect system performance. Bear these in mind when composing the scene. Direct3D may perform somewhat faster than OpenGL on some systems.

To choose a different background image, choose a new image file in the View Mode dialog box as explained in steps 2 through 4.

The background image is saved as part of the 3D workspace.

The background image and floor shadow do not appear in CloseUp windows.

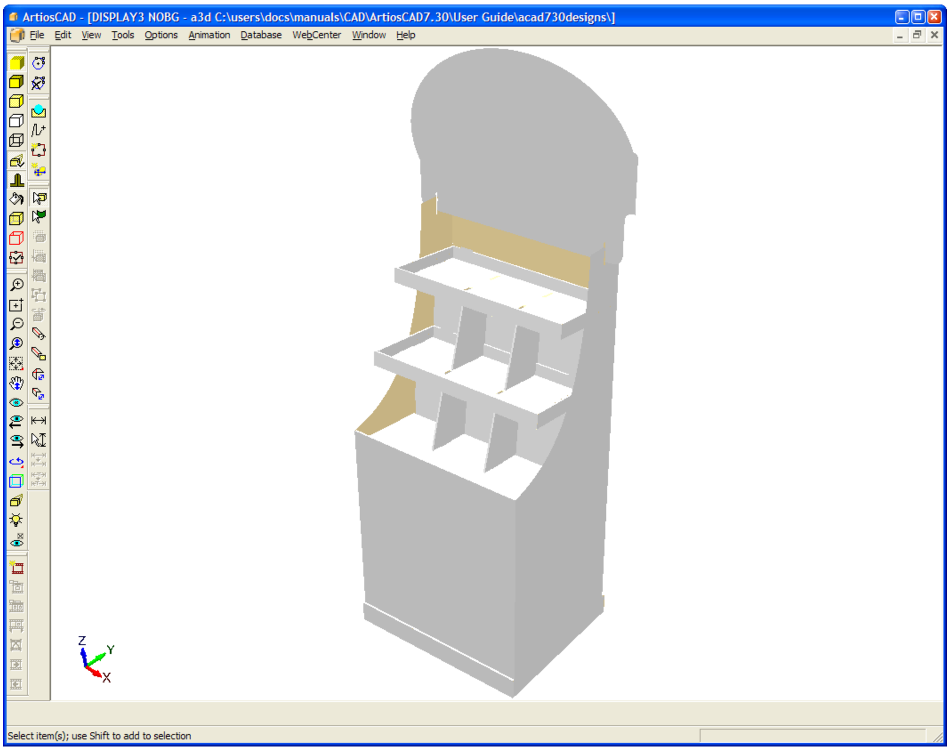
Using an aligned background image

Aligning a background image with a 3D workspace involves indicating a rectangle on the floor of the image (assuming you are using a picture of a place) and then entering the size of that rectangle. This tells ArtiosCAD how to set the vanishing point for the perspective in the workspace.

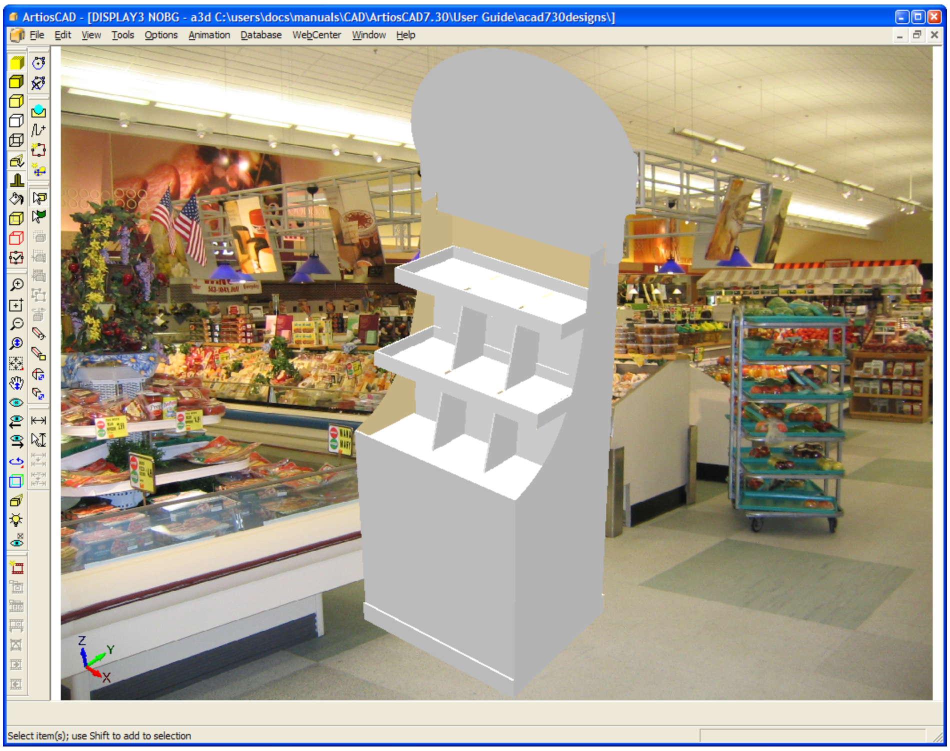
The **Align With Background** tool aligns the longest X/Y dimension of the bounding box of the designs with the longest dimension in the background image.

To use a background image and align the workspace with it, do the following:

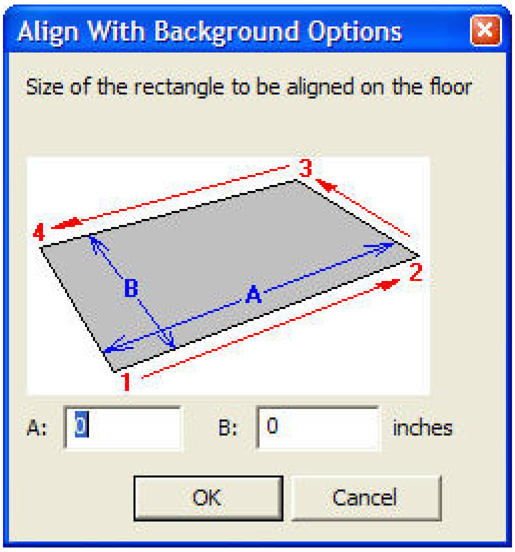
1. In ArtiosCAD, assemble the complete workspace.



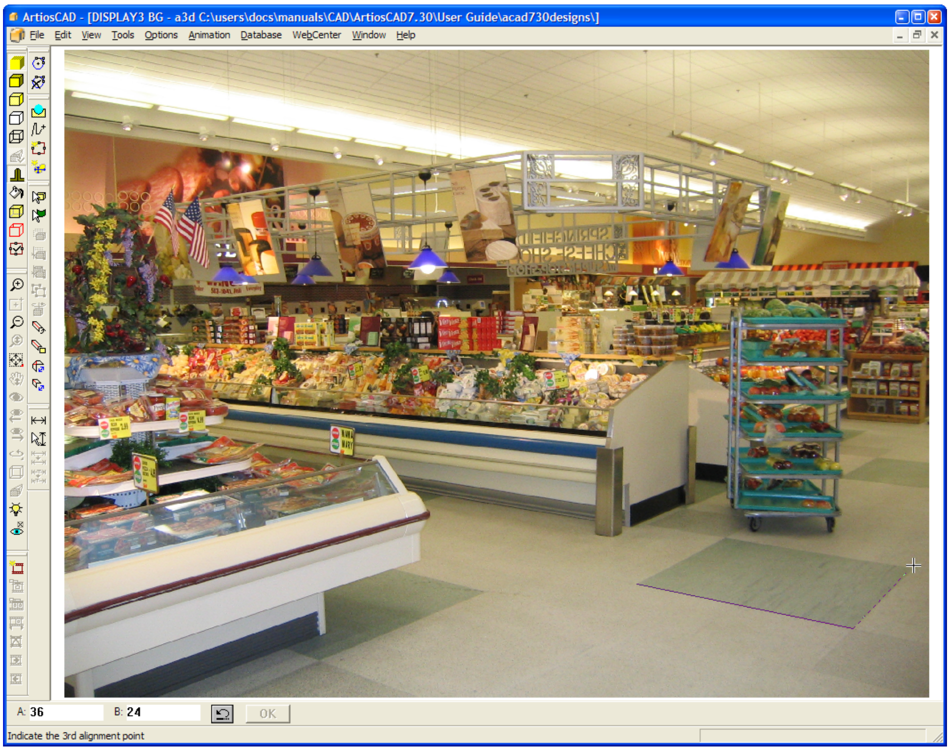
2. View the background image in an image-viewing application and decide upon the scene.
3.  Click **View Mode** to open the View Mode dialog box.
4. In the View Mode dialog box, select the **Background image** checkbox, and use the **...** (Browse) button to select the desired image.
5. In the View Mode dialog box, click **OK**. The background image appears in the workspace.



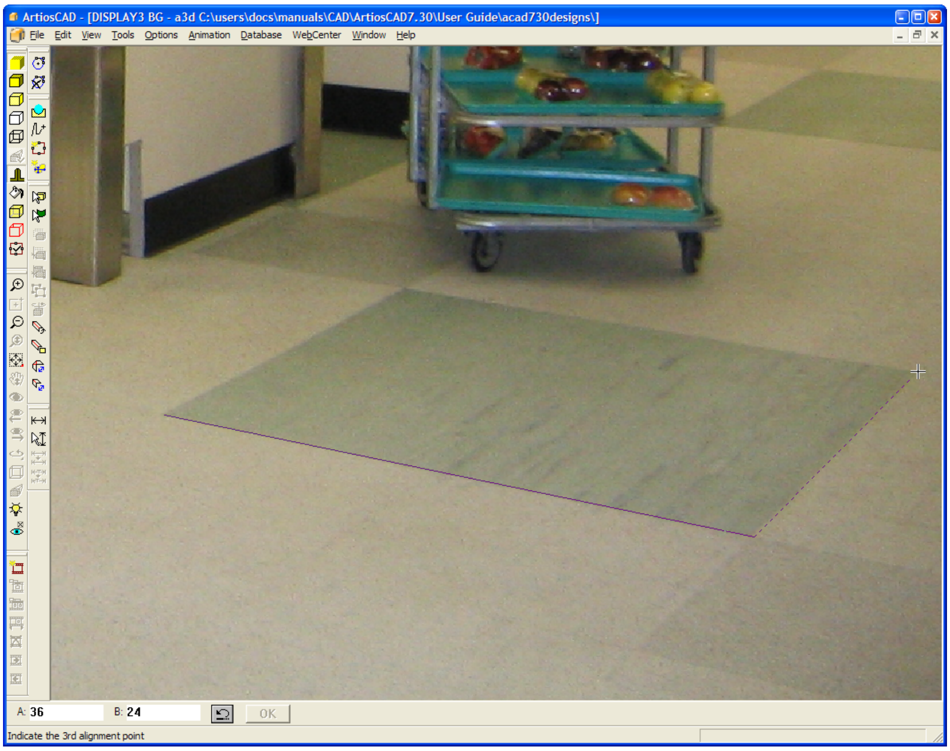
6. Click **View** > **Align with background**. The Align With Background Options dialog box appears.



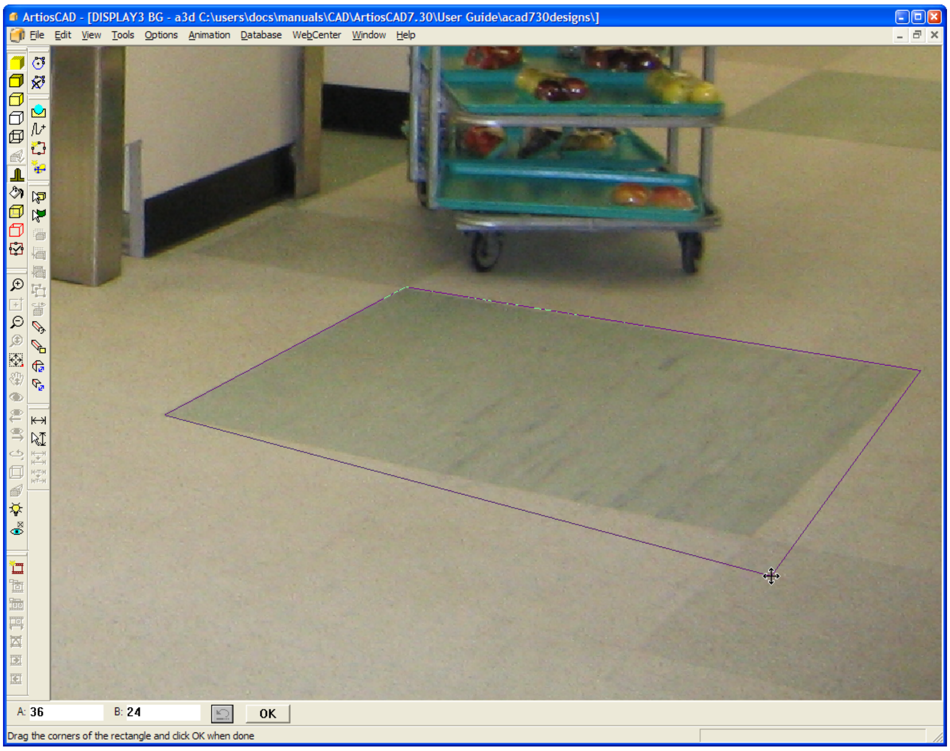
7. The picture in the dialog box shows the order in which to click the corners of the alignment rectangle in ArtiosCAD to set its position. In the fields, enter the longer dimension of the rectangle in the **A:** field and the shorter dimension in the **B:** field. Using a rectangle with a definite proportion between length and width works better than a squarer rectangle. Also, the line formed by points 1 and 2 should go across the front of the designs in the workspace for best results.
8. Click **OK** in the Align With Background Options dialog box. The 3D workspace will be turned off temporarily to show the entire background image.
9. Click the points of the rectangle in order. Click the lower left corner, then the lower right, then the upper right, and finally the upper left. ArtiosCAD automatically adds the line from the last corner to the first corner. If you make a mistake, click the **Undo** button on the Status bar. Shown below is the setting of the second corner.



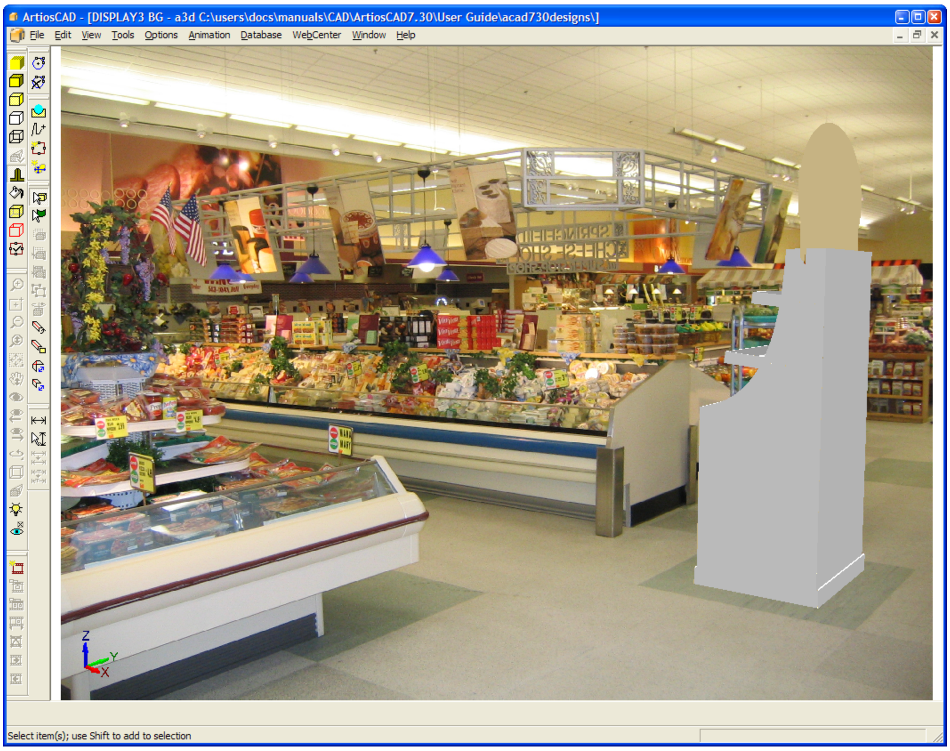
10. You may zoom as needed using the **Zoom In**, **Zoom Out**, and **Scale to Fit** tools. Larger alignment rectangles generally yield better results than smaller ones. Precise alignment is critical, as differences as small as a pixel can greatly affect the position of the vanishing point. Shown below is a zoomed-in view in which you can see that the right edge of the rectangle is a few pixels off; that can be corrected later in the process.



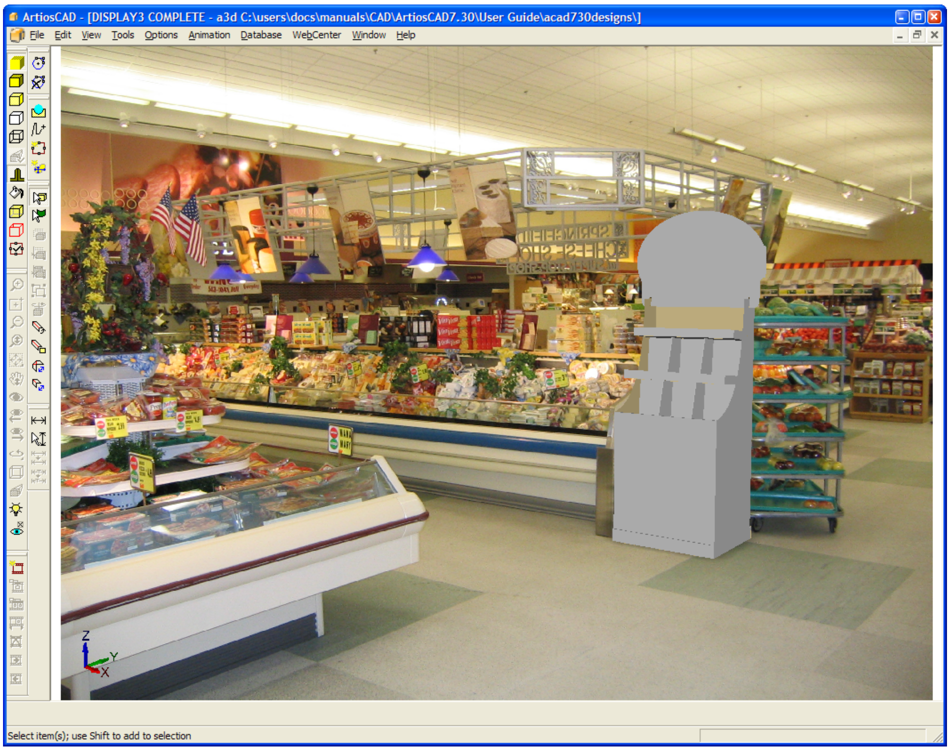
11. Once the rectangle is drawn, you can click a corner and drag it as needed to adjust it. Shown below is an adjustment for emphasis;



12. When you are satisfied with the size and position of the alignment rectangle, click **OK** on the status bar. The 3D workspace will be turned back on and aligned inside the alignment rectangle as shown below.



13. ArtiosCAD is now in **Align with background** view mode. In this mode, the **View Angle**, **Zoom**, **Scale to Fit**, and **Pan** tools are unavailable in order to not lose the alignment with the background. Use the **Move** and **Rotate** tools to further align the workspace objects against the background.
14. If desired, turn on the floor shadows in the View Mode dialog box and use the **Light Source** tool to adjust the position of the light sources to change the shadows.
15. Shown below is the completed image.



16. To exit **Align with background** mode and make the View Angle, Zoom, Scale-to-Fit, and Pan tools active again, click **View Mode** on the toolbar and deselect **Align with background**.

Note: Once you change the view, the alignment with the background is lost, and you will have to repeat the entire procedure to realign the background with the workspace.

Note: Wide-angle images from digital cameras may suffer from fish-eye lens distortion at their edges. In this case, the alignment of ArtiosCAD's perspective with the background image perspective will be less accurate, with the error becoming larger as the design is moved further from its original placement.

Outputs using background images

JPEG and PNG outputs include the background image at the same aspect ratio as shown in ArtiosCAD.

VRML output includes the background image as a rectangular object positioned behind the designs and grouped with the toolbar (if it is selected for output as well).

Animations with a background image using the Scale to Fit feature for captured frames may zoom in and out when output to VRML. The background image stays the same size when the animation is played in ArtiosCAD. Combining use of the Scale to Fit feature in an animation, using a background image, and exporting to VRML may lead to unpredictable results.

VRML viewers may perform more robustly with background images when set to use Direct3D instead of OpenGL.

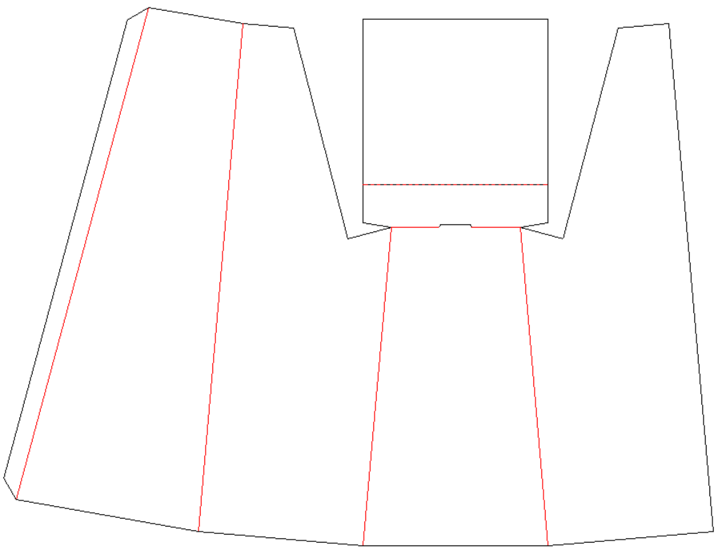
VRML outputs containing a background image also contain two extra rectangles for proper image placement. These extra rectangles may appear in other programs when opening the VRML file. To avoid this situation, before outputting a VRML file, turn off the background image in the View Mode dialog box, and then in the VRML dialog box, do not select **With toolbar**.

Level Designs tool

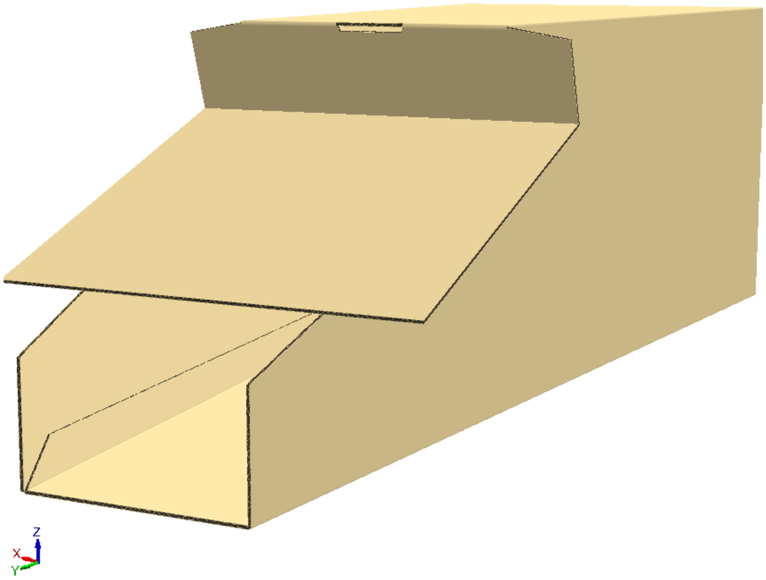
Use the **Tools** > **Level Designs** tool to make a design look level on the floor when it has no horizontal base face.

To use the **Level Designs** tool, do the following:

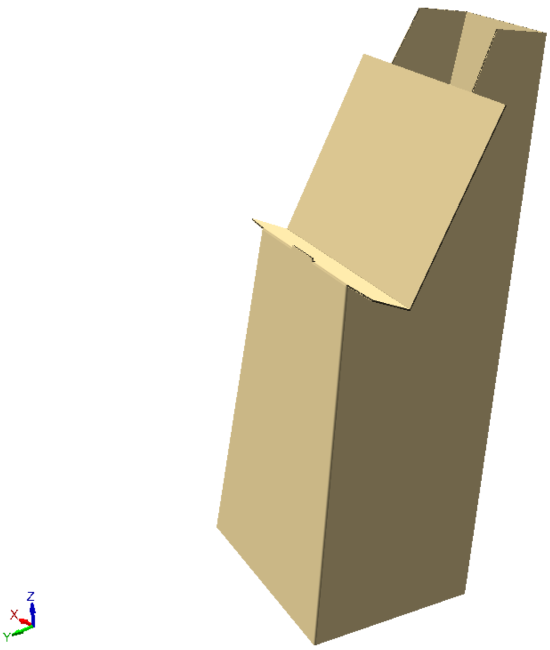
1. Create or open a 3D workspace that has no horizontal base face. For example, this simple display base has no panels that would sit flat on the floor.



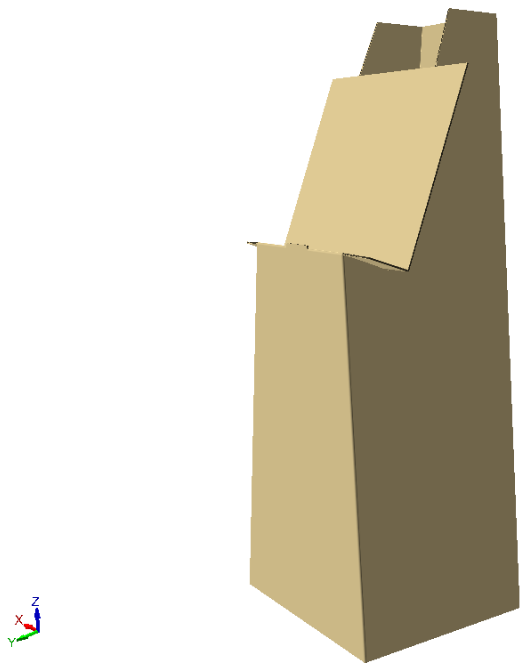
2. Use the back as the base face when converting to 3D.



3. Rotate the design so that it is close to upright.



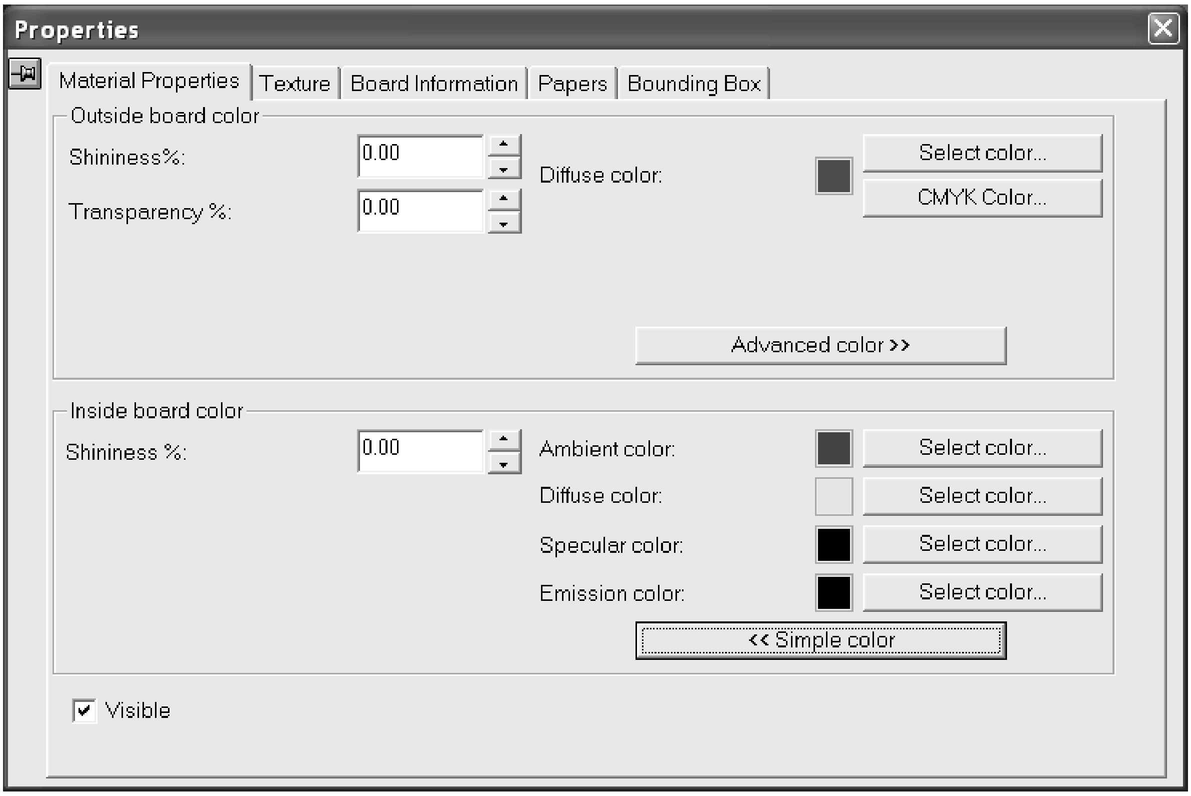
4. Click **Tools** > **Level Designs**. ArtiosCAD changes the orientation to make the display look more level.



Changing visible properties of objects in 3D

To change the colors of objects in 3D, double-click them with the appropriate Select tool to open the Properties dialog box for the type of object. To select more than one item to change at once, hold down CTRL while clicking to select items.

Use the **Select Designs** tool and double-click a container to change its properties.



The **Shininess %** and **Transparency %** fields help to simulate different materials. Shininess is most visible on round objects; it has little effect on flat objects such as containers. Transparency of the board works independently of the **Transparent** checkbox in the View Mode dialog box. Use the values in the table below to approximate common objects.

Table: Common Amounts of Shininess and Transparency

Material	Shininess	Transparency
Matte board	0	0
Glossy board	40	0
Beer bottle	90	50
Glass	100	80

Surface colors are split into four components: **ambient** color, which reflects the ambient light; **diffuse** color, which reflects the movable lights in an airy, non-shiny way; **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; and **emission** color, which is the color the object emits.

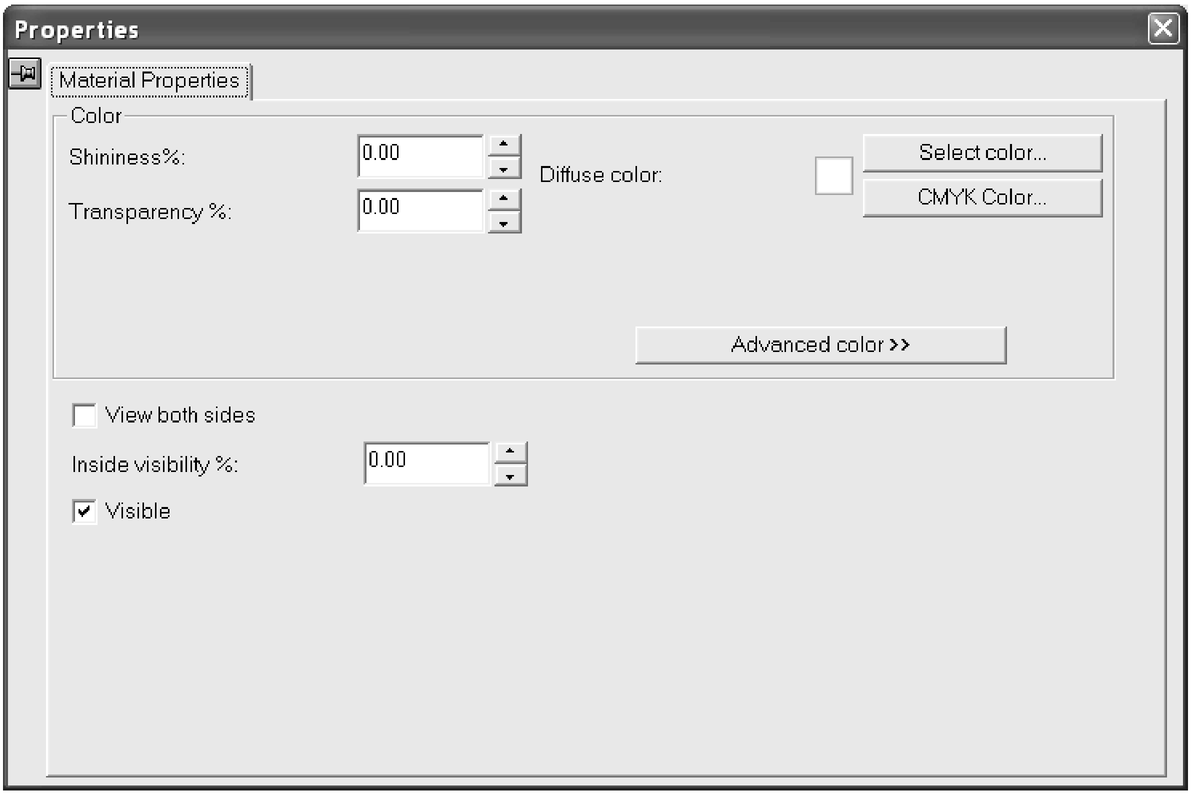
By default, the Material Properties dialog box only displays Diffuse color in Simple color mode. In Simple color mode, choose an RGB color by clicking **Select color**, or choose a CYMK color by clicking **CYMK color**. To set the other three colors, click **Advanced color**, and then click **Select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

Visible controls whether the current selection is included in VRML exports and also if it is included when ArtiosCAD calculates the view scale of the workspace. **Show invisible designs** in the View

Mode dialog box controls whether invisible designs are shown at 90% transparency (when visible) or not at all (when invisible). If any part of a design is visible, the entire design is visible; the visibility of the part affects the visibility of the whole. Visibility can be set independently for each design at the start of each animation frame.

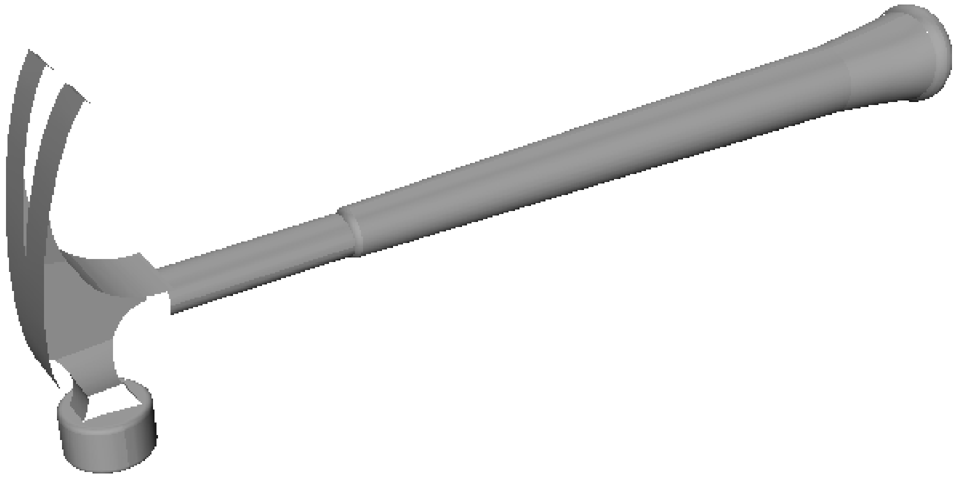
Changes you make in this dialog box are applied as they are made. Close the dialog box by clicking the X at the end of the title bar.

 Use the **Select Labels or Parts** tool and double-click a label or part of a solid to change its properties, or to delete selected parts by pressing del on the keyboard. You may also click and drag to select more than one item, or hold down CTRL while clicking to select more than one item.

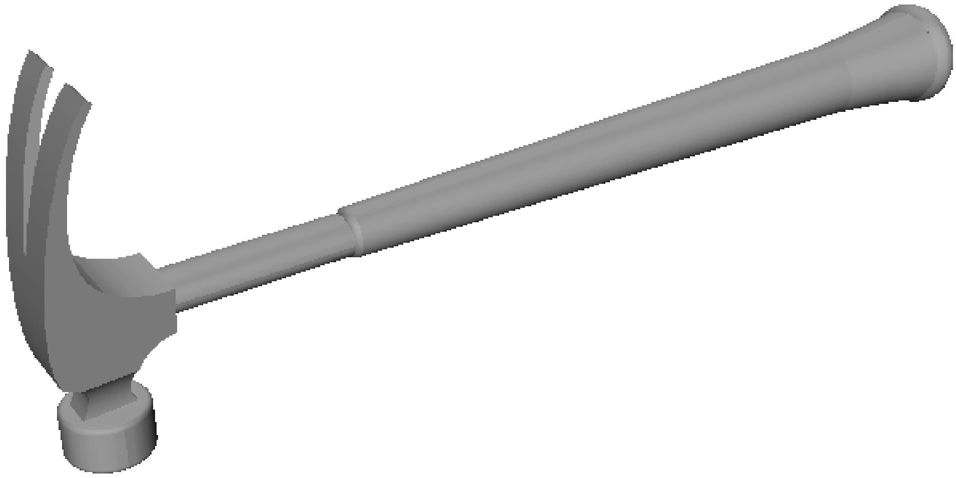


The color controls for a label or parts of a solid work the same way as they do for containers, but there is just one color group. The differences for labels and parts of solids are the **View both sides** checkbox and the **Inside visibility %** field.

View both sides shows all facets of a solid so that it is shown correctly. Shown below is a hammer model with **View both sides** turned off.



With **View both sides** turned on, the head of the hammer is now complete.



Inside visibility % sets the percentage of labels that show through transparent objects. The label on the bottle below is set to 0% inside visibility.

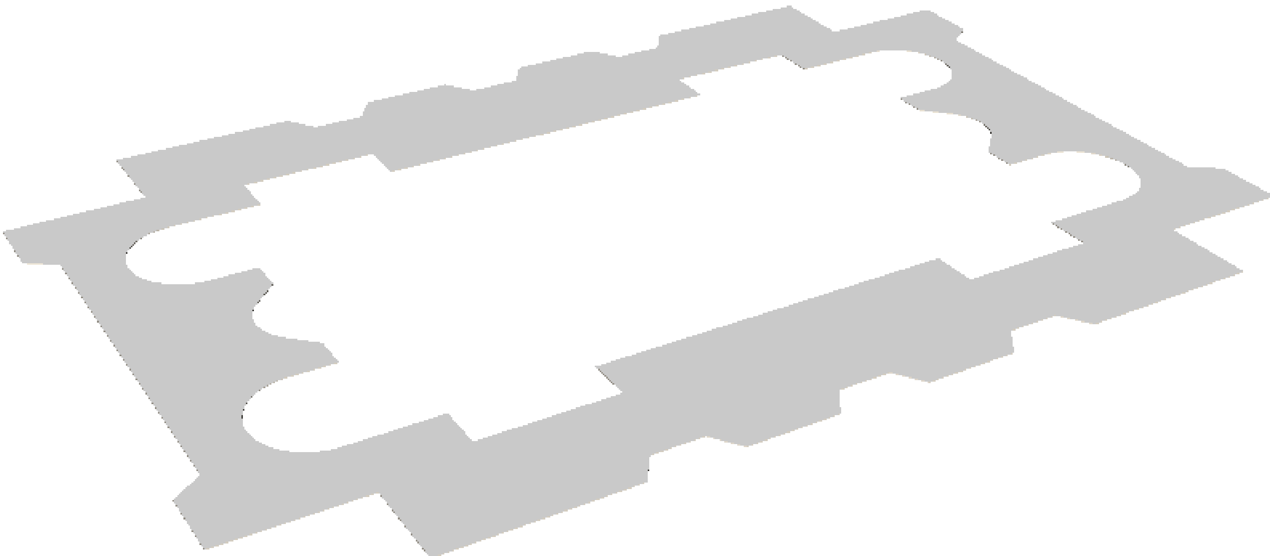
The same label is now set to 70% inside visibility:

Visible controls whether the current selection is included in VRML exports and also if it is included when ArtiosCAD calculates the view scale of the workspace. **Show invisible designs** in the View Mode dialog box controls whether invisible designs are shown at 90% transparency (when visible) or not at all (when invisible). If any part of a design is visible, the entire design is visible; the visibility of the part affects the visibility of the whole. Visibility can be set independently for each design at the start of each animation frame.

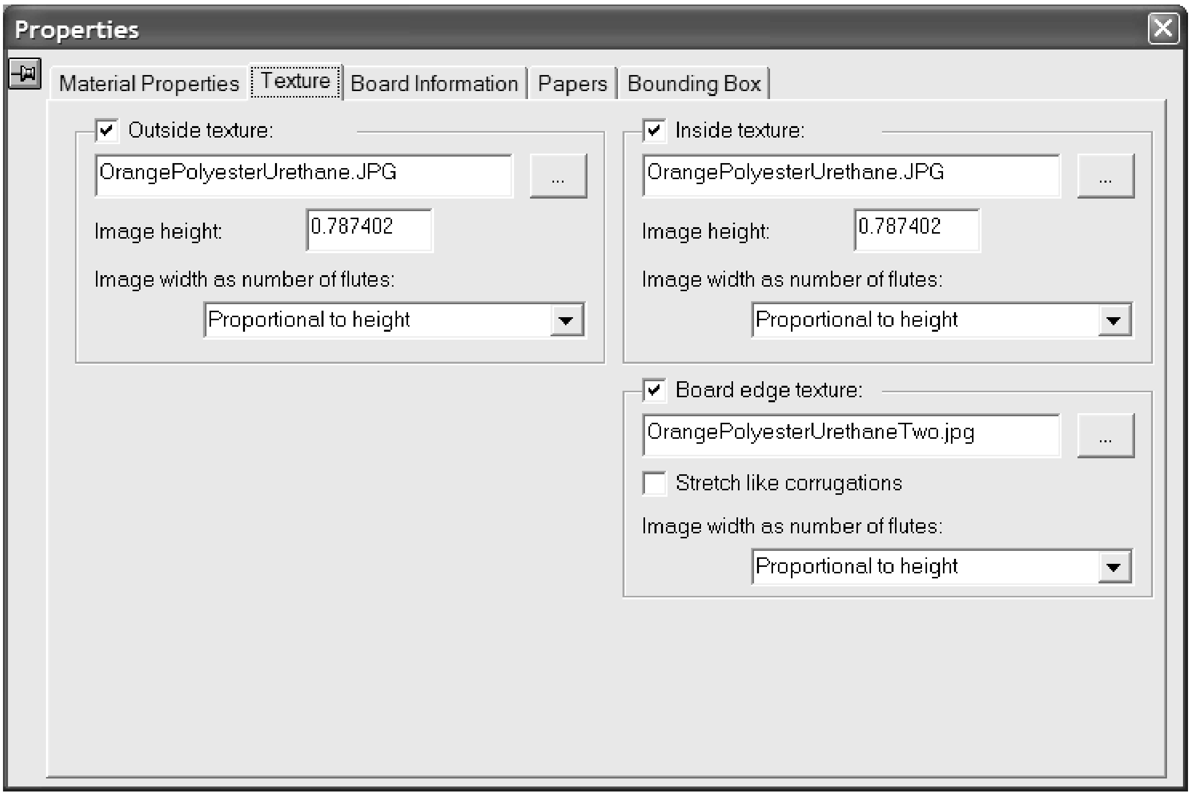
How to change the board information in 3D

To change the board information dynamically in 3D without changing the board information in the flat workspace, do the following:

1.  Click **Select Designs** and double-click the object to change, or single-click and select the design(s) to change, holding down `CTRL` to select more than one object.



2. If you selected more than one object, click **Edit** > **Properties**.
3. To change the color, shininess, or transparency of the boards, click the Material Properties tab, and change the values as desired. *#id873818U3D24*
4. To change the textures of the board from those defined by the board code, click the Texture tab. Shown below are the textures used for the Orange Urethane Two foam board in the **Artios** > **Foam catalog**.



5. To add an inside or outside texture to the board, click the checkbox for it, and 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 are stored in **\Artios\Common**. 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.

   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.

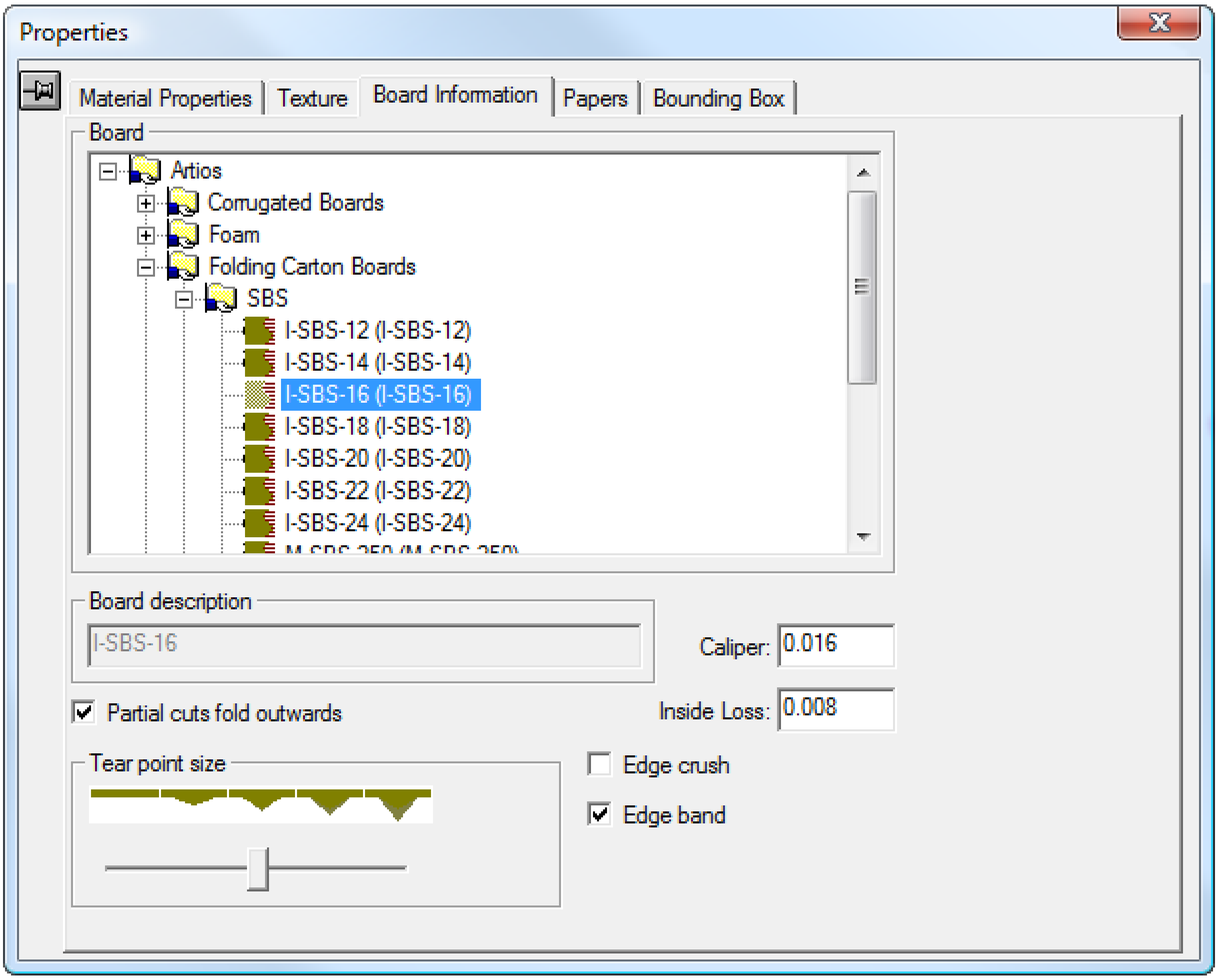
   Set the value in the **Image width as number of flutes**: drop-down list box to the number of flutes the image represents. The image should be of a whole number of flutes. It may also be set to **Proportional to height**, in which case the height cannot be set to 0.

   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 **\Artios\Common**. 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.

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

   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. This option should not be selected for hexacomb flutes.

6. To change the board code, the caliper, or the inside loss measurement of the selected objects, click the Board information tab in the Properties dialog box and change the values as desired. This tab is dynamic and only shows controls for the features that exist in the workspace. Change the value in the **Inside Loss:** field to make small adjustments to the position of the board in 90-degree folds.



Partial cuts fold outwards specifies the direction in which partial cuts act as creases. They generally fold away from the side of the board in which they were cut. Corrugated partial cuts generally fold to the outside of the container, while folding carton partial cuts generally fold to the inside of the container. If the selected board has flutes defined, this option is checked automatically.

Note:

Select this option to cut V-notches from the print side when using Re-board®. Cutting some V-notches from the inside and some from the outside in the same design is not supported.

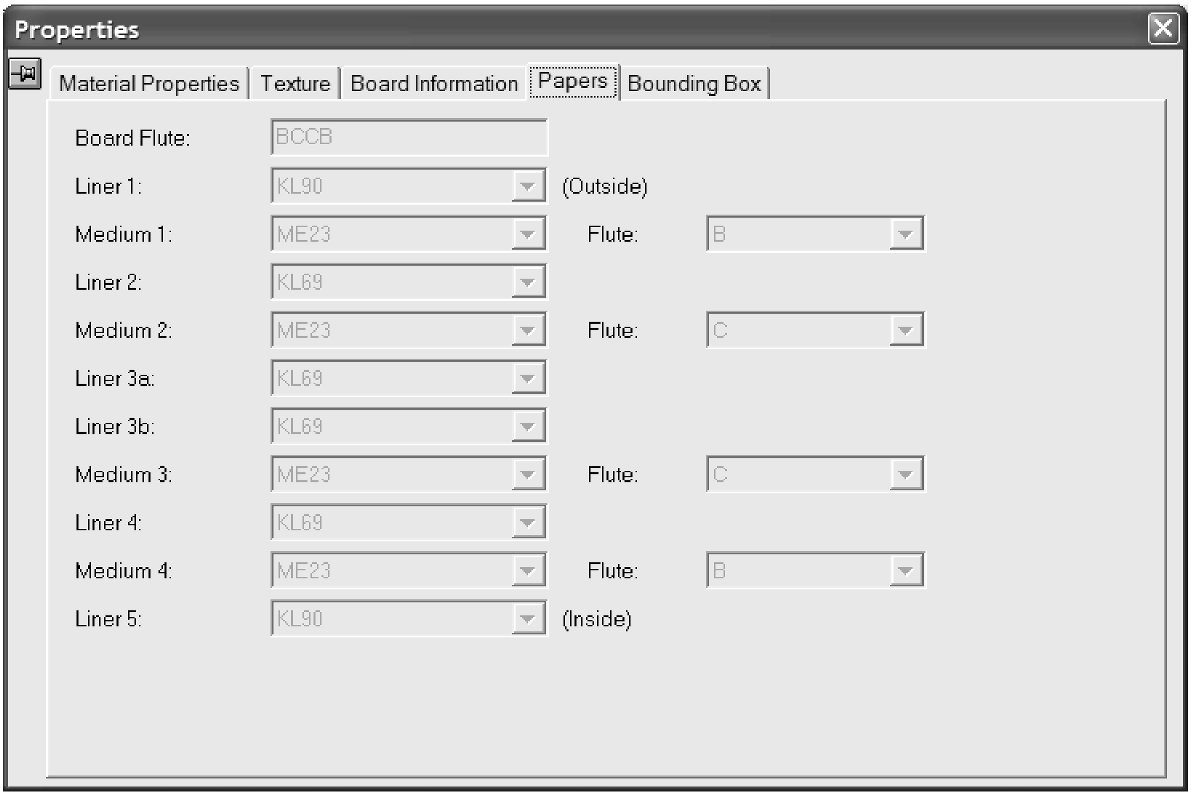
Thin corrugated boards such as E-, F-, N-, and G-flutes may be creased as if they were folding carton boards; therefore, depending on your workflow, turn it off if selecting one of those boards.

Use the **Tear point size** slider to adjust the appearance of tear points for perf in 3D.

Edge crush, when enabled, renders a rounded edge on cuts about half a millimeter wide that represents the beveled edge of cut rules. It is always shown for perfs between parts that tear, but you can control its appearance for other lines with this option.

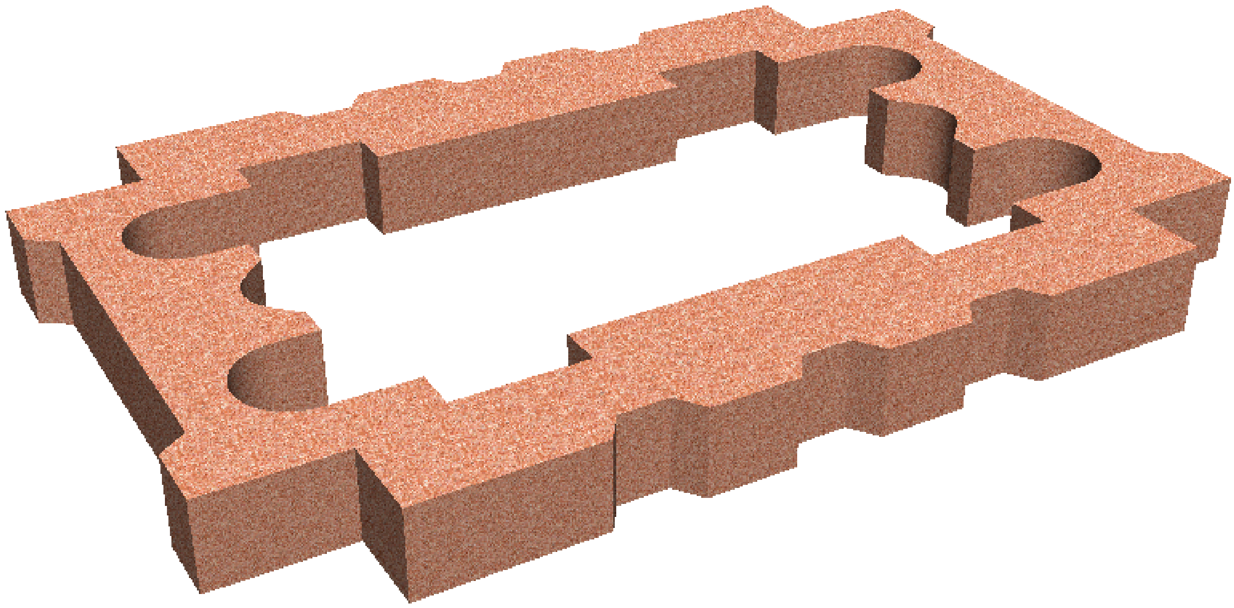
Edge band toggles the display of edge band rule.

Click the Papers tab to view the details about the papers comprising the board. No fields on this tab may be changed; they are for reference only.



Click the **X** in the upper right corner of the Properties dialog box to make any changes and close it.

7. The changes made will occur immediately. If board thickness was not turned on, it will be turned on when a new board is chosen.



Notes on board textures

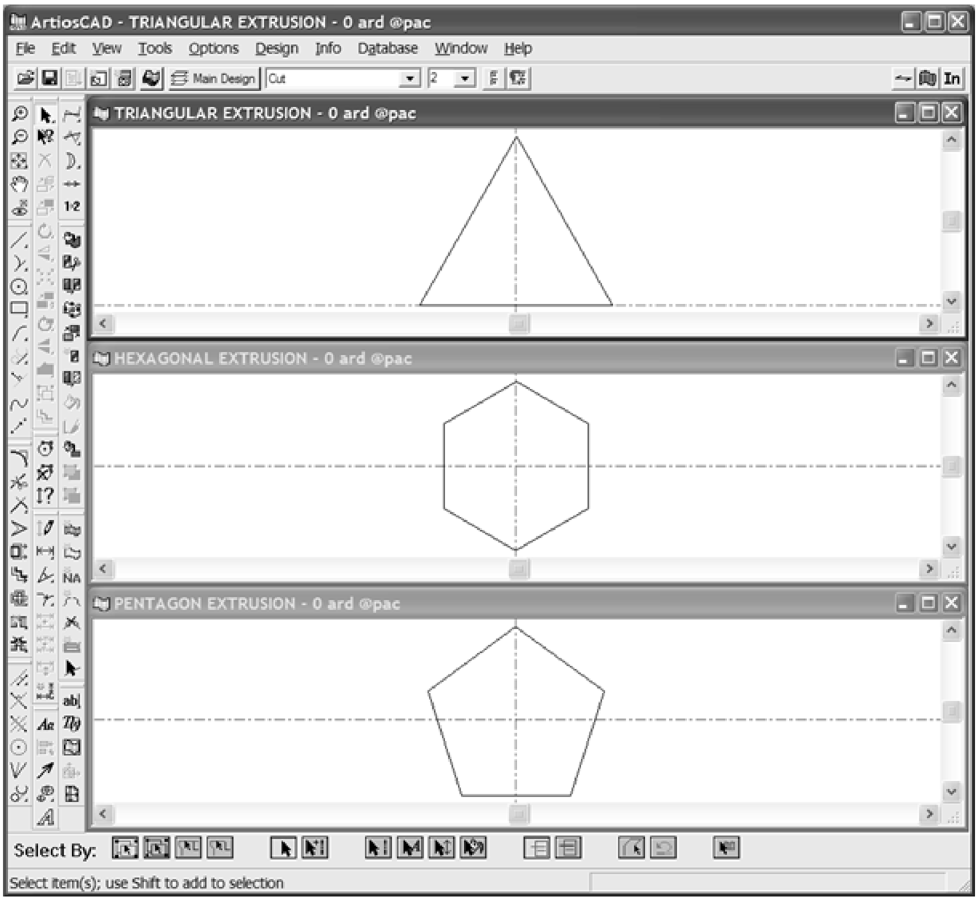
If a design has graphics, the graphics are drawn on top of the board texture or board color. If the graphics are PNG or JPG files, they will completely replace the board texture or board color. If the graphics are PDF files, the board image or board color will show through the parts of the PDF files that have no ink or are partially transparent.

Board textures are included in 3D workspaces sent to other sites. They should appear correctly even if the recipient's system does not have the same boards, papers, and textures defined.

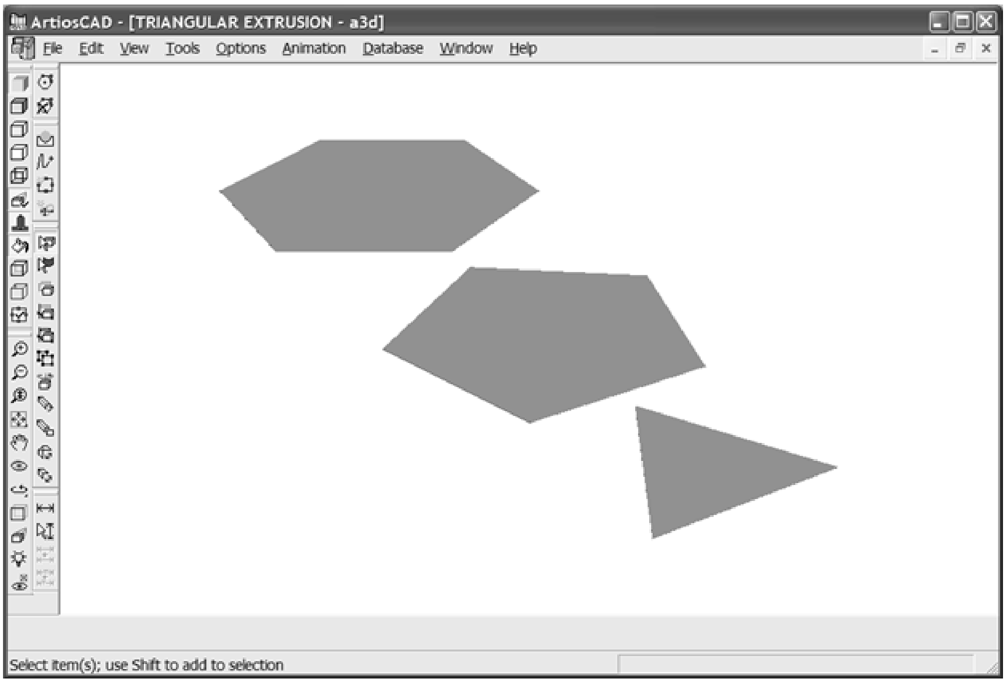
Making an extrusion

Making an extrusion is as simple as designing its shape in Single Design, converting it to 3D, and then changing the caliper of the board to the desired size of the extrusion.

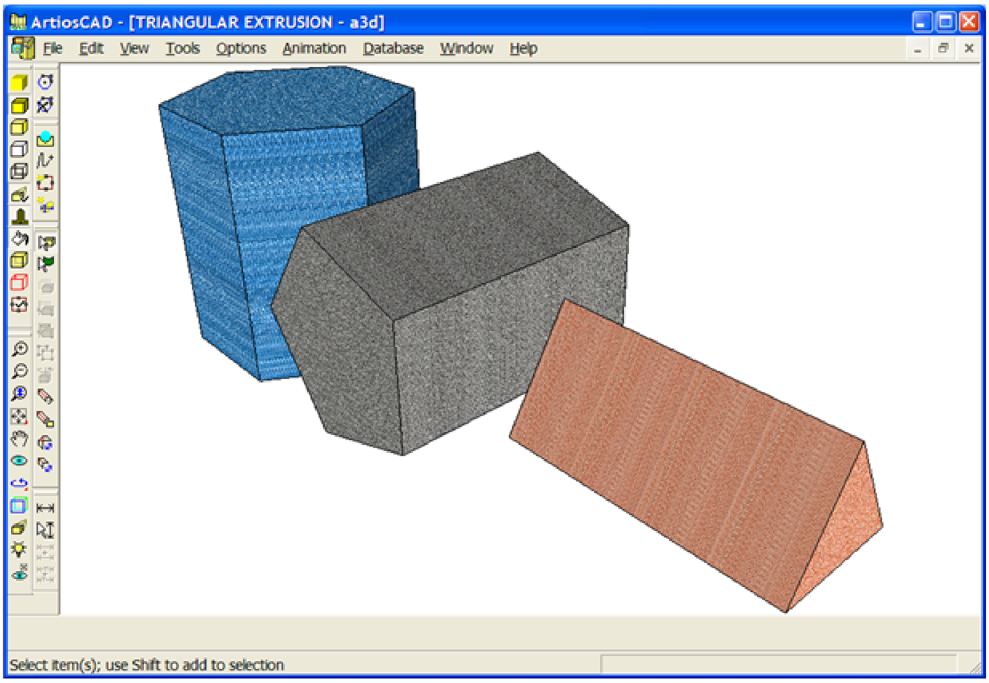
Shown below are three flat workspaces.



After converting one to 3D and adding the other two, this is the result having selected a normal folding carton board.



Shown below are the finished extrusions with different color foam boards selected and the caliper increased to 600 mm.



Refreshing the screen

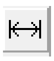
From time to time, refresh the screen to show the latest changes you have made to the workspace. Click **Refresh** on the View menu or press F2 to redraw the screen.

You can also hold down `SHIFT` and click the middle mouse button if you have one.

Working with dimensions in 3D

Use the tools in this section to add, select, modify, align, and remove dimensions in 3D.

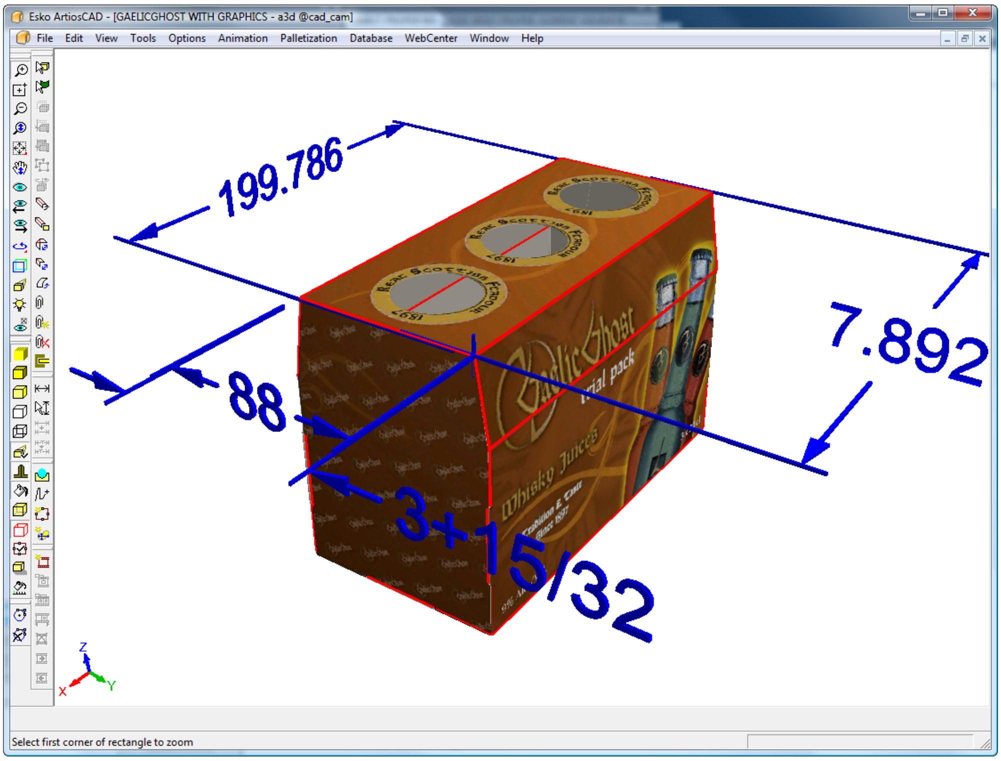
Dimension tool

 The **Dimension** tool allows you to insert dimensions into your workspace.

1.  Click the **Dimension** tool.
2. Select a point or line that will be the beginning of the measured distance.
3. If **Set Direction** is checked on the Status bar, set the direction for the dimension.
4. Indicate a point, a parallel line, or a collinear line that is the end of the measured distance
5. Indicate the extension point. This is where the text of the dimension is put in relation to the design.

6. The dimension is drawn. If **Set Direction** is checked on the Status bar, an extra extension line may be drawn indicating the direction.

Shown below is a design with both Imperial and metric dimensions. The Imperial dimensions were created first, and then the units were changed to metric and the rest of the dimensions were added. Unlike Single Design, 3D dimensions do not update when the units of the workspace are changed.

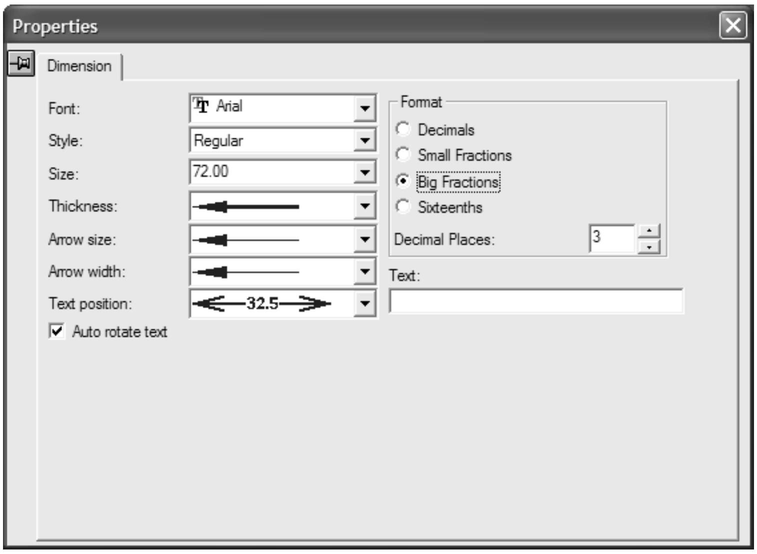


Set the default state of the **Set Direction** checkbox in **Options** > **Defaults** > **Property Defaults** > **3D Dimensions**.

Select Dimensions tool

This tool allows you to select dimensions so that you can change their appearance.

1.  Click the **Select Dimensions** tool.
2. Select the dimension(s) to change. To select more than one dimension, hold down the `SHIFT` key as you select items. To select many dimensions at once, make a window selection.
3. If you selected only one dimension, double-click it to open the Properties dialog box; otherwise, click **Edit** > **Properties**.



4. Change the style and format options as desired. **Auto-rotate text** enables the selected dimensions to automatically rotate to be parallel with an axis as the view angle changes. In some situations, the dimensions could be obscured by parts of the design when this option is selected, so deselect this option for affected dimensions as desired.

   The **Text**: field lets you replace the dimension text with text of your choosing. Use **Undo** to revert to the original dimension text if needed.

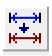
   The selected dimension(s) are updated with changes as you make them.

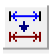
To set the dimension defaults for the current workspace, right-click while over a design, click **Property Defaults** on the context menu, and set the options as desired. The changes only affect dimensions created after the property defaults are changed.

To set the defaults for the arrows, text position, and auto-rotation of text for 3D dimensions, click **Options** > **Defaults** > **Property defaults** > **3D Dimensions**.

To view dimensions in a different color, change the plotting style in the View Mode dialog box to one which has a different color for dimensions.

Change Dimension Alignment tool

 The **Change Dimension Alignment** tool lets you change the extension point of a dimension within the plane of its extend lines. To use it, do the following:

1.  Select a dimension with the **Select Dimension** tool.
2.  Click the **Change Dimension Alignment** tool.

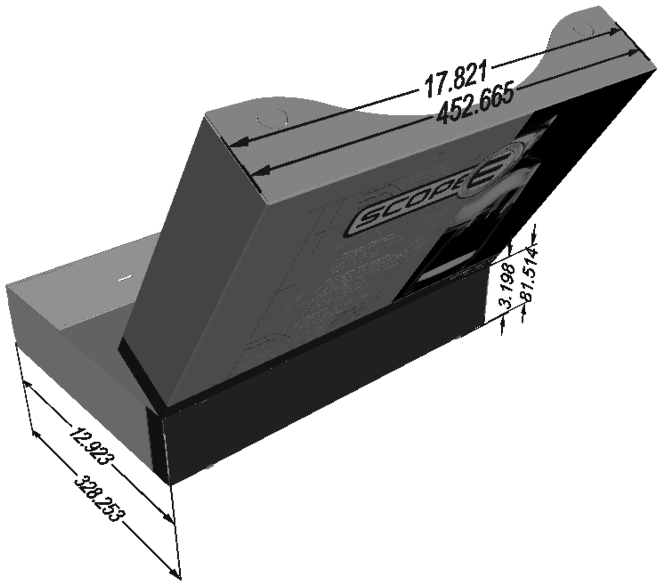
3. Click the desired location for the dimension. The dimension moves to the new location and the extend lines change length as needed.

Change Text Position tool

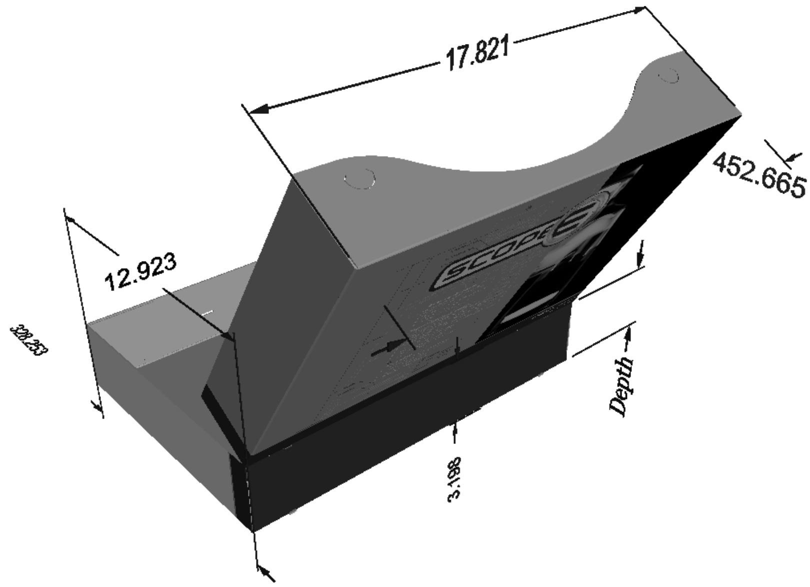
 The **Change Text Position** tool lets you change the position of text within a dimension. To use it, do the following:

1.  Select a dimension with the **Select Dimension** tool.
2.  Click the **Change Text Position** tool.
3. Drag the text and the extend point of the dimension to its new position and click to set it.

Shown below is a workspace before changing the text positions of dimensions.



Shown below is the same workspace after changing the text positions and extend points of the dimensions.



Deleting dimensions

To delete dimensions, select them with the **Select Dimensions** tool, and then press `Delete` on the keyboard.

Working with Edge Band and Tear Tape

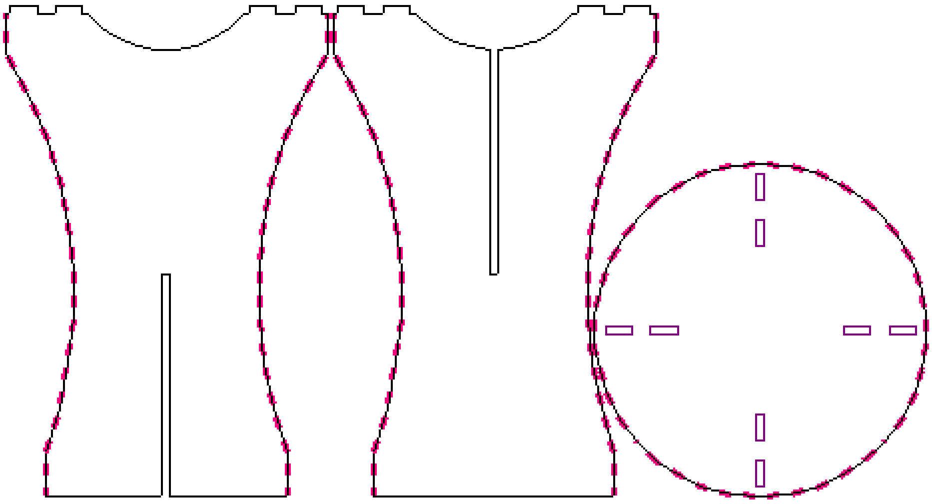
Short reference description.

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.

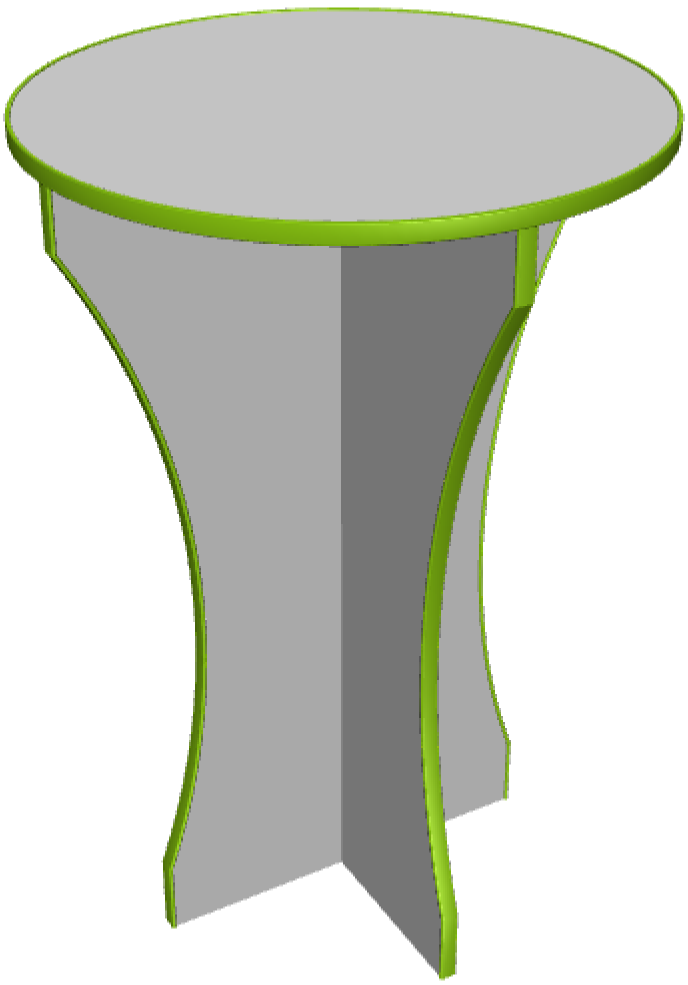
Both edge band and tear tape are special rules.

Edge band

To use edge band, set some edges in the 2D design to use it.



When you convert the 2D to 3D, the edge band appears. Turn it on and off by checking or clearing **Edge band** in the Properties dialog box on the Board Information tab.

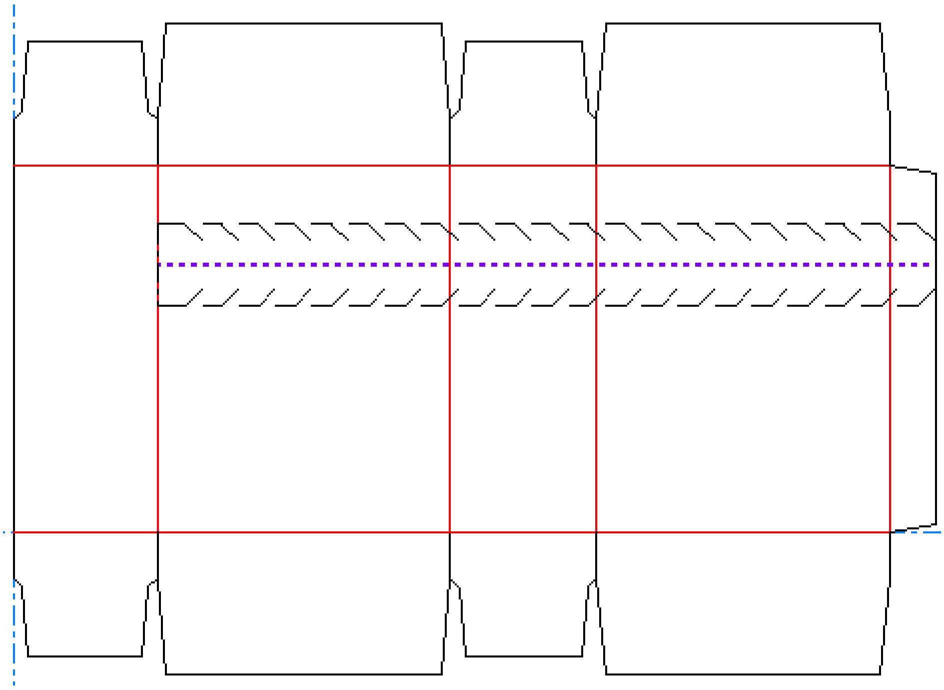


Notes about edge band:

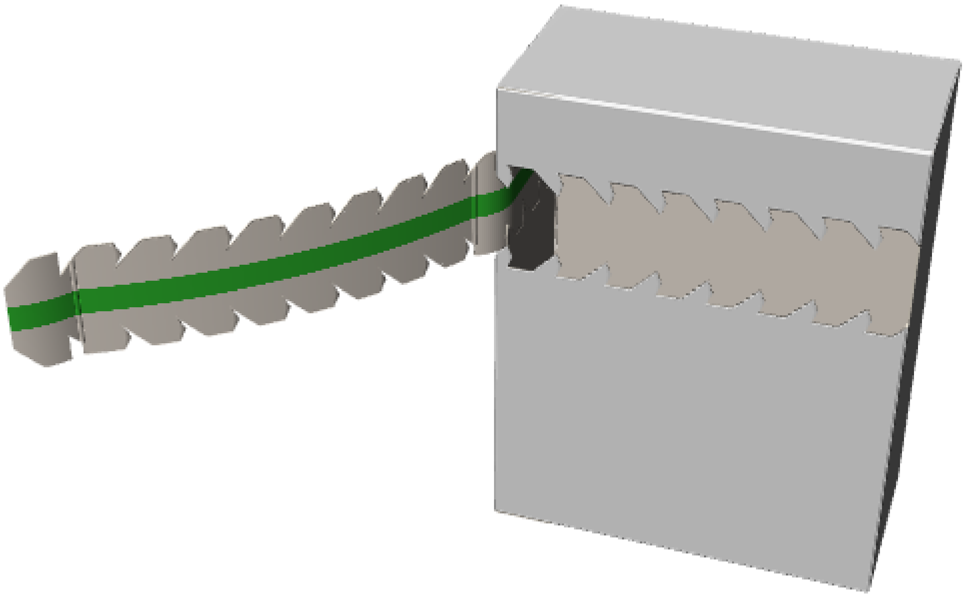
- If edge band goes around a corner, ArtiosCAD shows separate pieces with no mitering.
- You may only use one type of edge band per single design added to 3D.
- To change the color of the edge band, change it in 2D on the Special rule tab of the Properties dialog box for the edge band rule and reconvert to 3D. You cannot change the edge band color in 3D.
- ArtiosCAD shows only solid-color edge band and does not support textures on it.
- Edge band does not turn on or off within an animation.

Tear Tape

To use tear tape, add it as you would any other line to the design. In the example below, it is between the zipper rules.



When you convert it to 3D, it looks like the following example.



Note: Tear tape does not tear by itself, so to show it tearing without the aid of something like a zipper, surround it with tear lines in the 3D Assist layer.

Animation in 3D

Animation in 3D lets you record a folding sequence and output it as a **V**irtual **R**eality **M**arkup **L**anguage (VRML) file, which can then be played in any Web browser with a VRML plug-in. It will play back as you recorded it. The animation is defined by recording each change to the view in a new **frame** or snapshot.

When you export to VRML, multiple files are created by default, although this can be changed at time of export. The files are pictures of the faces of the objects in the workspace and a file containing the geometric information. Make sure to group the files together when sending them to another person or computer.

Esko recommends the Cortona VRML plug-in, available at `http://www.parallelgraphics.com/products/cortona`.

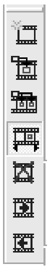
This is an optional feature and must be specifically purchased.

The workflow to making an animation is:

1. Create a single design and add any desired graphics, stock colors, or symbols.
2. Convert the single design to 3D. It should be flat.
3. Click **Add Frame** on the Animation toolbar.
4. Change the fold angles or move/rotate a design, and click **Add Frame** after each change.
5. Repeat until the design is folded and positioned as desired.
6. Export to VRML, setting the desired options in the VRML options dialog box.
7. Open the resulting file in a web browser which has a VRML plug-in installed.

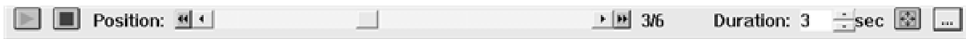
If there are multiple single designs in the 3D workspace, you might achieve better results by turning off **Scale to Fit** on the Status bar while making the animation.

The tools to make animations are on the Animation toolbar and the Animation menu. The Animation toolbar is shown below.



The Status bar

When any of the Animation tools are active, the Status bar contains controls as shown below.

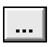


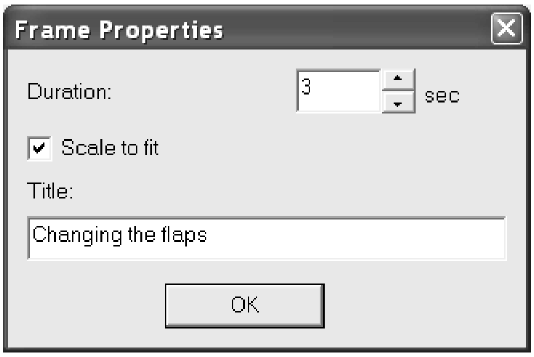
The green triangle button is the **Play** button and starts the animation playback at the current frame. The red square button is the **Stop** button and stops the animation playback.

The numbers between the **Position** slider and the **Duration** field show the current frame and the total number of frames. On the **Position** slider, use the inner direction buttons to move frame by frame; clicking the outer direction buttons moves to the first frame and the last frame respectively. Graphics are turned off while you drag the slider button, but are turned on when you release the mouse button. Graphics are not turned off when the frame-by-frame buttons are used.

The value in the **Duration:** field determines the timing of the animation by controlling the time taken to change from the previous frame to the current frame. The duration for the first frame sets the pause between repeats in VRML Outputs and may be set to 0 for the first frame only. By default, each frame's duration is set to three seconds; you may achieve better results by testing different values. The value in the **Duration**: field is specific to each frame.

 **Scale to fit** may be set for each frame. When clicked, ArtiosCAD centers visible designs in the view and adjusts the view to fit them. Upon playback, the field of view changes smoothly between frames that have this option selected; it may appear that the designs drift, but this is so that the view is centered in each frame that has this option selected.

 The **Options** button opens the Frame Properties dialog box.



Duration: and **Scale to fit** have the same values as their counterparts on the Status bar. Enter text in the **Title**: field to use when exporting the animation to another format. Click **OK** when done making changes to the dialog box.

The animation in a VRML file will use the view angle and elevation set at the time the VRML file is exported from 3D. Changing the view angle and elevation for each frame has no effect.

To save the animation in the corresponding single design, such as saving it to a standard, close the 3D file while the single design is still open. When a single design is created using that standard, the animation settings will be included.

Animation tools

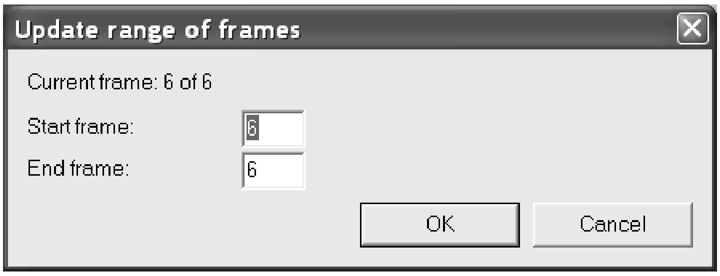
 The first button on the Animation toolbar is **Add Frame**. Use this command to take a snapshot of the current state of the workspace and add it to the animation. For changes to fold angles, you do not need to change each angle incrementally and then capture a new frame. Change the fold angle in the complete amount desired and ArtiosCAD will create the intermediate steps on export. Only changes in fold angles, moves, and rotations are captured in frames. To simulate a change in the view angle or elevation, move or rotate the designs as desired.

 The second button on the Animation toolbar is **Update Frame**. Use this to change the current animation frame to match what is on the screen without adding another frame.

 The third button on the Animation toolbar is **Update Range of Frames**. Use this to update more than one frame at once, for example if you have added another design to the 3D workspace and then moved it after having made an animation. Changes that can be applied to multiple frames are moving designs, rotating designs, changing the visible property on or off, and changing fold angles.

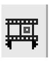
To use this tool, do the following:

1. Make the desired change(s) to the 3D workspace.
2.  Click **Update Range of Frames** to open the Update Range of Frames dialog box.



3. Enter the number of the start frame in the **Start frame**: field, and then enter the number of the ending frame in the **End frame**: field. The current frame must be included in the range.
4. Click **OK** to update the frames.

If designs that were moved or rotated already have different positions in different frames in the animation, ArtiosCAD adjusts the position and rotation of each design by the same relative amount in each frame.

 The fourth button on the Animation toolbar is **Animation Playback**. Use this to step through the animation in 3D by using the slider, or to start playing it at the current frame using the Play button.

 The fifth button on the Animation toolbar is **Delete Frame**. Use this to delete the current frame. This tool is available only when the Animation Playback tool is active.

 The sixth button on the Animation toolbar is **Shuffle Frame Forward**. This tool moves the current frame forward by one frame in the animation sequence. This tool is unavailable for the last frame, or if there is no animation defined. This tool is available only when the Animation Playback tool is active.

 The seventh button on the Animation toolbar is **Shuffle Frame Backward**. This tool moves the current frame back by one frame in the animation sequence. This tool is unavailable for the first frame, or if there is no animation defined. This tool is available only when the Animation Playback tool is active.

Note that when shuffling frames, their duration and Scale to Fit setting are kept with them.

The **Reverse Animation** command on the Animation menu reverses the order of all the frames in the animation. When reversing an animation, note that the first duration is a pause before the animation begins, so an animation with durations 2, 3, 4, 5 would have durations 2, 5, 4, 3 when reversed.

The **Delete Animation** command on the Animation menu deletes the entire animation sequence.

Animation notes and restrictions

The animation feature will not play back adding or deleting designs, or changing the base face. If you delete a design while making an animation, it will disappear from all frames created up to that point.

An animation with **Scale to Fit** turned on and with a background image keeps the background image the same size in ArtiosCAD but the VRML version has a zoomed-in image.

When rotating designs in an animation, do not rotate them through 360 degrees. Break the rotation into three 120 degree steps or four 90 degree steps. Otherwise, the designs may rotate the wrong way during animation playback.

If you find that the design(s) in the VRML file move a lot during playback, try turning off **Scale to Fit** on the Status bar before creating an animation.

Dimensions are not output to an animation, nor are they animated with the designs. They are turned off at the start of the animation playback and turned on when the playback completes.

When an animation has a background image, the **Fit** button in Cortona causes the design to disappear. Use the **Restore** button to bring it back.

Tear-away Animations

Tear-away animations show how the design separates. They are different from normal animations and there is a dedicated tool to make them.

Tear-away animations require that the panels can bend. If the panels do not bend, ArtiosCAD makes a pull-away animation instead in which you can only set the pull-off duration and distance of the pulled-away part.

Note: Tear-away animations only work for the first instance of a design; they do not work on copies.

