ArtCAM

# User Guide

*Issue: 2010.SP3 (build 472) Released: 03/07/2010*

## Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

## Patents

The functionality of preparing a 3D relief of the side profile of a human face direct from a photograph in ArtCAM Pro is subject to a patent.

Patent No: GB 2 403 883 "Photo to 3D"

The functionality of the 3D layers used to design and machine an article in ArtCAM Pro are subject to a patent application.

Patent application: GB 0600873.4 "3D Layers"

ArtCAM 2010 SP3. Published on 3 July 2010.

# Contents

# The 2D design process 96



# The 3D Design Process 166

# Introduction

ArtCAM is a unique software program which enables you to create impressive, high quality three-dimensional products starting out as two-dimensional bitmap or vector-based artwork. ArtCAM transforms ideas into finished products far more quickly than is possible using conventional methods. Even in cases where a hand-finished look is desired, ArtCAM accelerates production. Using ArtCAM and a machine tool or router to machine most of the job leaves you more time to concentrate on the original design and the fine details, which together help to differentiate products in today's competitive markets.

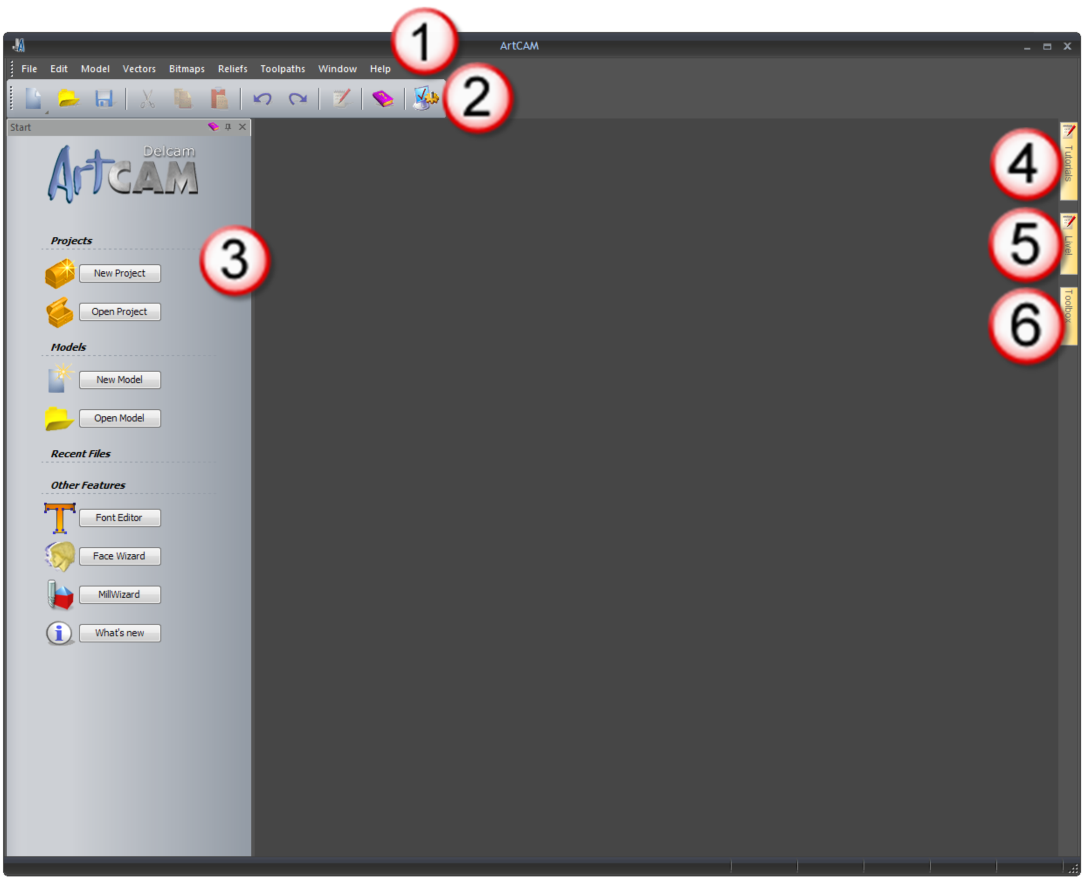
# The ArtCAM interface

Exactly what is included in your ArtCAM layout depends on whether or not you have a model or project open.

*If you are working in ArtCAM Insignia, you cannot create or open a project.*

## *The starting layout*

When you start ArtCAM, the layout includes six key areas:



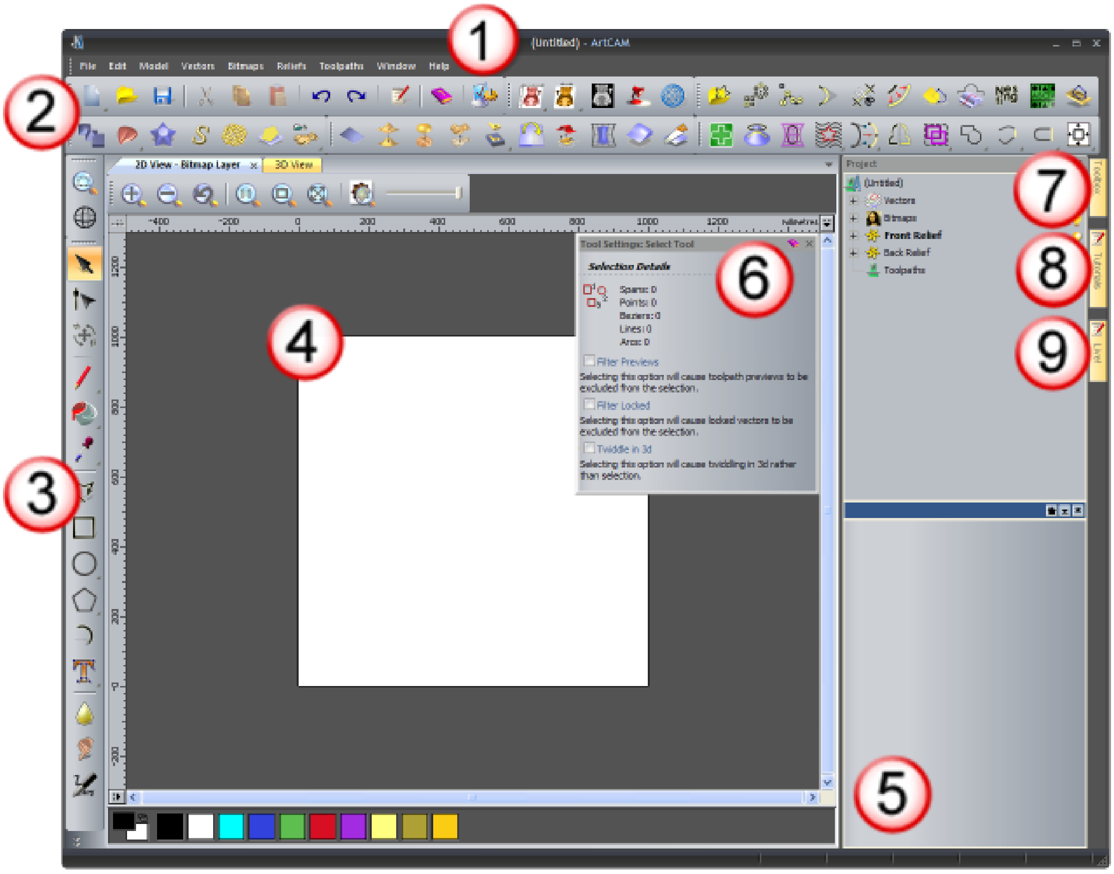
1. **Menu Bar** - Select a menu item associated with a particular part of the ArtCAM's design process to display a drop-down menu of sub-menus and commands. Most of these are inactive until you create a model or project.
2. **Top docking area** - The **File** toolbar is docked horizontally here. This enables you to create a new project or independent model, or open a previously saved project or independent model. You can also set your preferences and display the Reference Help.
3. **Start** - This docked panel enables you to create a new model or project, or open a previously saved model or project. Your recent files are listed and can be reopened. You can also start the **Font Editor** and **MillWizard** standalone tools, or use the **Face Wizard** tool to create a new model from a photographic image containing a side profile of a person's head and neck.

   *When a model or project is created or opened, the **Start** panel is hidden.*

   *If you are working in ArtCAM Insignia, the **Projects** area and **Face Wizard** tool are not included.*
4. **Tutorials** - This tab displays a panel that provides information on where to download tutorial packs. Tutorial packs included in your installation are listed.
5. **Live!** - This tab displays a panel that provides online videos you can watch to learn about working in ArtCAM.
6. **Toolbox** - This tab displays a panel that contains compatible 'plug-in' tools. Plug-ins included in your installation are listed.

## *The open model layout*

When you create a new, independent model, the layout includes nine key areas:




1 **Menu Bar** - As described previously, although more sub-menus and commands are active.

2 **Top docking area** - Seven toolbars are docked horizontally here: **File**, **Model**, **Bitmap Tools**, **Vector Creation**, **Relief Creation**, **Vector Editing** and **Relief Editing**.

Up to four additional toolbars are hidden by default: **Simulation**, **Rotary Relief Tools**, **Gem Tools** and **Back Relief Tools**. When displayed, these are also docked.

*If you are working in ArtCAM Insignia, the* ***Rotary Relief Tools****,* ***Gem Tools*** *and* ***Back Relief Tools*** *toolbars are not available. If you are working in ArtCAM Pro, the* ***Gem Tools*** *toolbar is not available.*

*Almost all of the toolbars' tools are also available from the* ***Assistant*** *panel, which is hidden by default so to maximise the viewing area. When displayed, this panel is docked on the left.*

3 **Left docking area** - The **View Manipulation** and **Design Tools** toolbars are docked vertically here. The former enables you to twiddle and zoom, while the latter contains four tool groups: modes, painting, drawing and sculpting. These include many of the most frequently used tools in ArtCAM.

4 **Viewing area** - The viewing area contains two types of tabbed windows: the **2D View** and **3D View**. These display the layers of two-dimensional vector and bitmap artwork in your model. The **3D View** window also displays the relief layers containing your three-dimensional shapes, calculated toolpaths and toolpath simulations. Each window has a toolbar and colour palette.

5 **Project** - This docked panel displays the Project Tree and tools associated with its items. The Project Tree maps the different elements of your model. Its items provide context menus that enable you to create models, layers, triangle meshes, assemblies and replica meshes, as well as calculate, edit and simulate toolpaths.

*The most frequently used tools from the **Toolpaths** panel, which is hidden by default, are available from the **Project** panel. When displayed, the **Toolpaths** panel is docked on the left.*

Up to five panels hidden by default: **Toolpaths**, **Relief Layers**, **Bitmap Layers**, **Vector Layers** and **Assistant**. When displayed, each panel is docked and pinned. The layers panels are also tabbed.

*If you are working in ArtCAM Insignia, the **Relief Layers** panel is not included.*

6 **Tool Settings** - This floating panel displays settings associated with a selected tool that requires direct interaction with the **2D View** or **3D View** window. The selected tool's name is shown on the panel's header. For example, **Tool Settings: Select Tool**.

*When you select a tool that requires little or no interaction with the **2D View** or **3D View** window, its settings are typically displayed on their own floating panel. The selected tool's name is displayed on the panel's header. For example, **Two Rail Sweep**.*

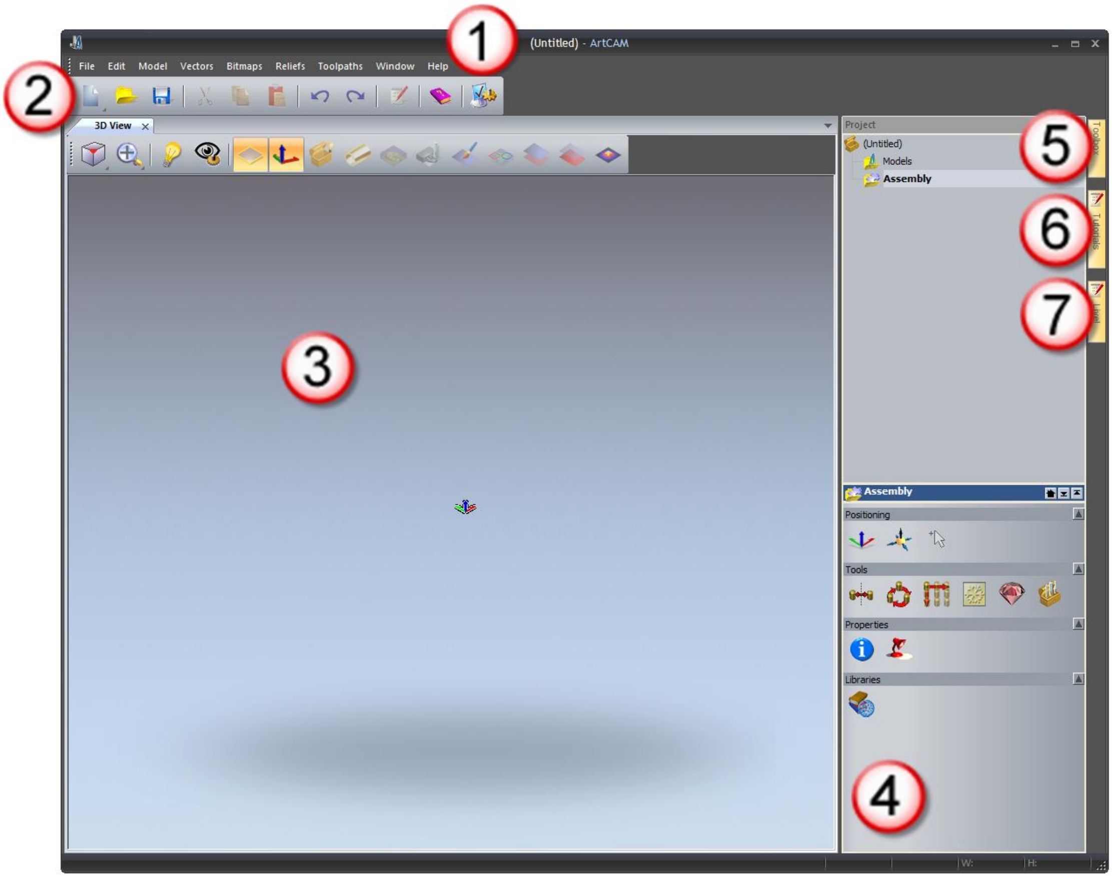
7 **Toolbox** - As described previously.

8 **Tutorials** - As described previously.

9 **Live!** - As described previously.

## *The open project layout*

When you create a new project, the layout includes seven key areas:




*If you are working in ArtCAM Insignia, you cannot create a project.*

1. **Menu Bar** - As described previously, although fewer sub-menus and commands are active compared to when a model is opened.
2. **Top docking area** - The **File** toolbar is docked horizontally here.
3. **Viewing area** - The viewing area is filled by a tabbed **3D View** window. This displays the assemblies and replica meshes included in your project. When a model is opened as part of your project, in the **3D View** window, you can toggle between the display of your project's assemblies and replica meshes, as well as the model's relief layers, calculated toolpaths and toolpath simulations.
4. **Project** - As described previously, although the Project Tree displays only two items: one that enables you to import ArtCAM models, triangle models and surface models as assemblies and replica meshes, the other to create and import ArtCAM models.
5. **Toolbox** - As described previously.

6 **Tutorials** - As described previously.

7 **Live!** - As described previously.

# Understanding the framework

The ArtCAM framework comprises a:

- **Title Bar**;
- docking area on all four sides;
- a viewing area; and
- **Status Bar**.

The framework:

- displays information about what you are working on and where;
- controls how and where docked toolbars are positioned;
- contains a tab for each auto-hidden panel; and
- affects the size of the viewing area.

## *The Title Bar*

The **Title Bar** enables you to close or resize ArtCAM, and displays information about what you are working on and where.

From left to right, the following is displayed along the **Title Bar**:

- a control icon, which displays the control menu;


- the project's name, provided a project is open;
- the model's name, provided a model is open independently or as part of a project;
- the product's name;

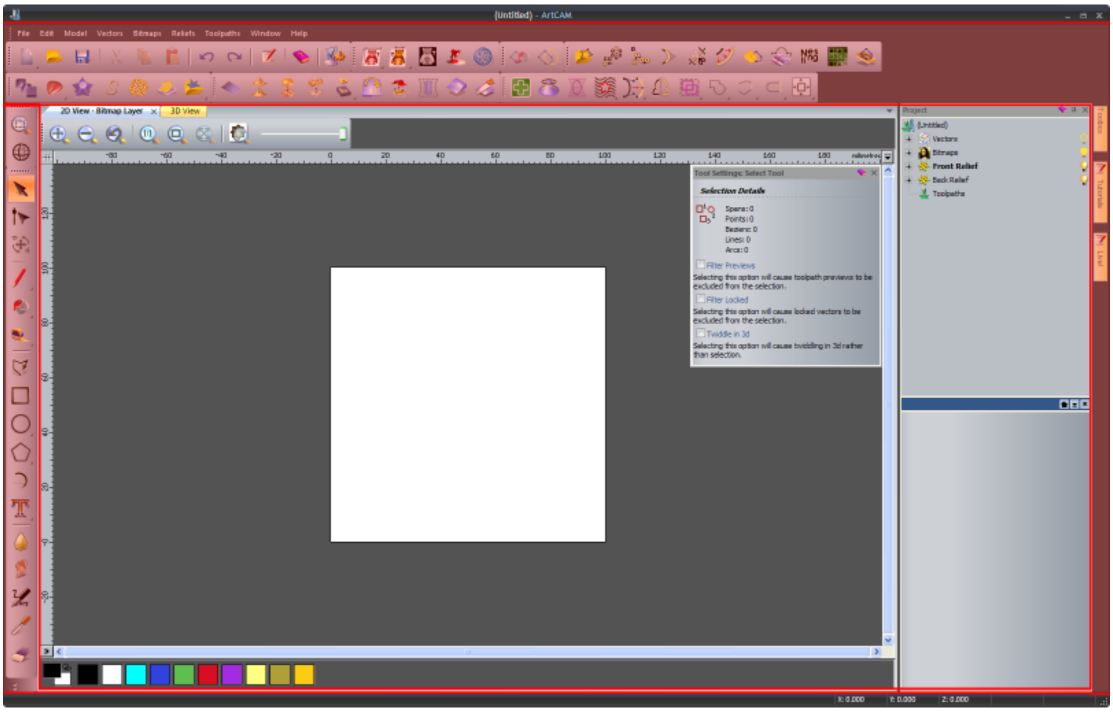
- the active view's name, provided a cascaded or tiled **2D View** or **3D View** window is maximised;
- the currently active bitmap layer's name, provided a cascaded or tiled **2D View** window is maximised;
- a **Minimise** or **Maximise** button, depending on whether or not fullscreen mode is used;
- a **Restore** button; and
- a **Close** button.

## *The docking areas*

There are four docking areas, and these are located:

- between the **Title Bar** and viewing area;
- to the left of the viewing area;
- to the right of the viewing area; and
- between the **Status Bar** and the viewing area.

The following image shows the ArtCAM layout when a new model is created. The docking areas are highlighted in red:



When you start ArtCAM or create a project, the top docking area contains:

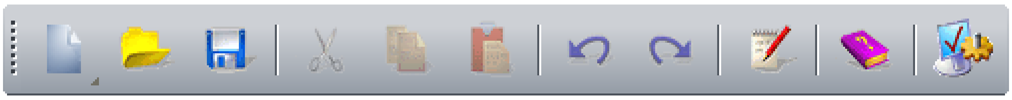
- the **Menu Bar**, which is docked horizontally below the **Title Bar**; and


- the **File** toolbar, which is docked horizontally below the **Menu Bar**.

When you create a model, the top docking area contains:

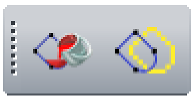
- the **Menu Bar**, which is docked horizontally below the **Title Bar**;
- the **File**, **Model**, **Bitmap Tools**, **Vector Creation**, **Vector Editing**, **Relief Creation** and **Relief Editing** toolbars, which are docked horizontally below the **Menu Bar**;

  **File** *toolbar*

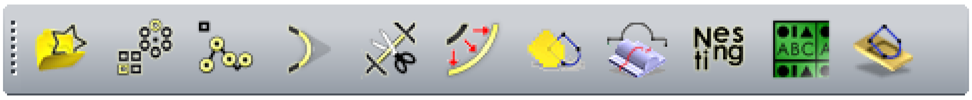


  **Model** *toolbar*

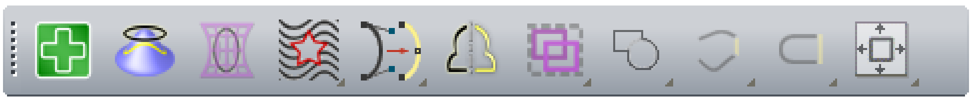
  **Bitmap Tools** *toolbar*



  **Vector Creation** *toolbar*



  **Vector Editing** *toolbar*

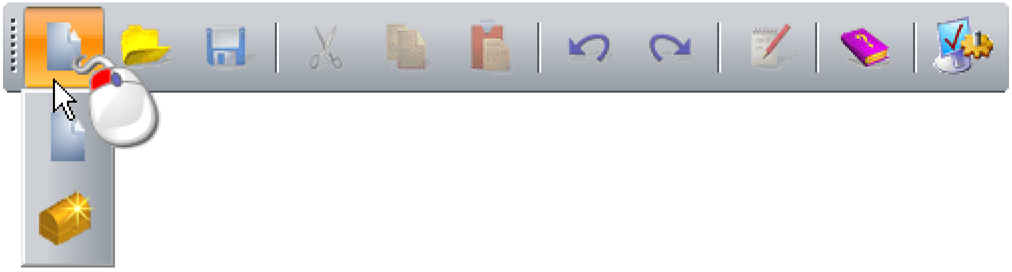


  **Relief Creation** *toolbar*

  **Relief Editing** *toolbar*

*If you are working in ArtCAM Insignia, several tools are not included in these toolbars.*

Many toolbars include toolsets, which are identified by ◢ in a button's bottom right corner. For example, the **File** toolbar includes the toolset shown below:



A toolset's most recently selected button is displayed in the toolbar. The other buttons are hidden.

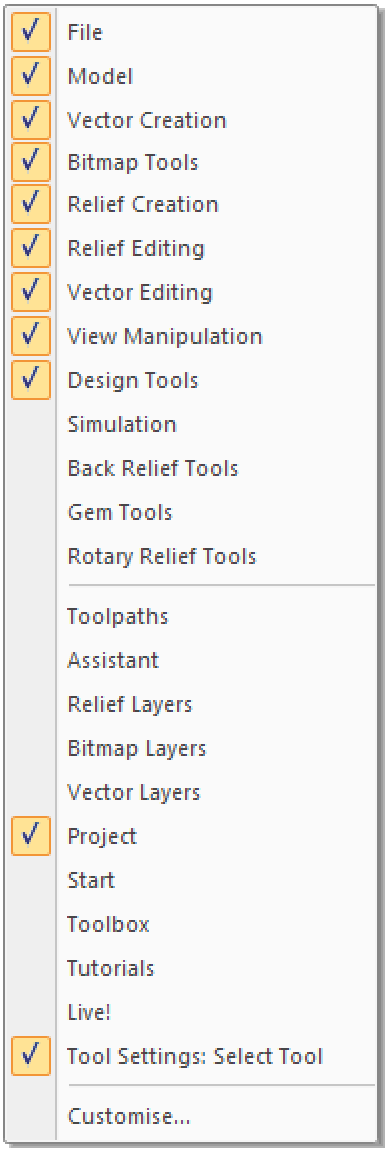
If a complete toolbar cannot fit within a docking area:

- is displayed at the end, when the toolbar is horizontal; and
- is displayed at the bottom, when the toolbar is vertical.

Click this to display the toolbar's other buttons as a list of options.

For example, when the **Transfer** tool is not displayed in the vertical **Design Tools** toolbar, it is included in the options list:

Docking areas, toolbars and auto-hidden panels share a common context menu that you can use to control what toolbars (see page 309) and panels (see page 302) are displayed:

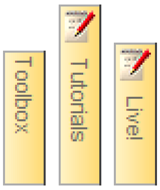



*If you are working in ArtCAM Insignia, the* ***Rotary Relief Tools****,* ***Back Relief Tools*** *and* ***Relief Layers*** *options are not included. If you are working in ArtCAM Pro, the* ***Gem Tools*** *option is not included.*

Pinned toolbars and panels are selected . The top half of the context menu comprises toolbar names, and the bottom half comprises panel names.

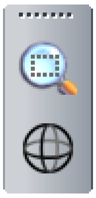
When you start ArtCAM or create a project, the right docking area contains:

- the **Toolbox**, **Tutorials** and **Live!** panel tabs (see page 299);

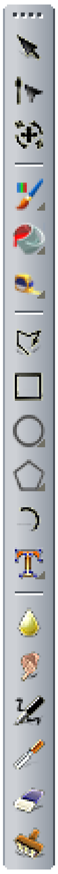



When you create a model, the left docking area contains:

- the **View Manipulation** toolbar, which is docked vertically:

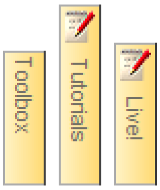


- the **Design Tools** toolbar, which is docked vertically;



*If you are working in ArtCAM Insignia, the bottom six **Sculpting** tools are not included in the **Design Tools** toolbar.*

When you create a model, the right docking area contains the **Toolbox**, **Tutorials** and **Live!** panel tabs, which are vertical:




## *The viewing area*

When you start ArtCAM, the viewing area is empty.

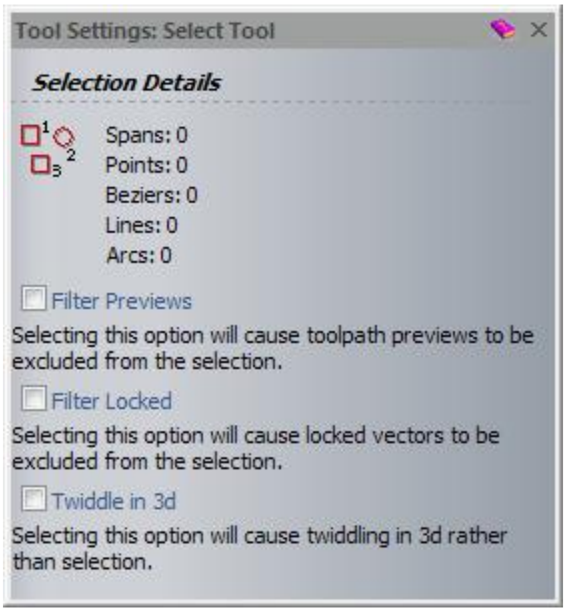
When you create a model, the viewing area contains:

- the **2D View** and **3D View** windows, tabbed with the **2D View** window active; and

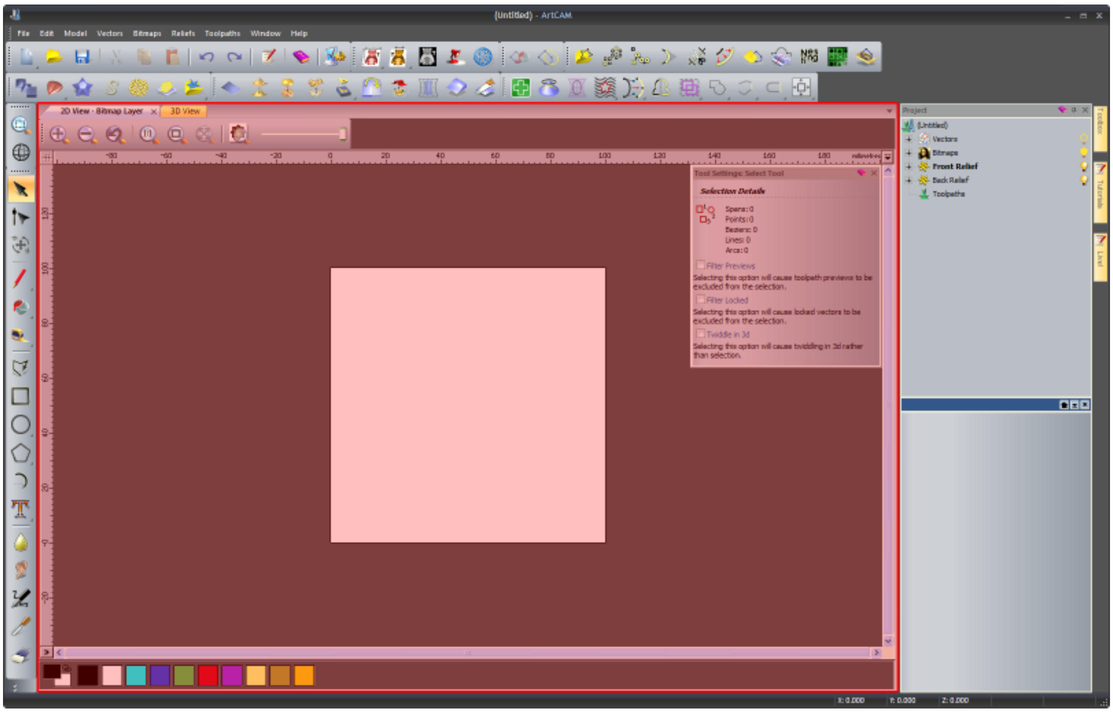
- the floating **Tool Settings** panel, which is positioned in the top right.

  On the **Tool Settings** panel, the **Select** tool's settings are displayed by default:

  *The **Tool Settings** panel is floating by default so that it is clear how certain tools use it. You can toggle its display using the **F6** key, and, like all panels, it can be docked (see page 296) and auto-hidden (see page 299).*

The following image shows the ArtCAM layout when a new model is created. The viewing area is highlighted in red:



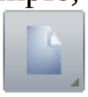
When you create a project, the viewing area contains the **3D View** window, tabbed and active.


*If you are working in ArtCAM Insignia, you cannot create a project.*

## *The Status Bar*

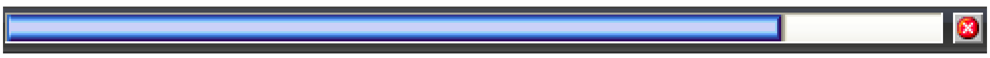
From left to right, the following are displayed along the **Status Bar**:

- a tool description, when the mouse cursor is over a toolbar button or menu;

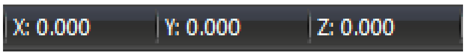
  For example, when the mouse cursor is over the **File** toolbar's **New Model**  button, *Create a new model* is displayed in the bottom left:

- a progress bar, when toolpaths and three-dimensional shapes are being calculated;

- the mouse cursor's XYZ coordinates, when the mouse cursor is over the **2D View** or **3D View** window;




- the bounding box's height and width, when vector artwork or toolpath previews are selected; and


- a resizing grip.

# Understanding the Start panel

On starting ArtCAM, and when no model or project is open, the **Start** panel is docked and pinned on the left. The panel is divided into four areas: **Projects**, **Models**, **Recent Files** and **Other Features**.

*If you are working in ArtCAM Insignia, the **Projects** area is not included. This means that the **Start** panel has only three areas.*

The **Projects** area enables you to:

- create a new ArtCAM Project (see page 89); or
- open an ArtCAM Project (see page 92).

The **Models** area enables you to:

- create a new ArtCAM model of a specific size (see page 57); or
- open a compatible file as an ArtCAM model (see page 61).

The **Recent Files** area enables you to open one of the last four ArtCAM Model files (***.art**) or ArtCAM Project files (***.3dp**) you have been working on.

*If you are working in ArtCAM Insignia, the last four ArtCAM Model files (***.art**) are listed.*

The **Other Features** area enables you to:

- create new fonts or modify them;
- create a relief in the shape of a face from a photographic image (see page 69);

  *If you are working in ArtCAM Insignia, this option is not included.*

- open MillWizard; or

- find out about the latest features and enhancements (see page 54).

When you create or open a project or model, the **Start** panel is closed (see page 302). The space previously occupied by the **Start** panel is then used by the viewing area.

# Understanding the Project panel

The **Project** panel is the hub of working in ArtCAM, and this is why it is the only docked and pinned panel displayed when you are working with a model or project.

The **Project** panel:

- is docked and pinned;
- is divided into two areas by a splitter bar;
- contains the Project Tree; which maps the structure of your ArtCAM projects and models; and
- provides access to a collection of context tools.

The layout of the **Project** panel is controlled by its splitter bar, which separates the Project Tree and its context tools:

The area above the splitter bar contains the Project Tree, while the area below displays all context tools and information associated with the currently selected item in the Project Tree. The name of the currently selected item in the Project Tree is displayed on the splitter bar.

To control what is shown on the **Project** panel, click:

to display the context tools and information associated with the currently selected item in the Project Tree;

to maximise the area below the splitter bar, and hide the Project Tree;

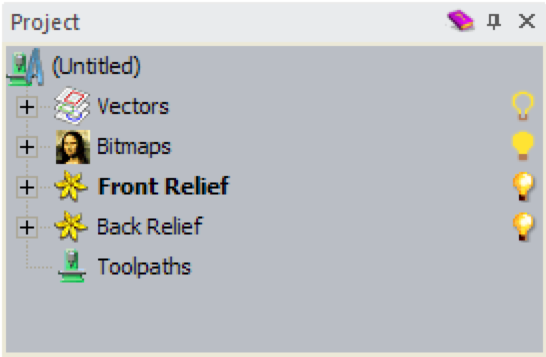
to maximise the area above the splitter bar, and hide the context tools and information associated with the currently selected item in the Project Tree; or

to move the splitter bar to its previous position.

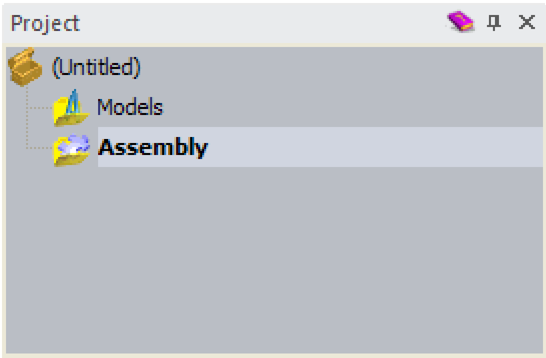
The layout of the Project Tree depends on whether an ArtCAM project or an independent ArtCAM model is created.

*If you are working in ArtCAM Insignia, you cannot create or open a project.*

For example, when working with a new independent model, the Project Tree looks as follows:




When working with a new project, the Project Tree looks as follows:




If you are working with a new ArtCAM project, the Project Tree includes:

- , at its root. Provided your project is saved, its name is shown here.

  There is no context menu associated with this item.

  *Click beside any item in the Project Tree to show its associated items. Click to hide them.*

- **Models**, which enables you to add new or saved ArtCAM models.

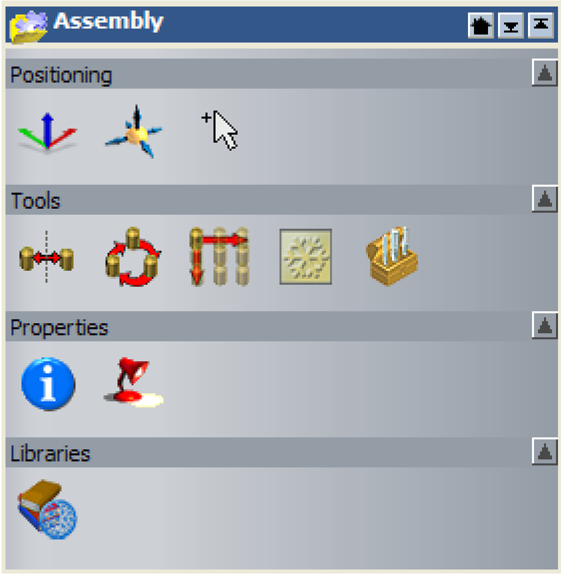
  Right-click **Models** to display its context menu:

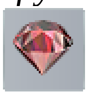
  
  

  You can use this to create or import a model.

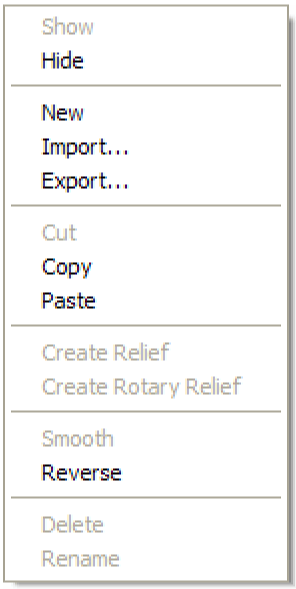
- **Assembly**, which enables you to add triangle and surface models.

When selected, displayed below the splitter bar are tools that enable you to nudge, reposition, select, mirror, copy, and render all associated assemblies, import components, display properties and calculate multi-sided machining toolpaths:




 *If you are working in ArtCAM JewelSmith, the* ***Add Gem***  *button is also displayed in the* ***Tools*** *area.*

Right-click  **Assembly** to display its context menu:



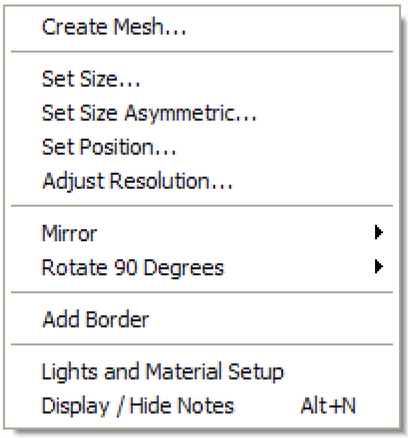

You can use this to toggle its visibility, create an empty assembly, import a triangle or surface model, export, copy or paste the root **Assembly**, and reverse the direction of triangles.

If you are working with an independent ArtCAM model, the Project Tree includes:

- , at its root. Provided your model is saved or imported, its name is shown here.

  When selected, the model's dimensions and resolution are displayed below the splitter bar, along with the composite relief dimensions.

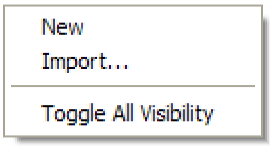
  Right-click  to display its context menu:


  You can use this to create a triangle mesh, set the model's size, position and resolution, mirror, rotate or add a border to the model, specify its lights and material settings, or display your notes.

- **Bitmaps**, which controls the bitmap layer stack.

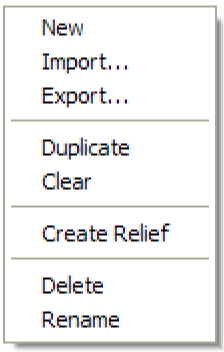
  Right-click **Bitmaps** to display its context menu:


  You can use this to create a new bitmap layer, import bitmap artwork, and toggle the visibility of bitmap layers.

- **Bitmap Layer**, which is the default bitmap layer and displayed by clicking ⊞ beside **Bitmaps**.

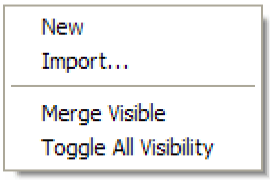
  Right-click **Bitmap Layer** to display its context menu:

You can use this to create a new bitmap layer, import bitmap artwork, export bitmap artwork, duplicate the layer, clear the layer, create a relief layer, delete the layer, and rename the layer.

- **Vectors**, which controls the vector layer stack.

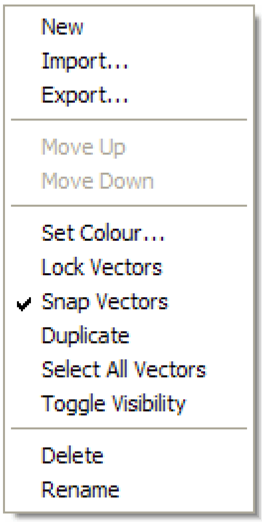
  Right-click **Vectors** to display its context menu:

  You can use this to create a new vector layer, import vector artwork, toggle the visibility of vector layers, and merge them.

- ● **Default Layer**, which is the default vector layer and displayed by clicking ⊞ beside **Vectors**.

  Right-click ● **Default Layer** to display its context menu:

  You can use this to create a new vector layer, import vector artwork, export, lock or select its vector artwork, set the stacking order, set the colour applied to the layer's vector artwork, toggle snapping or its visibility, duplicate the layer, delete or rename the layer.

- **Front Relief**, which controls the relief layer stack that makes up the model's front surface.

  *If you are working with a rotary relief,* *is displayed in place of* .

  *If you are working in ArtCAM Insignia, this item is not included in the Project Tree.*

Right-click **Front Relief** to display its context menu:


You can use this to create a new relief layer, import triangle models, surface models or relief clipart, and toggle the visibility of relief layers.

- **Back Relief**, which controls the relief layer stack that makes up the model's back surface.

  *If you are working with a rotary relief, is displayed in place of .*

  *If you are working in ArtCAM Insignia, this item is not included in the Project Tree.*

  Right-click **Back Relief** to display its context menu:

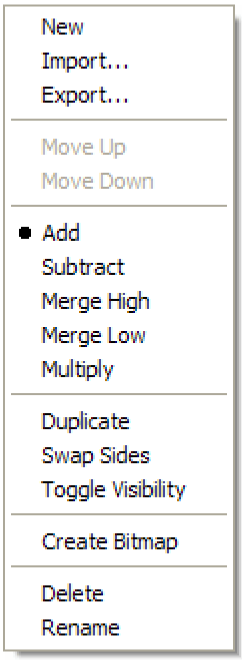

  You can use this to create a new relief layer, import triangle and surface models onto a relief layer, export a relief, duplicate a relief layer, move a relief layer to the opposite stack, merge relief layers, create a greyscale bitmap layer, delete a relief layer, and toggle the visibility of relief layers in the stack associated with the model's back surface.

- **Relief Layer**, which is the default relief layer and displayed by clicking beside **Front Relief** and **Back Relief**.

  *If you are working with a rotary relief, is displayed in place of .*

  *If you are working in ArtCAM Insignia, this item is not included in the Project Tree.*

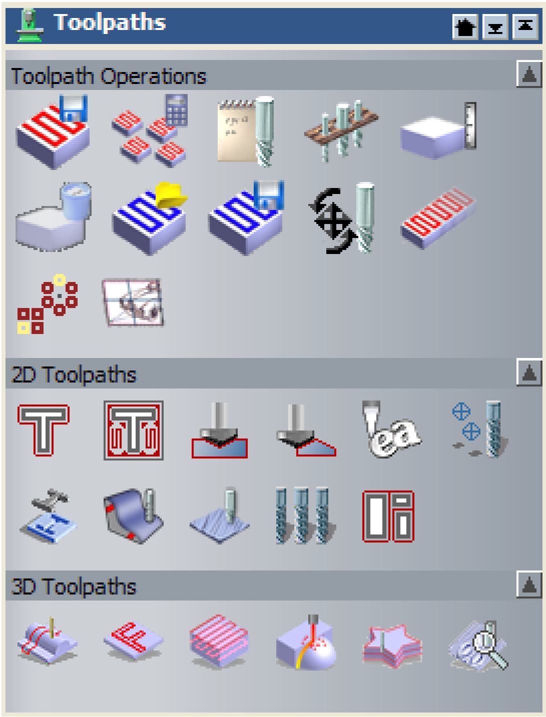
Right-click **Relief Layer** to display its context menu:



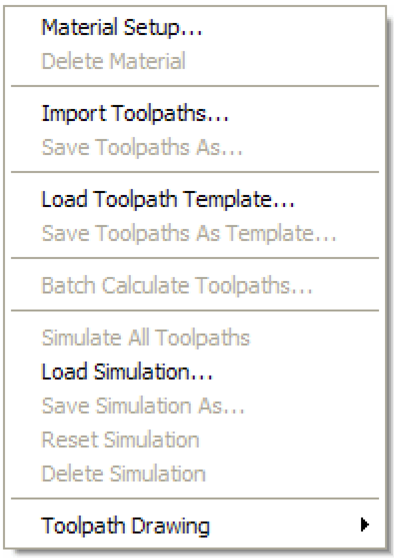

You can use this to create a new layer, import triangle, surface models or relief clipart onto the layer, export a relief, set the layer's combine mode and it's position in the stack, duplicate the layer, move the layer to the opposite stack, create a greyscale bitmap layer, delete the layer, and rename the layer.

- **Toolpaths**, which enables you to control your calculated and created toolpaths, and the tools they use.

When selected, displayed below the splitter bar are tools that enable you to create 2D and 3D toolpaths, display the **Tool Database**, specify the material block, import toolpaths from an ArtCAM model, import a toolpath template or simulation, and control how toolpaths are displayed.




Right-click **Toolpaths** to display its context menu:

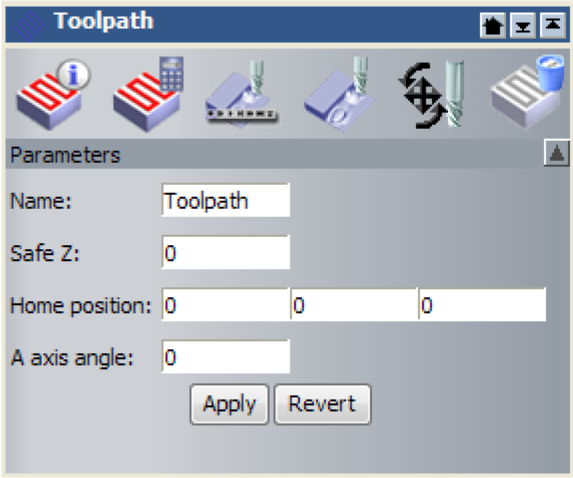



You can use this to specify the material block, import toolpaths from an ArtCAM model, import a toolpath template or simulation, and control how toolpaths are displayed.

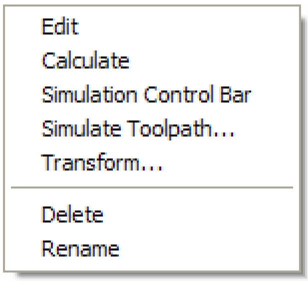
As you continue working in a model, other items displayed in the Project Tree include:

- for each of the toolpaths you create; beside which is shown the toolpath's name.

  When selected, displayed below the splitter bar are tools that enable you to display the **Simulation Control** toolbar, edit, calculate, simulate, transform or delete the toolpath. The toolpath's parameters are also shown.

  Right-click to display its context menu:

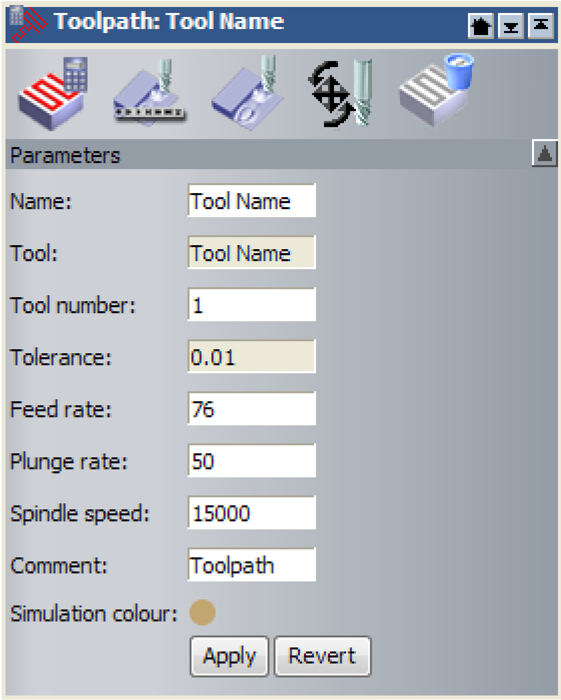
  You can use this to display the **Simulation Control** toolbar, edit, calculate, simulate, transform, delete or rename the toolpath.

  *Double-click to display the panel of settings used to create the toolpath. You can then edit or re-calculate the toolpath.*

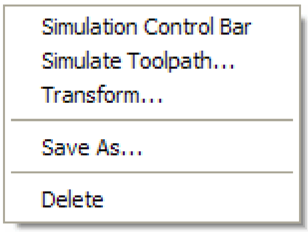
- for each of the tools used in your calculated toolpaths; beside which is shown the tool's name.

  *is sometimes replaced with another icon to indicate the tool's strategy. For example, where a spiral strategy is used, is displayed instead, and where feature machining is used, is displayed instead.*

When selected, displayed below the splitter bar are tools that enable you to display the **Simulation Control** toolbar, edit, simulate, transform or delete the toolpath. The tool's parameters are also shown.




Right-click to display its context menu:




You can use this to display the **Simulation Control** toolbar, simulate, transform, save or delete the tool passes.

*When right-clicking a tool used as part of a Profile toolpath, the* ***Profile Options*** *and* ***Profiling Machining Order*** *options are also displayed in the context menu.*

- beside **Vectors**, which you can use to toggle the display of that artwork on all vector layers in the **2D View** and **3D View** window.
- beside each vector layer, which you can use to toggle the display of the layer's vector artwork in the **2D View** and **3D View** window.

- beside each vector layer, which you can use to toggle snapping (see page 127).
- beside each vector layer, which you can use to toggle the locking (see page 127) of its vector artwork.
- beside **Bitmaps**, which you can use to toggle the display of the currently active bitmap layer in the **2D View** and **3D View** window.
- beside **Front Relief**, which you can use to toggle the display of all its associated relief layers in the **3D View** window.
- beside **Back Relief**, which you can use to toggle the display of all its associated relief layers in the **3D View** window.
- beside each relief layer, which you can use toggle its display in the **3D View** window.
- beside **Toolpaths**, which you can use to toggle the display of the wireframe toolpath preview associated with all calculated 2D toolpaths in the **2D View** window.
- beside **Toolpaths**, which you can use to toggle the display of the solid colour simulation associated with all calculated 2D toolpaths in the **2D View** window.
- beside **Toolpaths**, which you can use to toggle the display of all calculated toolpaths in the **3D View** window.
- beside each calculated 2D toolpath, which you can use to toggle the display of the wireframe toolpath preview in the **2D View** window.
- beside each calculated 2D toolpath, which you can use to toggle the display of the solid colour simulation in the **2D View** window.
- beside each calculated toolpath, which you can use to toggle the display of the toolpath in the **3D View** window.

As you continue working in a project, other items displayed in the Project Tree include:

- for each closed ArtCAM model; beside which is shown the model's name.

Right-click to display its context menu:

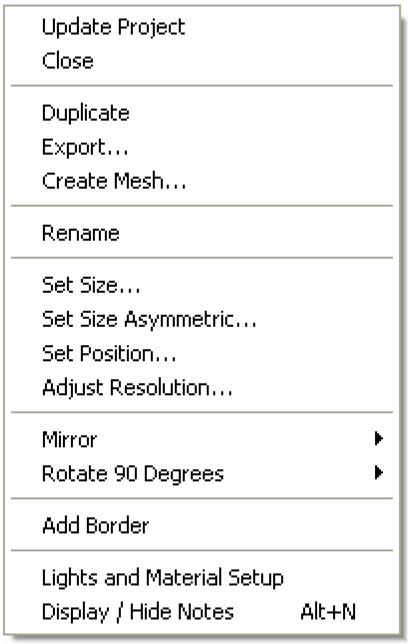

You can use this to edit, duplicate, export, delete and rename a model.

- for the ArtCAM model currently open; beside which is shown the model's name (provided it has already been saved).

  *Although there can be several models in a project, they cannot be opened concurrently.*

  When selected, displayed below the splitter bar is information regarding the model's dimensions and resolution, and the composite relief's dimensions.

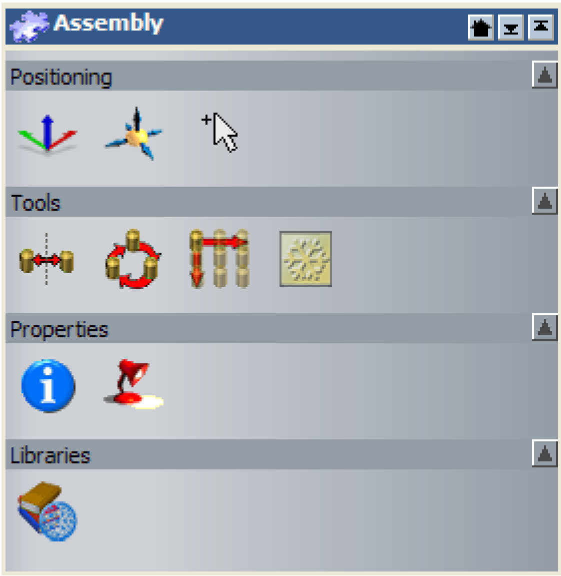
  Right-click to display its context menu:

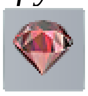



  You can use this to close, duplicate, export, rename, resize, reposition, mirror and rotate the model, adjust its resolution, lights and material settings, and toggle the display of its associated notes.

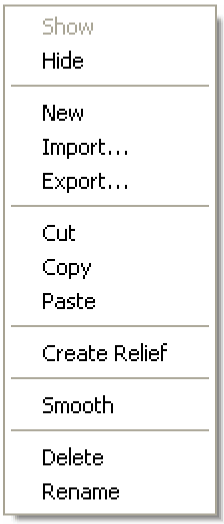
- for each assembly; beside which is shown the assembly's name.

When selected, displayed below the splitter bar are tools that enable you to nudge, reposition, select, mirror, copy, and render all associated assemblies, import components, add gems and display properties.




 *If you are working in ArtCAM JewelSmith, the* ***Add Gem***  *button is also displayed in the* ***Tools*** *area.*

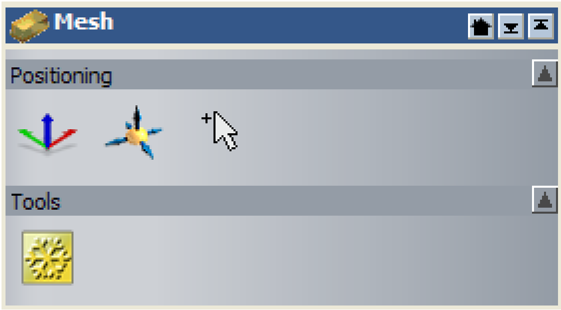
Right-click  to display its context menu:




You can use this to toggle its visibility, create another, import, export, cut, copy, paste, smooth, delete, rename or create a relief from the assembly.

-  for each of the replica meshes; beside which is shown the mesh's name.

When selected, displayed below the splitter bar are tools that enable you to nudge, reposition and select, or create an embossed relief:

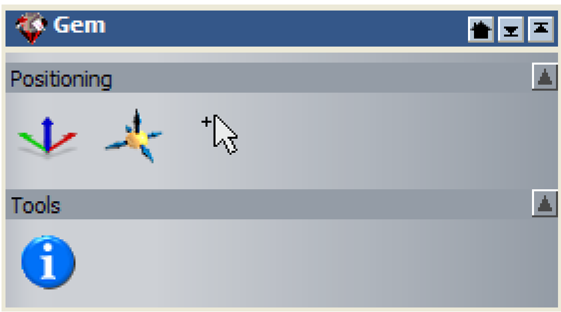



Right-click  to display its context menu:


You can use this to toggle its visibility, edit an associated model, export, cut, copy, smooth, delete, rename or create a relief from the replica mesh.

-  for each of the gems; beside which is shown the gem's name.

When selected, displayed below the splitter bar are tools that enable you to nudge, reposition and select, or display properties:




*If you are working in ArtCAM Pro, the **Tools** area is not displayed.*

Right-click  to display its context menu:


You can use this to toggle its visibility, export, cut, copy, reverse, delete or rename the gem.

When working in the Project Tree, the name of:

- the currently selected item is highlighted and displayed on the splitter bar.
- each active item is displayed in bold text.
- a calculated toolpath is displayed in black text.
- an uncalculated toolpath is displayed in red text.
- an unlicensed toolpath is displayed in orange text.

# Understanding the Tool Settings panel

One of the most important panels, particularly when creating bitmap and vector artwork, or sculpting a relief, is the **Tool Settings** panel.

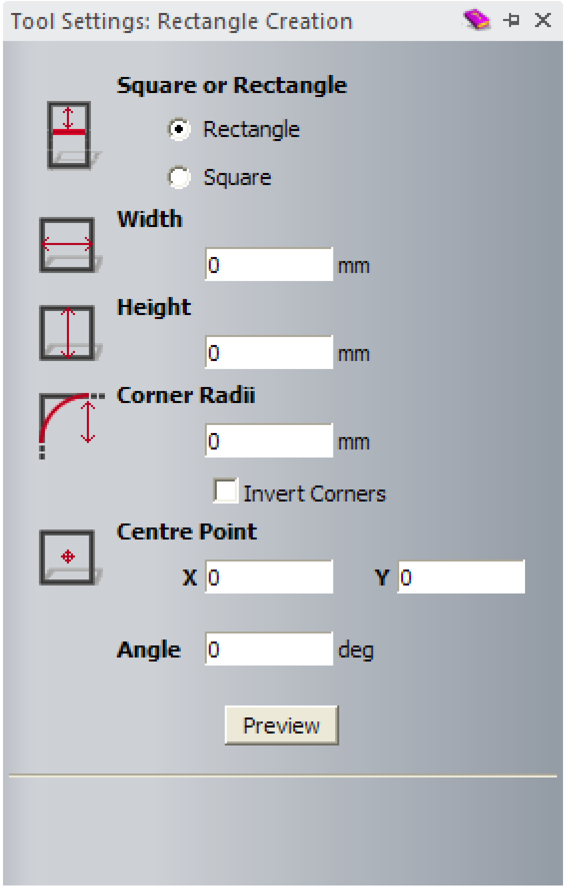
The **Tool Settings** panel:

- is floating over the top-right corner of the viewing area;
- displays the name of your selected tool on it's header and tab; and
- contains the settings associated with your currently selected tool.

Tools that require direct interaction with the **2D View** or **3D View** windows typically display settings on the **Tool Settings** panel. This includes the following tools:

| Tool Category | Tools |
|---|---|
| Design Tools | Select, Node Editing, Transform, Draw Tool, Paint Tool, Magic Wand Tool, Measure Tool, Create Polyline, Create Rectangles, Create Circles, Create Ellipse, Create Polygons, Create Stars, Create Arcs, Create Vector Text, Wrap text round a curve, Smooth, Smudge, Deposit, Carve, Erase and Transfer.<br>*If you are working in ArtCAM Insignia, the Smooth, Smudge, Deposit, Carve, Erase and Transfer tools are not available.* |
| Vector Creation | Fillet Vectors, Trim Vectors and Create Cross Section. |
| Vector Editing | Envelope Distortion |
| Relief Editing | Relief Envelope Distortion.<br>*If you are working in ArtCAM Insignia, the Relief Envelope Distortion tool is not available.* |
| 2D Toolpaths | Profile Options and Profile Machining Order. |

When you select a tool that uses the **Tool Settings** panel, its name is displayed on the panel's header. For example, with the **Create Rectangle** tool selected, the **Tool Settings** panel looks as follows:




If you select a tool that displays a dialog box or its own floating panel of settings, the settings last displayed on the **Tool Settings** panel stay shown. This means that you can choose to work with different combinations of tools displayed at the same time.

To toggle the display of the **Tool Settings** panel, use one of the following methods:

- Press the **F6** key; or
- From the **Menu Bar**, click **Window > Display Tool Settings**.

# Understanding the Toolpaths panel

When a model is created or opened in ArtCAM, the **Toolpaths** panel is hidden by default. This is because:

- its **Toolpath Operations**, **2D Toolpaths** and **3D Toolpaths** tools are displayed on the **Project** panel;

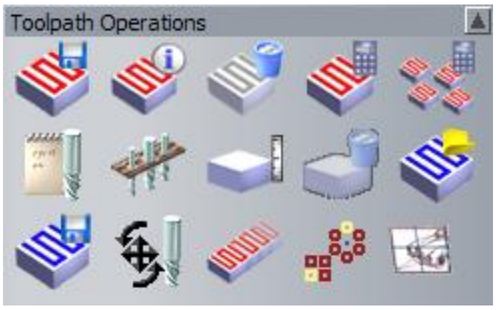
- its **Toolpath Simulation** tools are displayed on the **Simulation** toolbar;
- its simulation settings are displayed on the **3D Graphics Options** panel;
- your created and calculated toolpaths are included in the Project Tree.

When displayed, the **Toolpaths** panel is:

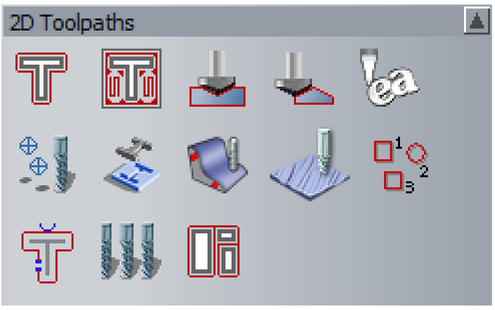
- docked and pinned on the left; and
- divided into four separate areas, each of which is shown by default.

These areas are as follows:

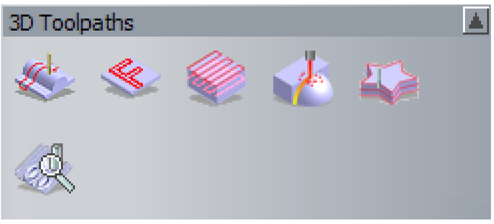
- **Toolpath Operations** - This area contains a collection of buttons you can use to manage the toolpaths you have created, the block or sheet of material in which you want to machine your vector artwork or composite relief, manage the content of your **Tool Database**, create toolpath templates, merge separate toolpaths, copy or transform them.




- **2D Toolpaths** - This area contains a collection of buttons you can use to create the toolpaths needed to machine the two-dimensional vector artwork drawn on the ArtCAM model's vector layers.



- **3D Toolpaths** - This area contains a collection of buttons you can use to create the toolpaths needed to machine all or part of the ArtCAM model's composite relief:




*If you are working in ArtCAM Insignia, the third, fifth and sixth buttons are not included.*

- **Toolpath Simulation** - This area contains a collection of buttons that enable you to simulate your calculated toolpaths. You can also control the way in which the simulation is shown in the **3D View** window.


These tools (see page 272) are also available between the **Simulation** toolbar and **3D Graphics Options** panel.

*If you are working in ArtCAM Insignia, the last three buttons are not included.*

You can control which of the four areas of the **Toolpaths** panel are displayed. Click:

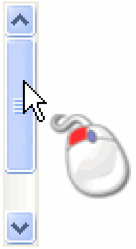
on the area control bar to hide its set of buttons currently shown below:

on the area control bar to display its set of buttons directly below:

All toolpaths that you create or calculate are listed above the **Toolpath Operations** area. For each toolpath you create, its name is listed and a description of each tool used is shown below.

When the toolpath list cannot fit in the space above the **Toolpaths Operation** area, a scrollbar is displayed along the **Toolpaths** panel's right edge.

Click and drag the scrollbar's slider to scroll through the toolpath list:



If you click the scrollbar's button, you can gradually scroll upwards to the top of the list. If you click and hold on the button you can scroll straight to the top of the list:

If you click the scrollbar's button, you can gradually scroll downwards to the bottom of the list. If you click and hold on the button you can scroll straight to the bottom of the list:

You can control the order in which the toolpaths are listed on the **Toolpaths** panel using the and buttons above the **Toolpath Operations** area.

To the right of each calculated 2D toolpath listed on the **Toolpaths** panel are icons to:

- control whether or not the toolpath is displayed in the **2D View** as a wireframe preview or solid colour simulation ;
- set the colour of the toolpath when displayed as a solid colour simulation ; and
- toggle its visibility in the **3D View** window .

To the right of each calculated 3D toolpath, there is only an icon to control whether or not it is displayed in the **3D View** window .

When the name of a tool associated with a specific toolpath is selected from the list, its parameters are shown below the **Toolpath Simulation** area. You can edit the toolpath parameters displayed on the panel.

# Understanding the Assistant panel

When a model is created or opened in ArtCAM, the **Assistant** panel is hidden by default. This is because:

- all of its tools are displayed on their own toolbars; and
- the model information is displayed on the **Project** panel.

When displayed, the **Assistant** panel is:

- docked and pinned on the left; and
- divided into up to nine separate areas, each of which is shown by default.

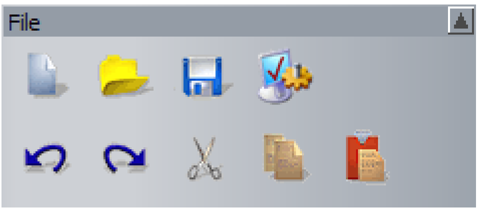
These areas are as follows:

- **Model Information** - displays your ArtCAM model's dimensions, represented by the white area shown in the **2D View** window, and the dimensions of the composite relief shown in the **3D View** window.

  *The dimensions shown depend on which relief layers are visible, as well as which relief side is currently active. For details, see Understanding the Project panel (see page 16) and Understanding the Layers panels (see page 41).*

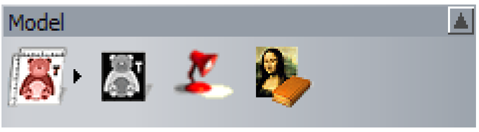
  This information is also displayed on the **Project** panel, when an open model is selected in the Project Tree.

- **File** - contains a collection of buttons you can use to manage your model files, and edit the content of the vector, bitmap and relief layers within them. You can also adjust many of the default settings used in ArtCAM from here.

  These tools are also displayed on the **File** toolbar.

- **Model** - contains a collection of buttons you can use to manage your model layout, its resolution and how it is displayed in the **3D View** window.

*If you are working in ArtCAM Insignia, the last three of these buttons are not available.*

These tools are also displayed on the **Model** toolbar.

- **Bitmap Tools** - contains a collection of buttons you can use to draw and paint artwork on bitmap layers, as well as manage the colour palette's content, and a brush's size and shape. You can also convert bitmap artwork to vector artwork.

  *If you are working in ArtCAM Insignia, the last of these buttons is not available.*

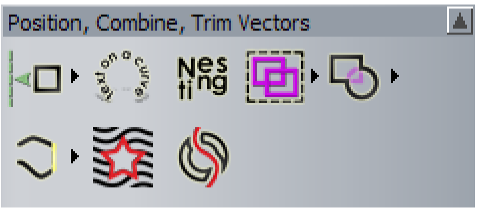
  These tools are also available from the **Design Tools** toolbar displayed vertically in the left docking area, and the **Tool Settings** panel floating over the viewing area.

- **Vector Tools** - contains a collection of buttons you can use to create artwork on vector layers as shapes or text, as well as measure and manipulate them. You can also import vector clipart, and convert vector artwork to bitmap artwork.

  These tools are also available from the **Vector Creation** and **Vector Editing** toolbars displayed horizontally in the top docking area.

- **Position, Combine, Trim Vectors** - contains a collection of buttons you can use to align and centre vector artwork, as well as wrap vectors around a curve, or nest them within a specified area. You can also group, merge, join, trim, clip and slice the vector artwork in your model.

  These tools are also displayed on the **Vector Editing** toolbar displayed horizontally in the top docking area, and the **Design Tools** toolbar displayed vertically in the left docking area.

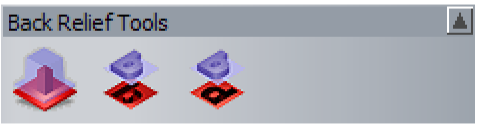
- **Relief Tools** - contains a collection of buttons you can use to scale, smooth, invert, offset, sculpt, slice, reset or add texture to a relief layer. You can also create an angled plane or a blended shape, distort, copy and paste all or a selected area of a relief layer. Furthermore, you can load, save and calculate shapes or text, create a triangle mesh, cross-section or vector boundary, and import relief clipart.

  *If you are working in ArtCAM Insignia, only the first, second, fifth and last of these buttons are available.*

  These tools are also available from the **Relief Creation** and **Relief Editing** toolbars displayed horizontally in the top docking area.

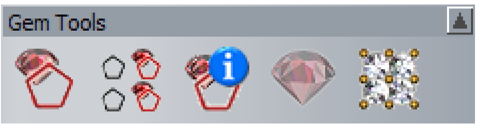
- **Back Relief Tools** - contains a collection of buttons you can use to offset or invert the composite relief to the opposite relief layer stack, or create a flip machining layer.




*If you are working in ArtCAM Insignia, these tools are not available.*

These tools are also available from the **Back Relief T**ools toolbar, which is hidden (see page 309) by default.

- **Gem Tools** - contains a collection of buttons you can use to create gems, gem vectors and pave settings, convert vectors to gem vectors and display gem vector properties:




*If you are working in ArtCAM Pro or Insignia, these tools are not available.*

These tools are also available from the **Gem Tools** toolbar (see page 309), which is hidden by default.

You can control which of the **Assistant** panel's areas are displayed:

- Click ▲ on the area control bar to hide the set of buttons displayed below:

- Click ▼ on the area control bar to display its set of buttons:

Seven of the **Assistant** panel's areas include buttons, each of which either:

- display settings on the **Tool Settings** panel;
- display settings on a floating panel; or
- complete a direct action.

Several of these buttons are combined together in toolsets. Toolsets are collapsed by default to minimise the space in the **Assistant** panel occupied by buttons. Several of the buttons across the different areas of the **Assistant** panel are not used frequently, and so it is better to keep them hidden. Of course, you may prefer to work with all of the buttons displayed at once.

You can control the extent to which the buttons belonging to a particular area are shown or hidden:

1. Click ▸ on the button's right edge to display its associated toolset.

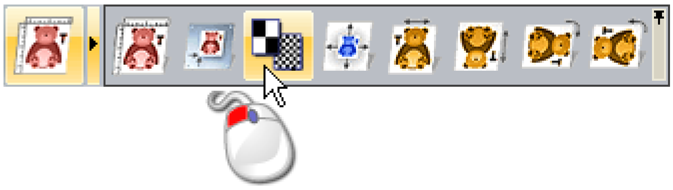
   For example, click ▸ in the **Model** area of the **Assistant** panel:

2. To use a specific tool from the toolset:

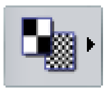
   - Click the tool's button.

     For example, if you needed to set the model's resolution, you would click the **Adjust Model Resolution** button as shown below:

     The toolset collapses and the selected button replaces that which was previously displayed.

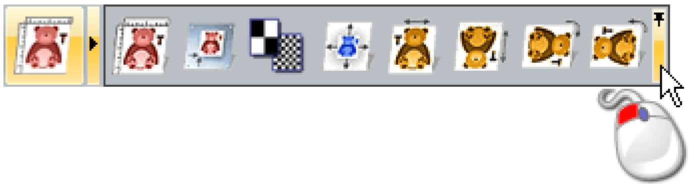
     In our example, the **Adjust Model Resolution** button is displayed as follows:

     The same button is displayed until such time that another button from the same toolset is used.

3. To pin the toolset so that all of it's buttons stay visible:

- Click on the far right edge of the expanded toolset, as shown below:



In our example, the toolset in the **Model** area of the **Assistant** is displayed as follows:

*If you are working in ArtCAM Insignia, the last three buttons are not available.*

- Click on the far right edge of the pinned toolset's end button to collapse the toolset.

# Understanding the Layers panels

When a model is created or opened in ArtCAM, the **Bitmap Layers**, **Vector Layers** and **Relief Layers** panels are hidden by default. This is because:

- their layer stack is included in the **Project** panel's Project Tree; and
- their associated tools are available from the Project Tree's context menus.

When displayed, the **Bitmap Layers**, **Vector Layers** and **Relief Layers** panels are:

- docked and pinned on the right; and
- embedded in a single container, and therefore tabbed.

*If you are working in ArtCAM Insignia, the* ***Relief Layers*** *panel is not available.*

The layers panels are as follows:

- **Bitmap Layers** - This panel is used to manage your model's bitmap layers (see page 96). When you create a new model, a default bitmap layer named *Bitmap Layer* is displayed here. The artwork drawn on the currently active bitmap layer can be displayed in the **2D View** and **3D View** window.

  The panel's toolset enables you to control the stack of bitmap layers:

- **Vector Layers** - This panel is used to manage your model's vector layers (see page 117). When you create a new model, a default layer named *Default Layer* is displayed here. The artwork drawn on vector layers can be displayed in the **2D View** and **3D View** window.

  The panel's list box enables you to choose which sheet of vector artwork is active. Each new model has a default sheet, and this is active. Additional sheets are created when you nest vector artwork or create plates.

  The panel's toolset enables you to control the stack of vector layers:

- **Relief Layers** - This panel is used to manage your model's relief layers (see page 167). When you create a new model, the *Front Relief* option is selected and a default layer named *Relief Layer* is displayed. This layer is part of a stack used to build a composite relief representing the model's front surface.

  The panel's list box enables you to choose which composite relief you want to create. With the *Back Relief* option selected, a default layer named *Relief Layer* is displayed. This layer is part of a stack used to build a composite relief representing the model's back surface.

  The three-dimensional shapes you create on a relief layer are displayed in the **3D View** window.

  The panel's toolset enables you to control the stack of relief layers:

The changes you make using these panels update the layers in the **Project** panel's Project Tree. Similarly, the changes you make using the Project Tree update the **Bitmap Layers**, **Vector Layers** or **Relief Layers** panels.

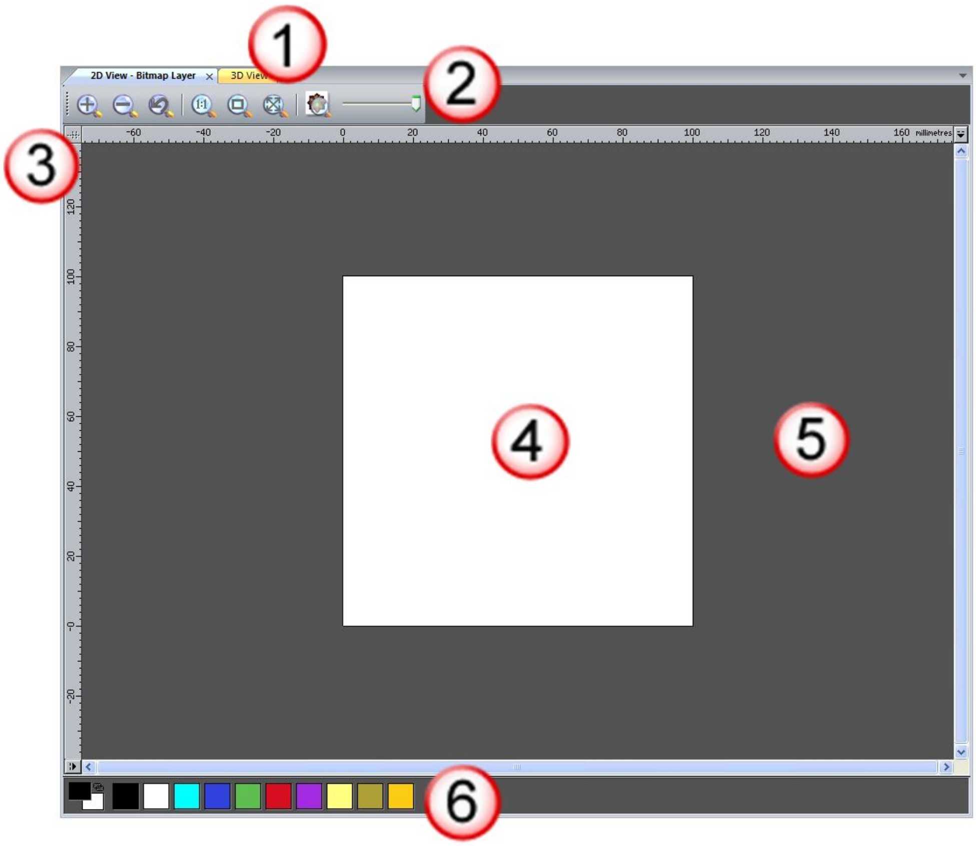
# Understanding the design windows

There are two default design windows in every new ArtCAM model: the **2D View** and the **3D View**. These design windows occupy the viewing area. You can create as many **2D View** windows as you like, but you cannot create any additional **3D View** windows.

## Using 2D View windows

You can create multiple **2D View** windows in a model. Each of the **2D View** windows are shown in the viewing area and include a:

1. tab, which displays the view's name and the currently active bitmap layer's name;
2. toolbar;
3. vertical and horizontal ruler;
4. model area;
5. workspace area; and
6. colour palette.




When working in an open model, the **2D View** window can display:

- the artwork on the currently active bitmap layer;
- the artwork on all visible vector layers;
- sheets;
- a preview of all calculated 2D toolpaths;
- a preview of the currently active relief layer; and
- a greyscale of the composite relief.

*If you are working in ArtCAM Insignia, you cannot create a greyscale of the composite relief.*

For each of the **2D View** windows that you create, you can change its:

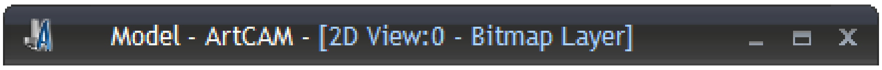
- name (see page 48); and
- workspace colour (see page 49).

## Understanding the Title Bar

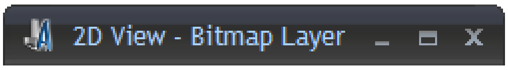
When a cascaded or tiled **2D View** window is minimised, its **Title Bar** shares almost all of the features of ArtCAM's **Title Bar**. ArtCAM's control icon and standard Windows buttons are displayed at each end of the **Title Bar**.

When working in an independent model and looking at a tiled or cascaded window's **Title Bar** when maximised, you can see the model's name, the product name, and information about the window's contents.

*Maximised...*

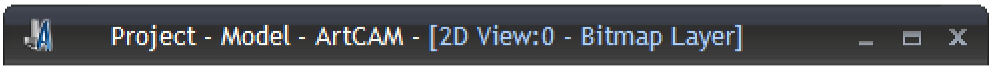



*Minimised...*

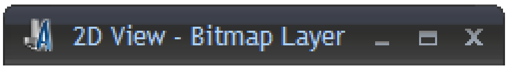



When working in a model as part of a project and looking at the window's **Title Bar** when maximised, you can see the project's name, the open model's name, the product name and information about the window's contents.

*Maximised...*




*Minimised...*



*If you are working in ArtCAM Insignia, you cannot create a project.*

In addition to the control icon and standard Windows controls, the window's **Title Bar** includes the following information: view name and currently active bitmap layer name.

**View Name** - The name of the **2D View** window is displayed to the right of the control icon . Each subsequent view is named *2D View* and numbered sequentially. The name and number information is separated by a ":" (colon). For details on changing the name of the view, see Renaming a 2D View window (see page 48).

**Active Bitmap Layer** - The name of the currently active bitmap layer (see page 99) is shown along with the view's name in brackets. This information is separated by a "-" (dash).

