ArtCAM Pro 6.0

Reference Manual

By Delcam plc

Issued 01/05/03

THE QUEEN'S AWARDS FOR ENTERPRISE 2003

Disclaimer

Delcam plc has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

Information contained in this manual is subject to change without notice and does not represent a commitment by Delcam plc. The software described in this manual is furnished under licence agreement and may be used or copied in accordance with the terms of such licence. No part of this manual may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying and recording, for any purpose without the express permission of Delcam plc.

Delcam plc
Talbot Way
Small Heath Business Park
Birmingham B10 OHJ
England

Tel: (UK) 0121-766-5544
(Int) +44 (0) 121-766-5544

Contents

Working with Vectors 119

Overview

ArtCAM Pro Overview

ArtCAM Pro is a unique software program which allows users to easily create impressive, high quality 3D products starting from 2D 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 the artisan more time to concentrate on the original design and the fine details, which together help to differentiate products in today's competitive markets.

Information about ArtCAM Pro

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

1. The ArtCAM Pro Assistant (In-line Help).
2. The ArtCAM Pro Reference Manual (this manual).
3. The On-line Help pages. You can find a list of all shortcuts that can be used in ArtCAM Pro here.

ArtCAM Pro Reference Manual

This manual provides you with instructions on how to create and manipulate both 2D and 3D models, and then machine them. Further information can be found in the On-line Help pages.

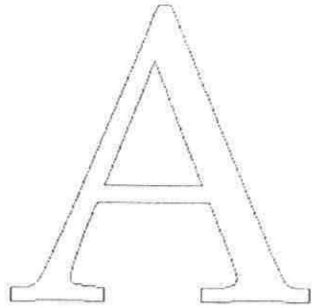
This Reference Manual is divided into the following sections:

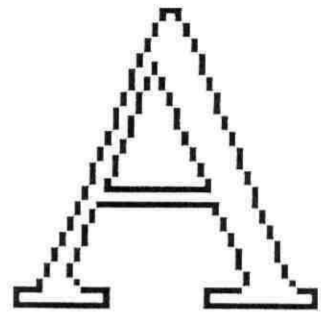
- **Overview**. This section provides an overview of what ArtCAM Pro can be used for.
- **ArtCAM Pro Layout**. This section explains the layout of ArtCAM Pro, how to use design windows and adjust its default settings.
- **Working with Models**. This section explains how to create and manage an ArtCAM model.
- **Working with Bitmaps**. This section explains how you can use a bitmap image to create a model itself, to create a vector object or a 3D shape as all or part of a relief in a model.
- **Working with Vectors**. This section explains how to create and use vector objects in a model. These can then be used to create a three-dimensional shape as all or part of a relief or to machine a 2D model.
- **Working with Reliefs**. This section explains how to create different types of shapes and combine them with any existing relief, how to transform, manipulate, edit and manage the relief. The relief can then be used to machine a 3D model.
- **Machining Models**. This section explains how to machine a model from the vector artwork or three-dimensional relief that you have created.

Comparing Bitmaps, Vectors and Reliefs

Vectors and bitmaps are two different methods of reproducing particular types of images. ArtCAM Pro uses both bitmap images and vector objects to create three-dimensional shapes. In ArtCAM Pro, a vector object can be created from a bitmap image, and vice versa. For more details, see the Working with Bitmaps and Working with Vectors chapters.

Vector Data *Bitmap Data*





The three-dimensional shapes created from the bitmap images and vector objects in ArtCAM Pro make up a relief. This relief is used to machine a 3D model.

What is a Vector?

Vector data is defined mathematically. Objects are geometrical, being made up of a number of points connected by lines or curves. Vectors are extremely flexible and can be manipulated with both ease and accuracy.

The amount of data required to display all of the attributes of a vector object is very small, so graphics made up of vector objects can have very small file sizes. As the objects become more complex, the sizes increase.

Vector data is ideal for producing smooth features. A vector object not only has greater definition than the bitmap, the geometry of the vector object can also be used to drive an assigned machining tool directly, which, in turn, can produce a better finish.

ArtCAM Pro can read files containing vector data saved in the following formats:

- Drawing Interchange files, including PowerSHAPE and AutoCAD (**.dxf**)
- Encapsulated PostScript format (**.eps**)
- AutoCAD Drawing files (**.dwg**)
- Adobe Illustrator image (**.ai**)
- Windows Metafile (**.wmf**)
- Lotus, PC Paint or DUCT picture (**.pic**)

What is a Bitmap?

Bitmap data is made up of a set of values specifying the colour of individual pixels (picture elements) that make up an image. Bitmap data is characterised by resolution and bit depth.

Resolution relates to the detail in an image and is expressed in dots per inch (**.dpi**) or pixels per inch (**.ppi**). The higher the resolution (i.e. the more dots used to describe the image), the more detail possible.

Bit depth has to do with the number of colours the image can display. Bits are the building blocks of binary data. A black and white image is 1 bit, meaning it can be off or on, black or white. As bit depth increases, more colours are available.

Unlike vector data, bitmap data is large. For example, a simple object like the letter in the previous image is 32,838 bytes as vector data in ArtCAM Pro. When rasterized (changed to a bitmap), the file size changes to 40,078 bytes in ArtCAM Pro. For small compositions the increase in file size may not matter greatly, but for larger compositions the difference in file size seriously impacts machining times.

ArtCAM Pro allows three-dimensional shapes to be created from areas of bitmap colour. It can also read bitmap files generated by other drawing and desktop publishing packages or those scanned from paper based artwork saved in any of the following formats:

- Windows Bitmap (**.bmp**)
- TIFF image (**.tif**)
- PCX image (**.pcx**)
- CompuServe image (**.gif**)
- JPEG image (**.jpg**)

What is a Relief?

A relief is made up of one or more three-dimensional shapes created in ArtCAM Pro. A three-dimensional shape becomes all or part of a relief when the relief combination method selected for the shape has been calculated.

Ultimately, a relief is made up of a grid of points in a similar way to a bitmap image. However, instead of colour, each point is assigned a specific height.

When a new model is created in ArtCAM Pro, you are required to set its size in millimetres or inches. The model's resolution is also specified here. The number of points in the model determines the resolution of the relief and any associated bitmap image. For further information, see "Creating a Model" in the Working with Models chapter.

A compromise must be made between the quality of a model, including any relief within it, and the speed of the processor in your computer. For most jobs, a model of 1,000,000 points is a reasonable value. Since a relief is made up of a number of points, even the quality and smoothness of the three-dimensional shapes created from vector objects depends on the resolution of the model, although this is to a lesser extent than with those shapes created from bitmap images.

A relief can be saved or loaded in the **.rlf** format. If there is no bitmap image of the same resolution as the relief, ArtCAM Pro creates a greyscale bitmap representation of the 3D model when it is loaded.

ArtCAM Pro Layout

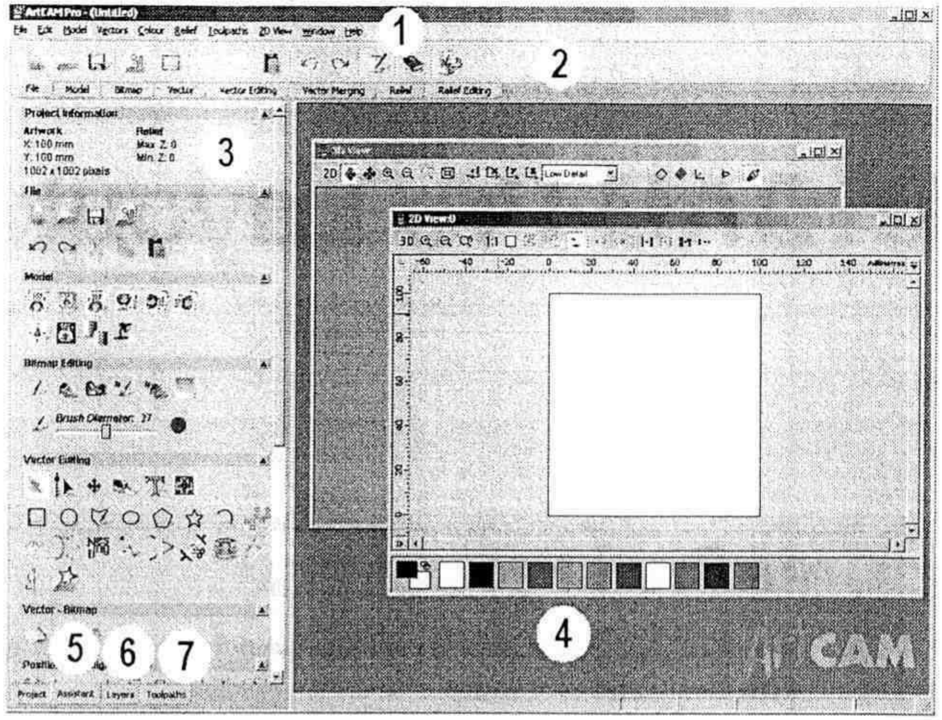
ArtCAM Pro Layout

The ArtCAM Pro screen layout has been designed to let you work effectively and efficiently.

The hub of ArtCAM Pro is the innovative **Assistant** window. When you have an open model, the **Assistant**'s Home page displays a collection of tools to help you create, edit and machine a model. These tools appear in the form of buttons.

Grouped in a logical arrangement, you can to navigate to the button that you require quickly. To find out more about most of the buttons in ArtCAM Pro, click on it to display a page in the **Assistant** window showing In-line Help. The In-line Help informs you of exactly where, when and how to use the button, thus reducing the learning curve.

The screen layout is divided into seven regions:



1. **Main menu bar** – Click on a menu item to display a drop-down menu that contains a range of sub-menus and commands relating to the features in ArtCAM Pro. If an item in a menu is greyed-out, then it is does not apply to the currently active **2D View**.
2. **Top toolbar** – Click on one of the eight available tabs to display a toolbar that contains a range of buttons relating to the features in ArtCAM Pro.
3. **Assistant Window** – On starting ArtCAM Pro, the **Assistant**'s **Getting Started** page is displayed. This page contains four main icons. These allow you to create a new model or a font, and work on an existing model. You can also open any of the last four models that you have been working on. When a model has been created or opened, the **Assistant**'s Home page is displayed.

   The **Assistant**'s Home page contains buttons for almost all of the features in ArtCAM Pro, as well as a summary of your model and relief dimensions. For almost every button you click on, a corresponding interactive In-line Help page is automatically displayed in its place. These buttons are divided into nine areas:

   - **File** – You can use these buttons to manage your model files, as well as the image, vector and triangle model files that you want to use.

- **Model** – You can use these buttons to manage the appearance of your model, as well as adjust how a model appears in the **3D View** window.
- **Bitmap Editing** – You can use these buttons to draw and paint bitmap shapes, as well as manage the content of the Colour Palette and the size and shape of the brush you use.
- **Vector Editing** – You can use these buttons to create vector objects in the form of shapes or text, as well as measure and manipulate the vector objects you create as part of your model.
- **Vector – Bitmap** – You can use these buttons to convert a vector object in your model to a bitmap or vice versa.
- **Group Merge Join Vectors** – You can use these buttons to group, merge and join the vector objects you create as part of your model.
- **Relief Operations** – You can use these buttons to load, save and calculate reliefs, and create a triangle mesh or cross-section.
- **Vector Based Relief Creation** – You can use these buttons to create three-dimensional shapes from the vector objects in your model, and paste an imported relief along a vector object.
- **Relief Editing** – You can use these buttons to scale, smooth, invert, offset, sculpt, reset or add texture to a relief. You can also create an angled plane or a blended shape.

You can change the appearance of the **Assistant**'s Home page in the following way:

- Click on the appropriate arrow to hide or reveal  options and reference material in any given area of the page.

You can also change the appearance of each page displayed in the **Assistant** window when you click on any of the buttons on the **Assistant**'s Home page:

- Click on **Hide Help** to hide all In-line Help shown on the page. You can also click on the **?** icon in the top-right corner of the page to hide In-line Help shown.
- Click on **Show Help** to display In-line Help on the page. You can also click on the **?** icon in the top-right corner of the page to display the In-line Help.
- Click on the **X** icon in the top-right corner of the page to return to the **Assistant**'s Home page.

You can play a video tutorial on how you can use the page that is currently displayed in the **Assistant** window if you:

- Click on the video icon on the In-line page.

4. **Design Windows** - This is the central area of the screen. ArtCAM Pro uses two types of view. The **2D View** window displays the two-dimensional vector and bitmap artwork that you create, while the **3D View** window displays a three-dimensional relief and any calculated or simulated toolpaths.
5. **Project** – This tab displays the **Project** page in the **Assistant** window. You can use this page to view information about your model, to create, delete or edit design windows, and calculate reliefs. You can also edit, calculate, delete and simulate toolpaths, or create a template from them.
6. **Layers** – This tab displays the **Layers** page in the **Assistant** window. You can use this page to organise the vector objects in your model into layers. Each layer you create can be given its own name and colour. You can also toggle the visibility, snapping and locking for the vector objects on each layer.
7. **Toolpaths** – This tab displays the **Toolpaths** Home page in the **Assistant** window. You can use this page to create machining toolpaths, drill holes, add bridging and lead moves, manage the machining order, produce a toolpath summary, and manage the tools database.

   You can also use this page to simulate toolpaths, save toolpaths, edit toolpaths, load or save a toolpath template, and edit the machining parameters relating to a tool.

Using the Design Windows

When you create or open a model in ArtCAM Pro, a **2D View** and a **3D View** design window are displayed. You use the **2D View** window to create the vector and bitmap artwork you need to produce a three-dimensional relief in ArtCAM Pro. This relief is shown in the **3D View** window.

Note: You can maximise the design windows by default using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" on page 31.

Adjusting the Window View

You can adjust the appearance of the **2D View** and **3D View** windows in ArtCAM Pro:

1. From the Main menu bar, click on the **Window** menu, followed by the option for how you want the design windows to be displayed:

   - **Cascade** - Click on this option to display both the **2D View** and **3D View** windows overlapping one another.
   - **Tile** - Click on this option to display both the **2D View** and **3D View** windows beside one another.
   - **2D View** - Click on this option to display the **2D View** window only.

Note: You can also display the **2D View** window by pressing the **F2** key on your keyboard.

   - **3D View** - Click on this option to display the **3D View** window only.

Note: You can also display the **3D View** window by pressing the **F3** key on your keyboard.

You can also select which design window is displayed from the **Project** page:

1. Click on the **Project** tab Project to display the **Project** page.

2. With the **Views** element shown, click on the **2D View** or **3D View** window that you want to view.

Opening a New 2D View Window

To open a new **2D View** window:

- From the Main menu bar, click on the **2D View** menu, and then on the **New View** option.

A **2D View** window appears named as **2D View: 1** by default. You can change the name of the new window if you want to. For details, see "Labelling a 2D View Window" on page 12.

The content within the new **2D View** window is identical to that of the previous window.

The Colour Palette beneath the new **2D View** window is the same as that which is below the previous **2D View** window, although any colour links or relief attributes used are removed.

You can also open a new **2D View** window from the **Project** page:

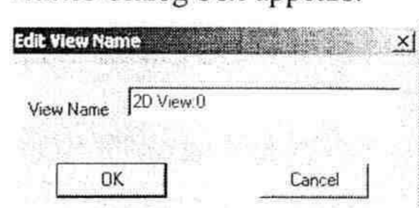
1. Click on the **Project** tab to display the **Project** page.
2. With the **Views** element shown, right-click on the last **2D View** created to display the context menu, and then click on the **New View** option.

Labelling a 2D View Window

The **2D View** window that automatically appears when you open or create a model is named **2D View: 0** by default. You can edit the name of any **2D View** window that is open in your current ArtCAM Pro session.

To edit the name of a **2D View** window:

1. Make sure that the **2D View** window you want to rename is active by clicking on it.
2. From the Main menu bar, click on the **2D View** menu, and then on the **Edit View Name** option. The **Edit View Name** dialog box appears:



3. Click in the **View Name** box and then type the name you want for the **2D View** window in it.
4. Click on the **OK** button to close the **Edit View Name** dialog box.

You can also edit the name of any **2D View** window from the **Project** page:

1. Click on the **Project** tab 

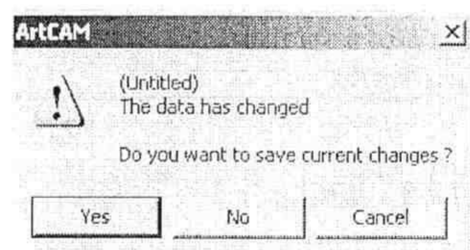
to display the **Project** page.
2. With the **Views** element shown, right-click on the **2D View** that you want to name to display the context menu, and then click on the **Edit View Name** option.
3. Type the name you want to use for the **2D View** window, then click.

Deleting a 2D View Window

You can delete any active **2D View** window in a model:

1. Make sure that the **2D View** window you want to delete is active by clicking on it.
2. From the Main menu bar, click on the **2D View** menu, and then on the **Delete View** option.

   If you have not previously saved the current model, or any changes that you have made since it was saved last, the following message box appears:



3. If you want to save the changes in the model:

   - Click on the **Yes** button to open the **Save As...** dialog box. For details on using the **Save As...** dialog box, see "Saving a Model" in the Working with Models chapter.

   If you do not want to save the model you are working on:

   - Click on the **No** button to close the message box and the **2D View** window. The **Getting Started** page appears in the **Assistant** window. For details, see

"Getting Started" in the Working with Models chapter.

If you have decided that you do not want to delete the **2D View** window:

- Click on the **Cancel** button to close the message box.

You can also delete any **2D View** window from the **Project** page:

1. Click on the **Project** tab to display the **Project** page.
2. With the **Views** element shown, right-click on the **2D View** that you want to delete to display the context menu, then click on the **Delete View** option. For further details, see "Viewing Model Information" in the Working with Models chapter.

2D View Manipulation

You can change the **2D View** of a model using the tools above the window:

3D View

Click on the **3D View** button to display the **3D View** window.

Note: You can also display the **3D View** window by pressing the **F3** key on your keyboard.

Zoom In Tool

You can use the **Zoom In Tool** button in two ways. You can:

- Enlarge the area directly beneath the cursor by 50%.
- Zoom in on a defined area of the model.

To enlarge an area directly beneath the cursor by 50%:

1. Click on the **Zoom In Tool** button, move the magnifying cursor over the area of the model that you want to zoom in on, and then click.

Note: You can also magnify the area directly beneath the cursor by 50% if you hold down the **Ctrl** key on your keyboard and right-click.

To zoom in on a defined area of the model:

1. Click on the **Zoom In Tool** button, and then move the magnifying cursor over the area of the model that you want to zoom in on.
2. Click and drag to create a bounding box around the area of the model that you are interested in.
3. Release the left mouse button to view the area defined by the bounding box you have drawn.

Zoom Out Tool

Each click on the **Zoom Out Tool** button reduces the size of the image by 50%.]

Note: You can also reduce the area directly beneath the cursor by 50% if you hold down the **Shift** key on your keyboard and right-click.

Zoom Previous

Click on the **Previous Zoom** button to restore the previous zoom setting.

Window Fit

Click on the **Window Fit** button to automatically adjust the zooming to view the whole model.

Zoom Object(s)

Click on the **Zoom Object(s)** button to focus in on a selected vector object within a model. For details, see "Selecting Vectors" in the Working with Vectors chapter.

1:1 Zoom 1:1

Click on the **Zoom 1:1** button 1:1 to return to a zoom aspect ratio of 1:1.

Vectors On/Off

Click on the **Vectors On/Off** button to hide the vector objects drawn in a model from view.

Click on the **Vectors On/Off** button to restore the vector objects drawn in a model.

Bitmap On/Off

Click on the **Bitmap On/Off** button to hide the bitmap images drawn in a model from view.

Click on the **Bitmap On/Off** button to restore the bitmap images drawn in a model.

Undo

Click on the **Undo** button to cancel your last action.

Redo

Click on the **Redo** button to repeat the last action that you had made before clicking on the **Undo** button.

Link All Colours

Click on the **Link All Colours** button to simultaneously link all colours in the current Colour Palette, other than the Secondary Colour, to the Primary Colour.

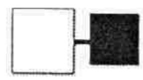
Unlink All Colours

Click on the **Unlink All Colours** button to unlink all colours currently linked together in the Colour Palette.

Link/Unlink Colours

Click on the **Link/Unlink Colours** button to link the Secondary Colour to the Primary Colour.

You can see that the Secondary and Primary Colours are linked when they appear in the Colour Palette as follows:



When the Secondary Colour is linked to the Primary Colour, it is displayed as the Primary Colour in the bitmap image.

Click on the **Link/Unlink Colours** button again to unlink the Secondary Colour from the Primary Colour.

Note: You can also link or unlink the Secondary and Primary Colours if you click on the **Link/Unlink Colours** icon in the Colour Palette or double right-click on the Secondary Colour itself.

Merge Colours

Click on the **Merge Colours** button to merge the current Secondary Colour with the current Primary Colour.

Scrolling the 2D View

If you cannot see all of an open model in the **2D View** window, often as a result of zooming in, ArtCAM Pro automatically displays scrollbars along the vertical and horizontal axes:

To view a different area of the open model, you can:

- Click on the arrow at either end of the scrollbar.
- Drag the thumb in the scrollbar to the appropriate position.
- Click directly in the scrollbar on either side of the thumb to page back and forth.

2D View Options

When drawing vector objects in ArtCAM Pro, there are three invaluable features available to help you create a vector object according to an exact set of measurements. These are:

- Rulers. For details, see "Using Rulers" on page 18.
- The Snap Grid. For details, see "Using the Snap Grid" on page 18.

- Horizontal and vertical guidelines. For details, see "Using Guidelines" on page 20.

These features help consistently align and size vector objects.

Using Rulers

You can use the rulers adjacent to the X and Y-axis of the **2D View** window to make measurements.

The ruler along the X-axis also displays the units of measurement (millimetres or inches) selected for the model that is shown in the **2D View** window. For details, see "Creating a Model" in the Working with Models chapter.

To turn the rulers off:

1. From the Main menu bar, click on the **2D View** menu, followed by the **Show Rulers** option to deselect it. The rulers along the X and Y-axis of the **2D View** window are hidden.

To turn the rulers on:

1. From the Main menu bar, click on the **2D View** menu, followed by the **Show Rulers** option to select it. Rulers appear adjacent to the X and Y-axis of the **2D View** window.

Using the Snap Grid

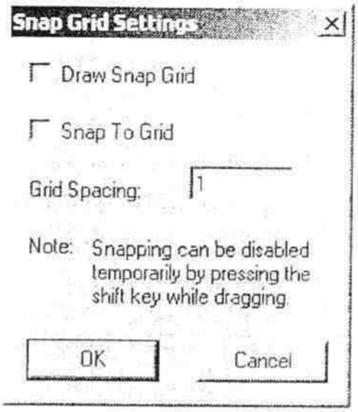
The snap grid is a network of fixed points that govern the placement of vector objects, making it easier to consistently size and align them.

You can use the **Snap Grid Settings** dialog box to adjust the spacing of gridlines, to activate or deactivate the grid, or to force vector objects to automatically align themselves to the snap grid.

Displaying the Snap Grid

To display the snap grid in an active **2D View** window

1. From the Main menu bar, click on the **2D View** menu, followed by the **Snap Grid Settings** option to display the **Snap Grid Settings** dialog box:



2. Click on the **Draw Snap Grid** option to select it ☑.
3. Click on the **OK** button to close the **Snap Grid Settings** dialog box and draw the snap grid.

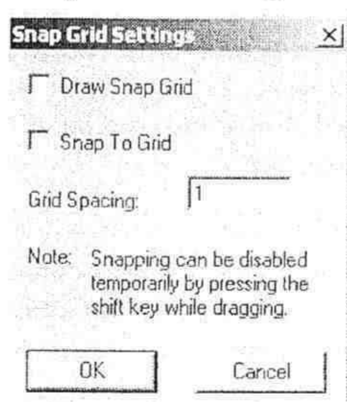
To hide the snap grid in an active **2D View** window, repeat the previous steps, clicking on the **Draw Snap Grid** option to deselect it ☐.

Setting the Grid Spacing

You can set the spacing between grid points using the model's units of measurement selected in the **Setup Job Dimensions** dialog. For details, see "Creating a Model" in the Working with Models chapter.

To set the grid spacing:

1. From the Main menu bar, click on the **2D View** menu, followed by the **Snap Grid Settings** option to display the **Snap Grid Settings** dialog box:

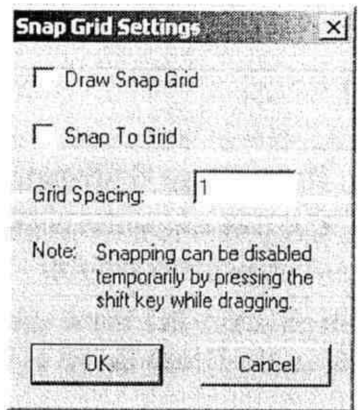


2. Define the distance between each grid point in the **Grid Spacing** box.
3. Click on the **OK** button to close the **Snap Grid Settings** dialog box and set the grid spacing.

Snapping to the Grid

To make the cursor snap to the nearest point on the grid when creating or editing a vector object:

1. From the Main menu bar, click on the **2D View** menu, followed by the **Snap Grid Settings** option to display the **Snap Grid Settings** dialog box:



2. Click on the **Snap To Grid** option to select it.
3. Click on the **OK** button to close the **Snap Grid Settings** dialog box and activate grid snapping.

Note: To temporarily disable the snapping behaviour, press and hold the **Shift** key on your keyboard.

Using Guidelines

While a **Snap Grid** is made up of a network of fixed points, a guideline is a solid line parallel to either the horizontal (X) or vertical (Y) axis that can be moved to any position in the **2D View** window. To create guidelines, the rulers must be checked on. For details, see "Using Rulers" on page 18.

To create a horizontal guideline:

- Click and hold down the left mouse button on the top ruler to display a guideline in the **2D View** window, and then drag it into position.

To create a vertical guideline:

- Click and hold down the left mouse button on the left-hand ruler to display a guideline in the **2D View** window, and then drag it into position.

To reposition a guideline:

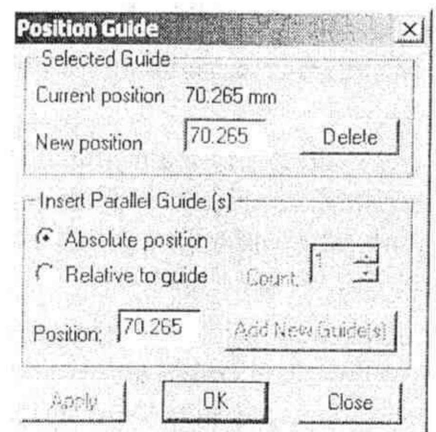
- Move the cursor over the guideline you want to move. When the cursor changes to a double-headed arrow ↔, click and drag the guideline into its new position.

Warning: You must have the both the **Show Rulers** and the **Show Guidelines** options turned on to see guidelines in the **2D View** window.

Defining a Guideline's Position

You can define the position of a guideline using the **Position Guide** dialog box. This is a more accurate method than clicking and dragging a guideline into position.

To display the **Position Guide** dialog box, move the mouse cursor over a guideline and then right-click:



The **Selected Guide** area of the **Position Guide** dialog box allows you to:

- Reposition a guideline.
- Delete a guideline.

To reposition a guideline:

- Type the new position for the guideline in the **New Position** box, and then click on the **Apply** button.

To delete a guideline:

- Click on the **Delete** button.

The **Insert Parallel Guide(s)** area of the **Position Guide** dialog box allows you to:

- Create a guideline at an absolute position.

- Create any number of guidelines at a position relative to the selected guideline.

To insert a parallel guideline at an absolute position:

1. Click to select the **Absolute position** option .
2. Type the position of the new guideline in the **Position** box.
3. Click on the **Add New Guide(s)** button to draw the guideline.
4. Click on the **OK** button to close the **Position Guide** dialog box.

To insert any number of parallel guidelines at a relative position:

1. Click to select the **Relative to guide** option .
2. Type the number of guidelines that you want to draw in the **Count** box.
3. Define the distance between each of these parallel guidelines in the **Position** box. You can set the relative position for the new guidelines using a positive or negative value.

   For example, if you want to create a new guideline to the right of an existing vertical guideline, or above an existing horizontal guideline, you must type a positive value in the **Position** box e.g. 10.

   If you want to create a new guideline to the left of an existing vertical guideline, or below an existing horizontal guideline, you must type a negative value in the **Position** box e.g. -10.
4. Click on the **Add New Guide(s)** button to draw the guidelines.

Tip: If you want to carry out more than one action in the **Position Guide** dialog box, instead of clicking on the **OK** button, click on the **Apply** button. This applies the change and keeps the dialog box open.

Snapping to Objects

Using snapping, you can align a vector object relative to another vector object on any visible layer or guideline shown in the **2D View** window. For further information on guidelines, see "Using

Guidelines" on page 20. For further details on layers, see "Snapping on a Layer" in the Working with Models chapter.

Snapping is checked on by default. You can, however, toggle snapping off and on:

1. From the Main menu bar, click on the **2D View** menu, followed by the **Snap to Objects** option. When turned on, the **Snap to Objects** option is checked ✓ Snap To Objects , and vice versa.

Note: Make sure that the **Toggle Snapping** button for the layer is also checked on. For details, see "Snapping on a Layer" in the Working with Models chapter.

Tip: To temporarily disable snapping, press and hold the **Shift** key down on your keyboard.

With the **Snap to Objects** option checked on, the cursor shown in the **2D View** changes when it snaps to any of the following places:

- A point (node) in another vector object. This is shown by the cursor changing to a .
- The mid of a linear or arc span within another vector object. This is shown by the cursor changing to a .
- The centre of another vector object, defined by a bounding box. This is shown by the cursor changing to a .

Tip: If you hold the **X** key down, the cursor also changes to where two vector objects intersect.

- The point at which two guidelines intersect. This is shown by the cursor changing to a .
- A horizontal guideline. This is shown by the cursor changing to a .
- A vertical guideline. This is shown by the cursor changing to a .
- Directly above or below the X co-ordinate of a polyline's Start Point (node). This is shown by the cursor changing to a .

Directly above or below the Y co-ordinate of a polyline's Start Point (node). This is shown by the cursor changing to a.

Note: The and cursors only appear when you are creating polylines. For details, see "Creating a Polyline" in the Working with Vectors chapter.

3D View Manipulation

You can change the view of a model using the tools above the **3D View** window:

2D View

Click on the **2D View** button to display the **2D View** window.

Note: You can also move from the **3D View** to the **2D View** window by pressing the **F2** key on your keyboard.

Twiddle Tool

The **Twiddle Tool** button allows you to rotate the model around a central point, adjust the viewpoint (pan) and zoom in on or out from the current viewpoint.

To rotate the model:

1. Click on the **Twiddle Tool** button.
2. Move the cursor over the model, click and hold down the left mouse button, and then drag the model to the required viewing angle.

To pan the model:

1. Click on the **Twiddle Tool** button.
2. Move the cursor over the model, click and hold down both mouse buttons, and then drag the model to the required viewing position.

To zoom in on the model:

1. Click on the **Twiddle Tool** button .
2. Move the cursor over the model, click and hold down the right mouse button, and then drag upwards.
3. Release the mouse button to set the viewpoint.

To zoom out from the model:

1. Click on the **Twiddle Tool** button .
2. Move the cursor over the model, click and hold down the right mouse button, and then drag downwards.
3. Release the mouse button to set the viewpoint.

Pan View

The **Pan View** button allows you to adjust the viewing position of the model:

1. Click on the **Pan View** button .
2. Move the cursor over the model, then click and drag the model into the required viewing position.

Note: You can also use the **Twiddle Tool** to adjust the viewpoint of a model. For details, see "Twiddle Tool" on page 24.

Zoom

You can use the **Zoom** button to zoom in on a defined area of the model:

1. Click on the **Zoom** button , then move the magnifying cursor over the area of the model that you want to zoom in on.
2. Click and drag the mouse to create a bounding box around the area of the model that you are interested in.
3. Release the left mouse button to view the area defined by the bounding box.

Note: You can also use the **Twiddle Tool** to zoom in on the model shown in the **3D View** window. For details, see "Twiddle Tool" on page 24.

Zoom Out

Click on the **Zoom Out** button to reduce the size of the image by 50%.

Previous View

Click on the **Previous View** button to restore the previous zoom setting.

Scale To Fit

The **Scale To Fit** button resizes the model so that it fits in the **3D View** window.

Isometric View

The **Isometric View** button displays the model in the standard isometric view. The viewing angle is shown by the XYZ representation on the button.

View Along X

The **View Along X** button displays the model from the X-axis.

View Along Y

The **View Along Y** button displays the model from the Y-axis.

View Along Z

The **View Along Z** button displays the model from the Z-axis.

High Detail Select Relief Detail

Click on the **Select Relief Detail** list box to display the list of options you can use when colour shading the relief, and then click on the option that you want to use:

- **Low Detail** – Select this option to colour shade the relief at a quarter of the model's pixel resolution.
- **Medium Detail** – Select this option to colour shade the relief at half of the model's pixel resolution.

- **High Detail** – Select this option to colour shade the relief equal to the model's pixel resolution. This option produces excellent visual clarity, but can mean that the relief takes longer to render.

Draw Zero Plane

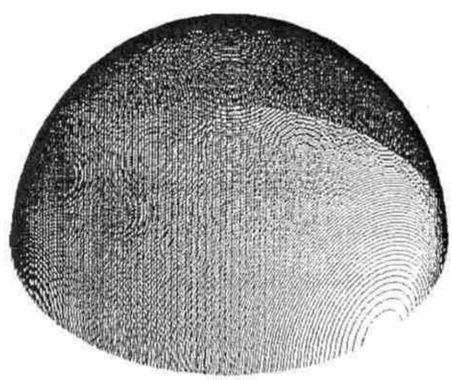
Click on the **Draw Zero Plane** button to hide the zero level of the relief from view.