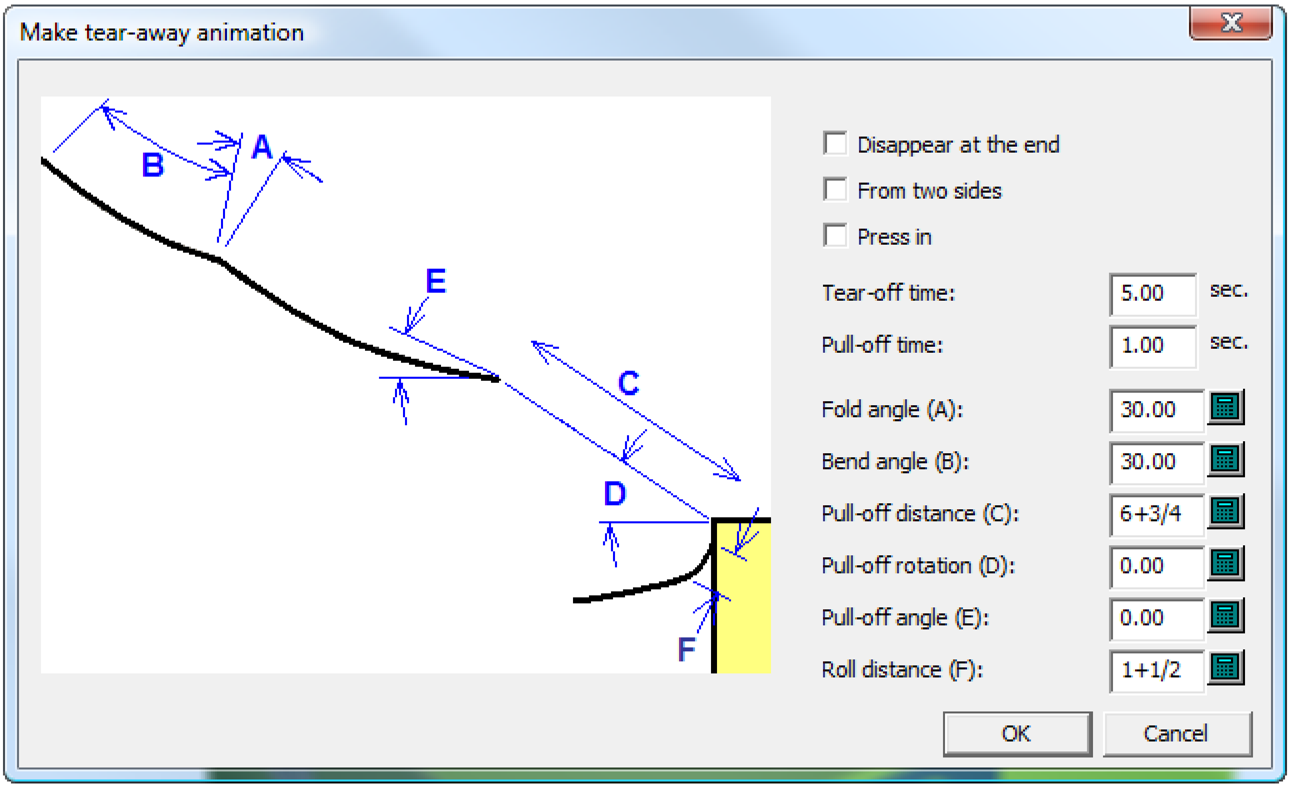
Making a Tear-away Animation

To make a tear-away animation, do the following:

1. Create a 3D workspace and make sure all the designs you want in it are arranged properly. The designs must also have the 3D property of the tearing lines set correctly in Single Design. For example, if you want to show potato chip bags in a retail-ready shipping container, arrange the bags inside the container and fold the container so that it appears closed.

2.  Click the **Make tear-away animation** tool and click where you want the tear-away to start in the part that is torn away. In the example below, the cursor is in the thumb hole and you can see the perf forming the left part of the tear.

3. Set the fields as desired in the Tear-away animation dialog box.



a) **Disappear at the end** causes the torn-away piece to disappear from the scene when the animation finishes.

b) **From two sides** shows two ends of the tear-away being torn away simultaneously before the tear-away is detached from the middle. Use this option only when you have specifically designed the tear-away to start in two places at once.

c) **Press in** causes a beginning thumb notch or other similar construction to be pressed into the container before it is torn away.

d) **Tear-off time** and **Pull-off time**, respectively, set the duration of those parts of the animation.

e) **Fold angle (A)** sets the angle at which a fold in the part being torn away is folded at the end of the animation.

f) **Bend angle (B)** sets the angle at which a bend in the part being torn away is bent at the end of the animation.

g) **Pull-off distance (C)** controls how far the torn-away piece is pulled from the main piece of the design at the end of the animation.

h) **Pull-off rotation (D)** determines the rotation at which the torn-off piece turns.

i) **Pull-off angle (E)** determines the direction in which the torn-off piece moves.

j) **Roll-off distance (F)** sets how far the torn-away pieces rolls off the main body and thus how tight the bend is. If you have to pull hard to do the tear-away in real life, set a small distance to have a tight bend.

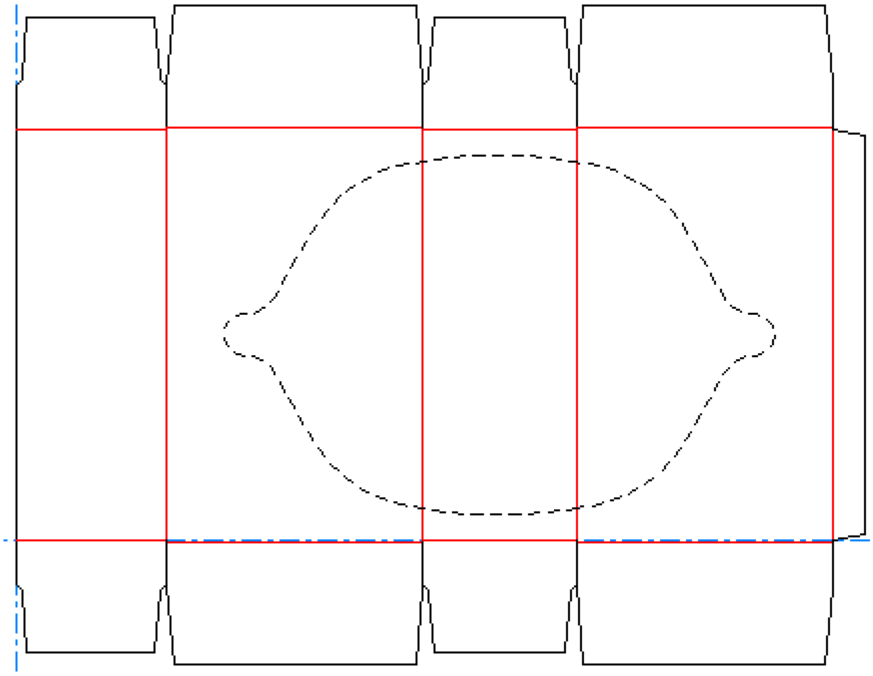
4. Click **OK** when you are done setting the values as desired.

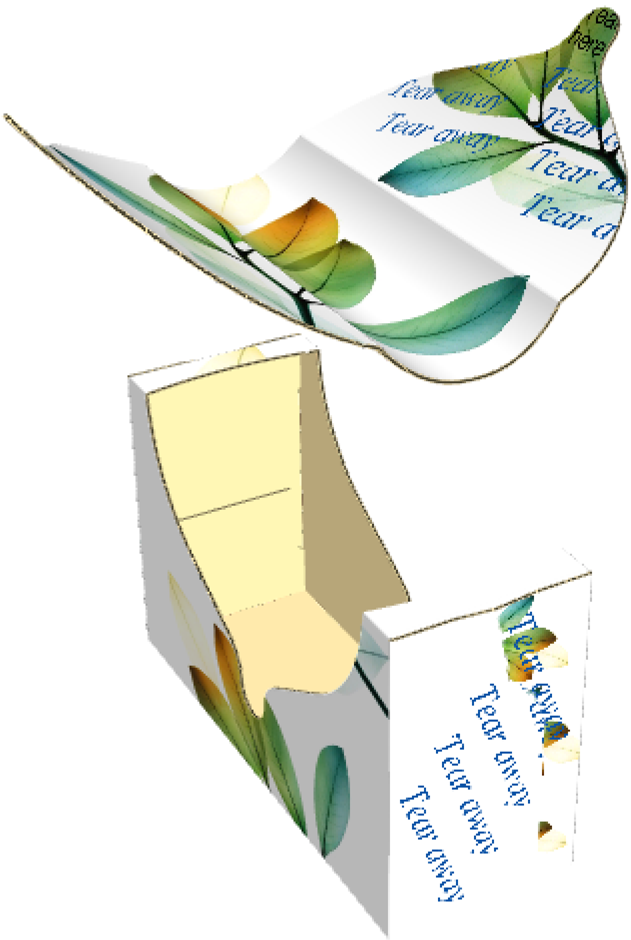
ArtiosCAD creates the animation and starts the **Animation Playback** tool. Click the green triangle to play the animation.

Note: If you use **Scale to Fit** on any of the animation frames, you will not be able to make any changes to the animation.

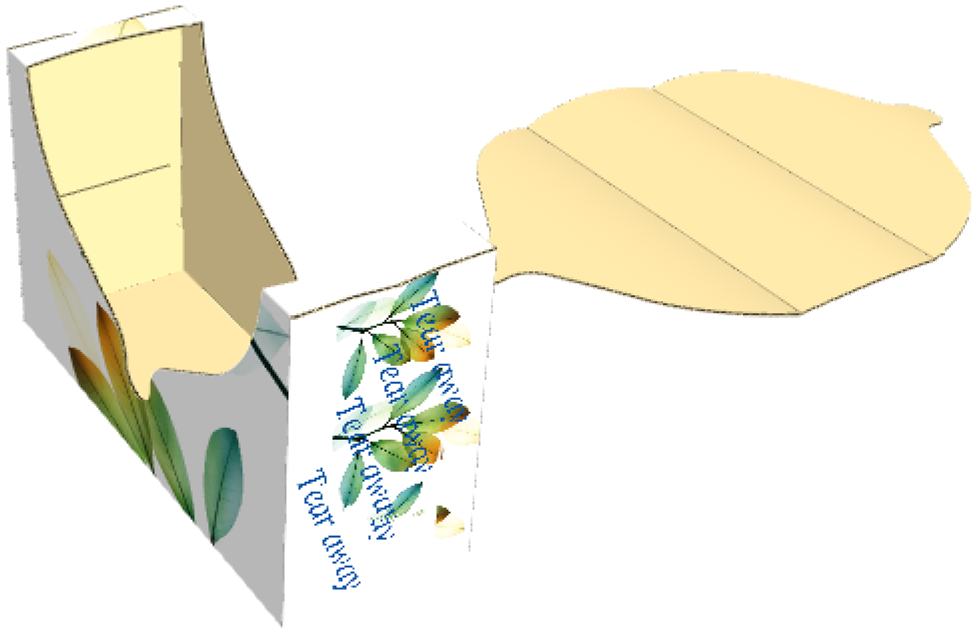
Example of Tearing from Two Sides

Make sure to check **From two sides** in the Make tear-away animation dialog box if you have designed a box similar to the one shown below.



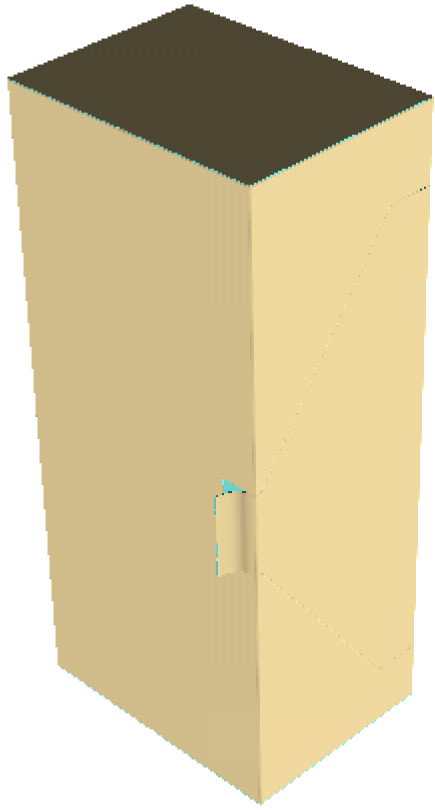

Tear away
Tear away
Tear away
Tear away
Tear away

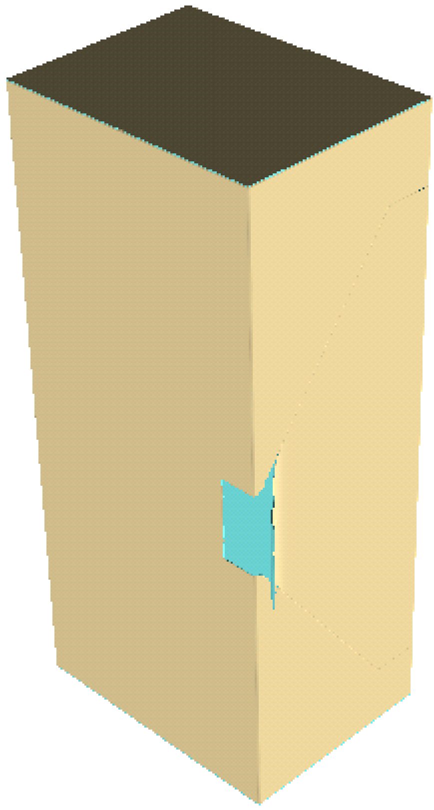
If you do not check the **From two sides** checkbox, the result will look something like this:

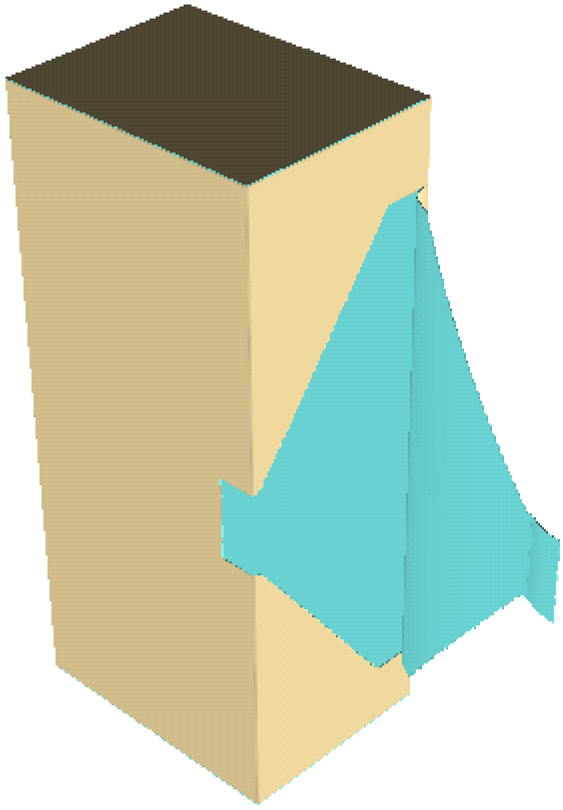


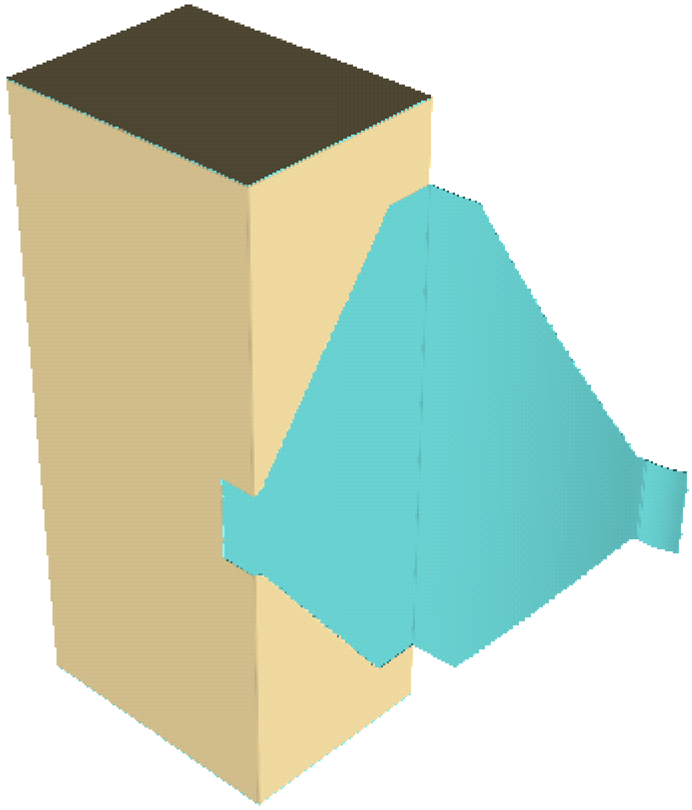
Example of an Attached Tear-Away Animation

Tear-away animations do not actually have to be torn away - as long as the design has lines that will tear in it, you can show them tearing. The example below has a flap edged with perf and ending on a crease. This allows access to its contents, but also means the box can be closed after its contents are removed.







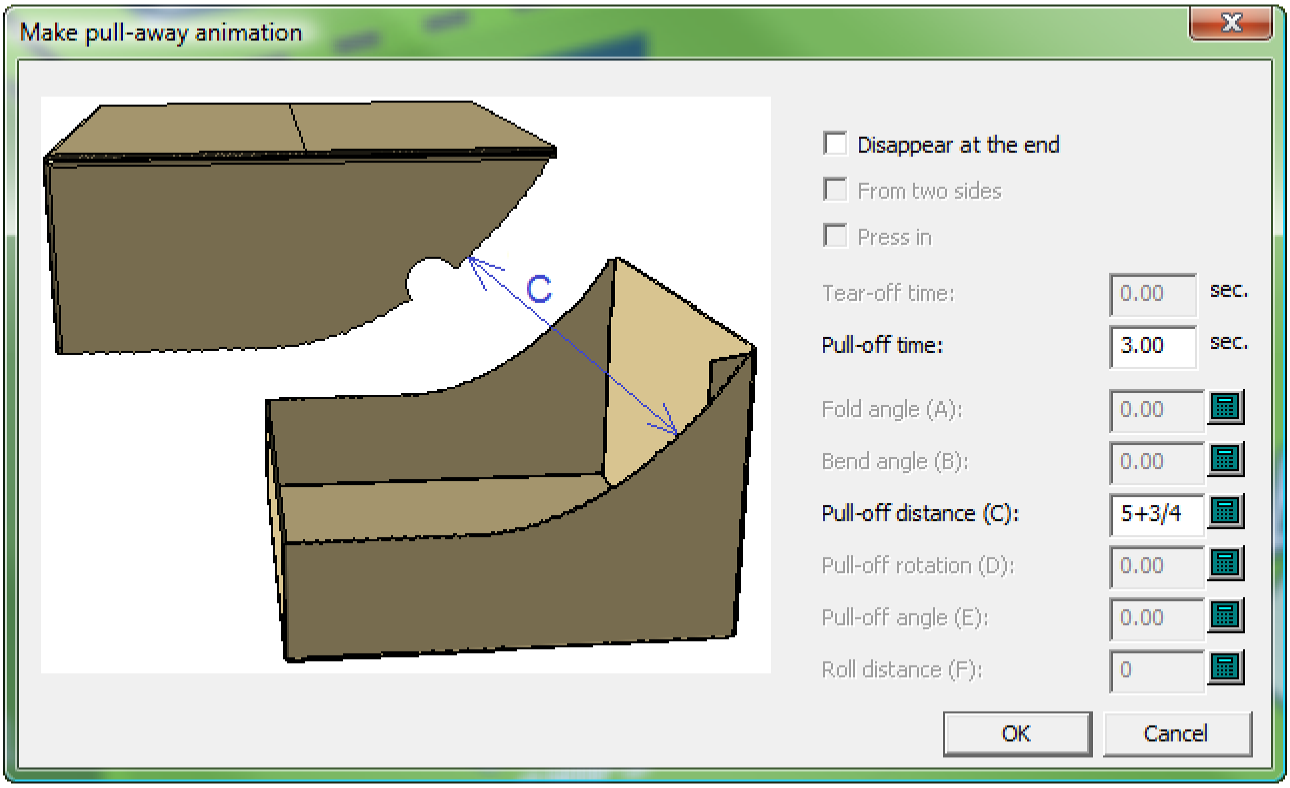


Making a Pull-Away Animation

If the panels of the box do not bend, ArtiosCAD removes the torn-away part from the main body of the container in a straight line.

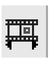
1. Design the container, making sure to set the 3D property of the tearing lines to **Tear** or **Tear and fold**.
2. Convert the design to 3D.
3. Click **Make tear-away animation** and then click the start point for the animation.

4. In the Make pull-away animation dialog box, set the options as desired and click **OK**.



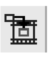
5. ArtiosCAD creates the animation and starts the **Animation Playback** tool. Click the green triangle to play the animation.

6. To change the direction in which ArtiosCAD pulls the part away, do the following:

a)  Use **Animation Playback** to go to the last frame of the animation.

b)  Use **Select designs** to select the torn-away part.

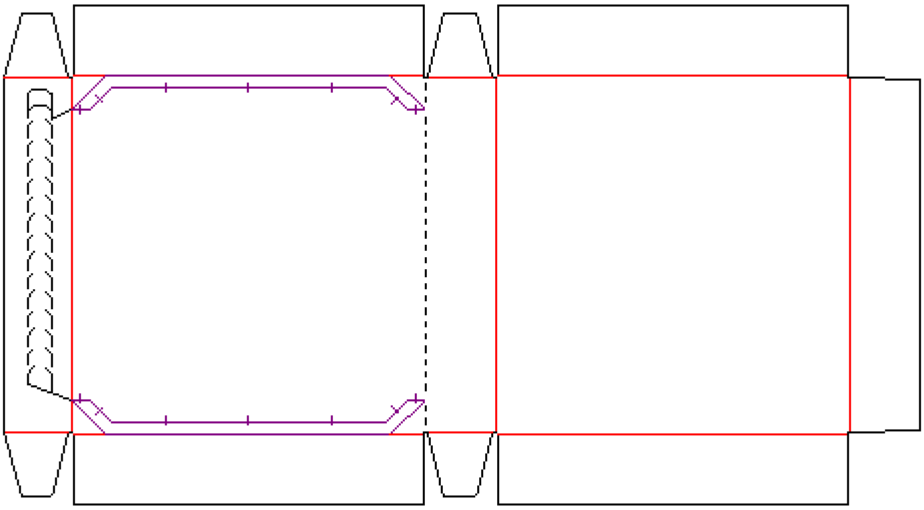
c)  Use **Move Designs** to move it in the desired direction.

d)  Click **Update Frame**.

e) When you replay the animation with the **Animation Playback** tool, the pulled-away part will move in the direction you specified.

Substrate Tears

Some designs use partial cuts and reverse partial cuts instead of perfs to tear. This type of tear is called a *substrate* tear, and is shown below as the lines in the second panel that connect the zipper rule to the perf.



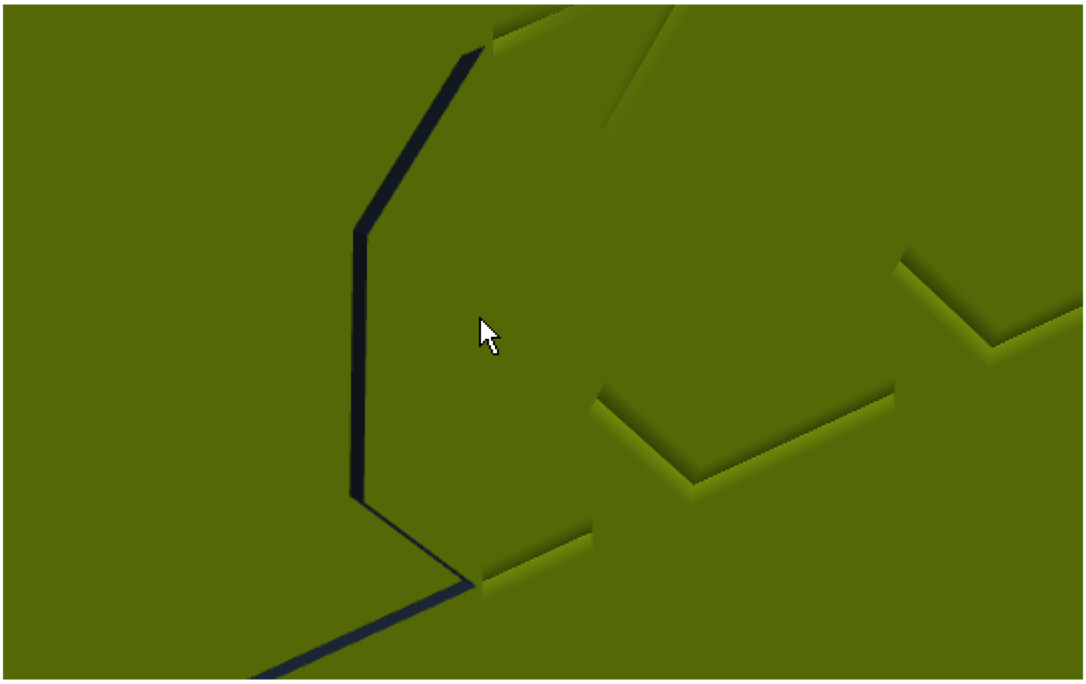
In order for 3D to recognize a substrate tear:

- The substrate tear is made by partial cuts and reverse partial cuts
- The partial cuts and reverse partial cuts have the **Tear** option selected on the 3D tab of the Properties dialog box in Single Design
- There are no gaps in the lines
- The substrate tear area is a maximum of 1/2 inch or 12.7 millimeters wide.

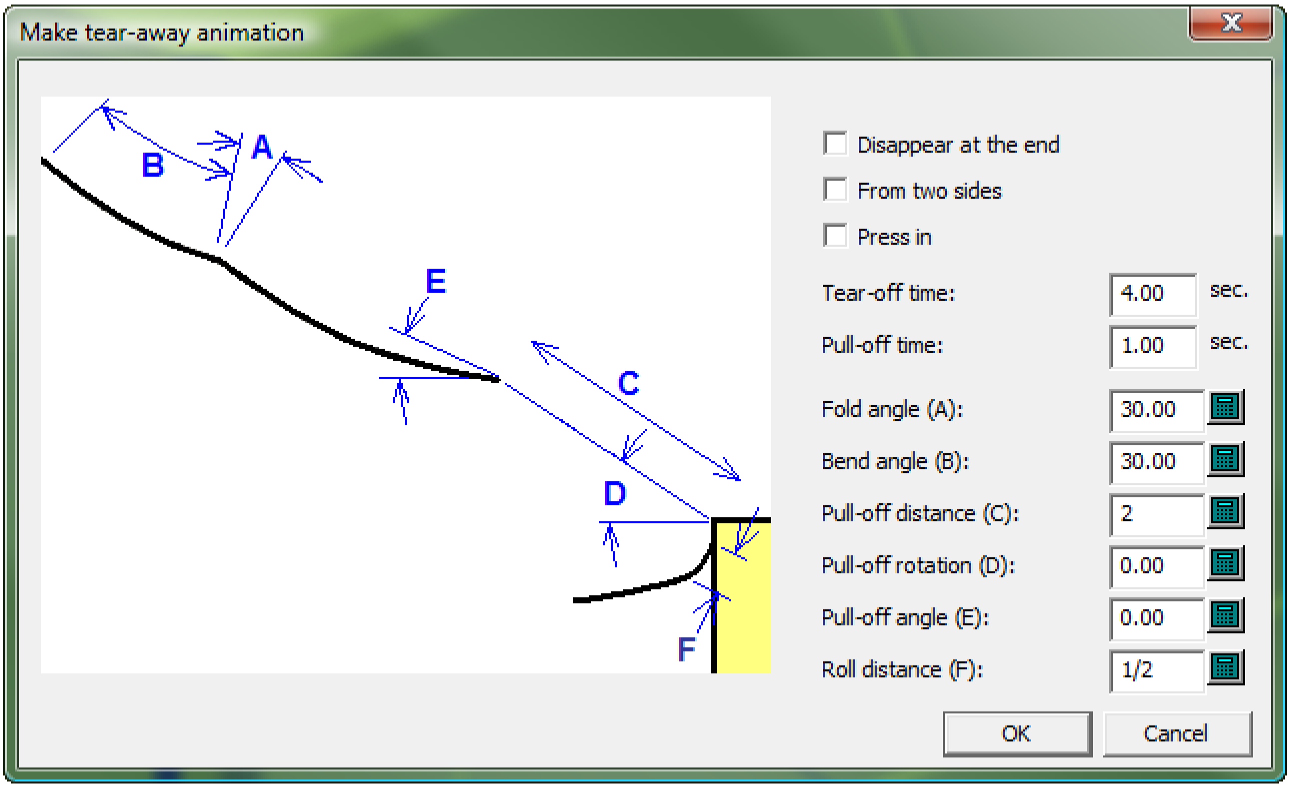
Example of Two Tear-away Animations

To have two tear-away animations in the same design, use **Make tear-away animation** twice consecutively.

1. Design the container, making sure to set the 3D property of the tearing lines to **Tear** or **Tear and fold**.
2. Convert the design to 3D.
3. Click **Make tear-away animation** and then click the start point for the first animation.



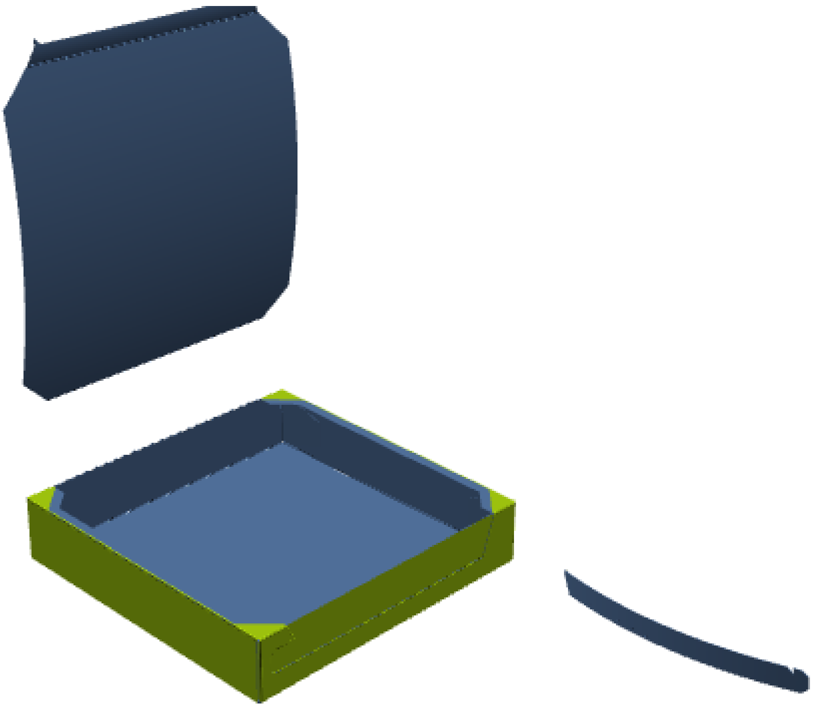
4. In the Make tear-away animation dialog box, set the values as desired and click **OK**.



5. Click **Make tear-away animation** and then click the start point for the second animation.

6. In the Make tear-away animation dialog box, set the values as desired and click **OK**.

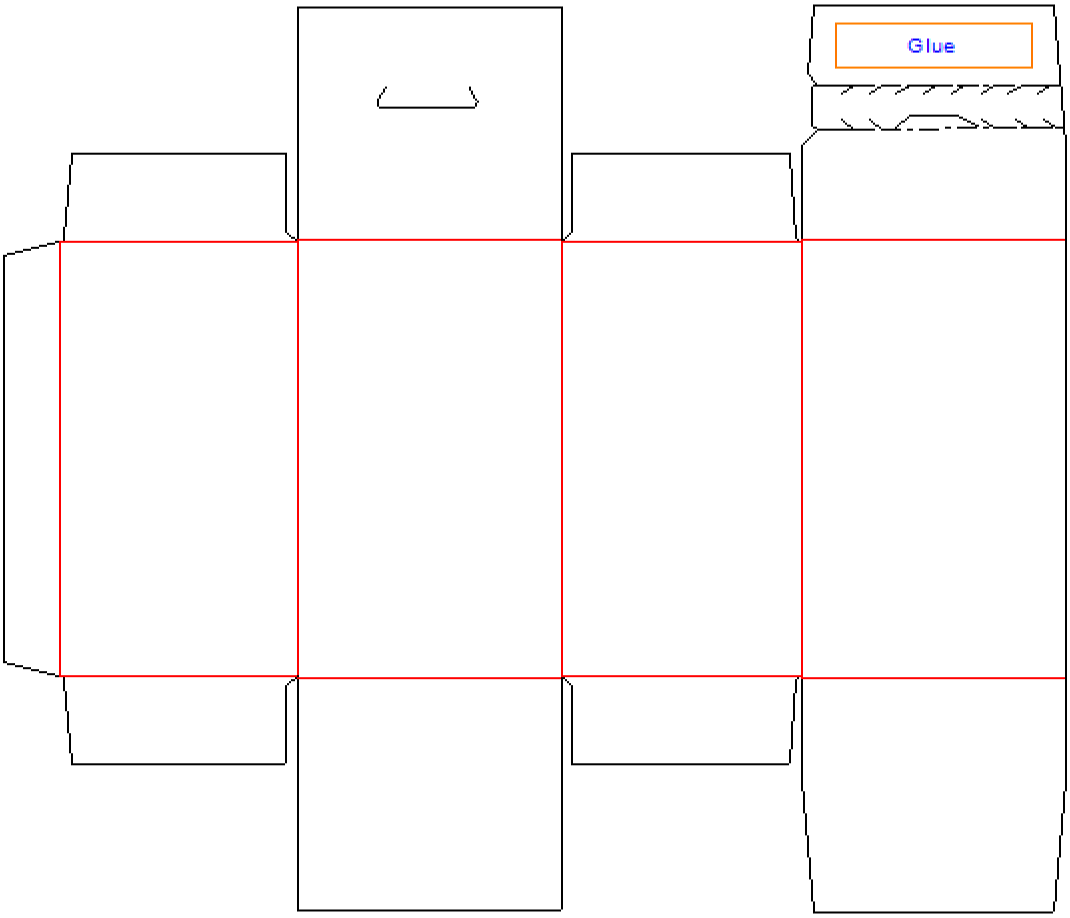
ArtiosCAD makes the second animation and starts the Animation Playback tool. Slide the frame slider all the way to the left to start at the beginning, and then click the green triangle to play both animations.



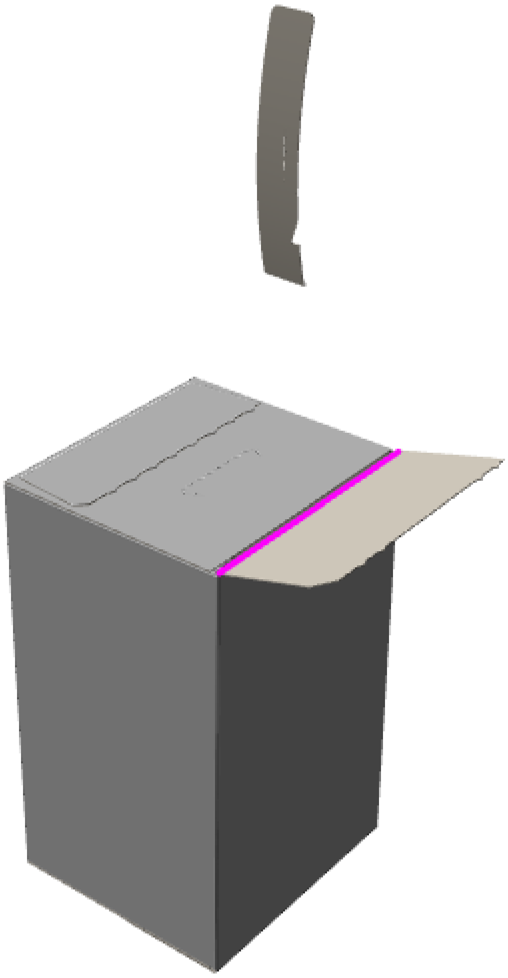
Example of a Tear Strip with an Attached Part

Usually torn-away parts are separated from the main part of the design. To leave the remainder of a part attached after part of it is torn off, such as a flap with a zipper glued to it, do the following:

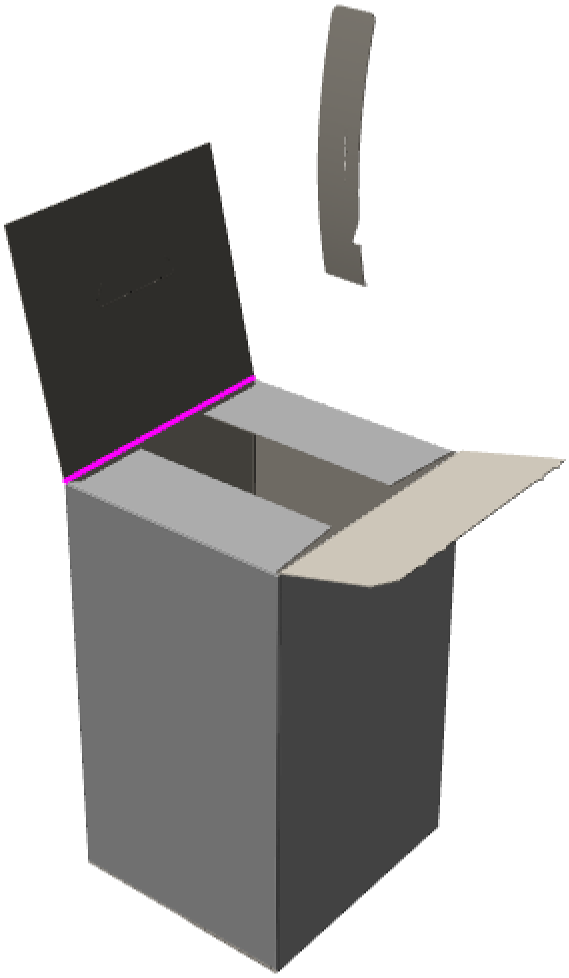
1. Design the container, making sure to set the 3D property of the tearing lines to **Tear** or **Tear and fold**.
2. Make a Mate area on the part of the design that will remain after the tear-away is torn off. Only one Mate area is needed; ArtiosCAD automatically creates the target area in 3D.



3. Convert the design to 3D.
4.  Click **Make tear-away animation** and then click the start point for the animation. ArtiosCAD creates the animation and activates the **Animation Playback** tool.
5. In the **Animation playback** tool, if the frame slider is not already on the last frame, slide it all the way to the right to go to the end of the animation.
6.  Use **Fold Angle** to unfold the flap that does not have the Mate area.



7.  Click **Add Frame**.
8.  Use **Fold Angle** to unfold the opposite panel, the one that has the Mate area. Note that the part with the Mate area is still attached and was not torn away.



9.  Click **Add Frame** to complete the animation.

You can edit the duration of each frame as desired in the **Animation Playback** tool.

Adding a Tear-away Animation to an Existing Animation

If you already have an existing animation when you use **Make tear-away animation**, ArtiosCAD adds frames to the existing animation. For example, you could create the following animation:

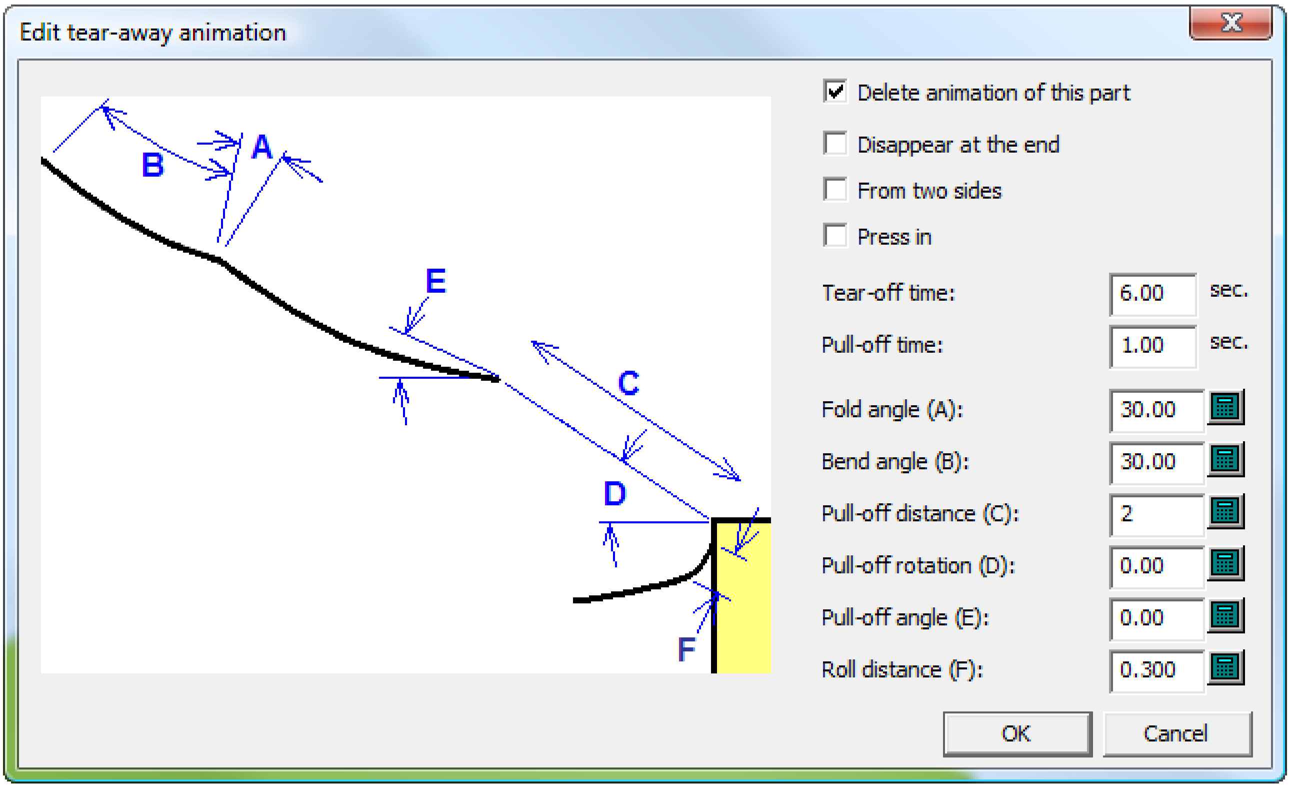
1. Fold a container, leaving the top open.
2. Put contents in the container.
3. Close the container.
4. Tear away the top of the container.
5. Remove the box contents.

To accomplish this, make steps 1, 2, and 3 manually, then use **Make tear-away animation** for step 4, and then add step 5 manually.

Deleting a Tear-away Animation

To delete an entire tear-away animation, click **Animation** > **Delete animation** . To delete just the tear-away portion of the animation, do the following:

1.  Click **Make tear-away animation**.
2. Click the part to delete.
3. In the Edit tear-away animation dialog box, check **Delete animation of this part** and click **OK**.



ArtiosCAD deletes the animation for the part you selected.

3D Designer

Use **3D Designer** to make a solid of revolution. A **solid of revolution** is a flat ArtiosCAD design with special layers defined. When converted to 3D, cross-sections and graphics defined in the single design workspace are rendered as a solid model. Shown below in the top pane is a flat single design., and in the bottom pane is the 3D workspace with the solid of revolution.

Solids of revolution

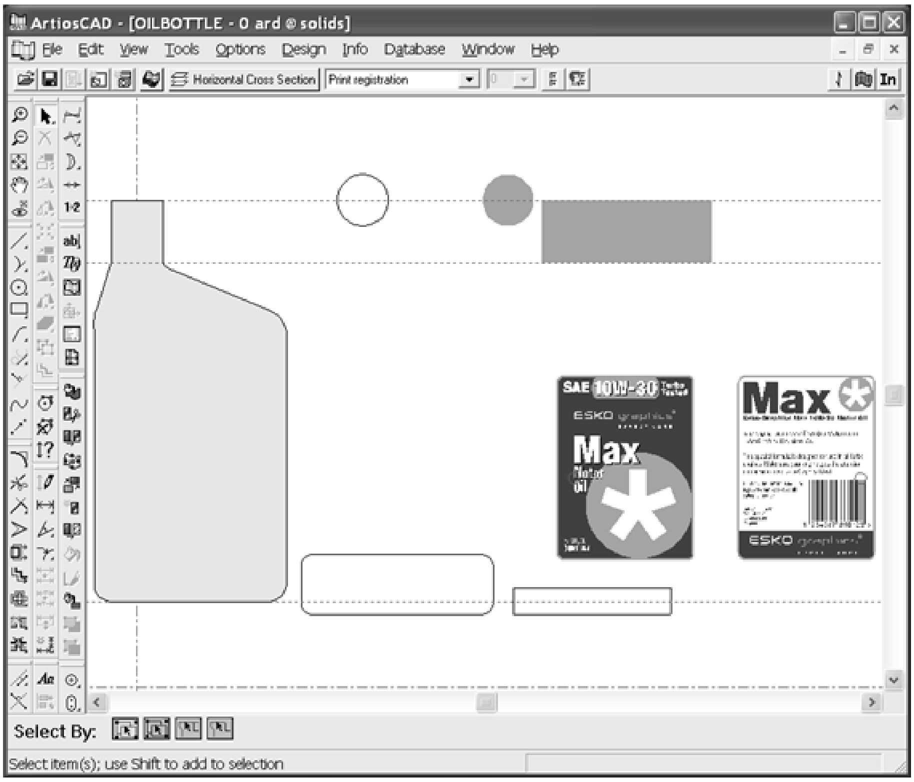
Working with the single design

To be converted to a solid of revolution, a single design has these properties:

- A vertical cross section positioned on the Y axis of the workspace in the **Main Design** layer (required)
- A color defined (optional)
- A set of horizontal cross sections at different heights along the vertical axis in a layer of class **Horizontal Cross Section** (optional)
- Graphics for labels defined in a layer of class **Label Graphics** (optional)
- Shrink-wrap graphics defined in a layer of class **Shrink Wrap Graphics** (optional)

Note: At least one layer of the three layer classes described above must exist for the design to be converted to a solid of revolution. The layer can be empty. Solids of revolution may not have holes that go completely through them.

Shown below is the single design workspace for the oil bottle. On the left is the vertical cross section in the **Main Design** layer. To the right of that, the open circle and two open rectangles are horizontal cross sections in the **Horizontal Cross Section** layer. To the right of that, the filled-in circle, filled-in rectangle, and two graphics are in the **Label Graphics** layer.



Vertical cross section

The vertical cross section is an outline of the shape using cut lines which straddles the Y axis in the **Main Design** layer. If it is filled with color (using the **Fill** tool), the color becomes the surface color when the single design is converted to a solid of revolution. The vertical cross section does not have to be symmetrical, but the solid of revolution will revolve around the Y axis, so place the vertical cross section accordingly.

Horizontal cross section

Horizontal cross sections must be in a layer of class **Horizontal Cross Section**. If there is no layer of this class, the solid of revolution will have a round cross section. Each horizontal cross section is an outline made of cut lines of the exterior shape of the solid of revolution.

The center of the horizontal cross section's bounding box is aligned vertically with the corresponding point in the vertical cross section. To use a different alignment point, use print registration lines to make a small cross at the desired alignment point. Non-symmetrical shapes may require alignment points for proper positioning.

The horizontal placement of the horizontal cross sections within the single design workspace is not important, but they must align vertically at the correct height along the vertical cross section, and none of the horizontal cross sections should overlap or touch.

If the size of the horizontal cross section differs from that indicated by the vertical cross section, upon conversion to a solid of revolution, the width of the horizontal cross section is scaled to match that of the vertical cross section. A warning dialog box will appear if the size difference is greater than 33 percent.

Graphics for labels

Graphics for labels must be in a layer of class **Label Graphics**. There can be more than one graphic in the same layer. Each label graphic should be a single group.

As with horizontal cross sections, the alignment point for a label is the middle of the bounding box for the graphic. To use an off-center alignment point, make a small cross or circle from print registration lines and group it with the graphic. This point is then aligned with the center of the solid.

Graphics in this layer are positioned on the solid of revolution as follows:

- If it extends above the top of the vertical cross section, the graphic is centered on the top.
- If it extends beneath the bottom of the vertical cross section, the graphic is centered on the bottom.
- If it is positioned between the top on the bottom of the vertical cross section, it is centered on the front.
- If there are two graphics that overlap vertically, the graphic on the right goes on the back.

Shown below are the label graphics for the oil bottle. Note the print registration circles through the M and O on the front label and through the bar code on the back label. They are aligned with the Y axis and thus ensure the labels are placed properly.

Shrink-wrap graphics

Shrink-wrap graphics must be in a layer of class **Shrink Wrap Graphics**. A shrink-wrap graphic wraps around the front and back of the design; it is not intended to cover the top or bottom. A shrink-wrap graphic must be rectangular. If it is made of multiple parts, group them together into a single group.

As with label graphics, the center of the bounding box for the shrink-wrap graphic defines the vertical position of the graphic on the solid of revolution. Any portion of the graphic extending beyond the top or bottom of the vertical cross section is discarded.

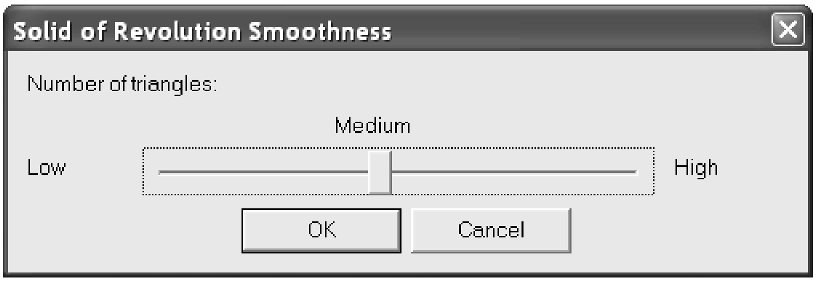
If the shrink-wrap graphic is not wide enough to wrap around the design, it is horizontally scaled to do so.

Do not mix label graphics and shrink-wrap graphics; use one or the other.

Do not turn on a Label Graphics layer if using shrink-wrap graphics.

Converting the single design to a solid of revolution

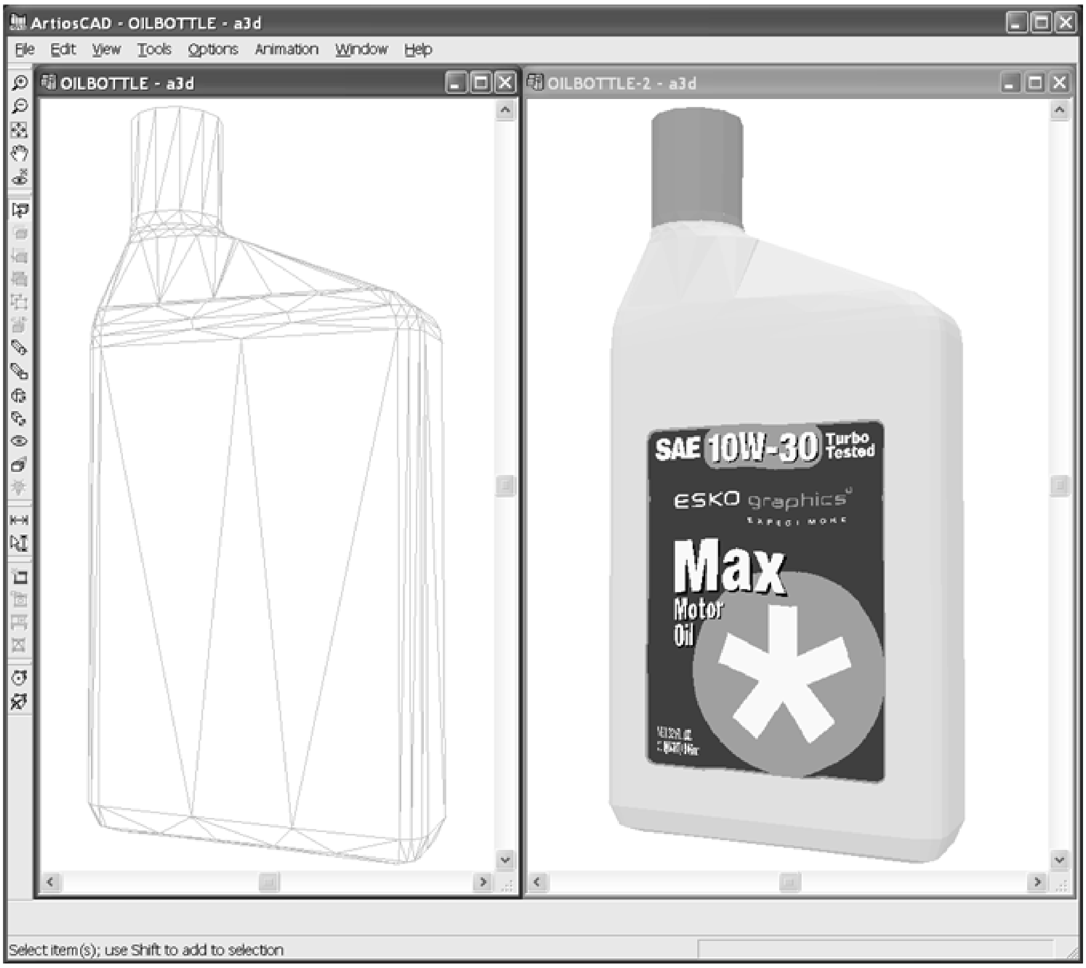
Once the single design is constructed with the layers defined and graphics added, click **File** > **Convert to 3D** > **Convert to 3D** button on the View bar. The Solid of Revolution Smoothness dialog box appears as shown below.



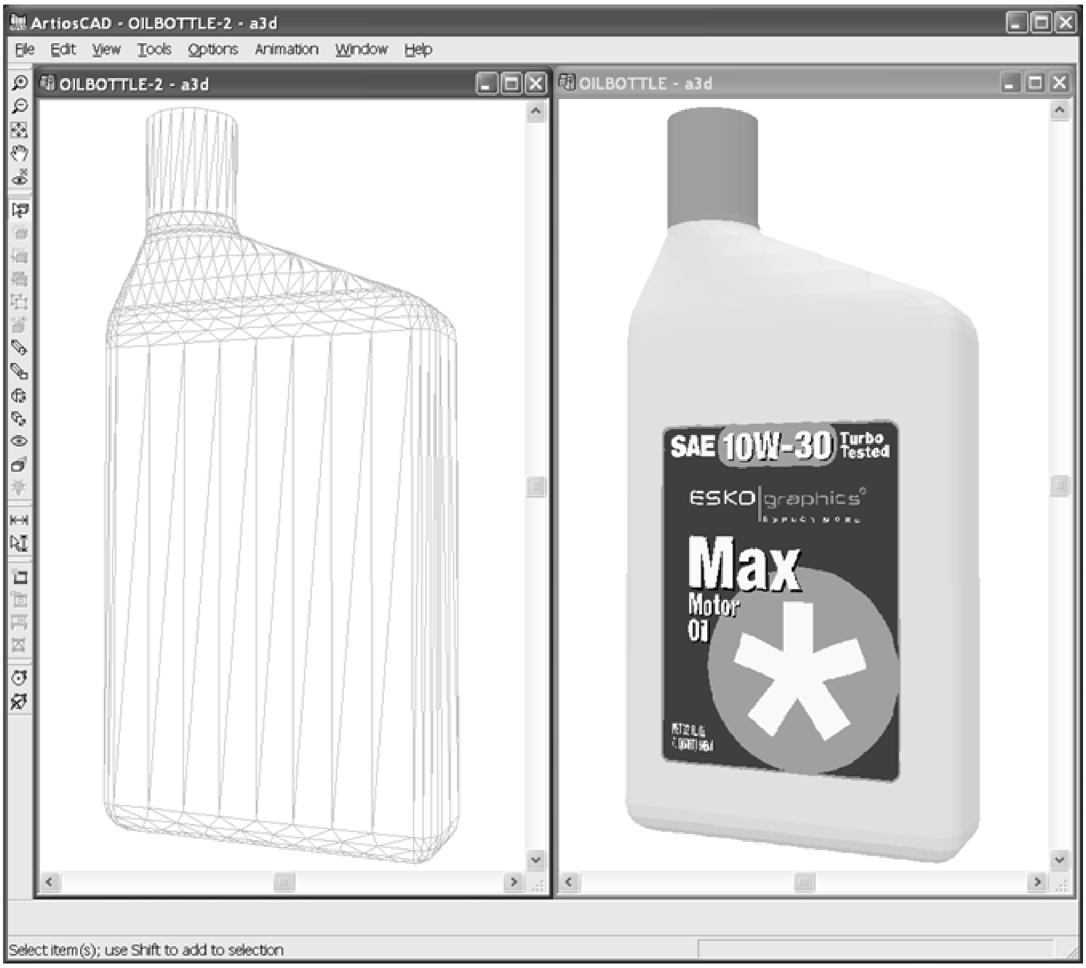
Solids of revolution are composed mainly of triangles. The quality and smoothness of the solid of revolution is directly proportional to the number of triangles. **However, the more triangles, the longer it will take to render the design and the longer it will take drag to refresh.**

Medium smoothness is recommended for most solids of revolution. The smoothness can not be adjusted once the solid of revolution is created. To adjust the smoothness, discard the 3D workspace and reconvert the single design, choosing a different smoothness when prompted.

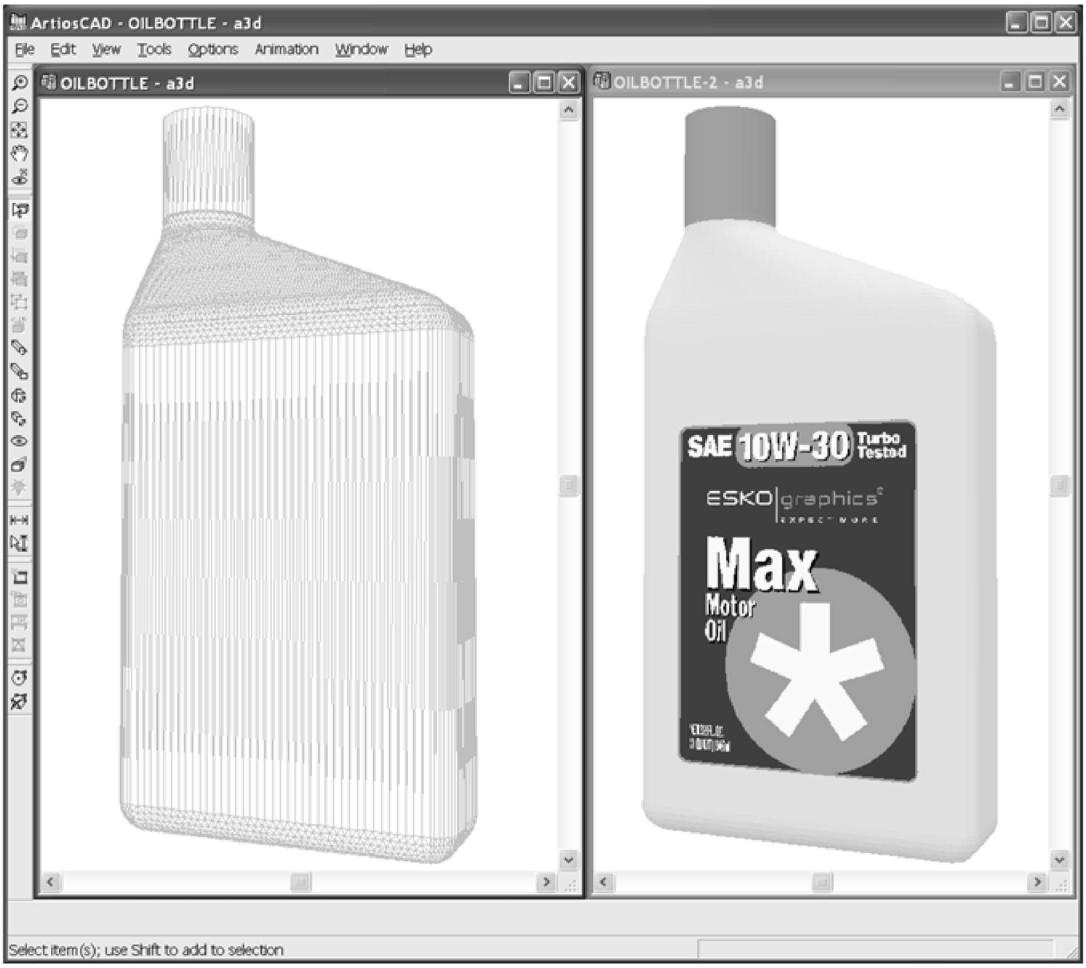
To set the smoothness, adjust the slider in the Solid of Revolution Smoothness dialog box as desired and click **OK**. The solid of revolution is created and presented in wireframe form. Shown below is the oil bottle at low smoothness in both wireframe and rendered forms.



At medium smoothness:



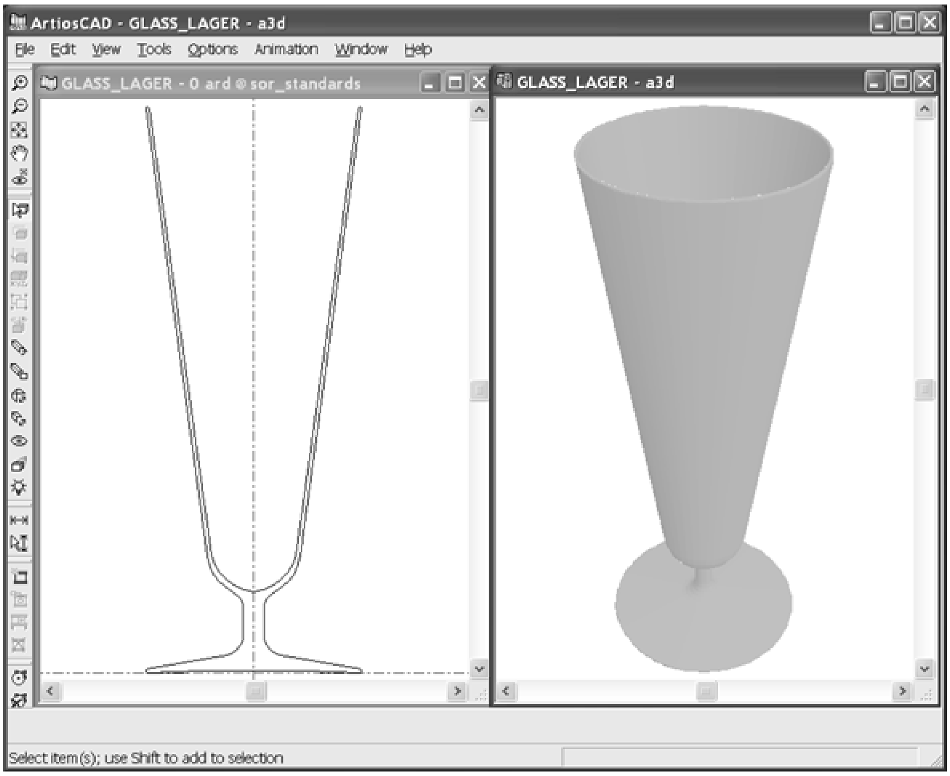
At high smoothness:

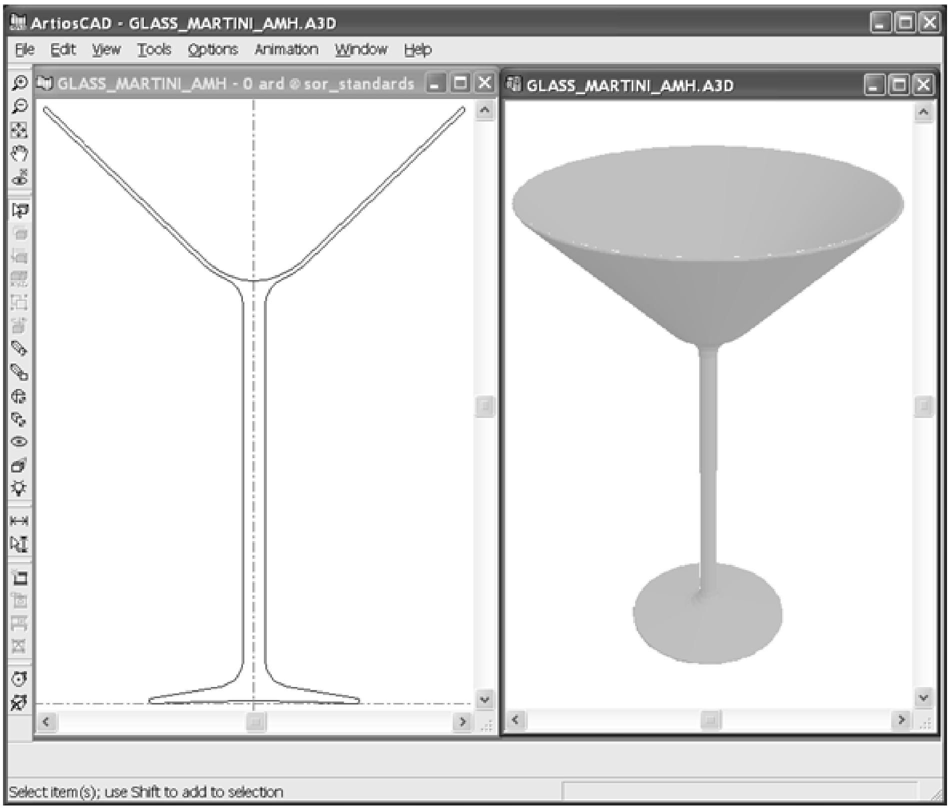


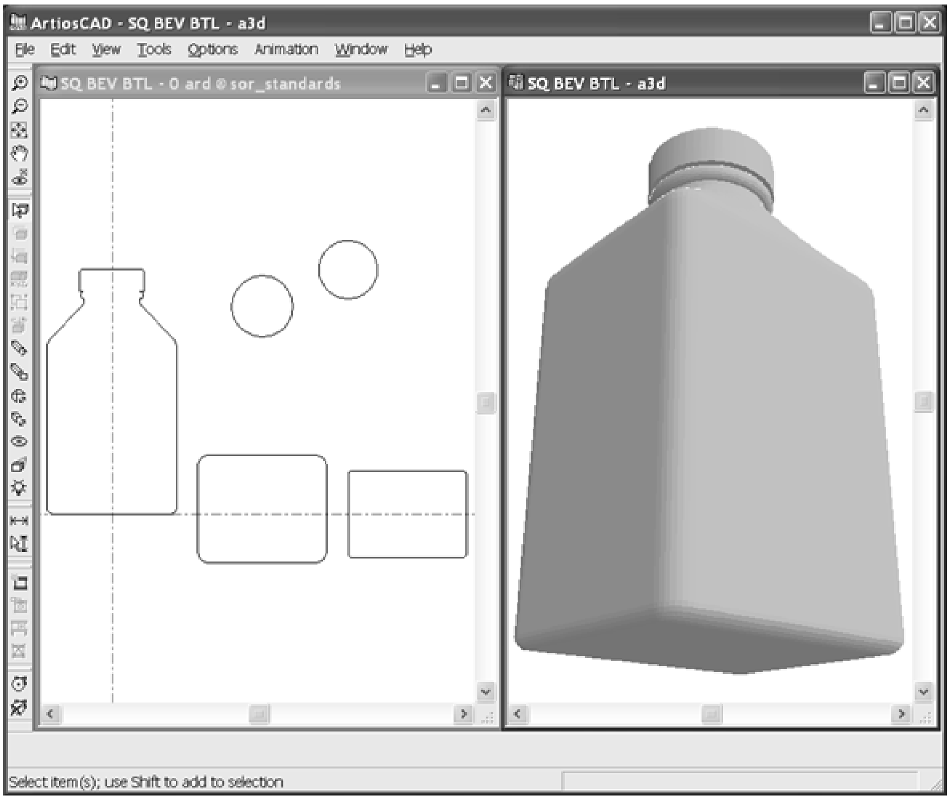
Once the solid of revolution is created, treat it like any other 3D object. Note that to snap to it, **Snap to solids** must be checked in **Options** > **Snap Options**.

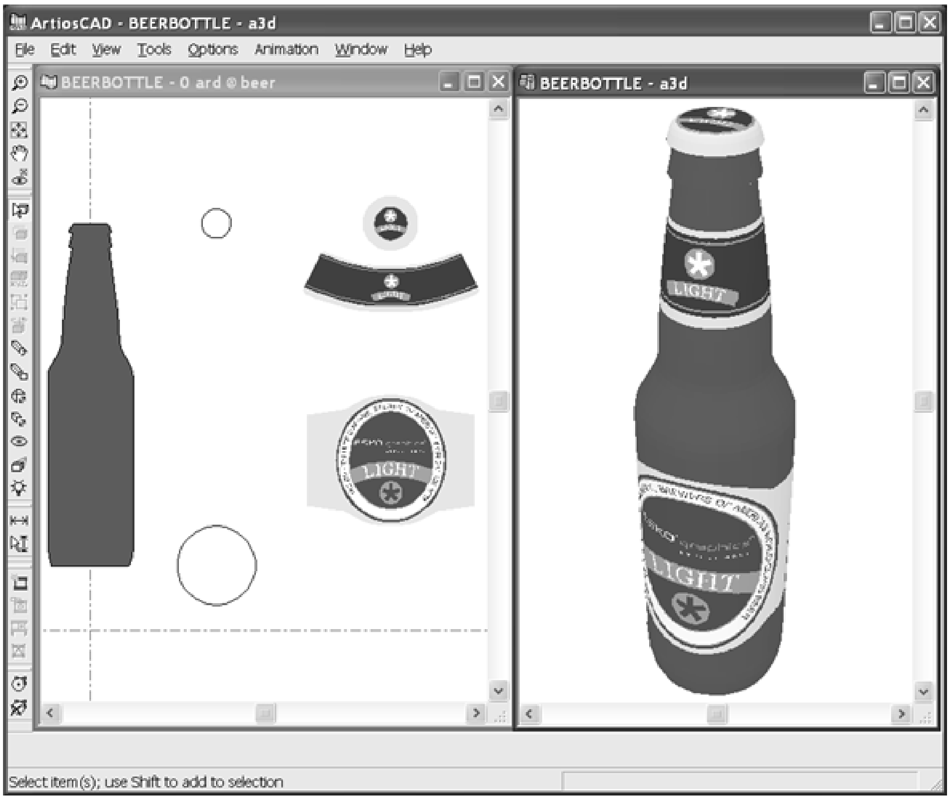
Examples

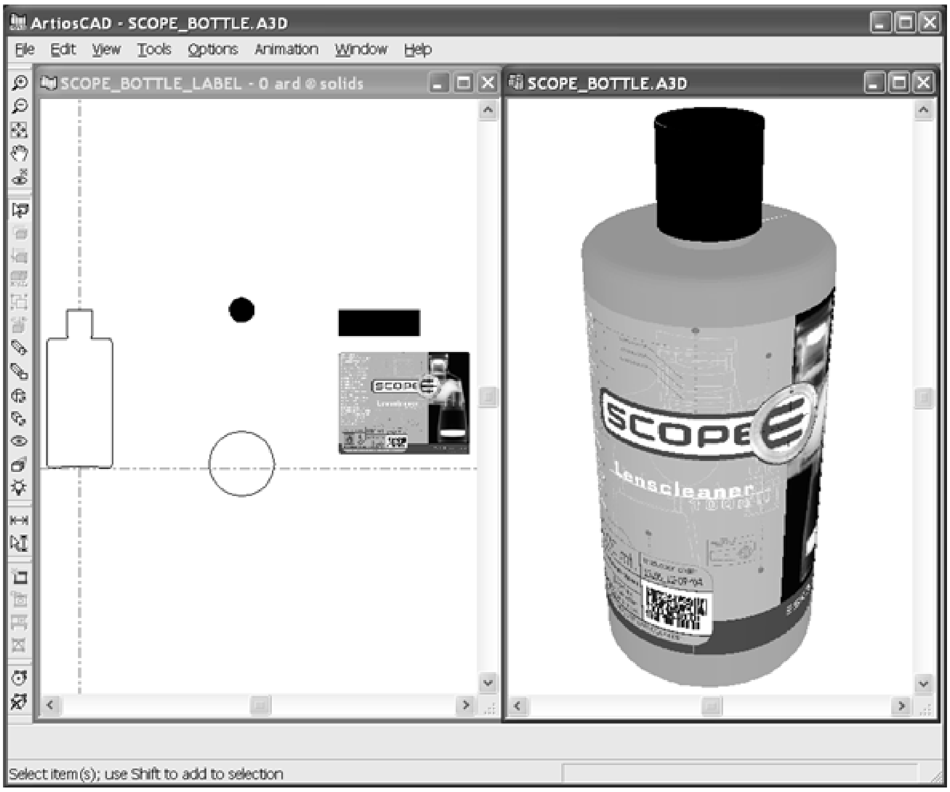
The power of ArtiosCAD lends itself to many different kinds of solids of revolution. Solids of revolution can have lines that reverse direction, such as glasses. They also can be made from resizable designs - resize the design and create a completely different shape. Both the lager glass and the martini glass shown below were made from the same single design workspace.


ArtiosCAD - GLASS_LAGER - a3d
File Edit View Tools Options Animation Window Help
GLASS_LAGER - 0 ard @ sor_standards
GLASS_LAGER - a3d
Select item(s); use Shift to add to selection


ArtiosCAD - GLASS_MARTINI_AMH.A3D
File Edit View Tools Options Animation Window Help
GLASS_MARTINI_AMH - 0 ard @ sor_standards
GLASS_MARTINI_AMH.A3D
Select item(s); use Shift to add to selection


ArtiosCAD - SQ BEV BTL - a3d
File Edit View Tools Options Animation Window Help
SQ BEV BTL - 0 ard @ sor_standards
SQ BEV BTL - a3d
Select item(s); use Shift to add to selection


ArtiosCAD - BEERBOTTLE - a3d
File Edit View Tools Options Animation Window Help
BEERBOTTLE - 0 ard @ beer
BEERBOTTLE - a3d
LIGHT
Select item(s); use Shift to add to selection



Solids of revolution notes and warnings

Use medium smoothness whenever possible instead of high smoothness, as solids of revolution made with high smoothness takes a long time to render and manipulate, even in wire frame view.

Do not make handles on single designs you intend to convert to solids of revolution. Make a handle as a separate design and add it to the 3D workspace.

Other solids

3D Designer also can import many different 3D models from other programs. Each format is an option that must be purchased, except for ACIS and VRML which are included with 3D Connection and 3D Designer respectively.

Table: Importable Solid File Formats

Format	File types supported
VRML	.wrl
ACIS and STEP	.acis, .sat, .step, .stp, .asat, .asab
IGES	.igs, .iges
CATIA version 4	.model, .exp, .session

Format	File types supported
CATIA version 5	.CATpart, .CATProduct
ProEngineer version 17 - version 2001, Wildfire versions 1 and 2	.prt, .prt.**x**, .asm, .asm.**x**
SolidWorks	.sldprt, .sldasm
Inventor	.ipt, .iam
Unigraphics	.prt

Importing VRML files

VRML files contain polygons, lines, and images. ArtiosCAD can directly import the polygons from most VRML 1.0 and 2.0 files created by other CAD programs. ArtiosCAD also resolves links to external files and uses them if it finds them.

The tables below list what is supported and unsupported, with features and entities being in normal type and keywords being in bold.

Table: Supported VRML features, entities, and keywords

Polygons	Surface color	Shininess
gzip compression	DEF/USE	Units from ArtiosCAD VRML export
scale	**Transform**	**Cube**
translation	**Group**	**Cylinder**
rotation	**Separator**	**Sphere**
matrix	**children**	**Cone**
Shape	**geometry**	**width**
Material material	**diffuseColor**	**height**
Appearance appearance	**ambientColor**	**depth**
coord	**emissiveColor**	**radius**
point	**specularColor**	**bottomRadius**
Coordinate	**ambientIntensity**	
Coordinate3	**shininess**	
coordIndex	**transparency**	

Table: Unsupported or Ignored VRML features, entities, and keywords

Animations	JavaScript	Lines
Points	Images	
WorldInfo	**ShapeHints**	**vector**
NavigationInfo	**Texture2Transform**	**solid**
Background	**IndexedLineSet**	**eventIn**
Viewpoint	**Normal normal**	**eventOut**
exposedField	**ROUTE**	**WWWInline**

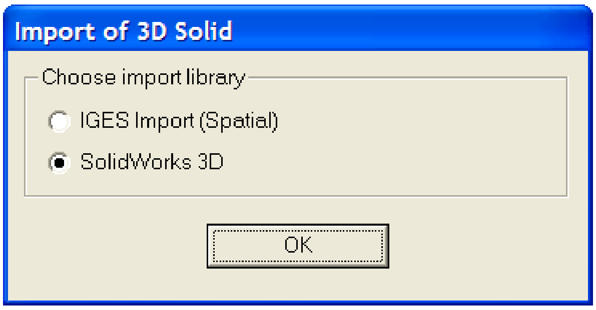
If the directory **C:\TEMP** exists, ArtiosCAD puts a log file called **VRMLLOG.TXT** in it for diagnostic purposes each time a VRML file is imported. The file is overwritten each time.

Importing non-VRML files

When you import an ACIS, STEP, IGES, CATIA, ProEngineer, or SolidWorks file, ArtiosCAD performs three steps before the workspace opens in the design window:

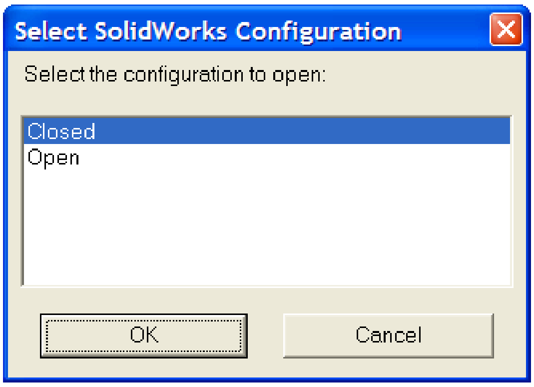
1. The file is preprocessed by either the Spatial or SolidWorks library to convert the 3D data into polygons. If it is using the Spatial library, see the next section for more information.
2. The polygons are written to file <system temp directory>\TEMP.MSH or TEMP.WRL depending on the library used.
3. ArtiosCAD reads the temporary file and opens it in a 3D workspace.

When importing an IGES or ProEngineer file, depending on the options set in Defaults, ArtiosCAD prompts you to choose the conversion library. While the Spatial library is included in the 3D Connection license, you must have purchased the SolidWorks option separately and installed it in order for that method to work.



Choose the desired conversion library and click **OK**.

SolidWorks files can contain multiple configurations. If opening such a file, select the desired configuration and click **OK**.



SolidWorks assembly files (ending in **.sldasm**) may also contain references to other parts files (ending in **.sldprt**). If any parts file is missing, ArtiosCAD ignores the missing parts and continues the import.

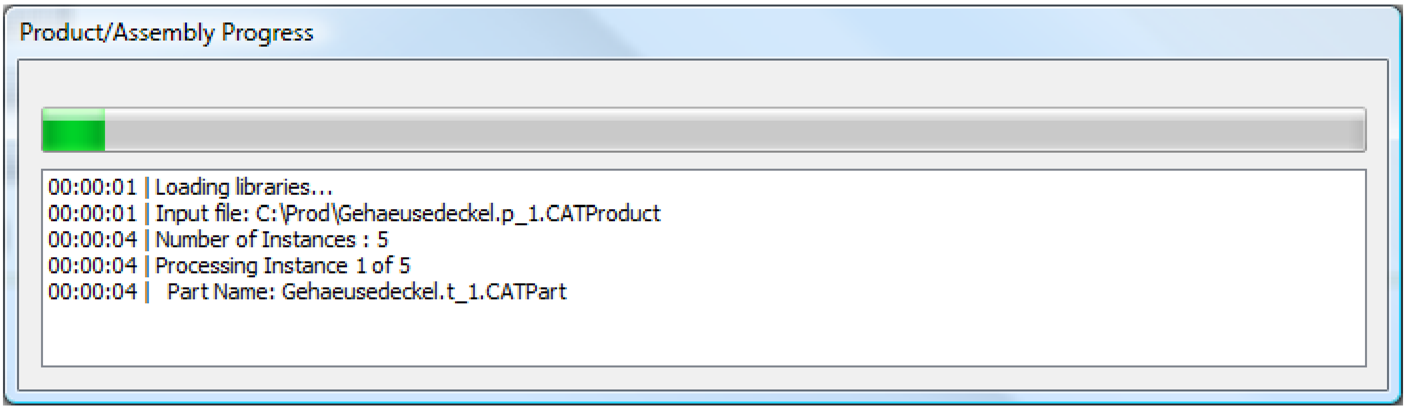
Opening large files of 50 to 100 MB or hundreds of thousands of polygons can take a lot of time (up to a half hour) depending on the capabilities of your computer. **Save the 3D workspace as soon as it opens**. If the import or save fails for any reason, try opening the `TEMP.MSH` or `TEMP.WRL` file to avoid repeating the initial conversion. Files larger than 50 to 100 MB may fail due to lack of memory or too many polygons for the display adapter to process. To show the number of polygons, click **Help** **Diagnostics** > **List Embedded Designs**.

Use the **Select Labels or Parts** tool to select parts of solids to change their position or properties, or to delete selected parts by selecting them and pressing `Delete` on the keyboard.

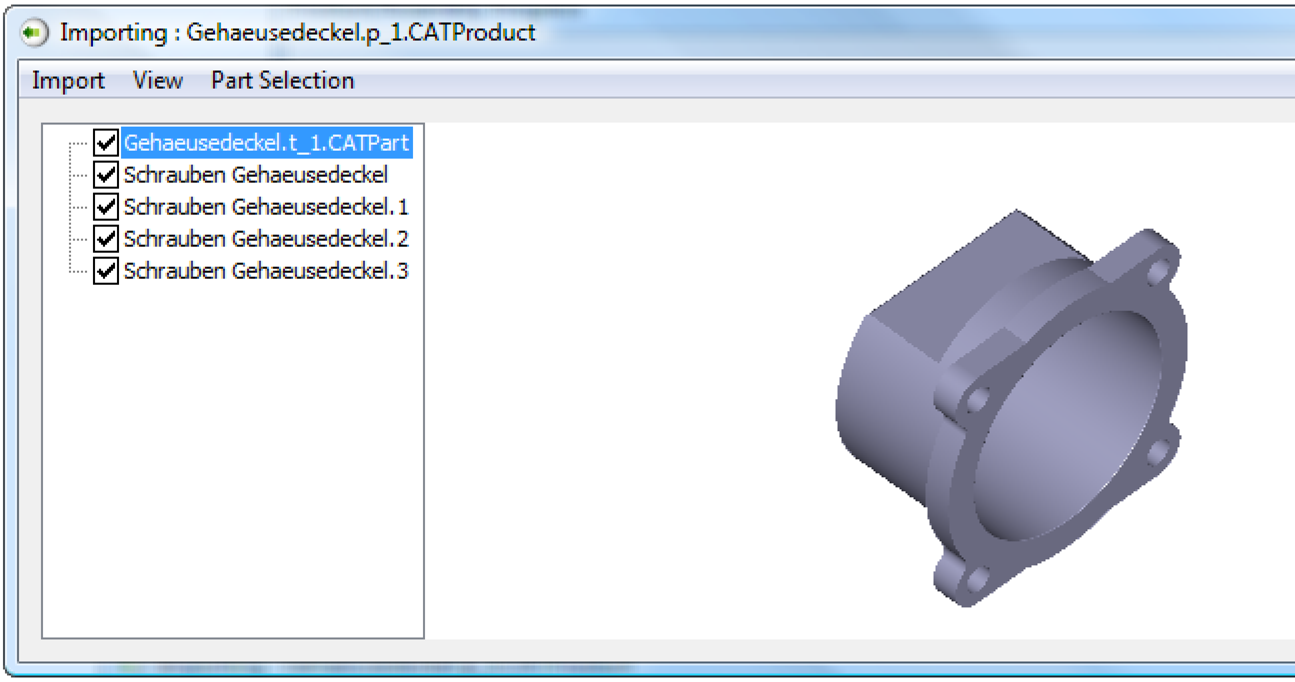
Importing a Solid using the Spatial Library

Do the following to import a solid using the Spatial library:

1. Import the solid into an open 3D workspace, or open it directly. A progress bar appears.



2. A preview of the solid appears. If it has parts defined, you can turn them on and off with the checkboxes next to their names, or on the **Part Selection** menu. To change to a defined angle in the view, choose an option on the **View** menu. You can also use the left mouse button to drag and rotate the camera, the scroll wheel to zoom in and out, and the right mouse button to drag and move the camera. Changing the view in the preview does not change the initial view of the solid when it opens in ArtiosCAD.



3. Click **Import** > **OK** to finish importing the solid.

Collada file format support

ArtiosCAD now supports the Collada intermediate file format for use in 3D.

The Collada format has two file types: `.DAE` and `.ZAE`. `.DAE` files are XML files that do not contain included graphics or other binary data. `.ZAE` files are compressed archives containing a `.DAE` file and the necessary textures, graphics, and other files to render the `.DAE` file properly.

The default Collada entry in the Outputs-3D Catalog creates a .DAE file.

Collada notes and warnings

When you export a foldable design from ArtiosCAD in Collada format that does not contain any graphics, Visualizer treats the entire design as if it were a print area, thereby allowing you to apply graphics.

When you load a Collada file with printable areas into Esko ArtiosCAD and no graphics are available, ArtiosCAD shows a checkerboard texture in those areas. When you export back to the Collada format, that checkerboard texture will output with the Collada file as if it was the true graphic.

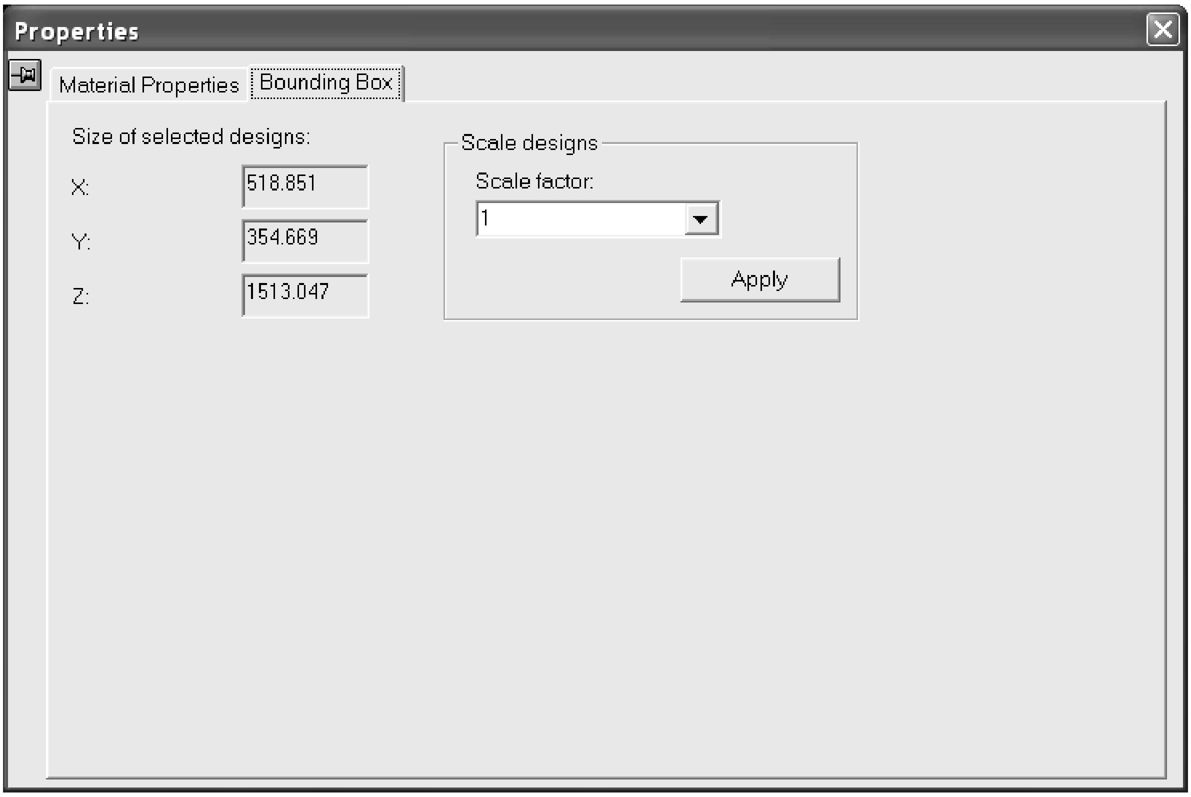
When you load a Collada file into Visualizer or Studio, the first printable area will be white, while the remaining printable surfaces will render normally.

For printable areas in a Collada file to be double-sided, there must be at least one transparent object in the scene.

Changing the scale of a solid

To change the scale of a solid, do the following:

1.  Click the **Select Designs** tool and select the solid(s) to modify. If you are only selecting one solid, double-click it to open the Properties dialog box; if you are selecting more than one solid, click **Edit** > **Properties**.
2. Click the **Bounding Box** tab in the Properties dialog box.



3. Choose the new scale from the **Scale factor** drop-down list box, or enter a scale factor greater than 0 in the field.
4. Click **Apply**. The selected objects are updated along with the values in the **Size of selected designs** fields.
5. Close the dialog box by clicking the X at the end of the title bar.

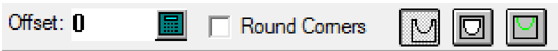
The scale is set to 1 each time you open the Properties dialog box; previous sizes and scale factors are not retained.

Cross Sections

The tools on the Cross Section toolbar let you make and adjust pieces of folding carton or corrugated board that intersect with objects in the 3D workspace. These tools require the 3D Designer license.

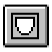
Intersect Design tool

 The **Intersect Design** tool creates holes in a corrugated or folding carton design where it intersects other designs or solids. The tool has the following controls on the Status bar:



Enter a value in the **Offset**: field to specify a gap between the intersecting object and the cutout in the intersected object.