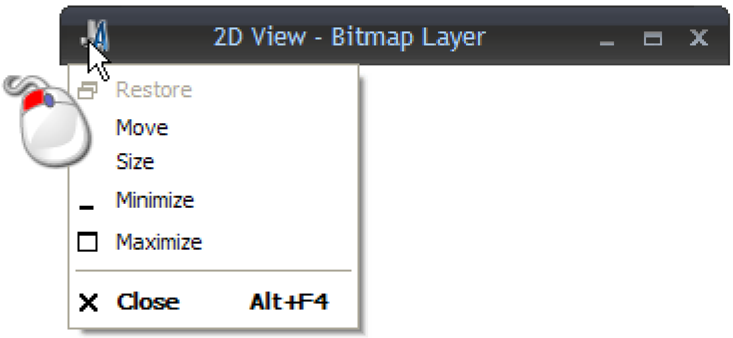
*Double-clicking the view's name on the **Title Bar** mimics clicking the standard Windows **Maximize** or **Restore** buttons also on the **Title Bar**.*

## Using the control menu

When your windows are cascaded or tiled, you can use the control icon on a window's **Title Bar** to control their layout in the viewing area (see page 7).

To display the control menu:

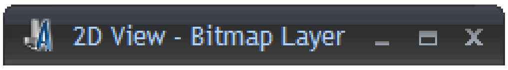
1. From the window's **Title Bar**, click on the far left to display the control menu:



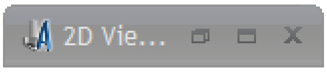

2. To minimise, maximise or restore the window to it's previous size, click:
   - **Minimize** to minimise the window so that its **Title Bar** is positioned along the bottom edge of the viewing area.

   *You can also click the **Minimize** button on the window's **Title Bar** to minimize the window.*

*Before...*




*After...*




- **Maximize** to maximise the window so that it fills the viewing area.

*You can also click the* ***Maximize*** *button or double-click the name on the window's* ***Title Bar*** *to maximise it.*

- **Restore** to restore the minimised window to its previous size.

*You can also click the* ***Restore*** *button or double-click the name on the window's* ***Title Bar*** *to restore the window to its previous size.*

3. To move a resized window:

*You cannot move a window when it is maximised (fills the viewing area).*

a. Click **Move**, then click and drag anywhere on its **Title Bar**.


b. Release the mouse button to set the window's position.

4. To resize the window, click **Size**, then adjust the window using one of the following methods:

*You cannot resize a window when it is maximised (fills the viewing area).*

- To change the width of the window, move the mouse cursor over to the left or right window border. When the mouse cursor changes into a horizontal double-headed arrow ↔ click and drag the border to the right or left.
- To change the height of the window, move the mouse cursor over the top or bottom window border. When the mouse cursor changes into a vertical double-headed arrow ↕, click and drag the border up or down.

- To change the height and width at the same time, move the mouse cursor over any window corner. When the mouse cursor changes to a double-headed diagonal arrow ↘, click and drag the border in any direction.

5. To close the window, click **Close**.

*You cannot close the **3D View** window.*

*You can also click the **Close** button on the window's **Title Bar** to close it.*

If you close the only **2D View** window before saving the ArtCAM model, a message box is displayed asking if you want to save the changes. To save the changes, click **Yes**. To close the model without saving your changes, click **No**. The **Start** panel is displayed.

## Adjusting the window layout

In the viewing area, the **2D View** and **3D View** windows are tabbed by default. You can set the layout of the windows.

From the **Menu Bar**, click **Window**, then:

- **Cascade** to display the **2D View** and **3D View** window overlaying one another;
- **Tile Vertically** to display the **2D View** and **3D View** window beside one another;
- **Tile Horizontally** to display the **2D View** and **3D View** window on top of one another;
- **Tabbed Views** to display a tabbed **2D View** and **3D View** window;
- **2D View** to display the **2D View** window only; or

  *You can also display the **2D View** window by pressing the **F2** key when the **3D View** window is displayed.*

- **3D View** to display the **3D View** window only.

  *You can also display the **3D View** window by pressing the **F3** key when the **2D View** window is displayed.*

## Opening a new 2D View window

To open a new **2D View** window, click **Bitmaps > Views > New View** from the **Menu Bar**.

Each new window is named **2D View** by default, and numbered sequentially. For example, **2D View:1**.

*You can change the **2D View** window's name (see page 48).*

The tab associated with each window is given a different colour. This makes it easier to distinguish the windows, as shown below:

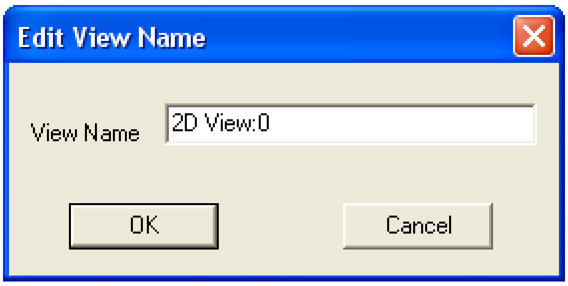

The same vector and bitmap artwork is displayed in the windows. The same colour palette is also displayed below the windows, although each window can have its own colour links and shape attributes applied to the colours within the palette.

## Renaming a 2D View window

The **2D View** window displayed when you create a model is named **2D View:0** by default. You can change the name of a **2D View** window.

To change a **2D View** window's name:

1. Make sure that the **2D View** window you want to rename is active by clicking:
   - it's tab;
   - it's title bar, if tiled or cascaded; or
   - anywhere in the window.
2. From the **Menu Bar**, click **Bitmaps > Views > Rename**. The **Edit View Name** dialog box is displayed:

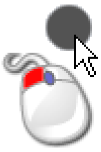
3. Click the **View Name** box, then type the name you want to give to the **2D View** window.
4. Click **OK** to close the **Edit View Name** dialog box and set the **2D View** window's name.

## *Changing the workspace colour*

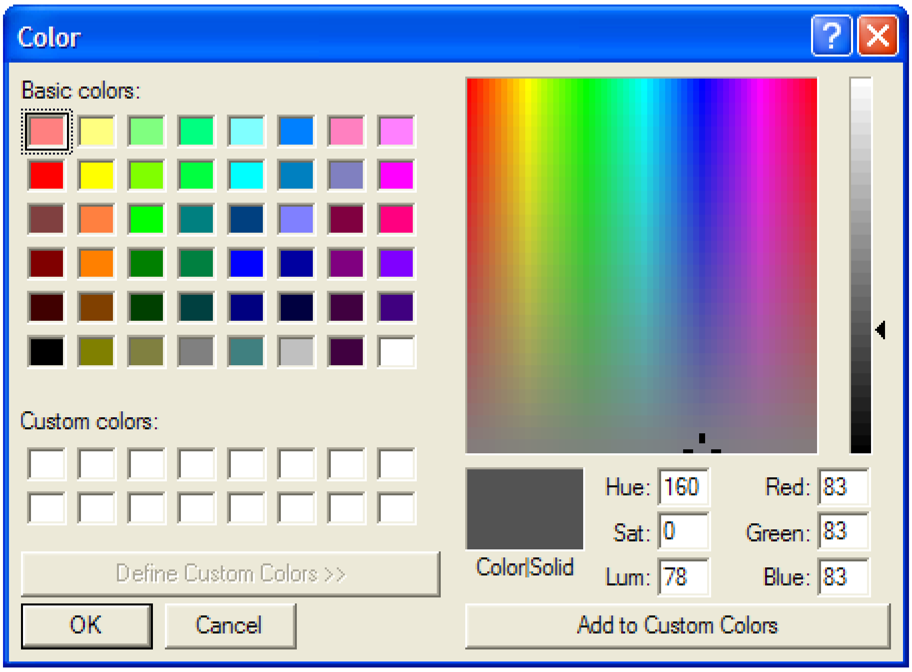
You can control the colour of the workspace area in the **2D View** window. Your chosen colour is applied to the workspace area in all **2D View** windows currently open.

To set the workspace area's colour:

1. Use one of the following methods to display the **Options** panel:
   - In the **File** toolbar, click the **Options**  button;
   - From the **Menu Bar**, click **Edit > Options**; or
   - From the **Assistant** panel, click the **Options**  button in the **File** area.
2. From the **Options** panel, click the **Drawing Colours** control bar to display its settings.
3. Select the **Custom Workspace Colour** check box. The **Workspace** colour swatch is displayed.
4. Click the **Workspace** colour swatch:

   The **Color** dialog box is displayed:

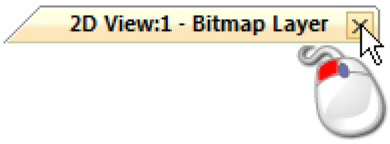
   
   

5. Select the colour you want to use, and then click **OK**. The **Color** dialog box is closed and your chosen colour is shown on the **Workspace** swatch.
6. Click **Apply** to close the **Options** panel and colour the workspace area.

### *Deleting a 2D View window*

You can delete a **2D View** window from your ArtCAM model.

1. Make sure that the **2D View** window you want to delete is active by clicking:
   - it's tab;
   - it's title bar, if tiled or cascaded; or
   - anywhere in the window.
2. Use one of the following methods to delete the **2D View** window:
   - From the **Menu Bar**, click **Bitmaps > Views > Delete**; or
   - On the window's tab, click ;

   If you are working with an independent model, are deleting the only **2D View** window and have not yet saved the model, or the changes you have made since it was saved previously, a message box is displayed asking if you want to save your changes.

   *If you do not want to delete the **2D View** window, click **Cancel** to close the message box.*

   To save the changes in the model:

   - Click **Yes.** The **Save Model As...** dialog box is displayed. For details, see Saving a model (see page 84).

   If you do not want to save the changes:

   - Click **No**. The message box and the model are closed, and the **Start** panel is displayed.

   If you are working in an independent model, are deleting the only **2D View** window, and have already saved any changes you have made to the model, the model closes immediately.

If you are working in a model that is part of a project, are deleting the only **2D View** window, and have not yet saved the model or the changes you have made since it was saved previously, a message box is displayed asking if you want to update the project with your changes.

*If you are working in ArtCAM Insignia, you cannot create or open a project.*

To save the changes in the model:

- Click **Yes**. The message box and the model are closed. On the **Project** panel, the updated model in the Project Tree is closed.

*The updated model is not saved as part of the project until the project is also saved. For details, see Saving a project (see page 93).*

If you do not want to save the changes:

- Click **No**. The message box and the model are closed. On the **Project** panel, the model in the Project Tree is closed.

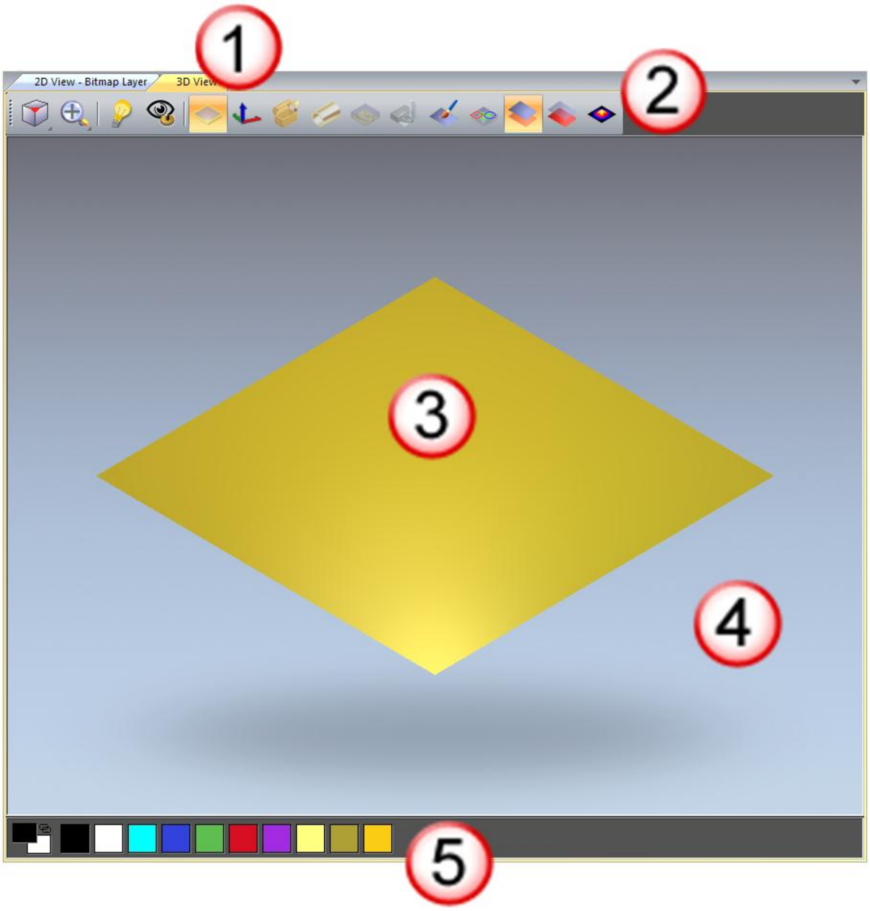
If you do not want to delete the **2D View** window, click **Cancel** to close the message box.

If you are working with a model that is part of a project, are deleting the only **2D View** window, and have already saved the changes you have made to the model, the model closes immediately.

## Using the 3D View Window

The **3D View** window is shown in the viewing area and includes a:

1. tab, which displays the view's name;
2. toolbar;
3. composite relief;
4. background; and
5. colour palette.




When working with an independent model, the **3D View** window can display:

- the composite relief resulting from all visible relief layers;

  *If you are working in ArtCAM Insignia, although there is a relief, there are no relief layers.*

- calculated toolpaths;
- the material block;
- a toolpath simulation;
- the artwork on the currently active bitmap layer;
- the artwork on all visible vector layers; and
- a triangle mesh.

  *If you are working in ArtCAM Insignia, you cannot create a triangle mesh.*

When working in a project, the **3D View** can also display:

- the root **Assembly**;
- assemblies;

- replica meshes; and
- gems.

*If you are working in ArtCAM Insignia, you cannot create a project.*

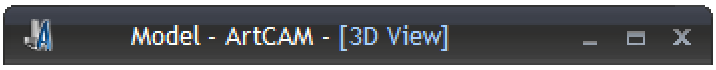
All of the visible items in the Project Tree are shown in the **3D View** window when a project is first opened. For details, see Understanding the Project panel (see page 16).

## Understanding the Title Bar

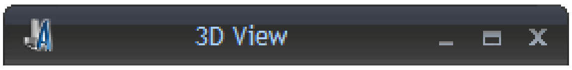
When a cascaded or tiled **3D View** window is minimised, its **Title Bar** shares almost all of the features of ArtCAM's **Title Bar**. ArtCAM's control icon and standard Windows buttons are displayed at each end of the **Title Bar**. However, by looking at the window's **Title Bar** you cannot see any information other than the name of the view.

When working in an independent model and looking at a tiled or cascaded **3D View** window's **Title Bar** when maximised, you can see the model's name, the product name and the name of the view.

*Maximised...*

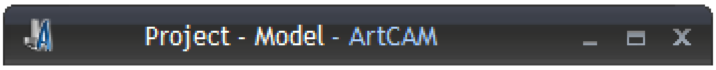



*Minimised...*

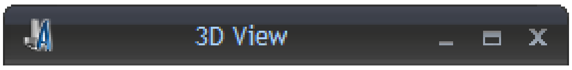



When working in a model as part of a project and looking at the **3D View** window's **Title Bar** when maximised, you can see the project's name, the open model's name, the product name and the window's name.

*Maximised...*




*Minimised...*



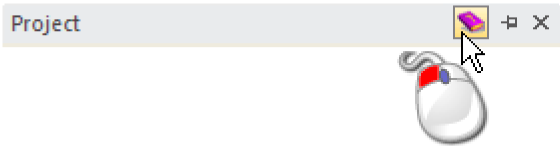

For details on using the control icon and the standard Windows controls, see Understanding the Title Bar (see page 44) in association with **2D View** windows.

# Information about ArtCAM

You can find information about the features in ArtCAM from the following sources:

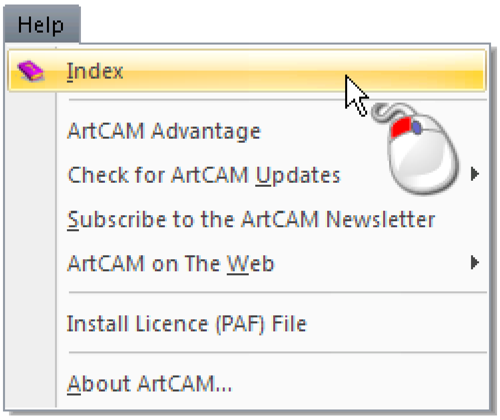
1. Tool panels (in-line help).

   Click  on a panel's header to toggle the display of its in-line help.

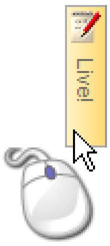
2. The ArtCAM Reference Help system.

   From the **Menu Bar**, click **Help > Index**.

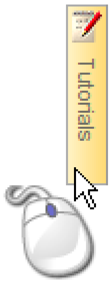
3. The **Live!** panel.

   Move the mouse cursor over the **Live!** tab in the right docking area:

4. The **Tutorials** panel.

Move the mouse cursor over the **Tutorials** tab in the right docking area:




5. The **Start** panel.

   Click in the **Other Features** area to display details of new features and enhancements.

6. The printed ArtCAM User Guide.
7. The ArtCAM website.

   From the **Menu Bar**, click **Help > ArtCAM on The Web > ArtCAM Home Page**.

8. The ArtCAM User Forum.

   From the **Menu Bar**, click **Help > ArtCAM on The Web > ArtCAM Forum**. You can also access the forum at http://forum.artcam.com.

   *If you have not yet registered as a forum member, click the **Join** option on the forum's home page to do so. Registered users are able to download images, example ArtCAM models and relief clipart files attached to posts.*

9. Subscribe to the ArtCAM Newsletter.

   From the **Menu Bar**, click **Help > Subscribe to the ArtCAM Newsletter** to send an e-mail requesting subscription to the quarterly newsletter.

# Creating models

A model is an ArtCAM proprietary file which hosts the different aspects of your design: the vector artwork, bitmap artwork, reliefs and toolpaths.

You can create an ArtCAM model that is independent or part of a project.

*If you are working in ArtCAM Insignia, you can only create an independent model.*

When you start ArtCAM, the **Start** panel is displayed on the left. To create a new independent model from the **Start** panel, click:

- **New Model**, then specify your material block's dimensions (see page 57);

  *You can also press the **Ctrl+N** keys on your keyboard to create a model in this way.*

- **Open Model**, then select a compatible file (see page 61); or
- **Face Wizard**, then choose a photographic image containing a side-profile of a person's head and neck (see page 69).

  *If you are working in ArtCAM Insignia, this option is not available.*

To create a new independent model from the **Menu Bar**, click:

- **File > New > Model**, then specify your material block's dimensions (see page 57);
- **File > New > Model (Specify Pixel Size)**, then specify the number of pixels (see page 67); or

- **File > New > From Image File**, then select a compatible file (see page 61).

To create a new model that is part of a project, from the **Project** panel:

- Right-click **Models** in the Project Tree, then click:
  a. **New > Model** in the context menu and specify your material block's dimensions (see page 57);
  b. **New > Model (Specify Pixel Size)** and specify the number of pixels (see page 67); or
  c. **New > From Image File** and select a compatible file (see page 61).
- Right-click a closed model in the Project Tree, then select **Duplicate** in the context menu. This enables you to create a duplicate copy of the model, which you can then edit.

# Creating a new model

When creating a model in ArtCAM, you must specify the:

- units of measurement you want to use;
- model's dimensions;
- model's zero origin; and
- model's resolution.

The model's dimensions typically represent the sheet or block of material you want to use when manufacturing your finished design.

To create a model using a set of imperial or metric measurements:

1. If you are working in ArtCAM Pro or JewelSmith and creating a model as part of a project:
   - From the **Project** panel, right-click **Models** in the Project Tree, then click **New > Model...** from the context menu. The **Size For New Model** dialog box is displayed.
2. If you are creating an independent model, use one of the following methods to display:
   a. the **Size For New Model** dialog box if you are working in ArtCAM Pro or JewelSmith; or
   b. the **Setup Job Dimensions** dialog box if you are working in ArtCAM Insignia.

- From the **Start** panel, click **New Model** in the **Models** area;
- From the **Menu Bar**, click **File > New Model**;
- In the **File** toolbar, click the **New Model** button; or
- Press the **Ctrl+N** keys.

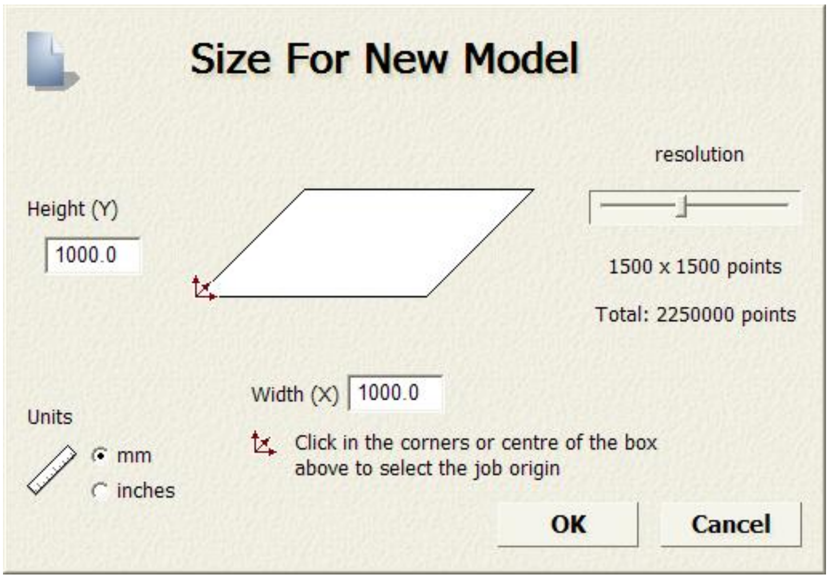
3. If you are creating a new model with another already open, use one of the following methods to display:

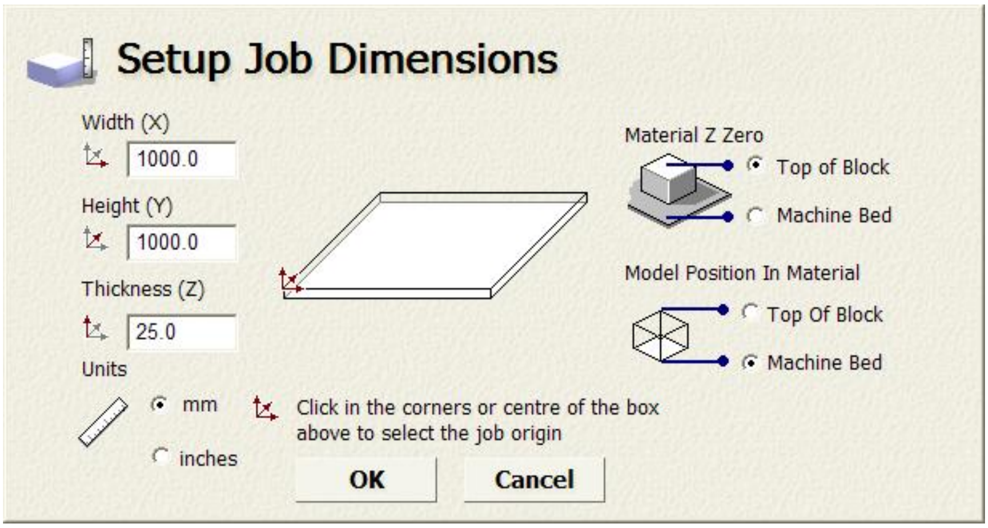
   a. the **Size For New Model** dialog box if you are working in ArtCAM Pro or JewelSmith; or

   b. the **Setup Job Dimensions** dialog box if you are working in ArtCAM Insignia.

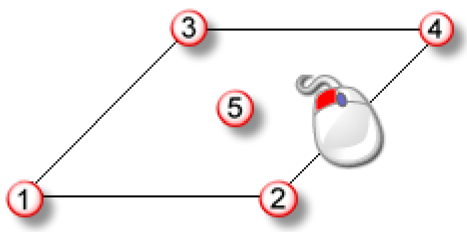
   - In the **File** toolbar, click the **New Model** button;
   - From the **Menu Bar**, click **File > New > Model**; or
   - Press the **Ctrl+N** keys.

If you have not saved any changes you have made to the open model before creating another, a message box is displayed asking if you want to save these changes. For further details, see Saving a model (see page 84). If you have not made any changes at all or since it was saved previously, the model is closed.




4. In the **Units** area, select **mm** or **inches**.
5. In the **Height (Y)** and **Width (X)** boxes, specify the height and width of the model you want to create.
6. Specify the model's origin by clicking one of the numbered positions shown below:



The icon is displayed on the box diagram in your selected position.

7. If you are working in ArtCAM Pro or JewelSmith:

   a. In the **Resolution** area, click and drag the slider to set the model resolution:

   A resolution of approximately 1500 x 1500 points is suitable for most jobs.

*After a model is created, you can use the* ***Adjust Model Resolution*** *button on the* ***Model*** *toolbar to change the resolution.*

8. If you are working in ArtCAM Insignia:

   a. In the **Thickness (Z)** box, specify the thickness of the material you are using for machining.

   b. In the **Material Z Zero** area, specify the Z-axis zero level. This is the position of the cutting tool relative to the material block's surface. Click:

   **Top of Block** to position the cutting tool on the material surface; or

   **Machine Bed** to position the cutting tool on the machine bed.

   The icon is displayed in either the top or bottom front left corner of the box diagram to mark the job origin.

   c. In the **Model Position In Material** area, set the position of the model relative to the material block. Click:

   **Top Of Block** to align the model's zero plane with the material surface; or

   **Machine Bed** to align the model's zero plane with the machine bed.

9. Click **OK** to create your model.

   If you are creating a model that is part of a project, an open model is shown below **Models** in the **Project** panel's Project Tree. The model is named *(Untitled)*.

If you are creating an independent model, the open model is the root of **Project** panel's Project Tree. The model is named *(Untitled)*.

A tabbed **2D View** and **3D View** window is created in the viewing area. The **2D View** window is typically the place in which you draw or edit bitmap and vector artwork, although you can do much of the same in the **3D View** window. The **3D View** window is the area in which you view a three-dimensional impression of your model, known as the composite relief.

Each new model has a single default vector, bitmap and relief layer. For further information, see Understanding the Project panel (see page 16) and Understanding the Layers panels (see page 41).

*If you are working in ArtCAM Insignia, there is no relief layer.*

# Creating a new model from a file

You can create a model using the ArtCAM **Open** dialog box, which contains standard Windows features.

You can create a new model by opening a file in ArtCAM saved in any of the following file formats:

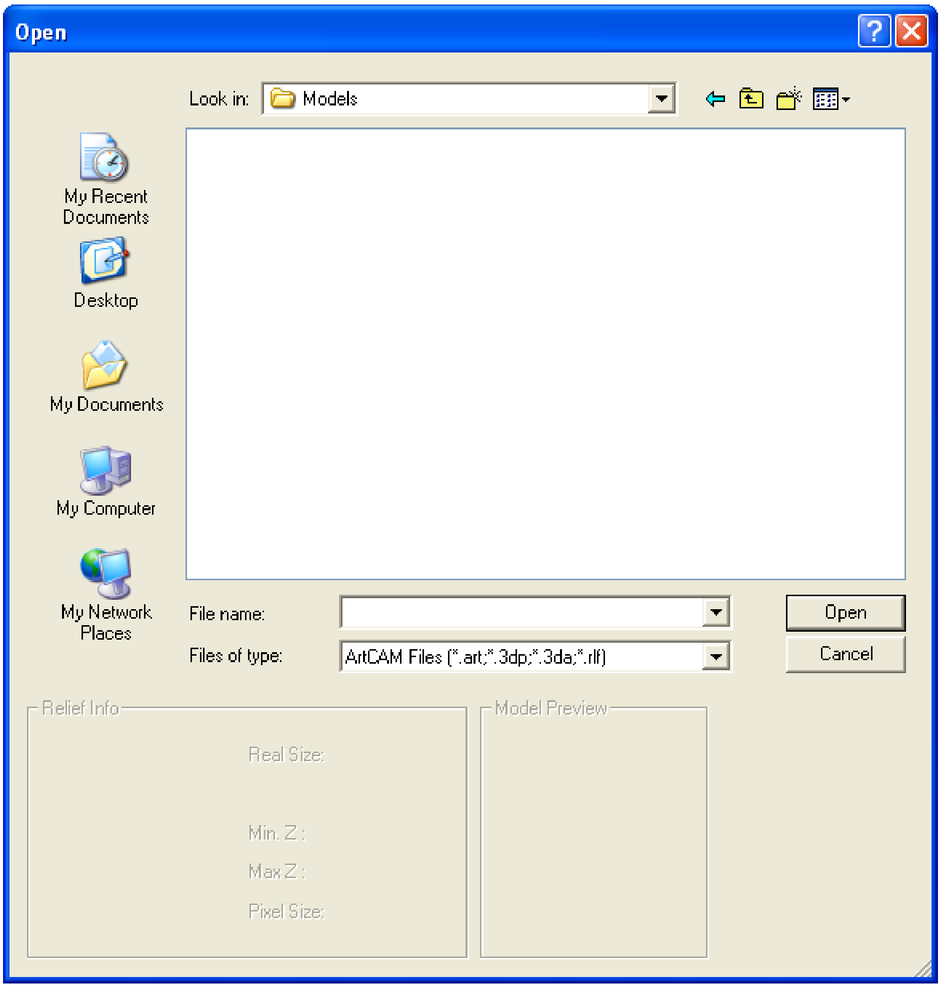
- ArtCAM Model (***.art**)
- ArtCAM Relief (***.rlf**)
- Windows or OS/2 Bitmap (***.bmp**)
- Windows or OS/2 DIB (***.dib**)
- Windows or CompuServe RLE (***.rle**)
- JPEG Image - JFIF Compliant (***.jpg**, ***.jpeg**, ***.jpe** and ***.jfif**)
- CompuServe Graphics Interchange (***.gif**)
- Windows Enhanced Meta File (***.emf**)
- Windows Meta File (***.wmf**)
- Tagged Image File Format (***.tif** and ***.tiff**)
- Portable Network Graphics (***.png**)
- Windows Icon (***.ico**)
- Drawing Interchange Format, including PowerSHAPE and AutoCAD (***.dxf**)

- AutoCAD 2D Drawing (***.dwg**)
- Lotus, PC Paint or DUCT picture (***.pic**)
- Delcam DGK (***.dgk**)
- Portable Document Format (***.pdf**)

To create a new model from a compatible file:

1. If you are creating a model with none currently open, use one of the following methods to display the **Open** dialog box:

   - From the **Start** panel, click  **Open Model** in the **Models** area;
   - From the **Menu Bar**, click the **File > Open** option; or
   - Press the **Ctrl+O** keys.

   If you are creating a model with another already open, use one of the following methods to display the **Open** dialog box:

   - In the **File** toolbar, click the **Open File**  button;

- From the **Menu Bar**, click **File > Open**;
- Press the **Ctrl+O** keys; or
- From the **Assistant** panel, click the **Open File** button in the **File** area;

If you have not saved any changes that have been made to the open model before attempting to open another, a message box is displayed asking if you want to save the changes to the model. For further details, see Saving a model (see page 84). If you have not made any changes to the current model, either whatsoever or since it was saved previously, it is closed immediately.

2. Select the file you want to use to create the model.

In the **Relief Info** area, the file's dimensions are displayed in actual measurements and pixels where possible.

*When choosing a bitmap image, the greyscale representation shown in the* ***Relief Info*** *area provides an indication of the relief that will be created. Consider that the brightest areas of the greyscale image reflect the highest areas in the relief, while the darkest areas reflect the lowest.*

The **Model Preview** area displays a thumbnail image of the file's contents where possible. If you select an ArtCAM model file (***.art**), the thumbnail image shown here contains the contents of the active **2D View** window when the model was last saved.

3. If you are creating a model from a ***.bmp**, ***.gif**, ***.jpg**, ***.jpeg**, ***.jpe**, ***.jfif**, ***.tif** or ***.tiff** file, specify the maximum depth of the resulting relief in the **Max Z** box.

*If you specify a high Z height, the detail in the resulting relief is likely to be poor. You should only specify a low Z height.*

*If you are using a Drawing Interchange File (*****.dxf****), Delcam DGK (*****.dgk****) or Portable Document File (*****.pdf****), a relief cannot be created from the artwork within the file.*

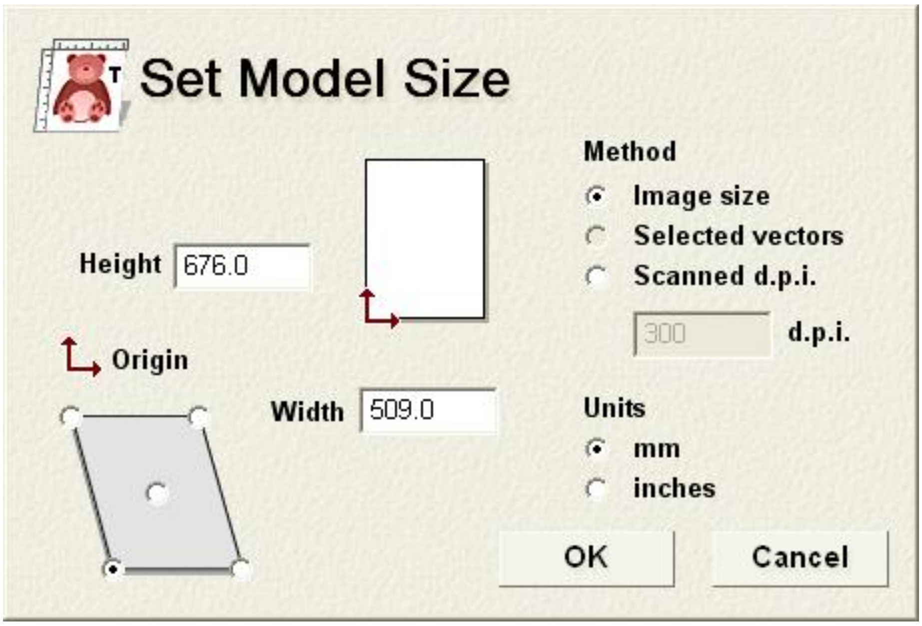
4. Click **Open**.

If you have selected an ArtCAM model file (***.art**), the **Open** dialog box closes, its design windows are displayed in the viewing area, and all layer information is shown on the **Project** panel.

If you have selected an ArtCAM Relief file (***.rlf**), the **Open** dialog box closes, and a greyscale image of the relief is displayed in the **2D View** window. A single default vector, bitmap and relief layer is included in the **Project** panel's Project Tree. For details, see Understanding the Project panel (see page 16).

*If you are working in ArtCAM Insignia, no relief layer is included.*

If you have selected a ***.bmp**, ***.gif**, ***.jpg**, ***.jpeg**, ***.jpe**, ***.jfif**, ***.tif** or ***.tiff** file, the **Open** dialog box closes, and the **Set Model Size** dialog box is displayed:



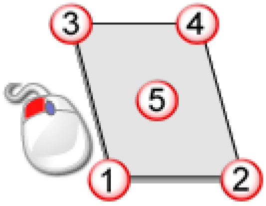

In the **Method** area, the **Image size** option is selected by default. The image's dimensions are displayed in the **Height** and **Width** boxes.

a. If you know the resolution at which the image was originally scanned, select the **Scanned d.p.i.** option, and then type this in the **d.p.i.** box.

b. In the **Units** area, select **mm** or **inches**.

c. In the **Height** and **Width** boxes, specify the height and width of the model you want to create.

d. Specify the model's origin by clicking one of the positions on the box diagram, as shown below:

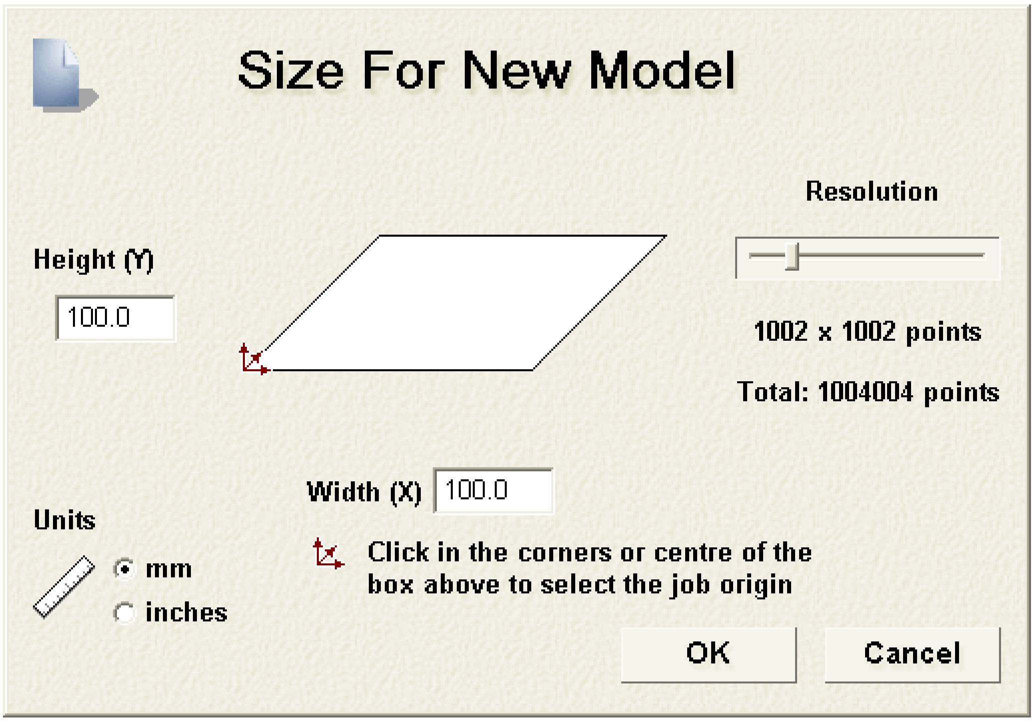



The icon is displayed on the box diagram in your selected position.

e. Click **OK** to create your model. A single default vector, bitmap and relief layer is included in the **Project** panel's Project Tree. For details, see Understanding the Project panel (see page 16) and Understanding the Layers panels (see page 41).

*If you are working in ArtCAM Insignia, no relief layer is included.*

If you select a ***.dxf**, ***.pic**, ***.dgk** or ***.pdf** file, the **Open** dialog box closes, and the **Size For New Model** dialog box is displayed:

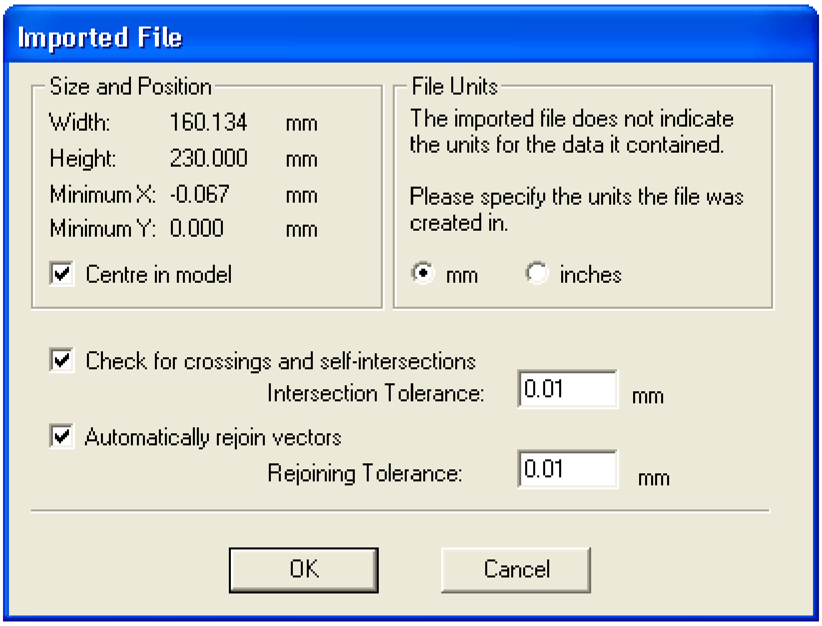



For details on how to use the **Size For New Model** dialog box, see Creating a new model (see page 57).

Click **OK** to close the **Size For New Model** dialog box.

*If you are opening a Delcam DGK file (****.dgk***) and it does not contain any vector artwork, a message box is displayed warning that the file is not suitable. Click* ***OK*** *to close the message box.*

If you are opening a ***.dxf** or ***.dwg** file, the **Imported File** dialog box is displayed:




If you click **Cancel**, a model is created equal to the dimensions previously displayed in the **Size For New Model** dialog box, but the vector artwork in the original file is not included.

To include the vector artwork, make sure the settings are correct:

- To position the artwork in the model's centre, select the **Centre in model** check box.
- In the **File Units** area, make sure the units of measurement for the artwork is the same as those used in your ArtCAM model by selecting **mm** or **inches**.
- To identify all self-intersecting spans in the artwork within tolerance, make sure the **Check for crossings and self-intersections** check box is selected, and that the correct tolerance is specified in the **Intersection Tolerance** box.

*When selected, self-intersecting spans are shown in red, with white circular shapes marking the positions where spans overlap.*

- To rejoin any spans in the artwork that have been *exploded* within tolerance, make sure the **Automatically rejoin vectors** check box is selected, and that the correct tolerance is specified in the **Rejoining Tolerance** box.

Click **OK** to close the dialog box, create the model and import the vector artwork.

*Any layer information within Drawing Interchange (***.dxf**) or Delcam DGK (***.dgk**) files is preserved in ArtCAM. A separate vector layer is created for each layer within the original file.*

If you are opening a Portable Document File (***.pdf**), ArtCAM creates a new model with a separate vector and bitmap layer for each of the pages within the PDF document and names each of the layers using the *file title (page number)* convention. For example, the layer relating to the first page of the document might be labelled *Brochure (Page 1)*.

*ArtCAM does not currently support layers (or 'Optional Content Groups') within Portable Document Format files (*.pdf).*

# Creating a new model using pixels

*If you are working in ArtCAM Insignia, you cannot create a model using an exact number of pixels.*

To create a new model using an exact number of pixels:

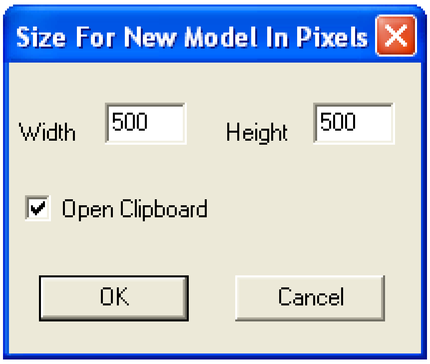
1. If you are creating a model that is part of a project:

   - From the **Project** panel, right-click **Models** in the Project Tree, then click **New > Model (Specify Pixel Size)** in the context menu.

   If you are creating an independent model:

   - From the **Menu Bar**, click the **File > New > Model (Specify Pixel Size)** option.

The **Size For New Model In Pixels** dialog box is displayed:




If you are using the **Size For New Model In Pixels** dialog box for the first time, the default value displayed in both the **Width** and **Height** boxes is *500*. Thereafter, ArtCAM uses your previously specified pixel values.

If there is a compatible image on the Windows clipboard, the **Open Clipboard** option is active. If there is not, the option is greyed out.

*You can create a model from any Windows or OS/2 Bitmap (***.bmp**), Windows or OS/2 DIB (***.dib**), Windows or CompuServe RLE (***.rle**), JPEG Image - JFIF Compliant (***.jpg**, ***.jpeg**, ***.jpe** and ***.jfif**), CompuServe Graphics Interchange (***.gif**), Windows Enhanced Meta File (***.emf**), Windows Meta File (***.wmf**), Tagged Image File Format (***.tif** and ***.tiff**), Portable Network Graphics (***.png**) or Windows Icon (***.ico**) file currently on the Windows clipboard.*

2. To set the model size according to the exact number of pixels in the image on the Windows clipboard, select the **Open Clipboard** check box. The number of pixels in the clipboard image is shown in the **Width** and **Height** boxes.

   *If you want a larger or smaller model than that resulting from the pixels found in the image on the Windows clipboard, specify the new number of pixels in the **Width** and **Height** boxes.*

   If you do not want to use the image on the Windows clipboard to set the model size:

   a. Deselect the **Open Clipboard** check box.

*If there is no compatible image on the Windows clipboard to begin with, the* ***Open Clipboard*** *option is already greyed-out.*

b. In the **Width** box, specify the model's width (X).

c. In the **Height** box, specify the model's height (Y).

3. Click **OK** to create the model according to the specified number of pixels.

   If you are creating a model that is part of a project, an open model is associated with **Models** in the Project Tree. Each new model is named *(Untitled)* by default.

   If you are creating an independent model, the open model is the root of the Project Tree. The model is named *(Untitled)* by default.

# Creating a model using the Face Wizard

The **Face Wizard** enables you to create a face model, which comprises a relief layer generated from a photographic image containing a side-profile of a person's head and neck. You can use a photographic image saved in any of the following file formats:

- Windows or OS/2 Bitmap (***.bmp**)
- Tagged Image File Format (***.tif**)
- CompuServe Graphics Interchange (***.gif**)
- JPEG image (***.jpg**)

The process is almost entirely automated. To produce a detailed face shape on a relief layer, your image should ideally be captured by a digital source using a resolution of at least 1024 x 768 pixels. Colour photographic images can be imported, but they are displayed as black and white in ArtCAM.

*You cannot use the* ***Face Wizard*** *as part of an open model. The* ***Face Wizard*** *requires that a new model is created each time that it is used. If you use the* ***Face Wizard*** *with a model already open in ArtCAM, a message box is displayed asking if you want to save the model before it is closed.*

To use the **Face Wizard** to create a model:

1. If you are creating a face model from the **Start** panel:

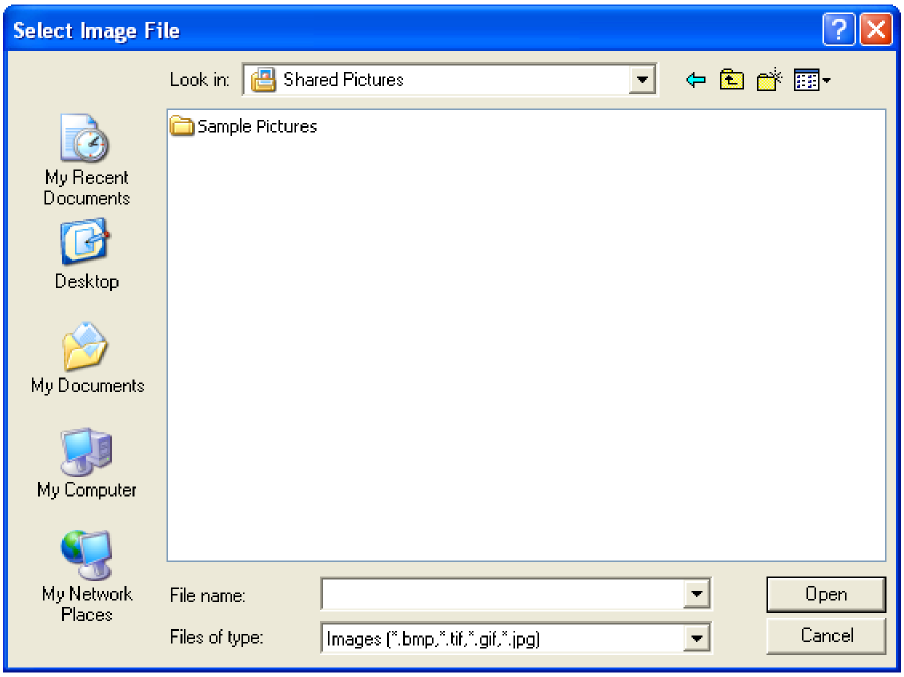
- In the **Other Features** area, click **Face Wizard**. The **Toolbox** panel is docked and pinned on the right of the interface, on which is displayed the **Face Wizard**'s first settings page.

If you are creating a face model with a project or a model already open:

- From the **Toolbox** panel, click **Face Wizard**. The **Toolbox** panel is docked and auto-hidden, on which is displayed the **Face Wizard**'s first settings page.

*To pin the **Toolbox** panel, click on the panel's header.*

2. From the **Toolbox** panel, click **Open Photo** to display the **Select Image File** dialog box:




3. Click the **Look in** list box, then the folder on your computer in which the image file from which you want to create a face shape is stored.
4. When you have found the image file, click its name. Its name is displayed in the **File name** area.
5. Click **OK** to import the image.

*If you are using the* **Face Wizard** *with a model already open and have not saved any changes that have been made to the open model, a message box is displayed asking if you want to save the model. For further details, see Saving a model (see page 84). If you have not made any changes to the current model either whatsoever or since it was saved previously, it is closed immediately.*

*If you are using the* **Face Wizard** *with both a project and model already open, and have not saved any changes that have been made to the model, a message box is displayed asking if you want to update the project with the changes. For further details, see Saving a project (see page 93). If you have not made any changes to current model either whatsoever or since it was saved previously, it is closed in the Project Tree.*

A new ArtCAM model is created using the image's dimensions. On the **Toolbox** panel, the **Face Wizard**'s second page of settings is displayed.

*If you are working with a project, the new model is associated with* ***Models*** *in the Project Tree. The model is named* (Untitled) *by default.*

*If you are working with an independent model, the new model is the root of the Project Tree. The model is named* (Untitled) *by default.*

In the **2D View** window, the imported image is shown. This is hosted on the default bitmap layer, which is associated with **Bitmaps** in the Project Tree displayed on the **Project** panel.

*You can change the name (see page 101) of the default bitmap layer.*

Also in the Project Tree is a default vector layer associated with **Vectors**. The colour red is assigned to this layer, on which you draw the vector artwork required by the **Face Wizard**.

*You can change the name (see page 125) and the colour (see page 126) associated with the default vector layer.*

A relief layer named *Relief Layer* is also created and this is the relief layer on which the three-dimensional face shape is created using the image and its associated vector artwork.

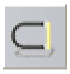
The TIFF image shown below is a good example of what you can import:

You are now ready to create a vector to mark the outline of the head and neck shown in your imported image.

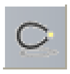
6. From the **Toolbox** panel, click the **Create Polyline**  button to enter Polyline Creation mode. Its settings are displayed on the **Tool Settings** panel.
7. To join each of the polyline's nodes with a bezier span, select the **Draw Smooth Polyline** check box.
8. In the **2D View** window, click and drag your mouse to create a freeform polyline around the head and neck of the person in your imported image.

 *You can press the **Alt+B** keys to toggle the display of the imported image. This enables you to view the polyline more clearly.*

9. Use one of the following methods to close the polyline:

   - Click the **Close Vector With A Line**  button to close the polyline with a linear span.

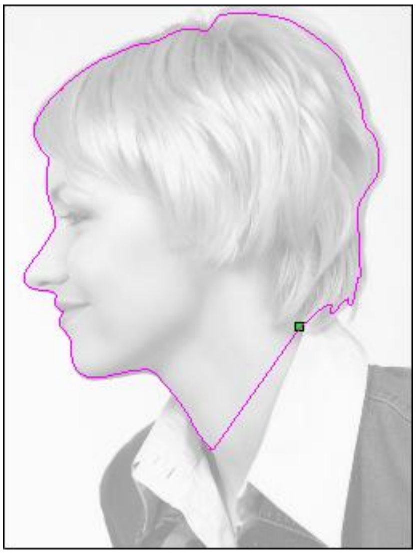
   - Click the **Close Vector With A Curve**  button to close the polyline with a bezier curve.

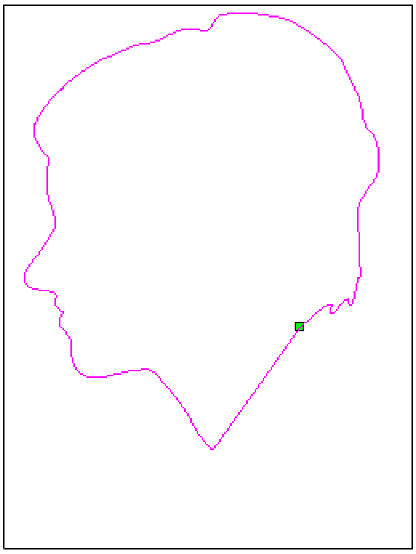
- Click the **Close Vector - Move End Nodes**  button to close the polyline by joining its start and end nodes.

The polyline is shown in magenta and surrounded by a bounding box, indicating that it is closed and selected.

For example, the polyline might look something like this:

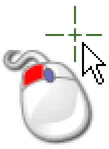
*Polyline, image displayed...* *Polyline, image hidden...*





10. Press the **N** key to enter Node Editing mode. You can see the nodes, spans and control points that make up the polyline you have drawn.
11. Edit the shape of polyline (see page 142) so that it follows the outline of the face area perfectly, or is even a pixel distance inside of the face area.
12. From the **Toolbox** panel, click **Next** to display the third settings page. You are now ready to specify the positions of facial features in the image.

    If you click **Next** before creating a vector outline of the head and neck, a message box is displayed warning that you must have one closed vector. Click **OK** to close the message box.
13. From the **Toolbox** panel, click and drag the top target over to the position in the image you want to specify as the centre-front of the person's eyebrow. This must be on or inside of the polyline that you have drawn.

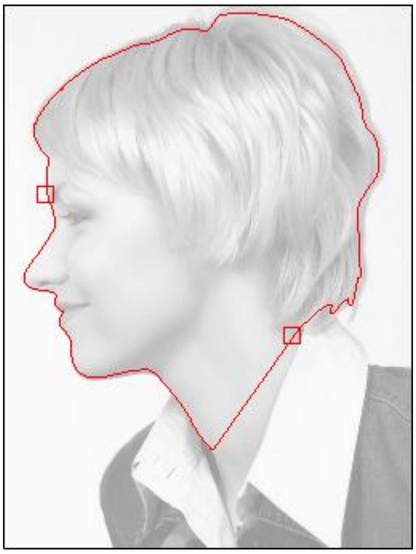


14. Release the mouse to set the position. The position is marked by a red square.
15. From the **Toolbox** panel, click and drag the bottom target over to the position in the image you want to specify as the nape of the neck. This must be on or inside of the polyline that you have drawn.
16. Release the mouse to set the position. The position is marked by a red square.

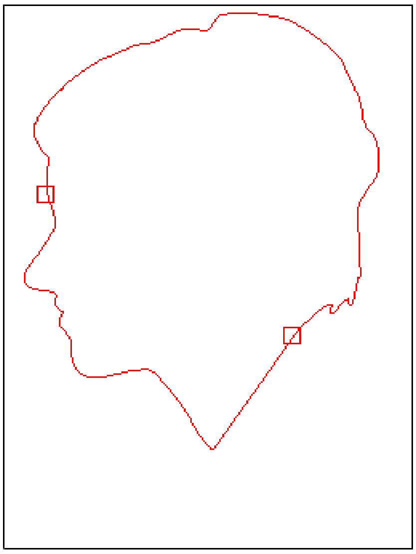
*You can click beside* ***Bitmaps*** *in the* ***Project*** *panel's Project Tree to toggle the display of the imported image. This enables you to view your selected positions more clearly.*

For example, the specified positions might look something like this:

*Positions, image displayed...*



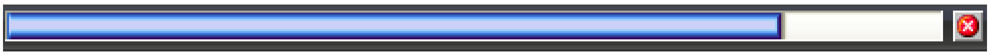
*Positions, image hidden...*



17. From the **Toolbox** panel, click **Next** to display the fourth page of settings.
18. In the **Head Dimensions** area, specify the height and width of the face shape you want to create.
19. In the **Model Dimensions** area, specify the height and width of the ArtCAM model in which you want to create the face relief.
20. Click **Next** to calculate the face relief.

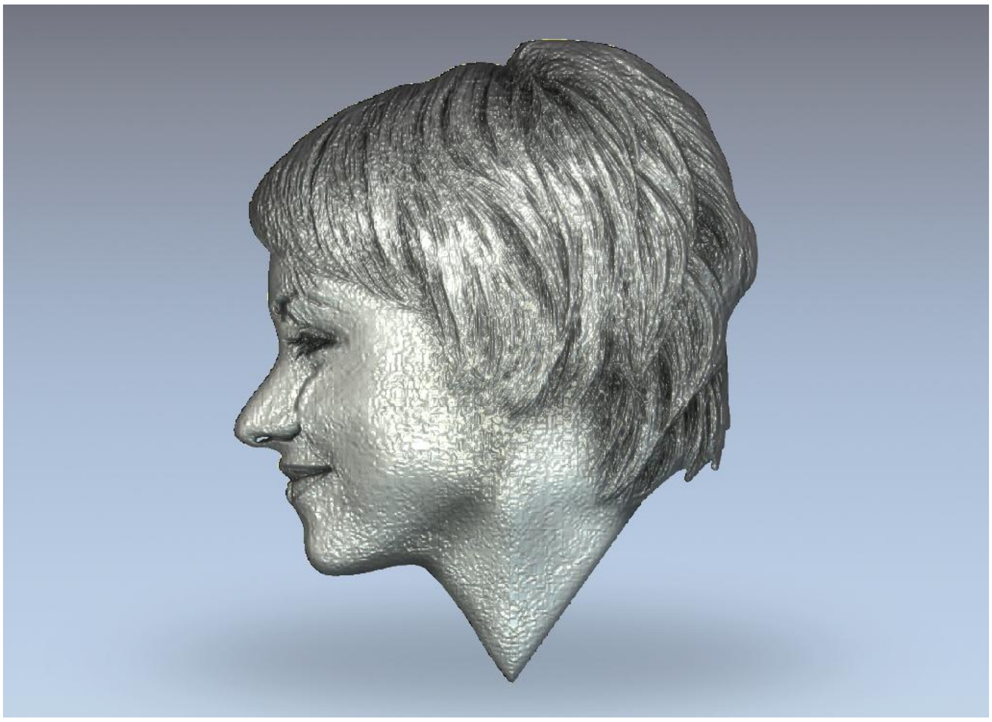
If you click **Next** without having specified the centre-front of the person's eyebrow and the nape of the neck, a message box is displayed warning that you must specify these points. Click **OK** to close the message box.

During the relief calculation process, a progress bar and cancel button are displayed on the **Status Bar**:



The calculated face relief is displayed in the **3D View** window, and the **Smoothing Tool** settings are displayed on the **Tool Settings** panel.

For example, the face shape created on the default relief layer is shown in the **3D View** window might look something like this:



The area of the imported image shown in the **2D View** window within the vector outline you have drawn is projected onto the surface of the relief layer.

21. From the **Design Tools** toolbar, use the sculpting tools to edit the shape of the face on the default relief layer shown in the **3D View** window.

22. Click **Close** to exit the **Face Wizard**.

For example, the sculpted face shape on the default relief layer might look something like this:

# Opening a model

You can open an ArtCAM model as part of a project or independently.

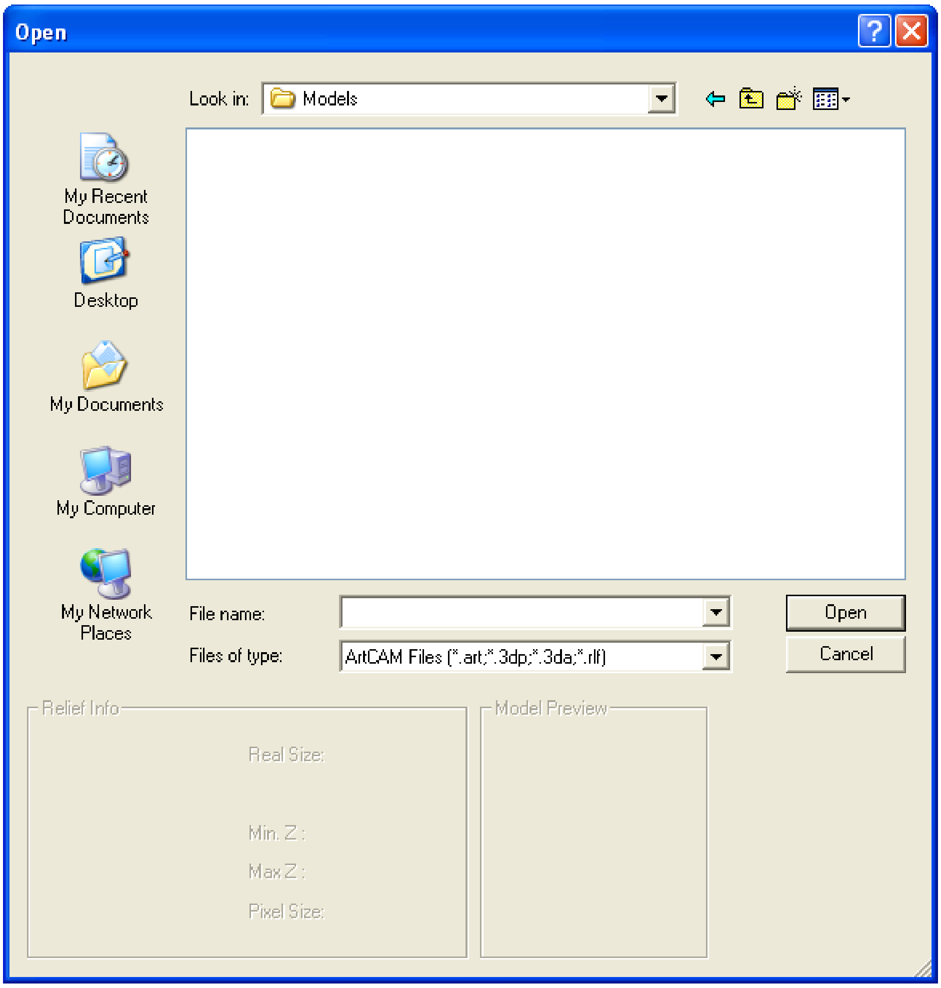
*If you are working in ArtCAM Insignia, you can only open a model independently.*

To open an ArtCAM model independently:

1. If you are opening an ArtCAM model with no other currently open, use any of the following methods to display the **Open** dialog box:

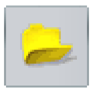
   - Click **Open Model** in the **Model** area;
   - From the **Menu Bar**, click **File > Open**; or

- Press the **Ctrl+O** keys.




*You cannot open* ***ArtCAM Project Files (*.3dp)*** *or* ***3D Assembly Files (*.3da)*** *as a model. If you select a file in either of these file formats, a project is opened instead. For details, see Opening a project (see page 92) and Creating a new project from a file (see page 90).*

If you are opening an ArtCAM model with a model already open in ArtCAM, use one of the following methods to display the **Open** dialog box:

- From the **Model** toolbar, click the **Open File**  button
- Press the **Ctrl+O** keys; or
- From the **Menu Bar**, click **File > Open**.

If you have not saved any changes that have been made to the open model before attempting to open another model, a message box is displayed asking if you want to save the changes to the model. For further details, see Saving a model (see page 84). If you have not made any changes to the model at all, or since it was saved previously, it is closed immediately.

2. Use one of the following methods to find the file you want to open as a model:

   - You can use the **Look in** list box and the **Up One Level** button to find the folder on your computer containing the file from which you want to create the new bitmap layer.
   - By default, ArtCAM lists **All Supported Files**. You can limit the files to a particular format by clicking the **Files of type** list box, followed by the specific file format you want to use.

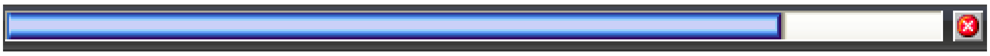
3. When you have located the file that you want to open, click the file name to select it. When you select the file, the **Relief Info** area of the dialog displays the file's dimensions in actual measurements and pixels. The **Model Preview** area displays a thumbnail image of the file's contents.

   If you select an ArtCAM model file (***.art**), the thumbnail image shown in the **Model Preview** area contains the contents of the active **2D View** window when the model was saved.

   *When choosing a bitmap image, the greyscale representation shown in the* ***Relief Info*** *area provides an indication of the form that the resulting relief shall take. Consider that the brightest areas of the greyscale image reflect the highest areas in the resulting relief, while the darkest areas reflect the lowest.*

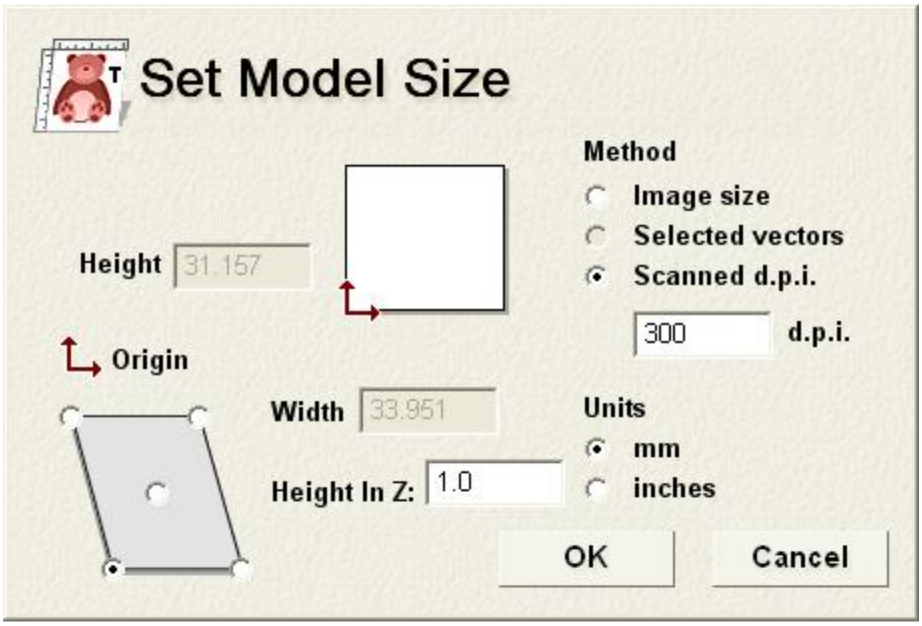
4. Click **Open**. Depending on which file you have selected, ArtCAM either opens the model immediately or begins the process of creating a model.

   When opening large files, a progress bar is displayed in the **Status Bar**:

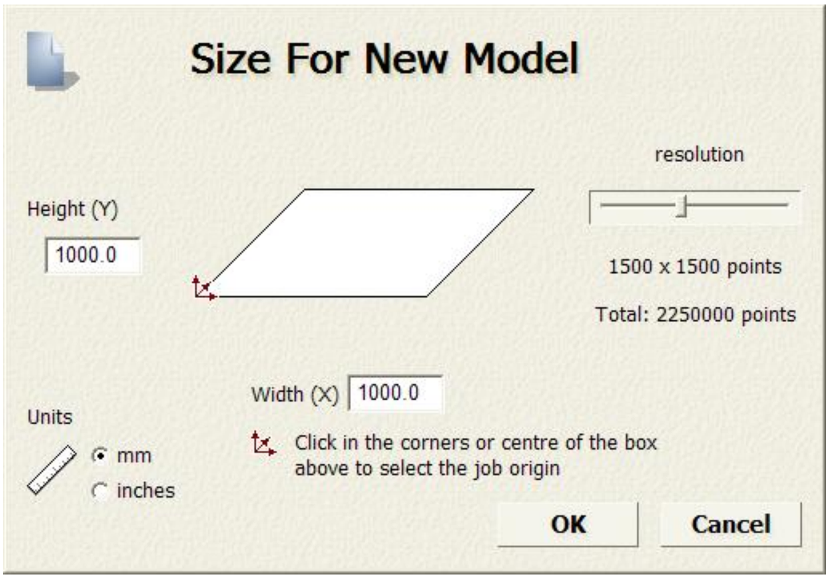
   

   If you select an ArtCAM Model file (***.art**), the model is opened.

If you select an image file, the **Set Model Size** dialog box is displayed:




If you are opening a Drawing Interchange (***.dxf**), AutoCAD 2D drawing (***.dwg**), DUCT picture (***.pic**) or Portable Document Format (***.pdf**) file, the **Size For New Model** dialog box is displayed:

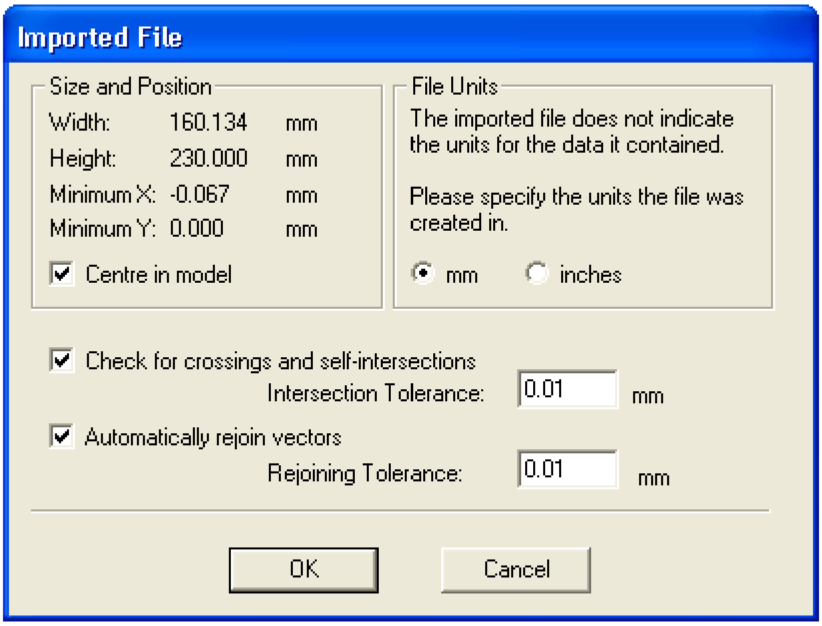



Click **OK** to close the **Size For New Model** dialog box.

If you are opening a Portable Document File (***.pdf**), ArtCAM creates a new model with a separate vector and bitmap layer for each of the pages within the PDF document and names each of the layers using the *file title (page number)* convention. For example, the layer relating to the document's first page might be named *Brochure (Page 1)*.

*ArtCAM does not currently support layers (or 'Optional Content Groups') within Portable Document Format files (*.pdf).*

If you are opening a Drawing Interchange (***.dxf**) or AutoCAD 2D drawing (***.dwg**) file, the **Imported File** dialog box is displayed:




If you click **Cancel**, a model is created equal to the dimensions previously seen in the **Size For New Model** dialog box, but the vector artwork in the original file is not included in the model.

To include the vector artwork, make sure that the settings are correct:

- To position the artwork in the model's centre, select the **Centre in model** check box.
- In the **File Units** area, make sure the units of measurement for the artwork is the same as those used in your ArtCAM model by selecting **mm** or **inches**.
- To identify all self-intersecting spans in the artwork within tolerance, make sure the **Check for crossings and self-intersections** check box is selected, and that the correct tolerance is specified in the **Intersection Tolerance** box.

*When selected, self-intersecting spans are shown in red, with white circular shapes marking the positions where spans overlap.*

- To rejoin any spans in the artwork that have been *exploded* within tolerance, make sure the **Automatically rejoin vectors** check box is selected, and that the correct tolerance is specified in the **Rejoining Tolerance** box.

Click **OK** to close the dialog box, create the model and import the vector artwork.

The layers in the imported file are added to the vector layer stack above the previously active layer. Each new vector layer uses the name of the layer in the imported file, as well as any colour.

If no colour has been applied to a layer in the imported file, the new vector layer is coloured red by default.

If the name of a layer in the imported file is already used by a vector layer in the model, the artwork on the file's layer is added to the vector layer with the same name.

If the imported file does not contain any vector artwork, a message box is displayed warning that the file cannot be used. Click **OK** to close the message box.

To open an ArtCAM model within a project:

1. From the **Project** panel, right-click the closed model in the Project Tree that you want to open, then click **Edit** in the context menu.

   *You can also double-click the closed model in the Project Tree to open the model.*

   The model's icon in the Project Tree changes from to , indicating that it is now open.

   The open model is always last in the list of models associated with **Models** in the Project Tree.

## Opening recent models

When working on models independently, you can quickly open any of your four most recent models.

Use one of the following methods to open a recent model:

- From the **Start** panel, click the model's name displayed below **Open Model** in the **Models** area; or
- From the **Menu Bar**, click **File** followed by the model's name listed above the **Exit** option.

A combination of up to four models and projects are listed.

*If you are working in ArtCAM Insignia, projects cannot be opened and therefore are not listed.*

*If you position the mouse cursor over the icon beside any of the four models listed in the* **Models** *area of the* **Start** *panel, its location on your computer is displayed.*

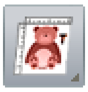
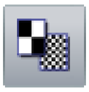
# Setting the model resolution

You can adjust the resolution of an open model. This enables you to disassociate the bitmap resolution from the model resolution.

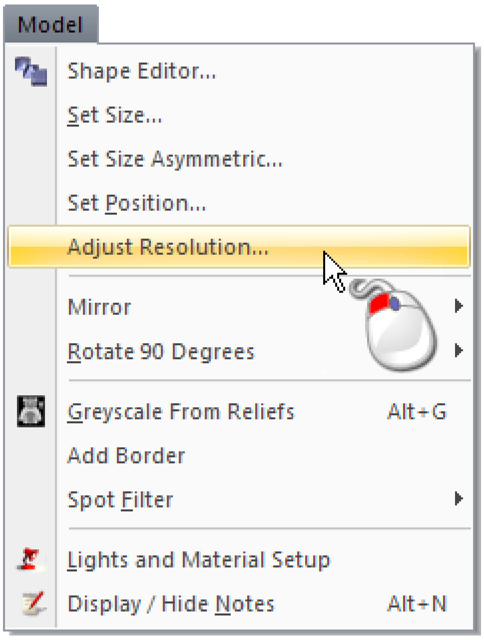
In pictures, the size of the pixels determines the resolution of the picture. Resolution is the number of pixels per inch (ppi) or centimetre. A picture is created at a specific resolution. You select the resolution based on how you will use the image in your model. Consider that too low a resolution causes pixelation, or large pixels that cause a coarse relief, while too high a resolution add to picture's memory requirements without producing a significant increase in its overall finish.

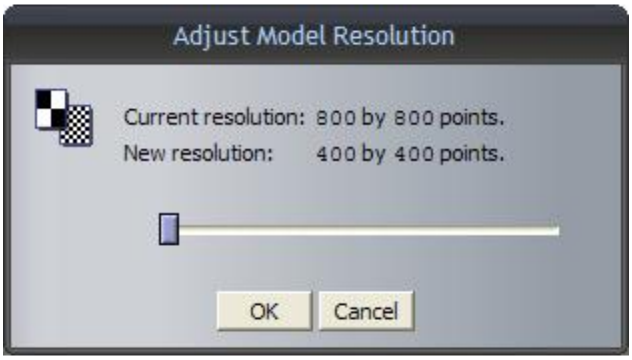
To set the resolution of an open model:

1. Use one of the following methods to display the **Adjust Model Resolution** dialog box:
   - From the **Project** panel, right-click the model in the Project Tree, then click **Adjust Resolution...** in the context menu;

- From the **Model** toolbar, click and hold on the **Set Model Size**  button, move the mouse cursor over the **Adjust Model Resolution**  button in the toolset displayed, then release the mouse button:

- From the **Menu Bar**, click the **Model > Adjust Resolution...** option;






The model's resolution is displayed in the **Current Resolution** area.

2. Click and drag on the slider to set the model's resolution according to your job requirements.

Drag the slider to the right to increase the resolution of the model. Drag the slider to the left to decrease the resolution. The new resolution of the model is displayed in the **New Resolution** area.

3. Click **Apply** to set the model's resolution.

*You cannot use the **Undo** button to restore the previous resolution of the model. You must be certain that you want to adjust the resolution to that shown in the **New Resolution** area before clicking **Apply**.*

Alternatively, click **Cancel** to close the dialog box and keep the current model resolution.

# Saving a model

You can save a model as an ArtCAM Model file (***.art**) only.

To save a model you are working on as part of the current project:

- From the **Project** panel, right-click the open model in the Project Tree, then select **Update Project** in the context menu.

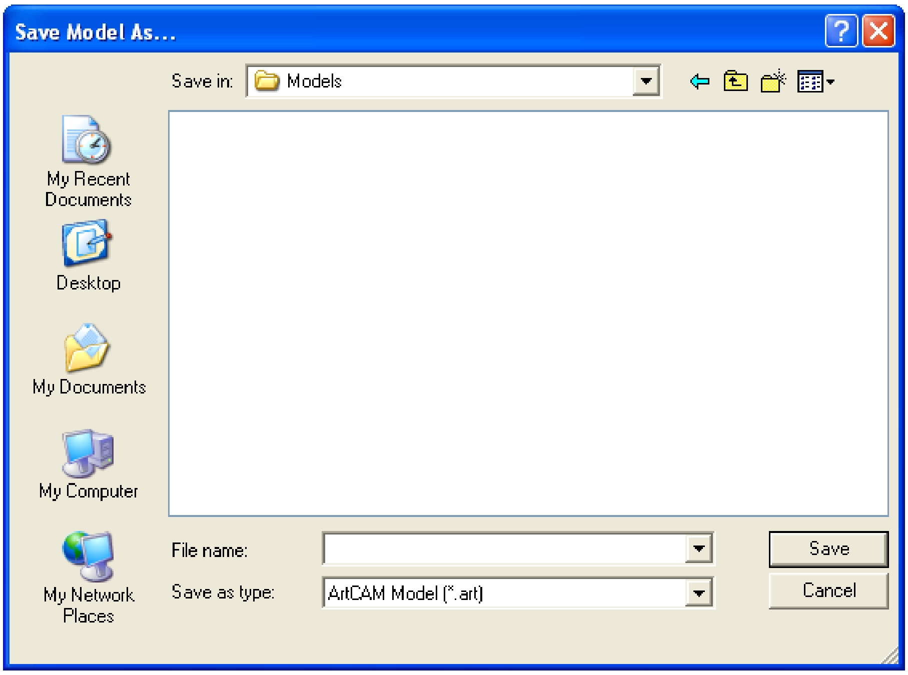
*The updated model is not saved as part of the project until the project itself is saved. For details, see Saving a project (see page 93).*

Use one of the following methods to save an independent model:

- From the **File** toolbar, click the **Save**  button;
- From the **Menu Bar**, click **File > Save**; or
- Press the **Ctrl + S** keys.

If you have previously saved the model, your changes are saved immediately; overwriting the previous model file (***.art**).

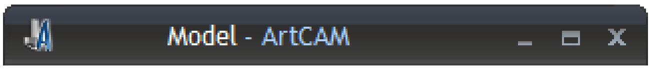
If you are saving the model for the first time, the **Save Model As...** dialog box is displayed:



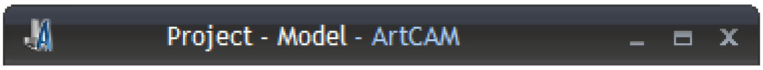

*The **ArtCAM Model (*.art)** option shown in the **Save as type** list box is the only file format in which you can save the model.*

1. Click the **Save in** list box or use the **Up One Level**  button to navigate to the folder on your computer in which you want to save the model.
2. In the **File name** box, type the name you want to give to the model.
3. Click **Save** to save the model and close the dialog box.

*If you are working on an independent model, it's name is shown to the right of the control icon in the* ***Title Bar****:*




*If you are working on a model as part of a project, the model's name is shown to the right of the project's name in the* ***Title Bar****:*




*For further information, see Understanding the Title Bar (see page 44).*

# Closing a model

To close the model that you are working on as part of a project:

1. From the **Project** panel, right-click the open model in the Project Tree, then select **Close** in the context menu.