If you want to restore the zero level of the relief, cliCck on the **Draw Zero Plane** button again.

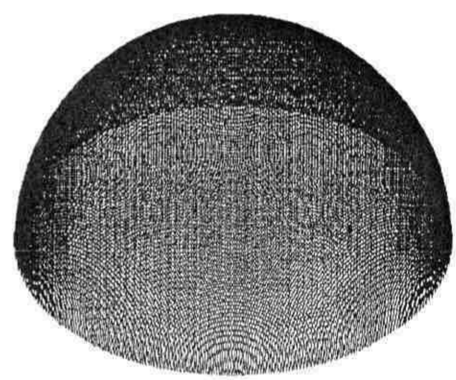
Draw X Y

Click on the **Draw X Y** button to draw vertical lines across the relief. Horizontal lines are drawn across the relief by default.

Draw X Y Off...



Draw X Y On...



Origin

The **Origin** button shows the X, Y and Z positions in the co-ordinate system used for the model.

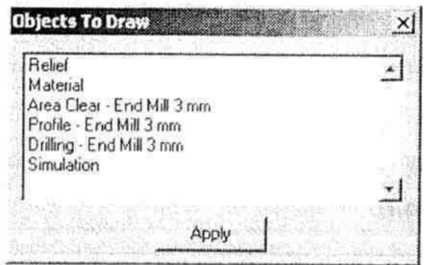
Click on the **Origin** button to display the origin in the **3D View** window.

The X-axis is red, the Y-axis is green and the Z-axis is blue.

If the origin is displayed in the **3D View** window, you can also click on the **Origin** button to hide the origin.

Objects To Draw

Click on the **Objects To Draw** button to display the **Objects To Draw** list box. The **Objects To Draw** list box lists the block of material, all of your calculated toolpaths and all of your toolpath simulations:



All items in the **Objects To Draw** list box are highlighted in blue in the **Objects To Draw** list box. If an item is not highlighted, it is not shown in the **3D View** window.

You can hide any of the objects displayed in the **3D View**:

1. Click to select the object in the list box that you want to hide. It is highlighted in blue.
2. Click on the **Apply** button to show the object in the **3D View** window.
3. Click on the in the top right corner of the **Objects To Draw** list box to close it.

To show any of the hidden objects in the **3D View**:

1. Click to select the object in the list box that you want to show. It is highlighted in blue.
2. Click on the **Apply** button to show the object in the **3D View** window.
3. Click on the in the top-right corner of the **Objects To Draw** list box to close it.

Colour Shade

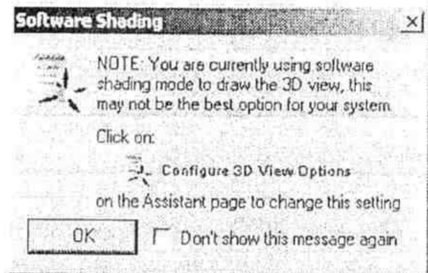
Click on the **Colour Shade** button to replace the relief or toolpath simulation with a colour shaded view.

3D View Options

You can choose whether ArtCAM Pro itself, or a graphics accelerator card using a set of fully Open GL compliant drivers, is used for shading the **3D View** window.

To adjust the 3D view options in ArtCAM Pro:

1. Start ArtCAM Pro. The **Software Shading** message box appears:

   If you do not want this message box to appear the next time that you start ArtCAM Pro, click to select the **Don't show this message again** option. Click on the **OK** button to close the message box.

2. Click on the **Configure 3D View Options** icon at the bottom of the **Assistant**'s **Getting Started** page to display the **3D View Options** page.

3. Click on either of the **Open GL Drawing Mode** radio buttons to select the method of shading that you want ArtCAM Pro to use when rendering your 3D models:

   - **Software Shading** – Select this option if you want ArtCAM Pro to shade a model using its own algorithms, independent of the graphics card that you have installed.

     If you have a fast processor installed alongside a relatively poor graphics card (common on high-end notebooks), you should select this option.

     If you select this option, ArtCAM Pro re-renders the model every time the viewpoint is adjusted or the model is changed in any way.

   - **Complete Open GL Support** – Select this option if you want ArtCAM Pro to shade a model using small triangles.

This option should only be selected if you have a fast processor, a graphics accelerator card and a set of fully compliant Open GL drivers installed. For example, NVIDIA's GeForce2 Ultra, GeForce 3 Graphics Processing Unit (GPU) or higher.

If you select this option, the model is always shaded. It also produces very fast rendering times and excellent visual clarity.

Warning: Many graphics accelerator cards have a limited subset of Open GL, used only to meet the system requirements of selected games. Please check with the manufacturer of your graphics accelerator card to confirm that it is fully Open GL compliant before selecting this option.

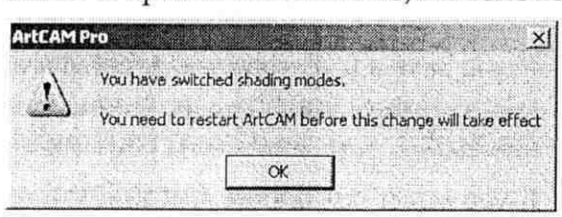
The size of the triangles that ArtCAM Pro uses to render a model in the **3D View** window is controlled by the list box in the **3D View** toolbar. The higher the detail, the smaller the triangles that are used. For details, see "3D View Manipulation" on page 24.

4. Click on the **Apply** button to confirm your **3D View** settings.
5. Click on the **Close** button to close the **3D View Options** page and return to the **Assistant**'s **Getting Started** page.

You can also display the **3D View Options** page in the following way:

1. From the Main menu bar, click on the **Window** menu, and then on the **3D View Options** option.

If you attempt to change the **Open GL Drawing Mode** while a model is open in ArtCAM Pro, the following message box appears:



Click on the **OK** button to close the message box, then shut down and restart ArtCAM Pro for the new **Open GL Drawing Mode** to take effect. For further details, see "Shutting Down ArtCAM Pro" in the Working with Models chapter.

Using the Top Toolbar

You can hide or view the Top toolbar which contains the **File**, **Model**, **Bitmap**, **Vector**, **Vector Editing**, **Vector Merging**, **Relief** and **Relief Editing** toolbars.

Hiding the Top toolbar increases the size of the central area of the screen, which in turn allows you to increase the size of the design windows.

If the Top toolbar is hidden, you can still access the options on the **File**, **Model**, **Bitmap**, **Vector**, **Vector Editing**, **Vector Merging**, **Relief** and **Relief Editing** toolbars from the Main menu bar. The Top toolbar is shown by default.

To hide the Top toolbar:

- From the Main menu bar, click on the **Window** menu, and then on the **Show Top Toolbar** option.

To view the Top toolbar:

- From the Main menu bar, click on the **Window** menu, and then on the **Show Top Toolbar** option.

Managing ArtCAM Pro's Preferences

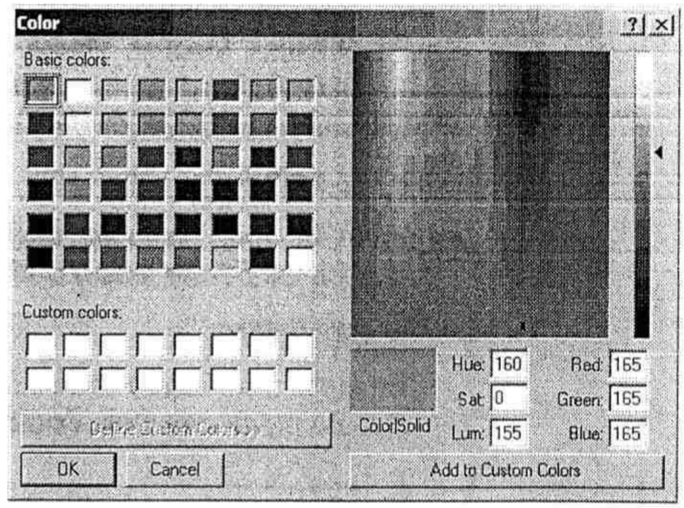
Many of the default settings in ArtCAM Pro can be controlled from one central location. Using the **ArtCAM Options** page, you can:

- Choose the colours associated with vector objects used to indicate their current state.
- Define the speed at which the relief rotates in the **3D View** window when spun.
- Define the default settings used for machining strategies within 2D and 3D toolpaths.
- Instruct ArtCAM Pro to check for or ignore self-intersections in imported vector artwork.
- Control the size of the design windows when ArtCAM Pro is started.

To adjust the aforementioned display settings in ArtCAM Pro:

1. Click on the **File** tab File in the Main toolbar to display the **File** toolbar.

2. Click on the **Options** button to display the **ArtCAM Options** page. The default colours and settings currently used in ArtCAM Pro are shown on the page.
3. In the **2D Drawing Colours** area, click on the arrow to display its settings:
    - To change the colour associated with a selected ungrouped vector object, click on the **Selected** option to display the **Color** dialog box:



      For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.

    - To change the colour associated with two or more selected ungrouped vector objects, click on the **Mulitiple Selected** option to display the **Color** dialog box. For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.
    - To change the colour associated with two or more overlapping vector objects, click on the **Overlapping** option to display the **Color** dialog box. For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.
    - To change the colour associated with 2D toolpath previews, click on the **Toolpath Preview** option to display the **Color** dialog box. For details on how to

select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.

- To change the colour associated with a selected grouped vector object, click on the **Selected** option in the **Grouped** area to display the **Color** dialog box. For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.
- To change the colour associated with locked vector objects when selected, click on the **Selected** option in the **Locked** area to display the **Color** dialog box. For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.
- To change the colour associated with locked vector objects when deselected, click on the **Unselected** option in the **Locked** area to display the **Color** dialog box. For details on how to select a colour, see "Assigning a Colour to a Layer" in the Working with Models chapter.

4. In the **3D View** area, click on the arrow to display its settings:

    - To change the frame rate at which a relief or triangle model in the **3D View** window rotates when spun, type the new speed in the **Spin Update Rate** box. For details on how to rotate a relief or triangle model, see "Rotating a Relief or Triangle Mesh" in the Working with Reliefs chapter.

5. In the **Machining** area, click on the arrow to display its settings:

    - Climb Milling rotates the toolbit in the same direction as the feed motion. To set climb milling as the default cutting direction used in toolpaths, click to select the **Use Climb Milling by Default** option.
    - A raster machining strategy machines in passes back and forth along a specified angle. To set the default angle for toolpaths that use a raster machining

strategy, type the angle in the **Default Raster Angle** box.

- When using an offset machining strategy in a toolpath, machining inwards from the outside edge of the block of material by default, click to select the **Start Offset Machining at Edge** option ☑.
- To show the cutting direction in all 2D toolpath previews shown in the **2D View** window, click to select the **Cutting Direction** option ☑.
- ArtCAM Pro shows rapid and plunge moves in all simulated toolpaths by default. To hide all rapid and plunge moves, click to deselect the **Rapid & Plunge Moves** option ☐.

6. In the **File Import** area, click on the ▼ arrow to display its settings:
   - ArtCAM Pro identifies all self-intersections within imported vector artwork by default. To ignore any self-intersections, click to deselect the **Check for Crossings** option ☐.

Note: You can use the **Vector Doctor** tool to check for self-intersections in vector artwork. For details, see "Using the Vector Doctor" in the Working with Vectors chapter.

7. In the **2D Drawing Options** area, click on the ▼ arrow to display its settings:
   - ArtCAM Pro replaces a selection of points (nodes) with a single vertical or horizontal linear span by default. To align a selection of points (nodes) with either the X or Y axis whilst preserving the points, click to deselect the **Align Nodes – replace with a single line** option ☐.
8. In the **Miscellaneous** area, click on the ▼ arrow to display its settings:
   - To maximise both of the design windows when ArtCAM Pro is started, click to select the **Maximise Views on Startup** option ☑.
9. Click on the **Apply** button to confirm your settings.

10. Click on the icon to return to the **Assistant**'s Home page.

Note: You can also press the **Esc** key on your keyboard or the icon at the top of the page to return to the **Assistant**'s Home page.

Working with Models

Getting Started

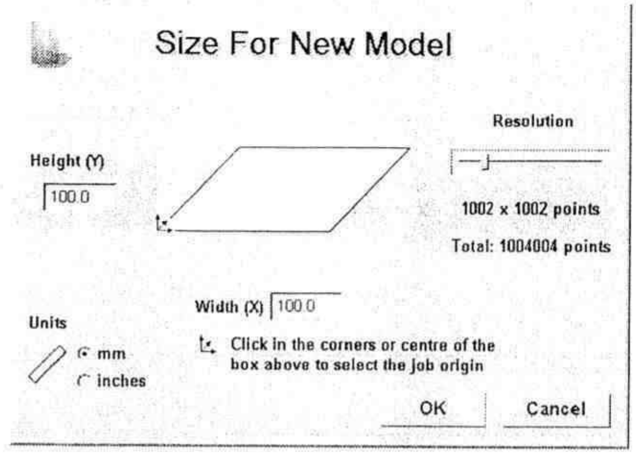
When you start ArtCAM Pro, there are four icons on the **Assistant**'s **Getting Started** page. These four icons are commands that enable you to:

- **Create a New Model** – Define the dimensions of a block of material that you want to produce a model from. For details, see "Creating a Model" on page 37.
- **Create a Model From an Image** – Open an image that you want to produce a model from. For details, see "Creating a Model from an Image" on page 39.
- **Open an Existing Model** – Open a previously created model or return to any of the previous four models that you have opened in ArtCAM Pro. For details, see "Opening an Existing Model" on page 40.
- **Font Editor** - Create your own fonts for use in ArtCAM Pro by editing characters within any installed fonts and then saving them. For details, see "Creating Fonts" on page 42.

Creating a Model

To create a model:

1. Click on the **Create New Model** icon  on the **Assistant**'s **Getting Started** page to display the **Size For New Model** dialog box:



2. Type in the **Height (Y)** and **Width (X)** values according to the physical size of the model you want to create.
3. Click on the centre or any of the four corners of the white box to define the X-axis zero and Y-axis zero origin.

   The **Job Origin** icon appears in the position you have clicked on.
4. Make sure that the **Units** option is set according to those you are working in, either millimetres or inches.
5. Click and drag on the slider to set the **Resolution** of your model.

   Drag the slider to the right to increase the resolution of the model. Drag the slider to the left to decrease it. Generally, a resolution of approximately 1,000,000 points is reasonable to work at.
6. Click on the **OK** button to create the model. ArtCAM Pro displays the **2D View** and **3D View** windows.

   The **2D View** window is the area in which you draw or import the bitmap images and vector objects, and import or create the reliefs, that make up your model. The **3D View** window is the area in which you can view a three-dimensional impression of your model.

   The model has a single default layer. For details, see "Creating a New Layer" on page 45.

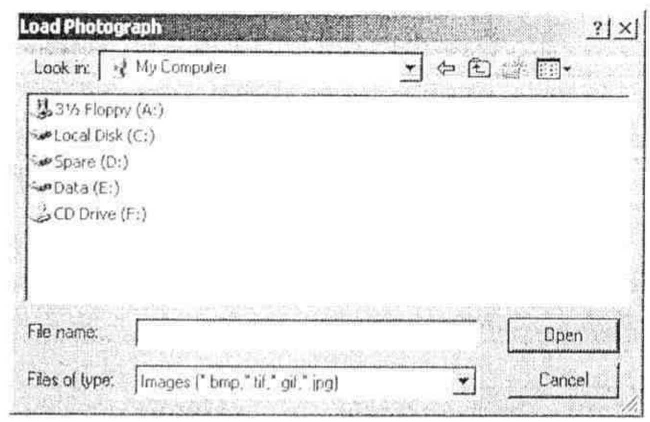
Note: To create a new model when ArtCAM Pro has already been started, click on the **New Model** button in the **File** area of the **Assistant**'s Home page.

Creating a Model from an Image

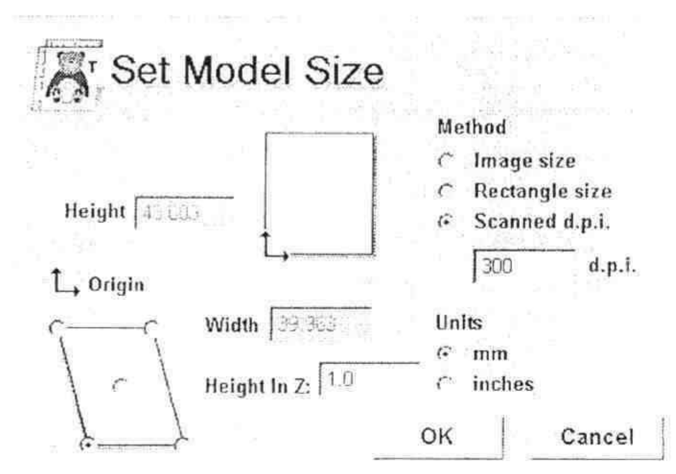
You can create a model by importing any the following types of image files as bitmaps into ArtCAM Pro: ***.bmp**, ***.tif**, ***.gif** and ***.jpg**. A relief is automatically calculated when the image is imported.

To create a model from a saved image file:

1. Click on the **Create Model From Image** icon on the **Assistant**'s **Getting Started** page to display the **Load Photograph** dialog box:



2. Click on the **Look in** list box and select the directory where the image file that you want to create a model from is stored.
3. Click to select the image file from which you want to create a model. The file name appears in the **File name** box.
4. Click on the **Open** button to display the **Set Model Size** dialog box:



5. Make sure that the **Units** option is set according to those you are working in, either millimetres or inches.
6. If you know the resolution (dpi) at which the image was scanned at, type this value in the **d.p.i.** box. If you do not, go straight to the next step.
7. You are now ready to set the size of the image. In the **Method** area, click on the **Image size** radio button to select it.
8. Type the value that you want to set the height (Y) of the image at in the **Height** box.
9. Type the value that you want to set the width (X) of the image at in the **Width** box.
10. To define the maximum depth of the relief that is to be created, type a value in the **Height In Z** box.

Tip: If you use an image with a high Z height, the resulting model is likely to be poor. You should only use images with a low Z height, such as textures.

Note: The default value in the **Height In Z** box is 1.0 irrespective of the units you are using. Make sure the Z height is correct.

11. Define the X-axis zero and Y-axis zero origin in the model. For further details, see "Editing the Model Dimensions" on page 79.
12. Click on the **OK** button to close the **Set Model Size** dialog box and create your model. The model has a single default layer. For details, see "Creating a New Layer" on page 45.

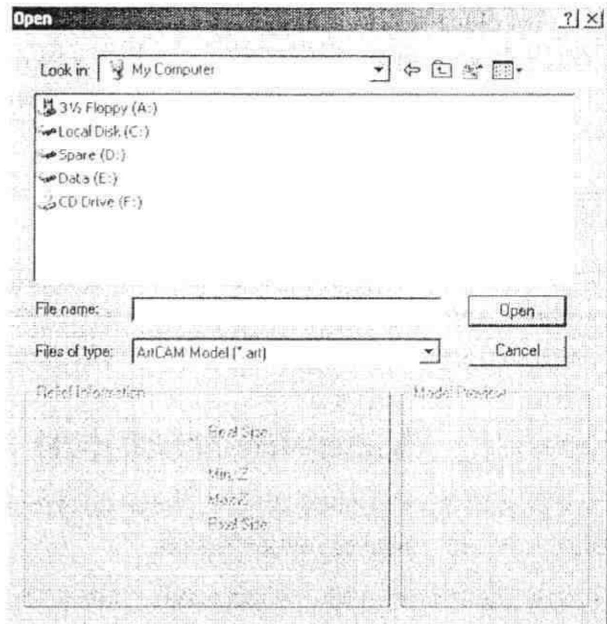
Note: If you have imported a colour image file, it is displayed as a greyscale image in the **2D View** window.

Note: You can also create a new model from a saved image file (***.bmp**, ***.tif**, ***.gif** or ***.jpg**) using the **File > New (From Image)** option from the Main menu bar.

Opening an Existing Model

To open an ArtCAM model that you have previously saved:

1. Click on the **Open Existing Model** icon on the **Assistant**'s **Getting Started** page to display the **Open** dialog box:



2. Click on the **Files of type** list box, and then on the file type of the model you want to open.

Note: You can open a model saved as an ***.art**, ***.bmp**, ***.tif**, ***.pcx**, ***.gif** or ***.jpg** file only.

3. Click on the **Look in** list box and select the directory where the model file that you want to open is stored.
4. Once you have found the model file, click to select it from the main window of the **Open** dialog box. Its name appears in the **File Name** box.
5. Click on the **Open** button to open the model.

Note: To open an existing model file when ArtCAM Pro has already been started, click on the **Open File** button in the **File** area of the **Assistant**'s Home page.

Opening Recent Models

To open any of the last four ArtCAM models (***.art**) that you have been working on:

1. Beneath the **Open Existing Model** icon on the **Assistant**'s **Getting Started** page, click on the name of the model that you want to open.

Tip: You can also open recently saved ArtCAM models by clicking on the **File** option in the Main menu bar, and then on the appropriate file name listed above the **Exit** option. Up to four models are listed.

Creating Fonts

You can create your own fonts by editing characters within any installed fonts and then saving them. For further information, see Creating Fonts in the Working with Vectors chapter.

Viewing Model Information

You can use the **Project** page to view information about the ArtCAM model you are currently working on.

To view information about the model that you are currently working on:

- Click on the **Project** tab to display the **Project** page in the **Assistant** window.

The **Project** page contains a tree that is divided into four model elements.

The icon denotes the root of the tree. If the ArtCAM Pro model has been saved, the name of the model file is shown beside the icon.

You can show the information in a model element by clicking on the ⊞ icon, or hide it by clicking on the ⊟ icon. The four model elements in the Project Tree are as follows:

- **Artwork** – This element shows the size of the model.
- **Views** – This element shows all of the open design windows in the model.

  Using this element, you can switch between the design windows open in the model. You can also create, delete or name a **2D View** window and calculate a relief. For details, see "Using the Design Windows" in the ArtCAM Pro Layout chapter and "Calculating a Relief" in the Working with Reliefs chapter.

- **Relief** – This element shows the dimensions of the relief in the model, its minimum and maximum height and its origin.
- **Machining** – This element shows the thickness of the material and all of the toolpath strategies that you have created.

  Using this element, you can adjust the material setup, and edit, calculate, delete or simulate a toolpath. You can also create a toolpath template. For details, see "Managing and Modifying Toolpaths" in the Machining Models chapter.

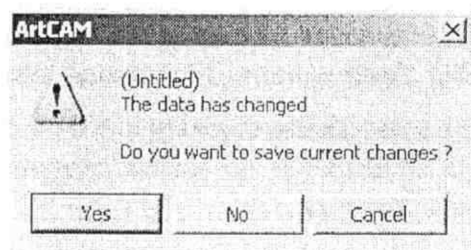
Closing a Model

To close the ArtCAM model that you are currently working on:

1. Click on the **File > Close** option from the Main menu bar.

   If you save the model before closing it, ArtCAM Pro returns to the **Getting Started** page.

   If you click on the **Close** option before saving a new model, or any changes that you may have made to an open model, the following message box appears:



   If you want to save the model:

   - Click on the **Yes** button to display the **Save As...** dialog box. For details, see "Saving a Model" on page 56.

   If you do not want to save the model you are working on:

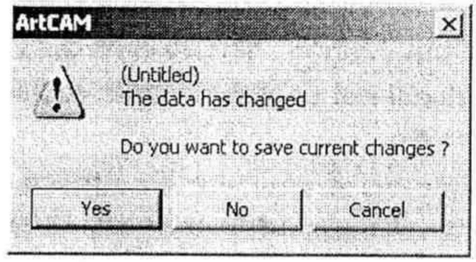
   - Click on the **No** button to close the message box. ArtCAM Pro returns to the **Getting Started** page.

Shutting Down ArtCAM Pro

To shut down ArtCAM Pro:

1. Click on the **File > Exit** option in the Main menu bar.

If you click on the **Exit** option before saving a new model, or any changes that you may have made to an open model, the following message box appears:



If you want to save the model:

- Click on the **Yes** button to display the **Save As...** dialog box. For further details, see "Saving a Model" on page 56.

If you do not want to save the model you are working on:

- Click on the **No** button to close the message box.

Working with Layers

All models created in ArtCAM Pro have a single default layer. You can, however, add as many as 990 layers to a model. You can turn layers on and off to view only the vector objects you are interested in, clearing the **2D View** window of all unnecessary detail.

The **Default Layer** for the model is listed on the **Layers** page. Vector objects on this layer are black by default. You can change a layer's colour at any time.

All newly created vector objects are created on the currently selected layer. You can select a layer listed on the **Layers** page by clicking on its name.

Layers are seen in the **2D View** window only. You can:

- Create layers to represent different parts of a model. For details, see "Creating a New Layer" on page 45.
- Name a layer to identify the vector objects on it. For details, see "Naming a Layer" on page 45.
- Assign a colour to a layer to be used for all the layer's vector objects. For details, see "Assigning a Colour to a Layer" on page 46.

- Hide layers, allowing you to isolate the part of the model that you want to work on. For details, see "Viewing a Layer" on page 48.
- Lock the vectors on a layer into position, or unlock them. For details, see "Locking the Vectors on a Layer" on page 49.
- Align a vector object on a layer with another vector object on any visible layer or guideline using snapping. For details, see "Snapping on a Layer" on page 49.
- Transfer selected vector objects from one layer to another. For details, see "Transferring Vectors Between Layers" on page 51.
- Merge the contents of one layer with another. For details, see "Merging Layers" on page 51.
- Delete a layer. For details, see "Deleting Layers" on page 52.

Creating a New Layer

You can create up to 990 layers in a model. For each layer that you create, the Visiblity and Snapping options are turned on by default, the Locking option is turned off, and the colour assigned to it is black. For further information, see "Manipulating Layers" on page 47.

To create a new layer:

1. Click on the **Layers** tab Layers to display the **Layers** page.
2. Click on the **Create** button to add a layer to the model. It is named *Layer* by default, and is numbered sequentially.
3. Type a name for the new layer.
4. Click on the button to apply the name to the layer. If you want to change the name, click on button, then follow the steps in "Naming a Layer" on page 45.

Naming a Layer

The name of a layer should reflect what the vector objects on it are used for in a model. You can change the name of any layer, other than the **Default Layer**, at any time. To do so:

1. Click on the **Layers** tab Layers to display the **Layers** page.
2. Click to select the layer that you want to rename. It is highlighted in blue.

Note: You cannot rename the **Default Layer** or layers that are not visible.

3. Click on the **Rename** button.

Note: You can also double-click on the existing layer name if you want to change it.

4. Type the new name for the layer over the current name.
5. Click on the ✓ button to apply the name to the layer. If you want to change the name, click on ✕ button and repeat the previous step.

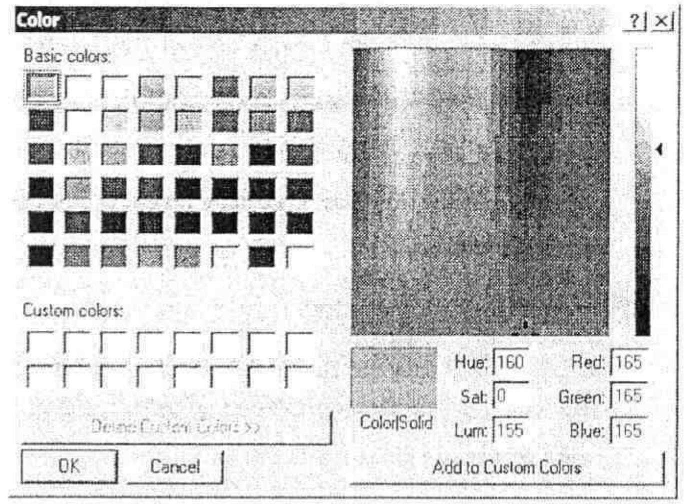
Assigning a Colour to a Layer

You can assign a colour to each layer. When you have done so, all of the vector objects on the layer are shown in that colour. This allows you to easily recognise the vector objects in the **2D View** window belonging to a specific layer in the model.

Tip: Avoid using blue, magenta or red when assigning a colour to a layer. These colours are used in ArtCAM Pro when you select vector objects or view toolpath previews.

1. Click on the **Layers** tab Layers to display the **Layers** page.

   Each layer is coloured black by default.
2. Click on the ● button for the layer whose colour you want to change. The **Color** dialog box is displayed:



3. Select the colour that you want to use for the layer. To select a Basic colour:

   - Click on a colour in the **Basic Colors** palette.

   To select a Custom colour:

   - Click on an approximate colour in the Colour Matrix, then click and drag on the slider at the right of the dialog box to adjust the colour's attributes.
   - Type values in the **Hue**, **Sat** (Saturation) and **Lum** (Luminosity), or the **Red**, **Green** and **Blue** boxes to specify the colour.
   - Click on the **Add To Custom Colors** button to add the colour to the **Custom Colors** palette.

4. Click on the **OK** button to close the **Color** dialog box and set the colour of the vector objects in the layer.

Manipulating Layers

You can change the settings for a layer as follows:

- Selecting a layer. For details, see "Selecting a Layer" on page 48.
- Hide a layer's content from view. For details, see "Viewing a Layer" on page 48.
- Lock vector objects on a layer into position. For details, see "Locking the Vectors on a Layer" on page 49.
- Snap one vector object on a layer in relation to another. For details, see "Snapping on a Layer" on page 49.

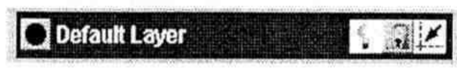
- Move vector objects from one layer to another. For details, see “Transferring Vectors Between Layers” on page 51.
- Merge one layer with another. For details, see “Merging Layers” on page 51.
- Delete a layer. For details, see “Deleting Layers” on page 52.

Selecting a Layer

You can select any layer listed on the **Layers** page. All newly created vector objects are drawn on the selected layer.

To select a layer:

1. Click on the **Layers** tab Layers to display the **Layers** page. The name of the currently selected layer is highlighted in blue, as shown below:



2. Click on the name of the layer that you want to select. It is highlighted in blue.

Viewing a Layer

You can turn the visibility for a layer on and off. When on, the vector objects are shown in the **2D View** window in the colour assigned to the layer. You can select vector objects from any of the visible layers in a model.

Note: You only hide the vector objects on a layer. If you also want to hide a toolpath preview associated with a vector object, see "Hiding a Toolpath" in the Machining Models chapter.

Note: You cannot merge a layer with another when its visibility is turned off. For details, see “Merging Layers” on page 51.

Note: If you try to create a vector object on the currently selected layer, for which the visibility is off, the visibility is automatically turned on.

1. Click on the **Layers** tab Layers to display the **Layers** page. All of the layers in the current model are listed.
2. To hide the content on any visible layer:

- Click on the **Toggle Visibility** button to the right of the appropriate layer's name.

To show the contents of any hidden layer:

- Click on the **Toggle Visibility** button to the right of the appropriate layer's name.

Locking the Vectors on a Layer

You can lock or unlock selected vector objects on a layer, so that they cannot be moved within it. Locking, however, does not prevent you from transferring vector objects from one layer to another.

To lock vector objects into position:

1. Click on the **Layers** tab Layers to display the **Layers** page. All of the layers in the current model are listed.
2. Hold the **Shift** key down, and then click to select the vector objects you want to lock into position. The vector objects turn magenta.
3. Click on the **Toggle Locking** button to the right of each of the layer's names to which the selected vector objects belong. The vector objects turn grey.

To unlock all of the vector objects on a layer:

1. Click on the **Layers** tab Layers to display the **Layers** page. All of the layers in the current model are listed. All locked vector objects appear as grey in the **2D View** window.
2. Click on the **Toggle Locking** button to the right of the appropriate layer's name. The vector objects turn to the colour assigned to the layer.

Snapping on a Layer

Using snapping, you can align a vector object on the active layer with another vector object on any visible layer, or a guideline. For further information on guidelines, see "Using Guidelines" in the ArtCAM Pro Layout chapter.

The **Snap To Objects** option in the **2D View** menu must be turned on ✓ Snap To Objects to use the **Toggle Snapping** button on the **Layers** page.

You can turn snapping off and on:

- From the Main menu bar, click on the **2D View** menu, followed by the **Snap to Objects** option.

  When turned on, the **Snap to Objects** option is checked ✓ Snap To Objects , and vice versa.

You can turn the snapping for a layer on and off.

- Click on the **Toggle Snapping** button to turn off snapping on the layer.
- Click on the **Toggle Snapping** button to turn on snapping on the layer.

Tip: To temporarily disable snapping, hold down the **Shift** key on your keyboard.

To use snapping on a layer:

1. Make sure that the **Snap to Objects** option is checked on ✓ Snap To Objects .
2. Make sure that the **Toggle Snapping** button is turned on for each of the layers on which you want to reposition and align vector objects.
3. Click to select a vector object from any visible layer, and then drag it into its new position. You can tell when it is aligned correctly as follows:
   - If it is aligned with a point (node) in another vector object, the cursor changes to a .
   - If it is aligned with the mid-point of a linear or arc span within another vector object, the cursor changes to a .
   - If it is aligned with the centre of another vector object, as defined by a bounding box, the cursor changes to a .

Tip: If you hold the **X** key down, the cursor also changes to where two vector objects intersect.

   - If it is aligned with a point at which two guidelines intersect, the cursor changes to a .

- If it is aligned with a horizontal guideline, the cursor changes to a .
- If it is aligned with a vertical guideline, the cursor changes to a .
- If it is aligned directly above or below the X co-ordinate of a polyline's Start Point (node), the cursor changes to a .
- If it is aligned directly above or below the Y co-ordinate of a polyline's Start Point (node), the cursor changes to a .

Note: The and cursors only appear when you are creating polylines. For details, see "Creating a Polyline" in the Working with Vectors chapter.

Transferring Vectors Between Layers

You can move all or a selection of vector objects from one layer to another.