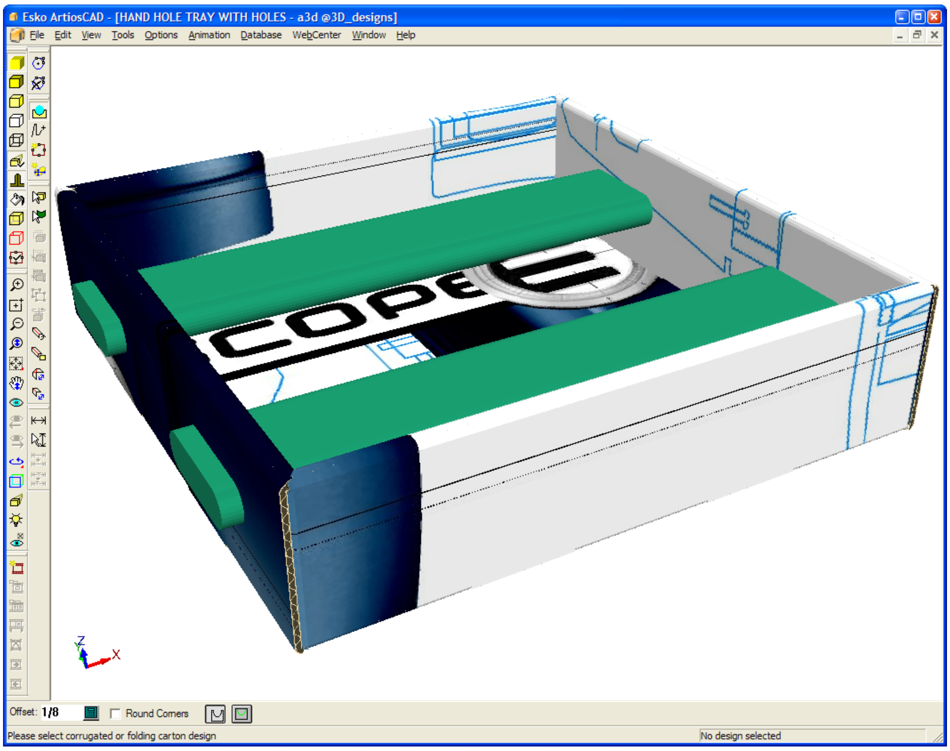
 The first button on the Status bar, **Intersect Designs**, sets the tool to use cut lines to make the hole boundaries in the flat design.

 The second button on the Status bar, **Intersect Designs All The Way Through**, sets the tool to make complete cut-outs for objects such as wine glasses.

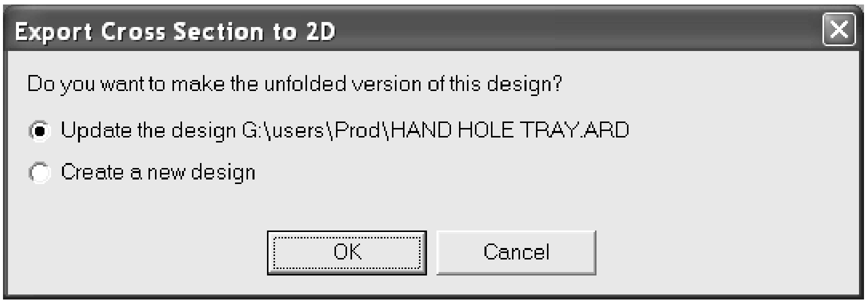
 The third button on the Status bar, **Intersect Designs with Annotations**, sets the tool to use annotation lines to make the hole boundaries in the flat design.

To use the tool, do the following:

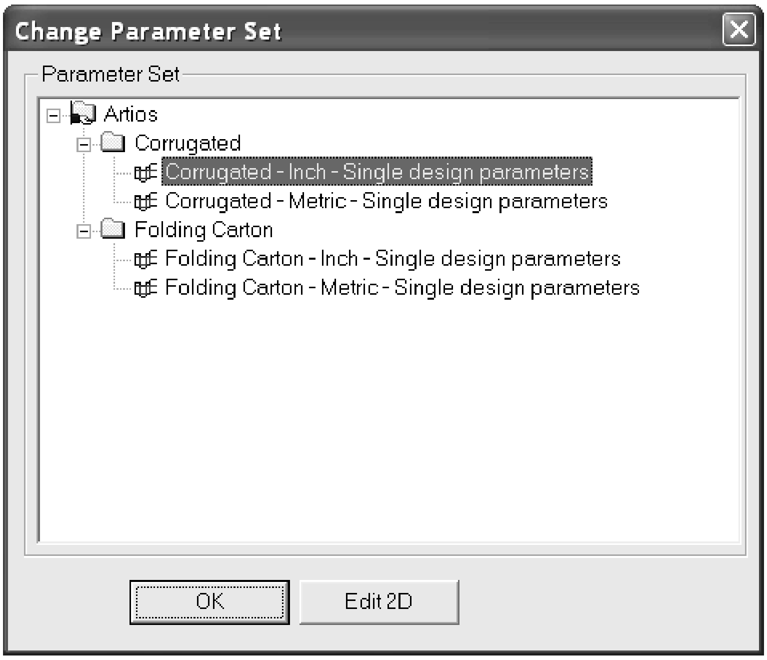
1. Position the objects together.
2.  Click **Intersect Design** on the Cross Section toolbar.
3. On the Status bar, enter an offset if desired, select or clear the **Round corners** checkbox (to create rounded corners in the holes), and choose the mode for the intersecting lines.



4. Click the intersected corrugated or folding carton design.
5. If the single design workspace corresponding with the selected 3D object exists, ArtiosCAD prompts you to update it or create a new single design. Set the option button as desired and click **OK**. Clicking **Cancel** stops the tool.



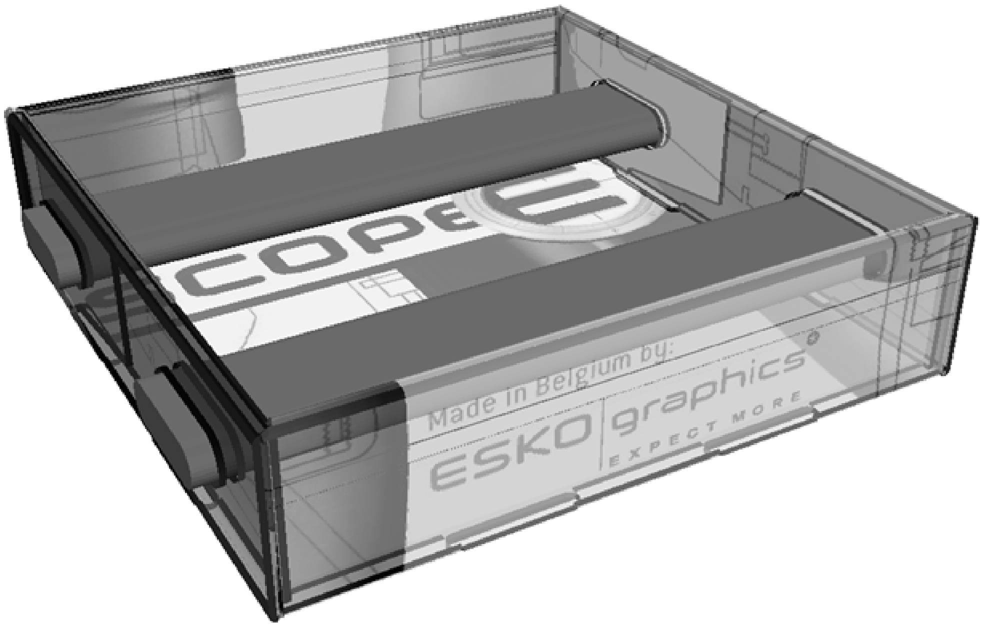
6. Choose a parameter set for the design. If you are updating the existing design, the previously-selected parameter set is selected.



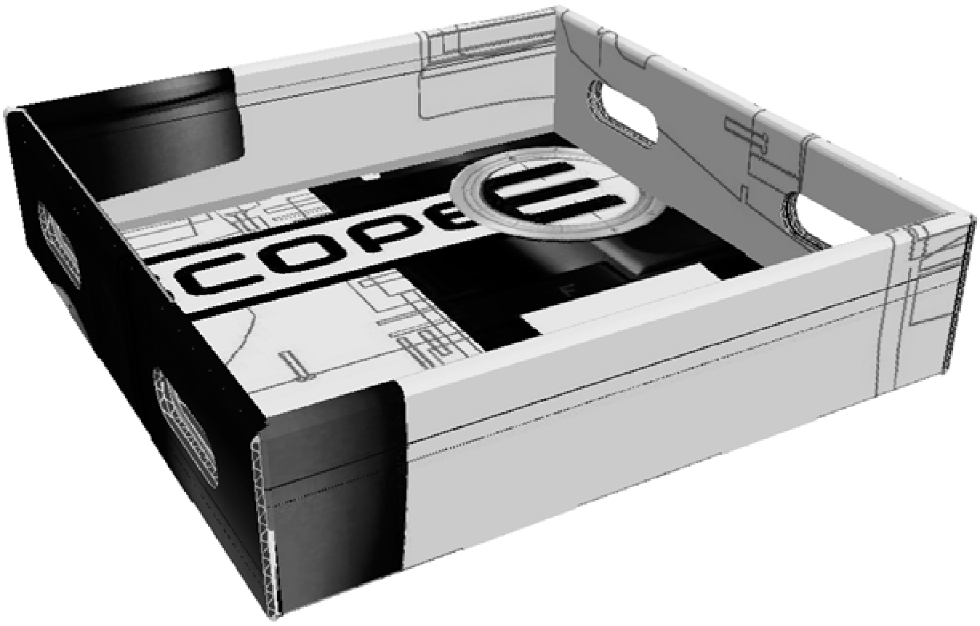
To accept the changes to the single design and return to 3D, click **OK**. You may want to move the dialog box to see more of the design.

To make changes to the single design, click **Edit 2D**, make the changes, click **Convert to 3D** on the View bar, and update the 3D workspace.

7. Once you return to 3D, the view changes to show transparency with purple edges to make the new holes more visible. Clicking any Select tool turns off the temporary transparency. Shown below is tray with temporary transparency turned on and the 1/8” offset around the solids.



Shown below is the final result with the bars removed.



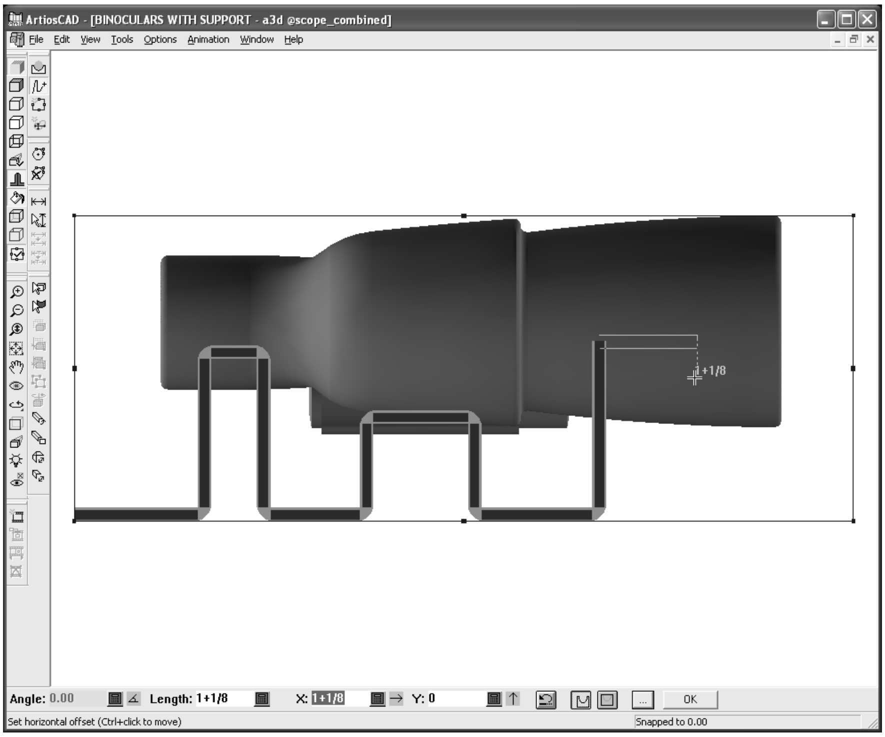
Note: Any copies of the design will have the same holes made in them.

The filename for the flat workspace containing the intersected design is generated from the filename of the 3D workspace, plus a number that increments by one for each intersected design.

Cross Section tool

 The **Cross Section** tool creates a piece of folded board with cutouts for the portions of objects it intersects. The size of the bounding box determines the size of the folded board created by the tool. Before using this tool, adjust the bounding box as needed using the instructions in the next section.

To use this tool, click it, select the start point of the piece of folded board, and make the desired geometry using drag similar to that of the Line tool in Single Design. Shown below is a step in the construction of a cross section.



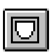
Shown below are the controls on the Status bar in greater detail.

The **Angle**:, **Length:**, **X**:, and **Y:** fields all behave as they do for the Line tool in Single Design.

Round Corners, when selected, has the tool create rounded corners in the holes it creates.

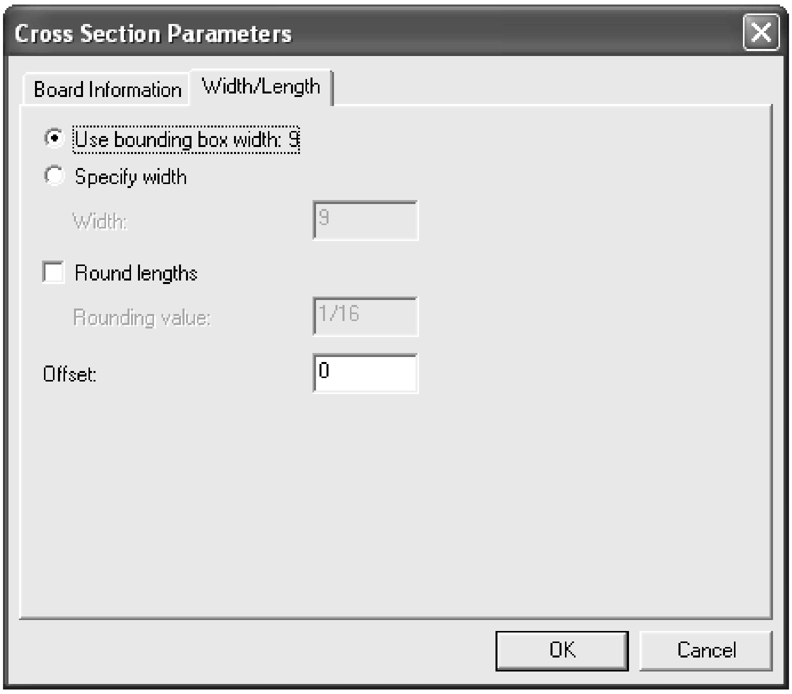
 The **Undo** button undoes the last piece of geometry created. It is very important to use this button instead of pressing `CTRL-Z`, as pressing `CTRL-Z` will undo everything the tool has made up to that point, not just the last segment.

 The **Intersect Designs** button has the tool create cut lines in the folded board where it passes through other objects.

 The **Intersect Designs All The Way Through** button has the tool create complete cut-outs for objects such as wine glasses.

The **Intersect Designs with Annotations** button has the tool create annotation lines in the folded board where it passes through other objects.

... Clicking **More Options** opens the Cross Sections Parameters dialog box:



The Board Information tab lets you choose the board and set the caliper of the folded board. The default board is used initially, but if you change it, the new board becomes the default for the tool in the current workspace.

On the Width/Length tab, choose to **Use bounding box width** to set the size of the folded board, or click **Specify width** and enter a value in the **Width:** field.

Round lengths rounds the board lengths to the nearest increment specified in the **Rounding value**: field. When checked, it is initially set to the rounding value set for the board.

A value in the **Offset:** field adds a gap between the piece of folded board and the object being cross sectioned.

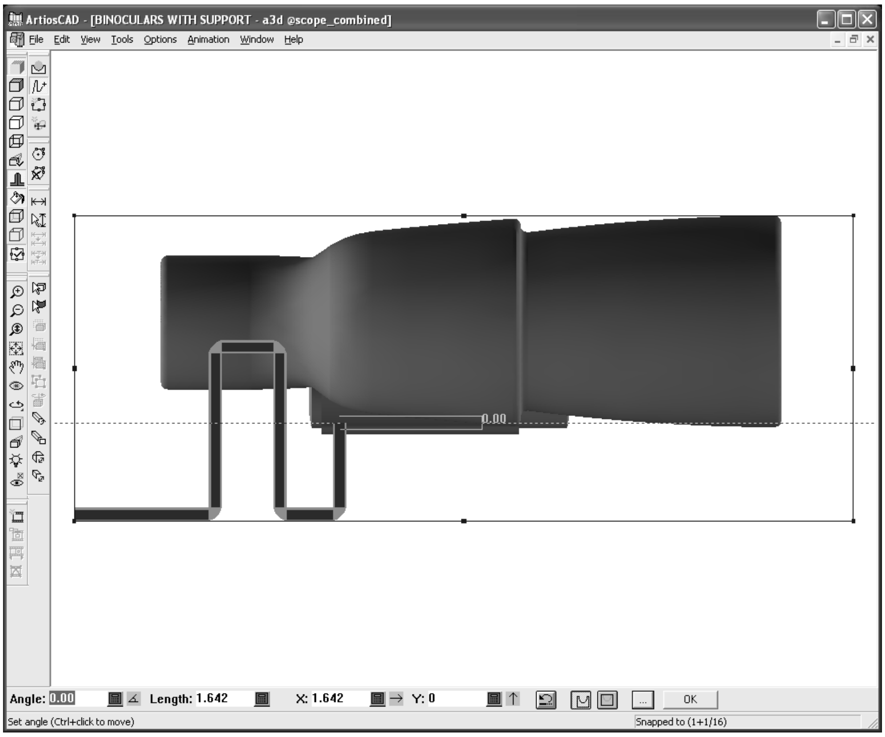
Note: Defaults for the **Intersect Design** and **Cross Section** tools are in **Options** > **Defaults** > **Shared defaults** > **Startup defaults** > **3D Tools Defaults**.

OK Clicking **OK** on the Status bar completes the geometry and calculates the cross section. The amount of time it takes to make the cross section depends on the complexity of the design as well as your computer hardware. Clicking another tool before clicking **OK** cancels the cross section completely.

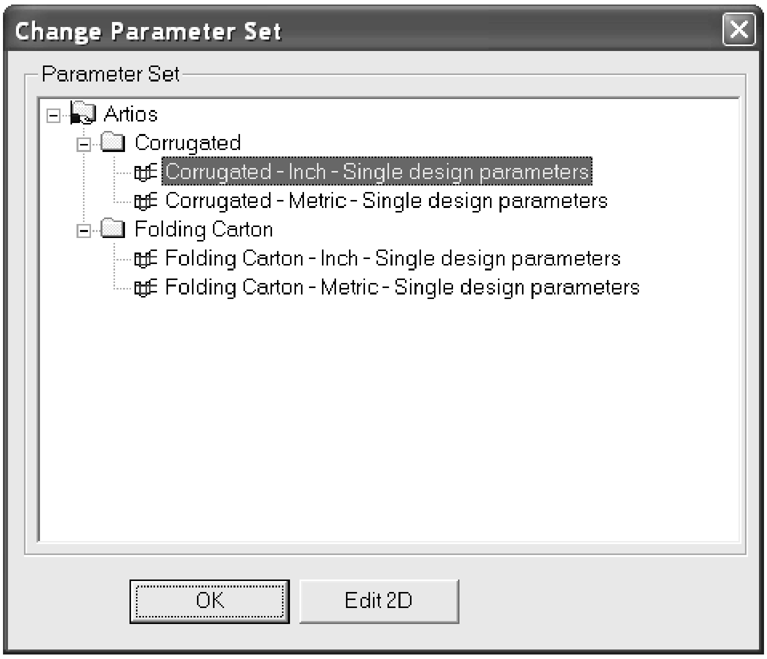
To make a cross section, do the following:

1. Open the design that will have a cross section made around it.
2. Use the **Bounding Box** tool to modify the size of the bounding box.
3. Click the **Cross Section** tool.

4. Click the start point for the piece of folded board. To move by an offset from that point, hold down CTRL and click in the desired location.
5. Click and drag to make the desired geometry. The drag snaps to existing lines, but also has enough of a cutback to allow for the board thickness depending on where you click. You can hold down CTRL while clicking to move the start point for the geometry, but make sure all pieces are connected.



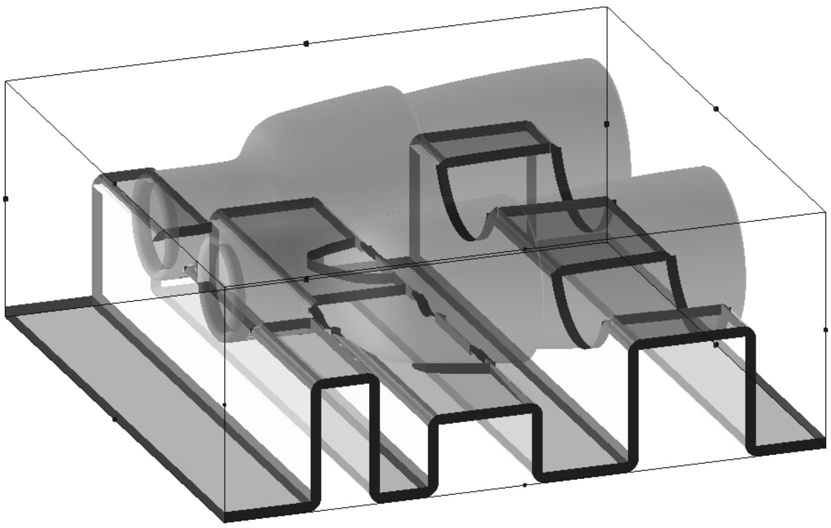
6. When done creating geometry, click **OK** to create the piece of folded board.
7. Choose a parameter set for the folded board.



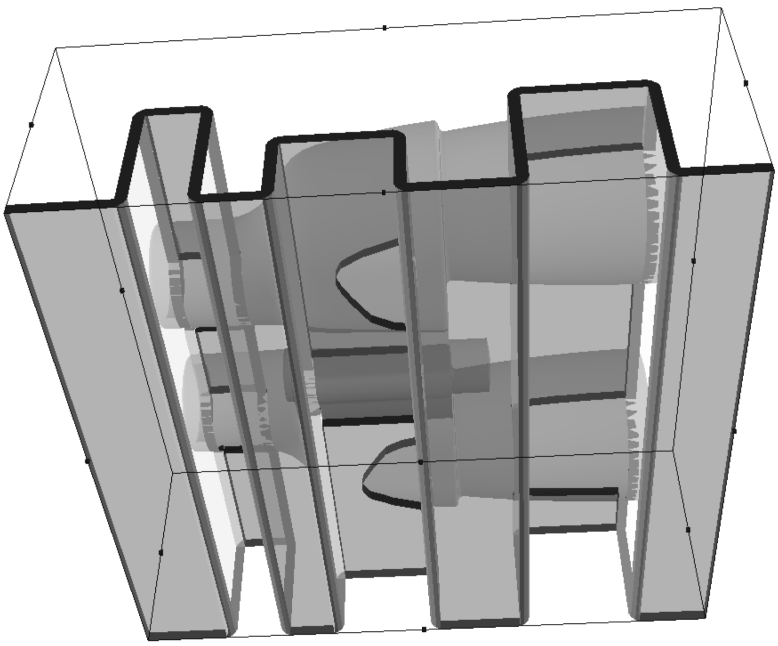
To accept the folded board and return to 3D, click **OK**. You may want to move the dialog box to view more of the design.

To make changes to the folded board, click **Edit 2D**, make the changes, click **Convert to 3D** on the View bar, and update the 3D workspace.

8. Once you return to 3D, the view changes to show the folded board in transparency with purple edges and no perspective, and the View Angle tool is activated. Clicking any Select tool turns off the temporary transparency.



Shown below is a view from underneath to show more detail.



The folded piece of board is a series of panels connected with creases or reverse creases depending on the fold angle. ArtiosCAD positions the design so there are more creases than reverse creases.

The filename for the flat workspace containing the folded piece of board is generated from the filename of the 3D workspace, plus a number that increments by one for each cross section workspace.

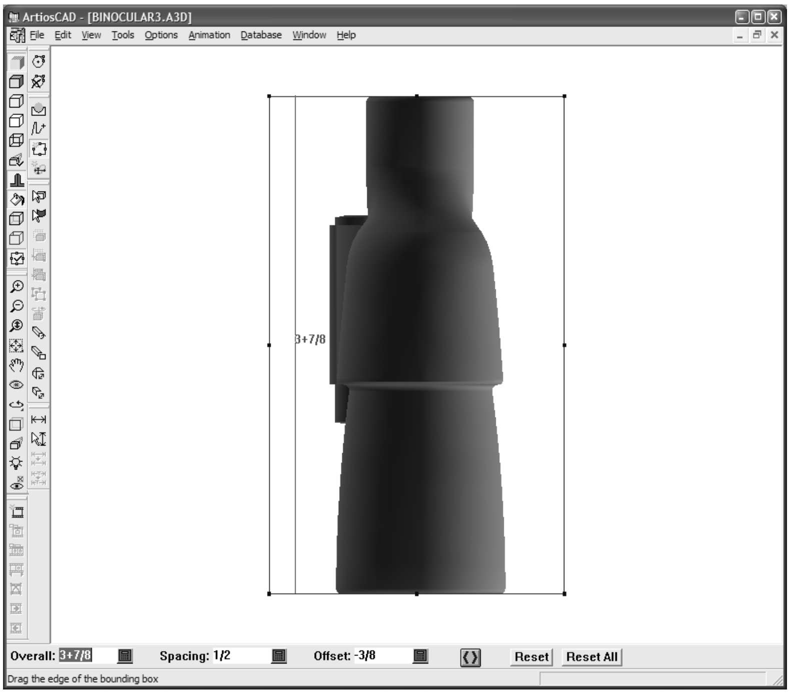
Bounding Box tool

 The **Bounding Box** tool creates a bounding box around all the objects in the 3D workspace, and lets you adjust the size of the bounding box. The **Cross Section** tool uses it to define the size of the flat design created as a cross section, and the **Run a Standard** tool uses it for the dimensions of the standard it creates.

The bounding box grows to accommodate designs added to the 3D workspace, but is unaffected when objects are removed.

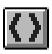
Note: The Bounding Box tool does not work with CloseUp windows.

When you click the tool, the view changes to the closest orthogonal view, and perspective is turned off. The fields on the Status bar become available when you click a side of the bounding box. The drag is for the active field in the Status bar.



Shown below are the controls on the Status bar:

The value in the **Overall**: field shows the distance from the opposite side of the bounding box. **Spacing**: refers to the distance from the closest edge of a design. **Offset**: is the distance from the previous position of the side being adjusted.

 The **Move Both Sides** button toggles the adjustment of both parallel sides at once.

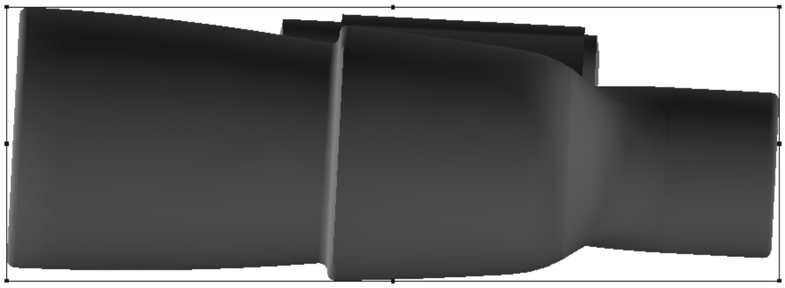
Reset snaps the last bounding box edge adjusted back to its minimum spacing from the edge of the object.

Reset all adjusts the bounding box edges to encompass all items in the 3D workspace.

Only two dimensions are shown in the orthogonal view. Use the **Rotate View** tools to adjust the view so that the other dimension of the bounding box can be adjusted.

Rotating solids using the bounding box

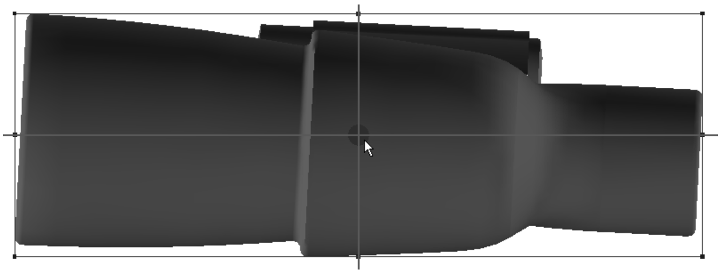
When importing a solid, it may not be defined to be square to its bounding box as shown below. Use the bounding box and the **Align With** button in the **Rotate** tool to make it square to the bounding box.



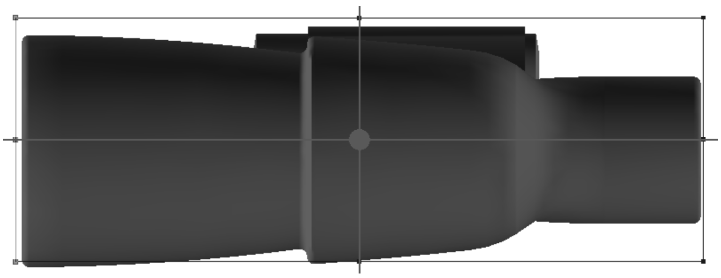
Note: Snap to solids must be selected in **Options** > **Snap Options** for this procedure to work.

To make the binoculars square to the bounding box, and to resolve similar situations with other solids, do the following:

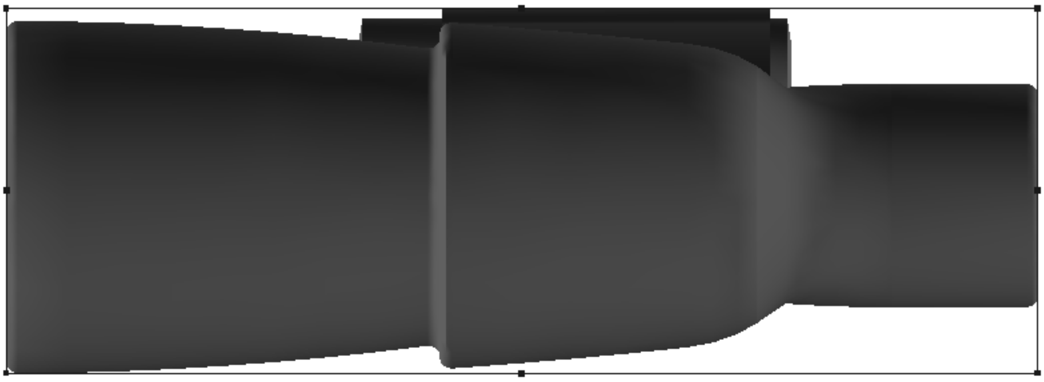
1.  Click the **Bounding Box** tool to show the bounding box and change to orthogonal view.
2. Select the item to adjust.
3.  Click the **Rotate** tool and click the axis around which to rotate the object. In the example below, the axis orthogonal to the view is selected.



4. Click the edge of the solid to align with the bounding box. For the binoculars, the bottom edge of the bigger lens was used.
5. Click the edge of the bounding box with which to align the object.
6. The object rotates.



7. Note how the bounding box is no longer correct. Click the **Bounding Box** tool and use the **Reset** button to snap the edges to their minimum clearances. Shown below is the final result.

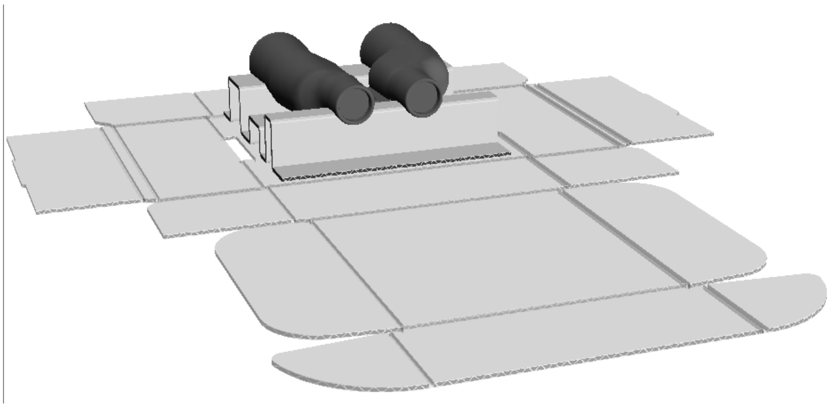


Run a Standard tool

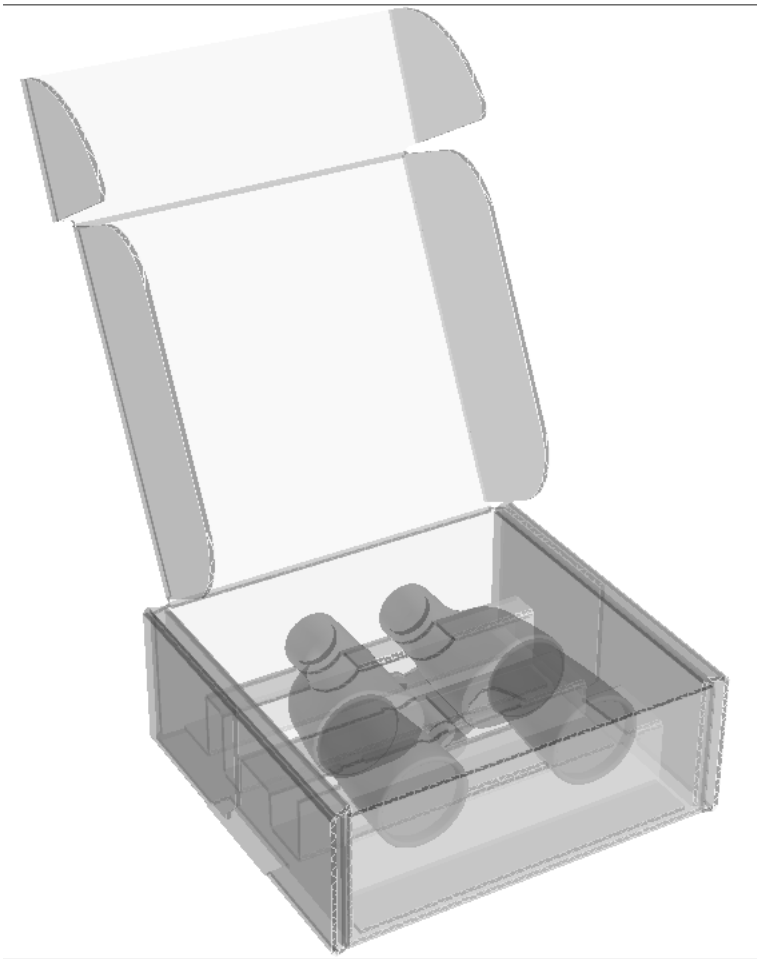
The **Run a Standard** tool creates a design from a Standards Catalog entry using the dimensions of the bounding box as the dimensions of the design.

To run a standard, do the following:

1. Adjust the bounding box as desired. You may want to make it slightly bigger than the object it will contain to allow for board thickness.
2. Click the **Run a Standard** tool.
3. Click the side of the bounding box that will be the length dimension of the new design.
4. Click the side of the bounding box that will be the width dimension of the new design.
5. Choose a standard from the Standards Catalog and click **OK**.
6. Choose a parameter set and board and click **OK**.
7. If the standard does not contain any of the L, W, or D variables, map the existing variables to the length, width, and depth and click **OK**.
8. Proceed through the standard prompts as usual, clicking **Next** to go to the next menu or **OK** to skip the remaining menus and make the design.
9. Choose a base face for the new design. It appears in the 3D workspace. If the standard contains fold angles, they are used; if not, the design is shown flat.



10. Fold it as desired.



11. The single design will be open in another ArtiosCAD window. Save it as desired.

Note: Designer WorkBench standards, or those that reference them, may not be used with this tool.

Note: If the standard contains a crease that crosses more than one panel, a warning will appear. If you proceed, panels may be missing from the 3D design created. For best results, split creases in standards that cross panels.

Convert to 2D tool

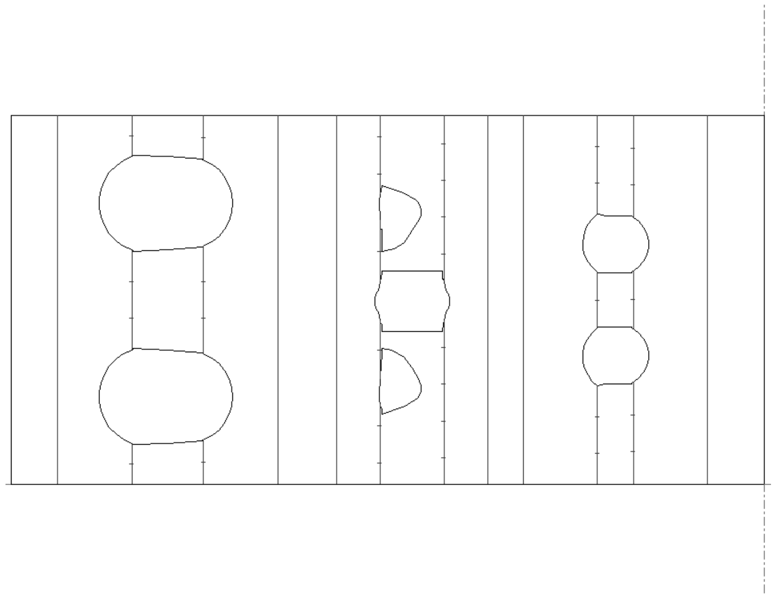
The **Convert to 2D** tool on the Tools menu lets you create a flat design out of a corrugated or folding carton design in a 3D workspace. This is useful when you receive a 3D workspace from someone else without the corresponding flat design(s) and you want to make changes to the design(s).

When you convert a design to 2D from 3D, the following restrictions apply:

- Curved lines are converted from a series of straight lines into arcs. Arcs smaller than 3 millimeters may not be converted exactly as they were in the 3D design.
- The 2D design line pointages are all set to 2.
- Cut lines should have most of their bridges but crease lines will have none.
- The 2D design will have no graphics.

For example, if you received the workspace shown in the Cross Section tool documentation, and needed to modify the piece of corrugated board supporting the binoculars, you would do the following:

1. Click **Tools** > **Convert to 2D**.
2. Choose a parameter set and click **Edit 2D**.
3. Edit the flat design as desired in Single Design.



4. Click **Convert to 3D** on the View bar, choose **Update 3D**, click **OK**, select the base face, and click **OK**. The updated workspace is shown.

Splitting an imported design into multiple parts

To split an imported design into multiple parts, do the following.

1. Use the **Select Labels or Parts** tool to select the elements of the design to split into a separate design.
2. Click **Group**.
3. ArtiosCAD will warn you that you are grouping the selected parts into a separate design. Click **OK** to split the parts into a separate design. The selected parts will be a single separate design; they will not be separated into individual designs.

Note: You cannot undo this procedure.

Palletization in 3D

You can round-trip data between CAPE/TOPS and ArtiosCAD to palletize a 3D design, make a secondary (shipping) container in CAPE/TOPS starting in 3D, and also to make a pallet load in 3D starting in CAPE/TOPS.

For answers to some frequently asked questions about palletization, see *Palletization FAQ* in the *Outputs* chapter.

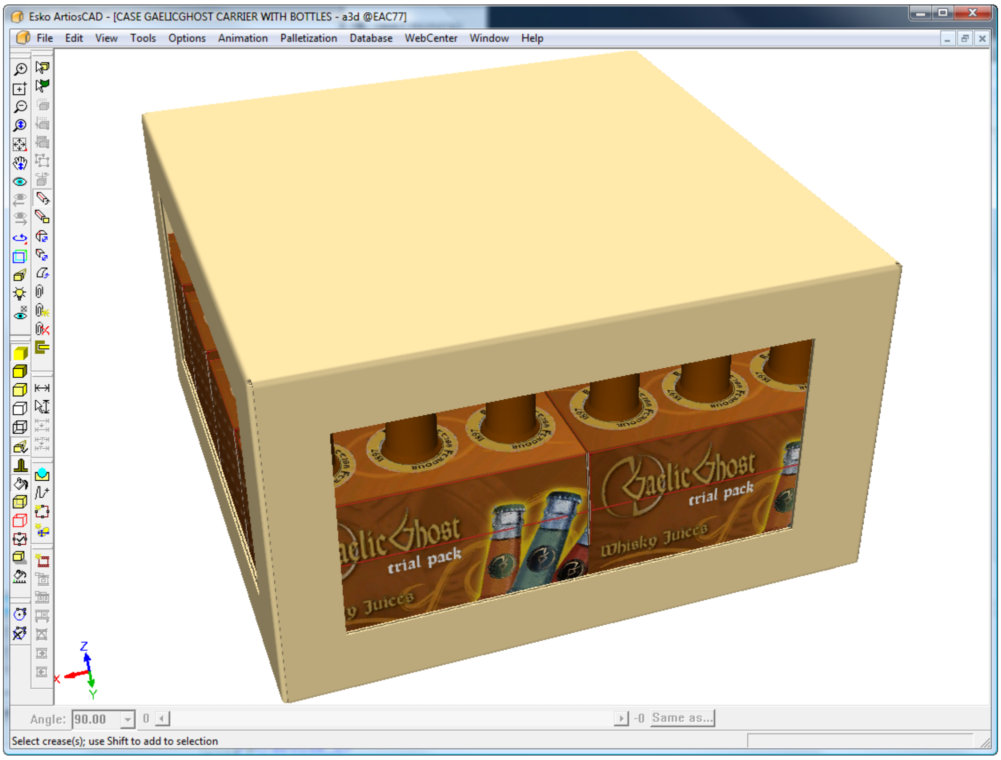
Note:

Before using these features for the first time, make sure you have mapped CAPE/TOPS styles and materials to ArtiosCAD standards and boards as described in the *Builder* chapter. You may also want to set CAPE/TOPS defaults as described in the *Defaults* chapter of the Administrator Guide.

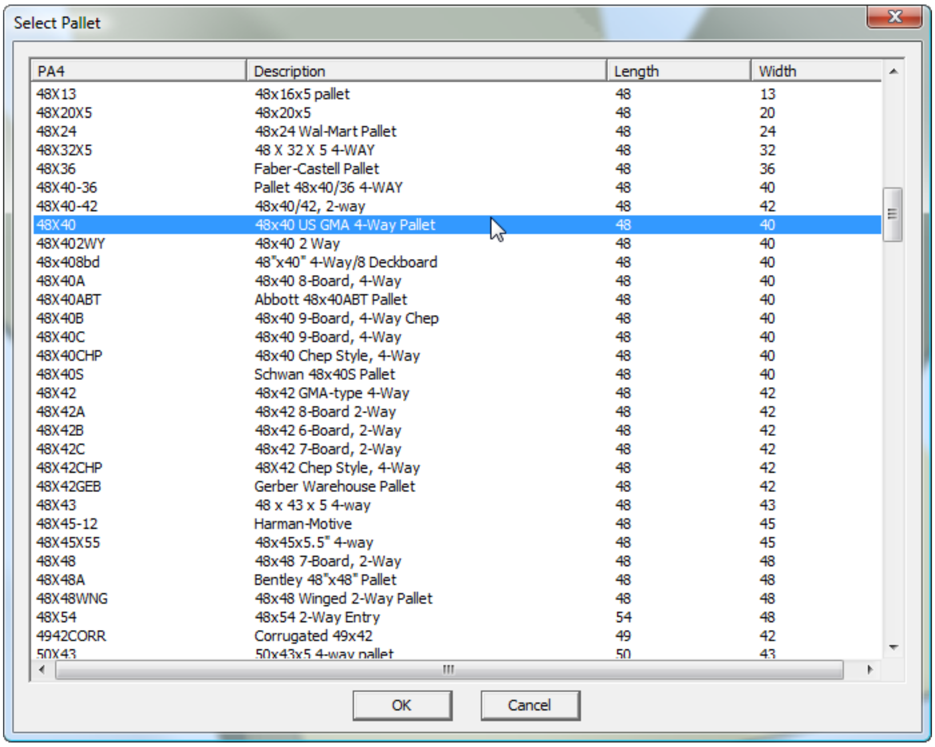
Palletizing a 3D workspace

To send a 3D workspace to CAPE for palletizing and then view the CAPE solution back in 3D, follow the instructions below. If you are using TOPS, the workflow is similar:choose a pallet and send the information to TOPS, which will palletize the design and send the information back to 3D.

1. In 3D, design a new workspace or open an existing one, and assemble its components as desired. Make sure to fold it into its final form as its bounding box will be used for the sizing information in CAPE.



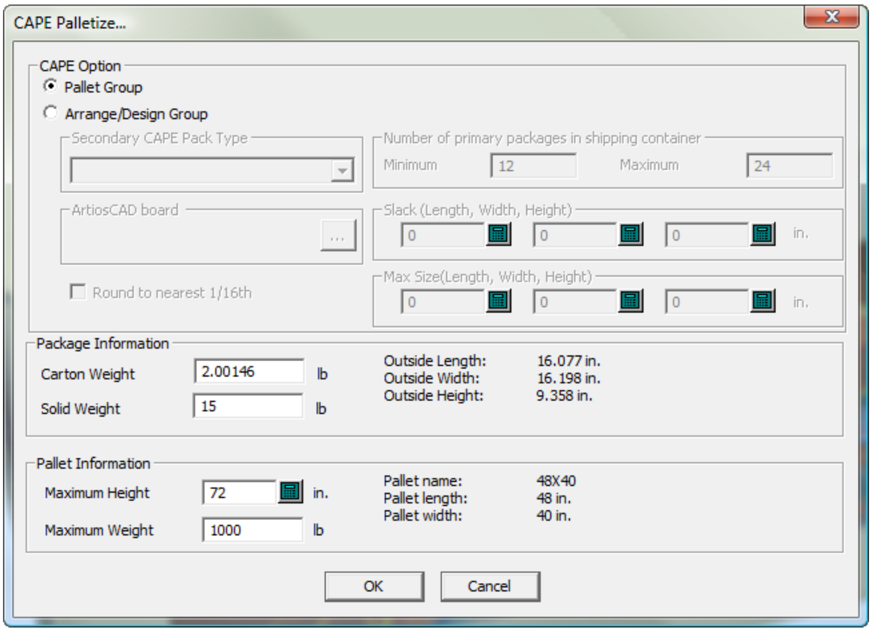
2. Click **Palletization** > **Change Pallet.**
3. In the Select Pallet dialog box, select a pallet and click **OK**.



PA4	Description	Length	Width
48X13	48x16x5 pallet	48	13
48X20X5	48x20x5	48	20
48X24	48x24 Wal-Mart Pallet	48	24
48X32X5	48 X 32 X 5 4-WAY	48	32
48X36	Faber-Castell Pallet	48	36
48X40-36	Pallet 48x40/36 4-WAY	48	40
48X40-42	48x40/42, 2-way	48	42
48X40	48x40 US GMA 4-Way Pallet	48	40
48X402WY	48x40 2 Way	48	40
48x408bd	48"x40" 4-Way/8 Deckboard	48	40
48X40A	48x40 8-Board, 4-Way	48	40
48X40ABT	Abbott 48x40ABT Pallet	48	40
48X40B	48x40 9-Board, 4-Way Chep	48	40
48X40C	48x40 9-Board, 4-Way	48	40
48X40CHP	48x40 Chep Style, 4-Way	48	40
48X40S	Schwan 48x40S Pallet	48	40
48X42	48x42 GMA-type 4-Way	48	42
48X42A	48x42 8-Board 2-Way	48	42
48X42B	48x42 6-Board, 2-Way	48	42
48X42C	48x42 7-Board, 2-Way	48	42
48X42CHP	48X42 Chep Style, 4-Way	48	42
48X42GEB	Gerber Warehouse Pallet	48	42
48X43	48 x 43 x 5 4-way	48	43
48X45-12	Harman-Motive	48	45
48X45X55	48x45x5.5" 4-way	48	45
48X48	48x48 7-Board, 2-Way	48	48
48X48A	Bentley 48"x48" Pallet	48	48
48X48WNG	48x48 Winged 2-Way Pallet	48	48
48X54	48x54 2-Way Entry	54	48
4942CORR	Corrugated 49x42	49	42
50X43	50x43x5 4-way pallet	50	43

4. Click **Palletization** > **Palletize Design**.

   ArtiosCAD creates a copy of the workspace and works with the copy in order to not infringe on your original.
5. In the Cape Palletize dialog box, choose **Pallet Group**.
6. Set the values in the **Package Information** and **Pallet Information** groups as appropriate and click **OK**.

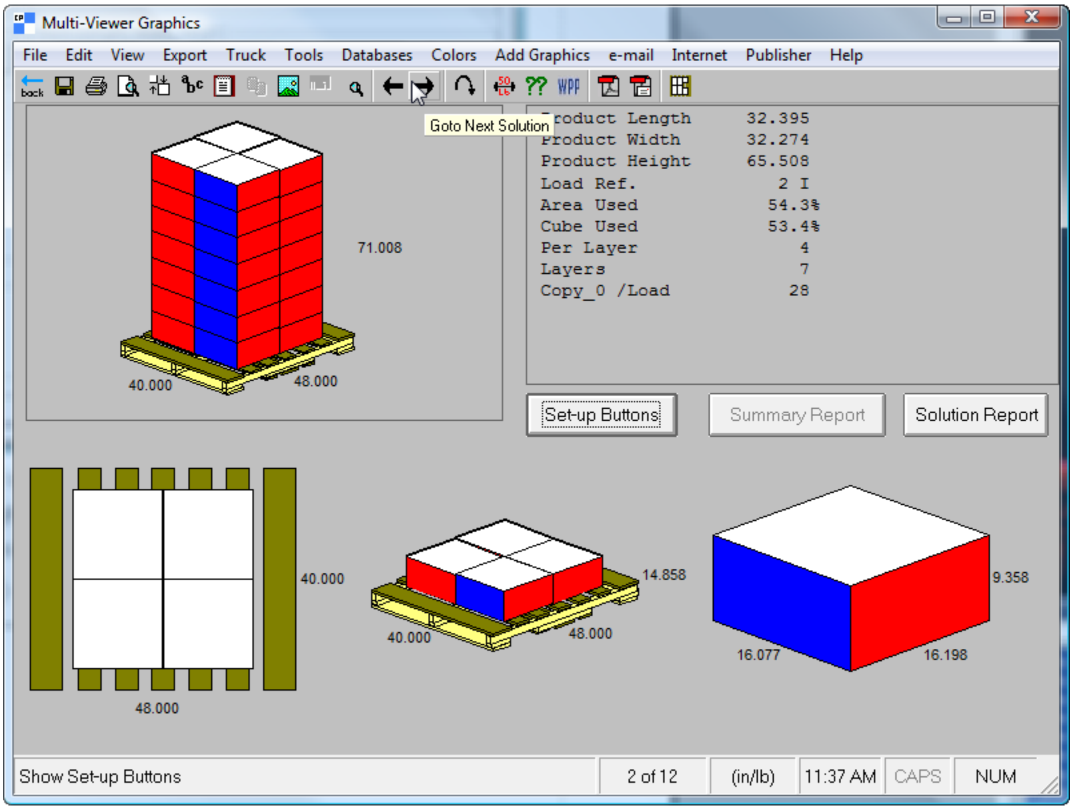


Click **OK** through any alerts. CAPE may prompt you that it is swapping the dimension directions.

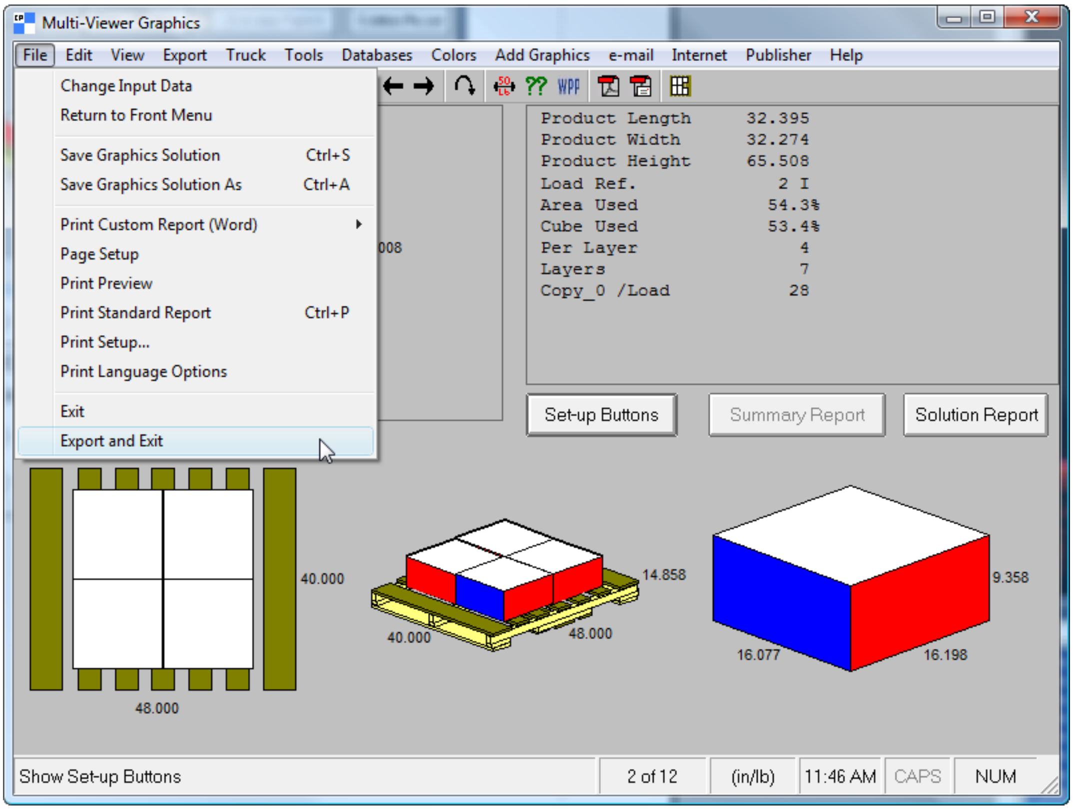
Note:

ArtiosCAD is unusable while waiting for a response from CAPE.

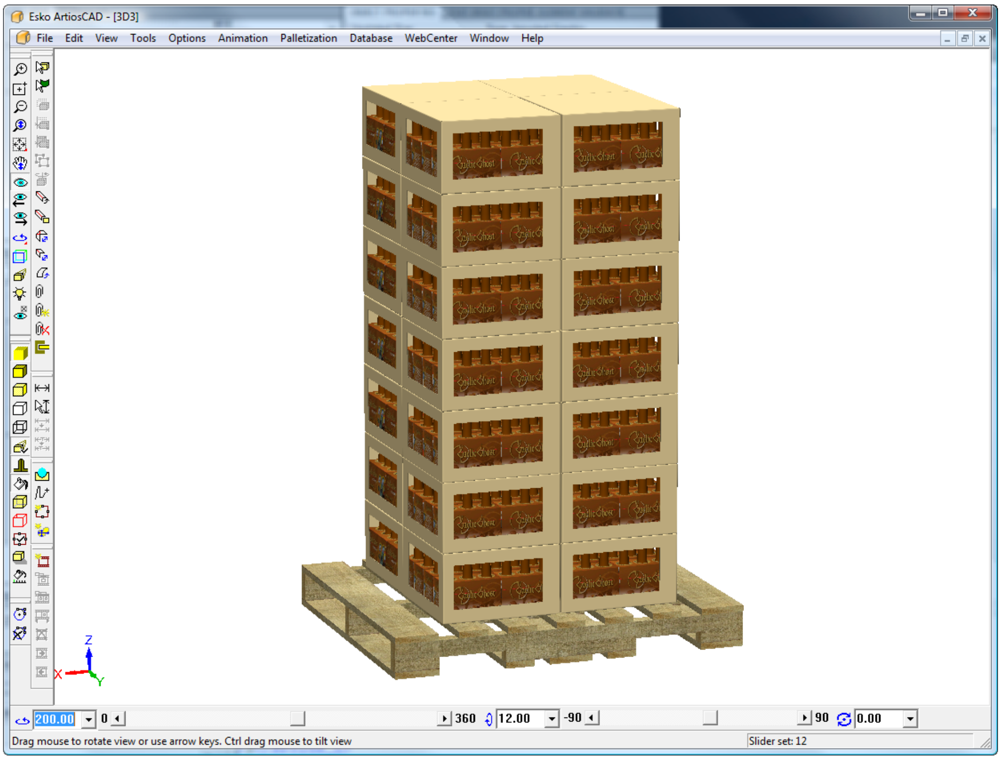
7. In CAPE, use the **Goto Next Solution** and **Goto Previous Solution** arrows to find the best solution.



8. When you have found the best solution, click **File** > **Export and Exit**.



9. The palletization solution appears in 3D; save it and work with it as a normal 3D workspace. Remember that this is a copy and not your original 3D workspace.



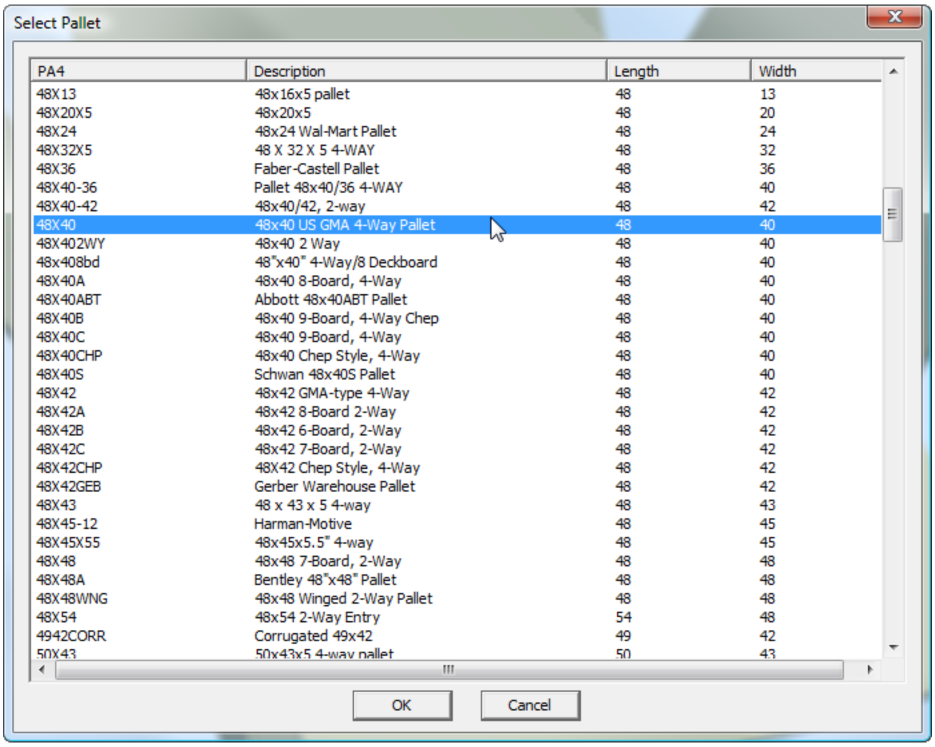
Arranging a Group in CAPE around a 3D workspace

In addition to palletizing a workspace from 3D in CAPE, you can also create secondary packaging in CAPE for your primary package in 3D.

To create secondary packaging around your primary package, do the following:

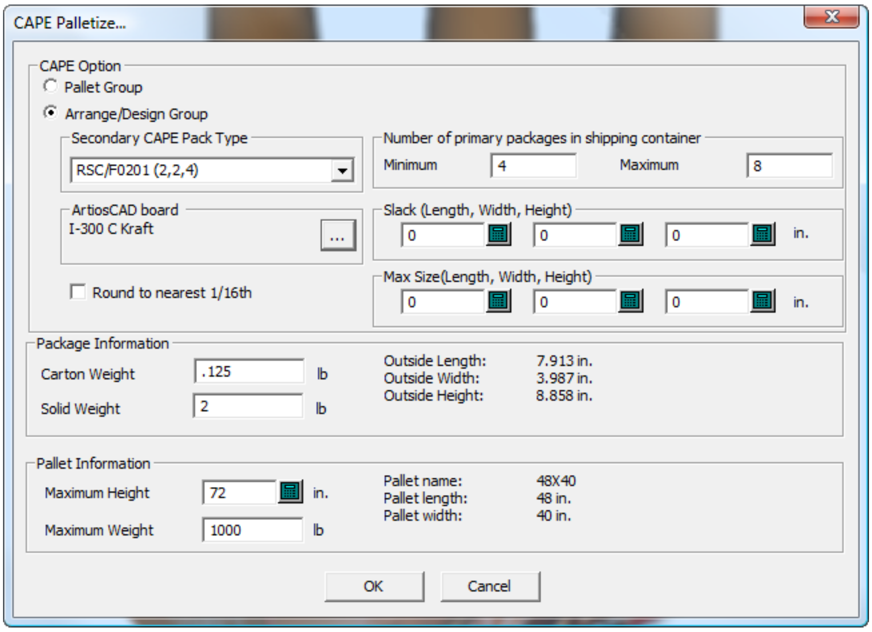
1. In 3D, design a new workspace or open an existing one, and assemble its components as desired.

2. Click **Palletization** > **Change Pallet**.
3. In the Select Pallet dialog box, select a pallet and click **OK**.



4. Click **Palletize** > **Palletize Design**.

5. In the Cape Palletize dialog box, choose **Arrange/Design Group**.



6. Enter appropriate values in the fields:

a) Select the secondary package from the **Secondary CAPE Pack Type** drop-down list box.

b) In the **ArtiosCAD board** group, click Properties (**...**) and select a board code.

c) In the Minimum and Maximum fields, enter the number of primary packages to create secondary packages for. If you have more than one primary package in your 3D workspace, divide these numbers accordingly; for example, if you had 4 primary packages in 3D, you would only want 1 or 2 of them in CAPE.

d) Enter values for the slack (gaps between the primary packages) and the maximum size of the secondary packaging, if desired.

e) Check **Round to nearest 1/16th** if desired.

f) In the **Package Information** group, enter the weight of the primary package and any solids it contains.

g) In the **Pallet Information** group, enter the maximum height of the stacked pallet and its maximum weight.

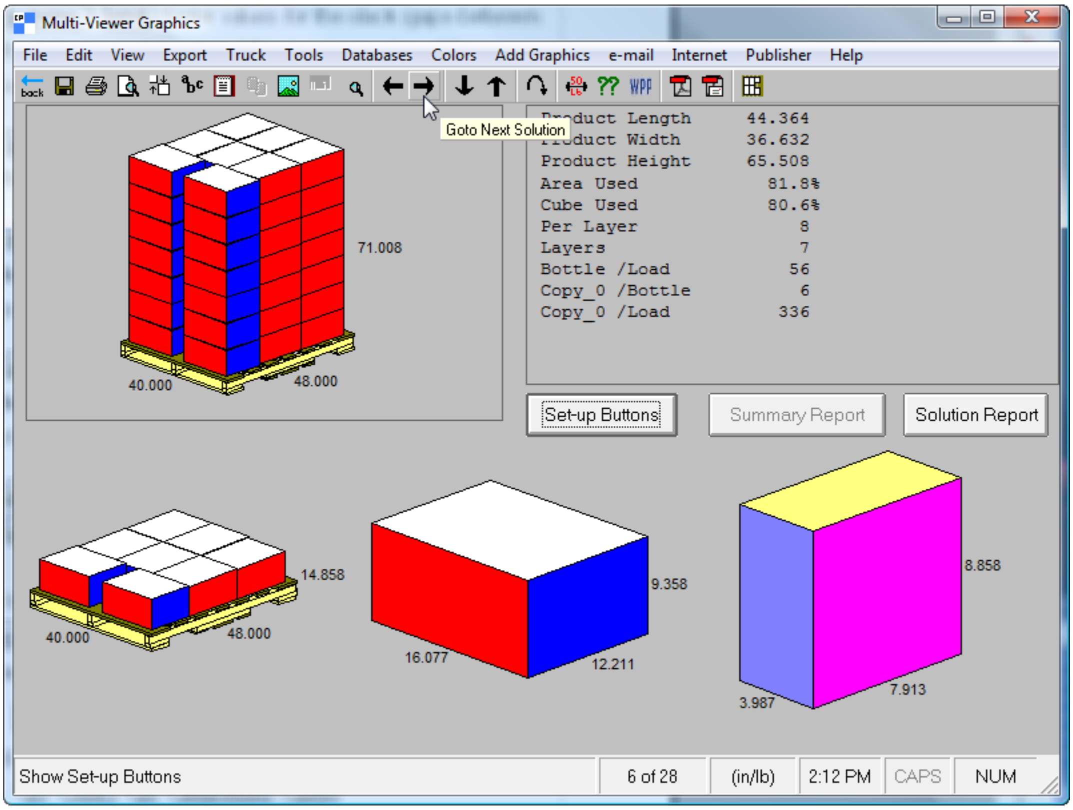
h) Click **OK**.

CAPE will launch. Click **OK** through any alerts. CAPE may prompt you that it is swapping the dimension directions.

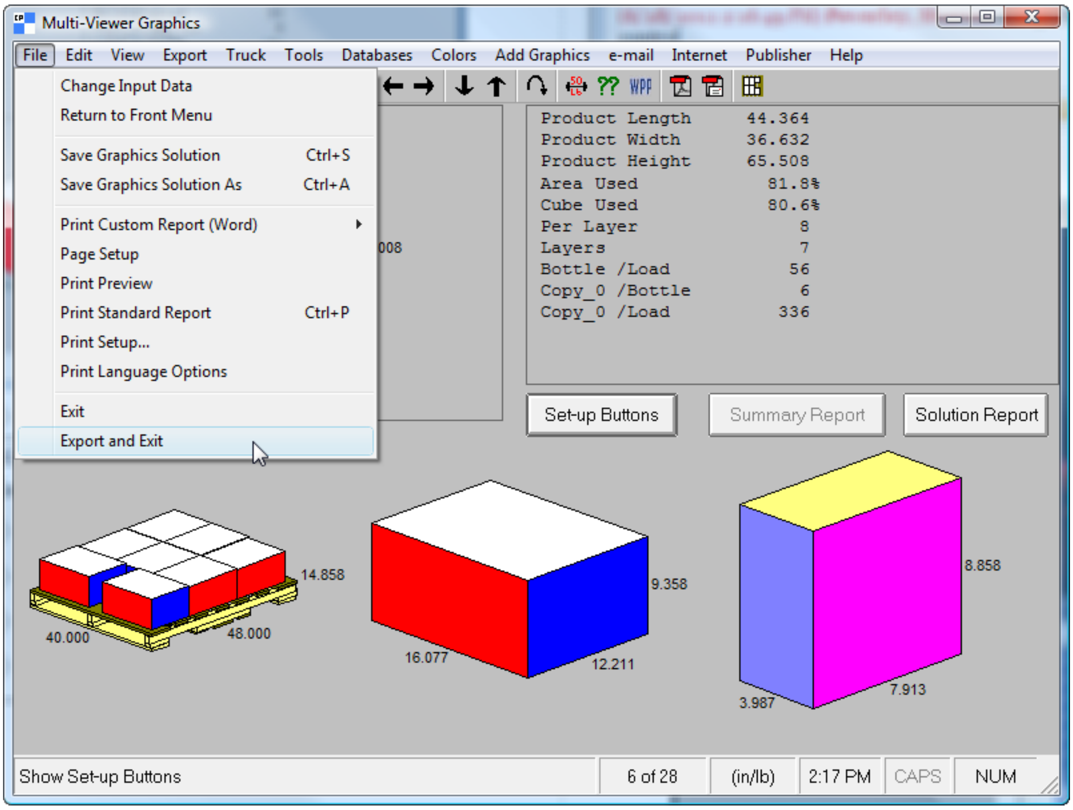
Note:

ArtiosCAD is unusable while waiting for a response from CAPE.

7. Use the **Goto Next Solution**, **Goto Previous Solution**, **Goto Next Pattern**, and **Goto Previous Pattern** arrows to find the best solution. The primary package is the one with yellow and magenta faces, while the secondary package has red and blue faces.

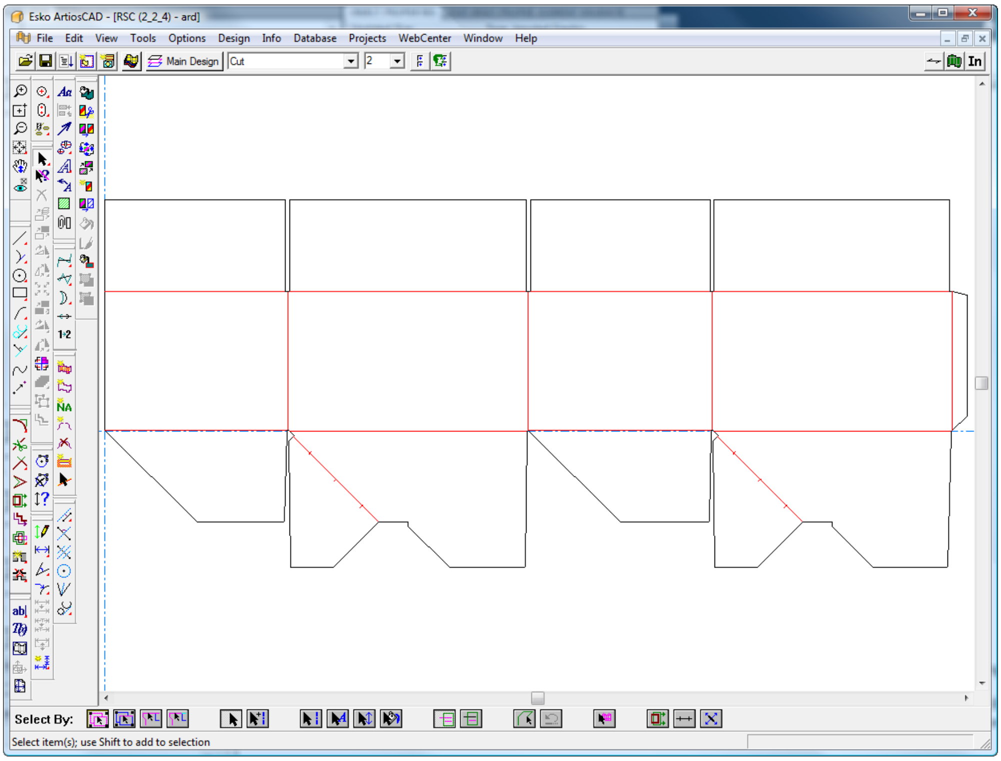


8. When you have found the best solution, click **File** > **Export and Exit**.

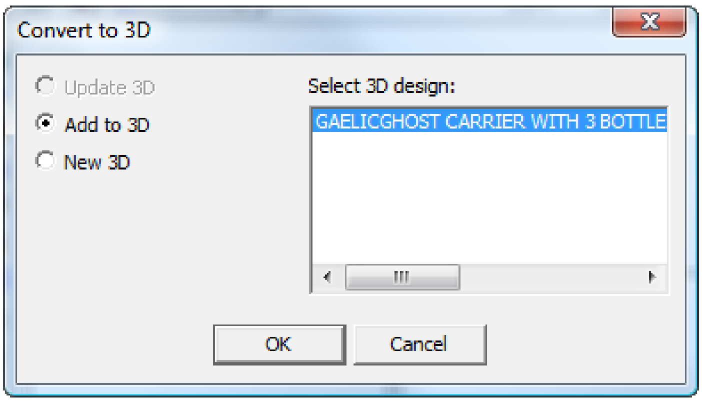


If you do not have either the styles or boards mapped as described previously, ArtiosCAD will prompt you to choose the missing elements.

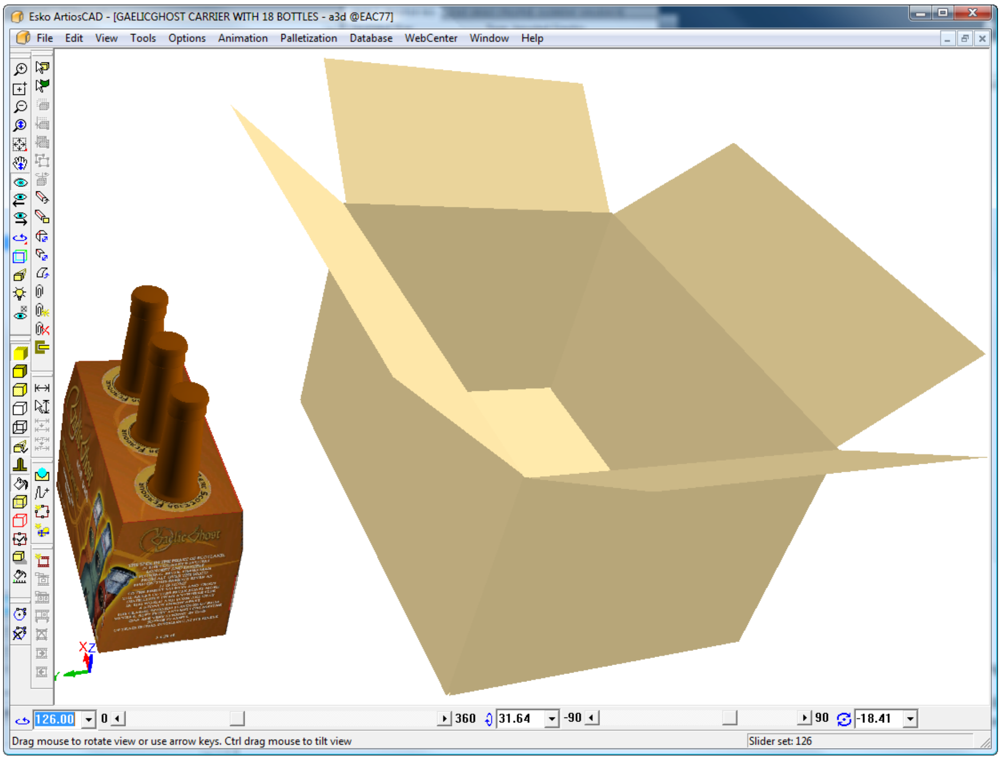
9. ArtiosCAD displays the standard it constructed using the CAPE data. Work with the single design as you would any other workspace.



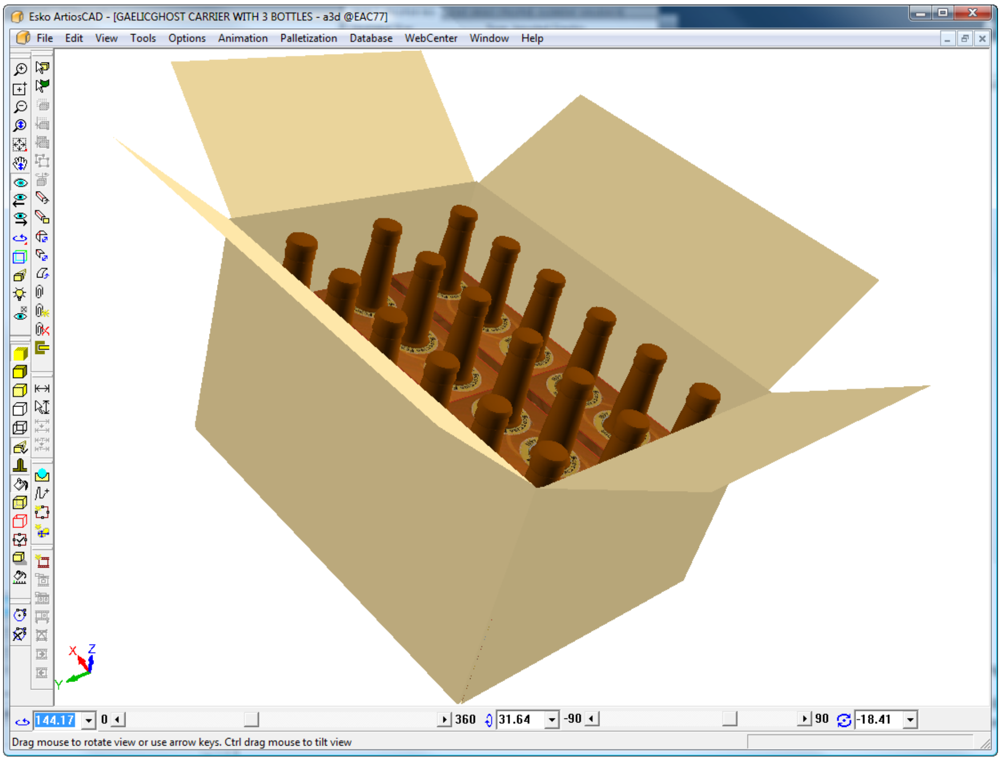
10.  Click **Convert to 3D** and then **Add to 3D** with the primary package's workspace selected, and then click **OK**.



The secondary package appears in the 3D workspace.



11. Use the tools in 3D to duplicate and align the primary packaging inside the secondary packaging as desired.

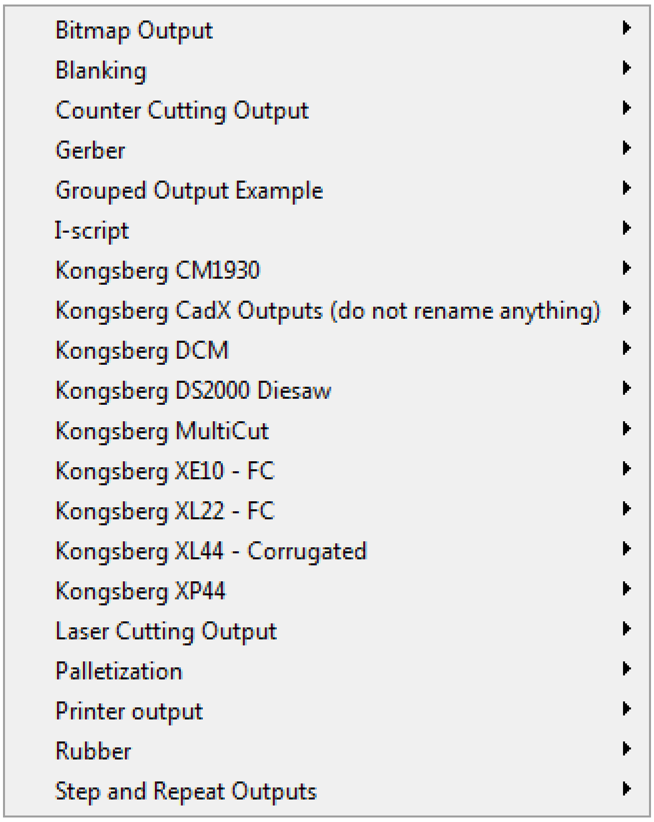


8. Outputs

Introduction to Outputs

Outputs are processes that transform your electronic drawing data into something tangible that you can share with the rest of the world. You can make an output using a report, a simple printout, a sample, or an electronic file. Outputs are configured in the Defaults section of ArtiosCAD. You can customize your own output methods in addition to using those on the server.

Shown below are the default outputs in the default configuration.



There are many options in Outputs that are configurable only in Defaults. For more information on configuring outputs, see the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

Note: The ability to use **Plot to PDF** in the Printer Output folder is an option and must be purchased.

Concepts and Ideas in Outputs

There are two conceptual types of outputs – those that are printed on a printer or plotter, and those created on a machine that is not a printer or plotter. Windows printer drivers are used for most printer outputs. ArtiosCAD CAM (Computer Aided Manufacturing) drivers are used for most plotter and machine output.

You can also create **Grouped outputs**, which as the name implies is a group of outputs. This way you can (for example) output to a printer, send a design via e-mail, and cut the sample all with a few clicks of the mouse. For more information on configuring Grouped outputs, see the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

Multiple workspace outputs support outputting more than one workspace at a time through the same report or to the same device. See the section at the end of the chapter for more information.

Sending data to output devices

ArtiosCAD communicates with each output device using special software called a driver. ArtiosCAD can use its own drivers, as in the case of sending output to a sample maker, or it can use the drivers that came with the operating system, as in the case of a desktop printer.

For more information on configuring devices, refer to the *Peripherals* and *Defaults* chapters of the *ArtiosCAD Administrator Guide.*

Notes and warnings when printing bitmaps

Below are answers to frequently asked questions regarding printing bitmaps from ArtiosCAD.

How do I make a JPEG/PNG/TIFF/Bitmap Output?

Create an Output in Defaults with **Output type Plot** on the Output Type tab, **Driver type Bitmap** on the Device tab, and choose the appropriate format in the **Bitmap Format**: group. Set the output size in pixels in the **Device Size**: fields on the Position tab.

On the Directories tab of the Output, set the extension, such as .JPG for JPEG, .PNG for PNG, .TIF for TIFF, etc.

My bitmap output is very small.

The default bitmap size is 100 by 100 pixels. Increase this to the required size in pixels.

My bitmap output is fuzzy. Why?

JPEG is a lossy compression format (as opposed to lossless) where there is a trade-off between image quality and file size, which blurs the image a little. If your output has dimensions and lines but no graphics, PNG format will work better. PNG compression produces crisp lines and dimensions, but will blend the colors in the graphics.

Poor quality print output. Why?

Make the bitmap size larger, for example 3000 pixels, to have sufficient resolution for printing. Set the width on the Position tab of the Output to match the size of the page in current units.

There is a loss of resolution when I zoom in on the JPEG/PNG. Why?

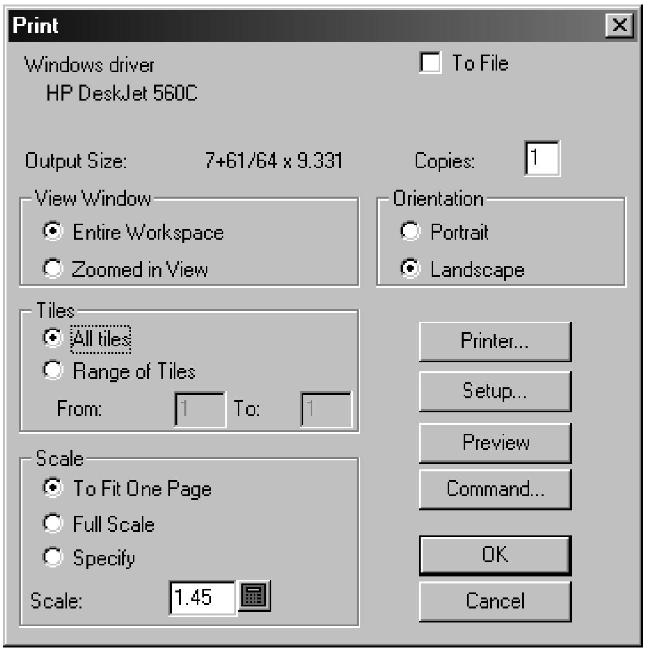
JPEG and PNG are each bitmap formats. A bitmap is a particular size in pixels and does not scale well. If you want an image that can be scaled, try using EPS format instead if you have a program which supports it (such as Adobe Illustrator). A large bitmap with thick lines scales down better than a small bitmap scales up, but bitmaps in general do not scale well.

Why can't I make a GIF file?

There are licensing limitations on software that makes GIF files. ArtiosCAD uses the PNG (Portable Network Graphics) format instead.

Printing

To print the active workspace, click **File** and then click **Print**.



To print the entire workspace on the default printer, click **OK**.

The **To File** checkbox sends the printer data to a file instead of to a printer. Enter the number of copies to print in the **Copies**: field. If you are currently zoomed in on a portion of the workspace, the options in the **View Window** group determine what will print – either the **Entire Workspace** or the **Zoomed-in View** you currently have on the screen.

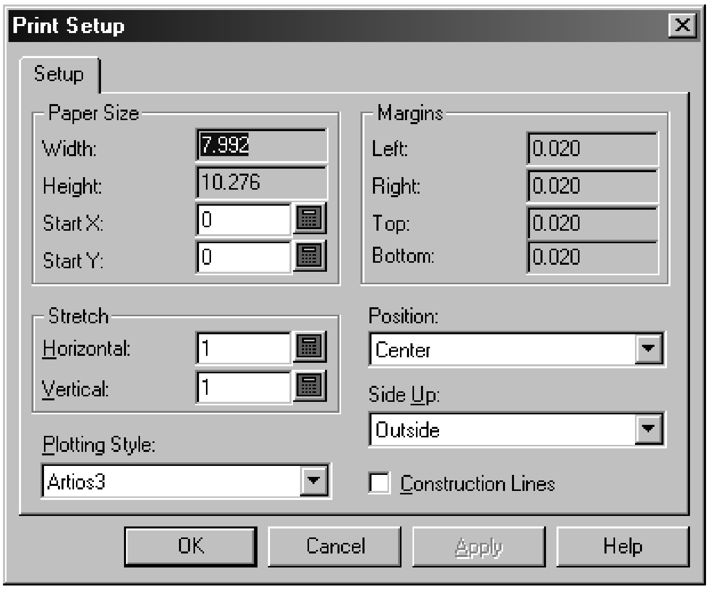
The **Scale** options control the physical size of the printout on the output media.

If the scale of the design (set in the **Scale** field) results in a print larger than the printer can handle, **All tiles** and **Range of Tiles** control what is printed.Tiled outputs have alignment marks on them for easy assembly of the full-size plot. As with other Output options, tiles are configurable in Defaults. The extent to which the alignment marks are printed depends on the margins of your output device.

Portrait and **Landscape** in the **Orientation** group determine how the long axis of the workspace relates to the long axis of the paper.

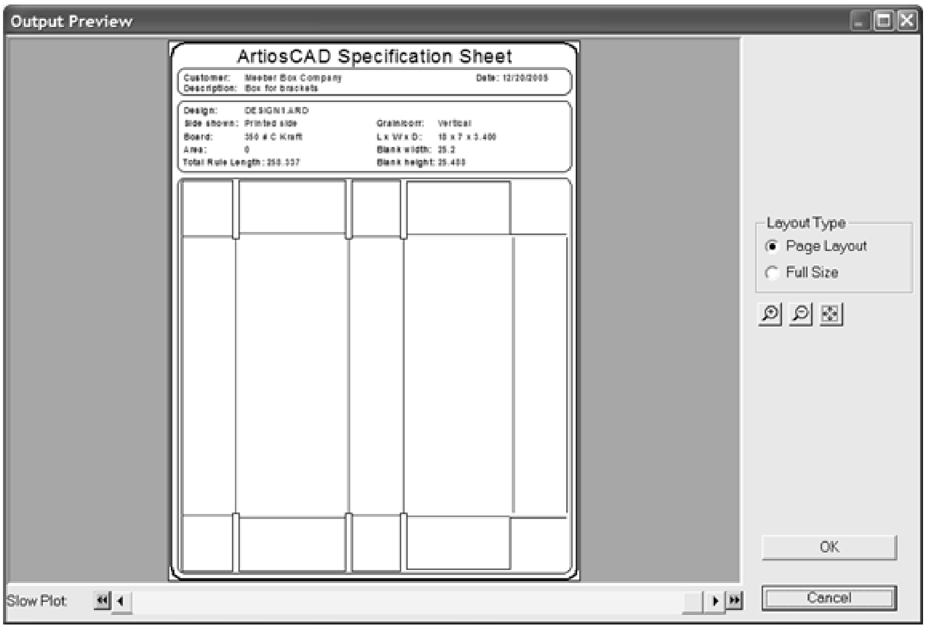
The **Printer** button lets you choose the driver to use with the printer. Choose between printers or plotters defined in the operating system (Windows printer) and printers or plotters driven using native ArtiosCAD drivers (CAM driver).

Clicking **Setup** leads to the Print Setup dialog box where you can adjust settings such as the stretch factor and position of the output on the page.



Preview shows you the data you are sending to the printer or plotter as it will look once printed or plotted. You can set the preview to show automatically in the Preview Control group in the Output's properties in Defaults. Step through the individual line order output by using the **Slow Plot** slider at the bottom of the preview window. An arrow shows the current line. If you zoom in, only the current lines shown are stepped through. **Command** lets you view and modify the actual ArtiosCAD command used to make the output. Do not modify the command unless you are sure of what you are doing, or are told to modify it by Esko Support personnel.

Shown below is the Output Preview dialog box for a Report. Click **OK** to make the output, or **Cancel** to return to the Output dialog box.



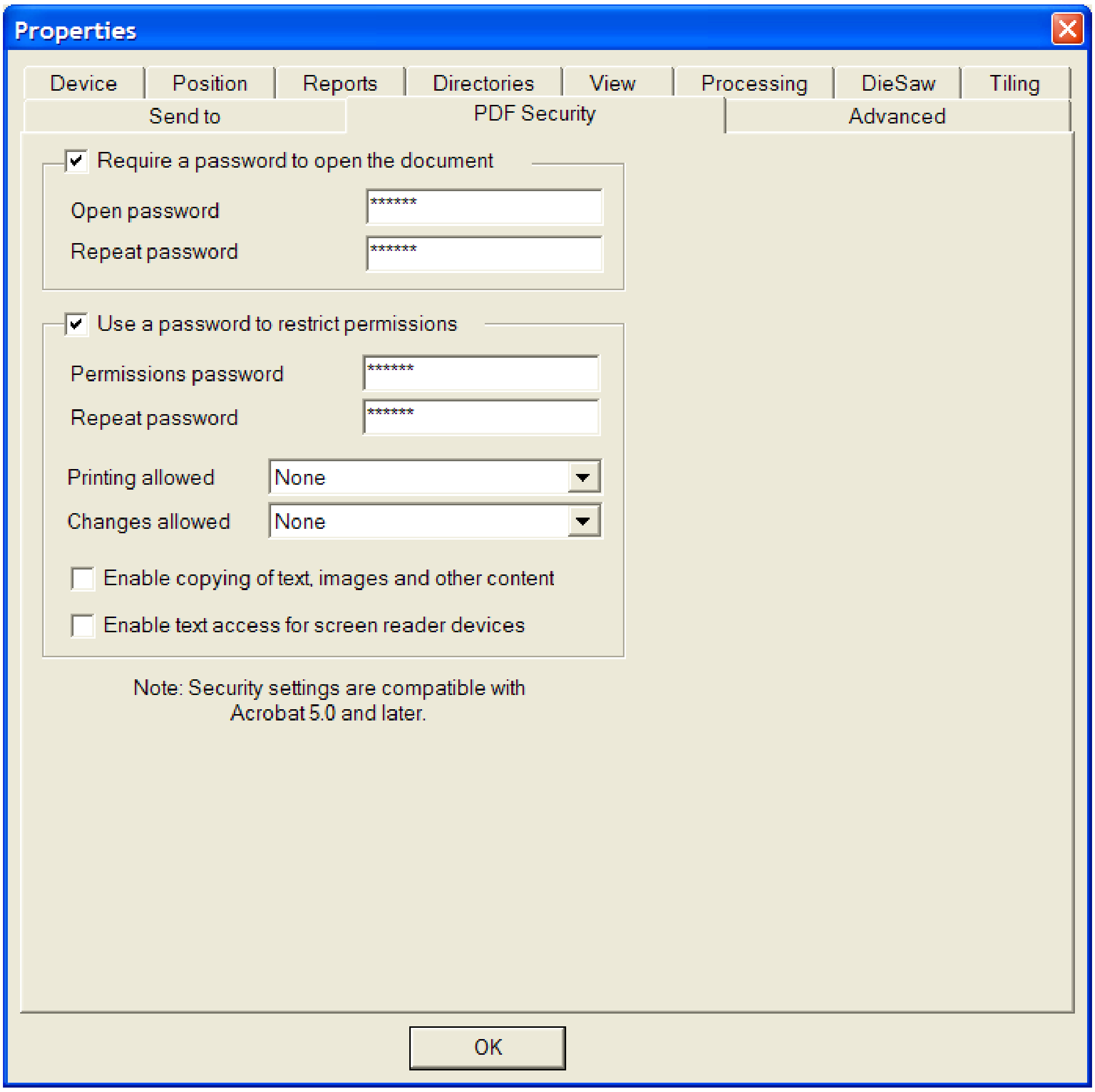
Creating a PDF file

The ability to create PDF files using the native ArtiosCAD driver is an option which much be purchased and may or may not be loaded on your system. Use of third-party software or drivers may lead to unpredictable results.