   If you have not made any changes to the model since it was opened, it closes immediately.

   The model icon in the Project Tree changes to ; indicating that it is now closed.

   If you attempt to close the model before saving any changes that you have made since it was opened, a message box is displayed asking if you want to update the project with your changes.

   To save the changes to the model before it is closed:

   - Click **Yes**. The message box closes and the model icon in the Project Tree changes to ; indicating that it is now closed.

   To close the model without saving any changes since it was either created or saved previously:

   - Click **No**. The message box closes and the model icon in the Project Tree changes to ; indicating that it is now closed.

To close the independent model that you are working on:

1. From the **Menu Bar**, click the **File > Close Model** option.

   If you click the **File > Close Model** option before saving a new model, or any changes that you may have made to a previously saved model, a message box is displayed asking if you want to save your changes.

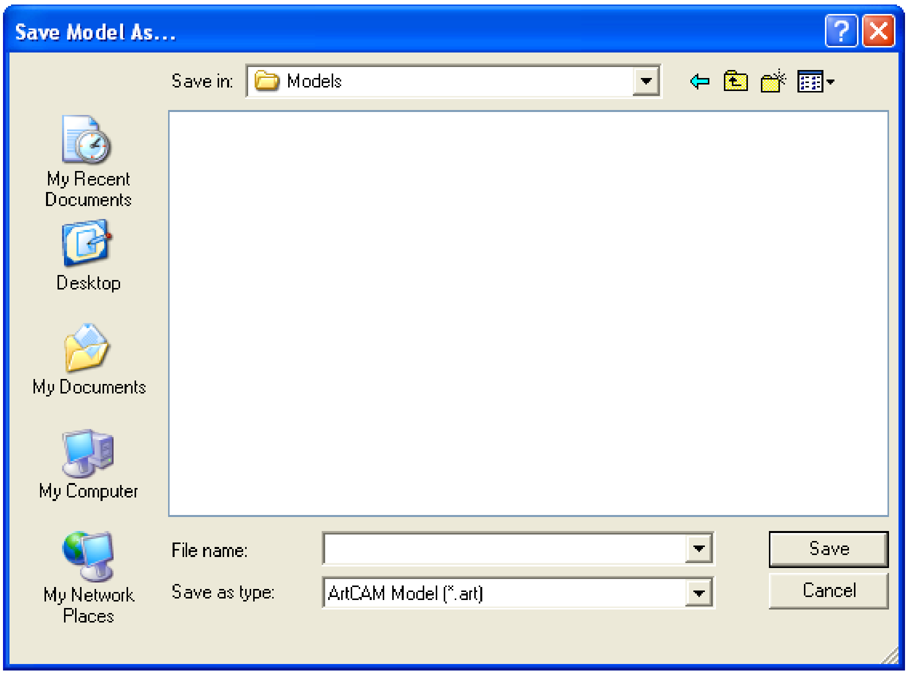
If you do not want to save the changes to the model before it is closed:

- Click **No**. The message box and model closes, and the **Start** panel is displayed.

To save the changes to the model before it is closed:

- Click **Yes**.

If you are saving the model for the first time, the **Save Model As...** dialog box is displayed:




To save the model:

a. Click the **Save in** list box, followed by the folder on your computer in which you want to save the model.

b. In the **File name** box, specify the name you want to give to the model.

c. Click **Save** to close the dialog box and the model. The **Start** panel is displayed.

*The **ArtCAM Model (*.art)** option shown in the **Save as type** list box is the only file format in which you can save the model.*

If you are working with a previously saved model, any changes made are saved immediately; overwriting the previous model file (***.art**).

*The names of the four most recent saved models are listed below the* ***Open Model*** *in the* ***Models*** *area of the* ***Start*** *panel, as well as in the* ***File*** *menu on the* ***Menu Bar****.*

If a previously saved model has not been changed since it was opened, the model closes and the **Start** panel is displayed.

# Creating projects

*If you are working in ArtCAM Insignia, you cannot create a project.*

An ArtCAM project hosts any number of ArtCAM models, assemblies and triangle mesh replicas. Each ArtCAM model within a project contains vector and bitmap artwork, reliefs and toolpaths. Each assembly hosts replica triangle meshes of ArtCAM models, material information or additional assemblies. Provided a replica mesh has been created from an ArtCAM model stored as part of the same project, it hosts a link to the ArtCAM model file from which it originates.

When you start ArtCAM, the **Start** panel is displayed on the left. There are two ways to create a project from the **Start** panel. These are:

- In the **Projects** area, click **New Project**. This enables you to create a new project (see page 89).
- In the **Projects** area, click **Open Project**. This enables you to choose a compatible file from which you can create a project (see page 90).

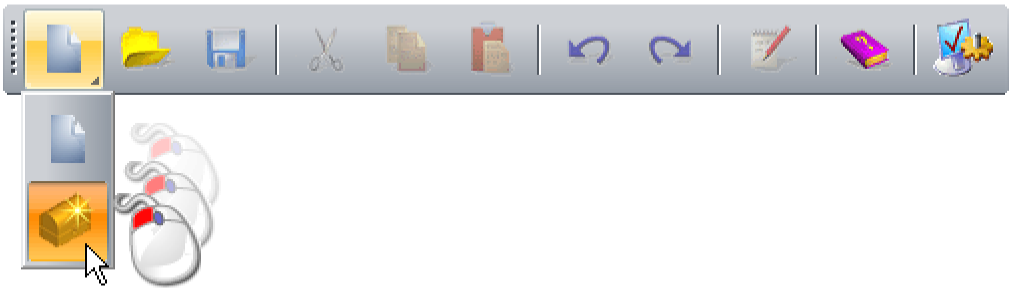
You can also create a project from the **Menu Bar**:

- Click **File > New > Project**. This enables you to create a new project (see page 89).

## Creating a new project

To create a new ArtCAM Project, use one of the following methods:

- From the **Start** panel, click **New Project** in the **Projects** area;
- From the **Menu Bar**, click **File > New > Project**; or
- In the **File** toolbar, click and hold on the **New Model** button, move the mouse cursor over the **New Project** button in the toolset displayed, and then release the mouse button.



The **Project** panel is docked and pinned on the right, in which is displayed the Project Tree. The Project Tree comprises three default items:

- **Project**, which is named *(Untitled)* by default.

*You can give the project a name when it is saved for the first time. For details, see Saving a project (see page 93).*

- **Models**, which hosts any number of ArtCAM models. For further details, see Creating models (see page 56).
- **Assembly**, which hosts any number of assemblies and their associated replica meshes.

You cannot delete or rename these items.

# Creating a new project from a file

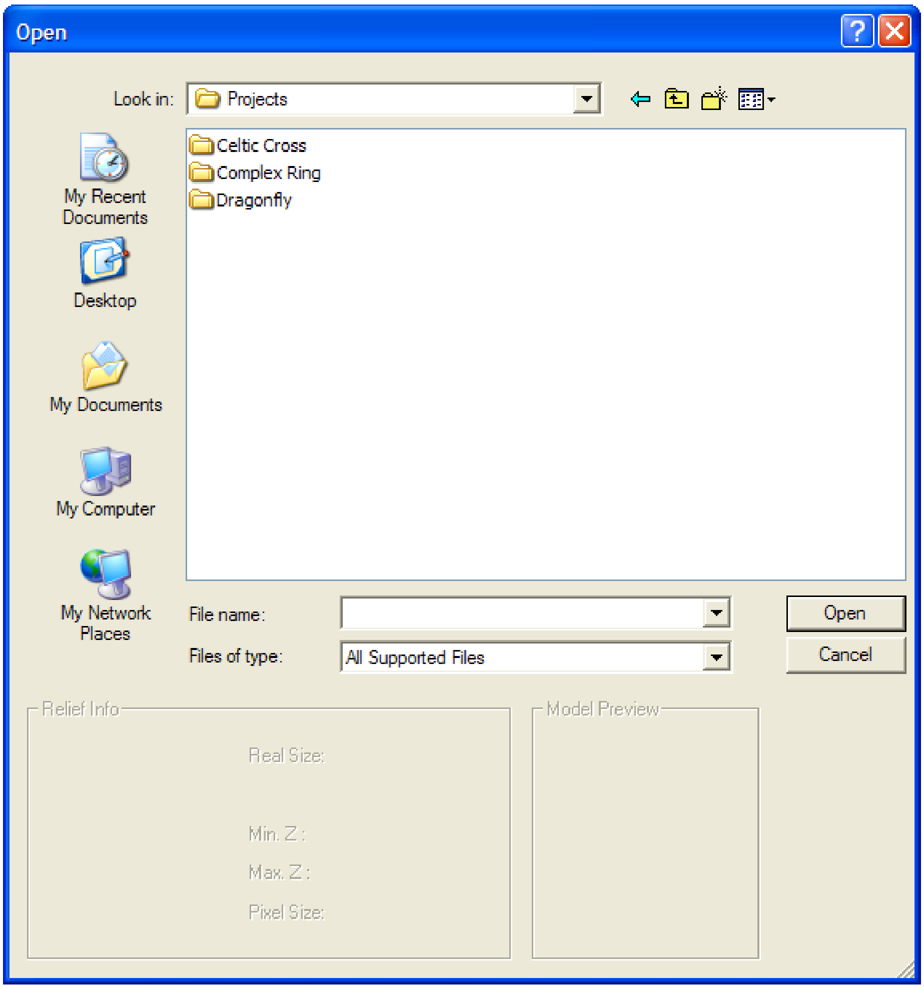
You can create a project using the ArtCAM **Open** dialog box, which contains standard Windows features.

You can create a new project by opening an ArtCAM Assembly (***.3da**) file.

To create a new project from an ArtCAM Assembly (***.3da**) file:

1. Use one of the following methods to display the **Open** dialog box:

- From the **Start** panel, click  **Open Project** in the **Projects** area;
- From the **Menu Bar**, click the **File > Open...** option; or
- Press the **Ctrl+O** keys.




2. In the **Files of Type** list box, make sure the **ArtCAM File (*.art; *.3dp; *.3da; *.rlf)** option is selected
3. Click the **Look in** list box, and then select the folder on your computer in which the file that you want to open is stored.
4. When you have found the file, click its file name. Its name is displayed in the **File name** box.
5. Click the **Open** button to open the file and display the **Project** panel.

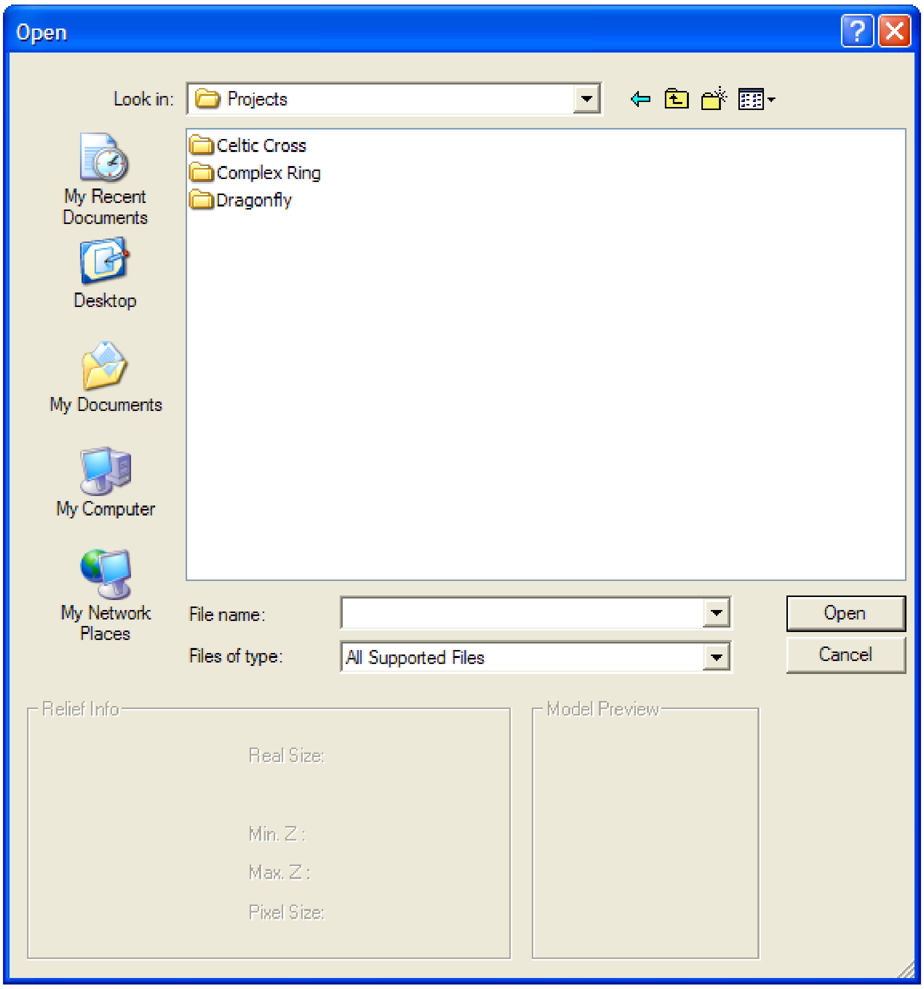
   The **Project** panel contains the Project Tree and a selection of tools that can be used to manipulate the default items in the Project Tree, as well as the items you subsequently create or import.

# Opening a project

To open an ArtCAM project:

1. Use one of the following methods to display the **Open** dialog box:

   - From the **Start** panel, click **Open Project** in the **Projects** area.
   - From the **Menu Bar**, click **File > Open**; or
   - Press the **Ctrl+O** keys.



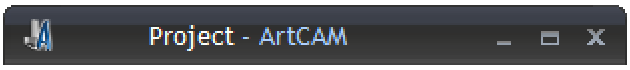

2. Select an **ArtCAM Project File (*.3dp)** or **ArtCAM Assembly File (*.3da)** you want to work with, and then click **Open**.

   On the **Project** panel, the assemblies and replica meshes in your project are displayed in the Project Tree.

The root **Assembly** is selected and its associated tools are displayed below the splitter bar.

The project's name is displayed:

- in the **Title Bar** to the right of the control icon ; and




- on the **Project** panel, beside at the root of the Project Tree.

## Opening recent projects

You can open one of the last four projects you have been working on.

*If you are working in ArtCAM Insignia, you cannot open a project.*

Use one of the following methods to open a recent project:

- From the **Start** panel, click the project's name in the **Recent Files** area.

*If you position the mouse cursor over the icon beside any of the projects listed in the* ***Recent Files*** *area of the* ***Start*** *panel, its location on your computer is shown.*

- From the **Menu Bar**, click the **File** option, then the project's name listed above the **Exit** option. A combination of up to four ArtCAM projects and models are listed here.

# Saving a project

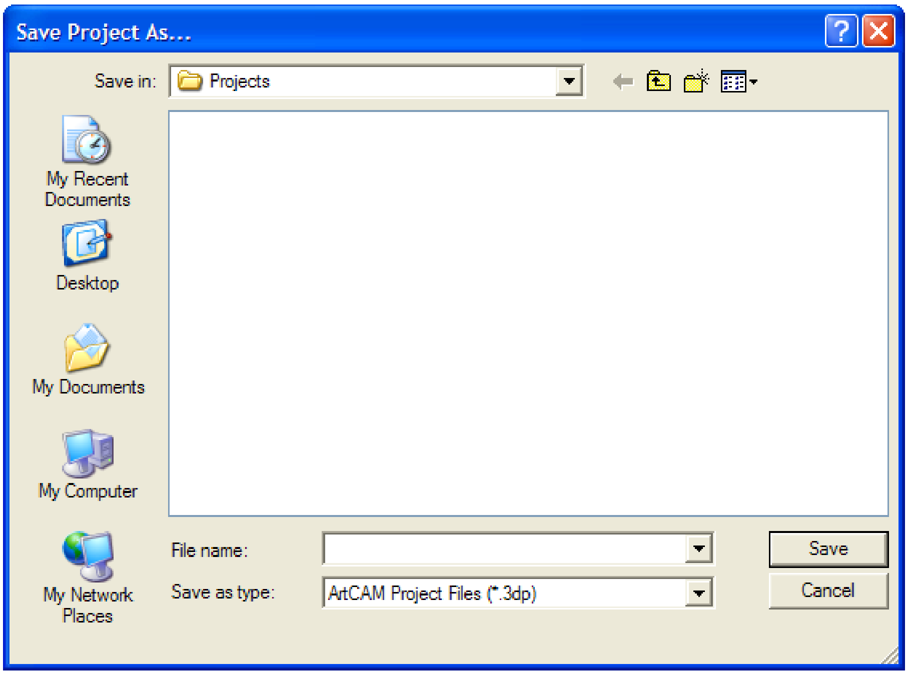
You can save a project as an ArtCAM Project file (***.3dp**) only.

Use one of the following methods to save the project you are currently working on:

- From the **File** toolbar, click the **Save** button;
- From the **Menu Bar**, click **File > Save**; or
- Press the **Ctrl+S** keys.

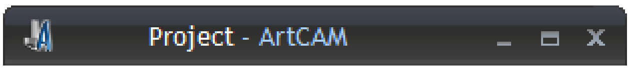
If you have previously saved the project, your changes are saved immediately; overwriting the previous project file (***.3dp**).

If you are saving the project for the first time, the **Save Project As** dialog box is displayed:




1. Click the **Save in** list box, followed by the folder on your computer in which you want to save the project.
2. In the **File name** box, type the name you want to give to the project.
3. Click **Save** to save the project and close the **Save Project** dialog box.

*The project's name is shown to the right of the control icon*  *in the* ***Title Bar****:*




*For further information, see Understanding the framework (see page 7).*

To save a previously saved project with a different file name:

1. From the **Menu Bar**, click the **File > Save As...** option. The **Save Project As** dialog box is displayed.
2. Click the **Save in** list box, followed by the folder on your computer in which you want to save the project.
3. In the **File name** box, type the new name you want to give to the project.
4. Click **Save** to save the project and close the **Save Project As** dialog box.

The new name given to the project replaces that which was originally shown to the right of the control icon in the **Title Bar**.

# Closing a project

To close the project that you are working on, click **File > Close Project** from the **Menu Bar**.

If you click **File > Close Project** before saving a new project, or any changes that you may have made to the currently open project, a message box is displayed asking if you want to save the project or not.

To save and close the project, click **Yes**. The saved project's name is listed:

- on the **Start** panel, in the **Recent Projects** area; and
- in the **Menu Bar**'s **File** menu.

If you close the project with a model still open, the model is also closed.

If you do not want to save the project before it is closed, click **No**.

If a previously saved project has not been changed since it was opened, the project closes and the **Start** panel is displayed.

# The 2D design process

After a new ArtCAM model is created, you must import or create the 2D artwork that is the foundation of your 2D/2.5D toolpaths or 3D reliefs.

Bitmap artwork in an ArtCAM model can start as an imported image comprising a large number of colours. For example, a photograph, downloaded graphic, or scanned document. The level of detail in an imported image is most often not required, and ArtCAM provides several tools that allow you to quickly reduce, replace or remove colours, so that only the essential artwork is left. When you are not using an imported image as the basis of your design, there are many painting and drawing tools that you can use to create your own bitmap artwork.

Vector artwork in an ArtCAM model can be imported or created from a model's bitmap artwork. Just as with bitmaps, there are many powerful tools that enable you to edit vectors. If you are not using imported vector or bitmap artwork as part of your model, there are a variety of drawing tools that you can use to create original vector designs.

The process of preparing your artwork can be accelerated by using a Portable Document Format (***.pdf**) file as the basis of a new model; provided that it already contains both vector and bitmap images. ArtCAM extracts the vectors and bitmaps from the pages of the PDF file and places the artwork on separate layers.

## Using bitmap layers

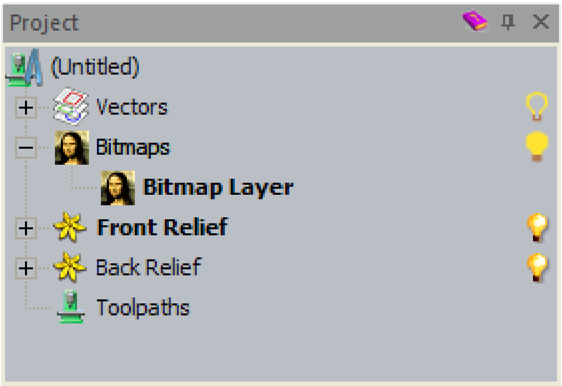
Bitmap layers are contained in a stack. The stack is displayed on the **Project** panel and the **Bitmap Layers** panel.

You can use one of the following methods to display the bitmap layer stack:

- On the **Project** panel, click beside **Bitmaps** in the Project Tree; or
- Click the **Bitmap Layers** panel.

*To display the **Bitmap Layers** panel, right-click a docking area, toolbar or the **Status Bar**, then select **Bitmap Layers** from the context menu.*

When working from the **Project** panel, the bitmap layer stack is displayed in the Project Tree:




Every ArtCAM model contains an empty default bitmap layer, named *Bitmap Layer*. However, you can create as many bitmap layers as you want; either empty, or from imported artwork.

## Importing bitmap artwork

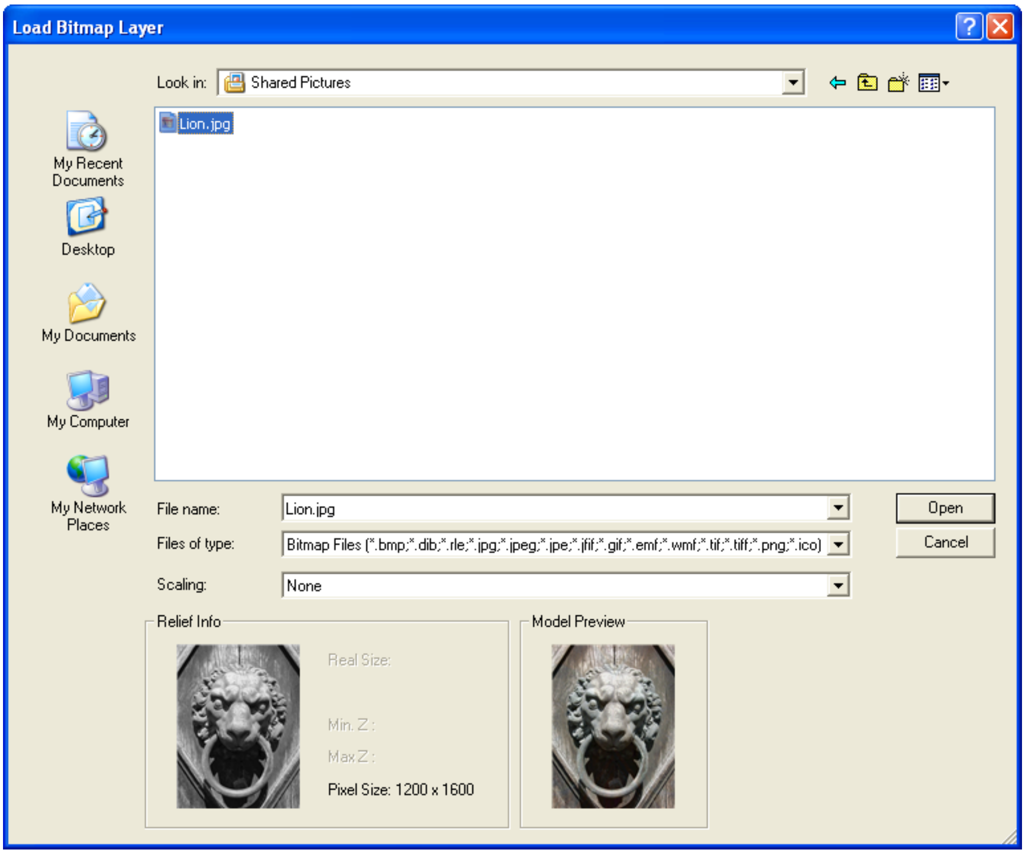
You can import an image saved in one of the following file formats as a new bitmap layer:

- ArtCAM Model (***.art**);
- ArtCAM Relief (***.rlf**);
- Windows or OS/2 Bitmap (***.bmp**);
- Windows or OS/2 DIB (***.dib**);
- Windows or CompuServe RLE (***.rle**);
- JPEG Image - JFIF Compliant (***.jpg**, ***.jpeg**, ***.jpe** and ***.jfif**);
- CompuServe Graphics Interchange (***.gif**);
- Windows Meta File (***.wmf**);
- Windows Enhanced Meta File (***.emf**);
- Tagged Image File Format (***.tif** and ***.tiff**);

- Portable Network Graphics (***.png**); or
- Windows Icon (***.ico**).

To import an image file as a new bitmap layer:

1. Use one of the following methods to display the **Load Bitmap Layer** dialog box:
   - From the **Project** panel, right-click **Bitmaps** in the Project Tree, then select **Import** in the context menu;
   - From the **Menu Bar**, click the **Bitmaps > Load Layer** option; or
   - From the **Bitmap Layers** panel, click the **Import Image** button.




2. Use one of the following methods to find the file that you want to import:
   - You can use the **Look in** list box and the **Up One Level** button to find the folder on your computer containing the file from which you want to create the new bitmap layer.
   - By default, ArtCAM lists **All Supported Files**. You can limit the files to a particular format by clicking the **Files of type** list box, followed by the specific file format you want to use.

3. When you have found the file, click to select the file listed in the main window of the **Load Bitmap Layer** dialog box. Its name is displayed in the **File name** box.

   In the **Model Preview** area, you can see a preview of the selected file. In the **Relief Info** area, a greyscale representation of the selected file is displayed, along with its dimensions in pixels.

   *The greyscale representation in the **Relief Info** area provides an indication of the form a resulting relief layer might take. The brightest areas of the greyscale image are the highest areas in the resulting relief layer, while the darkest areas are the lowest.*

4. To scale the image, click the **Scaling** list box and select:
   - **None** to leave the image unscaled.
   - **Fit** to maintain the aspect ratio of the image. If the image is equal to, or larger than, the model area (the white area in the **2D View** window), the image fills it without being cropped. If the image is smaller than the model area, ArtCAM adds borders around the image to compensate. The border uses the colour found in the top-left pixel of the image.
   - **Fill** to crop areas of the image outside of the model area. The selected image fills the model area and its original aspect ratio is maintained.
   - **Stretch** to fit the image to the model area. The image is resized to fill the model area; its original aspect ratio is not retained.

5. Click **Open** to import the image into the model as a new bitmap layer.

   The new bitmap layer is created directly above the previously active layer in the stack, and uses the name of the file from which it is created.

   A unique colour palette comprising the colours within the imported artwork is also created. When the bitmap layer is active, its associated colour palette is displayed below the **2D View** and **3D View** windows.

## Choosing the active layer

Only one bitmap layer can be active at a time, and it is the artwork on this layer which is displayed.

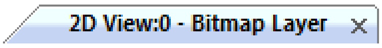
To choose the active bitmap layer:

1. Use one of the following methods to display the bitmap layer stack:
   - From the **Project** panel, click ⊞ beside **Bitmaps** in the Project Tree; or
   - Click the **Bitmap Layers** panel.

   *To display the* ***Bitmap Layers*** *panel, right-click a docking area, toolbar or the* ***Status Bar****, then select* ***Bitmap Layers*** *from the context menu.*

2. Click the layer you want to work with. Its name is bold and highlighted.

   The bitmap layer's name is displayed on the **2D View** tab:

   This is useful if you are working with the **Project** and **Bitmap Layers** panels auto-hidden or closed.

   The colours within the bitmap layer's artwork are displayed in their own colour palette, shown below the **2D View** and **3D View** windows.

## Creating a new layer

In an ArtCAM model, you can create as many bitmap layers as you like. This enables you to include many different images in a single model.

You can use one of the following methods to create a new bitmap layer:

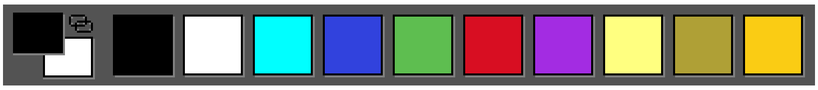
- From the **Project** panel, right-click **Bitmaps** in the Project Tree, then click **New** in the context menu;
- From the **Menu Bar**, click the **Bitmaps > Create New Layer** option; or
- From the **Bitmap Layers** panel, click the **New Bitmap Layer** button.

By default, each new bitmap layer you create is:

- named *Bitmap Layer*;
- numbered sequentially;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active; and

- visible in the **2D View** window.

*A default colour palette comprising ten basic colours is associated with each of the empty bitmap layers that you create, and this is displayed below the* ***2D View*** *and* ***3D View*** *windows:*



*When a bitmap layer results from an imported image file, a new colour palette with a greater range of colours is displayed.*

## Renaming a layer

In every new ArtCAM model, the default bitmap layer is named *Bitmap Layer*. Each bitmap layer you create uses this name together with a unique number. When you import bitmap artwork, the new bitmap layer uses the name of the image file from which it is created. You can rename any of the bitmap layers in a model.

To rename a bitmap layer:

1. Use one of the following methods to display the layer's name box:
   - From the **Project** panel, right-click the bitmap layer you want to rename in the Project Tree, then click **Rename** in the context menu.
   - From the **Bitmap Layers** panel, double-click the layer you want to rename in the stack.
2. In the name box, type the name you want to give to the layer.

   *To revert to the name previously given to the layer, press the* ***Esc*** *key on your keyboard.*
3. Use one of the following methods to set the name of the layer:
   - Press the **Enter** key; or
   - Click in the empty space below the layer stack.

## Viewing a layer

You can control the bitmap artwork shown in the **2D View** and **3D View** windows by choosing a specific bitmap layer and controlling whether or not it is visible. When a bitmap layer is created, it is active and visible by default.

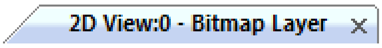
To control a bitmap layer's visibility:

1. If you are working in the **3D View** window, select the **Colour Shade** button in the **3D View** toolbar.

   *In the **3D View** window, all bitmap artwork is hidden unless the **Colour Shade** button is selected.*

2. Choose the layer (see page 99) containing the bitmap artwork you want to use.

   *The name of the currently active bitmap layer is displayed on the **2D View** tab:*


3. Use one of the following methods to control whether the bitmap artwork can or cannot be seen:

   - To hide the layer's artwork when working from the **Project** panel, click the icon beside **Bitmaps** in the Project Tree;
   - To hide the layer's artwork when working from the **Bitmap Layers** panel, click the **Toggle All Visibility** button;
   - To display the layer's artwork when working from the **Project** panel, click the icon beside **Bitmaps** in the Project Tree; or
   - To display of the layer's artwork when working from the **Bitmap Layers** panel, click the **Toggle All Visibility** button.

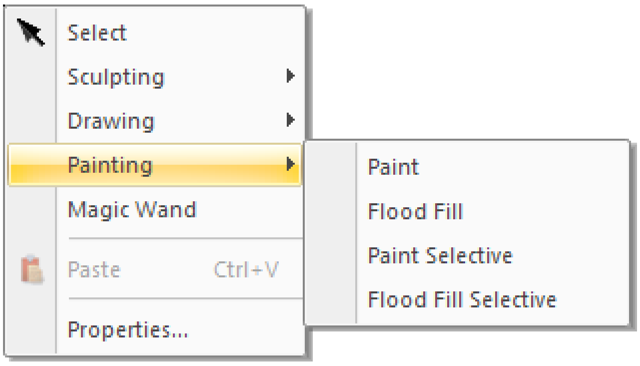
# Creating and editing bitmap artwork

Using the colour palette displayed below the **2D View** and **3D View** windows, and the bitmap tools available in ArtCAM, you can create an original design on an empty bitmap layer, or edit the artwork currently on a bitmap layer.

You can choose a bitmap painting or drawing tool from one of these places:

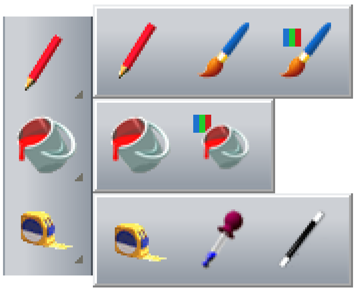
- The **2D View** or **3D View** window context menu; displayed by right-clicking in either window.

  For example, with the **2D View** window empty, its context menu includes the following options:

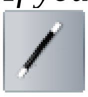



*If you are working in ArtCAM Insignia, the* ***Sculpting*** *and* ***Magic Wand*** *options are not displayed.*

- The **Design Tools** toolbar.



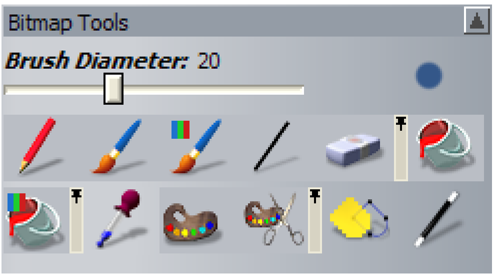
*If you are working in ArtCAM Insignia, the* ***Magic Wand***

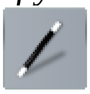


*button is not included.*

To display all of the buttons within a toolset on a toolbar, click and hold the button with ◢ displayed in the bottom-right. Release the mouse when its cursor is over the button that you want to select. You cannot pin any toolset on a toolbar.

- The **Bitmap Tools** area of the **Assistant** panel.

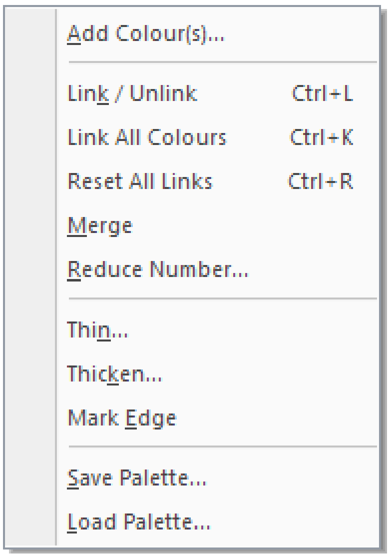



*If you are working in ArtCAM Insignia, the* ***Magic Wand***  *button is not included.*

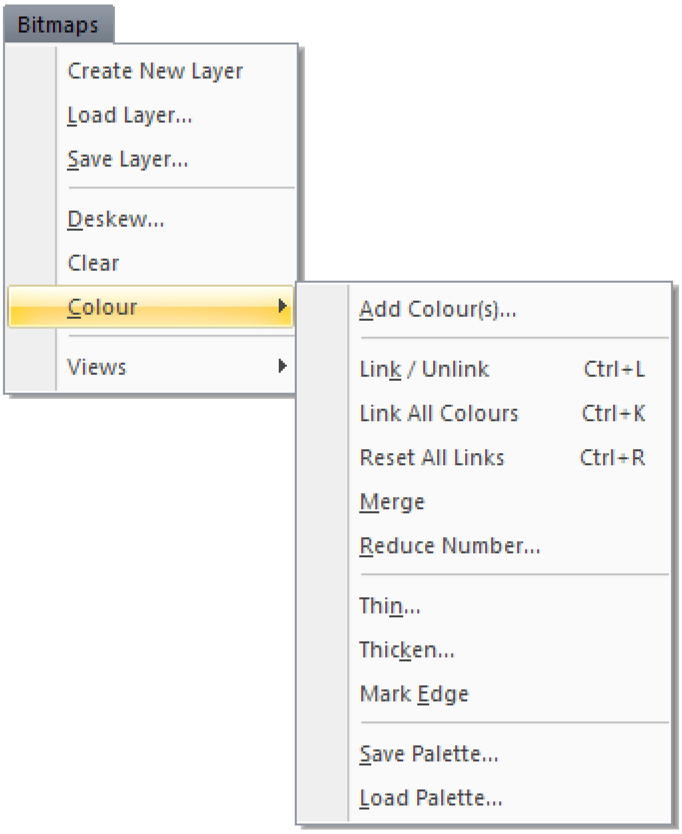
On the **Assistant** panel, to display all of the buttons within a toolset, click the ▸ icon on the right-edge of the button which is currently shown. To pin the toolset, click the icon along the right-edge of the last button in the toolset.

You can manage the colours in the colour palette using:

- The colour palette context menu; displayed by right-clicking the empty space immediately surrounding the colour palette:



- The **Bitmaps > Colour** menu; displayed from the **Menu Bar**:




## Reducing colours

You can reduce the number of colours in the artwork on a selected bitmap layer. ArtCAM takes colours of a similar shade in the artwork and produces an averaged colour from them. The averaged colour replaces the originals.

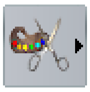
Reducing the number of colours helps to:

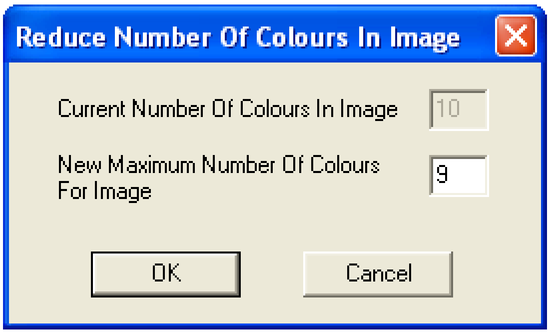
- remove unwanted colours;
- control the number of vectors created when converting bitmap artwork;
- control the shape of vectors created when converting bitmap artwork; and
- control the size of 3D shapes.

*Colour reduction should be done as an iterative process so that the number of colours can be minimised whilst preserving the detail in the original bitmap artwork.*

Reducing the number of colours in the bitmap artwork resets all colour links and shape attributes.

To reduce the number of colours in your bitmap artwork:

1. Select the bitmap layer (see page 99) in which you want to reduce the number of colours.
2. Use one of the following methods to display the **Reduce Number Of Colours In Image** dialog box:
   - Right-click the empty space surrounding the colour palette, and select **Reduce Colours** from the context menu;
   - From the **Menu Bar**, click the **Bitmaps > Colour > Reduce Number** option; or
   - Click the **Reduce Colours**  button from the Colour toolset shown in the **Bitmap Tools** area of the **Assistant** panel.

   The number of colours currently in the bitmap artwork is shown in the **Current Number Of Colours In Image** box.
3. In the **New Maximum Number Of Colours For Image** box, enter a number of colours between 2 and the current number of colours.

   *If you type 1 in the **New Maximum Number Of Colours For Image** box, a message box is displayed warning that you must use a number between 2 and the value equal to the total number of colours currently in the bitmap artwork. Click **OK** to close the message box, and then adjust the number in the box accordingly.*
4. Click **OK** to close the dialog box and reduce the number of colours in the bitmap artwork.

## Colour linking

Colour linking helps you to:

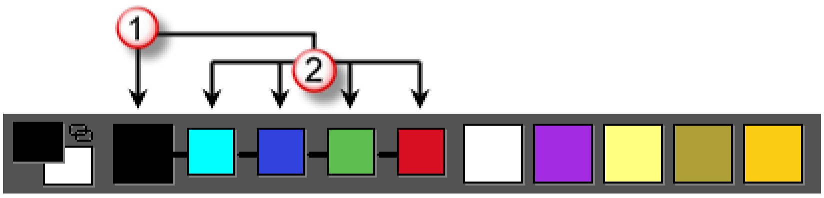
- change how different aspects of your bitmap artwork are displayed; and
- reduce the number of colours in your bitmap artwork, without removing them altogether.

You can link and unlink any of the colours within the colour palette shown below the **2D View** and **3D View** window.

When a colour is linked to the primary colour, it is:

- treated as the primary colour for as long as it remains linked;
- seen in the **2D View** and **3D View** windows as the primary colour; and
- shown in the colour palette as a smaller swatch, connected to the primary colour by a horizontal line.

For example, colours linked with the primary colour are shown as follows:




1 *Primary colour.*

2 *Colours linked to primary colour.*

You can one of the following methods to toggle the linking of the primary and secondary colours:

- From the **Menu Bar**, click the **Bitmaps > Colour > Link/Unlink** option;
- In the colour palette, double right-click the secondary colour's swatch;
- In the colour palette, click the icon; or
- Press the **Ctrl+L** keys.

You can also use your mouse to link and unlink any of the colours in the colour palette.

Colour links cannot be shared between colour palettes. Any colour links in the current colour palette are associated with the currently active bitmap layer only. If you choose another bitmap layer, its own colour palette is displayed along with its own set of colour links.

To link colours:

1. In the colour palette, click and drag an unlinked colour swatch onto the colour swatch with which you want to link.

   *When over a colour swatch, the mouse cursor is displayed as* .

   *When outside of a colour swatch, the mouse cursor is displayed as* .

2. Release the mouse button to link the colours. In the colour palette, the swatch of the linked colour is now smaller, and joined to the other colour swatch by a short horizontal line.

To unlink colours:

1. In the colour palette, click and drag a linked colour swatch onto the colour swatch with which it is already linked.
2. Release the mouse to unlink the colours. In the colour palette, the unlinked colour swatch returns to its original position.

## Linking All Colours

You can use one of the following methods to link all colours in the colour palette, with the exception of the secondary colour and primary colour:

- Right-click the empty space surrounding the colour palette, then select **Link All Colours** from the context menu;
- From the **Menu Bar**, click the **Bitmaps > Colour > Link All Colours** option; or
- Press the **Ctrl+K** keys.

## Unlinking All Colours

You can use one of the following methods to unlink all linked colours in the colour palette:

- Right-click the empty space surrounding the colour palette, then select **Reset All Links** from the context menu;
- From the **Menu Bar**, click the **Bitmaps > Colour > Reset All Links** option; or
- Press the **Ctrl+R** keys.

## Selecting the primary and secondary colours

There are two colours to consider when working with bitmap artwork: the primary colour and the secondary colour.

The primary colour influences how the following are created from bitmap artwork:

- vector artwork; and
- three-dimensional shapes.

ArtCAM can create vectors around the outline of all areas of bitmap artwork in the primary colour; as well as any other areas in colours currently linked to the primary colour.

You can also use the primary and secondary colours to specify particular areas of the relief that you do or do not want to edit.

The primary colour controls the colour of:

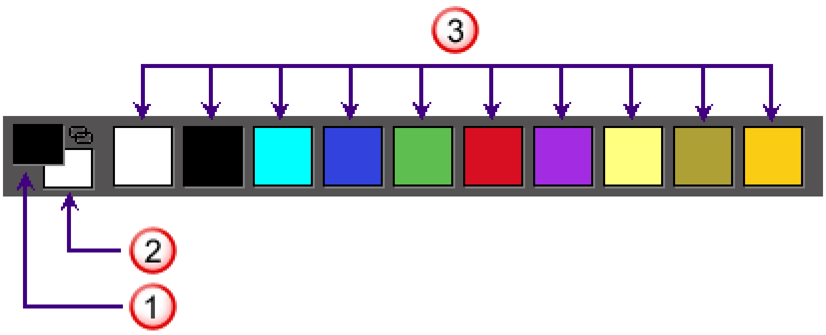
- the **Paint** tool;
- the **Paint Selective** tool;
- the **Draw** tool;
- the **Flood Fill** tool;
- the **Flood Fill Selective** tool; and
- the **Depth Colour**, when used in a toolpath simulation.

The secondary colour is used:

- as another colour choice for the **Paint** tool;
- as another colour choice for the **Draw** tool;
- to restrict the **Paint Selective** tool; and
- to restrict the **Flood Fill Selective** tool.

You can specify the primary and secondary colours:

- Using your mouse, by clicking the colours from the colour palette shown beneath the **2D View** and **3D View** window:



1 *Primary colour.*

2 *Secondary colour.*

3 *Available colours.*

- Using the **Pick Colour** tool, by clicking the colours directly from the artwork on the selected bitmap layer.
- If you are working in the **3D View** window, make sure that the **Colour Shade** button in the **3D View** toolbar is toggled on.

Use one of the following methods to select the primary colour:

- In the colour palette, click the colour you want to use;
- In the **Design Tools** toolbar, click and hold on the **Measure Tool** button, move the mouse cursor over the **Pick Colour** button in the toolset displayed, and then release the mouse button. Move the dropper over a colour in the bitmap artwork, then click; or
- Click the **Pick Colour** button in the **Bitmap Tools** area of the **Assistant** panel, move the dropper over a colour in the bitmap artwork, then click.

Use one of the following methods to select the secondary colour:

- In the colour palette, right-click the colour you want to use;
- In the **Design Tools** toolbar, click the **Pick Colour** button, move the dropper over a colour in the bitmap artwork, hold down the **Shift** key, then click; or
- Click the **Pick Colour** button in the **Bitmap Tools** area of the **Assistant** panel, move the dropper over a colour in the bitmap artwork, hold down the **Shift** key, then click.

## Using the Paint tools

The **Paint** tool enables you to apply the primary or secondary colour to the currently active bitmap layer. The **Paint Selective** tool enables you to restrict your painting to the areas using the current secondary colour. You can use these tools in the **2D View** or **3D View** window.

*When using the **Paint** and **Paint Selective** tools in the **3D View** window, the **Colour Shade** button in the **3D View** toolbar must be toggled on.*

To paint in the primary or secondary colour:

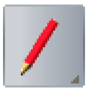
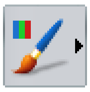
1. Choose the bitmap layer (see page 99) on which you want to paint.
2. Make sure the bitmap layer is visible (see page 101).
3. Use one of the following methods to select the **Paint** tool:
   - In the **Design Tools** toolbar, click and hold on the **Draw** button, move the mouse cursor over the **Paint** button, then release the mouse button.
   - Right-click the **2D View** or **3D View** window, then click **Painting > Paint** in the context menu;
   - From the Assistant panel, click the **Paint** button in the Drawing toolset shown in the **Bitmap Tools** area.

   *For information on how to display any of the toolsets in the **Assistant** panel, see Understanding the Assistant panel (see page 36).*
4. Change the primary and secondary colours (see page 109) to the colours you want to use.
5. Click the Toggle Brush Shape icon to set the brush shape. You can toggle between a round and a square tip.
6. Use one of the following methods to increase or decrease the size of the brush:
   - Hold down the **Shift** key, then roll the middle wheel on your mouse;
   - Drag the **Brush Diameter** slider on the **Tool Settings** panel; or

- Drag the **Brush Diameter** slider in the **Bitmap Tools** area of the **Assistant** panel.

7. Position the brush cursor over the area of the bitmap artwork in which you want to paint:
   - To paint in the primary colour, click and drag.
   - To paint in the secondary colour, hold down the **Shift** key, then click and drag.

To restrict your changes to those areas currently shown in the secondary colour:

1. Choose the bitmap layer (see page 99) in which you want to paint.
2. Make sure the bitmap layer is visible (see page 101).
3. Use one of the following methods to select the **Paint Selective** tool:
   - In the **Design Tools** toolbar, click and hold on the **Draw**  button, move the mouse cursor over the **Paint Selective**  button, then release the mouse button.
   - Right-click the **2D View** or **3D View** window, then click **Painting > Paint Selective** in the context menu; or
   - From the **Assistant** panel, click the **Paint Selective**  button in the Drawing toolset shown in the **Bitmap Tools** area.
4. Change the primary colour (see page 109) to the colour you want to use.
5. Change the secondary colour (see page 109) to the colour you want to replace.
6. Click the Toggle Brush Shape icon to set the brush shape. You can toggle between a round and a square tip.
7. Use one of the following methods to increase or decrease the brush diameter:
   - Hold down the **Shift** key, then roll the middle wheel on your mouse;
   - Drag the slider on the **Tool Settings** panel; or
   - Drag the slider in the **Bitmap Tools** area of the **Assistant** panel.

8. Position the brush cursor over the area of the bitmap layer in which you want to paint, then click and drag.

   All areas of the bitmap layer currently in the secondary colour, and beneath the brush cursor, are replaced with the primary colour. All other areas of colour remain unchanged.

## Using the Draw tool

The **Draw** tool enables you to apply a line of one pixel wide in the primary or secondary colour to the currently active bitmap layer. You can use this tool in the **2D View** or **3D View** window.

*When using the **Draw** tool in the **3D View** window, the **Colour Shade** button in the **3D View** toolbar must be toggled on.*

To use the **Draw** tool:

1. Choose (see page 99) or create (see page 100) the bitmap layer on which you want to draw.
2. Make sure the bitmap layer is visible (see page 101).
3. Use one of the following methods to select the **Draw** tool:
   - In the **Design Tools** toolbar, click the **Draw** button; or
   - Click the **Draw** button in the Drawing toolset shown in the **Bitmap Tools** area of the **Assistant** panel.

   *For information on how to display any of the toolsets in the **Assistant** panel, see Understanding the Assistant panel (see page 36).*
4. Select the colour (see page 109) in which you want to draw as the primary or secondary colour.
5. Position the draw cursor over the area of the bitmap layer in which you want to draw:
   - To use the primary colour, click and drag.
   - To use the secondary colour, hold down the **Shift** key, then click and drag.

## Using the Flood Fill tools

The **Flood Fill** and **Flood Fill Selective** tools enable you to replace areas of colour on the currently active bitmap layer. You can use these tools in the **2D View** or **3D View** window.

*When using the **Flood Fill** and **Flood Fill Selective** tools in the **3D View** window, the **Colour Shade** button in the **3D View** toolbar must be toggled on.*

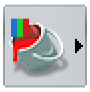
To replace a specific colour with the current primary colour:

1. Choose the bitmap layer (see page 99) containing the artwork that you want to edit.
2. Make sure the bitmap layer is visible (see page 101).
3. Select the colour (see page 109) you want to use for flood filling as the primary colour.
4. Use one of the following methods to select the **Flood Fill** tool:
   - In the **Design Tools** toolbar, click the **Flood Fill** button;
   - Right-click the **2D View** or **3D View** window, and click **Painting > Flood Fill** from the context menu;
   - Click the **Flood Fill** button in the Flood Fill toolset shown in the **Bitmap Tools** area of the **Assistant** panel.

   *For information on how to display any of the toolsets in the **Assistant** panel, see Understanding the Assistant panel (see page 36).*
5. Position the roller cursor over the colour in the bitmap layer you want to replace with the current primary colour, and then click.

To replace all colours, except those in or surrounded by the secondary colour:

1. Choose the bitmap layer (see page 99) containing the artwork you want to edit.
2. Make sure the bitmap layer is visible (see page 101).
3. Use one of the following methods to select the **Flood Fill Selective** tool:

- In the **Design Tools** toolbar, click and hold on the **Flood Fill**  button, move the mouse cursor over the **Flood Fill Selective**  button, then release the mouse button.
- Right-click the **2D View** or **3D View** window, and click **Painting > Flood Fill Selective** from the context menu; or
- Click the **Flood Fill Selective**  button in the Flood Fill toolset shown in the **Bitmap Tools** area of the **Assistant** panel.

*For information on how to display any of the toolsets in the* ***Assistant*** *panel, see Understanding the Assistant panel (see page 36).*

4. Select the colour (see page 109) you want to use for flood filling as the primary colour.
5. Select the colour (see page 109) you want leave untouched as the secondary colour.
6. Position the roller cursor  over a colour in the bitmap layer you want to replace with the primary colour, and then click.

   All colours, except those in or surrounded by the secondary colour, are replaced with the primary colour.

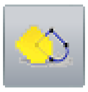
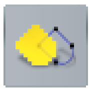
   *To replace any areas of colour surrounded by the secondary colour, you must click each of the colours inside the boundary shown in the current secondary colour.*

# Converting bitmap artwork into vector artwork

The **Bitmap To Vector** tool enables you to create a vector boundary around areas of the currently active bitmap layer shown in the current primary colour; as well as any other areas in colours currently linked to the primary colour. For further details about colour linking, see Colour linking (see page 106).

When converting bitmap artwork, the resulting vectors follow the pixelated outline of the colours. You can smooth these vectors by replacing their linear spans with bezier curves. For details, see Smoothing nodes (see page 154).

To create a vector boundary using bitmap artwork:

1. Choose the bitmap layer (see page 99) containing the artwork from which you want to create vectors.
2. Choose (see page 124) or create (see page 121) the vector layer on which you want to create the vectors.
3. Make sure that the active bitmap (see page 101) and vector (see page 130) layers are both visible.
4. Make sure that the colour around which you want to create vectors is selected as the primary colour (see page 109).
5. Use one of the following methods to display the **Vectors From Bitmap** panel:
   - In the **Vector Creation** toolbar, click the **Bitmap To Vector**  button;
   - From the **Menu Bar**, click the **Vectors > Fit Vectors To Colour Boundaries** option; or
   - Click the **Bitmap To Vector**  button in the **Bitmap Tools** area of the **Assistant** panel.
6. In the **Pixel Tolerance** box, specify the tolerance you want to use. This controls how closely the spans in the resulting vectors follow the pixelated outline of the bitmap artwork.
7. Select the conversion method you want to use when calculating the vector artwork. Click:
   - **Spline all nodes** to fit bezier curve spans between all nodes in the resulting vector artwork; or
   - **Keep lines** to fit bezier curve spans between all nodes, except where the number of consecutive pixels specified in the **Min Pixel Length** box forms a straight line.
8. Select the **Create Boundary** option.
9. Click the **Create Vectors** button to calculate the vector artwork. It is magenta and surrounded by a bounding box; indicating that it is selected.

*You can use the **Contrast** slider on the **2D View** toolbar to fade the artwork on the currently active bitmap layer and view clearly the resulting vector artwork. Alternatively, you can toggle the bitmap layer's visibility (see page 101) so that the image is hidden completely.*

# Using vector layers

Vector layers, like bitmap layers, are contained in a stack. This is displayed on the **Project** panel and the **Vector Layers** panel.

You can use one of the following methods to display the vector layer stack:

- From the **Project** panel, click ⊞ beside **Vectors** in the Project Tree; or

  *The **Project** panel is docked and pinned on the right of the interface.*

- Click the **Vector Layers** panel.

  *To display the **Vector Layers** panel, right-click a docking area, toolbar or the **Status Bar**, then select **Vector Layers** from the context menu.*

Every new ArtCAM model contains an empty default vector layer, named *Default Layer*. However, you can create as many vector layers as you want; either empty, or from imported artwork.

## Importing vector artwork

You can import vector artwork saved in any of the following formats onto a selected vector layer:

- Adobe Illustrator image (***.ai**);
- Encapsulated PostScript (***.eps**);
- Drawing Interchange Format, including PowerSHAPE and AutoCAD (***.dxf**);
- AutoCAD 2D Drawing (***.dwg**);
- Lotus, PC Paint or DUCT picture (***.pic**);
- Delcam DGK (***.dgk**); and
- Windows MetaFile (***.wmf**)

When choosing Drawing Interchange (***.dxf** ) or AutoCAD Drawing (***.dwg**) files, you can:

- position its vector artwork in the centre of the ArtCAM model;
- specify the units of measurement;
- instruct ArtCAM to search for loops within a specified tolerance; and
- rejoin all coincident nodes within a specified tolerance.

*ArtCAM also reads all layer information saved within these files.*

When choosing Encapsulated PostScript (***.eps**) or Adobe Illustrator image (***.ai**) files with an origin outside of the ArtCAM model, you can position their vector artwork:

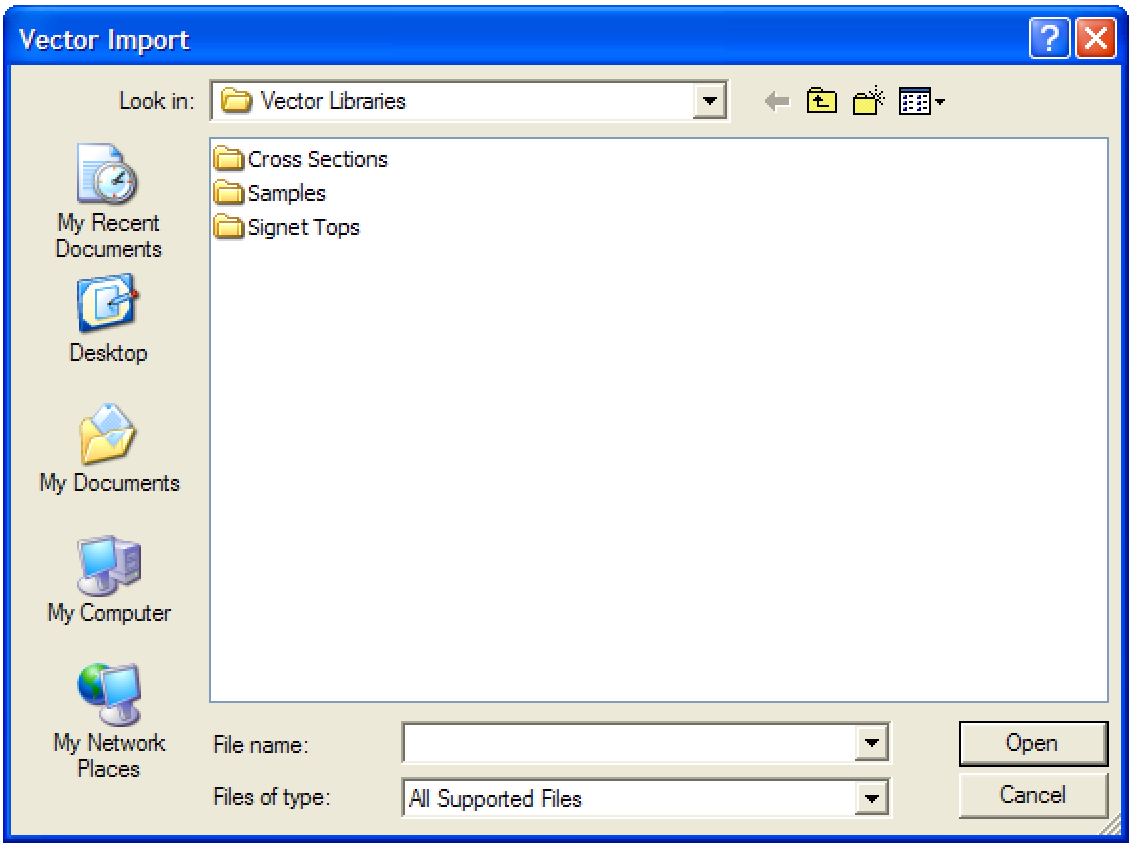
- in the centre of your model; or
- using the X and Y coordinates of the file's origin.

When choosing Windows Metafiles (***.wmf**), Enhanced Metafiles (***.emf**) and Lotus, PC Paint or DUCT picture files (***.pic**), their vector artwork is imported directly.

To import vector artwork:

1. Choose the vector layer (see page 124) on which you want to store the imported vector artwork.

2. From the **Menu Bar**, click the **Vectors > Import** option to display the **Vector Import** dialog box:



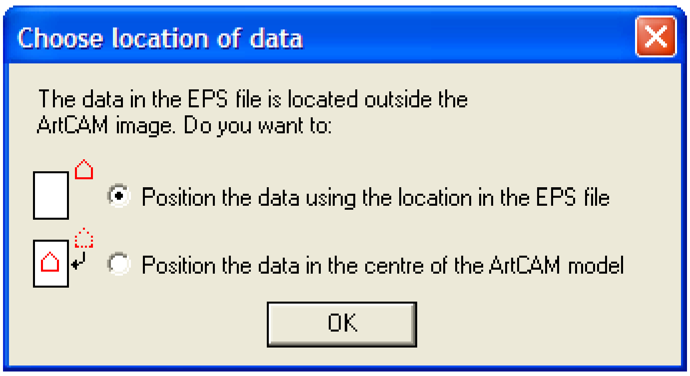

3. In the **Vector Import** dialog box, select the file you want to work with, and click **Open**.

   For ***.pic**, ***.dgk** and ***.wmf** files, or ***.eps** and ***.ai** files with an origin inside of the model area:

   the artwork is imported onto the currently active vector layer.

   For ***.eps** and ***.ai** files with an origin outside of the model area:

   the **Choose location of data** dialog box is displayed:

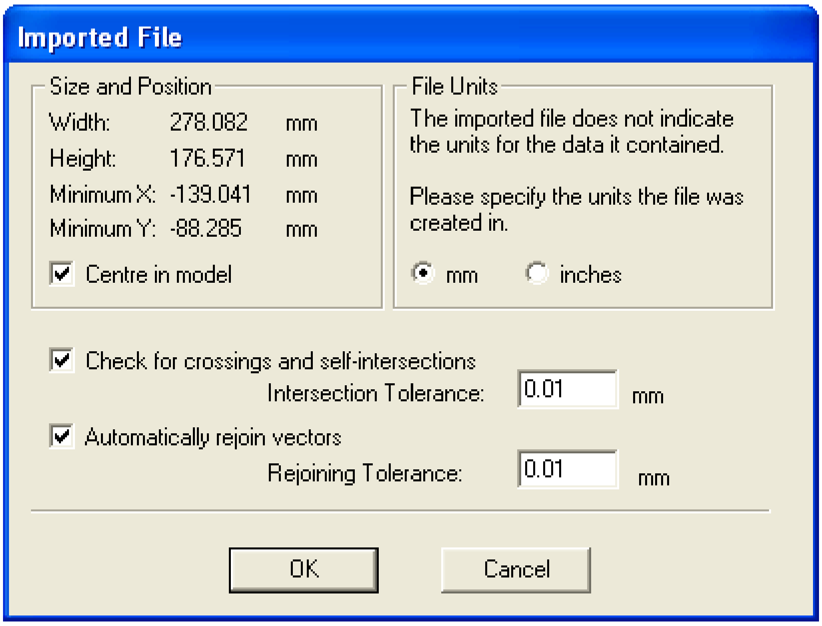



   Select the method you want to use to position the artwork, and click **OK**.

- To position the artwork using the X and Y coordinates stored in the file, select the **Position the data using the location in the EPS file** option.
- To position the artwork in the centre of your model, select the **Position the data in the centre of the ArtCAM model** option.

For ***.dxf** and ***.dwg** files:

The **Imported File** dialog box is displayed:




Choose the options that you want to apply to the artwork, and click **OK**.

- To position the artwork in the centre of your model, select the **Centre in model** check box.
- In the **File Units** area, make sure the units of measurement for the artwork is the same as those used in your ArtCAM model by selecting **mm** or **inches**.
- To identify all self-intersecting spans in the artwork within tolerance, make sure the **Check for crossings and self-intersections** check box is selected, and that the correct tolerance is specified in the **Intersection Tolerance** box.

*When selected, self-intersecting spans are shown in red, with white circular shapes marking the positions where spans overlap.*

- To rejoin any spans in the artwork that have been *exploded* within tolerance, make sure the **Automatically rejoin vectors** check box is selected, and that the correct tolerance is specified in the **Rejoining Tolerance** box.

The layers in the imported file are added to the vector layer stack above the previously active layer. Each new vector layer uses the name of the layer in the imported file, as well as any colour.

*If the name of a layer in the imported file is already used by a layer in the vector stack, the artwork on the file's layer is added to the vector layer with the same name.*

*If no colour has been applied to a layer in the imported file, the new vector layer is coloured red by default.*

All imported artwork is selected by default. For further information, see Selecting vector artwork (see page 122).

## Creating a new layer

In an ArtCAM model, you can create as many vector layers as you like. This enables you to separate your vector artwork and control what is displayed and when.

You can use one of the following methods to create a new vector layer:

- From the **Project** panel, right-click **Vectors** in the Project Tree, then click **New** in the context menu;
- From the **Vector Layers** panel, click the **New Vector Layer** button;
- From the **Menu Bar**, click **Vectors > Create New Layer**; or
- Right-click a selected vector, then click **Move Vectors To > New Layer** in the context menu.

Each new vector layer that you create is:

- named *Vector Layer*;
- numbered sequentially;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active;
- given a default colour of black;

- unlocked;
- using snapping; and
- visible in the **2D View** window.

*If you are working in the **3D View** window, click the **Toggle Vector Visibility** button in the **3D View** toolbar to display the artwork drawn across all visible vector layers.*

## Selecting vector artwork

You can select one or more vectors drawn across the vector layers in an ArtCAM model. Selected vector artwork can be used to create:

- bitmap artwork (see page 140);
- 3D shapes (see page 181); and
- toolpaths (see page 240).

To select one or more vectors:

1. Make sure that the vector layers from which you want to select vectors are visible (see page 130).
2. Use one of the following methods to enter Select mode:
   - In the **Design Tools** toolbar, click the **Select** button;

   *If the **Select** button is shown as in the **Design Tools** toolbar, then you are already in Select mode.*

   - Right-click the **2D View** or **3D View** window, then click **Select** in the context menu;
   - Press the **Esc** key; or
   - Click the **Select** button in the Vector Editing Mode toolset shown in the **Vector Tools** area of the **Assistant** panel.
3. Click the vector you want to select.

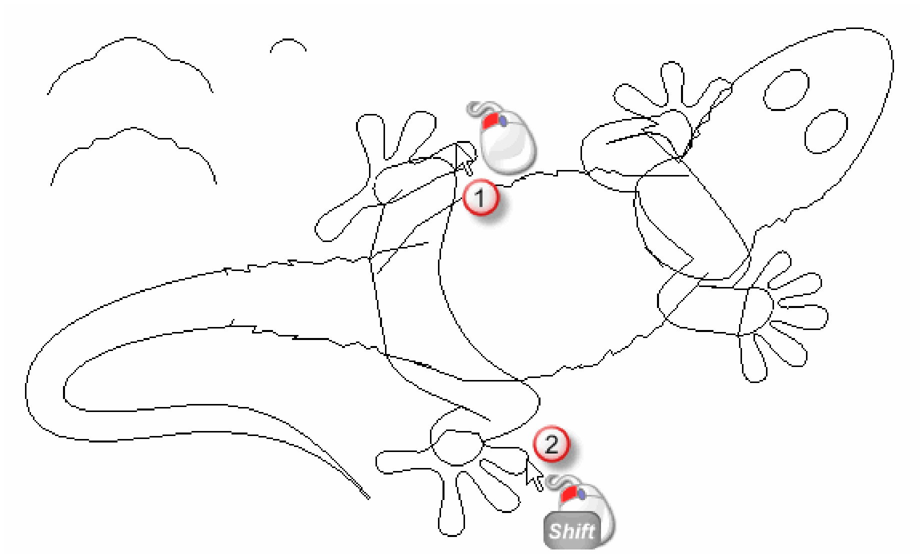
   The number of spans, nodes, bezier curve spans, linear spans and arc spans in the selected vector are displayed on the **Tool Settings** panel.

*Press the **F6** key to display the **Tool Settings** panel.*

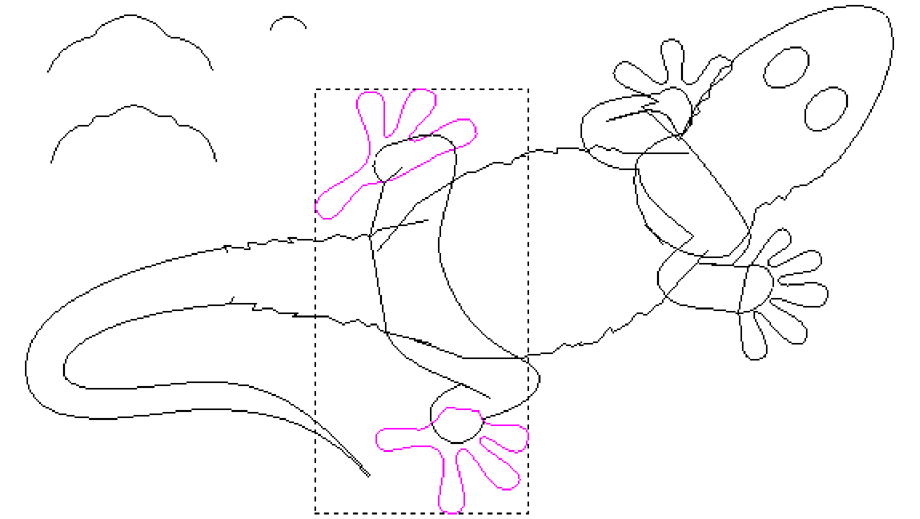
4. To add more vectors to your selection, use one of the following methods:

   ▪ Hold down the **Shift** key, then click each of the vectors; or

   *Before...*




   *After...*



   ▪ Click and drag to create a red selection rectangle around the vectors, and then release the mouse to select them.

*To exclude locked vectors from your selection, select the* **Filter Locked** *check box on the* **Tool Settings** *panel. To exclude the wireframe previews associated with calculated 2D toolpaths, select the* **Filter Previews** *check box. To use your mouse to twiddle the* **3D View** *instead of selecting vector artwork, select the* **Twiddle in 3d** *check box.*

A bounding box surrounds the selected vectors.

*To deselect a vector, hold down the* **Shift** *key, and click the vector.*

### Understanding the colour-coding of vectors

Vector artwork is not always shown in the colour assigned (see page 126) to the vector layer on which it is drawn or imported.

- One or more ungrouped vectors are magenta when selected. When deselected, they are displayed in the colour assigned (see page 126) to the vector layer on which they are hosted.
- Grouped closed vectors are purple when selected. When deselected, they are displayed in the colour assigned (see page 126) to the vector layer on which they are hosted.
- Grouped, closed, self-intersecting vectors are red when deselected. A ○ marks the position of each intersection, whether the vectors are selected or not.
- Locked vectors are teal when selected. When deselected, they are grey regardless of the colour assigned (see page 126) to the vector layer on which they are hosted.
- Grouped open vectors are blue when deselected, regardless (see page 126) of the colour assigned to the vector layer on which they are hosted.
- Vectors with one or more overlaying copies are red when selected (see page 122) using the click and drag method. When deselected, they are displayed in the colour assigned (see page 126) to the vector layer on which they are hosted.

## Choosing the active layer

Only one vector layer can be active at a time, although the artwork drawn across all vector layers which are currently visible is displayed.

Using the active vector layer you can:

- control where vector artwork is drawn in your model;
- export its vector artwork as a ***.eps**, ***.dxf**, ***.dgk** or ***.pic** file;
- create a duplicate copy of the layer and its vector artwork; or
- delete the layer and its vector artwork.

To choose the active vector layer:

1. Use one of the following methods to display the vector layer stack:
   - From the **Project** panel, click ⊞ beside **Vectors** in the Project Tree; or
   - Click the **Vector Layers** panel.

   *To display the* ***Vector Layers*** *panel, right-click a docking area, toolbar or the* ***Status Bar****, then select* ***Vector Layers*** *from the context menu.*
2. Click the layer you want to work with. Its name is bold and highlighted.

## Renaming a layer

In every new ArtCAM model, the default vector layer is named *Default Layer*. All vector layers you create are named *Vector Layer*, and given a unique number. When you import vector artwork, the new vector layer uses the name of the vector file from which it is created. You can rename any of the vector layers in a model.

To rename a vector layer:

1. Use one of the following methods to display the layer's name box:
   - From the **Project** panel, right-click the vector layer you want to rename in the Project Tree, then click **Rename** in the context menu.
   - From the **Vector Layers** panel, double-click the layer you want to rename in the stack.
2. In the name box, type the name you want to give to the layer.

   *To revert to the name previously given to the layer, press the* ***Esc*** *key on your keyboard.*
3. Press the **Enter** key, or click in the empty space below the layer stack to set the name of the layer.

## Assigning a colour to a layer

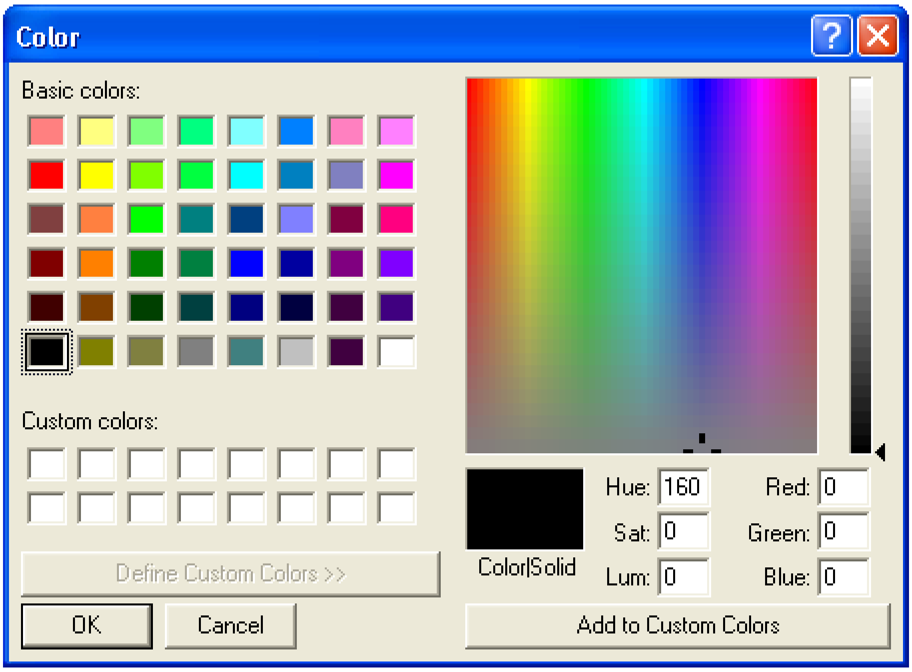
All new vector layers are shown in black; meaning that all artwork on a vector layer is displayed in black.

You can change the colour associated with a vector layer. Giving a unique colour to each layer makes it easier to identify where vectors are stored in your model, and what they are used for.

*Avoid using blue, magenta or red when choosing a vector layer's colour. These colours are used when vectors and toolpath previews are selected in the* ***2D View*** *window.*

To change the colour associated with a vector layer:

1. Use one of the following methods to display the **Color** dialog box:
   - From the **Project** panel, click the layer's swatch ● in the Project Tree;
   - From the **Project** panel, right-click the vector layer in the Project Tree, then select **Set Colour...** from the context menu; or
   - From the **Vector Layers** panel, click the layer's swatch ● in the stack.




2. In the **Color** dialog box, select the colour you want to use for the vector layer, and then click **OK**. The colour is applied to the vector layer's artwork.
3. To add a colour to the **Custom colors** palette:

a. If you know the specific HSL or RGB values of the colour you want to add, type them in the **Hue**, **Sat** (Saturation) and **Lum** (Luminance) boxes, or the **Red**, **Green** and **Blue** boxes.

b. If you do not know the HSL or RGB values, in the Colour Matrix displayed on the right, click the colour that closest resembles what you want to add. In the **Hue** and **Sat** (Saturation) boxes, the colour values are displayed.

   To change the overall colour balance, use the **Red**, **Green** and **Blue** boxes or the vertical slider to increase or decrease the red, green and blue colour channels by a percentage of themselves.

   To brighten the colour, use a higher value in the **Lum** box or click and drag the slider upwards. To darken the colour, use a lower value in the **Lum** box or click and drag the slider downwards. The default value is zero.

c. To add the colour, click the **Add to Custom Colors** button.

## Locking a layer

Locking enables you to prevent the artwork on a vector layer from being moved, deleted or edited. Locking does not prevent you from transferring artwork between vector layers.

Use one of the following methods to lock a layer:

- From the **Project** panel, click the icon beside the vector layer in the Project Tree;
- From the **Project** panel, right-click the vector layer in the Project Tree, then click **Lock Vectors** in the context menu; or
- From the **Vector Layers** panel, click the layer's icon displayed beside the vector layer in the stack.

Use one of the following methods to unlock a layer:

- From the **Project** panel, click the icon beside the vector layer in the Project Tree;
- From the **Project** panel, right-click the vector layer in the Project Tree, then click **Unlock** in the context menu; or
- From the **Vector Layers** panel, click the layer's icon displayed beside the vector layer in the stack.

## Snapping on a layer

Snapping enables you to align a selected vector with:

- vectors on another, visible layer; or
- a vertical or horizontal guideline.

By default, snapping is enabled on all vector layers in a model. You can restrict its use to specific vector layers, or choose whether it is used at all.

To toggle snapping on a vector layer:

1. From the **Menu Bar**, make sure that the **Bitmaps > Views > Snap To Objects** option is selected.

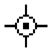
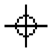
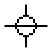
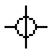
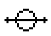
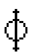
   *If the **Bitmaps > Views > Snap To Objects** option is deselected, snapping is not used even when snapping for a vector layer is toggled on.*

2. Use one of the following methods to toggle snapping off:
   - From the **Project** panel, click the icon displayed beside the vector layer in the Project Tree;
   - From the **Project** panel, right-click the vector layer in the Project Tree, then click **Snap Vectors** in the context menu; or
   - From the **Vector Layers** panel, click the icon displayed beside the vector layer in the stack.
3. Use one of the following methods to enable snapping:
   - From the **Project** panel, click the icon displayed beside the vector layer in the Project Tree;
   - From the **Project** panel, right-click the vector layer in the Project Tree, then click **Snap Vectors** in the context menu; or
   - From the **Vector Layers** panel, click the icon displayed beside the vector layer in the stack.

   *To temporarily disable snapping, hold down the **Shift** key.*

The mouse cursor changes in the following ways:

| Cursor | Displayed when |
| --- | --- |
| | over a node in a vector in the **2D View** or **3D View** window. |
| | over the mid of a linear or arc span within a vector in the **2D View** or **3D View** window. |

| Cursor | Displayed when |
|---|---|
|  | over the centre of a vector, as defined by its bounding box, in the **2D View** or **3D View** window; and<br><br>over the point at which two vectors intersect in the **2D View** or **3D View** window, provided the **X** key is held down on your keyboard. |
|  | over the point at which two guidelines intersect in the **2D View** window.<br><br>*This cursor is displayed only when guidelines are shown, and the* ***Bitmaps > Views > Snap To Guidelines*** *option is selected in the* ***Menu Bar****.* |
|  | over a horizontal guideline in the **2D View** window.<br><br>*This cursor is displayed only when guidelines are shown, and the* ***Bitmaps > Views > Snap To Guidelines*** *option is selected in the* ***Menu Bar****.* |
|  | over a vertical guideline in the **2D View** or **3D View** window.<br><br>*This cursor is displayed only when guidelines are shown, and the* ***Bitmaps > Views > Snap To Guidelines*** *option is selected in the* ***Menu Bar****.* |
|  | the cursor position has the same Y value as the polyline's start node in the **2D View** or **3D View** window.<br><br>*This cursor is displayed only when working in Polyline Creation (see page 137) mode.* |
|  | the cursor position has the same X value as the polyline's start node in the **2D View** or **3D View** window.<br><br>*This cursor is displayed only when working in Polyline Creation (see page 137) mode.* |

## Viewing a layer

You can control what vector artwork is shown in the **2D View** and **3D View** windows by specifying which vector layers are visible. When a vector layer is created, it is visible by default.

To control a vector layer's visibility:

1. If you are working in the **3D View** window, select the **Toggle Vector Visibility** button in the **3D View** toolbar.

   *In the **3D View** window, all vector artwork is hidden unless the **Toggle Vector Visibility** button is selected.*

2. Use one of the following methods to control what vector artwork can or cannot be seen:

   - To hide a layer's artwork, click its icon;
   - To hide the artwork on all but one layer, right-click the icon associated with the layer you want to keep visible;
   - To hide the artwork on all layers when working from the **Project** panel, click the icon beside **Vectors** in the Project Tree;
   - To hide the artwork on all layers when working from the **Vector Layers** panel, click the **Toggle All Visibility** button;
   - To display a layer's artwork, click its icon;
   - To display the artwork on all but one layer, right-click the icon associated with the layer you want to keep hidden;
   - To display the artwork on all layers when working from the **Project** panel, click the icon beside **Vectors** in the Project Tree;
   - To display the artwork on all layers when working from the **Vector Layers** panel, click the **Toggle All Visibility** button.