To transfer a selection of vector objects from one layer to another:

1. Click on the **Layers** tab Layers to display the **Layers** page.
2. Hold the **Shift** key down, and then click to select each of the vector objects shown in the **2D View** window that you want to transfer. The vector objects turn magenta and are surrounded by a bounding box.

Tip: If you want to transfer all of the vector objects in the currently selected layer only, click on the **Select All** button.

---

3. In the list box at the bottom of the **Layers** page, click on the layer to which you want to transfer the selected vector objects.

   The vector objects are transferred to the chosen layer and are redisplayed using the chosen layer's colour.

Merging Layers

You can merge all the visible layers with the currently selected layer.

1. Click on the **Layers** tab Layers to display the **Layers** page. All of the layers in the current model are listed.
2. Click to select the layer you want all visible layers in the model to be merged with. It is highlighted in blue.

   For example, if you select the default layer, it appears as follows:

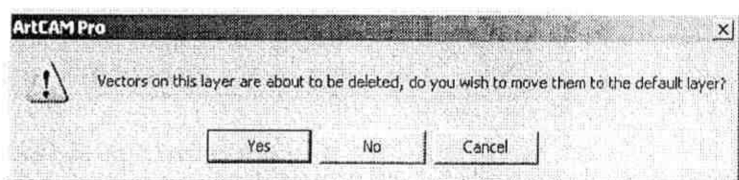
3. Make sure that the visibility is turned off for all of the layers that you do not want to merge with the currently selected layer. For details, see "Viewing a Layer" on page 48.
4. Click on the **Merge Visible** button.

   The visible layers are merged with the currently selected layer. The vector objects are redisplayed using the chosen layer's colour.

Deleting Layers

You can delete any layer, choosing either to delete the vector objects on it or to transfer them to the default layer:

1. Click on the **Layers** tab Layers to display the **Layers** page. All of the layers in the current model are listed.
2. Click to select the layer that you want to delete. It is highlighted in blue.
3. Click on the **Delete** button to display the following message box:

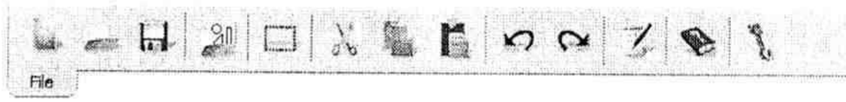
4. If you want to delete both the layer and its vector objects, click on the **No** button.

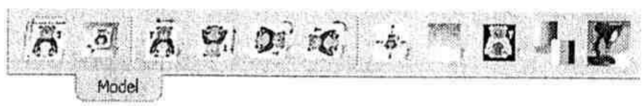
   If you want to transfer the layer's vector objects to the default layer, click on the **Yes** button.

Managing a Model

When you have created or opened a model in ArtCAM Pro, the **Assistant**'s Home page is automatically displayed.

You can use the buttons in the **File** and **Model** areas of the **Assistant**'s Home page to help you manage a model. All of these buttons can also be found on the **File** and **Model** toolbars:





Using the buttons on the **File** toolbar you can:

- Create a new model. For details, see "Creating a New Model" on page 54.
- Open a saved model. For details, see "Opening a Model" on page 56.
- Save an open model. For details, see "Saving a Model" on page 56.
- Import vector files created in other applications. For details, see "Importing Vector " on page 63.
- Correct a mistake that you have made. For details, see "Correcting an Action" on page 78.
- Display or hide a note file attached to a model. For details, see "Using the ArtCAM Pro Notepad" on page 96.
- Open the Help for ArtCAM Pro.

Note: Click on the **Help Index** button  in the **File** toolbar to display ArtCAM Pro's Online Help.

- Adjust the settings for ArtCAM Pro. For details, see "Adjusting Light and Material Settings" on page 87.

Using the buttons on the **Model** toolbar you can:

- Edit the dimensions of a model. For details, see "Editing the Model Dimensions" on page 79.

- Set the position of a model. For details, see "Setting the Position of a Model" on page 82.
- Mirror a model. For details, see "Mirroring a Model" on page 78.
- Rotate an open model. For details, see "Rotating a Model" on page 79.
- Add a border to a model. For details, see "Adding a Border to a Model" on page 85.
- Clear the contents of a model. For details, see "Clearing a Model" on page 84.
- Create a greyscale image from a relief. For details, see "Creating a Greyscale Image from a Relief" on page 85.
- Reduce the colours in a model. For details, see "Reducing Colours" in the Working with Bitmaps chapter.
- Adjust the lights and material settings for a relief. For details, see "Adjusting Light and Material Settings" on page 87.

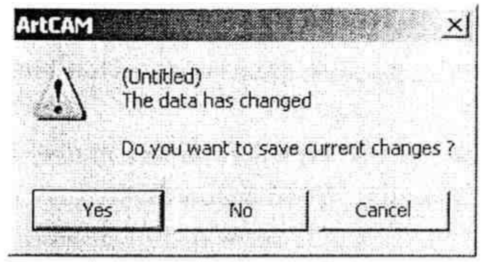
Creating a New Model

To create a new model:

1. Click on the **New Model** button  in the **File** area to open the **Size For New Model** dialog box. For details, see "Creating a Model" on page 37.

Note: You can also display the **Size For New Model** dialog box by clicking on **File** in the Main menu bar, and then on the **New...** option.

If you click on the **New Model** button whilst working on an unsaved model, and then on the **OK** button in the **Size For New Model** dialog box, the following message box appears:



If you want to save the model:

- Click on the **Yes** button to display the **Save As...** dialog box. For further details, see "Saving a Model" on page 56.

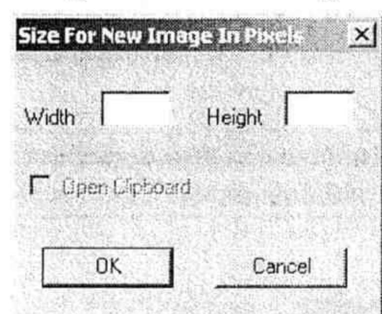
If you do not want to save the model you are working on:

- Click on the **No** button to close the message box.

Creating a New Model Using Pixels

To create a model made up of an exact number of pixels:

1. From the Main menu bar, click on the **File > New (Specify Pixel Size)** option to display the **Size For New Image In Pixels** dialog box:



   If you have a bitmap image copied to the Windows clipboard, the **Open Clipboard** option is available. If you do not, the option is greyed out.

2. If you want to use the image currently on the Windows clipboard as the new model, make sure that the **Open Clipboard** option is checked on ☑.

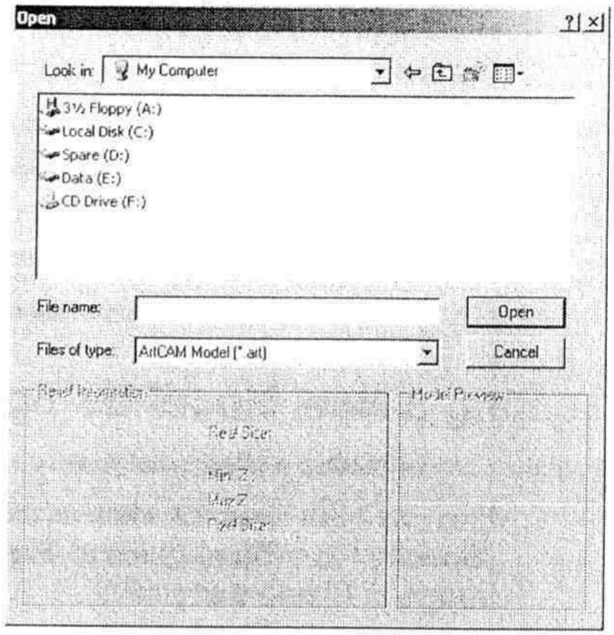
   The number of pixels in the clipboard image appears in the **Width** and **Height** boxes.

3. Type the width (X) of the model in the **Width** box.
4. Type the height (Y) of the model in the **Height** box.
5. Click on the **OK** button to create the model.

Opening a Model

To open a model that you have previously saved:

1. Click on the **Open File** button in the **File** area to display the **Open** dialog box:



For further details, see "Opening an Existing Model" on page 40.

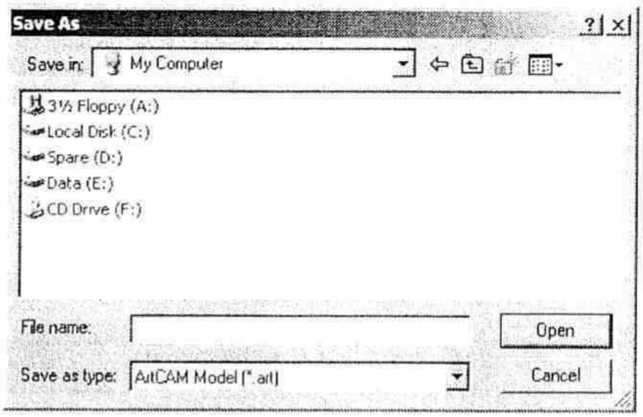
Note: You can also open the **Open** dialog box if you click on **File** in the Main menu bar, and then on the **Open** option.

Saving a Model

You can save a model as an ArtCAM model file (***.art**), Windows Bitmap (***.bmp**), TIFF (***.tif**), PCX (***.pcx**), GIF (***.gif**), or JPEG (***.jpg**) image.

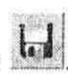
To save a model for the first time:

1. Click on the **Save** button in the **File** area to display the **Save As...** dialog box:



2. Click on the **Save In** list box and select the directory in which you want to save the model.
3. Click on the **Save as type** list box, and then on the file type you want to save the model as.
4. Type the file name you want to use for the model in the **File name** box.
5. Click on the **Save** button.

To save the changes you have made to a model since it was last saved:

1. Click on the **Save** button .

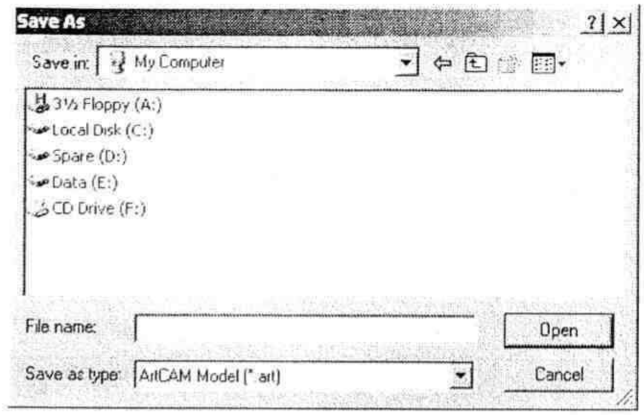
Note: You can also save a model if you click on **File** in the Main menu bar, and then on the **Save** option.

Saving the 3D View as an Image

You can save the contents of the **3D View** window as a Windows Bitmap (***.bmp**), TIFF (***.tif**) or JPEG (***.jpg**) image.

To save the model displayed in the **3D View** window as an image:

1. Click on the **3D View** window that you want to save as an image to make sure that it is active.
2. From the Main menu bar, click on the **Windows** option to display the **Windows** menu, and then on the **Save 3D View Image** option. The **Save As** dialog box appears:



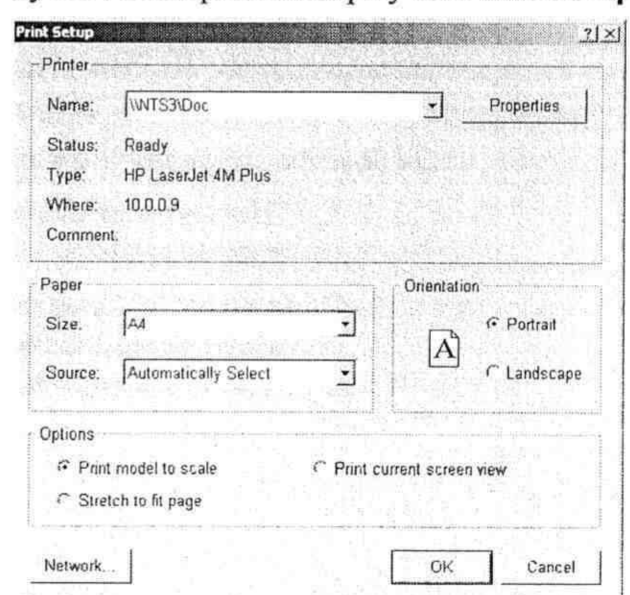
3. Click on the **Save In** list box and select the directory in which you want to save the image.
4. Type a name for the image in the **File Name** box.
5. Click on the **Save As Type** list box, and then on the type of image that you want to save the contents of the **3D View** window as (*.bmp, *.tif or *.jpg).
6. Click on the **Save** button to save the image.

Printing a Model

You can print a model as it is shown in the active design window.

To print a model:

1. Click to select the design window that you want to print.
2. From the Main menu bar, click on the **File** option, followed by the **Print** option to display the **Print Setup** dialog box:



3. Make sure that your printer settings are correct. For details, see "Print Setup" on page 61.
4. In the **Options** area, click on the print option that you want to use:

Note: If you are printing the **3D View** window, the print options are automatically greyed-out. In this instance, ArtCAM Pro prints to fit the **3D View** window in its current orientation.

- **Print model to scale** – Click on this radio button if you want to print the model according to its physical dimensions.

  Before printing, make sure that the model can fit onto the paper onto which it is to be printed. For details, see "Print Setup" on page 61.

- **Stretch to fit page** – Click on this radio button if you want to print the model according to the dimensions of the paper onto which it is to be printed.
- **Print current screen view** – Click on this radio button if you want to print the model as it currently appears in the **2D View** window.

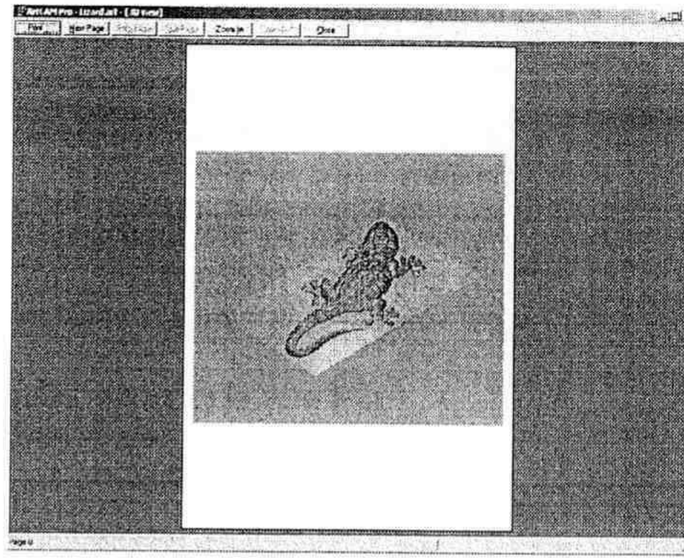
5. Click on the **OK** button to close the **Print Setup** dialog box and print the active design window.

Print Preview

You can view the content of the active design window as it would appear when printed.

To create a print preview:

1. Click to select the design window that you want to print.
2. From the Main menu bar, click on the **File** option, followed by the **Print Preview** option to display a preview image of the active design window:



3. You can use the buttons in the bar at the top of the **ArtCAM Pro** window to view the preview image in more detail:

   - To show the next page of the preview image, click on the **Next Page** button. If this is greyed out, there are no further pages.
   - To show the previous page in the preview image, click on the **Prev Page** button. If this is greyed out, there are no further pages.
   - To show two pages in the preview image at the same time, click on the **Two Page** button. If this is greyed out, there is only one page in the preview image.
   - To enlarge the size of the preview image, click on the **Zoom In** button.

Note: You can also move the cursor over the area you want to enlarge, and then click.

   - To reduce the size of the preview image, click on the **Zoom Out** button.

4. You are now ready to print the preview image:

   - If you want to print the preview image, click on the **Print** button. For details, see "Printing a Model" on page 58.
   - If you do not want to print the preview image, click on the **Close** button.

Print Setup

You can control the settings for the printer to which the content of the active design window is printed in ArtCAM Pro.

To adjust the printer settings:

1. Click to select the design window that you want to print.
2. From the Main menu bar, click on the **File** option, followed by the **Print Setup** option to display the **Print Setup** dialog box:
3. Click on the **Name** list box and then on the printer that you want to use.

   If you want to change any of the default settings of the printer, such as the paper size and orientation, click on the **Properties** button.
4. In the **Paper** area, click on the **Size** list box and select the size of paper that you want to print to.
5. In the **Paper** area, click on the **Source** list box and then on the tray or paper feed option that you want to use when printing.
6. Select the orientation that you want to use when printing:
   - **Portrait** - Click on this radio button if you want to print the model vertically. Typically, portrait orientation is used for models that are taller than they are wide.
   - **Landscape** - Click on this radio button if you want to print the model horizontally. Typically, landscape orientation is used for models that are wider than they are tall.
7. In the **Options** area, click on the print option that you want to use:
   - **Print model to scale** – Click on this radio button if you want to print the model according to its physical dimensions.

     Before printing, make sure that the model can fit onto the paper to which it is to be printed.
   - **Stretch to fit page** – Click on this radio button if you want to print the model according to the

dimensions of the paper onto which it is to be printed.

- **Print current screen view** – Click on this radio button if you want to print the model as it currently appears in the **2D View** window.

Note: If you are printing the **3D View** window, the print options are automatically greyed-out. In this instance, ArtCAM Pro prints to fit the **3D View** window in its current orientation.

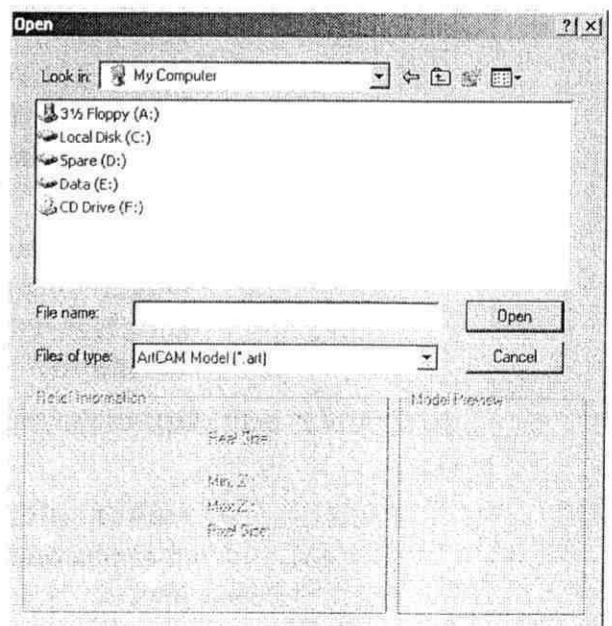
8. Click on the **OK** button to print the active design window and close the **Print Setup** dialog box.

Importing Images

In ArtCAM Pro, you can use image files created in other drawing packages as part of a model.

If you want to import an image saved as files of type ***.bmp**, ***.tif**, ***.pcx**, ***.gif** or ***.jpg** into a model:

1. Click on the **Open** option from the **File** menu to display the **Open** dialog box:



2. Click on the **Files of type** list box, and then on the image file type that you want to import.
3. Click on the **Look In** list box and find the image file that you want to import.

4. Once you have found the image file, click to select the file listed in the main window of the **Open** dialog box. Its name appears in the **File Name** box.
5. Click on the **Open** button to import the image file into ArtCAM Pro.

Importing Vector Artwork

In ArtCAM Pro, you can import vector artwork saved as files of type ***.dxf**, ***.eps**, ***.dwg**, ***.ai**, ***.wmf** or ***.pic** into a model.

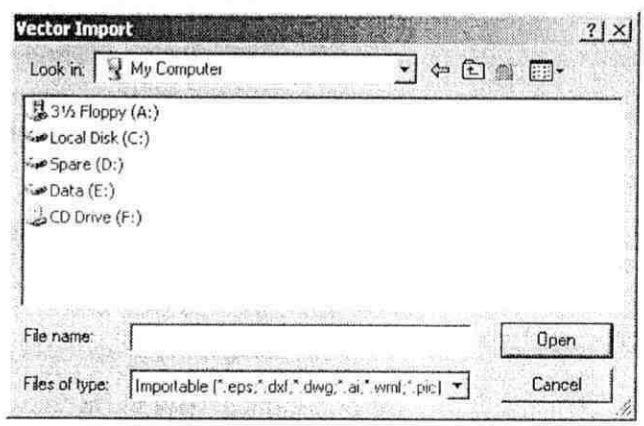
When importing Drawing Interchange (***.dxf**) or AutoCAD Drawing (***.dwg**) files, you can position the artwork in the centre of the ArtCAM model, define the units of measurement used, instruct ArtCAM Pro to identify loops within tolerance and rejoin all coincident points (nodes) within tolerance. ArtCAM Pro also reads all layer information saved within these files.

When importing Encapsulated PostScript (***.eps**, ***.ai**) files with an origin outside of an ArtCAM model, you can position the artwork in the centre of the ArtCAM model or according to the origin saved in the files.

Windows Meta (***.wmf**) and Lotus or PC Paint PIC (***.pic**) files import into ArtCAM Pro directly.

To import vector artwork:

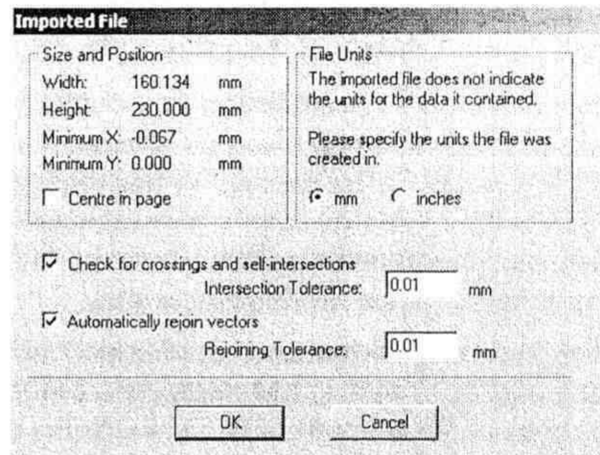
1. Click on the **Import Vector Data** button  in the **File** area of the **Assistant**'s Home page to display the **Vector Import** dialog box:



Note: You can also open the **Vector Data** dialog box by clicking on **File > Import > Vector Data** from the Main menu bar.

2. Click on the **Look in** list box and select the directory where the file containing the vector artwork that you want to import is stored. Once you have found the file, click on its name.
3. Click on the **Open** button to import the vector artwork.

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



   You can use the **Imported File** dialog box as follows:

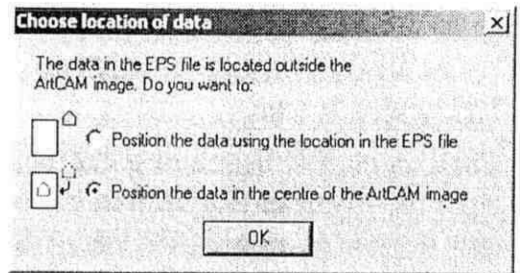
   - To position the artwork in the centre of the current ArtCAM model, click to select the **Centre In Page** option in the **Size and Position** area.
   - Define the units of measurement used in the artwork by clicking on either of the radio buttons in the **File Units** area.
   - If you want ArtCAM Pro to identify all loops within any self-intersecting vector objects in the artwork and then remove them, make sure that the **Check for Crossings and Self-Intersections** option is selected.

     You can define the size of loops that you want ArtCAM Pro to ignore in the **Intersection Tolerance** box.
   - If you want ArtCAM Pro to identify all coincident points in the artwork and then join them together, make sure that the **Automatically Rejoin Vectors** option is selected.

If you only want ArtCAM Pro to join coincident points in the artwork within a specific distance of one another, type the distance in the **Rejoining Tolerance** box.

- Click on the **OK** button to import the vector artwork.

If you are importing an Encapsulated PostScript file (***.eps**, ***.ai**) with its origin is outside of the current ArtCAM model, the **Choose Location of Data** dialog box is displayed:



Click on either of the radio buttons to select where you want to position the imported data in the ArtCAM model, and then click on the **OK** button to import the vector artwork.

Using a Vector Library

You can use the **Vector Library** tool to manage vector artwork saved as Encapsulated PostScript (***.eps** and ***.ai)** files that you want to import into an ArtCAM model. ArtCAM Pro recalls the position in which the vector object was saved.

A vector library is made up of at least one folder within a parent directory that contains one or more *.eps or *.ai files.

ArtCAM Pro creates an area on the **Vector Library** page for each of the folders within a library that contain *.eps and *.ai files. If an *.eps or *.ai file within a library shares the same name as a *.jpg or *.gif file also within the same library, ArtCAM Pro creates an icon next to its filename. If there are no *.eps or *.ai files in a selected directory, a message is displayed on the page indicating that no library has been found.

You can click on the arrow on the page to hide the list of files within a library's folder, and on the arrow to reveal the list.

To select a folder as a vector library:

1. Click on the **Import From a Vector Library** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Vector Library** page.
2. If you cannot see the vector library settings, click on the **Choose Library** arrow to display them. Any existing vector libraries are listed in the **Libraries** box.
3. Click on the **New** button to display the **Browse For Folder** dialog box.
4. Click to select the directory in which you want to store your vector artwork files, or in which they are already kept. It is highlighted in blue.
5. Click on the **OK** button to select the directory, and then close the **Browse For Folder** dialog box. The directory is now defined as an ArtCAM vector library.
6. Click on the icon to return to the **Assistant**'s Home page.

Note: You can also press the **Esc** key on your keyboard or the icon at the top of the page to return to the **Assistant**'s Home page.

To import a file from a vector library:

1. Click on the **Import From a Vector Library** button in the **Vector Editing** area to display the **Vector Library** page.
2. If you cannot see the vector library settings, click on the **Choose Library** arrow to display them. All existing vector libraries are listed in the **Libraries** list box.
3. Move the cursor over the filename in the list containing the vector artwork that you want to import, and then click. The vector artwork appears in the **2D View** window.
4. Click on the icon to return to the **Assistant**'s Home page.

Note: You can also press the **Esc** key on your keyboard or the icon at the top of the page to return to the **Assistant**'s Home page.

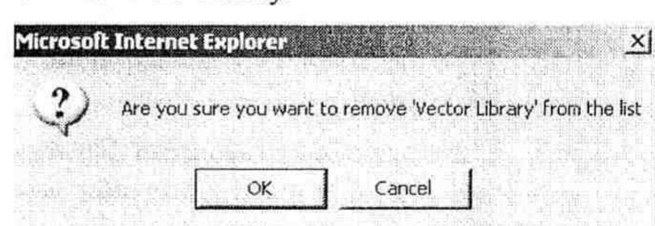
To refresh the list of files identified within a library:

1. Click on the **Import From a Vector Library** button in the **Vector Editing** area to display the **Vector Library** page.
2. If you cannot see the vector library settings, click on the **Choose Library** arrow to display them. All existing vector libraries are listed in the **Libraries** box.
3. Click on the **Libraries** list box, and then on the library for which you want to update the listing.
4. In the **Options** area, click on the **Rescan** button to refresh the list of folders and files within the current library.
5. Click on the icon to return to the **Assistant**'s Home page.

Note: You can also press the **Esc** key on your keyboard or the icon at the top of the page to return to the **Assistant**'s Home page.

To remove a library:

1. Click on the **Import From a Vector Library** button in the **Vector Editing** area to display the **Vector Library** page.
2. If you cannot see the vector library settings, click on the **Choose Library** arrow to display them. All existing vector libraries are listed in the **Libraries** box.
3. Click on the **Libraries** list box, and then on the library you want to remove.
4. In the **Options** area, click on the **Remove** button to remove the selected library.
5. A message box appears asking you to confirm your decision to remove the library:



Click on the **OK** button to close the message box and remove the library.

6. Click on the icon to return to the **Assistant**'s Home page.

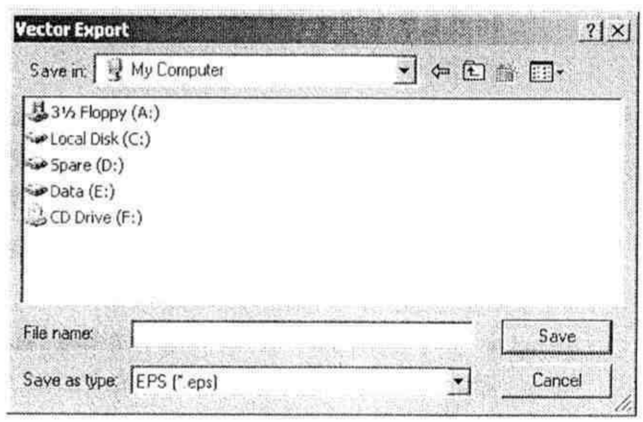
Note: You can also press the **Esc** key on your keyboard or the icon at the top of the page to return to the **Assistant**'s Home page.

Exporting Vector Data

You can export vector objects as data. This data can then be used in other drawing packages supporting ***.eps**, ***.dxf** and ***.pic** file formats.

To export vector objects as data:

1. Make sure that you are in **Select Vectors** mode . For details, see
2. Hold the **Shift** key down, and then click to select each of the vector objects that you want to export.
3. From the Main menu bar, click on the **File** option followed by the **Export** option to open the **Vector Export** dialog box:

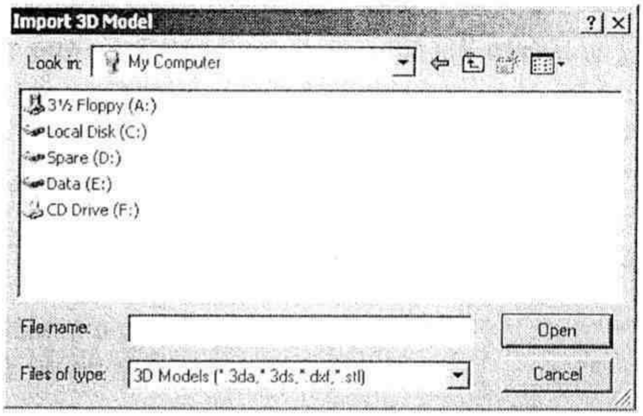


4. Click on the **Save In** list box and select the directory in which you want to save the vector data.
5. Type the file name you want to use for the vector data in the **File name** box.
6. Click on the **Save as type** list box, and then on the file type you want to save the vector object as.
7. Click on the **Save** button.

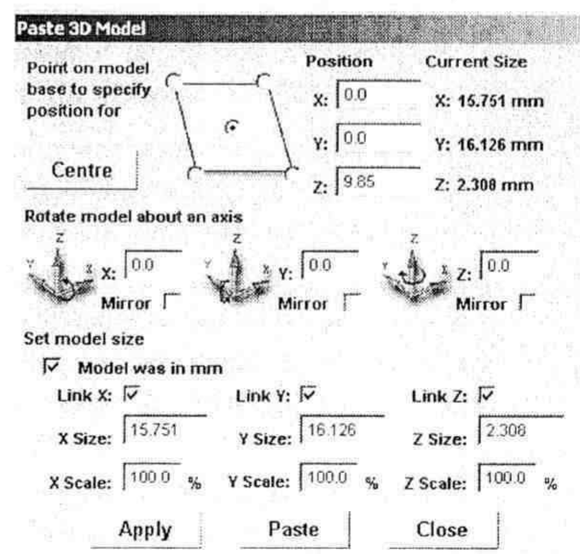
Importing a 3D Model File

You can import triangle models saved as 3D Assembly (***.3da**) 3D Studio (***.3ds**), Drawing Interchange Format (***.dxf**) and STL (***.stl**) files into an open ArtCAM model:

1. From the Main menu bar, click on the **File > Import > 3D Model** option to display the **Import 3D Model** dialog box:



2. Click on the **Look In** list box and find the file that you want to import.
3. Once you have found the file, click to select the file listed in the main window of the **Import 3D Model** dialog box. Its name appears in the **File Name** box.
4. Click on the **Open** button to import the file into the open model and display the **Paste 3D Model** dialog box:



The current size of the model is shown and the original co-ordinate system used when the triangle model was constructed is kept.

5. You can now use the **Paste 3D Model** dialog box to set the position, size and origin of the triangle model before it is pasted into the relief within the ArtCAM model.

Using the Paste 3D Model Dialog Box

The **Paste 3D Model** dialog box allows you set the position, size and origin of an imported three-dimensional model before it is pasted into the relief in an ArtCAM model.

To define how the triangle model is pasted into the relief:

1. Set the position of the triangle model within the ArtCAM model:
    - To position the triangle model in the centre of the ArtCAM model, click on the **Centre** button.
    - To position the triangle model in the centre or one of the four corners in the ArtCAM model, click on the appropriate **Point on base to specify position for** radio button on the model diagram.
    - To position the triangle model in a specific location within the ArtCAM model, type the co-ordinates in the **X**, **Y** and **Z** boxes.
2. To rotate the triangle model on any of the three axes:
    - Type a value in the **X**, **Y** and/or **Z** box in the **Rotate model about an axis area**.
    - To mirror the model in any of the three axes, make sure that the **Mirror** option below the **X**, **Y** and/or **Z** box is selected.
3. If you are working in inches, the **Model was in inches** option is selected by default. If the triangle model appears too large, either click on the option to turn it off and scale the 3D model between inches and mm, or go straight to the next step to resize or scale the triangle model.

    If you are working in millimetres, the **Model was in mm** option is selected by default. If the triangle model appears too small, either click on the option to deselect it and scale the triangle model between mm and inches, or go straight to the next step to resize or scale the triangle model.

4. Set the size of the triangle model:

    - To resize or scale the triangle model in a specific axis, click on the **Link** options for the axes that you do not want to adjust to deselect them ☐. For example, if you only want to resize the model in the X-axis, click on the **Link Y** and **Link Z** options to deselect them.

      All three **Link** options are checked on ☑ by default to keep the dimensions of the triangle model in proportion.

    - Type the new size for the axis you want to adjust in its **Size** box.

    - Type the new scale for the axis you want to adjust in its **Scale** box.