To create a PDF file, do the following:

1. Create the workspace as usual and turn on the layers to be included in the Output.
2. Click **File** > **Outputs** > **Printer output** > **Plot to PDF**. If you have added any Outputs to the root level, there will be an **Artios** folder to select after you click Outputs. The Plot to PDF dialog box appears similar to the one shown below.

3. Click **Properties** to change the size, position, security, and other aspects of the Output.



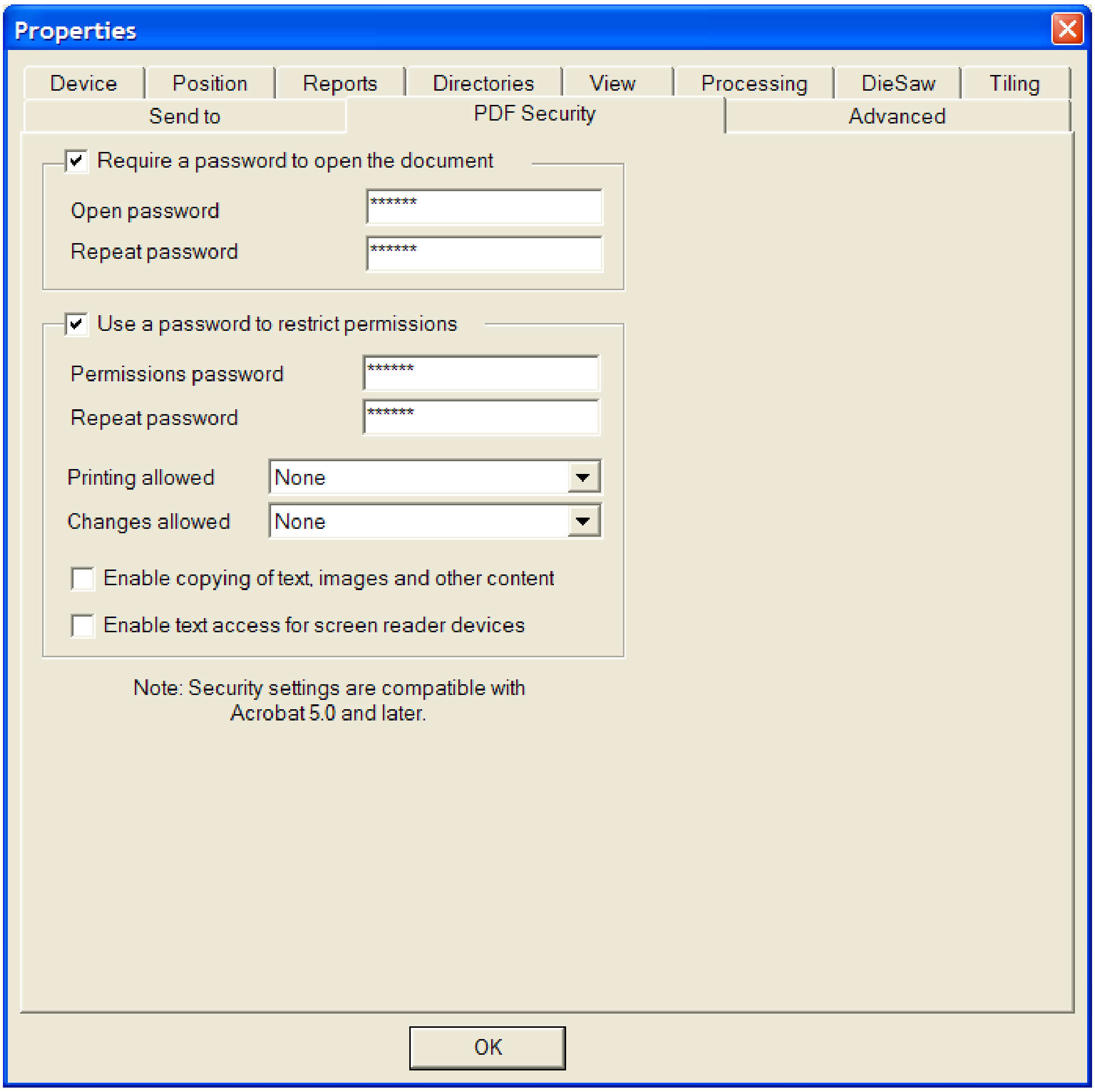
Click **OK** once you have set the options as desired to return to the Plot to PDF dialog box.

4. Click **OK** in the Plot to PDF dialog box.
5. In the Save As dialog box, navigate to the directory in which to save the Output and enter a filename for the Output in the **File name:** field.
6. Click **Save** to make the Output.

Depending on the state of the **Automatically Open** checkbox on the Directories tab of the Output definition, if you have a PDF-viewing utility installed, the PDF file may automatically open once it is created.

Setting PDF security options

The PDF security options available in an ArtiosCAD Output correspond to those used in Adobe Acrobat. They may be set in Defaults just like any other properties.



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.

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 ArtiosCAD with security options set must use Adobe Acrobat 5.0 or later to open the file.

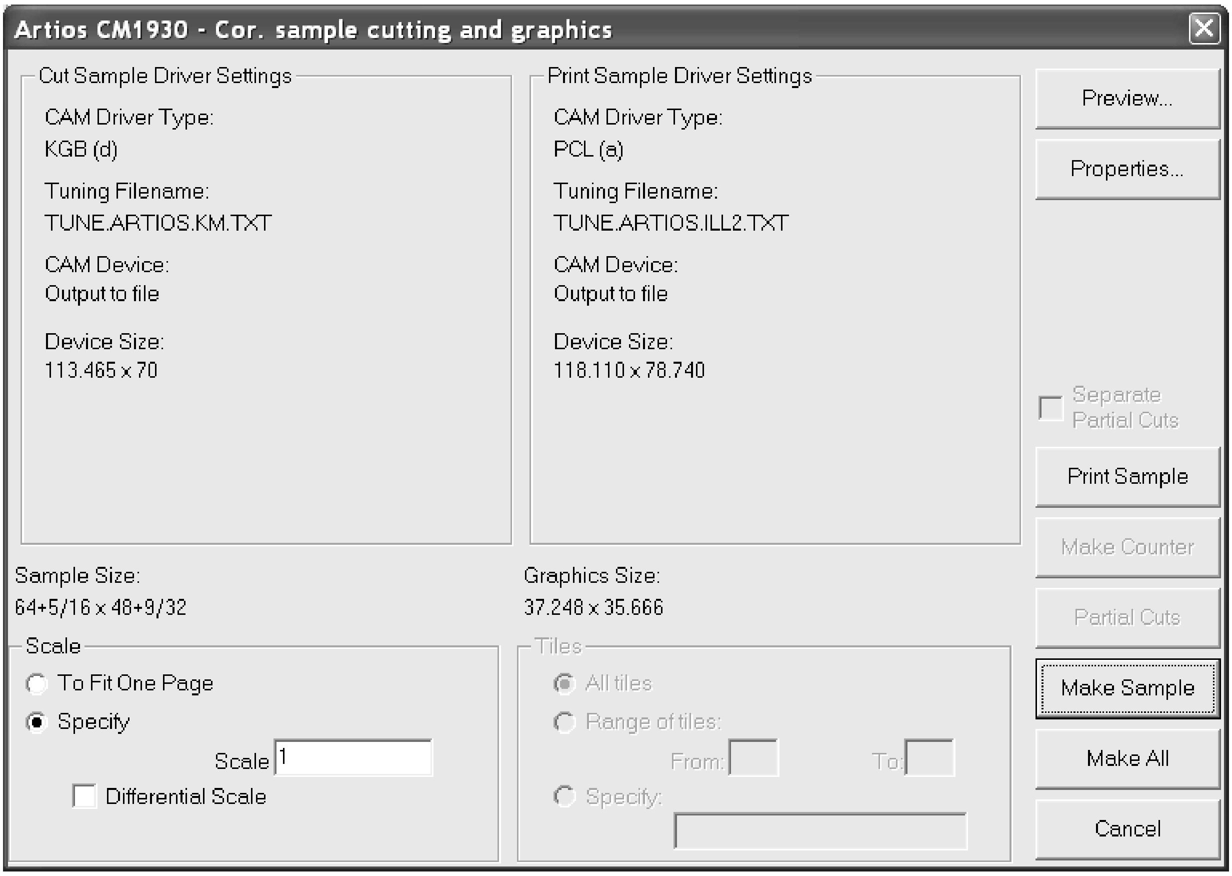
Note: ArtiosCAD cannot open PDF files with an Open password, nor can it open PDF files that have restricted permissions and that do not have content copying enabled.

The **Plot to PDF/U3D** Output is meant to be used only in the optional 3D module, as it requires 3D data.

Samplemaking

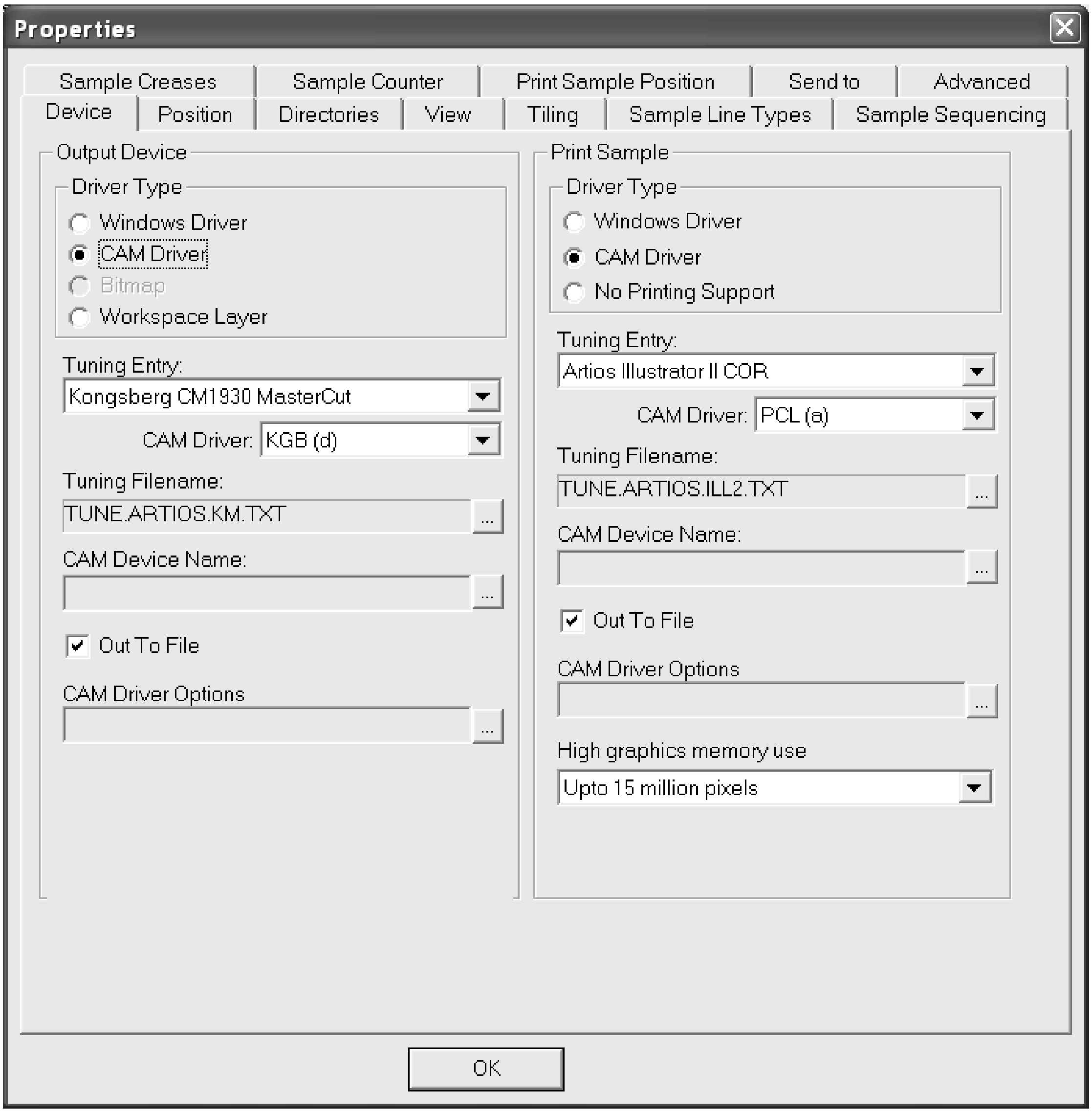
To print or cut a sample, click **File**, then **Outputs**, and then navigate through the menu items until you find your samplemaker. To make navigation easier, you should move your machinery outputs higher up in the hierarchy in Defaults.

Clicking **Artios CM1930 – Cor. sample cutting and graphics** leads to the dialog box shown below.



Preview lets you see what will be output before it is actually output.

Properties leads to the Properties dialog box where all samplemaking and sampleprinting options can be set.



You can customize the behavior of your samplemaker by changing its information in Defaults. All the same options in the Properties dialog box are configurable in Defaults.

Print Sample (back in the samplemaking dialog box) sends the graphic portions of the workspace to the print head on the samplemaker. Note: Text which is in a graphics layer will be printed in black if not given a color. Also, sample printing of Inside layers is not supported.

Make Counter sends the counter information in the Counter layer to a file which can then be interpreted by a samplecutter. The line types in the Counter layer must be set to countercutter line types. Note that this is not used to make a phenolic resin counter - it is used to make a pseudo-counter out of carton stock on a samplecutter. A counter made out of carton stock is also called a **matrix**.

Make Sample cuts the sample.

Make All makes everything that is supported by the selected output device.

Note that some output drivers straighten nearly-straight arcs. Use the GNC driver to preserve nearly-straight arcs. The GNC driver can also facet small arcs to raise and lower the knife for smoother cuts. Contact Support for more information about the GNC driver.

Note for DieSaw users

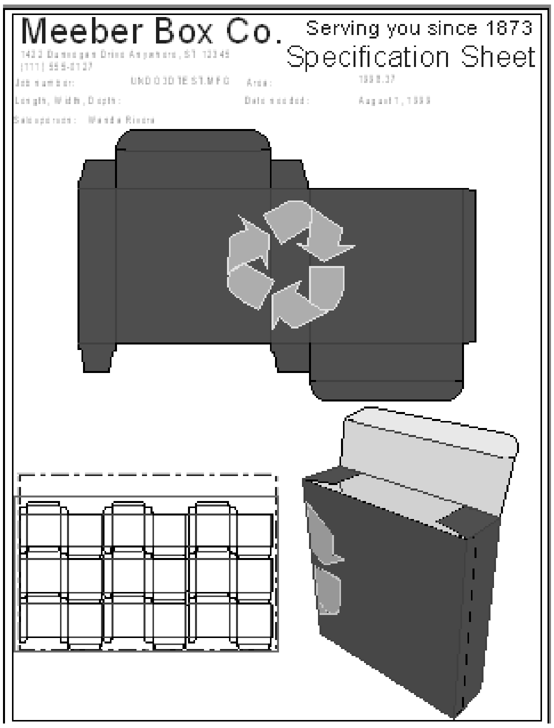
When outputting to a DieSaw, the DieSaw layer is the only layer which is output. Make sure you are not trying to output a different layer when performing a DieSaw output.

Reports

What is a report?

A report is a way to print pictures of workspaces along with information about those workspaces. For example, you could have the name of the design, its dimensions, the customer name, and the job order number on the same printout as a flat view of the design, the design folded in 3D, and the design laid out on the die board.

You must have the ReportMaker feature to modify design windows or work with variables. However, you can change fixed text on pre-made forms if you do not have ReportMaker.



Making a custom report

Although ArtiosCAD comes with several pre-made reports, you will probably want to make some that serve your unique needs. For example, you may want to put your company's logo on one, along with certain items of text that aren't on the pre-made forms.

Making a report usually involves doing the following steps:

- Deciding what types of designs and information you want on your report;
- Defining the size of the report;
- Drawing lines to separate areas from each other;
- Adding text and graphics to the report;
- Designating the part of the report that the design images will go into;
- Adding the report to a Report Catalog;
- Defining an Output that uses the report.

Tools specific to the creation of reports are on the ReportMaker toolbar. The first button on the toolbar is in the Toolbar Master Control and toggles the display of the toolbar.

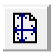
Deciding the contents of the report

Different circumstances call for different information on reports. For example, information going to your sales force probably requires different information than that going to the printing plate machine operators. Customers would probably like to see a folded version of their design along with its pertinent details. Know your audience -- too much information can be just as bad as too little information. You probably already have some sort of shop ticket, job information sheet, or order tracker that can be turned into a report.

You should also decide which file views you want to be on the report. Would a 3D representation help? A manufacturing file view would be for a diemaker. A view of both sides of the workspace could be useful as well. The possibilities are infinite.

Defining the size of the report

Once you have decided what will be on the report, start ArtiosCAD and make a blank design. Turn on the ReportMaker toolbar if it is not on already. The size of a report is usually based on the printer used to print it.

 Click the **Report Size** button on the ReportMaker toolbar.

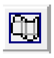
Choose the printer from the **Device** list box. The size of the report will automatically be displayed in the **Report width**: and **Report height**: fields. If you want to rotate the rectangle used to form the edge of the report, check the **Rotate** report checkbox. Click **OK** to accept the settings. A rectangle will appear in the design that is the edge of the report.

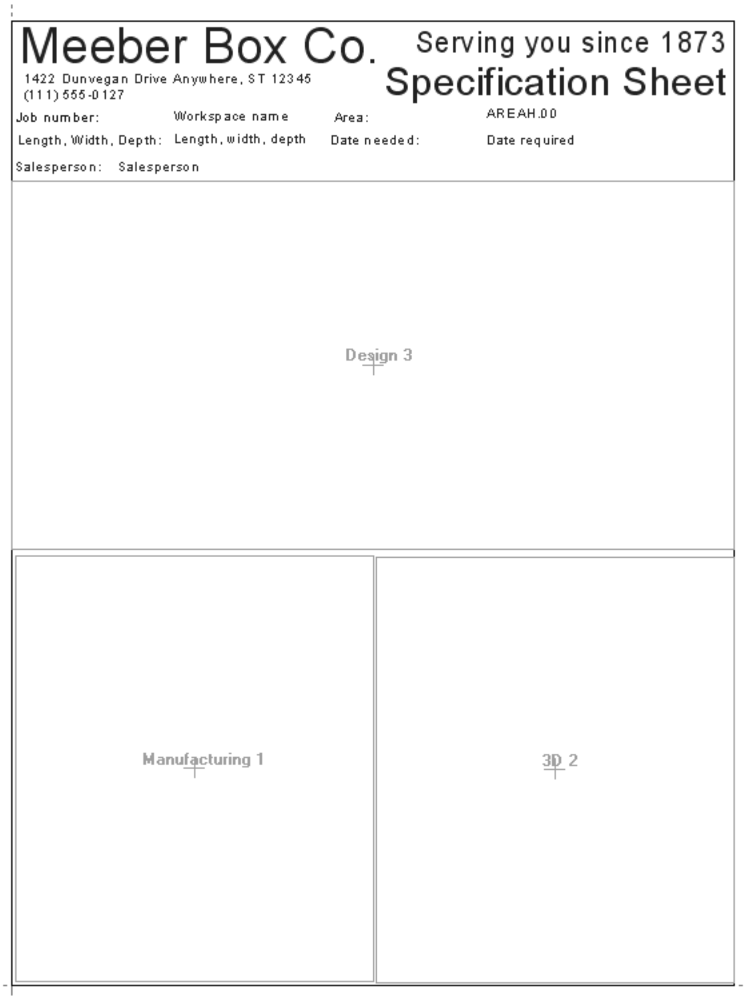
Drawing lines to separate areas

If you want to have lines defining the different areas on your form (for example, you may wish to separate text from graphics), use the **Line** tool to draw them. Lines are optional, as the only thing required on a report is a File Window.

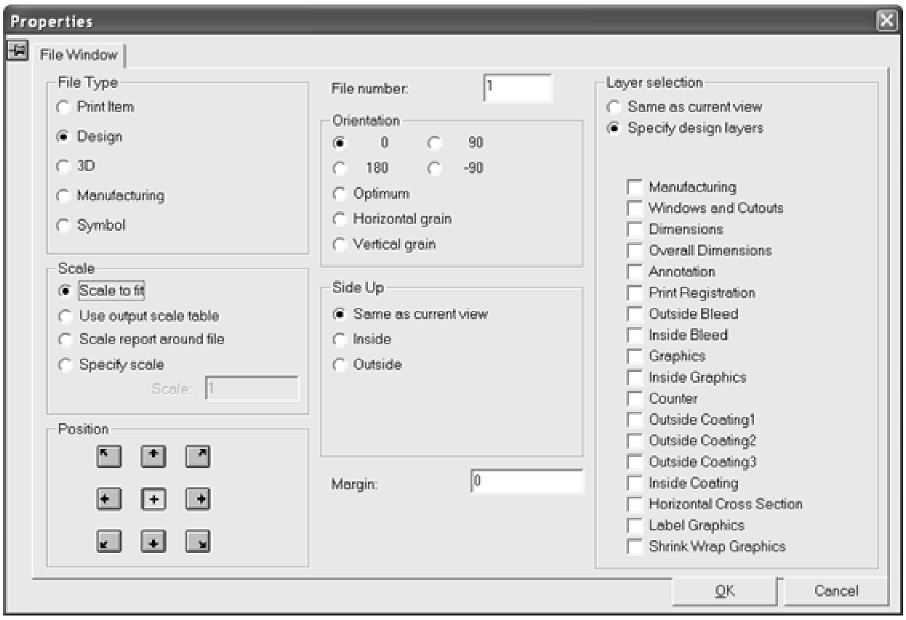
Creating a File Window

File Windows contain symbols, pictures of flat designs, manufacturing files, print items, and 3D files. You can have many File Windows on the same report, each having different contents.

 To create a File Window, click the **File Window** tool on the ReportMaker toolbar. Use the drag to set the diagonal starting and ending points of the rectangle. Alternately, enter values in the **X** and **Y** offset fields on the Status bar to set the corners of the rectangle. After creating a rectangle, the tool is still active in order to facilitate making another. Shown below is a report with three File Windows.



Double-click a File Window with the **Select** tool to change its properties.



Choose the option button in the **File Type** group that corresponds with the type of file which will be shown in the File Window.

The option buttons in the **Scale** group determine the size of the file view within the File Window. **Scale to fit** makes the contents of the window as large as possible while still fitting within the File Window.

Use output scale table instructs ArtiosCAD to set the scale of a File Window according to a predefined table in Defaults. ArtiosCAD will set the scale in that File Window to the closest scale that is less than or equal to the scale derived from the Scale to Fit command. For example, if Scale to Fit would use a scale of 0.8753, but **Use output scale table** is selected, ArtiosCAD will use the scale in the table that is closest to that value without exceeding it, for example 3/4.

Scale report around file expands the report to fit around the file window if the file window contents are initially larger than the report, but does not shrink the report if the file window contents are initially smaller than the report. This option is valid only for Design and Manufacturing type File Windows. If more than one File Window in a report uses this method, the report scale is calculated to the size needed to encompass all the occupied windows. It is meant for use with an Output set to scale 1. If you use a report with a File Window set to this scale option in an earlier version of ArtiosCAD, the File Window will behave as if it were set to **Scale to Fit**. If you open the report workspace in an earlier version of ArtiosCAD, the **Scale** group option is set to **Specify Scale** of -2.

The option buttons in the **Scale** group are not available when the File Window type is 3D or Symbol.

The buttons in the **Position** group control how the file view is positioned within the File Window. The buttons around the edge of the square align the design with the corresponding point of the File Window. The center button (+) aligns the center of the design with the center of the File Window.

The **File number** indicates which file will be displayed in the File Window. The **File Number** prompt changes to **Print Item Number** prompt if **Print Item** is selected in the File Type group. When you create a report containing more than one File Window, increment the File number by 1 for each File Window. When outputting the Report, the current design is file number 1; any other open designs are assigned numbers 2 onward. When using a Manufacturing report, make sure the manufacturing file is the active workspace - so that it is in File Window 1.

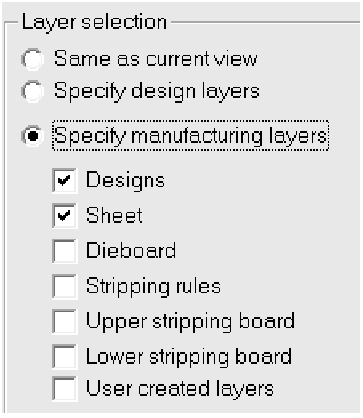
Note: Make sure each File Window has a unique number, except if you have multiple Design File Windows of the same number that are meant to show different layers of the same design, as long as those layers are not Manufacturing or Windows and Cutouts, as those might cause problems with calculated expressions. For example, multiple Design File Windows with the same number would be permissible when one is showing the Outside Graphics layer, another is showing the Dimensions layer and the flat view, and yet another is showing the Inside Graphics layer. Do not assign File Windows of differing types the same number - for example, do not have Design 1, 3D 1, and Manufacturing 1. The report may not work correctly if you have same-numbered File Windows for different types of workspaces. If you are planning a report with a manufacturing file and single designs, make the manufacturing file as File Window 1 and make the other File Window types 2, 3, 4, and so forth.

The option buttons in the **Orientation** group determine the rotation of the design in the File Window. **0**, **90**, **180**, and **-90** all rotate the contents by a fixed amount. **Optimum** sets the orientation so that the largest scale is obtained. **Horizontal grain** and **vertical grain** set the design so that the grain direction chosen is drawn along the longer axis of the File Window.

The **Margin** specified indicates the distance from the edge of the File Window to the outermost edge of design.

Options buttons in the **Side Up** group control which side of the design is shown.

The option buttons in the **Layer Selection** group control which layers are shown. **Same as current view** shows the design on the report the same way it is being viewed in ArtiosCAD. **Specify layers** lets you choose which layers to show on the report regardless of the view at the time of output. If the **File Type** is set to Manufacturing, another option button appears which allows you to specify those Manufacturing or Design layers to output.



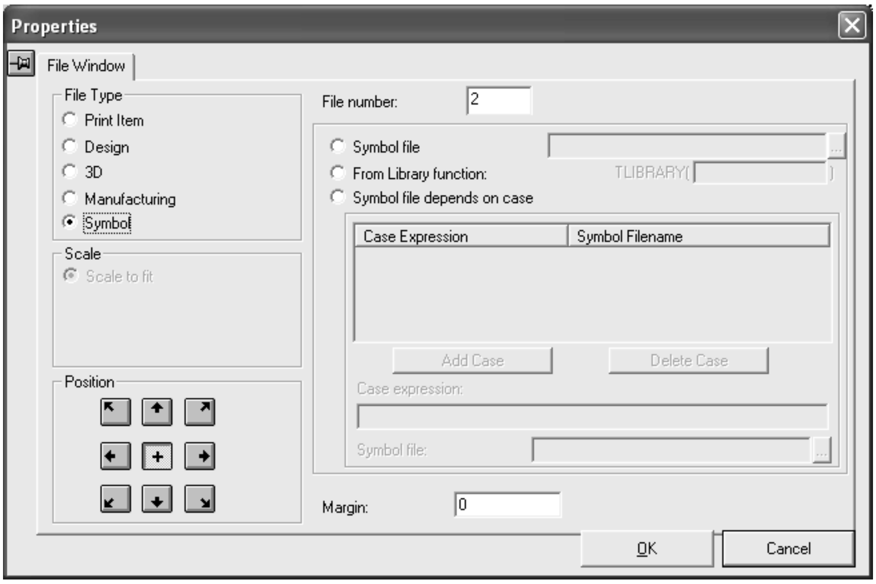
The layers selected when **Specify design layers** and **Specify manufacturing layers** are active are remembered when the other option button is selected. For example, if you select **Inside Bleed** and **Graphics** when **Specify design layers** is active, they will be remembered when **Specify manufacturing layers** is active. If you do not select any design layers for a Manufacturing file window, only the Main Design layer is shown when **Designs** is selected in the Manufacturing layers.

Using a Symbol file window

The **Symbol** type of file window is used to hold a symbol, which is a graphic. This graphic can be lines and arcs drawn and filled in ArtiosCAD, or it can be a graphic file of the following format and extension: BMP, DIB, JPG, PNG, TIFF, and PDF (only if you have purchased the PDF option).

You can:

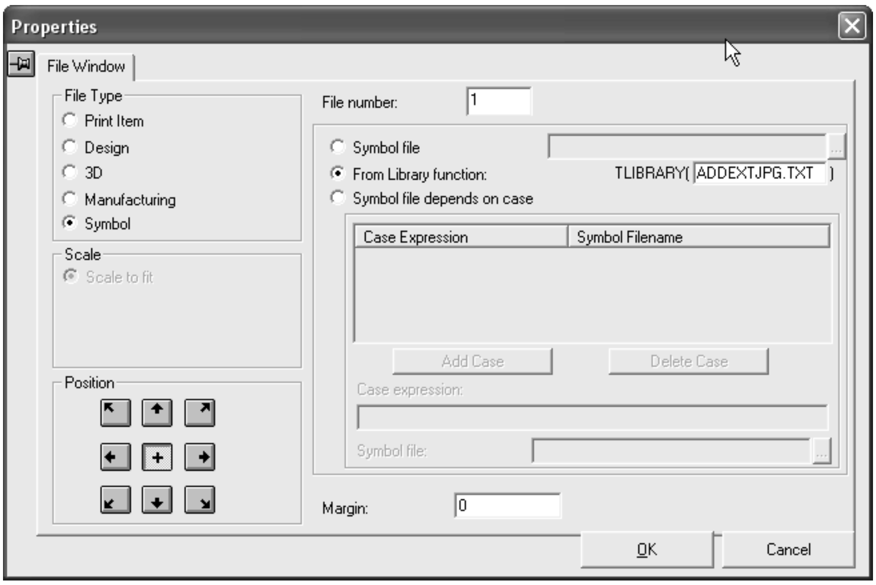
- specify the exact symbol to use;
- specify a library function which will determine the name of the symbol;
- build a conditional statement that instructs ArtiosCAD to use a different symbol when the specified conditions are met.



When the File Window is set to type **Symbol**, the Scale is automatically set to **Scale to fit** and the Position is automatically set to centered horizontally and vertically. As with other types of File Windows, the value in the File Number field controls which element of the current workspace corresponds to the symbol window.

To use the same symbol file each time the report is accessed, click the **Symbol File** option button and enter the filename of the ArtiosCAD workspace containing the symbol.

The **From library function** option button lets you use a library function to return the symbol filename to ArtiosCAD.



Perhaps you have corresponding JPG files for each design you make that have the same name as the design file. Including them automatically on a report can be done using a library function as described below.

A library function such as ADDEXTJPG.TXT is a short program allowing the filename for the contents of a Symbol-type file window to be defined automatically as the Report is run.

Note: This library function is not part of the ArtiosCAD installation; to use it, you must create it in a text editor such as Notepad and save it in `ServerLib`.

The library function will typically use the function `#PATHNOEXT$` to get the pathname of the current design without the .ARD extension. The last line of the command file returns the pathname to be used. The following example would return `pathname.JPG` for a design `pathname.ARD`.

```
DEFINE &FNAME$
```

```
SET &FNAME$ #PATHNOEXT$,”.JPG”
```

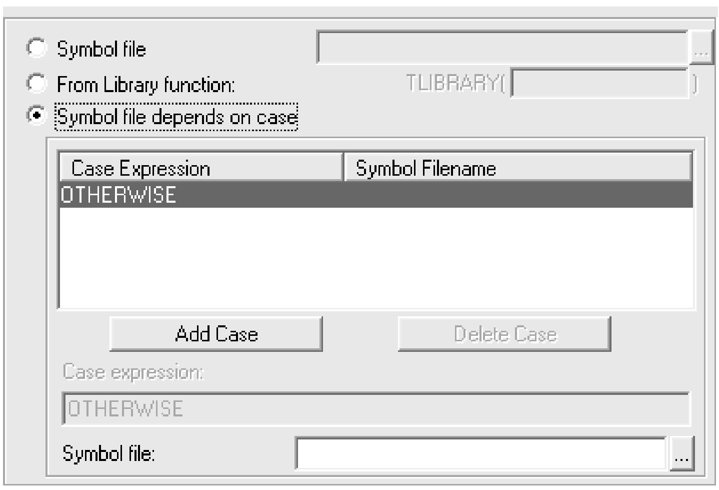
When the Report runs in an Output, the library function for the pathname is evaluated for each file window, and the image from that pathname is scaled to fit in the rectangle defined by the file window item. The pathname may refer to an ArtiosCAD design, such as **GRAIN.ARD,** or an external image such as **1234.JPG**.

The report may contain several file windows, each referring to a different image, such as a picture of the product, and a palette diagram. For each image file that does not exist, a text message is centered in the file window with the pathname of the missing file. This diagnostic is useful for instances such as a missing grain symbol. To avoid this message for designs that do not have a corresponding image file, the library function that sets the filename can be set to ignore any missing files as shown below:

```
DEFINE &FNAME$
SET &FNAME$ #PATHNOEXT$,".JPG"
IF FILEEXISTS(&FNAME$)
SET &FNAME$ &FNAME$
ELSE
SET &FNAME$ ""
ENDIF
```

Again, to use the library function, you have to enter it in a text editor and save it in `ServerLib` before you specify it in the Properties dialog box for a symbol-type file window on a Report.

To configure the report to use a different symbol depending on the result of an evaluated expression, select the **Symbol file depends on case** option button. When you select this option button, the **Case** group becomes available.



To add a case, do the following:

1. Select the **Symbol file depends on case** option button.
2. Click **Add Case**.
3. Enter the expression to be evaluated in the **Case expression** field. Use the same syntax here as that used in any Edit Expression dialog box.
4. Enter the full name and path of the desired symbol file in the **Symbol file** field. Click the button at the end of the field to browse for the file.
5. Add more case expressions by clicking **Add Case**. Delete a case expression by selecting the one to delete and then clicking **Delete Case**.

There will always be an `OTHERWISE` case listed when **Symbol file depends on case** is selected. The expression entered in this field is the one ArtiosCAD will use when none of the other case expressions are valid. This field can be left blank if you know that the condition will always be met.

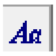
Adding text to the report

Three types of text are used on reports:

- Fixed text, which never changes;
- Prompted text, which you can change each time you use the report;
- Calculated expressions, which is text generated automatically each time you use the report.

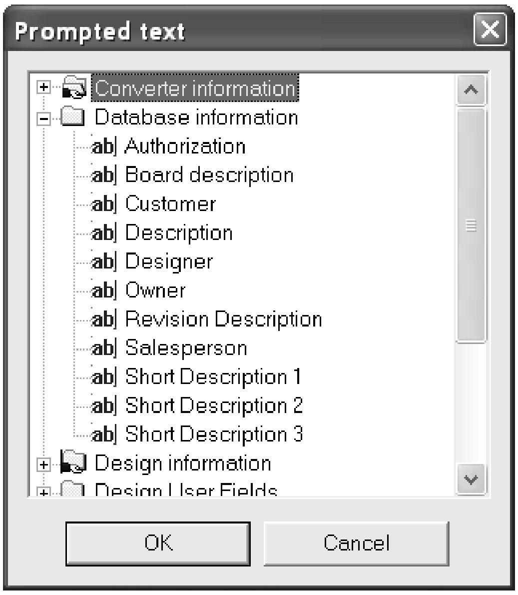
The maximum amount of text allowed in an item of text is 4000 characters.

Adding fixed text


Adding fixed text is accomplished by using the normal **Paragraph Text** tool on the Annotation toolbar. See **Paragraph Text tool** in the *Builder* chapter for more information.

Adding prompted text


Add prompted text by using the **Prompted Text** tool on the ReportMaker toolbar. Click the tool and place the text in the desired location. The Prompted Text Catalog will appear as shown below; yours may be different if you have different defaults defined. Click the plus sign to the left of a category to expand it. Then select an entry and click **OK**.



The cursor will still be the one for prompted text, so if you click in another location, the Prompted Text Catalog will reappear so that you may add another entry. When you have finished adding prompted text, click Cancel in the Prompted Text Catalog.

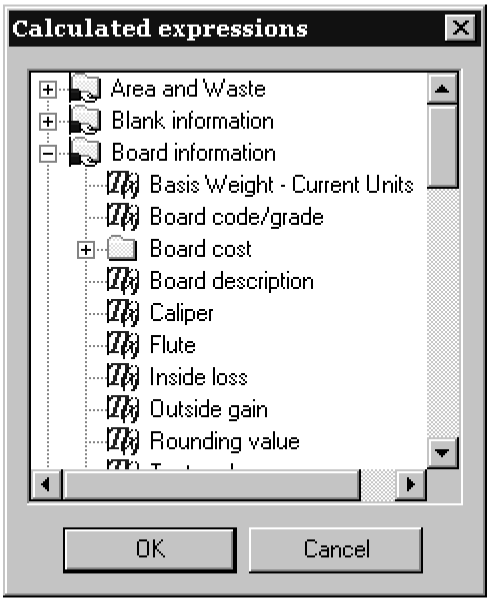
If you place a prompted text variable within a rectangle of design lines, the text entered for that prompted variable will wrap to the confines of the rectangle. If there is more text than will fit, ArtiosCAD will print as much as possible and end with an asterisk.

The **Database Information**, **Design User Fields**, and **Manufacturing User Fields** Catalogs link to the database, so that when you output the Report, you are prompted to set those fields. The values you set are saved in the database. Reports with database prompted text items may not be used in versions of ArtiosCAD lower than ArtiosCAD 6.0.

To copy database variables into another design or standard style for use when rebuilding a design, use the **Copy** button in Advanced StyleMaker as described in the *Designer* chapter, *Adding the Inside Dimensions menu*, except use the name of the report workspace containing the variables instead of VARMASTER.

Adding calculated text

 Add calculated text by using the **Calculated Text** tool on the ReportMaker toolbar. Place the indicator box in the desired spot and click the mouse button. The Calculated Expressions Catalog will appear.



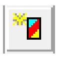
Choose the calculated expression to add and click **OK**.

The tool will remain active; clicking again will place another item of calculated text and will cause the Calculated Expressions Catalog to reappear. When you are done adding calculated expressions to your report, click **Cancel** in the Calculated Expressions Catalog.

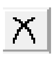
If you place a calculated text expression within a rectangle of design lines, the text generated for that calculated expression will wrap to the confines of the rectangle. If there is more text than will fit, ArtiosCAD will print as much as possible and end with an asterisk.

A special calculated text entry for the length, width, and depth of the design where not all three variables are required to be set is the `[L] [xW][xD]` entry in the **Design information** > **LxWxD sizes** folder. Use this entry to print those values if you are not sure if they will be set when the Report is run. Values which are not set will be blank.

Adding graphics to the report

 Adding graphics to a report is as simple as using the **Add Graphics** tool on the Graphics toolbar. Navigate to the directory the graphic file is in and double-click its name. The graphic will appear in the report and its handles will be on. Click a handle to use to move the graphic to the desired location, and then drag the graphic to its new position.

Modifying items on the report

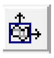
 To delete an item from a report, select it and click the **Delete** tool on the Edit toolbar.

Move text by selecting it with the **Select** tool and dragging it to its new position. The Align Text tools are handy for aligning fixed text and prompted text items into columns. Move graphics with the **Move** tool.

To change the attributes of any text, double-click it with the **Select** tool. The Properties dialog box will appear. Do not change the **Text** field of Prompted Text items or Calculated Expressions.

Use the tools on the Edit toolbar to modify a graphic if desired. The tools on the Graphics toolbar also modify any graphic item on a report.

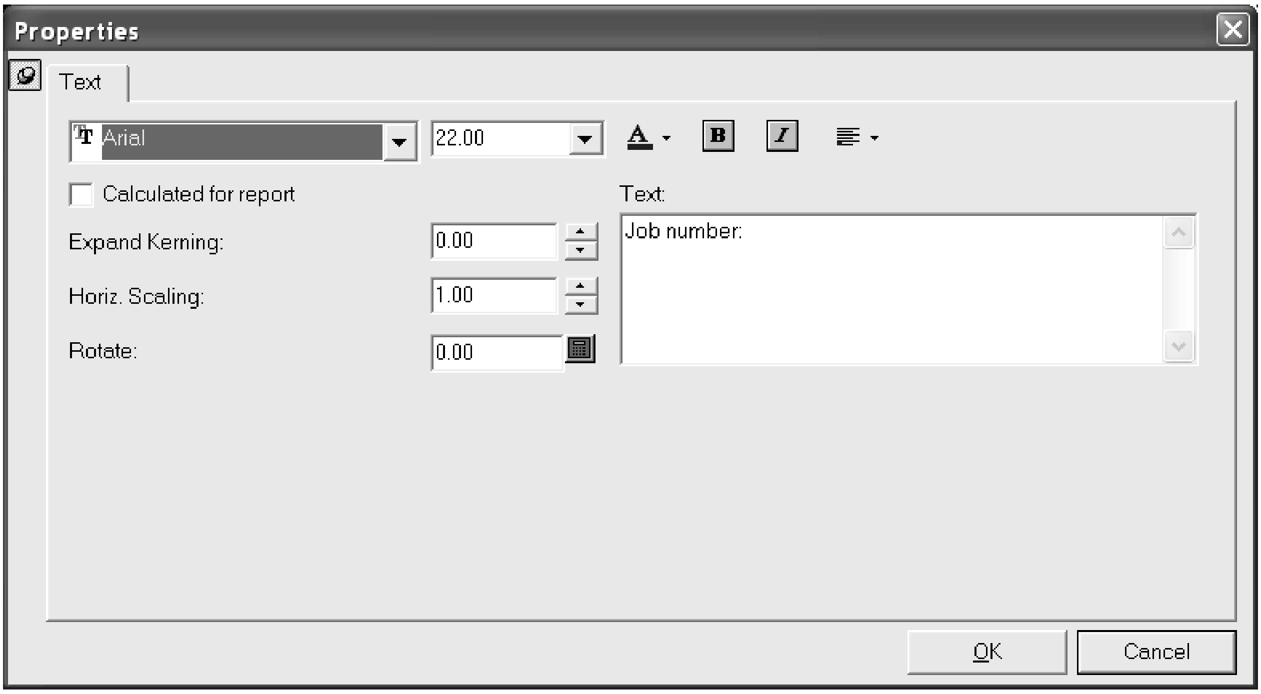
Changing the size of a File Window

 Use the **Adjust File Window** tool to change the size of a File Window by doing the following:

1. Click the **Select** tool and then select the File Window to modify.
2. Click the **Adjust File Window** tool.
3. Click the handle point to use. This is the point you will be moving to change the size of the file window.
4. Move the handle point using drag or by entering new values in the Status bar fields.

Modifying fixed or prompted text

To change an item of fixed or prompted text, double-click it with the **Select** tool to invoke the Properties dialog box.

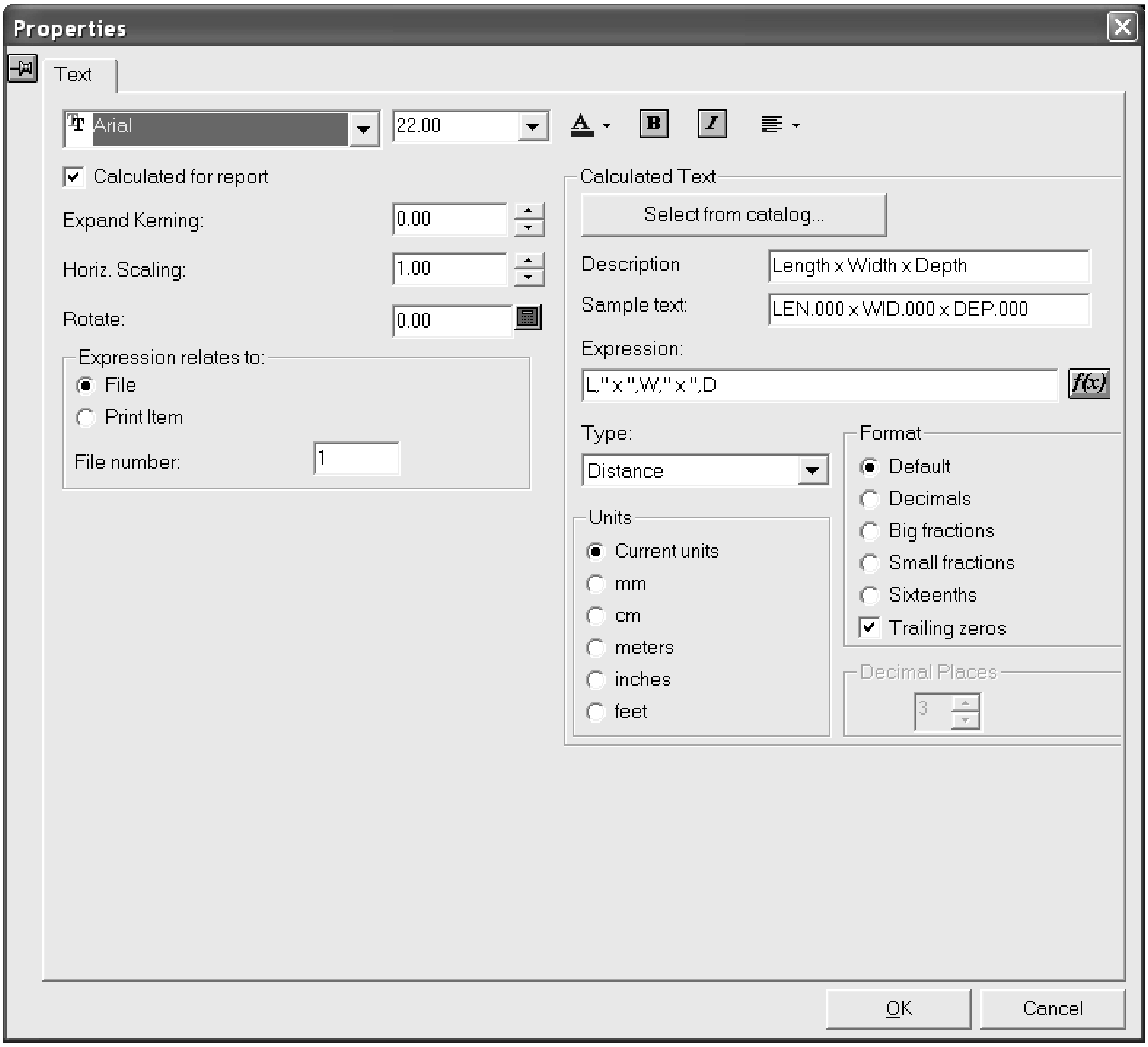


Modify the text appearance and justification by using the controls along the top of the dialog box. **Kerning** controls the amount of space ArtiosCAD inserts between letters. **Horizontal scaling** increases the horizontal size of the characters without changing the vertical size. **Rotate** changes the angle of the text. Note that changing the text justification may be required to position the text exactly as desired.

To change the text itself, change the text in the **Text** field.

Modifying calculated text

To modify calculated text, double-click it with the **Select** tool to invoke its Properties dialog box.



Modify the text attributes in the usual ways. **Kerning** controls the amount of space ArtiosCAD inserts between letters. **Horizontal scaling** increases the horizontal size of the characters without changing the vertical size. **Rotation** sets the angle of the text from horizontal.

In the **Expression relates to** group, make sure you have the correct file or print item selected, otherwise incorrect information will print on the report.

The **Calculated Text** group contains the options which affect the actual information displayed when the report is run, not just the font options of the information. **Description** is what shows in the menu when you select the item of calculated text. **Sample text** is what appears as a placeholder in the design window while you are building the form. **Expression** is the ArtiosCAD expression which calculates the value. The **f(x)** button lets you change the way the expression is calculated by invoking the Edit Expression dialog box. Do not use invisible variables in calculated expressions. **Select from**

catalog lets you replace the current item of calculated text with a new one you choose from the Calculated Text Catalog.

The choice in the **Type** drop-down list box determines the availability of the **Units** and **Format** groups as well as the options within them. For instance, if the Type is **Distance**, the Units are plain linear measurements, but if the Type is **Area**, the Units are squared. If the Type is **Angle or Number** the Units option buttons are unavailable, and if the Type is **Text** both the Units and Format groups are unavailable. Use **Distance** for expressions which are units of length. Use **Area** for units which are squared. **Angle or Number** should be used for numbers with no units such as number up (on a die). **Text** should be used for text or for expressions which contain more than a single number, such as **Length, Width, Depth**.

Decimal Places controls how many decimal places are used and is only effective when anything but Default is chosen in the **Format** group.

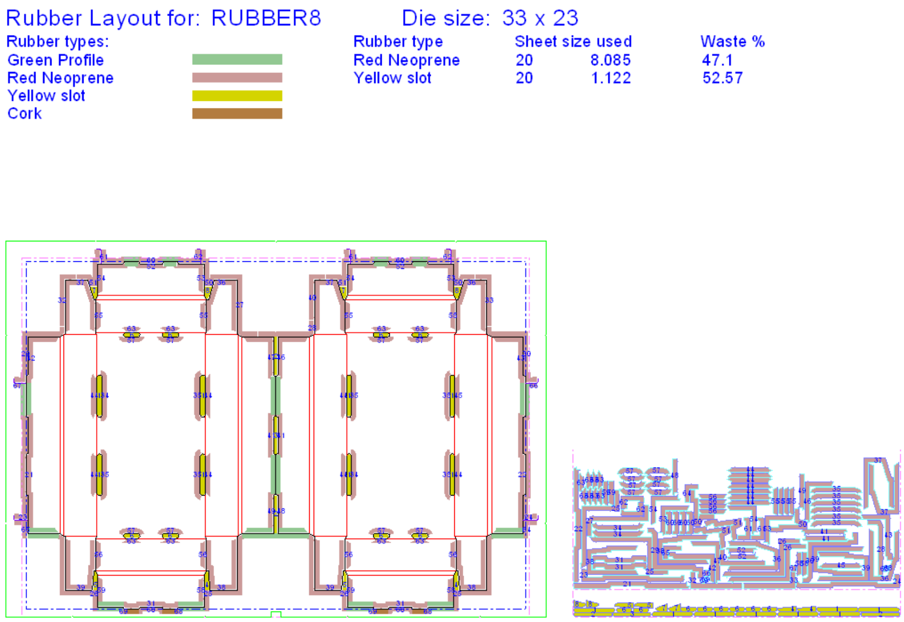
Rubber on Reports

You can create a custom Rubber Types legend on a Report using the calculated text items below. An example Rubber Types legend is included in the **Artios** example outputs.

Table: Calculated expressions for a Rubber Types legend

Description	Calculated text expression	Example
Rubber type name in the list of rubber types	`RTNAME(` **rtype** `), rtype =1,2,3,4,...`	`RTNAME(1)`
Rubber color block in the list of rubber types	`RTLINE(` **rtype** `)`	`RTLINE(1)`
Rubber type name in the list of rubber sheets	`RUBT.R[RUBS.S` **rtype** `.RT].NAME$`	`RUBT.R[RUBS.S1.RT].NAME$`
Rubber sheet width	`RUBS.S` **rtype** `.DX*#M`	`RUBS.S1.DX*#M`
Rubber sheet height	`RUBS.S` **rtype** `.DY*#M`	`RUBS.S1.DY*#M`
Waste %	`RUBS.S` **rtype** `.WASTE`	`RUBS.S1.WASTE`

Shown below is an example layout with rubber sheets on a Report with a Rubber Types legend.



To have each rubber sheet on a separate page, use the reports in **File** > **Outputs** > **Artios** > **Rubber** > **Rubber Separate Sheets**. A separate Report is provided for rubber sheets 1 through 5. The rubber sheet number for the file window is derived from the text item called **rubbertypename5** with calculated text `RUBT.R[RUBS.S5.RT].NAME$`.

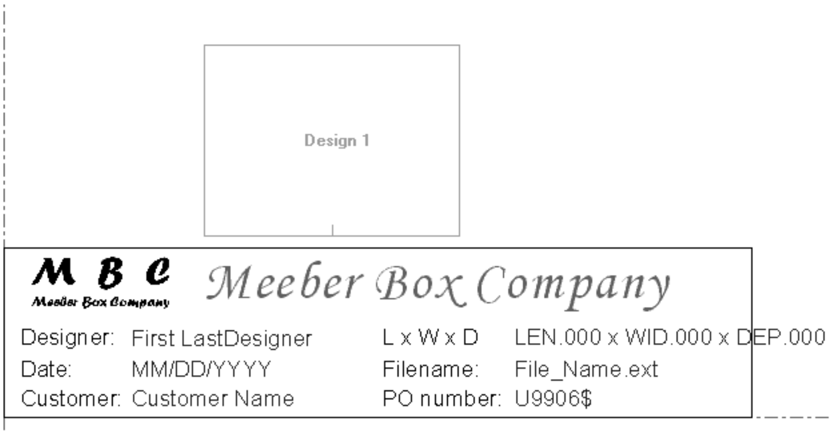
Saving the report

When you are done constructing the report, save it to either the `ServerLib` or `ClientLib` directory of the ArtiosCAD release you are using in order to be able to easily add it to the Reports catalog. Also, saving it in one of those directories preserves it through software upgrades.

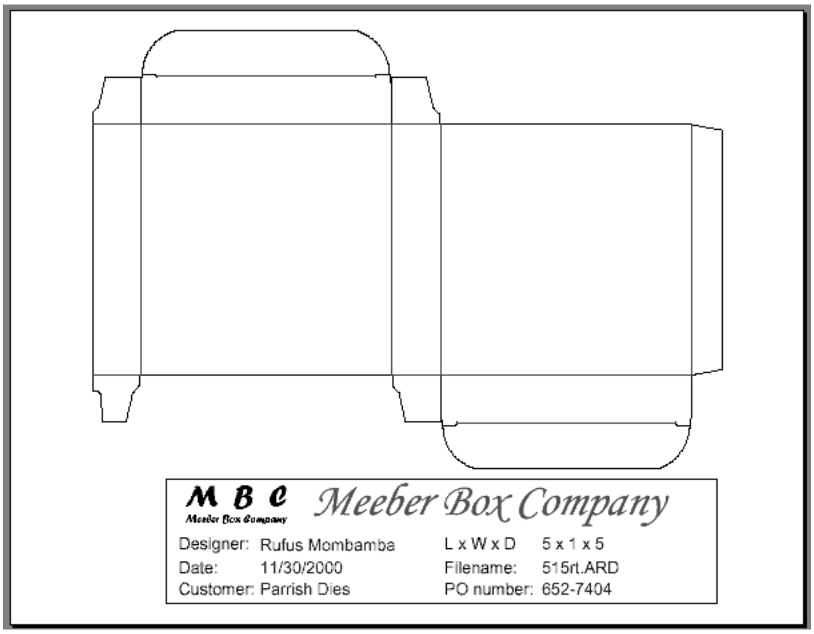
Making a title block report

Most reports scale the design to fit inside a rectangle in the report. A **title block** report has the design at a specified scale, usually scale 1, with the report text positioned relative to the edge of the design.

Create a title block report by designing the title block, and adding the file window for the design with a fixed scale. Set the justification of the file window to position the title block at the required position relative to the design. For example, to position the title block at the bottom center of the design, use bottom center justification for the file window. Shown below is an example.



When a title block Report is used in an Output, the design is positioned in the file window according to the justification of the file window. Set the scale of the output to fit one page. The actual size of the file window is not used; it does not make any difference if the design is bigger or smaller than the file window. Shown below is an example of a completed Output.



Notes and warnings about Reports

When you want to output a report which will show a manufacturing file and its embedded single designs, do the following:

1. Assign File Window 1 the **Manufacturing** file type in its Properties dialog box.
2. Assign File Windows 2..**x** the **Design** file type in the Properties dialog box.
3. When you Output the report, **make sure** the manufacturing file is the active document. File Windows 2..onward are filled in the order in which they are displayed in the Oneups Dialog in Manufacturing. Thus, the first embedded file would appear in File Window 2, the second embedded file would appear in File Window 3, and so forth.

4. If you have more file windows on the report than you have embedded designs in the manufacturing file, the remaining file windows will be empty. You can assign other open files to them if you wish.

When putting text on a Report, you should always put it inside a closed shape made out of cut lines. The text is truncated against the perimeter of the report if the scale gets too large on Output. This prevents text from running off the page if the scale is too big.

Using the ITEMORD function on Reports

The ITEMORD() function returns the ordered quantity of a print item. The function argument refers to the quantity number. If you have a multiple quantity report for use with manufacturing files, the ordered quantities should have calculated text:

ITEMORD(1) print item 1, ITEMORD(2) print item 1, ITEMORD(3) print item 1, and so on.

If you have a report for a mixed layout, the ordered quantities on the report should have calculated text:

ITEMORD(1) print item 1, ITEMORD(1) print item 2, ITEMORD(1) print item 3, and so on.

Adding a report to the Reports Catalog

Once the report workspace is saved, click **Defaults** on the **Options** menu of ArtiosCAD.

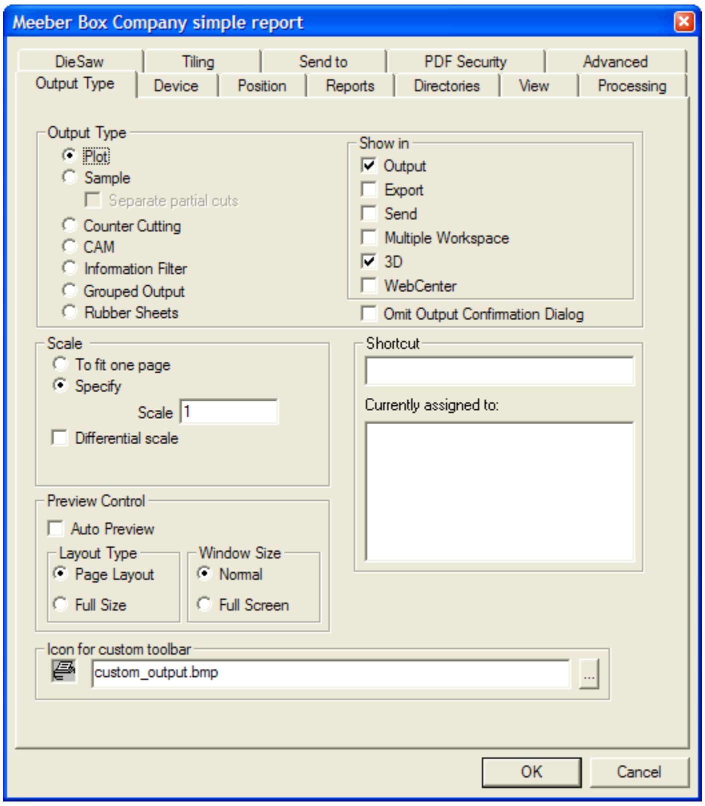
To add the report to the Shared defaults, which will let everyone using this installation of ArtiosCAD use it, do the following.

1. Open the Report Catalog in the Shared Defaults window.
2. Right-click **Report Catalog**, click **New**, click **Data**, and enter the name of the re-port as you want it to appear in the Report Catalog.
3. Then double-click its icon and enter the name of the report work-space. It is a good idea to use a copy of the report workspace which you have saved in `ServerLib` as this will be preserved through future upgrades.
4. Click **OK** when you have entered the name.
5. Save your changes to Defaults by clicking **File** > **Save**.

Once the report has been added to the Reports Catalog, it must be added to Outputs in order for it to appear in the Outputs menu.

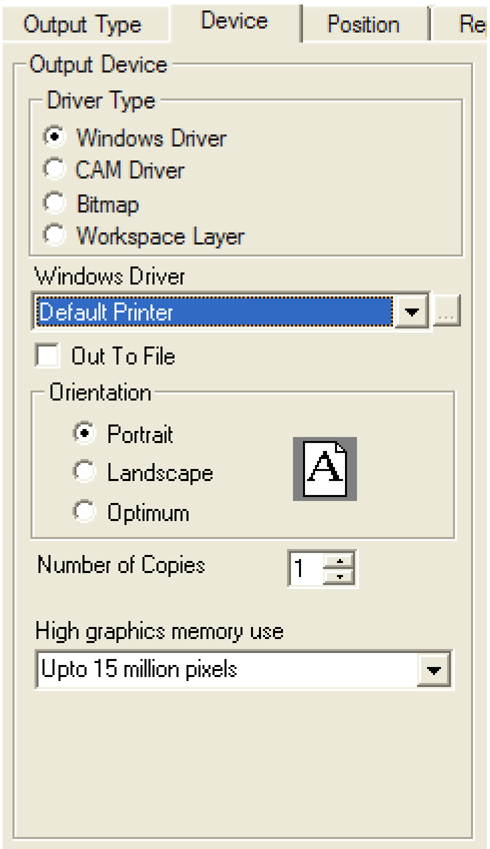
Defining an Output that uses the report

To define an Output using the report, while still in Defaults, right-click Outputs, click **New**, then click **Data**. Enter the name of the new output, e.g. `Meeber Box Company simple report`. Double-click its icon. A dialog box will appear with the name of the report as the title.



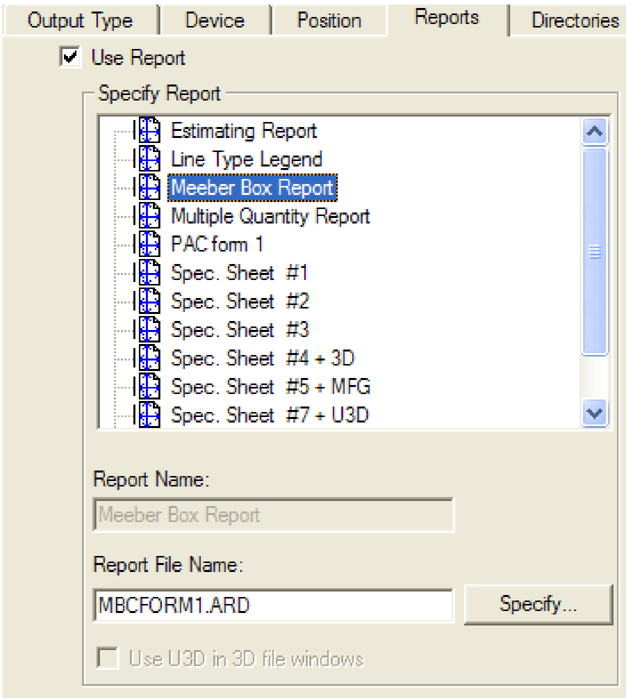
In the **Shown in** group, select those areas of ArtiosCAD (or, optionally, WebCenter) in which the new Output will appear. You must select at least one area; if only one checkbox is selected, that checkbox may not be deselected until another is selected.

If you will print this report on a printer driven by Windows, choose the printer the report will be printed on in the **Windows Driver** drop-down list box on the the Device tab, or choose **Default Printer** to output to the default printer.



If you are using an ArtiosCAD internal driver for the printer, choose CAM on the **Device** tab and choose the device from the CAM Driver list box. Leave the **Output Type** as **PLOT** and set the **Scale** to **To Fit One Page**.

Once the options on the Device tab are set correctly, click the Reports tab. Check the **Use Report** checkbox and select your report.



If the Report has at least one 3D window on it and the Output is a PDF file, the **Use U3D in 3D file windows** checkbox becomes available. U3D is an industry-standard 3D output format that can be read by many applications. For example, if you embed U3D data in a PDF file, Acrobat Reader 7.0 or greater lets you zoom in and out, rotate, and otherwise change the view of the objects in the 3D file window. Select this checkbox as desired.

To change other settings, click the other tabs and set the options as desired. When done, click **OK** to apply the changes you made to Defaults. Then click **File**, and then click **Save** to save the shared defaults. Click **Yes** in the Warning dialog box about overwriting the shared defaults.

To give only yourself access to the report, follow the procedure listed above for Shared Defaults, except do it in the User Defaults pane of the Defaults dialog box.

Using a Report

Once you have added your report to the Reports Catalog and have created an Output for it, using it is quite easy. The procedure for using a Report in Single Design or Manufacturing differs from using one in 3D; see the next section for the 3D procedure.

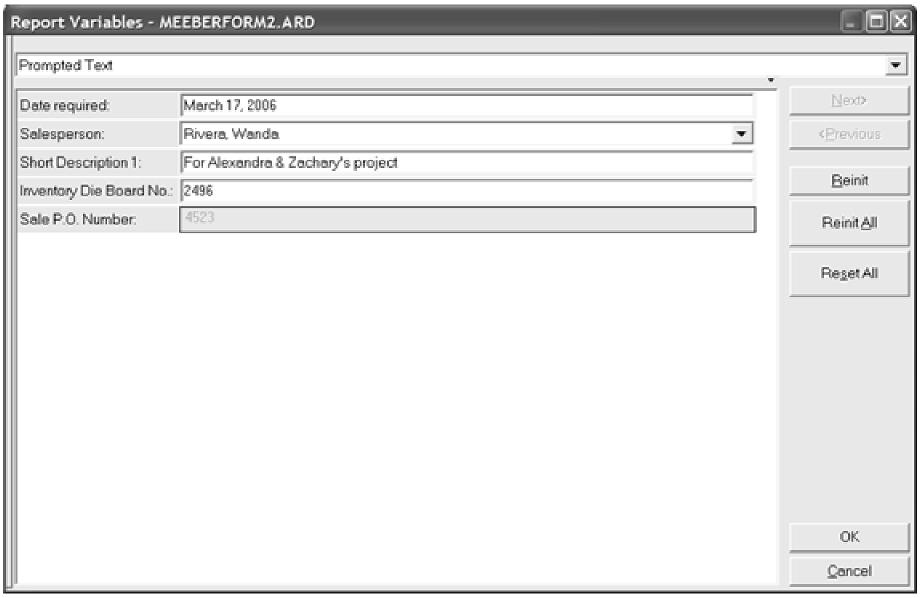
Single Design and Manufacturing

To use a Report, open the workspace(s) you want to place on the report. Turn on the layers that you want to be printed on the report along with the workspace(s). Click File, then Outputs, and then the name of the Output you created. If you used prompted text on the Report, you will be prompted to set values for each prompted text item.

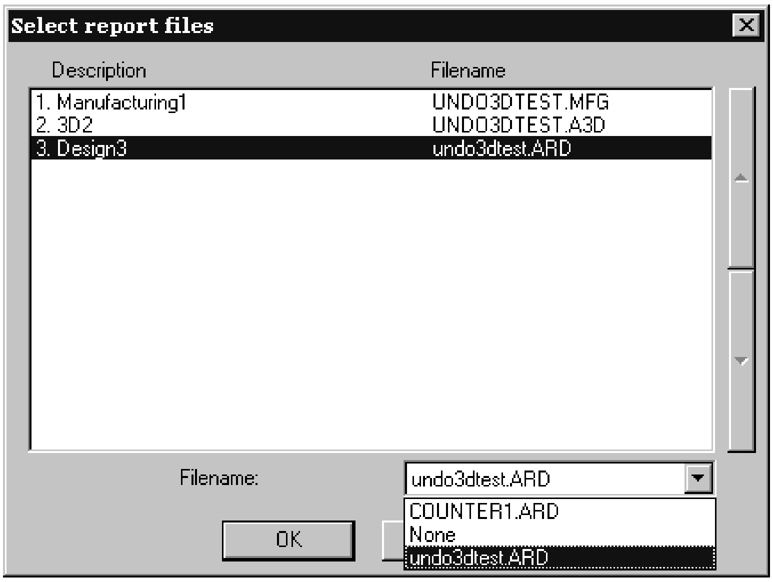
The values you choose for any database-linked prompted text items will be set in the database upon completion of the output. You may only set user fields for the type of document being Output; those user fields with the opposite type are listed but unavailable.

Variables used in both the report and the workspace(s) being used on the Report may be listed when outputting the Report, but may not be changed. To change them, change them in the workspace(s) before outputting the Report.

Set the values as desired and click **OK**.

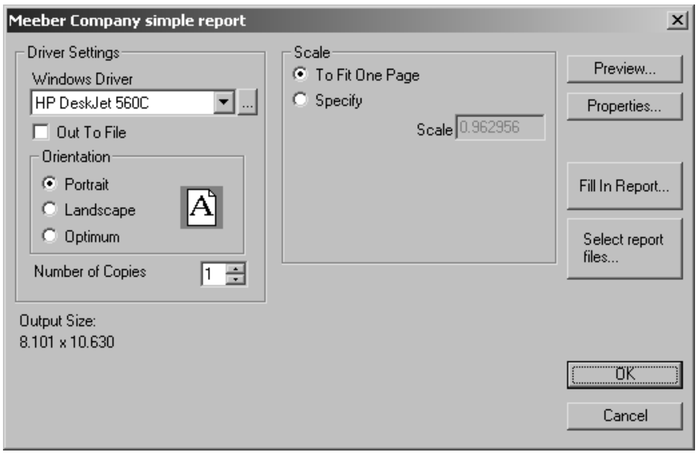


If you have more than one File Window on the report, the Select Report Files dialog box will appear. In this dialog box you can review which file is associated with which File Window and change it if necessary. You can only choose from among open files, so make sure you have already opened the workspaces you want to output before you start the Output. Click **OK** when the files are assigned to their proper File Windows.

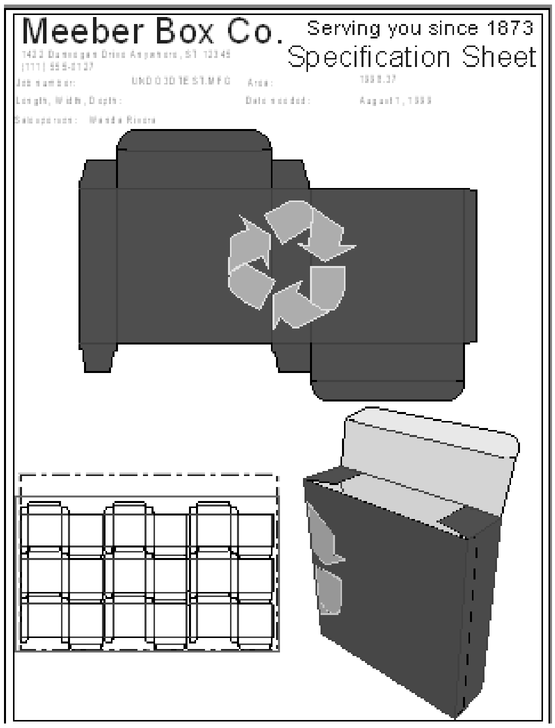


Note: If you have a manufacturing file window on a report, **you must make the manufacturing file the active document** in ArtiosCAD (so that its title bar is highlighted) for it to print correctly on the report.

The output dialog box for the report will appear, and because you defined all the options in Defaults when configuring this Output, everything should be set correctly.



Preview shows a preview of the form as it will print.



Properties lets you adjust the properties of the Output itself, not of the report. **Fill in Report** lets you adjust the answers you gave to prompted items. **Select report files** invokes the Select report files dialog box as shown on the previous page.

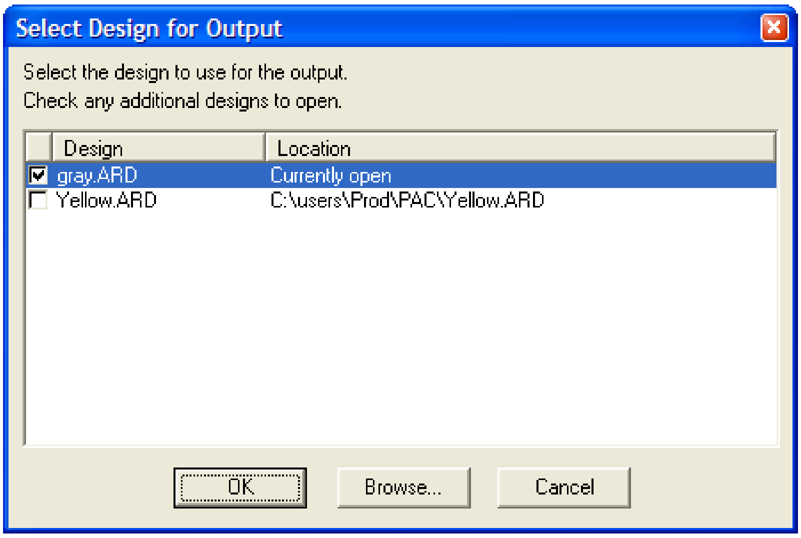
Click **OK** to print the report. Click **Cancel** to dismiss the output dialog box.

3D

Using a Report from 3D is as easy as using one in Single Design or Manufacturing. The difference is that in order to get the design information for variables on the Report, ArtiosCAD references the single design workspaces that are shown folded in the 3D workspace. The Output will not work without the associated single design workspaces, so to use a Report on a 3D workspace on a different computer or at a different site, you must also copy or send the associated single design workspaces along with the 3D workspace.

If the 3D workspace contains exactly one structural single design, ArtiosCAD checks to see if the single design is already open. If it is not already open, ArtiosCAD looks in the location holding the 3D workspace, and opens it if it is found. If it still cannot be found, ArtiosCAD prompts you to browse for the file.

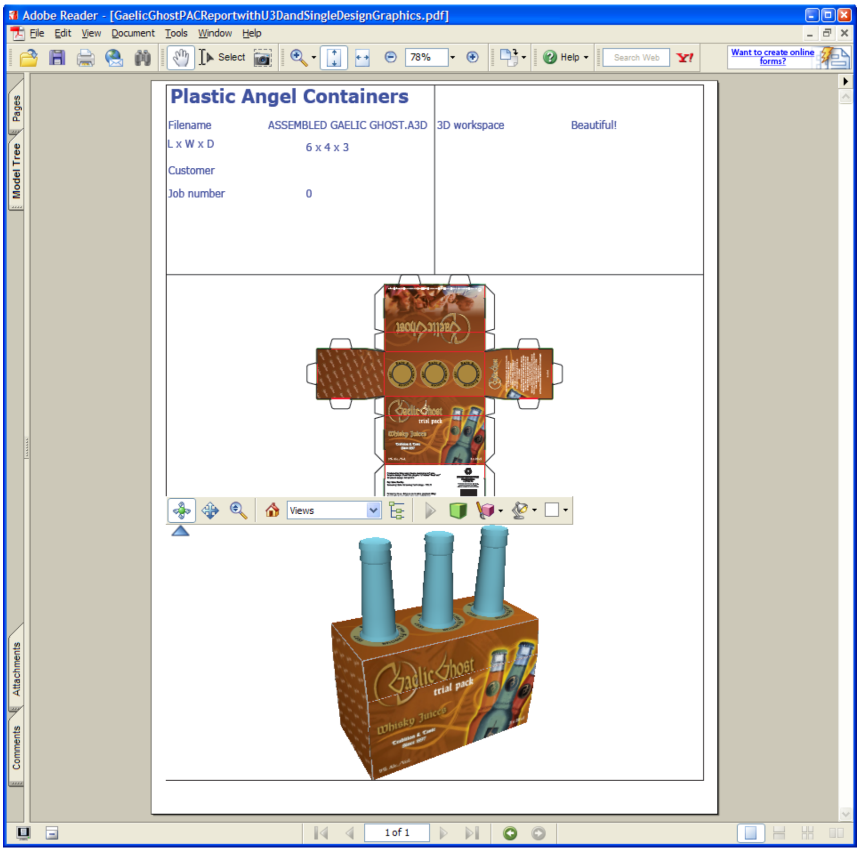
If the 3D workspace contains more than one structural single design, ArtiosCAD prompts you for the workspace(s) to use with the Report.



Select the workspace(s) to use by checking their checkboxes and click **OK**. If **File not found** is displayed in the **Location** column, highlight that line and click **Browse** to find the file. The Report will then be created in the same way as in Single Design or Manufacturing.

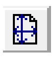
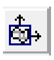
Note: You may not use a Report with a workspace containing only embedded solids or Solids of Revolution; there must be a structural component to the 3D workspace.

Shown below is an example of a Report with a Single Design window and a 3D window exported to a PDF file that uses U3D.



Example - Using multiple instances of the grain/corrugation direction indicator on a report

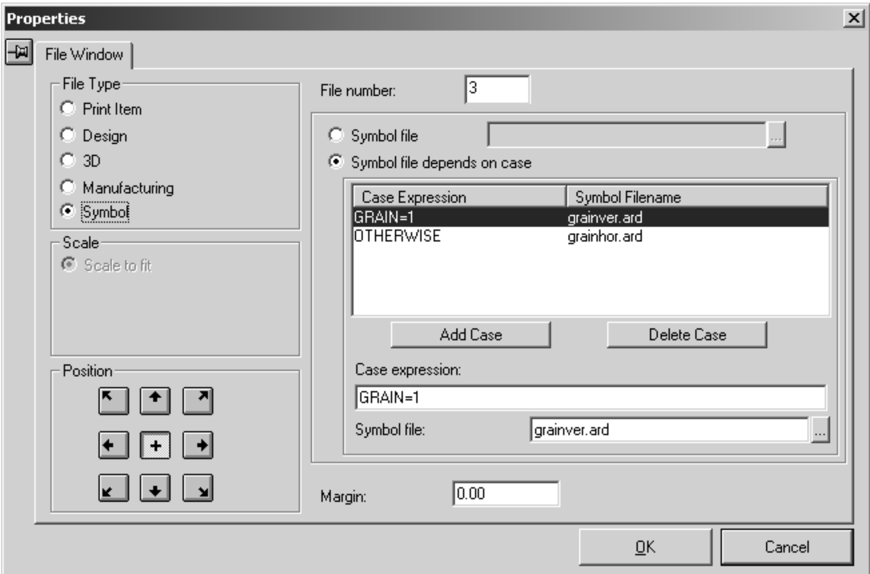
A practical use of the conditional symbol functionality is to put a grain/corrugation direction symbol on a report for each file window. To start, do the following:

1. Create a new Single Design file and decide how many designs will be on the report.
2.  Use the **Report Size** tool to set the size of the report based on the output device.
3.  Use the **File Window** tool to make twice the number of File Windows as decided on in step 1, e.g. if there will be three designs on the report, make six File Windows. Make large windows for the designs and small windows for the direction indicators.
4. Double-click each small window to invoke its Properties page. Three things need changing in the Properties page for each small window - the file type, the file number, and the contents of the Case expression group.

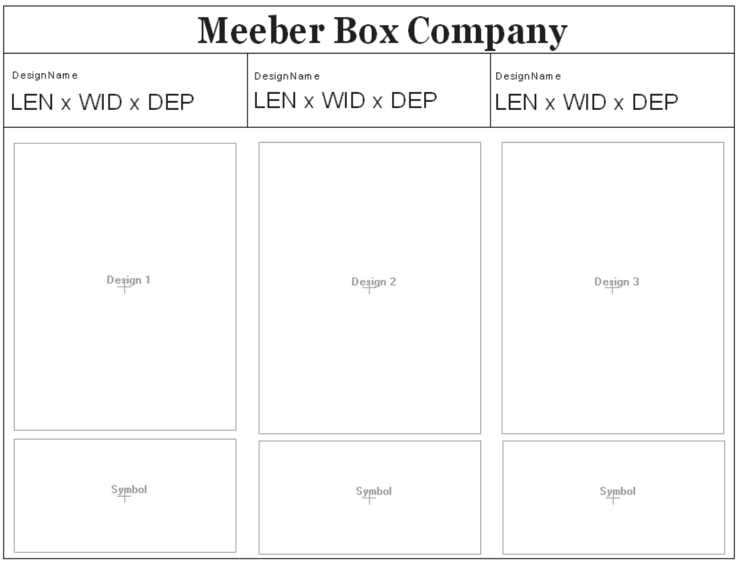
   - Change the File Type to **Symbol**.
   - Change the file number to match its corresponding Design window. This means that the Symbol window for Design 1 should have a file number of 1, the Symbol window for Design 2 should have a file number of 2, and the Symbol window for Design 3 should have a file number of 3.

- Select **Symbol file depends on case**.
- Click **Add Case**. In the **Case expression** field, enter **GRAIN=1**. Click the **...** button at the end of the **Symbol file** field to browse for the symbol file. Navigate to the `InstLib` directory of your ArtiosCAD installation and choose `GRAINVER.ARD`.
- Select the OTHERWISE case. Click the **...** button at the end of the **Symbol file** field to browse for the symbol file. Navigate to the `InstLib` directory of your ArtiosCAD installation and choose `GRAINHOR.ARD`.

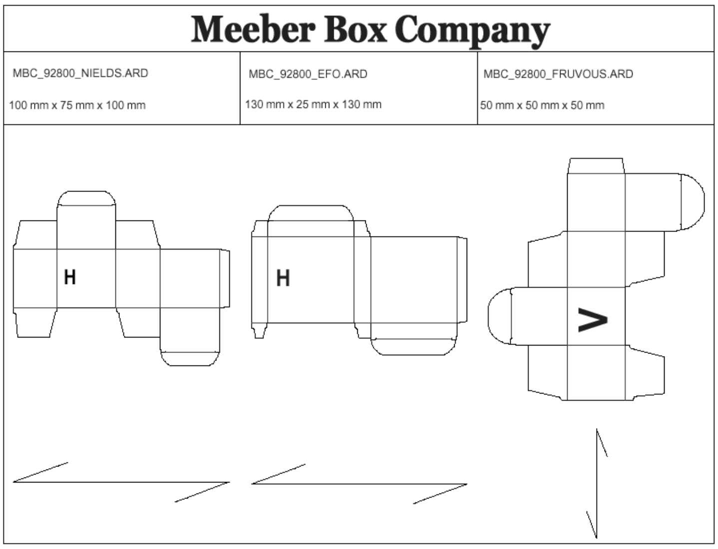
After repeating this step for each Symbol file window, the Properties dialog box for each Symbol file window should look similar to the one shown below, except each window will have a different number in the **File number** field.



5. Make any other desired changes to the report - adding text, a logo, and so on. Shown below is the form in the design stage

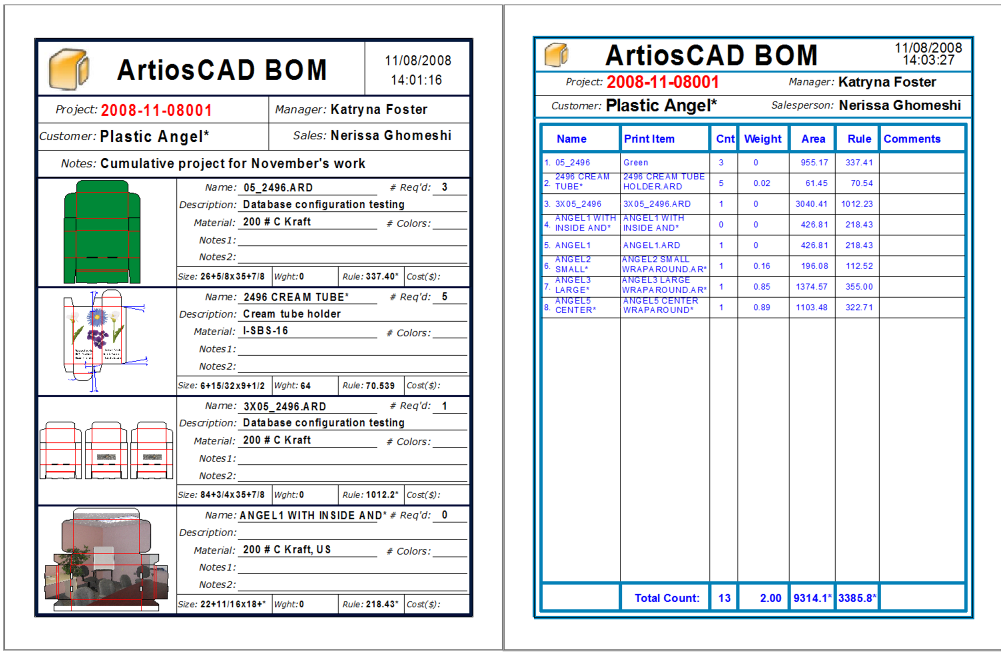


6. Save the report, add it to the Report Catalog, and create an Output for it. When you run it with three designs open, the output will be similar to the picture below.



Bill of Materials Reports for Projects

ArtiosCAD can create Bill of Materials Reports, or BOM Reports. BOM Reports list different designs that make up a Project. Shown below are the first pages of two example BOM Reports run against the same Project.



The basic idea behind using BOM Reports is that you design a Report that repeats information down the page for each individual document in the Project. You can show as much or as little information as you like about the designs. To help show you what is possible, a few example BOM Reports are included in Shared defaults in **Options** > **Shared defaults** > **Outputs** > **Artios** > **Project Bill of Materials**.

Note:

BOM Reports do not need to be added to the Report Catalog in Defaults.

The **Project Documents** dialog box and the **Documents** pane of the Project Browser are where you configure the order and quantity in which the designs will appear on BOM Reports.

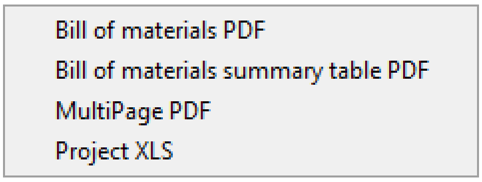
To run an existing Output against every document in a Project, simply check **Project-Bill of Materials** in the **Show in** group on its Properties dialog box in Defaults. When it is time to run that Output, select it from the **Projects** > **Bill of Materials** menu instead of the **Outputs** menu.

Using an example BOM Report

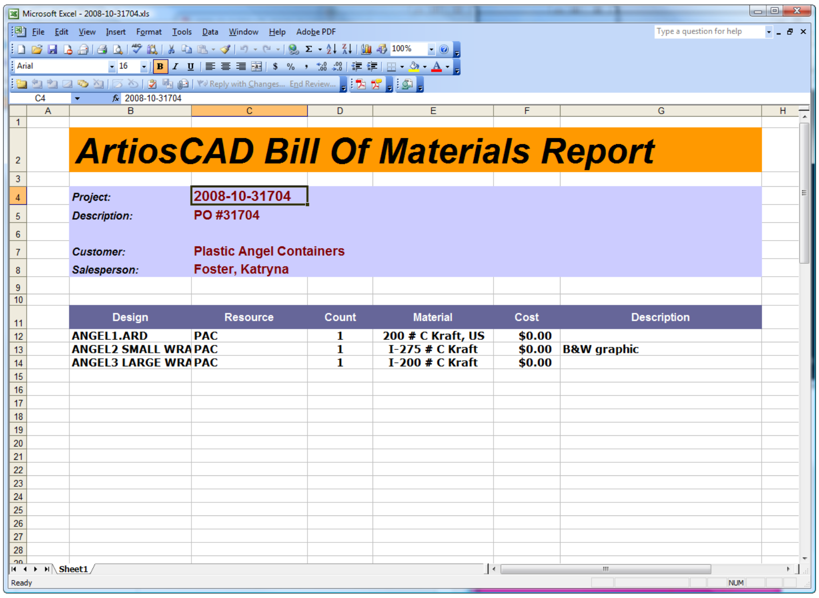
Note:

To use most of the example BOM Reports, you must have the PDF option installed on your system.

ArtiosCAD ships with several example BOM Reports on the **Projects** > **Bill of Materials** menu. Note that BOM Reports **do not** appear in **File** > **Outputs**.

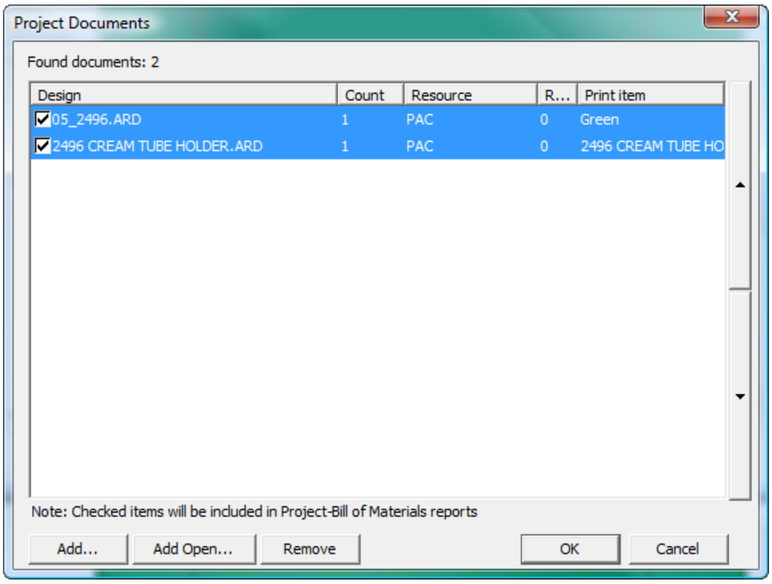


- **Bill of materials PDF** and **Bill of materials summary table PDF** are the two BOM reports shown in the previous section.
- **Multipage PDF** prints each Project document to a separate page of a PDF file.
- **Project XLS** exports Project data to a Visual Basic script file which is then used to create an .XLS file. If you have Microsoft Excel on your system, it automatically opens the .XLS file as a spreadsheet. If you are interested in changing this Report, contact the System Integration team at your local Esko office.