## Setting the stacking order

You can change the stacking order of vector layers by moving a layer. This enables you to display the vector layers in your preferred order. You can move any vector layer in the stack; even when it is locked.

Use one of the following methods to set a vector layer's position in the stack:

- From the **Project** panel, right-click the vector layer in the Project Tree, then select **Move Up** or **Move Down** in the context menu.

  *If the layer is at the bottom of the stack, the* ***Move Down*** *option is greyed-out. If the layer is at the top of the stack, the* ***Move Up*** *option is greyed-out.*

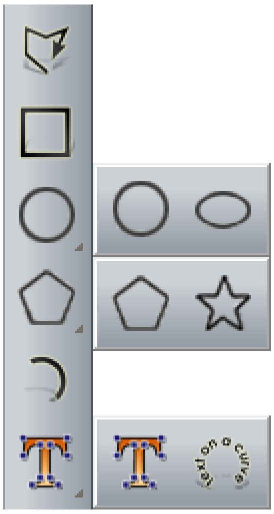
- From the **Vector Layers** panel, click and drag the layer up or down in the stack. When the layer is in your required position, release the mouse button.

# Creating artwork on a vector layer

The artwork drawn on vector layers can be used to create two-dimensional toolpaths or three-dimensional shapes on the currently active relief layer. For further information, see Creating simple shapes using closed vectors (see page 181) and Creating complex shapes using vectors (see page 194).

You can draw simple shapes or more complex free-form shapes on a vector layer using the Vector Creation tools located in:

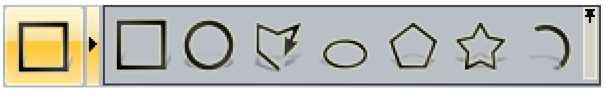
- the **Design Tools** toolbar.



The **Design Tools** toolbar is displayed on the left of the interface.

To display all of the buttons within a toolset on a toolbar, click and hold the button with ◢ displayed in the bottom-right. Release the mouse when its cursor is over the button that you want to select. You cannot pin any toolset on a toolbar.

- the **Vector Tools** area of the **Assistant** panel.



The **Assistant** panel is hidden by default. To display the panel, right-click a docking area, then select **Assistant** from the context menu.

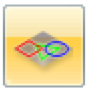
To display the buttons within a toolset on the **Assistant** panel, click ▸ on the right-edge of the button which is currently shown. To pin the toolset, click ⊤ displayed along the right-edge of the end button in the toolset. Repeat this step to collapse the toolset.

The available preset shapes are: rectangles, squares, circles, ellipses, polygons, stars and arcs. You can also create free-form shapes from polylines, which are one or more linear or bezier curve spans joined together by nodes.

In addition to drawing, you can import artwork into a vector layer, and convert bitmap artwork into vector artwork. For details, see Converting bitmap artwork into vector artwork (see page 115).

## Creating preset vector shapes

To create a preset shape:

1. Choose the vector layer (see page 124) on which you want to create a vector.
2. Make sure the vector layer is visible (see page 130).
3. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on. Otherwise, although you can still draw vector artwork, it cannot be seen.
4. To create an exact shape:
   a. Click the button for the shape you want to create.
   b. On the **Tool Settings** panel, specify its properties.
   c. On the **Tool Settings** panel, click the **Preview** button to update the shape's preview.

   Each of the shapes and its associated properties are as follows:

| Shape Button | Properties |
|---|---|
|  Rectangle | ▪ Height;<br>▪ Width;<br>▪ Corner radii;<br>▪ Angle; and<br>▪ Centre point's X and Y coordinates. |
|  Square | ▪ Height;<br>▪ Corner radii;<br>▪ Angle; and<br>▪ Centre point's X and Y coordinates. |

| Shape Button | Properties |
|---|---|
| Circle | ▪ Radius or Diameter; and<br>▪ Centre point's X and Y coordinates. |
| Ellipse | ▪ Height;<br>▪ Width;<br>▪ Angle; and<br>▪ Centre point's X and Y coordinates. |
| Polygon | ▪ Number of sides;<br>*A polygon must have a minimum of three sides.*<br>▪ Angle;<br>▪ Radius; and<br>▪ Centre point. |
| Star | ▪ No. of points;<br>▪ Angle;<br>▪ Radius of first points;<br>▪ Radius of second points; and<br>▪ Centre point. |
| Arc | ▪ Centre point, start point and end point;<br>▪ Start point, end point and mid point; or<br>▪ Start point, end point and radius.<br>*When creating an arc, you must specify the coordinates of 3 separate points. What these points are depends on your chosen method for creating the arc.* |

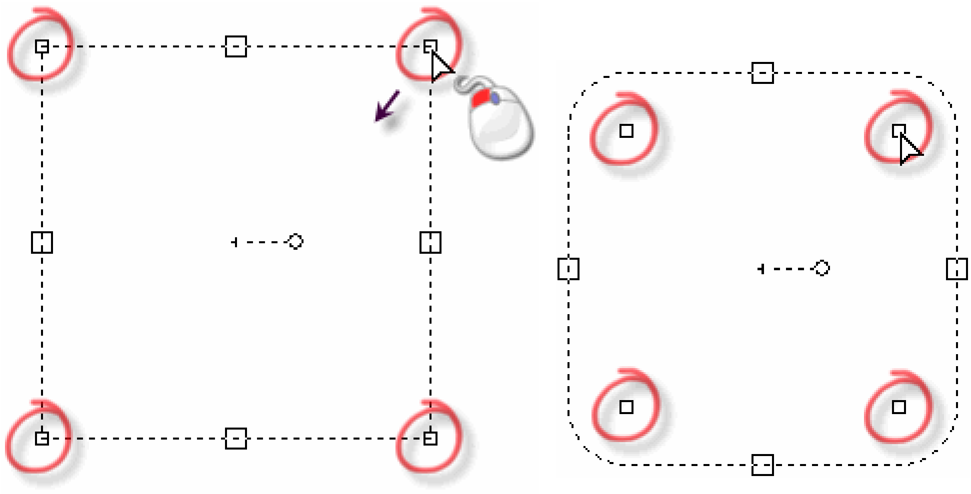
5. To create an approximate shape:
   a. Click the button for the shape you want to create.
   b. In the **2D View** or **3D View** window, move the mouse cursor + to the position in which you want to create your shape.

c. Left-click and drag the mouse. When the shape reaches the required size, release the mouse button. The shape's properties are displayed in the **Tool Settings** panel.

6. If you are creating a **Rectangle** or **Square**, drag the corner handles to shape the corners. On the **Tool Settings** panel, the radius of each corner is shown in the **Corner Radii** box.

   For example, in the **2D View** window, a square looks as follows before and after its top right corner is dragged inwards:

   *Before...* *After...*

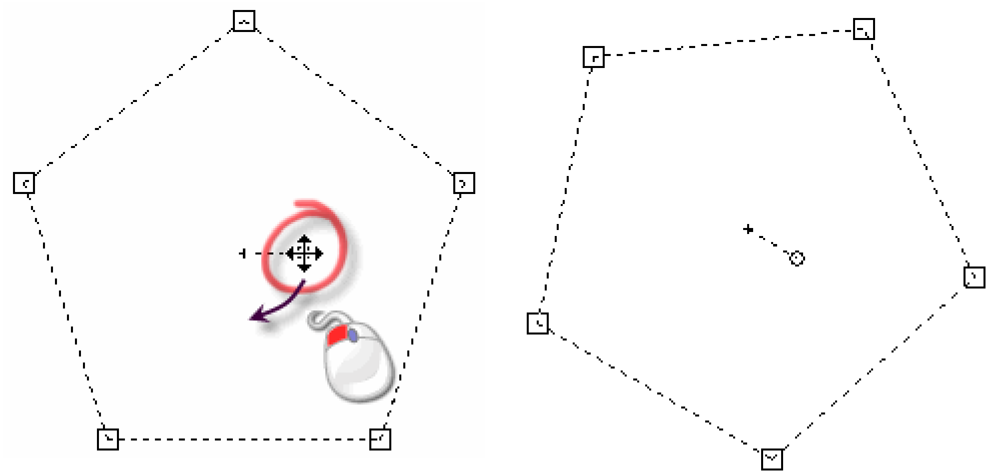


*In the **3D View** window, corner handles are displayed as a red dot .*

7. If you are creating a **Rectangle**, **Square**, **Polygon**, **Ellipse** or **Star**, click and drag the rotation handle to set its angle. Drag clockwise to apply a positive angle, or drag anti-clockwise to apply a negative angle. On the **Tool Settings** panel, the shape's angle is displayed in the **Angle** box.

   For example, in the **2D View** window, a polygon looks as follows before and after its rotation handle is dragged clockwise:

*Before...* *After...*

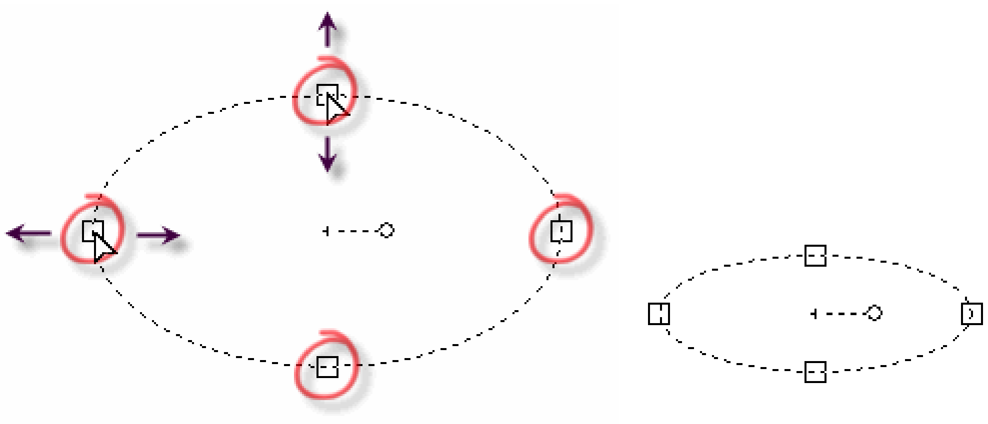


*In the **3D View** window, a rotation handle is displayed as a green dot .*

8. If you are creating a preset shape other than an **Arc**, you can use the resizing handles to adjust its size. Click and drag either the top or bottom resizing handle to adjust its height. Click and drag either the left or right resizing handle to adjust its width.

   For example, in the **2D View** window, an ellipse looks as follows before and after its top and left resizing handles have been adjusted:

   *Before...* *After...*



*In the **3D View** window, resizing handles are displayed as an orange dot .*

9. To create your shape and keep the current tool selected, left-click. This enables you to create another instance of the same shape.

To create your shape, leave the current preview displayed and keep the current tool selected, press the **Enter** key. This enables you to create multiple copies or variations of the same shape using the preview already shown.

To create your shape and deselect the current tool, right-click. The **Select** tool is reselected, and the properties of the selected vector artwork are displayed on the **Tool Settings** panel.

*To edit a preset shape, select the vector, then press the **E** key. The Vector Creation tool and your specific settings used to create the shape are displayed on the **Tool Settings** panel, so that you can edit them. Other than polylines and arcs, any shape created by a Vector Creation tool can be edited.*

## Creating free-form vector shapes

You can create free-form vector artwork on any vector layer using polylines, which are one or more linear or bezier curve spans joined together by nodes.

When creating a polyline, the following information is displayed on the **Tool Settings** panel:

- the coordinates of the current mouse cursor position;
- the angle and length of the span;
- the coordinates of the last point; and
- the change in position from the last point in both the X and Y direction.

To create a polyline:

1. Choose the vector layer (see page 124) on which you want to create your polyline.
2. Make sure the vector layer on which you want to create your polyline (see page 130) is visible.
3. Use one of the following methods to display the **Poly Line Creation** settings on the **Tool Settings** panel:
   - In the **Design Tools** toolbar, click the **Create Polyline**  button;
   - Right-click the **2D View** or **3D View** window, then select **Drawing > Polyline** in the context menu; or

- In the **Vector Tools** area of the **Assistant** panel, click the **Create Polyline** button from the Vector Creation toolset.

*The **Assistant** panel is hidden by default. To display the panel, right-click a docking area and select **Assistant** from the context menu.*

*To display all of the buttons within a toolset on the **Assistant** panel, click on the right-edge of the button which is currently shown. To pin the toolset, click displayed along the right-edge of the end button in the toolset. Repeat this step to collapse the toolset.*

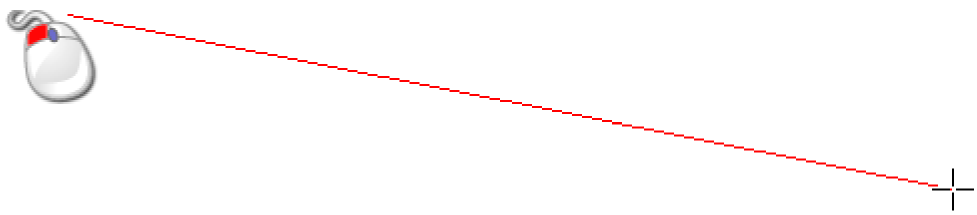
4. In the **2D View** or **3D View** window, move the mouse cursor to the position in which you want to create the polyline's start node, then left-click.

*If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility** button in the **3D View** toolbar is toggled on. Otherwise, although you can draw the polyline, it cannot be seen.*

*If you want to display the coordinates on the mouse cursor, click to select the **Window > Show Cursor Information** option from the **Menu Bar**.*

5. Move the mouse cursor to the position at which you want to create the next node, then left-click.

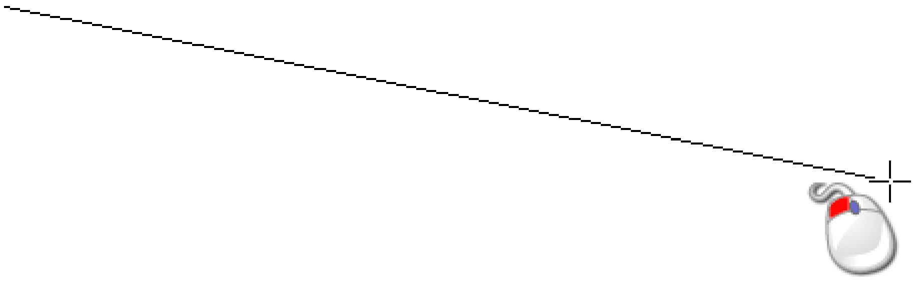
   When moving, a preview of a linear span is displayed as a red line:



*To join nodes with a bezier curve span instead, click and hold down the left mouse button before dragging. Release the mouse button to create the node.*

*To constrain the angle of the linear span drawn between points (nodes) to 15 degree increments, hold down the **Ctrl** key when moving the mouse.*

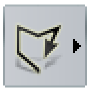
After clicking the mouse, the span connecting the nodes is drawn as a black line:



6. Repeat the previous two steps to create more nodes joined by spans.
7. Use one of the following methods to end the polyline:
   - To end the polyline and enter Select mode, right-click;
   - To end the polyline and stay in Polyline Creation mode, press the **Space Bar**;
   - To join the polyline's start and end nodes and stay in Polyline Creation mode, press the **Tab** key; or
   - To end the polyline, create a closed polygon and stay in Polyline Creation mode, click the polyline's start node.

You can also create a polyline using specific values:

1. Choose the vector layer (see page 124) on which you want to create the polyline.
2. Make sure the vector layer on which you want to create your polyline (see page 130) is visible.
3. Use one of the following methods to display the **Poly Line Creation** settings on the **Tool Settings** panel:
   - From the **Design Tools** toolbar, click the **Create Polyline**  button;
   - Right-click the **2D View** or **3D View** window, then select **Drawing > Polyline** in the context menu; or

- In the **Vector Tools** area of the **Assistant** panel, click the **Create Polyline**  button from the Vector Creation toolset.

4. To create a polyline from bezier curve spans linked by smoothed nodes, select **Draw Smooth Polylines**.
5. Use one of the following methods to specify the position of the polyline's start node:
    - In the **X** and **Y** boxes, type the node's X and Y coordinates;

    *Move the mouse cursor to the position where you want to create the node in the polyline. The X and Y coordinates of its position are shown in the **Cursor Position** area and on the **Status Bar**.*

    - In the **degs** box, type the angle of the polyline's span, and then, in the **L** box, type the polyline's length; or
    - In the **dx** and **dy** boxes, type the distance of the node along the X and Y axes.
6. Click **Add** to create the node and its associated span.
7. Repeat steps 5 to 6 to create more nodes joined by spans.
8. Position the mouse cursor over the **2D View** or **3D View** window, then use one of the following methods to end the polyline:
    - To end the polyline and enter Select mode, right-click;
    - To end the polyline and stay in Polyline Creation mode, press the **Space Bar**;
    - To join the polyline's start and end nodes and stay in Polyline Creation mode, press the **Tab** key; or
    - To end the polyline, create a closed polygon and stay in Polyline Creation mode, click the polyline's start node.

## Converting vector artwork into bitmap artwork

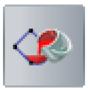
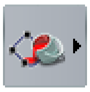
You can create bitmap artwork on the currently active bitmap layer using selected vector artwork. The bitmap artwork is created in the current primary colour:

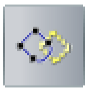
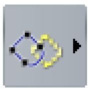
- around the boundary of the selected vector artwork; or
- within the boundary of the selected vector artwork.

You can use the bitmap artwork to:

- create 3D shapes (see page 177); or
- restrict the reach of Sculpting tools (see page 232).

To convert vector artwork into bitmap artwork:

1. Select the vectors (see page 122) you want to convert into bitmap artwork.
2. Choose the bitmap layer (see page 99) on which you want to create the bitmap artwork.
3. Make sure the bitmap layer is visible (see page 101).
4. Change the primary colour (see page 109) to the colour in which you want to create the bitmap artwork.
5. To flood fill the selected vectors, use one of the following methods:
   - From the **Bitmap Tools** toolbar, click the **Flood Fill Vectors**  button.
   - From the **Menu Bar,** click **Vectors > Flood Fill Vectors**; or
   - In the **Vector Tools** area of the **Assistant** panel, click the **Flood Fill Vectors**  button in the Vector Bitmap toolset.

   The colour filled area is created on the currently active bitmap layer.
6. To convert the selected vectors into bitmap artwork, use one of the following methods:
   - From the **Bitmap Tools** toolbar, click the **Vector To Bitmap**  button;
   - From the **Menu Bar**, click the **Vectors > Copy Vectors Into Bitmap** option; or
   - In the **Vector Tools** area of the **Assistant** panel, click the **Vector To Bitmap**  button in the Vector Bitmap toolset.

   A one pixel wide outline is created on the currently active bitmap layer.
7. Use one of the following methods to toggle the visibility of all vector layers off:

- From the **Project** panel, click the icon beside **Vectors** in the Project Tree;
- From the **Project** panel, right-click **Vectors** in the Project Tree, then click **Toggle All Visibility** in the context menu; or
- From the **Vector Layers** panel, click the **Toggle All Visibility** button.

You can now see clearly the bitmap artwork created from the vector artwork. For details, see Viewing a layer (see page 130).

# Editing vector artwork

Almost all of the preset vector shapes created using the Vector Creation tools can be edited in the same way: square, rectangle, circle, ellipse, polygon and star. For further information on these shapes, see Creating preset vector shapes (see page 133).

These vectors can be edited:

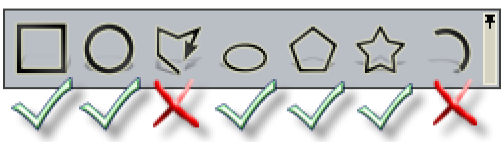
- using specific properties on the **Tool Settings** panel; or
- working in Node Editing mode, and using the vector's spans, nodes and control points.

All other vectors can be edited only when working in Node Editing mode, and using their spans, nodes and control points.

To edit a preset vector shape:

1. Select the vector (see page 122) you want to edit.
2. Use one of the following methods to display the selected vector's properties on the **Tool Settings** panel, and its manipulation handles in the **2D View** and **3D View** windows:
   - Press the **E** key; or
   - Right-click the selected vector, then click **Edit** in the context menu. For example, **Edit Ellipse**.

*You can only edit vectors drawn using the following tools available in the Vector Creation toolset:*



3. On the **Tool Settings** panel, change the selected vector's properties as required. For example, specify its new height in the **Height** box. For details, see Creating preset vector shapes (see page 133).

Vectors created using any of the methods listed below cannot be edited using specific properties:

- The **Create Polyline** tool. For details, see Creating preset vector shapes (see page 133);
- The **Create Arcs** tool. For details, see Creating free-form vector shapes (see page 137);
- The **Create Rectangle**, **Create Circles**, **Create Ellipse**, **Create Polygons** or **Create Stars** tool, and where a node or span has been moved or edited;
- the **Vector Library** or **Vector Import** tool; or
- the **Bitmap To Vector** tool. For details, see Converting vector artwork into bitmap artwork (see page 140).

To edit vector artwork created using any of these methods, you must use its spans and nodes. When working in Node Editing mode, you can use the options on the selected vector's context menu to change its spans and nodes.

When a vector is selected, use one of the following methods to enter Node Editing mode:

*You cannot work in Node Editing mode in the **3D View** window.*

- In the **Design Tools** toolbar, click the **Node Editing** button;

  *If the **Node Editing** button is shown as in the **Design Tools** toolbar, then you are already in Node Editing mode.*

  *If the **Node Editing** button is shown as in the **Design Tools** toolbar, then you are working in the **3D View** window. Press the **F2** key to work in the **2D View** window.*

- Right-click the **2D View** window, then click **Node Editing** from its context menu;
- Press the **N** key; or
- In the **Vector Tools** area of the **Assistant** panel, click the **Node Editing** button from the Vector Editing Mode toolset.

## Selecting nodes and control points

You can select single or multiple nodes and control points in the vector artwork drawn on a vector layer.

You can select nodes and control points only from the **2D View** window.

To select nodes and control points:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

   A bounding box surrounds the selected vector, within which you can see its spans, nodes and control points.
3. To select a single node or control point:

   a. Move the mouse cursor over the node or control point you want to select.

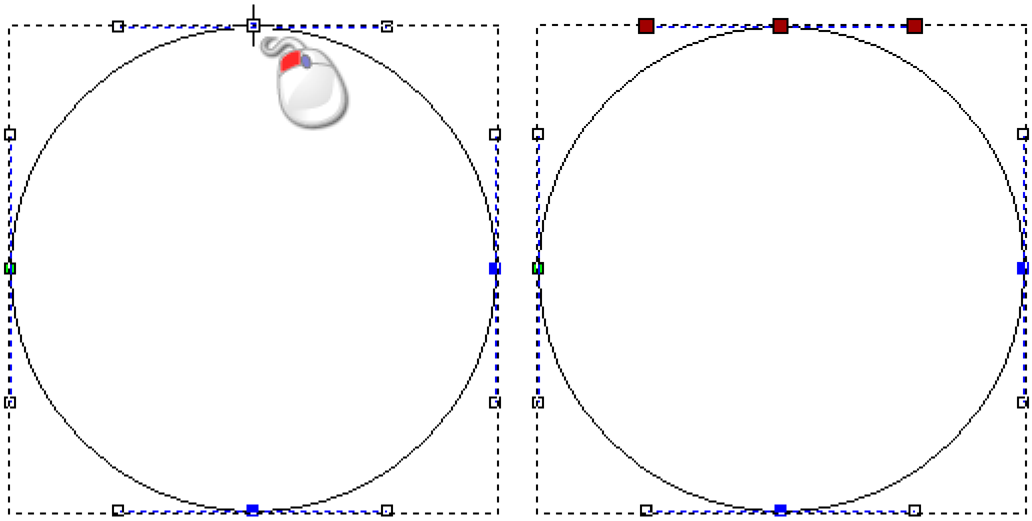
   b. When the cursor changes to , click the node or control point beneath the cursor.

   *To deselect a node or control point, position the mouse cursor over the node or control point, hold down the* ***Shift*** *key, and then click.*

   If you select a node that has adjoining control points, these are also selected; all of which are red.

   For example, when the node shown below is selected, we can see that its adjoining control points are also selected:

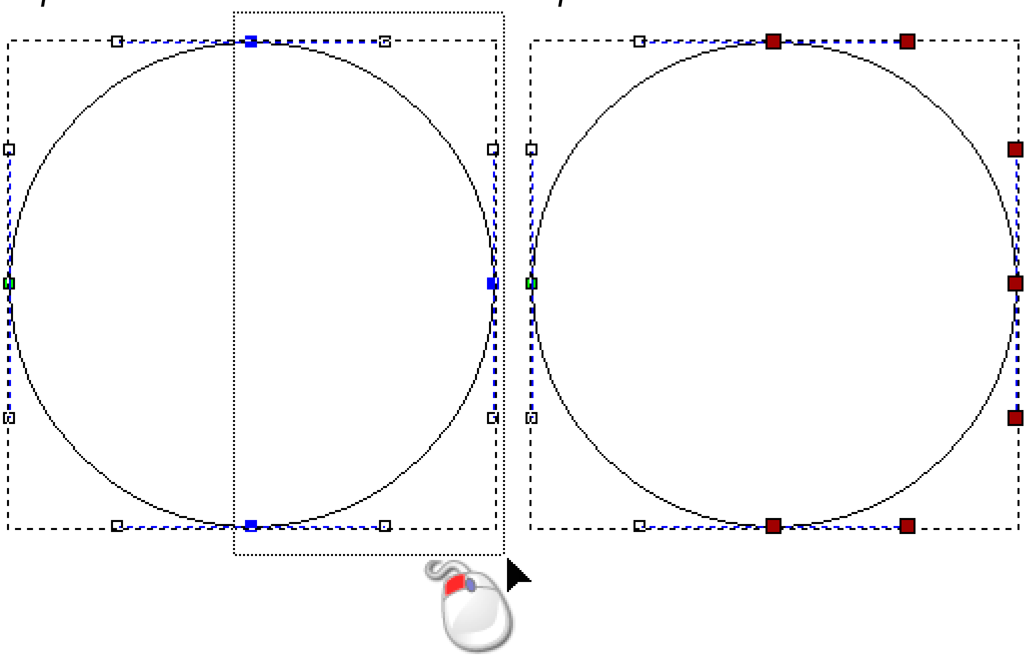
*Before...* *After...*



4. To select more than one node or control point, use one of the following methods:

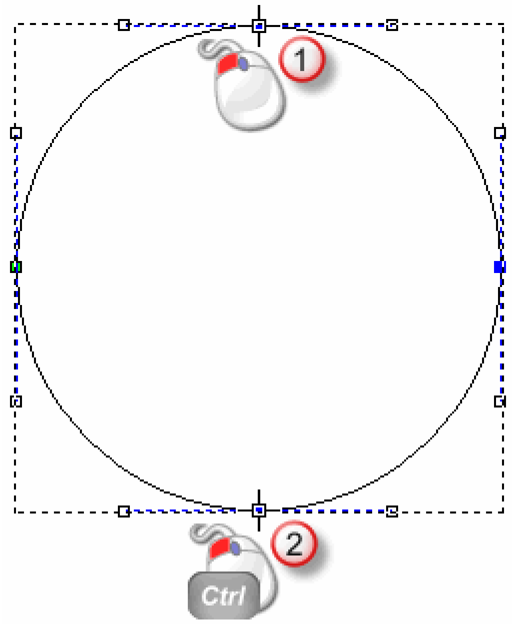
   - Click and drag to create a bounding box around the nodes and control points.

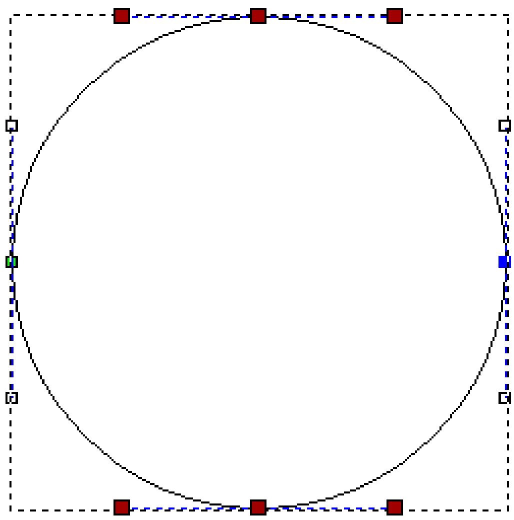
     *Before...* *After...*

   - Click to select a node or control point. Hold down the **Ctrl** key, then click the additional nodes and control points. The selected nodes and control points are red.

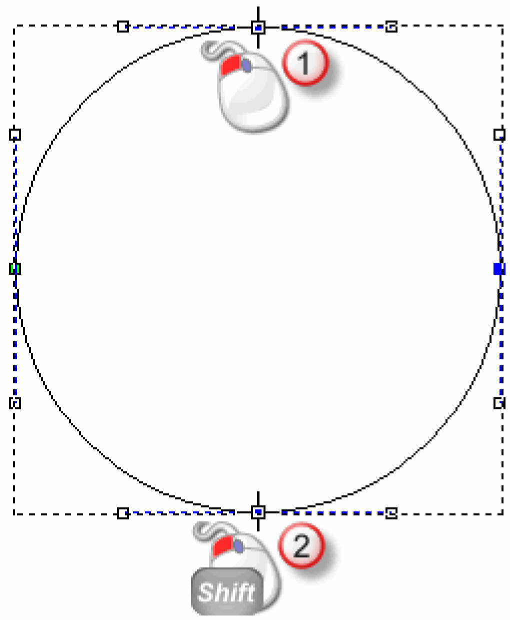
*Before...* *After...*

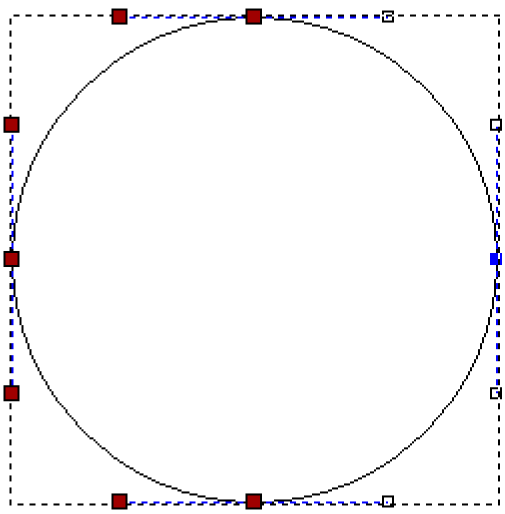


- Click to select a node or control point. Hold down the **Shift** key, and then click to select a second node or control point. ArtCAM calculates the shortest distance between the selected nodes, and then selects the other nodes and control points between them. The selected nodes and control points are red.

*Before...* *After...*



## Converting spans

Spans within vector artwork can be linear, arcs or bezier curves. A span is what joins two nodes in a vector. When working in Node Editing mode, you can use the options on the selected vector's context menu to change the type of span.

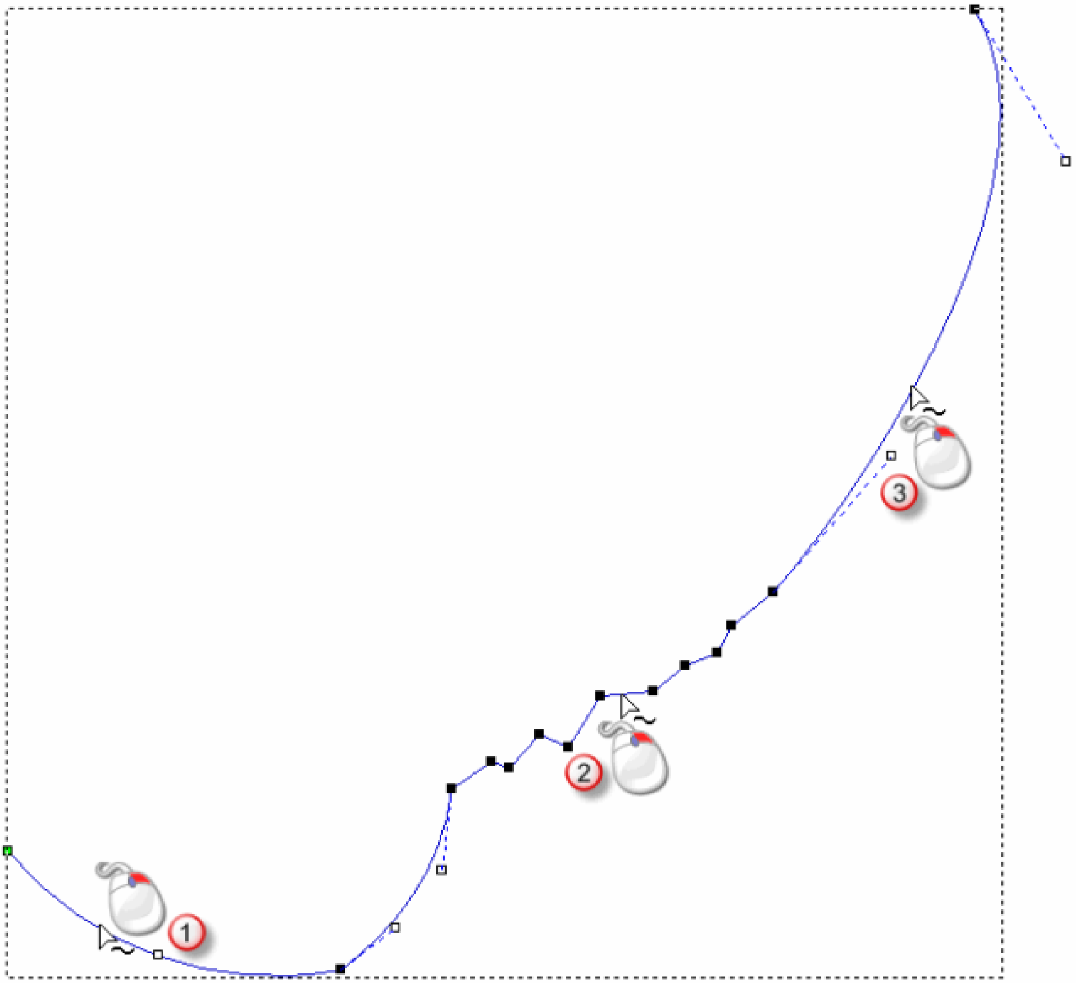
To convert a span:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

   *There are no control points associated with a linear span. There are two control points associated with a bezier curve span, and each of them is attached to the node located at either end of the span. There is one control point associated with an arc span, and this is located midway between the nodes located at either end of the span.*

For example, the first numbered span shown below is an arc, the second a line and the third a bezier curve:



3. Right-click the span you want to change, then click either of the **Convert span** options in the context menu:
   - If you are converting a linear span, click **Convert span to bezier** or **Convert span to arc**.

     *You can also press the* ***A*** *key to convert a linear or bezier span to an arc span.*

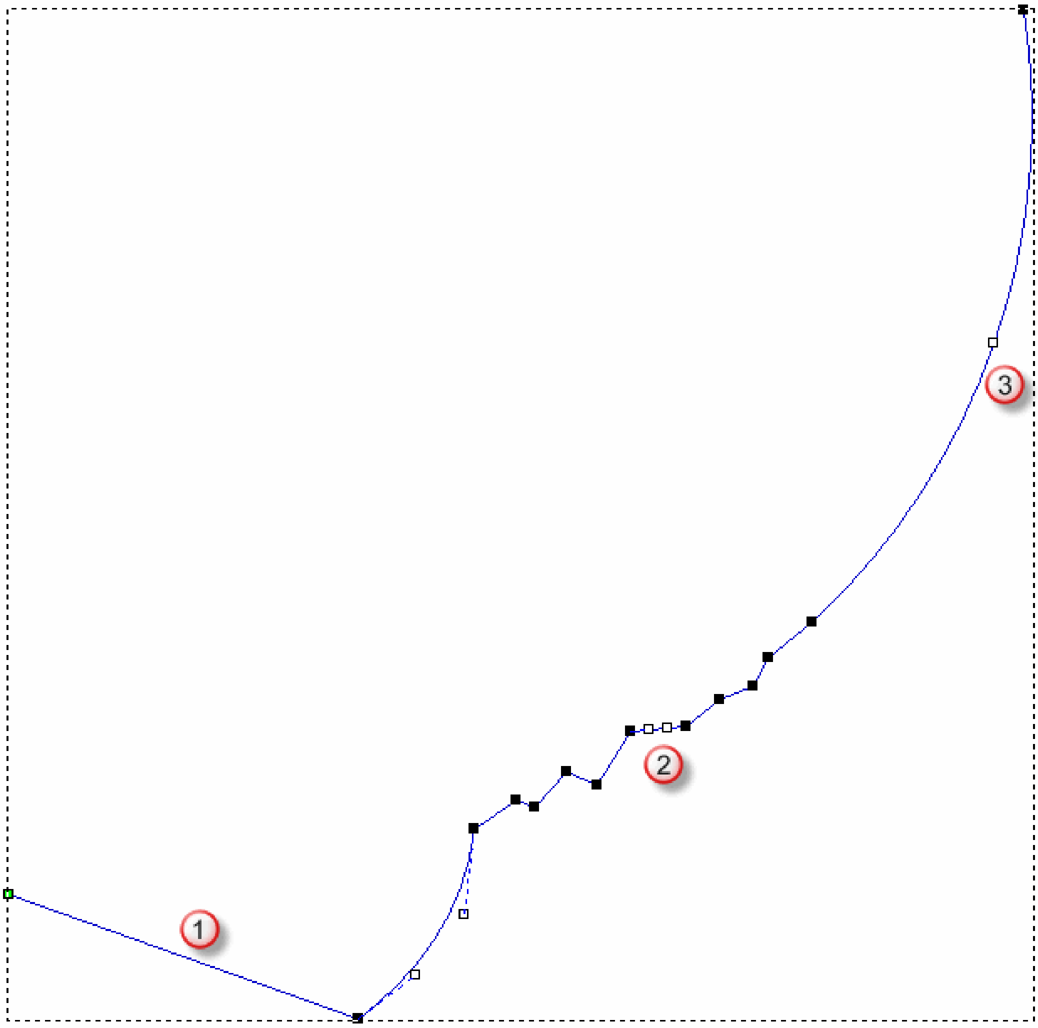
   - If you are converting a bezier curve span, click **Convert span to line** or **Convert span to arc**.

     *You can also press the* ***L*** *key to convert a bezier or arc span to a linear span.*

   - If you are converting an arc span, click **Convert span to line** or **Convert span to bezier**.

     *You can also press the* ***B*** *key to convert an arc or linear span to a bezier span.*

In our example, if we convert the first numbered span from an arc to a line, the second from a line to a bezier and the third from a bezier to an arc, we can see how the overall shape of the selected vector changes quite considerably:

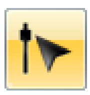



## Moving nodes

You can move one or more nodes in a selected vector to a new position. This changes the vector's shape and structure.

To move a single node or control point:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

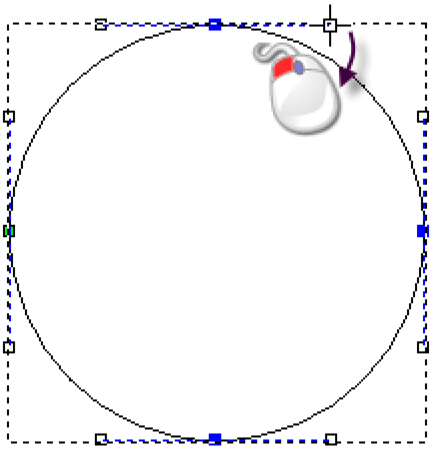
*If the **Node Editing** button is shown as*  *in the **Design Tools** toolbar, then you are already in Node Editing mode.*

A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

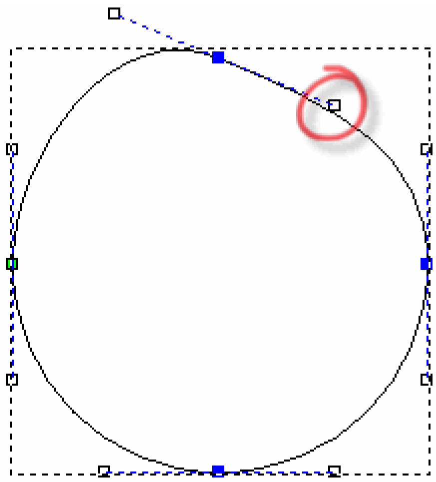
3. Move the mouse cursor over the node or control point you want to move.
4. When the cursor changes to , click and drag the node or control point into its new position. Release the mouse button to set its position.

   For example, when the control point in the circle shown below is dragged diagonally to the left, we can see that the overall shape of the vector changes:

   *Before...*

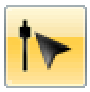
   

   *After...*

To move a selection of nodes simultaneously:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

   *If the **Node Editing** button is shown as*  *in the **Design Tools** toolbar, then you are already in Node Editing mode.*

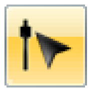
   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

3. Hold down the **Ctrl** key, and then select the nodes (see page 144) you want to move. The selected nodes and any adjoining control points are red.
4. Use one of the following methods to move the selected nodes:

- Move the mouse cursor over any of the selected nodes, click and drag the node to move all of the selected nodes, then release the mouse button to set their new position; or
- Use the four arrow keys on your keyboard to nudge the selected nodes into their new position.

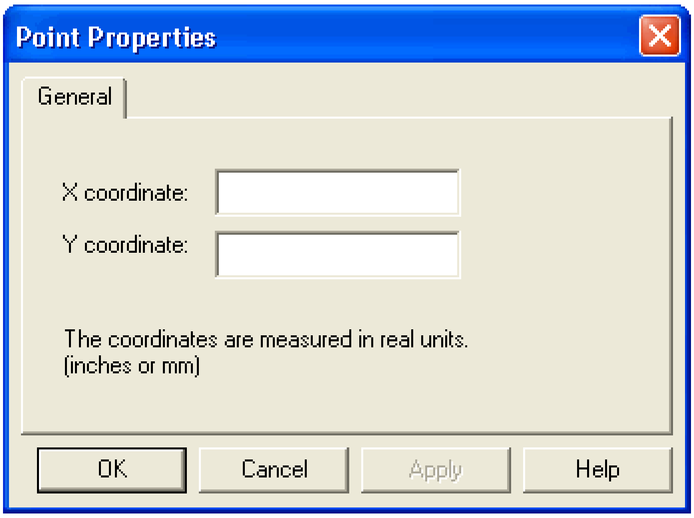
Alternatively, you can move a node to an exact position:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

   *If the **Node Editing** button is shown as*  *in the **Design Tools** toolbar, then you are already in Node Editing mode.*

   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

3. Right-click any node, then select **Properties** from the context menu. The **Node Properties** dialog box is displayed:

   The node's current X and Y co-ordinates are shown in the **X coordinate** and **Y coordinate** boxes.

4. In the **X coordinate** and **Y coordinate** boxes, specify the X and Y coordinates of the node's new position.
5. Click **OK** to move the node and close the dialog box.

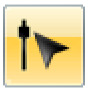
## Inserting nodes

You can insert a node in any span within open or closed ungrouped vectors drawn on a vector layer. Inserting a node divides a span into two new spans of the same type as the original. Inserting nodes gives you greater freedom to change a vector's overall shape.

You can also insert a start node in any span within closed vectors drawn on a vector layer. This divides a span into two new spans of the same type as the original. The start node is shown in green.

To insert a node in a span:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

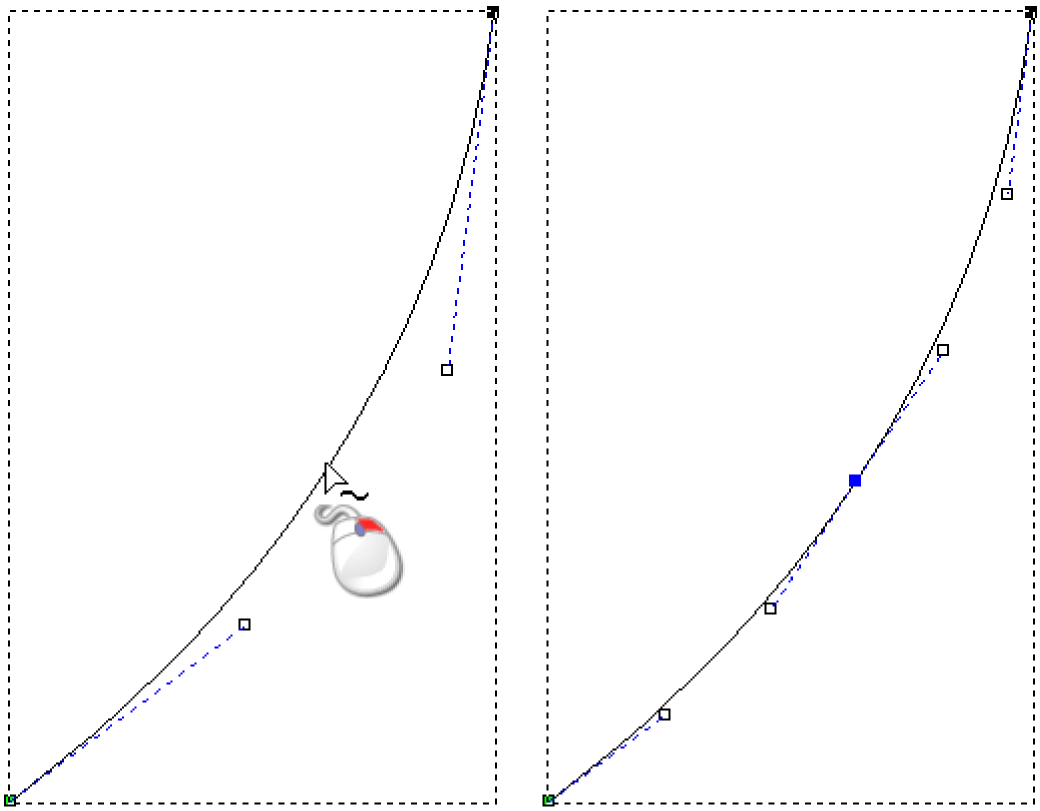
   *If the **Node Editing** button is shown as*  *in the **Design Tools** toolbar, then you are already in Node Editing mode.*

   A bounding box surrounds the selected vector, within which you can see its spans, nodes and control points.
3. Move the mouse cursor over the position in the span in which you want to insert a node.
4. When the mouse cursor changes to :
   - right-click, then select **Insert a node** from the context menu; or
   - press the **I** key.

   A new node is created in the span beneath the mouse cursor, dividing it into two separate spans.

   For example, when a node is inserted into the bezier curve span shown below, you can see that a second bezier curve span with two adjoining control points is created:

*Before...* *After...*



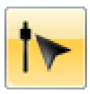
*Adding nodes to vector artwork can be a time consuming process. Instead, you can use the* ***Create Polyline*** *tool to create new, more complicated shapes. With the* ***Create Polyline*** *tool settings displayed in the* ***Tool Settings*** *panel, each corresponding movement and mouse click creates a new node and connects it to the last with a linear or bezier curve span. For details, see Creating free-form vector shapes (see page 137).*

To insert a start node in a span:

1. Select the vector (see page 122) you want to edit.

   *You cannot insert a start node into an open vector.*

2. Enter Node Editing mode (see page 142).

   *If the* ***Node Editing*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Node Editing mode.*

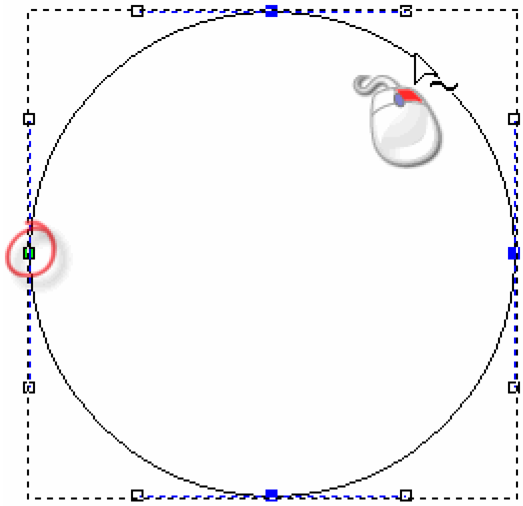
   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

3. Position the mouse cursor over the span in which you want to insert a start node.
4. When the mouse cursor changes to :
   - right-click, then select **Insert Start Node** from the context menu; or
   - press the **P** key.

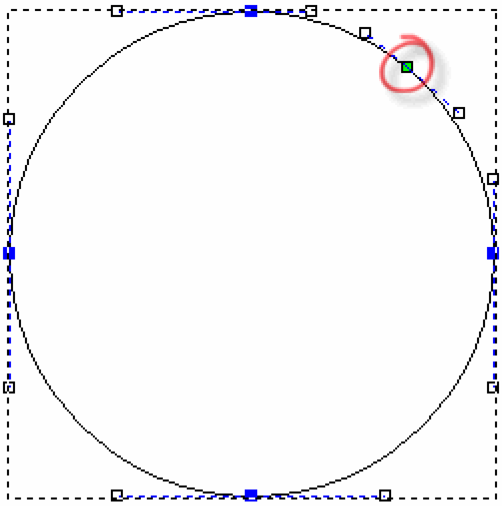
   A new node is created in the span beneath the mouse cursor, dividing it into two separate spans. The previous start node in the selected closed vector changes to a node.

   For example, when a start node is inserted into the bezier curve span shown below, you can see that a second bezier curve span with two adjoining control points is created, and the previous start node changes to a node:

*Before...*



*After...*



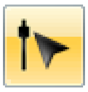
## Smoothing nodes

You can smooth any node in a vector, other than the start or end node in an ungrouped open vector. For details, see Selecting nodes and control points (see page 144) and Selecting vector artwork (see page 122).

When smoothing, the span on either side of a node is converted to a bezier curve span. ArtCAM appends control points to the node which enable you to control the curvature of the selected vector.

To smooth a single node:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

*If the* ***Node Editing*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Node Editing mode.*

A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

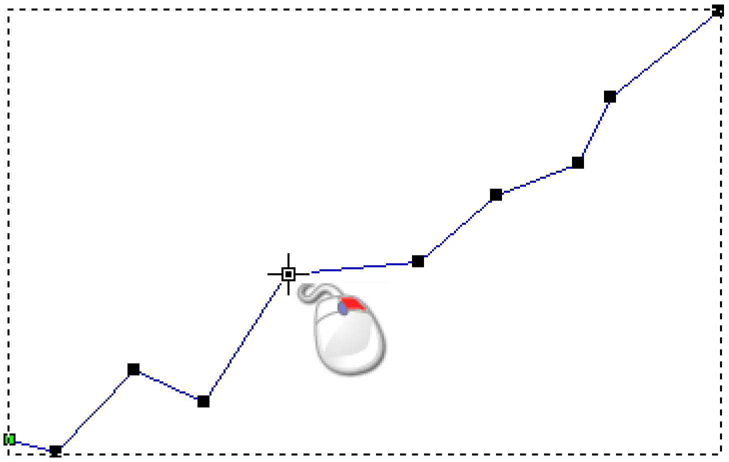
3. Use one of the following methods to smooth a node:

   - Right-click the node you want to smooth, then select **Smooth node** from the context menu; or
   - Select the node (see page 144), then press the **S** key.

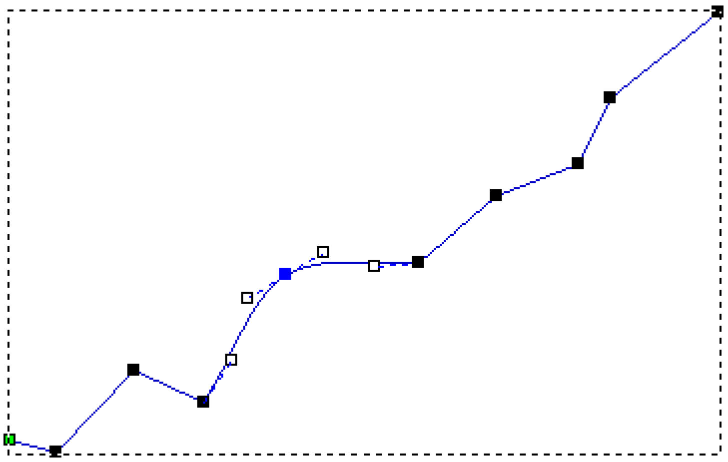
   The spans on either side of the node are converted to bezier curves. The node changes from black to blue.

   For example, a node in a polyline is displayed as shown when smoothing is applied:

   *Before...*



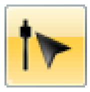
   *After...*



If you move one of the control points joined to the smoothed node, the other moves too. This simultaneous movement preserves the tangency between the two bezier curve spans.

To smooth a selection of nodes simultaneously:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

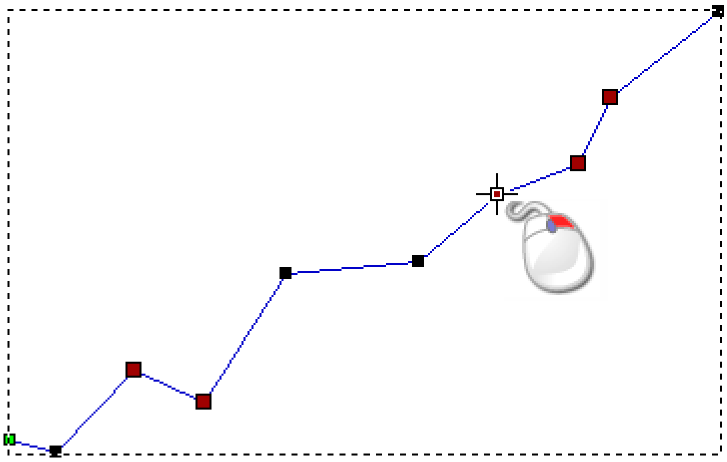
   *If the* ***Node Editing*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Node Editing mode.*

   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.
3. Hold down the **Ctrl** key, and then select the nodes (see page 144) you want to smooth. The selected nodes are red.
4. Use one of the following methods to smooth the selected nodes:
   - Right-click any of the selected nodes, then select **Smooth Points** from the context menu; or
   - Press the **S** key.

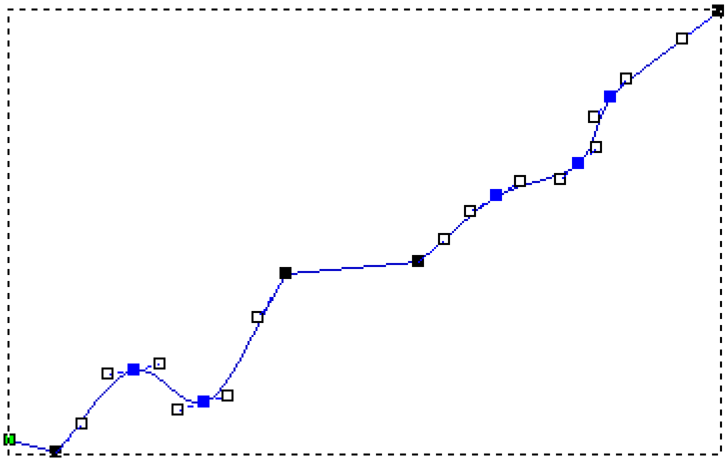
   The spans on either side of the selected nodes are converted to bezier curves.

   For example, a selection of five nodes in a polyline is displayed as shown below when smoothing is applied:

   *Before...*

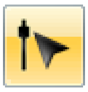


*After...*



To remove the smoothing applied to a node:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

   *If the* **Node Editing** *button is shown as*  *in the* **Design Tools** *toolbar, then you are already in Node Editing mode.*

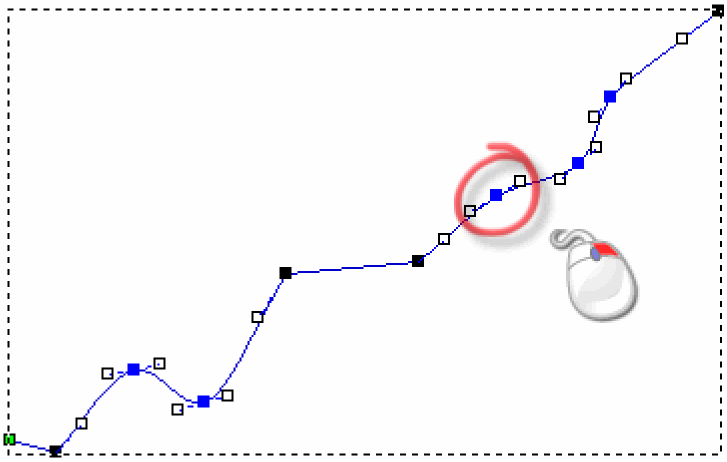
   A bounding box surrounds the selected vector, within which you can see the spans, nodes and control points that make up the selected vector.

3. Position the mouse cursor over the node from which you want to remove smoothing.
4. When the mouse cursor changes to :
   - right-click and select **Smooth node** from the context menu; or
   - press the **S** key.

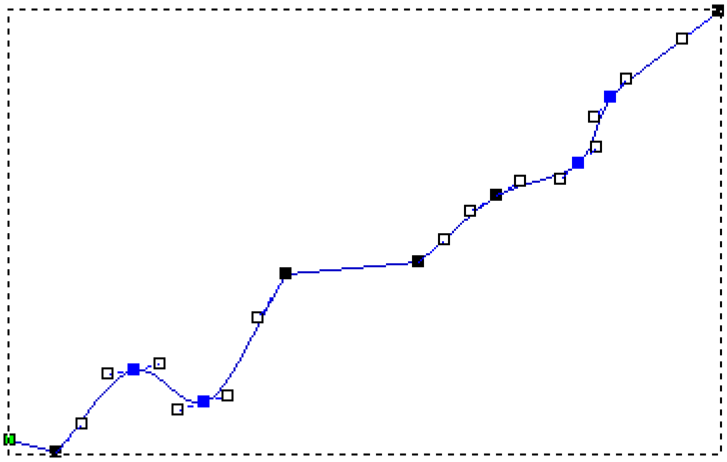
   The smoothing is removed, and the node changes from blue to black.

   For example, a node in a polyline is displayed as shown below when its smoothing is removed:

*Before...*



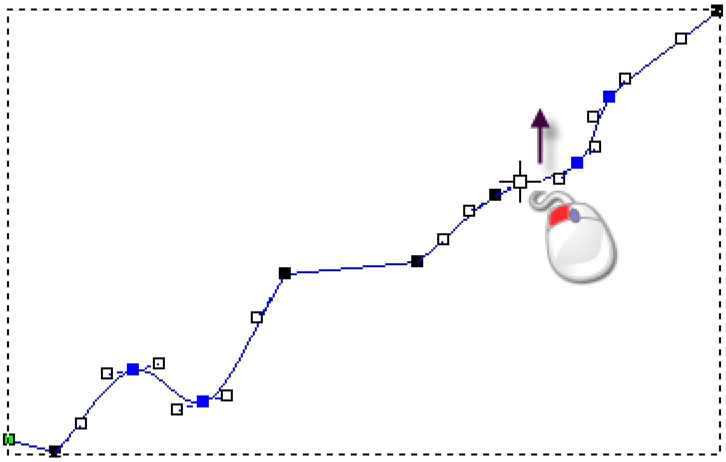
*After...*



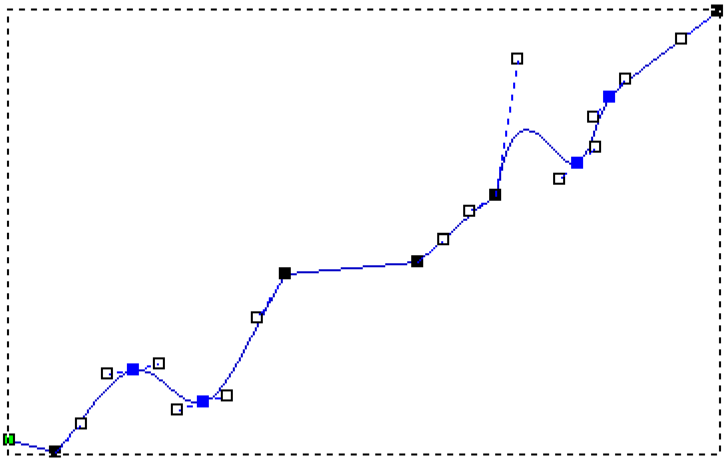
Although the span on either side of the node remains as a bezier curve span, removing its smoothing causes the control point on either side of the node to affect its adjoining bezier curve span only, rather than the whole of the selected vector.

For example, when we move the control point on the right of the node shown below, we can see that the control point on its left remains in the same position:

*Before...*



*After...*



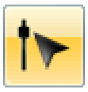
## Aligning nodes

You can align a selection of nodes and control points in either the horizontal (X) or vertical (Y) axis. All selected nodes and control points are replaced with a horizontal or vertical linear span.

*To keep intermediate nodes and control points when aligning nodes, deselect the* ***Align Nodes - replace with a single line*** *option on the* ***Options*** *panel (see page 49).*

To align a selection of nodes and control points in a vector:

1. Select the vector (see page 122) you want to edit.
2. Enter Node Editing mode (see page 142).

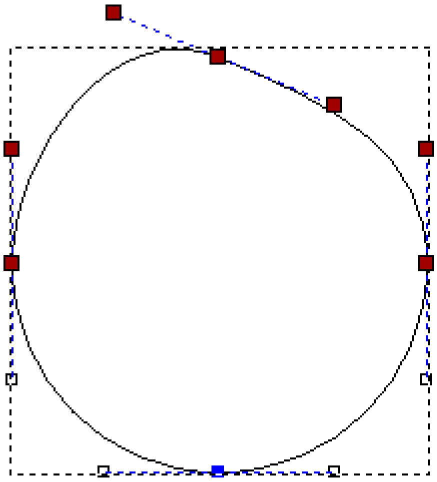
*If the* ***Node Editing*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Node Editing mode.*

A bounding box surrounds the selected vector, within which you can see its spans, nodes and control points.

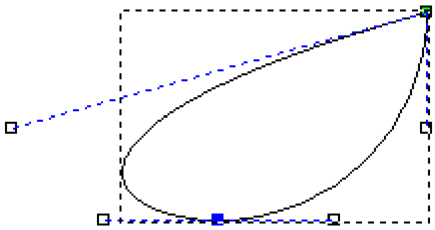
3. Hold down the **Ctrl** key, then select the nodes (see page 144) and control points you want to align. The selected nodes and control points are red.
4. To align the selected nodes and control points, right-click any of the selected nodes to display the context menu, then select:
   - **Align Nodes > in X** to align them horizontally
   - **Align Nodes > in Y** to align them vertically.

   For example, a selection of nodes and control points look as shown below when aligned horizontally and vertically:

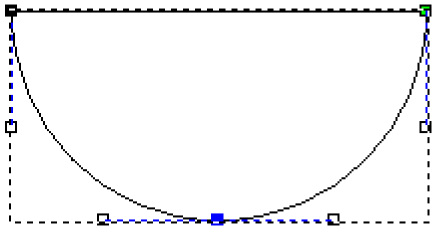
   *Before...*



*Align in X...*



*Align in Y...*



# Importing artwork from PDF files

Adobe's Portable Document Format (***.pdf**) files can contain both bitmap and vector artwork. Rather than gradually importing separate pieces of vector and bitmap artwork into a model, you can use a PDF file to:

- batch import several pieces into an open model; or
- create a new model containing several pieces of artwork.

The artwork from the PDF file can be edited in the same way as if it was created using ArtCAM's vector and bitmap tools.

Every page in a PDF file has a crop box that specifies the region to which its contents are clipped when displayed or printed. When creating a new model from a PDF file, ArtCAM uses the crop box on its first page to set the model dimensions. This ensures that all vector and bitmap artwork in the PDF file is included in the model.

When importing a PDF file into an open model with dimensions equal to or greater than those of the crop box, all of its vector and bitmap artwork is imported. If the model's dimensions are smaller than the crop box, all vector artwork is imported, but any bitmap artwork outside of the model area is lost.

The vector and bitmap artwork extracted from each page in the PDF file is placed on separate vector and bitmap layers. Each of these vector and bitmap layers is:

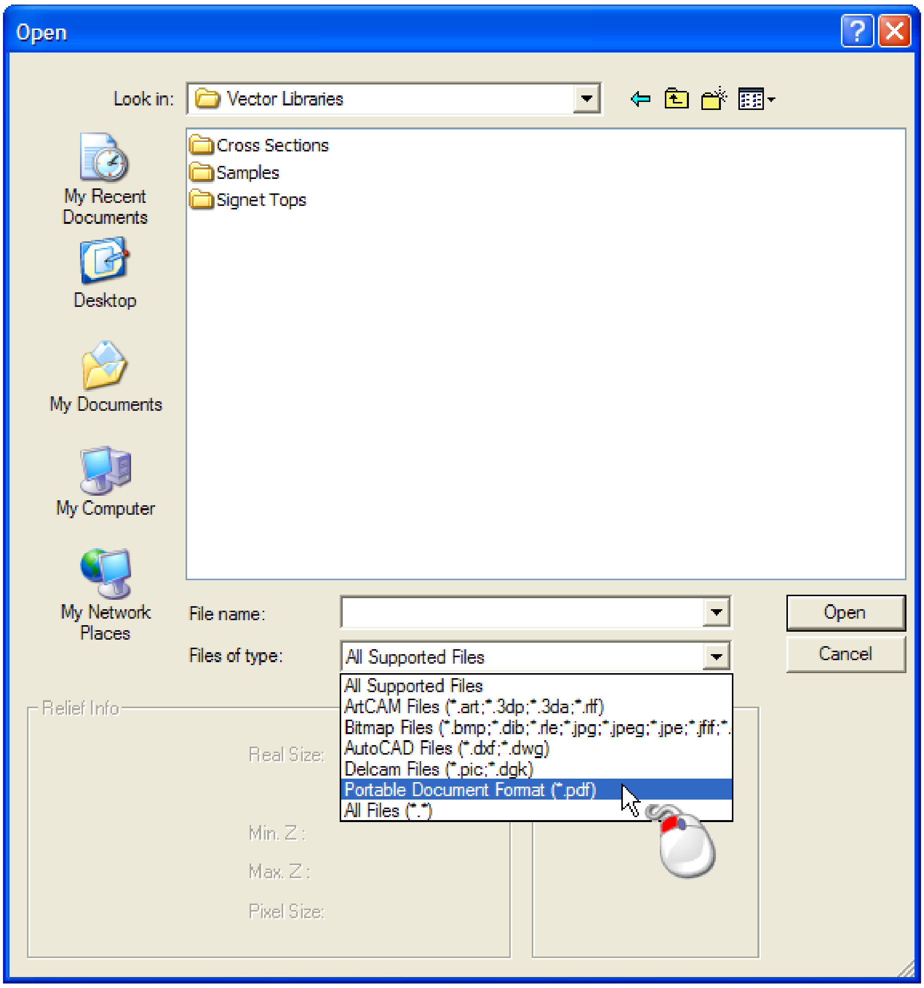
- named with the PDF file's title and the page number from which its artwork was extracted;
- added to the layer stack directly above the layer which was previously active;
- visible; and
- unlocked.

### *Creating a model from a PDF file*

To create a new model using a PDF file:

1. Use one of the following methods to display the **Open** dialog box:
    - From the **Start** panel, click **Open Model**;
    - From the **Menu Bar**, click **File > Open**; or

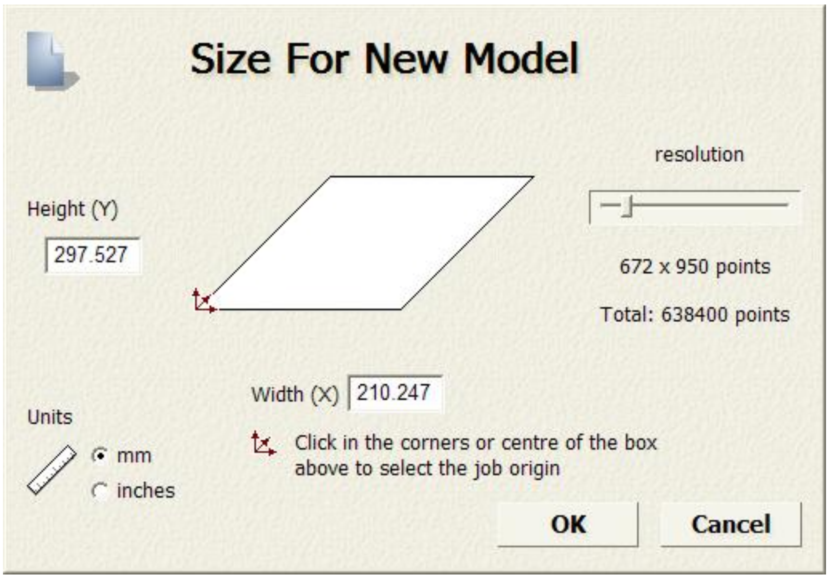
- Press the **Ctrl+O** keys.



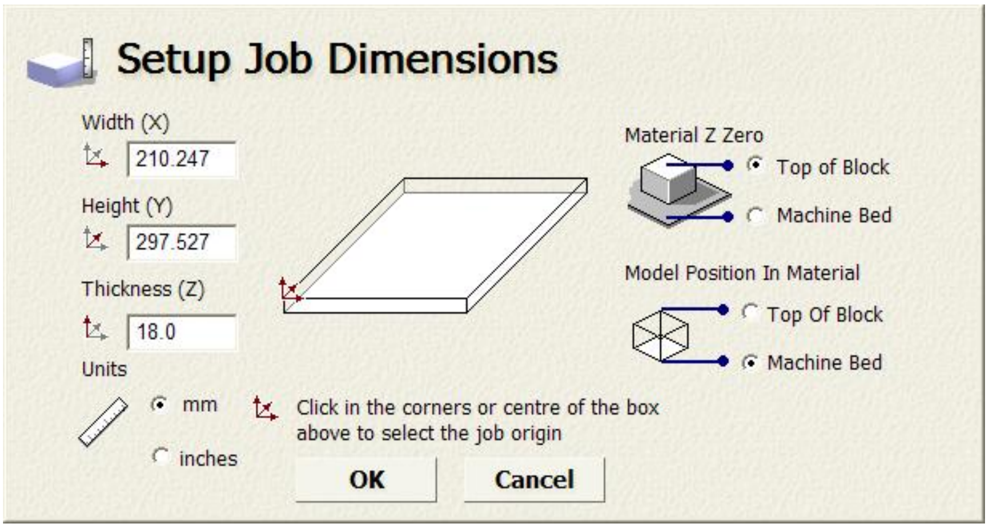

*To list only PDF files, click the **Files of type** list box, then select the **Portable Document Format (*.pdf)** option.*

2. Select the PDF file you want to work with, and then click **Open**.

If you are working in ArtCAM Pro or JewelSmith, the **Size For New Model** dialog box is displayed:



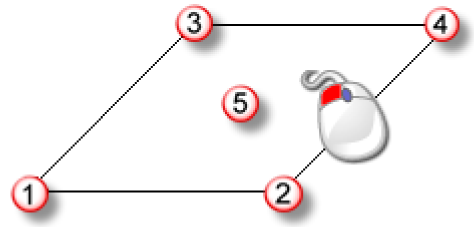

If you are working in ArtCAM Insignia, the **Setup Job Dimensions** dialog box is displayed:




3. In the **Units** area, select **mm** or **inches**.
4. In the **Height (Y)** and **Width (X)** boxes, specify the height and width of the model you want to create. The default values are taken from the PDF file's crop box.

   If you adjust the model size so that it is smaller than the PDF file's crop box, the bitmap artwork from the PDF file's first page outside of the model area is lost. The PDF file cannot be scaled to fit inside the model area.

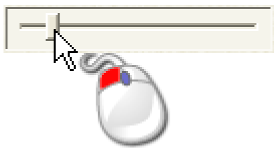
5. Specify the model's origin by clicking one of the numbered positions shown below:



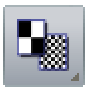

The  icon is displayed on the box diagram in your selected position.

6. If you are working in ArtCAM Pro or JewelSmith:

   a. In the **Resolution** area, click and drag the slider to set the model resolution:

   A total resolution of approximately 1,500,000 points is suitable for most jobs. When creating vectors or reliefs using the bitmap artwork extracted from a PDF file, choosing a higher resolution helps to produce cleaner results.

 *After a model is created, you can use the* ***Adjust Model Resolution***  *button on the* ***Model*** *toolbar to change the resolution.*

7. If you are working in ArtCAM Insignia:

   a. In the **Thickness (Z)** box, specify the thickness of the material you are using for machining.

   b. In the **Material Z Zero** area, specify the Z-axis zero level. This is the position of the cutting tool relative to the material block's surface. Click:

   **Top of Block** to position the cutting tool on the material surface; or

   **Machine Bed** to position the cutting tool on the machine bed.

   The  icon is displayed in either the top or bottom front left corner of the box diagram to mark the job origin.

   c. In the **Model Position In Material** area, set the position of the model relative to the material block. Click:

**Top Of Block** to align the model's zero plane with the material surface; or

**Machine Bed** to align the model's zero plane with the machine bed.

8. Click **OK** to create your model.

   A tabbed **2D View** and **3D View** window is created in the viewing area. The **2D View** window is selected by default, in which you can see the vector artwork extracted from each of the PDF file's pages.

9. On the **Project** panel, click ⊞ beside:

   - **Vectors** to display the vector layers created from each of the PDF file's pages; or
   - **Bitmaps** to display the bitmap layers created.

## Importing a PDF file into an open model

To import a PDF file into an open model:

1. Use one of the following methods to display the **Load Vector Layer** dialog box:

   - From the **Project** panel, right-click **Vectors** in the Project Tree, then click **Import** from the context menu;
   - From the **Vector Layers** panel, click the **Import Vectors** button; or
   - From the **Menu Bar**, click the **Vectors > Load Layer...** option.

2. In the **Load Vector Layer** dialog box, select the PDF file you want to import, and then click **Open**.

# The 3D Design Process

After you have created or imported your vector and bitmap artwork, and completed any necessary editing, you are ready to create:

- 2D toolpaths (see page 240); or
- reliefs.

A relief is essentially one or more three-dimensional shapes in a model, although there are two different concepts to consider when working with them: the 'relief layer' and the 'composite relief'.

The content of a *relief layer* comprises one or more three-dimensional shapes typically created:

- from attributes (see page 177) applied to vector artwork and bitmap colours;
- by importing (see page 226) a triangle model, surface model or piece of relief clipart;
- using the Sculpting tools (see page 232).

The content on these relief layers combine to form a *composite relief*. Exactly how each of the relief layers interact depends on the combine mode (see page 169) assigned to each of them, and whether or not they are visible (see page 172).

A composite relief can be built from two separate layer stacks in the model; one representing the front surface of your design, the other representing the back. You can view the composite relief resulting from these layer stacks separately or concurrently.

When you have created or imported your composite relief, you are ready to create and calculate the 3D toolpaths (see page 240) needed to machine it. These toolpaths can then be simulated (see page 272) so that you can check for potential issues during the machining process, the surface finish, and envision your finished piece.

# Using relief layers

Relief layers, like vector and bitmap layers, are contained in a stack. Unlike vector and bitmap layers, there are two stacks of relief layers; one representing the front surface and another representing the back surface of a three-dimensional model.

*If you are working in ArtCAM Insignia, there are no relief layers.*

When working from the **Project** panel, both layer stacks are displayed simultaneously in the Project Tree. When working from the **Relief Layers** panel, only one of the relief layer stacks can be displayed at a time.

You can use one of the following methods to display either of the relief layer stacks:

- From the **Project** panel, click ⊞ beside **Front Relief** or **Back Relief** in the Project Tree; or

  *The **Project** panel is docked and pinned on the right of the interface.*

- From the **Relief Layers** panel, click the list box, then **Front Relief** or **Back Relief**.

  *To display the **Relief Layers** panel, right-click a docking area then select **Relief Layers** in the context menu.*

Every ArtCAM model contains an empty default relief layer on each of the stacks, named *Relief Layer*. However, you can create as many relief layers as you want; either empty, or from imported clipart.

## Choosing the active layer

Only one relief layer can be active at a time, although the content across all visible relief layers is displayed in the **3D View** window.

Using the active relief layer you can:

- create a bitmap layer;

- export its relief as a ***.rlf**, ***.bmp** or ***.tif** file;
- create a duplicate copy of the layer and its content; or
- delete the layer and its content.

To choose the active relief layer from the **Project** panel:

1. Click ⊞ beside:
   - **Front Relief** in the Project Tree to display the layer stack that makes up the model's front surface; or
   - **Back Relief** in the Project Tree to display the layer stack that makes up the model's back surface.
2. In the Project Tree, click the layer you want to work with. Its name is bold and highlighted.

To choose the active relief layer from the **Relief Layers** panel:

*To display the **Relief Layers** panel, right-click a docking area, toolbar or the **Status Bar**, then select **Relief Layers** from the context menu.*

1. Click the list box, then select:
   - **Front Relief** to display the layer stack that makes up the model's front surface; or
   - **Back Relief** to display the layer stack that makes up the model's back surface.
2. In the layer stack, click the layer you want to work with. Its name is highlighted.

## Creating a new layer

In an ArtCAM model, you can create as many relief layers as you like. This gives you greater control over changing the appearance of the composite relief.

You can use one of the following methods to create a new relief layer:

- From the **Project** panel, right-click **Front Relief** or **Back Relief** in the Project Tree, then select **New** from the context menu;
- From the **Relief Layers** panel, click the **New Relief Layer** button; or
- From the **Menu Bar**, click the **Reliefs > Create New Layer** option.

Each new relief layer that you create is:

- named *Relief Layer*;
- numbered sequentially;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active;
- given the **Add** combine mode; and
- visible in the **3D View** window.

## Setting the combine mode

In the **3D View** window, the relief you see is the composite relief. It is made from a stack of one or more visible relief layers. You can significantly affect how the composite relief takes shape by changing a relief layer's combine mode. ArtCAM starts with the layer at the bottom of the stack, then moves upwards, applying the combine mode for each visible layer.

To set a relief layer's combine mode:

1. Make sure the relief layer is visible (see page 172).

   When working from the **Project** panel, the combine mode icon in the Project Tree is greyed-out when a relief layer is hidden. For example, .

2. Press the **F3** key to display the **3D View** window so you can see how the composite relief changes as a result of setting the layer's combine mode.

3. Use one of the following methods to specify how you want to combine the relief layer's content with each of the visible layers below in the stack:

   - From the **Project** panel, right-click the relief layer in the Project Tree, then select the combine mode option in the context menu.

- From the **Relief Layers** panel, click the combine mode button currently shown to display the combine mode toolset, then click the button you want to use.


- From the **Relief Layers** panel, click the combine mode button currently shown to toggle through each of them in the toolset.