5. Click on the **Apply** button.

Tip: Click on the **Apply** button whenever you type in a new value in the **Paste 3D Model** dialog box so you can see how your model is changing.

---

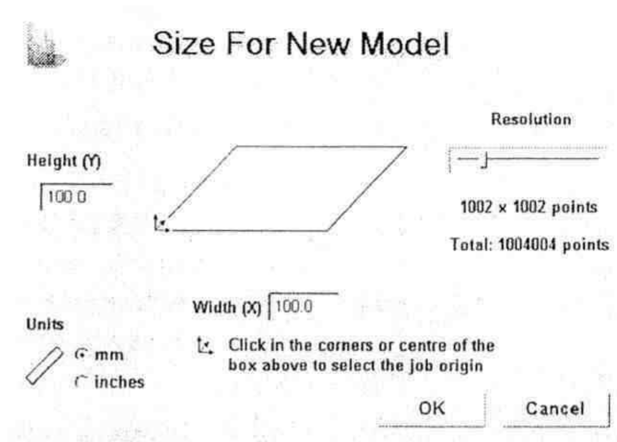
6. Click on the **Paste** button to paste the triangle model into the relief within the ArtCAM model using the **Merge Highest** option. For details, see

7. Click on the **Close** button to close the **Paste 3D Model** dialog box and paste the triangle model into the relief within the ArtCAM model.

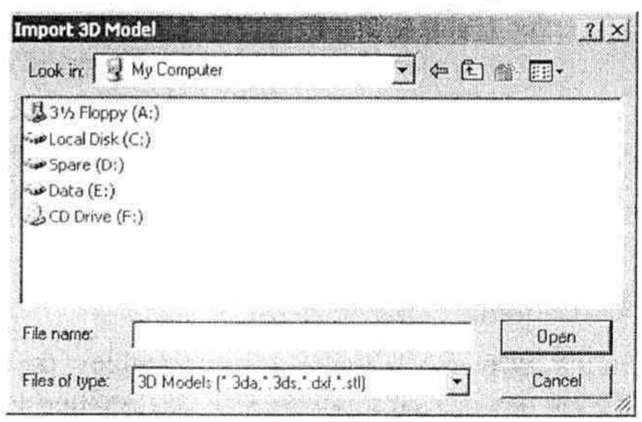
Example

In the following example, you can see how the **Paste 3D Model** dialog box is used to set the size and position of an imported triangle model, and then paste it into the relief in an ArtCAM model.

1. Click on the **Create New Model** icon on the **Getting Started** page to display the **Size For New Model** dialog box:

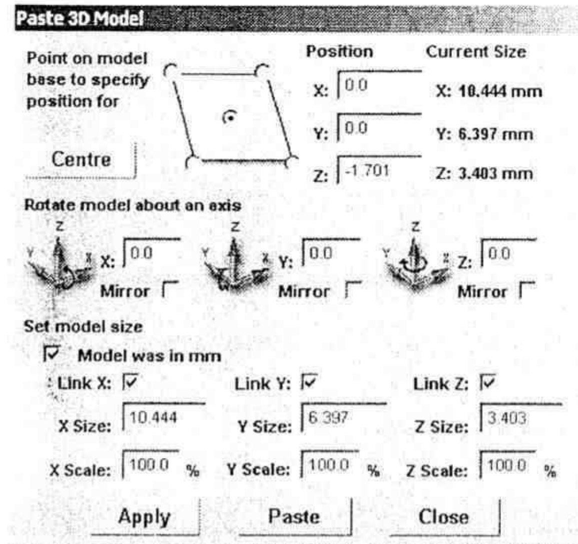


2. Click on the **mm** radio button in the **Units** area to set the units of measurement as millimetres.
3. Type a value of *100* in the **Height (Y)** and **Width (X)** boxes.
4. Click and drag on the **Resolution** slider and set it to *1004004* points.
5. Click on the **OK** button to close the **Size For New Model** dialog box and create the ArtCAM model.
6. From the Main menu bar, click on the **File** option to display the **File** menu, then on the **Import > 3D Model** option to display the **Import 3D Model** dialog box:

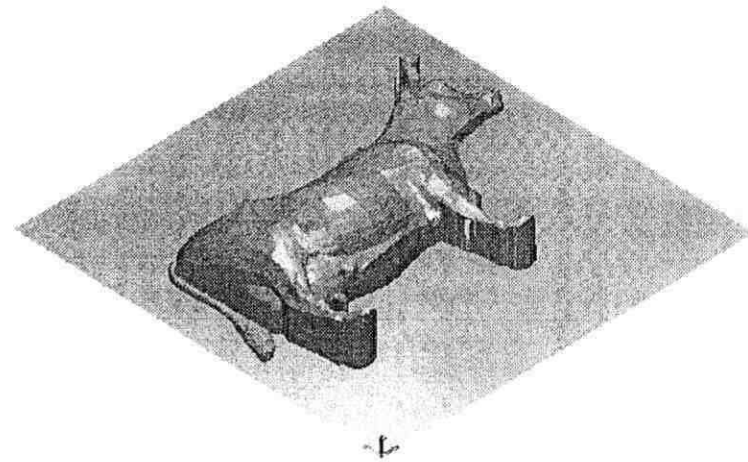


7. Click on the **cow.dxf** file in ArtCAM Pro 6.0\Examples\3d models to select it, and then click on the **Open** button to import the triangle model into your ArtCAM model.

   The cow triangle model appears on the origin of the ArtCAM model in the **3D View** window. The **Paste 3D Model** dialog box also appears:



8. You are now ready to set the position of the cow triangle model in the ArtCAM model. In the **Position** area of the **Paste 3D Model** dialog box:
    - First, type a value of *50* in the **X** box.
    - Next, type a value of *50* in the **Y** box.
    - Finally, type a value of *-14.499* in the **Z** box.
9. You are now ready to resize the cow triangle model. In the **Set Model Size** area, type a value of *89.999* in the **X Size** box.
10. Click on the **Apply** button to resize and position the cow triangle model.
11. Click on the **Paste** button to add the cow triangle model to the relief in the ArtCAM model. The relief should appear in the **3D View** window something like the image shown below:



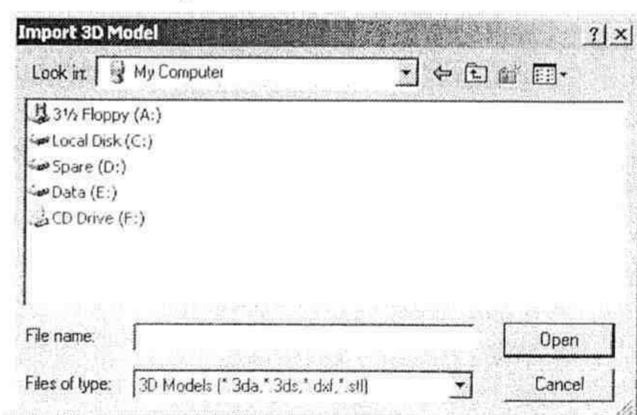
Importing a 3D Model File for Unwrapping

You can import wrapped triangle models saved as 3D Assembly (***.3da**), 3D Studio (***.3ds**), Drawing Interchange Format (***.dxf**) and STL (***.stl**) files into an open ArtCAM model.

The imported model can then be unwrapped along a cylindrical axis into a flat relief within an ArtCAM model. This command is intended for unwrapping ring designs created in other CAD systems into flat reliefs for machining within ArtCAM Pro.

To import a triangle model that you want to unwrap:

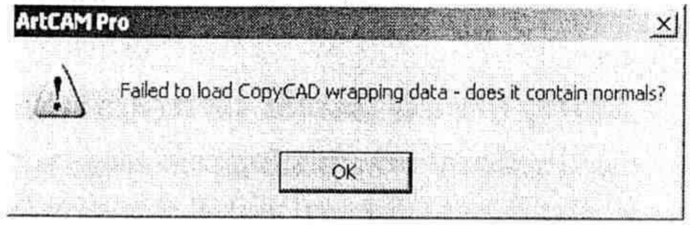
1. From the Main menu bar, click on the **File > Import > 3D Model For Unwrapping** option to display the **Import 3D Model** dialog box:

2. Click on the **Look In** list box and find the file that you want to import.
3. Once you have found the file, click to select the file listed in the main window of the **Import 3D Model** dialog box.

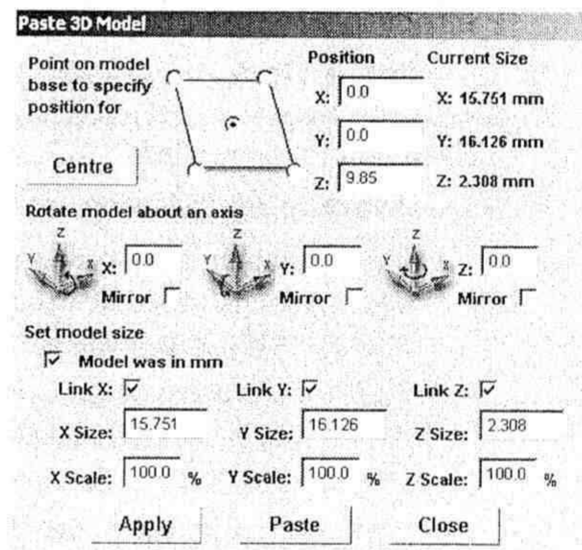
4. Click on the **Open** button to import the file into the open ArtCAM model.

   If you select a file that does not contain normals, the following message box appears:



   Click on the **OK** button to close the message box.

   If the file contains normals, the **3D Model Unwrapping** page is displayed in the **Assistant** window and the **Paste 3D Model** dialog box appears:



Note: Normals show the direction from which the points in the 3D model were taken and are perpendicular to the model surface.

   The current size of the model is shown and the original co-ordinate system used when the triangle model was constructed is kept.

5. You can now use the **Paste 3D Model** dialog box and/or the **3D Model Unwrapping** page to set the position, size and origin of the unwrapped model and exactly how it is unwrapped:

- For details on using the **Paste 3D Model** dialog box, see "Using the Paste 3D Model Dialog Box" on page 70.
- For details on using the **3D Model Unwrapping** page, see "Using the 3D Model Unwrapping page" on page 76.

Using the 3D Model Unwrapping page

The **3D Model Unwrapping** page allows you to unwrap an imported triangle model along a cylindrical axis into a flat relief within an ArtCAM model.

To unwrap a triangle model along a cylindrical axis:

1. In the **3D Model Origin** area, type a value in the **X**, **Y** and **Z** boxes to set the position of the triangle model's origin within the ArtCAM model.
2. In the **Rotate Model** area, use the +90° and -90° buttons to rotate the triangle model along any of the three axes in the ArtCAM Pro model in 90° increments.
3. In the **Unwrapping Cylinder** area:
   - Type a value in the **Diameter** box to set the diameter of the cylindrical axis along which the triangle model is unwrapped.
   - Click on one of the **Axes** radio buttons to select the axis along which the centreline in the triangle model is aligned.
   - Type a value in the **Border Width** box to set the width of the border, which is added above and below the triangle model.

     If you want to add a border later or in a different way, see "Adding a Border to a Model" on page 85 for details.
4. Click on the **Create New Model** button.
5. Click on the **Close** button to return to the **Assistant**'s Home page.

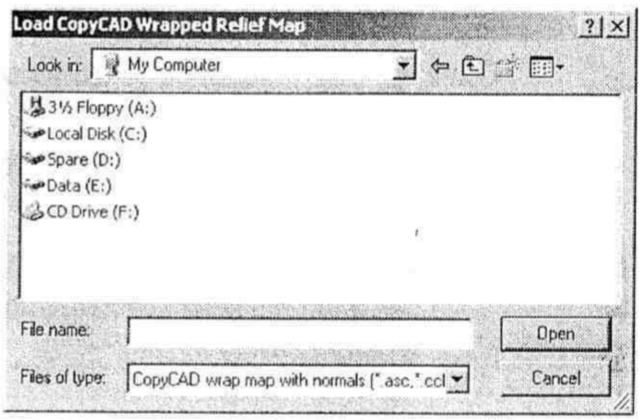
Importing CopyCAD Relief Data

You can import a wrapped relief map with normals saved as American Standard Code for Information Interchange (***.asc**) and CopyCAD Binary (***.ccb**) format files into an open model. The imported relief data is attached to the base of a relief in the open ArtCAM model.

Note: Normals show the direction from which the points in the 3D model were taken and are perpendicular to the model surface. The size of the wrapped relief map with normals must match that of the relief in the ArtCAM model.

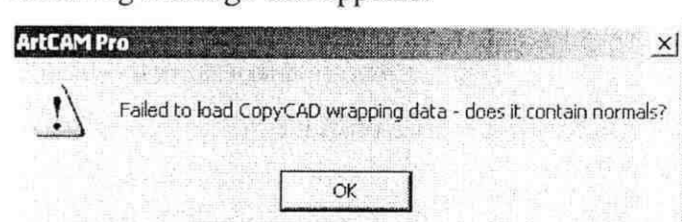
To import relief data:

1. From the Main menu bar, click on the **File > Import > Load CopyCAD Wrapping Data** option to display the **Load CopyCAD Wrapped Relief Map** dialog box:



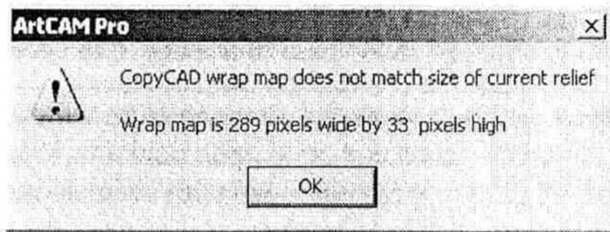
2. Click on the **Look In** list box and find the file that you want to import.
3. Once you have found the file, click on the file name listed in the main window of the **Load CopyCAD Wrapped Relief Map** dialog box.
4. Click on the **Open** button to import the file into the ArtCAM model.

   If you select a file that does not contain normals, the following message box appears:



Click on the **OK** button to close the message box.

If the size of the imported wrapped relief map with normals does not match that of the relief in the ArtCAM model, the following message box appears:



Click on the **OK** button to close the message box.

Correcting an Action

There are two buttons you can use to correct your mistakes when working in ArtCAM Pro:

- Click the **Undo** button  to cancel your last action.
- Click the **Redo** button  to repeat the last action that you had made before clicking on the **Undo** button.

Note: You can also **Undo** or **Redo** an action by clicking on **File** in the Main menu bar, and then selecting the relative option listed. For example, *Undo Draw Ellipse*.

Mirroring a Model

You can mirror an open model both vertically and horizontally.

To mirror a model horizontally:

- Click on the **Mirror Model Horizontally** button  in the **Model** area of the **Assistant**'s Home page.

You can also mirror a model horizontally in the following way:

- From the Main menu bar, click on the **Model** option to display the **Model** menu, then click on the **Mirror > Horizontal** option.

To mirror a model vertically:

- Click on the **Mirror Model Vertically** button  in the **Model** area of the **Assistant**'s Home page.

You can also mirror a model horizontally in the following way:

- From the Main menu bar, click on the **Model** option to display the **Model** menu, then click on the **Mirror > Vertical** option.

Rotating a Model

You can rotate an open model both clockwise and anti-clockwise.

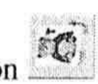
To rotate a model clockwise:

- Click on the **Rotate Model Clockwise** button  in the **Model** area of the **Assistant**'s Home page to turn the model through 90° in a clockwise direction.

You can also rotate a model clockwise in the following way:

- From the Main menu bar, click on the **Model** option to display the **Model** menu, then click on the **Rotate 90 Degrees > Clockwise** option.

To rotate a model anti-clockwise:

- Click on the **Rotate Model AntiClockwise** button  in the **Model** area of the **Assistant**'s Home page to turn the model through 90° in an anti-clockwise direction.

You can also rotate a model clockwise in the following way:

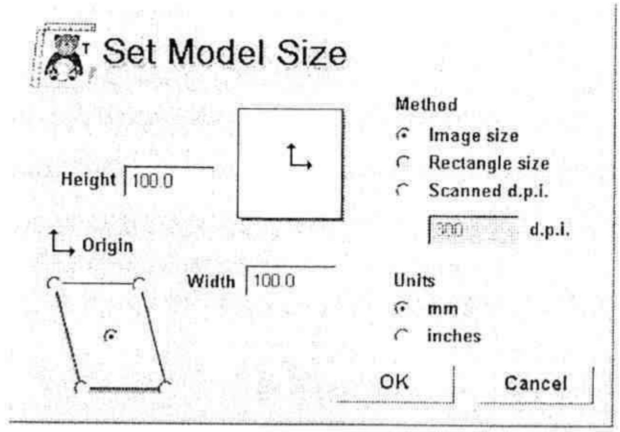
- From the Main menu bar, click on the **Model** option to display the **Model** menu, then click on the **Rotate 90 Degrees > AntiClockwise** option.

Editing the Model Dimensions

You can edit the dimensions that you had originally defined for an ArtCAM model when creating it.

To edit the model dimensions:

1. Click on the **Set Model Size** button  in the **Model** area of the **Assistant**'s Home page to display the **Set Model Size** dialog box:



2. Click on the **Method** radio button that you want to use to set the new size of the model. If you have selected the **Image Size** option:

    - Type in the **Height** and **Width** values according to the physical size of the model you want to create.

Warning: If you have not created a selection rectangle before clicking on the **Set Model Size** button, the **Rectangle Size** option is greyed out. For details, see "Using the Selection Rectangle" on page 95.

If you have selected the **Rectangle Size** option:

- The dimensions of the selection rectangle that you have drawn in the model are displayed in the **Height** and **Width** boxes.

  If you want to change these dimensions, type in the new **Height** and **Width** values according to the physical size of the model you want to create.

If you have selected the **Scanned d.p.i.** option:

- Type the resolution you want for the model in the **d.p.i.** box.

Note: Reducing the dots per inch increases the size of the model.

3. Click on the centre or any of the four corner radio buttons in the **Origin** area to define the X-axis zero and Y-axis zero origin.

4. Make sure that the **Units** option is set according to those you are working in (millimetres or inches).

5. Click on the **OK** button to set the size of the model according to the dimensions shown.

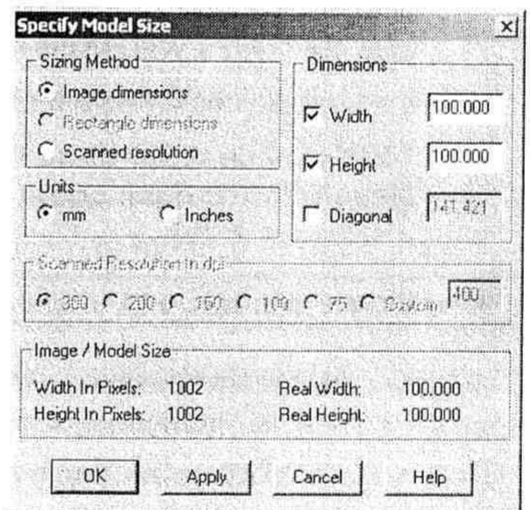
Editing Asymmetrical Dimensions in a Model

Warning: Editing the asymmetrical dimensions of a model only affects the size of a relief within it.

You can edit any asymmetrical dimensions that you find in an open model.

To edit a model's asymmetrical dimensions:

1. From the Main menu bar, click on the **Model > Set Size Asymmetric** option to display the **Specify Model Size** dialog box:



2. Click on the **Sizing Method** radio button that you want to use:

   - **Image dimensions** – This option allows you to set the size of a model using specific measurements.
   - **Rectangle dimensions** – This option allows you to set the size of a model to an area defined by a **Selection Rectangle**. For details, see "Using the Selection Rectangle" on page 95.
   - **Scanned resolution** – This option allows you to set the size of a model by defining the size of a pixel within it.

3. If you have selected the **Image dimensions** or the **Rectangle dimensions** option:

    - To define both the **Width** (X) and the **Height** (Y) as a specific size, make sure that both of these options is selected in the **Dimensions** area, then type a value in each of the boxes.
    - To preserve the ratio between the **Width** (X) and the **Height** (Y) of the model, make sure that only one of these options is selected in the **Dimensions** area, then type a value in the adjacent box.

      All other boxes in the **Dimensions** area are automatically greyed out when only one of the available options is selected .
    - To define the distance between the origin and the top right corner of the model, make sure that only the **Diagonal** option is selected in the **Dimensions** area, then type a value in the adjacent box.

    If you have selected the **Scanned resolution** option:

    - Click on of the radio buttons in the **Scanned Resolution in d.p.i.** area to define the size of a pixel in the model.

Note: Reducing the dots per inch increases the size of the model.

4. Make sure that the **Units** option is set according to those you are working in (millimetres or inches).
5. Click on the **Apply** button to preview the new model size.

    The **Real Height** and **Real Width** values in the **Image/Model Size** area are updated according to your new settings.
6. Click on the **OK** button to set the size of the model.

Setting the Position of a Model

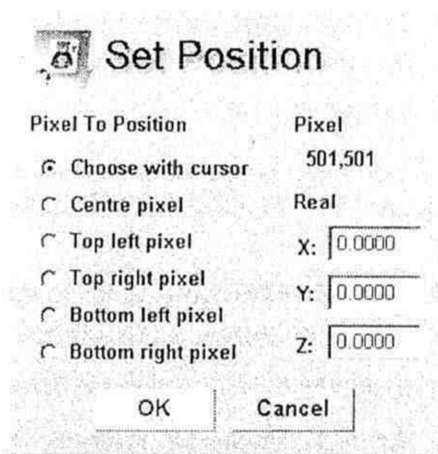
You can define the position of the X-axis zero and Y-axis zero origin as one of five pre-defined locations using the **Set Model Size** dialog box. For details, see "Editing the Model Dimensions" on page 79.

To define the position of the X-axis zero and Y-axis zero origin as somewhere other than the five available locations in the **Set Model**

Size dialog box, you can use the **Set Position** dialog box. You can also define the Z-axis zero origin using the **Set Position** dialog box.

To set the position of a model:

1. Click on the **Set Model Position** button in the **Model** area of the **Assistant**'s Home page to display the **Set Position** dialog box:



2. Click on one of the radio buttons in the **Pixel To Position** area to select which pixel in the model is to be the origin:

   - **Choose with cursor** – If you select this option, move the cursor over the pixel in the model that you want to define as the origin and then click.

     The co-ordinates of the pixel are displayed in the **Pixel** area.

   - **Centre pixel** – Select this option to define the origin as the centre of the model.
   - **Top left pixel** – Select this option to define the origin as the top left corner of the model.
   - **Top right pixel** – Select this option to define the origin as the top right corner of the model.
   - **Bottom left pixel** – Select this option to define the origin as the bottom left corner of the model.
   - **Bottom right pixel** – Select this option to define the origin as the bottom right corner of the model.

If you want to define the exact location of the origin, type the appropriate X, Y and Z values in the boxes in the **Real** area.

3. Click on the **OK** button to close the **Set Position** dialog box and to define the new origin in the model.

Clearing a Model

You can delete all bitmap data within an open model and restore the background to the current Primary Colour.

To clear an open model:

1. Make sure that the colour you want to use for the background of the model is selected as the Primary Colour.
2. Click on the **Clear** button  in the **Model** toolbar to restore the background to the current Primary Colour.

You can also clear an open model in the following way:

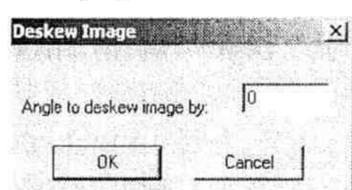
1. From the Main menu bar, click on the **Model** option to display the **Model** menu, and then on the **Clear** option.

Deskewing a Model

You can adjust an open model to compensate for justification errors that may have occurred when scanning the bitmap images used within it. This operation is called deskewing.

To deskew a model:

1. From the Main menu bar, click on the **Model** option to display the **Model** menu, and then on the **Deskew** option to display the **Deskew Image** dialog box:



2. Define the deskewing angle in the **Angle to deskew image by** box.
3. Click on the **OK** button to deskew the model and close the **Deskew Image** dialog box.

Creating a Greyscale Image from a Relief

A greyscale is an image in which the only colours are shades of grey. You can create a greyscale image from any relief in an open model.

It is recommended that you use this option to view a swept profile shape more clearly, particularly when you want to position vector text or 3D clipart relative to it. For details, see "Creating a Swept Profile Shape" in the Working with Reliefs chapter.

This option is particularly useful for identifying areas of a model to which you want to add texture, as the entire relief is painted in a single colour. For details, see "Adding Texture to a Relief" in the Working with Reliefs chapter.

To create a greyscale image from a relief:

1. Click on the **Grayscale From Relief** button  in the **Model** area of the **Assistant**'s Home page.

You can also create a greyscale image from a relief in either of the following ways:

- Press the **F10** key on your keyboard.
- From the Main menu bar, click on the **Model** option to display the **Model** menu, then click on the **Grayscale From Relief** option.

Note: If you want to restore the full colour image of a relief, click on the **Edit** option in the Main menu bar to display the **Edit** menu, and then on the **Undo Create Grayscale Image** option.

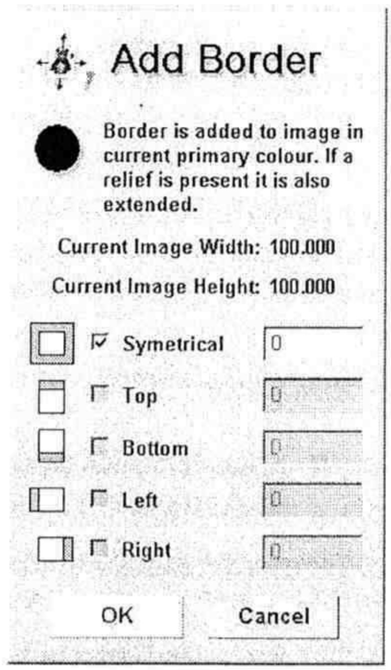
Adding a Border to a Model

You can create a border around an open model in the current Primary Colour. Any existing relief in the model is extended by the width of the border you define.

To create a border around a model:

1. Make sure that the colour you want to create the border in is selected as the Primary Colour. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.

2. Click on the **Add Border** button in the **Model** area of the **Assistant**'s Home page to display the **Add Border** dialog box:



The current Primary Colour is shown, as is the width and height of the model. The **Symmetrical** border option is selected by default.

3. If you want to create a symmetrical border, type a value in the adjacent box to define the width of all four sides simultaneously.

   If you want to define the width of the border on each of the four sides independently:

   - First, click on the **Symmetrical** option to deselect it.
   - Next, click on each of the side options (**Top**, **Bottom**, **Left**, **Right**) that you want to add a border to in order to select them, and then type a value in the box adjacent to each of the selected side options.

     If a side option is not selected, it is greyed out.

4. Click on the **OK** button to close the **Add Border** dialog box and create the border around the model.

Using a Spot Filter on a Bitmap Image

You can remove any isolated pixels from within a scanned bitmap image using a spot filter.

A spot filter performs better with black and white or greyscale images. It may be necessary to apply a spot filter several times on the same image to remove all unwanted pixels (spots) in the bitmap image.

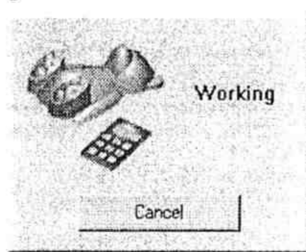
To use a spot filter:

1. From the Main menu bar, click on the **Model** option to display the Model menu. Move over the **Spot Filter** option to display the filters available.
2. Click on the **Spot Filter** option that you want to apply to the bitmap image:
   - **Photo Image** – This option is the maximum filter that is most likely to remove unwanted spots in the image.

Warning: The **Photo Image** filter may also remove areas of the bitmap image that you want to keep.

   - **General** – This option is the medium filter.
   - **Line Drawing** – This option is the minimum filter, which removes only a few spots in the image. You may have to apply this filter several times before removing any unwanted spots.

   The **Working** dialog box appears during the filtering process:



Adjusting Light and Material Settings

You can alter the appearance of a model in the **3D View** window by:

- Using up to four different lights at once to illuminate selected areas of the model.

- Changing the colour of the relief to match that of the actual block of material used in the manufacturing process.
- Changing the background colour or filling it with a bitmap image.
- Projecting an image or reflection map onto the relief.

This allows you to create an authentic impression of the machined model.

Light and Shading Settings

You can illuminate selected areas of the relief shown in the **3D View** window using up to four types of light in a number of combinations.

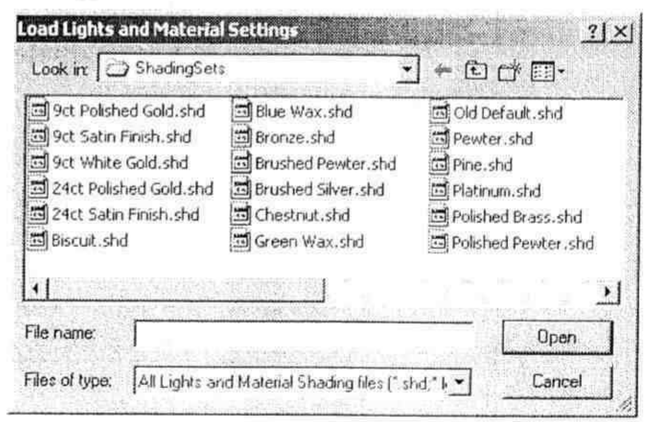
To change the light or shading settings:

1. Click on the **Lights and Material** button in the **Model** area of the **Assistant**'s Home page to display the **Lights and Material** page.

Note: You can also display the **Lights and Material** page if you click on the **Model** option in the Main menu bar, and then on the **Lights and Material Setup** option.

---

2. If you want to load a light (***.lgt**) or shading (***.shd**) setting file that you have previously saved, click on the **Load** button to open the **Load Lights and Material Settings** dialog box:



Click on the **Look in** list box and select the directory where the file that you want to load is stored, and then click on the setting file that you want to load to select it. The file name appears in the **File name** box

Click on the **Open** button to load the settings into the **Lights and Material** page.

3. If you want to use any of the preset shading settings in ArtCAM Pro, click on the list box in the **Shading Setup** area, and then on the shading setting that you want to use.
4. To adjust the amount of background lighting, click and drag on the **Ambient** slider.

Note: To hide any of the **Shading Setup** settings, or any of the other groups of settings on the page, click on the arrow.

---

5. In the **Lights** area, there are four light options that you can use to expose light to the relief. To expose light to a relief, click on the **Enabled** option beside the light options that you want to select .

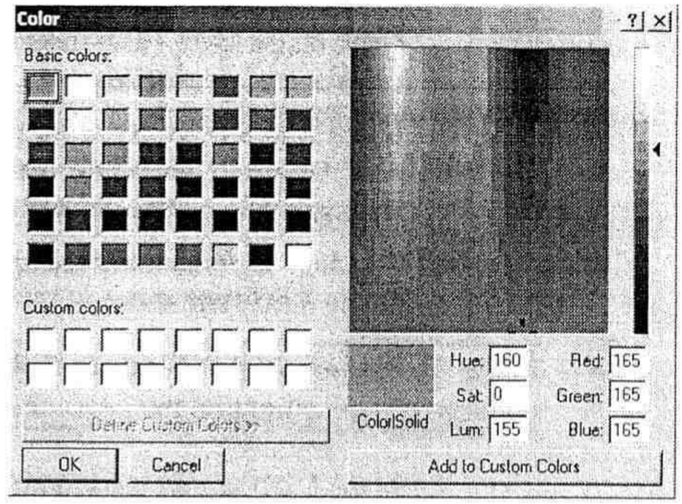
   The light icon beside the **Enabled** box appears as on when selected. For example, the point light icon changes from to .
6. To edit the settings for any of the lights you have enabled, click on the arrow to display them.

   If you want to change the type of light, click on the **Type** list box, and then on the light option that you want to use:

   - **Distant** – This option allows you to add the effect of a light at a distance from the relief. The icon appears if this type of light is selected.
   - **Point** – This option allows you to add the effect of a light bulb held over the relief. The icon appears if this type of light is selected.
   - **Spot** – This option allows you to add the effect of a spot light held over the relief. The icon appears if this type of light is selected.

   Next, set the colour of the light:

   - First, click on the **Colour Palette** button or the colour shown beside it to display the **Color** dialog box:



- Next, select the colour that you want to use for the light. For details on how to select Basic and Custom colours, see "Assigning a Colour to a Layer" on page 46.
- Finally, click on the **OK** button to close the **Color** dialog box.

You can now adjust the contrast of the light by clicking and dragging on the **Intensity** slider.

If you have selected a **Spot Light**, you can adjust its focus by clicking and dragging on the **Edge** slider. You can also adjust its scope by typing a value in the **Width** box.

If you have selected a **Point Light** or a **Spot Light**, you can adjust its position relative to the relief:

- Click on the bulb icon on the **Lights and Material** page and drag it into position over the relief in the **3D View** window.

  The value displayed in the first box is shown as a percentage of the width (X) of the model.

  The value displayed in the second box is shown as a percentage of the height (Y) of the model.

- The value displayed in the third box is shown as a percentage of half the height of the longest side in the model. This value is then used for the Z height of the light.

If you have selected a **Distant Light** or a **Spot Light**, you can set the direction of the light:

- To align the direction of the light to one of seventeen points on the sphere, click on the **Use Snap points on sphere** option to select it.

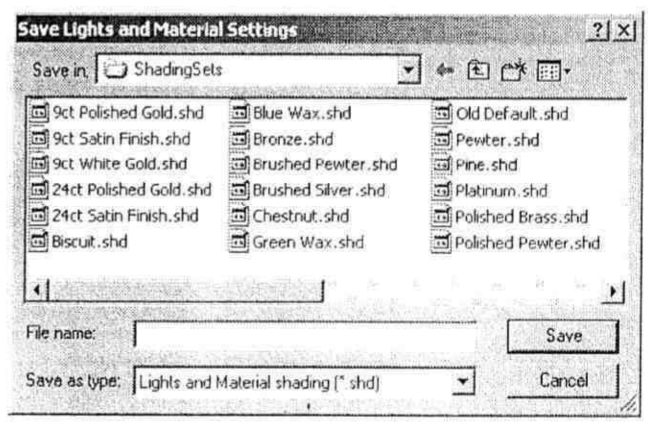
  Click and drag on the sphere and release the mouse button when the cursor has snapped to the point that you want to use as the light direction.