Configuring Project documents before using a BOM Report

Before you Output a BOM Report, use the **Project Documents** dialog box to set up which documents in the Project will appear on the BOM Report, the order in which they will appear, and the quantity of each component required by the Project as a whole.



To configure the documents before running the BOM Report, do the following:

1. Create a new Project and add documents to it, or open an existing Project.
2. Click **Projects** > **Project Documents**.
3. To include a document on the BOM Report, check the checkbox to the left of its name in the **Design** column. To omit the document from the BOM Report, clear the checkbox.

4. Change the number in the **Count** column using the increment buttons to the number of items represented by the document that are required by the Project as a whole.
5. The documents will appear on the BOM Report in the order they are shown in this dialog box from the top down. To change the position of a document, select it, and then use the up and down arrows at the right edge of the list to change its position.
6. Click **OK** in the **Project Documents** dialog box to return to ArtiosCAD.

Outputting a BOM Report

To Output a BOM Report, do the following:

1. Create a new Project and add documents to it or open an existing Project.
2. Configure the presence, the order, and the count of the documents on the BOM Report using the **Project Documents** dialog box as explained previously.
3. Click **Projects** > **Bill of Materials**and click the BOM Report to Output.
4. If there is a **Properties** dialog box for this BOM Report, set the options as desired and click **OK**.
5. ArtiosCAD Outputs the chosen BOM Report.

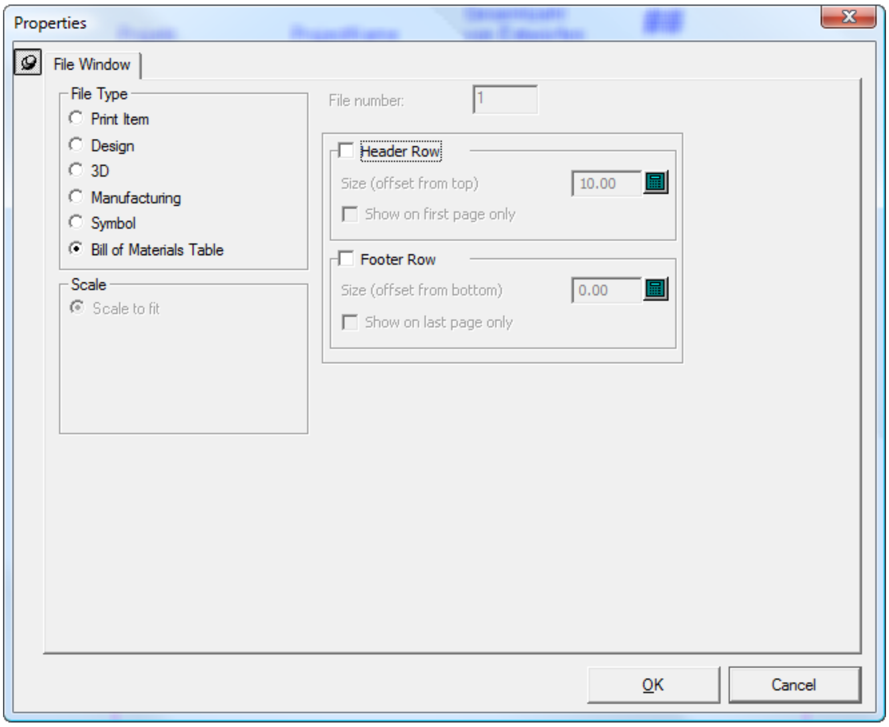
Creating a BOM Report

Creating a BOM Report is just like creating a regular Report, except you define a special area called the **Bill of Materials table** and you do not have to add the Report to the Report Catalog when done. Note that your system must have the Reportmaker option in order for you to create new Reports.

Before you create your own BOM Reports, open the example ones (`..\InstLib\BOMReport.ARD` and `..\InstLib\BOMTABLEREPORT.ARD`) to see how they work. You may modify these as desired, but save the modified versions to `..\ServerLib`.

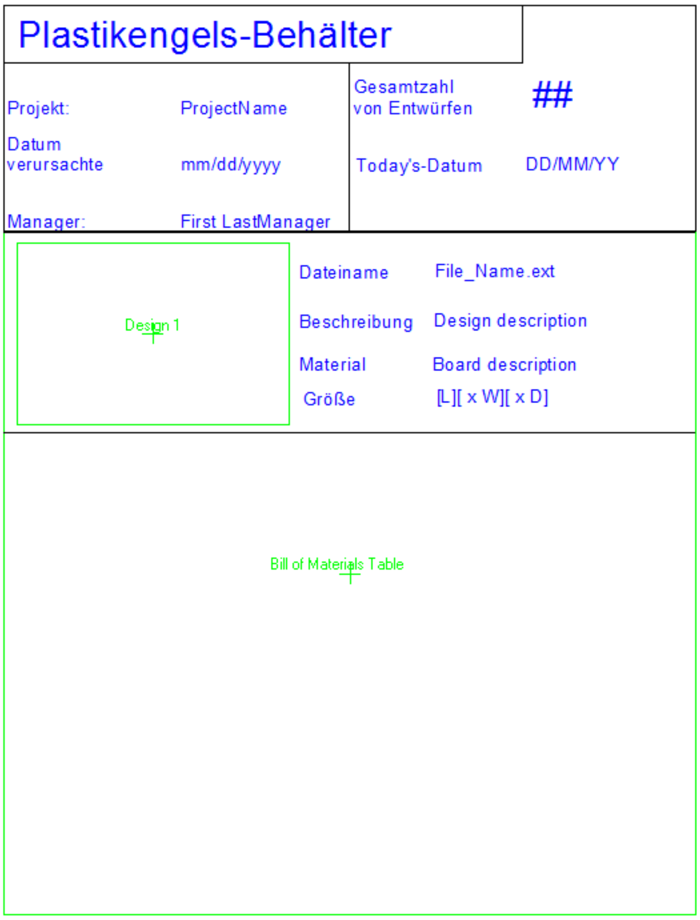
The steps below describe the general procedure for creating a BOM Report.

1. Start ArtiosCAD and create a new single design.
2. Using the tools on the Reportmaker and Geometry toolbars, define the report size and draw lines to separate areas on the form as desired.
3. Create a File Window that will be the main repeating area of the BOM Report, and double-click it after you have created it to open its **Properties** dialog box.
4. Change the **File Type** to **Bill of Materials Table**.
5. If desired, leave space for a header and a footer by checking **Header Row** and **Footer Row**, respectively. Set their sizes as desired as well as the pages on which they will be shown.



6. Click **OK** to close the dialog box. Shown below is an example of a possible BOM Report in mid-construction.

7. Next, define the actual repeating File Window that will show the information about each document in the Project. Create a template small enough to be repeated down the page several times, and fill it with a File Window and the desired static and calculated text. Prompted text is not available for BOM Reports. Note that design lines will only be drawn on parts of the Report that are repeated. Use Form Lines 1 and Form Lines 2 linetypes to draw lines in the BOM Table window in non-repeated areas (for example, table column separators that go all the way down the page even if there aren't repeated rows of the table).

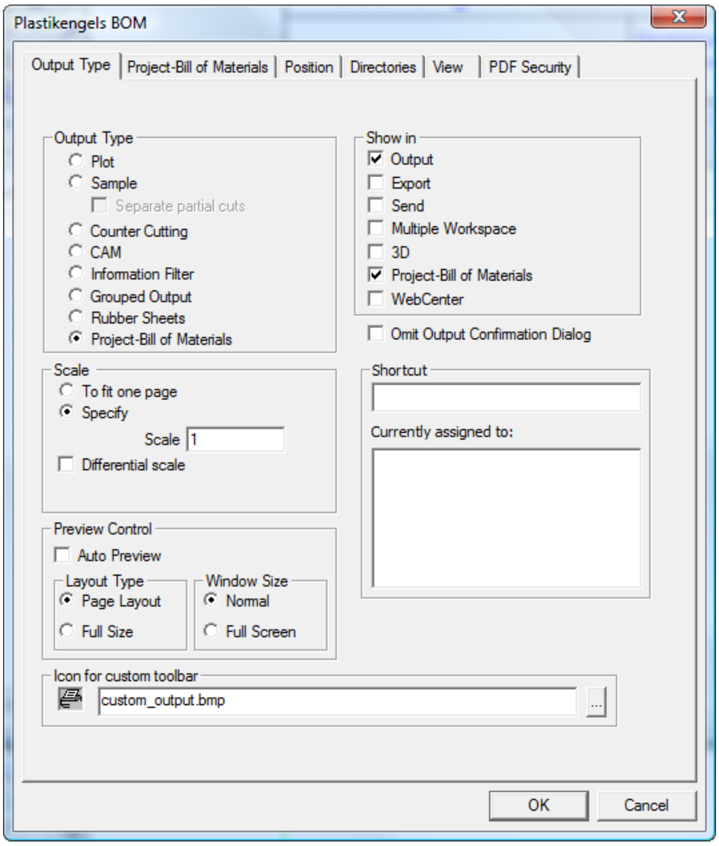


8. Save the BOM Report workspace to `..\ServerLib`.

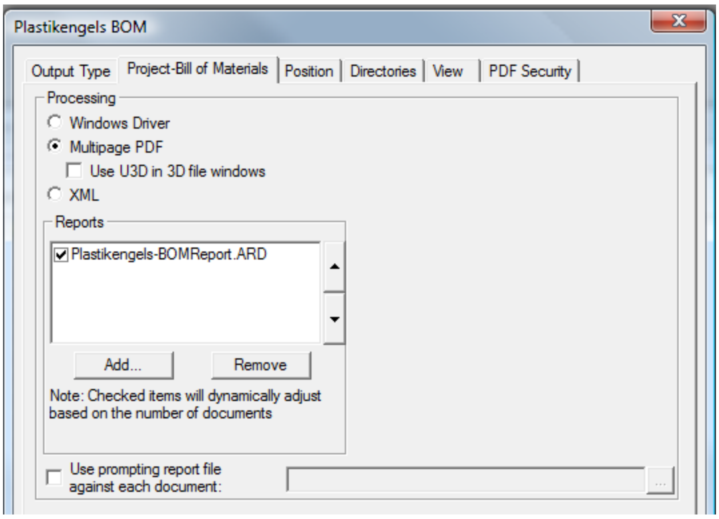
Adding a BOM Report to the Bill of Materials menu

Once you have created a BOM Report and saved it to `..\ServerLib`, do the following to add it to the Bill of Materials menu:

1. Start ArtiosCAD if it is not already started.
2. Click **Options** > **Defaults**, and then expand **Shared defaults** > **Outputs** > **Artios** > **Project Bill of Materials**.
3. Right-click **Project Bill of Materials**, click **New** > **Data**, and enter the name of the new BOM Report.
4. Double-click the new entry you just created to open its **Properties** dialog box.
5. On the **Output Type** tab, in the **Output Type** group, select **Project-Bill of Materials**, and in the **Shown In** group, select **Project-Bill of Materials**.



6. On the **Project-Bill of Materials** tab, in the Processing group, choose the destination for the BOM Report. **Windows Driver** generally directs the output to printer; set the options in the **Windows Driver** group as appropriate. **Multipage PDF** creates a PDF file of the Report and requires your system to have the PDF option; select the **Use U3D in 3D file windows** checkbox as desired. **XML** is reserved for use by the Esko Systems Integration group.
7. Still on the **Project-Bill of Materials** tab, in the **Reports** group, click **Add**, navigate to `..\ServerLib`, select the workspace for the BOM Report, and click **Open**. If there is another Report you want to run when this BOM Report is run, add it here. If you want the Report to run against every document in the Project, leave its checkbox selected. If you clear its checkbox, it will run only once when this Bill of Materials is run.

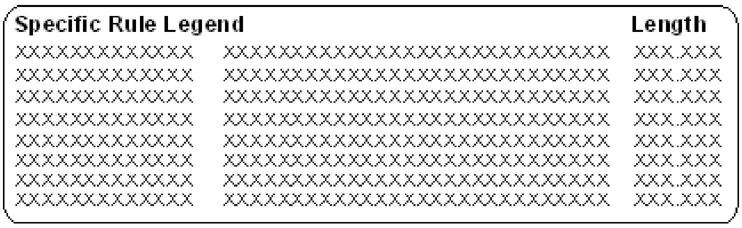


8. Individual prompted expressions do not work in BOM Reports, but you can specify a special prompting Report to be run against every document in the Project by checking **Use prompting report file against each document** and specifying the workspace in the associated field. In this Report, use calculated expressions that ArtiosCAD will prompt for when running the Report. If the Project documents are open, the expressions will be stored in the open designs, but the documents will need to be saved manually for the values to be preserved.
9. Click the other tabs and review the settings on each, changing them as desired.
10. Click **OK** to close the **Properties** dialog box of the new BOM Report.
11. Save and exit Defaults as usual. The new BOM Report is now available for use on the **Projects** > **Bill of Materials** menu.

Line type legends

Line type legends are used on reports to show the names, lengths, and examples of the rules used in the design or manufacturing file being output on the report. They are built by adding calculated text fields to the report.

Shown below is the Specific Rule Legend from `SPECSHEET6.ARD` in the `InstLib` directory.



In the first column are the fields for the name of the rules. These are instances of the **Line Name** calculated expression. Add these as any other calculated expression; they are in the Line Type Legend catalog of Calculated expressions.

When you add a calculated expression from the **Line Type Legend** catalog, you are prompted for the **Line type index** in the Function Parameter dialog box. This is a number that gets incremented by one each time a new line type is added to the workspace being output. When adding the calculated expressions, increment this number by 1 for each row in the legend.

In the second column are the fields for the examples of the rules. These are instances of the **Line Representation** calculated expression.

In the third column are the fields for the lengths of the rules. These are instances of the **Total Line Length** calculated expression.

Line type legend expression properties

The properties of each calculated expression are important as different options can be set in the **Expression** field. The amount of text in the **Sample text**: field sets the length of the field on the screen and on the report.

Line Name expression

When adding a Line Name item, the expression is `LTNAME(line index, ruletype option)`. If you double-click the first example of the name in the Specific Rule legend to view its properties, the expression is `LTNAME(1,2)`. The Line Index was set in the Function Parameter dialog box - but the ruletype option uses a value of 1 by default.

For each line in the Line type index, when the ruletype option is 1, and the rule is a generic rule, the name of that rule in the index is shown using its generic name. For example, the names of 1/2 by 1/2 generic perf and 1/4 by 1/4 generic perf would be combined into one line, Perf.

When the ruletype option is 2, a separate entry is made in the legend for each type of specific rule.

Line Representation expression

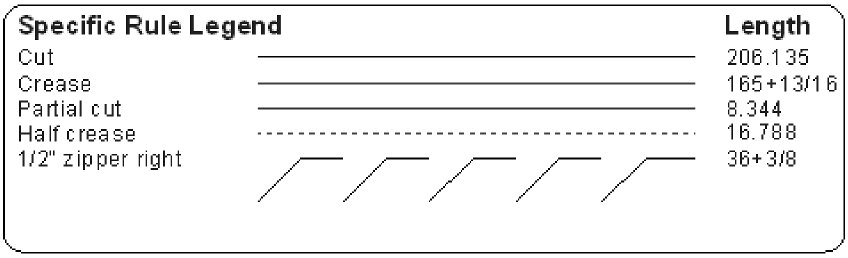
As with the Line Name expression, the **Expression** field controls how the calculated expression is displayed on the report. The Expression is in the format `LTLINE(line index, ruletype option)`. The line type index should always increment by one. When the ruletype option is 1, a picture of the generic rule is drawn. When the ruletype option is 2, a picture of the specific rule type is drawn.

Total Rule Length expression

As with the other expressions used to make line type legends, the Total Rule Length expression uses a ruletype option of 1 or 2. When set to 1, the rule length shown by the expression is that of all special rules of that type combined with generic rules of that type. When the ruletype option is set to 2, the length of special rule and generic rule is listed separately.

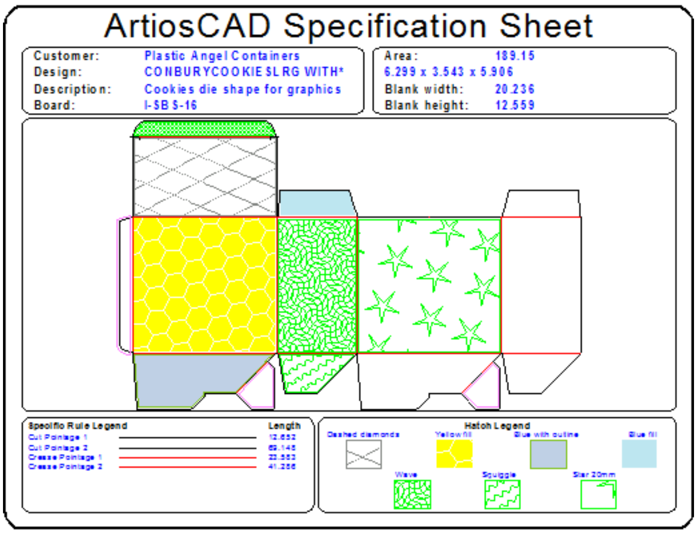
Line type legend results

Once the legend is configured, when the report is output, the legend should look similar to the one shown below, substituting the line types in the workspace being output.



Line and Hatch Legend Output

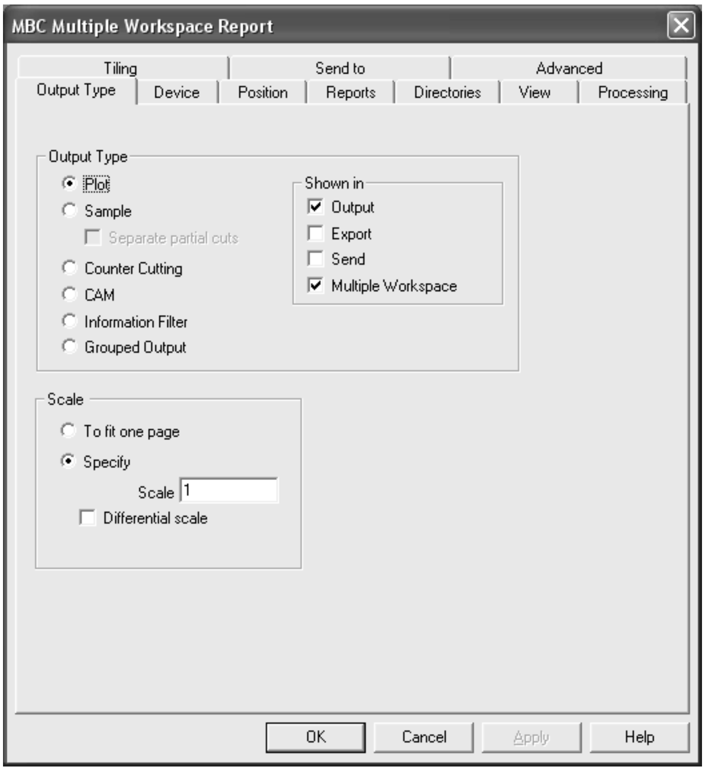
The **Line and Hatch Legend** Report in **Outputs** > **Printer Output** shows the single design, some basic information about the design, a Line Type legend, and a Hatch Legend. The Hatch Legend in this Output shows samples of up to 7 hatches used in the workspace. If two hatches have the same name but are functionally different, they are both shown in the legend with the same label. Shown below is the preview of a sample workspace used with the Output. The Hatch Legend is shown at the bottom right.



To build a custom Hatch Legend, use the entries in the **Hatch Legend** folder of the Calculated Expressions catalog when creating a Report.

Making a multiple workspace output

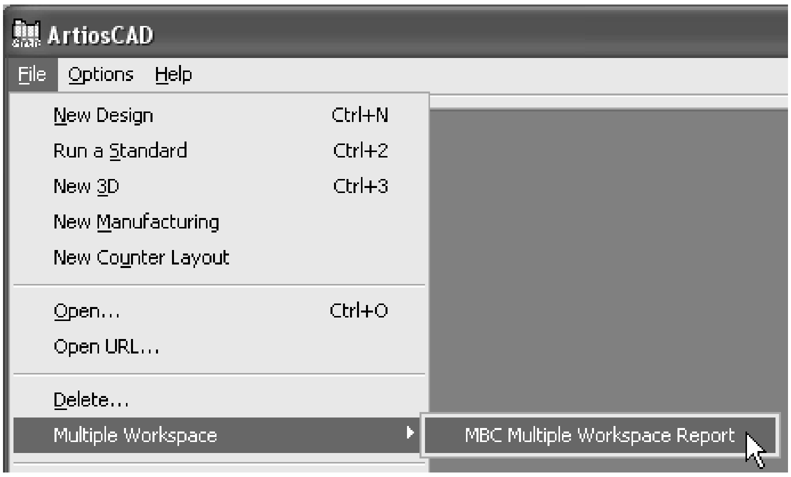
You can set up an Output which outputs more than one workspace at a time by checking the Multiple Workspace checkbox in the **Shown in**: group on the Output Type tab of an Output's Properties dialog box. This Output can also be used in the Database browsers.



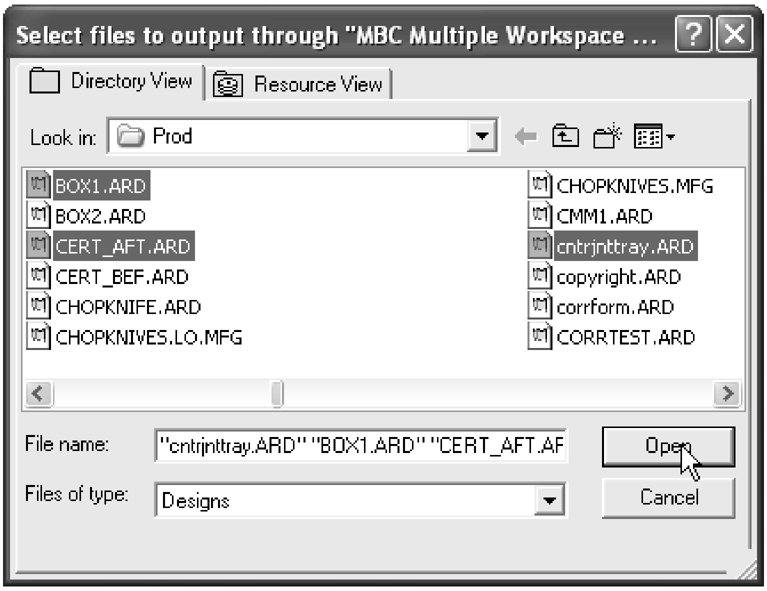
To use a Multiple workspace output, select it from the **Multiple Workspace** folder in **File** > **Outputs** and then select the files to use with that output. You must have at least one Output designated as a Multiple Workspace output for the folder to appear on the File menu. Multiple Workspace outputs appear on the File menu when A) a Single Design is open; B) a Manufacturing file is open; or C) when nothing is open.

To create and run a Multiple Workspace output, do the following:

1. Create a Report and configure an Output for it in Defaults, making sure to select the **Multiple Workspace** checkbox on the Output Type tab of its Properties dialog box.
2. Save and exit Defaults.
3. Click **File** > **Multiple Workspaces** > **Your output name**.



4. Using the **Directory View** in the Open dialog box, select the files to output by holding down CTRL and clicking their names. Click **Open** when you are done selecting files.



5. As each workspace is output, its path and name will appear in a dialog box.

Palletization integration

You can create palletized solutions directly from layouts. CAPE PACK and TOPS Pro have been enhanced by their developers to take advantage of this.

Note:

You must have CAPE PACK v2.09 or TOPS Pro v6.503 or newer installed on your system to use the palletization features. They can use either client/server or workstation/standalone mode.

Note:

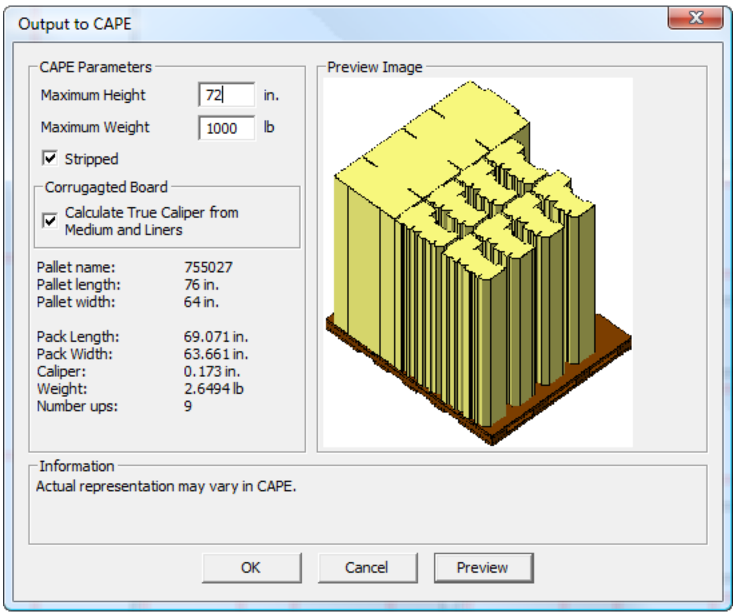
Set the CAPE/TOPS Defaults before using any palletization features. See the *Defaults* chapter of the *ArtiosCAD Administrator Guide* for more information.

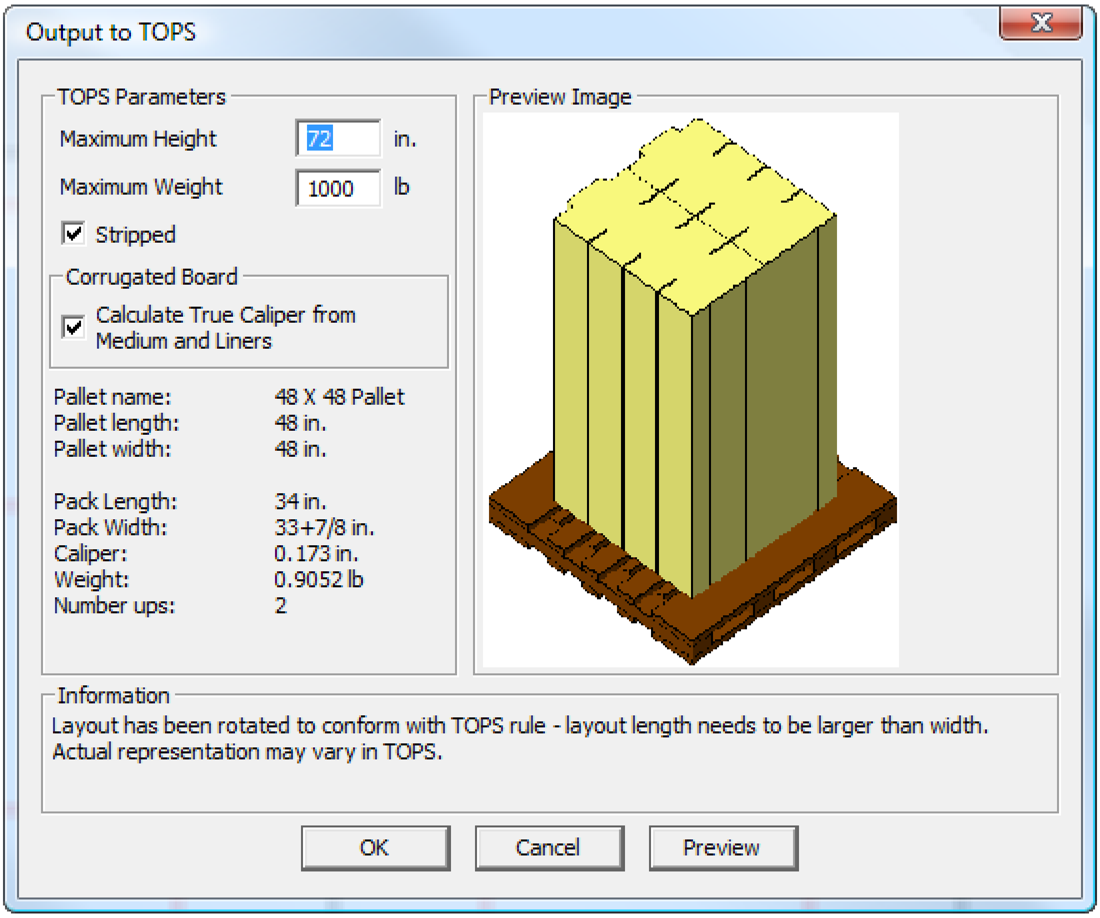
Creating a new CAPE/TOPS palletization solution

Before creating a new palletization solution, create and save the single design(s) to use in it, then do the following:

1. In ArtiosCAD, click **File** > **New Palletization**.
2. Select the pallet to use from the list shown in the **Select Pallet** dialog box and click **OK**. ArtiosCAD extracts the list of pallets from CAPE/TOPS.
3. Use the **Add Oneup** tool to add the single designs to the layout.
4. Use the **Nest** tools to arrange the single designs.
5. Click **Palletization** > **Output to CAPE/TOPS**. CAPE/TOPS must not already be running; if it is, ArtiosCAD will prompt you to close it and try again.

   The Output dialog box will appear showing a preview of how the pallet might look in CAPE/TOPS. CAPE/TOPS may show the pallet differently than ArtiosCAD depending on the orientation of the longer dimension and other factors. Shown below is a stripped pallet.





6. In the **Output to CAPE/TOPS** dialog box:

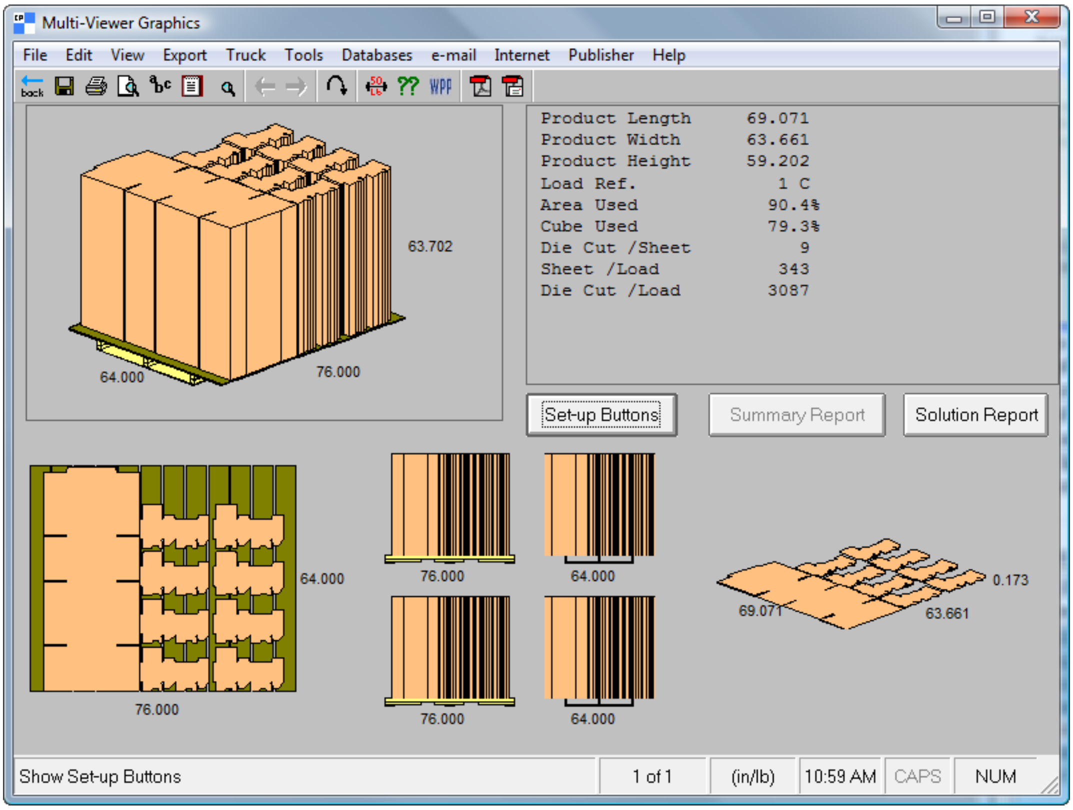
a) Set the **Maximum Height** and **Maximum Weight** fields with appropriate values.

b) To have only the cartons and not the cartons and the waste appear on the pallet, check the **Stripped** checkbox. When this checkbox is selected, stacks of the carton are shown in the preview and in CAPE PACK, as in the example above. When this checkbox is not selected, just the flat top-down view is shown in the preview and in CAPE PACK.

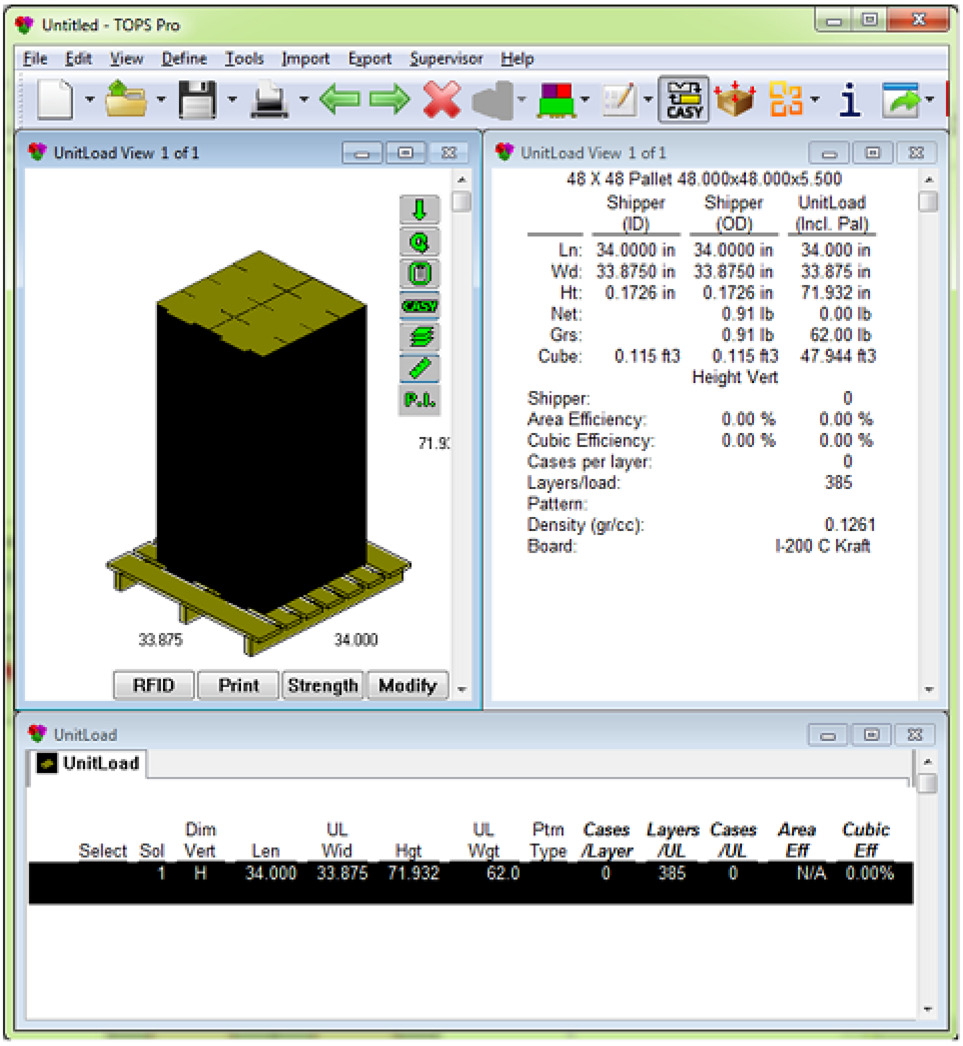
c) Set the **Calculate True Caliper from Medium and Liners** checkbox to force ArtiosCAD to measure the caliper based on the board components along the flute rather than using the caliper variable. This checkbox is cleared and unavailable for non-corrugated boards.

7. Click **OK** to perform the Output.

CAPE will open showing the solution.



In TOPS, it looks like this:



8. Use CAPE/TOPS as desired to complete the palletization solution.

Converting a layout to a CAPE/TOPS palletization solution

To convert a layout to a CAPE/TOPS palletization solution, do the following:

1. Open the layout in ArtiosCAD.
2. Click **Palletization** > **Output to CAPE/TOPS**.
3. ArtiosCAD will prompt that no pallet is defined. Click **OK** to have ArtiosCAD select the closest matching pallet.
4. In the **Select Pallet** dialog box:
   a) Select a different pallet if desired.
   b) Select or deselect the **Stripped** checkbox to control if waste will be on the pallet.
   c) Select the **Adjust sheet size to match pallet size** checkbox to change the size of the existing sheet to match the pallet.
5. Click **OK** in the **Select Pallet** dialog box to continue.

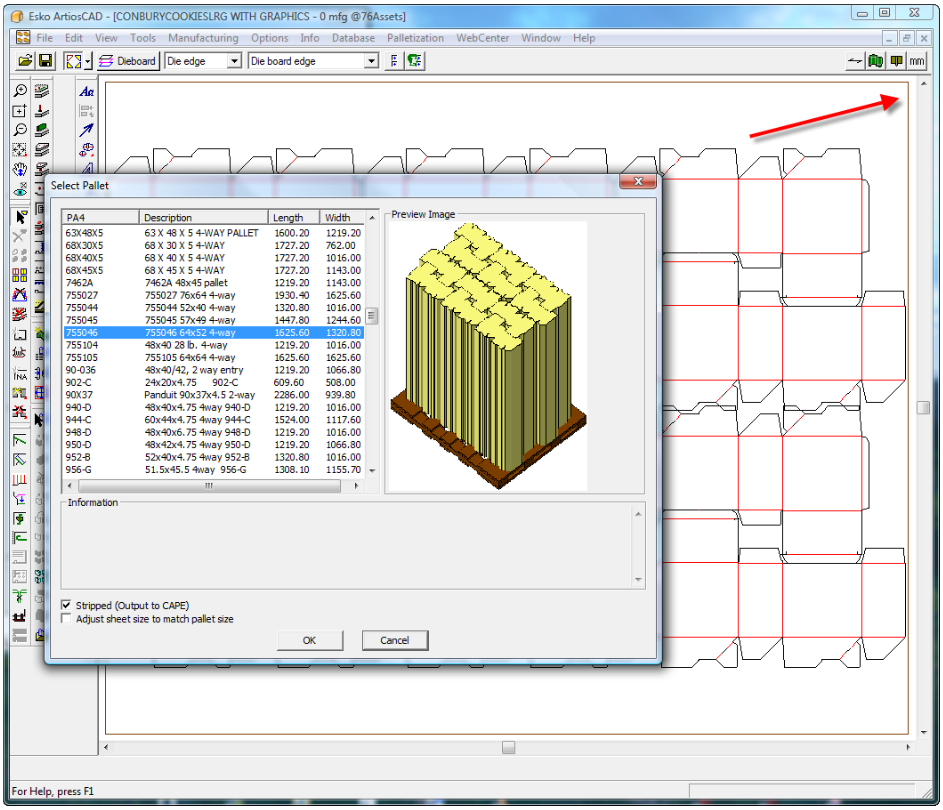
6. In the **Output to CAPE/TOPS** dialog box:
    a) Enter the maximum height and weight of the pallet.
    b) Select or deselect the **Stripped** checkbox.
    c) Select of deselect the **Calculate True Caliper from Medium and Liners** checkbox (if available).
7. Click **OK** in the **Output to CAPE/TOPS** dialog box to perform the Output.

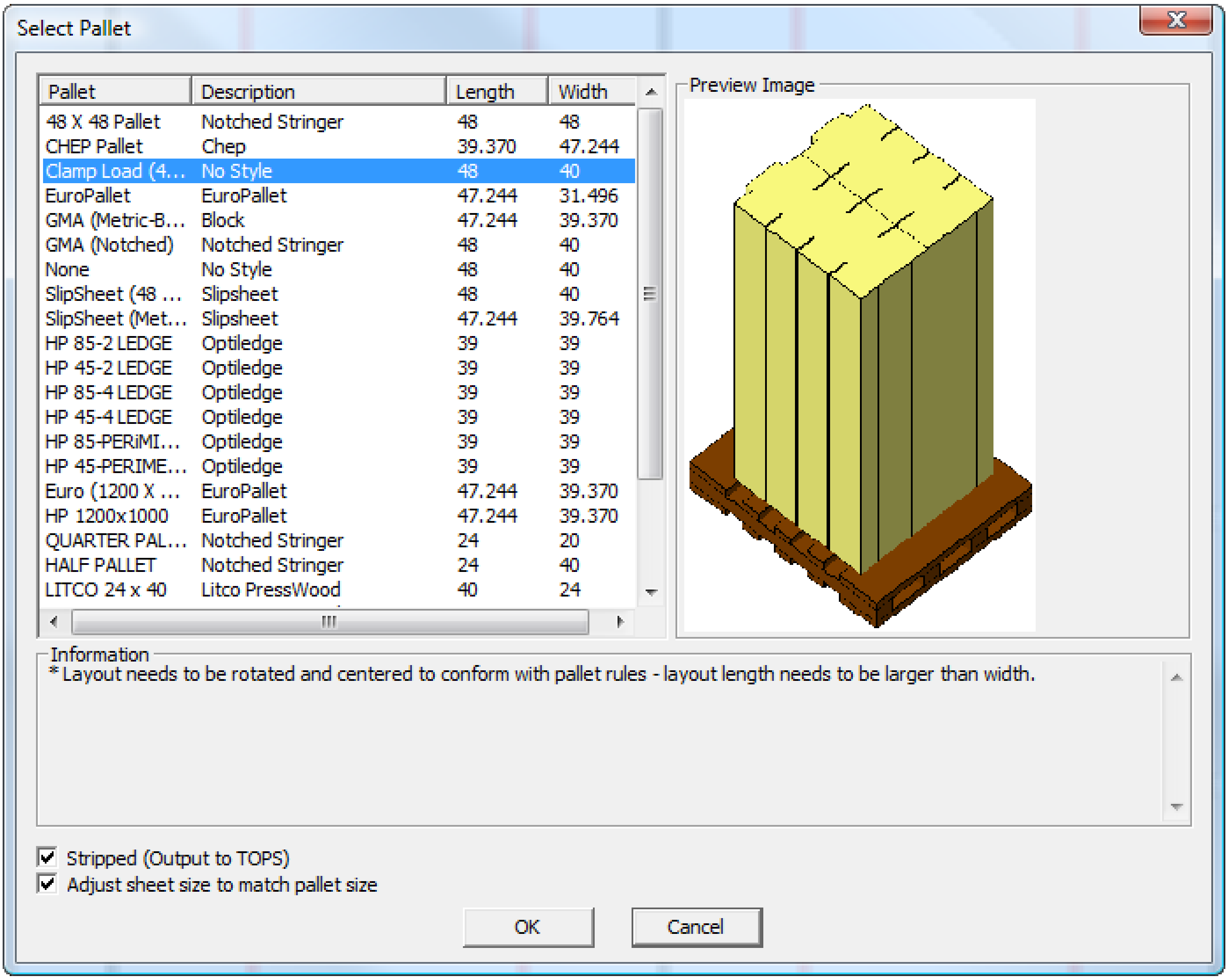
    CAPE/TOPS will open showing the solution.
8. Use CAPE/TOPS as desired to complete the palletization solution.

Changing the pallet in a CAPE/TOPS palletization solution

To change the pallet in a layout that already has been used in a palletization solution, do the following:

1. Open the layout in ArtiosCAD.
2. Click **Palletization** > **Change Pallet**.
3. In the **Select Pallet** dialog box, select a different pallet. A brown bounding box represents the selected pallet in the drawing area as shown below.





4. Set the **Stripped** and **Adjust sheet size to match pallet size** checkboxes as desired.
5. Click **OK** to make the change.

Palletization FAQ

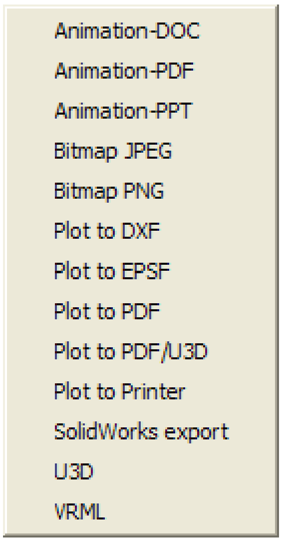
Question	Answer
Why is **OK** unavailable in the CAPE/ TOPS Output dialog box after I create a new palletization workspace?	No board information has been defined yet. Either add a design or click **Info** > **Board Information** and select a board.
Why does the sheet size keep changing when I change pallets?	**Adjust sheet size to match pallet size** is selected in the **Select Pallet** dialog box. Deselect it to stop this.

Question	Answer
Why isn't the sheet size changing to match the pallet size when I choose a new pallet?	**Adjust sheet size to match pallet size** is not selected in the **Select Pallet** dialog box. Select it to have the sheet size change with the pallet.
Why is **Calculate True Caliper from Medium and Liners** unavailable?	The current board is not a corrugated board and thus has no medium or liners defined. Change to a corrugated board by clicking **Info** > **Board Information**.
Why does the preview look different than my layout?	CAPE/TOPS requires the length of the layout to be greater than the width. ArtiosCAD will rotate the layout by 90 degrees if required to meet this rule.
There's a brown edge when I select a pallet and sometimes the sheet edges vanish. Why?	The brown edge represents the pallet edge. The pallet is placed centered on the layout. When **Stripped** is selected, only the single designs determine the pallet placement and the sheet edges are not shown. When **Stripped** is not selected, both the single designs and the sheet edge determine the pallet placement.
What are the brown and purple lines in the preview?	When **Stripped** is not selected, the brown lines are the pallet edge, and the purple lines are the sheet edges which are not output to CAPE/TOPS.
My layout has an overhang; why isn't it showing?	CAPE/TOPS handles overhangs differently from ArtiosCADas everything is centered on the pallet. An uneven overhang will be recalculated. Also, if the blank is smaller than the pallet, there will be no overhang in CAPE/TOPS.
I made my layout the way I want it and CAPE/TOPS is nesting it differently. Why?	CAPE/TOPS maximizes the pallet usage and thus may re-nest the layout if the either the blank width or height are less than half of the pallet width or height.
How does ArtiosCAD calculate the inside dimensions from CAPE/TOPS dimensions?	ArtiosCAD uses the inside dimensions to build a design, while CAPE/TOPS can use either inside dimensions plus the caliper or outside dimensions. If CAPE/TOPS specifies inside dimensions, so does ArtiosCAD. If CAPE specifies outside dimensions, the package type defines the number of folds along length/width/height and

Question	Answer
	ArtiosCAD subtracts these folds (times the board caliper) along each side to calculate the inside dimensions.
A design in ArtiosCAD is not the same size when I convert it to 3D even though the inside dimensions and caliper match.	ArtiosCAD's outside dimensions on a standard may not always match those from CAPE/TOPS due to the use of inside loss and outside gain causing bulges on folds.
Why do I have to reselect a board when I have mapped it properly?	In CAPE/TOPS, most outside dimensions are specified and thus the material thickness is sometimes ignored, in which case it is set to **0.0** with name **undefined**. There is no material by that name in the CAPE/TOPS mapping table and thus ArtiosCAD does not recognize it. All ArtiosCAD designs must have a board code and inside dimensions, and to set the inside dimensions properly from CAPE's/TOPS' outside dimensions, you may have to reselect the board.
Do I need to fold up my 3D design or will ArtiosCAD do it automatically for me before palletizing it?	You must fold up your designs before palletization so that ArtiosCAD sends the correct size bounding box to CAPE/TOPS.
Can I customize the textures shown on the pallet?	Yes. Put your own versions of `palletwood.jpg` and `palletwoodend.jpg` in `ClientLib` or `ServerLib` if desired.
I got an error about mismatched package size. What is this?	This is due to the difference between ArtiosCAD's and CAPE's/TOPS' methods for calculating outside dimensions. Generally click **No** if you see this message.
I have done a lot of work in CAPE/TOPS and now I have many designs open in ArtiosCAD. Which one is correct?	They are all correct since you supplied the parameters each time and clicked **File** > **Export and Exit** . If you only wanted to check the palletization in CAPE/TOPS, click **File** > **Exit** instead without exporting.

3D Outputs

To output a 3D workspace, click **File** > **Outputs-3D**.

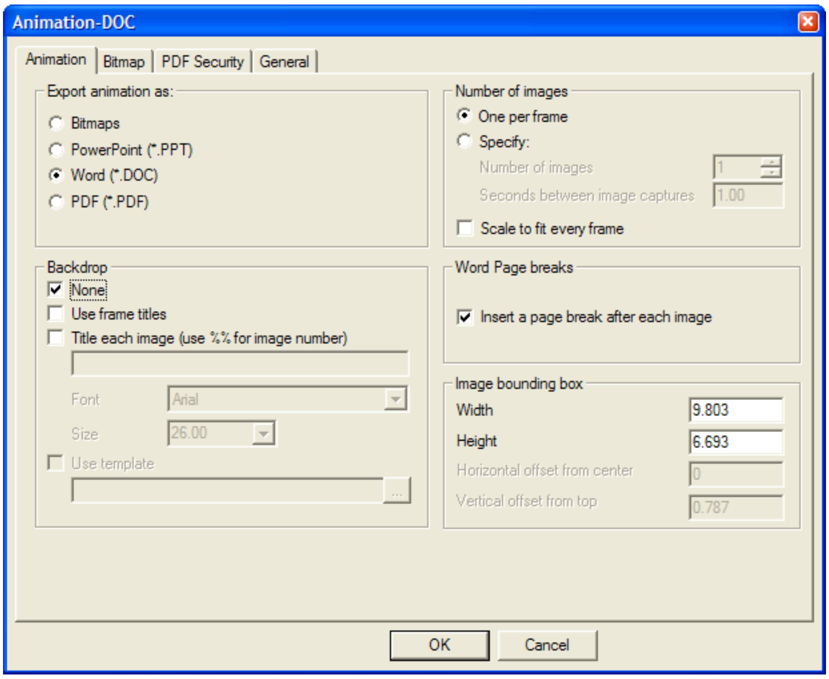


Note: Using any of the Animation outputs requires purchasing the 3D Animation feature. Contact your Esko salesperson for more information about purchasing this option. Using the **Animation-PDF**, **Plot to PDF**, and **Plot to PDF/U3D** outputs require purchasing the PDF option. Contact your Esko salesperson for more information about purchasing this option. Using the **Animation-DOC** output requires Microsoft Word to be installed on your system. Using the **Animation-PPT** output requires Microsoft PowerPoint to be installed on your system. Dimensions are turned off in animation outputs. Using the SolidWorks export output requires purchasing the SolidWorks option.

Outputting a 3D animation as a Microsoft Word document

To output a 3D animation as a Microsoft Word document, do the following:

1. Make sure Microsoft Word is loaded on your system and that the animation is saved and ready for output.
2. Click **File** > **Outputs-3D** > **Animation-DOC**. The Animation-DOC dialog box appears as shown below.



Animation tab

1. On the Animation tab, the options in the **Export animation as:** group control the type of output created. Leave **Word (*.DOC)** selected.

   The options in the **Backdrop** group determine what is shown along with the animation. **None** turns off frame titles; to turn off the background image, deselect it in the View Mode dialog box before starting the Output. **Use frame titles** includes the frame titles set for each frame in the Frame Properties dialog box when making the animation. **Title each image** adds a title for each frame if none were defined during creation. Use %% to include the image number. Using both title options simultaneously is possible; frames with specific titles defined in the animation use them, while frames with no title defined use the one specified in the **Title each image** field. **Font** and **Size** set the typeface and size of the title for each frame. **Use template** is not applicable for this output type.

   In the **Number of images** group, **One per frame** uses a page of Word document per frame of animation. **Specify** enables the **Number of images:** field, which controls how many animation frames are captured and exported, and **Seconds between image captures**: field, which controls the length of animation playback time between each image capture. Setting a value in one field sets the value in the other field.

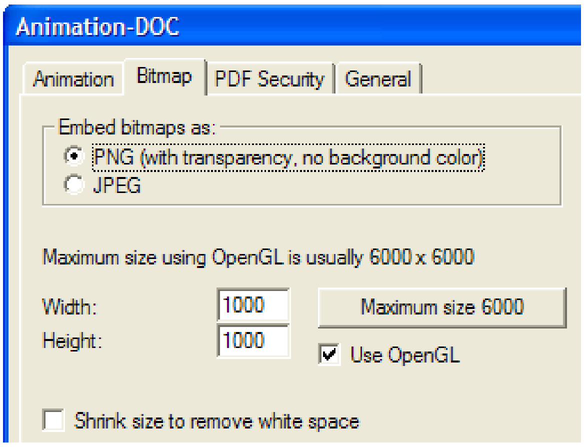
   Scale to fit every frame scales each frame to fit the output size defined in the **Image bounding box** group regardless of the frame properties set in the animation.

   Insert a page break after each image in the **Word Page breaks** group controls whether or not hard page breaks are inserted after each image. When this option is not selected, more than one image per page is possible depending on the size of the bounding box.

   The fields in the **Image bounding box** group control the size of the image placed on the page.

   Set the options on this tab as desired, and then click the Bitmap tab.

Bitmap tab



1. The settings on the Bitmap tab control the format of the exported images, their pixel size, and the rendering method used to generate them.

   In the **Embed bitmaps as**: group, choose either **PNG** (Portable Network Graphics) or **JPEG** (Joint Photographic Experts Group) format. PNGs ignore background color and support alpha channel transparency, which can improve their appearance in programs that support alpha transparency.

   JPEGs support quality settings via the **Quality** field, with 1 being lowest quality and smallest image file size and 100 being highest quality and largest image file size. JPEGs support background color but do not support transparency.

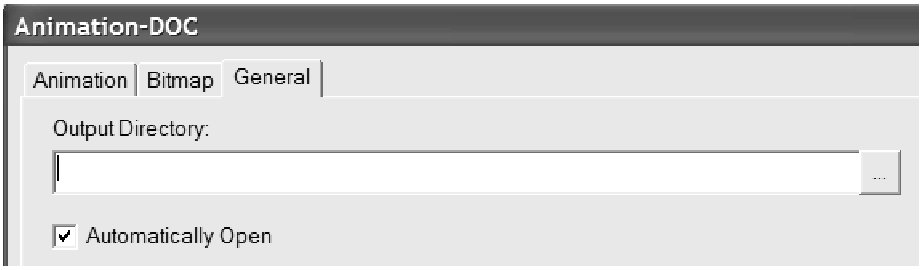
   The maximum bitmap size using Direct3D is different for each display adapter. The **Width:** and **Height:** fields all set the number of pixels that in total determine the resolution of the exported images. Do not forget to consider the size of the pages in Microsoft Word when setting the pixel size of the images, as the images will be clipped to fit the margins of the page if they are too large. To set the maximum size, click **Maximum size** <number>.

   Setting both the **Width**: and the **Height**: to 1000 pixels provides sufficient resolution for most workspaces. Use a higher resolution if the exported animation will be printed, but do not forget that image size is also controlled by the bounding box size set on the Animation tab and must fit on the page in Microsoft Word.

   To use OpenGL and thus make higher-resolution images, check the **Use OpenGL** checkbox and change the number of pixels; this option is selected by default. **Shrink size to remove white space** removes the background pixels surrounding the workspace from the exported images, and is automatically selected and disabled if a background image is visible in the workspace.

   Set the options on this tab as desired, then click the General tab.

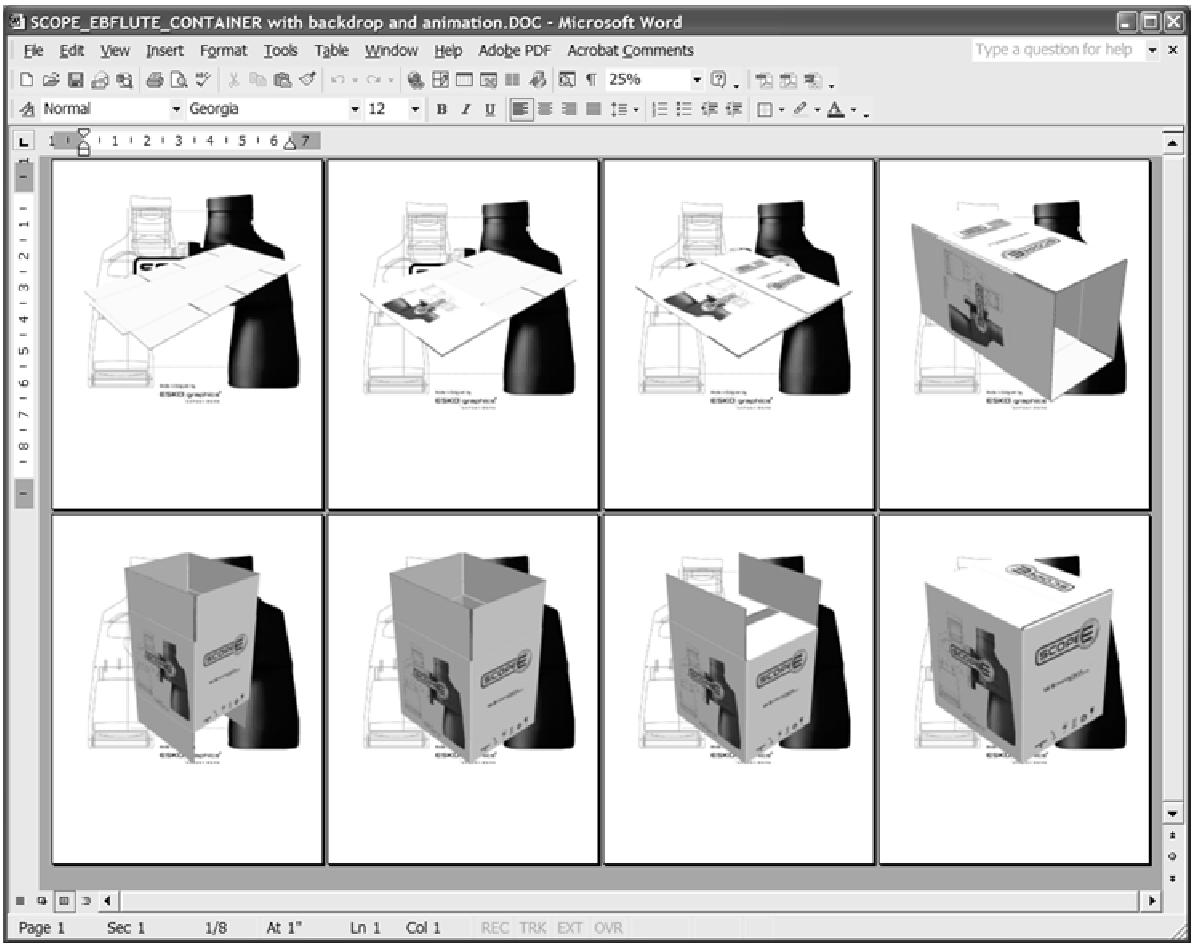
General tab



1. On the General tab are the **Output Directory:** field and the **Automatically Open** checkbox. If you enter a directory in the **Output Directory**: field or use the **Browse** button to specify one, that directory is used by the Save As dialog box when you click **OK**. **Automatically Open** controls whether ArtiosCAD launches Microsoft Word after the Output is done and opens the document.

Finishing the Output

1. Once you have set the options on all the tabs as desired, click **OK**.
2. In the Save As dialog box, choose the directory in which to save the file and enter a filename. Click **Save** to perform the Output. Depending on the state of the **Automatically Open** checkbox, ArtiosCAD may launch Microsoft Word and open the document. Shown below is an example Output.

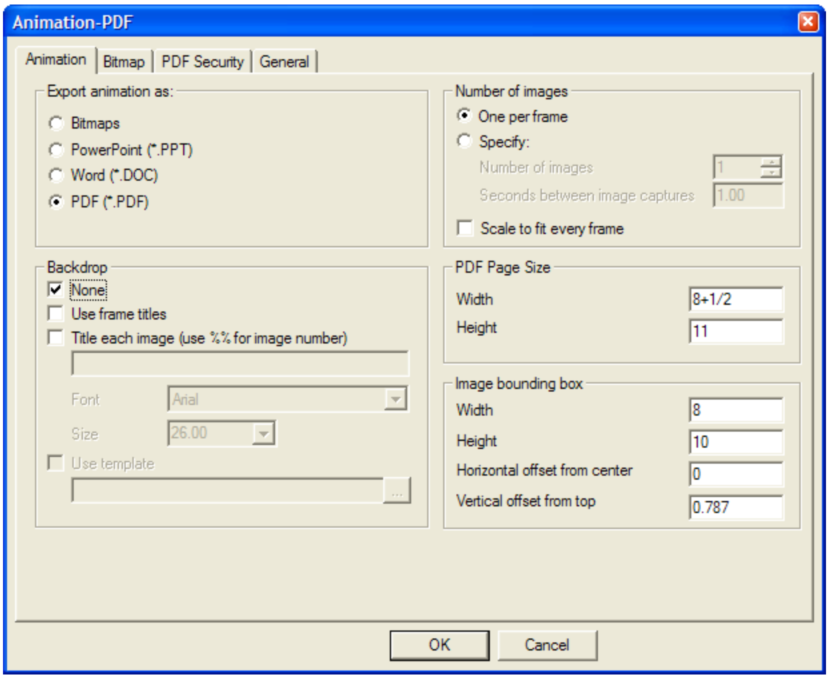


Outputting a 3D animation as a PDF document

Outputting a 3D animation as a PDF document is similar to outputting an animation as a Microsoft Word document. The only differences are on the Animation tab of the Animation-PDF dialog box. Using this Output requires having purchased the PDF option.

To output a 3D animation as a PDF document, do the following:

1. Make sure the animation is saved and that the workspace is ready for output.
2. Click **File** > **Outputs-3D** > **Animation-PDF**. The Animation-PDF dialog box appears as shown below.



In the **Export animation as:** group, leave **PDF (*.PDF)** selected.

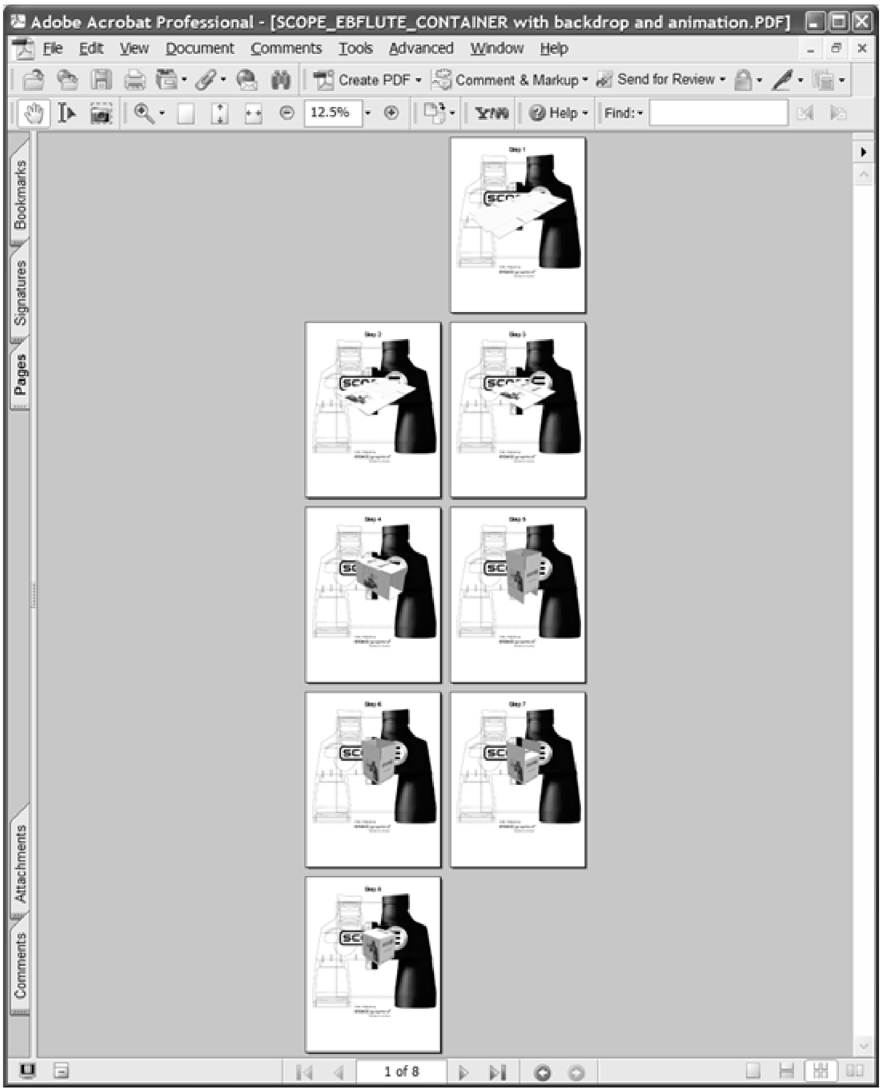
The options in the **Backdrop** and **Number of images** groups work the same way as in the Animation-Doc Output.

The **Width** and **Height** fields in the **PDF Page Size** group set the size of the PDF pages in the outputted file.

The fields in the **Image bounding box** group control the size and placement of the images on the pages; set them as desired. Ensure the bounding box is smaller than the page size to avoid clipping the graphics in the outputted file.

Set the options on this tab as desired.

1. Set the options on the Bitmap, PDF Security, and General tabs as desired, and click **OK**.
2. In the Save As dialog box, choose the directory in which to save the file and enter a filename. Click **Save** to perform the Output. If you have a PDF-viewing utility (such as Adobe Acrobat or Adobe Acrobat Viewer) and the **Automatically Open** checkbox on the General tab was selected at time of Output, the PDF file will be opened automatically as shown below.

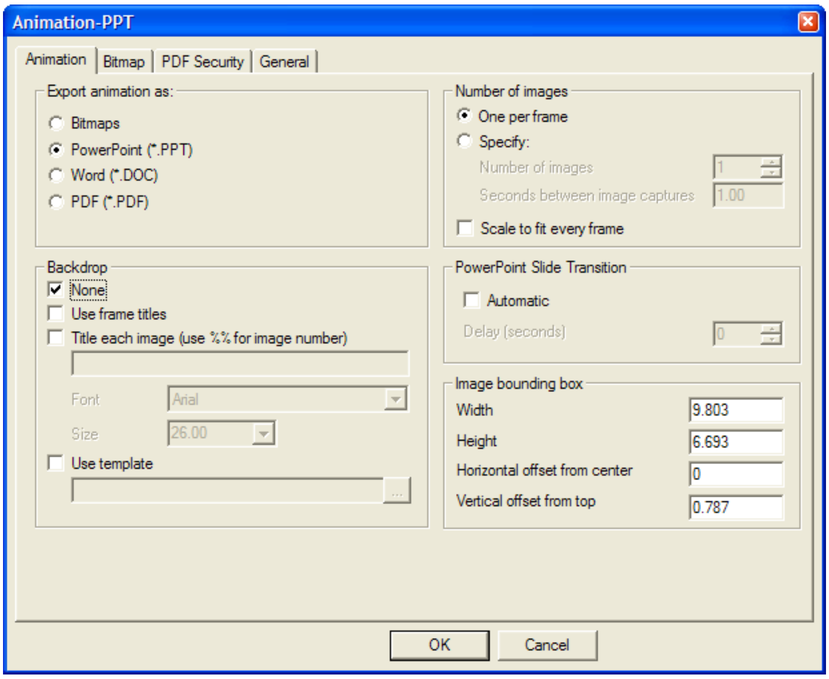


Outputting a 3D animation as a Microsoft PowerPoint document

Outputting a 3D animation as a Microsoft PowerPoint document is similar to outputting an animation as a Microsoft Word or PDF document. The only differences are on the Animation tab of the Animation-PPT dialog box.

To output a 3D animation as a Microsoft PowerPoint document, do the following:

1. Make sure Microsoft PowerPoint is loaded on your system and that the animation with at least 2 frames is saved and ready for output.
2. Click **File** > **Outputs-3D** > **Animation-PPT**. The Animation-PPT dialog box appears similar to the one shown below.



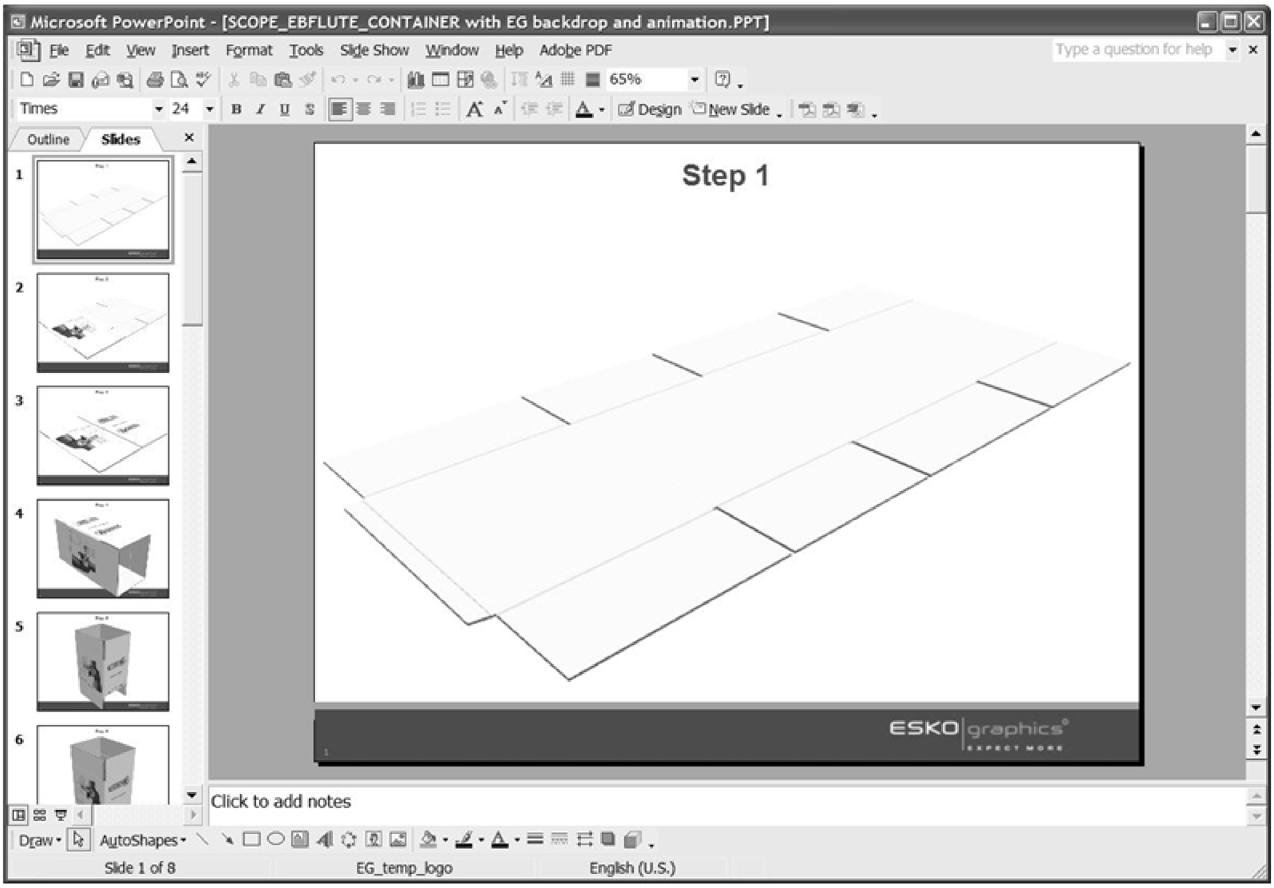
3. In the **Export animation as:** group, leave **PowerPoint (*.PPT)** selected.

   The options in the **Backdrop** and **Number of images** groups work the same way as in the Animation-Doc and PDF Outputs. **Use template** lets you specify or browse for a Microsoft PowerPoint template to use instead of having the images placed on a series of blank slides. If you plan to use a template, turning off the background image in the View Mode dialog box before performing the Output may yield more desirable results.

   The options in the **PowerPoint Slide Transition** group control if the output file runs as a slide show when opened. Click **Automatic** to have the slides automatically advance using the duration specified in the **Delay (seconds)** field.

   The fields in the **Image bounding box** group control the size and placement of the images on the slides; set them as desired. Ensure the bounding box is smaller than the page size to avoid clipping the graphics in the outputted file.

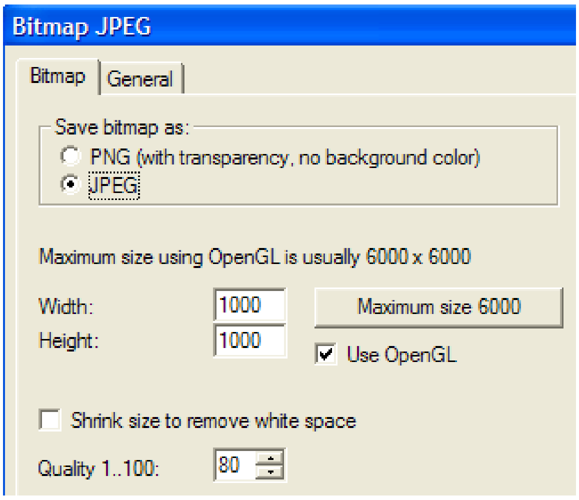
   Set the options on this tab as desired.
4. Click the Bitmap and General tabs and set the options on them as desired, and then click **OK**.
5. In the Save As dialog box, choose the directory in which to save the file and enter a filename. Click **Save** to perform the Output. If **Automatically Open** was checked on the General tab, ArtiosCAD will launch Microsoft PowerPoint and open the presentation.



Outputting a 3D workspace as a JPEG or PNG bitmap

To output a 3D workspace or frame of an animation as a bitmap, do the following:

1. Save the workspace and position it as you would like it to appear in the outputted file. If outputting an animation frame, use the **Animation Playback** tool to show the desired frame of the animation.
2. Click **File** > **Outputs-3D** > **Bitmap JPEG** or **Bitmap PNG**. The Bitmap JPEG or Bitmap PNG dialog box will appear similar to the one shown below.



3. The settings on the Bitmap tab control the format of the exported image, its pixel size, and the rendering method used to generate it.

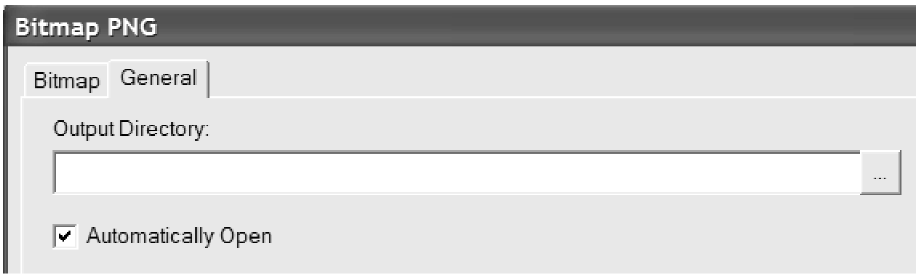
   In the **Save bitmaps as**: group, choose either **PNG** (Portable Network Graphics) or **JPEG** (Joint Photographic Experts Group) format. PNGs ignore background color and support alpha channel transparency, which can improve their appearance in programs that support alpha transparency. They are the only Output type which supports the transparency features in ArtiosCAD.

   JPEGs support quality settings via the **Quality** field, with 1 being lowest quality and smallest image file size and 100 being highest quality and largest image file size. PNGs do not support a quality setting, so that option is not available when PNG is the selected format. JPEGs support background color but do not support transparency.

   The maximum bitmap size using Direct3D is different for each display adapter. The **Width:** and **Height:** fields set the number of pixels that determine the resolution of the exported images. To set the maximum size, click **Maximum size** <number>. Setting both the **Width**: and the **Height**: to 1000 pixels provides sufficient resolution for most workspaces. Use a higher resolution if the exported animation will be printed.

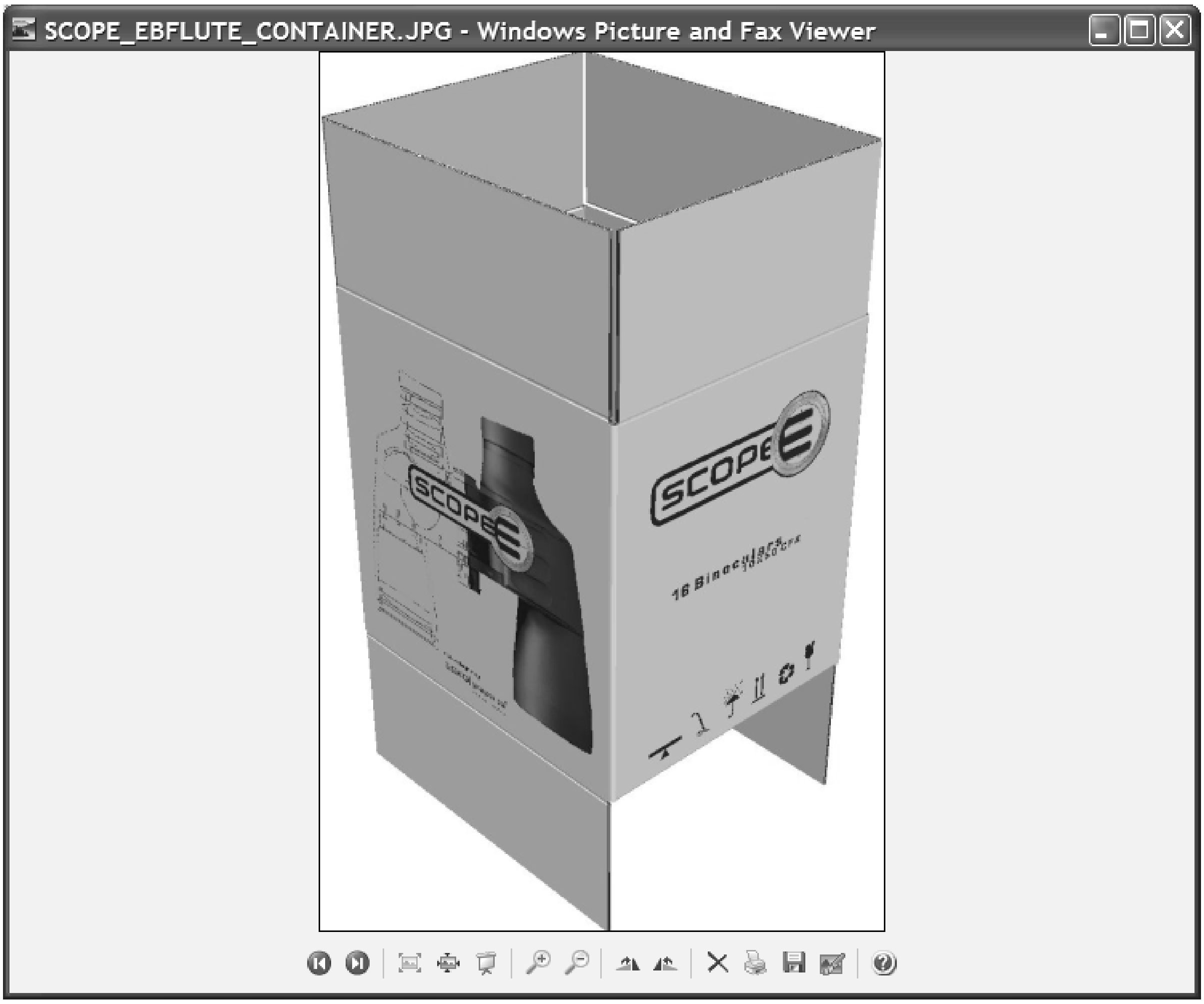
   To use OpenGL and thus make higher-resolution images, check the **Use OpenGL** checkbox and change the number of pixels; this option is selected by default. **Shrink size to remove white space** removes the background pixels surrounding the workspace from the outputted image, and is automatically selected and disabled if a background image is visible in the workspace.

   Set the options on this tab as desired, then click the General tab.



4. On the General tab are the **Output Directory:** field and the **Automatically Open** checkbox. If you enter a directory in the **Output Directory**: field or use the **Browse** button to specify one, that directory is used by the Save As dialog box when you click **OK**. **Automatically Open** controls whether ArtiosCAD launches an image viewer after the Output is done and opens the document.

5. In the Save As dialog box, choose the directory in which to save the file and enter a filename, and click **Save** to perform the Output. If **Automatically Open** was checked on the General tab, ArtiosCAD will launch the default image viewer associated with the bitmap file type and show the file.

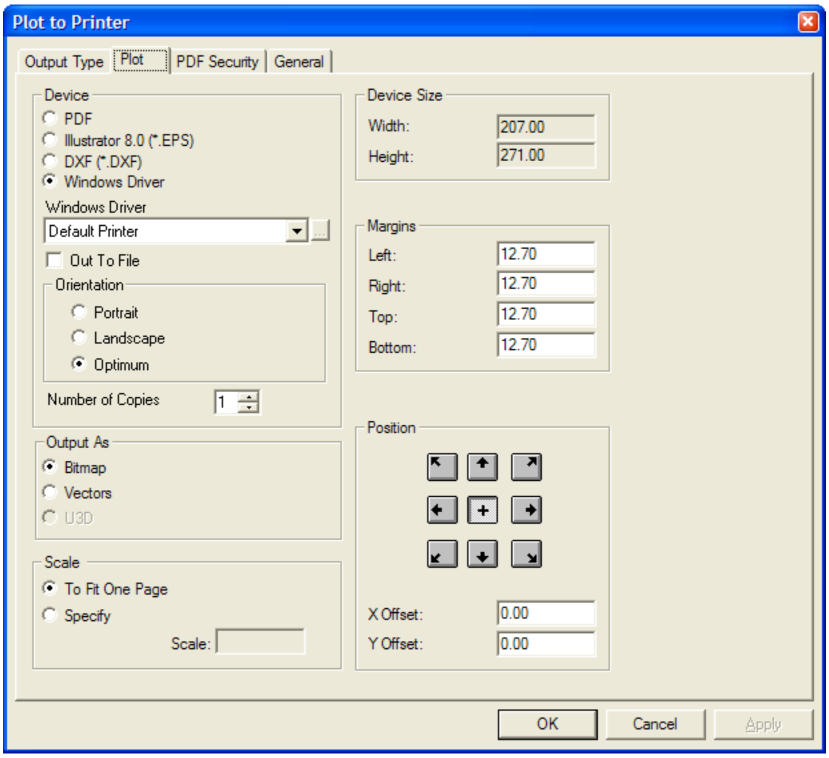


Outputting a 3D workspace to an EPSF file, PDF file, or printer

The **Plot to EPSF**, **Plot to PDF**, and **Plot to Printer** Outputs in **File** > **Outputs-3D** all work similarly, just as the animation outputs work similarly.

To perform one of these three Outputs, do the following:

1. Save the workspace and position it as you would like it to appear in the outputted file. If outputting an animation frame, use the **Animation Playback** tool to show the desired frame of the animation.
2. Click **File** > **Outputs-3D**, then click either **Plot to EPSF**, **Plot to PDF**, or **Plot to Printer**. The Plot to **Choice** dialog box will appear.



The options in the **Device** group control the Output type and any associated options for that type. **PDF** and **Illustrator 8.0 (*.EPS)** have no additional options in this group, but choosing **Windows Driver** lets you choose the device, the page orientation, and the number of copies to make.

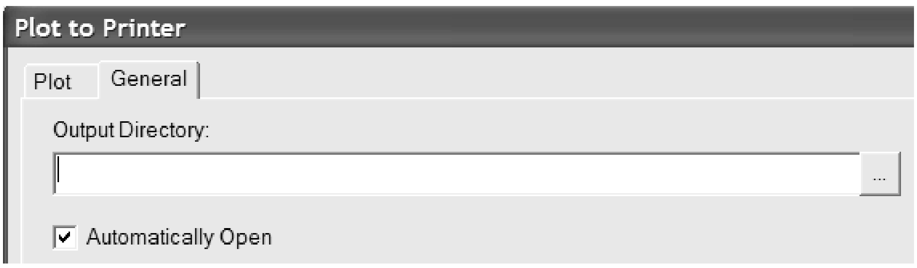
In the Output As group, **Bitmap** makes the output as a snapshot of the screen. **Vectors** takes a snapshot but adds the edges as lines. **U3D** is unavailable as this is not a PDF or U3D Output.

When choosing **Vectors**, the degree of complexity of the workspace may cause issues with hidden line removal, and if there are more than 10 designs or solids with more than 10,000 polygons, the Output may take a few minutes to complete. Vector outputs do not contain bounding boxes. The color of the lines is set by the plotting style for plots and PDF Outputs. Line color in EPSF Outputs are controlled by the tuning file `TUNE.EPSF.3D.TXT`.

The options in the **Scale** group control the size of the Output as compared to its original size in the workspace. **To Fit One Page** is most useful when using **Plot to Printer**. **Specify** lets you enter a desired value in the **Scale**: field.

The **Device Size**, **Margins**, and **Position** groups all work as they do in other Outputs. **Device Size** is unavailable for Outputs of type **Windows Driver**.

Set these options and values as desired, and then click the General tab.

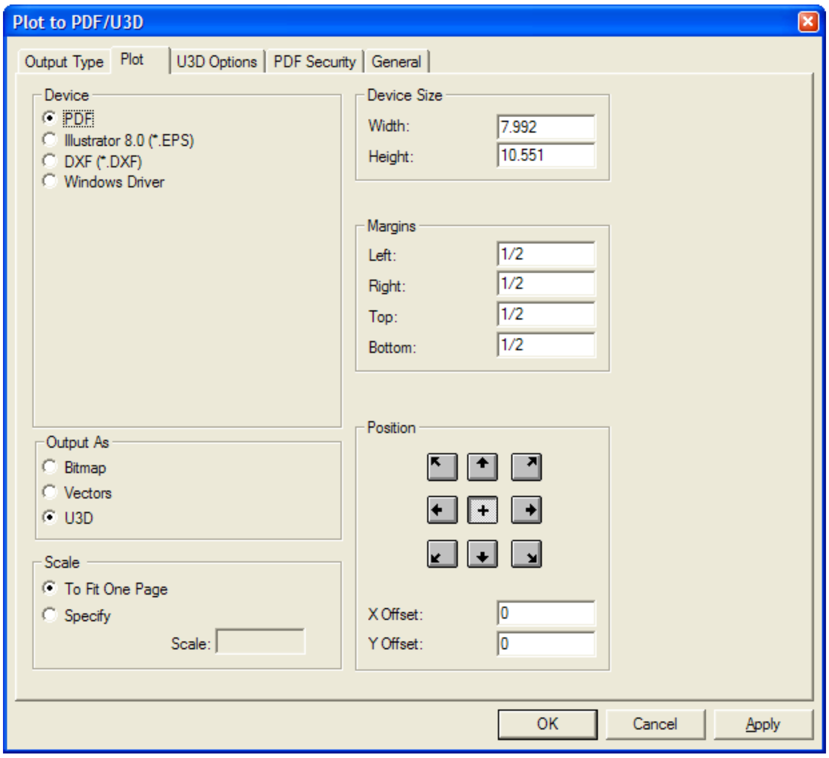


1. On the General tab, the **Output Directory** field lets you specify the directory in which the Output file will go. The **Automatically Open** checkbox controls whether the outputted file is automatically opened by a program associated with its file type, such as Adobe Illustrator for .EPS files. **Automatically Open** has no effect for **Plot to Printer** unless **Out to File** is also selected and the filename includes an extension associated with a program such as .txt.
2. Once you have set the options on the Plot and General tabs as desired, click **OK**.
3. In the Save As dialog box, choose the directory in which to save the file and enter a filename, and click **Save** to perform the Output. If **Automatically Open** was checked on the General tab, ArtiosCAD will launch the default application associated with the outputted file type and show the file.

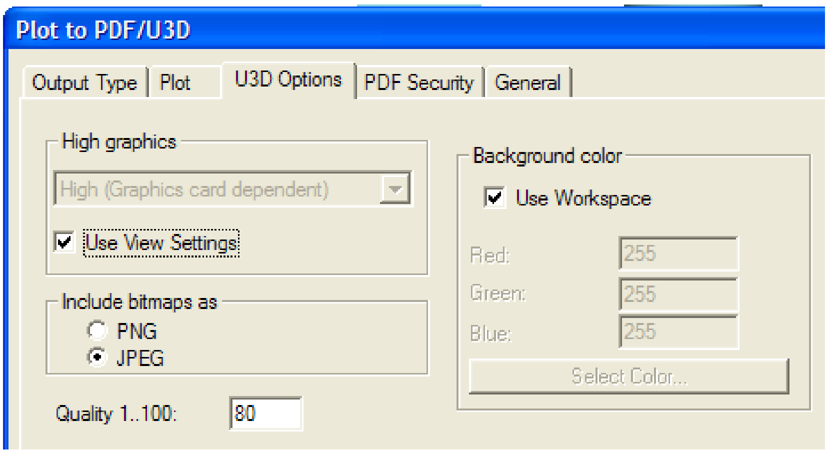
Outputting a 3D workspace to a PDF file with U3D

U3D is an industry-standard 3D output format that can be read by many applications. For example, if you embed U3D data in a PDF file, Acrobat Reader 7.0 or greater lets you zoom in and out, rotate, and otherwise change the view of the objects in the 3D file window.

Creating a PDF file with embedded U3D data is similar to creating a PDF file without it, in that in 3D, you click **File** > **Outputs** > **Plot to PDF/U3D,** except that instead of choosing to output as a bitmap or vectors, you choose **U3D** in the **Output As** group on the Plot tab of the Output dialog box. Also, the U3D Options tab is added to the Output dialog box as shown below.



On the U3D Options tab, in the **High graphics** group, **Use View Settings** uses the current high graphics mode of the workspace. Clear this checkbox to enable the drop-down list box in which you can choose the desired high graphics mode setting.