Select:

**Add** to add the relief layer's content to the visible layers below in the stack.

**Subtract** to subtract the relief layer's content from the visible layers below in the stack.

**Merge High** to merge the relief layer's content with the visible layers below in the stack, so that only the highest points remain.

**Merge Low** to merge the relief layer's content with the visible layers below in the stack, so that only the lowest points remain.

**Multiply** to multiply the height of the relief layer's points by the coincident (matching) points on the visible layers below in the stack.

*If you create a new relief layer* *(see page 168)**, the* ***Add*** *combine mode is selected by default.*

*When you import relief clipart, you can choose the layer's combine mode.*

*To create a feasible composite relief, a relief layer with the* ***Multiply*** *combine mode applied should have a maximum Z height of between 0 and 1.*

## Renaming a layer

In every new ArtCAM model, the default relief layer is named *Relief Layer*. All relief layers you create are named *Relief Layer*, and given a unique number. When you import relief clipart, the new relief layer uses the name of the file from which it is created. You can rename any of the relief layers in a model.

To rename a relief layer:

1. Use one of the following methods to display the layer's name box:
   - From the **Project** panel, right-click the layer in the Project Tree, then click **Rename** in the context menu; or
   - From the **Relief Layers** panel, double-click the layer in the stack.
2. In the name box, type the name you want to give to the layer.

   *To revert to the name previously given to the layer, press the* ***Esc*** *key.*
3. Press the **Enter** key, or click in the empty space below the layer stack to set the name of the layer.

## Creating a bitmap layer from a relief layer

You can create a new bitmap layer containing a greyscale image of the currently active relief layer's content.

The new bitmap layer is:

- named with the relief layer from which the greyscale image was created;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active; and
- visible.

Use one of the following methods to create the bitmap layer:

- From the **Project** panel, right-click the relief layer in the Project Tree, then click **Create Bitmap** in the context menu; or
- From the **Relief Layers** panel, click the **Create Bitmap Layer** button.

## Viewing a layer

You can change the appearance of the composite relief by choosing a specific relief layer and controlling whether or not it is visible. When a relief layer is created, it is visible by default.

To control a relief layer's visibility:

1. If you are working with the layer stack associated with the **Front Relief**, select the **Toggle Front Relief** button in the **3D View** toolbar.
2. If you are working with the layer stack associated with the **Back Relief**, select the **Toggle Back Relief** button in the **3D View** toolbar.
3. Use one of the following methods to control whether or not a layer's content can or cannot be seen:
   - To hide a layer's content, click its icon;

     When a relief layer is hidden from the **Project** panel, its combine mode icon in the Project Tree is greyed-out. For example, changes to .
   - To hide the content on all but one layer, right-click the icon associated with the layer you want to keep visible;
   - To hide the content on all layers in either stack when working from the **Project** panel, click the icon beside **Relief** in the Project Tree;
   - To hide the content on all layers when working from the **Relief Layers** panel, click the **Toggle All Visibility** button;
   - To display a layer's content, click its icon;
   - To display the content on all but one layer, right-click the icon associated with the layer you want to keep hidden;
   - To display the content on all layers in either stack when working from the **Project** panel, click the icon beside **Relief** in the Project Tree; or
   - To display the content on all layers when working from the **Relief Layers** panel, click the **Toggle All Visibility** button on the **Relief Layers** panel.

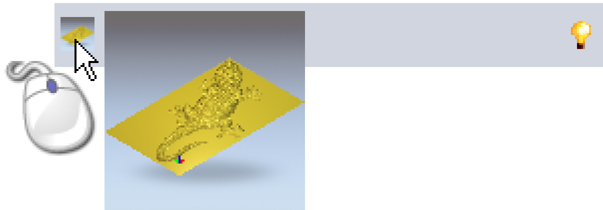
## Previewing a relief layer's content

On the **Relief Layers** panel, each relief layer has its own thumbnail image that enables you to check its content whether it is currently visible in the **3D View** window or not. This is particularly valuable when a composite relief is made up of a large number of relief layers.

*The **Relief Layers** panel is hidden by default. To display the panel, right-click a docking area then select **Relief Layers** from the context menu.*

To preview the content on a relief layer:

1. On the **Relief Layers** panel, move the mouse cursor over the relief layer's thumbnail image to the left of its relief combine mode toolset.

   The thumbnail image is magnified to display the content on the relief layer more clearly.

   *The dimensions of the preview thumbnail image associated are 17 x 17 pixels. The dimensions of the magnified thumbnail image are 100 x 100 pixels.*

   *The thumbnail image, either in its original or magnified state, is a scaled-to-fit isometric view of the relief layer as if it were shown in isolation in the **3D View** window. The thumbnail view does not use the view orientation currently applied to the **3D View** window.*

   *If you move the mouse cursor up or down vertically in line with the column of thumbnail images in the layer stack, the magnified thumbnail image is updated.*

## Transferring relief layers between stacks

The composite relief can be built from one of two layer stacks in the model; one representing the front surface of your piece, the other its back.

Although you can only work with one stack at a time, you can use the **Toggle Front Relief** and **Toggle Back Relief** tool in the **3D View** toolbar to concurrently view the composite relief resulting from both stacks.

When you transfer a relief layer from one stack to the other, the composite relief resulting from both layer stacks is affected. It is prudent to check the composite relief shown in the **3D View** window whenever you transfer a relief layer.

In every new model, each of the stacks includes an empty relief layer. There must always be at least one layer in each of them.

To transfer a relief layer to the opposite layer stack:

1. Use one of the following methods to display the layer stack from which you want to transfer a layer:
   - From the **Project** panel, click **Front Relief** or **Back Relief** in the Project Tree; or
   - From the **Relief Layers** panel, click the list box, then select the layer stack: **Front Relief** or **Back Relief**.
2. Choose the layer (see page 167) you want to transfer to the opposite stack. Its name is bold and highlighted.

   *Rather than transferring the active relief layer to the opposite stack, you might prefer to duplicate the layer and transfer this instead. This means that an identical layer is included in both stacks.*

   *Where there is only one layer in the current stack, you cannot transfer this to the opposite stack. If you try to transfer the only relief layer, a message box is displayed warning that this cannot be done. Click **OK** to close the message box.*
3. Use one of the following methods to transfer the currently active layer to the opposite layer stack:
   - From the **Project** panel, right-click the relief layer in the Project Tree, then click **Swap Sides** from the context menu; or
   - From the **Relief Layers** panel, click the **Transfer Opposite Side** button.

   The relief layer is no longer included in the layer stack currently displayed.
4. Use one of the following methods to confirm that the layer has been transferred to the opposite stack,

- From the **Project** panel, click beside the opposite **Relief** in the Project Tree; or
- From the **Relief Layers** panel, click the list box, then select the opposite layer stack: **Back Relief** or **Front Relief**.

The opposite layer stack is displayed and you can see the transferred relief layer. The composite relief made up from this layer stack is displayed in the **3D View** window.

## Creating a relief layer from bitmap artwork

You can create a new relief layer from a bitmap layer. Each colour in the bitmap layer's artwork produces a three dimensional shape with a particular height. The height is calculated using the colour's RGB values. In general, bright colours produce high shapes, while dark colours produce low shapes. You can restrict the height of these shapes.

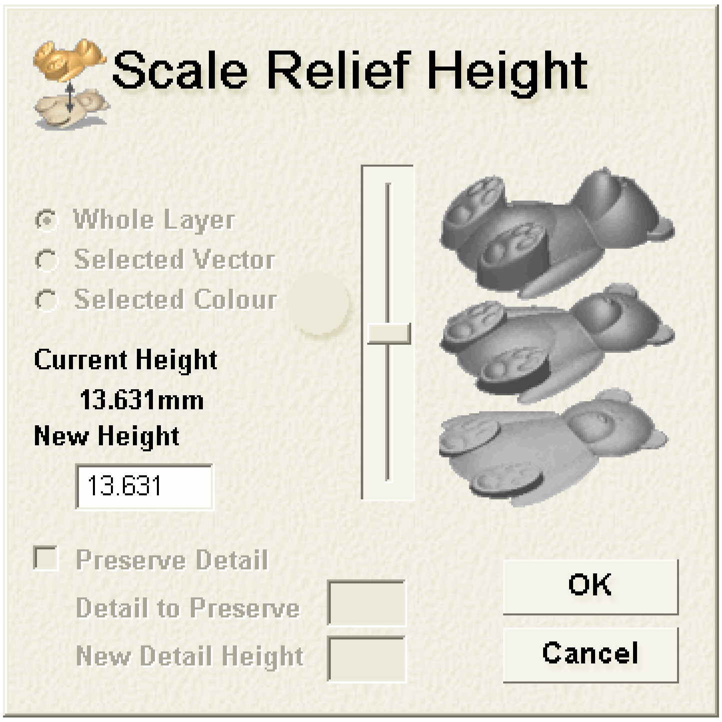
The new relief layer is:

- given the same name as the currently active bitmap layer;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active;
- given the **Add** combine mode ; and
- visible.

To create a relief layer from a bitmap layer:

1. Choose the bitmap layer (see page 99) you want to use to create a relief layer. Its name is bold and highlighted.
2. Use one of the following methods to display the **Scale Relief Height** dialog box:
    - From the **Project** panel, right-click the bitmap layer in the Project Tree, then select **Create Relief** from its context menu; or

- From the **Bitmap Layers** panel, click the **Create Relief Layer** button.




*The current height value displayed is not the actual Z height of the composite relief. This is an arbitrary value calculated at 1/4 of either the model's height (Y) or width (X); whichever is the smallest at the time of creating the relief.*

3. In the **New Height** box, specify the maximum Z height of the relief.
4. Click the **OK** button. The dialog box is closed, and the relief layer is created.

# Using the Shape Editor

You can create simple three-dimensional shapes on relief layers using the **Shape Editor** tool together with:

- colours drawn on bitmap layers (see page 103); or
- closed vectors drawn on vector layers (see page 132).

*If you are working in ArtCAM Insignia, the* ***Shape Editor*** *tool is not available.*

Using the **Shape Editor** tool, you can control:

- the shape's profile;
- the shape's angle;
- the shape's height; and
- how the shape is combined with any shapes already on the currently active relief layer.

## Creating simple shapes using bitmap colours

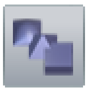
You can create a simple three dimensional shape using all areas of bitmap artwork displayed in the primary colour. The shape is created on the currently active relief layer.

To create a shape using bitmap artwork:

1. Choose the bitmap layer (see page 99) containing the artwork you want to use to create a shape.
2. Make sure the bitmap layer is visible (see page 101).
3. Choose the relief layer (see page 167) on which you want to create the shape.
4. If you are working in the **3D View** window, click the **Colour Shade**  button in the **3D View** toolbar to display the artwork on the currently active bitmap layer.
5. Make sure that you are working in Select mode (see page 122).

   *If the* ***Select*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Select mode.*

6. Select the colour you want to use to create the shape as the primary colour (see page 109).
7. Use one of the following methods to display the **Shape Editor**:

   a. In the **Relief Creation** toolbar, click the **Shape Editor**  button;

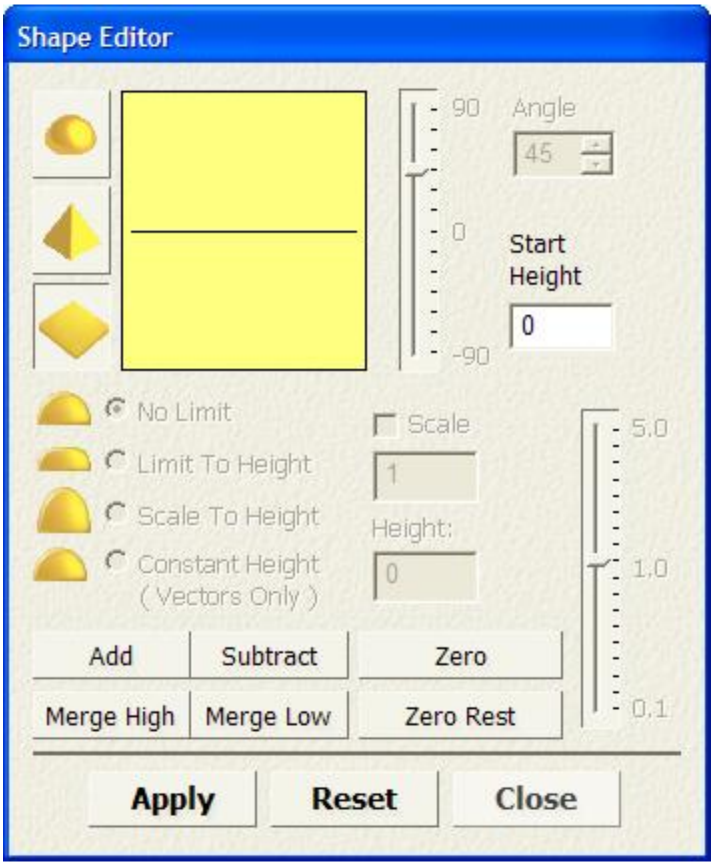
   b. From the **Menu Bar**, click **Model > Shape Editor** option; or

   c. Press the **F12** key;

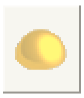
*You can double-click a colour to select it as the primary colour and display the* ***Shape Editor****.*

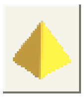
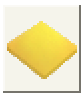
The primary colour is displayed in the **Shape Editor**'s window. The default settings enable you to create a flat shape with a start height of zero; this is represented by the horizontal line shown.

For example, with yellow selected as the primary colour, the **Shape Editor** looks as follows:




8. Select the shape's profile by clicking one of the following buttons:

- Click  to select a rounded profile;

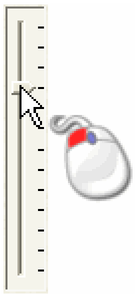
- Click  to select an angled profile; or
- Click  to select a flat profile.

  The image shown in the dialog's window changes according to your selected profile.

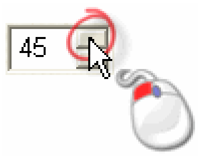
9. In the **Start Height** box, specify the Z height at which the shape is created. This creates a vertical side wall in the shape. If you are creating a flat shape, this also controls the height of the plane.
10. If you are creating a rounded or angled shape, specify its angle using one of the following methods:
    - In the **Angle** box, type the angle; or

    *Type a positive value to create a convex shape, a negative value to create a concave shape, or 0 to create a plane.*

    - Click and drag the leftmost slider; or

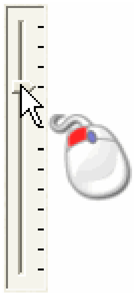
    

    - Click or on the right of the **Angle** box:

11. If you are creating a rounded or angled shape, choose the method you want to use to control the shape's height:
    - To allow the shape to grow to the height that it would naturally reach, click the **No Limit** option. This is selected by default.

*To apply a scaling factor to the shape in its Z-axis, click* ***Scale****, then specify the scaling factor either by typing it in the* ***Scale*** *box or clicking and dragging the rightmost slider:*



*Drag the slider upwards to increase the scaling factor, or downwards to decrease the scaling factor. The scaling factor is shown in the* ***Scale*** *box.*

- To allow the shape to grow to a specific height and then plateau, click the **Limit To Height** option, then specify the maximum height in the **Height** box.

*If the natural height of the shape exceeds that of the value specified in the* ***Height*** *box, a flat top is created.*

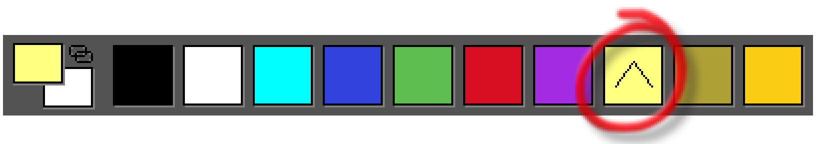
*To apply a scaling factor to the shape in its Z-axis, click* ***Scale****, then set the scaling factor either by typing it in the* ***Scale*** *box or clicking and dragging the rightmost slider.*

- To allow the shape to grow to a specific height by applying a scaling factor in its Z-axis, click the **Scale To Height** option, then specify the height in the **Height** box. This does not create a flat top in the shape.

12. To apply the shape attributes to the colour, click **Apply**.

*You can revert to the* ***Shape Editor****'s default settings by clicking* ***Reset****.*

The shape profile applied to the colour is displayed on it's swatch in the colour palette. For example, if an angled profile is applied to yellow, it is shown as follows:



13. Select the relief combine mode (see page 185) you want to use to combine the shape with the currently active relief layer:

*The selected combine mode controls how the shape is combined with the active relief layer only. It does not control how the shape is combined with the composite relief. How the shape is combined with other relief layers to make the composite relief is determined by the relief layer's combine mode (see page 169).*

*To reset all areas beneath the colour to zero, click* ***Zero****.*

*To reset all areas to zero, other than those beneath the colour, click* ***Zero Rest****.*

14. Click **Close** to close the **Shape Editor**.

*If you click* ***Close*** *before applying your specified shape attributes, a message box is displayed confirming they have changed and asking if you want to save them. Click* ***Yes*** *to apply the shape attributes or* ***No*** *to delete them before closing the* ***Shape Editor****.*

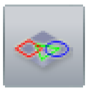
15. If you are working in the **2D View** window, press **F3** to display the **3D View** window and see your shape.

*Click the* ***Colour Shade***  *button in the* ***3D View*** *toolbar to hide the artwork on the currently active bitmap layer.*

## Creating simple shapes using closed vectors

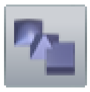
You can create a simple three dimensional shape using closed vector artwork. The shape is created on the currently active relief layer.

To create a shape using closed vector artwork:

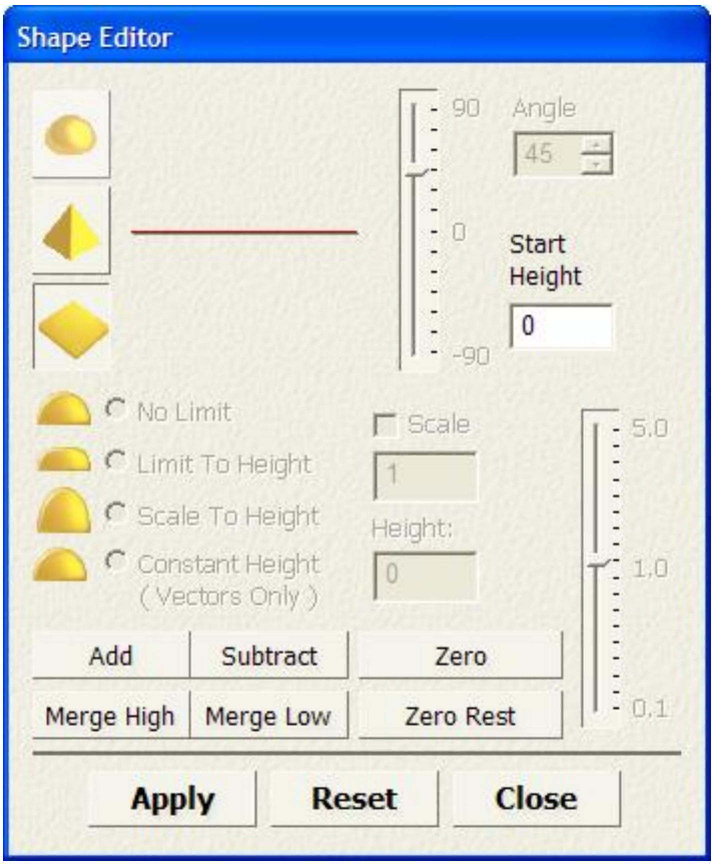
1. Make sure the vector layer containing the closed vector you want to (see page 130) use is visible.
2. If you are working in the **3D View** window, click the **Toggle Vector Visibility**  button in the **3D View** toolbar to display the artwork on all visible vector layers.
3. Make sure that you are working in Select mode (see page 122).

*If the* ***Select*** *button is shown as*  *in the* ***Design Tools*** *toolbar, then you are already in Select mode.*

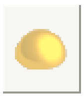
4. Select the closed vector (see page 122) you want to use to create a shape.
5. Use one of the following methods to display the **Shape Editor** dialog box:

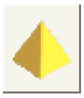
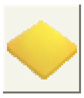
   - In the **Relief Creation** toolbar, click the **Shape Editor**  button;
   - From the **Menu Bar**, click the **Model > Shape Editor** option;
   - Press the **F12** key; or
   - Double-click the selected vector.

    *If you use the last method, be careful not to move the vector at the same time.*


   The default settings enable you to create a flat shape with a start height of zero; this is represented by the horizontal line shown.
6. Select the shape's profile by clicking one of the following buttons:

   - Click  to select a rounded profile;

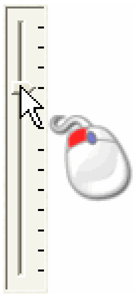
- Click  to select an angled profile; or
- Click  to select a flat profile.

The image shown in the dialog's window changes according to your selected profile.

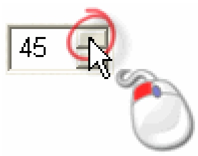
7. In the **Start Height** box, specify the Z height at which the shape is created. This creates a vertical side wall in the shape. If you are creating a flat shape, this also controls the height of the plane.
8. If you are creating a rounded or angled shape, specify its angle using one of the following methods:
   - In the **Angle** box, type the angle; or

   *Type a positive value to create a convex shape, a negative value to create a concave shape, or 0 to create a plane.*

   - Click and drag the leftmost slider; or

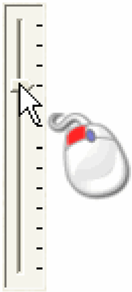
   

   - Click or on the right of the **Angle** box:


9. If you are creating a rounded or angled shape, choose the method you want to use to control the shape's height:
   - To allow the shape to grow to the height that it would naturally reach, click the **No Limit** option. This is selected by default.

*To apply a scaling factor to the shape in its Z-axis, click* ***Scale****, then specify the scaling factor either by typing it in the* ***Scale*** *box or clicking and dragging the rightmost slider:*



*Drag the slider upwards to increase the scaling factor, or downwards to decrease the scaling factor. The scaling factor is shown in the* ***Scale*** *box.*

- To allow the shape to grow to a specific height and then plateau, click the **Limit To Height** option, then specify the maximum height in the **Height** box.

*If the natural height of the shape exceeds that of the value specified in the* ***Height*** *box, a flat top is created.*

*To apply a scaling factor to the shape in its Z-axis, click* ***Scale****, then set the scaling factor either by typing it in the* ***Scale*** *box or clicking and dragging the rightmost slider.*

- To allow the shape to grow to a specific height by applying a scaling factor in its Z-axis, click the **Scale To Height** option, then specify the height in the **Height** box. This does not create a flat top in the shape.
- To allow the shape to grow to a specific height, click the **Constant Height (Vectors Only)** option, then specify the height in the **Height** box. This creates a shape where its angle or curvature changes in order to maintain a constant height; even where its width varies.

10. To apply the shape attributes to the vector, click **Apply**.

*You can revert to the* ***Shape Editor****'s default settings by clicking* ***Reset****.*

11. Select the relief combine mode (see page 185) you want to use to combine the shape with the currently active relief layer:

*The selected combine mode controls how the shape is combined with the currently active relief layer only. It does not control how the shape is combined with the composite relief. How the shape is combined with other relief layers to make the composite relief is determined by the relief layer's combine mode (see page 169).*

*To reset all areas within the vector boundary to zero, click* ***Zero****.*

*To reset all areas to zero, other than those within the vector boundary, click* ***Zero Rest****.*

12. Click **Close** to close the **Shape Editor.**

*If you click* ***Close*** *before applying your specified shape attributes, a message box is displayed confirming they have changed and asking if you want to save them. Click* ***Yes*** *to apply the shape attributes or* ***No*** *to delete them before closing the* ***Shape Editor****.*

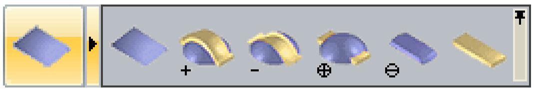
13. If you are working in the **2D View** window, press **F3** to display the **3D View** window and see your shape.

# Calculating a relief

The Relief Calculation tools control how the shape attributes applied to a colour in a bitmap layer's artwork calculate a new relief and combine it with the currently active relief layer.

The Relief Calculation tools are available from:

- the **Menu Bar**, by clicking **Reliefs > Calculate**.
- the **Shape Editor**; and
- the **Relief Tools** area of the **Assistant** panel, which is hidden by default.



*If you are working in ArtCAM Insignia, the Relief Calculation tools are not available.*

Using one of four combination methods, you can:

- **replace** (see page 186) the currently active relief layer's content with a new relief.
- **add** (see page 187) a new relief to the currently active relief layer.
- **subtract** (see page 189) a new relief from the currently active relief layer.
- **merge** (see page 191) a new relief with the currently active relief layer, so that only their highest or lowest points are kept.

## Replacing the relief

You can replace the currently active relief layer's content with a new relief.

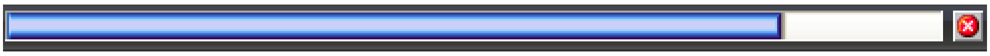
To replace one relief with another:

1. Choose the bitmap layer (see page 99) containing the artwork you want to use to create the relief.
2. Make sure the bitmap layer is visible (see page 101).
3. If you are working in the **3D View** window, make sure the **Colour Shade** button in the **3D View** toolbar is toggled on.
4. Apply shape attributes (see page 177) to a colour in the bitmap layer's artwork.
5. Choose the relief layer (see page 167) containing the relief you want to replace.

   *You can use the **Preview Relief Layer** button in the **2D View** toolbar to check the contents of the currently active relief layer.*

6. Use one of the following methods to replace the currently active relief layer's content with a new relief calculated from the attributes applied to the colours in your bitmap artwork:
   - From the **Menu Bar**, click **Reliefs > Calculate > Replace**; or
   - Click the **Replace Relief** button in the **Relief Tools** area of the **Assistant** panel;

During the calculation process, a progress bar is displayed in the **Status Bar**:



*You can cancel the relief replacement process by clicking* .

7. If you are working in the **2D View** window, press the **F3** key to display the composite relief in the **3D View** window.

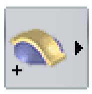
## Adding to the relief

Using bitmap artwork, you can add a new relief to the currently active relief layer's content.

To add a shape to the currently active relief layer:

1. Choose the bitmap layer (see page 99) containing the artwork you want to use to create the relief.
2. Make sure the bitmap layer is visible (see page 101).
3. If you are working in the **3D View** window, make sure the **Colour Shade**  button in the **3D View** toolbar is toggled on.
4. Apply shape attributes (see page 177) to the colours in the bitmap layer's artwork.
5. Choose the relief layer (see page 167) on which you want to add a new relief.

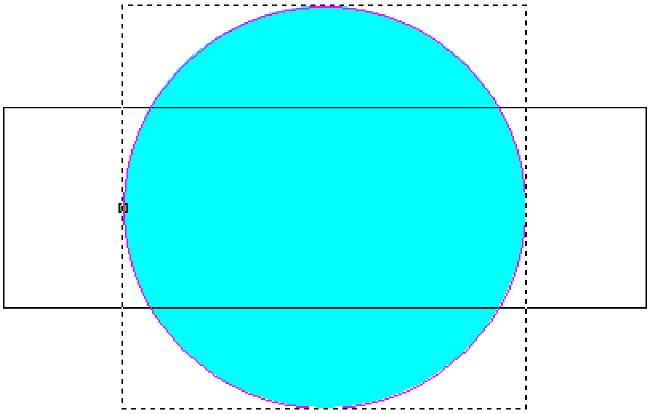
   *If you are working in the **2D View** window, you can use the **Preview Relief Layer***  *button in the **2D View** toolbar to check the contents of the currently active relief layer.*

6. Use one of the following methods to add the new relief calculated from the attributes applied to the colours in your bitmap artwork to the currently active relief layer's content:
   - From the **Menu Bar**, click **Reliefs > Calculate > Add**; or
   - Click the **Add Relief**  button in the **Relief Tools** area of the **Assistant** panel.

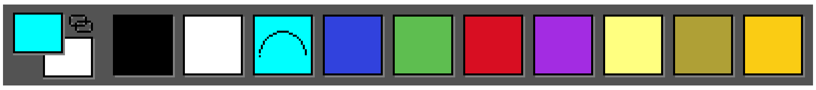
*The combine mode selected here controls how the shape is added to the currently active relief layer only. It does not control how the shape is combined with the composite relief. How the shape is combined with other relief layers to form the composite relief is determined by the combine mode assigned to the layer itself. For details, see Setting the combine mode (see page 169).*

7. If you are working in the **2D View** window, press the **F3** key to display the composite relief in the **3D View** window.

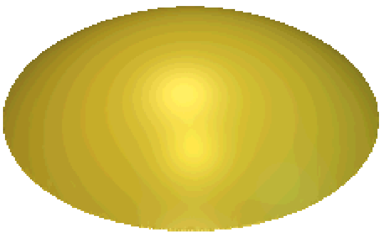
   For example, a cyan circle is painted on a bitmap layer:

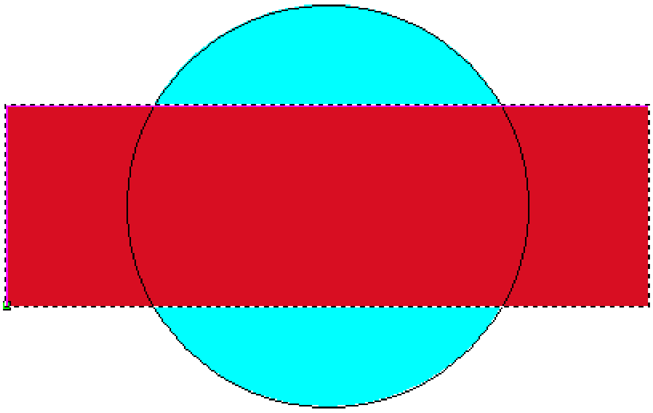
   Using the **Shape Editor**, a rounded shape is applied to the circle's colour, as you can see by looking at the colour palette below the **2D View** and **3D View** window:

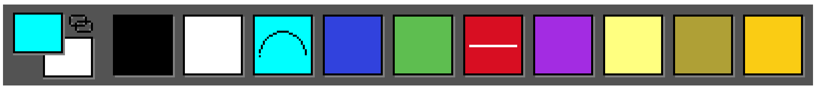
   The rounded shape is added to the currently active relief layer, and displayed in the **3D View** window as follows:

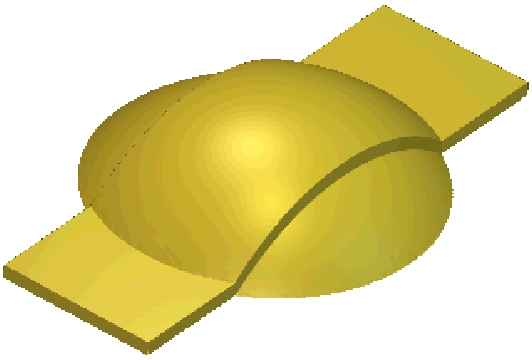
The rectangular vector is then flood-filled in red:



Using the **Shape Editor**, a plane shape is applied to the rectangle's colour, as you can see by looking at the colour palette below the **2D View** and **3D View** window:



The plane is added to the rounded shape on the currently active relief layer to create the following new relief:



## Subtracting from the relief

You can subtract a shape from the currently active relief layer.

To subtract a shape from the currently active relief layer:

1. Choose the bitmap layer (see page 99) containing the artwork you want to use to create the relief.
2. Make sure the bitmap layer is visible (see page 101).
3. If you are working in the **3D View** window, make sure the **Colour Shade**  button in the **3D View** toolbar is toggled on.
4. Apply shape attributes (see page 177) to a colour in the bitmap layer's artwork.

5. Choose the relief layer (see page 167) from which you want to subtract a new relief.

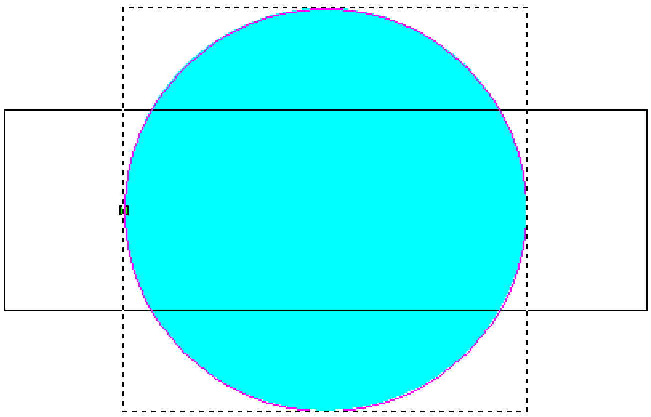
   *You can use the* ***Preview Relief Layer*** *button in the* ***2D View*** *toolbar to check the contents of the currently active relief layer.*

6. Use one of the following methods to subtract the relief from the currently active relief layer:
   - From the **Menu Bar**, click **Reliefs > Calculate > Subtract**; or
   - Click the **Subtract Relief** button in the **Relief Tools** area of the **Assistant** panel;

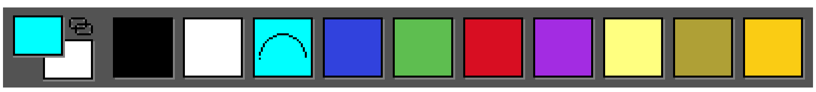
   *The combine mode selected here controls how the shape is subtracted from the currently active relief layer only. It does not control how the shape is combined with the composite relief. How the shape is combined with other relief layers to form the composite relief is determined by the combine mode assigned to the layer itself. For details, see Setting the combine mode (see page 169).*

7. Press the **F3** key to display the composite relief in the **3D View** window.

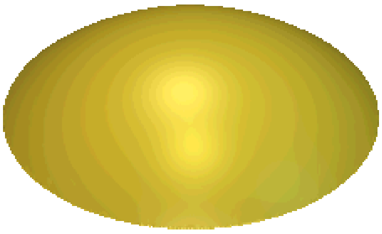
   For example, a cyan circle is painted on a bitmap layer:



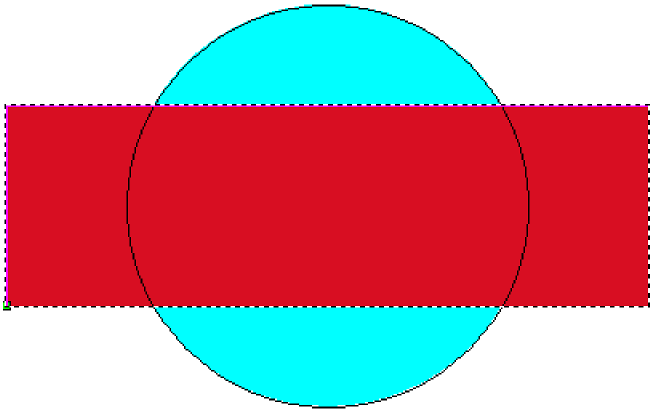
   Using the **Shape Editor**, a rounded shape is applied to the circle's colour, as you can see by looking at the colour palette below the **2D View** and **3D View** window:



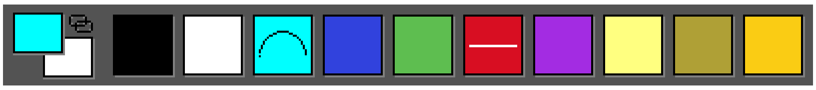
The rounded shape is added to the currently active relief layer, and displayed in the **3D View** window as follows:



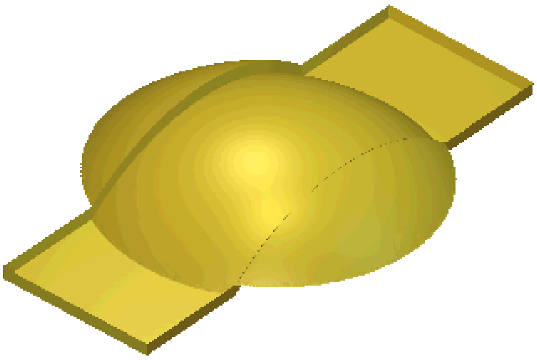
The rectangular vector is then flood-filled in red:



Using the **Shape Editor**, a plane shape is applied to the rectangle's colour, as you can see by looking at the colour palette below the **2D View** and **3D View** window:



The plane is subtracted from the rounded shape on the currently active relief layer to create the following new relief:



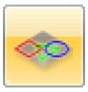
## Merging with the relief

You can merge a new relief with the currently active relief layer, so that only their:

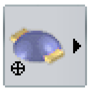
- highest points are kept; or

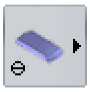
- lowest points are kept.

To merge a new relief with the currently active relief layer:

1. Choose the bitmap layer (see page 99) containing the artwork you want to use to create the relief.
2. Make sure the bitmap layer is visible (see page 101).
3. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on.
4. Apply shape attributes (see page 177) to a colour in the bitmap layer's artwork.
5. Choose the relief layer (see page 167) with which you want to merge a new relief.

 *You can use the* ***Preview Relief Layer*** *button*  *in the* ***2D View*** *toolbar to check the contents of the currently active relief layer.*

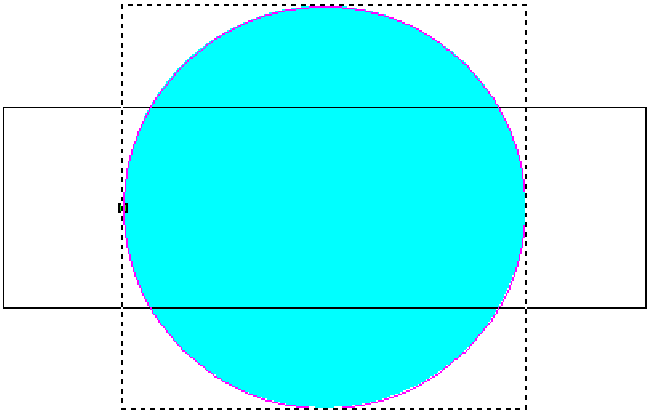
6. Use one of the following methods to merge the new relief calculated from the attributes applied to the colour with the currently active relief layer's content, so that only the highest points show:
   - From the **Menu Bar**, click **Reliefs > Calculate > Merge Highest**; or
   - Click the **Merge High**  button in the **Relief Tools** area of the **Assistant** panel.

   Use one of the following methods to merge the new relief calculated from the attributes applied to the colour with the currently active relief layer's content, so that only the lowest points show:
   - From the **Menu Bar**, click **Reliefs > Calculate > Merge Lowest**; or
   - Click the **Merge Low**  button in the **Relief Tools** area of the **Assistant** panel.

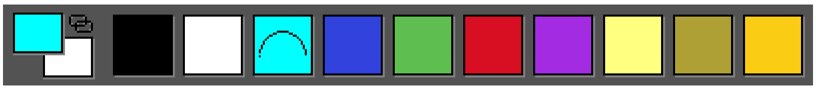
*The combine mode selected here controls how the shape is merged with the currently active relief layer only. It does not control how the shape is combined with the composite relief. How the shape is combined with other relief layers to form the composite relief is determined by the combine mode assigned to the layer itself. For details, see Setting the combine mode (see page 169).*

7. Press the **F3** key to display the composite relief in the **3D View** window.

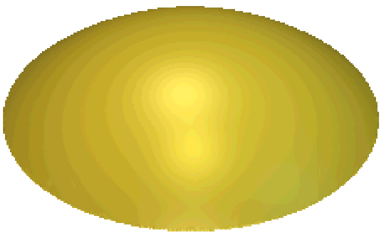
   For example, a cyan circle is painted on a bitmap layer:

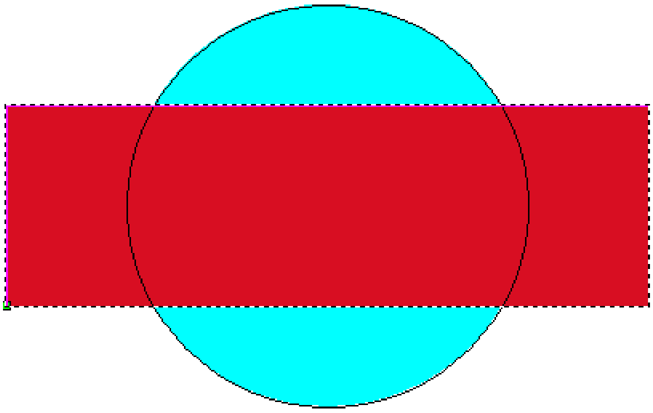
   Using the **Shape Editor**, a rounded shape is applied to the circle's colour as you can see by looking at the colour palette below the **2D View** and **3D View** window:

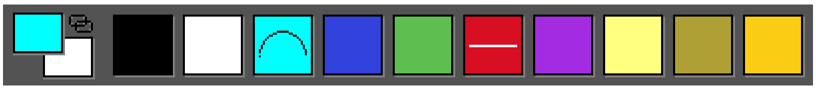
   The rounded shape is added to the currently active relief layer, and displayed in the **3D View** window as follows:

The rectangular vector is then flood-filled in red:

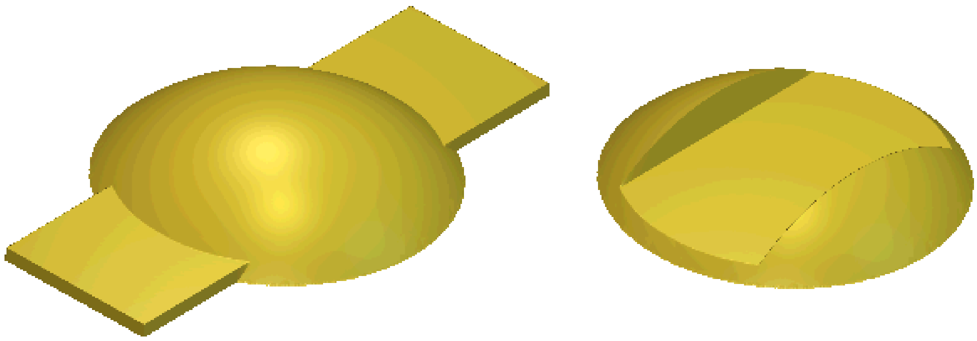


Using the **Shape Editor**, a plane shape is applied to the rectangle's colour, as you can see by looking at the colour palette below the **2D View** and **3D View** window:



The plane is merged with the rounded shape on the currently active relief layer to create a new relief. The different results from using the **Merge High** and **Merge Low** options are shown below:

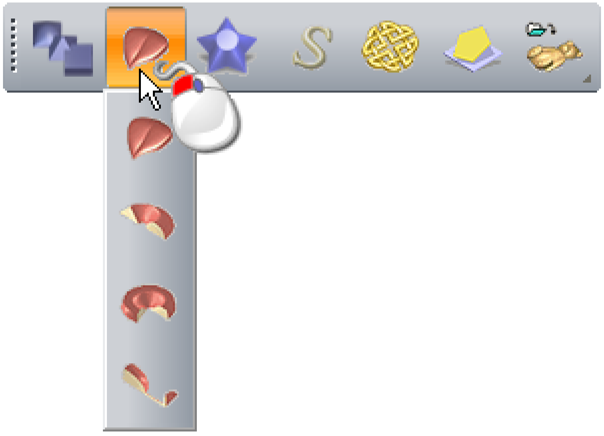
*Merge High…* *Merge Low…*



# Creating complex shapes using vectors

You can create complex free-form shapes on relief layers using vector artwork and the Swept Profiles toolset available:

- on the **Relief Creation** toolbar;



- from the **Menu Bar**, by clicking **Reliefs > Swept Profiles**;
- in the **Relief Tools** area of the **Assistant** panel.

*If you are working in ArtCAM Insignia, the Swept Profiles toolset is not available.*

The four shapes you can create are:

- extrude (see page 195)
- spin (see page 200)
- turn (see page 205); and
- two-rail sweep (see page 208).

## Creating an extruded shape

Extruded shapes are created on relief layers using vector artwork. An extruded shape uses at least two vectors:

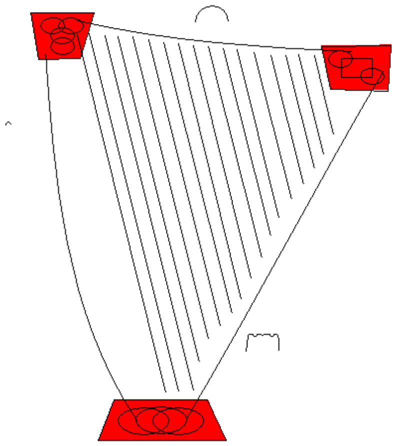
- one to specify the path along which the shape is extruded, known as a drive curve; and
- another to control the shape's profile at its start, known as the start profile.

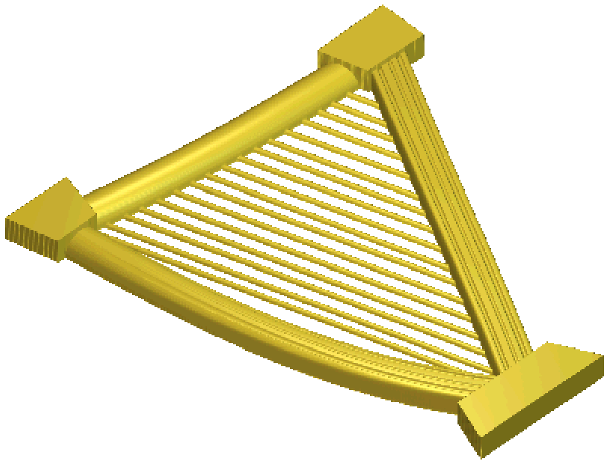
Up to two additional vectors can be used:

- one to control the shape's profile at its end, known as the end profile; and
- another to control the shape's height in the Z direction, known as the Z modulation profile.

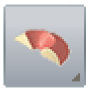
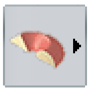
For example, the vector artwork shown below can be used to extrude a harp frame shape:

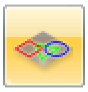
*Before...* *After...*





To create an extruded shape:

1. Choose (see page 167) or create (see page 168) the relief layer on which you want to extrude the shape.
2. Use one of the following methods to display the **Extrude** panel:
   - From the **Relief Creation** toolbar, click the **Extrude**  button;
   - From the **Menu Bar**, click the **Reliefs > Swept Shapes > Extrude** option;
   - Click the **Extrude**  button in the Swept Profiles toolset shown in the **Relief Tools** area of the **Assistant** panel.

    *For information on how to display any of the toolsets in the **Assistant** panel, see Understanding the Assistant panel (see page 36).*
3. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on.
4. Select the vector (see page 122) you want to sweep the cross section along. This is referred to as the drive curve.
5. In the **Drive Curve** area, click the **Select** button.

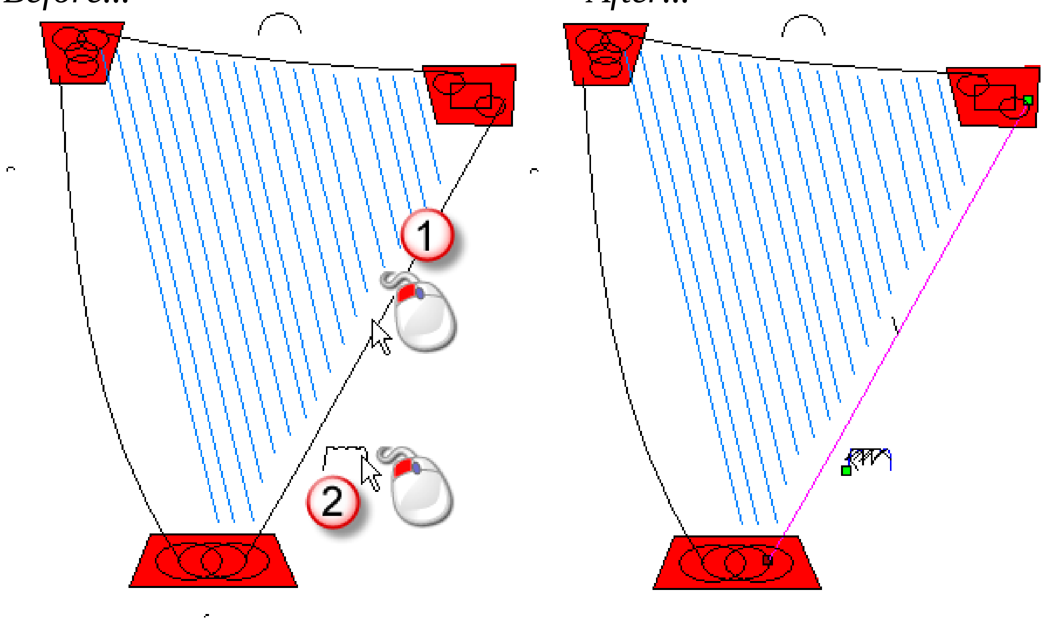
   In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector to show the direction of the drive curve and on which side the cross section is to be attached.

6. To change the drive curve's properties, select the:
    - **Reverse curve direction** check box to reverse the direction of the drive curve. The direction arrows along the selected vector now face the opposite direction.
    - **Use other side** check box to change the side of the drive curve along which the cross-section is extruded. The direction arrows along the selected vector are now displayed on the opposite side.
    - **Use as a centreline** check box to use the drive curve as the centreline for the extrusion. The direction arrows are now displayed along both sides of the selected vector.
    - **Create square corners** check box to create sharp corners in the extruded shape.
7. Select the open, ungrouped vector (see page 122) you want to use as the cross section at the start of the extruded shape. This is referred to as the start profile.
8. In the **Start Profile** area, click the **Select** button.

    In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector to show the current Z-axis direction. The side on which the direction arrows are displayed on the selected vector is the side on which the shape is extruded.

    For example, a selected drive curve and start profile in the **2D View** window might look as shown below:

*Before...* *After...*



9. To change the start profile's properties, select the:
   - **Move anchor point to other end** check box to change the start node's position in the selected vector. The start node is green and controls which end of the start profile is attached to the drive curve.
   - **Invert curve in Z** check box to invert the profile in the Z-axis direction.
10. To use a different profile at the end of the extruded shape:
    a. Select the **use separate end profile** check box.
    b. Select the open, ungrouped vector (see page 122) you want to use as the cross-section. This is referred to as the end profile.
    c. In the **End Profile** area, click the **Select** button.
11. To change the end profile's properties, select the:
    - **Move anchor point to other end** check box to change the start node's position in the selected vector. The start node is green and controls which end of the end profile is attached to the drive curve.
    - **Invert curve in Z** check box to invert the profile in the Z-axis direction.

    In our example, the vector used for the end profile is the same as that which was selected as the start profile.
12. To scale the extruded shape in the Z-axis along its length:
    a. Select the **Use a z modulation profile** check box.
    b. Select the vector (see page 122) you want to use to scale the extruded shape. This is referred to as the z modulation profile.
    c. In the **Z Modulation** area, click the **Select** button.
13. To change the z modulation profile's properties, select the:
    - **Move anchor point to other end** check box to change the position of the start node in the selected vector. The start node is green and controls which end of the start profile is attached to the z modulation vector.
    - **Invert curve in Z** check box to invert the selected vector in the Z-axis direction.
14. Select the combine mode you want to use. Click:
    - **Add** to add the extruded shape to the layer's content.

- **Subtract** to subtract the extruded shape from the layer's content.
- **Highest** to merge the extruded shape with the layer's content, so that only the highest points show.
- **Lowest** to merge the extruded shape with the layer's content, so that only the lowest points show.

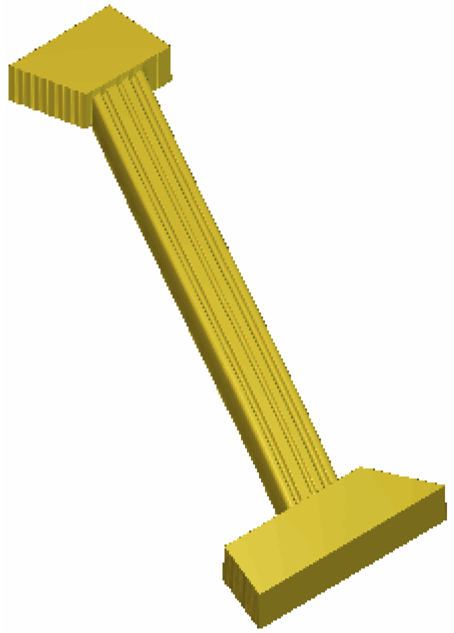
*The combine mode (see page 185) selected here controls how the extruded shape is combined with the currently active relief layer only. It does not control how the extruded shape is combined with the composite relief. How the extruded shape is combined with other relief layers to form the composite relief is affected by the combine (see page 169) mode assigned to the layer.*

15. Click **Calculate** to create the extruded shape. The extruded shape is displayed in the **3D View** window.
16. Click **Close** to close the **Extrude** panel.

*If you are working in the **2D View** window, click the **Preview Relief Layer** button in the **2D View** toolbar to display a greyscale image of the extruded shape.*

*To view only the relief layer on which you have created the extruded shape, right-click the currently active layer's icon. For details, see Viewing a layer (see page 172).*

For example, the extruded shape might look something like this when the **Add** combine mode is used:



Prior to calculating the extruded shape, the blocks at the top and bottom of the harp frame were created by applying a flat profile with a shallow height to the red bitmap colour using the **Shape Editor**.

## Creating a spun shape

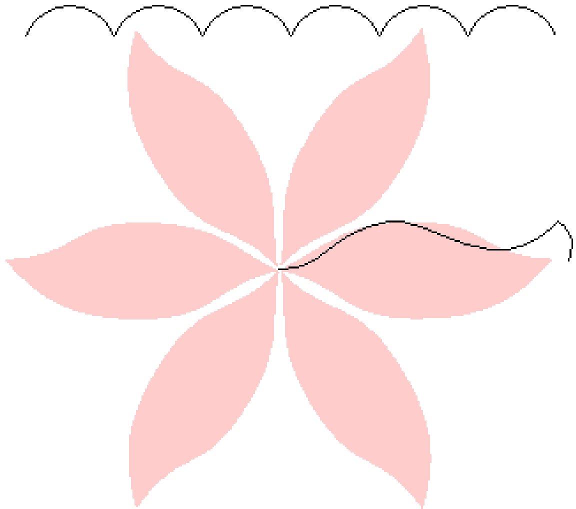
Spun shapes are created on relief layers using vector artwork. A spun shape uses at least one vector to control the shape's profile, known as the start profile.

Up to two additional vectors can be used:

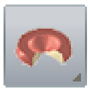
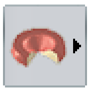
- one to control the shape's end cross section, known as the end profile; and
- another to control the shape's height in the Z direction, known as the Z modulation profile.

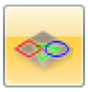
For example, the vector artwork shown below can be used to create a spun shape, and the bitmap artwork to produce leaf shapes:

*Before...* *After...*



To create a spun shape:

1. Choose (see page 167) or create (see page 168) the relief layer on which you want to create the spun shape.
2. Use one of the following methods to display the **Spin** panel:
   - From the **Relief Creation** toolbar, click the **Spin**  button;
   - From the **Menu Bar**, click **Reliefs > Swept Shapes > Spin**;
   - Click the **Spin**  button in the Swept Profiles toolset shown in the **Relief Tools** area of the **Assistant** panel.

    *For information on how to display any of the toolsets on the **Assistant** panel, see Understanding the Assistant panel (see page 36).*
3. If you are working in the **3D View** window, make sure the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on.
4. Select the open, ungrouped vector (see page 122) you want to use as the cross section at the start of the spun shape. This is referred to as the start profile.
5. In the **Start Profile** area, click the **Select** button.

In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector to show which side the cross-section is to be attached and the direction in which it is spun.

6. To change the start profile's properties, select the:
   - **Move anchor point to other end** check box to change the start node's position in the selected vector. The start node is green and controls the point around which the start profile is spun.
   - **Invert curve in Z** check box to invert the selected vector in the Z-axis direction.
7. To use a different profile at the end of the spun shape:
   a. Select the **use separate end profile** check box.
   b. Select the open, ungrouped vector (see page 122) you want to use as the cross-section. This is referred to as the end profile.
   c. In the **End Profile** area, click the **Select** button.

   In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector to show which side the cross-section is to be attached and the direction in which it is spun.

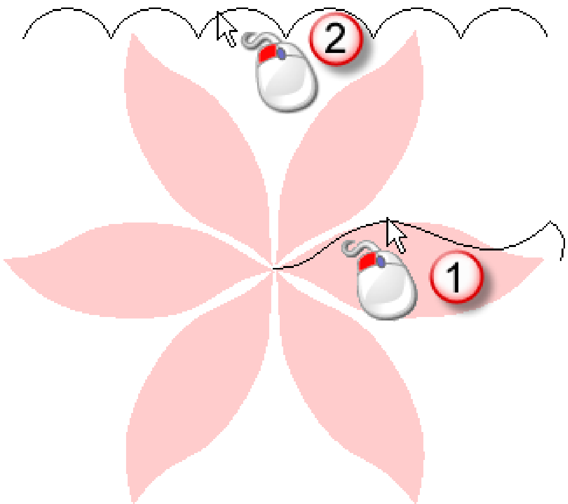
   In our example, the vector used as the end profile is the same as that which was selected as the start profile.
8. To change the end profile's properties, select the:
   - **Move anchor point to other end** check box to change the start node's position in the selected vector. The start node is green and controls the point around which the end profile is spun.
   - **Invert curve in Z** check box to invert the profile in the Z-axis direction.

   In our example, the vector used for the end profile is the same as that which was selected as the start profile.
9. To scale the spun shape in the Z-axis along its length:
   a. Select the **Use a z modulation profile** check box.
   b. Select the vector (see page 122) you want to use to scale the spun shape. This is referred to as the z modulation profile.
   c. In the **Z Modulation** area, click the **Select** button.

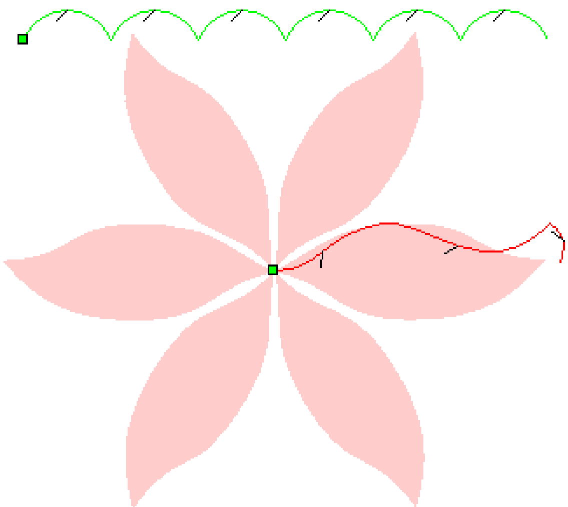
In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector to show which side the cross-section is to be attached and the direction in which it is spun.

For example, a selected start and Z modulation profile in the **2D View** window might look something like this:

*Before…*




*After…*



10. To change the z modulation profile's properties, select the:

    - **Move anchor point to other end** check box to change the start node's position in the selected vector. The start node is green and the direction of the Z modulation vector is reversed.
    - **Invert curve in Z** check box to invert the profile in the Z-axis direction.

11. To sweep the selected profiles clockwise through 360 degrees starting from a specific angle:

a. Select the **Sweep through 360 degrees** check box.

b. In the **Start Angle** box, specify the angle at which to begin sweeping.

12. To sweep the profile through a specific angle and in a specific direction:

    a. Deselect the **Sweep through 360 degrees** check box.

    b. In the **Start Angle** box, specify the angle at which to begin sweeping.

    c. In the **End Angle** box, specify the angle at which to stop sweeping.

    d. Specify the sweep direction by clicking the **Clockwise** or **Anticlockwise** option.

13. Select the combine mode you want to use. Click:

    - **Add** to add the spun shape to the layer's content.
    - **Subtract** to subtract the spun shape from the layer's content.
    - **Highest** to merge the spun shape with the layer's content, so that only the highest points show.
    - **Lowest** to merge the spun shape with the layer's content, so that only the lowest points show.

    *The combine mode (see page 185) selected here controls how the spun shape is combined with the currently active relief layer only. It does not control how the spun shape is combined with the composite relief. How the spun shape is combined with other relief layers to form the composite relief is affected by the combine mode (see page 169) assigned to the layer.*

14. Click **Calculate** to create the spun shape. The spun shape is displayed in the **3D View** window.

15. Click **Close** to close the **Spin** panel.

For example, a profile swept from 0 through 360 degrees in a clockwise direction might look something like this when the **Add** combine mode is used:

The wave effect is achieved by using the Z modulation vector. The overall shape of the leaves is achieved by resetting the relief outside of the leaf-shaped bitmap artwork to zero.

*If you are working in the **2D View** window, click the **Preview Relief Layer** button in the **2D View** toolbar to display a greyscale image of the spun shape.*

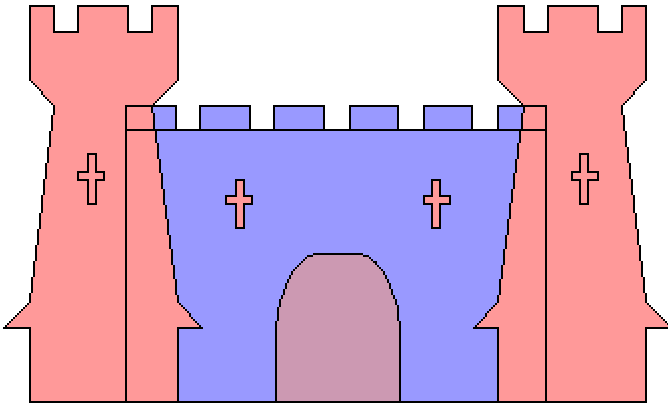
*To view only the relief layer on which you have created the spun shape, right-click the currently active layer's icon. For details, see Viewing a layer* *(see page 172).*

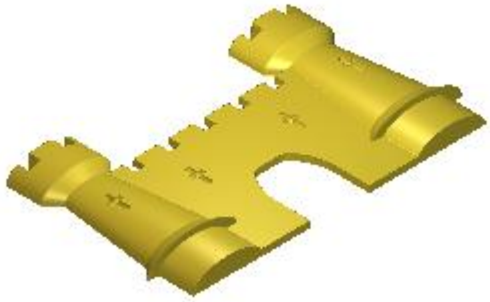
## Creating a turned shape

You can turn a shape using a vector. An imaginary line between the start node and end node in the selected vector acts as an axis about which it is turned to create the shape's cross section.

For example, you can see how a shape can be turned using vectors to form castle turrets:

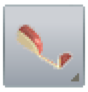
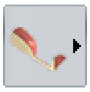
*Before...* *After...*



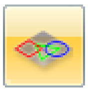


To create a turned shape:

1. Choose (see page 167) or create (see page 168) the relief layer on which you want to create the turned shape.
2. Use one of the following methods to display the **Turn** panel:

   - From the **Relief Creation** toolbar, click the **Turn**  button;
   - From the **Menu Bar**, click **Reliefs > Swept Shapes > Turn**;
   - Click the **Turn**  button in the Swept Profiles toolset shown in the **Relief Tools** area of the **Assistant** panel.

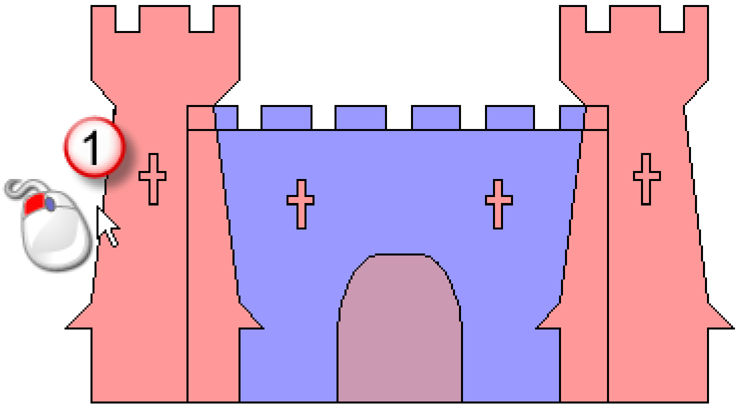
   *For information on how to display any of the toolsets on the* ***Assistant*** *panel, see Understanding the Assistant panel (see page 36).*

3. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on.
4. Select the open, ungrouped vector (see page 122) you want to use as the turned shape's cross section. This is referred to as the profile.
5. In the **Profile** area, click the **Select** button.

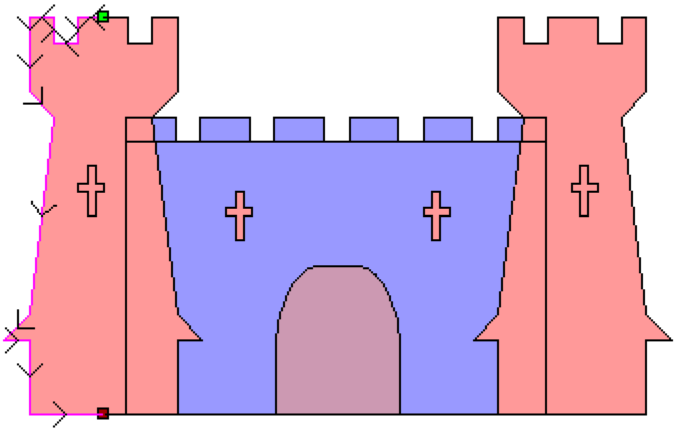
   In the **2D View** and **3D View** window, direction arrows are displayed about the selected vector to show the direction from the start node to the end node.

For example, a selected profile in the **2D View** window might look something like this:

*Before…*



*After...*



6. In the **Z Scale Factor** box, specify the scaling factor that you want to apply to the profile in the Z-axis direction. The default value of 1 produces a semi-circular cross section.

   In our example, a scale factor of 0.5 is applied to the profile.

7. To change any of the settings before calculating the turned shape, click **Reset**.

8. Select the combine mode you want to use. Click:

   - **Add** to add the turned shape to the layer's content.
   - **Subtract** to subtract the turned shape from the layer's content.
   - **Highest** to merge the turned shape with the layer's content, so that only the highest points show.
   - **Lowest** to merge the turned shape with the layer's content, so that only the lowest points show.

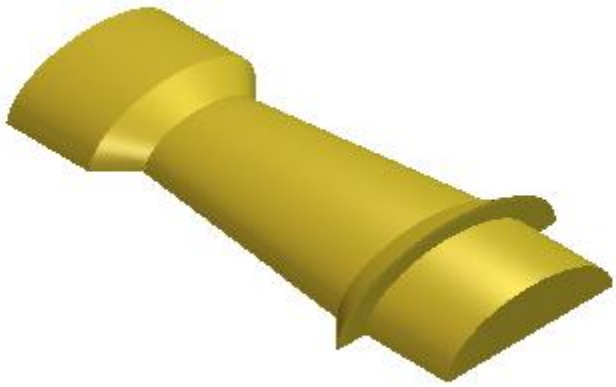
*The combine mode (see page 185) selected here controls how the turned shape is combined with the currently active relief layer only. It does not control how the turned shape is combined with the composite relief. How the turned shape is combined with other relief layers to form the composite relief is affected by the combine mode (see page 169) assigned to the layer.*

9. Click **Calculate** to create the turned shape. The turned shape is displayed in the **3D View** window.

10. Click **Close** to close the **Turn** panel.

    In our example, the **Add** option is used.

    In our example, the currently active relief layer is displayed in the **3D View** window as follows:



The crenellations within the castle turrets are created by selecting white as the Primary Colour, and then using the **Zero Relief Under Colour** button.

*If you are working in the **2D View** window, click the **Preview Relief Layer** button in the **2D View** toolbar to display a greyscale image of the turned shape.*

*To view only the relief layer on which you have created the turned shape, right-click the currently active layer's icon. For details, see Viewing a layer (see page 172).*

## Creating a two-rail sweep

Swept shapes are created on relief layers using vector artwork. A two-rail sweep uses at least three open, ungrouped vectors:

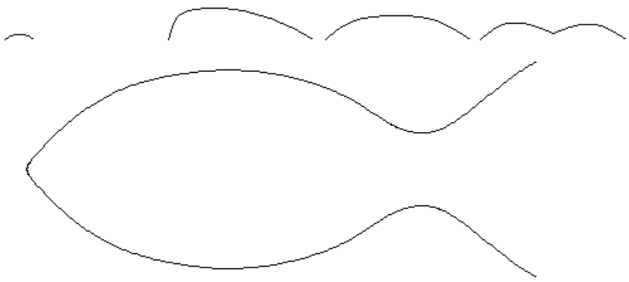
- two to set the boundary of the sweep, known as drive rails; and

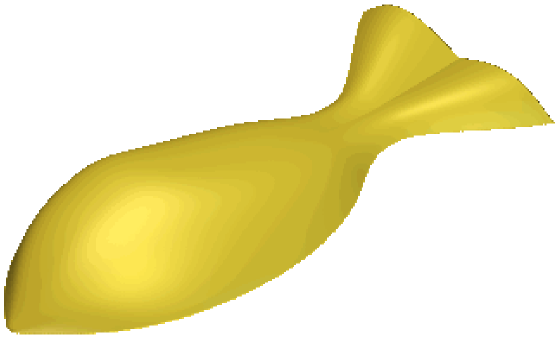
- one or more to control the sweep's height and profile, known as cross sections.

When you create a two-rail sweep, an additional vector to restrict the shape's height can also be used.

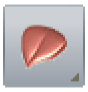
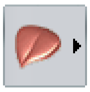
For example, the vector artwork shown below can be used to create a sweep in the shape of a fish:

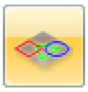
*Before...* *After...*





To create a two-rail sweep:

1. Choose (see page 167) or create (see page 168) the relief layer on which you want to create the sweep.
2. Use one of the following methods to display the **Two Rail Sweep** panel:
   - From the **Relief Creation** toolbar, click the **Two Rail Sweep**  button;
   - From the **Menu Bar**, click **Reliefs > Swept Shapes > Two Rail Sweep**;
   - Click the **Two Rail Sweep**  button in the Swept Profiles toolset shown in the **Relief Tools** area of the **Assistant** panel.

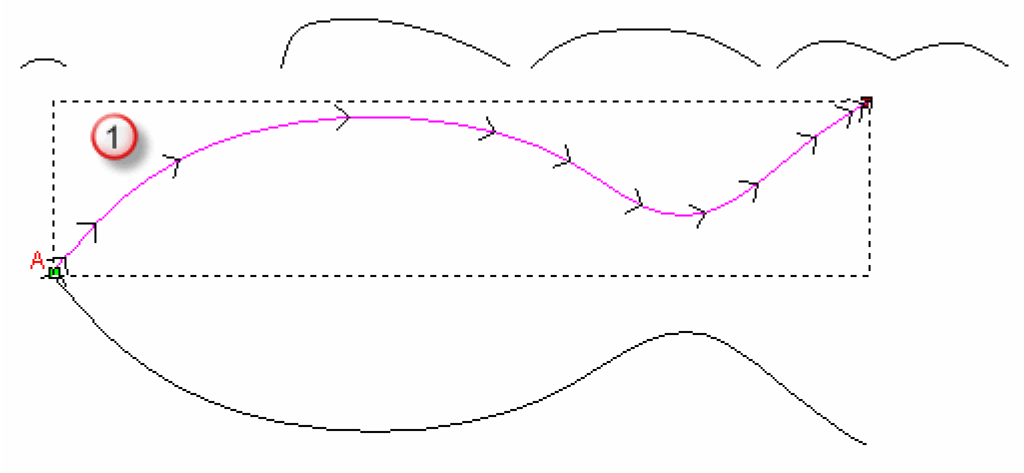
    *For information on how to display any of the toolsets on the* ***Assistant*** *panel, see Understanding the Assistant panel* *(see page 36)**.*
3. Choose the two drive rails:
   a. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility**  button in the **3D View** toolbar is toggled on.

b. Select the first vector (see page 122) you want to sweep cross sections along.

c. In the **Select Control Vectors** area, click the **Top Drive Rail**'s **Select** button.

   In the **Status** area, the **First Drive Rail** changes from *Not Selected* to *Valid*. In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector. In the **2D View** only, a red letter *A* is also shown beside it's start node.

   For example, the first drive rail might look as shown below:

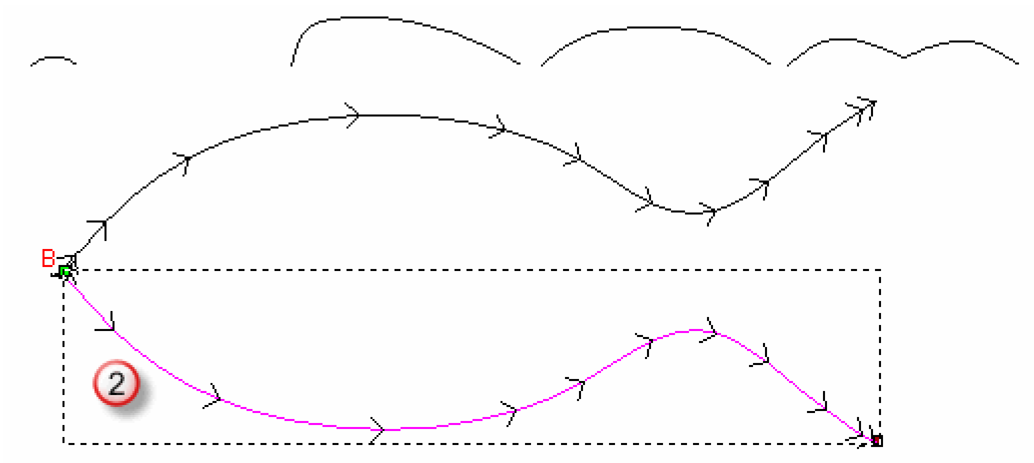



d. Select the second vector (see page 122) you want to sweep cross sections along.

e. In the **Select Control Vectors** area, click the **Bottom Drive Rail**'s **Select** button.

   In the **Status** area, the **Second Drive Rail** changes from *Not Selected* to *Valid*. In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector. In the **2D View** only, a red letter *B* is also shown beside it's start node.

   For example, the second drive rail might look as shown below:



4. Make sure that the drive rails share the same direction. To reverse the direction of:
   - the first drive rail, click to select the **First** check box in the **Reverse Direction of Drive Rails** area.
   - the second drive rail, select the **Second** check box.
5. Choose the cross-section vectors:
   a. Select the open, ungrouped vectors (see page 122) you want to use as cross sections.

   When selecting two or more cross sections, each of the selected vectors ideally should contain the same number of spans. This helps to ensure a smooth transition is made between each of the shape's cross sections. The order in which you select the cross sections can affect the shape.

   *When using the* ***Select*** *tool , the number of spans in a selected vector are displayed on the* ***Tool Settings*** *panel.*

   a. In the **Status** area, click the **Add Cross Section** button.

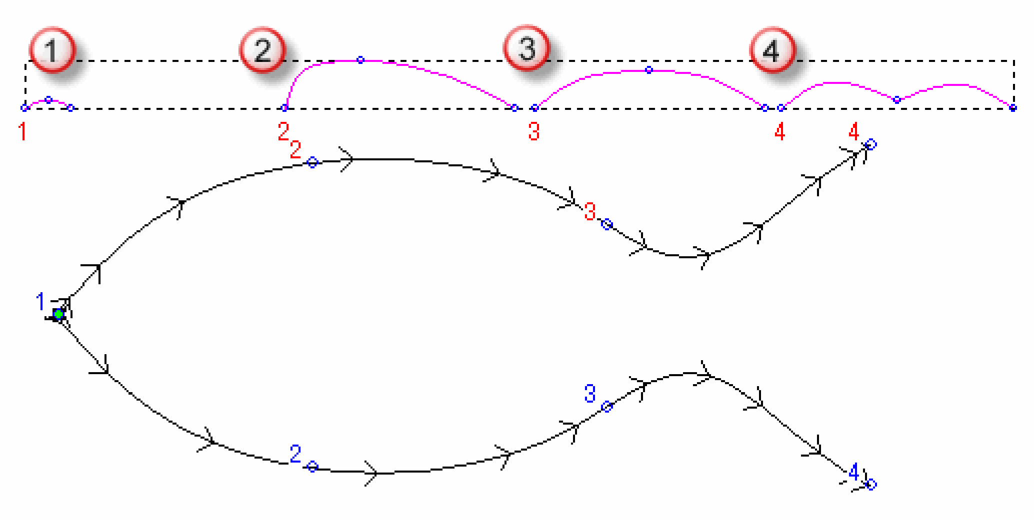
   In the **Cross Section** area, a list of cross sections is shown. Each of the cross-sections has a:

   - number;
   - *Valid* status;
   - **Smooth Blend** icon;
   - **Set Position** icon;
   - **Delete** × icon; and
   - displays its number of spans.

   In the **2D View** window only, red sequential numbers are displayed:

   - on the vectors selected as cross-sections;
   - along the first drive rail to mark the position of the cross sections; and
   - blue sequential numbers are displayed along the second drive rail to mark the position of the cross sections.

For example, the cross sections might look as shown below:




Provided the vectors selected as cross sections contain an equal number of spans, the **Sweep between spans** check box is enabled and selected. The spans and nodes in each subsequent cross section along the drive rails are paired, and the sweep is extruded between them.

When the vectors selected as cross-sections do not contain an equal number of spans, or you deselect the **Sweep between spans** check box, although the sweep is extruded between each subsequent cross section along the drive rails, their spans and nodes are not used.

6. To insert a cross section:

   a. Select the open, ungrouped vector (see page 122) you want to insert as a cross section.

   b. In the **Status** area, click the **Insert Cross Section** button.

      In the **2D View** window, a red sequential number is displayed on the selected vector.

      In the **Cross Sections** area, a new cross section is added to the list with **1st Rail** displayed.

   c. In the **2D View** window, move the + cursor over the position in the first drive rail in which you want to insert the cross section, then click.

      The cross section's **1st Rail** label is replaced with **2nd Rail**.

   d. Move the + cursor over the position in the second drive rail, then click to complete the cross section.

The cross section's **2nd Rail** label is replaced with **Set Position**.

7. To reposition a cross section on the drive rails:

   a. Click to toggle on. **1st Rail** replaces its **Position** label.

   b. On the first drive rail, click the cross section's new position. **2nd Rail** replaces its **1st Rail** label.

   c. On the second drive rail, click the cross position's new position. is displayed, and **Position** replaces its **2nd Rail** label.

8. Specify how the shape is blended between each cross-section. Click:

   - to toggle on and use a linear blend; or
   - to toggle on and use a smooth blend.

9. To remove a cross-section from the list, click . The cross section vector is deselected, and the numbers are updated:

   - in the **Cross Sections** area; and
   - on the cross sections and drive rails shown in the **2D View** window.

10. Choose the scaling options.

    To use a vector to scale the sweep in the Z-axis along its length:

    a. Select the open, ungrouped vector (see page 122) you want to use to control the Z height; known as a Z control vector.

    b. In the **Select Control Vectors** area, click the **Z Control Vector**'s **Select** button.

    In the **Status** area, the **Z Control Vector** changes from *Not Selected* to *Valid*. In the **2D View** only, a red letter *Z* is shown beside it's start node.

    c. To match the height of each cross section with the Z control vector, select the **Vector Controls Exact Height** check box.