- If you want to align the direction of the light with your field of vision when focused directly at your computer monitor, click on the **Fix light along viewing direction** option to select it.

Note: This option is only available if you have enabled a **Distant Light**.

To hide the settings for a light you have enabled, click on the arrow.

7. Click on the **Apply** button to apply the changes you have made to the light and/or shading settings.
8. You can now save or reset your light and shading settings:
   - If you want to save these light and shading settings, click on the **Save** button to display the **Save Lights and Material Settings** dialog box:



     Click on the **Save in** list box and select the directory where you want to save the file. Type a name for the file in the **File Name** box, and then click on the **Save** button.

   - If you are not happy with these light and shading settings, you can either continue to adjust them or click on the **Reset** button to restore the default settings.

9. If you want to set your light and shading settings as default, click on the **Set As Default** button.
10. Click on the **Done** button to close the **Lights and Material** page and return to the **Assistant**'s Home page.

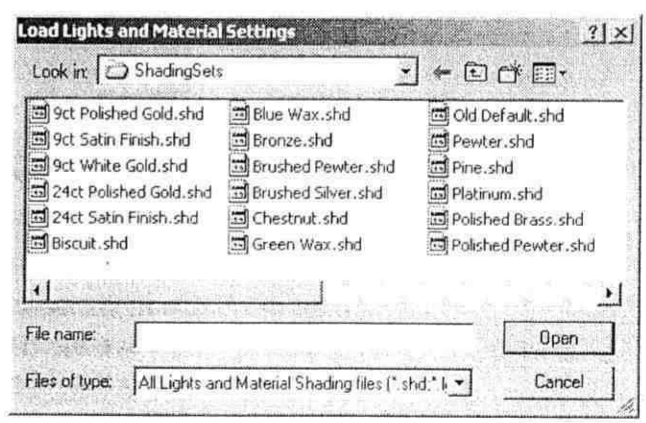
Material and Background Settings

You can change the colour used to shade the relief in the **3D View** window, the brightness of the lit areas on the relief surface, and the reflectivity of the relief surface. You can also select a background colour or image for the **3D View** window, and project a reflection map onto the relief surface.

You can use the sphere in the **Material** area of the page to preview the effects that the settings you create have on the relief shown in the **3D View** window:

To change the material settings:

1. Click on the **Lights and Material** button  in the **Model** area of the **Assistant**'s Home page to display the **Lights and Material** page.
2. If you want to load a material shading (***.mtl**) setting file that you have previously saved, click on the **Load** button to open the **Load Lights and Material Settings** dialog box:



Click on the **Look in** list box and select the directory where the file that you want to load is stored, and then click on the setting file that you want to load to select it. The file name appears in the **File name** box

Click on the **Open** button to load the settings into the **Lights and Material** page.

3. To change the colour of the material shown in the **3D View** window, click on the list box in the **Colour** area, and then on the option that you want to use:

   - **Selected Colour** - This option allows you to select the colour of the relief using the **Color** dialog box. For further information, see "Assigning a Colour to a Layer" on page 46. You can then edit the chosen colour using the sliders in the **Colour** area.

   - **2D View** - This option allows you to project the bitmap colours or image, as shown in the **2D View** window, onto the relief.

   If you want to use any of ArtCAM Pro's preset materials, click on the list box in the **Colour** area, and then on the material that you want to use. When using any of these preset settings:

   - If you want to tile the material across the relief, click to select the **Tile** option and then define its size using the **Size X** and **Size Y** boxes.

   - If you want to replace areas of the relief under the current Primary Colour with the selected material, click to select the **Only Show Image Under Primary Colour** option.

Note: If you want to use your own material, copy the TIF image file (***.tif**) representing the material to the ArtCAM Pro\Rendering\Materials folder and then restart ArtCAM Pro. Your image will then be listed in the **Colour** list box when the **Lights and Material** page is next displayed.

---

4. To adjust the reflectivity of the material surface, click and drag on the **Shininess** slider.

5. To adjust the brightness of the lit areas in the model, click and drag on the **Highlight Intensity** slider.

6. If you want to project a reflection map onto the relief surface, click on the list box in the **Reflection Map** area, and then on the reflection map that you want to use.

7. You can now fill the background of the **3D View** window with an image or a colour. To fill the background with a colour:

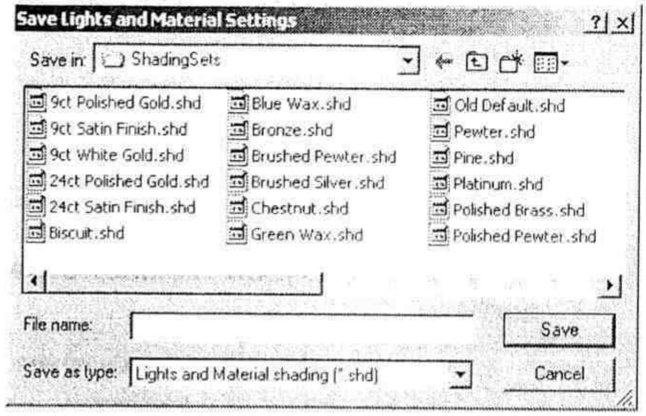
    - In the **Background** area, click on the list box and then on the **Selected Colour** option. This is the default option when the **Lights and Material** page is displayed.
    - Next, click on the **Colour Palette** button to display the **Color** dialog box.
    - Select the colour that you want to use as the background. For details on how to select Basic and Custom colours, see "Assigning a Colour to a Layer" on page 46.
    - Finally, click on the **OK** button to close the **Color** dialog box.

    To fill the background with a bitmap image:

    - In the **Background** area, click on the list box and then on the image that you want to use.

Note: If you want to use your own bitmap image as a background in the **3D View** window, copy the bitmap file (***.bmp**) to the ArtCAM Pro\Backgrounds folder and then restart ArtCAM Pro. Your image will then be listed in the **Background** list box when the **Lights and Material** page is next displayed.

8. Click on the **Apply** button to apply the material settings that you have chosen.
9. You can now save or reset your material settings:
    - If you want to save these material settings, click on the **Save** button to display the **Save Lights and Material Settings** dialog box:



Click on the **Save in** list box and select the directory where you want to save the file. Type a name for the file in the **File Name** box, and then click on the **Save** button.

- If you are not happy with these light and shading settings, you can either continue to adjust them or click on the **Reset** button to restore the default settings.

10. If you want to set your material settings as default, click on the **Set As Default** button.
11. Click on the **Done** button to close the **Lights and Material** page and return to the **Assistant**'s Home page.

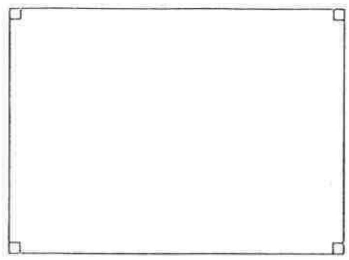
Using the Selection Rectangle

You can use a selection rectangle to define an area in an open model that you want to:

To create a selection rectangle:

1. Click on the **Selection Rectangle** button in the **File** toolbar.
2. Move the cursor over a corner of the area in the model that you want to select, then click and drag diagonally.

   The selection rectangle appears in the **2D view** window as follows:



If you want to move the selection rectangle to another area in the model:

- Move the cursor inside of the selection rectangle, then click and drag it into position.

If you want to edit the size of the selection rectangle:

- Move the cursor over a corner or side of the selection rectangle, then click and drag into position.

Using the ArtCAM Pro Notepad

You can add comments concerning an open model to the ArtCAM Pro notepad.

To use the ArtCAM Pro notepad:

1. Click on the **Display/Hide Notes** button  in the **File** toolbar to display the **ArtCAM Notes** window:

2. Click on the **ArtCAM Notes** window and then type the comments that you want to make about the open model in it using your keyboard.

Tip: If you type **..** and then press the **Enter** key on your keyboard before typing your comments, the **ArtCAM Notes** window is automatically displayed when you open the saved model again.

3. Click on the **Display/Hide Notes** button to close the **ArtCAM Notes** window.

When the open model is saved, any comments that have been typed into the **ArtCAM Notes** window are also saved.

If you do not save the model, any comments that you have made corresponding to it are automatically deleted from the **ArtCAM Notes** window.

Working with Bitmaps

Drawing using Bitmaps

When you have created or opened an ArtCAM model, you can use the drawing and painting tools in the **Bitmap Editing** area of the **Assistant**'s Home page and on the **Bitmap** toolbar to create your own bitmap images.

The bitmap images you create can be used to create the three-dimensional shapes that make up a relief, to edit the relief in a variety of ways, and ultimately machine a 3D model. For more information, see the Working with Reliefs chapter.

Bitmap Drawing Tools

A bitmap image, or raster graphic, is made up of pixels in a grid. Pixels are picture elements; tiny dots of individual colour that make up the images that you see on your screen. Bitmap images are resolution dependent. Resolution refers to the number of pixels in an image and is usually stated as d.p.i. (dots per inch). Since bitmap images are resolution dependent, it is difficult to scale them to a larger size without sacrificing a degree of quality.

In ArtCAM Pro, you can edit imported bitmap images or create your own. The group of bitmap tools is found on the **Bitmap** toolbar:

You can also find a selection of the tools featured in the **Bitmap** toolbar in the **Bitmap Editing** and **Vector – Bitmap** areas of the **Assistant**'s Home page.

Creating a Model from a Bitmap

You can create a model by importing any the following types of image files as bitmaps into ArtCAM Pro: ***.bmp**, ***.tif**, ***.gif** and ***.jpg**. A relief is automatically calculated when the image is imported. For details, see "Creating a Model from an Image" in the Working with Models chapter.

Importing a Bitmap into a Model

In ArtCAM Pro, you can import images saved as files of type ***.bmp**, ***.tif**, ***.pcx**, ***.gif** or ***.jpg** into a model. For details, see "Importing Images" in the Working with Models chapter.

Setting a Bitmap's Size and Origin

You can change the size of a bitmap image imported into ArtCAM Pro. In resizing the bitmap image, you also set the size of the vector objects that can be created from it.

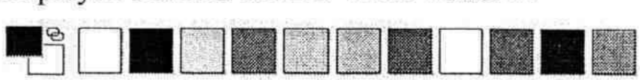
There are three ways to set the size of a bitmap image imported into ArtCAM Pro, all of which are done via the **Set Model Size** dialog box.

Note: Changing the size of the image does not alter the resolution of the original bitmap you imported into ArtCAM Pro. The image retains the same number of pixels as the original.

You can also set the origin of an imported bitmap image to any one of five preset positions in ArtCAM Pro. Each of these positions determines where the vector objects you create from the bitmap image are positioned in the model. For details on how to use the **Set Model Size** dialog box, see "Editing the Model Dimensions" in the Working with Models chapter.

Working with Colours

When you create or open an existing model, a default Colour Palette is displayed beneath the **2D View** window:



When you have imported a bitmap image, all of the colours within the bitmap image are displayed in the Colour Palette below the **2D View** window.

ArtCAM Pro allows you to manipulate the colours within a bitmap image. You can:

- Select the Primary and Secondary colours from the Colour Palette. For details, see "Selecting the Primary and Secondary Colours" on page 101.
- Reduce the number of colours in a bitmap image. For details, see "Reducing Colours" on page 102.
- Link colours within a bitmap image. For details, see "Colour Linking" on page 104.
- Merge colours within a bitmap image. For details, see "Colour Merging" on page 103.
- Mark the edge of a bitmap image. For details, see "Edge Marking" on page 105.
- Thicken colours within a bitmap image. For details, see "Colour Thickening" on page 106.
- Thin colours within a bitmap image. For details, see "Colour Thinning" on page 105.
- Add colours to the Colour Palette. For details, see "Adding Colours" on page 107.
- Save a custom Colour Palette. For details, see "Saving a Custom Colour Palette" on page 109.
- Load a custom Colour Palette. For details, see "Loading a Custom Colour Palette" on page 109.

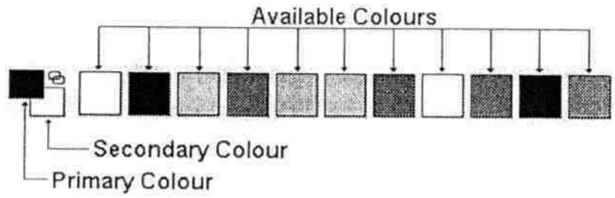
Selecting the Primary and Secondary Colours

The Primary Colour defines the shape of the vector objects or the three-dimensional shapes you can create from a bitmap image. ArtCAM Pro creates vector objects around the outline of all areas in the Primary Colour, along with those colours linked to it.

The Primary Colour controls the colour of the **Paint** and **Draw** tools. You can use the Secondary Colour only when painting, drawing or editing a bitmap image.

The Primary and Seconday colours are also used to define the areas in a relief that you do or do not want to edit.

You can select the Primary and Secondary colours from the Colour Palette beneath the **2D View** window:



You can also select the Primary and Secondary colours from the bitmap image itself using the **Pick Colour** tool.

To select the Primary Colour, you can either:

- Click on the colour in the Colour Palette you want to use.
- Click on the **Pick Colour** button in the **Bitmap** toolbar, move the dropper over the colour in the bitmap image that you want to select, and then click.

To select the Secondary Colour, you can either:

- Right-click on the colour in the Colour Palette you want to use.
- Click on the **Pick Colour** button in the **Bitmap** toolbar, move the dropper over the colour in the bitmap image that you want to select, hold the **Shift** key down on your keyboard, and then click.

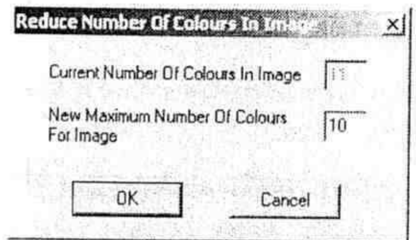
Reducing Colours

You can reduce the number of colours in a bitmap image. ArtCAM Pro takes colours of a similar shade and produces an averaged colour. This averaged colour is displayed in the Colour Palette in place of the original colours.

Note: 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 image.

To reduce the number of colours in a bitmap image:

1. Click on the **Reduce Colours** button in the **Model** area of the **Assistant**'s Home page to display the **Reduce Number Of Colours In Image** dialog box.



2. Type a value in the **New Maximum Number Of Colours For Image** box.

   The default value is one less than the value in the greyed-out **Current Number Of Colours In Image** box.

3. Click on the **OK** button to close the **Reduce Number Of Colours In Image** dialog box and reduce the number of colours in the bitmap image to that specified in the last step.

Warning: Reducing the number of colours in a bitmap image resets all colour links and attributes. If you want to keep any new or linked colours you have created, do not click on the **OK** button.

Colour Merging

You can merge all the pixels in the Secondary Colour in a bitmap image with the Primary Colour. The Secondary Colour is then removed from the Colour Palette beneath the **2D View** window.

To do so:

1. Click on the colour within the Colour Palette you want to select as the Primary Colour.

2. Right-click on the colour within the Colour Palette you want to select as the Secondary Colour.

3. Click on the **Merge Colours** button in the **2D View** toolbar to merge the current Secondary Colour with current the Primary Colour. The Secondary Colour is removed from the Colour Palette altogether.

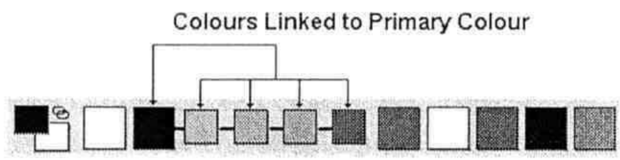
Colour Linking

You can both link and unlink colours within the Colour Palette beneath the **2D View** window. When a colour in the Colour Palette is linked to the Primary Colour, it is treated as the Primary Colour until it is unlinked from it.

There are three ways you can link a colour in the Colour Palette with the current Primary Colour:

- Click on the **Link/Unlink** button to link the Secondary Colour to the Primary Colour.
- Click on the **Link/Unlink** icon in the Colour Palette to link the Secondary Colour to the Primary Colour.
- Double right-click on the colour in the Colour Palette that you want to link to the Primary Colour.

You can see that colours are linked to the Primary Colour when they appear in the Colour Palette as follows:



When colours are linked to the Primary Colour, they are displayed in the Primary Colour in the image.

Linking All Colours

By clicking on the **Link All Colours** button, you can simultaneously link all colours currently in the Colour Palette, other than the Secondary Colour, to the Primary Colour.

Tip: Depending on how many colours there are you want to link, it may be quicker to use the **Link All Colours** method of linking and then use the **Link/Unlink** button to unlink the individual colours that you do not want.

Unlinking All Colours

By clicking on the **Unlink All Colours** button, you can unlink all colours currently linked in the Colour Palette.

Edge Marking

You can paint a line around the edge of a bitmap image. Marking the outline of a bitmap image in this way means that its shape is clearly defined when it is converted into vector objects or three-dimensional shapes.

To mark the edges:

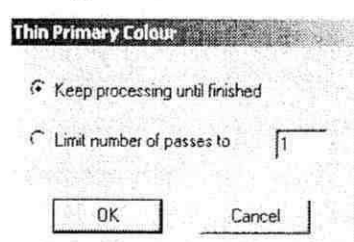
1. Click on the colour in the Colour Palette around which you want to paint a line. This colour is now set as the Primary Colour.
2. Right-click on the colour in the Colour Palette that you want to use for painting the outline. This colour is now set as the Secondary Colour.
3. Click on the **Mark Edge** option from the **Colour** menu. A line is painted in the Secondary Colour around all regions in the bitmap image made up of the Primary Colour.

Colour Thinning

Thinning reduces the width of a colour area, replacing edges with the Secondary Colour. If you select the full thinning process, this reduces the area to a single pixel skeleton. Rather than manually changing an area of colour pixel by pixel, you can automatically thin areas in the Primary Colour.

To thin areas in the Primary Colour:

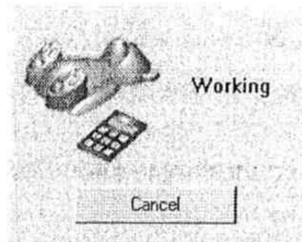
1. Click on the colour within the Colour Palette you want to select as the Primary Colour.
2. Right-click on the colour within the Colour Palette you want to select as the Primary Colour.
3. From the Main toolbar, click on the **Colour** menu followed by the **Thin...** option to display the **Thin Primary Colour** dialog box:



4. Click to select the thinning method you want to use:

- **Keep processing until finished** - This option continues to thin all regions in the Primary Colour until they are reduced to a width of one pixel.
- **Limit number of passes to** - This option sets a limit on the number of thinning passes according to the value you type in the box. Each pass converts one edge pixel in the Primary Colour to the Secondary Colour.

5. Click on the **OK** button to close the **Thin Primary Colour** dialog box and to thin all regions in the Primary Colour. The **Working** box is displayed while ArtCAM Pro completes the colour thinning process:



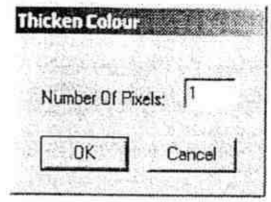
The region of removed pixels is marked in the current Secondary Colour. If you do not need to mark this region, ensure that the Secondary Colour is the same as the background colour in the bitmap image before colour thinning.

Colour Thickening

Thickening increases the width of a colour area, expanding the borders with the Secondary Colour. Rather than manually changing an area of colour pixel by pixel, you can automatically thicken areas in the Primary Colour.

To thicken areas in the Primary Colour:

1. Click on the colour within the Colour Palette you want to select as the Primary Colour.
2. Right-click on the colour within the Colour Palette you want to select as the Secondary Colour.
3. From the Main toolbar, click on the **Colour** menu followed by the **Thicken...** option to display the **Thicken Colour** dialog box:



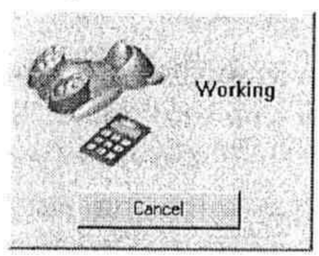
4. Define the brush diameter in the **Number of Pixels** box.

Warning: The brush diameter should be an odd number. Even numbers are automatically rounded up to an odd number.

---

5. Click on the **OK** button to close the **Thicken Colour** dialog box and to apply a round brush of the diameter you have already defined in the current Secondary Colour to each pixel in the current Primary Colour.

   The **Working** box is displayed while ArtCAM Pro completes the colour thickening process:

   If you want to cancel the colour thickening process at any time, click on the **Cancel** button.

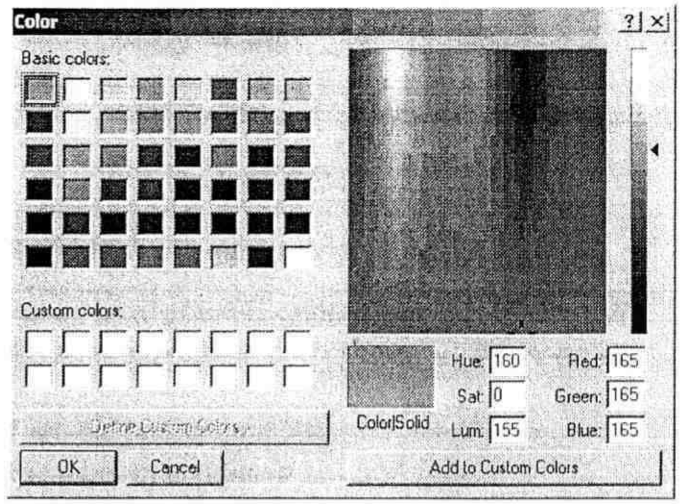
   The original pixels in the bitmap image remain in the Primary Colour, while the thickened regions appear in the Secondary Colour. If you do not need this distinction, ensure that both the Primary and Secondary Colours are identical before colour thickening.

Adding Colours

You can increase the range of colours within the Colour Palette beneath the **2D View** window. You can add Basic Colours and/or create your own Custom Colours for you to paint in.

To add to the Colour Palette:

1. Click on the **Add Colour** button in the **Bitmap Editing** area of the **Assistant**'s Home page to open the **Color** dialog box:



2. Select a colour that you want to add to the Colour palette. To select a Basic colour:

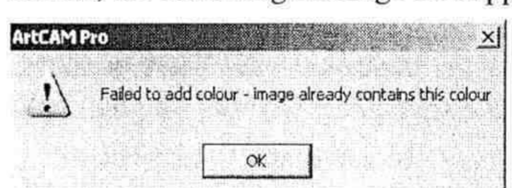
   - Click on a colour in the **Basic colors** chart.

   To select a Custom colour:

   - Click on an approximate colour in the Colour Matrix, and then click and drag the slider ◂ at the right of the dialog box to adjust the colour's attributes.
   - Type values in the **Hue**, **Sat** (Saturation) and **Lum** (Luminosity) or the **Red**, **Green** and **Blue** boxes to specify the colour. The colour appears in the **Color|Solid** area of the **Color** dialog box.

3. Click on the **Add To Custom Colors** button to add the colour you have defined to the Custom Colours palette. The colour you have selected appears in the **Custom colors** area of the **Color** dialog box.
4. Click on the **OK** button to close the **Color** dialog box.

   If you try to add a colour that is already in the Colour Palette, the following message box appears:



   If so, click on the **OK** button to close the message box and add a different colour.

The Basic and/or Custom Colours that you have selected appear in the Colour Palette beneath the **2D View** window.

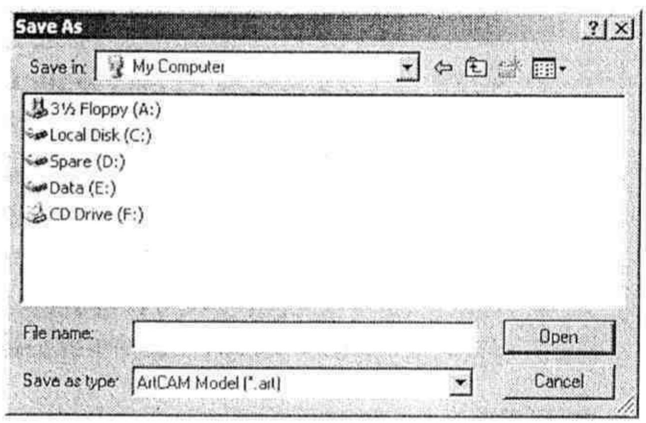
Saving a Custom Colour Palette

You can save any custom arrangement of colours you have made in the Colour Palette beneath the **2D View** window as an ArtCAM Palette file (***.pal**).

You can replace the current Colour Palette displayed beneath the **2D View** window with another custom Colour Palette by loading a previously saved ArtCAM Palette file.

To save the Colour Palette that you are currently using:

1. From the Main menu bar, click on the **Colour** menu, and then on the **Save** option to display the **Save As** dialog box:



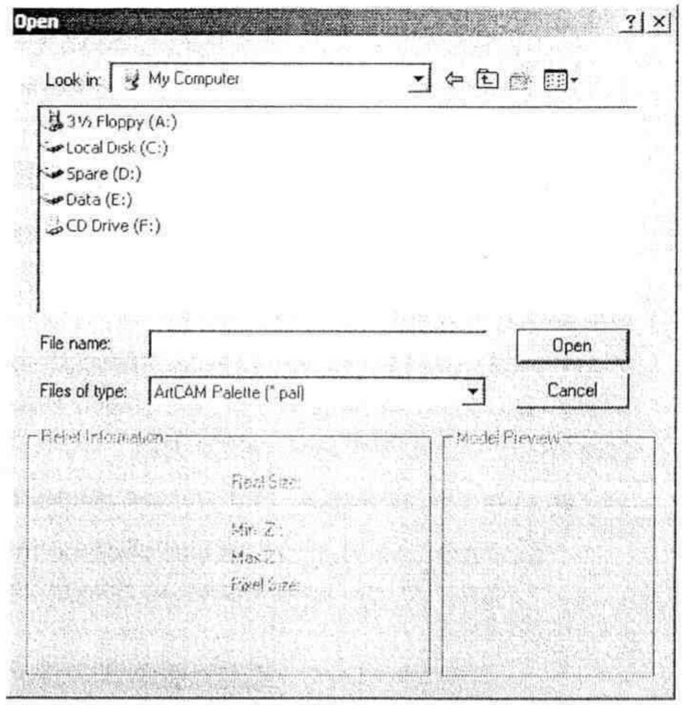
2. Click on the **Save In** list box and then on the directory in which you want to save the Colour Palette.
3. Type the file name that you want to use for the Colour Palette in the **File name** box.
4. Click on the **Save** button to save the Colour Palette.

Loading a Custom Colour Palette

You can replace the Colour Palette currently displayed beneath the **2D View** window with another custom Colour Palette by loading a previously saved ArtCAM Palette file (***.pal**).

To load an ArtCAM Palette file:

1. From the Main menu bar, click on the **Colour** menu, and then on the **Load** option to display the **Open** dialog box:



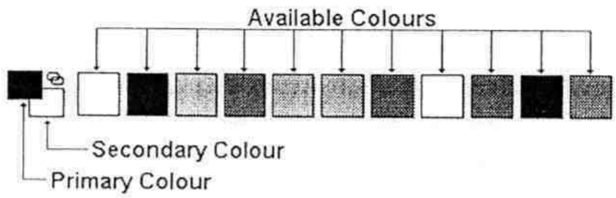
2. Click on the **Look In** list box in the **Open** dialog box and find the ArtCAM Palette file that you want to load.
3. Once you have found the file, click on the file name listed in the main window of the **Open** dialog box. Its name appears in the **File name** box.
4. Click on the **Open** button to display the custom Colour Palette beneath the **2D View** window.

Editing a Bitmap Image

You can change a bitmap image using the painting and drawing tools and the Colour Palette. The painting tools in ArtCAM Pro are a brush and a flood-fill tool. The drawing tools are a pencil and a bitmap-line tool.

Using the Paint Brush

The paint brush allows you to paint in the Primary and Secondary Colours, as shown on the left of the Colour Palette below the **2D View** window:

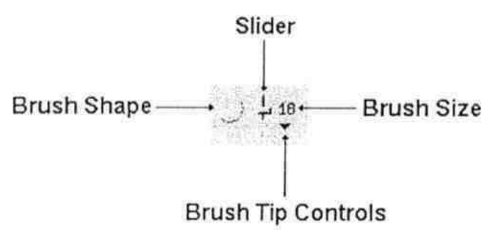


You can use the paintbrush to modify any bitmap image that you have created or imported into ArtCAM Pro:

1. Click on the **Paint** button in the **Bitmap Editing** area of the **Assistant**'s Home page to enter Paint mode.
2. Set the brush size and shape, as described in "Setting the Brush Size and Shape" on page 111.
3. Move the cursor over the area of the bitmap image you want to edit, and then click and drag to paint in the Primary Colour.
4. Click on the **Select Vectors** button in the **Vector Editing** area of the **Assistant**'s Home page to exit from Paint mode.

Setting the Brush Size and Shape

You can change the size and shape of the brush used for painting at any time with the **Brush Size** tool displayed in the **Bitmap** toolbar:



You can change between a square and a circle brush tip simply by clicking on the Brush Shape icon, as shown above.

You can increase the number of pixels that make up the Brush Size by clicking and dragging the slider upwards, or decrease them by clicking and dragging the slider downwards.

You can also change the size and shape of the brush used for painting at any time with the **Brush Diameter** slider and the brush shape icon displayed in the **Bitmap Editing** area of the **Assistant**'s Home page:

You can change between a square and a circle brush tip simply by clicking on the Brush Shape icon, as shown above.

You can increase the number of pixels that make up the Brush Size by clicking and dragging the slider upwards, or decrease them by clicking and dragging the slider downwards.

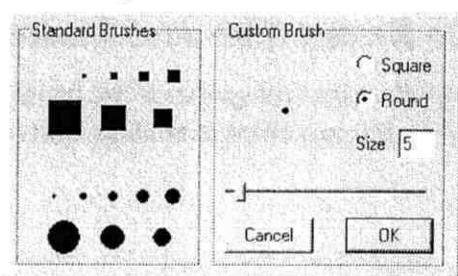
You can also change the size and shape of the brush used for painting at any time with the **Brush Diameter** slider and the brush shape icon displayed in the **Bitmap Editing** area of the **Assistant**'s Home page:

You can change between a square and a circle brush tip simply by clicking on the Brush Shape icon, as shown above.

You can increase the number of pixels that make up the Brush Size by clicking and dragging the slider to the right, or decrease them by clicking and dragging the slider to the left.

Alternatively, you can use the Brush Tip controls in the **Bitmap** toolbar to set all of the brush properties:

1. Click on the ▼ arrow of the **Brush Size** tool to open the Brush Tip controls:



2. Select or create the brush properties. If you want to select a Standard brush:

    - Click on any of the square or circle icons.

   If you want to create a Custom brush:

    - Click on either of the radio buttons to select the shape of the brush.
    - Type the number of pixels in the **Size** box, or drag the slider to set the size of the brush.

3. Click on the **OK** button to set the size and shape of the brush and close the Brush Tip controls. You are now ready to paint with your new brush.

Selective Painting

You can use the **Paint Selective** tool to paint with your brush in the current Primary Colour in all areas of a bitmap image made up of the current Secondary Colour.

To do so:

1. Click on the **Paint Selective** button in the **Bitmap Editing** area of the **Assistant**'s Home page.
2. Click on the colour within the Colour Palette you want to select as the Primary Colour.
3. Right-click on the colour within the Colour Palette you want to select as the Secondary Colour.
4. Move the cursor over the area you want to paint over, and then click and drag.

All areas of the bitmap image that are of the Secondary Colour are replaced with the Primary Colour. All other colours in the bitmap image remain unchanged.

Using the Draw Tool

The **Draw** tool allows you to draw a line of one pixel wide in the Primary and Secondary Colours, as shown on the left of the Colour Palette shown beneath the **2D View** window:

You can use the draw tool to create a new bitmap image or to modify a bitmap image that you have imported into ArtCAM Pro:

1. Click on the **Draw** button in the **Bitmap Editing** area of the **Assistant**'s Home page to enter Drawing mode.
2. Move the cursor over the area of the model that you want to draw in, and then click and drag to draw in the Primary Colour.
3. If you want to draw in the Secondary Colour, hold the **Shift** key down on your keyboard, and then click and drag.
4. Click on the **Select Vectors** button in the **Vector Editing** area of the **Assistant**'s Home page to exit from Draw mode.

Using the Bitmap Line Tool

The bitmap line tool allows you to draw a line in a bitmap image in the current Primary Colour.

To draw a bitmap line:

1. Click on the **Draw Bitmap Line** button in the **Bitmap** toolbar to enter Bitmap Line mode.
2. If you want to change the size and shape of the bitmap line tool, you can do so as described in "Setting the Brush Size and Shape" on page 111.
3. Move the cursor over the area of the bitmap image that you want to draw a bitmap line in, and then click and drag in the appropriate direction. A preview line appears in the **2D View** window.
4. Release the mouse button to draw the bitmap line in the model.
5. Click on the **Select Vectors** button in the **Vector Editing** area of the **Assistant**'s Home page to exit from Bitmap Line mode.

Using the Erase Tool

The erase tool allows you to remove all or part of your last painting or drawing action from a bitmap image.

To erase all or part of your last painting or drawing action:

1. Click on the **Erase** button in the **Bitmap** toolbar to enter Erase mode.
2. If you want to change the size and shape of the erase tool, you can do so as described in "Setting the Brush Size and Shape" on page 111.
3. Move the cursor over the area of the bitmap image that you have drawn last, and then click and drag to erase it.