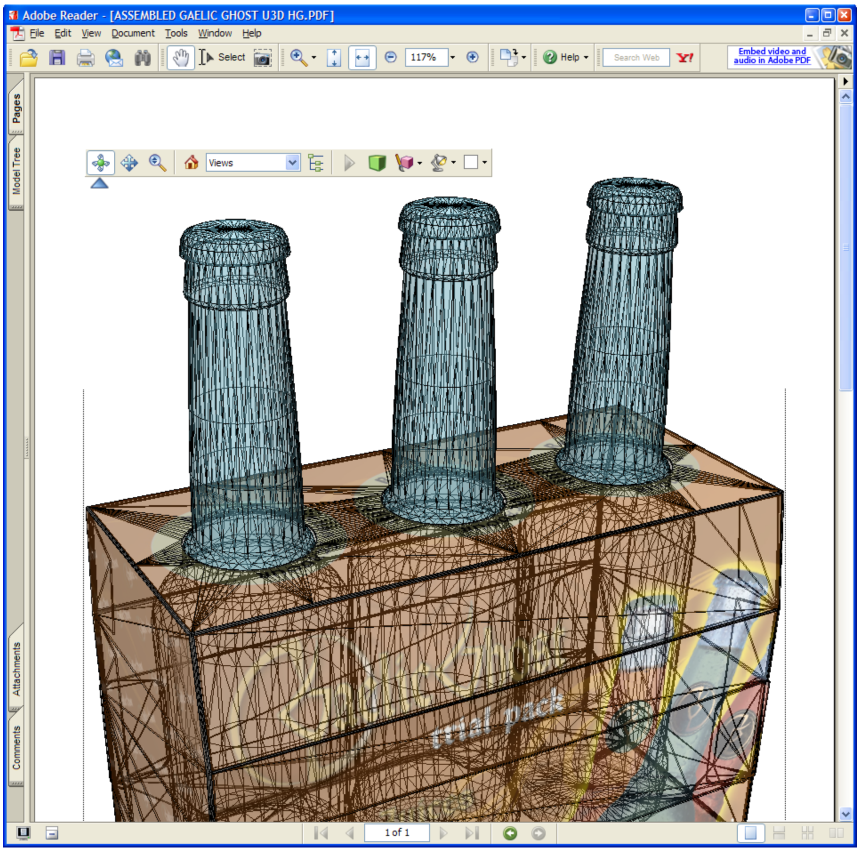
In the **Include bitmaps as** group, choose the file type for the images embedded in the PDF file, either **PNG** or **JPEG**. When JPEG is the selected choice, the **Quality 1..100** field is available. Use a higher number for better-quality images.

In the **Background color** group, **Use Workspace** includes the background color defined in the View Mode dialog box, if there is one. Clearing this checkbox enables the **Red:**, **Green:**, and **Blue:** fields, and also the Select Color button, which leads to the standard operating system color selector.

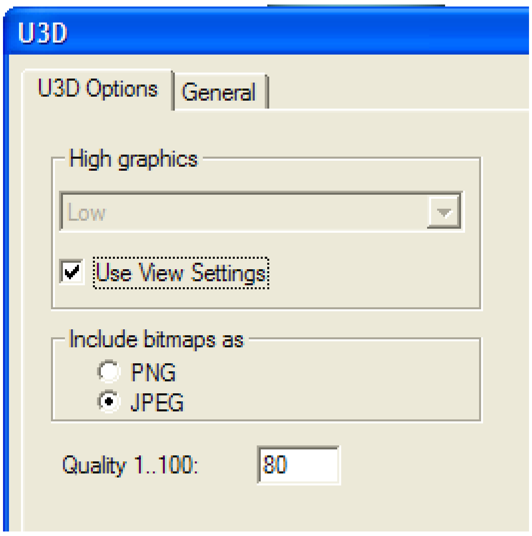
On the PDF Security and General tabs, set the options as desired.

When you have completed setting the various options, click **OK** to make the PDF file with embedded U3D data, or click Cancel to return to ArtiosCAD.

Shown below is a 3D workspace output to a PDF file using U3D in transparent wireframe view.



To output a 3D workspace to plain U3D for use with other applications, use the **U3D** entry on the Outputs-3D menu. Clicking the **U3D** entry opens the U3D dialog box as shown below.



On the U3D Options tab, in the **High graphics** group, **Use View Settings** uses the current high graphics mode of the workspace. Clear this checkbox to enable the drop-down list box in which you can choose the desired high graphics mode setting.

In the **Include bitmaps as** group, choose the file type for the images embedded in the PDF file, either **PNG** or **JPEG**. When JPEG is the selected choice, the **Quality 1..100** field is available. Use a higher number for better-quality images.

On the General tab, the **Output Directory** field lets you specify the directory in which the Output file will go. The **Automatically Open** checkbox controls whether the outputted file is automatically opened by a program associated with its file type, such as Adobe Illustrator for .EPS files.

Once you have set the options on the Plot and General tabs as desired, click **OK**.

In the Save As dialog box, choose the directory in which to save the file and enter a filename, and click **Save** to perform the Output. If **Automatically Open** was checked on the General tab, ArtiosCAD will launch the default application associated with the outputted file type and show the file. However, most systems do not have an application associated with U3D files.

Notes and warnings when using U3D

U3D-enabled Outputs do not follow the 3D View mode settings in the current view, except in the one case of High graphics mode if **Use View Settings** is checked on the U3D Options page of the Output. In particular, U3D files are always created as if **Solid**, **Perspective**, **Show board thickness**, and **Graphics** are turned on in the View mode.

For PDF Reports with embedded U3D data, the U3D models are centered and fill the appropriate 3D file window, regardless of window settings in the Report. The scene is scaled to fit the 3D file window. Also, 3D content appears as a static bitmap of the initial view in the Outputs Preview dialog box.

For Plot to PDF Outputs, the U3D window matches the placement as if **Bitmap** were selected in the **Output As** group. With an edge-on view this will generally result in a skinny U3D window in the PDF file and the model will appear cut off as soon as it is rotated. To get around this, use a PDF Report with a 3D file window of the desired shape and size. Size issues can be somewhat counteracted by using the Marquee Zoom tool in Acrobat to make the 3D window fill the screen.

For both PDF Reports with embedded U3Ds and Plot to PDF Outputs, background images are not embedded in PDFs. Background images for 3D objects are not supported by the current PDF standard (PDF 1.6).

For plain U3D Outputs, the background image is included in the U3D file but will only match an ArtiosCAD view if the viewing window can be made the same size relative to the background image as it is in ArtiosCAD. It may be possible to achieve nearly the same alignment as viewed in ArtiosCAD by not manipulating the object and resizing the window carefully. This means that a model aligned to a background in ArtiosCAD can not be Output in a way that it will necessarily appear aligned to the background when opened in another U3D-enabled application.

U3D background images can only be 24-bit color images; other format background images are ignored, not exported.

Reports with more than one 3D file window with the same file number do not work correctly with U3Ds.

Adobe Acrobat versions 7.0.9 and 8.0 have an issue where the first of several copies of a 3D object does not appear initially. For example, an beer carrier with 6 bottles will initially appear to have only 5 bottles. The missing object(s) appears as soon as you clicks in the 3D area, or you scroll the Acrobat Reader window, or do something else to cause it to update the view. With Acrobat 7.0.9 the temporary disappearance will happen again each time Acrobat's 3D Toolbar is used (such as to change the rendering mode or the lighting).

Adobe Acrobat Professional 7.0.9 does not embed ArtiosCAD-generated U3D data. Users wishing to embed raw U3D from ArtiosCAD in PDF files are currently recommended to use Adobe Acrobat Professional 8.0 (best) or 8.1. In Adobe Acrobat, use **Tools** > **Advanced Editing** > **3D Tool** to embed the U3D data into an existing PDF file.

As of this writing, the best viewer for ArtiosCAD-generated PDFs with U3D models embedded is Adobe Acrobat Reader (or Professional) version 8.0. Version 8.1 has introduced an issue where replicated objects sometime move over other copies of the same object. Adobe Acrobat aggressively updates itself to the highest version available with the same major release number (7.**x** -> 7.0.9 and 8.x goes to 8.1, at the moment). As a workaround, Adobe Acrobat Reader 7.0.9 can still be downloaded from the Adobe website for free and works better than Adobe Acrobat Reader 8.1 for viewing ArtiosCAD files with copied design and solids.

To download Adobe Acrobat Reader 7.0.9, point a Web browser to `http://www.adobe.com/products/acrobat/readstep2_allversions.html`.

Outputting a 3D workspace to a VRML file

Outputting a VRML file from 3D lets you share your work in 3D with people who do not have either ArtiosCAD or the ArtiosCAD Viewer.

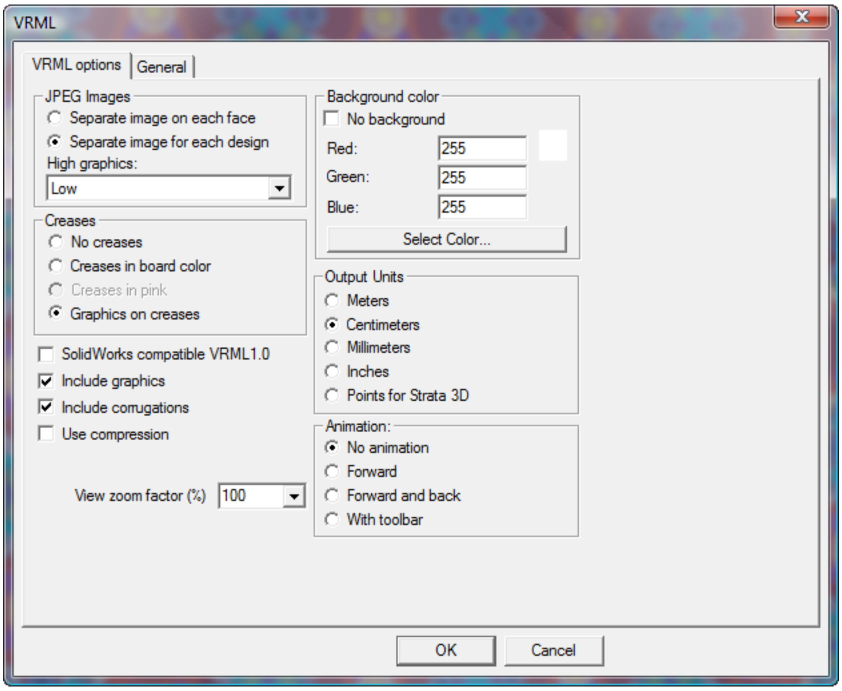
Note: If your workspace contains a tear-away animation, do not output it to VRML as VRML does not support the type of bends used. Use a different Output format.

To output a 3D workspace to a VRML file, do the following:

1. Set the view angle and elevation of the 3D workspace as you want them to be in the VRML file.
2. Click **File** > **Outputs-3D** > **VRML**.

3. Navigate to the directory in which to save the files, and name the main file as desired. The picture files will share the same root name as this main file. As the **Save as type**: field defaults to VRML when exporting from 3D, do not change it. Click **OK** when done naming the file to be saved in the desired location.
4. The VRML Options dialog box appears. The options in it are explained at the end of this procedure. Set the options as desired and click **OK**.
5. The VRML files are created in the desired location. If the **Automatically open** checkbox in the VRML Options dialog box was checked, the Web browser launches and the animation opens.

Shown below is the VRML Options dialog box. All the options in it are configurable in **Defaults** > **Outputs-3D** > **Artios** > **VRML** > **VRML options**.



SolidWorks compatible VRML 1.0 outputs a VRML version 1.0 file for use with SolidWorks and other programs that do not support VRML version 2.0. VRML 1.0 does not support graphics or animation.

Include graphics controls the export of graphics. Turn this checkbox off for the smallest VRML files with the most robust performance possible in the browser; turn it on for rich detail. This checkbox must be checked for the **JPEG Images** and **Creases** groups to be available.

Note: Exporting a design to VRML with graphics turned on creates separate .JPG files. Make sure to keep them with the .WRL file so that the graphics are displayed correctly when the .WRL file is viewed.

Include corrugations, when checked, draws corrugations on the edges of corrugated boards. Turn this checkbox off for smaller VRML files with more robust performance in the browser.

Use compression controls if the VRML geometric information file is compressed or not. A smaller file is faster to download, but some third-party programs do not support compression and may error if this option is checked. The JPEG image files are already compressed according to the **JPEG Quality** setting in Property Defaults and are unaffected by the state of this checkbox.

Automatically open, when checked, launches the Web browser and opens the VRML file as soon as it is created.

View zoom factor sets the initial field of vision in the VRML file. 100 percent uses the same scale factor as in ArtiosCAD. If you find that animations are extending outside of the browser window, try setting this value to a smaller number. The zoom factor can be between 40 percent and 250 percent.

JPEG Images group

On export, the image of each face of the box is converted to a JPEG graphic. **Separate image on each face** creates a separate JPEG file for each face. The size of each file is limited by the resolution of the graphics in ArtiosCAD.

Separate image for each design makes a single large JPEG file for each design in the workspace instead of separate JPEG files for each face. Use this option when using the VRML file in other 3D graphics applications that need to substitute or manipulate the graphics.

The **High graphics:** drop-down list box lets you specify the resolution of the exported graphics as **Low**, **Medium**, **High**, or **Maximum**. **High** and **Maximum** depend on the abilities of the computer's display adapter.

Creases group

There are several ways to handle creases when exporting to VRML. **No creases** simply does not include them in the output. **Creases in board color** makes them the same color as the board, but the lighting in the VRML rendition shows them as slightly contrasting. **Creases in pink** turns the creases pink for easy visibility. **Graphics on creases** wraps the graphics around the creases on the inside and outside of the design(s). This is the most sophisticated option, but it increases file size by 50 percent.

Background color group

The settings in the **Background color** group control the color shown by the browser in the empty space as the VRML file plays. **No background** allows the preferences set in the VRML plug-in to be used. When checked, the rest of the options in the group are unavailable.

The values in the **Red:**, **Green:**, and **Blue:** fields allow you to set custom values for each color. The results of the settings are shown next to the **Red**: field.

Clicking **Select Color** leads to the standard Windows color selection dialog box. Click **OK** when you have chosen a color.

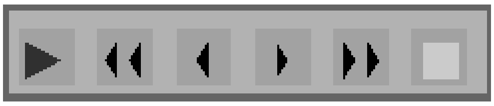
Output Units group

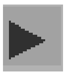
As there is no measurement ability in the browser window, the options in this group control the granularity of the VRML plug-in's zoom tools. Selecting **Meters** provides coarse zoom while selecting **Millimeters** provides fine zoom. **Centimeters** is the default, and should be selected when choosing **With toolbar** in the **Animation** group. **Points for Strata 3D** should be selected when the VRML file will be opened in Strata 3D; select **No animation** in the **Animation** group as well for files destined for Strata 3D.

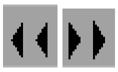
Animation group

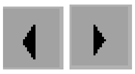
The options in the **Animation** group control how the animation behaves in the browser.

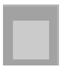
No animation creates an unmoving view of the design. **Forward** runs the animation continuously. **Forward and back** runs the animation forward, then backward, and repeats. **With toolbar** inserts a toolbar into the VRML file allowing you to manually step through the file, as well as toggle design transparency. The toolbar is shown below:

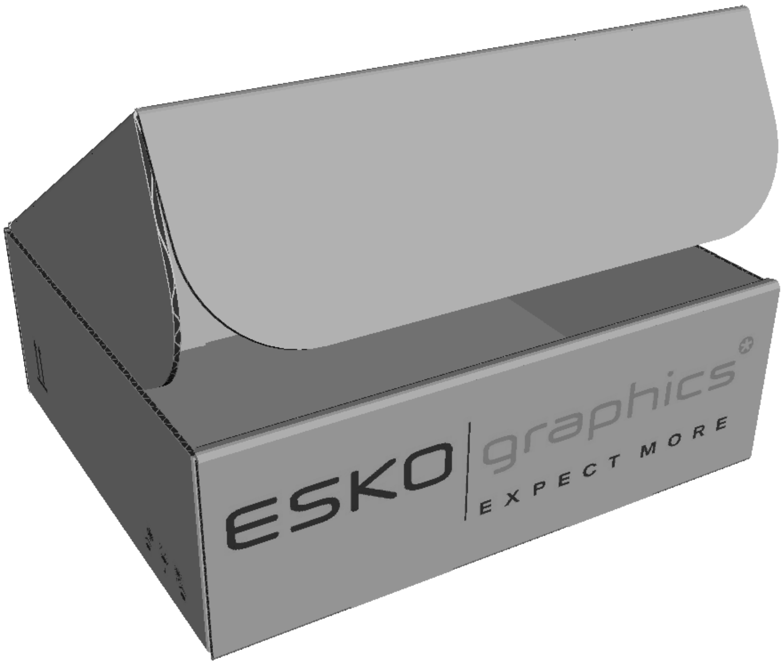


 The first button plays the animation.

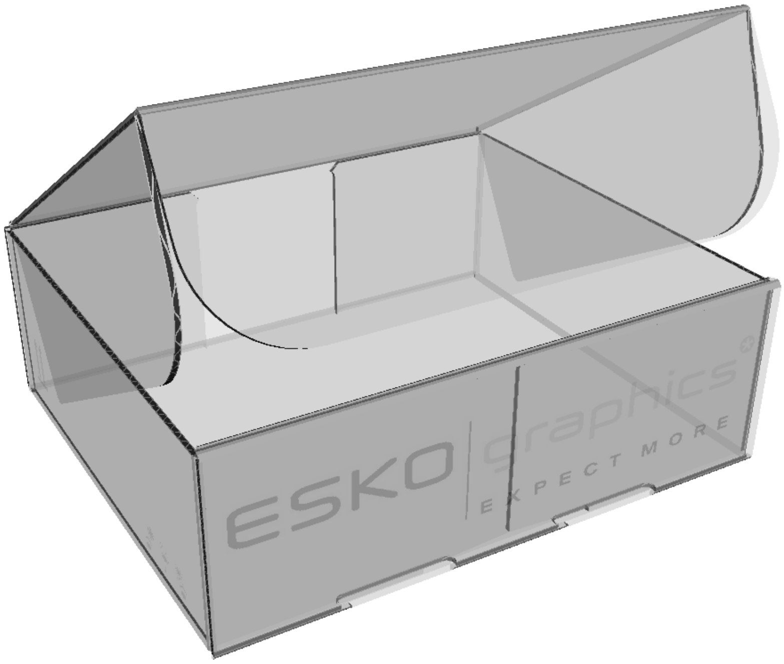
The second and fifth buttons move to the first and last frames of the animation, respectively.

 The third and fourth buttons move back or forward one frame in the animation sequence, respectively.

 The sixth button toggles transparency. Shown below is a frame of an animation with transparency off.



Shown below is the same frame with transparency on.

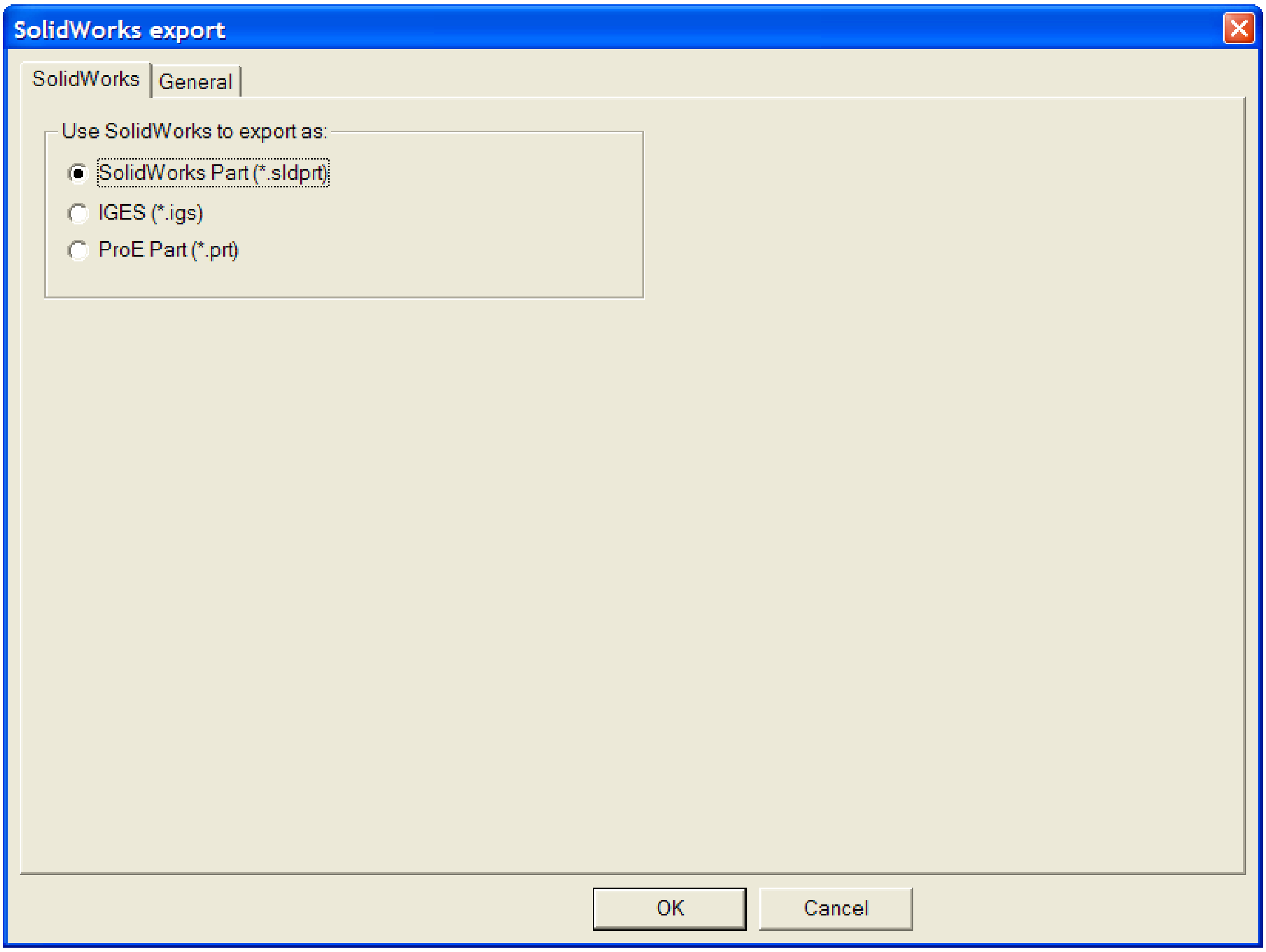


Outputting a 3D workspace to a SolidWorks file

The **SolidWorks export** command on the Outputs-3D menu lets you output a 3D workspace as a SolidWorks part, an IGES file, or a ProEngineer part. You must have purchased the SolidWorks option and installed the SolidWorks Import/Export component for this feature to work.

To perform the SolidWorks export, do the following:

1. Position the items in the 3D workspace as desired and save the workspace.
2. Click **File** > **Outputs-3D** > **SolidWorks export**.



3. Choose the type of file to create.
4. On the General tab are the **Output directory** field and the **Automatically Open** checkbox. Set them or leave them blank as desired.
5. Click **OK**.
6. In the Save As dialog box, navigate to the desired directory (if necessary) and enter the filename in the **File name:** field.
7. Click **Save**. The Output is performed.

Notes and warnings

The exported file has a SolidWorks body for each for each face and crease in each carton, as well as for each solid. Each SolidWorks body has the outside color of the ArtiosCAD 3D design, but there is no separate inside color, and no graphics.

Graphics, dimensions, and bounding boxes are not exported.

ProEngineer and IGES exports do not include color.

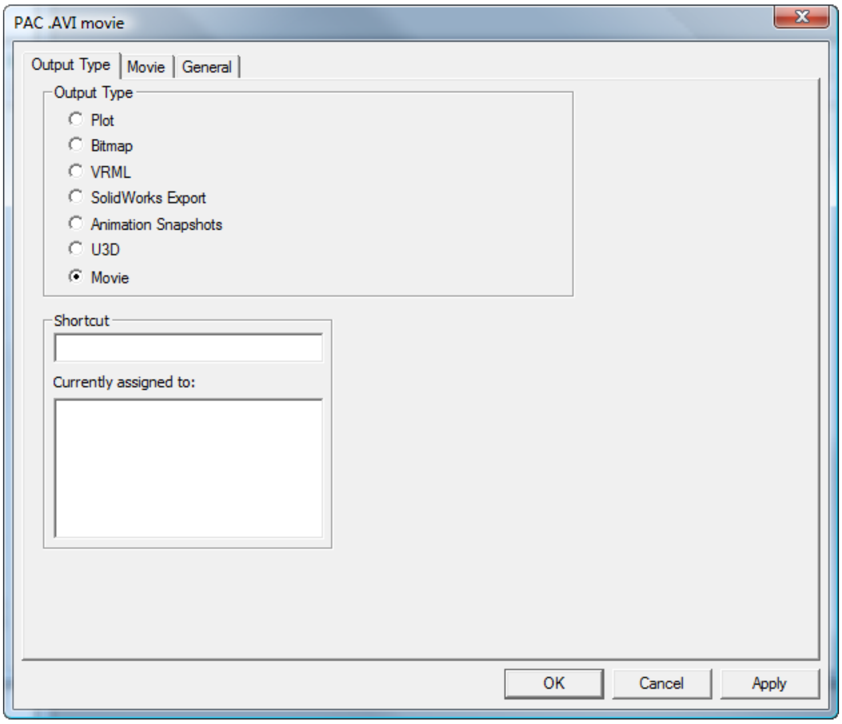
Exporting a 3D workspace as a ProEngineer part automatically appends a version number to the filename based on the contents of the output folder. The export process also replaces any non-ASCII characters with an underscore (_).

Some imported VRML and IGES files might not export correctly from ArtiosCAD if they do not have well-formed solid bodies. Imported CATIA and SolidWorks files should have well-formed solid bodies.

SolidWorks files are not backward-compatible between versions of SolidWorks. To show the version of SolidWorks that ArtiosCAD uses, click **Help** > **Diagnostics** > **Show SolidWorks Version**.

3D Movie Outputs

If you have the 3D Animation module, you can now export .AVI and .MOV movies from ArtiosCAD. The **Movie** output type has been added to Outputs-3D entries.



Note:

This feature is highly dependent on the multimedia codecs on your computer. Esko cannot guarantee that the codecs on your personal computer will work as anticipated. If you have trouble using this feature, you should make sure you have the newest codecs available from your operating system vendor or other third-party software vendor.

Note:

Outputting a .MOV file requires the Apple QuickTime Player to be installed on your computer.

See the *Defaults* chapter of the *ArtiosCAD Administrator Guide* for more information on creating a Movie output.

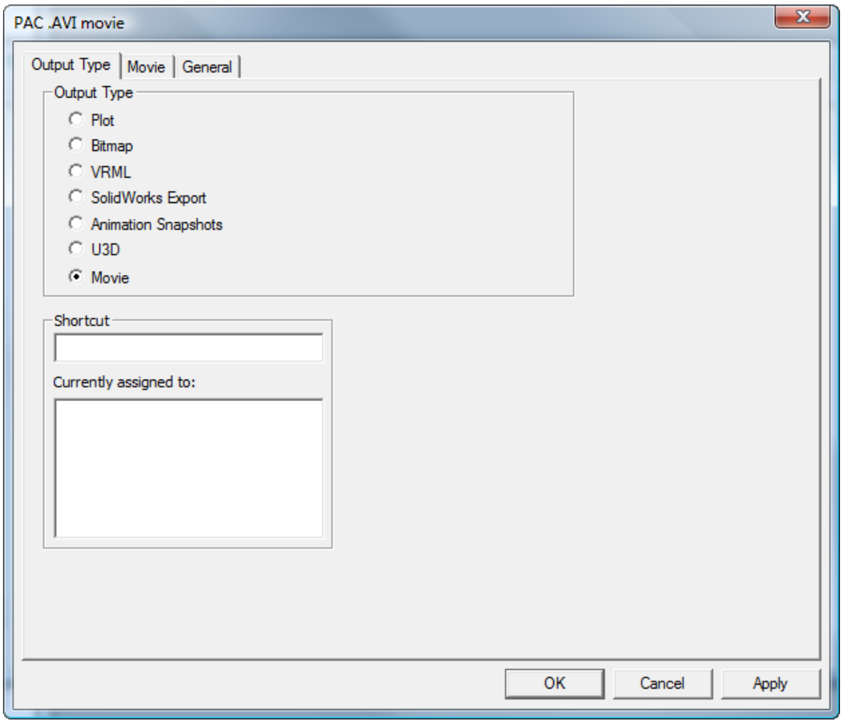
Creating a 3D Movie Output in Defaults

Before you create a 3D Movie output, keep these considerations in mind:

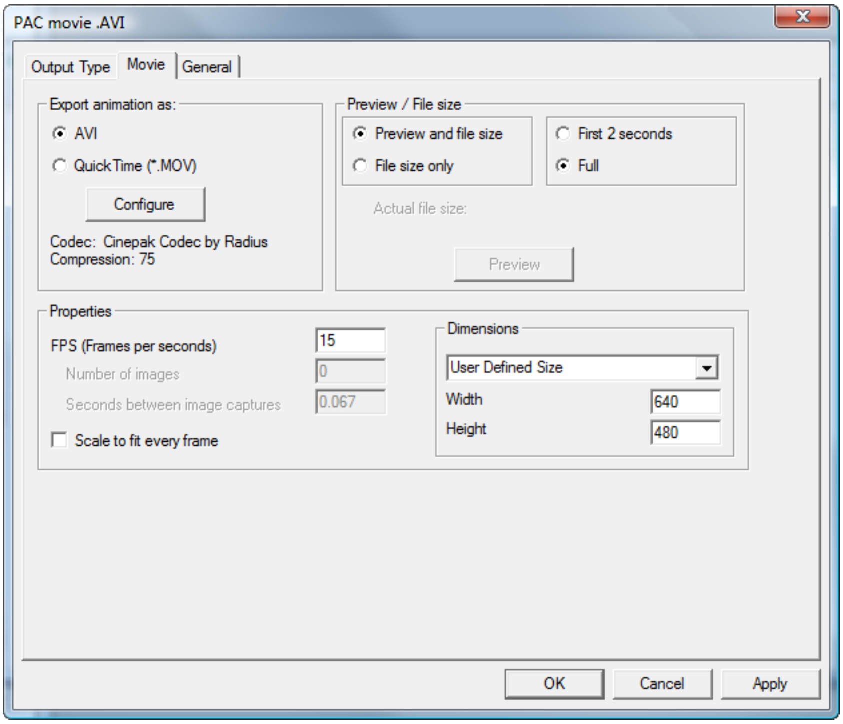
- Multimedia codecs vary between computers and there is no guarantee that a movie you make on your machine will work flawlessly the first time on someone else's computer. If you encounter problems, you may need third-party support from your operating system vendor or codec supplier.
- You need the 3D Animation module on your computer to use this Output.
- You need the free Apple® QuickTime® Player Installed on your system to output files in .MOV format. This software is available for download from at `http://www.apple.com/quicktime`.

Create a 3D Movie Output definition by doing the following:

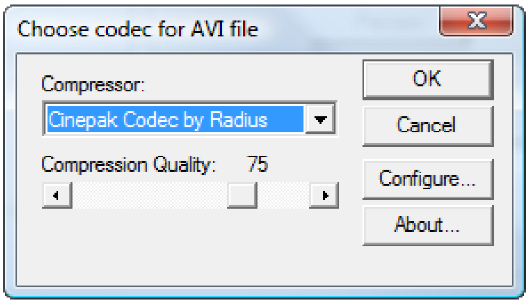
1. Start ArtiosCAD, open Shared Defaults, and expand the Outputs-3D catalog.
2. Right-click **Outputs-3D**, then click **New** > **Data**.
3. Enter a name for the Output, press `Enter`, and double-click the new entry.
4. In the **Output Type** group, select **Movie**.

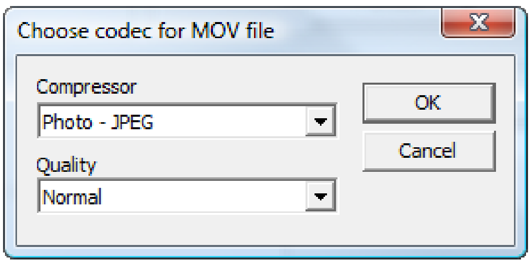


5. Click the **Movie** tab.



6. In the **Export animation as** group, choose either **AVI** or **QuickTime (*.MOV)**. AVI files are generally used on computers using the MIcrosoft Windows family of operating systems while QuickTime files are generally used on the Apple Macintosh.
   a) Supplemental information about the codec used by the selected format is shown beneath the **Configure** button. The default settings are sufficient for most situations, but if you are not achieving the desired results, click **Configure**.





b) in the **Choose codec for AVI file** or **Choose codec for MOV file** dialog box, set the codec ArtiosCAD uses to create the movie in the **Compressor** field. The list of compressors available is dependent upon the software loaded on your system.

c) For .MOV files, each compressor may have different configuration options available to it via the **Configure** button. Depending on the output format and compressor you chose, you may also be able to set the compression quality using the slider for .AVI files or the drop-down list for .MOV files.

d) When you have selected the compressor and quality settings to use, click **OK** to return to the **Movie** tab in the Output definition.

7. In the **Preview/File size** group, set the default selection for these options when creating the Output. **Preview and file size** will make a preview of the movie and compute its file size based on the selected codec configuration. **File size only** will only compute the file size. The times in the other group either preview the **First 2 seconds only** or the **Full** movie length. **Get file size** is available only when creating the Output.

8. In the **Properties** group, set the **FPS (Frames per second)** value to a number between `1` and `30`. 15 is the default. A higher number results in smoother video. **Scale to fit every frame** performs a Scale to Fit for every frame of the video.

   a) **Number of images** and **Seconds between image captures** are read-only fields that display their relevant information when creating the Output.

9. In the **Dimensions** group, set the width and height in pixels of the movie manually by selecting **User Defined Size** from the drop-down list box, or choose one of the pre-defined sizes. The sizes must both be exactly divisible by 4.

   a) **NTSC** (National Television Standards Committee) is the video format used for televisions in the United States and Japan. **PAL** (Phase Alternating Line) is the format used in some of the rest of the world.

   b) DVD, SVCD, and VCD refer to the size of the frame in pixels - Digital Versatile Disk being large, Super Video CD being medium, and Video CD being small.

   c) DivX, XviD, and WMV are all standard sizes for movies intended to be watched on a computer.

   a) DV stands for digital video which has a standard width of 720 pixels. The height varies by country standard.

10. Once you have set the options on the **Movie** tab as desired, click the **General** tab, on which you can set the **Output Directory** (if desired) and also the **Automatically open** checkbox, which is checked by default.

11. Click **OK** to finish defining the Output.

12. Save and exit Defaults as usual.

Outputting a Movie from 3D

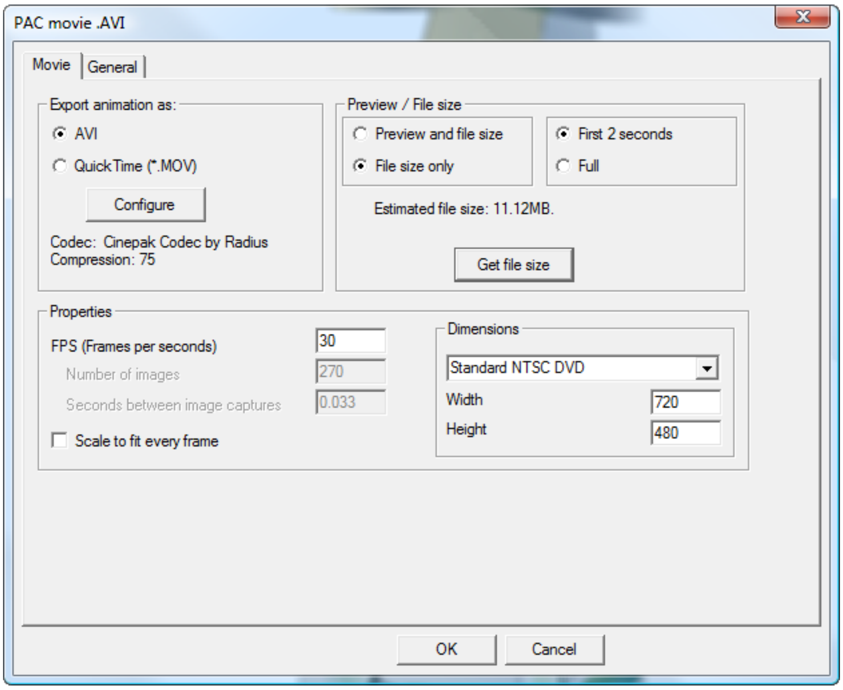
Because VRML and U3D outputs do not support the methods used for curved creases, you can now output .AVI and .MOV movie files of your animation sequences. You must have the 3D Animation module to output movies from 3D.

.AVI files can be played on the Apple Macintosh using **VLC Player** which can be downloaded from: `http://www.videolan.org/vlc/`, and MOV files can be played on computers using the MIcrosoft Windows family of operating systems using the Apple® QuickTime® Player which can be downloaded from `http://www.apple.com/quicktime`. Both players are free.

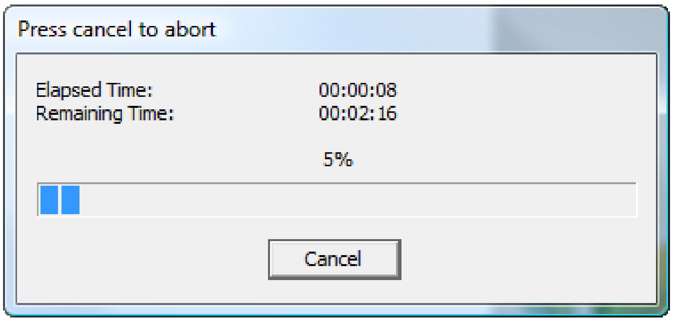
Before you can Output a movie from 3D, you must have configured a movie Output in the **Outputs-3D** catalog in Defaults. See the *Defaults*chapter of the *ArtiosCAD Administrator Guide* for more information.

To Output a movie from 3D, do the following:

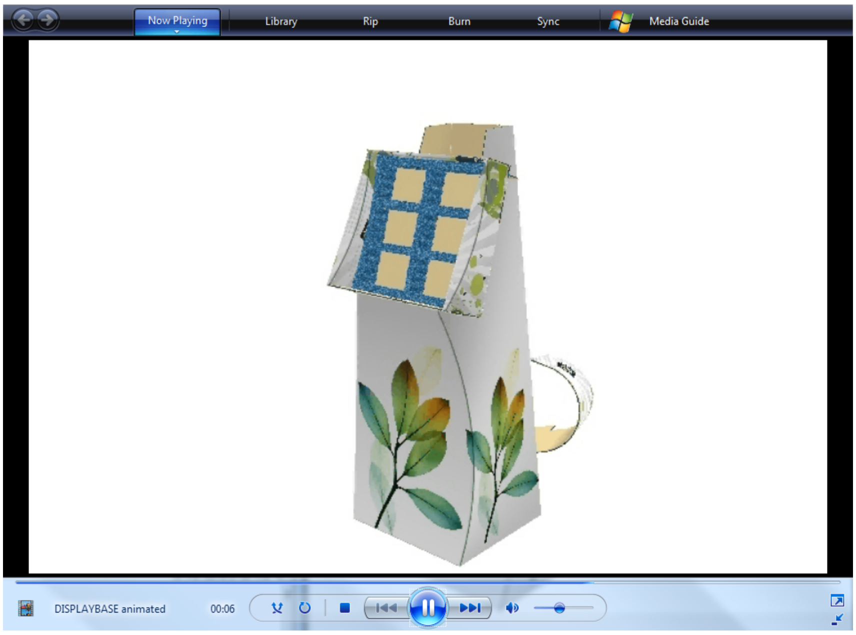
1. Create or open a 3D workspace that has an animation sequence defined.
2. Click **File** > **Outputs-3D** and click the desired movie output.
3. Set the options on the **Movie** tab as desired. For more information about each option, see the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

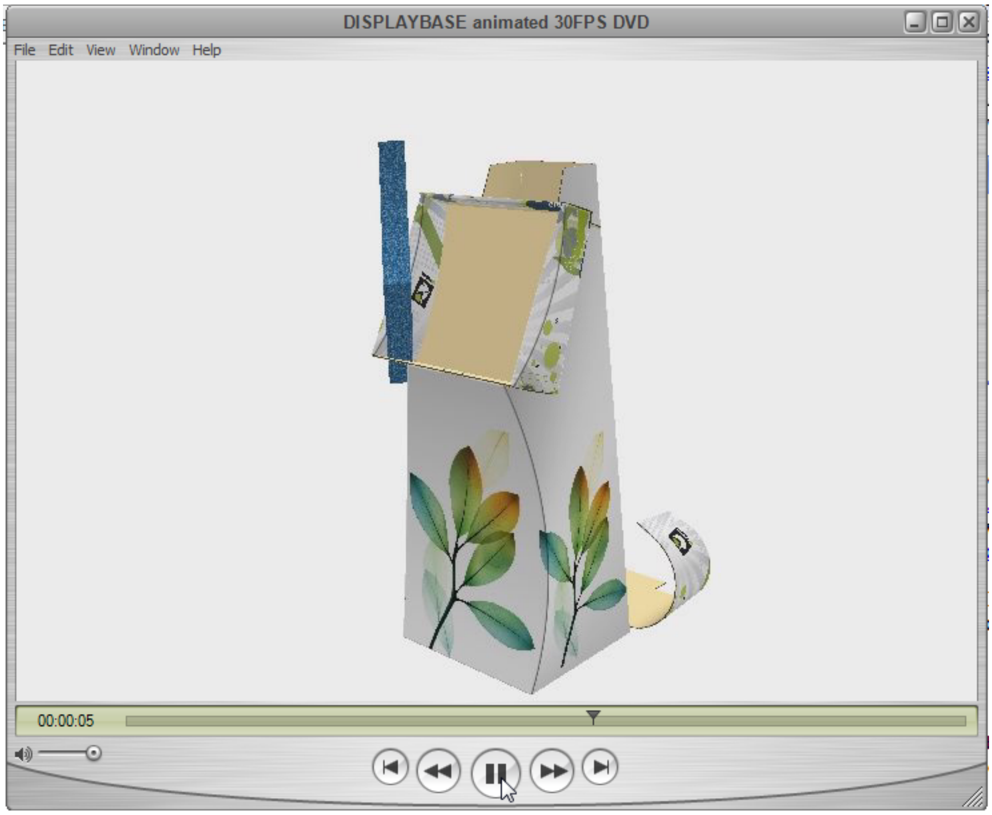


4. If desired, click **Preview/Get file size** in the **Preview/File size** group to generate a preview of the movie or find out how large the file will be. Adjust the settings if the file size is too large; some email systems still have 5 MB attachment limits.
5. Click the **General** tab and set the options as desired.
6. Click **OK** and enter a name for the movie file in the **Save As** dialog box to begin the Output.
7. A progress dialog box appears. If it is going to take too long to Output the movie, click **Cancel**. In that case you should retry the Output with different settings - a lower FPS setting, a smaller frame size, greater compression - that will result in a smaller, faster-to-generate file.



8. If **Automatically Open** on the General tab was checked, the movie file opens automatically in the associated media player.




DISPLAYBASE animated 30FPS DVD
File Edit View Window Help
00:00:05

9. DataCenter

Introduction to DataCenter

DataCenter is a program which makes finding specific information about ArtiosCAD designs quick and easy. As you work, DataCenter automatically records facts about every single design and every manufacturing file you create or modify.

DataCenter has two intertwined parts – the information-gathering engine in ArtiosCAD and the browsers in which the information is viewed. The engine sends information to a database using MSDE, Microsoft SQL Server, SQL Server 2005 Express Edition, or Oracle.

Within ArtiosCAD, you can search for single designs and manufacturing files, as well as viewing information for embedded designs in manufacturing files. Other browsers, for information such as board codes, customers, userfields, and so forth are contained within the **DataCenter Admin** program in the **Esko** > **ArtiosCAD** program group.

Concepts and Ideas in DataCenter

Browsers

There are fifteen main areas of DataCenter called **browsers** that display information. They are:

- The **Design** browser (in ArtiosCAD)
- The **Manufacturing** browser (in ArtiosCAD)
- The **Embedded Design** browser (designs saved in manufacturing files, shown on a per-manufacturing file basis in ArtiosCAD)
- The **Projects** browser (in ArtiosCAD)
- The **Auto-Number** browser (in DataCenter Admin)
- The **Board** browser (in DataCenter Admin)
- The **Characteristics** browser (in DataCenter Admin).
- The **Company** browser (in DataCenter Admin)
- The **Company Type** browser (in DataCenter Admin)
- The **Counter** browser (in DataCenter Admin)
- The **Person** browser (in DataCenter Admin)
- The **Resource** browser (in DataCenter Admin)
- The **Server** browser (in DataCenter Admin)
- The **Userfield** browser (in DataCenter Admin; requires Information Enhancement module)

- The **Restricted Userfield Set** browser (in DataCenter Admin)

Viewing database information

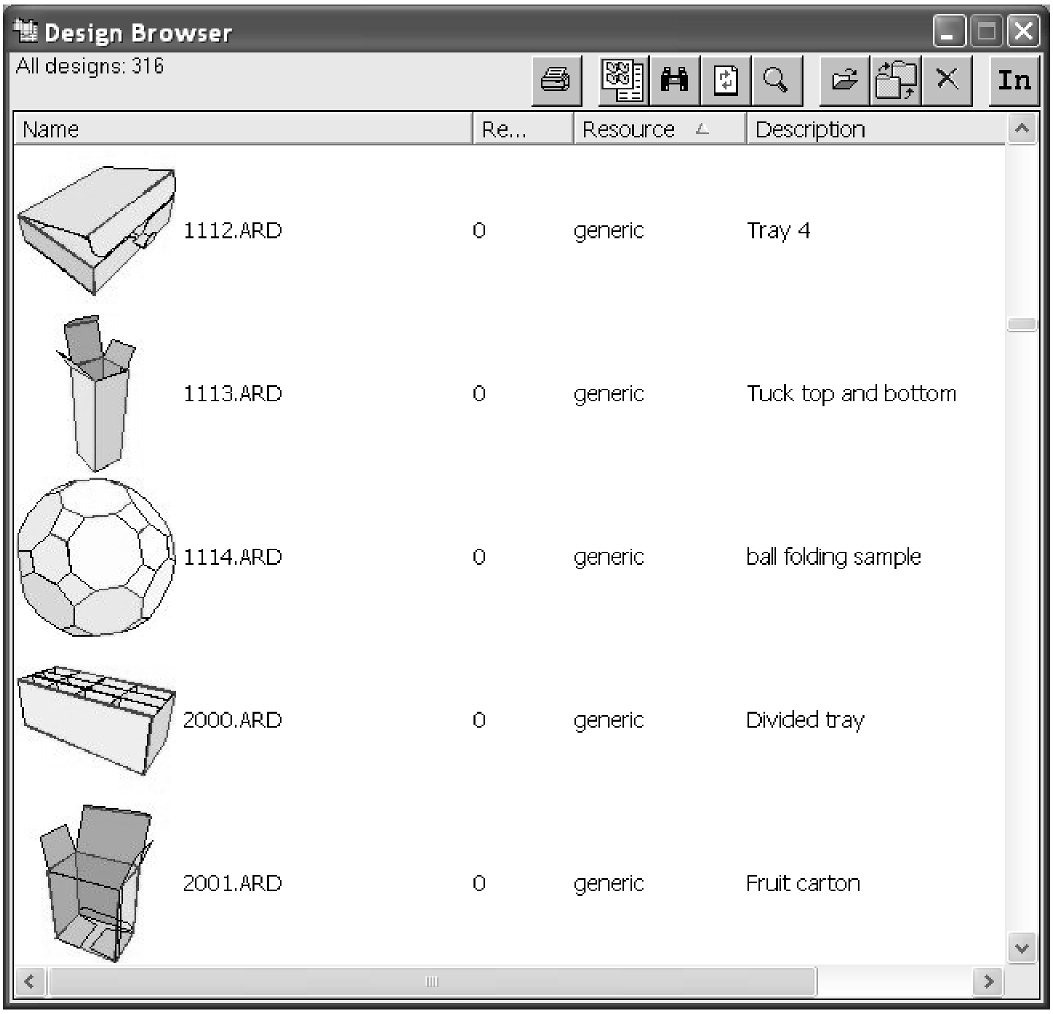
In ArtiosCAD, to view all the single designs or manufacturing files in the database without searching for them, open the appropriate browser from the **Database** > **Browsers** menu.

The browser windows are independent of the main ArtiosCAD window.

To view all non-design information, open **DataCenter Admin** in the ArtiosCAD program folder or on the desktop.

Sorting the list of information

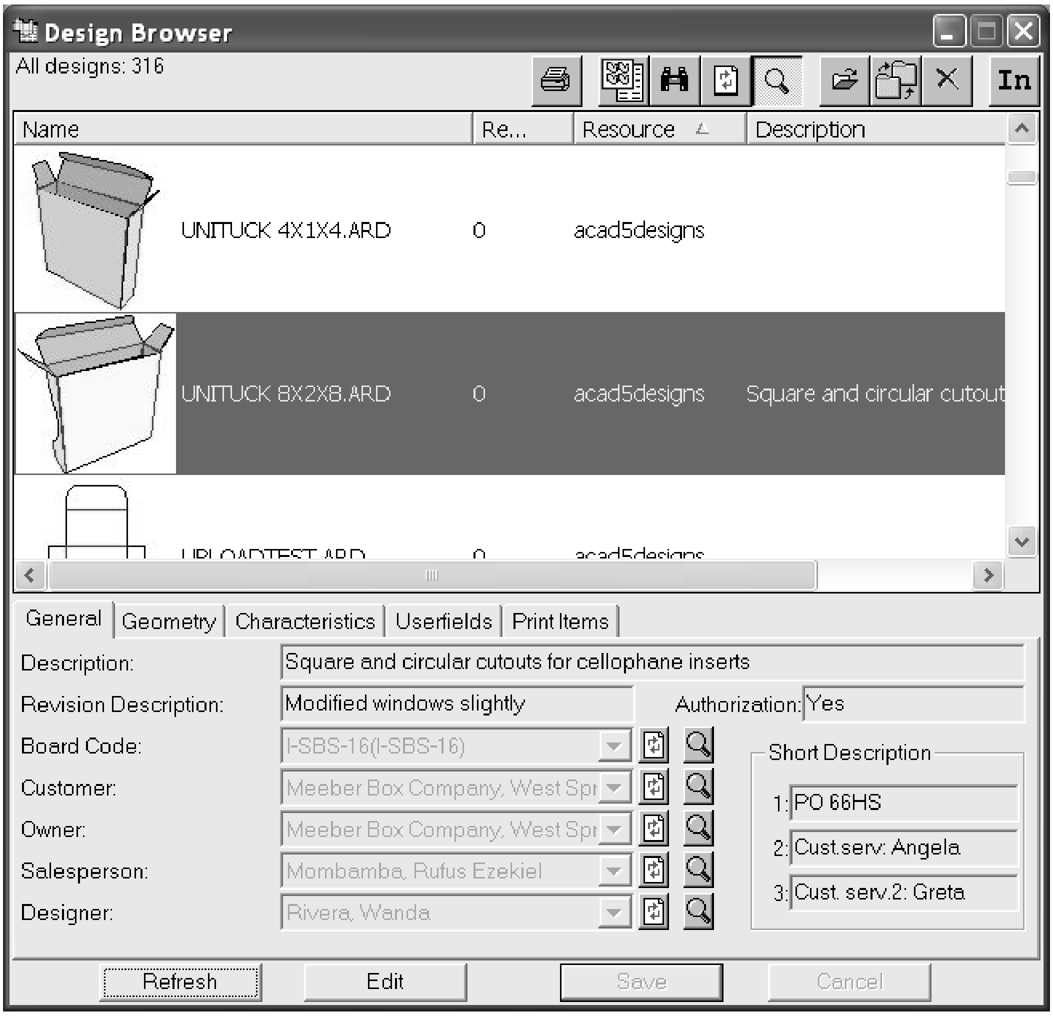
All browsers except for Characteristics show a list of items in the database that can be sorted by column. To sort the information, click the button with the column name at the top of each column. Successive clicks of the same button change the sort order from ascending to descending and vice-versa.



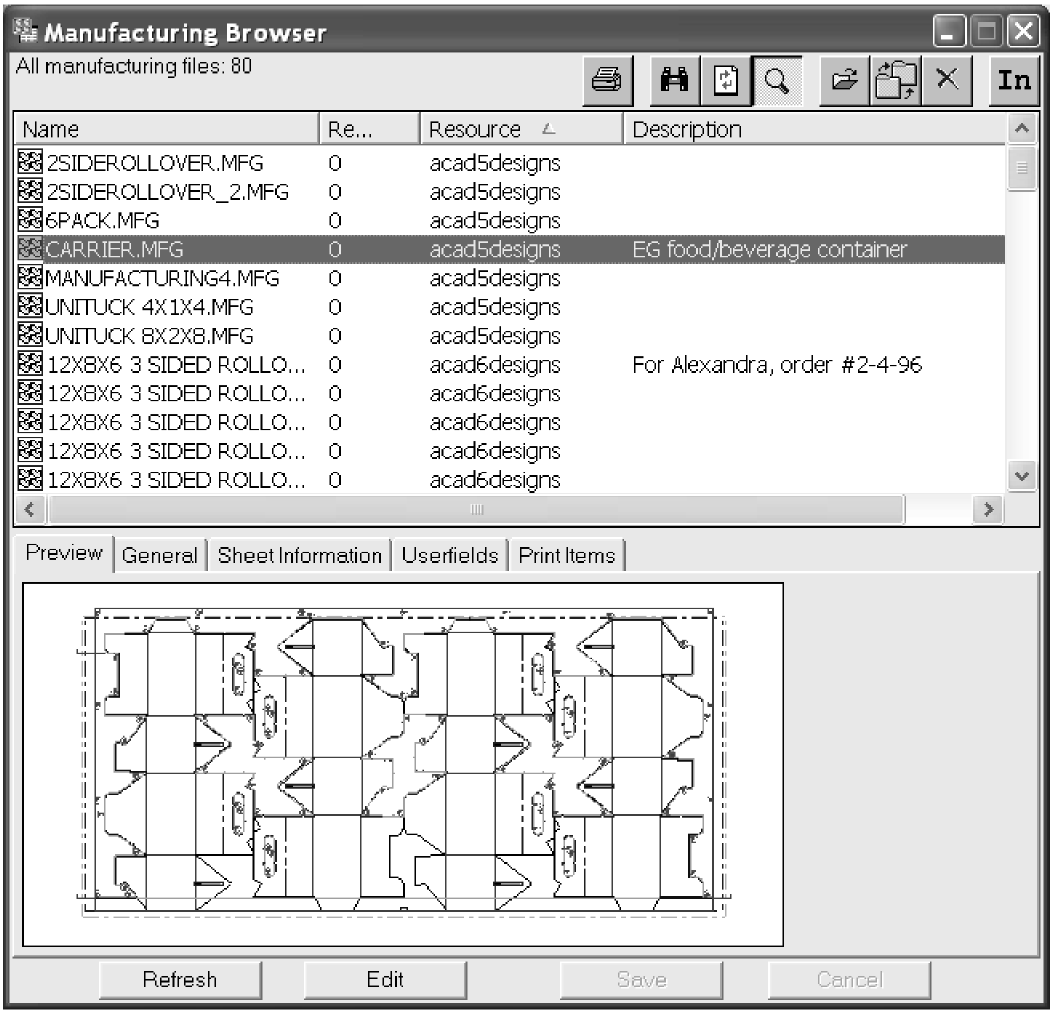
Selecting an item from the list and viewing its information

Double-click an item in the list to select it and view its details. The clicked item's information will appear beneath the list. Select more than one item by holding down CTRL and clicking the desired items.

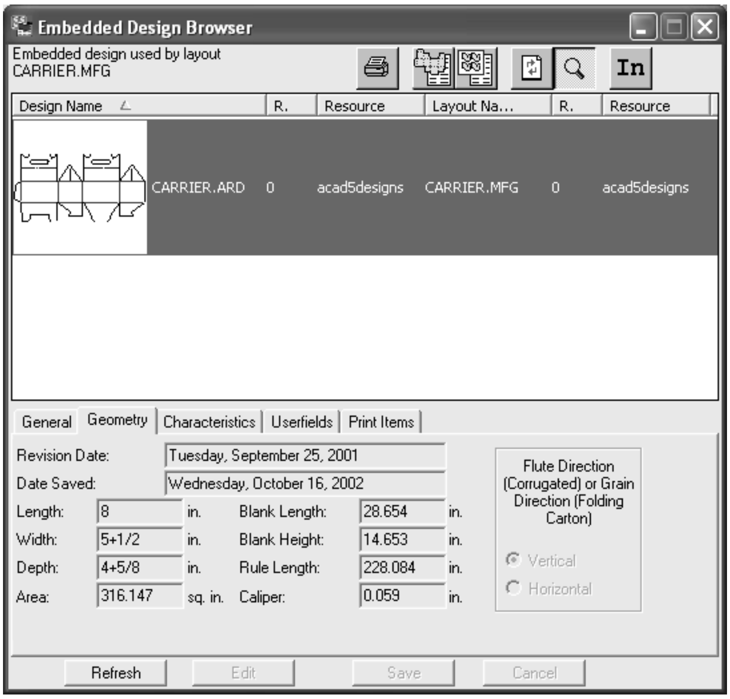
The details area of the window usually has more than one tab. Switch to a different tab by clicking it. Shown below is the Design browser in ArtiosCAD.



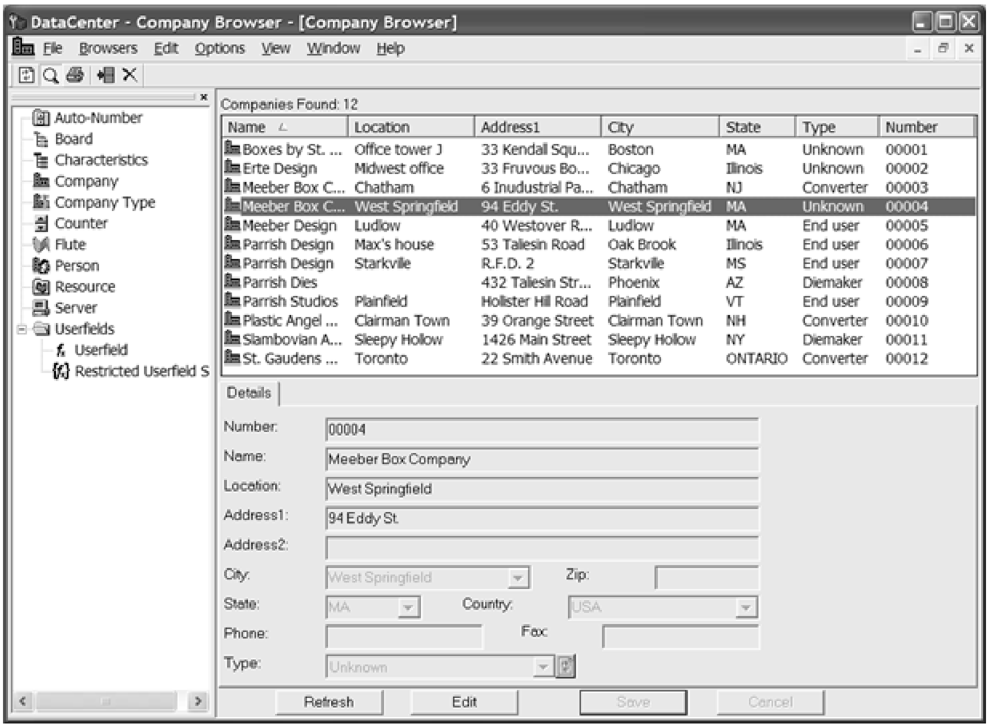
Shown below is the Manufacturing browser in ArtiosCAD showing the details for the selected manufacturing file.



Shown below is the Embedded Design browser in ArtiosCAD showing the details for the selected item.



Shown below is the Company browser in DataCenter Admin showing the details for the selected item.



 To view specific information for a field, click **Show Item Details**. A dialog box appears showing details for the selected item. Click **Cancel** to close the dialog box.

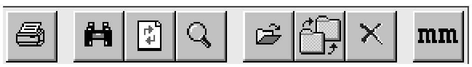
The toolbars

Each browser in ArtiosCAD has a unique toolbar, while the browsers in DataCenter Admin share the same toolbar.

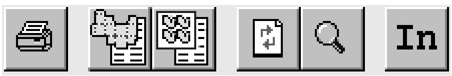
Toolbars in ArtiosCAD Browsers

Shown below is the Design browser toolbar.

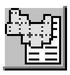
Shown below is the Manufacturing browser toolbar.

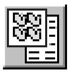


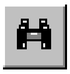
Shown below is the Embedded Design browser toolbar.

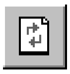


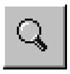
 Clicking **Print** prints a report for the current browser.

 **Find Associated Design File** in the Embedded Designs browser finds the non-embedded version of the design.

 **Show Associated Manufacturing file** shows the manufacturing files containing an embedded copy of the selected design. Only one record may be selected when using this feature.

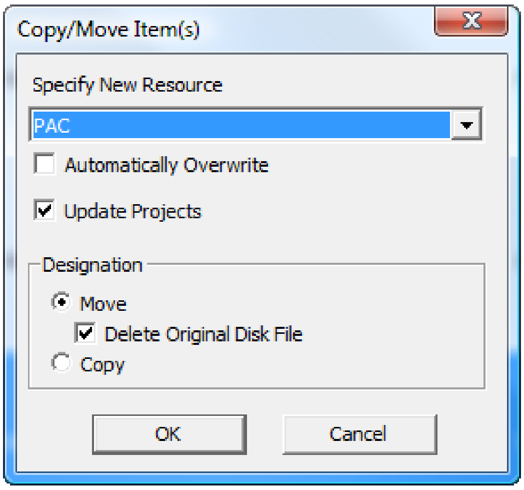
 **Edit Search Criteria** opens the Search dialog box so that you can refine the search results.

 **Refresh Browser** reloads the browser after clearing any search criteria.

 **Show Details** toggles the display of the details area beneath the browser window.

 **Open file** opens the selected file(s) in ArtiosCAD.

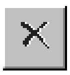
 **Move/Copy to New Resource** opens the Copy/Move Item(s) dialog box, which lets you copy or move the current selection to a new resource.



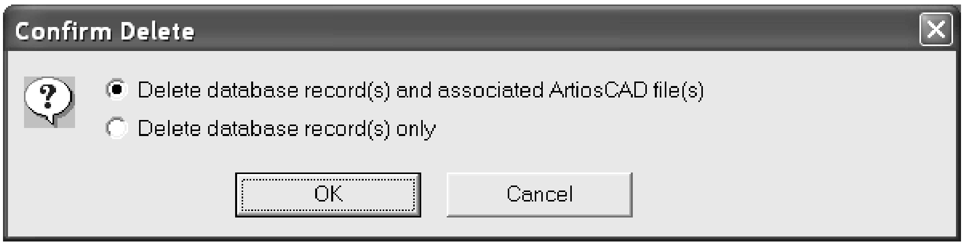
Specify the new resource in the **Specify New Resource:** drop-down list box. To overwrite files in the new resource without being prompted for each file, check the **Automatically Overwrite** checkbox. **Update Projects** prevents accidentally losing Project data when moving items between resources.

Note: Remember to back up your database regularly. Restoring it is the only way to recover from making such a mistake with Project data.

In the **Designation** group, choose to **Move** or **Copy**. If moving, check the **Delete Original Disk File** checkbox to remove the original file, leaving just the file(s) in the new resource. Click **OK** to perform the move or copy, or **Cancel** to return to the browser.

 **Delete** deletes the current selection. When clicked, you are prompted to choose whether to delete both the database record and associated ArtiosCAD file or just the database record as shown

below. Choose the desired option button and click **OK** to delete or **Cancel** to return to the browser with no deletion.



Units changes the units between imperial and metric.

Toolbar in DataCenter Admin

Shown below is the toolbar in DataCenter Admin.

 **Refresh** resets the search criteria and reloads the browser.

 **Details** toggles the display of details for the selected item(s).

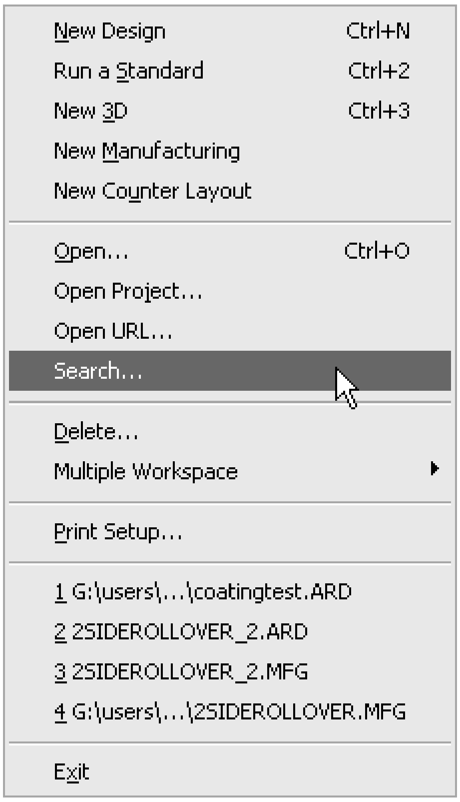
 **Reports** prints a report of the information in the current browser, or, if more than one item is selected, asks if the report is to encompass just the selected records or the entire browser.

 **Insert New Record** adds a new record to the current browser.

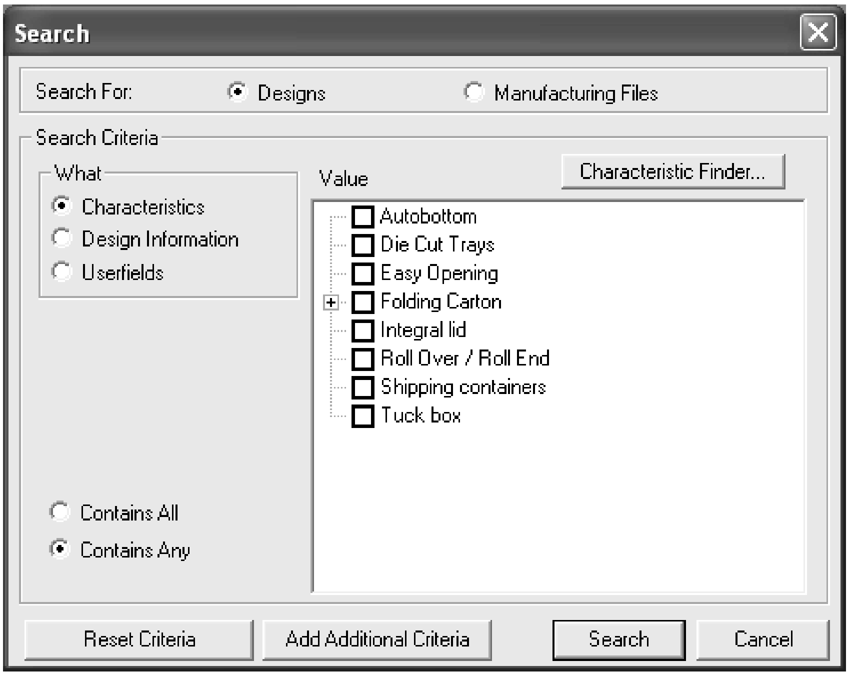
 **Delete** deletes the currently selected records. It is available only when there is a current selection. You are asked to confirm the deletion before it takes place.

Searching for information

A database is only as good as the information in it. To be useful, that information must be easy to retrieve. To retrieve the information quickly and easily, use the **Search** command on the File menu in ArtiosCAD.



Shown below is the Search dialog box.



To search for information, do the following:

1. Start ArtiosCAD, and click **File** > **Search** to open the Search dialog box.
2. Select **Designs** or **Manufacturing files** in the **Search For**: group.
3. In the **What** field, choose the kind of data to use as search criteria.

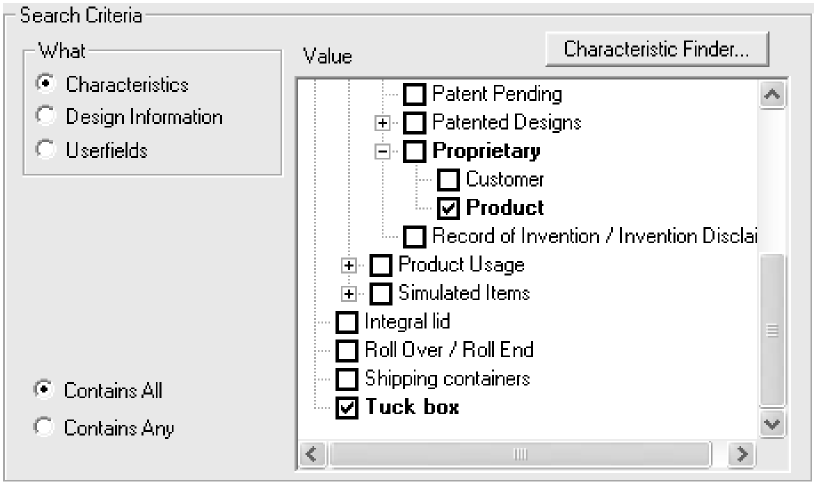
Characteristics are attributes assigned to the design, such as the style of container, or its intended usage.

Design Information is information such as resource name, owner name, width, and so on. This option is unavailable when searching for manufacturing files.

Userfields are customizable variables whose values are saved in a single design or manufacturing file.

4. The option button selected in the **What** field controls the appearance of the rest of the **Search Criteria** group.

   When it is set to **Characteristics**, the Characteristic tree appears, as do the **Characteristic Finder** button and the **Contains All** and **Contains Any** option buttons. As manufacturing files do not use characteristics, this option is unavailable when searching for them.

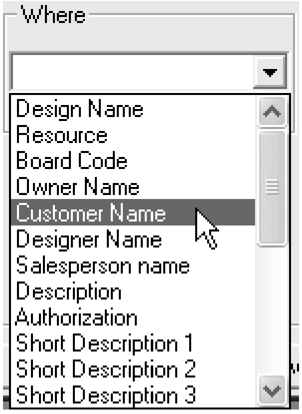


   The Characteristics tree shows data hierarchically for easy categorization. Check the checkboxes for characteristics to search for. **Contains All** searches for single designs or manufacturing files which have all the selected characteristics defined. **Contains Any** searches for single designs or manufacturing files which have at least one of the selected characteristics.

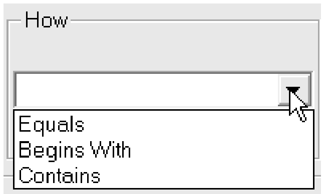
   The Characteristics Finder is described later in this chapter.

   When the option button in the **What** field is set to **Design Information** or **Userfields**, the **Where**, **How**, and **Value** fields appear. The **How** field appears after the **Where** field is set, and the **Value** field appears after the **How** field is set.

   In the **Where** field, choose the type of item to search for.

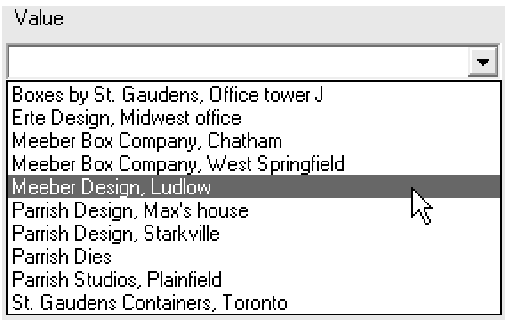


For text items, in the **How** field, choose **Equals** to set the **Value** using something already in the database. Choose **Begins With** to match items that begin with the text you enter. Choose **Contains** to type a custom value.

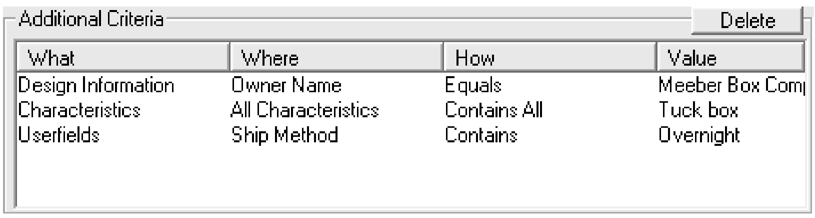


For numeric selections in the **Where** field, **>=** (greater than or equal to), **<=** (less than or equal to), and **Is Between** are available in the **How** field. When using **Is Between**, enter the lower limit in the **From** field and the higher limit in the **To** field.

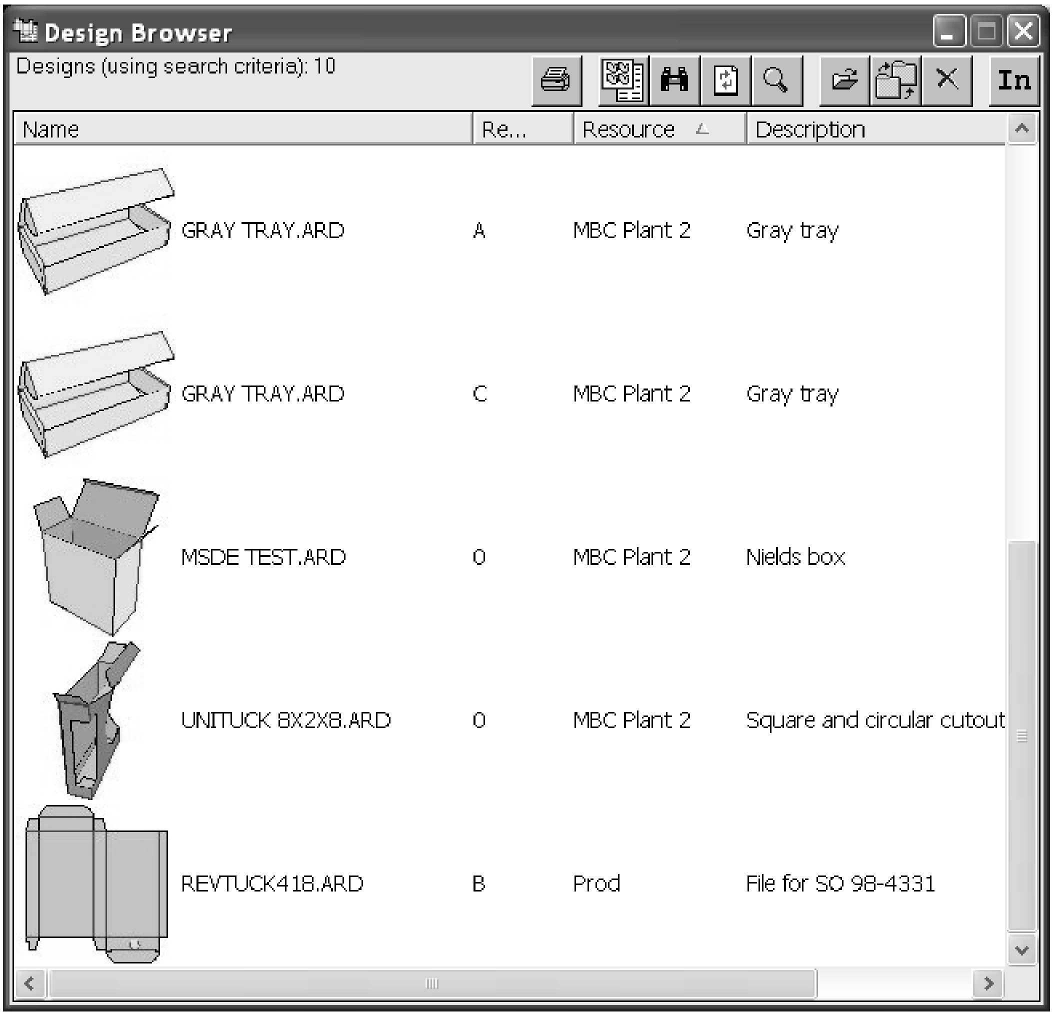
In the **Value** field, either choose an item from the dropdown list (if **Equals** is selected in the **How** field) or type the desired information to search for (if **Contains** is selected in the **How** field).



5. Once all the fields have values entered, you can Search or click **Add Additional Criteria** to add more search conditions. Add the new search conditions just as you created the first set, making sure to click **Add Additional Criteria** after choosing the last set to add it to the criteria list. To delete a set of criteria, select it and click **Delete**.



Once you are satisfied with all the search criteria, click **Search**. The Design Browser or the Manufacturing Browser appears containing the results of the search.



You can return to the Search dialog box by clicking the **Search** button on the browser toolbar.

Servers and resources

A fundamental concept of DataCenter is the idea of resources and servers. Each *resource* tracks all the files in one directory. To track the files in a directory in the database, a resource needs to be created. Resources will automatically be created for files placed in any subdirectory of a master resource. The directory for each resource is defined relative to locations called *servers*. A server is typically a hard drive.

For the beginner, Esko recommends setting up one master resource per server. Master resources make resources for each subdirectory created inside them after they are designated as master resources. This ensures that all designs are tracked in the database. If you set up multiple regular resources and have no master resource, and someone creates a new directory but does not create a resource to go along with that directory, any work saved in that directory will not be tracked.

When you load ArtiosCAD for the first time, a database server is created for the drive ArtiosCAD is loaded on, and a master resource is created for the default file-saving location (e.g. **\Esko\Artios\Designs**). In this manner, all work that you save in the default directories is tracked in the database. Any subdirectories made under the master resource will have resources created automatically, so the work saved in those directories will be tracked in the database. If you install ArtiosCAD yourself, the database server and resource are set to the drive and user directory you specified while loading.

Configuring DataCenter

When you start using DataCenter with ArtiosCAD, only the work saved under the **\Users** or **\Esko\Artios\Designs** directory is saved into the database by default. If you create new directories on drives other than where you installed ArtiosCAD, and want DataCenter to track the work you save in those directories, you have to create a database server for that drive in DataCenter Admin as well as resources for those directories.

All configuration is done in the DataCenter Admin program in the ArtiosCAD program folder.

Working with servers and resources

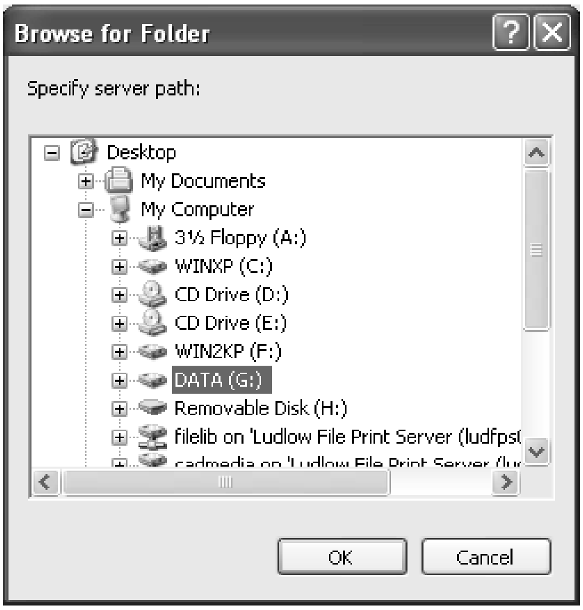
As explained earlier in this chapter, servers and resources are the places on your computer where you store work done in ArtiosCAD.

You must create database servers before configuring resources.

Creating a server

To create a database server, do the following:

1. Share the drive or directory you wish to be a database server.
2. Start DataCenter Admin.
3. Double-click the **Server** browser entry to open it.
4.  Click **Insert New Record** on the toolbar.
5. Enter the name of the new server in the **Name**: field.
6. Click in the **Path**: field and type the UNC locator for that drive, or click the **Browse** button to navigate to the proper directory. If you type the entry, make sure to enter it correctly as no error-checking is done for typed entries.



7. Choose the drive or directory to be the new server and click **OK**.
8. Click **Insert** to add the new server or click **Cancel** to cancel.
9. Exit the Server Browser, if desired, by closing its window.

Creating a resource

To create a resource, do the following:

1. Start DataCenter Admin.
2. Double-click the **Resource** browser entry to open it.
3.  Click **Insert New** Record on the toolbar.
4. Enter a name for the resource in the **Name** field.
5. Click in the **Server**: field and select the server for the new resource from the drop-down list.
6. Click in the **Directory**: field and either type the directory name or click the Browse button and select the directory for the new resource. If you type the entry, make sure to enter it correctly as no error-checking is performed for typed entries.

7. Once you have selected the desired directory, click **OK**.
8. To create the new resource, click **Insert**; to cancel its creation, click **Cancel**.
9. Exit the Resource Browser, if desired, by closing its window.

It is prudent to leave the **Master** checkbox checked when creating a resource. This ensures that all new work in any subdirectories that may be created will be tracked. The **Active** checkbox specifies whether DataCenter tracks the work saved in the resource. **Inherit Auto-Number** controls whether or not the resource will use Auto-Numbering if its parent resource uses it.

Deleting a server

You cannot delete servers containing resources. If a server has resources in it, you must either delete them or change the server they reference. Also, when you delete a server, only the information in the database is deleted. The actual designs are not deleted.

To delete a server, do the following:

1. Start DataCenter Admin.
2. Double-click the **Server** browser to open it.
3. Select the server to delete.
4.  Click Delete.
5. If you really want to delete the server, click **Yes** when asked.
6. The server entry will be deleted. If the server still contains resources, you will be told that servers cannot be deleted which contain resources. You will have to delete the resources and then try again.

Deleting a resource

When you delete a resource, the ArtiosCAD workspaces stored within them are also deleted. **Use this procedure with great caution.**

To delete a resource, do the following:

1. Start DataCenter Admin.
2. Double-click the **Resource** browser to open it.
3. Select the resource to delete.
4.  Click **Delete**.
5. Click **Yes** to delete the resource. If there are files in the resource, DataCenter Admin asks you to confirm the deletion. Click **Yes**.
6. The resource is deleted.

Userfields

Userfields are customizable fields for designs and manufacturing files that can contain almost any kind of information.

Userfields:

- Can contain text, integers, decimal numbers (called *floats*), distances, areas, volumes, or weights.
- Are specific to the type of workspace; design userfields cannot be used in manufacturing files, and vice-versa.
- Can be included on reports.
- Can have their values restricted to a list defined in the Restricted Userfield Set browser.

For designs, 36 userfields come pre-created in DataCenter. There are no pre-made userfields for manufacturing files.

When you make your own userfields, you can specify that they are required userfields. Required userfields are stored automatically in each workspace. For example, if you have a piece of information you want stored in every single design you make, you can add a required userfield via the Userfield browser in DataCenter Admin. The information contained in the userfield will be saved in each workspace; you cannot change the value of a required userfield in a workspace.

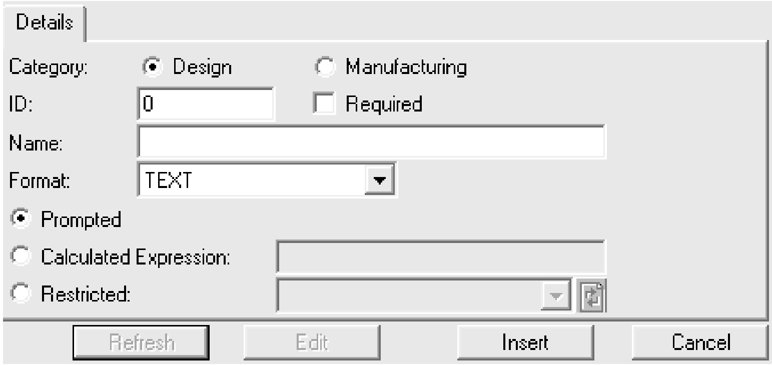
There are four steps to using userfields.

1. Tell DataCenter which userfields to track. You can immediately start using the pre-created userfields.
2. Configure restricted sets if desired.
3. Set values for specific userfields in designs or manufacturing files.
4. Set up formulae to automatically set values for userfields when they are added to designs or manufacturing files.

Adding userfields to DataCenter

To add userfields to DataCenter, do the following:

1. Start DataCenter Admin.
2. Open the Userfields folder if necessary and double-click the **Userfield** browser to open it.
3.  Click **Insert New Record**. The Details tab appears in the browser as shown below.



4. Choose the appropriate type of userfield in the **Category** field.
5. Enter an identification number for the userfield in the **Userfield ID** field. This can be any number.
6. To have this userfield present in every design or manufacturing file, check the **Required** checkbox.
7. Enter a name for the userfield in the **Name** field.
8. Choose the type of userfield from the **Format** drop-down list box.
9. Choose the way the userfield value is supplied. To enter a value for the userfield every time it is used, select the **Prompted** option button.

   To have it supplied by ArtiosCAD automatically, select **Calculated Expression** and enter the expression in the associated field. Use this method for setting a default value for the userfield - just enter the desired value in the field without using variables. If entering a text item, enclose it in quotes to prevent ArtiosCAD from treating it as variables.

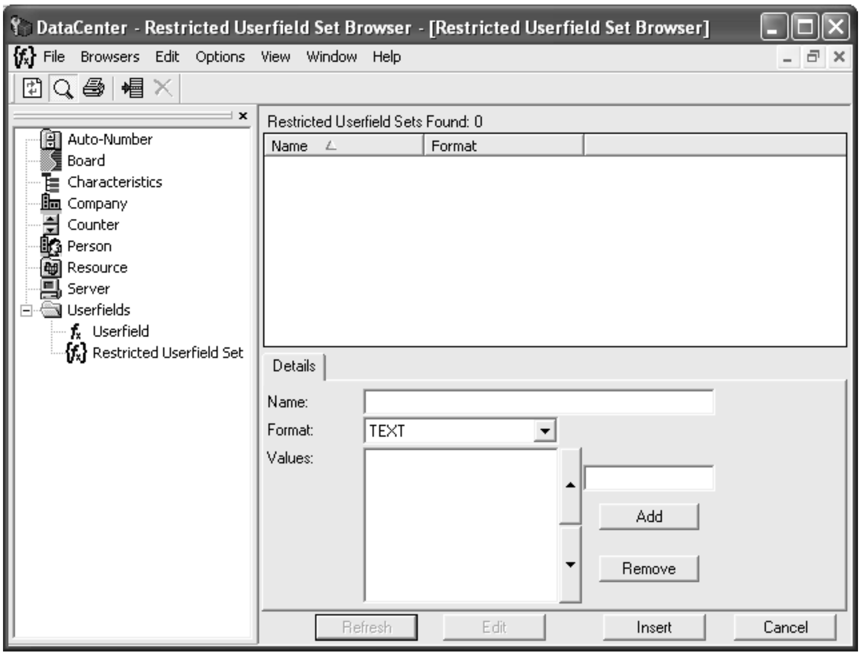
   To have it use a Restricted Set, select **Restricted Set** and choose the set from the drop-down list box. (You must have a restricted set already defined in the Restricted Userfield Set browser.)
10. Click **Insert** to insert the new record into the database.

Setting up a restricted userfield

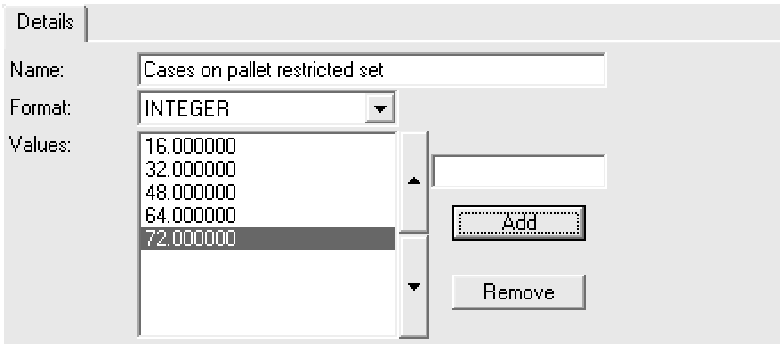
Restricted userfields are userfields that have a set range of values from which you can choose when setting them. These sets of values are configured in the Restricted Userfield Set browser. Sets of values can be used with as many userfields as desired.

To configure a restricted set, do the following:

1. Start DataCenter Admin.
2. Open the Userfields folder if necessary and double-click the **Restricted Userfield Set** browser to open it. It contains no data by default, so when it is opened for the first time, it is as if **Insert New Record** was clicked.



3.  If there are already restricted sets configured, click **Insert New Record**.
4. Enter a name for the set in the **Name** field.
5. Set its format using the **Format** drop-down list box. Make sure it is the same format as its corresponding userfield(s); for example, sets of type INTEGER cannot be used with userfields of type FLOAT.
6. Enter the first value for the set in the field above **Add**. It will appear in the **Values** list. Continue adding values by entering them in the field and clicking **Add**. To reorder them, select a value and use the directional arrows to change its position in the list.



7. When done entering values, click **Insert** to insert the record into the database.

Deleting userfields from DataCenter

To delete userfields, do the following:

1. Start DataCenter Admin.
2. Open the Userfields folder if necessary and double-click the **Userfield** browser to open it.
3. Select the userfield to delete.
4.  Click **Delete**.

5. You are asked to confirm the deletion of the userfield. Click **OK** if you really want to delete it; otherwise, click **Cancel**.

Configuring companies

The **Company** browser manipulates information about companies. No companies are defined by default.