    *The **Scale Height with Width** and **Scale Final Height** check boxes are greyed-out.*

    If you are not using a vector to control the sweep's height, you can choose one or more of the following scaling options instead:

    - To scale the sweep's height proportionally to the width of each cross section across the drive rails, leave the **Scale Height with Width** check box selected.

With **Scale Height with Width** selected, narrow cross sections produce low heights in the sweep, and wide cross sections produce high heights.

- To keep a constant height throughout the sweep, deselect the **Scale Height with Width** check box.
- To scale the height of the sweep during the calculation process, select the **Scale Final Height** check box, then specify the maximum Z height in the adjacent box.

11. To create a flat base beneath the sweep, specify its Z height in the **Start Height** box.
12. Select the combine mode you want to use. Click:
    - **Add** to add the sweep to the layer's content.
    - **Subtract** to subtract the sweep from the layer's content.
    - **Highest** to merge the sweep with the layer's content, so that only the highest points show.
    - **Lowest** to merge the sweep with the layer's content, so that only the lowest points show.

*The combine mode (see page 185) selected here controls how the sweep is combined with the currently active relief layer only. It does not control how the sweep is combined with the composite relief. How the sweep is combined with other relief layers to form the composite relief is affected by the combine mode (see page 169) assigned to the layer.*

13. Click **Calculate** to create the sweep. The sweep is displayed in the **3D View** window.
14. Click **Close** to close the **Two Rail Sweep** panel.

*If you are working in the **2D View** window, click the **Preview Relief Layer** button in the **2D View** toolbar to display a greyscale image of the sweep.*

*To view only the relief layer on which you have created the sweep, right-click the currently active layer's icon. For details, see Viewing a layer (see page 172).*

## Creating a two-rail ring sweep

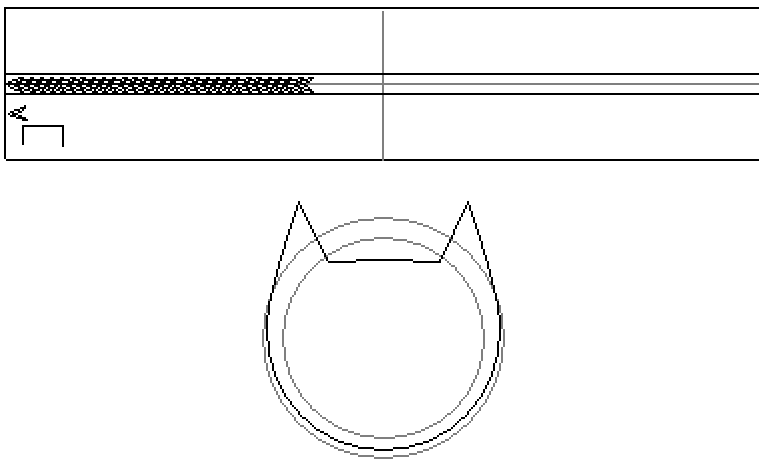
Swept ring shapes are created on relief layers using vector artwork, and are typically used to create or edit a shank. These can only be created in ArtCAM JewelSmith where a model containing a rotary relief is currently open.

A two-rail ring sweep uses at least three open, ungrouped vectors and one closed, ungrouped vector:

- two open, ungrouped vectors to set the boundary of the sweep, known as drive rails;
- one closed, ungrouped vector to control the sweep's height, known as the ring silhouette; and
- one or more open, ungrouped vectors to control the sweep's height and profile, known as cross sections.

For example, the vector artwork shown below can be used to create a rotary axis shank:

*Before…* *After…*



To create a two-rail ring sweep:

1. Choose (see page 167) or create (see page 168) the relief layer on which you want to create the sweep.
2. Use one of the following methods to display the **Two Rail Ring Sweep** panel:
   - In the **Rotary Relief Tools** toolbar, click the **Two Rail Sweep - Ring**  button;

*The **Rotary Relief Tools** toolbar is hidden by default. To display the toolbar, right-click a docking area, then select **Rotary Relief Tools** from the context menu.*

- Click the **Two Rail Sweep - Ring** button in the **Rotary Relief Tools** area of the **Assistant** panel.

*The **Assistant** panel is hidden by default. To display the panel, right-click a docking area, then select **Assistant** from the context menu.*

3. Choose the two drive rails:

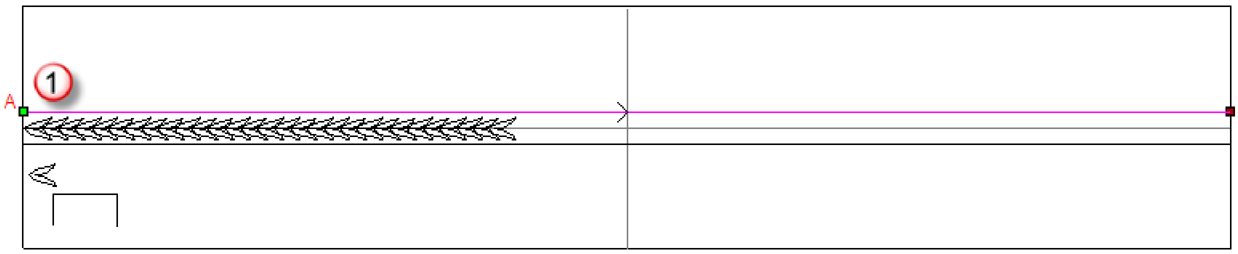
   a. If you are working in the **3D View** window, make sure that the **Toggle Vector Visibility** button in the **3D View** toolbar is toggled on.

   b. Select the first vector (see page 122) you want to sweep cross sections along.

   c. In the **Select Control Vectors** area, click the **Top Drive Rail**'s **Select** button.

   In the **Status** area, the **Top Drive Rail** changes from *Not Selected* to *Valid*. In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector. In the **2D View** only, a red letter *A* is also shown beside it's start node.

   For example, the top drive rail might look as shown below:

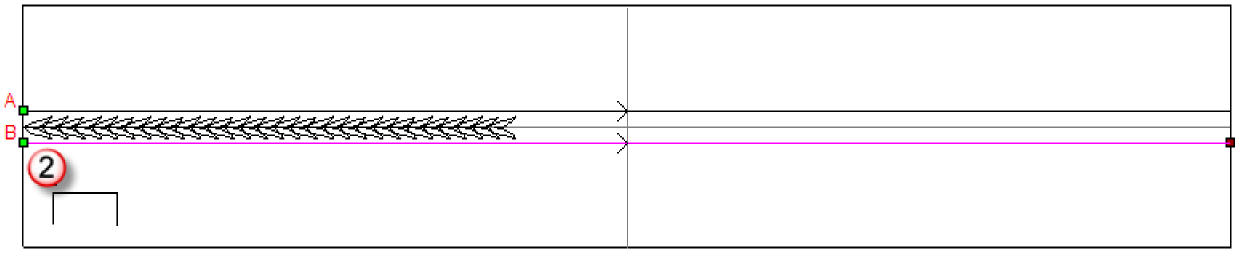



   d. Select the second vector (see page 122) you want to sweep cross sections along.

   e. In the **Select Control Vectors** area, click the **Bottom Drive Rail**'s **Select** button.

   In the **Status** area, the **Bottom Drive Rail** changes from *Not Selected* to *Valid*. In the **2D View** and **3D View** window, direction arrows are displayed along the selected vector. In the **2D View** only, a red letter *B* is also shown beside it's start node.

For example, the bottom drive rail might look as shown below:




4. Make sure that the drive rails share the same direction. To reverse the direction of:
   - the top drive rail, click to select the **First** check box in the **Reverse Direction of Drive Rails** area.
   - the bottom drive rail, select the **Second** check box.
5. Choose the cross section vectors:
   a. Select the open, ungrouped vectors (see page 122) you want to use as cross sections.

   When selecting two or more cross sections, each of the selected vectors ideally should contain the same number of spans. This helps to ensure a smooth transition is made between each of the sweep's cross sections. The order in which you select the cross sections can affect the shape.

   *When using the* ***Select*** *tool , the number of spans in a selected vector are displayed on the* ***Tool Settings*** *panel.*

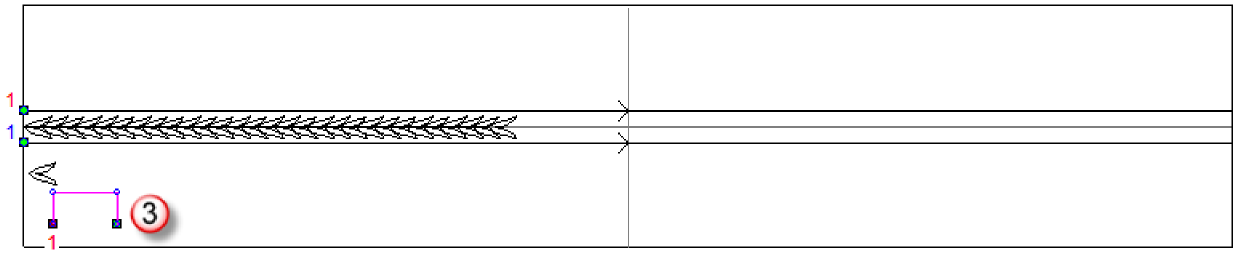
   b. In the **Status** area, click the **Add Cross Section** button.

   In the **Cross Section** area, a list of cross sections is shown. Each of the cross sections has a:
   - number;
   - *Valid* status;
   - **Set Position** icon;
   - **Smooth Blend** icon;
   - **Delete** × icon; and
   - displays its number of spans.

   In the **2D View** window only, red sequential numbers are displayed:
   - on the vectors selected as cross sections;

- along the first drive rail to mark the position of the cross sections; and
- blue sequential numbers are displayed along the second drive rail to mark the position of the cross sections.

For example, the cross section might look as shown below:




Provided the vectors selected as cross sections contain an equal number of spans, the **Sweep between spans** check box is enabled and selected. The spans and nodes in each subsequent cross section along the drive rails are paired, and the sweep is extruded between them.

When the vectors selected as cross-sections do not contain an equal number of spans, or you deselect the **Sweep between spans** check box, although the sweep is extruded between each subsequent cross section along the drive rails, their spans and nodes are not used.

The **Ring is symmetrical** check box is selected by default. This positions the first cross-section across the start nodes in the selected drive rails, the last cross section where the drive rails intersect the vertical reference guideline vector, with any intermediate cross sections in between. This ensures a symmetrical sweep is created.

When deselected, each cross section is positioned along the length of the top and bottom drive rails, with the first across the start nodes and the last across the end nodes. This can result in an asymmetrical sweep being created.

6. To insert a cross section:

   a. Select the open, ungrouped vector (see page 122) you want to insert as a cross section.

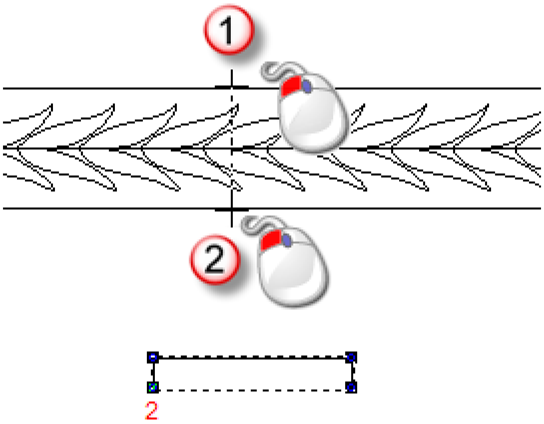
   b. In the **Status** area, click the **Insert Cross Section** button.

      In the **2D View** window, a red sequential number is displayed by the selected vector's start node. In the **Cross Section** area, a new cross section is added to the list with its **1st Rail** label displayed.

c. In the **2D View** window, move the cursor over the position in the top drive rail in which you want to insert the cross section, then click. The cross section's **1st Rail** label is replaced with **2nd Rail**.

d. Move the cursor over the position in the bottom drive rail, then click to complete the cross section. The cross section's **2nd Rail** label is replaced with **Set Position**.

For example, the cross section could be inserted as follows:




7. To reposition a cross-section on the drive rails:

   a. In **Cross Section** area, click the cross section's **Set Position** icon. The **1st Rail** label is displayed.

   b. In the **2D View** window, click the cross section's new position on the top drive rail. The **1st Rail** label is replaced with **2nd Rail**.

   c. Click the cross section's new position on the bottom drive rail. The **2nd Rail** label is replaced with **Set Position**.

8. Specify how the shape is blended between each cross-section. Click:

   - to toggle on and use a linear blend; or
   - to toggle on and use a smooth blend.

9. To remove a cross section from the list, click . The cross section vector is deselected, and the numbers are updated:

   - in the **Cross Sections** area; and
   - on the cross sections and drive rails shown in the **2D View** window.

10. Choose the scaling options.

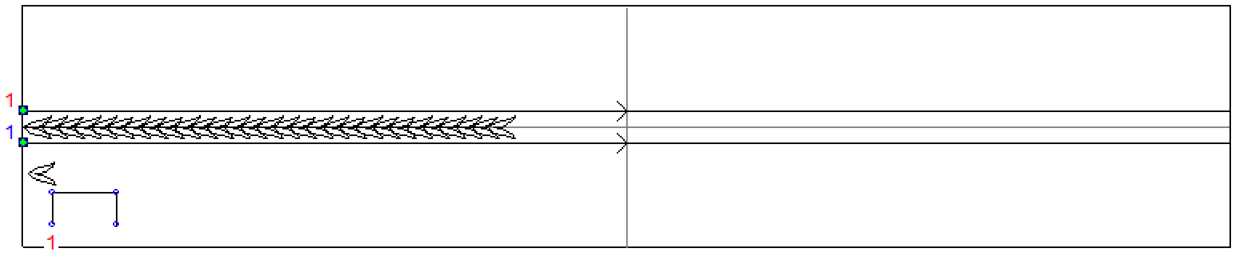
To use the ring silhouette to scale the shape in the Z-axis along its length:

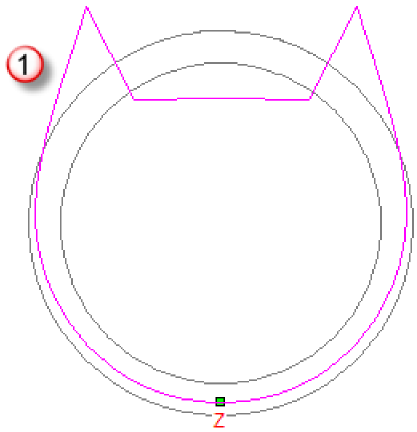
a. Select the closed, ungrouped vector (see page 122) you want to use to control the Z height; known as a ring silhouette vector.

b. In the **Select Control Vectors** area, click the **Ring Silhouette**'s **Select** button.

   In the **Status** area, the **Ring Silhouette** changes from *Not Selected* to *Valid*. In the **2D View** only, a red letter *Z* is shown beside it's start node.

   For example, the ring silhouette vector might look as shown below:




c. To match the height of each cross section with the ring silhouette, leave the **Silhouette Vector Controls Z** check box selected.

*The **Scale Height with Width** and **Scale Final Height** check boxes are greyed-out.*

If you are not using the ring silhouette vector to control the sweep's height, deselect the **Silhouette Vector Controls Z** check box, then select one or more of the following scaling options instead:

- To scale the sweep's height proportionally to the width of each cross section across the drive rails, leave the **Scale Height with Width** check box selected.

  With **Scale Height with Width** selected, narrow cross sections produce low heights in the sweep, and wide cross sections produce high heights.

- To keep a constant height throughout the sweep, deselect the **Scale Height with Width** check box.
- To scale the height of the sweep during the calculation process, select the **Scale Final Height** check box, then specify the maximum Z height in the adjacent box.

11. To create a flat base beneath the sweep, specify its Z height in the **Start Height** box.
12. Select the combine mode you want to use. Click:
    - **Add** to add the sweep to the layer's content.
    - **Subtract** to subtract the sweep from the layer's content.
    - **Highest** to merge the sweep with the layer's content, so that only the highest points show.
    - **Lowest** to merge the sweep with the layer's content, so that only the lowest points show.

*The combine mode (see page 185) selected here controls how the sweep is combined with the currently active relief layer only. It does not control how the sweep is combined with the composite relief. How the sweep is combined with other relief layers to form the composite relief is affected by the combine mode (see page 169) assigned to the layer.*

13. Click **Calculate** to create the sweep. The sweep is displayed in the **3D View** window.
14. Click **Close** to close the **Two Rail Ring Sweep** panel.

*If you are working in the **2D View** window, click the **Preview Relief Layer** button in the **2D View** toolbar to display a greyscale image of the unwrapped sweep.*

*To view only the relief layer on which you have created the sweep, right-click the currently active layer's icon. For details, see Viewing a layer (see page 172).*

# Creating an Embossed Relief

To create an embossed relief, you must be working in an open project with at least one visible replica mesh included in the Project Tree. These replica meshes can be created:

- from a triangle mesh of the composite relief;
- by importing compatible triangle or surface models (see page 226); or
- by importing components (see page 238) from the **Component Library**.

*If you are working in ArtCAM Insignia, you cannot create an embossed relief.*

Using the visible replica meshes in the Project Tree, the **Emboss Relief Wizard** produces a bas-relief, or low relief, which, despite having a particularly shallow depth, preserves:

- the detail and illusion of depth that would be lost through conventional scaling; and
- perspective in the direction of embossing.

Reproducing the results of the **Emboss Relief Wizard** using relief creation, editing and sculpting tools would be extremely difficult and time consuming, if not impossible.

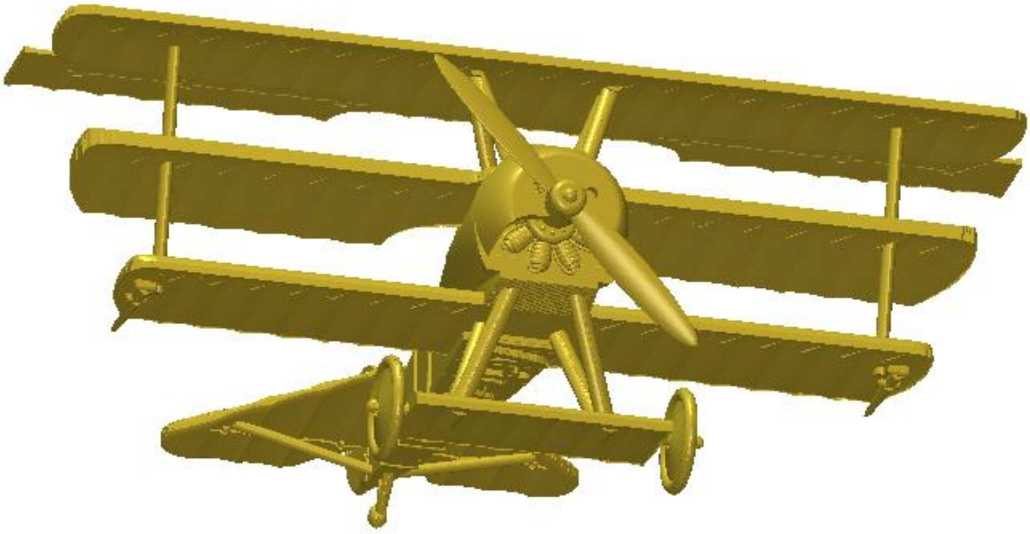
To create an embossed relief:

1. From the **Project** panel, select one of the following items in the Project Tree:
   - root **Assembly**;
   - assembly ; or
   - replica mesh .

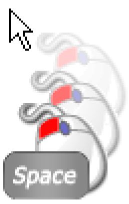
   The selected item's name is highlighted, and is also displayed on the splitter bar.
2. All of the visible replica meshes associated with your selected item in the Project Tree are included in the embossed relief. Make sure that only the items you want to use are visible in the **3D View** window.

*To toggle the display of an assembly or replica mesh in the Project Tree, right-click either the assembly or replica mesh, then click **Show** or **Hide** in the context menu, depending on whether it is currently visible or not.*

For example, a replica mesh created from an imported triangle model of a triplane is shown in the **3D View** window as follows:



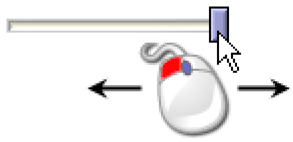
3. From the **Project** panel, click the **Emboss Relief Wizard** button in the **Tools** area below the splitter bar. The **Select View Direction** settings are displayed.
4. Set the viewing angle you want to use for the embossed relief:
   a. In the **3D View** window, position the mouse cursor over one of the replica meshes displayed.
   b. Hold down the **Space Bar**, then click and drag.

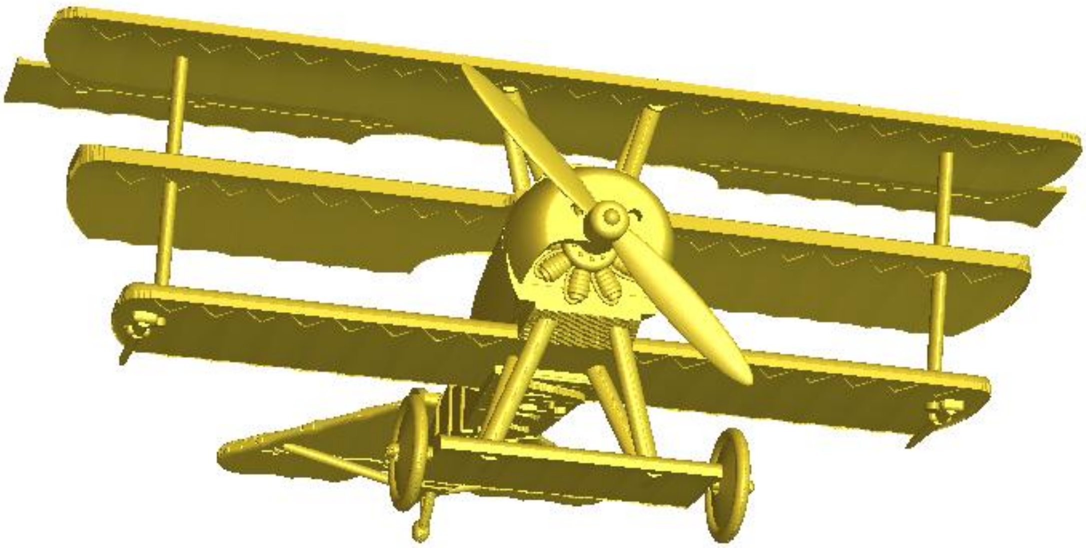
   c. Release the mouse button and **Space Bar** to set the viewing angle.
5. Click **Next**. The **Apply Perspective** settings are displayed.

   If any gems are associated with the selected item in the Project Tree, these are now hidden in the **3D View** window. Gems cannot be included in the embossed relief.

6. Click and drag the **Apply Perspective** slider to set the perspective factor you want to apply to the Z-axis of all replica meshes shown in the **3D View** window.



For example, by moving the slider to its middle position, the illusion of depth is noticeable in the triplane's tail section:



7. Click **Next >**. The **Select Detail** settings are displayed.

In the Project Tree, a new model is created beneath **Models**. The model is open, and given the name of the item selected in the Project Tree. For example, *TriPlane*. The embossed relief is eventually created in this model.

8. In the **Detail Height** box, specify the maximum Z height of the detail, edges and features from the visible items in the Project Tree you want to include in the embossed relief.

This setting enables the embossed relief's shape to vary in order to create the illusion of depth.

9. In the **Shape Height** box, specify how much of the Z height from the visible items in the Project Tree you want to use in the embossed relief.

This setting produces the same result as scaling the items in the Project Tree; preserving their original shape in the new relief.

10. Click **Finish** to create the embossed relief.

During the calculation process, a:

- progress bar is displayed in the **Status Bar** area; and
- a relief layer named *Temp* is created in the active relief layer stack.

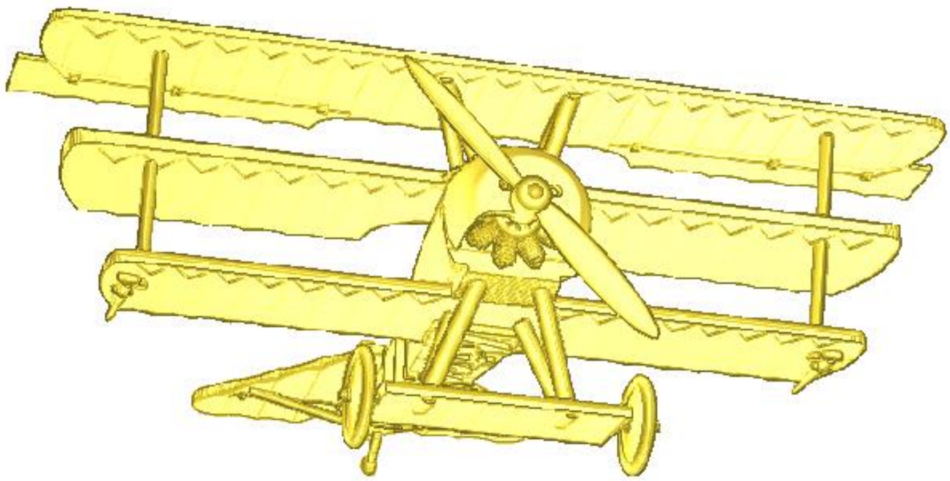
When the calculation process is complete, the:

- *Temp* relief layer is deleted from the layer stack.
- embossed relief is created on a new relief layer, which is selected and given the name of the currently selected item in the Project Tree.
- embossed relief is shown in the **3D View** window looking down the Z-axis.

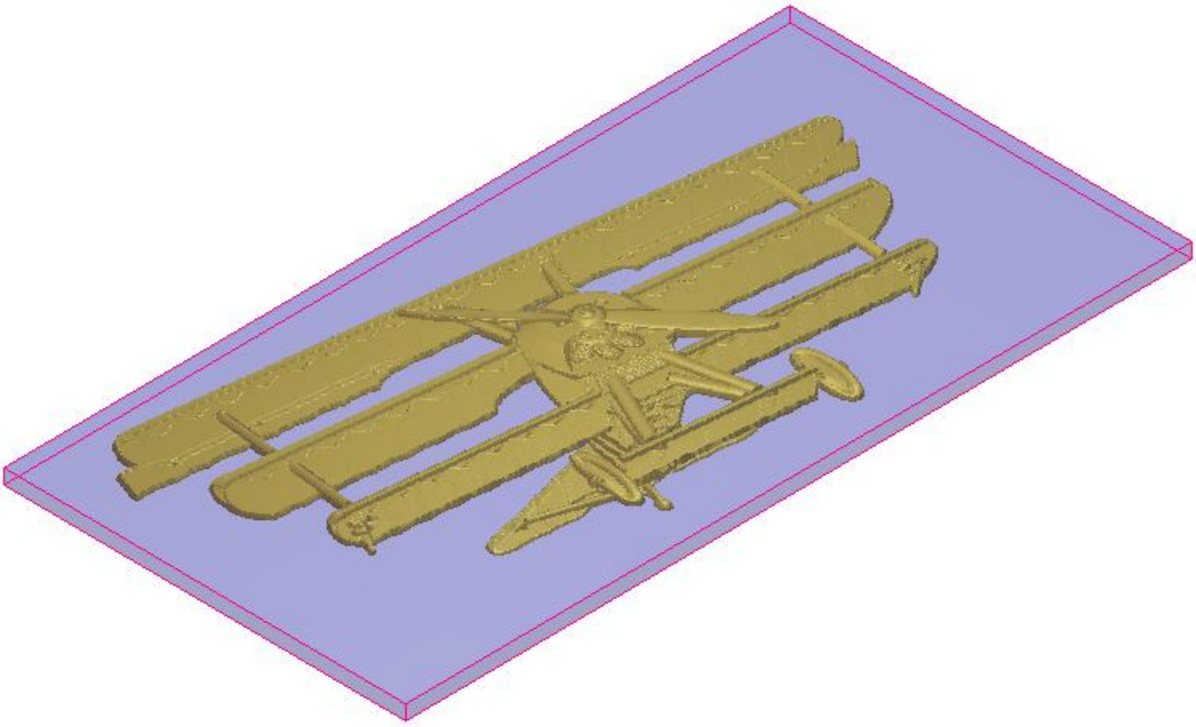
The embossed relief includes the original detail in the replica meshes and your chosen viewing angle.

11. In the **3D View** toolbar, click to deselect the **Toggle Assembly Visibility**  button and the **Draw Zero Plane**  button. This leaves only the embossed relief displayed in the **3D View** window.

For example, the embossed relief created from the triplane replica mesh is shown as follows:



The maximum Z height of this embossed relief is *0.529* mm, and is best illustrated displayed in a material block with a thickness of only *1* mm:



# Importing a triangle or surface model

You can import a triangle or surface model into an open ArtCAM model or project. When importing into an open model, the 3D model is pasted onto the currently active relief layer. When importing into an open project, the 3D model is added to the Project Tree as a replica mesh or an assembly; either of which can be used to create a new model containing:

- a relief;
- a rotary relief; or
- an embossed relief (see page 222).

*If you are working in ArtCAM Insignia, you can only import a triangle or surface model into an open ArtCAM model. The 3D model is combined with the relief.*

The 3D models you can import are as follows:

### Triangle Models

- ArtCAM Assembly (***.3da**);
- ArtCAM Project (***.3dp**);
- 3D Studio (***.3ds**);
- Drawing Interchange File (***.dxf**);
- Binary or ASCII STL (***.stl**);

- Universal 3D File (***.u3d**);
- Wavefront Object File (***.obj**); and
- Delcam Machining Triangles (***.dmt**)

**Surface Models**

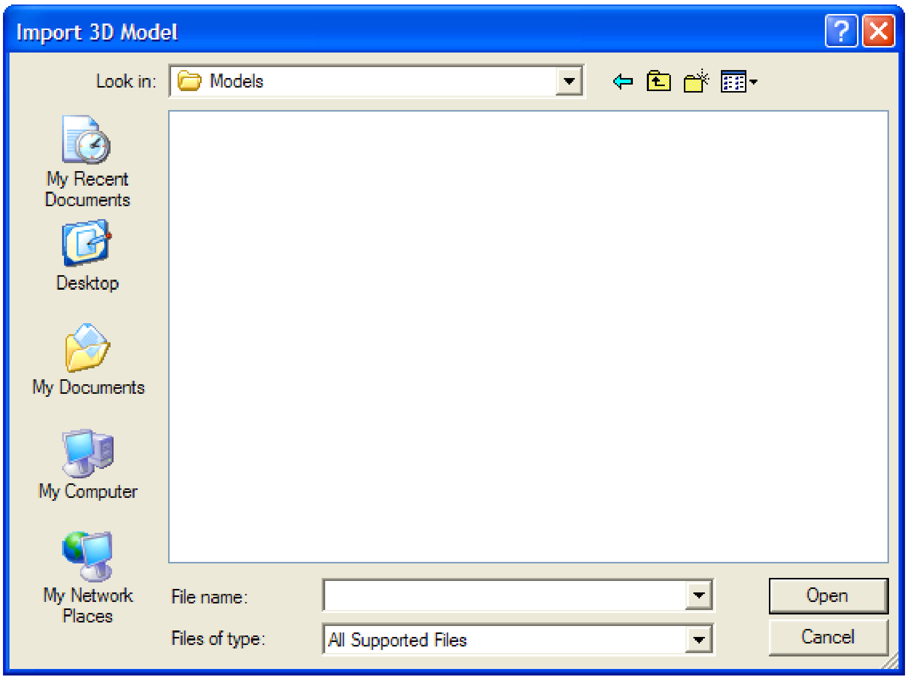
- 3D NURBS Modeller - Rhinoceros (***.3dm**);
- Delcam DGK (***.dgk**);
- SolidWorks Part File (***.sldprt**);
- IGES Format (***.igs**, ***.ige** and ***.iges**);
- ACIS File (***.sat**);
- CATIA File (***.fic**);
- CATIA5 (***.catpart** and ***.catproduct**);
- Cimatron File (***.pfm**);
- Elite File (***.elt**);
- Ideas File (***.mf1** and ***.prt**);
- Inventor File (***.ipt**);
- Parasolid Files (***.x_t**, ***.xmt_txt**, ***.x_b** and ***.xmt_bin**);
- Parts File (***.psmodel**);
- DDX Files (***.ddx** and ***.ddz**);
- Pro/Engineer Files (***.asm** and ***.par**);
- SpaceClaim File (***.scdoc**);
- Step Files (***.stp** and ***.step**);
- Unigraphics File (***.prt**); and
- VDAFS File (***.vda**)

### *Importing into a model*

To import a triangle or surface model into an open model:

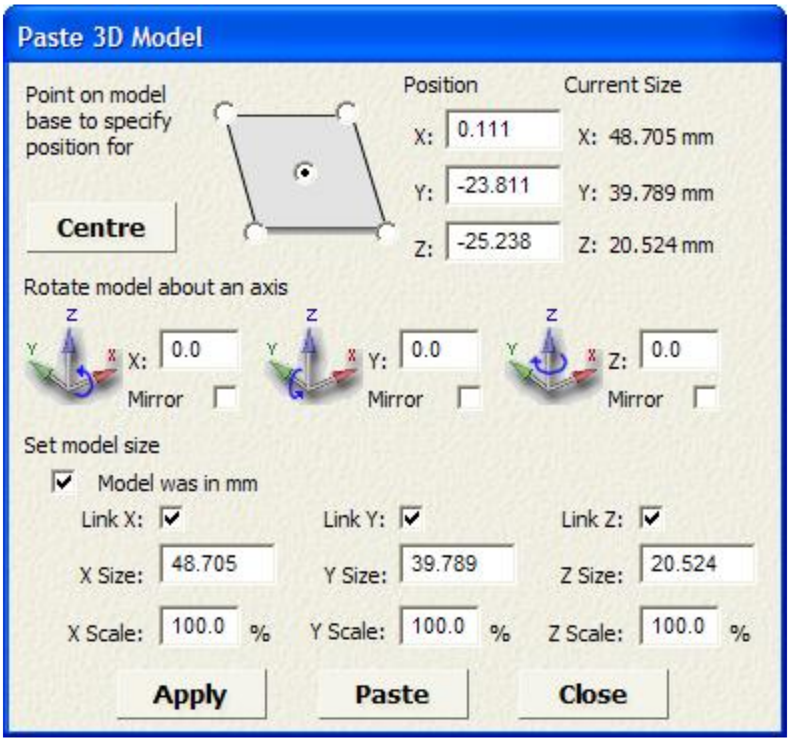
1. Choose the relief layer (see page 167) onto which you want to import the triangle or surface model.

2. From the **Menu Bar**, click **Reliefs > Import 3D Model**. The **Import 3D Model** dialog box is displayed:




3. Click the **Look in** list box, and then select the folder on your computer containing the triangle or surface model you want to use.
4. Select the file you want to import, and then click **Open**. During the import process:
   - a progress bar is displayed in the **Status Bar**;
   - the **Import 3D Model** dialog is closed;
   - the imported model is displayed in the **3D View** window; and

- the **Paste 3D Model** dialog box is displayed.




In the **Current Size** area, the imported model's dimensions are shown. Whether these are metric or imperial measurements depends on the units you are using in your ArtCAM model.

5. If you are working in the **2D View** window, press the **F3** key to display the imported model in the **3D View** window. Although the imported model is displayed, it is not yet part of your ArtCAM model.

6. Set the imported model's position in the ArtCAM model:

   - To align the model's origin with the ArtCAM model's origin, click **Centre**.

   - To position the model in the centre or a corner of the ArtCAM model, click one of the box diagram's five preset positions.

   - To position the model in a specific location, type the coordinates in the **X**, **Y** and **Z** boxes.

   When using either of the first two methods, the coordinates in the **Position**'s **X** and **Y** boxes are updated.

7. To rotate the imported model:

   a. In the **Rotate model about an axis** area, specify the angle of rotation for each axis in its box. You can use positive and negative values.

b. Click **Apply**. The model's size and position are updated, and the angle in the **X**, **Y** and **Z** boxes is reset to 0.0.

For example, if you rotate the model by 45 degrees in the X-axis only, the coordinates in the **Position**'s **Y** and **Z** boxes and the measurements in the **Set model size**'s **Y Size** and **Z Size** boxes change.

8. To mirror the imported model:

   a. Select the **Mirror** check box for each of the axes in which you want to mirror.

   b. Click **Apply**. All three check boxes are deselected.

9. If you are working with metric measurements, the **Model was in mm** check box is shown and selected by default. If you are working with imperial measurements, the **Model was in inches** check box is shown and selected by default.

   If the imported model was not created in the units of measurement used in your ArtCAM model, deselect the check box. The model is resized in the **3D View** window, and its dimensions are updated.

10. To scale the imported model or resize its dimensions:

   a. Select the **Link** check boxes for each of the plane's axes.

   For example, to use the X-Y plane, select the **Link X** and **Link Y** check boxes and deselect the **Link Z** check box.

   b. For the axis in which you want to resize the model, specify the new measurement in its **Size** box. The percentage in its **Scale** box is updated proportionally. For all other linked axes, the measurement in the **Size** box and percentage in the **Scale** box is updated proportionally.

   For example, to resize along the X-axis, type the new measurement in the **X Size** box. The percentage in the **X Scale** box is updated proportionally. Provided the **Link X** and **Link Y** check boxes are selected, the measurement in the **Y Size** box and percentage in the **Y Scale** box is adjusted proportionally.

   c. For the axis in which you want to scale the model, specify the percentage in its **Scale** box. The measurement in its **Size** box is updated proportionally. For all other linked axes, the measurement in the **Size** box and percentage in the **Scale** box is updated proportionally.

For example, to scale the model by 50% in the Z-axis, make sure that only the **Link Z** check box is selected, and then type the percentage in the **Z Scale** box.

d. Click **Apply**. The model's size and position are updated, and the percentage in each **Scale** box is reset to 100.0.

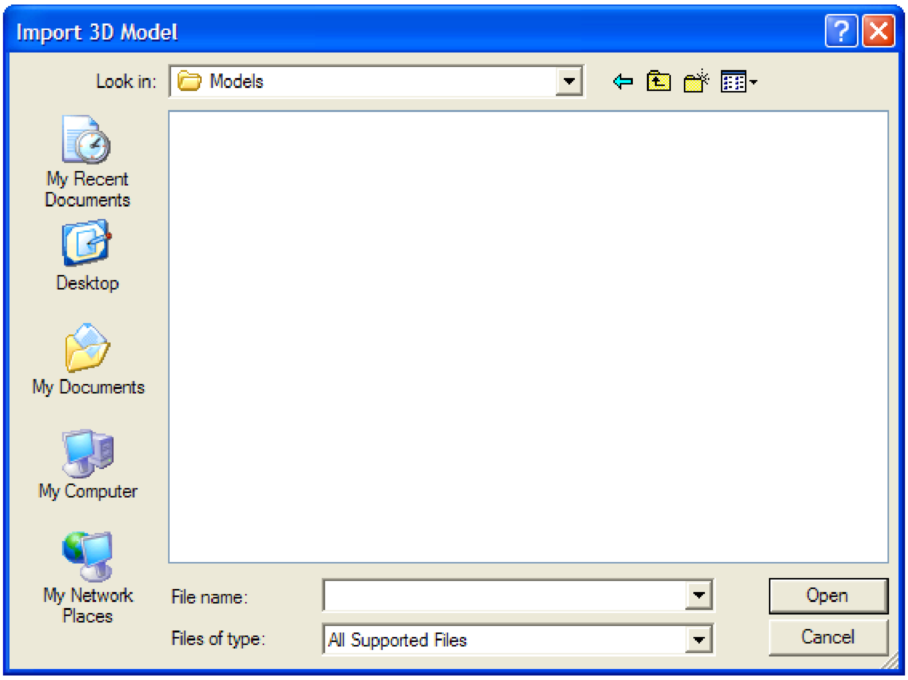
11. Click **Paste** to paste the imported model as it is shown in the **3D View** window into the currently active relief layer using the merge highest (see page 191) combine method.

12. Click **Close** to close the dialog box.

## *Importing into a project*

To import a triangle or surface model into an open project:

1. From the **Project** panel, right-click the root **Assembly** or the assembly in the Project Tree below which you want to import the triangle or surface model, then click **Import** in the context menu. The **Import 3D Model** dialog box is displayed:




2. Click the **Look in** list box, and then select the folder on your computer containing the triangle and surface model you want to use.

3. Select the file you want to import, and then click **Open**. During the import process:

   - a progress bar is displayed in the **Status Bar**;

- a replica mesh is added to the Project Tree, below the root **Assembly** or currently active assembly ; and
- the **Import 3D Model** dialog is closed.

The imported model is displayed in the **3D View** window.

# Sculpting a relief

You can use the Sculpting tools to:

- create free-form shapes on a new relief layer; or
- edit the content on a relief layer.

*If you are working in ArtCAM Insignia, the Sculpting tools are not available.*

The settings you choose for the Sculpting tools are stored between ArtCAM sessions.

To sculpt:

1. Select the relief layer (see page 167) on which you want to sculpt.
2. In the **Design Tools** toolbar, select the sculpting tool you want to use. Click:

   
   **Smooth** for smoothing an area of the relief layer by blending one area with the next;

   
   **Smudge** for extending or reducing an area of the relief layer by dragging it;

   
   **Deposit** for adding material to the relief layer;

   
   **Carve** for removing material from the relief layer;

   
   **Erase** for gradually restoring or removing material on the relief layer to zero; or

   
   **Transfer** for copying one area of the relief layer and pasting it to another.

You can also select any of these tools from the context menu displayed by right-clicking in either the **2D View** or **3D View** window.

*When one of the sculpting tools is currently selected, you can toggle through them all using the* ***1*** *to* ***6*** *number keys.*

In the **Design Tools** toolbar, the selected tool's button is highlighted. On the **Tool Settings** panel, the selected tool's name and its settings are displayed.

By default, the selected sculpting tool has:

- a rounded profile; and
- no relief layer attached.

3. To attach a relief layer to your chosen tool, click the list box on the **Tool Settings** panel, followed by its name. All of the layers included in the currently selected relief layer stack (see page 167) are included.

   When using a tool with a relief layer attached, all areas of the attached layer above or below the zero plane are used. The zero plane is ignored.

   When the sculpting cursor is positioned over the relief shown in the **3D View** window, a greyscale image of the attached relief layer's content when viewed down the Z axis is displayed beneath the cursor.

   *If you cannot see the greyscale image, click the* ***Colour Shade*** *button in the* ***3D View*** *toolbar.*

   *If you are working in* ***Software Shading*** *mode, no thumbnail image of the relief layer is shown in the list box, and no greyscale image of the relief layer is displayed beneath the sculpting cursor. An orange outline surrounds the sculpting cursor instead.*

   *If you are working in* ***Complete OpenGL 1 Support*** *mode, the thumbnail image of the relief layer shown in the* ***Tool Settings*** *list box is lower-quality, and no greyscale image of the relief layer is displayed beneath the sculpting cursor. An orange outline surrounds the sculpting cursor instead.*

4. Specify the settings you want to apply to the selected sculpting tool.

   To set the number of pixels that make up the tool's width:

   - Click and drag the **Diameter** slider; or
   - Hold down the **Shift** key, then roll the middle mouse wheel.

     Click and drag to the right to increase the width of the tool. Click and drag to the left to reduce its width. As you move the slider, the image to the left changes to illustrate the overall tool size.

     *If your mouse has a middle wheel, when the sculpting cursor is positioned over the relief, hold down the **Shift** key, and then roll it backwards to reduce the diameter of the tool, or forwards to increase its diameter.*

     The **Diameter** controls how much material is deposited onto, or removed from, the currently selected relief layer when using the selected tool.

     If a relief layer is attached to the selected tool, this is scaled according to the specified diameter.

     When the sculpting cursor is positioned over the relief in the **3D View** window, the greyscale image of the attached relief layer displayed beneath the cursor is scaled according to the specified diameter.

   - To set the height of the deposit or the depth of the removal as a percentage of the tool width, click and drag the **Strength** slider.

     Click and drag to the right to increase the height of the deposit or depth of the removal. Click and drag to the left to reduce the height of the deposit or depth of the removal. As you move the slider, the image to the left changes to illustrate the height of the deposit or the depth of the removal.

     *If your mouse has a wheel, when the sculpting cursor is positioned over the relief, roll it backwards to reduce the strength of the tool, or forwards to increase its strength.*

   - To set the sharpness of the tool, click and drag the **Smoothness** slider.

Click and drag to the right to smooth the tool. Click and drag to the left to sharpen the tool. As you move the slider, the image to the left of the **Diameter** and **Strength** sliders changes to illustrate the softness or sharpness of the tool.

If a relief layer is attached to the selected tool, this is scaled according to the specified diameter.

When the sculpting cursor is positioned over the relief in the **3D View** window, the greyscale image of the attached relief layer displayed beneath the cursor is faded from the outside-edge according to the specified smoothness.

*If your mouse has a middle wheel, when the sculpting cursor is positioned over the relief, hold down the* ***Ctrl*** *key, and then roll it backwards to reduce the sharpness of the tool, or forwards to increase its sharpness.*

5. To sculpt a specific area of the selected relief layer only, you can use a combination of options to control your sculpting.

   To sculpt a specific area of the selected relief layer using a bitmap colour:

   a. Make sure that the bitmap layer containing the artwork you want to use is selected (see page 99) and visible (see page 101).

   b. Make sure that the **Colour Shade** button in the **3D View** toolbar is toggled on. The artwork on the selected bitmap layer is displayed in the **3D View** window.

   c. Select the primary colour (see page 109) you want to use to control your sculpting.

   d. Specify how you want to use your chosen colour:

      To ignore the colour and sculpt anywhere in the selected relief layer, select **Ignore**.

      To sculpt only in those areas of the selected relief layer beneath the colour, select **Sculpt only under colour**.

      To sculpt only in those areas of the selected relief layer outside of the colour, select **Sculpt excluding colour**.

   To prevent material from being removed below a specific height in the selected relief layer:

- Select the **Material Safety Plane** check box, then, in the adjacent box, specify the Z height below which you want to prevent all sculpting.

To sculpt a specific area of the selected relief layer using a mask:

- Click the **Mask Settings** list box, followed by the name of the relief layer you want to use as the mask.

  All of the layers in the currently selected relief layers stack are included in the **Mask Settings** list box.

  The relief layer selected as the mask overlays the relief layer selected for sculpting. A low contrast greyscale image of the relief layer selected as the mask is also displayed in the **3D View** window.

  The selected mask works almost as a stencil, where your use of the tool is restricted to those areas of the mask that are above or below the zero plane only.

6. Specify how you want the effects of the selected sculpting tool to combine with the selected relief layer.

   If you have selected the **Smooth**  tool or **Smudge**  tool, select a **Relief Combine Mode** in the **Modelling Settings** area:

   - To use the average result to raise and lower the area of the selected relief layer beneath the sculpting cursor, select **Normal**.
   - To raise only the lowest points in the selected relief layer beneath the sculpting cursor, select **Raise Only**.
   - To lower only the highest points in the selected relief layer beneath the sculpting cursor, select **Lower Only**.

   If you have selected the **Deposit**  tool, select a **Relief Combine Mode** in the **Modelling Settings** area:

   - To deposit material onto the area of the selected relief layer beneath the sculpting cursor, select **Add**.
   - To deposit a single layer of material onto the area of the selected relief layer beneath the sculpting cursor, regardless of how many times the cursor passes over the area, select **Merge**.

If you have selected the **Carve** tool, select a **Relief Combine Mode** in the **Modelling Settings** area:

- To carve material from the area of the selected relief layer beneath the sculpting cursor, select **Add**.
- To carve a single layer of material onto the area of the selected relief layer beneath the sculpting cursor, regardless of how many times the cursor passes over the area, select **Merge**.

If you have selected the **Transfer** tool, select a **Relief Combine Mode** in the **Modelling Settings** area:

- To deposit the cloned material onto the area of the selected relief layer beneath the sculpting cursor, select **Add**.
- To replace the material in the area of the selected relief layer beneath the sculpting cursor with the cloned material, select **Merge**.

*If you have selected the **Erase** tool, there is no **Modelling Settings** area.*

7. In the **3D View** window, click and drag the sculpting cursor over the area of the selected relief layer to sculpt using your chosen tool with the settings that you have specified.

   If you are using the **Transfer** tool:

   a. Hold down the **Shift** key, then left-click the position in the relief layer you want to use as the reference point for cloning.

   b. Click and drag from the position in which you want to clone the area of the relief layer taken from your reference point.

8. If you want the sculpting cursor to display a boundary equal to the specified diameter of the selected sculpting tool, select the **Show tool outline** check box in the **Common Settings** area.
9. If you want the sculpting cursor to display a brush equal to the specified strength and smoothness of the selected sculpting tool, as well as a greyscale thumbnail of any relief layer attached as a mask, select the **Show tool** check box in the **Common Settings** area.

10. If you want to hide the sculpting cursor's boundary and brush when clicking and dragging, select the **Hide tool mid-stroke** check box in the **Common Settings** area.

# Importing from the Component Library

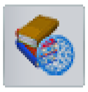
A component library is essentially a folder on your computer containing a collection of 3D assembly (***.3da**) files, each of which you can import separately. In this way, it is similar to using a vector library or relief clipart library.

*If you are working in ArtCAM Insignia, the Component Library is not available.*

Each component:

- can only be imported into an open project;
- has up to three parametric dimensions;
- can be resized proportionally; and
- creates at least one new assembly in the Project Tree, each with its own associated replica mesh.

To import a component:

1. From the **Project** panel, select the root **Assembly** or assembly in the Project Tree below which you want to import a component.
2. Below the panel's splitter bar, click the **Open Mesh Library**  button in the **Libraries** area. The **Component Library** settings are displayed.
3. Click the **Libraries** list box, then the name of the library you want to use. The library's components are listed below the **Advanced Options** area.
4. Click to select the component you want to import. Its name and dimensions are displayed.

   For example, with the **Primitives** library selected, clicking **Cone**  displays the **Component Name**, **Base Diameter** and **Height**.
5. To control the dimensions and set their parameters:

a. Click the **Advanced Options** control bar. Its settings are displayed below.

b. Click the **Dimension** list box, then the dimension you want to add or modify.

   Each dimension is numbered. If your chosen dimension is already used by the component, the **Active** check box is selected and three parameters are displayed: **Description**, **Reference Size** and **Links**.

c. If you are using a new dimension, select the **Active** check box, and then go to the next step.

   If you are editing parameters, go to the next step.

   If you are removing the dimension, deselect the **Active** check box, then go to step g.

d. In the **Description** box, type the dimension's name.

   *If you type* ShankDiameter *or* ShankWidth, ***Standard*** *and* ***Size*** *list boxes alongside* ***Diameter*** *and* ***Width*** *boxes are included in the component's settings.*

e. In the **Reference Size** box, type its distance.

f. In the **Link** area, make sure the check box for each of the plane's axes is selected.

   For example, the first of the Cone's dimensions is its base diameter. Here, the distances along the X-axis and Y-axis on the X-Y plane are 10 mm respectively.

g. Click **Apply**.

6. In the **Component Name** box, specify the component's name.
7. Specify each of the component's dimensions in the boxes.

   For example, the Cone has two dimensions, **Base Diameter** and **Height**, and both of these are set to 10 mm.
8. Click **Import** to add the component to the Project Tree.

   One or more assemblies are created in the Project Tree, each with its own associated replica mesh . Any gems associated with the component are also imported.
9. Click on the splitter bar to close the **Component Library** settings.

# Creating Toolpaths

ArtCAM provides several 2D/2.5D and 3D toolpaths that you can use to machine the vector artwork or composite relief that you have created as part of your ArtCAM model.

The toolpaths you can use to machine vector artwork are available from the:

- **Project** panel, displayed in the **2D Toolpaths** area below the splitter bar when **Toolpaths** is selected in the Project Tree;
- **Menu Bar**, displayed by clicking **Toolpaths > 2D Toolpaths**; and
- **Toolpaths** panel, displayed in the **2D Toolpaths** area.

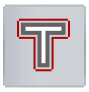
The toolpaths that can be used to machine a composite relief are available from the:

- **Project** panel, displayed in the **3D Toolpaths** area below the splitter bar when **Toolpaths** is selected in the Project Tree;
- **Menu Bar**, displayed by clicking **Toolpaths > 3D Toolpaths**; and
- **Toolpaths** panel, displayed in the **3D Toolpaths** area.

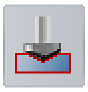
## *Selecting a toolpath*

Select the type of toolpath you want to create:

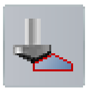
1. To create a 2D toolpath, in the **2D Toolpaths** area of the **Project** or **Toolpaths** panel, click the:

- **Create 2D Profiling Toolpath**  button to display the **Profiling** panel, which enables you to create a toolpath that machines along, inside or outside the boundary of a selection of vectors.

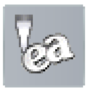
- **Create Area Clearance Toolpath**  button to display the **2D Area Clearance** panel, which enables if you to create a toolpath that clears an area of material within a selected vector or between selections of vectors.

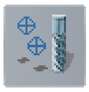
- **Create V-Bit Carving Toolpath**  button to display the **V-Bit Carving** panel, which enables you to create a toolpath that reproduces an engraved or recessed look when machining vector text or a selection of vectors.

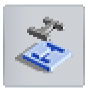
*You cannot machine a V-Bit Carving toolpath without using a 3-axis machine. For further information, consult a member of staff or your machine tool supplier.*

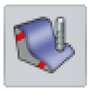
- **Create Bevel Carving Toolpath**  button to display the **Bevel Carving** panel, which enables you to create a toolpath that reproduces a bevelled (angle-edged) look when machining vector text or a selection of vectors.

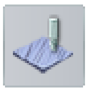
*You cannot machine a Bevel Carving toolpath without using a 3-axis machine. For further information, consult a member of staff or your machine tool supplier.*

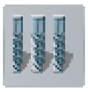
- **Create Engraving Toolpath**  button to display the **Smart Engraving** panel, which enables you to create a toolpath that engraves within or around the boundary of vector text or a selection of vectors.

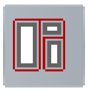
- **Create Drilling Toolpath**  button to display the **Drilling** panel, which enables you to create a toolpath that drills holes using a selection of vectors or a 2D toolpath preview.

- **Create Inlay Toolpath**  button to display the **Inlay Wizard** panel, which enables you to create a toolpath that machines a selection of vectors that represent corresponding inlays (female) and inserts (male).

- **Create Raised Round Toolpath**  button to display the **Raised Round** panel, which enables you to create a toolpath that enables you to create precise recessed letters and shapes using vector text or a selection of vectors.

- **Create Texture Toolpath**  button to display the **Texture Toolpath** panel, which enables you to create a toolpath that machine a texture across the whole surface, or a specific area, of your model using only a tool's geometry and machining parameters.

- **Create Drill Bank Toolpath**  button to display the **Drill Banks** panel, which enables you to machine a pattern such as L, + or T shapes containing any number of drills to match the Drill Bank on your machine. ArtCAM considers the selected geometry and applies a toolpath for all X and Y axis holes containing a 32 mm (1.26") pitch (standard pitch value for most machines).

- **Open 2D Machining Wizard**  button to display the **2D Machining Wizard** panel, which provides a fully automated nested-based manufacturing solution for makers of custom furniture and cabinetry.

2. To create a 3D toolpath, in the **3D Toolpaths** area of the **Project** or **Toolpaths** panel, click the:

   - **Create Machine Relief Toolpath**  button to display the **Machine Relief** panel, which enables you to create a toolpath that machines the whole of the composite relief or a particular area within a vector boundary.

- **Create Feature Machining Toolpath** button to display the **Feature Machining** panel, which enables you to create a toolpath that machines Raised, Recessed or Centreline Engraved features created from vector artwork.

- **Create Z Level Roughing Toolpath** button to display the **Z Level Roughing** panel, which enables you to create a toolpath that removes unwanted material in planar slices when machining the composite relief.

*If you are working in ArtCAM Insignia, the **Create Z Level Roughing Toolpath** button is not included.*

- **Create Laser Machining Toolpath** button to display the **Laser Machining** panel, which enables you to create a toolpath that removes layers of unwanted material when machining the composite relief using a laser-engraving machine.

- **Create 3D Cut Out (Profile) Toolpath** button to display the **3D Cut Out** panel, which enables you to create a toolpath that machines the area of the composite relief either inside or outside the boundary of a selection of vector artwork.

*If you are working in ArtCAM Insignia, the **Create 3D Cut Out (Profile) Toolpath** button is not included.*

- **Create 3D Rest Machining Toolpath** button to display the **3D Rest Machining** panel, which enables you to identify all areas of the composite relief that cannot be machined when using a chosen tool, and create a toolpath to machine all remaining areas in the composite relief using a second smaller tool.

*If you are working in ArtCAM Insignia, the **Create 3D Rest Machining Toolpath** button is not included.*

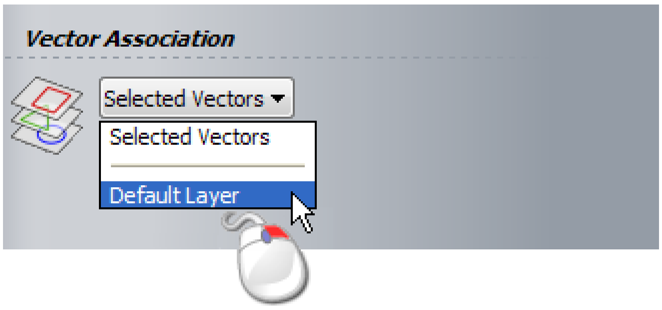
## *Specifying what you are machining*

Almost all of the toolpath panels include a **Vector Association** or **Area To Machine** area. This contains at least one list box that enables you to control what vector artwork or which part of the composite relief is machined by the toolpath.

Specify what part of your model you want to machine:

1. To machine vector artwork directly or use vectors to control which area of the composite relief is machined:

   a. Click the **Vector Association** or **Area To Machine** list box, then **Selected Vectors** or the vector layer's name. For example, *Default Layer*.

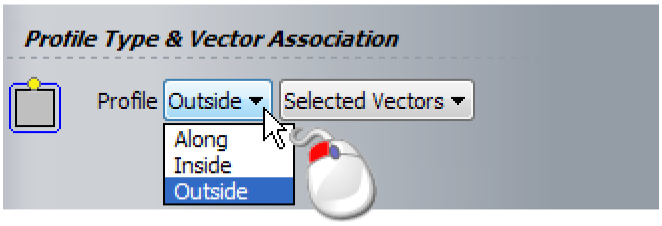
   
   

   b. If you have selected **Selected Vectors**, select the vector artwork (see page 122) you want to use.

   c. If you have chosen a specific vector layer (see page 124), make sure it contains all of the vector artwork you want to use.

   The **Profiling** and **3D Cut Out** panels also include a **Profile Type** list box that enables you to control how the vector artwork is machined. Select:

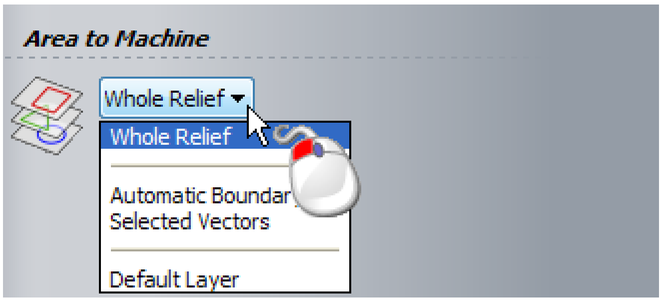
   - **Along** to machine along the vector boundary;
   - **Inside** to machine within the boundary; or
   - **Outside** to machine outside the vector boundary.

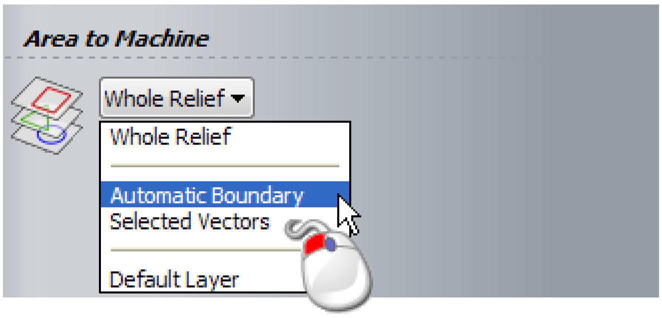
2. To machine the composite relief:

   a. Make sure that only the relief layers that make up the composite relief you want to machine are visible (see page 172).

b. Click the **Area To Machine** list box, then **Whole Relief**.

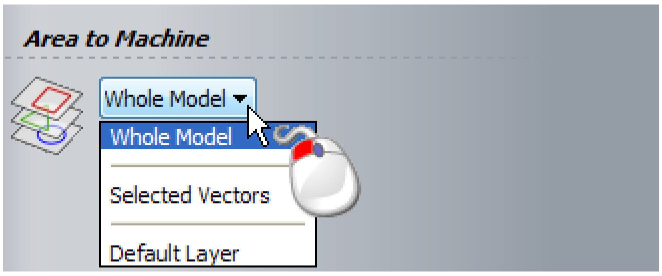



The **Machine Relief** panel's **Area To Machine** list box includes an additional option that enables you to machine as closely as possible to the composite relief above or below its base height using a chosen tool: **Automatic Boundary**.




A boundary vector is calculated from the tool's geometry and is drawn on the currently active vector layer (see page 124).

The **Texture Toolpath** panel's **Area To Machine** list box includes an additional option that enables you to machine the model area: **Whole Model**.




## *Specifying the cutting parameters*

Almost all of the toolpath panels include a **Cutting Parameters** area with one or more of the following settings:

- **Start depth** or **Surface Z** sets the surface depth at which a tool begins machining.
- **Finish depth** or **Finish Z** sets a tool's final cutting depth.

- **Allowance** or **Offset Allowance** specifies an offset around the vector boundary or the composite relief. This leaves additional material during intermediate machining passes for a tool to clear during the last pass.
- **Tolerance** controls how closely and accurately a tool follows the curvature of a vector boundary or composite relief. Although a tight tolerance produces a quality finish, it can also significantly increase toolpath calculation and machining times.

Some toolpath panels include additional allowances and cutting parameters for particular types of tools.

Specify your toolpath's cutting parameters:

| Panel | Cutting Parameters |
|---|---|
| Profiling | Start Depth, Finish Depth, Allowance and Tolerance.<br>Provided the **Profile Type** is set to **Outside** or **Inside**, you can specify a Final Pass Thickness and Final Pass Allowance. |
| 2D Area Clearance | Start Depth, Finish Depth, Allowance, Final Tool Allowance, Tolerance and Independent Finish Depth. |
| V-Bit Carving | Start Depth and Tolerance.<br>Limit Tool Maximum Depth is optional. |
| Bevel Carving | Start Depth, Wall Height, Finish Depth, Allowance and Tolerance. |
| Smart Engraving | Start Depth, Finish Depth, Tolerance, Allowance and Independent Finish Depth. |
| Drilling | Start Depth, Finish Depth and Plunge Clearance. |
| Inlay | Start Depth, Finish Depth, Allowance and Tolerance.<br>Depth and Width if the inlay is stepped. |
| Raised Round | Start Depth, Finish Depth and Tolerance. |
| Texture Toolpath | Start Depth, Minimum Finish Depth and Maximum Finish Depth. |

| Panel | Cutting Parameters |
|---|---|
| Drill Banks | Start Depth, Finish Depth, Plunge Clearance, Circle Tolerance and Pitch Tolerance. |
| Machine Relief | Allowance and Tolerance. |
| Feature Machining | Feature Height or Depth, Feature Allowance and Allowance. Overcut Distance if Area Clear strategy is used. |
| Z Level Roughing | Surface Z, Allowance and Last Slice Z. |
| Laser Machining | Surface Z, Slice Thickness, Last Slice Z, Stepover and Angle Increment. |
| 3D Cut Out | Surface Z, Finish Z, Allowance and Tolerance. |
| 3D Rest Machining | Cusp Tolerance. |

*If you are working in ArtCAM Insignia, the **Z Level Roughing**, **3D Cut Out** and **3D Rest Machining** toolpaths are not available.*

### *Using tools from the Tool Database*

All of the toolpath panels require that you select at least one tool from the **Tool Database**. Some toolpaths enable you to select several tools.

When a toolpath requires only one tool, there is one tool area on the panel. When a toolpath requires or enables you to use two tools or more, the panel includes:

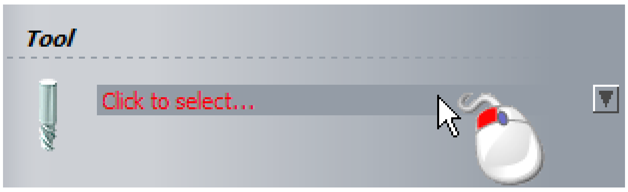
- a **Tools List**; or
- a separate area for each tool.

For example, the **2D Area Clearance** and **Smart Engraving** toolpath include a **Tools List**, and the **Bevelled Carving** toolpath includes a **Carving Tool** and **Profiling Tool** area.

Choose the tools you want to use:

1. If there is a **Tool** area on the panel:

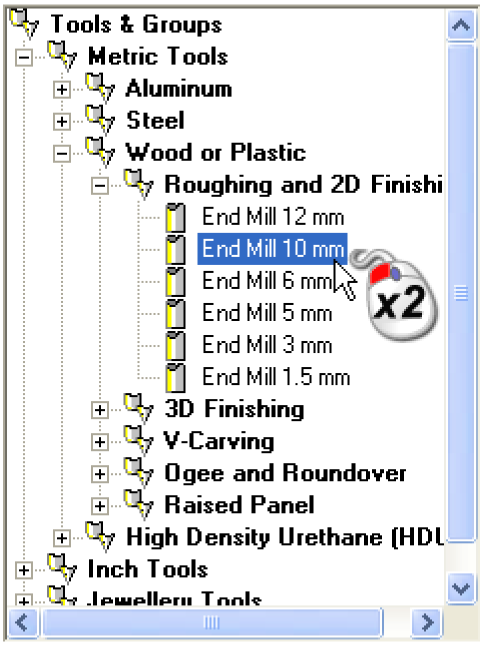
a. Click the control bar to display the **Tool Database**.




The area's name often includes the type of tool you need. For example, **Profiling Tool** or **Carving Tool**.

On the control bar, *Click to select...* or *Undefined* is displayed when no tool is selected.

b. In the **Tool Database**, double-click the name of the tool you want to use:




The **Tool Database** closes, and your selected tool's description is displayed on the control bar. For example, *End Mill 10 mm*. For further details on using the **Tool Database**, see Using the Tool Database (see page 263).

2. To replace the tool in the **Tool** area:

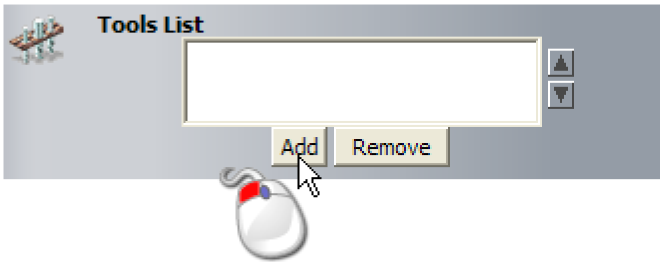
   a. Click the control bar to display the currently selected tool's parameters.

   b. Click **Select...** to display the **Tool Database**.

   c. From the **Tool Database**, double-click the name of the tool you want to use instead. The **Tool Database** is closed, and your selected tool's description replaces what was previously shown on the control bar. For example, *End Mill 3 mm*.

3. If there is a **Tools List** on the panel:

a. Below the **Tools List**, click **Add**.




The **Tool Database** is displayed.

b. From the **Tool Database**, double-click the name of the tool you want to use. The **Tool Database** is closed, your selected tool's description is displayed in the **Tools List**, and the tool is selected.

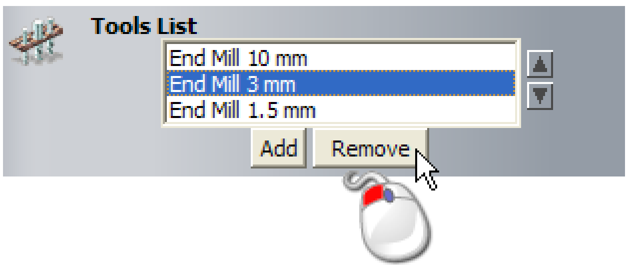
*If you are working in ArtCAM Insignia, you can only add one tool to the* ***Tools List****.*

A control bar is displayed below the **Tools List**, on which the tool's description is shown. Below the control bar, the tool's parameters are displayed. For example, **Stepover** and **Stepdown**. You can change the tool's parameters by specifying them in the boxes. Updating the parameters here does not affect those in the **Tool Database**.

4. To remove a tool from the **Tools List**:

a. In the **Tools List**, click the tool's description. For example, *End Mill 3 mm*. It is highlighted.

b. Click **Remove**.




If there are no other tools included in the **Tools List**, the control bar is no longer displayed. If there is at least one other tool left in the **Tools List**, the tool which was previously below the deleted tool is now selected, and its description is shown on the control bar instead. The tool is not removed from the **Tool Database**.

### *Selecting a strategy*

The strategy controls the way in which your selected vector artwork or composite relief is machined.

When creating a toolpath, you are sometimes given a choice between two instances of a single strategy. For example, **Raster** or **Raster (Classic)**. The "(Classic)" suffix typically indicates that a particular strategy:

- does not make use of multithreaded technology;
- is less memory-intensive;
- is better suited where radiused conical tools are used; and
- produces a surface finish familiar to users of ArtCAM 2009 and earlier.

To choose a strategy:

1. If there is a **Tools List** on the panel, click the tool's description. For example, *End Mill 3 mm*.
2. Click the **Tool Clearance Strategy** list box, then:
   - **Raster** or **Raster (Classic)** to machine in passes back and forth along the X-axis at a specified angle.

     *If you are working in ArtCAM Insignia, these strategies are named **Raster in X** and **Raster in X (Classic)**, and you cannot specify the angle.*

   - **Raster X & Y** or **Raster X & Y (Classic)** to machine in passes back and forth along the X and Y axes at a specified angle. Machining in two perpendicular directions increases the machining time, but typically produces a superior surface finish.

     *If you are working in ArtCAM Insignia, you cannot specify the angle.*

   - **Offset** or **Offset (Classic)** to machine in repeated passes, each time moving inwards or outwards by the tool's **Stepover**.

     *If you are working in ArtCAM Insignia and creating a **Machine Relief** toolpath, these strategies are not available.*

   - **3D Offset** to machine using a consistent stepover, regardless of direction. This technique is best suited to machining steep and near vertical surfaces.

*If you are working in ArtCAM Insignia, this strategy is not available.*

- **3D Offset Spiral** to machine in a spiral motion, where the stepover is consistent regardless of direction. This minimises the number of lifts of the tool and maximises cutting time while maintaining more constant load conditions and deflections on the tool.

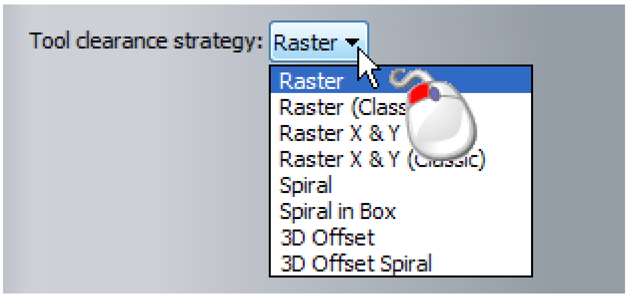
*If you are working in ArtCAM Insignia, this strategy is not available.*

- **Spiral** to machine in a spiral motion. When the tool reaches the first edge of the composite relief, it stops. If the composite relief is rectangular in shape, this means that only part of it is machined.

*If you are working in ArtCAM Insignia, this strategy is not available.*

- **Spiral in Box** to machine in a spiral motion. When the tool reaches the first edge of the composite relief it retracts, traverses along the edge of the composite relief, and then plunges to continue machining the next area of the composite relief. This strategy allows you to machine the entire composite relief but can also be time consuming.

*If you are working in ArtCAM Insignia, this strategy is not available.*




*If you cannot see the tool's **Stepover** on the panel, click ▼ on the control bar displaying the tool's description.*

3. If you have selected **Raster**, **Raster (Classic)**, **Raster X & Y** or **Raster X & Y (Classic)**, specify the angle in the **Angle** box.

   *If you are working in ArtCAM Insignia, the **Angle** box is not displayed.*

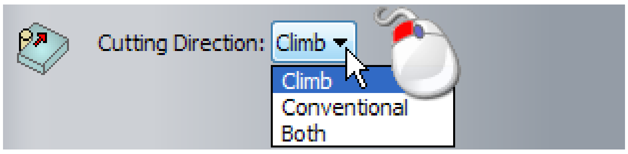
4. If you have selected **Offset** or **Offset (Classic)**:

   a. Click the **Cutting Direction** list box, then:

      **Climb** to revolve the tool in the same direction as the table feed. The tooth meets the material at the top of the cut, producing the thickest part of the chip first. Typically, this improves the surface finish and increases tool life.

      **Conventional** to revolve the tool in the opposite direction to the table feed. The width of the chip starts at zero, and increases to a maximum at the end of the cut. This can lead to accelerated tool wear and an inferior surface finish.

      **Both** to revolve the tool in either direction, as required.



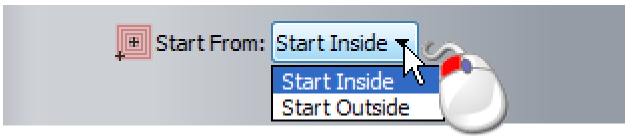

*In ArtCAM Pro and JewelSmith, the option to use both climb and conventional milling is limited to the* ***Machine Relief*** *toolpath. If you are working in ArtCAM Insignia, the* ***Both*** *option is not available.*

*Some toolpaths use nothing other than an offset strategy, therefore only the* ***Cutting Direction*** *list box is displayed on its panel. For example,* ***Profiling*** *and* ***Bevel Carving****.*

   b. Click the **Start From** list box, then:

      **Start Outside** to machine inwards from the boundary; or

      **Start Inside** to machine outwards from the centre, towards the boundary.




*Although several toolpaths with an* ***Offset*** *option include* ***Start From*** *settings, such as* ***2D Area Clearance*** *and* ***Machine Relief****, others do not. For example,* ***Smart Engraving****.*

### *Setting Z slices*

When calculating a **Machine Relief** or **Laser Machining** toolpath, you can control the number of Z passes used to machine between the top of the material block and the composite relief's lowest point.

*If your tool's **Allowance** is set to zero, dwell marks may remain on the material surface after the final machining pass.*

If you have selected **Raster (Classic)**, **Raster in X (Classic)**, **Raster X & Y (Classic)** or **Offset (Classic)** as your tool clearance strategy:

1. Click the **Multiple Z Passes** control bar to display the settings.
2. Select the **Do Multiple Z Passes** check box.
3. In the **Start Z** box, specify the Z height of the first pass.
4. In the **Finish Z** box, specify the Z height of the last pass.

If you have selected any other tool clearance strategy:

1. Click the **Z Slices** control bar to display the settings.
2. To enable ArtCAM to calculate the number of slices using the composite relief's Z dimensions, material block dimensions and your **Roughing Tool**'s specified **Stepdown**, **Allowance** and **Tolerance**, select the **Automatic** check box. The:
   - **Start/Surface Z** is equal to the top of the material block or, where you have not specified your material block dimensions, the composite relief's maximum Z height.
   - **Last Slice Z** is equal to the composite relief's minimum Z height, together with your **Roughing Tool**'s specified **Allowance** and **Tolerance**.
   - **Number of Slices** is the result calculated from the Z heights and the **Roughing Tool**'s **Stepdown**, **Allowance** and **Tolerance**.
   - **Slice Thickness** is equal to or less than the selected **Roughing Tool**'s **Stepdown**.

   *If you cannot see the tool's **Stepdown** on the panel, click ▼ on the control bar displaying the tool's description. For example, End Mill 10 mm.*
3. To control the number of slices and their thickness:

   a. Deselect the **Automatic** check box. The **Start/Surface Z** and **Last Slice Z** boxes are enabled.

b. In the **Start/Surface Z** box, specify the Z height of the first slice.

c. In the **Last Slice Z** box, specify the Z height of the last slice.

d. Click **Apply** to calculate the number of slices and their thickness. Your **Roughing Tool**'s specified **Stepdown**, **Allowance** and **Tolerance** are also used during the calculation process.

## *Adding lead moves*

Lead moves are a precautionary measure to prevent dwell marks on a machined piece at the point where the tool enters and leaves the material block when machining a profile pass.

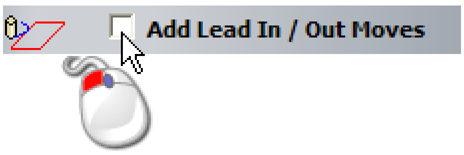
Instead of plunging into the material block, machining the profile, and then retracting, the tool enters and leaves the block at a particular position in the profile from a specified distance. The entry and exit positions can be the profile's start and end points, or the mid of its longest span.

You can apply lead moves to a:

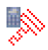
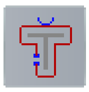
- new or previously calculated **Profiling** or **3D Cut Out** toolpath; or
- previously calculated **Bevel Carving**, **Female Inlay** or **Male Insert** toolpath.

To add lead moves:

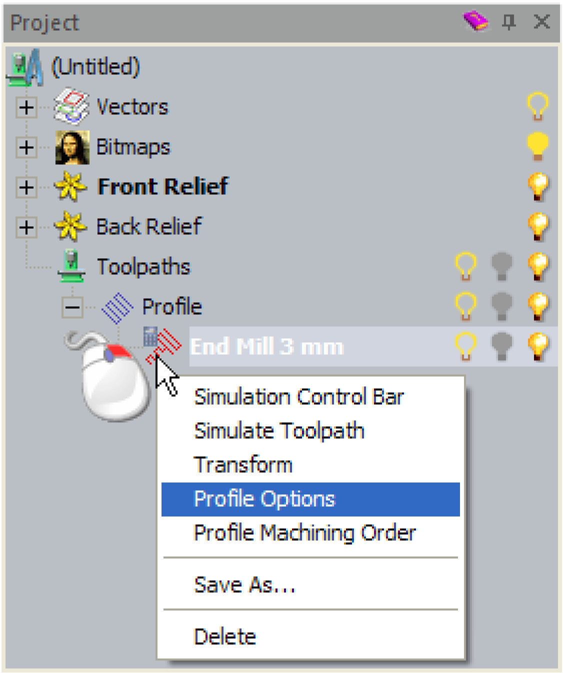
1. If you are creating a new toolpath, select the **Add Lead In / Out Moves** check box to display its associated settings.




2. If you are adding lead moves to a previously calculated toolpath, use one of the following methods to display the **Profile Options** settings on the **Tool Settings** panel:

   - From the **Project** panel, select the tool's name  in the Project Tree, then click the **Profile Options**  button displayed below the splitter bar;

- From the **Project** panel, right-click the tool's name in the Project Tree, then select **Profile Options** in the context menu; or




- From the **Toolpaths** panel, select the tool's name from the list above the **Toolpath Operations** area, then click the **Profile Options** button in the **2D Toolpaths** area.