4. Click on the **Select Vectors** button in the **Vector Editing** area of the **Assistant**'s Home page to exit from Erase mode.

Flood Filling

You can replace areas of any selected colour with the current Primary Colour, using the **Flood Fill** tool.

To do so:

1. Click on the **Pick Colour** button in the **Bitmap** toolbar.
2. Move the dropper over the colour you want to select as the Primary Colour and click.
3. Click on the **Flood Fill** button in the **Bitmap Editing** area of the **Assistant**'s Home page, and then move the roller over the colour you want to flood fill with the Primary Colour and click. The block of colour you click on is replaced with the Primary Colour.

Selective Flood Filling

You can flood fill areas of any colour with the current Primary Colour, until it meets pixels of the current Secondary Colour, using the **Flood Fill Selective** tool.

To do so:

1. Click on the **Flood Fill Selective** button in the **Bitmap Editing** area of the **Assistant**'s Home page.
2. Click on the colour within the Colour Palette you want to select as the Primary Colour.
3. Right-click on the colour within the Colour Palette you want to select as the Secondary Colour.
4. Move the roller over the bitmap image and click to flood fill it with the Primary Colour. The Primary Colour continues flooding an area until it meets pixels of the Secondary Colour.

Note: If colours are surrounded by the Secondary Colour they are not filled in the Primary Colour. You must flood fill these areas separately.

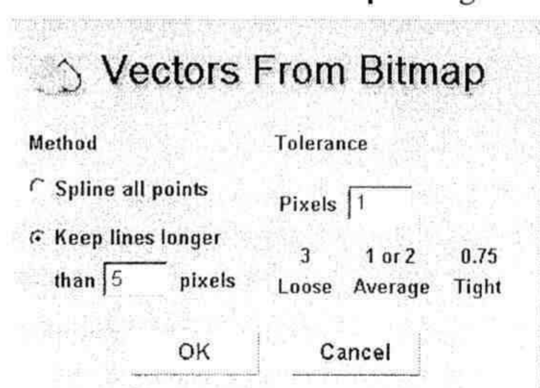
Converting a Bitmap into Vectors

You can convert any bitmap image into vector objects. ArtCAM Pro creates vector objects around the outline of all areas in the Primary Colour, together with those colours that are linked to the Primary Colour. For further details about colour linking, see "Colour Linking" on page 104.

When converting a bitmap image into vector objects, the vector objects follow the pixellated outline of the bitmap. You can smooth the outline of the vector objects by replacing the linear spans within them with bezier curves. For details, see "Smoothing Vector Objects" in the Working with Vectors chapter.

To convert a bitmap image into vector objects:

1. Click on the colour within the Colour Palette you want to select as the Primary Colour.
2. Click on the **Bitmap to Vector** button in the **Vector – Bitmap** area of the **Assistant**'s Home page to display the **Vectors From Bitmap** dialog box:

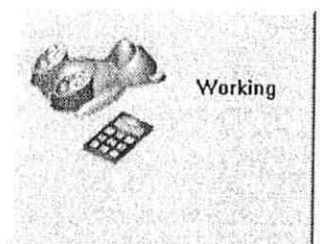


3. Click on the **Method** radio button you want to use:
   - **Spline all points** - This option fits bezier curve spans between all points (nodes) in the vector objects.

- **Keep lines longer than** [ ] **pixels** - This option fits bezier curve spans between all points (nodes), except where the number of consecutive pixels you type in the box form a straight line.

4. Type a value in the **Pixels** box to set the **Tolerance**. This sets how closely the bezier curve spans follow the points (nodes) in the vector objects.
5. Click on the **OK** button to close the **Vectors From Bitmap** dialog box and produce the vector objects in the **2D View** window.

   The **Working** box appears while ArtCAM Pro converts the bitmap image into vector objects:

6. Click on the **Bitmap On/Off** button in the **2D View** toolbar to hide the original bitmap image, allowing you to see the new vector objects clearly.

Creating a Shape from a Bitmap

You can generate three-dimensional shapes from any colour within a bitmap image using the **Shape Editor** tool. For more details, see "Creating a Shape from a Bitmap" in the Working with Reliefs chapter.

Working with Vectors

Overview

A vector object, or object-oriented graphic, is made up of a sequence of commands or mathematical statements that places lines and shapes in a given two-dimensional or three-dimensional space. Vector objects are fully scalable without loss of resolution, and file sizes are independent of resolution.

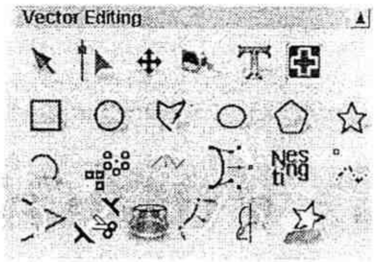
In ArtCAM Pro, you can both import vector artwork from other applications and convert a bitmap image into vector objects. For details, see "Importing Vector Artwork" in the Working with Models chapter and "Converting a Bitmap into Vectors" in the Working with Bitmaps chapter.

Once you have created or opened a model file, you can use the drawing tools in the **Vector Editing** area of the **Assistant**'s Home page and/or the **Vector** toolbar to create your own vector objects that can then be used to:

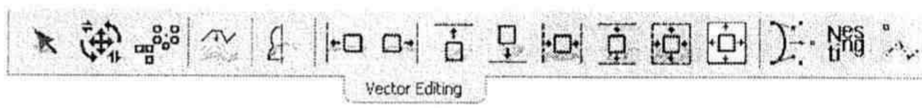
- Machine a 2D model. For details, see "2D Toolpath Strategies" in the Machining Models chapter.

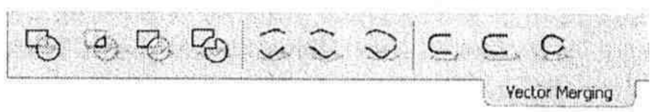
  Create a shape that can be used to machine a 3D model. For details, see "Creating a Shape from a Closed Vector" and "Creating a Shape Using Vectors" in the Working with Reliefs chapter, then "3D Toolpath Strategies" in the Machining Models chapter.

The group of vector drawing tools are found in the **Vector Editing** area of the **Assistant**'s Home page and on the **Vector** toolbar:



You can also find tools in the **Vector Editing** and **Vector Merging** toolbars that assist you in editing and manipulating the vector objects you create:





Drawing with Polylines

You can create freeform vector objects from polylines, which are one or more linear or bezier curve spans joined together by points (nodes).

Creating a Polyline

When creating a polyline, ArtCAM Pro displays the co-ordinates of the current cursor position, the angle and length of the span you are creating, the co-ordinates of the last point, and the change in position from the last point in both the X and Y direction on the **Polyline Creation** page.

To create a polyline:

1. Click on the **Create Polyline** button  in the **Vector Editing** area of the **Assistant**'s Home page to display the **Polyline Creation** page.
2. Click once in the **2D View** window to create the start point (node) of the polyline.
3. Drag the mouse. A dotted line appears indicating where the polyline is to be drawn.

Tip: Click and hold the left button before dragging the mouse to create a bezier curve span. Release the mouse button to create the point (node) in the polyline.

4. Click at the position where you want to create another point (node) in the polyline.

Tip: Press and hold the **Ctrl** key on your keyboard during each click of the mouse to constrain the angle of the linear span drawn between points (nodes) to 15° increments.

A linear span is drawn connecting the two points (nodes).

Repeat this and the previous step if you want to create further points (nodes) in the polyline connected by linear spans.

5. Right-click to end the polyline and return to the **Assistant**'s Home page.

Note: You can also end the polyline if you press the **Esc** key on your keyboard or click on the **Close** button on the **Polyline Creation** page.

For example, if you click in the right of the **2D View** window, drag downwards and left, and then right-click, you can create a polyline that looks something like this:

You can also create a polyline using co-ordinates taken from the **2D View** window:

1. Click on the **Create Polyline** button in the **Vector Creation** area of the **Assistant**'s Home page to display the **Polyline Creation** page.
2. Click on the position in the **2D View** window where you want to create the polyline's Start Point.
3. Define the position of the next point, using any of the following methods:
    - Type the X and Y co-ordinates for the next point in the **X** and **Y** boxes.

Tip: Move the cursor to the position where you want to create the point in the polyline. The X and Y co-ordinates for the position are shown in the **Cursor Position** area of the page.

- Type the angle of the next span in the polyline in the ° box, followed by its length in the **L** box.
- Type the distance from the previous point in the X and Y axes in the **dx** and **dy** boxes.

4. Click on the **Add** button.
5. Repeat these steps until you have finished creating the polyline, and then click on the **Close** button to return to the **Assistant**'s Home page.

Completing Polyline Creation

To complete the polyline you are drawing, but remain in polyline creation mode:

- Press the **Space Bar** on your keyboard.

To finish creating a polyline and return to the **Assistant**'s Home page, leaving it as an open vector object, you can use any of the following methods:

- Right-click on your mouse.
- Press the **Esc** key on your keyboard.
- Press the **Enter** key on your keyboard.
- Click on the **Close** button on the **Polyline Creation** page.

By default, the polyline is black. If, however, you create a polyline on a layer to which a colour has been given, the polyline adopts the layer's colour. For further information, see "Selecting Vectors" on page 138 and "Assigning a Colour to a Layer" in the Working with Models chapter.

Closing a Polyline to Create a Polygon

If you are in **Polyline Creation** mode, you can finish creating a polyline and automatically join its Start Point with its last point to form a polygon if you:

- Press the **Tab** key on your keyboard.

If you have already created a polyline, you can join its Start Point with its last point (node) to form a polygon if you:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Objects" on page 137.
2. Click to select the polyline that you have created. By default, the polyline is black and surrounded by a bounding box.
3. Click on the Start Point and drag it to meet the last point in the polyline.

Amending a Polyline

A polyline is made up of spans and points (nodes). Spans can be lines, arcs or bezier curves. You can change the position of the points and the nature of the spans after the polyline is created. For example, a linear span can be changed to an arc or a bezier curve span.

You change a polyline's form using both the Node Editing and Vector Editing menu options. These menus are displayed when working in Node Editing mode and right-clicking on either a span or a point. For details, see "Editing Vector Spans" on page 141 and "Editing Vector Nodes" on page 148.

Creating Simple Shapes

In addition to the **Create Polyline** button , as detailed in "Creating a Polyline" on page 120, there are a further six buttons in the **Vector Editing** area of the **Assistant**'s Home page that allow you to create specific vector object shapes.

You can create:

- Rectangles or Squares. See "Creating a Rectangle" on page 124.
- Circles. See "Creating a Circle" on page 126.
- Ellipses. See "Creating an Ellipse" on page 128.
- Polygons. See "Creating a Polygon" on page 131.
- Stars. See "Creating a Star" on page 133.
- Arcs. See "Creating an Arc" on page 136.

Creating a Rectangle

You can create either an exact or an approximate square or rectangular shaped vector object.

To create an exact square or rectangle:

1. Click on the **Create Rectangle** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Rectangle Creation** page.
2. Click on the **Square** or **Rectangle** radio button to select the type of shape you want to create.
3. Type the height you want for the shape in the **Height** box.
4. Type the width you want for the shape in the **Width** box.
5. If you want filleted (rounded) corners, type the radius you want to apply to all four corners in the **Corner Radii** box. Otherwise, leave the default value of 0 in the **Corner Radii** box.
6. Define the centre point of your shape. To do so, you can either:
   - Type values in the **Centre Point**'s **X** and **Y** boxes.
   - Move the + cursor over the **2D View** window and click on the point. The X and Y co-ordinates of the point you click on appear in the **Centre Point**'s **X** and **Y** boxes.
7. If you want your shape to be drawn at a specific angle to the model block, type a value in the **Angle** box:
   - Type a positive value to rotate the shape clockwise.
   - Type a negative value to rotate the shape anti-clockwise.
8. Click on the **Preview** button to see the shape you are about to create in the **2D View** window.
9. If you want to draw your shape and remain in Rectangle Creation mode, click on the **Create** button. If you want to draw your shape and return to the **Assistant**'s Home page, right-click.

To create an approximate sized square or rectangle:

1. Click on the **Create Rectangle** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Rectangle Creation** page.
2. Click on the **Square** or **Rectangle** radio button to select the type of shape you want to create.
3. Move the + cursor over the **2D View** window, and then click and drag to create the size of shape you want.
4. Release the mouse button to draw a preview image of the shape you are creating in the **2D View** window.
5. If you want to draw your shape and remain in Rectangle Creation mode, click on the **Create** button. If you want to draw your shape and return to the **Assistant**'s Home page, right-click.

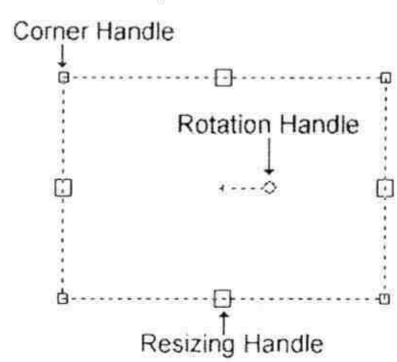
Editing a Square or Rectangle

You can edit the size, position, corners and/or angle of a square or rectangular shaped vector object.

To begin editing a square or rectangle:

1. Select the square or rectangle you want to edit. For details, see "Selecting Vectors" on page 138.
2. Right-click on the selected shape to display the Vector Editing menu.
3. Click on the **Edit Rectangle** option to display the **Rectangle Editing** page in the **Assistant** window.

   The selected shape now appears with resizing handles attached to it, and a rotation handle for its centre point:



Note: You can display the **Rectangle Editing** page by selecting a square or rectangle, and then pressing the **E** key on your keyboard.

You can now change the size, position, corners and/or angle of the selected square or rectangle using the **Rectangle Editing** page, or by manipulating the handles on the shape itself.

To edit the selected square or rectangle using the **Rectangle Editing** page:

- Type a new value in the appropriate box for each attribute of the shape that you want to adjust.

  For example, typing a new value in the **Height** box immediately increases or decreases the height of the selected square or rectangle.

You can edit the selected square or rectangle using the shape itself:

- Click and drag any of the resizing handles to adjust the size of the shape. The new height and width is shown in the **Height** and **Width** boxes on the **Rectangle Editing** page.
- Click and drag any of the corner handles to add filleting to all four corners of the shape. The new radius of the filleted corners is shown in the **Corner Radii** box on the **Rectangle Editing** page.
- Click and drag the rotation handle to adjust the angle of the shape. The new angle is shown in the **Angle** box on the **Rectangle Editing** page.
- Move the + cursor over the centre point of the shape. When it changes to the ✥ cursor, click and drag to move the whole shape. The new centre point appears in the **Centre Point**'s **X** and **Y** boxes on the **Rectangle Editing** page.

Tip: To create an exact square, hold down the **Shift** key down on your keyboard, and then drag the resizing handles on your rectangle to the required size.

To confirm the new shape shown in the **2D View** window and return to the **Assistant**'s Home page, right-click.

Creating a Circle

You can create a circular shaped vector object.

To create a circle of a specified size:

1. Click on the **Create Circles** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Circle Creation** page.
2. Define the centre point of your circle. To do so, you can either:
   - Type values in the **Circle Centre**'s **X** and **Y** boxes.
   - Move the + cursor over the **2D View** window and click on the point. The X and Y co-ordinates of the point you click on are shown in the **Circle Centre**'s **X** and **Y** boxes.
3. Type a value in the **Circle Radius** box.
4. Click on the **Preview** button to draw a preview image of the circle you are creating in the **2D View** window.
5. If you want to draw your circle and remain in **Circle Creation** mode, click on the **Create** button. If you want to draw your circle and return to the **Assistant**'s Home page, right-click.

To create an approximate sized circle:

1. Move the + cursor over the **2D View** window, and then click and drag to create the size of circle you want.
2. Release the mouse button to draw a preview image of the circle you are creating in the **2D View** window.
3. If you want to draw your circle and remain in Circle Creation mode, click on the **Create** button. If you want to draw your circle and return to the **Assistant**'s Home page, right-click.

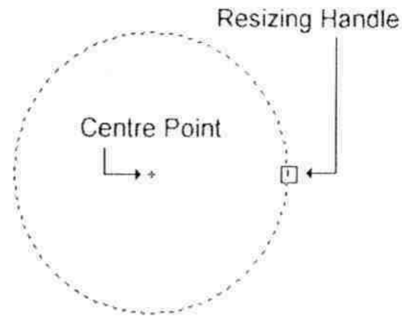
Editing a Circle

You can edit the radius and position of a circular shaped vector object.

To edit a circle:

1. Select the circle you want to edit. For details, see "Selecting Vectors" on page 138.
2. Right-click on the circle to display the Vector Editing menu.
3. Click on the **Edit Circle** option to display the **Circle Editing** page in the **Assistant** window.

The selected circle now appears as a preview image with a resizing handle:



You can now change the size and position of the selected circle using the **Circle Editing** page, or by manipulating the resizing handle and the centre point on the shape itself.

Note: You can display the **Circle Editing** page by selecting a circle, and then pressing the **E** key on your keyboard.

To edit the selected circle using the **Circle Editing** page:

- Type a new value in the appropriate box for either of the shape's attributes that you want to adjust.

  For example, typing a new value in the **Radius** box immediately increases or decreases the size of the selected circle.

You can edit the selected circle using the shape itself:

- Click and drag the resizing handle to adjust its radius. The new radius is shown in the **Radius** box on the **Circle Editing** page.
- Move the + cursor over the centre point of the circle. When it changes to the ✥ cursor, click and drag to move the circle. The new centre point is shown in the **Centre Point**'s **X** and **Y** boxes on the **Circle Editing** page.

To confirm the new circle shown in the **2D View** window and return to the **Assistant**'s Home page, right-click.

Creating an Ellipse

You can create an elliptical shaped vector object.

To create an ellipse of a specified size:

1. Click on the **Create Ellipse** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Ellipse Creation** page.
2. Define the centre point of your ellipse. To do so, you can either:
3. Type the height you want for the ellipse in the **Ellipse Height** box.
4. Type the width you want for the ellipse in the **Ellipse Width** box.
5. If you want your shape to be drawn at a specific angle to the model block, type a value in the **Angle** box:
6. Click on the **Preview** button to see the ellipse you are about to create in the **2D View** window.
7. If you want to draw your ellipse and remain in **Ellipse Creation** mode, click on the **Create** button.

To create an approximate sized ellipse:

1. Move the + cursor over the **2D View** window, and then click and drag to create the size of ellipse you want.
2. Release the mouse button to draw a preview image of the ellipse you are creating in the **2D View** window.
3. If you want to draw your ellipse and remain in Ellipse Creation mode, click on the **Create** button. If you want to draw your ellipse and return to the **Assistant**'s Home page, right-click.

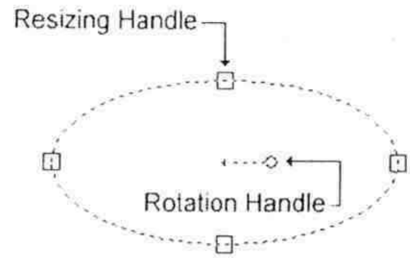
Editing an Ellipse

You can edit the size, position and/or angle of an elliptical shaped vector object.

To edit an ellipse:

1. Select the ellipse you want to edit. For details, see "Selecting Vectors" on page 138.
2. Right-click on the selected ellipse to display the Vector Editing menu.
3. Click on the **Edit Ellipse** option to display the **Ellipse Editing** page in the **Assistant** window.

The selected ellipse now appears as a preview image with resizing handles attached to it, and a rotation handle about its centre point:



Note: You can display the **Ellipse Editing** page by selecting an ellipse, and then pressing the **E** key on your keyboard.

You can now change the size, position and/or angle of the selected ellipse using the **Ellipse Editing** page, or by manipulating the handles on the ellipse itself.

To edit the selected ellipse using the **Ellipse Editing** page:

- Type a new value in the appropriate box for each attribute of the ellipse that you want to adjust.

  For example, typing a new value in the **Height** box immediately increases or decreases the height of the ellipse.

You can edit the selected ellipse using the shape itself:

- Click and drag any of the resizing handles to adjust the size of the ellipse. The new height and width is shown in the **Height** and **Width** boxes on the **Ellipse Editing** page.
- Click and drag the rotation handle to adjust the angle of the ellipse. The new angle is shown in the **Angle** box on the **Ellipse Editing** page.
- Move the + cursor over the centre point of the ellipse. When it changes to the cursor, click and drag to move the whole ellipse. The new centre point is shown in the **Start Point**'s **X** and **Y** boxes on the **Ellipse Editing** page.

To confirm the new ellipse shown in the **2D View** window and return to the **Assistant**'s Home page, you can either:

- Click on the **Apply** button, followed by the **Close** button on the **Ellipse Editing** page.
- Click the right mouse button.

Creating a Polygon

You can create a vast range of polygonal shaped vector objects.

To create an exact polygon:

1. Click on the **Create Polygons** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Polygon Creation** page.
2. Type the number of sides you want in the polygon in the **Settings' No. of Sides** box.

Warning: A polygon must have a minimum number of three sides. Typing a value of 3 in the **Settings' No. of Sides** box produces a triangle.

3. If you want your polygon to be drawn at a specific angle to the model block, type a value in the **Settings' Angle** box:
   - Type a positive value in the **Angle** box to rotate the polygon clockwise.
   - Type a negative value in the **Angle** box to rotate the polygon anti-clockwise.

   If you do not want your polygon to be drawn at a specific angle, leave the default value of 0 in the **Settings' Angle** box.
4. Define the centre point of your polygon. To do so, you can either:
   - Type values in the **Polygon Centre**'s **X** and **Y** boxes.
   - Move the + cursor to the point in the model you are interested in and then click. The X and Y co-ordinates of the point you click on appear in the **Polygon Centre**'s **X** and **Y** boxes.
5. Type a value in the **Geometry**'s **Radius** box.
6. Click on the **Preview** button to see the polygon you are about to create in the **2D View** window.
7. If you want to draw your polygon and remain in Polygon Creation mode, click on the **Create** button. If you want to draw your polygon and return to the **Assistant**'s Home page, right-click.

To create an approximate sized polygon:

1. Move the + cursor over the **2D View** window, and then click and drag to create the size of polygon you want.
2. Release the mouse button to draw a preview image of the polygon you are creating in the **2D View** window.
3. If you want to draw your polygon and remain in Polygon Creation mode, click on the **Create** button. If you want to draw your polygon and return to the **Assistant**'s Home page, right-click.

Editing a Polygon

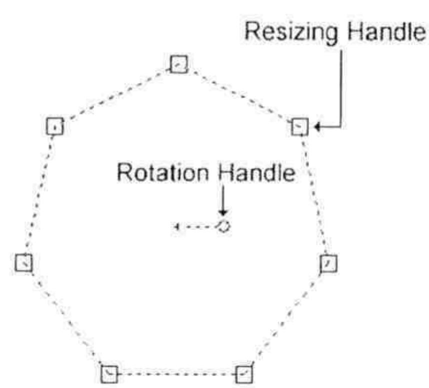
You can edit the size, position, sides and/or angle of a polygonal shaped vector object.

To edit a polygon:

1. Select the polygon you want to edit. For details, see "Selecting Vectors" on page 138.
2. Right-click on the polygon to display the Vector Editing menu.
3. Click on the **Edit Polygon** option to display the **Polygon Editing** page in the **Assistant** window.

Note: You can display the **Polygon Editing** page in the **Assistant** window by pressing the **E** key on your keyboard.

The polygon now appears as a preview image with resizing handles and a rotation handle:



You can now change the size, position, sides and/or the angle of the selected polygon using the **Polygon Editing** page, or by manipulating the handles on the polygon itself.

To edit the selected polygon using the **Polygon Editing** page:

- Type a new value in the appropriate box for each attribute of the polygon that you want to adjust.

  For example, typing a new value in the **No. of Sides** box immediately increases or decreases the number of sides in the selected polygon.

You can edit the selected polygon using the shape itself:

- Click and drag any of the resizing handles to adjust the size of the polygon. The new size is shown in the **Geometry**'s **Radius** box on the **Polygon Editing** page.
- Click and drag the rotation handle to adjust the angle of the polygon. The new angle is shown in the **Angle** box on the **Rectangle Editing** page.
- Move the + cursor over the centre point of the polygon. When it changes to the ✥ cursor, click and drag to move the whole polygon. The new centre point is shown in the **Polygon Centre**'s **X** and **Y** boxes on the **Polygon Editing** page.

To confirm the new polygon shown in the **2D View** window and return to the **Assistant**'s Home page, you can either:

- Click on the **Apply** button, followed by the **Close** button on the **Polygon Editing** page.
- Click the right mouse button.

Creating a Star

You can create a star shaped vector object.

To create an exact star:

1. Click on the **Create Stars** button ☆ in the **Vector Editing** area of the **Assistant**'s Home page to display the **Star Creation** page.
2. Type the number of sides you want in the polygon in the **Settings**' **No. of Points** box.

Warning: A star must have a minimum number of three points.

---

3. If you want your star to be drawn at a specific angle to the model block, type a value in the **Settings**' **Angle** box:

- Type a positive value in the **Angle** box to rotate the star clockwise.
- Type a negative value in the **Angle** box to rotate the star anti-clockwise.

If you do not want your star to be drawn at a specific angle, leave the default value of 0 in the **Settings' Angle** box.

4. Define the centre point of your star. To do so, you can either:
   - Type values in the **Star Centre**'s **X** and **Y** boxes.
   - Move the + cursor to the point in the model you are interested in and then click. The X and Y co-ordinates of the point you click on appear in the **Star Centre**'s **X** and **Y** boxes.
5. Type a value in the **Geometry**'s **Radius of first Points** box. This sets the radius of a polygon, upon which the outer points (nodes) in the star lie.
6. Type a value in the **Geometry**'s **Radius of second Points** box. This sets the radius of a polygon, upon which the inner points (nodes) in the star lie.
7. Click on the **Preview** button to see the ellipse you are about to create in the **2D View** window.
8. If you want to draw your star and remain in Star Creation mode, click on the **Create** button. If you want to draw your star and return to the **Assistant**'s Home page, right-click.

To create an approximate sized star:

1. Move the + cursor over the **2D View** window, and then click and drag to create the size of polygon you want, upon which the outer points in the star lie.
2. Click and drag to create the size of polygon you want, upon which the inner points in the star lie.
3. Release the mouse button to draw a preview image of the star you are creating in the **2D View** window.
4. If you want to draw your star and remain in Star Creation mode, click on the **Create** button. If you want to draw your polygon and return to the **Assistant**'s Home page, right-click.

Editing a Star

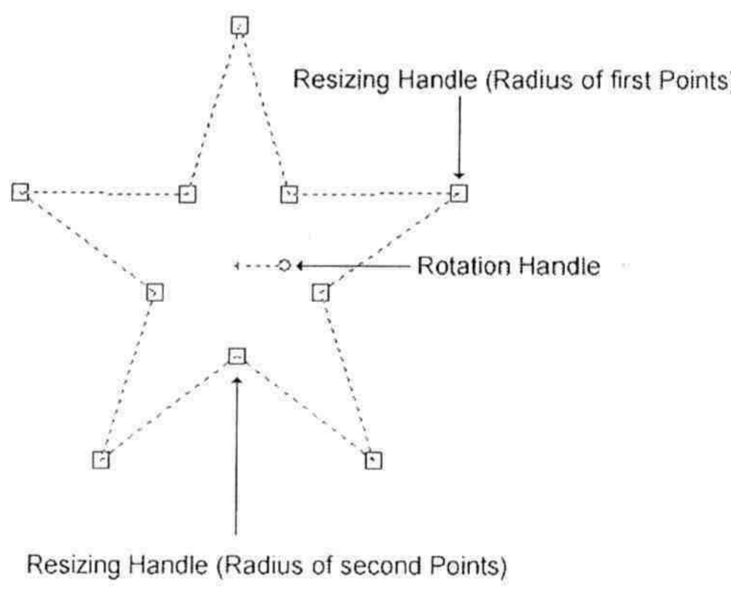
You can edit the size, position, sides and/or angle of a star shaped vector object.

To edit a star:

1. Select the star you want to edit. For details, see "Selecting Vectors" on page 138.
2. Right-click on the star to display the Vector Editing menu.
3. Click on the **Edit Star** option to display the **Star Editing** page in the **Assistant** window.

Note: You can display the **Star Editing** page in the **Assistant** window by pressing the **E** key on your keyboard.

The star now appears as a preview image with resizing handles attached to it, and a rotation handle about its centre point:



You can now change the size, position, sides and/or the angle of the selected star using the **Star Editing** page, or by manipulating the handles on the star itself.

To edit the selected star using the **Star Editing** page:

- Type a new value in the appropriate box for each attribute of the star that you want to adjust.

For example, typing a new value in the **No. of Sides** box immediately increases or decreases the number of sides in the selected star.

You can edit the selected star using the shape itself:

- Click and drag any of the inner or outer resizing handles to adjust the size and shape of the star. The new size is shown in the **Geometry**'s **Radius of first Points** and/or **Radius of second Points** boxes on the **Star Editing** page.
- Click and drag the rotation handle to adjust the angle of the star. The new angle is shown in the **Angle** box on the **Rectangle Editing** page.
- Move the + cursor over the centre point of the star. When it changes to the ✥ cursor, then click and drag to move the whole star. The new centre point appears in the **Star Centre**'s **X** and **Y** boxes on the **Star Editing** page.

To confirm the new star shown in the **2D View** window and return to the **Assistant**'s Home page, you can either:

- Click on the **Apply** button, followed by the **Close** button on the **Star Editing** page.
- Click the right mouse button.

Creating an Arc

You can create an arc shaped vector object.

To create an arc:

1. Click on the **Create Arcs** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Arc Creation** page.
2. Click on either of the **Arc Type** radio buttons to select the type of arc you want to create:
    - **Centre – Start – End** – A swept arc created by defining a centre point, a Start Point and then an end point.
    - **Start – End – Point On Arc** – A fitted arc created by defining a Start Point, an end point and then a mid point.

3. If you have selected the **Centre – Start – End** option:
4. If necessary, you can now change the position of any of the three defined points:
5. Click on the **Preview** button to see the arc you are about to create in the **2D View** window.
6. If you want to draw your arc and remain in Arc Creation mode, click on the **Create** button. If you want to draw your arc and return to the **Assistant**'s Home page, right-click.

Editing Vector Objects

You can edit any of the standard shapes of vector object in a model in the same way: square, rectangle, circle, ellipse, polygon and star.

To edit any of these standard shapes, select the vector object, then either:

- Press the **E** key on your keyboard.
- Right-click to display the Vector Editing menu and then click on the **Edit** option for the selected vector object e.g. **Edit Ellipse**.

ArtCAM Pro displays the Editing page for the selected vector object in the **Assistant** e.g. **Ellipse Editing**. You are then able to change the properties for the selected shape of vector object by modifying the appropriate values. For example, the **Height** of a rectangle.

You cannot edit the following vector objects using an Editing page in the **Assistant**, although they still can be edited:

- Created with the **Polyline Creation** tool. For details, see "Creating a Polyline" on page 120.
- Created with the **Arc Creation** tool. For details, see "Creating an Arc" on page 136.
- Created with one of the buttons in the **Vector Editing** area of the **Assistant**'s page, and in which you have moved or edited a point (node) or span. For details, see "Editing Vector Objects" on page 137.
- Imported directly into ArtCAM Pro. For details, see "Importing Vector Artwork" in the Working with Models chapter.

- Created from a bitmap image using the **Bitmap to Vector** tool. For details, see "Converting a Bitmap into Vectors" in the Working with Bitmaps chapter.

Selecting Vectors

You can select vector objects from any of the visible layers in a model. For further information on layers, see "Viewing a Layer" in the Working with Models chapter.

To select a single vector object from a layer:

1. Click on the **Select Vectors** button  in the **Vector Editing** area, and then on the vector object that you want to select.

Note: If the **Select Vectors** button is shown as  in the **Vector Editing** area, you are already in Select Vectors mode.

Note: You can also press the **Esc** key to enter Select Vectors mode, or right-click to display the Vector Editing menu, and then click on the **Select Vectors Mode** option.

Note: You can also use the **N** key to toggle between the different modes. For example, pressing the **N** key once when in Transform Vectors mode takes you into Node Editing mode. Pressing it twice takes you into Select Vectors mode.

To select more than one vector object from any visible layer:

1. Click on the **Select Vectors** button  in the **Vector Editing** area, and then on the vector object that you want to select.
2. Hold the **Shift** key down on your keyboard, and then click on each of the vector objects you want to select.

Note: You can also click and drag around any number of vector objects to select them. A bounding box surrounds all selected vector objects.

Tip: To deselect a vector object you have selected by mistake, hold the **Shift** key down on your keyboard and then click.

Vector objects are not always shown in the colour assigned to the layer from which they have been selected:

Note: You can set the default colour for selected vector objects using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- By default, one or more ungrouped vector objects are magenta when selected. When deselected, they appear in the colour assigned to the layer on which they are drawn.

  You can change the default colour used to indicate single and multiple ungrouped vector objects when selected using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- By default, all grouped closed vector objects are purple when selected, unless they are self-intersecting. Then they are shown in red with the ○ icon marking the position of all intersections. When deselected, all grouped closed vector objects appear in the colour assigned to the layer on which they are drawn, again unless they are self-intersecting. Then they are shown in red.

  You can change the default colour used to indicate grouped vector objects when selected using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- By default, any locked vector object is grey when selected. When deselected it is green, regardless of the colour assigned to the layer on which it is drawn.

  You can change the default colours used to indicate locked vector objects when selected and deselected using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- A grouped open vector object is blue when deselected, regardless of the colour assigned to the layer on which it is drawn.

- If you click and drag to around a vector object to select it, and it has one or more overlaying copies, the vector object turns red.

  You can change the default colour used to indicate overlaying vector objects using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Moving Vectors

You can move part or the whole of any selected vector object to another area of the layer, or to a different layer in a model. For further details, see "Transferring Vectors Between Layers" in the Working with Models chapter.