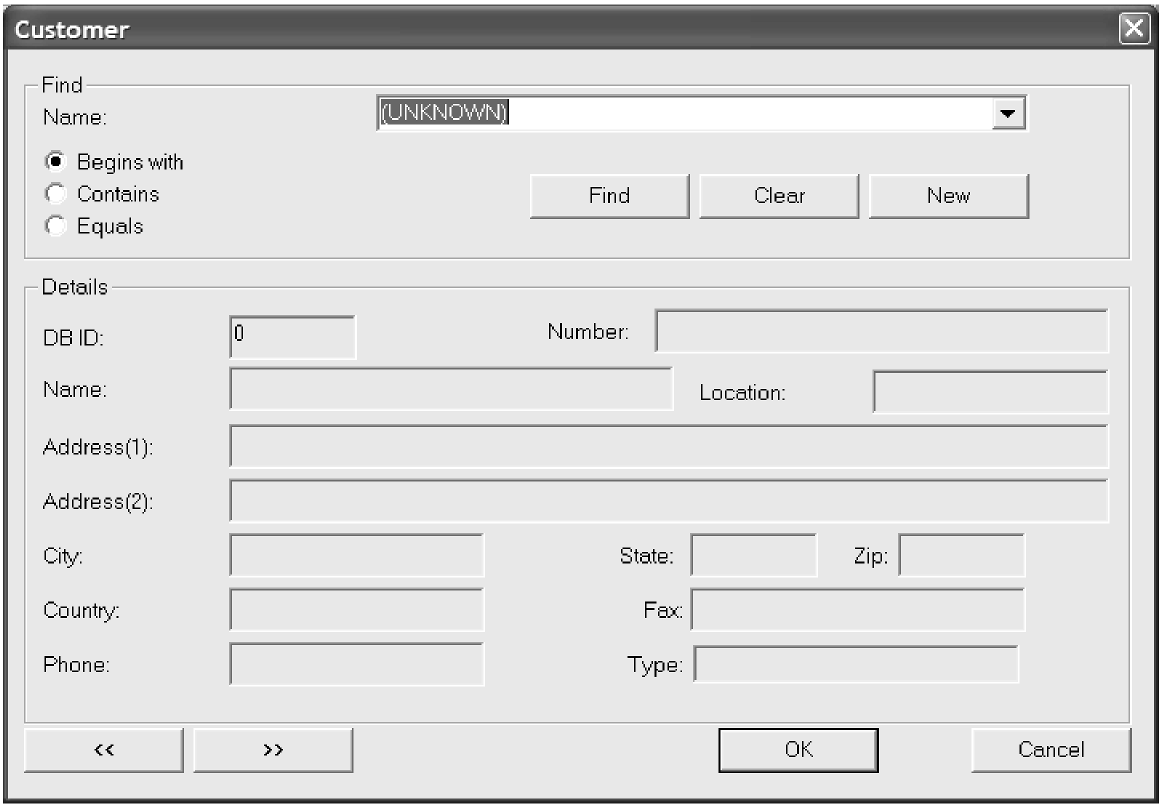
Adding a company

To add a company, do the following:

1. Start DataCenter Admin.
2. Double-click the **Company** browser to open it.
3.  Click **Insert New Record** on the toolbar.
4. Enter the information for the company in the appropriate fields. The **Number**: field is optional and is text only; it is not calculated.
5. Click **Insert**.

To add more companies, repeat steps 3 through 5.

You may also add a company when saving a design by clicking **New** in the Customer Browse dialog box.



Deleting a company

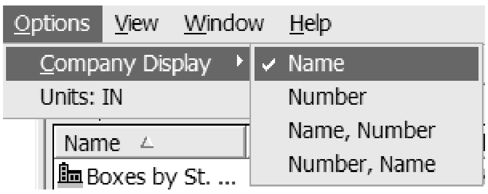
To delete a company, do the following:

1. Start DataCenter Admin.
2. Double-click the **Company** browser to open it.
3. Select the company to delete.
4.  Click **Delete**.
5. Click **Yes** to confirm the deletion.
6. If the company name is used in any design, you are prompted to **Replace** it with another, **Clear** it from the affected record(s), or **Cancel** the operation. If you are deleting more than one company at once, you can also replace all the companies with one by choosing **Replace All**.

To delete more companies, repeat steps 3 through 5.

Changing how company identifiers are shown

On the **Options** menu, you can choose different ways for how company identifiers are shown in DataCenter Admin.



Choose **Name**, **Number**, **Name, Number**, or **Number, Name** as desired.

Configuring company types

The **Company Type** browser lets you add and remove company types. Company types are used in company definitions. Examples of company types are **Converter**, **Diemaker**, and **End user**.

Adding a company type

To add a company type, do the following:

1. Start DataCenter Admin.
2. Open the **Company Type** browser by double-clicking it.

3. Click **Insert New Record**.
4. Enter a unique code and a description for the company type in the appropriate fields.
5. Click **Insert**.

Deleting a company type

To delete a company type from the database, do the following:

1. Start DataCenter Admin.
2. Open the **Company Type** browser by double-clicking it.
3. Select the company type to delete.
4. Click **Delete**.
5. Click **Yes** to confirm the deletion.
6. If the company type is in use, you are prompted to **Replace** it with another, **Clear** it from the affected record(s), or **Cancel** the operation. If you are deleting more than one company type at once, you can also replace all the company types with one by choosing **Replace All**.

To delete more company types from the database, repeat steps 3 through 5.

Configuring people

DataCenter tracks salespeople and designers as well as design information. Information about people is stored in the **Person** browser.

Adding a person

To add a person's information to DataCenter, do the following:

1. Start DataCenter Admin.
2. Open the **Person** browser by double-clicking it.
3. Click **Insert New Record**.
4. Enter the information about that person in the appropriate fields.
5. Click **Insert**.

Deleting a person's information

To delete a person's information from the database, do the following:

1. Start DataCenter Admin.
2. Open the **Person** browser by double-clicking it.
3. Select the person whose information is to be deleted.
4. Click **Delete**.
5. Click Yes to confirm the deletion.
6. If the person's information is used in any design, you are prompted to **Replace** it with another, **Clear** it from the affected record(s), or **Cancel** the operation. If you are deleting more than one

person's information at once, you can also replace all the people with one person by choosing **Replace All**.

To delete information about more people from the database, repeat steps 3 through 5.

Configuring board information

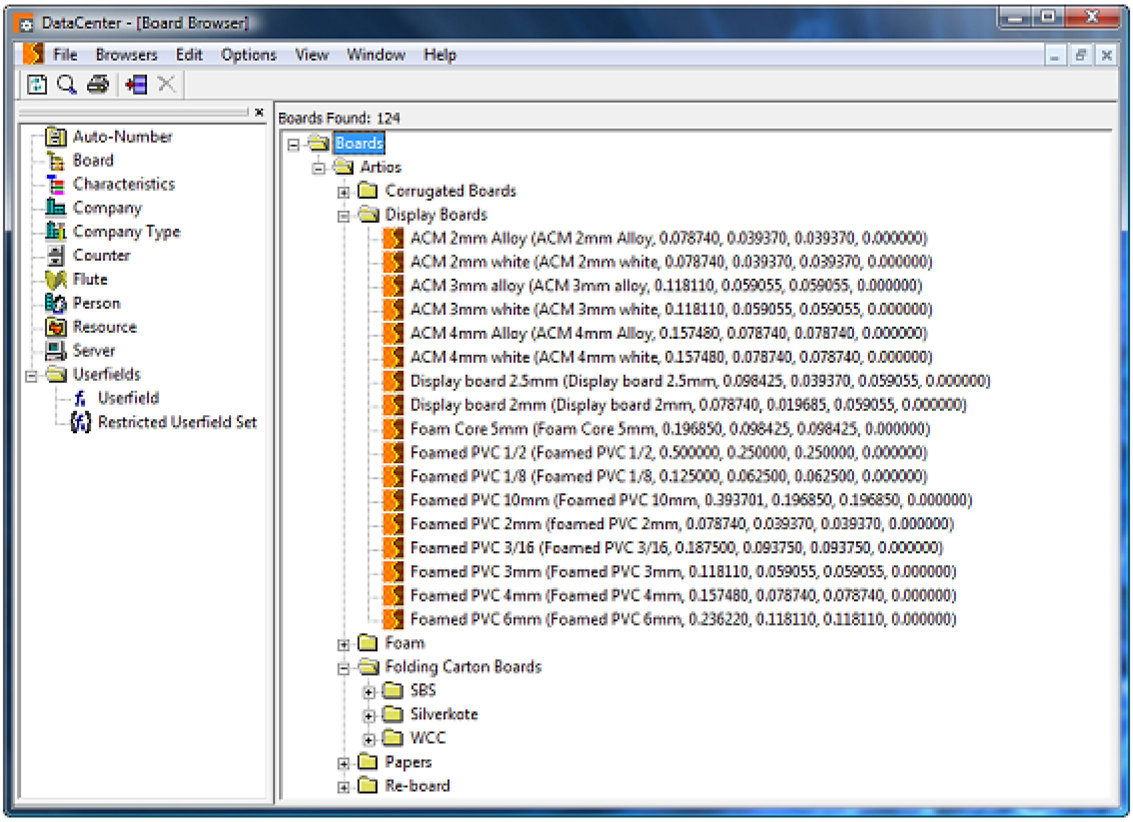
ArtiosCAD gets all information about boards from DataCenter, so it is important that the Flute and Board browsers contain accurate information. Papers and flutes must be defined prior to trying to use them to define corrugated boards.

Boards are **hierarchical**; that is, they may be organized in a hierarchy of folders for more effective organization and easier access.

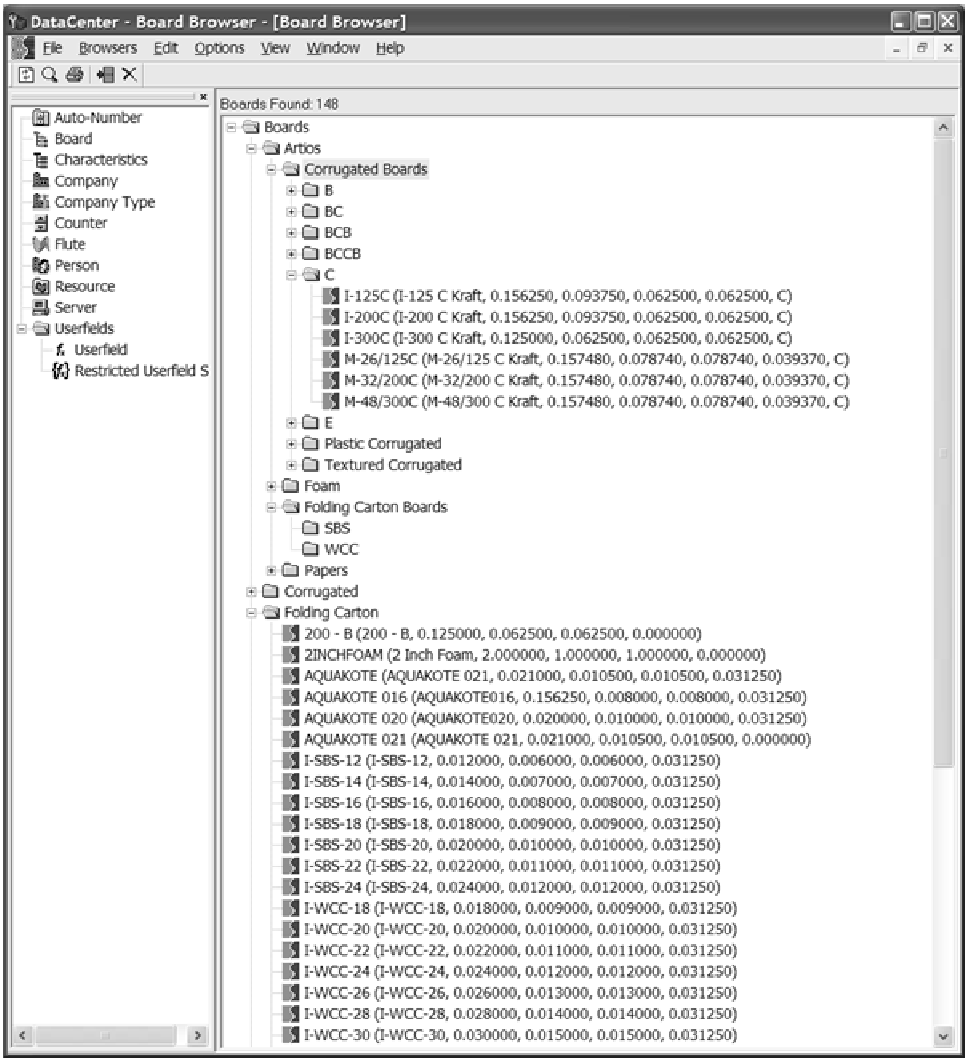
If this installation of ArtiosCAD is an upgrade from a previous version, the old board table is converted to the new hierarchical format during the upgrade, and new boards are added. New boards in the Artios folder hierarchy will not replace previously-existing boards with the same name and description; those will remain in the top-level Corrugated and Folding Carton folders.

AutoLoad does not rebuild 3D or paper information for boards. In the event of a database malfunction, this information can come either from a database backup that is restored through DataCenter Admin, or by using the ArtiosCAD Database Export and Import utilities to export and import the board table.

Your board browser may look different depending on if you loaded ArtiosCAD as a new installation or if you upgraded a previous installation. Shown below is an example Board browser for a new installation:



Shown below is an example Board browser for an upgraded system.



Corrugated board is built from wavy (corrugated) paper glued to flat paper. The flat paper is called a **liner**, and the wavy paper is called a **medium**. The medium is defined in terms of a **flute**, a letter code generally from A to F, with A being the thickest/largest and F being the smallest/thinnest.

Folding carton board is one layer of material, usually made from paper that is Solid Bleached Sulfate or White Clay Coated.

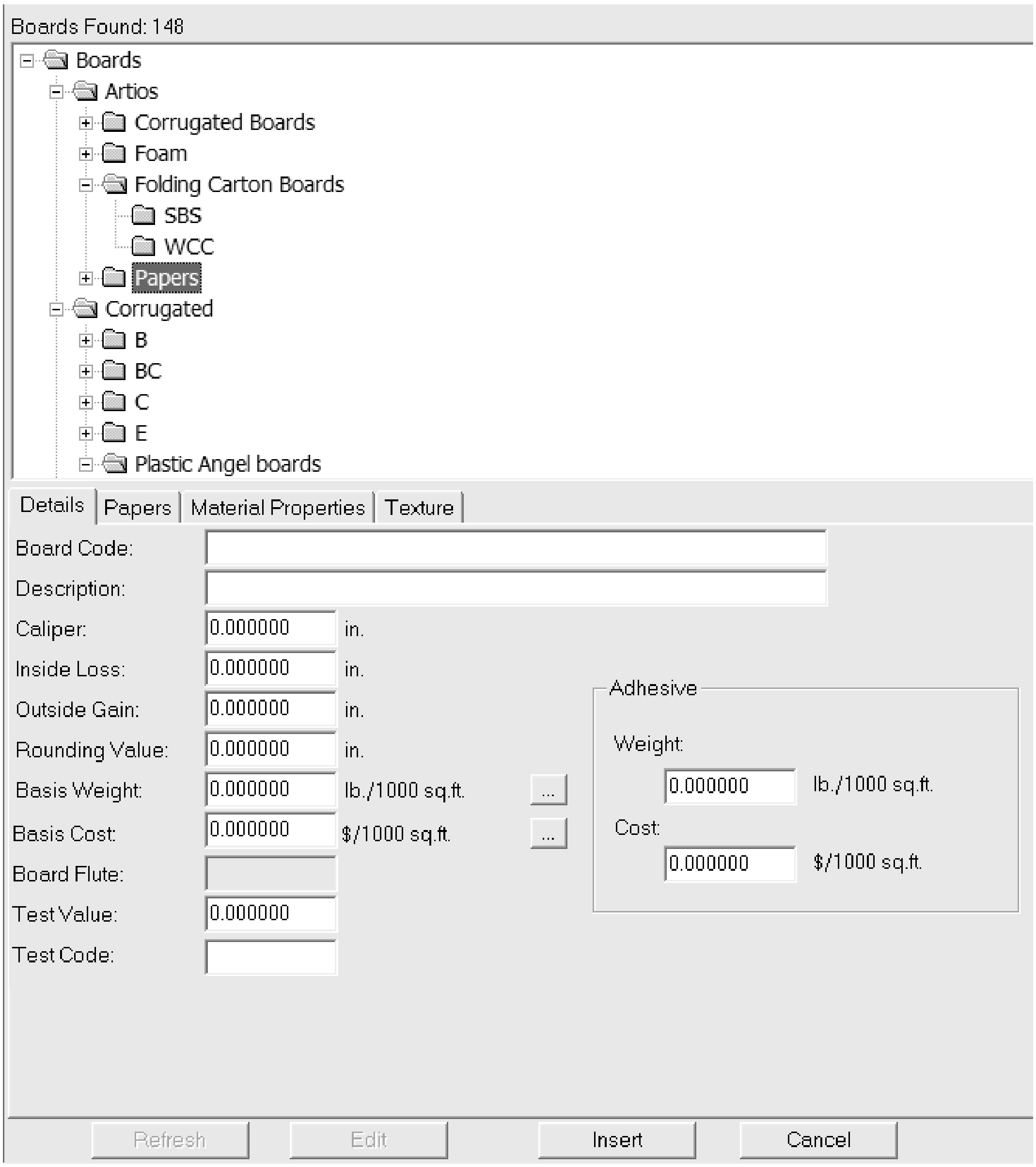
Because texture and paper information is stored in the database, it is important to back up the database as board information changes, because complete board information cannot be recreated only by using AutoLoad. For more information about backing up an MSDE or SQL Server 2005 Express Edition database, see the *Other Tasks* chapter of the *ArtiosCAD Administrator Guide*.

Creating a paper

To create a new paper, first examine an existing paper to determine what information you will need to define the new paper. 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. Start DataCenter Admin.
2. Open the Boards browser, and then open the Artios catalog by clicking the plus sign (**+**) next to it.
3. Click the word **Papers** to select it.
4. Click **Insert New Record**, or right-click and click **New** > **Board**. The Details pane will automatically appear with the fields enabled for editing.



5. Enter values in the fields on the Details tab as appropriate.

   Board Code: is the identifier used by the database to identify this new paper; each paper must have a unique code.

   Description: is the description of the paper.

   Caliper: is the thickness of the paper.

   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.

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.

Basis Weight: is the weight of the paper in local currency and units of measurement.

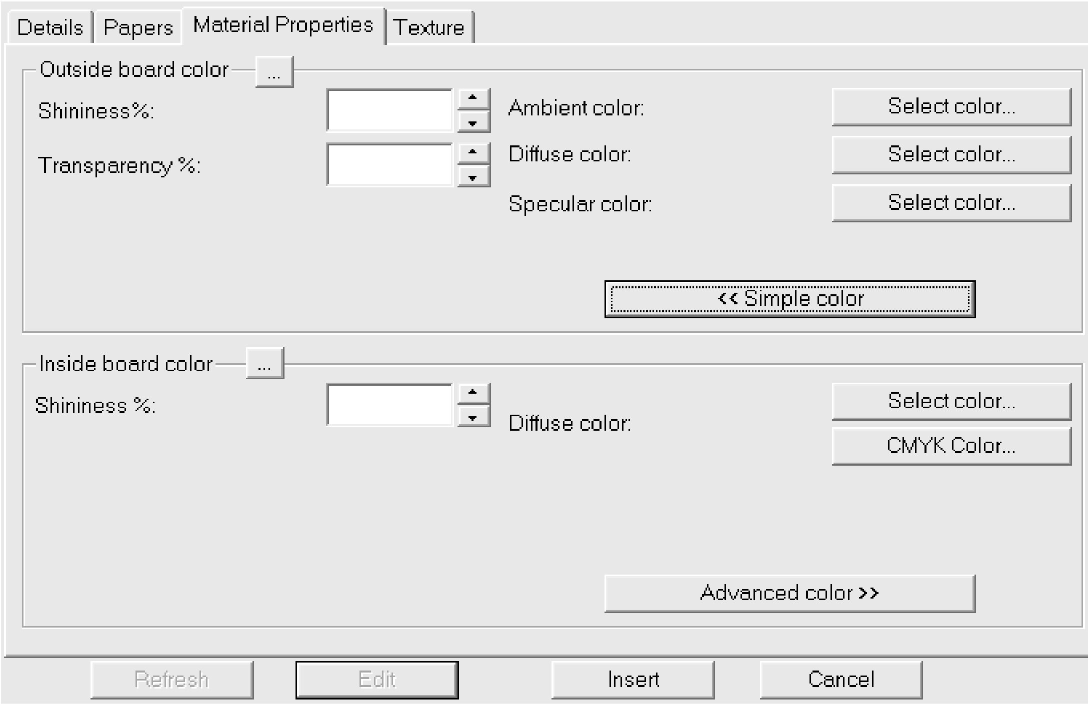
Basis Cost: is the cost of the paper in local currency and units of measurement.

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.

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.

Weight: and **Cost:** in the **Adhesive** group should be set for the complete board definition, not individual paper and flute definitions.

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



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.

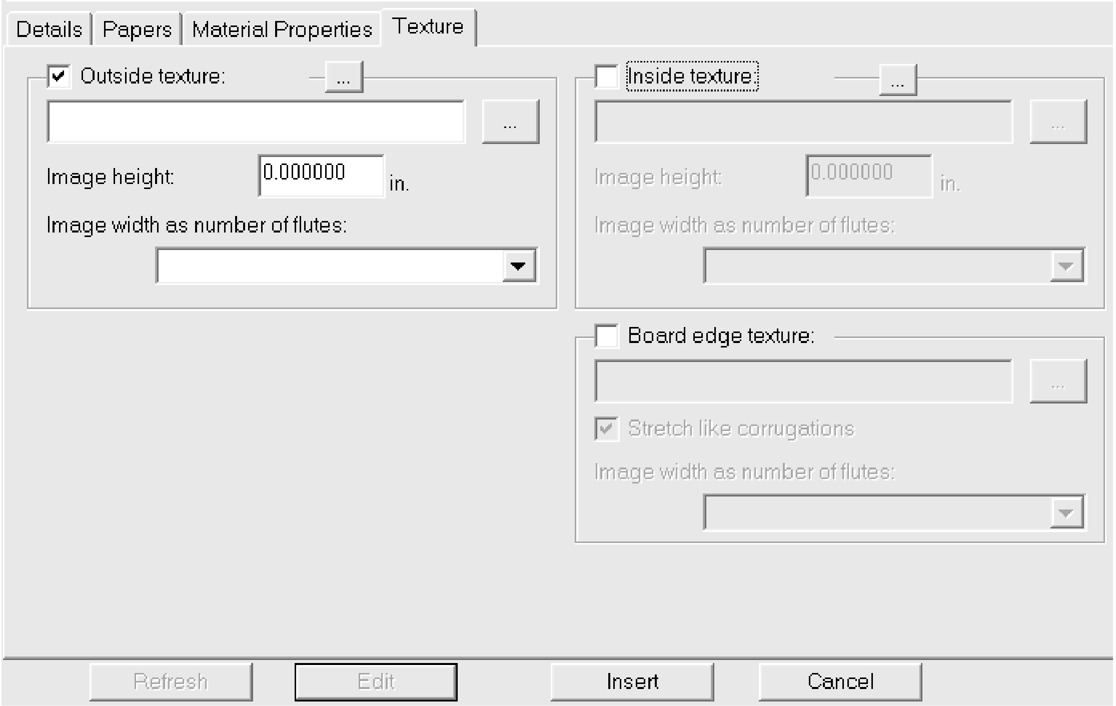
Surface colors for papers are split into three components: **ambient** color, which reflects the ambient light; **diffuse** color, which reflects the movable lights in an airy, non-shiny way; and **specular** color, which reflects movable lights for a shiny surface, with a shiny surface having a white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

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.

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.

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.

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 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.



To add an inside or outside texture to the paper, click the checkbox for it, and 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 are stored in **\Esko\Artios\Common**. 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.

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.

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.

Shown below is **\Esko\Artios\Common\boardbrown.jpg**, an example board texture three flutes wide:

9. When you have finished defining the paper, click **Insert** to add the new paper definition to the database.

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

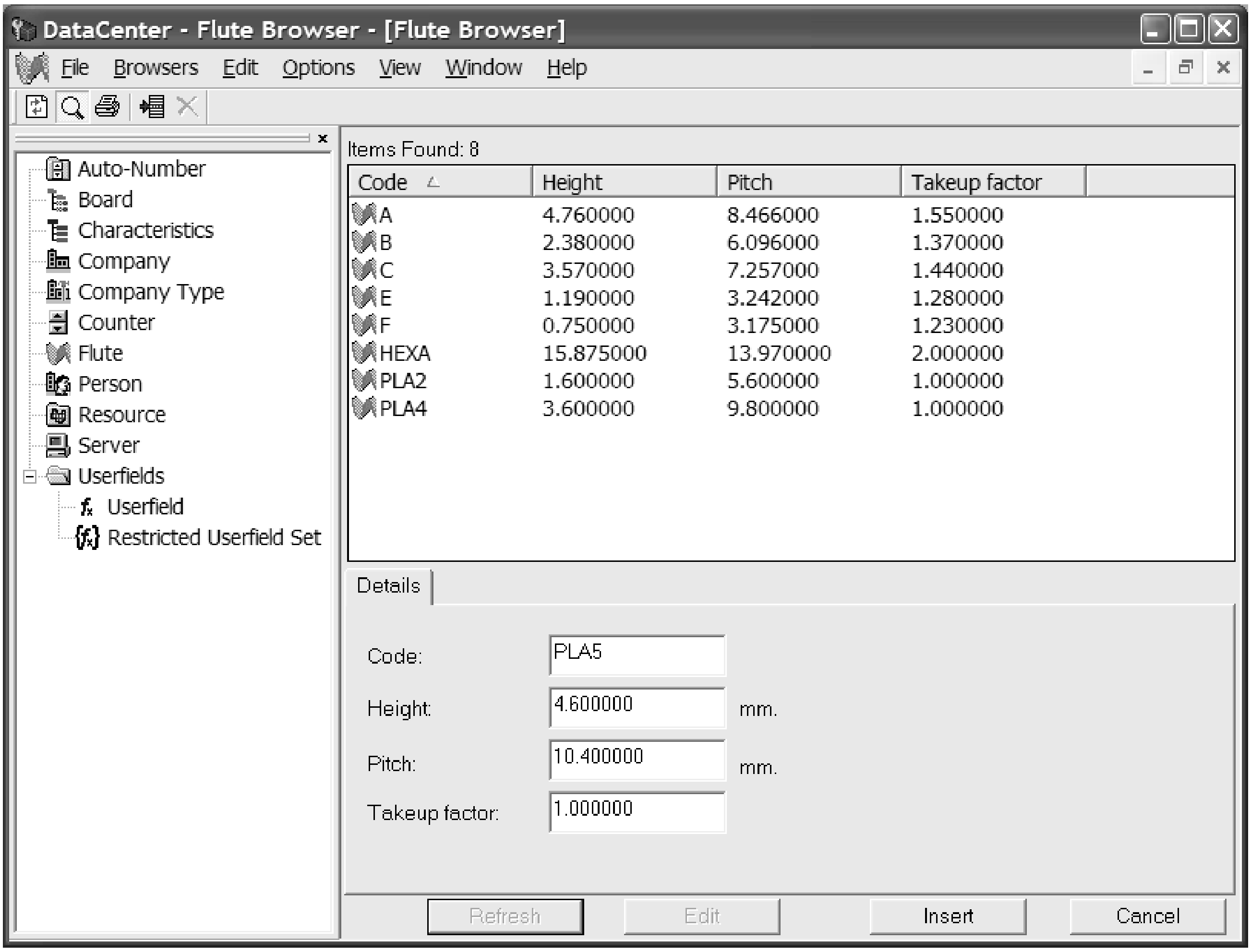
Creating a flute

A *flute* is the wavy paper that provides strength to corrugated board. A flute is defined by four pieces of information:

- A unique code to identify the flute. This code must be less than four characters long.
- The flute pitch, which is the 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 create a new flute, do the following:

1. Start DataCenter Admin.
2. Open the **Flute** browser by double-clicking it.
3.  Click **Insert New Record**. The Details pane will automatically appear with the fields enabled for editing.
4. Enter the new flute's code, height, pitch, and take-up factor in the appropriate fields as shown below.



5. Click **Insert** to add the new flute definition to the database.

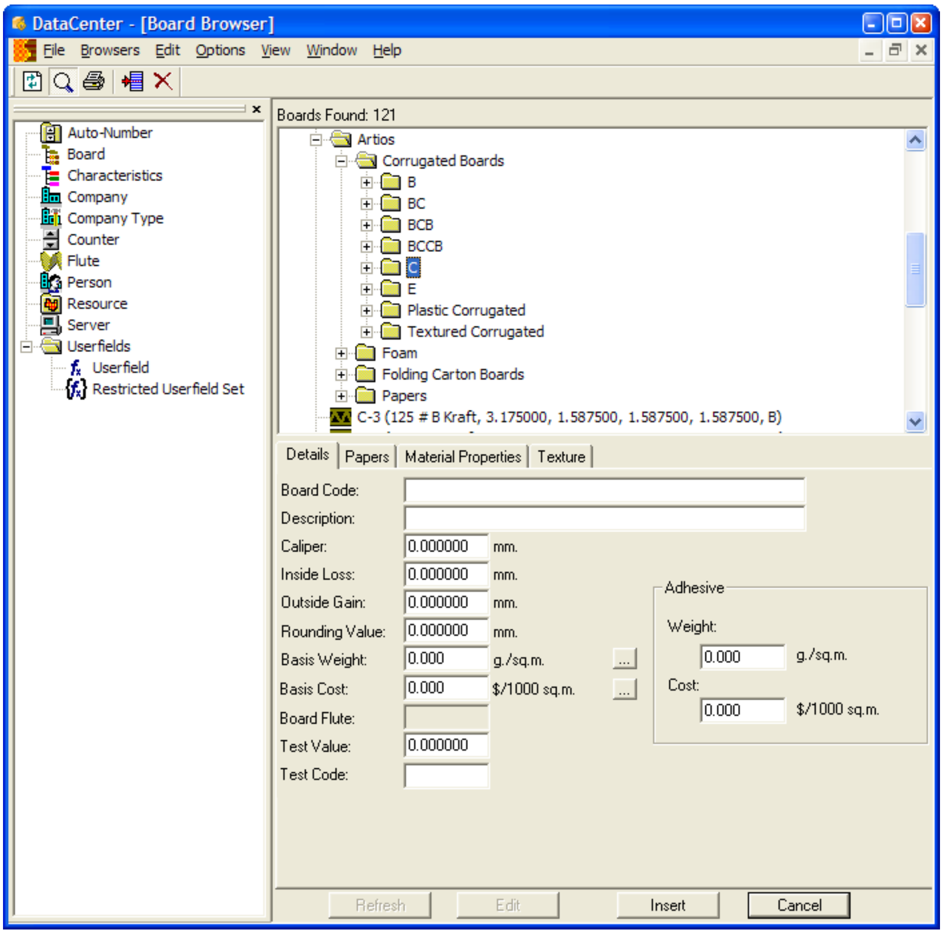
Creating a board

To create a board, do the following:

1. Start DataCenter Admin.
2. Open the **Board** browser by double-clicking it.
3. 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** > **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 New Record** 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.



4. Enter values in the fields on the Details tab as appropriate.

   Board Code: is the identifier used by the database to identify this new board; each board must have a unique code.

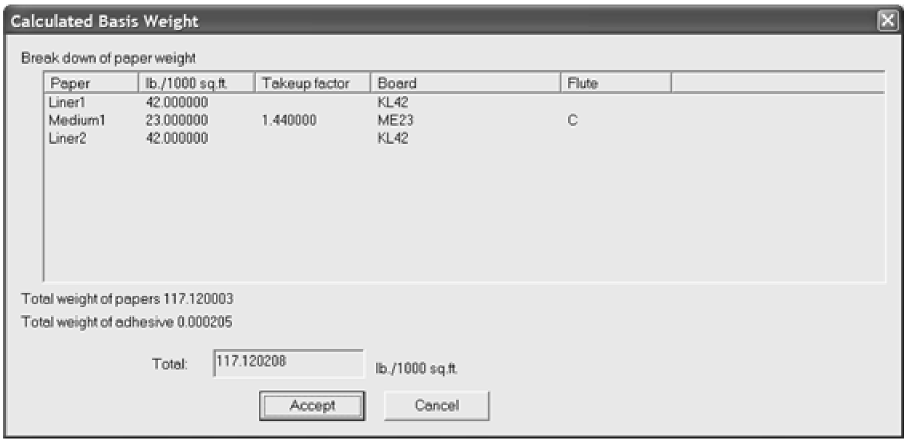
   Description: is the description of the board.

   Caliper: is the thickness of the board.

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

   Rounding Value: sets a value to which dimensions may be defined to round to when constructing a corrugated design, such as the nearest 1/64” (0.015625). This is variable CRRV in Advanced StyleMaker and can be used in the **Round to**: field when defining a new variable.

   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**.



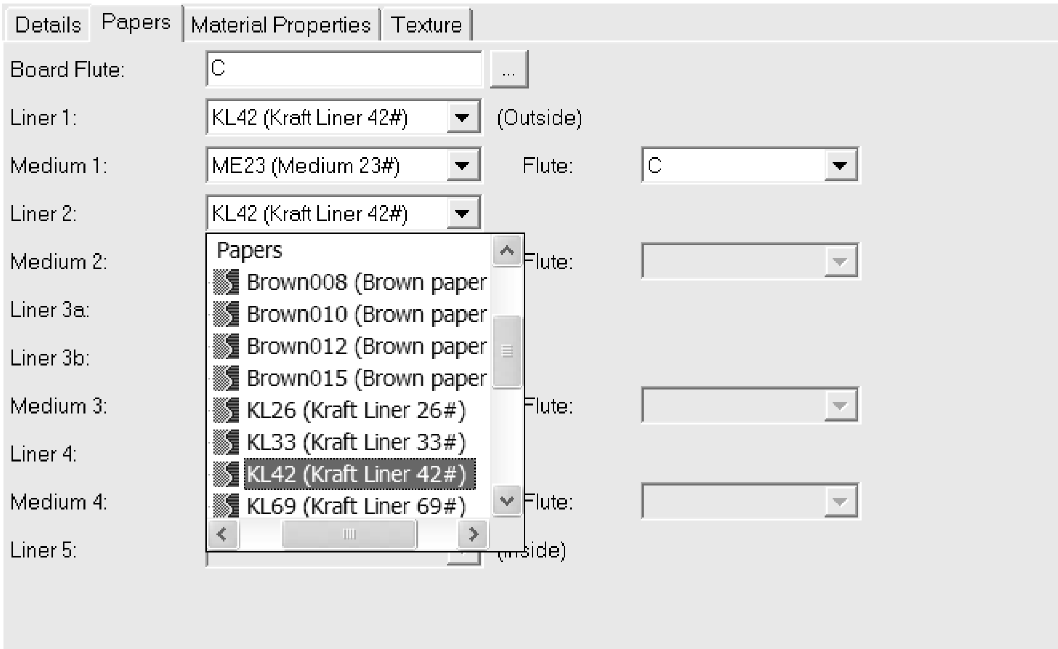
Basis Cost: is the cost of the board in local currency and units of measurement. As with the Basis **Weight:** field, the ... (Browse) button at the end of the field displays an analysis of the calculated cost of the board based on the papers used to make it. To replace a manually-entered cost with the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

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

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

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

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



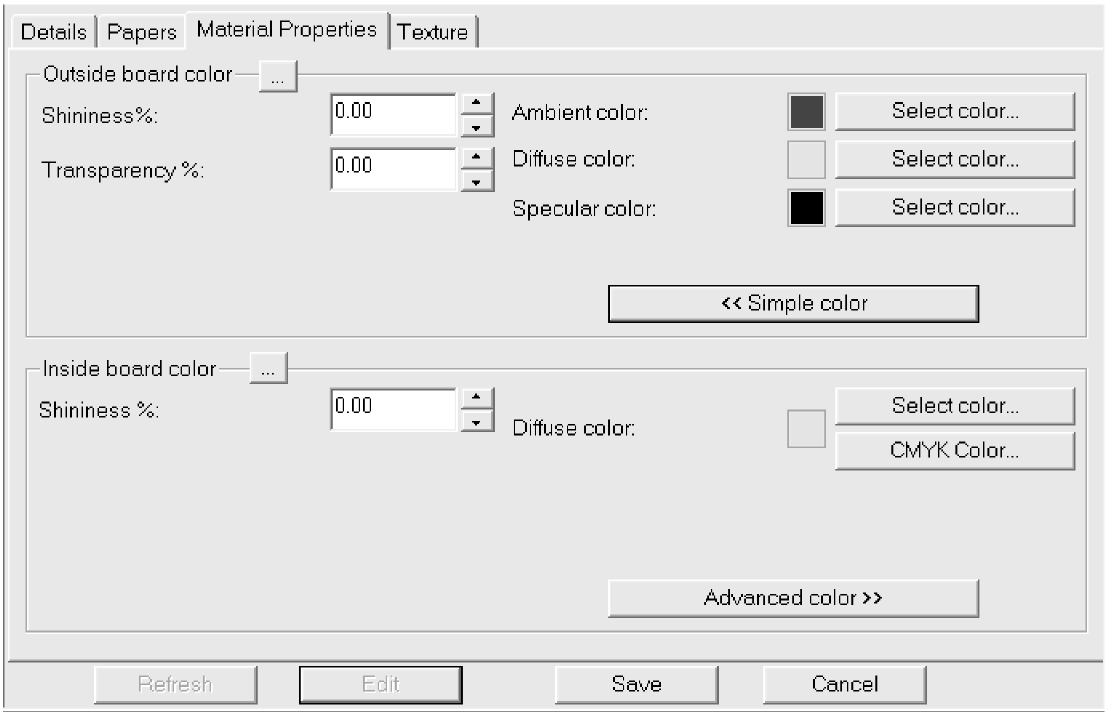
If you know it already, type the flute code of the new board in the **Board Flute:** field. At the end of the field is a **...** (Browse) button that opens the Calculated Flute dialog box. Use this to calculate the flute code after defining the liners and mediums for the board. If it is a single board,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

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.

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.

Complete the paper selection process with a liner.

6. 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.



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.

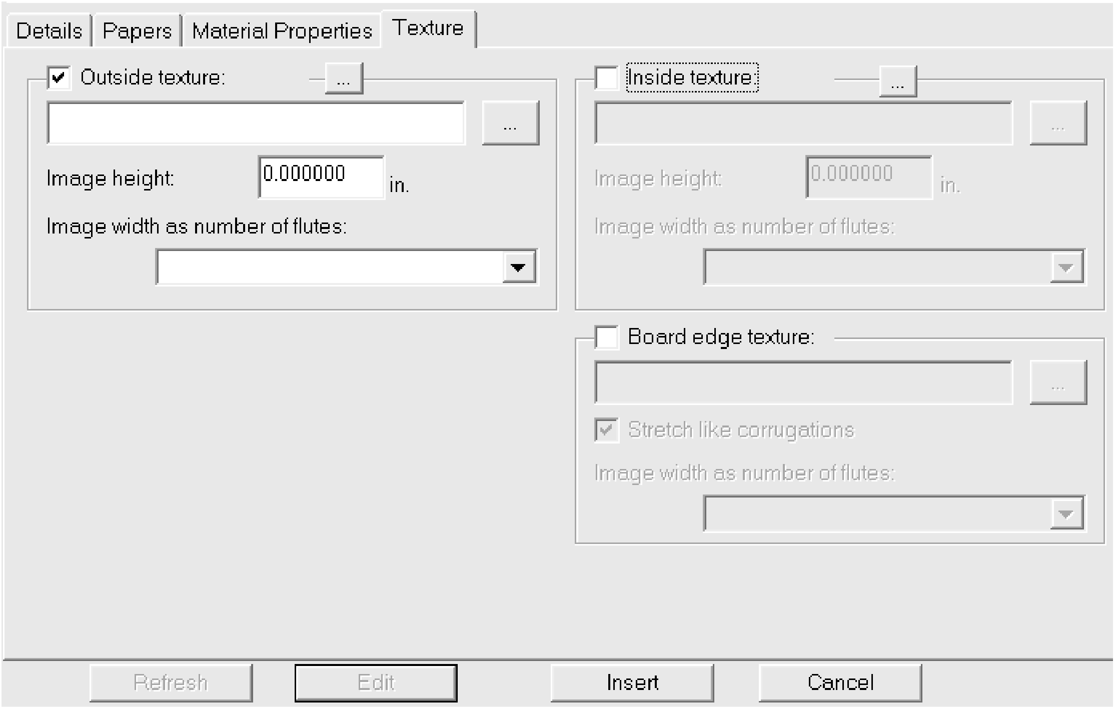
Surface colors for boards are split into three components: **ambient** color, which reflects the ambient light; **diffuse** color, which reflects the movable lights in an airy, non-shiny way; and **specular** color, which reflects movable lights for a shiny surface, with a shiny surface having a white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

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.

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.

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.

7. 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.



To add an inside or outside texture to the board, click the checkbox for it, and 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 are stored in **\Esko\Artios\Common**. The image should be of the board surface with vertical grain or corrugation direction.

In the **Image height**: field, enter the height of the image. You may have to use a separate graphic editing program to determine the size of the picture. 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.

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.

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.

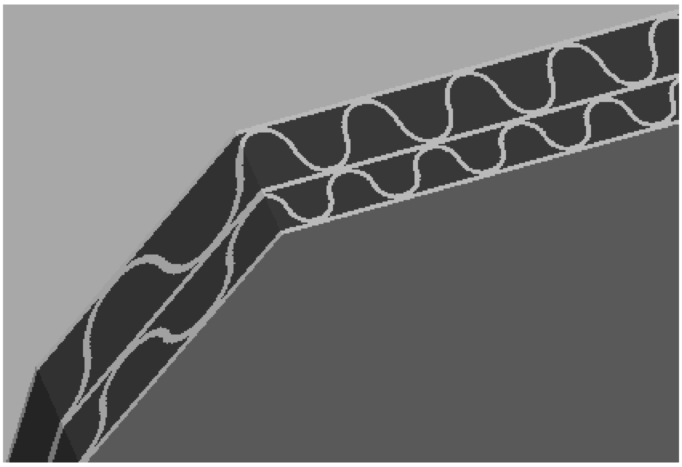
Shown below is **\Esko\Artios\Common\boardbrown.jpg**, an example board texture three flutes wide:

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 **\Esko\Artios\Common**. 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.

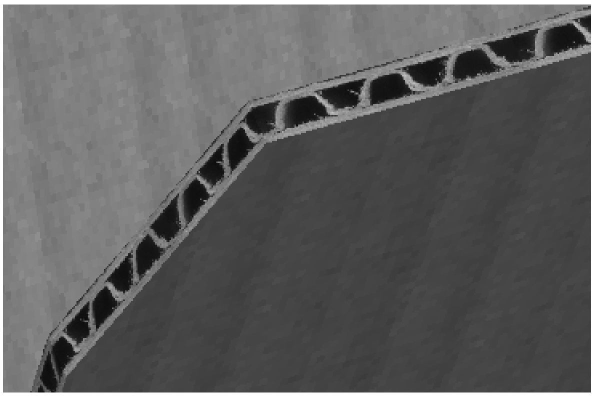
If no texture is specified for a corrugated board edge, a default picture based on the height and pitch of the flutes is used.

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.

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



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:



8. When you have finished defining the board, click **Insert** to add the new board definition to the database.

Renaming, deleting, moving, and copying board entries

In addition to the standard ways of manipulating entries in DataCenter Admin, 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 workspaces using this board.

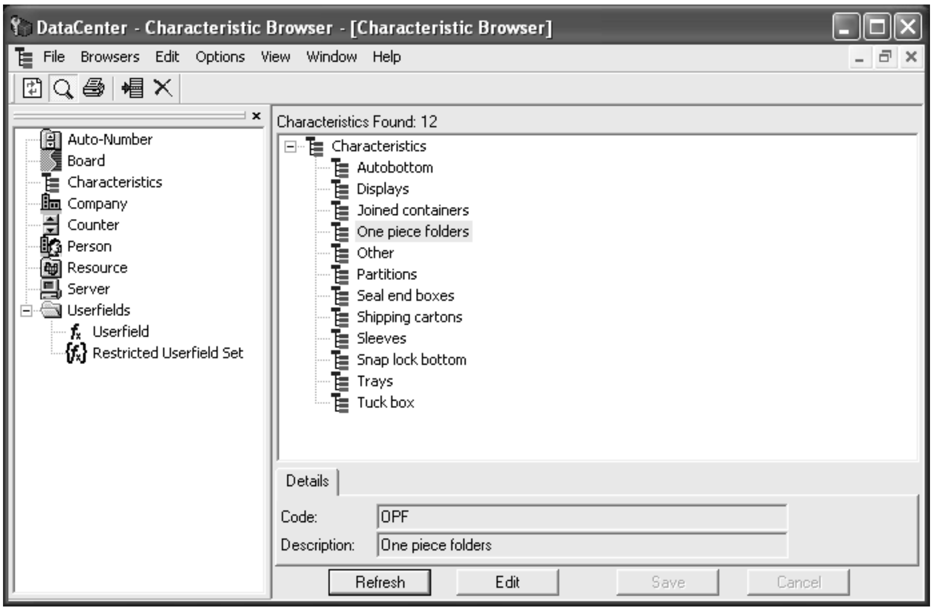
Boards may not be deleted if they are in use. You may replace the board definition in affected designs with another, clear the board information in those designs, or cancel the deletion.

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.

Configuring characteristics

Characteristics provide a way to describe the type of box referenced by an entry in the Design browser. Designs can have more than one characteristic, such as Autobottom and Tuck top. Characteristics are comprised of a **code**, which is the internal name of the characteristic, and a **description**, which is what is shown.



Adding a characteristic

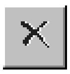
To add a characteristic, do the following:

1. Start DataCenter Admin.
2. Double-click the **Characteristics** browser to open it.
3.  Select the characteristic to add the new characteristic under and click **Insert New Record**.
4. Enter the code for the new characteristic in the **Code**: field.
5. Enter a description of the new characteristic in the **Description**: field.
6. Click **Insert**.

There are additional characteristics available beyond those shipped by default. See *Installing additional characteristics* later in this section.

Deleting a characteristic

To delete a characteristic, do the following:

1. Start DataCenter Admin.
2. Double-click the **Characteristics** browser to open it.
3.  Click Delete.
4. Click **OK** when asked to confirm the deletion. Deleting the characteristic from the database will also delete it from all files containing it.

Assigning characteristics is described in the *Using DataCenter* section.

Configuring Design Auto Numbering

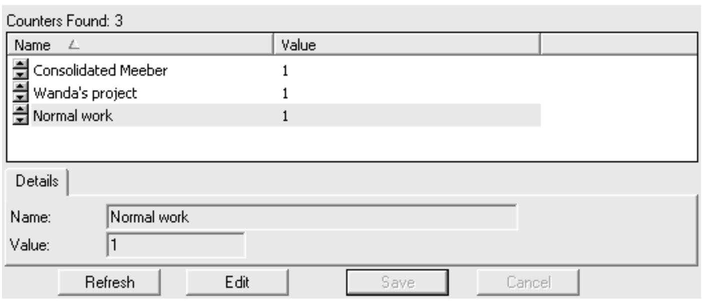
You can configure ArtiosCAD and DataCenter to use an automatic file numbering scheme for designs and manufacturing files. First you create a filename counter that will increment by one, and then you link that counter to a resource. Once that information is saved in DataCenter, when you use that resource in ArtiosCAD, you can use the **Save Next Name** button to automatically save the file with the next available number.

Each resource can have one counter for designs and one counter for manufacturing files. However, each counter can be used in more than one resource. The counter will span resources. For example, if two resources, **MBC Plant 1** and **MBC Plant 2**, use the same counter, auto-saving a file in **MBC Plant 1** will cause the next file auto-saved in **MBC Plant 2** to increment up a number from the name used in **MBC Plant 1**.

Configuring a filename counter

To set up a filename counter, do the following.

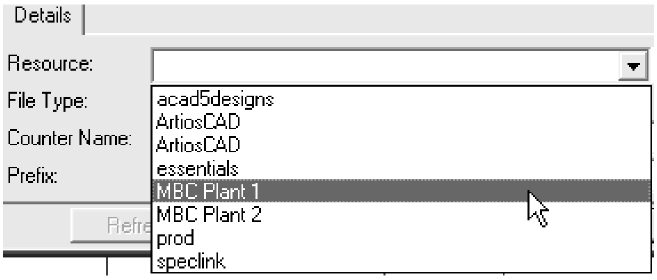
1. Start DataCenter Admin.
2. Double-click the **Counter browser** to open it.
3. Click **Insert New Record**.
4. Enter the name of the filename counter in the **Name:** field, and enter the starting value for the filename counter in the **Value:** field. The name should indicate the purpose of the counter.
5. Repeat steps 3 and 4 as desired.
6. Click **Insert**. When done, the Counter browser should look like the following picture (substituting the data you entered):



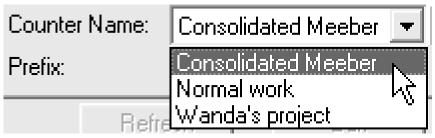
Configuring Auto Numbering

Once the filename counters have been added, do the following to set up automatic numbering:

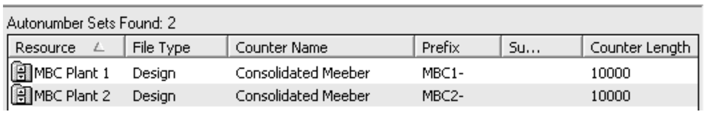
1. Start DataCenter Admin if it is not already open.
2. Double-click the **Auto-Number** browser to open it.
3. Click **Insert New Record**.
4. Click in the **Resource** field and choose the desired resource from the drop-down list box.



5. In the **File Type**: field, choose either the **Design** or **Manufacturing** option button according to the kind of file to use this auto-numbering.
6. Click in the **Counter Name** field and choose the desired counter from the drop-down list box.

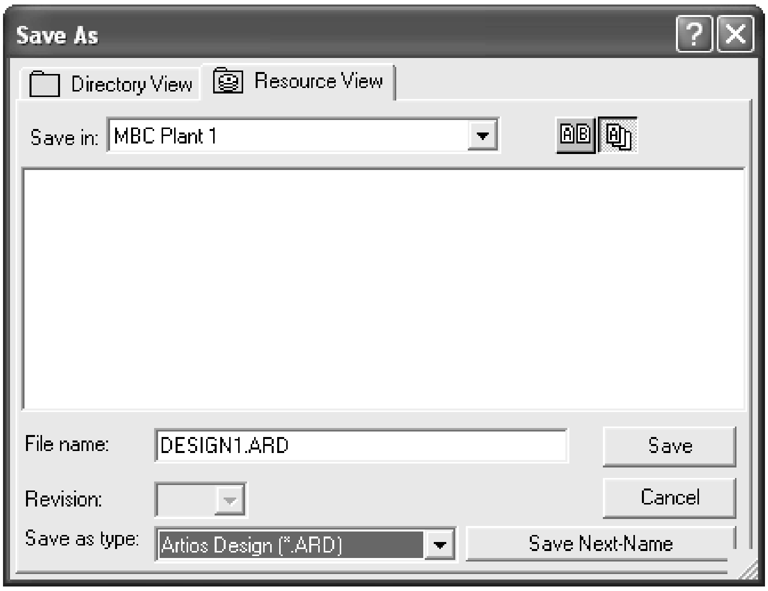


7. Enter the number of digits of the counter in the **Length** field. Extra zeroes will be added to the counter when a file is saved if the number of digits is smaller than specified here. If the counter is set to start at 1, and the Counter Length is set to 4, the counter will start at 0001, go up to 0002, 0003, and so forth.
8. In the **Prefix** field, enter non-changing text to come before the auto-incrementing number. You can leave this field empty if desired.
9. In the **Suffix** field, enter non-changing text to follow the counter. You can leave this field empty as desired.
10. Click **Insert**.
11. Repeat steps 3 through 10 as desired.
12. The Auto Numbering browser should look similar to the following picture (except that it will have the data you entered in it). Both resources share the same counter.


Autonumber Sets Found: 2

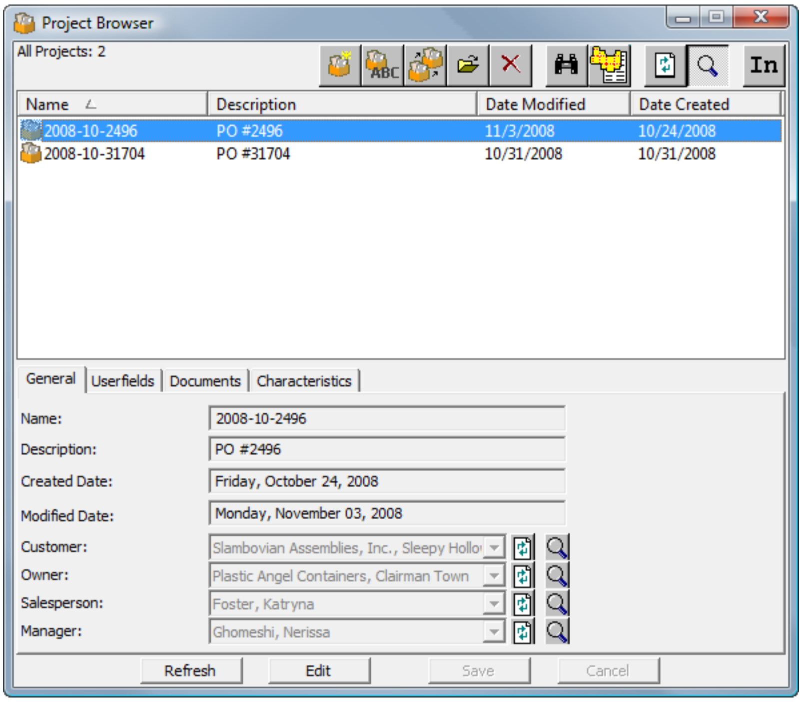
Resource	File Type	Counter Name	Prefix	Su...	Counter Length
MBC Plant 1	Design	Consolidated Meeber	MBC1-		10000
MBC Plant 2	Design	Consolidated Meeber	MBC2-		10000

Automatic design numbering is now fully configured and ready for use by clicking **Save Next Name** in ArtiosCAD.

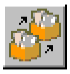
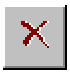
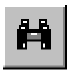
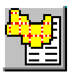


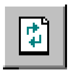
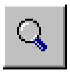
Using the Project browser

Along with single designs, manufacturing files, and single designs embedded in manufacturing files, there is a Project browser within ArtiosCAD that enables you to manage Projects separately from the other commands on the **Projects** menu. To open the Project browser, click **Projects** > **Project Browser**. Shown below is the Project browser with the **Show Details** toggle turned on and a Project selected.



Like the other database browsers, the first group of buttons along the top of the browser perform actions on the currently-selected Project, while the panes along the bottom of the browser show information specific to that Project. The panes are only shown when **Show Details** mode is turned on by clicking the magnifying glass icon on the toolbar.

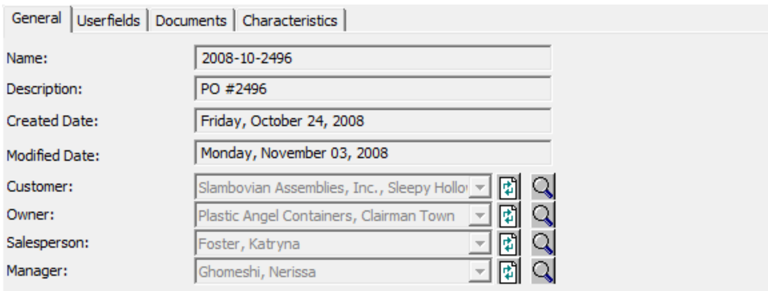
Button	Name	Function
	**New Project**	Creates a new Project just as if you clicked the menu command.
	**Rename Project**	Renames the currently-selected Project as long as it is not open in ArtiosCAD.
	**Copy Project**	Copies the currently-selected Project to a new Project using a dialog box similar to the **Create New Project** dialog box. The **Reset Project document counter** checkbox determines if the counter (for auto-numbering purposes) is reset or not. ArtiosCAD copies Project information, characteristics, userfields, and Project documents to the new Project.
	**Open Project**	Opens the currently-selected Project in ArtiosCAD and makes the ArtiosCAD window active.
	**Delete Selected Item(s)**	Deletes the currently-selected item(s).
	**Edit Search Criteria**	Searches all Projects with criteria you specify. See the section on searching below.
	**Show Project Documents**	Lists all documents in this Project in the Design Browser.

Button	Name	Function
	**Refresh Browser**	Re-queries the database and updates the list of Projects.
	**Toggle - Show Details**	Turns the information panes at the bottom of the browser on and off.
In or mm	**Units**	Changes the units of measurement between Imperial and metric.

To change information shown in one of the panes, as in the other database browsers, click **Edit**, make the desired changes, and then click **Save**. You may need to click **Refresh** to see the updated information.

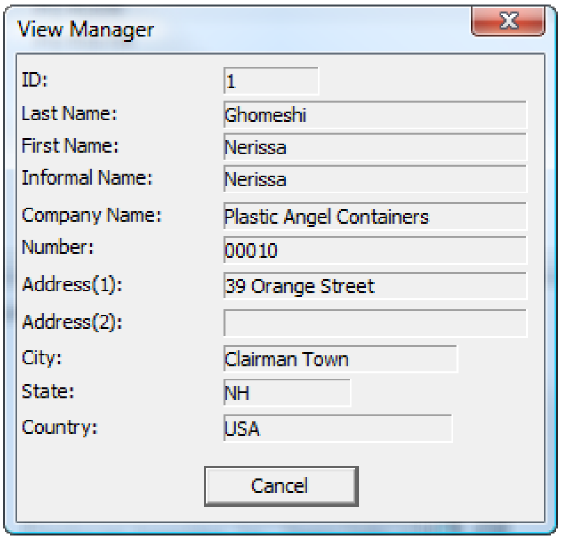
Project browser - General pane

The **General** pane of the Project browser shows basic information about the Project as shown below. All of the fields are in read-only mode; to change them, click **Edit** beneath the pane.



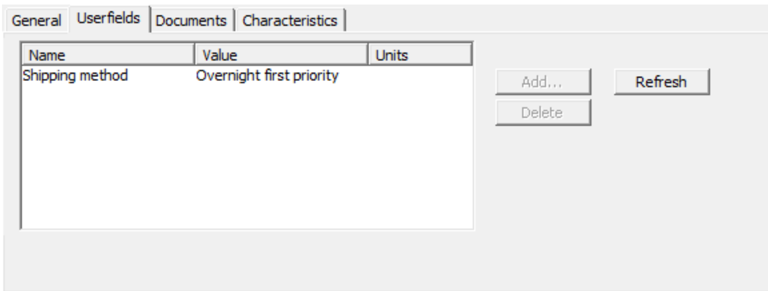
 **Refresh** requeries the database for the information in that field in case someone else changed it while you were looking at it.

 **Show Item Details** opens a dialog box which displays more information related to the associated field. Click **Cancel** to return to the Project browser.



Project browser - Userfields pane

The userfields assigned to a Project are shown in the **Userfields** pane of the Project browser. You may also add and remove userfields by using this pane as opposed to using the **Project Information** dialog box. Shown below is an example Userfields pane.

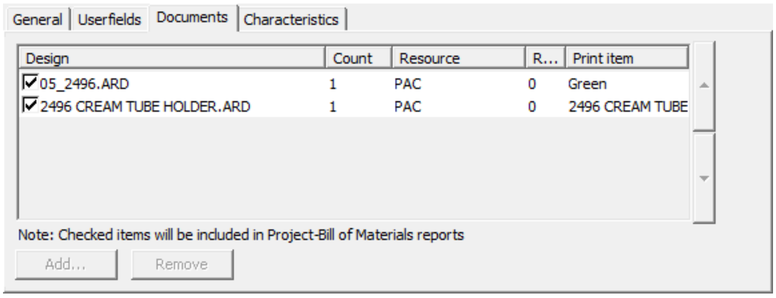


To add or remove userfields, click **Edit** beneath the pane, and then click **Add** or **Delete** (having first selected a userfield in the list shown) as desired. When you are done making changes to the userfields, click **Save** beneath the pane.

Refresh requeries the database and updates the list of userfields if anyone else has made changes to them since you opened this pane.

Project browser - Documents pane

The **Documents** pane of the Project browser shows the documents included in the Project and lets you organize how they appear on Bill of Materials Reports (BOM Reports).



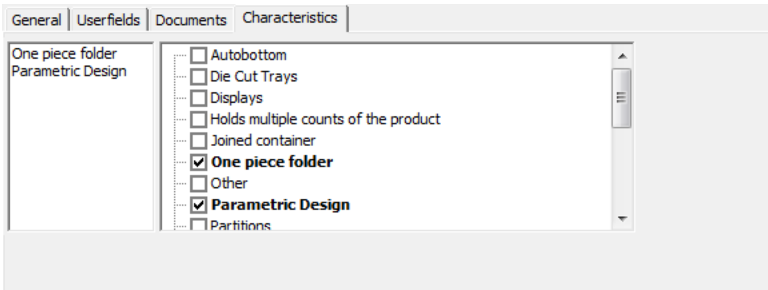
To add or remove documents from the Project, click **Edit** beneath the pane, and then click **Add** or **Remove** (having clicked to select at least one document beforehand) as desired. To include or exclude documents on BOM Reports, check or clear their checkboxes before their names in the list. To change the order in which documents appear on BOM Reports, select a document by clicking it and then click the up or down arrow at the right edge of the dialog box as appropriate. You may also change the Count as desired by clicking the up and down arrows next to the numeral in that field for a particular document.

For more information about BOM Reports, see the *Outputs* chapter.

When you are done making changes to the documents, click **Save** beneath the pane to return to the Project browser.

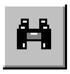
Project browser - Characteristics pane

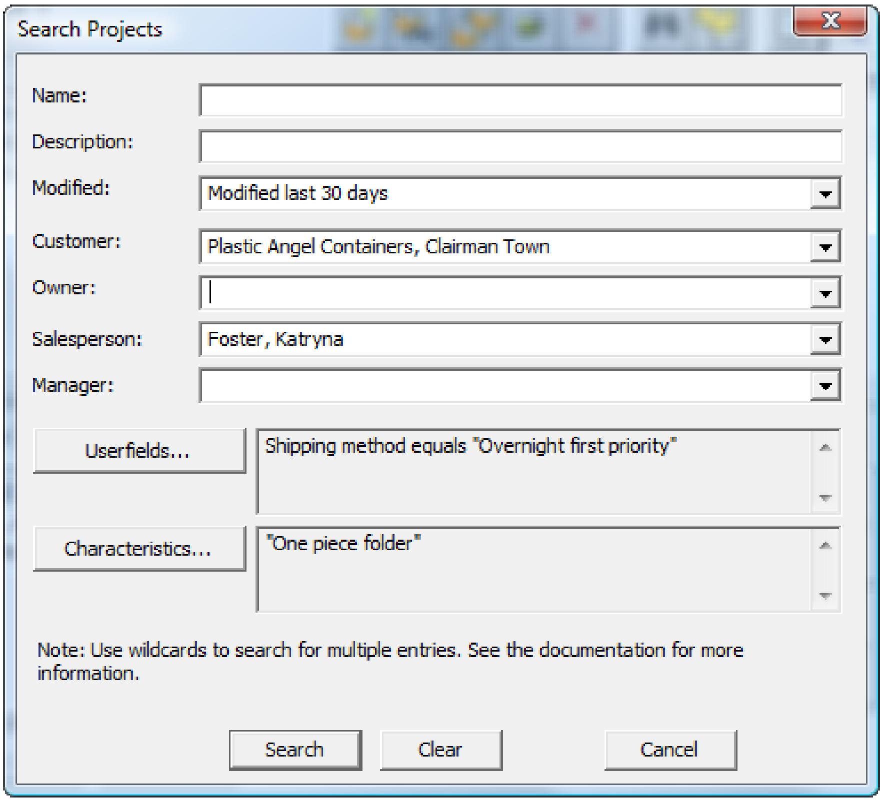
In the **Characteristics** pane of the Project browser, ArtiosCAD shows the characteristics currently assigned to the selected Project in the left side of the pane. You can also add and remove characteristics to and from the Project.



To add or remove characteristics, click **Edit** beneath the pane, and then select or deselect the desired characteristics. When you have finished modifying the characteristics for the selected Project, click **Save** beneath the pane to return to the Project browser.

Searching for Projects

 When you click **Edit Search Criteria**, the **Search Projects** dialog box appears as shown below (with sample data; initially it is empty).



You can enter text to search on in each text field, but the **Modified:**, **Customer:**, **Owner:**, **Salesperson:**, and **Manager:** fields all have drop-down list boxes containing cached entries from the database. All text that you enter must use wildcards as explained below:

- * and % mean match all characters.
- ? and _ mean match one character.
- To search on one of the wildcards, prefix it with a backslash (\ ), such as “100\% Pure”.
- To search specifically on a backslash, enter “\\”.

Examples of using wildcards are shown in the following table.

Search term	Search results
Project_1	All Projects starting with “Project” and then any single character between t and 1, such as ProjectX1, Project$1, and so forth.
Project_1	Project_1.
Project_*	All Projects starting with Project_, such as Project_1, Project_X, and Project_5BD2.
Project\1	Project\1.

In any of the fields with a drop-down list box, if you enter text to search on using wildcards, ArtiosCAD will:

- search by customer name for both the **Customer**: and **Owner**: fields.
- search by last/first/greeting names for the **Salesperson**: and **Manager**: fields. However, make sure to use wildcards if you manually modify an entry in these fields, as ArtiosCAD will not analyze the string to extract the first and last names separately. For example, if you select "Smith, John" from one of the drop-down list boxes and then add an asterisk after John, forming "Smith, John*" as the search string, ArtiosCAD will probably not find the desired result as it will not separate the words in the search string since you manually changed it. However, were you to search on "Smith*" or "John*" you would probably achieve the desired results.

To search on a userfield, do the following:

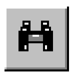
1. Click **Userfields**.
2. In the **Project Userfields** dialog box, select the userfield to search against.
3. Select the method to use for the pattern match in the **How**: field. For text userfields, use wildcards. For numeric userfields, use **=**, **<**, and **>**. For restricted set userfields, only **=** is available, and then the set of values becomes available in the **Value**: field.
4. Repeat steps 1-4 as desired to add more userfields to the search. Each userfield search argument is connected with AND logic, so only Projects meeting all the Userfield conditions will be returned in the search results.
5. Click **OK** to return to the **Search Projects** dialog box.

To search on Characteristics, do the following:

1. Click **Characteristics**.
2. In the **Characteristics** dialog box, select the desired characteristics to search against.
3. To find characteristics, click **Find** and perform the usual search for characteristics using the Characteristics Finder. Once you have found and selected the desired characteristics, click **OK** to return to the **Set Characteristics** dialog box.
4. Once you have set the characteristics to search against, click **OK** to return to the **Search Projects** dialog box.

In the **Search Projects** dialog box, click **Search** to search against the criteria you have entered.ArtiosCAD displays the results in the Project Browser.

Clicking **Clear** in the **Search Projects** dialog box clears the fields in the dialog box but keeps any search results.

To clear the search results and show the list of Projects again, click  (**Edit Search Criteria**), click **Clear**, and click **Search**.

Modifying information in browsers

To change information in browsers, simply select the record to change, open the Details tab, and click **Edit**.

For example, if you wanted to change the price of 200# B-flute board in inches, you would open the Board browser, double-click the entry for **I-200B** in **Artios** > **Corrugated Boards** > **B**, click **Edit**, change the information shown, and click **Save**.

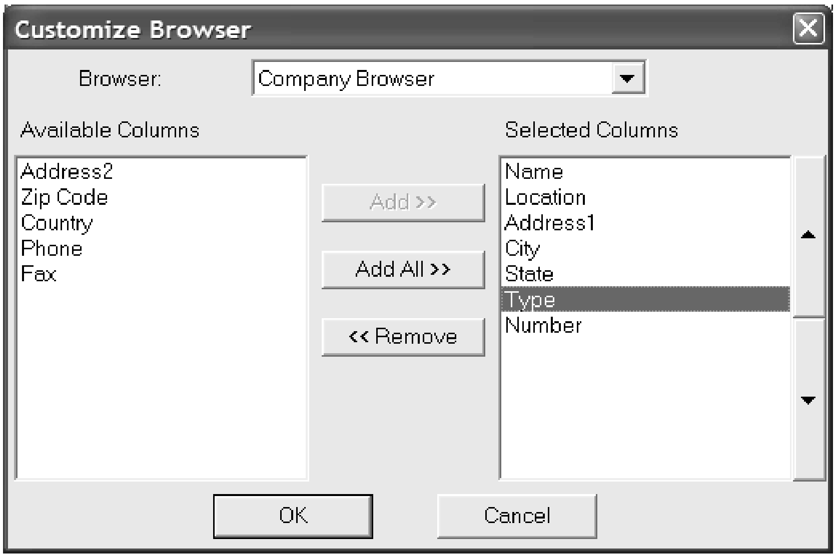
Refreshing fields in browsers

As you change information in the browsers, the information they have cached in memory may not be synchronized with the recent changes.

Click  **Refresh** to update the contents of the current field.

Customizing columns in DataCenter Admin

The **Customize** command on the Browsers menu in DataCenter Admin lets you choose the data shown in the browser windows and also output on Reports.



Choose the browser to modify from the **Browser**: drop-down list box.

Available columns which are not already shown in the browser are listed in the **Available Columns** list. The columns currently shown in the browser are list in the **Selected Columns** list. To move items between lists, select them and click either **Add**, **Add All**, or **Remove** as appropriate.

To change the order in which items are shown in the browser and on reports, select the item to move and then use the direction buttons. The top item is shown at the left of the browser and the bottom item is shown at the right of the browser.

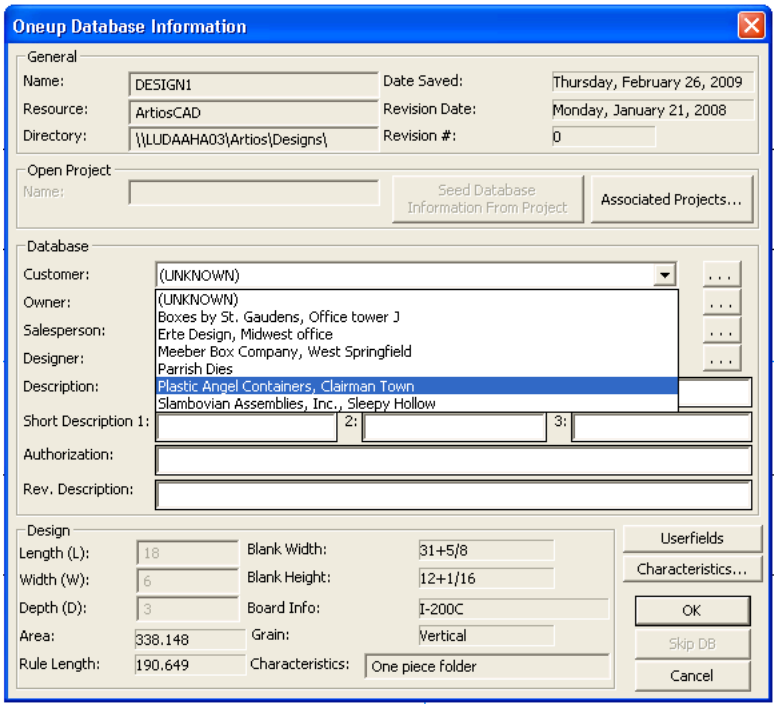
Using DataCenter

Once DataCenter is configured to match your unique needs, it is easy to use. Two situations make up the majority of times you use DataCenter: when saving designs or manufacturing files, and using the browsers to view information.

Entering database information when saving in ArtiosCAD

When you save a design, ArtiosCAD automatically supplies DataCenter with some information about the design, but you must specify other information such as the customer name and the designer. Specify the customer, owner, salesperson, and designer for the design by clicking the arrow at the end of the field and choosing an entry. When you are done specifying information, click **OK** to save the database information and the design. Standard userfields are automatically saved with each design or manufacturing file.

Shown below is the process of assigning customer information to a design using the default database dialog box.



You can enter anything you want in the Description and Authorization fields. The **Description** is limited to 80 characters. **Short Description 1** is shown under the design preview when opening the design and is limited to 20 characters. To add a user field, click **User Fields**. (You must have purchased the Information Enhancement module to access user fields.)

Choose a **customer**, **owner**, **salesperson**, and **designer** by selecting the appropriate entry from the drop-down list box for each field. You can also search for specific information by clicking **...** at the end of those fields. Your database administrator may have disabled the drop-down list boxes, in which case using the **...** button is the only way to enter information in those fields.

To skip adding this file to the database, click **Skip DB**. This button may be disabled in Defaults. Even when it is enabled, it is unavailable if a file with the same name already exists in the database to avoid having a conflict between the file data and the database data.

If the design is not rebuildable and does not already have these variables assigned, you can enter the length, width, and depth in the appropriate fields.

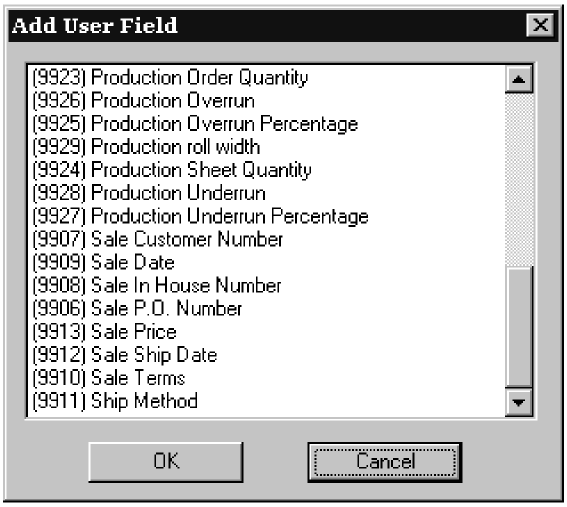
Your system administrator may have customized the database dialog box. For more information on customizing database dialog boxes, see the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

Setting values for userfields in designs or manufacturing files

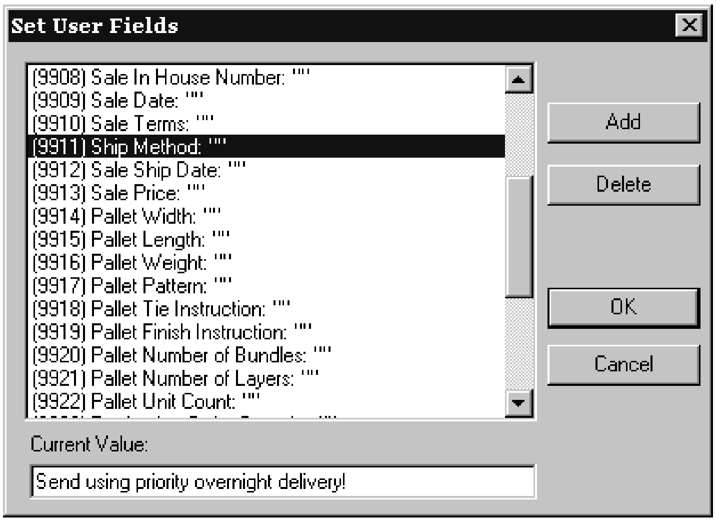
The easiest way to set values for userfields in ArtiosCAD is by clicking **User Fields** in the Database Information dialog box when saving a design or manufacturing file. You can also set them at any time in ArtiosCAD by clicking **Information** on the **Database** menu and then clicking **User Fields**. You will arrive at the same Set User Fields dialog box using either method.

To set a value for a userfield while in a design:

1. Use either of the methods described above to get to the Set User Fields dialog box.
2. Click **Add** to add a userfield and choose one from the list shown. Click **OK** after selecting the userfield to add.



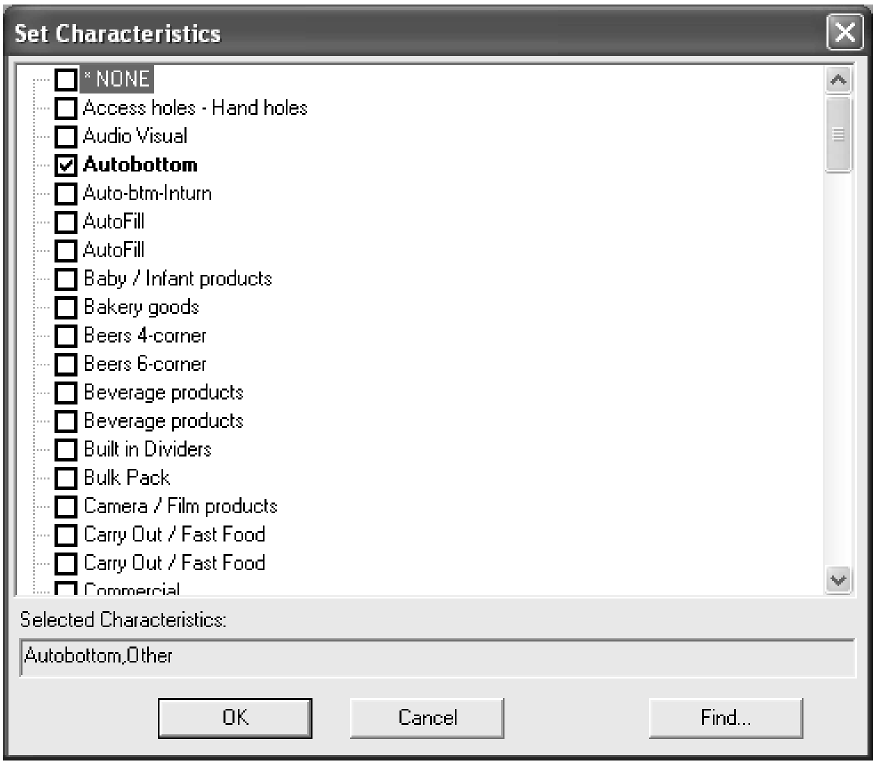
3. Enter the information you want the userfield to contain in the **Current Value**: field. In this example, the text `Send using priority overnight delivery!` will be in the Ship Method userfield. Click **OK** when you have entered the information.



4. The userfield will be set to the value you specified.

Characteristics

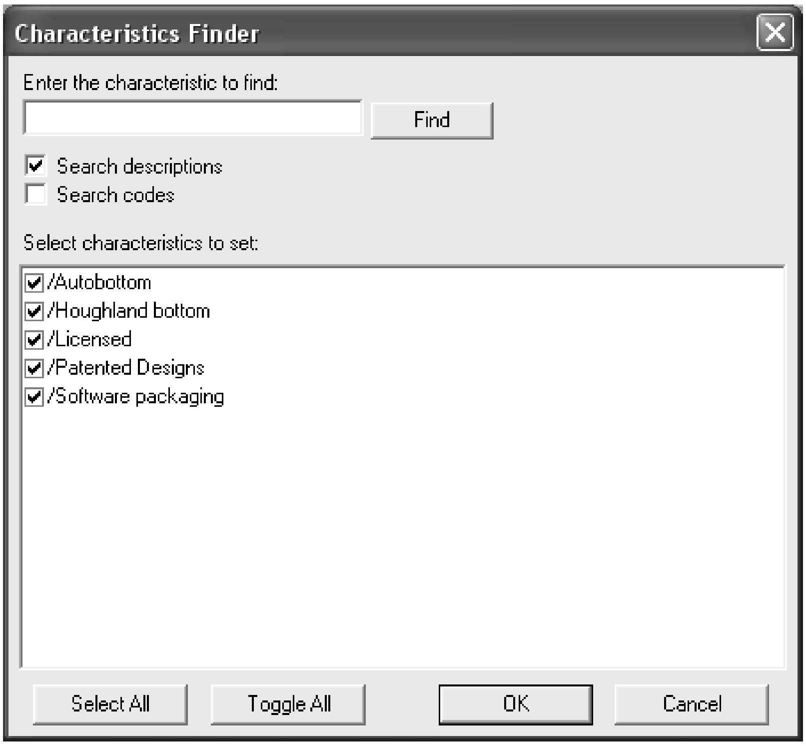
The **Characteristics** button allows you to set the characteristics of the design.



To set characteristics, check the checkboxes next to the desired entries and click **OK**.

To cancel assigning new characteristics, click **Cancel**. The dialog box closes without affecting previously-assigned characteristics.

Clicking **Find** opens the Characteristics Finder dialog box as shown below. Characteristics which were already set will appear in the new dialog box automatically.



Enter the characteristic to find and click **Find**. **Search descriptions** searches the descriptions for matches, while **Search codes** searches only the codes for matches. At least one of these checkboxes must be checked.

Any matching characteristics will appear in the **Select characteristics to set** field, but they will not be selected.

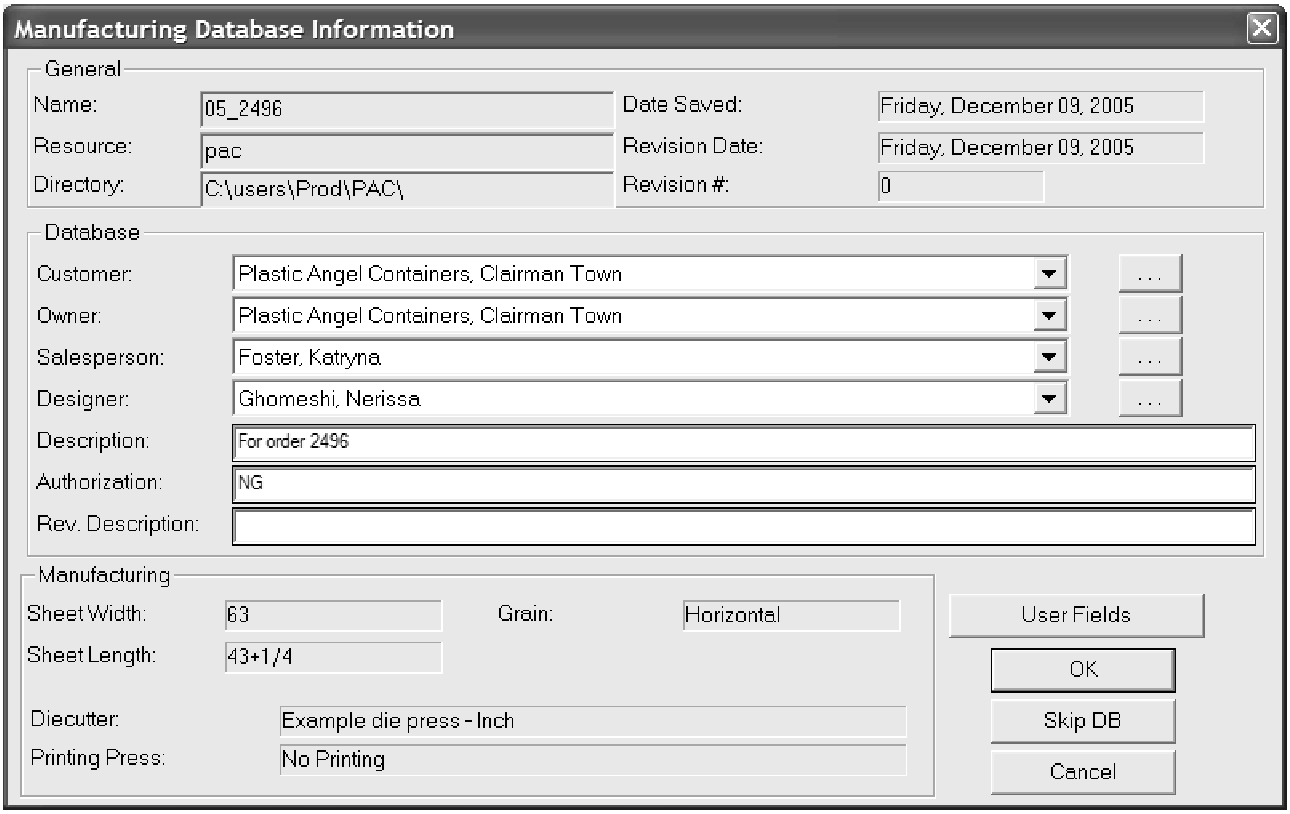
Select All selects all the characteristics. **Toggle All** deselects those characteristics which were previously selected and selects those which were previously not selected.

Combine searches by entering more search terms and clicking **Find**. Any non-selected characteristics from the previous search will be removed from the results box. The results of the new search will be shown along with those results of the previous search that were selected.

Saving a manufacturing file is done the same way as saving a design file. Specify a customer, owner, salesperson, designer, description, and an authorization message. Click User Fields to assign userfields to the manufacturing file.

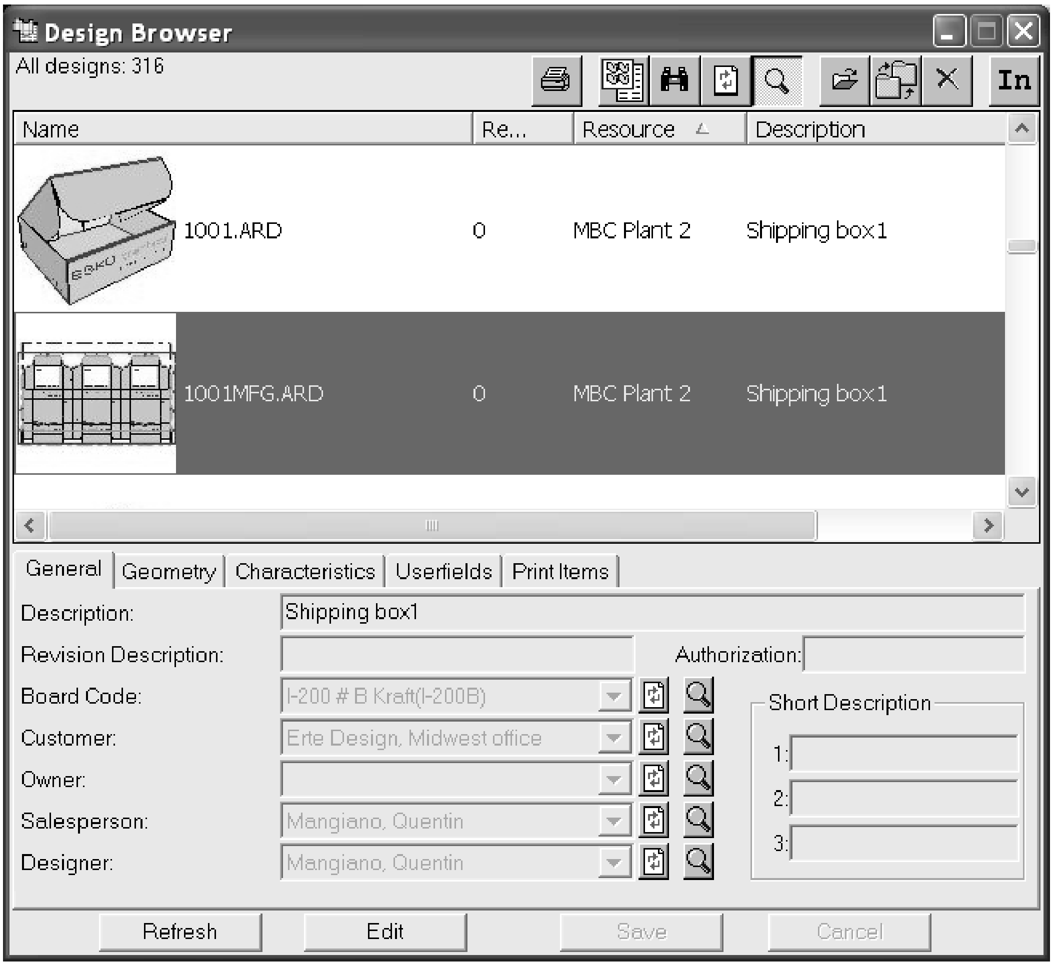
Note: The Characteristics Finder does not show hierarchical characteristics. All characteristics are displayed on the same logical level, so there may be apparent duplicate characteristics.

The following dialog box appears when saving a manufacturing file. It works the same way as saving a single design.

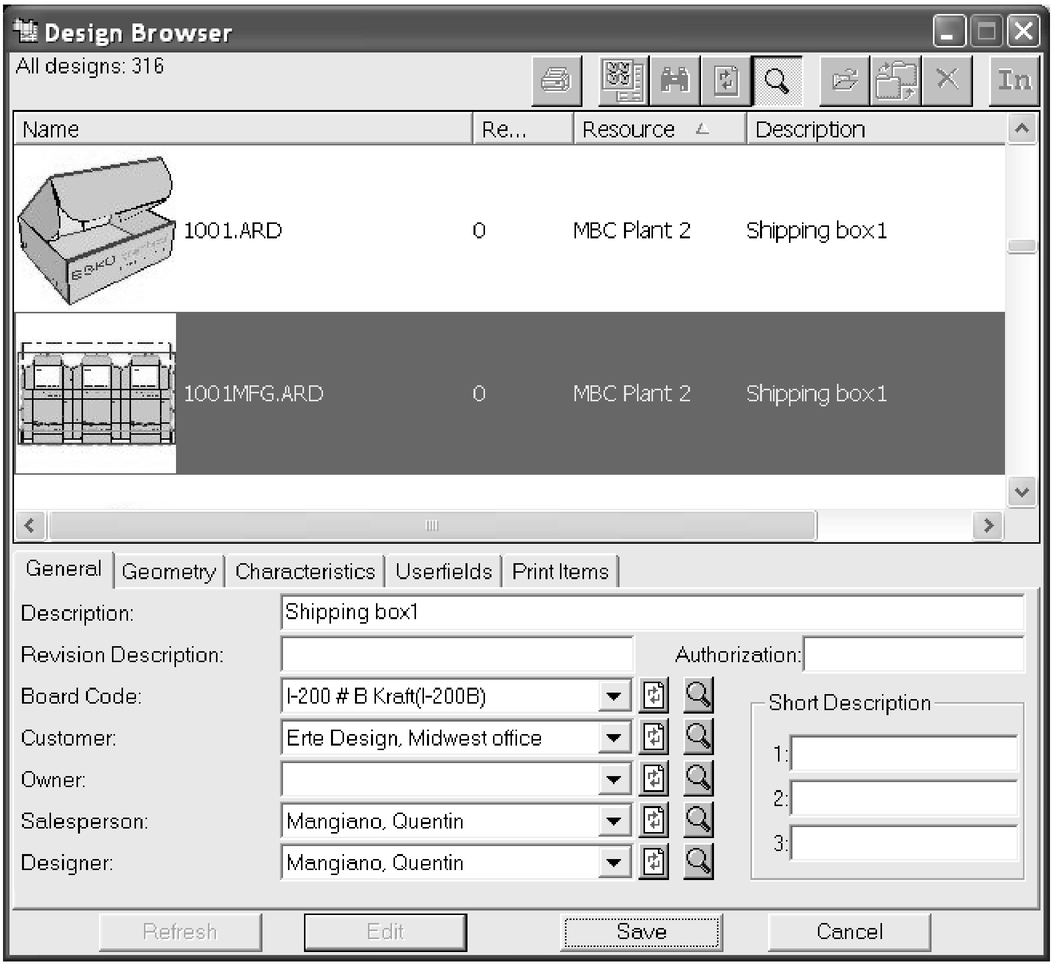


Changing existing database information in DataCenter

You can change information about a design or manufacturing file by clicking **Edit** in a browser.