3. In the **Distance (D)** box, specify the distance from the profile at which the tool cuts into and retracts from the material block.
4. In the **Overcut (O)** box, specify the distance you want the tool to machine beyond the profile's start and end point. This further helps to create a smooth finish.
5. Select:
   - the **Do not lead out** check box if you want to add lead in moves only.
   - **Linear** if you want the tool to lead into and out of the material block in a straight-line motion.

     When you calculate the toolpath, ArtCAM checks that the lead move's distance is within the profile's boundary. If your specified distance causes the lead move to intersect with the profile, it is clipped so that it does not.
   - **Circular Arc** if you want to instruct the tool to lead into and out of the material block in an arc motion.

When you calculate the toolpath, ArtCAM checks that the lead move's radius is within the profile's boundary. If your specified radius causes the lead move to intersect with the profile, a linear lead with a distance within the profile's boundary replaces the arc lead.

If ArtCAM makes any necessary changes to the lead moves, a message box is displayed during the calculation process.

6. If you are adding linear lead moves:
   a. In the **Angle In** box, specify the lead in move's angle.
   b. If you are adding lead out moves, specify their angle in the **Angle Out** box.
   c. If you are creating a new toolpath and want the tool to ramp from the surface depth to the profile rather than plunge, select the **Ramp from surface** check box. The surface depth is equal to your specified **Start depth** or **Surface Z** value. The ramp length is equal to your specified **Distance (D)** value.
   d. If you are creating a new toolpath and want to position lead moves at the profile's start and end points only, deselect the **Automatic positioning** check box. Otherwise, ArtCAM positions the lead moves at the mid of profile's longest span; which is also the longest span in the vector artwork used to create the toolpath.
   e. If you are creating a new toolpath and want to apply cutter compensation commands (*G41* - comp left and *G42* - comp right) to the NC code associated with the profile's start and end point, select the **Cutter compensation** check box.

      Cutter compensation enables your machine tool's controller to calculate the offset toolpath based on the profile that is specified in the NC code. The offset value is typically stored in the controller's memory. The main advantage of this approach is that changes to the offset value are completed on the machine without editing the NC code. This is very practical when making adjustments to compensate for tool wear or changing to a tool with different diameter.

*The **Ramp from surface**, **Automatic positioning** and **Cutter compensation** check boxes are not available on the **Tool Settings** panel. These are only available when creating a toolpath.*

7. If you are adding circular lead moves, specify the arc move's radius in the **Radius (R)** box. This must be less than or equal to the value in the **Distance (D)** box, otherwise a message box is displayed.
8. If you are adding lead moves to a previously calculated toolpath, click **Apply**.

   If the previously calculated toolpath includes ramping or cutter compensation and you have selected **Circular Arc**, a message box is displayed warning that the two are incompatible. To apply circular arc lead moves to the profile, you must:

   a. edit the toolpath.

   b. change your original lead move settings.

   c. recalculate the toolpath.

   If you have selected **Circular Arc** and your specified **Radius (R)** value is greater than the **Distance (D)** value, a message box is displayed warning that the distance must be less than or greater than the radius. To apply circular arc lead moves to the profile, you must:

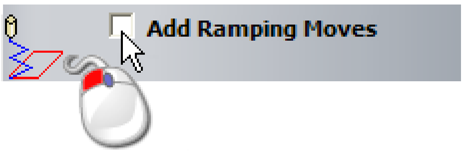
   - reduce their radius so that it is equal to or less than their distance; or
   - increase their distance so that it is equal to or greater than their radius.

## Adding ramping moves

When machining hard stock material, ramping the tool significantly reduces the load on the tool and the Z axis of your machine. You can add ramping moves to **Profiling**, **2D Area Clearance**, **Smart Engraving**, **Inlay**, **Machine Relief**, **Z Level Roughing** or **3D Cut Out** toolpaths.

To add ramping moves to your toolpath:

1. Select the **Add ramping moves** or **Add ramping** check box.




2. If you are creating a **Machine Relief** toolpath:

   a. In the **Max Zig Angle** box, specify the angle of descent as the tool ramps into the material block.

b. In the **Plunge Height** box, specify the Z height from which the ramp descends.

c. Select the **Limit length** check box, then, in the **Ramp Length (TDU)** box, specify the maximum ramp distance relative to the tool's diameter. Typically, the ramp length should be greater than the tool diameter to allow chips to clear from beneath the tool.

d. Select the **Independent zag angle** check box, then, in the **Zag Angle** box, specify the second angle of descent if your tool does not reach the start of the toolpath segment in a single pass.

*If you are working in ArtCAM Pro or JewelSmith and have selected **Raster (Classic)**, **Raster X & Y (Classic)** or **Offset (Classic)** as your tool clearance strategy, the **Independent zag angle** check box is not displayed. If you are working in ArtCAM Insignia and have **Raster in X (Classic)** selected as your tool clearance strategy, the **Independent zag angle** check box is not displayed.*

3. If you are creating another toolpath, specify the type of ramp you want to use. Click:

- **Spiral** to descend in a helical ramp motion around the profile's perimeter.
- **ZigZag** to descend in linear zig zag moves with a specified Z angle and distance.
- **Smooth** to add a short linear ramp to each toolpath segment.

If you have selected **ZigZag** or **Smooth**:

a. In the **Max Ramp Angle (A)** box, specify the maximum usable slope for ramping down to depth. Setting this to 0 causes a plunge cut.

*The ideal ramp angle should be between 0 and 20 degrees from the table surface. This angle enables the tool to enter into the material block at 100% of the feed rate. At any angle greater then 20 degrees, the feed rate should be reduced accordingly.*

b. In the **Max Ramp Length (L)** box, specify the maximum distance for each linear ramping move.

c. In the **Min Ramp Length (Lmin)** box, specify the minimum distance for each linear ramping move.

If you have selected **ZigZag**, specify the Z height from which the ramp descends in the **Zig Start Height (S)** box.

### *Setting the machining order*

If your toolpath contains segments, you can control the order in which they are machined. These toolpaths include **Profiling**, **V-Bit Carving** and **Drilling**.

You can instruct ArtCAM to calculate the machining order, or use a polyline drawn on a vector layer to map the order.

1. To use a preset machining order:
   a. Click the **Cut Sequence** control bar to display it settings.
   b. Click to select the **Automatic** option.
   c. Click the list box, then:

      **Optimise** to instruct ArtCAM to calculate the most efficient machining order.

      **Use Text Order** if you are machining vector text and want to machine in the order in which it was created.

      **Left to Right** to machine from the left of the model area to the right.

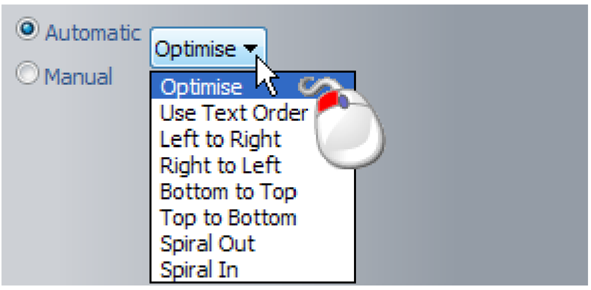
      **Right to Left** to machine from the right of the model area to the left.

      **Bottom to Top** to machine from the bottom of the model area to the top.

      **Top to Bottom** to machine from the top of the model area to the bottom.

      **Spiral Out** to machine from the centre of the model area outwards in a spiral motion.

      **Spiral In** to machine from the outside edge of the model area inwards in a spiral motion.



Your selected option is displayed on the control bar.

2. To set the order using an open, ungrouped vector:
   a. Click the **Cut Sequence** control bar to display it settings.
   b. Click to select the **Manual** option.
   c. Draw (see page 137) and select (see page 122) a polyline on a vector layer that follows the order in which you want to machine the toolpath segments. The polyline's start node should be positioned (see page 149) close to the toolpath segment at which you want to begin machining.

   *You must select an open ungrouped vector to represent the machining order.*

   d. Click the **Select Vector** button. The *No Vector Selected* message in red text changes to *Vector Selected* in blue text.
   e. Make sure all of the vectors you want to machine are selected (see page 122).

## Specifying the Safe Z height and Home position

All toolpaths require that you specify the:

- height at which your selected tool makes rapid moves between toolpath segments; and
- the X, Y and Z coordinates of the tool's start and end position.

To set these positions:

1. In the **Options** area, click the control bar to display the **Safe Z** box and the **Home**'s **X**, **Y** and **Z** boxes.
2. In the **Safe Z** box, specify the height at which your selected tool makes rapid moves between toolpath segments.

   *This must be sufficient to clear any clamps used to hold your material block or sheet in position.*

3. In the **Home**'s **X**, **Y** and **Z** boxes, type the X, Y and Z coordinates of the tool's start and end position.

   *This should be at a safe distance away from your material block or sheet.*

4. Click the control bar to hide the **Safe Z** box and the **Home**'s **X**, **Y** and **Z** boxes.

Your settings are displayed on the control bar.

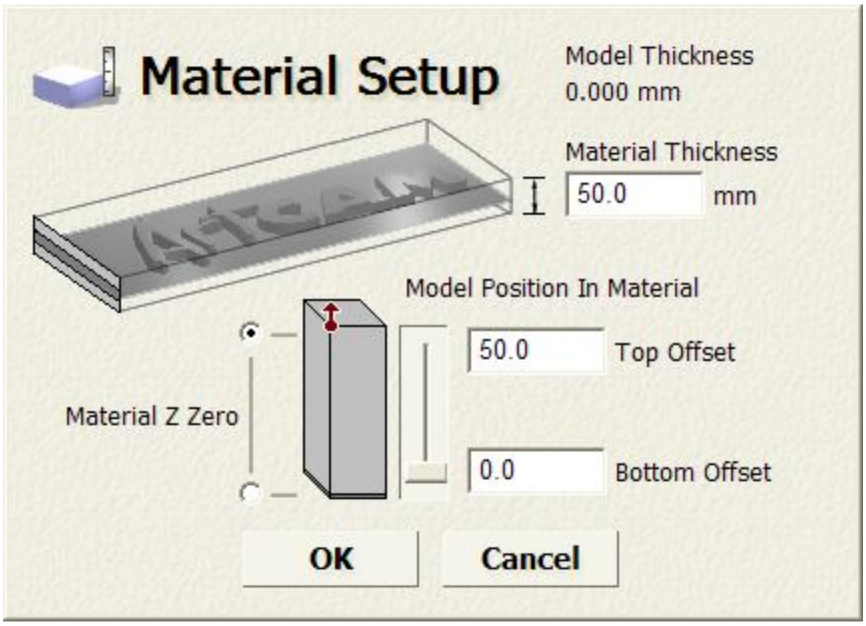
### *Specifying the material setup*

The material setup enables you to specify:

- the thickness of your material block or sheet; and
- how the model is positioned relative to your material block or sheet.

To specify your material's thickness and set the position of your model relative to the material:

1. In the **Material** area, click the control bar. The **Material Setup** dialog box is displayed:

2. In the **Material Thickness** box, make sure the value is equal to that of the material block or sheet you are using for your job.

   If you have already created a composite relief, you cannot set the **Material Thickness** to less than the Z height displayed in the **Model Thickness** area.

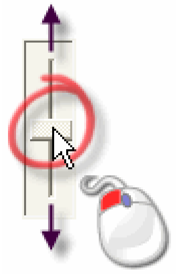
   *The composite relief's Z height is also displayed on the* ***Project*** *panel when the model is selected in the Project Tree.*

3. In the **Material Z Zero** area, make sure the Z zero position is correct by clicking the option beside the top or bottom left of the block diagram.

*The icon marks the position of the origin relative to the material block or sheet.*

4. Make sure that the position of the model relative to the material block or sheet is set correctly.
   - To specify the amount of stock material above the composite relief's surface, click the **Top Offset** option in the **Model Position In Material** area, then specify the offset in its associated box.

   *You can adjust the position of the composite relief relative to the material block or sheet by clicking and dragging the slider:*

   *The value shown in both the **Top Offset** and **Bottom Offset** boxes is adjusted when the slider is moved.*

   - To set the amount of stock material below the base of the composite relief, click the **Bottom Offset** option, then specify the offset in its associated box.

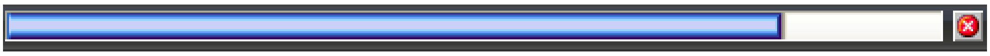
   *The box between the **Top Offset** and **Bottom Offset** boxes shows the current depth (Z) of the composite relief.*

### *Calculating the toolpath*

To calculate the toolpath:

1. In the **Name** box, type the name you want to give the toolpath.
2. Choose when you want to calculate the toolpath:
   - To calculate the toolpath at a later time, either by itself or as part of a batch of toolpaths, click the **Calculate Later** button.
   - To calculate the toolpath now, click the **Calculate Now** button.

A progress bar is displayed in the **Status Bar** area during the toolpath calculation process:



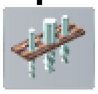
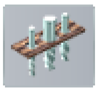
*You can click to cancel the toolpath calculation process at any time.*

3. Click **Close** to close the panel.

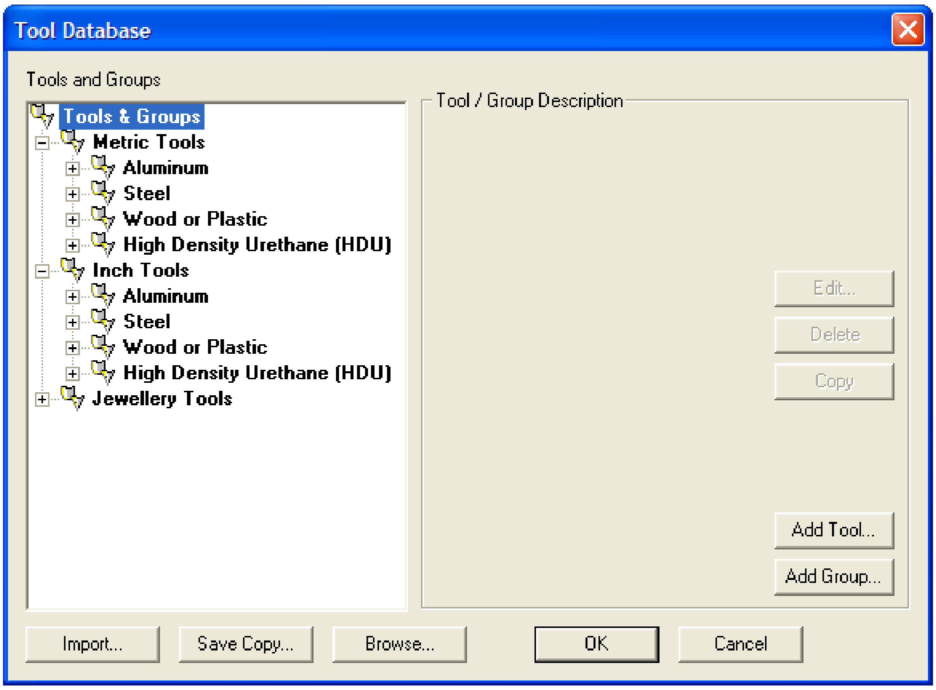
# Using the Tool Database

When you create a toolpath in ArtCAM, it is necessary to select the tool you want to use to machine your selected vector artwork or the composite relief. A broad range of pre-defined tools can be selected from the **Tool Database**. You can also specify your own custom tools and add them to the **Tool Database**.

To specify and add a custom tool to the **Tool Database**:

1. Use one of the following methods to display the **Tool Database**:
   - From the **Project** panel, select **Toolpaths** in the Project Tree, then click the **Tool Database**  button in the **Toolpath Operations** area displayed below the splitter bar; or
   - From the **Toolpaths** panel, click the **Tool Database**  button in the **Tool Operations** area.

*For information on using the* **Toolpaths** *panel, see Understanding the Toolpaths panel (see page 32).*




2. Make sure that the tool group in which you want to add a tool is selected by clicking its name. When selected, its name is emphasised in blue.

   To add a new tool group:

   a. In the **Tool/Group Description** area, click the **Add Group...** button. The new tool group is named *New Group* by default and is created below whichever item is currently selected in the **Tools and Groups** window.

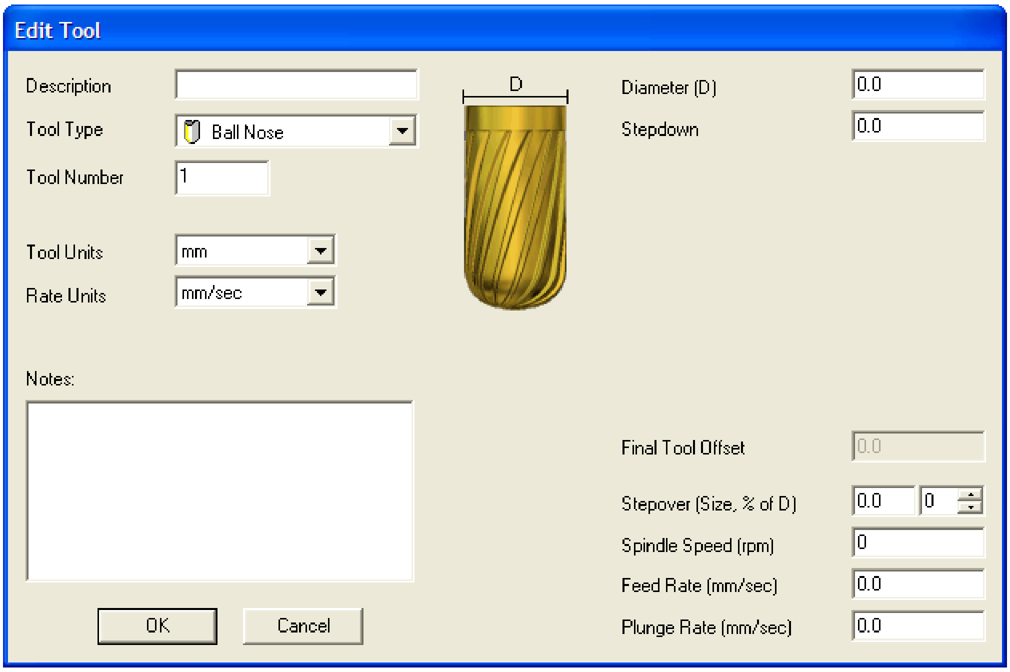
*A tool group is identified by the icon, while a tool is identified by the , , , , , or icon.*

   a. Type the name that you want to give to the new tool group.

   b. Click anywhere in the white area shown in the **Tools and Groups** window to apply the name to the tool group.

*You can click and drag any of the tools listed in the* ***Tools and Groups*** *window into the new tool group you have created.*

3. Click the **Add Tool...** button to display the **Edit Tool** dialog box:




4. In the **Description** box, type the name you want to give to your tool.
5. Click the **Tool Type** list box, and then select the type of tool you want to add to the **Tool Database**. An image representing the selected tool type is displayed alongside boxes in which you must specify the tool's machining parameters.
6. In the **Tool Number** box, specify the number of the tool according to its current position in your tool changer.
7. Click the **Tool Units** list box, followed by the unit of measurement associated with your tool.
8. Click the **Rate Units** list box, followed by the unit of speed associated with your tool.
9. In the **Notes** box, type any relevant information concerning the practical uses of your tool.
10. In the column of boxes on the right, specify your tool's machining parameters. For example, **Diameter**, **Stepdown** and **Feed Rate**.
11. Click **OK** to add your tool to the **Tool Database**. The tool is listed in the **Tools and Groups** window, and its details are displayed in the **Tool/Group Description** area.
12. Click **OK** to close the **Tool Database**.

# Saving toolpaths

You can save a toolpath as a machine-specific toolpath file or as a toolpath template file (***.tpl**).

*When you save an ArtCAM model file (***.art**), all of the toolpaths created or calculated as part of the model are saved in the file. For details, see Saving a model (see page 84).*

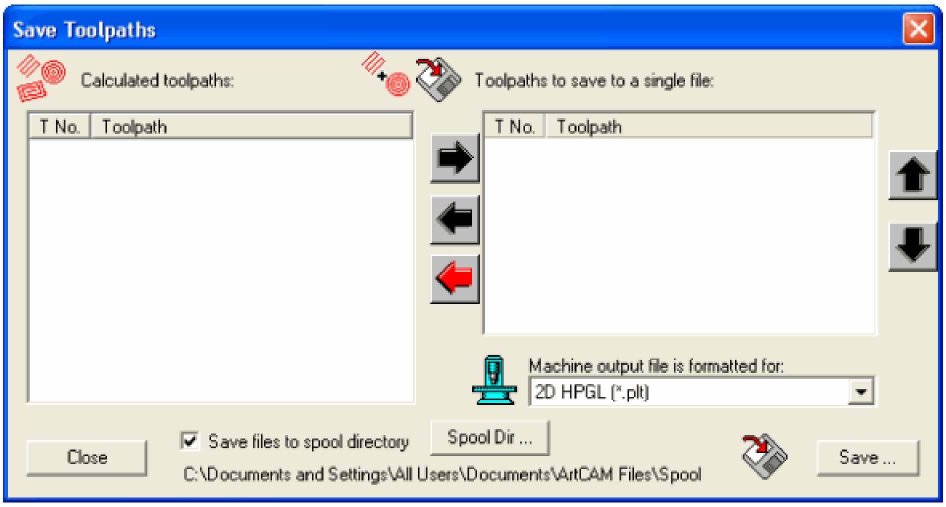
You can create a machine-specific toolpath file from any of your calculated toolpaths. The toolpath file can then be sent directly to your CNC machine tool.

You can create a toolpath template file (***.tpl**) from any of your created or calculated toolpaths. A template contains all of the original settings you have used when creating the toolpaths. The toolpath settings saved within a template can be applied to selected vector artwork drawn across the vector layers in an ArtCAM model. A toolpath template file can be imported into an ArtCAM model.

To save a toolpath as a machine-specific file:

1. Use one of the following methods to display the **Save Toolpaths** dialog box:
   - From the **Project** panel, select **Toolpaths** in the Project Tree, then click the **Save Toolpaths** button in the **Toolpath Operations** area below the splitter bar;
   - From the **Project** panel, right-click **Toolpaths** in the Project Tree, then select **Save Toolpaths As** from the context menu;
   - From the **Menu Bar**, click **Toolpaths > Save Toolpaths As**;
   - From the **Toolpaths** panel, click the **Save Toolpaths** button in the **Toolpath Operations** area.

*For information on using the **Toolpaths** panel, see Understanding the Toolpaths panel (see page 32).*




2. In the **Calculated toolpaths** window, select the toolpath you want to save as a single machine-specific toolpath file. The toolpath's name is highlighted.

   *To select more than one toolpath, hold down the **Shift** key then click each of the toolpaths' names.*

3. Click the right arrow button to transfer all selected toolpaths to the **Toolpaths to save to a single file** window.

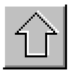
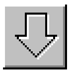
   *The greyed-out right arrow button is displayed when all toolpaths are listed in the **Toolpaths to save to a single file** window.*

4. Make sure that the toolpaths are listed in the correct machining order:

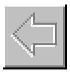
   - Click the up arrow or down arrow button to set the order in which the toolpaths are saved. Each click on these buttons moves the selected toolpath one position in the list. The name of the selected toolpath is highlighted.

   *The greyed-out down arrow button is displayed if the toolpath at the bottom of the list shown in the **Toolpaths to save to a single file** window is selected.*

   *The greyed-out up arrow button is displayed if the toolpath at the top of the list shown in the **Toolpaths to save to a single file** window is selected.*

*Both the*  *and*  *button are displayed when all toolpaths are listed in the* ***Calculated toolpaths*** *window.*

- Click  to transfer the toolpath selected in the **Toolpaths to save to a single file** window back to the **Calculated toolpaths** window. The name of the selected toolpath is highlighted.
- Click  to transfer all toolpaths listed in the **Toolpaths to save to a single file** window to the **Calculated toolpaths** window.

*The*  *and*  *buttons are greyed-out*  *when all toolpaths are listed in the* ***Calculated toolpaths*** *window.*

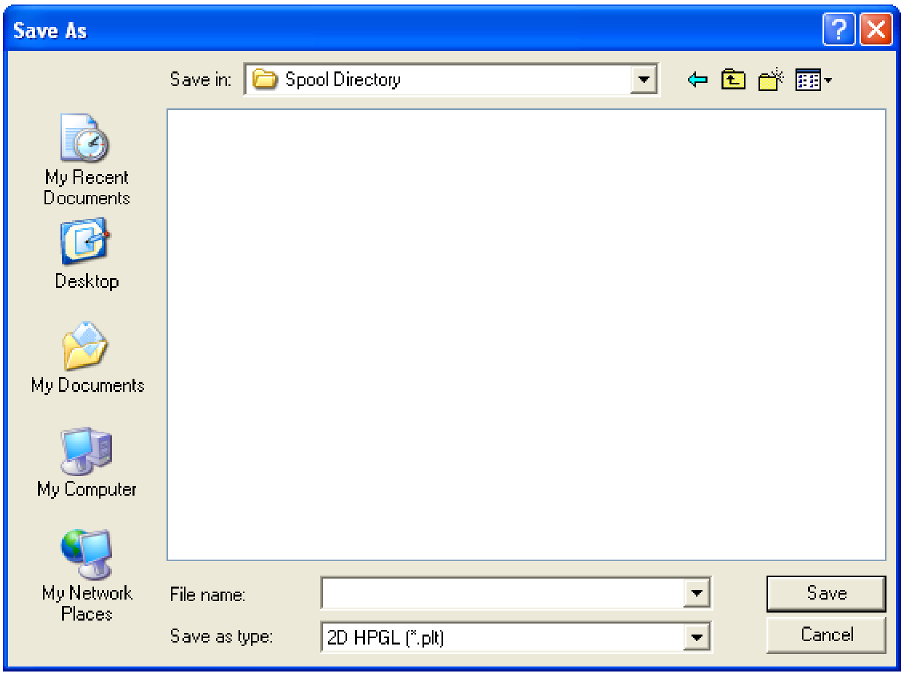
5. Click the **Machine output file is formatted for** list box, followed by the machine format you require.

   *If you are saving toolpaths that make use of a variety of tools in a single output file, you must use a machine that has been formatted with a tool changer.*

6. To save the machine-specific toolpath file to a specific spool directory on your computer, select the **Save files to spool directory** check box.

   *If you have not selected a spool directory, click the* ***Spool Dir...*** *button to display the* ***Browse For Folder*** *dialog box. Create or select the folder on your computer where you want to save your machine-specific toolpath files, and then click* ***OK*** *to confirm it as the spool directory.*

7. Click **Save** to display the **Save As** dialog box:




You can see that the machine format that you chose from the **Machine output file is formatted for** list box in the **Save Toolpaths** dialog box is shown in the **Save as type** list box.

8. Click the **Save in** list box or use the **Up One Level** button  to navigate to the folder on your computer in which you want to save the machine-specific toolpath file.

*If the **Save files to spool directory** option is selected in the **Save Toolpaths** dialog box, its location is shown in the **Save in** list box.*

9. In the **File name** box, type the name that you want to give to the machine-specific toolpath file.

10. Click **Save** to save the toolpaths listed in the **Toolpaths to save to a single file** window as a machine-specific toolpath file and close the **Save As** dialog box.

11. Click **Close** to close the **Save Toolpaths** dialog box.

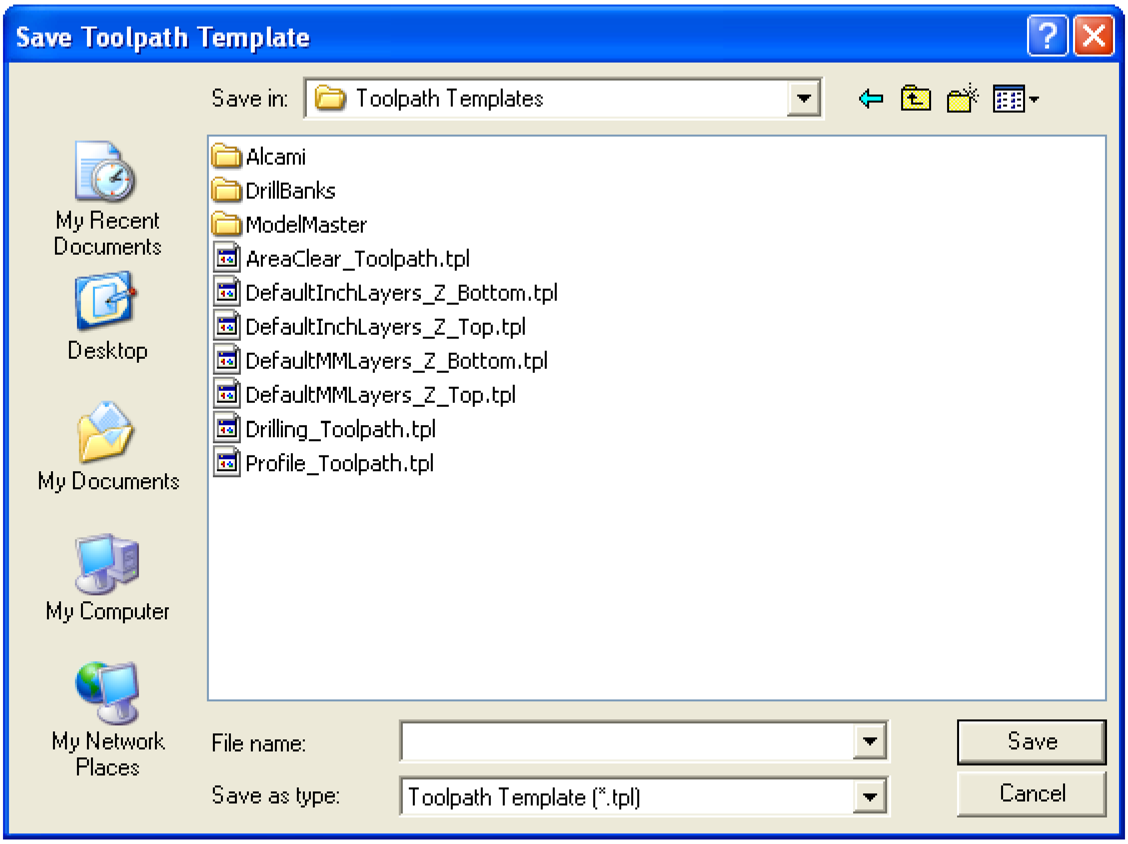
*To close the **Save As** dialog box without saving the machine-specific toolpath file, click **Cancel**.*

To save a toolpath as a template file:

1. On the **Project** panel or **Toolpaths** panel, make sure that only the toolpaths you want to save as a template are listed. The toolpaths can be created or calculated.

*In the **Project** panel's Project Tree and on the **Toolpaths** panel, the names of calculated toolpaths are listed in black. The names of uncalculated toolpaths are listed in red.*

2. Use one of the following methods to display the **Save Toolpath Template** dialog box:
   - From the **Project** panel, select **Toolpaths** in the Project Tree, then click the **Save Toolpaths As Template** button in the **Toolpath Operations** area below the splitter bar;
   - From the **Project** panel, right-click **Toolpaths** in the Project Tree, then select **Save Toolpaths As Template** from the context menu;
   - From the **Menu Bar**, click **Toolpaths > Save Toolpaths As Template**; or
   - From the **Toolpaths** panel, click the **Save Toolpath As Template** button in the **Toolpath Operations** area.



3. Click the **Save in** list box or use the **Up One Level** button to navigate to the folder on your computer in which you want to save the template.
4. In the **File name** box, specify the name you want to give to the template.
5. Click **Save** to close the **Save Toolpath Template** dialog box and save the template.

   *To close the* ***Save Toolpath Template*** *dialog box without saving the template, click* ***Cancel****.*

# Simulating toolpaths

You can simulate calculated toolpaths in the **2D View** and **3D View** window. This enables you to envision the machining passes used to create your finished piece.

*In the **Project** panel's Project Tree and on the **Toolpaths** panel, the names of calculated toolpaths are listed in black. The names of uncalculated toolpaths are listed in red.*

In the **2D View** window, you can simulate calculated 2D toolpaths as solid colours. These toolpaths include: **Profile**, **Area Clearance**, **V-Bit Carving**, **Bevel Carving**, **Smart Engraving**, **Drilling**, **Inlay**, **Raised Round**, **Texture** and **Drill Banks**.

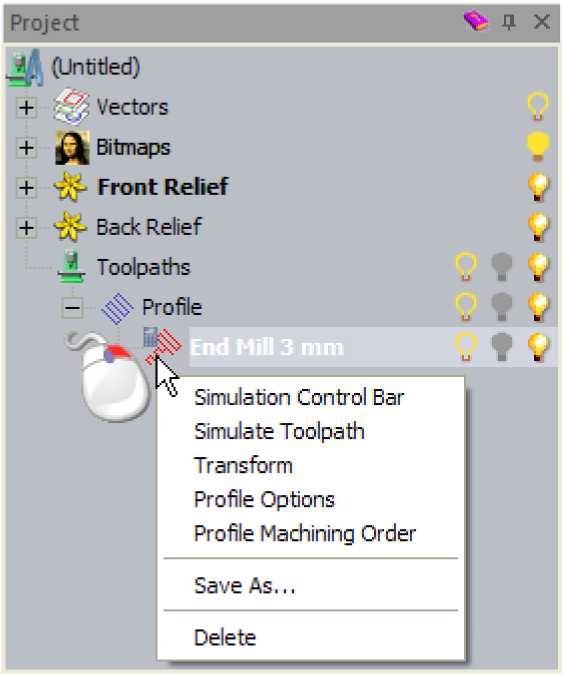
A solid colour toolpath simulation is a more informative display of a 2D toolpath than that offered by its default wireframe preview. For example, where each of the tools in a calculated toolpath is assigned a different colour, a solid colour simulation might look something like this:

In the **3D View** window, you can simulate calculated 2D or 3D toolpaths in a simulation block. You can control how the toolpath simulation is rendered and apply a depth colour to all areas of the simulation below the Z zero height. This enables you to clearly envision the finished piece. For example, the same simulated 2D toolpaths shown above might look something like this:

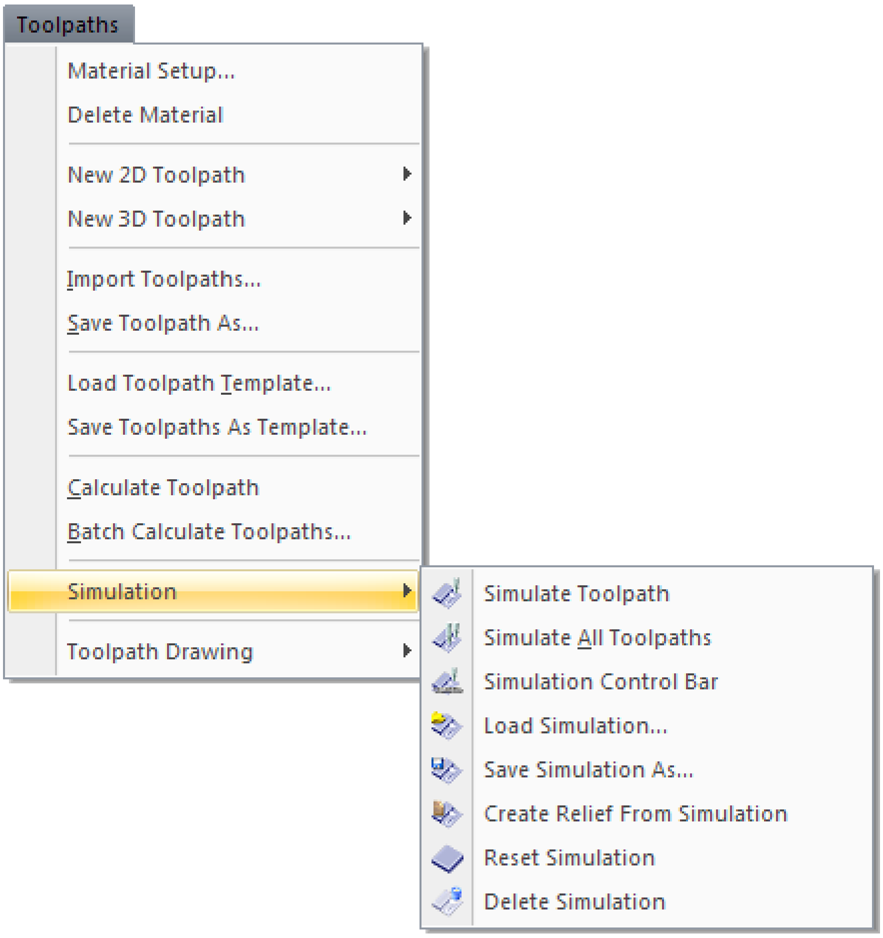

You can simulate calculated toolpaths:

- From the **Project** panel, using the context menu and tools associated with the **Toolpaths**, toolpath and tool items in the Project Tree;

For example, right-clicking a *Profile* toolpath's *End Mill 3 mm* tool displays the following context menu:




- From the **Menu Bar**, using the **Toolpaths > Simulation** menu.



*If you are working in ArtCAM Insignia, the* ***Load Simulation****,* ***Save Simulation As...****, and* ***Create Relief From Simulation*** *options are not included.*

- From the **Simulation** toolbar.


*The* ***Simulation*** *toolbar is hidden (see page 309) by default. When displayed for the first time, it is floating over the viewing area.*

*If you are working in ArtCAM Insignia, the last three buttons are not included.*

- From the **Toolpaths** panel, using the tools in the **Toolpath Simulation** area.


*If you are working in ArtCAM Insignia, the last three buttons are not available.*

*The* ***Toolpaths*** *panel is hidden (see page 309) by default. When displayed it is docked and pinned on the left.*

## Simulating 2D toolpaths in the 2D View window

Calculated 2D toolpaths can be displayed in the **2D View** window as a wireframe preview and a solid colour simulation. They are shown as a wireframe preview only by default. Displaying toolpaths as a solid colour simulation enables you to see the areas of the vector artwork machined by your chosen cutting tools, and how effectively the toolpath reproduces the integrity of your original design.

In the example below, the solid colour simulation on the left confirms you would need to use a tighter tolerance in your toolpath in order to preserve the letter's profile. The image on the right shows the simulation of the same toolpath with a tighter tolerance:

*Letter 'a' using a loose tolerance...*

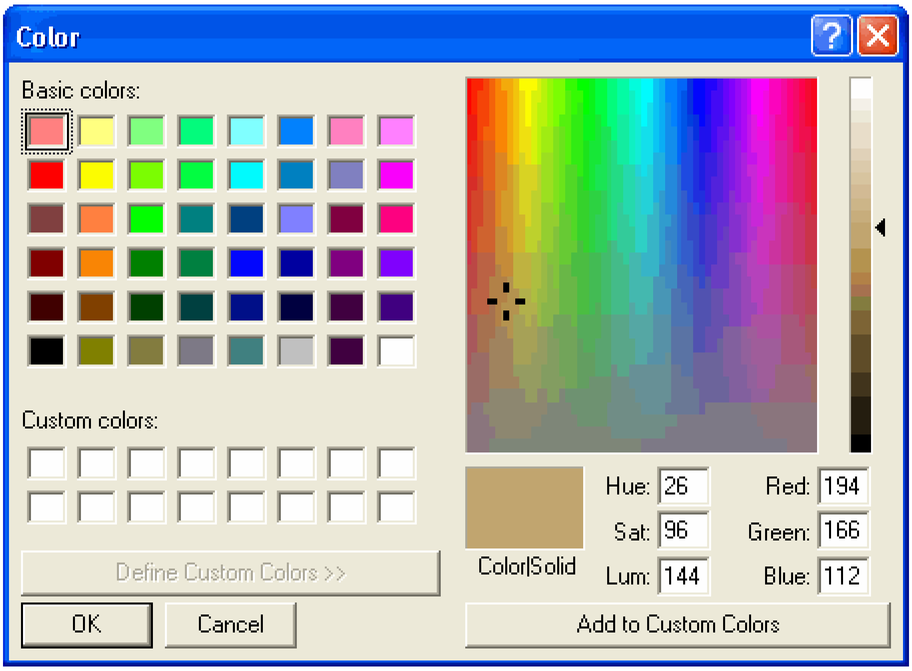
*Letter 'a' using a tighter tolerance...*

Each of the tools used in a calculated 2D toolpath is assigned the colour brown by default, but you can change this. When you assign a different colour to a tool, it is applied to all subsequent toolpaths where the same tool is used. For example, if you assign blue to a *3 mm End Mill* tool, then the simulation for all subsequent toolpaths using a *3 mm End Mill* tool will be displayed in blue. This does not apply retrospectively, so previous toolpaths using the same tool do not use the newly assigned colour.

To control how and what calculated 2D toolpaths are displayed in the **2D View** window:

1. Specify which calculated toolpaths you do not want to display as a wireframe preview, using one of the following methods:
   - From the **Project** panel, click beside the toolpaths or tool names in the Project Tree; or
   - From the **Toolpaths** panel, click beside the tool names listed above the **Toolpath Operations** area.
2. Specify which calculated toolpaths you want to display as a solid colour simulation using one of the following methods:
   - From the **Project** panel, click beside the toolpaths or tool names in the Project Tree; or
   - From the **Toolpaths** panel, click beside the tool names listed above the **Toolpath Operations** area.
3. To associate a colour with a tool, use one of the following methods to display the **Color** box:

- From the **Project** panel, select the tool in the Project Tree, then click the **Simulation colour** swatch in the **Parameters** area displayed below the splitter bar; or
- From the **Toolpaths** panel, click the swatch beside the tool's name in the list above the **Toolpath Operations** area.




For details on using this dialog box, see Assigning a colour to a vector layer (see page 126).

## *Simulating toolpaths in the 3D View window*

You can simulate calculated toolpaths in the **3D View** window in four different ways. You can simulate:

- a specific toolpath;
- a particular tool used in a toolpath;
- a particular region of a toolpath, specified by a selected vector;
- all toolpaths in succession.

*In the **Project** panel's Project Tree and on the **Toolpaths** panel, the names of calculated toolpaths are listed in black, while the names of uncalculated toolpaths are listed in red.*

You can choose between three toolpath simulation methods. Use the:

- **Simulate All Toolpaths** method to display a simulation block and quickly simulate all of your toolpaths.

- **Simulate Toolpath** method to display a simulation block and quickly simulate a specific toolpath or tool used as part of a toolpath.
- **Simulation Control Bar** method to display a simulation block and a toolbar you can use to simulate your toolpaths in greater detail and with far greater control.

When using the **Simulation Control Bar** method, a wireframe tool is displayed to simulate the tool moves. The simulation block is gradually updated to simulate the results of the machining process. For example, the image below illustrates the wireframe of a *V-Bit 32 mm 130 degree* tool machining bevel carved text into the simulation block:


To simulate a specific toolpath or particular tool used as part of a toolpath:

1. Use one of the following methods to select the calculated toolpath or tool you want to simulate:
    - From the **Project** panel, click beside **Toolpaths** in the Project Tree to display your toolpaths, then click the toolpath or tool's name.
    - From the **Toolpaths** panel, click the toolpath or tool's name in the list shown above the **Toolpath Operations** area.

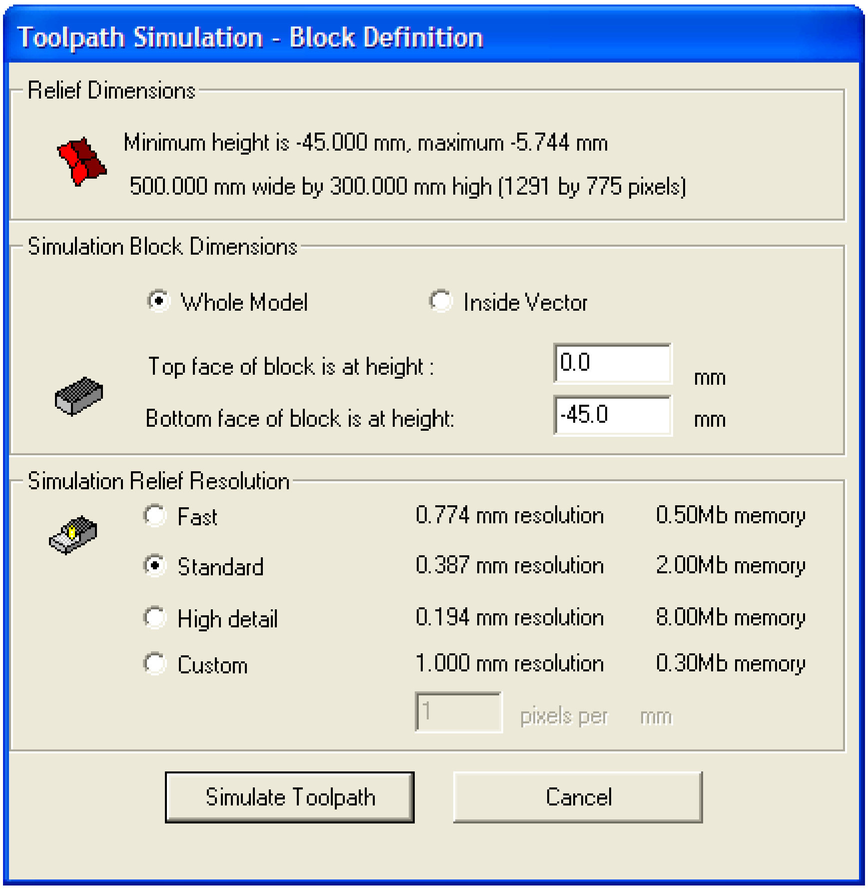
    The selected toolpath or tool's name is highlighted.

2. To simulate the toolpath quickly, use one of the following methods:

- From the **Project** panel, right-click the toolpath or tool's name in the Project Tree, then click **Simulate Toolpath** from the context menu;
- From the **Project** panel, click the **Simulate Toolpath** button in the toolset displayed below the splitter bar;
- From the **Menu Bar**, click **Toolpaths > Simulation > Simulate Toolpath**;
- From the **Simulation** toolbar, click the **Simulate Toolpath** button; or
- From the **Toolpaths** panel, click the **Simulate Toolpath** button in the **Toolpath Simulation** area.

3. To simulate the toolpath in more detail and with greater control, use one of the following methods:
   - From the **Project** panel, right-click the toolpath or tool's name in the Project Tree, then click **Simulation Control Bar** from the context menu;
   - From the **Project** panel, click the **Simulation Control Bar** button in the toolset displayed below the splitter bar;
   - From the **Menu Bar**, click **Toolpaths > Simulation > Simulation Control Bar**;
   - From the **Simulation** toolbar, click the **Simulation Control Bar** button; or
   - From the **Toolpaths** panel, click the **Simulation Control Bar** button in the **Toolpath Simulation** area.

The **3D View** window is displayed. If you are simulating for the first time, the **Toolpath Simulation - Block Definition** dialog box is also shown.




The **Relief Dimensions** area displays the composite relief's dimensions and resolution.

The **Simulation Block Dimensions** area enables you to control how much of the toolpath is simulated. By default, the simulation block's height (Y) and width (X) is equal to the model (see page 57) area, while its thickness and Z zero position are equal to specified in your material block settings.

The **Simulation Relief Resolution** area enables you to set the resolution of the toolpath simulation.

4. Set the simulation block's height (Y) and width (X). Select:
   - **Whole Model** to use the model's dimensions. This is selected by default.
   - **Inside Vector** to use a selected vector's dimensions. You can select the vector (see page 122) from either the **2D View** or **3D View** window.
5. Set the simulation block's thickness (Z):

a. In the **Top face of block is at height** box, specify the Z position of the block's top surface.

b. In the **Bottom face of block is at height** box, specify the Z position of the block's bottom surface.

6. Set the toolpath simulation's resolution. Click:
   - **Fast** to prioritise speed and memory consumption over quality and detail.
   - **Standard** to compromise between quality, detail, speed and memory consumption.
   - **High detail** to prioritise detail and quality over speed and memory consumption.
   - **Custom** to use a specific pixel resolution. The default of 1 pixels per mm equates to a 1.000 mm resolution, which makes use of 0.02Mb of memory. If you increase this to 100 pixels per mm, the resolution increases to 0.010 mm which makes use of 200.00Mb of memory.

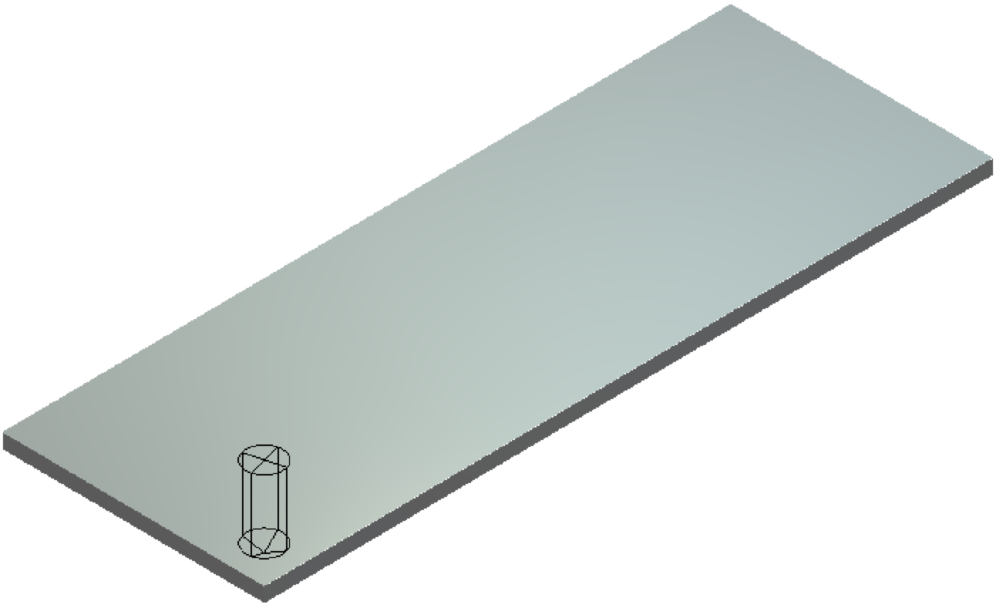
   *Increasing the resolution can increase the toolpath simulation's duration.*

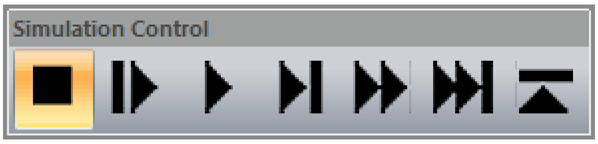
7. Click **Simulate Toolpath** to close the dialog box and begin the toolpath simulation.

   *To change your* ***Simulation Block Dimensions*** *or* ***Simulation Relief Resolution*** *settings, you must delete the toolpath simulation currently shown in the* ***3D View*** *window. For details, see Deleting a simulation (see page 284).*

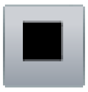
   If you have chosen the **Simulate Toolpath** method, the simulation block is displayed in the **3D View** window, the toolpath simulation begins immediately and continues until it is complete.

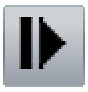
If you have chosen the **Simulate Control Bar** method, the simulation block and wireframe tool is displayed in the **3D View** window with the **Simulation Control** toolbar floating over it.

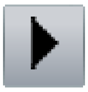


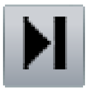



From the **Simulation Control** toolbar, click:

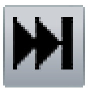
 to pause the simulation.

 to simulate the selected toolpath's next move, gradually updating the simulation block as it does.

 to simulate the selected toolpath, gradually updating the simulation block as it does.

 to simulate the selected toolpath until its next Z retract move, gradually updating the simulation block as it does.

 to quickly simulate the whole of the selected toolpath.

 to quickly simulate the selected toolpath up until its next retract move.

to stop the simulation and close the **Simulation Control** toolbar.

To simulate all of your calculated toolpaths, use one of the following methods:

- From the **Project** panel, right-click **Toolpaths** in the Project Tree, then click **Simulate All Toolpaths** in the context menu;
- From the **Menu Bar**, click **Toolpaths > Simulation > Simulate All Toolpaths**;
- From the **Simulation** toolbar, click the **Simulate All Toolpaths** button; or
- From the **Toolpaths** panel, click the **Simulate All Toolpaths** button in the **Toolpath Simulation** area.

# Resetting a simulation

You can reset a completed toolpath simulation displayed in the **3D View** window. You cannot reset a simulation when it is being calculated.

When a simulation is reset, the simulation block is kept, but your simulated toolpaths are lost.

Use one of the following methods to reset a simulation:

- From the **Project** panel, right-click **Toolpaths** in the Project Tree, then click **Reset Simulation** in the context menu;
- From the **Menu Bar**, click **Toolpaths > Simulation > Reset Simulation**;
- From the **Simulation** toolbar, click the **Reset Simulation** button; or

  *The **Simulation** toolbar is hidden by default. When displayed for the first time, it is floating over the viewing area.*

- From the **Toolpaths** panel, click the **Reset Simulation** button in the **Toolpath Simulation** area.

*The **Toolpaths** panel is hidden by default. When displayed for the first time, it is docked and pinned on the left.*

# Deleting a simulation

You can delete a completed toolpath simulation from the **3D View** window. You cannot delete a simulation when it is being calculated.

Use one of the following methods to delete a simulation:

- From the **Project** panel, right-click **Toolpaths** in the Project Tree, then click **Delete Simulation** in the context menu;
- From the **Menu Bar**, click **Toolpaths > Simulation > Delete Simulation**;
- From the **Simulation** toolbar, click the **Delete Simulation** button; or

  *The **Simulation** toolbar is hidden by default. When displayed for the first time, it is floating over the viewing area.*

- From the **Toolpaths** panel, click the **Delete Simulation** button in the **Toolpath Simulation** area.

  *The **Toolpaths** panel is hidden by default. When displayed for the first time, it is docked and pinned on the left.*

# Saving a toolpath simulation

When a toolpath simulation is complete, you can save the simulation in one of the following file types:

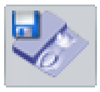
- ArtCAM relief (***.rlf**);
- ArtCAM 9 relief (***.rlf**);
- Windows or OS/2 Bitmap (***.bmp**); or
- 16-bit Tagged Image File Format (***.tif**).

*If you are working in ArtCAM Insignia, you cannot save a toolpath simulation.*

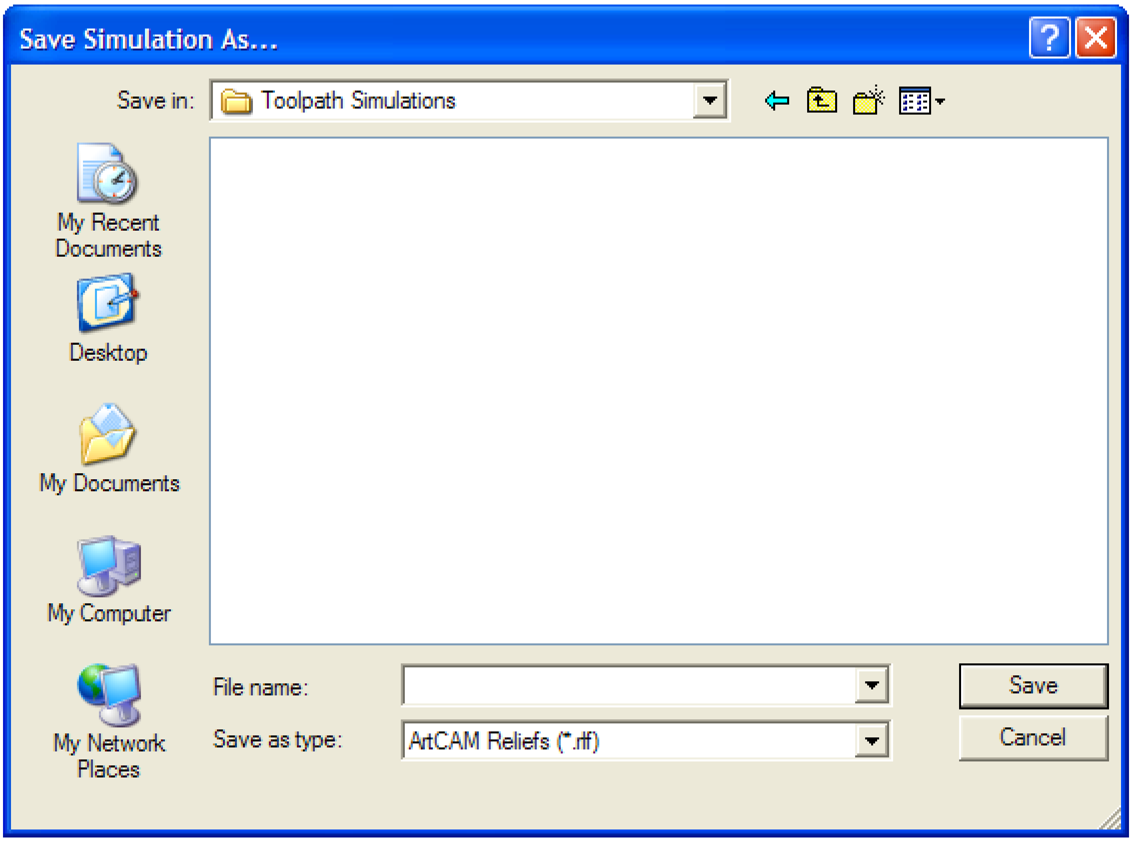
To save a toolpath simulation:

1. Use one of the following methods to display the **Save Simulation As...** dialog box:
   - From the **Project** panel, right-click **Toolpaths** in the Project Tree, then click **Save Simulation As...** in the context menu;
   - From the **Menu Bar**, click **Toolpaths > Simulation > Save Simulation As...** ;
   - From the **Simulation** toolbar, click the **Save Simulation**  button; or

     *The **Simulation** toolbar is hidden by default. When displayed for the first time, it is floating over the viewing area.*

   - From the **Toolpaths** panel, click the **Save Simulation**  button in the **Toolpath Simulation** area.

     *The **Toolpaths** panel is hidden by default. When displayed for the first time, it is docked and pinned on the left.*



2. Click the **Save in** list box, or use the **Up One Level** button, to navigate to the folder on your computer in which you want to save the toolpath simulation.
3. In the **File name** box, specify the name that you want to give to the file.
4. Click the **Save as type** list box, followed by the format in which you want to save the toolpath simulation (***.rlf**, ***.bmp** or ***.tif**).
5. Click **Save** to save the toolpath simulation and close the dialog box.

# Loading a toolpath simulation from a relief

You can load a simulation from an ArtCAM relief file (***.rlf**). The resulting simulation is displayed in the **3D View** window.

*If you are working in ArtCAM Insignia, you cannot load a simulation from an ArtCAM relief file.*

If you load a toolpath simulation when another is already displayed in the **3D View** window, the current simulation is replaced.

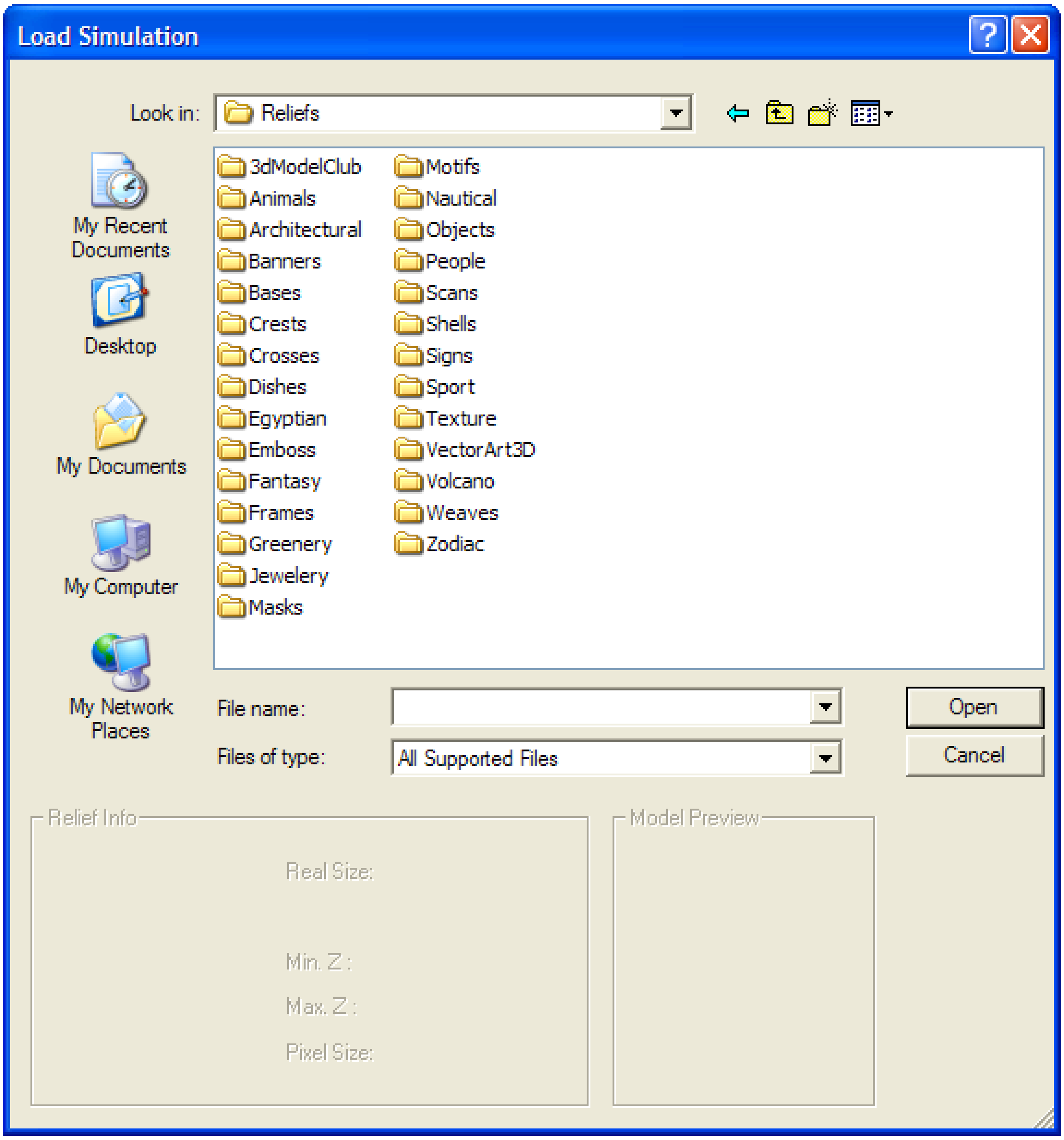
To load a simulation from an ArtCAM relief:

1. Use one of the following methods to display the **Load Simulation** dialog box:
   - From the **Project** panel, right-click **Toolpaths** in the Project Tree, then click **Load Simulation...** in the context menu;
   - From the **Menu Bar**, click **Toolpaths > Simulation > Load Simulation...**;
   - From the **Simulation** toolbar, click the **Load Simulation** button; or

   *The **Simulation** toolbar is hidden by default. When displayed for the first time, it is floating over the viewing area.*

   - From the **Toolpaths** panel, click the **Load Simulation** button in the **Toolpath Simulation** area.

*The **Toolpaths** panel is hidden by default. When displayed for the first time, it is docked and pinned on the left.*




2. Click the **Look In** list box, followed by the folder on your computer in which the ArtCAM relief file (***.rlf**) that you want to simulate is saved.
3. When you have found the ArtCAM relief file, click the file name listed in the main window of the dialog box. Its name is displayed in the **File name** box.

*You can only select ArtCAM relief files (***.rlf**). You can confirm this by clicking the **Files of Type** list box.*

In the **Model Preview** area you can see a preview of the selected relief file. A greyscale representation of the selected relief file is displayed in the **Relief Info** area along with its dimensions in pixels.

*When choosing a relief, the greyscale representation shown in the **Relief Information** area provides an indication of the form that a resulting simulation will take. Consider that the brightest areas of the greyscale image reflect the highest areas in the resulting simulation, while the darkest areas reflect the lowest.*

4. Click **Open** to load the ArtCAM relief as a simulation and close the dialog box.

# Creating a relief layer from a simulation

You can create a new relief layer from a toolpath simulation displayed in the **3D View** window. The relief's shape is identical to the simulation block.

*If you are working in ArtCAM Insignia, you cannot create a relief from a toolpath simulation.*

The new relief layer is:

- named *Simulation*;
- active;
- selected;
- added to the layer stack directly above the layer which was previously active;
- given the **Add** combine mode; and
- visible.

To create a relief layer from a toolpath simulation:

1. Use one of the following methods to display the layer stack on which you want to create the relief layer:
   - From the **Project** panel, click beside **Front Relief** or **Back Relief** in the Project Tree; or

- From the **Relief Layers** panel, click the list box, then select the layer stack: **Front Relief** or **Back Relief**.

2. Choose the relief layer (see page 167) above which you want to create the relief layer.
3. Use one of the following methods to create the relief layer from the toolpath simulation:
   - From the **Simulation** toolbar, click the **Create Relief Layer**  button;

     *The **Simulation** toolbar is hidden by default. When displayed for the first time, it is floating over the viewing area.*

   - From the **Menu Bar**, click **Toolpaths > Simulation > Create Relief From Simulation**; or
   - From the **Toolpaths** panel, click the **Create Relief Layer**  button in the **Toolpath Simulation** area.

     *The **Toolpaths** panel is hidden by default. When displayed for the first time, it is docked and pinned on the left.*

# Shading a toolpath simulation

You can control how a toolpath simulation is displayed in the **3D View** window. You can:

- render the simulation block with a material; and
- colour all areas of the toolpath simulation below the simulation block's top surface with the primary colour.

  *The simulation block's default material is* Simulation Default, *and the default primary colour is black.*

For example, in the toolpath simulation shown below, a *Light Oak (H)* material and custom depth colour is selected:


To specify how the simulation block is displayed in the **3D View** window:

1. In the **3D View** toolbar, click the **3D Graphics Options**  button to display the **3D Graphics Options** panel.
2. Click ▼ on the **Simulation Rendering** control bar to display its settings.
3. To specify the shading set used to render the simulation block in the **3D View** window:
    a. Click the **Material** list box, then the name of the shading set you want to use. For example, *Light Oak (H)*.
    b. Click **Apply** to render the simulation block with the selected shading set.
4. To colour all areas of the toolpath simulation below the simulation block's top surface:
    a. Select the primary colour (see page 109).
    b. Select the **Depth Colour** check box.
    c. Click **Apply** to colour the simulation block.
5. Click **Close** to close the **3D Graphics Options** panel.

# Toggling the simulation display

You can control the display of a:

- solid colour simulation in the **2D View** window associated with calculated 2D toolpaths and the tools they use.
- toolpath simulation in the **3D View** window associated with calculated 2D and 3D toolpaths and the tools they use.

### *Toggling solid colour simulations in the 2D View window*

When a 2D toolpath is calculated, you can control the display of solid colour simulations in the **2D View** window:

1. To toggle the visibility of all solid colour simulations:
   - From the **Project** panel, click beside **Toolpaths** in the Project Tree to hide all solid colour simulations.
   - From the **Project** panel, click beside **Toolpaths** in the Project Tree to display all solid colour simulations
   - From the **Toolpaths** panel, click the uppermost above the list of toolpaths to hide all solid colour simulations.
   - From the **Toolpaths** panel, click the uppermost above the list of toolpaths to display all solid colour simulations.
2. To toggle the visibility of a specific toolpath's solid colour simulation:
   - From the **Project** panel, click beside the toolpath's name in the Project Tree to hide its solid colour simulation.
   - From the **Project** panel, click beside the toolpath's name in the Project Tree to display its solid colour simulation.

   When you have two or more calculated toolpaths and one solid colour simulation is currently hidden, is displayed beside **Toolpaths** in the Project Tree and at the top of the **Toolpaths** panel.
3. To toggle the visibility of a specific tool's solid colour simulation:
   - From the **Project** panel, click beside the tool's name in the Project Tree to hide its solid colour simulation.

- From the **Project** panel, click beside the tool's name in the Project Tree to display its solid colour simulation.
- From the **Toolpaths** panel, click beside the tool's name to display its solid colour simulation.
- From the **Toolpaths** panel, click beside the tool's name to display its solid colour simulation.

When a toolpath uses two or more tools and at least one tool's solid colour simulation is currently hidden, is displayed beside the toolpath's name in the Project Tree and at the top of the **Toolpaths** panel.

### *Toggling toolpath simulations in the 3D View window*

When a toolpath simulation is complete, you can control the display of the simulation block in the **3D View** window.

You cannot control the display of the simulation block:

- before simulating a toolpath;
- during the toolpath simulation process; or
- after deleting the toolpath simulation.

Use one of the following methods to hide the simulation block:

- In the **3D View** toolbar, click the **Toggle Simulation Visibility**  button;
- Right-click the **3D View** window, then deselect **Objects To Draw > Simulation** in the context menu; or
- In the **3D View** toolbar, click the **Objects To Draw**  button, deselect **Simulation** in the list box, and then click **Apply**.

Use one of the following methods to display the simulation block:

- In the **3D View** toolbar, click the **Toggle Simulation Visibility**  button;
- Right-click the **3D View** window, then select **Objects To Draw > Simulation** in the context menu; or

- In the **3D View** toolbar, click the **Objects To Draw**  button, select **Simulation** in the list box, and then click **Apply**.

# Customising the user interface

The default user interface layout in ArtCAM 2010 is designed to help you work efficiently, find tools easily, and maximise the viewing area. However, using only a few mouse clicks you can adjust the layout to suit your particular needs and preferences.

You can:

- control how and which panels and toolbars are displayed;
- control the layout of panels and toolbars;
- create and edit toolbars;
- specify keyboard shortcuts; and
- choose a colour scheme.

## Floating a docked panel

The following panels, when displayed, are docked by default: **Toolpaths**, **Assistant**, **Project**, **Start**, **Toolbox**, **Tutorials**, **Live!**, **Relief Layers**, **Bitmap Layers** and **Vector Layers**.

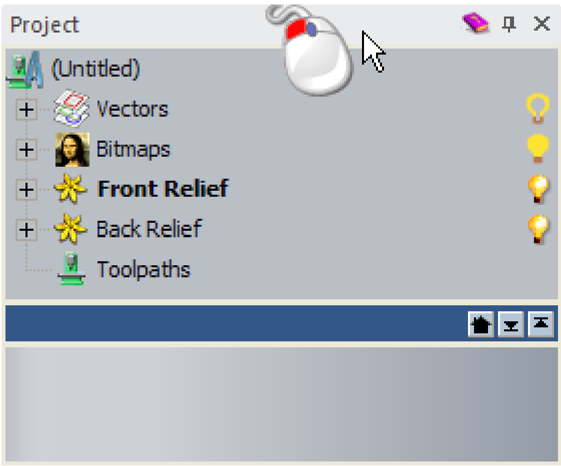
*The **Toolpaths**, **Assistant**, **Relief Layers**, **Bitmap Layers** and **Vector Layers** panel are hidden by default.*

*If you are working in ArtCAM Insignia, the **Relief Layers** panel is not available.*

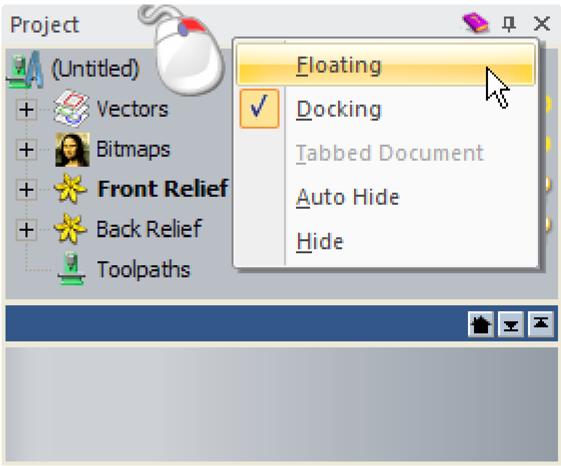
You can choose whether a panel is either docked or floating.

To float a docked panel, use one of the following methods:

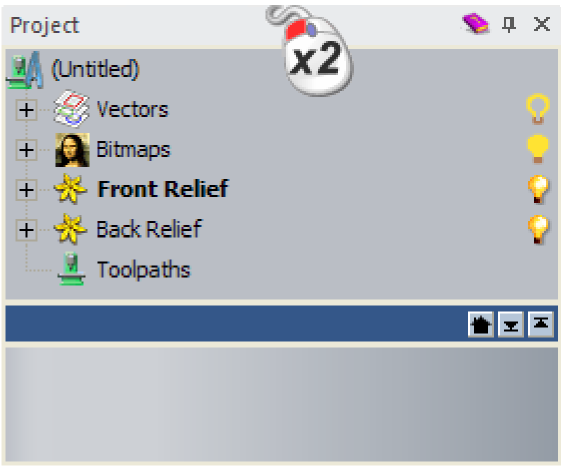
- Click the panel's header and drag to the viewing area, then release the mouse button;




- Right-click the panel's header, then click **Floating** from the context menu; or




- Double-click the panel's header.



# Docking a floating panel

When you create a model, the **Tool Settings** panel is floating by default. The following tools are also displayed on a floating panel when used:

| Tool Category | Panel |
|---|---|
| Model | Lights and Material, Mesh Creator, Options and 3D Graphics Options. |
| Vector Creation | Vector Library, Block and Rotate Copy, Paste Along a Curve, Offset Vectors, Vector From Bitmap, Nesting, Multi-Plate Tool, Create Border, and Create Vector Boundary. |
| Vector Editing | Vector Doctor, Wrap Vectors to Relief, Vector Clipping, Spline Vectors, Fit Arcs to Vectors, Convert to Shapes, and Mirror Vectors. |
| Relief Creation | Extrude, Spin, Turn, Two Rail Sweep, 3D Blend, Add Draft Angle, Create Angled Plane, Relief Clipart Library and 3D Model Unwrapping.<br>*In ArtCAM Insignia, only the Relief Clipart Library and 3D Model Unwrapping panels can be displayed.* |
| Relief Editing | Scale Relief, Fade Relief, Slice Relief and Mirror Merge Relief.<br>*In ArtCAM Insignia, only the Scale Relief panel can be displayed.* |

| Tool Category | Panel |
|---|---|
| Toolpaths | *Each toolpath uses the same panel container. Only one type of toolpath can be displayed on the panel at a time because toolpaths must be calculated sequentially.*<br>Profiling, 2D Area Clearance, V-Bit Carving, Bevel Carving, Smart Engraving, Drilling, Inlay Wizard, Raised Round, Texture Toolpath, Drill Banks, 2D Machining Wizard, Machine Relief, Feature Machining, Z Level Roughing, Laser Machining, 3D Cut Out and 3D Rest Machining.<br>*In ArtCAM Insignia, the Z Level Roughing, 3D Cut Out and 3D Rest Machining panels are not included.* |
| Rotary Relief | Two Rail Sweep - Ring, Create Flat Plane and Correct Vector For Height.<br>*In ArtCAM Pro and Insignia, the Two Rail Sweep - Ring panel is not included.* |
| Gem | Create Gem Vector, Vectors To Gem Vectors, Gem Vector Properties, Create Gems and Pave Wizard.<br>*In ArtCAM Pro and Insignia, none of these panels are included.* |

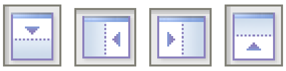
If a panel is floating, you can:

- move it to a new floating position; or
- return it to a previously docked position.

To dock a floating panel:

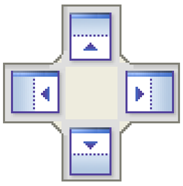
1. Click the panel's header and drag.

   You can use the drop-targets displayed on all four sides of the interface:

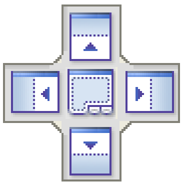


In addition, if the panel is floating over:

- the viewing area, you can use the Docking Assistant displayed in the centre:



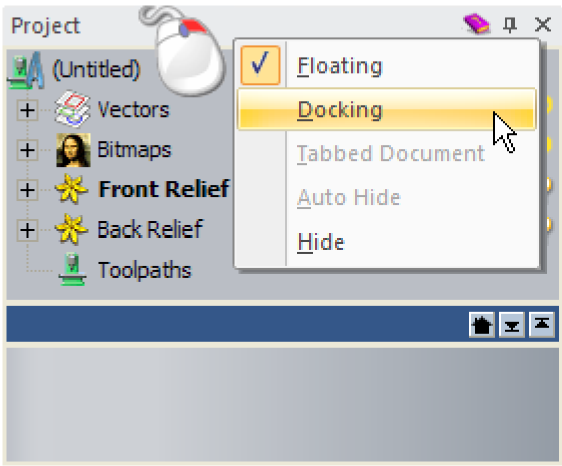
- a docked pinned panel, you can use the Docking Assistant displayed over the panel:



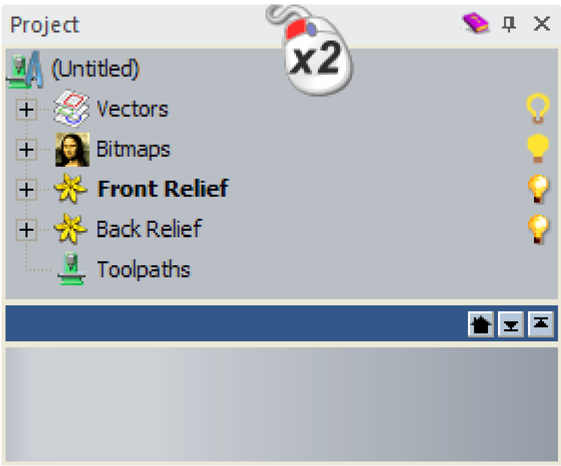
Release the mouse button when the cursor is over the drop-target you want to use.

To return a floating panel to its last docked position, use one of the following methods:

- Right-click the panel's header, then click **Docking** from the context menu; or



- Double-click the panel's header.




# Auto-hiding docked panels

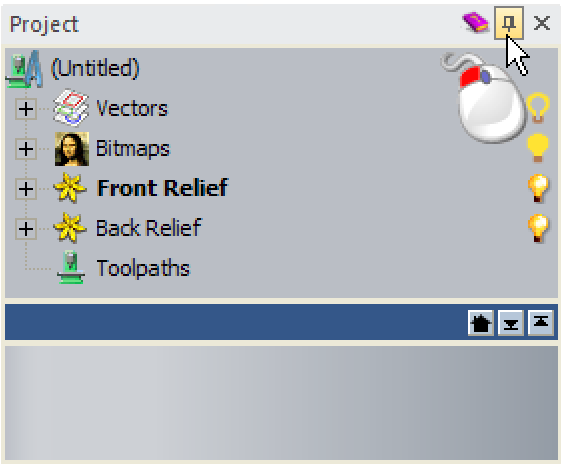
You can control whether or not docked panels are displayed or hidden.

The following panels are auto-hidden by default:

- **Tutorials**;
- **Live!**; and
- **Toolbox**.