Note: You cannot move a vector object if it is locked. For details, see

To move a vector object to another position in the current layer:

1. Select the vector object that you want to move. For details, see "Selecting Vectors" on page 138.
2. Move the cursor over the bounding box. The cursor changes from to .
3. Click and drag the vector object into its new position, and then release the mouse button.

To transfer a vector object from one layer to another:

1. Select the vector object that you want to transfer to another layer. For details, see "Selecting Vectors" on page 138.
2. Right-click on the selected vector object to display the Vector Editing menu, and then click on the **Move To Layer** option to display the list of layers in the model.
3. Click to select the layer from those listed to which you want to transfer the selected vector object. The selected vector object is transferred to the layer and is now shown in the colour assigned to it.

Alternatively, you can transfer a vector object from one layer to another in the following way:

1. Click on the **Layers** tab Layers to display the **Layers** page.
2. Select the vector object that you want to transfer to another layer. For details, see "Selecting Vectors" on page 138.

Tip: If you want to transfer all of the vector objects in the currently selected layer, click on the **Select All** button.

3. In the list box at the bottom of the **Layers** page, click on the layer to which you want to transfer the selected vector object. The selected vector object is transferred to the chosen layer and are shown in the colour assigned to it.

Editing Vector Spans

Spans within a vector object can be linear, arc or a bezier curve. You can change the type of span using the Vector Editing options, displayed when you right-click on a selected vector object.

The Vector Editing options allow you to:

- Convert a span to a line. For details, see "Converting a Span to a Line" on page 141.
- Convert a span to a bezier curve. For details, see "Converting a Span to a Bezier Curve" on page 142.
- Convert a span to an arc. For details, see "Converting a Span to an Arc" on page 143.
- Insert a point (node) into a span. For details, see "Inserting a Point" on page 144.
- Remove a span. For details, see "Removing a Span" on page 147.

Converting a Span to a Line

Converting a span to a line allows you to alter the appearance of an arc or a bezier curve span within a vector object.

To convert a span to a line:

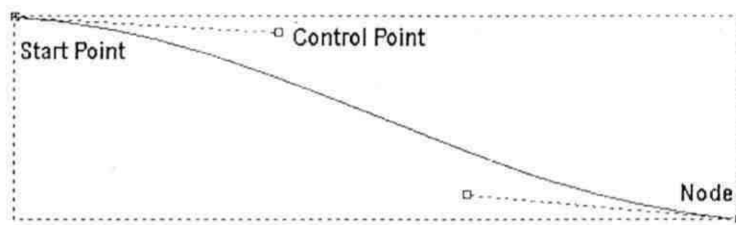
1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select a vector object containing an arc or bezier curve span. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.

3. Position the cursor over the arc or bezier curve span. When the cursor changes from ➤ to , right-click to display the Vector Editing menu.
4. Click on the **Convert span to a line** option to convert the existing span into a linear span.

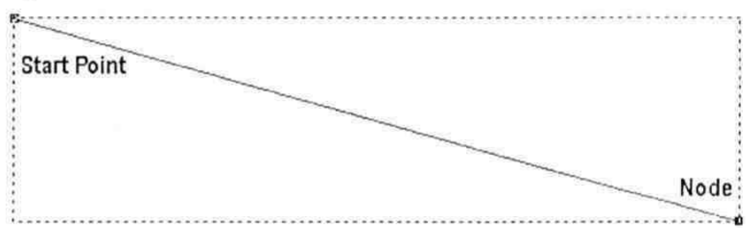
Note: Moving the cursor over the arc or bezier curve span and pressing the **L** key on your keyboard also converts it to a linear span.

For example, if you were to convert a bezier curve span to a linear span you would see that its adjoining control points have been removed along with any curvature, as shown below:

Before...



After...



Converting a Span to a Bezier Curve

Converting a span to a bezier curve allows you to alter the appearance of a linear or an arc span within a vector object.

Converting a span to a bezier curve places two control points on the span, both of which can be dragged to add or reduce the curvature within it.

To convert a span into a bezier curve:

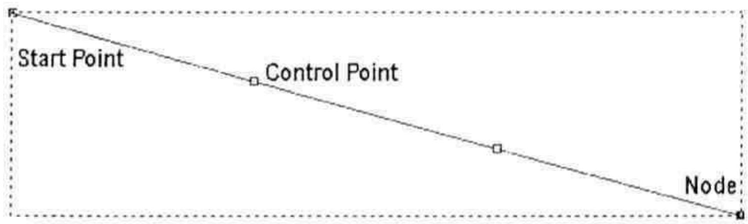
1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select a vector object containing an arc or linear span. A bounding box surrounds it, within which you can

see the spans, points (nodes) and control points that make up the selected vector object.

3. Position the cursor over the arc or linear span. When the cursor changes from ▶ to , right-click to display the Vector Editing menu.
4. Click on the **Convert span to bezier** option to convert the existing span into a bezier curve span.

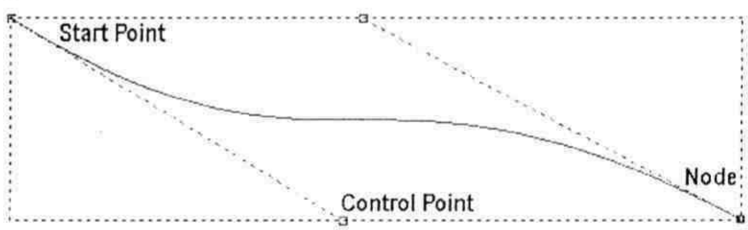
Note: Moving the cursor over the linear or arc span and pressing the **B** key on your keyboard also converts it to a bezier curve.

For example, your span may look something like that shown below:



Although this span may not look like a curve now, the two control points in it indicate that it is a bezier curve span.

In our example, by dragging the control points as shown below, a bezier curve span with a smooth curvature is produced:



Converting a Span to an Arc

Converting a span to an arc allows you to alter the appearance of a linear or bezier curve span within a vector object.

Converting a span to an arc places one control point in the middle of the span. This can be dragged to add or reduce the degree of curvature in the arc.

To convert a span into an arc:

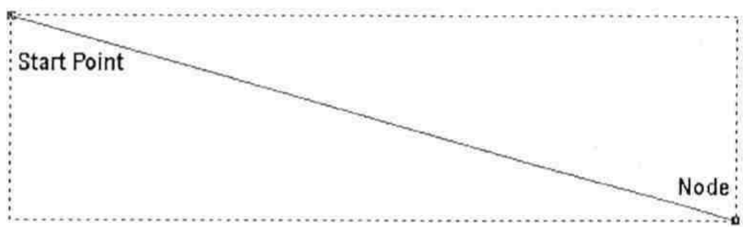
1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.

2. Click to select a vector object containing an bezier curve or linear span. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Position the cursor over the bezier curve or linear span. When the cursor changes from to , right-click to display the Vector Editing menu.
4. Click on the **Convert span to an arc** option to convert the existing span into an arc span.

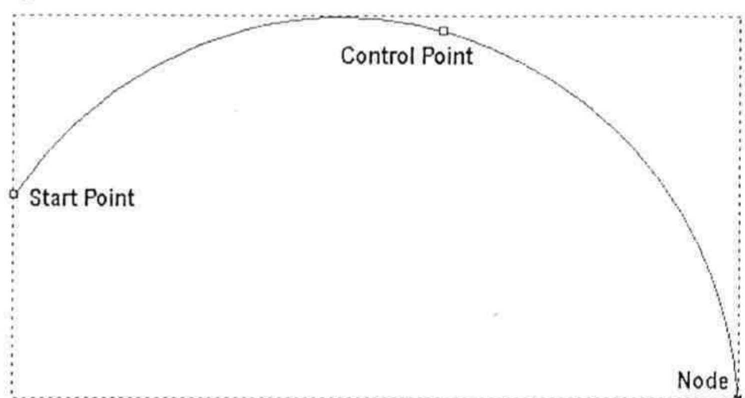
Note: Moving the cursor over the linear or bezier curve span and pressing the **A** key on your keyboard also converts it to an arc.

For example, if you were to convert a linear span to an arc span you would see that one control point is now positioned in the centre of the arc. You can drag the control point to create a curve, as shown below:

Before...



After...



Inserting a Point

You can insert a point (node) in any span. Inserting a point divides a span into two new spans of the same type as the original.

By inserting a point you add greater flexibility in changing the shape of the vector object.

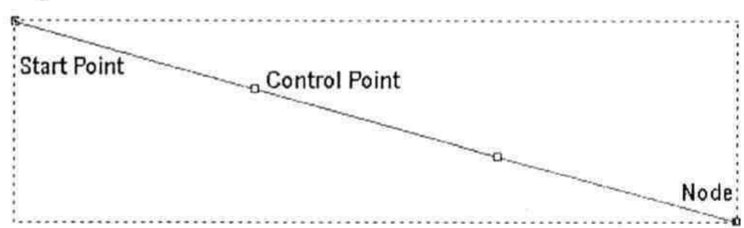
To insert a point in a span:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object into which you want to insert a point. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
5. Position the cursor over the span into which you want to insert a point. When the cursor changes from to , right-click to display the Vector Editing menu.
3. Click on the **Insert a point** option to produce a new point in the span, dividing it into two separate spans.

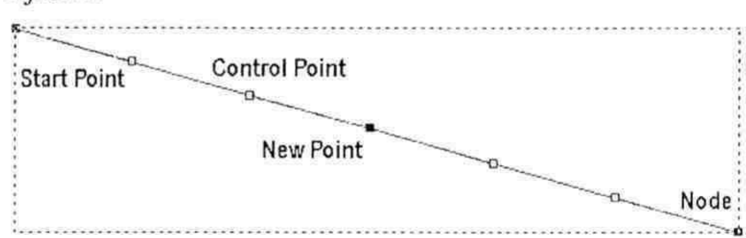
Note: Moving the cursor over the span and pressing the **I** key on your keyboard also inserts a point into it.

For example, a point has been inserted into the bezier curve span illustrated below. You can see that doing so has created a second bezier curve span with two adjoining control points:

Before...



After...



Tip: Adding points to a vector object can be a time consuming process. Alternatively, you can use the **Create Polyline** tool to create more complicated shapes from the beginning. With the **Create Polyline** page displayed, each corresponding movement and click of the mouse creates a new point and connects it to the last with a linear or bezier curve span. For details, see "Creating a Polyline" on page 120.

Inserting a Start Point

The green point shown within a selected vector object is the Start Point.

The Start Point in a vector object determines the point at which a tool enters the block of material when you are machining. For further details, see "Changing the Start Point Position" in the Machining Models chapter.

Inserting a Start Point divides a span into two new spans of the same type as the original. Alternatively, you can set an existing point (node) in a span as the Start Point. For details, see "Changing the Start Point" on page 153.

Note: For open vector objects, the Start Point must be at either end.

To insert a Start Point in a span:

1. Make sure that you are in Node Editing mode. For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object into which you want to insert a Start Point. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Position the cursor over the span into which you want to insert a Start Point. When the cursor changes from to, right-click to display the Vector Editing menu.
4. Click on the **Insert Start Point** option to produce a new Start Point in the span and divide it into two separate spans.

Note: Moving the cursor over the span and pressing the **P** key on your keyboard also inserts a Start Point into it.

Removing a Span

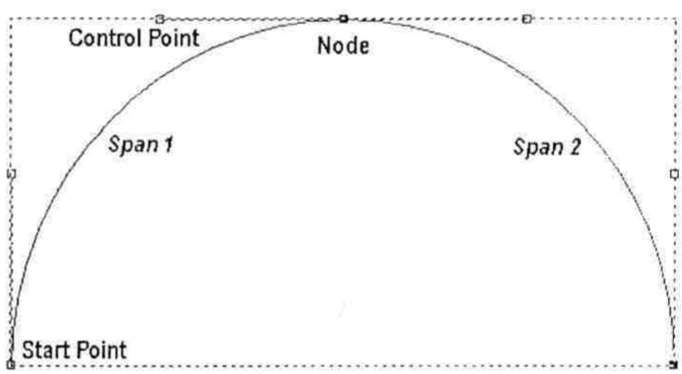
You can remove a linear, bezier curve or an arc span within any selected vector object in the same way.

To remove a span:

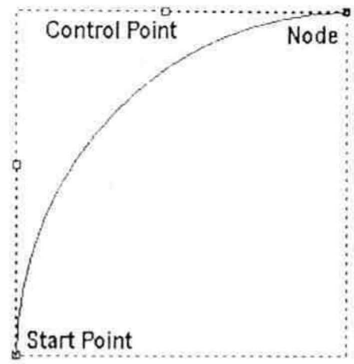
1. Make sure that you are in Node Editing mode. For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object from which you want to remove a span. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Position the cursor over the span. When the cursor changes from to , right-click to display the Vector Editing menu.
4. Click on the **Remove Span** option to remove the selected span.

Note: Moving the cursor over the span and pressing the **R** key on your keyboard also removes it.

For example, a vector object with two spans appears as follows:



In our example, if you move the cursor over *Span 2* and then remove it from the vector object, the remaining span is shown below:



Editing Vector Nodes

You can change the structure and appearance of a vector object using the Node Editing options. The Node Editing menu is displayed when you right-click on any point (node) within a selected vector object.

The Node Editing menu allows you to:

- Smooth points (nodes). For details, see "Smoothing Points" on page 150.
- Delete points (nodes). For details, see "Deleting Points" on page 152.
- Change the position of the start point. For details, see "Changing the Start Point" on page 153.
- Change the position of points (nodes) and/or control points. For details, see "Changing the Position of Points" on page 154.
- Align points (nodes). For details, see "Aligning Points" on page 155.

To edit points within a vector object, you must work in Node Editing mode. Use any of the following methods to enter Node Editing mode:

Note: If the **Node Editing** button is shown as in the **Vector Editing** area, you are already in Node Editing mode.

- Click on the **Node Editing** button in the **Vector Editing** area of the **Assistant**'s Home page.
- Press the **N** key on your keyboard.
- Hold the **Ctrl** key down on your keyboard, and then click once.

The **Node Editing** button changes to .

Note: If you click outside of the bounding box that surrounds the selected vector object once, ArtCAM Pro remains in Node Editing mode. If you click more than once, ArtCAM Pro returns to Select Vectors mode. For details, see "Selecting Vectors" on page 138.

Selecting Points and Control Points

You can select single or multiple points and/or control points in a selected vector object.

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the points and/or control points you want to edit. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. If you want to select a point (node) or control point:
   - Move the cursor over the point or control point that you want to select. When the cursor changes to a , click to select it. The selected point or control point turns red.

     If you select a point that has adjoining control points, these are selected along with the point itself. All of which turn red.

   If you want to select more than one point (node) or control point, you can use any of the following methods:

   - Click and drag to form a bounding box around them. The selected points and/or control points turn red.
   - Click to select a point or control point. Hold the **Ctrl** key down on your keyboard, and then click on further points and/or control points to select them. The selected points and/or control points turn red.
   - Click to select a point or control point. Hold the **Shift** key down on your keyboard, and then click to select a second point. ArtCAM Pro finds the shortest distance between these two points, and then selects

all of the other points in between them. The selected points and/or control points turn red.

Deselecting Points and Control Points

If you want to deselect a point or control point:

- Move the cursor over the point or control point that you want to deselect. When the cursor changes to a , hold the **Shift** key down on your keyboard and then click on the point or control point to deselect it.

Smoothing Points

You can smooth any of the points (nodes) in a vector object other than the Start Point or the last point in an ungrouped, open vector object.

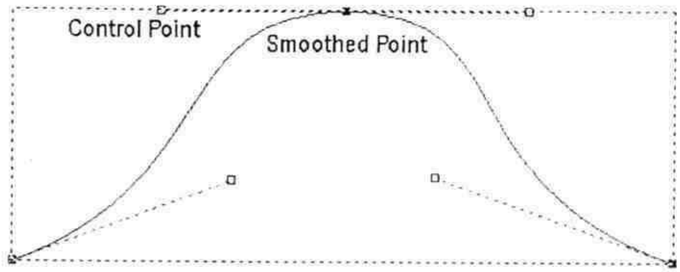
In smoothing, the span on either side of a point is converted to a bezier curve span. ArtCAM Pro positions control points next to the point, allowing you to control the degree of curvature in the whole vector object.

To smooth a single point:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the point you want to smooth. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object. All unsmoothed points are shown in black.
3. Move the cursor over the point that you want to smooth. When the cursor changes to a , right-click to display the Vector Editing menu.
4. Click to select the **Smooth Point** option to convert the spans on either side of the point to bezier curves. The point turns blue.

Note: Moving the cursor over the point and pressing the **S** key on your keyboard also smoothes it.

For example, the point in the middle of the vector object shown below has been smoothed:



If you move one of the control points next to the smoothed point, the other automatically moves with it. This simultaneous movement preserves the tangency between the two bezier curve spans.

To smooth a group of points:

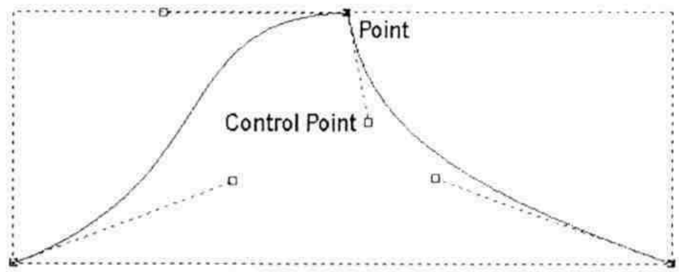
1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the points you want to smooth. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Select the points that you want to smooth. For details, see "Selecting Points and Control Points" on page 149.
4. Right-click on any of the selected points to display the Node Editing menu.
5. Click on the **Smooth Points** option to convert the spans on either side of the selected point(s) to bezier curves.

To remove the smoothing option from a smoothed point:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the smoothed points from which you want to remove the smoothing. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Move the cursor over a smoothed point (node). When the  cursor changes to a , right-click to display the Node Editing menu. The **Smooth Point** option is selected.
4. Click on the **Smooth Point** option to deselect it. The point (node) turns black.

Although the span on either side of the point (node) remains as a bezier curve span, deselecting the **Smooth Point** option causes the control point on either side of the point (node) to affect its adjoining bezier curve span only, rather than the whole vector object.

In our example, you can see that moving one control point no longer automatically moves the other:



Deleting Points

You can delete any of the points (nodes) within a vector object.

Deleting a point does not change the type of span on either side of the point but it does reduce your ability to manipulate the shape of the whole vector object.

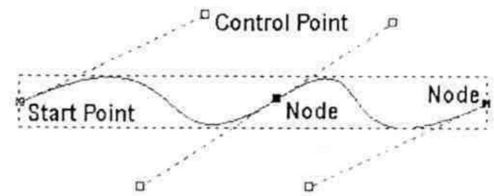
To delete a single point:

1. Make sure that you are in Node Editing mode. For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the point you want to delete. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Move the cursor over the point that you want to smooth. When the cursor changes to a , right-click to display the Vector Editing menu.
4. Click to select the **Delete Point** option to delete the point from the vector object.

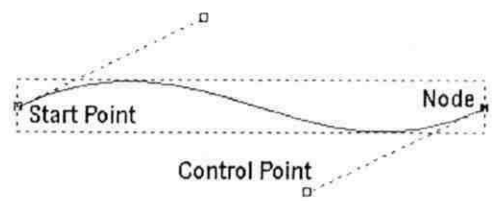
Note: Moving the cursor over the point and pressing the **D** key on your keyboard also deletes it.

In the following example, you can see that in deleting the middle point, the number of control points is reduced and its shape is changed considerably:

Before...



After...



To delete a group of points:

1. Make sure that you are in Node Editing mode. For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the points you want to delete. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Select the points that you want to delete. For details, see "Selecting Points and Control Points" on page 149.
4. Right-click on any of the selected points to display the Node Editing menu.
5. Click on the **Smooth Points** option to convert the spans on either side of the selected point(s) to bezier curves.

Note: Moving the cursor over any of the selected points and pressing the **D** key on your keyboard also deletes them.

Changing the Start Point

The Start Point in a vector object determines the point at which a tool enters the block of material when you are machining your model.

You can change the position of the Start Point in any vector object. The green point within a selected vector object is the Start Point.

To change the position of the Start Point:

1. Make sure that you are in Node Editing mode. For details, see "Editing Vector Nodes" on page 148.

2. Click to select the vector object in which you want to change the position of its Start point. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Move the cursor over the point that you want to set as the Start Point. When the cursor changes to a , right-click to display the Vector Editing menu.
4. Click on the **Start Point** option to make the point the Start Point. The point changes from either blue or black to green.

Note: Moving the cursor over the point and pressing the **P** key on your keyboard also changes it to the Start Point.

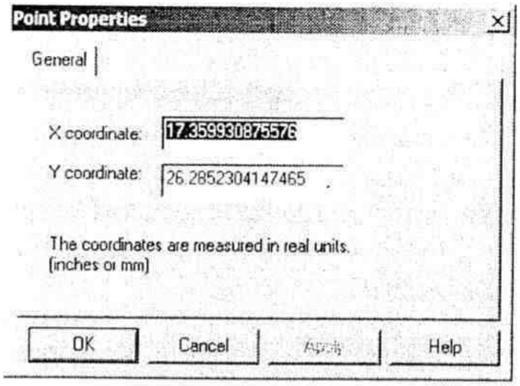
Using a toolpath preview in the **2D View** window, you can also set the position of the Start Point in any toolpath containing a profile pass. For further details, see "Changing the Start Point Position" in the Machining Models chapter.

Changing the Position of Points

You can change the position of any point (node) or control point within a selected vector object.

To change the exact position of a point or control point:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object in which you want to change the position of a point or control point. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Move the cursor over the point that you want to reposition. When the cursor changes to a , right-click to display the Vector Editing menu.
4. Click on the **Properties** option to display the **Point Properties** dialog box:



5. Define the point's new position using the **X coordinate** and **Y coordinate** boxes.
6. Click on the **OK** button to close the dialog box.

To change the approximate position of a point or control point:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object in which you want to change the position of a point or control point. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Click and drag on the point or control point into its new position. Release the mouse button to set its position.

Aligning Points

You can align a selection of points (nodes) within a selected vector object in either the horizontal (X) or vertical (Y) axis.

To align a selection of points:

1. Make sure that you are in Node Editing mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the points you want to align. A bounding box surrounds it, within which you can see the spans, points (nodes) and control points that make up the selected vector object.
3. Hold the **Shift** key down on your keyboard, and then select the points you want to align with that selected last. For details, see "Selecting Points and Control Points" on page 149.

4. Right-click on any of the selected points to display the Node Editing menu.
5. Click on the **Align Nodes** option, followed by either the **in X** or the **in Y** option, depending on the axis with which you want to align them. By default, the point selected first is aligned with the point selected last, and the intermediate points are replaced by a single horizontal (X) or vertical (Y) linear span.

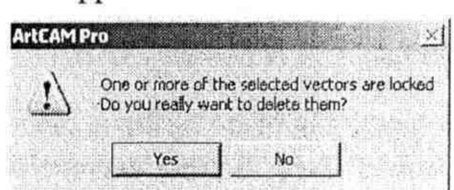
Note: If you want to preserve all of the intermediate points (nodes) when aligning them with the X or Y axis, click to deselect the **Align Nodes – replace with a single line** option on the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Deleting Vector Objects

You can delete a selected vector object or group of vector objects using either of the following methods:

- First, hold the **Shift** key down on your keyboard and then click to select the vector object that you want to delete. Next, right-click on any of the selected vector objects to display the Vector Editing menu, and then click on the **Delete** option.
- Select the vector objects that you want to delete, and then press the **Delete** key on your keyboard.

If you try to delete any locked vector objects, the following message box appears:



Click on the **Yes** button to close the message box and delete the selected vector objects. For further information, see "Locking and Unlocking Vector Objects" on page 172 and "Locking the Vectors on a Layer" in the Working with Models chapter.

Copying and Pasting Vector Objects

There are three buttons in the **File** area of the **Assistant**'s Home page that you can use to copy, paste and cut vector objects.

Using the **Block Copy / Rotate** button in the **Vector Editing** area, you can also create multiple copies of a vector object in two regular patterns:

- Block copy. For details, see "Block Copy" on page 157.
- Rotate copy. For details, see "Rotate Copy" on page 159.

To copy, paste or cut a vector object:

1. Select the vector object that you want to copy, paste or cut. For details, see "Selecting Vectors" on page 138.

   If you click and drag around a vector object and it has one or more overlaying copies, the selected vector object turns red. To move an overlaying copy, hold down the **Ctrl** key on your keyboard and then click and drag the copy into its new position.

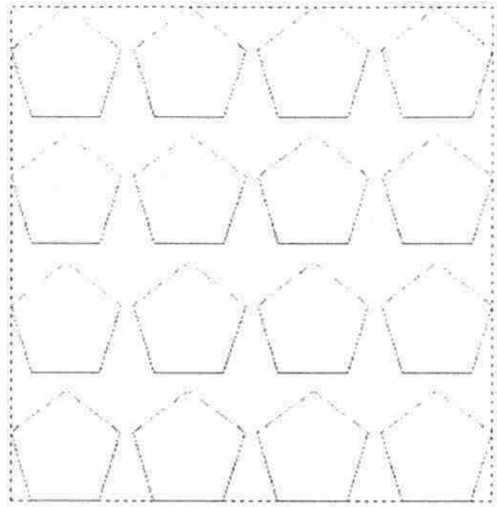
2. Use the appropriate button in the **File** area of the **Assistant**'s Home page:

   - Click on the **Cut** button to remove the selected vector object and place it on the clipboard.
   - Click on the **Copy** button to create a duplicate of the selected vector object and place it on the clipboard.
   - Click on the **Paste** button to copy the selected vector object back from the clipboard into its original position.

     If you want to move the pasted copy, move the cursor over the copied vector object. When the cursor changes to ✥, click and drag the copy into its new position.

Block Copy

You can produce multiple copies of a selected vector object in a grid format:



To make copies of a selected vector object in this way:

1. Select the vector object that you want to copy and paste. For details, see "Selecting Vectors" on page 138.
2. Click on the **Block Copy / Rotate** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Block and Rotate Copy** page.
3. Click on the **Block Copy** radio button to display the settings on the page.
4. Type the distance you want to set between each copy along the X-axis in the **X Offset** box.

   The values you type as X and Y offsets set the distance between the bottom left corner of each copy of the vector object.

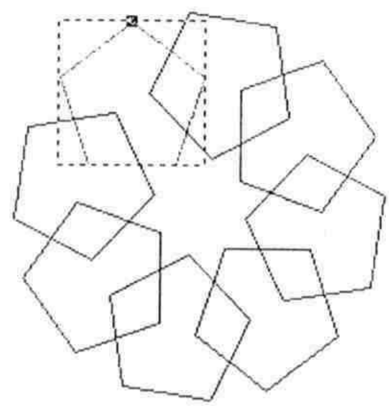
   You can set the distance you want between each copy using a positive or negative value:

   - To create a block of copies to the right of and above the selected vector object, type positive values in the offset boxes e.g. 90.
   - To create a block of copies to the left of and below of the selected vector object, type negative values in the offset boxes e.g. -90.
5. Type the distance you want to set between each copy along the Y-axis in the **Y Offset** box.

6. Define the number of copies by typing the number of rows and columns you want in the **Number of Rows** and **Number of Columns** boxes.
7. Click on the **Apply** button to create the new block of vector objects.
8. Click on the **Close** button to return to the **Assistant**'s Home page.

Rotate Copy

You can produce several copies of a selected vector object in a circular pattern by defining a rotation centre:



To make copies of a selected vector object in this way:

1. Select the vector object that you want to copy and paste. For details, see "Selecting Vectors" on page 138.
2. Click on the **Block Copy / Rotate** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Block and Rotate Copy** page.
3. Click on the **Rotate Copy** radio button to display the settings on the page.
4. Define the point around which you want to rotate the selected vector object. You can either:
   - Define the origin of rotation co-ordinates using the **Rotation Centre X** and **Rotation Centre Y** boxes.
   - Click to select the **Pick Centre with Mouse** option, move the cursor over the position in the model you want to use as the origin of rotation and then click to select.

5. Click on the appropriate **Angle** radio button :
   - **Incremental** - To rotate each subsequent copy of the selected vector object by a specified angle.
   - **Total** - To rotate each copy of the selected vector object evenly within a specified angle.
6. Define the angle of rotation in the **degrees** box:
   - Type a positive value to rotate the vector object clockwise.
   - Type a negative value to rotate the vector object anti-clockwise.
7. Type the number of copies you want to make in the **Number of Objects** box.
8. Click on the **Apply** button to create the circular pattern of copies.
9. Click on the **Close** button to return to the **Assistant**'s Home page.

Offsetting Vector Objects

You can create an offset vector from any vector object:

To create an offset vector from a vector object:

1. Select the vector object from which you want to create an offset vector. For details, see "Selecting Vectors" on page 138.
2. Click on the **Offset Vector(s)** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Offset Vector(s)** page.

Note: You can also display the **Offset Vector(s)** page from the Main menu bar. Click on the **Vectors** menu, followed by the **Offset...** option.

3. Set the distance between the selected vector object and the offset vector using the **Offset Distance** box.
4. Click on the appropriate **Offset Direction** radio button to set the position of the offset vector:
   - Select **Outwards/Right** if you want the offset vector drawn outside of the selected vector object if

it is closed, or to the right of the Start Point, looking down the selected vector object if it is open.

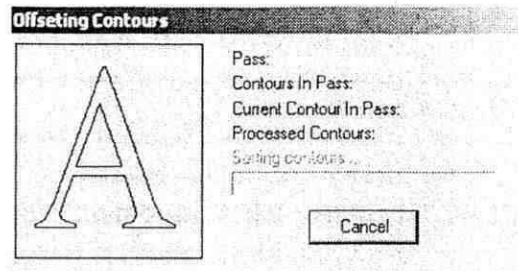
- Select **Inwards/Left** if you want the offset vector drawn within the selected vector object if it is closed, or to the left of the Start Point, looking down the selected vector object if it is open.
- Select **Both Sides (Ridge)** if you want two offset vectors drawn. One inside and another outside of the selected vector object if it is closed, or on either side of the Start Point, looking down the selected vector object if it is open.

5. Click on the appropriate **Offset Corners** radio button to set the shape of the corners in the offset vector:
   - Select **Radiused** if you want the corners of the offset vector to appear as arcs with a radius equal to the **Offset Distance**.
   - Select **Chamfered** if you want the corners of the offset vector to appear as chamfers.
   - Select **Sharp** if you want the corners of the offset vector to appear as sharp points.

     If you select the **Sharp** option, set the maximum distance a sharp corner can be offset before a chamfer is applied to it using the **Max. Sharp Offset Distance** box .

     The value you use should represent a percentage of the **Offset Distance**. For example, if the **Offset Distance** is 4 mm and the **Max. Sharp Offset Distance** is set to 50%, a chamfer is applied to the offset corners at 6 mm (4 + 50% of 4 = 6).
6. If you want to replace the original vector object, click to select the **Delete original vectors** option .
7. Click on the **Offset** button to draw the offset vector.

   The **Offsetting Contours** message box is displayed while ArtCAM Pro calculates the position of the offset vector:



Note: You may not see the **Offsetting Contours** message box if you are creating an offset vector from a simple shape of vector object. For example, a circle.

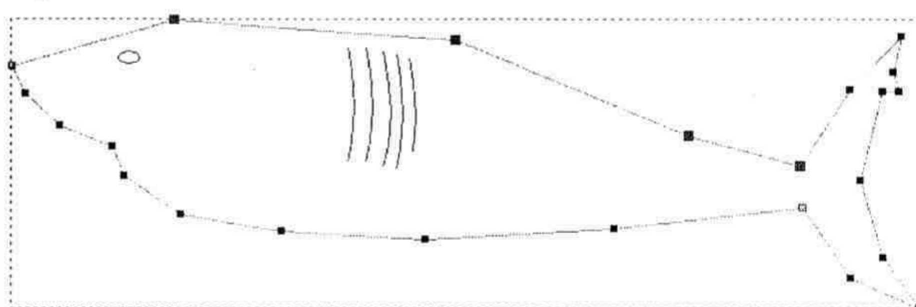
8. Click on the **Close** button to return to the **Assistant**'s Home page.

Splining Vector Objects

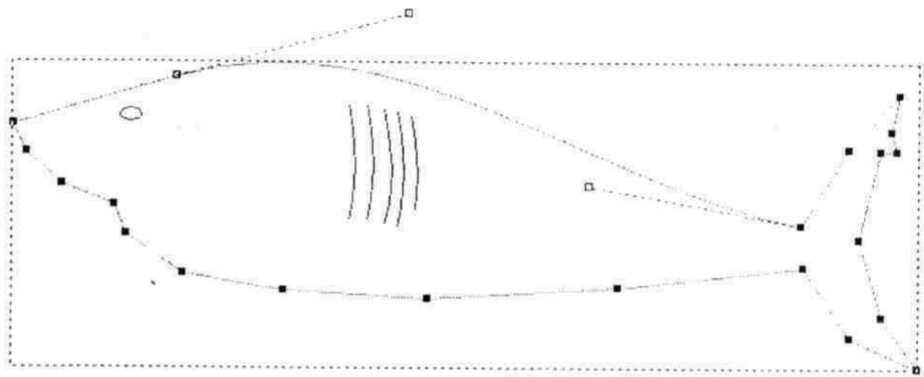
You can take a selection of points (nodes) within a selected vector object and convert their adjoining linear or arc spans into a single bezier curve span. This process is referred to as 'splining'. For further information on spans, see "Editing Vector Spans" on page 141. The vector object must be ungrouped.

In the following example, you can see vector artwork in the shape of a shark made up entirely of linear spans. When the linear spans between a selection of four points (nodes) along the top of the vector artwork are splined, you can see the its shape changes considerably:

Before...



After...