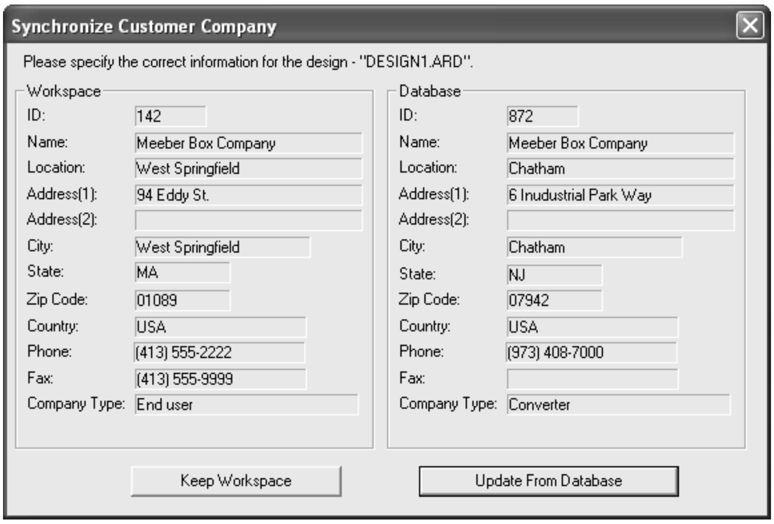


When **Edit** is clicked, the editable fields on the General, Characteristics, Userfields, and Print Items tabs turn white, and the **Save** and **Cancel** buttons become available as shown below. Make the desired changes and click **Save**.



Synchronizing information in DataCenter

On rare occasions, the information in the database will not match the information in ArtiosCAD, and you are asked to choose which set of information is correct – the information saved in the file or the information from DataCenter.



When a Synchronize dialog box appears, review the information shown in the **Workspace** group and **Database** group. Click **Keep Workspace** if the information in the Workspace group is correct. Click **Update From Database** if the database information is correct.

Exporting information from DataCenter

 Click the **Generate Report** button on the toolbar to export information from DataCenter. You can either export detailed information about the current selection(s) by choosing **Use Selected Records**, or print a list of the objects in the browser by choosing **Use All Records**. If there are any Multiple Workspace Outputs defined in Defaults, you can also output the selected workspace(s) through them as well.

There are two reports for each browser. The **AutoCol Browser List format** automatically detects which columns of data are selected 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.

All report types are created using XSLT transformations.

Customizing these reports involves changing the XML that is exported from ArtiosCAD and then modifying the XSLT transformation template to process the new XML information. Simple modifications, such as changing header columns can be performed by changing the Database Browser Reports entries in Defaults, but for more complex modifications, you should enlist the aid of the Esko Professional Services department or someone who has XML/XSLT editing experience.

Note: If Thumbnails are selected in the report Properties in Defaults, a JPG file for each thumbnail is stored in the same directory as the report output file. The thumbnails must be kept in the same directory as the report output file in order to appear in the report.

Other tasks

AutoLoad

AutoLoad is a utility which reads information about existing CAD designs and layouts and imports the data into the database. AutoLoad can read native ArtiosCAD (.ARD) workspaces, The Designer WorkBench (.DES) workspaces, and LASERPOINT IQ workspaces.

Use AutoLoad whenever you have multiple designs whose data you want to enter in the database without having to open each file and save it. Such a situation could be when you have loaded ArtiosCAD on a system which already contains ArtiosCAD, LASERPOINT IQ, or The Designer WorkBench designs. Another situation could be when you have copied many designs from another system onto your system.

AutoLoad can also convert batches of foreign file types (such as DXF or CF2) to ArtiosCAD workspaces. See the upcoming section *Recommended AutoLoad procedure for foreign file types* for more information.

Before using AutoLoad

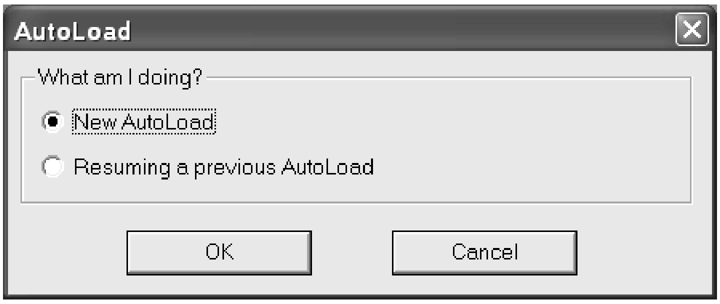
Before using AutoLoad, you must have created servers and resources as described earlier in this chapter. AutoLoad can only act upon files in resources.

You also should determine the type of AutoLoad to perform, **Simple** or **Advanced**. Simple AutoLoad lets you choose a resource or resources to load into the database and automatically saves all designs with an ArtiosCAD filename extension appropriate to the type of file. Advanced AutoLoad lets you configure AutoLoad options and the default database behavior.

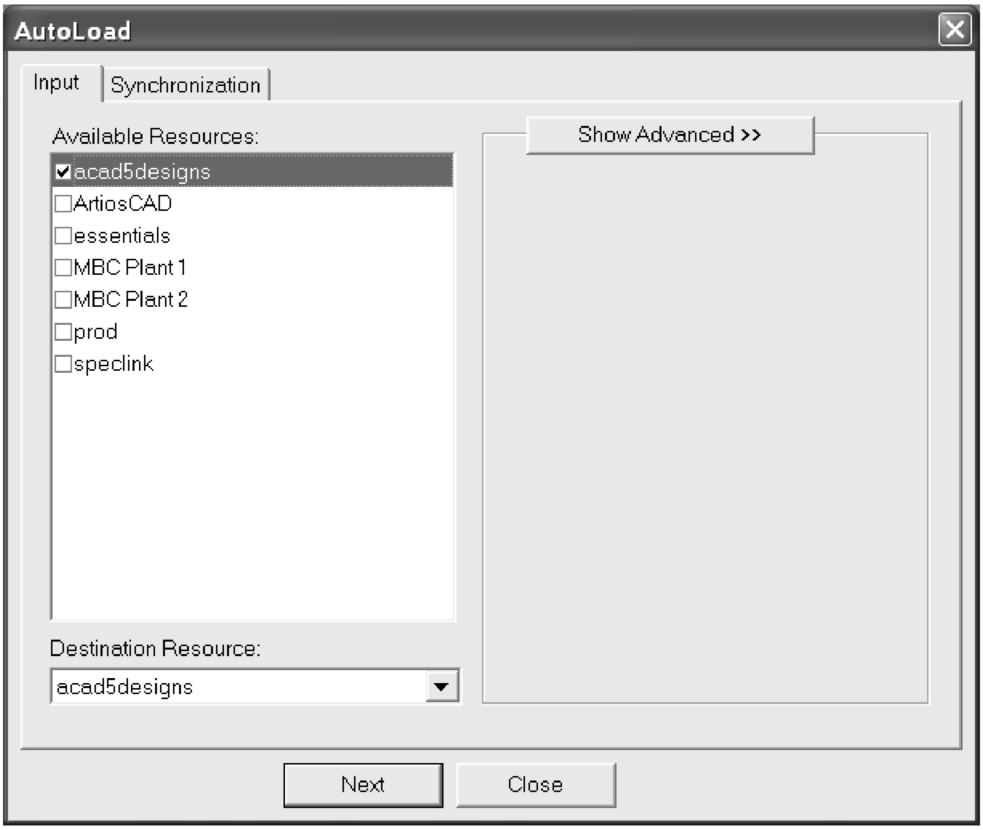
Simple AutoLoad

The procedure for using Simple AutoLoad is:

1. Make sure your database servers and resources are configured correctly.
2. Start ArtiosCAD.
3. Click **Database**, then **AutoLoad**.
4. In the **What am I doing?** group, choose **New AutoLoad** and click **OK**.



5. On the Input tab, check the checkboxes for the resource(s) you want to AutoLoad.



If you select only one resource, you can choose the resource into which the files are loaded by changing the entry in the **Destination Resource** list box. Selecting more than one resource to AutoLoad disables the Destination Resource list box.

6. Click the **Synchronization** tab.
7. Set the option in the **What am I doing?** group.

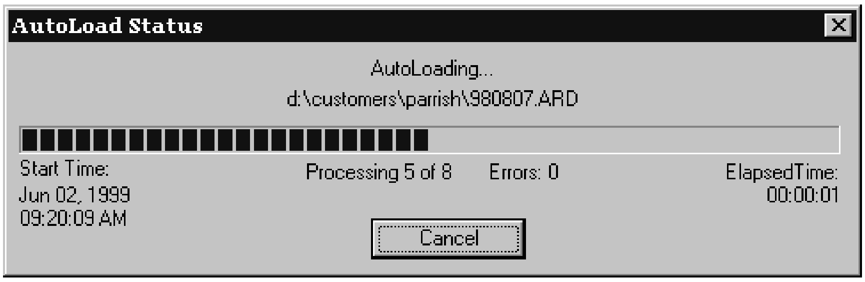
Use **Adding new files to the database** (the default) when adding completely new files to the database. These files have never been in a database in this ArtiosCAD installation before.

Use **Recovering database** when rebuilding a database, such as after a hard drive crash or accidental deletion. This reads the database information stored in the workspaces and rebuilds the database with it.

Use **Refreshing files after database changes** to update the workspaces after having made changes to their records in DataCenter Admin.

Click **AutoLoad** when you have chosen the option to use.

8. A progress indicator will appear.

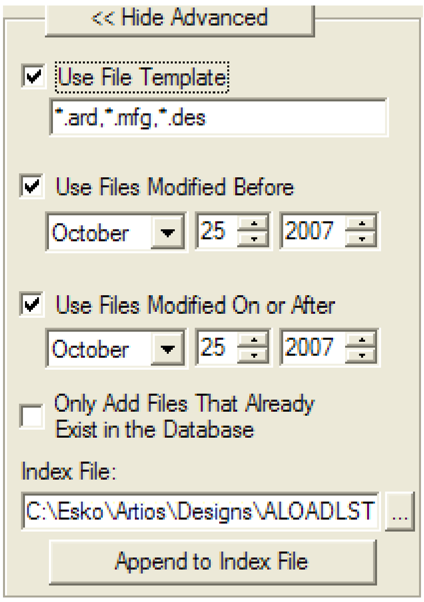


9. When AutoLoad completes, click **Close** to return to the AutoLoad dialog box, and then click **Close** to return to ArtiosCAD.

Advanced AutoLoad

Input tab Advanced pane

Clicking the **Show Advanced** unfold button on the Input tab of the AutoLoad dialog box shows more options than just the list of resources.



The **Use File Template** checkbox and field control which files are AutoLoaded. Any files matching the wildcards in the field are loaded. ArtiosCAD single designs, ArtiosCAD manufacturing files, and The Designer WorkBench workspaces have their wildcards set up by default. To AutoLoad all files in the selected resource(s), deselect the checkbox.

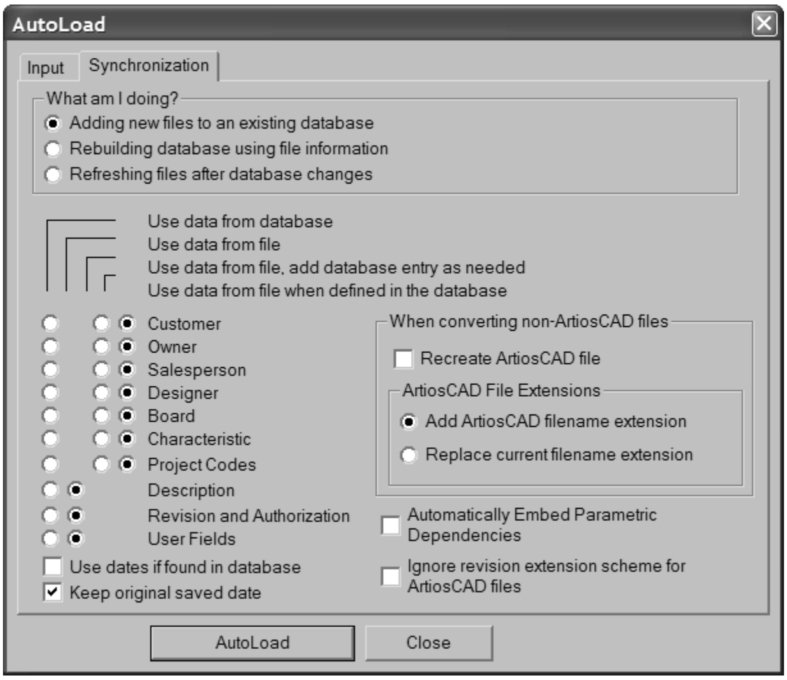
The **Use Files Modified Before** and **Use Files Modified On or After** checkboxes, when checked, let you specify the dates before, between, or after the dates you enter.

When checked, **Only Add Files That Already Exist in the Database** causes AutoLoad to skip those files whose records have been deleted from the database. This allows you to refresh the database or designs with new information without including all the designs whose records you manually deleted from the database on purpose.

When AutoLoad runs, it generates a list of all the files it is to load and writes the list to the file shown in the **Index File** field. This is usually `ALOADLST.TXT` in the default design directory. **Append to Index File** lets you link together multiple source and destination resources into one file and thus one AutoLoad.

Synchronization tab

The Synchronization tab is shown below.



The options in the **What am I doing?** group determine the way in which AutoLoad treats any existing database information in the files. Use **Adding new files to an existing database** (the default) when adding completely new files to the database. These files have never been in a database in this ArtiosCAD installation before. Use **Rebuilding database using file information** when rebuilding a database, such as after a hard drive crash or accidental deletion. This reads the database information stored in the workspaces and rebuilds the database with it. Use **Refreshing files after database changes** to update the workspaces after having made changes to their records in DataCenter Admin.

The data synchronization options control which data is kept when AutoLoad encounters a file which has a corresponding record in the database. These options are set by the option selected in the **What Am I doing**? group. They can also be set manually. If you make any manual changes, the option buttons in the **What am I doing?** group are cleared.

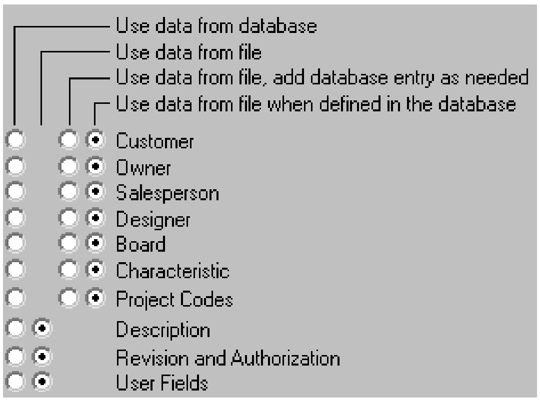
Use data from database updates the file on disc with the information saved in the database. This option is available for all categories of information stored in the database.

Use data from file reads the disc file for information and uses the data it finds to overwrite whatever is stored in that category of the database record. This option is available only for that category of data which does not have its own browser in DataCenter Admin.

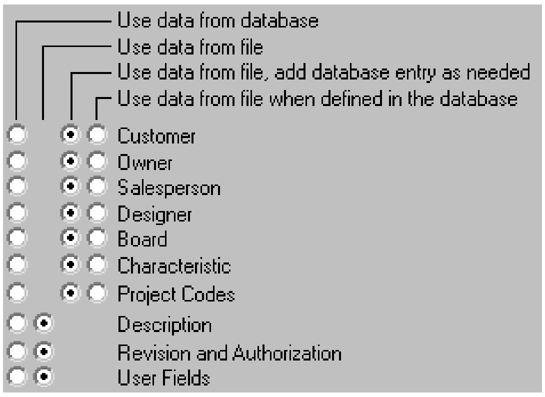
Use data from file, add database entry as needed reads the information from the disc file and automatically creates any needed entries in the database. This option is available only for those categories of information having their own browsers in DataCenter Admin.

Use data from file when defined in the database reads the information from the disc file and attempts to match it to already-existing entries in the database. This option is available only for those categories of information having their own browsers in DataCenter Admin.

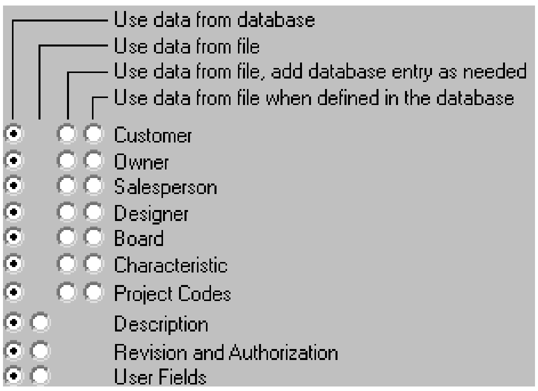
When **Adding new files to an existing database** is the selection in the **What am I doing?** group, the options are set as shown:



When **Rebuilding database using file information** is the selection in the W**hat am I doing?** group, the options are set this way:



When **Refreshing files after database changes** is the selection in the **What am I doing?** group, the options are set like this:



The **Use dates if found in database** checkbox uses the dates in the database if they are available. If they are not available, or this option is turned off, dates from the workspace are used.

The **Keep original saved date** checkbox, when checked, keeps as an element of the database record the date the file was originally saved, but the date and time of the physical file is changed in the filesystem because AutoLoad updates and saves the physical file. When this checkbox is cleared, the current date and time of the AutoLoad is used in both the database record and the filesystem.

The **Automatically embed parametric dependencies** checkbox controls whether ArtiosCAD automatically embeds in a design those components of a design which are required to rebuild it such as geometry macros and included designs.

The **Ignore revision extension scheme for ArtiosCAD files** checkbox, when checked, keeps AutoLoad from renaming revisions of ArtiosCAD files it processes.

The options in the **When converting non-ArtiosCAD files** group control what happens when AutoLoad encounters a foreign file with the same base name as an ArtiosCAD workspace. The ArtiosCAD workspace may have been modified since the last AutoLoad. The **Recreate ArtiosCAD file** checkbox, when checked, tells AutoLoad to delete the current ArtiosCAD file and recreate it from the foreign file; any modifications to the ArtiosCAD file since it was converted to an ArtiosCAD workspace will be lost. If the checkbox is clear, the foreign file is skipped and a warning is added to the log file.

The options in the **ArtiosCAD File Extensions** group control the filenames of non-native ArtiosCAD files. **Add ArtiosCAD filename extension** adds the appropriate .ARD or .MFG to the filename, which is useful when AutoLoading LASERPOINT IQ workspaces. **Replace current filename extension** deletes the current extension and replaces it with either .ARD or .MFG as appropriate, which is useful when AutoLoading .DES files from The Designer WorkBench. Note that .WBD files from The Designer WorkBench can not be AutoLoaded.

To perform an Advanced AutoLoad, set the options described above as desired and click **AutoLoad**. The files will be loaded into the database. When AutoLoad completes, click **Close** to return to the AutoLoad dialog box, and then click **Close** to return to ArtiosCAD.

AutoLoad notes and warnings

Only .DES files from The Designer WorkBench can be AutoLoaded; .WBD files can not be AutoLoaded.

AutoLoad assigns the default board code to designs that have no board code already defined.

When synchronizing design files with the database, and length, width, and depth variables are not set in the workspace and are not zero in the database, the variables in the workspace are set from the database.

AutoLoad does not rebuild 3D or paper information for boards. This information can come either from a database backup that is restored through DataCenter Admin, or by using the ArtiosCAD Database Export utility to export the board table before any problems occur and then by using the ArtiosCAD Database Import utility to import the board table back into the database.

Recommended AutoLoad procedure for foreign file types

If you want to use AutoLoad to automatically convert foreign files types (such as DXF and CF2) to ArtiosCAD workspaces, do the following:

1. Set up the default import tuning by configuring the Import Tuning entries in **Options** > **Defaults** > **Import Tuning Table** (adding or modifying the entries as appropriate).
2. Set the import tuning mapping in **Options** > **Defaults** > **Design Defaults** > **Default Import Tuning** to select the correct entry for the given file type.
3. Start to AutoLoad files starting with the default File Template file types. Make sure the **Recreate ArtiosCAD file** checkbox is checked on the Synchronization tab.
4. Following successful processing of these files, begin to AutoLoad any of the common format files separately by typing the appropriate entry into the file template text field, such as `*.cf2` or `*.dxf`.
5. Once AutoLoad has completed, exit and restart ArtiosCAD before AutoLoading further file types.

This procedure will minimize the overall time required to AutoLoad the files. Note that using any of the database synchronization options can significantly increase the time required to process the desired files.

AutoLoad and Projects

When AutoLoading, depending on if the design is already in the database and if the design came from the same database, project codes will be handled differently.

If **Use data from database** is selected, and the file exists in the database, then ArtiosCAD will synchronize the information using the database values. If the file does not exist in the database, but came from the same database (has the same database ID), it will AutoLoad normally. If the file came from a different database (has a different database ID), the Project codes will be cleared and the file will AutoLoad normally.

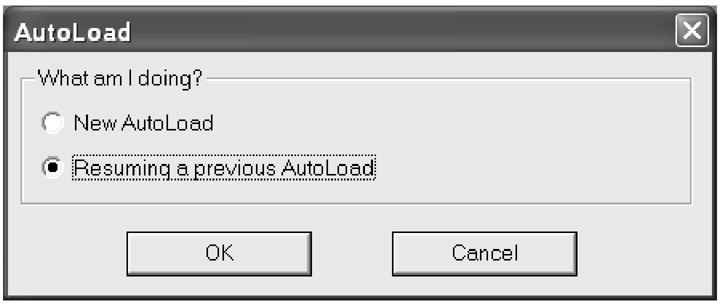
If **Use data from file, add database entry as needed** is selected, database entries will be added as needed. Select this option to add Project codes from foreign files to your database.

If **Use data from file when defined in the database** is selected, Project codes which do not already exist in the database will be cleared from the file.

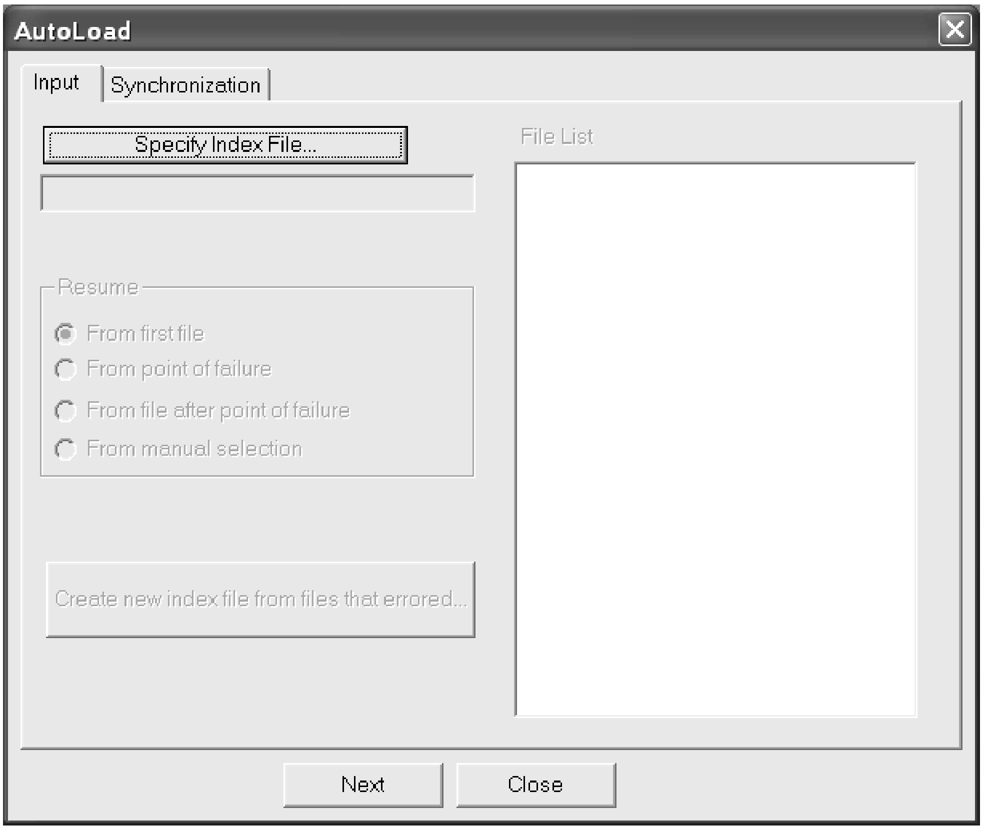
Resuming a previous AutoLoad

If AutoLoad terminates abnormally, you can resume it by doing the following:

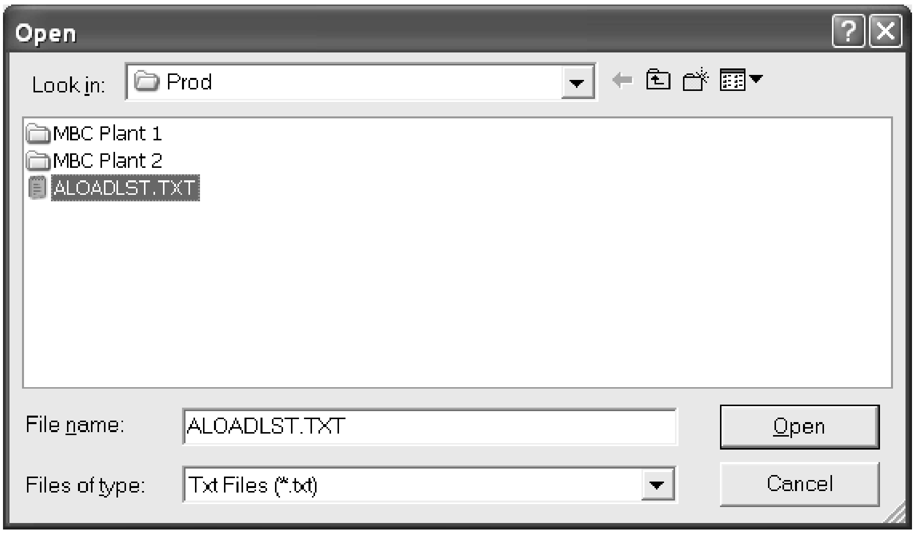
1. Restart ArtiosCAD and click **Database** > **AutoLoad**.
2. Choose **Resuming a previous AutoLoad** in the initial **What Am I Doing?** group of the AutoLoad dialog box and click **OK**.



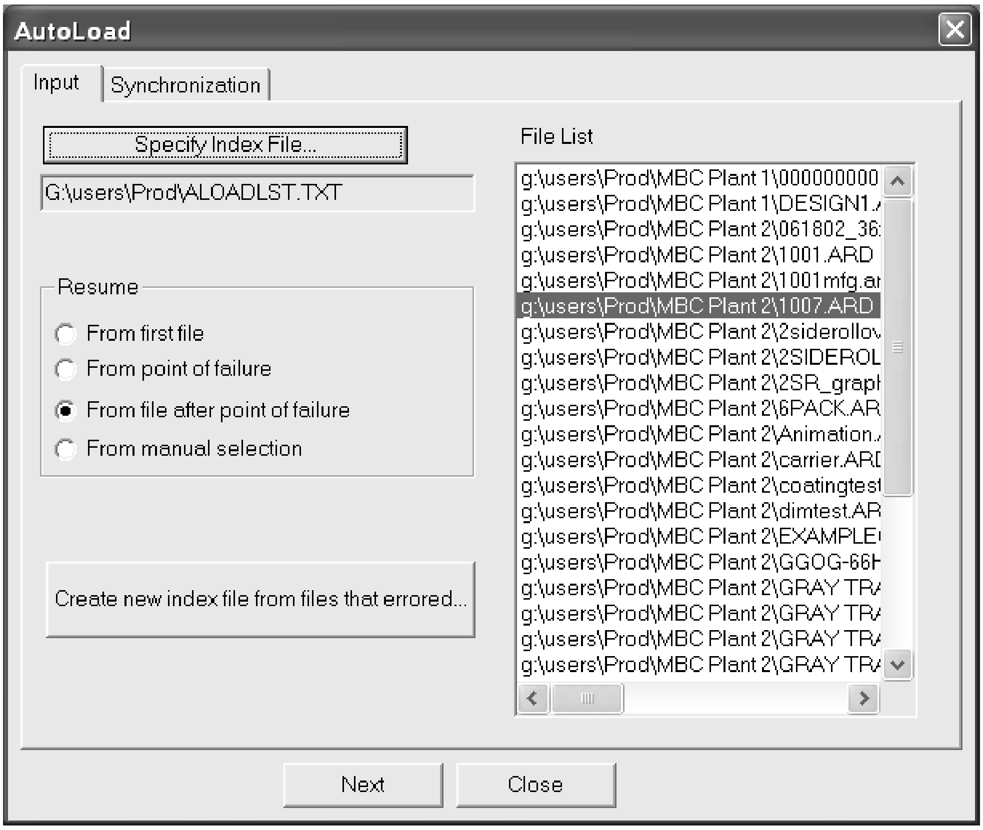
3. The AutoLoad dialog box appears with a different Input tab than for new AutoLoads.



4. Click **Specify Index File** and find the `ALOADLST.TXT` file generated by the previous AutoLoad. The Open dialog box opens in the default design directory; browse for the file if you specified a different location for the index file in the previous AutoLoad. Select the file and click **Open**.



5. The **File List** group shows the files that AutoLoad previously attempted to process. The first file after the point of failure is highlighted by default.



6. Choose the option in the **Resume** group at which AutoLoad is to resume:

   From first file chooses the first file in the list.

   From point of failure selects the last file the previous AutoLoad attempted to process. To avoid another abnormal termination, you should try to open this file manually in ArtiosCAD and save it before choosing this option.

   From file after point of failure is the default, and selects the first file after the one that caused an abnormal termination.

   From manual selection lets you select the file in the File List at which the AutoLoad resumes.
7. Click the **Synchronization** tab and set the options as desired.
8. Click **AutoLoad**. It will resume from the file chosen in step 6.
9. When the AutoLoad finishes, click **Close** to close the AutoLoad Status dialog box and then click **Close** to close the AutoLoad dialog box.

Create new index file from files that errored creates an index file of any files in the AutoLoad that errored, except for those loaded into a destination resource. Click it and then enter a sensible path and filename for the new index file and click **Save**. That index file is then automatically loaded into the file pane. Try AutoLoading again; if they error, examine the log file by clicking **View Log**. To resolve the error, try to open the files in ArtiosCAD and then save them.

Database network performance

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

The Customer cache is refreshed when the **New** button in the Customer dialog box is clicked.

Refreshing the cache may take a while on a slow WAN with very large entity lists.

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**.

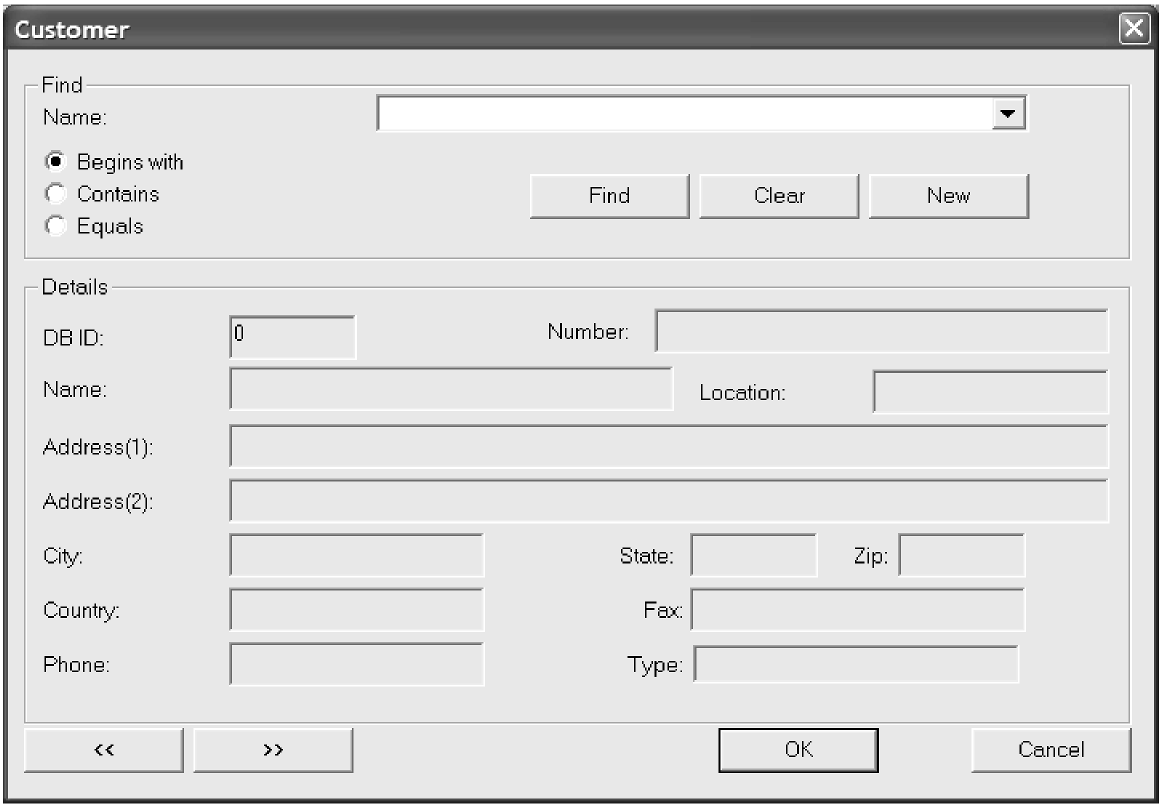
For information about configuring these options, refer to the *Defaults* chapter of the *ArtiosCAD Administrator Guide*.

Connecting to the database

Connecting to the database should be automatic. However, in the event of network trouble that severs the connection, click **Database** > **Connect**. This command is unavailable when there is a current connection.

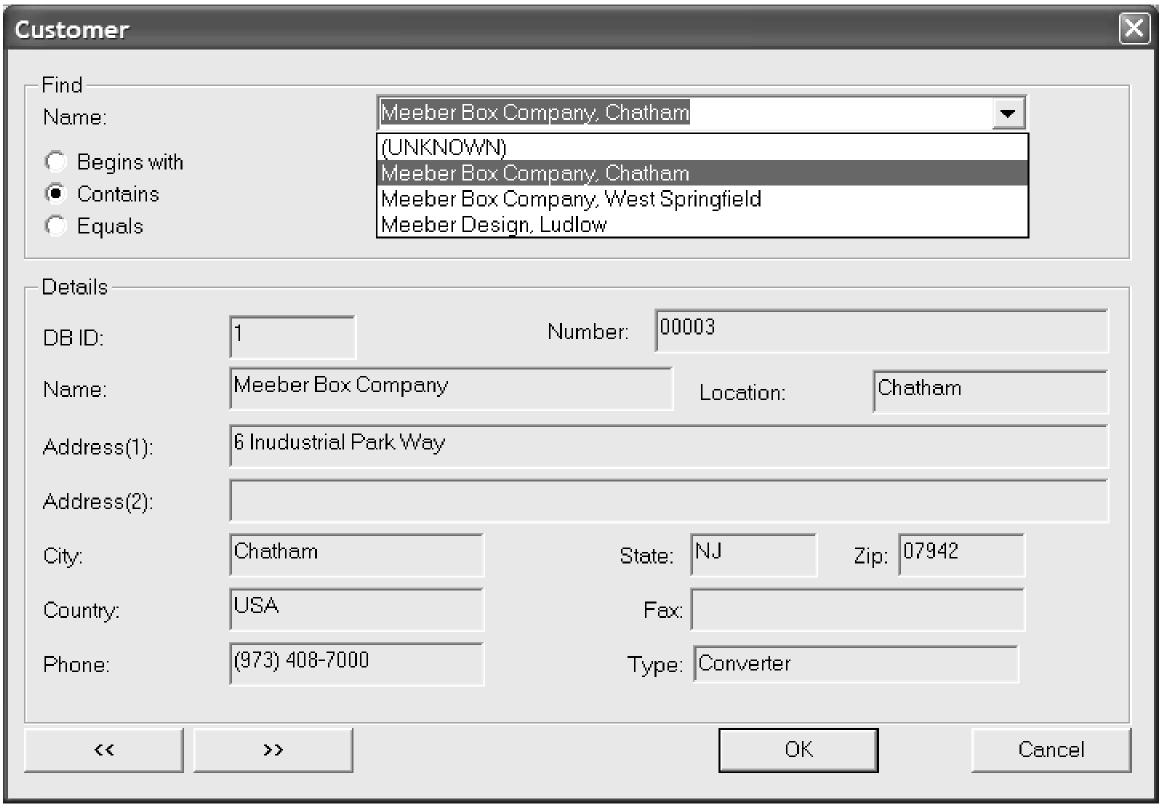
Pattern searching

When pattern searching is enabled in Defaults, 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.

For example, to find the Meeber Box Company, one could enter just the first few letters - **Meeb** - and choose **Begins with**. Or, one could enter **Box** and choose **Contains**. Finally, one could enter **Meeber Box Company** and choose **Equals**, and then click **Find**. Clicking the drop-down list box activator will display a list of results from which the desired entry is chosen.



Clear clears the current search results and lets you search for a new value.

New appears only in the Customer dialog box and lets you enter a new Customer database record.

The double chevrons (« and ») display the previous and next entry in the search results, respectively.

To accept the results of the search, click **OK**; otherwise, click **Cancel** to return to the previous dialog box.

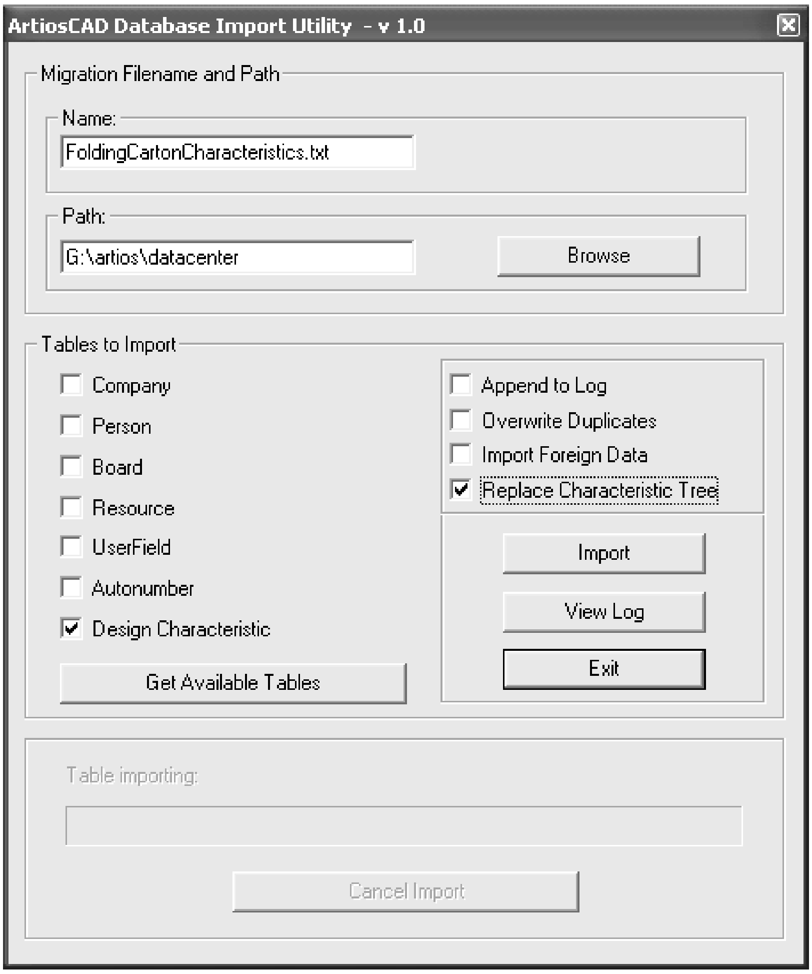
Installing additional characteristics

Two files containing many additional characteristics are installed to the `\Esko\Artios\DataCenter` directory when ArtiosCAD is installed. The two files are `FoldingCartonCharacteristics.txt` and `CorrugatedCharacteristics.txt`. You may load one of them, both of them, or neither at your discretion.

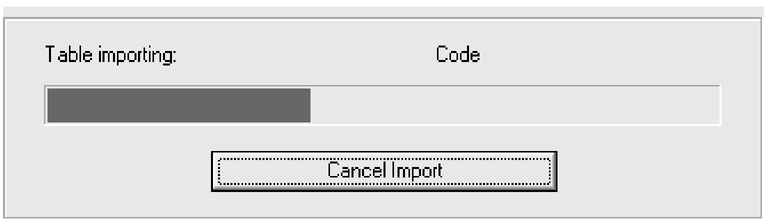
To load them into the database, do the following:

1. Make sure no one is using ArtiosCAD, DataCenter Admin, Esko SpecLink, or WebCenter (version 3 or below) to insure that the database is not being accessed.
2. Log on to the database server as Administrator or as a member of the **Administrators** group.
3. Start DataCenter Admin.
4. Click **File** > **Loader**.
5. When prompted for a username and password, enter **admin** for the username and do not enter a password.

6. The ArtiosCAD Database Loader Utility dialog box will appear. Enter the name of the new characteristics file to load in the **Name**: field, and enter the path (such as `C:\Esko\Artios\DataCenter`) in the **Path**: field. Or, click **Browse** to navigate to the file.
7. In the **Tables to Import** group, check the **Design Characteristic** checkbox. If you want to replace the existing characteristics, check the **Replace Characteristic Tree** checkbox. To merge the new characteristics in with the existing characteristics, leave the checkbox blank. The dialog box should look similar to the one shown below.



8. Click **Import**. The **Table Importing** progress bar will increment as the file loads.



9. When it is done, you may load the other new characteristics file by repeating steps 5 through 7, substituting the name of the other file in the **Name**: field, and making sure to not select the **Replace Characteristic Tree** checkbox (otherwise the characteristics from the file you just loaded would be overwritten).
10. To view the log file, click **View Log**. To exit the utility, click **Exit**.

Changing the database password

To change the database password ArtiosCAD uses internally to connect to the database when using MSDE, Microsoft SQL Server, SQL Server 2005 Express Edition, or Oracle, do not use any vendor-supplied utility. Instead, use the **Change Database Password** command on the File menu.

When running DataCenter Admin in an operating system supporting User Access Control, you must be logged in as an administrative user to change the database password. The **Change Database Password** command is unavailable if you are not an administrative user.

10. Appendix

Arithmetic functions

Included in ArtiosCAD are arithmetic functions which can be used anywhere that data is input. In the table below, the capitalized letters in the function are the minimum you have to enter into the data entry field.

Table: Arithmetic Functions

Function	Usage
ARCCos	Calculates the angle that has the specified cosine. Syntax: **ARCCos(cosine)** Result: Angle between 0 and 180. ArtiosCAD will error if the cosine value is outside range -1.1. For example, ARCCOS(0) = 90
ARCSin	Calculates the angle that has the specified sine. Syntax: **ARCSin(sine)** Result: Angle between -90 and 90. ArtiosCAD will error if the sine value is outside range -1..1. For example, ARCSIN(1) = 90
ARCTan	Calculates the angle that has the specified tangent. Syntax: **ARCTan(tangent)** Result: Angle between -90 and 90. For example, ARCTAN(1) = 45
COSine	Calculates the cosine of an angle. The angle must be in degrees but there are no limits on the value used for the angle. Syntax: **COS(angle)** Result: Number between -1.0 and 1.0 For example, COS(0) = 1 COS(60) = 0.5 COS(180) = -1.0

Function	Usage
POWer	Raises the number to the given power. Fractional powers are not permitted for negative numbers. Syntax: **POWer(value,exponent)** Result: Numeric. For example, POWER(3,2) = 9 POWER(64,1/3) = 4 POWER(2,3) = 8 POWER(64,-1/3) = 1/4 POWER(-2,3) = -8 POWER(5,3.6) = 328.316 Note: To calculate a hypotenuse, use the following expression: $C^2 = A^2 + B^2$ C = Square root of (A+B) C = POW((POW(A,2)+POW(B,2),.5)
SINe	Calculates the sine of an angle. The angle must be in degrees but there are no restrictions on the value used for the angle. Syntax: **SIN(angle)** Result: Number between -1.0 and 1.0. For example, SIN(0) = 0.0 SIN(30) = 0.5 SIN(360) = 0.0
SQRT	Calculates the square root of an arithmetic expression. Not valid for negative numbers. Syntax: **SQRT(expression)** Result: Numeric. For example, SQRT(36) = 6
TAN	Calculates the tangent of the angle. The angle must be in degrees but there are no limits on the value used for the angle. Syntax: **TAN(angle)** Result: Number. For example, TAN(0) = 0, TAN(45) = 1, TAN(60)=1.73205, TAN(-45) = -1

Reserved words

The following words and word fragments are reserved for use by ArtiosCAD. Do not use them for variable names or in command files.

Table: Reserved Words

ARC	AT	BOT	BOTT	BOTTO	BOTTOM	BRI
BRID	BRIDG	BRIDGE	BY	BYX	BYY	CEN
CENT	CENTE	CENTER	CENTR	CENTRE	CIR	CIRC
CIRCL	CIRCLE	COR	CORN	CORNE	CORNER	CS
DIA	DIAM	DIAME	DIAMET	EXPAND	EXT	EXTE
EXTEN	EXTEND	FCC	FCN	FIN	FINI	FINIS
FINISH	FON	FONT	FOR	FORM	FORMA	FORMAT
FRO	FROM	ITE	ITEM	LEF	LEFT	LEN
LENG	LENGT	LENGTH	LONG	MOV	MOVE	NEA
NEAR	OFF	OFFS	OFFSE	OFFSET	ORI	ORIE
ORIEN	ORIENT	POI	POIN	POINT	POINTA	POS
POSI	POSIT	POSITI	RAD	RADI	RADIU	RADIUS
RELAX	REV	REVE	REVER	REVERS	RIG	RIGH
RIGHT	SC	TCC	TCN	TEX	TEXT	THR
THRO	THROU	THROUG	TO	TOL	TOLI	TOLIN
TOLINE	TOP	TOX	TOY	TYP	TYPE	VAR
VARI	VARIA	VARIAB	WID	WIDT	WIDTH	WSCALE

List of line types

Below is the complete list of line types in the order in which they appear in ArtiosCAD. The **Grouped in** column indicates in which group the line type is found. The **Number** is the line designation in fundamental ArtiosCAD syntax.

Table: List of Line Types

Line type name	Grouped in	Number	Category / Notes
Cut	Design	1	Rules
Crease	Design	2	Rules
Partial cut	Design	4	Rules
Reverse partial cut	Design	9	Rules
Reverse crease	Design	8	Rules
Second height crease	Design	223	Rules
Matrix crease	Design	224	Rules
Half crease	Design	16	Rules
Unknifed diecut	Design	12	Center edge
Die etch	Design	18	Etch
Print registration	Design	10	Annotation
Inside Bleed	Design	212	Annotation
Outside Bleed	Design	11	Annotation
Inside Coating (Varnish)	Design	213	Annotation
Outside Coating (Varnish)	Design	85	Annotation
Annotation	Design	0	Annotation
Annotation2	Design	207	Annotation
Annotation3	Design	208	Annotation
Annotation4	Design	209	Annotation
Annotation5	Design	210	Annotation
Dimensions and text	Design	20	Annotation
Print image	Design	14	Annotation
Form lines 1	Design	23	Annotation
Form lines 2	Design	24	Annotation
Copy area	Design	262	Annotation
Copy free	Design	263	Annotation
Dynamic art	Design	264	Annotation

Line type name	Grouped in	Number	Category / Notes
Side bevel	Generic rules	19	Rules
Generic perf	Generic rules	3	Rules
Generic perf in channel	Generic rules	6	Rules
Generic cut and crease	Generic rules	5	Rules
Partial cut and crease	Generic rules	15	Rules
Glue assist	Generic rules	7	Rules
Generic special rule	Generic rules	13	Rules
Partial cut perf	Generic rules	251	Rules
Reverse partial cut perf	Generic rules	252	Rules
Partial cut perf in channel	Generic rules	253	Rules
Second height cut crease	Generic rules	254	Rules
Second height partial cut crease	Generic rules	255	Rules
Reverse partial cut crease	Generic rules	256	Rules
Usable limit of sheet	Utility	97	Annotation
Edge of sheet	Utility	98	Annotation
Axes	Utility	31	Annotation
Construction line 1	Utility	34	Annotation
Construction line 2	Utility	33	Annotation
Construction line 3	Utility	32	Annotation
Nick	Utility	17	Annotation
Tear	Utility	205	Annotation
Bend	Utility	206	Annotation
Size only	Utility	215	Used for Size information only
Stripping rule	Dieboard	93	Rules
Carrier rule	Dieboard	94	Rules

Line type name	Grouped in	Number	Category / Notes
Scrap/Chop knife	Dieboard	96	Rules
Balance knife	Dieboard	218	Rules
Unknifed diecut	Dieboard	12	Center edge Same as earlier versions of ArtiosCAD. Also used for floats.
Die board edge	Dieboard	170	Outside edge Also used for die splits. Milled parts of the dieboard edge for DieSaw output are distinguished using different subtypes.
Die registration hole	Dieboard	171	Inside edge Used for registration holes or chase holes.
Laser position holes	Dieboard	216	Inside edge
Die bolt holes	Dieboard	217	Inside edge. Drill holes for DieSaw output are distinguished using different subtypes.
Die etch	Dieboard	18	Etch For labels that are burned in the die.
Lead hole	Dieboard	172	Inside edge Lead hole for the DieSaw in the rule path layer.
Unknifed lead in	Dieboard	173	Center edge Lead in line which starts a cut by the DieSaw but is not part of the design and has no rule.
Die split	Dieboard	174	Outside edge Used to divide the die into pieces.
Gripper finger	Dieboard	219	Annotation
Lower board edge	Lower board	175	Outside edge
Lower stripping hole	Lower board	176	Inside edge The lower holes must be separate from other holes because the

Line type name	Grouped in	Number	Category / Notes
			processing to create the upper board depends on them.
Lower registration hole	Lower board	177	Inside edge
Lower etch	Lower board	178	Etch For labels that are burned into the lower board
Unknifed lower	Lower board	179	Center edge For a slot with a specified width, such as the end of a carrier rule outside the stripping hole.
Hole inside modifier	Lower board	180	Inside edge The inside edge of an interference made of wood which extends partly into the lower stripping hole.
Hole outside modifier	Lower board	181	Inside edge A slot in the edge of a lower stripping hole which holds the outside edge of an interference element.
Hole interference	Lower board	182	Inside edge The edge of the active part of an interference element.
Upper board edge	Upper board	183	Outside edge
Stripping block	Upper board	184	Outside edge
Block support pin	Upper board	185	Inside edge
Upper stripping rule	Upper board	186	Rules The base of the stripping rule which is burned in the upper board
Pusher base	Upper board	187	Inside edge The base of the push pin or pusher element which is burned in the upper board
Pusher active part	Upper board	188	Inside edge The active part which pushes the waste

Line type name	Grouped in	Number	Category / Notes
Pusher shoulder	Upper board	189	Annotation The shoulder that stops the element from being pushed through the upper board
Pusher clearance	Upper board	190	Annotation Shows the effective size of the pusher or pin.
Upper air hole	Upper board	191	Inside edge
Upper registration hole	Upper board	192	Inside edge
Upper etch	Upper board	193	Etch For labels which are burned in the upper board
Unknifed upper	Upper board	194	Center edge For a slot with a specified width.
Front waste edge	Front waste separator	195	Outside edge
Front waste registration hole	Front waste separator	196	Inside edge
Front waste etch	Front waste separator	197	Etch
Rubber inside edge	Rubber	220	Annotation. Distinguished from the rest of the outline so that it can be cut with a bevel on machines that have a tilting head.
Rubber outline	Rubber	221	Annotation
Rubber name	Rubber	222	Annotation. Used to cut numbers on each piece of rubber.
Blanker lower board edge	Blanker	227	
Blanker lower hole	Blanker	228	
Blanker lower registration hole	Blanker	229	
Blanker lower etch board	Blanker	230	

Line type name	Grouped in	Number	Category / Notes
Blanker upper board edge	Blanker	231	
Blanker upper air hole	Blanker	232	
Blanker upper registration hole	Blanker	233	
Blanker upper etch	Blanker	234	
Blanker pusher edge	Blanker	235	
Blanker pusher air hole	Blanker	236	
Blanker pusher registration hole	Blanker	237	
Blanker pusher etch	Blanker	238	
Blanker frame	Blanker	239	
Blanker non-stop sword	Blanker	240	
Blanker photocell	Blanker	241	
Blanker grid	Blanker	242	
Blanker support bar	Blanker	243	
Component active part	Blanker	244	
Component edge	Blanker	245	
Component clearance	Blanker	246	
Component annotation	Blanker	247	
Sample knife	Sample	120	Etch
Sample partial cut knife	Sample	121	Etch
Sample crease with grain	Sample	122	Etch
Sample crease cross grain	Sample	123	Etch
Sample pen 1	Sample	124	Etch
Sample pen 2	Sample	125	Etch
Sample pen 3	Sample	126	Etch
Sample pen 4	Sample	127	Etch

Line type name	Grouped in	Number	Category / Notes
Sample reverse partial cut	Sample	128	Etch
Sample reverse crease	Sample	129	Etch
Sample reverse crease cross grain	Sample	130	Etch
Sample counter knife	Sample	131	Etch
Sample router with grain	Sample	132	Etch
Sample router cross grain	Sample	133	Etch
Sample counter pen	Sample	134	Etch
Sample V-notch	Sample	135	
Sample V-notch cut	Sample	136	
Sample drill hole	Sample	137	
Sample 2nd height crease with grain	Sample	225	Etch
Sample 2nd height crease cross grain	Sample	226	Etch
Tool 1	Tooling	140	Etch
Tool 2	Tooling	141	Etch
Tool 3	Tooling	142	Etch
.	.	.	.
.	.	.	.
.	.	.	.
Tool 28	Tooling	167	Etch
Tool 29	Tooling	168	Etch
Tool 30	Tooling	169	Etch
With grain crease	Counter	100	Etch
Cross grain crease	Counter	101	Etch
Crease end tool	Counter	102	Etch
Periphery tool	Counter	103	Etch

Line type name	Grouped in	Number	Category / Notes
Chamfer tool	Counter	104	Etch
2nd height with grain crease	Counter	105	Etch
2nd height cross grain crease	Counter	106	Etch
Name tool	Counter	107	Etch
Positioning hole tool	Counter	108	Etch
Milling tool	Counter	109	Etch
Counter mill outline	Counter	110	Etch
Position holes 2	Counter	111	Etch

Glossary

Term	Definition
Accept ratio problems	A checkbox that allows the user to continue running a standard even if the ratio of the variables (length, width, depth, etc.) is not in correct proportion.
Account for Deflection	An adjustment for the deflection of a tool when cutting the counter. It is used to ensure channel width accuracy.
Advanced	A button that gives access to more options.
Air hole	Hole(s) made in the stripping board to help prevent jams caused by a vacuum effect.
Alignment hole	Hole(s) made in the die board to allow quick and easy alignment of a set of manufacturing tools.
Allow Rotation on Repeat	A menu checkbox in Manufacturing Layout that gives the option to consider rotated one-ups the same as non-rotated one-ups and therefore do the same operations in congruent areas.
allowance reduction	An addition to the bridge's width to compensate for the leader hole's reduction of the bridge's width when using a diesaw.
Angle	The number of degrees between two lines or between a line and an axis
Annotation	Details, arrows, and text that are typically placed on a drawing to provide more explanation.
Arc	A portion of a circle.

Term	Definition
Area	A value calculated by multiplying length and width.
Authorization	Approval by authority.
Authorization Msg	A field for marking a design. Typically initialed as “approved by” or marking it with an appropriate comment.
Auto Aligned Distance Dimension	A tool that dimensions a series of parallel lines in a design.
Auto Numbering	Automatically sequencing the next design and/or revision in ascending order based on configuration.
Auto-Detect	An importing option that decides if a file should be imported as a Layout or Oneup. This decision is made based on the contents of the file.
AutoLoad	A utility which gathers information about existing CAD designs and layouts. Then it imports the data into DataCenter and saves them with the appropriate ArtiosCAD file extension.
Auto-Repeat	Automatically repeats elements in manufacturing such as stripping rules, holes, and pins throughout similar areas of the layout.
AutoSave	A utility that will create temporary backup files that are available for recovery should the system crash.
AutoSave Recovery	A dialog box offering the option to recover the temporary backup files created by the AutoSave feature.
AutoTrace Bitmap	A tool that replaces a bitmap with lines and arcs that can be manipulated in ArtiosCAD.
Balance Knives	Steel rules that are used to balance the load on a dieboard for a platen press. They are also called leveling knives.
Base Face	The selected panel of the design around which all other panels fold. Used in 3D mode.
Basic Dimensions	The dimensions that are required to be entered to run a standard, typically length, width, and depth.
Beeps	A sound/noise made to inform the user of things such as data received or an error in snapping.
Belt speed	The number of feet per minute that a conveyer belt moves. The belt speed and travel distance are used in one method of determining the number of sheets per hour in the Intelligent Layout module.
Bend corners	A grouping of options to take a piece of rule and extend it around a corner at an angle on a rotary die. The options are to take the rule from rotary to flat or from flat to rotary.

Term	Definition
Bend Length	In a rotary die, the length of the chop knife that is used for transitioning corners from rotary rule to flat rule or from flat rule to rotary rule.
Between teeth	An option to have bridging start between the teeth on a rotary rule.
Bevel	An option for joining strokes that chamfers the outside edges of strokes that meet. Strokes are graphical additions to a line. The Bevel option can be used when auto-tracing a bitmap.
Bezier	A tool that creates a French curve with a start point, an end point, and two adjustable middle points.
Beziers to Arcs	An option that changes all bezier lines in the design to arcs and is used when exporting to programs that do not support beziers.
Blank	Any die cut section of boxboard in a flat configuration. The Blank is formed into a rigid box later. Also, a Blank is a folding carton after cutting and scoring but before folding and gluing.
Blanker	The actual board that breaks the nicks to separate the designs. It is the last station in the press.
Blanker pusher	The part of the blanking tool that traps the design and pushes it through to separate the designs.
Bleed	Defines the area of ink coverage when the sheet moves through the printing press.
Bleed Layer	A layer created for the Bleed Outline.
Bleed tool	A tool that creates a Bleed Outline with optionally excluded panels and an offset value.
Blend	A tool that creates an arc to connect two lines.
Block	A piece of wood or a font type for the name that is burnt into the die.
Board	A heavy thick sheet of paper or other fiber substance usually of a thickness of 0.006 inches or more.
Board Code	A group of settings that tells ArtiosCAD about the material used to manufacture the design.
Board Properties	The characteristics of the board that is being used. The properties include caliper (board thickness), costs, and test paper.
Bowtie	A hole in the counter to allow for the cut section of a cut crease line. The hole is shaped like a bowtie.
Boxboard	Paperboard of sufficient thickness to be used in the manufacture of paperboard boxes.

Term	Definition
Bridge	The small areas left uncut in a jig or laser die for the purpose of holding the die together.
Bridge Rule Path	Applies the bridging formulas to each rule path and creates the appropriate bridges.
Bridging	The placement of notches in a steel rule blade so that the blade can fit into it’s relative place in a dieboard.
Brightness Threshold	A control (slider) that determines how much of the chosen color will be vectorized in relation to the brightness of the color selected to vectorize. It is used when auto-tracing bitmaps.
Browse	To look for a file.
Build Rule Path	Applies the rule path algorithm to the entire design and creates a representation of the steel rule.
Builder	The component of ArtiosCAD which allows the user to create new designs based on standards.
Building a command	A message statement that informs the user that the digitizing command is not yet recognized.
Burn Name	A name to be inscribed by laser burn into the die board.
CAD-X	ArtiosCAD ActiveX control, a library of calls that will enable ArtiosCAD functions to be invoked from another program.
Calculated Expression	The text that is generated automatically from a predetermined formula.
Calculated Text	The tool that contains the options which affect the text (calculated expression) that is generated when printing a report.
Caliper (CAL):	The thickness of the type of board that is being used for this design.
CAM Device	A peripheral (output device) which outputs using a driver other than a Windows driver.
Carrier Rule	A steel rule used to support the sheet to prevent snagging when there are interior cutouts.
Case Expression	An action that is performed when conditions are met.
Chamfer	A tool that trims the edge of the counter with a gentler slope (except near creases) than the periphery tool.
Chamfer Gap	The distance between the inside edge of a chamfer and the crease channel in a counter.
Check Condition	An expression that limits the values that can be assigned to a variable (e.g. L>0). It is used in modules like StyleMaker.

Term	Definition
Chop Knife	The steel rule in a die that is used to cut up the scrap into smaller pieces.
Class	A category of the layer (main design, annotations, windows, cutouts, etc.).
Clear Extend	A tool that removes the handle points created by the Extend tool on the Extend/Measure toolbar.
Clearance	The space or distance between two items.
Clip Graphics	A tool to trim the extraneous part of a graphic away, by forming a clipping path.
Clip Path	A path (loop of lines) to cut along, by trimming the outside excess away.
CloseUp	A window showing a zoomed in view of the area surrounding the point clicked.
Coating blanket	A coating or thin layer of varnish used to protect those areas of the sheet not to be coated as the sheet moves through the printing press. Also, it can be called a varnish blanket.
Color Stock	A tool that adds a uniform color to all panels of the board design.
Color to Vectorize	An option which lets the user choose the Bitmap color that is converted to lines and arcs.
Colored	A tool which turns all hidden lines blue in 3D mode.
Compromise Factor	The ratio of the distance from the cut to the counter periphery. It is used when any cut and crease are too close together for the distances specified.
Conarc	Construction arc. An arc which is often drawn to aid in the actual drawing of the design.
Concatenate	An option which links text together as a series.
Condition	Composed of a type and the limit value for Cost Center expressions. The condition can be changed by using a modifier.
Conline	Construction line. A line that is often drawn to aid (guide) in the actual drawing of the design.
Contrast	The light/dark ratio (of graphics).
Control point	A point of reference to control the drawing of the curve between the start and end points. Two middle points are used as Control points in Beziers.
Convert Fills to Lines	An option to change fills (fills of color) to print image lines. It also places them in overlays whose names are the color of the fill. This option can be used when importing an EPSF file.

Term	Definition
Convert Strokes to Lines	An option to change strokes (graphical additions to lines) to print image lines. It also places them in overlays whose names are the color of the stroke and the thickness of the stroke in points. This option can be used when importing an EPSF file.
Corner angle sharpness threshold	“The parameter for determining how close a pin, block, or rule will travel into the narrow areas of the lower stripping board.
Corner Factor	A parameter that determines whether a corner is sharp enough to be considered the start of a new line.
Corner relief	The relief cut for a steel insert in a counter to make reverse creases or reverse cuts.
Counter	A special template made out of phenolic resin that aids in the production of creases by providing a solid surface. It is used in folding carton manufacturing.
Counter channel widths	A check box that when checked displays the actual paths made by the counter tools instead of just an outline.
Counter layout	A module of ArtiosCAD used to make many counters at once.
Counter Mill Outline	The outline of an area to be milled (cut out). It is used for reverse creases, reverse partial cuts, or for embossing.
Crease	A line indicating a fold in the carton or container.
Crease Channel Width	The width of the channel crease in a counter, measured at the top of the channel.
Cross Grain	The direction perpendicular to the grain (fiber) of the board.
Cume Dimension	A tool that displays the running total or accumulation of distances from a known point to an endpoint. Cume is short for cumulative.
Curve Straight	A tool that creates a curve followed by a straight line (defined by a start point, end point, angle, and radius for the arc).
Cut	A line indicting a cut, typically an outside edge or inside cut.
Database	Electronic storage (a place) where the information about the design and customer are saved.
Default	The setting of an option (property, variable, tool, etc.) that is used unless another choice is entered or the default is changed under Options. Also, Defaults are baseline settings.
Delimited	The separation of data by a comma or another item. This is a format option for the Digitizer.
Design Checks	A tool that checks the current design for double lines and gaps.

Term	Definition
Detail	A tool that allows selection of an area to be magnified. It also allows placement of the window which will show the magnification, called the detail window.
Device	A tab or dialog box referring to an output mechanism (e.g. printer).
Diagram	A documentation plot (small sketch) displayed in StyleMaker or when running a standard.
Diameter	2 * Radius, a measurement for a circle across the largest part.
Die	Any of the various steel rules used to cut, crease, or perforate Boxboard into a desired shape.
Die Cut	A cut made in some soft to semi-rigid material with a cutting die.
Die Press	A machine that holds and uses a die to cut material into pieces and parts.
Dieboard	A hard plywood used as the carrier for steel rule dies (e.g. stripping knives, balance knives, etc.).
Diecutter	A person who is employed in diecutting or operating a diecutting press.
Diemaker's Fix-it	A group of tools that check the design for small abnormalities and give the user the option of fixing them.
DieSaw	A saw used to cut the routed groove for the insertion of the steel rule. A Diesaw is used like a jig saw and has an attached drill to make starter holes for the saw blade.
Differential Scale	A tool that scales the selected items with different factors for horizontal and vertical directions.
Digitize	The process of transferring a design from a drawing or plot into a digital file using a digitizer.
Digitizer	An input device that normally consists of a flat tablet that the operator traces over with a pen like stylus or another cursor device. The patterns traced by the operator are automatically entered into the computer's memory for subsequent processing.
Dimension	A measurement of distance, angle, radius, etc. displayed.
Distance	A measurement from one point to another.
Double Knife Removal	A tool used to remove any knives that are too close together on a dieboard.
Drag Color	The color of the lines used to show the lines that will be created.
Drag Snap	A snap feature for dragging the cursor (mouse) that lets the user snap to specified angle and length increments. The cursor will round to the nearest snap increment when drag snap is on.

Term	Definition
Drill Hole	A hole made by drilling up and down in a one step process.
DWB	Designer WorkBench, a CAD program.
Dynamic	An option to automatically update the screen as values change.
Ear style	A group of options for the style of the ears (special flaps). They are created when running a standard with ears for a container or carton.
Edit	To change, typically text or geometry.
Edit Tools	Tools that help with the process of changing, deleting, coping, and/or processing.
Elevation	The vertical angle from which the design is viewed.
Ellipse	A tool that creates a set of arcs forming a loop defined by a length and width.
Embed dependent files	An option to embed the files that the selected file is dependent on. This option is in the information filter.
embedded design	A design that has been embedded in another file (e.g. designs saved in manufacturing files).
Emboss	A technique for impressing a design or texture into a sheet of material from the back of the sheet so that the design extends out towards the viewer.
Emboss Relief Width	The width of the relief from the half crease (crease on one side of the counter channel).
Enable subroutined output	An option that refers to support subroutines in the NC (numeric control) output. This option only works with the PUNCH command. Turning this checkbox off combines all elements of the layout into one set of data on output.
Enforced Check	A check that must be performed in Advanced StyleMaker. Unlike a check condition, an enforced check cannot be ignored.
DataCenter	A program which makes finding specific information about ArtiosCAD designs quick and easy by using a database structure.
Etch	A thin partial cut made with a laser die cutter or a mill tool. Typically, it is used for putting text on the die.
Export	To bring a file out of ArtiosCAD in a chosen format.
Expression	A mathematical statement.
Extend	To lengthen one geometry to meet another geometry (e.g. extend one line to meet another line).

Term	Definition
Exterior	A grouping of choices to determine the type of tooling for the outside (exterior) stripping of a dieboard.
External Varnish Offset	The offset (distance) that the varnish is to extend beyond the design.
Fast Block	A block type font that is designed to be burned into the die quickly.
Favor Horizontal	An option for when rule paths cross in which the vertical rule is broken and the horizontal rule is continued.
Favor Vertical	An option for when rule paths cross in which the horizontal rule is broken and the vertical rule is continued.
Fields	Data fields (specified spaces for information).
File Window	A window (pane) that displays pictures of flat designs, manufacturing files, print items, and 3D files.
Fill	A tool that fills or covers an area contained by a closed loop with the default color.
Filter	A method of sorting that uses selection criteria, typically it is used in database searches or data selection.
Fix All	A command to fix all of the problems that were found with Diemaker's Fix-it.
Fix minimum	To fit the sheet size to the minimum size defined by the die press or printing press selected.
Fixed Point	A point to which an item that the user is editing remains fixed (relative to).
Flat	A style of ending a stroke in which the stroke ends at the same point as the end of the object being stroked.
Flute (Corrugated)	The wave shaped formation of the inner component of corrugated fiberboard.
flute/grain direction	An indication of fiber direction (horizontal or vertical). The normal direction of flutes is parallel to the depth of the box, so that they are vertical when the box is stacked for shipment.
Fold 1 to Meet	A tool that changes the angle of a crease so that two lines meet.
Fold 2 to Meet	A tool that changes the angles of two creases so that two lines meet.
Fold All	A tool that changes the fold angles of all the creases in the selected design(s).
Fold Angle	A tool that changes the fold angle of the selected crease(s).
Follow	A tool that follows (parallels) the path of existing geometry with an optional offset.

Term	Definition
Freeform Editing	An option that allows the user to make changes to the edit log commands by typing over them. This option is for rebuilding the design and may affect the rebuild of the design.
Freehand Coordinates	An option which allows the user to snap to coordinates not on the grid.
Freehand Drag	To move objects by clicking and dragging them. There are two groups of tools to which freehand drag can be applied. The first group of tools is the annotations, dimensions, and text for which freehand drag is on by default. The second grouping is lines, arcs, beziers, etc.
Front waste	The remaining front trim after all stripping has been done.
Front Waste Separator	A tool that adds a separator to strip the waste from the lead edge of the layout.
Function Key	A shortcut key, a key which may be used to perform a task rather than clicking.
Geometry	Any item that is on a die press manufacturing tool. Dimensions, annotations, and graphics are not considered geometry.
Geometry Macro	A catalog of resizable shapes that can be added to the design. The shapes are composed of common geometry (lines and arcs).
Gets Device size	A tool that gets the size of the device for which the report is being made.
Ghosting Constraint	The restrictions on how the layout of the sheet may be done based on printing considerations, typically color sequencing.
Glue assist	A type of perforation rule which partially cuts the board to assist in embedding glue (similar to roughing a surface to hold glue).
Grain	In paper making, the direction in which most fibers lie. Normally, the grain corresponds with the direction that the paper travels through the machine.
Graphic	A picture or logo to be added to the design(s).
Greeting	A field in DataCenter used for the short name of a customer.
Gripper fingers	Metal fingers that clamp onto the board and control it’s flow as it passes through the press.
Gripper Image	A workspace that shows the placement of the gripper fingers that are used to pull the sheet through the press.
Group	A tool that makes ArtiosCAD treat more than one object as one object.
Group Layer Warning	A warning about the layer that a group of items is in (e.g. a group of items is not in your current layer).

Term	Definition
Group Output	Allows you to group (list) already defined (made) outputs to turn them into one output.
Guillotine	The piece(s) of wood that pushes out the external waste from the top and side of the sheet.
Gutter	The set distance between single designs in a layout of nested designs.
Half Crease (must form loop)	To crease on one side of the counter crease channel.
Half Cross Grain Channel Width	To adjust the channel width for the cross grain direction.
Hand Hold	A hole in the dieboard to aid in handling/carrying and storing.
Handle Point	A point used to grab and move an element. It can also be used to control an operation (e.g. copy rotate).
Header slot style	A group of choices for the slot style of a container. The choices are between full cut slot, 3 sides cut, and crushed.
Hole Pattern	A pattern of mounting holes in a dieboard.
Hook	A J-shape that can be formed at the end of the stripping rule. A hook is used to keep the stripping rule from moving in the dieboard.
HRMS/SCS transfer	To send information from ArtiosCAD to the Harry Rohdes Management System shop card server, a business system.
Image	A picture or graphic.
Import	To bring in a file or design from outside of ArtiosCAD.
Import as Graphics	An option to import strokes and fills unchanged when importing EPSF files.
Infold perf and score	The diagonal crease of the side panels to facilitate folding inward, typically on a 4 or 6 corner tray.
Information Enhancement	An option to synchronize some fields between ArtiosCAD and DataCenter.
Inner Diameter Subtype:	An input box that has a pull down menu that tells ArtiosCAD which manufacturing tool to use to create a hole.
Inner pin to pin distance	The density of pins in a large support or waste area.
In-Place Geometry Tool	A special advanced geometry tool for Manufacturing that runs within the context of the design (layout).
Inset count	In rotary rule, the number of teeth to skip before removing the first tooth for a bridge.

Term	Definition
Inset distance:	A user set distance for a bridge's position from the end of the rule.
Inside Loss (IL):	The distance lost on the inside measurement of a carton or container due to a fold in it. This loss is attributed to the thickness of the material being folded and the radius of the bend (fold) in the material.
Inside Side	The side of a carton or container that will be on the inside when the carton or container is folded, usually the unprinted side.
Intelligent Counters	A module of ArtiosCAD which easily creates counters for most designs.
Intelligent Layout	A module that performs automatic sheet layout calculation.
Interference	Also called Power Stripping. The process of adding a little bump into the outline of a hole. It is used to prevent things from clogging or jamming the hole due to suction.
Interior	A grouping of choices to determine the type of tooling for the inside (interior) stripping of a dieboard.
Intersection Point	The point where two geometries (e.g. lines or arcs) meet.
Invert Selection	An option which selects only those items that are unselected.
Item reference tool	A Diagnostic tool to display the full path name of an item on the design.
Jigged	An option for creating a separator wood edge in manufacturing (I.e. to create using a jig saw).
Join	A group of choices about how strokes that meet are connected (e.g. join by bevel).
Kerning	The amount of space ArtiosCAD inserts between letters or text.
Keypad	A display of a keypad on the monitor to enter values, variables, and expressions.
Land	The space between the start or end of a zipper cut and the next intersecting line.
Laser	An output type which makes laser setup available. It is also a method of manufacturing. It is also a line type subset.
Layer	The separate levels of design information grouped by functionality (e.g. Main Design, Dimensions, and Annotation).
Layout	A design or multiple designs stepped and repeated one or more times.
Lead -> Trail	bottom->top, an option to work the rule path from the bottom to the top of the dieboard.
Lead edge	The first portion of the blank to come in contact with the die.
Leader Hole	A starter (pilot) hole in which to insert the diesaw's dieblade.

Term	Definition
Leader Hole Tools	A set of tools for making starter (pilot) holes in the dieboard for the diesaw.
Length of Line	A button on the keypad that allows the creation of a new line using the same dimension as an existing line.
Library Functions	Any predefined function (set of instructions).
Licenses	The permissions to use a program or piece of a program. Certain restrictions and responsibilities apply.
Light source	A tool that changes the light source's angle and elevation in the 3D Design.
Line - Gap Sequence	A tool that turns a line - gap sequence into a line with bridges.
Line Type	The category (type) of a line (I.e. cut, crease, partial cut, etc.).
Liner	The non-corrugated part of the corrugated board, typically the inside and outside of the board.
Localization	The act of converting a version of software from the original language it was developed in to the language and conventions of another country.
Lock style	A group of options for the style of the lock (tab to keep the carton or container closed). It is created when running a standard with a lock for the container or carton.
Log Command	An individual command in the Edit Log (Logfile) that is used in rebuilding the design.
Logfile	The file of commands which were used to build the design. Also, a history file containing the history of the designs that have been autoloaded.
LWD values cannot be negative.	An error message meaning that length, width, and depth values cannot be negative.
Main Design	A layer for the design.
Make Counter	An option that sends the Counter information in the Counter Layer to a file which can then be interpreted by a samplecutter. A Counter is also called a Matrix.
Manufacturing	The module of ArtiosCAD that makes production tools from single designs.
Mapping	The transformation from one set of attributes to another set of attributes used when importing DXF files.
Measure Tool	A tool that measures the length, angle, radius, and the x and y coordinate.

Term	Definition
Medium	The corrugated part of the corrugated board, typically the middle of the board.
Merge Line Straight	A tool that merges lines into a single straight line.
Mill	To cut out or remove material using a milling bit.
Mill Hole	A hole made by milling (cutting) up, down, sideways, and/or in a spiral pattern.
Mill outline	A line type to represent the area to be milled.
Milling Tool	A tool that cuts an area or outline. It is like a drill, but has a sharp outer edge (flute) to allow a horizontal moving cut.
Miter	An option for joining strokes (graphical additions to lines) that makes the inside and outside angles the same.
Mounting Bar	A bar to mount the die wood to the die press.
Mounting Hole	A hole to fasten the manufacturing tools to the press.
Move	A tool that moves a selected item.
Name Tool	A tool that etches an identifying label into the counter.
Nest	A tool that takes a single design and steps and repeats it tightly. Nesting is used to serve as a pattern for die making.
Nick	A little piece of material connecting individual items that doesn't get cut in order to keep the sheet together.
Notch	A cut (notch) in the dieboard edge for registration and to help line up the dieboard and machine.
Number Expression	A text string comprised of numbers, called number text. This type of text can be used when editing an expression in Advanced StyleMaker.
Number up	The total number of times that a certain design is on the layout.
Oblong Hole	A hole stretched in one direction.
Off the Counter	An option referring to lines, cuts, and creases not being on the actual counter (I.e. when the shape of the counter does not include these lines.)
Offset	A spot or point that is displaced, or offset, with respect to another point.
On center of tooth	An option to have bridging start in the middle of a tooth on a rotary rule.
Oneup	A single design that is part of a manufacturing layout and is usually used for creating multiple or nested designs.
Open-out	A small slit cut in the corner of a rubber piece that allows it to open flat in a rubber layout for better nesting.

Term	Definition
Optimize	A process to make a method/selection (e.g. lines or rules) efficient. It is used in Double Knife Removal, Counter Cutting Order Tool Path, etc.
Out from die split	An option to work out from the edges of the die split when ruling a rotary dieboard.
Outfold perf and score	The diagonal crease of the side panels used to facilitate folding of the panels outward, typically on a 4 or 6 corner tray.
Outline Text	A tool that converts text into print image lines.
Output	The function in ArtiosCAD where reports, documents, samples, and data to drive the equipment is produced.
Output Layer	A layer that is a composite of the layers that are turned on. This layer is used for inspecting and editing the output.
Output to Layer	To output to a specific (chosen) workspace/design layer.
Outside Gain (OG)	The distance gained on the outside measurement due to a fold in a carton or container. This gain is attributed to the thickness of the material being folded and the radius of the bend (fold) in the material.
Overrun	Finished product quantities created in excess of the specified quantity.
Overwrite	To replace a file or something that already exists.
Pane	A window.
Panel	A side of a box.
Parametrics	The technique by which a design is defined by equations based on some independent variable and is used when rebuilding a design.
Peripherals	Auxiliary physical devices connected to a computer system (I.e. plotter, printer, and mouse).
Periphery	The edge of the counter.
Periphery Tool	A tool that cuts out the counter with a beveled edge.
Perspective	A tool that changes the amount of perspective (depth and width) in the 3D module.
Playback Tool	A tool that rebuilds a design step by step while looking at the sequence of the design as it is rebuilt.
Plot	A type of output in ArtiosCAD, typically a large printout.
Plotting Style	The appearance of the graphical attributes of all objects. A description of the way that various types of lines are being represented.
Pointage	The thickness of rules and lines.

Term	Definition
Post Run Geometry Tool	A tool to modify the current manufacturing die board.
Power stripping	Interference, the process of adding a little bump into the outline of a hole so that the waste stays where it is supposed to and doesn't come back into the press.
Press	A machine which performs the mechanical action of diecutting or printing.
Previous Fold Angles	An option in 3D to fold the design using the angles by which the design was last folded.
Prompted text	A type of text inserted into a report to have the report prompt you to fill that text in each time the report is printed.
Prompted Variable	A variable that the user is asked to enter a value for. Prompted variables are used in Reports and Cost Center.
Properties	The attributes or characteristics of an item.
Property Defaults	Typically startup items, they can change properties of an object during a program session.
Property Page	The page or window that displays the characteristics of a selected item.
Punch drivers	Software routines that control the punching out of material during the diecutting process.
Push Pins	A tool for stripping, usually metal, used to push out waste.
Pusher active part	The top piece of a push pin which actually pushes the material away.
Pusher base	The base of the push pin which is set into the dieboard.
Pusher clearance	The distance that a push pin is from any other object.
Pusher shoulder	The larger part of the push pin that keeps the push pin in place by not letting it fall through the die board.
Radius Sweep	The value of the angle an arc travels through. Half of a circle would be 180 degrees, and a quarter of a circle would be 90 degrees.
Rebuild	A tool that allows the building of a design that is based on variables so that it's size can be changed.
Rebuild Conflicts	A dialog box that warns of design problems that arise while rebuilding a design.
Rebuild Expression Tree	A command button to update a conditional statement. It is available in Advanced StyleMaker after editing the Current Expression dialog box.

Term	Definition
Rebuild Playback	A tool that rebuilds a design step by step while looking at the sequence of the design as it is rebuilt.
Record	A collection of related data items used in DataCenter.
Recovery Options	A group of choices on how to proceed with recovering a design after Rebuild (Design) has failed.
Reference Point	A point of origin.
Refresh	To redraw the active view.
Registration Hole	Mounting hole, a hole to fasten the manufacturing tool to the press.
Registration Marks	Marks where a registration hole will be made
ReInit	A function that resets a variable to its default value in a design
Reinit All	A function that resets all variables to their default values in a design
Relief Profile	The shape of the relief (extra cut out material) to be used to avoid squeezing the carton or container as it is pressed against the counter.
Relief Slot Base	The width at the bottom of the relief (extra cut out material) slot for ending a crease in a counter.
Relief Slot Opening	The width at the top of the relief (extra cut out material) slot for ending a crease in a counter.
Relief Slot Overcut	The distance between the end of the crease and the relief slot bottom. The relief is extra cut out material.
Report	A form with pictures of workspaces along with information about these workspaces.
Report Maker	A module that allows the user to create customized reports.
Reset	A command that sets values back to their original values (usually 0).
Resource	A directory tied to the database.
Rotary	A checkbox in manufacturing that informs ArtiosCAD that rotary rule is going to be used.
Rotary Rule	A rule that is built curved to be used on a rotary die press.
Round Limit	The accuracy with which to round. It is used to set the accuracy for the rounding function.
Round slot method	A method of dealing with sharp corners, so tearing doesn't occur.
Routed	An option when cutting a sample counter to route (grind away the material in) the crease channel.

Term	Definition
Rule	A steel blade on the die board which can create a variety of lines on a sheet, including cut and crease lines.
Rule Mapping	To map a particular rule size and type of your preference to a custom rule.
Rule Path	The groove that the rule follows in the cutting die.
Run a Standard	Loads a previously designed ArtiosCAD rebuildable design.
Safe separation	The minimum distance two lines must be from each other to be ruled individually.
Sample	A prototype (sample) for customer approval.
Samplemaking tool	A tool to cut a prototype (sample) for customer approval.
Save As Revision	A command that saves the active document as a new revision.
Scale	To change the size or proportion of an item.
Scallop	A rule type describing the scalloped shape of the rule. It is used to perforate a carton.
Scrap	Waste, extra material of no use.
Scrap rule	Also called chop knives, a steel rule used to cut up waste into small pieces.
Scrapped overrun	The extra items (e.g. cartons or containers) which the customer will not accept.
Send to Back	A tool that places the selected graphic behind all the other graphics.
Send to Front	A tool that places the selected graphic in front of all the other graphics.
Separator Board	Tooling to remove the front waste or trim.
Shared defaults	The defaults used (shared) by all the users who use the same ArtiosCAD server.
Sharp angle threshold	The parameter for determining how close a pin, block, or rule will travel into narrow areas of a lower stripping board.
Sheet	The raw material placed into a die press to be made into the final product. Also, it describes the individual piece (sheet) of container or carton board.
Side bevel	A type of cut rule that creates a bevel. More specifically, the serrated or flat edge cut rule that has points or flats to one side of the cross section as opposed to having points in the middle.
Single pass creasing	A process in which the creasing tool passes over the material once to make a crease.

Term	Definition
Size Threshold	Sets the size limit of items to vectorize.
Slot	A cutout in a carton or container. In Manufacturing, when representing the Die, the word slot refers to an option for the user to specify a nonstandard width (pointage) for the slot where the rule will be placed.
Slot style	A group of options for the style of the slot (opening or indent for a tab). It is created when running a standard with a slot(s) for the container or carton.
Smooth Corner Parameter	A parameter that determines whether a corner is sharp enough to be considered the start of a new line.
Smoothing Threshold	Controls the jaggedness of the new lines created when vectorizing a graphic.
Snap	An option that allows an exact placement of an object by set proximity.
Special Rule	A specialty rule or extra generic rule that has been defined by the user.
Split Line	A tool that separates a line at a point.
Split Rule Path	A tool that separates the specified rule path.
Standard	A box or folding carton design that is rebuildable and is located in the standards catalog.
Status Bar	A bar located at the bottom of the window where information that is pertinent to the user is displayed.
Step Distance	The distance between echelon rules.
Stock Color	The uniform color for all panels of the design.
Stop Bits	The last bit of framing for a byte of data, in asynchronous serial communication.
Straight Curve	A tool that creates a straight-curve by end point, angle, and radius.
Straight Nest	To nest (step and repeat a single design) without losing the orientation of the design (I.e. without rotating the design).
Straighten	A tool that replaces an arc or bezier with a straight line
Stretch Point	Moves a point and the lines, arcs, and beziers ending at the point
Strip rule	A steel rule that cuts waste into smaller pieces for easier removal. It is also a rule that aids in the removal of scrap during the manufacturing process.
Stripping	The removal of all scrap material from a cutting die.
Stripping board	A board that performs the removal of waste.

Term	Definition
Stripping Edge	The working area in which to place the stripping rule.
Stripping pins	Pins that are made of a base, shoulder, and active part. They are used to push out waste (scrap material).
Stroke	Adds a ribbon of color around the edge of a line (graphical addition to a line) or group of lines selected.
Structural Orientation	A dialog box to set or change the side of the design, the grain/flute direction, and the die side.
Style	A grouping of choices as to how something is done or represented.
Style Catalog	An organization mechanism (catalog) for styles also called Standards. Standard carton and container designs are stored in defaults.
StyleMaker	The module of ArtiosCAD that allows the user to make rebuildable designs.
Subtype Mapping	A way of differentiating rule or line types (mostly for manufacturing).
Support Bar	The bottom support(s) for the lower stripping board.
Support Holes	Holes by which the support bar is attached to the bottom stripping board.
Support Pins	Pins that consist of a base and a shoulder. They are used to hold up the block (wood block and guillotine).
Symbol	A graphic or graphic file in ArtiosCAD.
Synchronize	A command to bring the data in ArtiosCAD into agreement with the data in DataCenter.
Tack Bridge	A small gap in the edge of the material being cut to help hold it together. For example, a tack bridge in the wood die is used to keep the waste pieces from falling out into the laser bin.
Tack Bridge in Bowtie	A gap in the periphery to hold the bowtie while the counter is being cut.
Tangent tolerance	The number of degrees between two tangents. It is used to determine the best sequence of cutting.
Taper	Used to mean angle, the angle used to taper off the reverse crease in a counter.
Tear angle	The smallest angle allowed before considering that a fracture or rip may occur at that angle, which could require changing the sequence of cutting.
Tear distance	The smallest distance allowed across a small piece of material at a specified distance from the end of the material before considering that the material may rip or fracture, which could require resequencing the cutting order.

Term	Definition
Template	A pattern that is used to assist in formatting data (e.g. a filename template used for building a list of files).
Text Library Function	An ArtiosCAD command file that returns a text variable.
Text Position	The position of the text on a design.
Thumbnail	With reference to images, it is a smaller version of an image used as a preview.
Tile	To split a drawing (output) over one or more pages, when it is larger than a page or after it has been scaled to a size larger than a page.
Toggle	To switch back and forth.
Tolerance for gaps	Used in design checks, it defines the maximum gap that will be ignored when finding the perimeter of a design.
Tooling	Anything physically made from the manufacturing process or CAM output.
Trailing	Being the last thing following a path with regards to rule path direction.
Trim	To shorten a line against another line.
Trim Against Selection	A tool that shortens or extends a line or arc against the selection of another.
Trim Interior	A tool that cuts out the interior section between two other geometries (e.g. lines).
Trim/Extend One Line	A tool that shortens or extends a line or arc to meet another line or arc.
Trim/Extend Two Lines	A tool that shortens or extends two lines to a corner.
Tune File	A file containing parameters that modify and control the way a driver produces data or an import process interprets data.
Underrun	Number of finished products less than the specified quantity required.
User Defaults	The settings under defaults that the user has established.
Userfield	Customizable fields for designs and manufacturing files that can contain almost any kind of information.
Value	A quantity that is associated with an item.
Variable	A character or a string of characters up to six, representing a quantity whose value can be altered (changed).
Variable Conflicts	A dialog box that informs the user that there are problems because of illegal actions on variables. These problems are usually caused by moving or deleting variables that have dependent relationships.

Term	Definition
Varnish	A thin, protective coating applied to a printed sheet for protection or appearance.
Varnish Layer	The layer where the varnish is applied.
Varnish tool	A tool that creates varnish outlines of selected panels, with separate internal and external offset values.
View Angle	A tool that changes the angle and elevation at which a design is viewed in ArtiosCAD 3D.
View Mode	A tool that controls the choice of what is shown (lines, graphics, annotations, etc.) and the representation (plotting style) of how it is shown.
Wave	A rule type describing the wavy shape of the rule. It is used to perforate a carton.
Wood block	Any piece of wood used to push out waste during interior stripping.
Wood Edge	The outside edge of any board, the outside geometry of the tooling.
Workspace	A generic name for design files and/or the current design/work area.
Zigzag	A rule type describing the zigzagged shape of a rule. It is used to perforate a carton.
Zipper	A rule type describing the zippered shape of a rule. It is used to perforate a carton.
Zoom In	To increase the magnification from the center of the selected area.
Zoom Out	To decrease the magnification from the center of the screen in order to increase the viewing area.