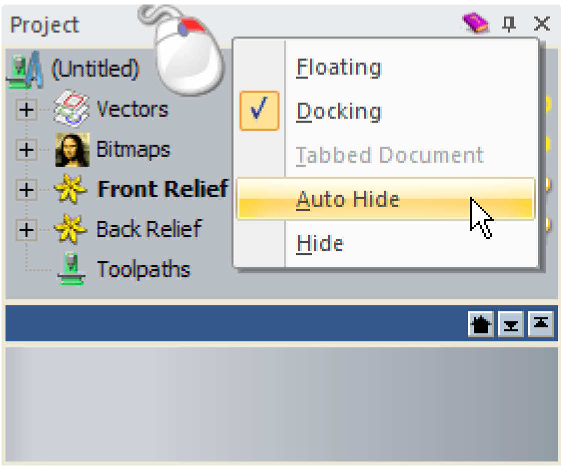
To collapse a docked panel, use one of the following methods:

- Click  on the panel's header; or




*If the docked panel contains tabbed pages, each of its tabs is now displayed as a group of tabs in the docking area. The tab used most recently is emphasised in orange.*

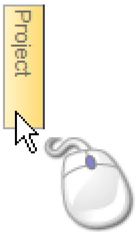
- Right-click the panel's header, then select **Auto Hide** from its context menu.




The panel collapses against its adjacent docking area, and a tab is displayed.

To pin a sliding panel:

1. In the docking area, move the mouse cursor over the tab displaying the name of the panel you want to pin.

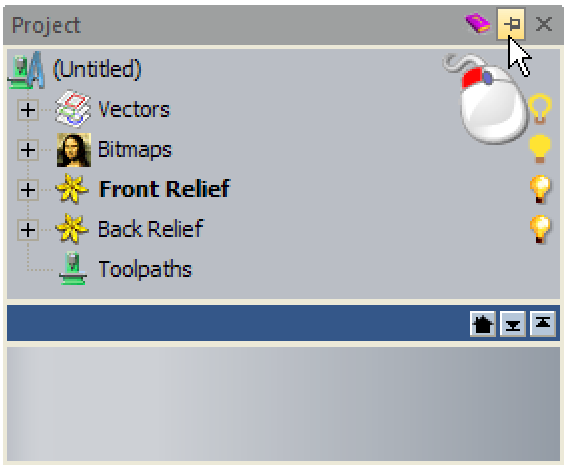



The panel slides out, and is visible for as long as the mouse cursor is over the tab or its associated panel.

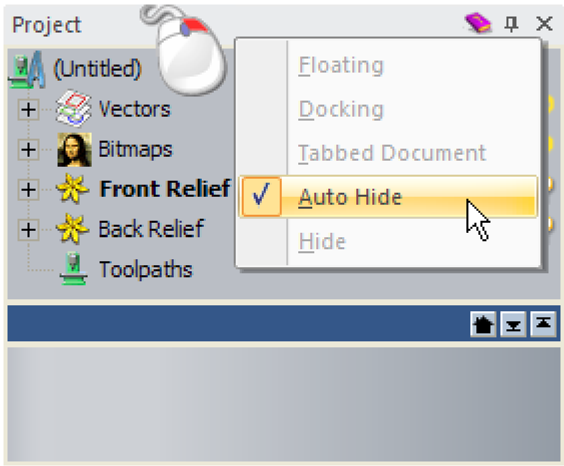
*If you move the mouse cursor outside of the panel or its associated tab, the panel collapses against its adjacent docking area.*

2. Use one of the following methods to pin the panel:

- Click  on the panel's header; or

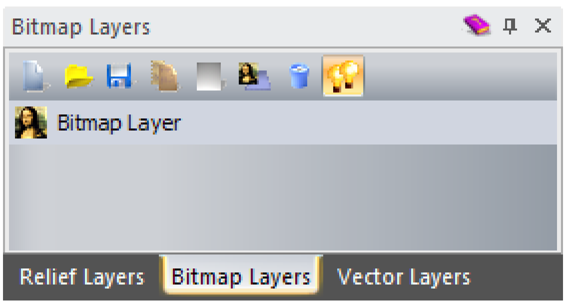



- Right-click the panel's header, then click **Auto Hide** in the context menu.




The panel is docked. If your chosen tab belongs to a group of tabs, the other tabs within the group are also pinned and displayed within the panel as separate tabs. The page associated with your chosen tab is shown in the panel, and its name is displayed on its header.

For example, with the **Bitmap Layers** panel sharing the same container as the **Relief Layers** and **Vector Layers** panels, it looks as follows:



# Hiding and displaying panels

You can control which of the panels are displayed or hidden. You can hide a panel, whether it is pinned or auto-hidden.

The following panels are displayed by default:

- **Start**;

  *The **Start** panel (see page 15) is only displayed before a model or project is created or opened.*

- **Project**;
- **Tool Settings**;
- **Tutorials**;
- **Live!**; and
- **Toolbox**.

The following panels are hidden by default:

- **Assistant**;
- **Relief Layers**;

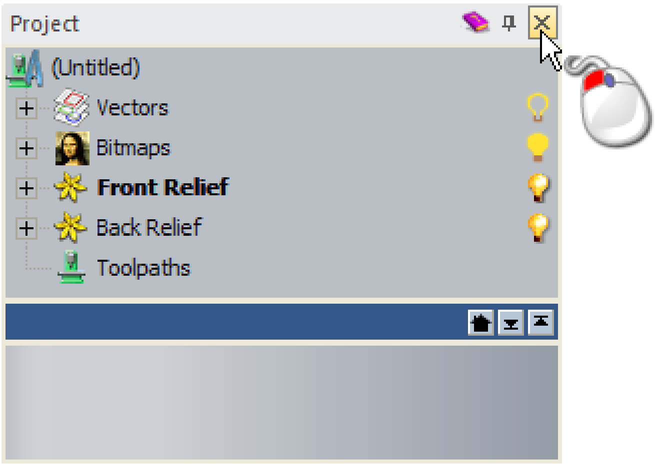
  *The **Relief Layers** panel (see page 41) is not included in ArtCAM Insignia.*

- **Vector Layers**;
- **Bitmap Layers**; and
- **Toolpaths**.

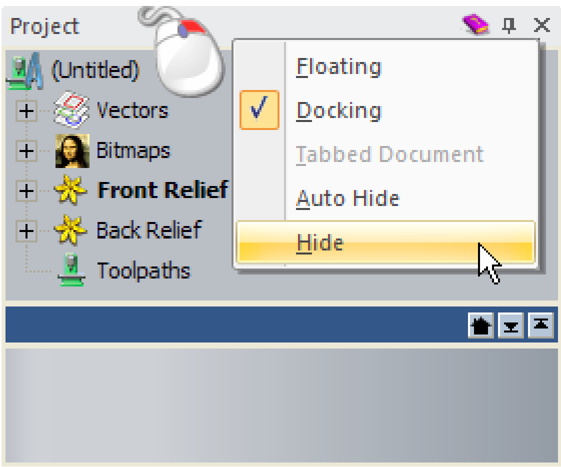
To hide a panel, use one of the following methods:

- Click on the panel's header;

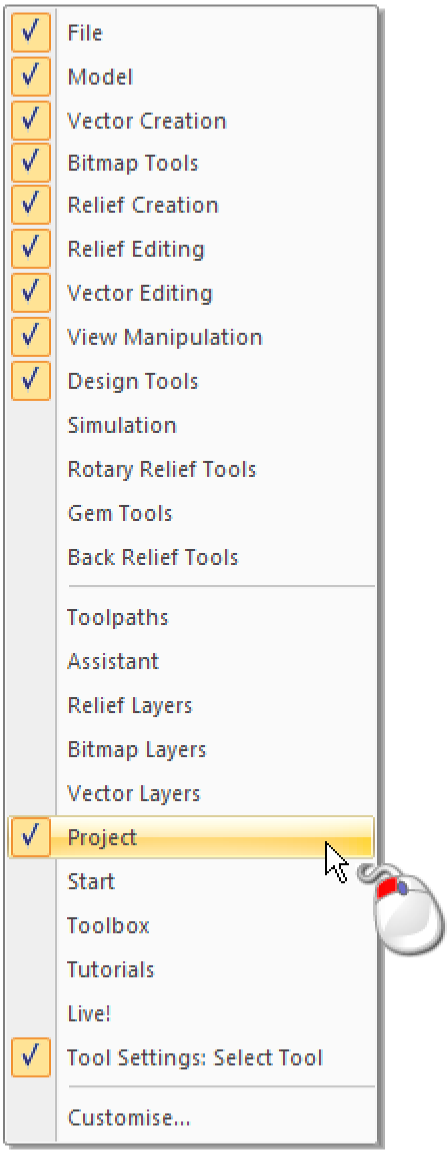
*If the panel is auto-hidden, move the mouse cursor over the tab displaying the panel's name. For details, see Auto-hiding docked panels* *(see page 299).*




- Right-click the panel's header, then click **Hide** in the context menu;



- Right-click a docking area, then click to deselect the panel's name in the context menu; or




*If a panel is auto-hidden, it is not selected in the context menu. Only floating or docked panels can be hidden or displayed in this way.*

- From the **Menu Bar**, click **Window > Toolbars and Docking Windows**, then the selected panel's name in the sub menu.

To display a hidden panel:

- Right-click a docking area, then click to select the panel's name in the context menu; or
- From the **Menu Bar**, click **Window > Toolbars and Docking Windows**, then the deselected panel's name in the sub-menu.

If the panel has not been displayed before, it is shown in its default layout position.

If the panel has been displayed before, it is shown in its last position; docked or floating.

# Embedding panels

Each panel, whether docked or floating, is a container that allows other panels to share the same space. This enables you to use the available workspace efficiently.

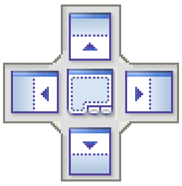
The following panels are embedded by default:

- **Bitmap Layers**;
- **Vector Layers**; and
- **Relief Layers**.

*The **Relief Layers** panel is not included in ArtCAM Insignia.*

To embed one panel within another:

1. Click the panel's header, and drag over to the destination panel. The destination panel's Docking Assistant is displayed:



2. Release the mouse button when the cursor is over:

 to embed the panel below those already displayed.

 to embed the panel above those already displayed.

 to embed the panel to the left of those already displayed.

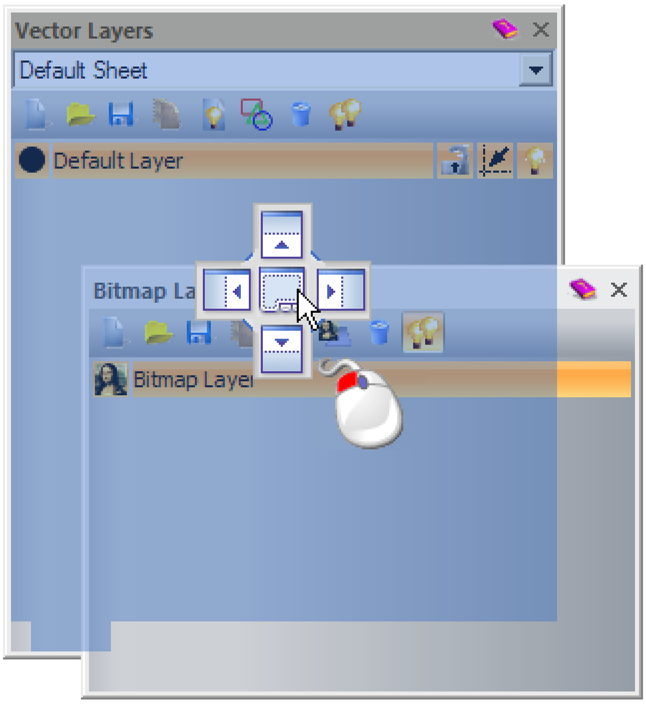
 to embed the panel to the right of those already displayed.

 to embed the panel as a tab.

*If the destination panel is already tabbed, the panel you are relocating is added as a new tab. If not, two new tabs are created. The tab associated with the relocated panel is selected.*

When the mouse cursor is over a drop target in the Docking Assistant, its corresponding space is shaded blue. This provides a preview of the new layout.

For example, positioning the **Bitmap Layers** panel over in the **Vector Layers** panel's Docking Assistant creates the following result:




# Resizing panels

You can resize a floating, docked or embedded panel.

When a panel is too short and narrow to display all of its content, a scrollbar is displayed along its right and bottom edge. You can use the scrollbar to control what of the panel's content is visible.

## *Floating panels*

To resize a floating panel:

1. Move the mouse cursor over a panel's edge or corner. When the cursor changes to:
   - ↔, click and drag left or right to adjust its width;
   - ↕, click and drag up or down ↕ to adjust its height;
   - ⤢ or ⤡, click and drag the corner inwards or outwards diagonally to adjust its height and width simultaneously.

## Docked panels

To resize a docked panel:

1. Move the mouse cursor over the panel's edge adjacent to the viewing area. When the cursor changes to:
   - , click and drag left or right to adjust its width;
   - , click and drag up or down to adjust its height.

## Embedded panels

To resize an embedded panel:

1. Move the mouse cursor over the solid splitter bar between two adjacent embedded panels.

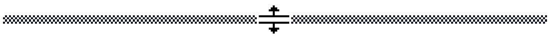
   The splitter bar is horizontal or vertical, depending on how the panels are arranged. For example, a horizontal splitter bar looks as follows:

   *The **Project** panel has its own splitter bar which is always displayed, separating the Project Tree from the tools associated with the currently selected item:*

2. When the cursor changes to:
   - , click and drag up or down to adjust the height of the panels above and below the splitter bar;
   - , click and drag left or right to adjust the width of the panels on the left and right of the splitter bar.

   When moved, the splitter bar is no longer solid. For example, a horizontal splitter bar looks as follows:

   *The **Project** panel's splitter bar includes tools you can use to adjust its layout. Click:*

   *to align the splitter bar with the panel's bottom edge;*

   *to align the splitter bar with the panel's top edge; or*

   *to return the splitter bar to its previous position.*

3. Release the mouse button to set the splitter bar's position. The panels on either side of the splitter bar are resized.

# Floating a docked toolbar

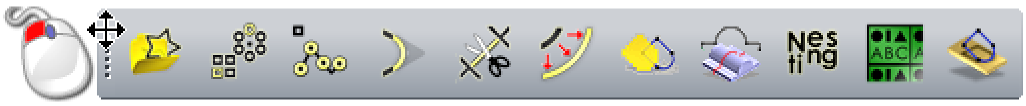
All toolbars are docked by default, but can be floated. When floating, a toolbar has:

- a header displaying its name and icons to edit  or hide ; and
- sharpened corners.

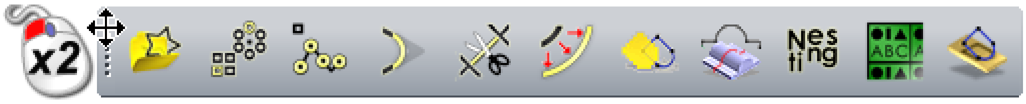
For example, the **Vector Creation** toolbar looks as shown below when floating:


To float a docked toolbar, use one of the following methods:

- Click the toolbar's grip,  or , and drag to the viewing area, then release the mouse button;



- Double-click the toolbar's grip:




  If the toolbar has not been floated before, it is displayed in the top left corner of the interface.

  If the toolbar has been floated before, it is displayed in its last floating position.

# Docking a floating toolbar

You can dock toolbars, default or customised, in the docking area:

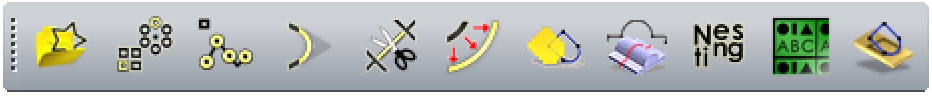
- between the **Title Bar** and viewing area;
- between the **Status Bar** and colour palette;
- to the left of the viewing area; or
- to the right of the viewing area.

Depending on where a toolbar is docked, it is vertical or horizontal. When docked, the toolbar's name is hidden. All toolbars are docked by default.

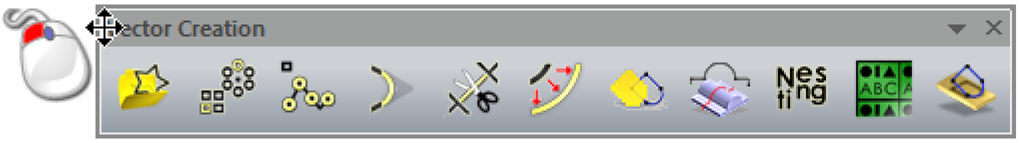
When docked, a toolbar has:

- a grip on its left  or top  edge, depending on whether it is horizontal or vertical respectively; and
- rounded corners.

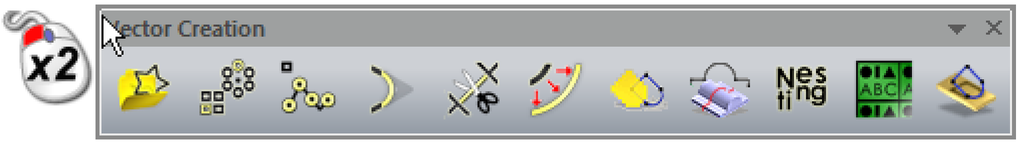
For example, the **Vector Creation** toolbar looks as shown below when docked:



To dock a floating toolbar, click the toolbar's header and drag, then release the mouse button when the toolbar is over the empty docking area.



To return a floating toolbar to its last docked position, double-click the toolbar's header.

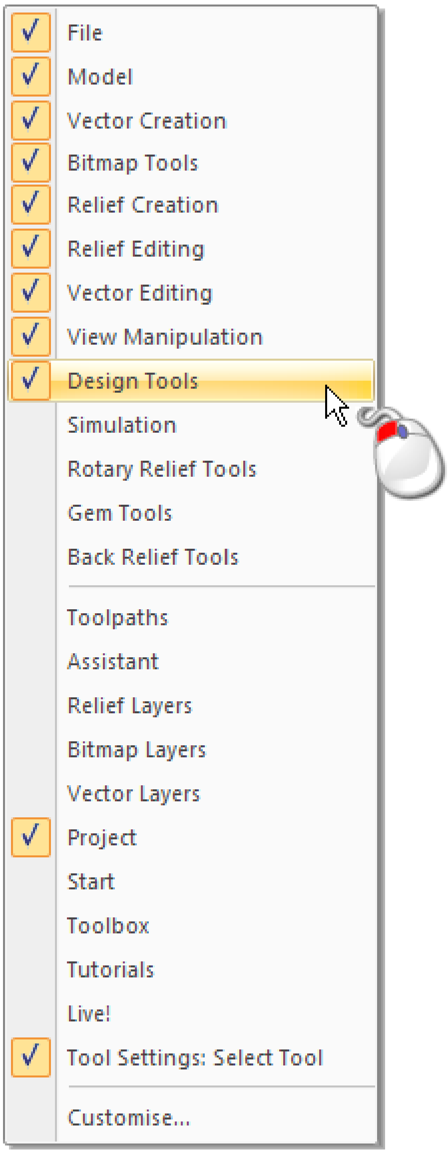



# Hiding and displaying toolbars

You can control which of the toolbars are displayed or hidden. You can hide a toolbar, whether it is currently docked or floating.

To hide a toolbar, use one of the following methods:

- Right-click a docking area, then click to deselect the toolbar's name in the context menu;




- From the **Menu Bar**, click **Window > Toolbars and Docking Windows**, then the selected toolbar's name in the sub-menu; or
- Click ☒ on the toolbar's header.

*A toolbar's header is displayed only when the toolbar is floating* *(see page 308)**.*

To display a hidden toolbar:

- Right-click a docking area, then click to select the toolbar's name in the context menu.
- From the **Menu Bar**, click **Window > Toolbars and Docking Windows**, then the deselected toolbar's name in the sub menu.

If the toolbar has not been displayed before, it is shown in its default layout position.

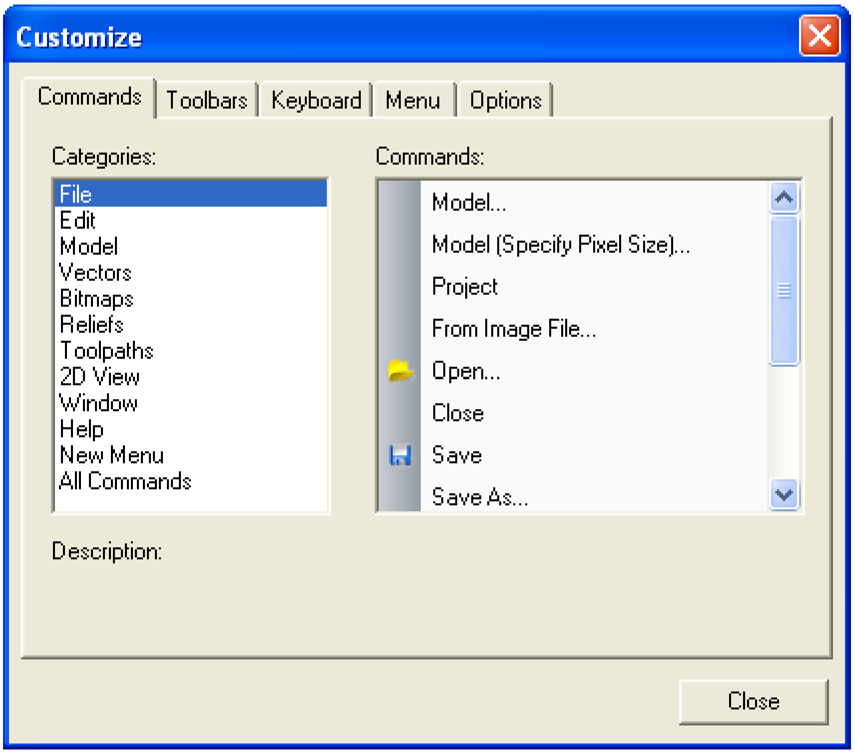
If the toolbar has been displayed before, it is shown in its last position; docked or floating.

# Creating a custom toolbar

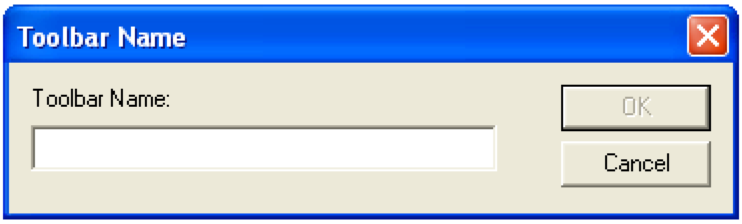
You can create your own toolbars, and add your choice of tools as buttons.

To create a custom toolbar:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Click the **Toolbars** tab to display its settings.
3. Click **New...** to display the **Toolbar Name** dialog box:




4. In the **Toolbar Name** box, type the name that you want to give to the toolbar. For example, *Favourites*.

*The toolbar's name is only displayed when it is floating (see page 308).*

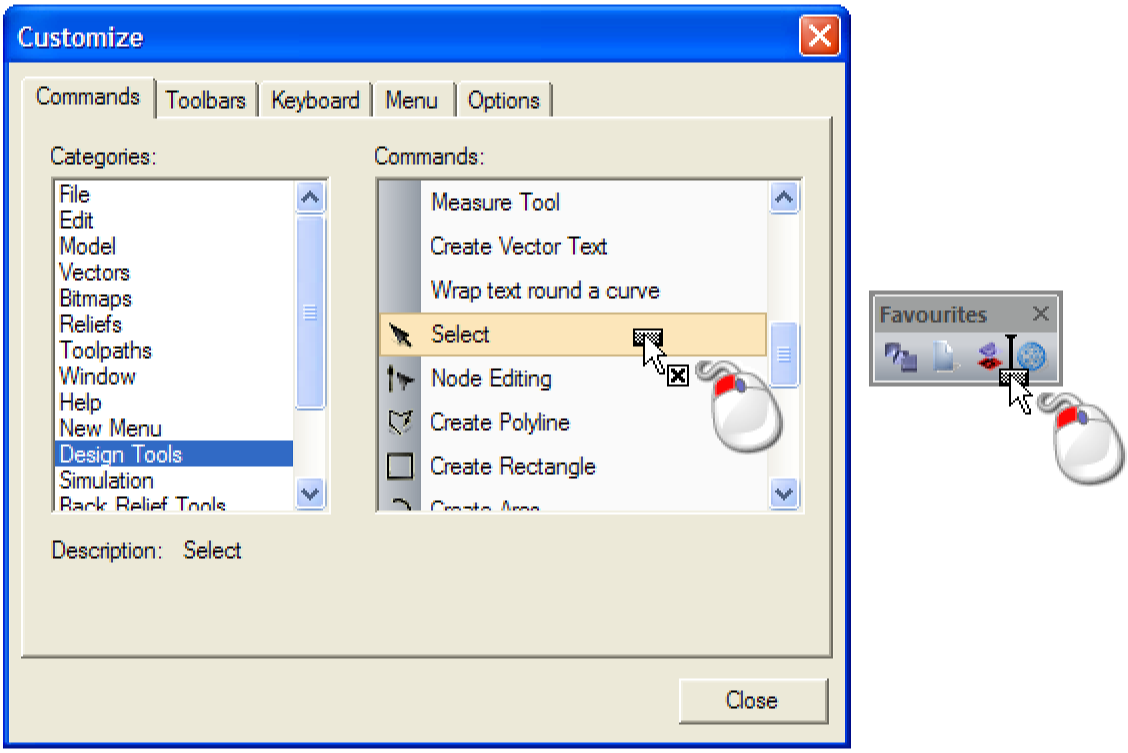
5. Click **OK** to close the **Toolbar Name** dialog box and create the new toolbar. The toolbar is floating and empty. On the **Toolbars** tab of the **Customize** dialog box, the name of the toolbar is selected in the **Toolbars** list.
6. Click the **Commands** tab to display its settings.
7. In the **Categories** list, click to select the category of commands that you want to browse. Its associated commands are displayed in the **Commands** list.

   For example, selecting the **Design Tools** category displays all of the commands you would associate with the buttons on the **Design Tools** toolbar.
8. From the **Commands** list, click and drag the command to the toolbar.

*When clicking and dragging a command, the mouse cursor changes to .*

*When positioned over a toolbar, the mouse cursor changes to .*

For example, click and drag **Select** from the **Commands** list to a custom **Favourites** toolbar.




9. When the mouse cursor is over the toolbar, release the mouse button to add the command as a new button.

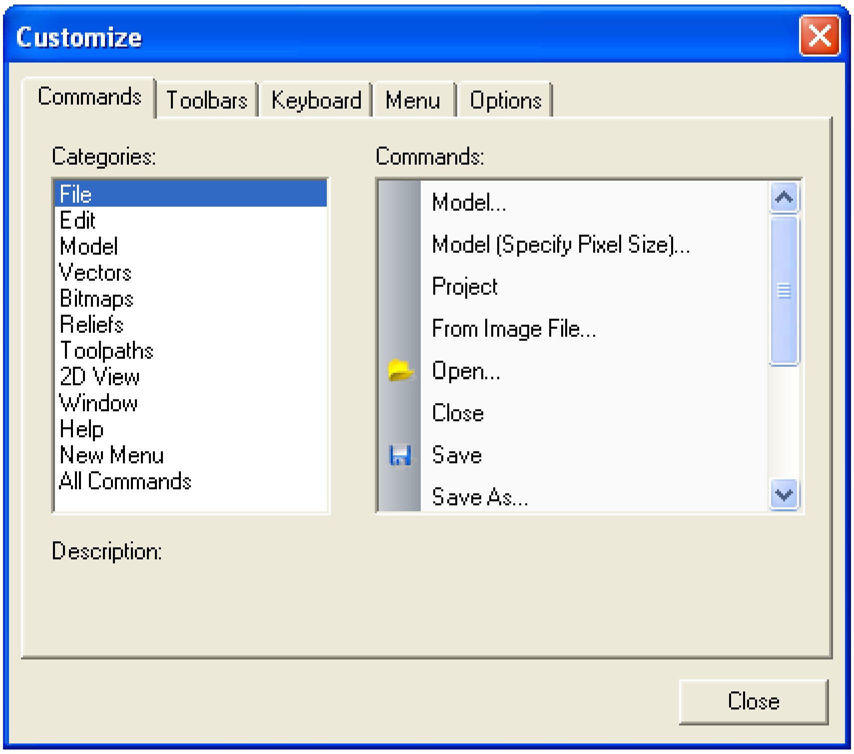
*The* I *icon in the toolbar indicates the position in which the command is added as a button.*

Where there is an icon corresponding with the command, this is used as the button. Where there is no icon, only a text label is displayed.

# Deleting a custom toolbar

To delete a custom toolbar:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Click the **Toolbars** tab to display its settings.
3. In the **Toolbars** list, click to select the toolbar you want to delete. Its name is emphasised in blue.

   *The toolbar you want to delete does not have to be visible.*

4. Click **Delete**.

   *If you have selected any of the default toolbars,* ***Delete*** *is greyed-out.*

# Renaming a custom toolbar

You can rename any of your custom toolbars. You cannot rename any of the default toolbars, which are:

- **2D View**;
- **3D View**;
- **Back Relief Tools**;

  *This toolbar is not included in ArtCAM Insignia.*

- **Bitmap Tools**;

- **Design Tools**;
- **File**;
- **Gem Tools**;

  *This toolbar is not included in ArtCAM Insignia or Pro.*
- **Model**;
- **Relief Creation**;

  *This toolbar is not included in ArtCAM Insignia.*
- **Relief Editing**;

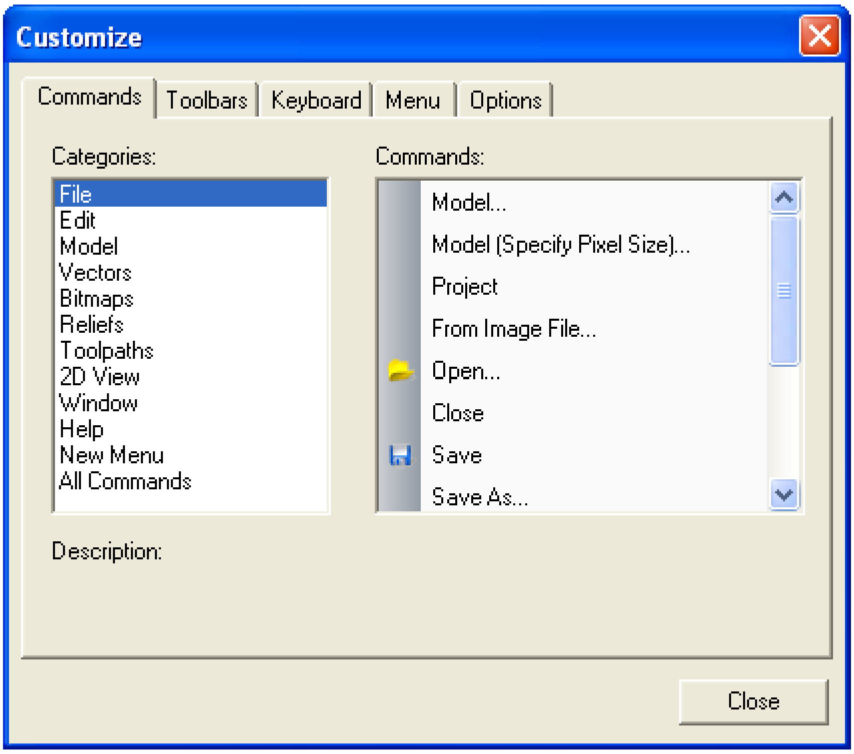
  *This toolbar is not included in ArtCAM Insignia.*
- **Rotary Relief Tools**;

  *This toolbar is not included in ArtCAM Insignia.*
- **Simulation**;
- **Simulation Control**;

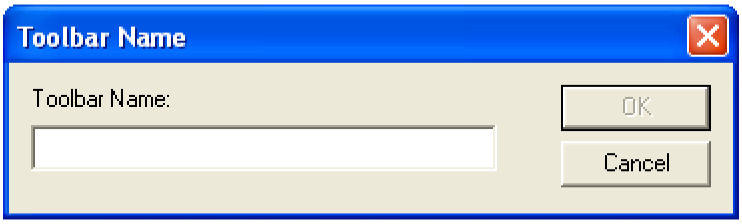
  *This toolbar is not included in ArtCAM Insignia.*
- **Vector Creation**;
- **Vector Editing**; and
- **View Manipulation**.

To rename a custom toolbar:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Click the **Toolbars** tab to display its settings.
3. In the **Toolbars** list, click to select the toolbar that you want to rename. Its name is emphasised in blue.
4. Click the **Rename...** button to display the **Toolbar Name** dialog box:




In the **Toolbar Name** box, the toolbar's name is emphasised in blue, which indicates that it is selected.

5. In the **Toolbar Name** box, type the new name that you want to give to the toolbar.
6. Click **OK** to apply the new name and close the **Toolbar Name** dialog box. The new name given to the toolbar is displayed in the **Toolbars** list. Provided that the toolbar is visible and floating, the new name is also displayed on its header.
7. Click **Close** to close the **Customize** dialog box.

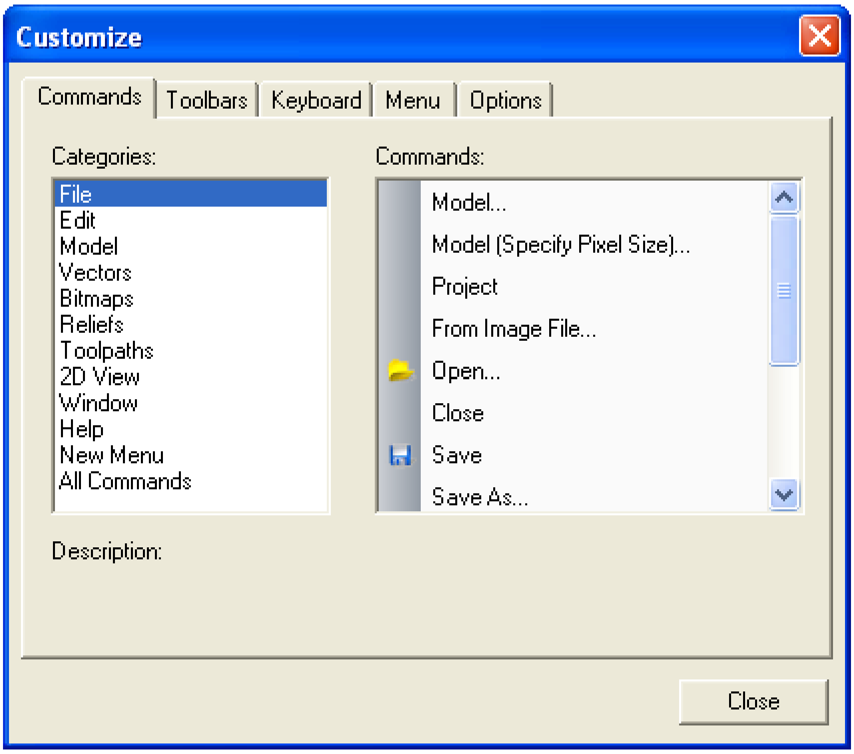
# Adding buttons to a toolbar

You can add buttons to any toolbar: default or custom. These buttons can be:

- copied from another toolbar; or
- created from a list of commands.

To create a copy of a button from one toolbar to another:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Hold down the **Ctrl** key, then click and drag a copy of the button from one toolbar to another.

   For example, copying the **File** toolbar's **New Model** button looks as follows:


3. When the mouse cursor is over the toolbar position in which you want the copied button, release the mouse button.

To add a button using a command:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box.
2. From the **Commands** tab, in the **Categories** window, click the category that includes the command you want to add as a button.
3. In the **Commands** window, click and drag the command to the toolbar.

   *When clicking and dragging a command, the mouse cursor changes to* .

4. When the mouse cursor is over the toolbar position in which you want the command as a button, release the mouse button.

   *The* I *icon in the toolbar indicates the position in which the button is added.*

   Where there is an icon associated with the command, this is used by the button. Where there is no icon, only a text label is shown.

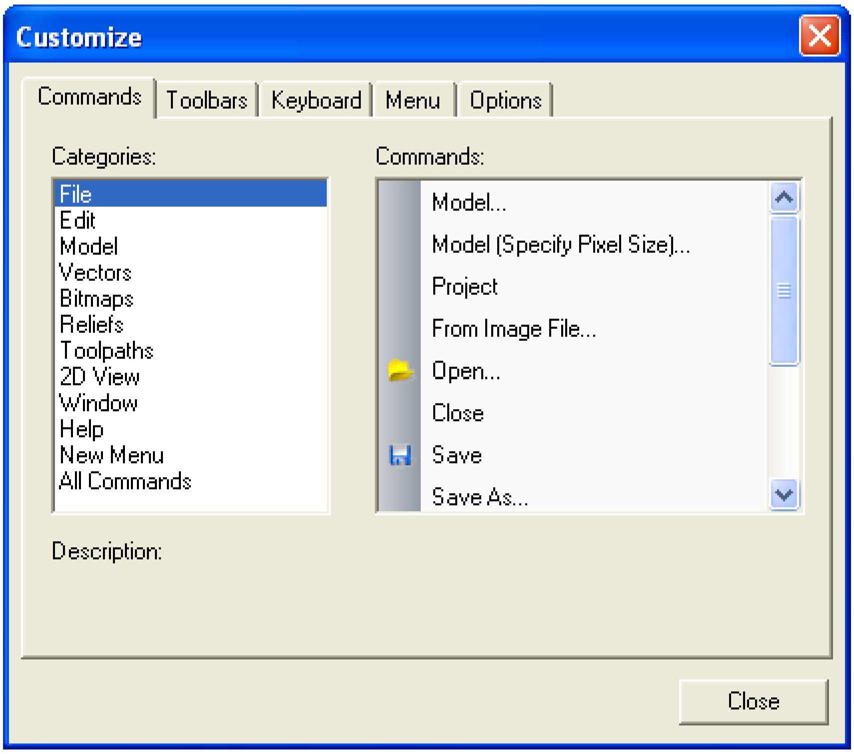
# Setting toolbar button properties

As well as controlling which buttons are included (see page 317) in a toolbar, you can also set their size and toggle the display of their:

- screen tips;
- shortcut keys; and
- text labels.

To set the toolbar button properties:

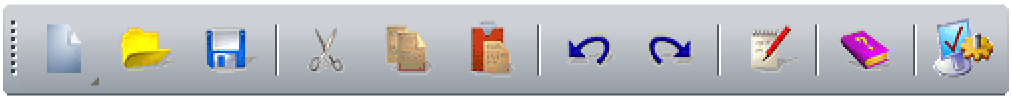
1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Click the **Options** tab, then the:
   - **Large Icons** check box to set the size of all toolbar buttons;

     For example, the **File** toolbar looks as follows:

   *Large...*

   *Small...*

   - **Show Screen Tips on toolbars** check box to toggle the display of screen tips when the mouse cursor is positioned over a toolbar button;

     For example, the **File** toolbar's **New Model** button looks as follows:

   *Screen tip with shortcut....*


*Screen tip only...*


- **Show shortcut keys in Screen Tips** check box to toggle the display of shortcut keys in a toolbar button's screen tip.

*If the **Show Screen Tips on toolbars** check box is deselected, the **Show shortcut keys in Screen Tips** check box is greyed-out.*

3. To toggle the display of text labels on each of the buttons in a specific toolbar:

   a. Click the **Toolbars** tab, then the name of the toolbar in the **Toolbars** window. It is emphasised in blue.

   b. Click the **Show text labels** check box.

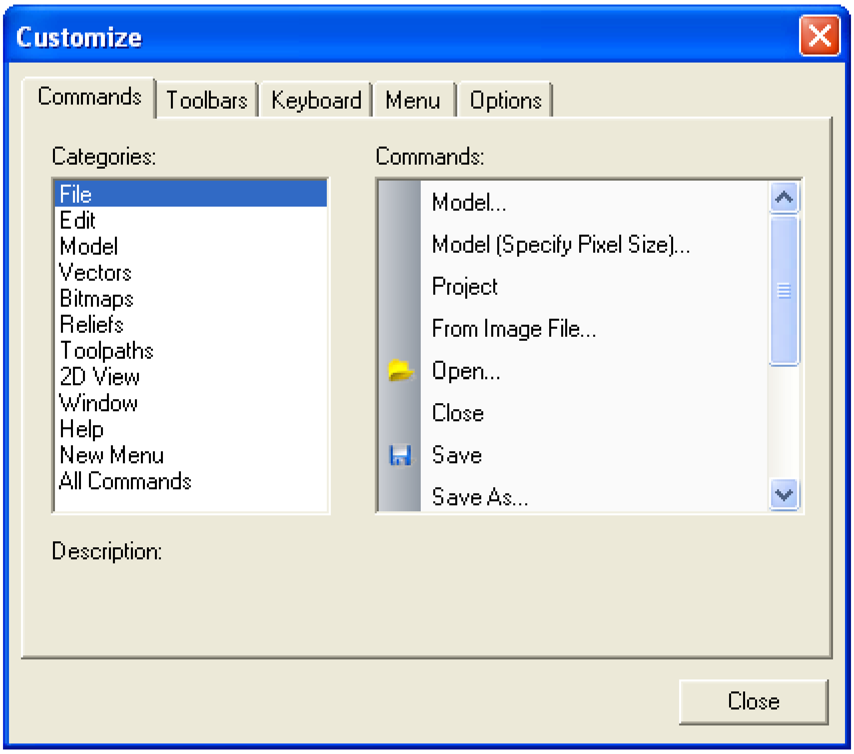
   For example, the **File** toolbar looks as follows with text labels displayed:


4. Click **Close** to close the **Customize** dialog box.

# Adding a custom menu to a toolbar or menu

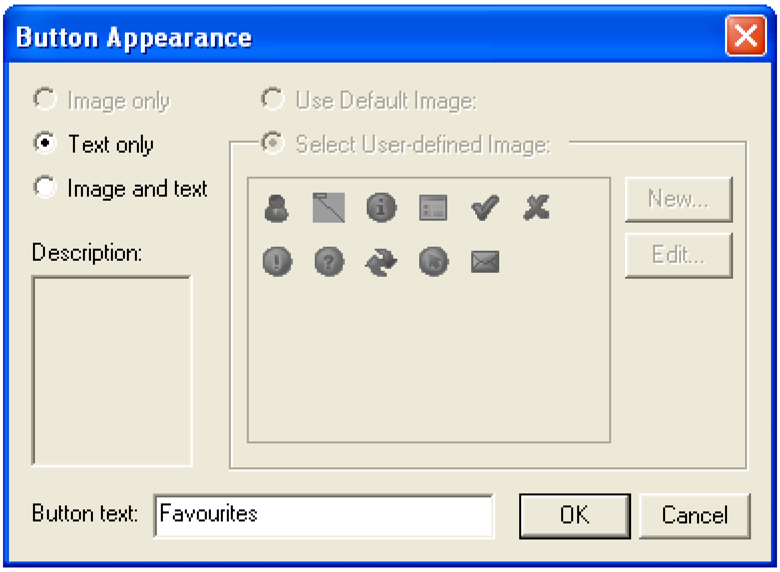
To add a custom menu to a toolbar or menu:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. In the **Categories** list, click **New Menu**. The **New Menu** command is displayed in the **Commands** list.
3. From the **Commands** list, click and drag the **New Menu** command to the toolbar or menu.
4. When the mouse cursor is over the toolbar or menu, release the mouse button. The command is added as a menu button.

5. On the toolbar or menu, right-click the **New Menu** button, then click **Button Appearance...** from the context menu. The **Button Appearance** dialog box is displayed:




6. In the **Button text** box, type the name that you want to give to the menu button. For example, *Favourites*.
7. Click **OK** to apply the new name to the menu button, and close the **Button Appearance** dialog box.
8. In the **Categories** list, click to select the category containing the command you want to add to the menu.
9. From the **Commands** list, click and drag the command to the menu button. An empty menu is displayed.
10. Move the mouse cursor to the empty menu, and release the mouse button to add the command.

    If the command has an associated icon, this is added to the menu. If there is no icon, the menu command is displayed as text only.

    *When clicking and dragging a command, the mouse cursor changes to* .

    *When positioned over a menu, the mouse cursor changes to* .

11. Repeat the last three steps until you have added all of the commands that you want to include in the menu.

    *The* *icon in the menu indicates the position in which the command is added to the menu.*

*The icon in the menu indicates that the command is added to the top of the menu.*

*The icon in the menu indicates that the command is added to the bottom of the menu.*

12.Click **Close** to close the **Customize** dialog box.

# Assigning or changing keyboard shortcuts

Using keyboard shortcuts accelerates your workflow. You can assign new keyboard shortcuts, or change the key combinations already used in shortcuts. Unless you choose a function key, **F1** through **F12**, your key combination must start with a modifier key and end with a regular key.

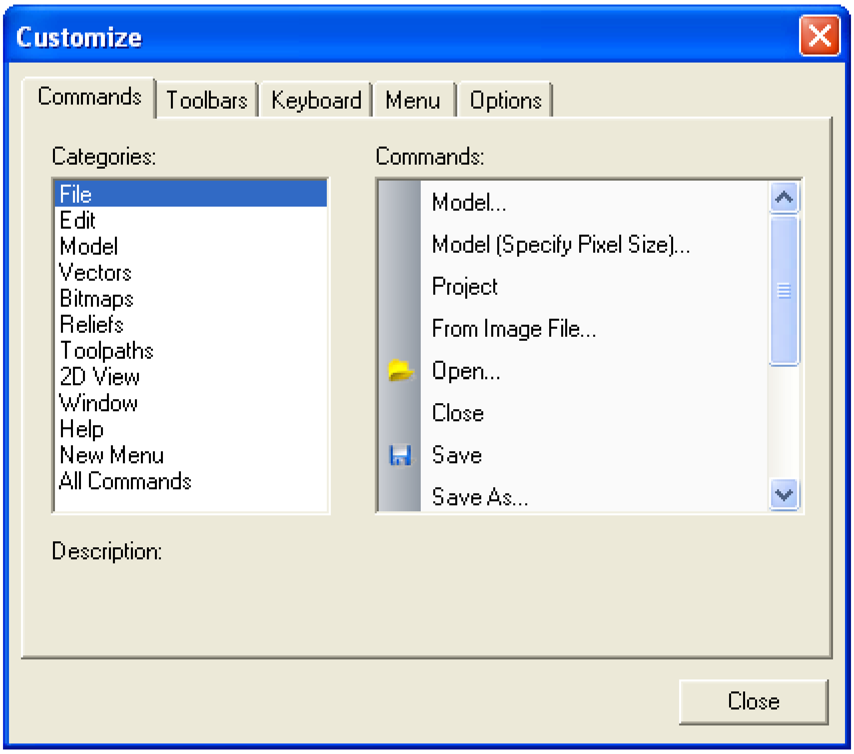
The modifier keys you can use are:

- **Alt**;
- **Ctrl**;
- **Alt Gr**, which is equivalent to using **Ctrl+Alt**; and
- **Shift**, provided it is also used with **Ctrl** or **Alt**.

You can use one or more modifier keys in a shortcut.

To assign or change a keyboard shortcut:

1. From the **Menu Bar**, click **Window > Toolbars and Docking Windows > Customise...** to display the **Customize** dialog box:




2. Click the **Keyboard** tab to display its settings.
3. Click the **Category** list box, followed by the menu that contains the command you want to assign or change the keyboard shortcut to.
4. Click the **Commands** list box, followed by the command you want to assign the shortcut to.

   A description of the selected command is shown below. The **Current Keys** area displays keyboard shortcuts already assigned to the selected command.
5. To delete a shortcut already assigned to the selected command:
   a. Click the key combination shown in the **Current Keys** area. It is emphasised in blue.
   b. Click **Remove**.
6. Click the empty **Press new shortcut key** box, then, using your keyboard, press the key combination you want assigned to the shortcut.

   If you choose a shortcut used elsewhere, its associated command is displayed below. In this instance you can:

- press the **Delete** key, followed by a different key combination; or
- select the command already using the shortcut, delete the shortcut, and then repeat this step.

7. Click **Assign** to assign the shortcut keys to the command.
8. To restore the default shortcuts, click **Reset All**.
9. Click **Close** to close the **Customize** dialog box.

## Using the keyboard

There are many different shortcuts available to help you complete your tasks in ArtCAM as efficiently as possible.

### View Control

The following keyboard shortcuts can be used to adjust the ArtCAM layout, the content of the **2D View** window:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Display Reference Help | **F1** |
| Display **2D View** | **F2** |
| Display **3D View** | **F3** |
| Toggle **Project** panel visibility | **F4** |
| Toggle **Tool Settings** panel visibility | **F6** |
| Preview currently active relief layer in **2D View** | **F10** |

*If you are working in ArtCAM Insignia, a preview of the relief is displayed.*

| | |
|---|---|
| Display cursor information | **Alt+C** |
| Toggle visibility of all vector layers | **Alt+V** |
| Toggle visibility of currently active bitmap layer | **Alt+B** |
| Create greyscale from composite relief | **Alt+G** |

*If you are working in ArtCAM Insignia, you cannot create a greyscale from the relief.*

| | |
|---|---|
| Toggle **Notes** visibility | **Alt+N** |

## Models

The following keyboard shortcuts can be used when working with ArtCAM models:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Create new model | **Ctrl+N** |
| Open model | **Ctrl+O** |
| Save model | **Ctrl+S** |
| Create new sheet in model | **Ctrl+Alt+Shift+S** |

## Editing

The following keyboard shortcuts can be used when editing:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Copy to ArtCAM clipboard | **Ctrl+C** |
| Paste from ArtCAM clipboard | **Ctrl+V** |
| Cut to ArtCAM clipboard | **Ctrl+X** |
| Undo last action | **Ctrl+Z** |
| Redo last action | **Ctrl+Y** |
| Delete | **Delete** |

## Vector Drawing

The following keyboard shortcuts can be used when drawing vector artwork:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Create vector and continue drawing | **Return** or **Space Bar** |
| **Select** tool | **Esc** |

| | |
|---|---|
| Constrain angle of linear span between nodes to 15 degree increments *(Create Polyline only)* | **Ctrl** |
| Keep aspect ratio *(Create Rectangle only)* | **Shift** |
| Close polyline to create polygon and continue drawing | **Tab** |

## Vector Editing

The following keyboard shortcuts can be used when editing vector artwork:

| **ArtCAM Function** | **Keyboard Shortcut** |
|---|---|
| **Select** tool | **Esc** |
| Select all vectors | **Ctrl+A** |
| Edit selected vector | **E** |
| **Node Editing** tool | **N** |
| Convert span *(linear or bezier)* to arc | **A** |
| Convert span *(linear or arc)* to bezier | **B** |
| Convert span *(bezier or arc)* to line | **L** |
| Cut span | **C** |
| Remove span | **R** |
| Insert node | **I** |
| Insert start node *or* change node to start node | **P** |
| Toggle smoothing on node | **S** |
| Delete node | **D** |
| Align selected nodes in X-axis | **X** |
| Align selected nodes in Y-axis | **Y** |
| Nudge selected vector up | ↑ |
| Nudge selected vector down | ↓ |
| Nudge selected vector right | → |

| | |
|---|---|
| Nudge selected vector left | ← |
| **Transform** tool (on **Tool Settings** panel) | **T** |
| **Measure** tool (on **Tool Settings** panel) | **M** |
| Display **Vector Clipping** panel | **Ctrl+Alt+Shift+C** |
| Display **Vector Slice** panel | **Ctrl+Alt+Shift+V** |

## Vector Alignment

The following keyboard shortcuts can be used when aligning vectors:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Centre in model | **F9** |
| Align left | **Ctrl+←** |
| Align horizontal centre | **Shift+←** |
| Align right | **Ctrl+→** |
| Align top | **Ctrl+↑** |
| Align vertical centre | **Shift+↓** |
| Align bottom | **Ctrl+↓** |

## Vector Grouping

The following keyboard shortcuts can be used when grouping vectors:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Group selected vectors | **Ctrl+G** |
| Ungroup vectors | **Ctrl+U** |

## Bitmap Colours

The following keyboard shortcuts can be used when working with bitmap artwork:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Toggle linking between primary and secondary colours | **Ctrl+L** |
| Link all colours | **Ctrl+K** |
| Reset all colour links | **Ctrl+R** |

## Reliefs

*If you are working in ArtCAM Insignia, these shortcuts are not available.*

The following keyboard shortcuts can be used when working with reliefs:

| ArtCAM Function | Keyboard Shortcut |
|---|---|
| Display **Shape Editor** *If you are working in ArtCAM Insignia, the Shape Editor is not available.* | **F12** |
| Reset currently active relief layer *If you are working in ArtCAM Insignia, you can reset the relief.* | **Shift+Delete** |
| Copy relief *If you are working in ArtCAM Insignia, you cannot copy a relief.* | **Ctrl+Shift+C** |
| Display **Offset Relief** dialog box *If you are working in ArtCAM Insignia, you cannot offset the relief.* | **Ctrl+Alt+Shift+O** |
| Select **Relief Envelope Distortion** tool *If you are working in ArtCAM Insignia, the Relief Envelope Distortion tool is not available.* | **Ctrl+Alt+Shift+R** |

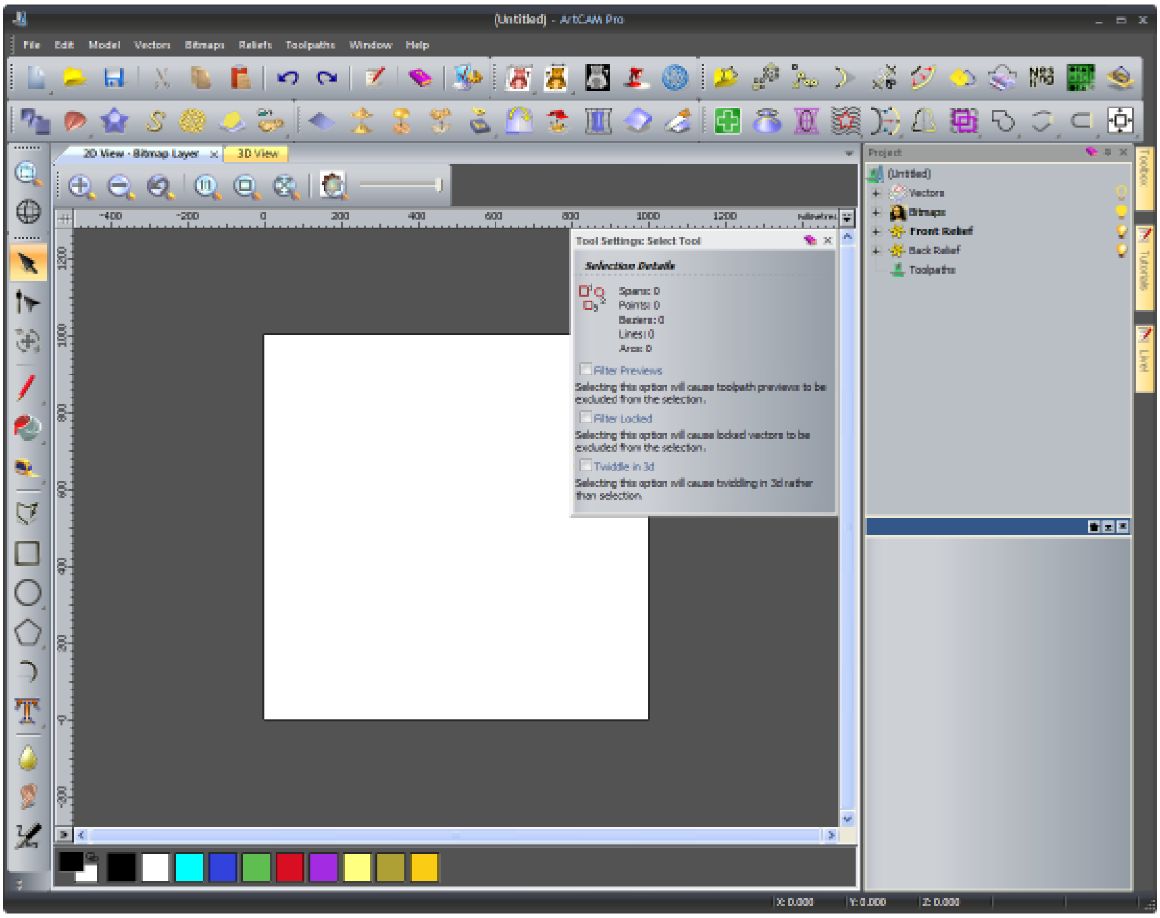
# Changing the theme

You can choose between two themes. This controls the user interface's colour scheme, but does not change its layout (see page 331).

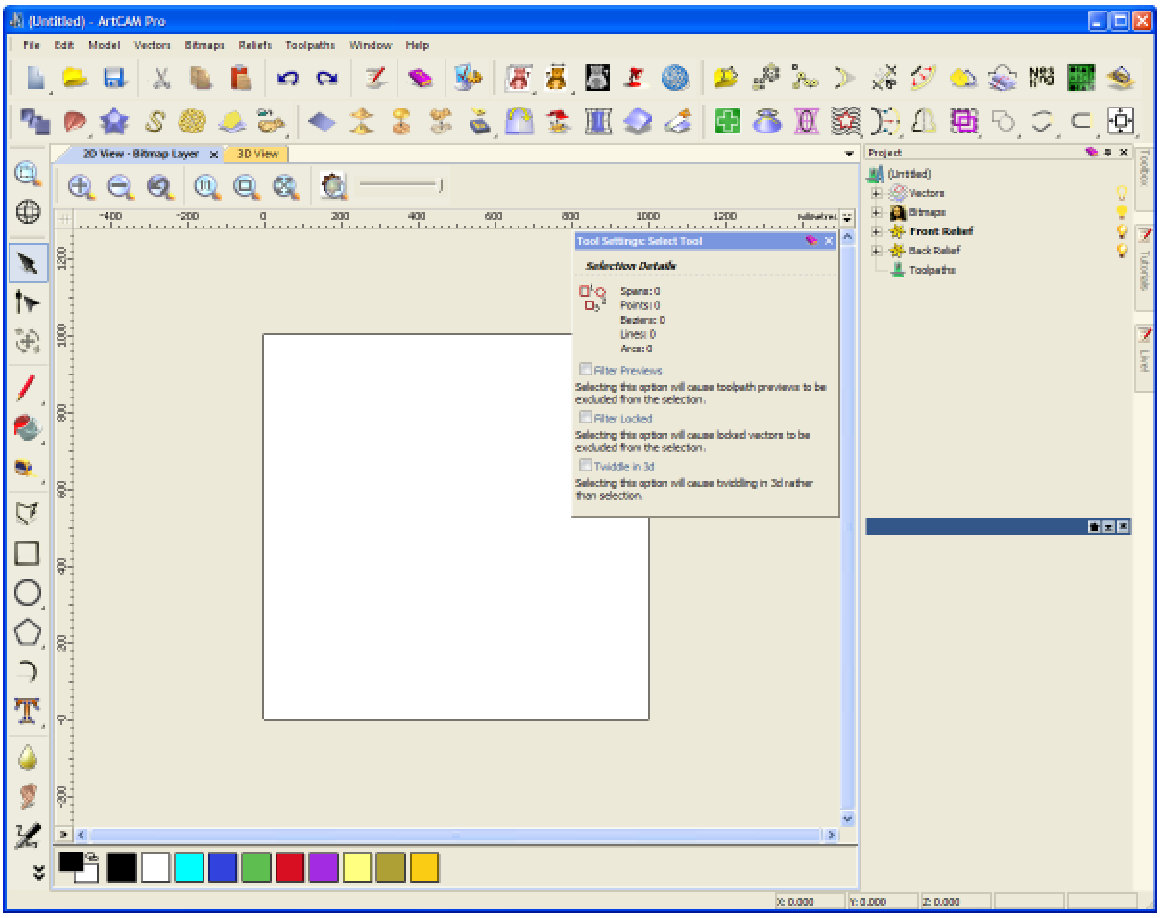
From the **Menu Bar**, click **Window > Theme**, then:

- **2010** to apply the obsidian black colour scheme. This is selected by default.
- **Classic** to apply the beige colour scheme. This is familiar to users of ArtCAM 2009 and earlier.

For example, a model open in ArtCAM Pro with the **2010** theme and layout looks as follows:



While a model open in ArtCAM Pro with the **Classic** theme and **2010** layout looks as follows:



# Resetting and changing the layout

You can choose between three layouts, each of which includes a different arrangement of panels and toolbars.

From the **Menu Bar**, click **Window > Reset Layout**, then:

- **2010** to access the most frequently used tools, without compromising the viewing area. This is selected by default.
- **2010 Advanced** to maximise the viewing area.
- **Classic** to keep the familiarity of ArtCAM 2009 and earlier.

When selecting a new layout, your:

- custom toolbars and menus are kept;
- toolbar changes are kept; and
- previous panel and toolbar arrangement is lost.

Although your chosen layout affects how toolbars and panels are arranged, whether you are working with a project or model also does.

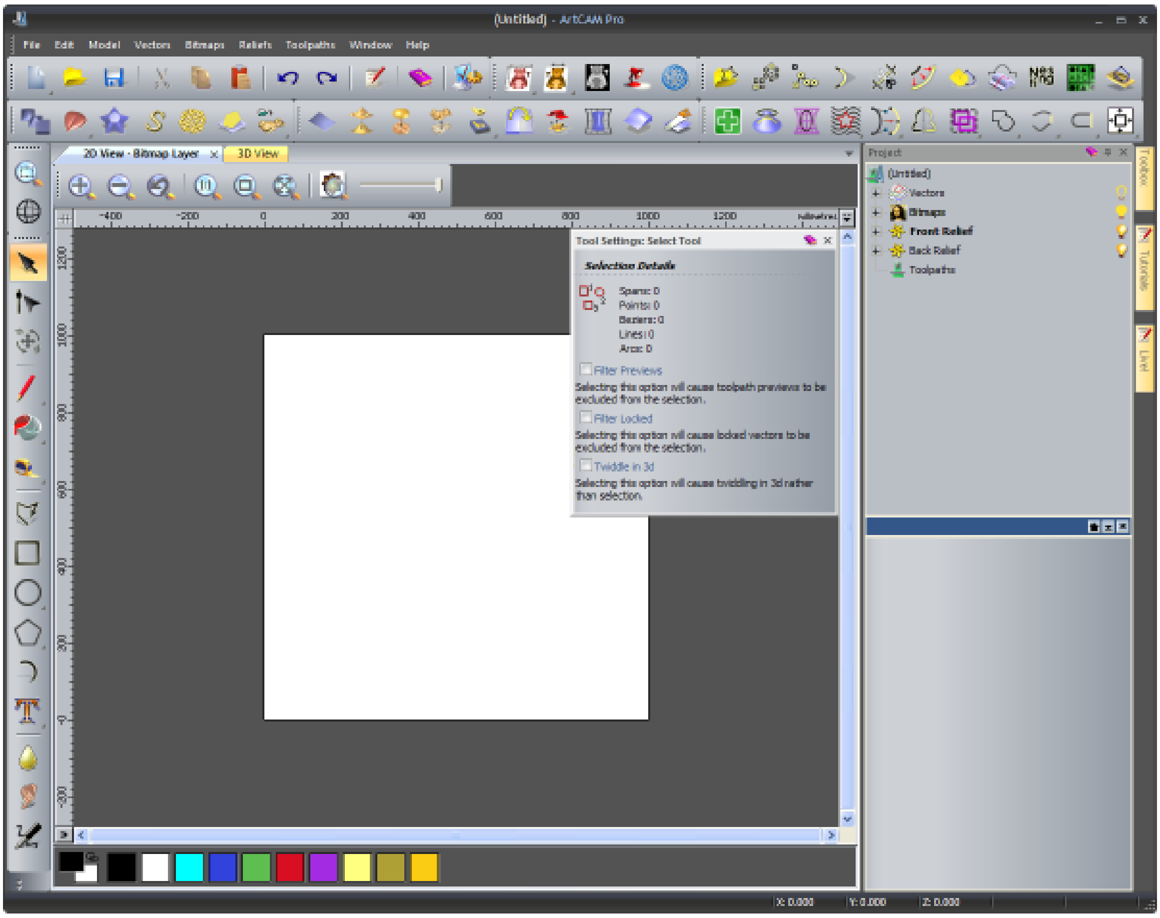
When working with a model in the **2010** layout:

- the **Project** panel is docked and pinned on the right;
- the **Tool Settings** panel is floating over the viewing area;
- the **Tutorials**, **Live!** and **Toolbox** panels are docked and auto-hidden on the right;
- the **File**, **Model**, **Bitmap Tools**, **Vector Creation**, **Vector Editing**, **Relief Creation** and **Relief Editing** toolbars are docked horizontally below the **Title Bar**;

  *The **Relief Editing** toolbar is not included in ArtCAM Insignia.*

- the **Design Tools** and **View Manipulation** toolbars are docked vertically on the left;
- the design windows are tabbed; and
- large buttons are displayed.

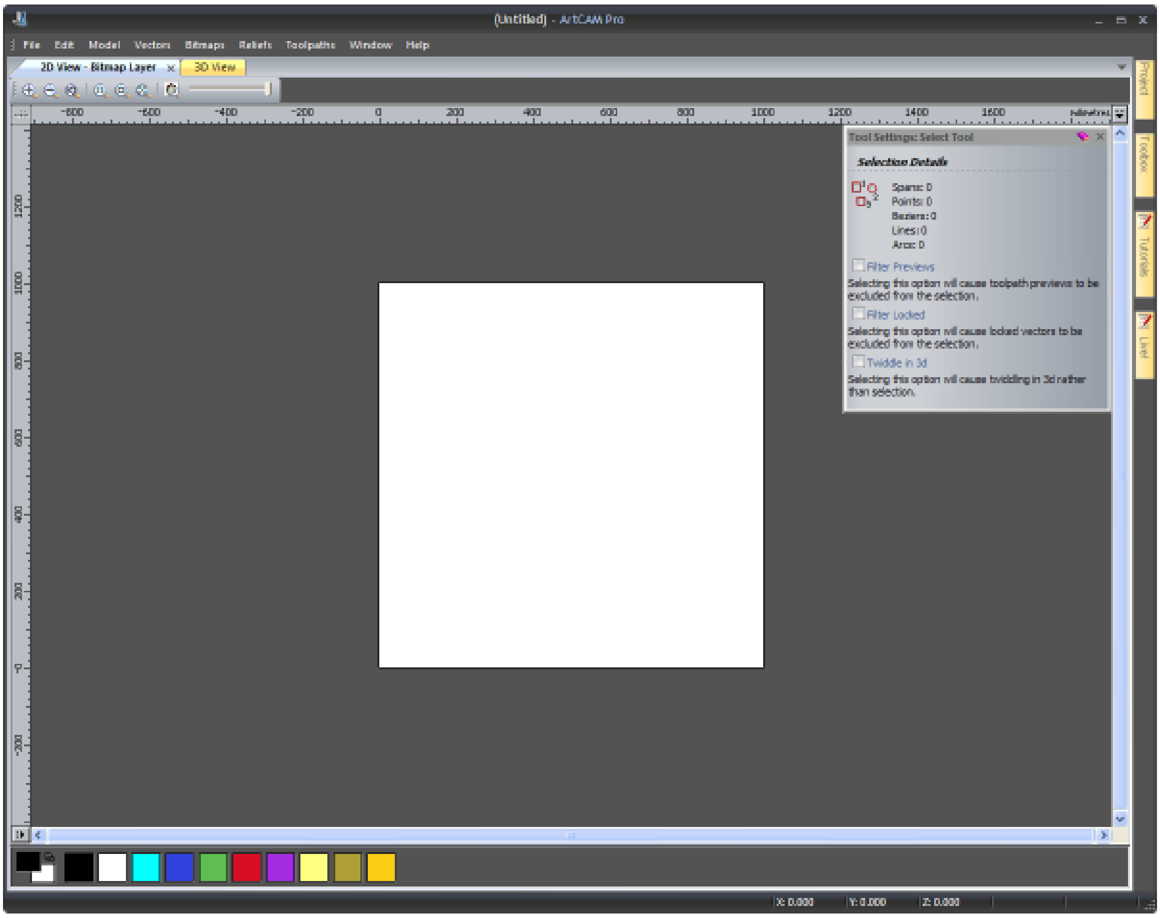
For example, a model open in ArtCAM Pro with the **2010** layout and theme selected looks as follows:




What distinguishes the **2010 Advanced** layout from the **2010** layout is:

- the **Project** panel is auto-hidden on the right;
- all toolbars other than those above the **2D View** and **3D View** windows are hidden; and
- small buttons are displayed.

For example, a model open in ArtCAM Pro with the **Advanced** layout and **2010** theme selected looks as follows:



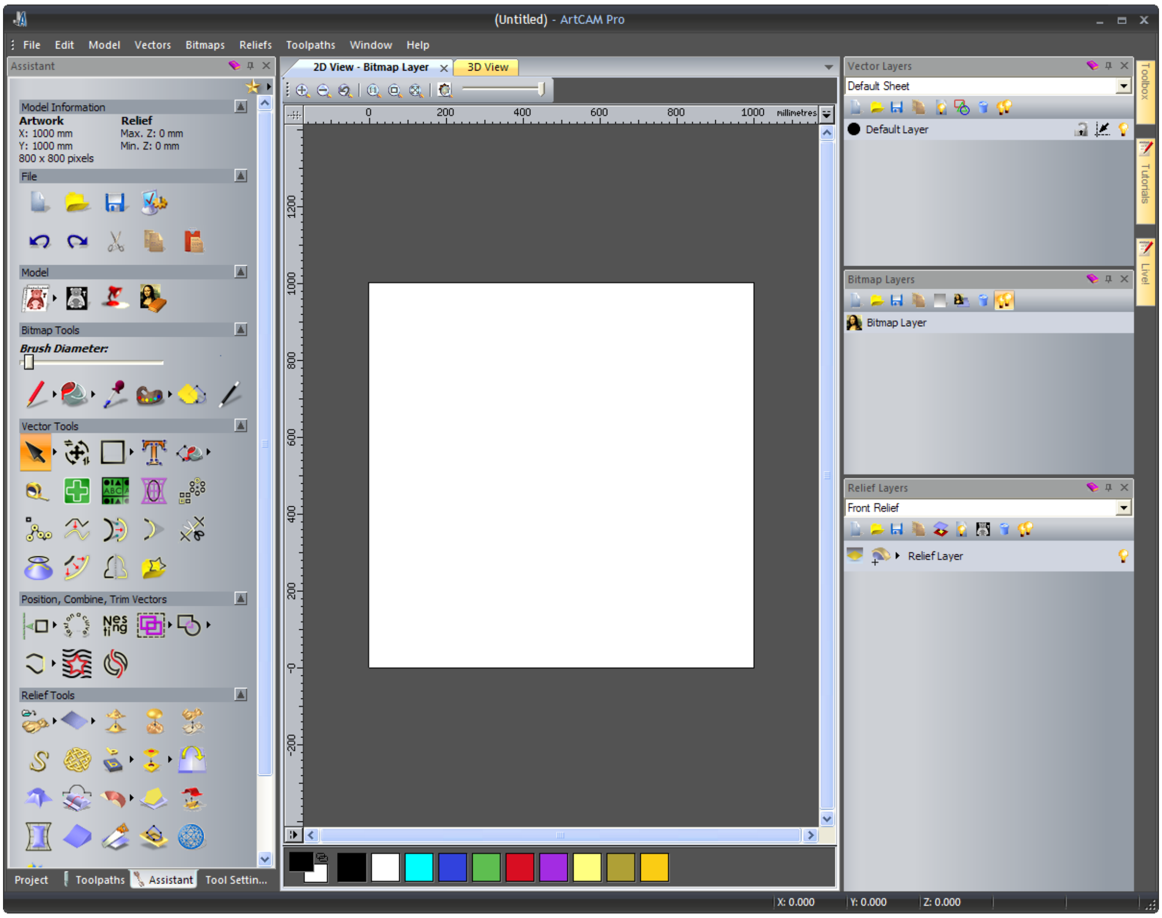

What distinguishes the **Classic** layout from the **2010** layout is:

- the **Vector Layers**, **Bitmap Layers**, and **Relief Layers** panels are docked and pinned on the right;

  *The **Relief Layers** panel is not included in ArtCAM Insignia.*

- the **Project** and **Tool Settings** panels are docked, pinned and tabbed on the left;
- the **Assistant** and **Toolpaths** panels are docked, pinned and tabbed on the left; and
- all toolbars, other than those above the **2D View** and **3D View** windows, are hidden.

For example, a model open in ArtCAM Pro with the **Classic** layout and **2010** theme selected looks as follows:




When working with a project in the **2010** layout:

- the **Project** panel is docked and pinned on the right;
- the **Tutorials**, **Live!** and **Toolbox** panels are docked and auto-hidden on the right;
- the **File** toolbar is docked horizontally below the **Title Bar**;
- the **3D View** window is tabbed; and
- large buttons are displayed.

What distinguishes the **2010 Advanced** layout from the **2010** layout is:

- the **Project** panel is auto-hidden on the right;
- the **File** toolbar is hidden; and
- small buttons are displayed.

What distinguishes the **Classic** layout from the **2010** layout is:

- the **Project** panel is docked and pinned on the left;
- the **File** toolbar is hidden; and

- small buttons are displayed.

# Choosing favourites

There is a **Favourites** toolbar at the top of the **Assistant** panel, which you can use to control which of the buttons, toolsets and areas on the **Assistant** panel are shown. This enables you to hide the tools that you confidently do not require during your workflow. You can also display labels for each of the buttons.

*Although you can choose which of a toolset's buttons are displayed or not, you cannot move a button from one toolset to another, or elsewhere on the* ***Assistant*** *panel.*

## Using the Favourites Toolbar

By default, all of the buttons, toolsets and areas that make up the **Assistant** panel are available, and Favourites mode is toggled off. The icon is displayed.

To choose your favourites:

1. Click on the **Favourites** toolbar, then **Edit Favourites** from the context menu:

   All of the buttons included in the **Assistant** panel are displayed, and the toolset layout is temporarily ignored. All buttons are active when selecting your favourites, although they cannot be used.

2. On the **Assistant** panel, click each of the buttons that you want to add or remove:

   - If a button is currently greyed-out, click to add the button to your favourites.
   - If a button is currently visible, click to remove the button from your favourites.

3. Click **Edit Favourites** to set your favourites.

   Either **Favourites On** or **Favourites Off** is displayed in the toolbar, depending on whether or not you were working in Favourites mode prior to choosing your favourites.

Only the buttons included in your favourites are displayed on the **Assistant** panel. The toolsets are restored, although which of their original buttons are available now depends on whether or not they are included in your favourites. If none of the buttons within a particular toolset are included in your favourites, the toolset is removed altogether. If none of the buttons in a particular area of the **Assistant** panel are included in your favourites, the area is removed altogether.

To toggle your favourites on, click:

- ; or
- , then **Favourites Off** from the context menu.

  When only your favourites are shown on the **Assistant** panel, the icon is displayed in the toolbar.

To toggle your favourites off, you can either:

- ; or
- , then **Favourites On** from the context menu.

  When the default layout is shown on the **Assistant** panel, is displayed.

## Using Labels

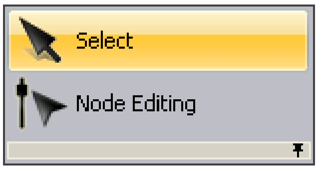
When you use ArtCAM for the first time, labels are displayed beside each of the buttons on the **Assistant** panel. The buttons are also arranged in such a way that their labels can be easily read. Thereafter, you can control whether to work with or without labels displayed.

To toggle labels on:

1. In the **Assistant** panel's header, click . The icon changes to .

   *When labels are hidden, the same description can be displayed as a tooltip whenever the mouse cursor is positioned over a button.*

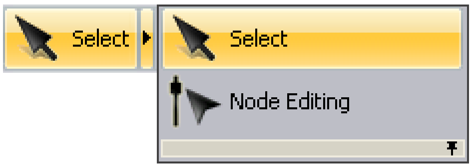
When a toolset is pinned and labels are displayed, all of the buttons belonging to the toolset are stacked vertically within a shaded backdrop. The icon used to pin the contents of a toolset is positioned along the bottom of the shaded area. For example, the Vector Editing toolset is displayed as shown below:




*To unpin the toolset, click along the bottom of the shaded area.*

When a toolset is unpinned and labels are displayed, only the most recently used button within the toolset is shown along with its corresponding label. For example, the Vector Editing toolset is displayed as shown below:


The icon beside the button displays all of the other buttons in its toolset, and these are stacked vertically within a shaded backdrop. The icon used to pin the contents of a toolset is positioned along the bottom of the shaded area. For example, the Vector Editing toolset is displayed as shown below:




*To pin the toolset, click along the bottom of the shaded area.*

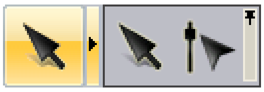
If a button is not currently available when labels are toggled on, both the button and its label are greyed-out.

To toggle labels off:

1. In the **Assistant** panel's header, click . The icon changes to .

When a toolset is unpinned and labels are hidden, only the most recently used button within the toolset is shown. The other buttons in the toolset are hidden by default. Clicking the icon displays all of the buttons in a toolset, and these are adjacent horizontally. The icon used to pin the contents of a toolset is positioned along the right-edge of the button currently shown.

For example, the Vector Editing toolset is displayed as shown below:



# Using the mouse

The way in which your mouse can be used in ArtCAM often changes according to which design window is displayed and the particular aspect of the model on which you are working. Furthermore, if you own a mouse with a wheel this also increases the range of options available to you.

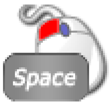
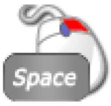
## 2D View

You can use the mouse to manipulate the **2D View** in the following ways:

| ArtCAM Function | Mouse Action |
| --- | --- |
| Magnify view by 50% |  |
| Reduce view by 50% |  |
| Zoom in |  |
| Zoom out |  |

## 3D View

You can use the mouse to manipulate the **3D View** in the following ways:

| ArtCAM Function | Mouse Action |
| --- | --- |
| Rotate view |   |
| Zoom |   |
| Pan view |   |
| Zoom in |  |
| Zoom out |  |

## Vectors

You can use the mouse relative to vector artwork in the following ways:

| ArtCAM Function | Mouse Action |
| --- | --- |
| Select vector *(also select node in Node Editing mode)* |  |
| Select multiple vectors |   |
| Copy vector |   *...on selected vector.* |
| Display context menu |  *...on selected vector.* |
| Display Shape Editor |   *...on selected vector.* |

## Bitmaps

You can use the mouse relative to bitmap images shown in the **2D View** in the following ways:

| ArtCAM Function | Mouse Action |
|---|---|
| Select primary colour |  ...*on colour swatch in colour palette.* |
| Select secondary colour |  ...*on colour swatch in colour palette.* |
| Link colour to primary colour |  ...*on colour swatch in colour palette.* |
| Display Shape Editor |  ...*on colour swatch in colour palette,* **2D View** *or* **3D View**. |

## Toolpaths

You can use the mouse relative to toolpaths in the following ways:

| ArtCAM Function | Mouse Action |
|---|---|
| Edit toolpath |  ...*on 2D toolpath preview or toolpath name.* |

# Index

## 3



## A



## C