To spline a vector object or selection of points (nodes) within a vector object:

1. Make sure that you are in **Node Editing** mode . For details, see "Editing Vector Nodes" on page 148.
2. Click to select the ungrouped vector object that you want to spline. A bounding box surrounds it, within which you can see the spans, points and control points that make up the selected vector object.
3. Click on the **Spline Vectors** button  in the **Vector Editing** area of the **Assistant**'s Home page to display the **Smooth Vectors** page.

Note: You can also display the **Spline Vectors** page from the Main menu bar. Click on the **Vectors** menu, followed by the **Spline Vectors...** option.

4. If you only want to spline specific spans within the selected vector object, select the points and/or control points to which these spans are attached. For details, see "Selecting Points and Control Points" on page 149.

   In the **Selected Vectors Information** area, ArtCAM Pro displays the total number of spans and points (including control points) within your selection, and the number of linear, arc and bezier spans that make up the total.
5. If you only want convert linear spans to bezier curve spans that are within a specific tolerance, type the maximum distance the point (node) can be from the selected span in the **Tolerance** box.

6. If you want to keep all linear spans within the current selection equal to or greater than a specific length, click on the **Preserve Straight Spans** option and then define the length in the **Keep lines longer than** box.

   Otherwise, leave the default **Ignore Straight Spans** option selected to replace all of the linear and arc spans within the current selection with bezier curve spans.

7. Click on the **Spline** button to spline all selected spans.

   The linear and/or arc spans joined to the selected points are converted to bezier spans. ArtCAM Pro displays the new total number of spans and points (including control points) within the selection, and the new number of linear, arc and bezier spans that make up the total.

8. Click on the **Close** button to return to the **Assistant**'s Home page.

You can also display the **Spline Vectors** page in the following way:

1. Make sure that you are in Node Editing mode .For details, see "Editing Vector Nodes" on page 148.
2. Click to select the vector object containing the points (nodes) you want to spline. A bounding box surrounds it, within which you can see the spans, points and control points that make up the selected vector object.
3. Select the points and/or control points to which the spans in the vector object you want to spline are attached. For details, see.
4. Right-click on any of the selected points to display the Node Editing menu, and then click on the **Spline Vectors...** option to display the **Spline Vectors** page.

Restoring the Original Shape of a Splined Vector

You can restore the original shape of a splined vector object using either of the following methods:

- From the Main menu bar, click on the **Edit** menu, followed by the **Undo Spline Vectors** option.
- Press the **Ctrl** + **Z** keys on your keyboard.

Filleting Vector Objects

A fillet is an arc, which is made up of two points (nodes) and a control point. What distinguishes a fillet from an arc in ArtCAM Pro, is that you can define the radius of a fillet.

You can close an open vector object with a fillet, or by joining the two points at the junction. In the latter instance, ArtCAM Pro extends the spans connected to the two points.

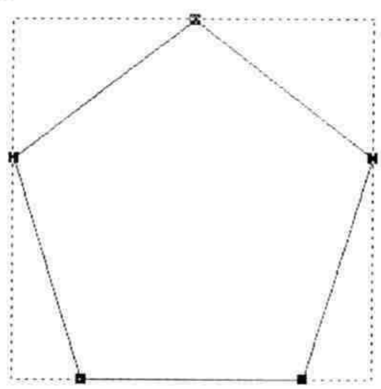
Note: Not all open vector objects can be closed with a fillet. For practical results, a vector object should contain at least three points (nodes).

You can now also convert a sharp corner within a vector object into a filleted, or rounded, corner.

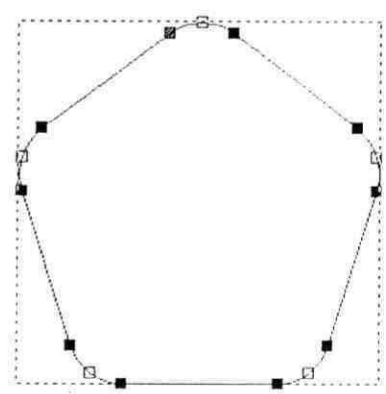
To convert a sharp corner into a filleted corner:

1. Select the vector object in which you want to insert filleted corners. For details, see “Selecting Vectors” on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.
3. Click on the **Insert Fillet** radio button to select this option.
4. Type the radius of the filleted corner you want to insert in the **Fillet Radius** box.
5. Move the cursor over the point (node) in the selected vector object, and then click to insert the fillet. The point is converted into two points and a control point.

Before...



After...

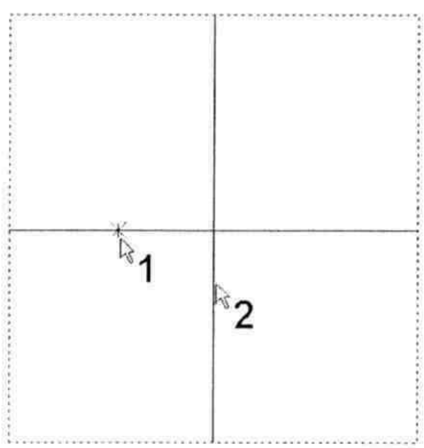


6. Click on the **Close** button to return to the **Assistant**’s Home page.

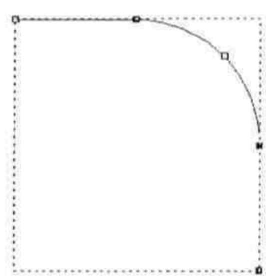
You can also convert a sharp corner into a filleted corner if you:

1. Select the vector object in which you want to insert filleted corners. For details, see "Selecting Vectors" on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.
3. Click on the **Insert Fillet** radio button to select this option.
4. Type the radius of the filleted corner you want to insert in the **Fillet Radius** box.
5. Click on a span on either side of a point (node) or intersection in the selected vector. The ✳ icon appears on the selected span.
6. Click on the span opposite to that already selected to insert the fillet. The point or intersection is converted into two points and a control point.

Before...



After...



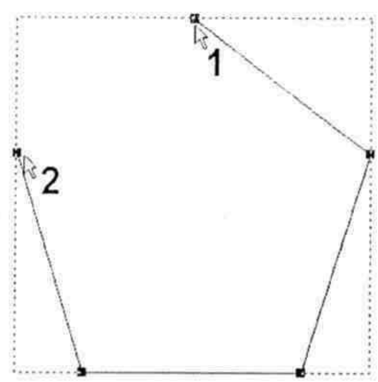
7. Click on the **Close** button to return to the **Assistant**'s Home page.

To close an open vector object with extended spans:

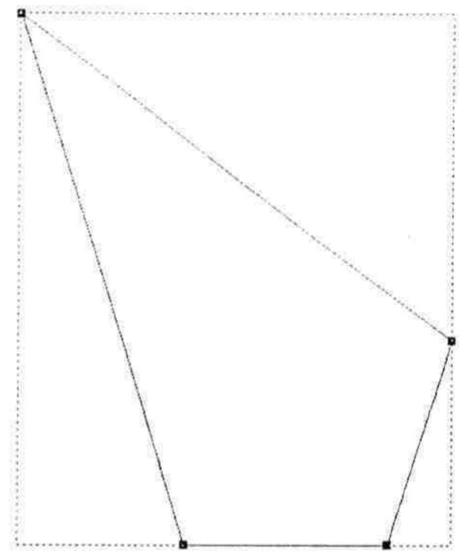
1. Select the open vector object that you want to close. For details, see "Selecting Vectors" on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.
3. Click on the **Extend/Trim Line to Intersection** radio button to select this option.

4. Click to select the first point (node) in the junction of the open vector object. The ○ icon appears around the selected point.
5. Click to select the second point in the junction. ArtCAM Pro joins the two points, extending their connecting spans whilst doing so.

Before...



After...

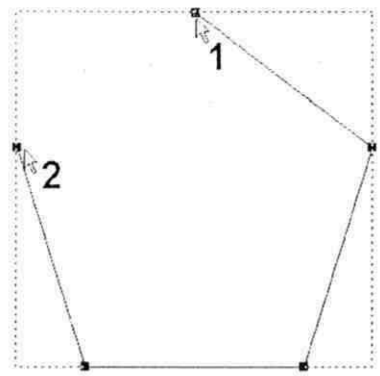


6. Click on the **Close** button to return to the **Assistant**'s Home page.

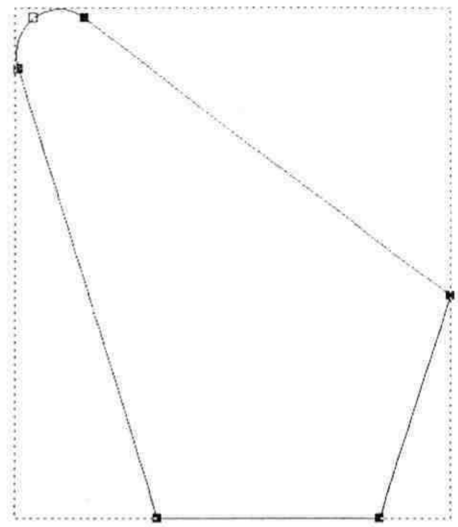
To close an open vector object with a fillet:

1. Select the open vector object that you want to close. For details, see "Selecting Vectors" on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.
3. Click on the **Insert Fillet** radio button to select this option.
4. Type the radius of the fillet with which you want to close the vector object in the **Fillet Radius** box. The wider the radius, the shorter the extended spans.
5. Click to select the first point in the junction of the open vector object. The ○ icon appears around the selected point.
6. Click to select the second point in the junction. ArtCAM Pro joins the two points with a fillet, extending their adjoining spans whilst doing so.

Before...



After...



7. Click on the **Close** button to return to the **Assistant**'s Home page.

Editing a Fillet

You can alter the radius of a fillet after it has been created:

1. Select the vector object containing the fillet that you want to edit. For details, see "Selecting Vectors" on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.
3. Click on the **Insert Fillet** radio button to select this option.
4. Type the new radius of the fillet in the **Fillet Radius** box.
5. Double-click on the control point between the two points in the fillet to alter its radius.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Editing a Fillet Arc

You can convert an arc to a fillet:

1. Select the vector object containing the fillet that you want to edit. For details, see "Selecting Vectors" on page 138.
2. Click on the **Fillet Tool** button in the **Vector Editing** area to display the **Fillet Tool** page.

3. Click on the **Insert Fillet** radio button to select this option.
4. Type the new radius of the fillet in the **Fillet Radius** box.
5. Click on the control point between the two points in the fillet to alter its radius.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Alternatively, you can click and drag the control point to alter the shape of the arc.

Trimming Vector Objects

You can control exactly which part of any number of overlapping vector objects you want to remove.

To trim two or more ungrouped overlapping vectors:

1. Make sure that you are in **Node Editing** mode. For details, see
2. Hold the **Shift** key down on your keyboard, and then click to select the ungrouped overlapping vector objects that you want to trim.
3. Click on the **Trim Tool** button in the **Vector Editing** area to display the **Trim Tool** page.
4. If you want to create a copy of the original vector object you are trimming, click on the **Keep Original** option to select it. The copy is overlaying the original vector object.
5. Move the cursor over a span within the area in which the vector objects overlap. When the cursor changes to , click to delete the span.

To trim two or more grouped overlapping vectors:

1. Make sure that you are in **Node Editing** mode. For details, see "Editing Vector Nodes" on page 148.
2. Hold the **Shift** key down on your keyboard, and then click to select the grouped overlapping vector objects that you want to trim.
3. Click on the **Trim Tool** button in the **Vector Editing** area to display the **Trim Tool** page.

4. Click on the **Explode Selection** button. ArtCAM Pro ungroups the vector objects and then converts the spans between every two points (nodes) within them into individual vector objects.
5. Move the cursor over a span that you want to remove. When the cursor changes to , click to delete.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Wrapping Vectors to a Relief

You can wrap vector objects across the surface of a relief. After being wrapped, the selected vector object is restructured in the **2D View** window. You can apply a three-dimensional shape to the restructured vector object, and then combine this with the existing relief. This allows you to maintain the surface area of the original vector object in the relief.

To wrap a vector object to a relief:

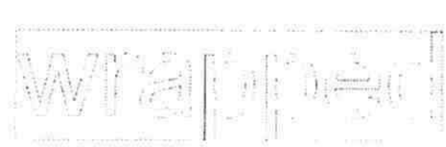
1. Select the vector object that you want to wrap across the relief surface. For details, see "Selecting Vectors" on page 138.
2. Click on the **Wrap Vectors To Relief** button in the **Vector Editing** area to display the **Wrap Vectors To Relief** page.
3. In the **Wrap Mode** area, click to select the wrapping method that you want to use:
   - **Radial** – This option allows you to wrap the clipart relief around a spherical shape, curved around the X and Y-axes.
   - **Cylindrical Wrap in X** – This option allows you to wrap the clipart relief around a cylindrical shape that is curved around the X-axis.
   - **Cylindrical Wrap in Y** – This option allows you to wrap the clipart relief around a cylindrical shape that is curved around the Y-axis.
4. In the **Wrap Origin** area, define the origin around which the selected vector object will be wrapped. You can use any of the following methods:

- Click on any of radio buttons on the rectangle diagram. The X and Y co-ordinates of the chosen origin appear in the **X** and **Y** boxes.
- Type the X and Y co-ordinates of the origin in the **X** and **Y** boxes.
- Click on the **Use Mouse** option, followed by the **Start** button. Move the cursor to the position in the **2D View** window that you want to use as the origin, and then click. The X and Y co-ordinates of the origin appear in the **X** and **Y** boxes.

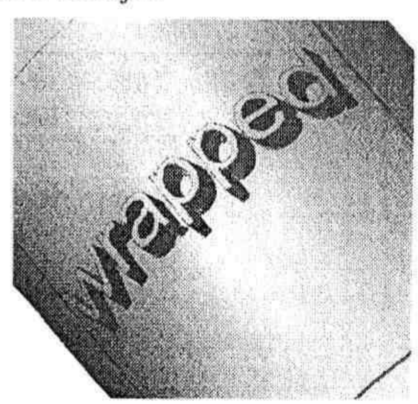
5. If you want to keep a copy of the selected vector object in its original form, make sure that the **Keep Original Vectors** option is checked on.
6. Click on the **Wrap** button to wrap the selected vector across the relief surface. ArtCAM Pro restructures the vector object in the **2D View** window.

   For example, the vector text *wrapped* is wrapped around the X-axis of a bottle-shaped relief. ArtCAM Pro restructures the vector text. A plane with a start height of 1 mm is applied to the wrapped text and added to the relief. The surface area of the vector text is maintained, unlike when it has been projected, as shown below:

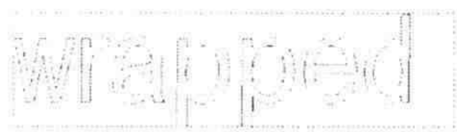
Projected Vector Text...



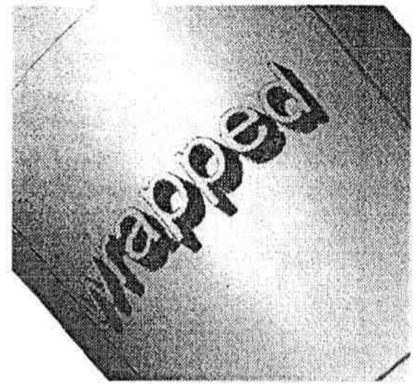
Projected Vector Text Combined with Relief...



Vector Text Wrapped in X...



Wrapped Vector Text Combined with Relief...



7. Click on the **Close** button to return to the **Assistant**'s Home page.

Locking and Unlocking Vector Objects

You can prevent a vector object from being accidentally moved to a different position by locking it.

To lock a vector object into position:

1. Select the vector object that you want to lock into position. For details, see "Selecting Vectors" on page 138.
2. Right-click to display the Vector Editing menu, and then click on the **Lock Vector(s)** option. The selected vector object turns green. When deselected, the vector object is shown in grey.

Note: You can also lock and unlock a selected vector object using the **Layers** page. For details, see

---

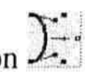
If you want to move any vector object that has previously been locked into position:

1. Select the vector object that you want to move. For details, see "Selecting Vectors" on page 138.
2. Right-click to display the Vector Editing menu, and then click on the **Unlock Vector(s)** option. The selected vector object turns magenta. When deselected, the vector object is shown in the colour assigned to the layer on which it is drawn.

Fitting Arcs to Vector Objects

You can replace all of a selected vector object's bezier curve spans with a series of arc spans whilst maintaining its original shape.

To fit arc spans to a selected vector object:

1. Select the vector object whose bezier curve spans you want to convert to arc spans. For details, see "Selecting Vectors" on page 138.
2. Click on the **Arc Fit Vectors** button  in the **Vector Editing** area of the **Assistant**'s Home page to display the **Fit Arcs To Vectors** page.

Note: You can also display the **Fit Arcs To Vectors** page from the Main menu bar. Click on the **Vectors** menu, followed by the **Fit Arcs To Vector(s)...** option.

In the **Selected Vectors Information** area, ArtCAM Pro displays the total number of spans and points (including control points) within the selected vector object, and the number of linear, arc and bezier spans that make up the total.

Note: The points and control points in a selected vector object are grouped together under **Points** in the **Selected Vectors Information** area of the page.

3. Define how closely you want the arc spans to follow the original shape of the bezier curve spans using the **Tolerance** box.
4. Click on the **Fit Arcs** button to convert all bezier curve spans to arc spans.

   ArtCAM Pro displays the new total number of spans and points (including control points) within the selection, and the new number of linear, arc and bezier spans that make up the total.

Note: If a bezier curve span in the selected vector object has no curvature, it is automatically converted to a linear span.

5. Click on the **Close** button to return to the **Assistant**'s Home page.

Pasting Vectors Along a Curve

You can paste copies of a selected vector object along another curved vector object.

To paste a selected vector object along another curved vector object:

1. Hold the **Shift** key down on your keyboard, click to select the vector object that you want to copy, followed by the curved vector object that you want to paste copies along. For details, see "Selecting Vectors" on page 138.
2. Click on the **Paste Along A Curve** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Paste Along A Curve** page.

Note: You can also display the **Paste Along A Curve** page from the Main menu bar. Click on the **Vectors** menu, followed by the **Paste Vector(s) on Curve...** option.

3. Select the pasting method that you want to use:
   - Click on the **Specify Number** radio button, then define the number of copies using the **Number of Copies** box.
   - Click on the **Specify Distance** radio button, then define the distance between each copy using the **Distance Between Copies** box.
4. If you want to make the spacing between the copies even, click to select the **Make Spacing Even** option.
5. Click on the **Paste** button to produce the copies along the selected curved vector object.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Working with Vector Text

You can create and manipulate vector text using three tools in ArtCAM Pro. These are:

- The **Font Editor** icon on the **Assistant**'s **Getting Started** page. For details, see "Creating Fonts" on page 175.

- The **Create Vector Text** button in the Vector Editing area of the **Assistant**'s Home page. For details, see "Creating Vector Text" on page 181.
- The **Wrap text round a Curve** button in the **Position Size Align Vectors** area of the **Assistant**'s Home page. For details, see "Wrapping Text Round a Curve" on page 186.

Creating Fonts

You can create your own fonts by editing characters within any installed fonts and then saving them. Using the **Font Editor** tool, you can:

- Adjust the contour of characters within an existing font.
- Adjust the offset distance and the ascent applied to all characters within an existing font.
- Set the order in which the vector objects that make up a character will be machined.
- Adjust the kerning (white space) between pairs of characters within an existing font.
- Save your customised font and then type in it using the **Text Tool.** For further details, see

Note: All customised fonts are listed with an '(AFN)' prefix in the **Font** list box on the **Text Tool** page.

To create a customised font:

1. Click on the **Font Editor** icon on the **Assistant**'s **Getting Started** page to display its settings.
2. Select the font that you want to edit. This is referred to as the base font.

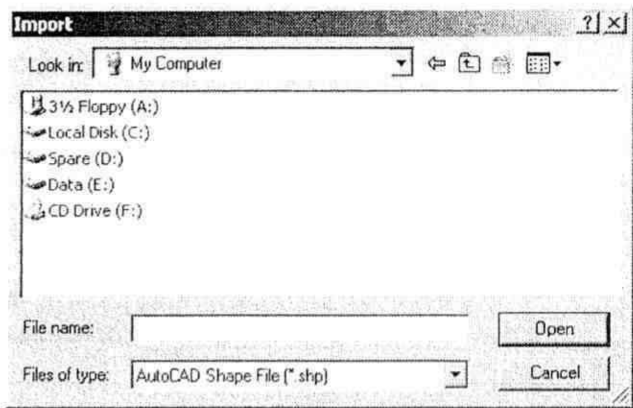
   If you want to edit a TrueType font:

   - First, click on the **Base Font** list box, and then on the font that you want to edit.
   - Next, click on the **Script** list box below, and then on the script option that you want to use for the font.

- Finally, click on the **Create** button.

If you want to edit an AutoCAD shape file (***.shp**):

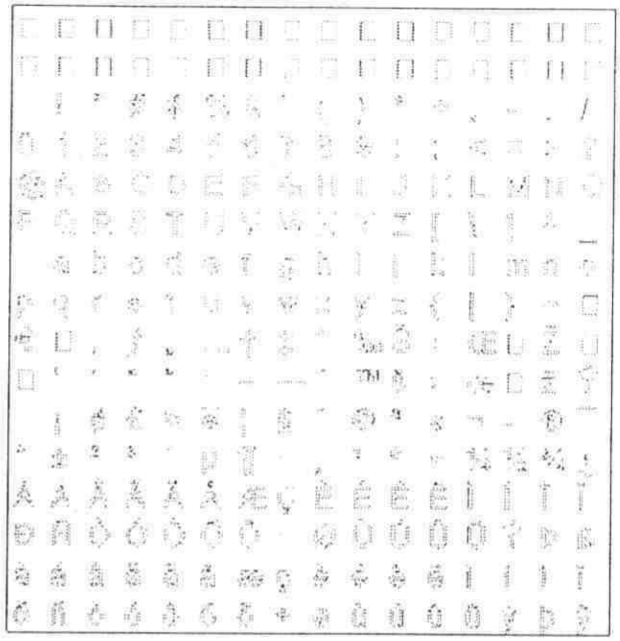
- First, click on the **Import** button to display the **Import** dialog box:



- Next, click on the **Files of type** list box, and then on the file type of the AutoCAD shape you want to open.
- Next, click on the **Look in** list box and select the directory where the AutoCAD shape file that you want to open is stored. Once you have found the file, click on its name.
- Finally, click on the **Open** button.

A new ArtCAM model is created and the **Font Creator** page is displayed. All of the characters within the selected font appear as vector objects in the **2D View** window.

For example, the 255 characters that make up the Arial Western font appear as shown below:



3. If you cannot see the character that you want to edit, click on the **View Page** list box, and then on the next page number. Repeat this step until you have found the character.

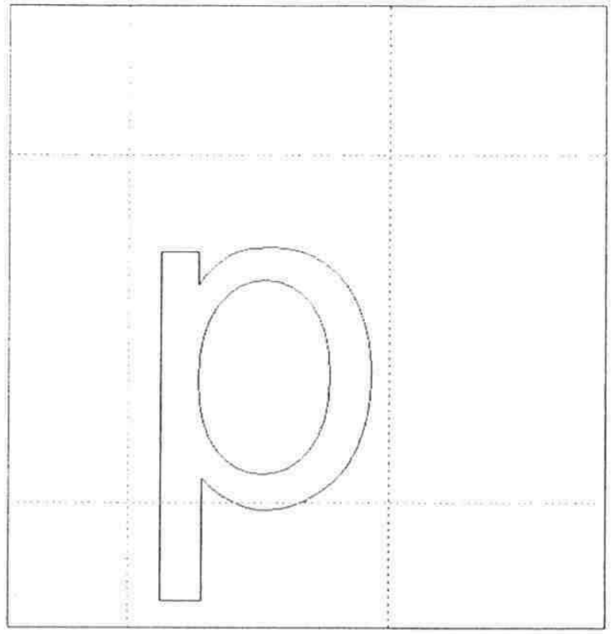
   For Unicode fonts, each page contains 256 characters.

4. Select the character that you want to edit. You can either:

   - Type the character or its code number in the **Choose Character** box. The character turns magenta, indicating that it is selected and that the vector objects within it are grouped.
   - Click on the character itself in the **2D View** window. The character turns magenta, indicating that it is selected and that the vector objects within it are grouped. Its code number is identified in the **Choose Character** box.

5. Click on the **Edit Character** button. Only the selected character is shown in the model area (the white area) of the **2D View** window. The vector objects that make up the character are now ungrouped. Red, purple and green guidelines are shown by default.

   In our example, the lower-case character 'p' is selected, as shown below:



6. Click on the **Assistant** tab and use the available tools to edit the selected character. A character can be edited in the same way as any other vector object in ArtCAM Pro. For details, see
7. Click on the **Font Creator** tab to display its settings. You can now adjust the offset and the ascent for the font using the currently selected character:
    - To adjust the offset distance between the current character and the next in the font, click and drag the red guideline into position.
    - To adjust the ascent of the font to which the current character belongs, click and drag the purple guideline at the top and the green guideline at the bottom of the font into position.
    - To restore the default position of all guidelines, click on the **Reset Guides** button.

Tip: Make sure that the purple guideline is aligned with the top edge of the capital 'H' or 'M' character in the font. The green guidelines should be aligned with the left and bottom edges of the capital character.

---

8. If you want to set the order in which the character's vector objects will be machined, click on the **Set Vector Order** button to display the settings in the **Assistant** window.

Each of the character's vector objects is numbered in the **2D View** window, showing the total number of vector objects and the current order in which they will be machined. You can either:

- Move the cursor over the vector, and then click on each of the vector objects that make up the character in turn.

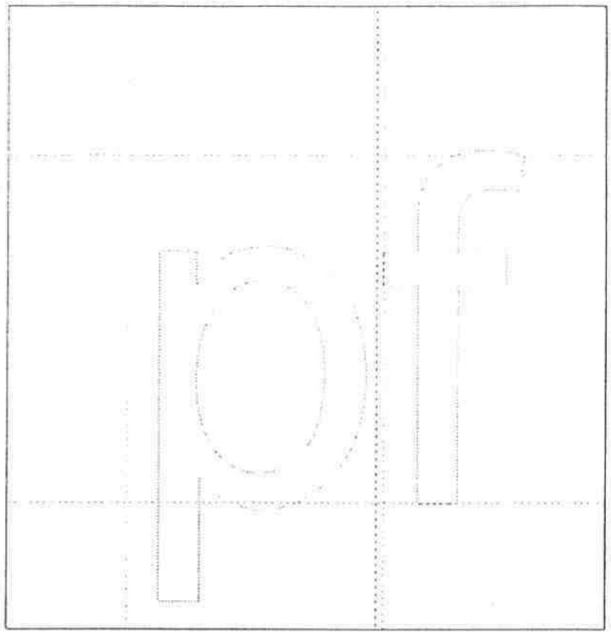
- Click on the **Swap Order** button. Move the cursor over the vector object whose position you want to alter, and then click. Its current position in the machining order is displayed on the mouse cursor, replacing its left question mark. Click on the vector object whose position you want to exchange with that which is already selected. The selected vector objects are renumbered according to their new positions in the machining order.

9. If you want to adjust the kerning between the selected character and another within the font:

Note: A list of characters that have been kerned by default along with the selected character are shown in the **Currently Kerned Characters** area of the page. You can click on any of the listed characters to display them along with the selected character in the **2D View** window.

- First, type the new character or its code number in the **Edit Kerning for Character** box, and then click on the **Edit Kerning** button. The new character appears alongside the previously selected character in the **2D View** window.

  In our example, the lower-case characters 'p' and 'f ' appear as follows when selected:

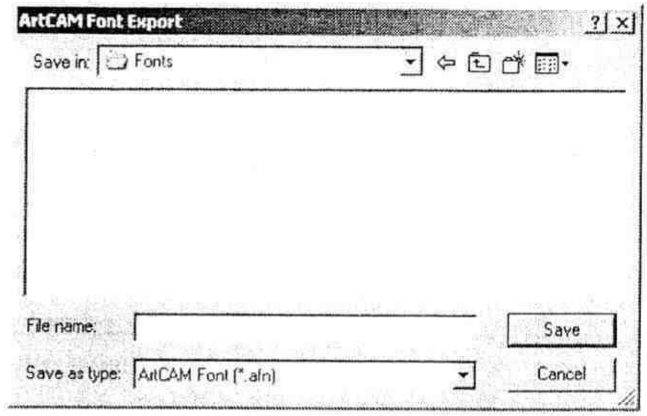


- Next, click and drag the blue guideline to set the distance between the pair of characters. Drag to the left to move the characters closer together, and to the right to move them further apart.

Note: This action only adjusts the kerning for the currently selected pair of characters in the font.

- Finally, click on the **Done** button to return to the **Edit Character** settings.

10. Click on the **Done** button. The **Choose Character** options are displayed on the page and all of the characters within the selected font are shown again in the **2D View** window.
11. Repeat these steps until you have finished editing characters in the font. When you have finished, click on the **Save Font** button to display the **ArtCAM Font Export** dialog box:



The ArtCAM Pro 6.0\Fonts folder is selected by default.

12. Type a name for the font in the **File Name** box, and then click on the **Save** button to close the **ArtCAM Font Export** dialog box and save the font (***.afn**).
13. Click on the **Quit** button to return to the **Assistant**'s **Getting Started** page.

Note: If you click on the **Quit** button before saving the font, a message box appears warning that the font has been modified. If you want to save the font, click on the **Yes** button. If you do not, click on the **No** button.

Creating Vector Text

You can create vector text with ease. Creating vector text in ArtCAM Pro is similar to typing text in most word processing packages. You can type in glyph (closed vector) or single-stroke (open vector) fonts.

To create vector text:

1. Click on the **Create Vector Text** button in the **Vector Creation** area of the **Assistant**'s Home page to display the **Text Tool** page.
2. Select the formatting for the text using the options on the **Text Tool** page. For details on the different options available, see "Formatting Vector Text" on page 184.
3. Move the I cursor over to the **2D View** window and click where in the model (the white area) you want to create the vector text. A text box appears in the **2D View** window and the mouse cursor changes to +.

4. Type the vector text you want to create. It is useful to remember the following keystrokes when typing in vector text:

   - Press the **Enter** key on your keyboard to start a new line of vector text.
   - Press the **Backspace** key on your keyboard to delete the last character typed.

5. Click on the **Done** button or press both the **Ctrl** + **Enter** keys to create the vector text and return to the **Assistant**'s Home page.

   By default, the block of vector text is purple, indicating that the vector objects that make up the text are grouped. A bounding box also surrounds the block:

Note: You can move the block of vector text if you position the cursor over it, and then click and drag it into its new position.

Selecting Vector Text

You can select vector text whilst the **Text Tool** page is displayed using either of the following methods:

- Click and drag over the character or block of vector text that you want to select.
- Hold the **Shift** key down on your keyboard, and then use the arrow keys on your keyboard to control your selection.

  The selected vector text is white and is highlighted in black:

Note: If you have selected a block of vector text containing more than one style of font, script and/or size, the relative list boxes in the **Style** area are blank.

When you have created the vector text, you can select it using the method described in "Selecting Vectors" on page 138.

Editing Vector Text

Text editing refers to making changes in vector text, such as adding and deleting, or copying and pasting individual characters or blocks of vector text.

You can cut, copy and paste a selected character or block of vector text when the **Text Tool** page is displayed using the editing buttons in the **Style** area:

- Click on the **Cut** button to remove the selected vector text and place it on the clipboard.
- Click on the **Copy** button to create a duplicate of the selected vector text and place it on the clipboard.
- Click on the **Paste** button to place a copy of the vector text, overlaying the currently selected text.

You can use either of the following methods to edit a selected character or block of vector text after it has been created:

- Right-click on the block of vector text to display the Vector Editing menu, and then click on the **Edit Text Block** option to display the **Text Tool** page.
- Move the cursor over the block of vector text, and then press the **E** key on your keyboard to display the **Text Tool** page.

You can also cut, copy and/or paste a block of vector text after it has been created using the editing buttons in the **File** area of the **Assistant**'s Home page. For details, see "Copying and Pasting Vector Objects" on page 157.

You can add or delete characters to a block of vector text when the **Text Tool** page is displayed using your keyboard:

- Use the character keys on your keyboard to add characters to the block of vector text.
- Press the **Backspace** key on your keyboard to delete the last character typed.

You can delete a selected block of vector text when the **Text Tool** page is not displayed using either of the following methods:

- Move the cursor over the block of vector text, and then press the **Delete** key on your keyboard.
- Right-click on the selected block of vector text to display the Vector Editing menu, and then click on the **Delete** option.

Formatting Vector Text

Text formatting refers to setting the way vector text looks using different fonts, bold or italic type, indents and alignments.

You can format a character or block of vector text both during and after its creation using the formatting buttons and the list boxes in the **Style** area of the **Text Tool** page.

You can use the formatting buttons as follows:

- Click on the **Bold** button to make the selected vector text bold, or to remove the bold formatting.
- Click on the **Italic** button to make the selected vector text italic, or to remove the italic formatting.
- Click on the **Align Left** button to align the selected vector text to the left of the text box with a ragged right edge.
- Click on the **Align Right** button to align the selected vector text to the right of the text box with a ragged left edge.
- Click on the **Centred** button to place the selected vector text in the centre of the text box.

To change the font used when typing:

- Click on the **Font** list box, and then on the font required.

  You can type in glyph (closed-vector) or single-stoke fonts.

Note: You can type in glyph (closed-vector) or single-stoke (engraving) fonts. All single-stroke fonts and fonts created using the **Font Editor** are listed with an '(AFN)' prefix in the **Font** list box.

To change the size of the font you are using:

- Define the size of the font using the **Size** box.

To adjust the angle of a character or block of vector text:

- Define the angle at which you want to position the vector text using the **Angle** box.

  For example, setting the angle at 45° places each character in a block of vector text as follows:

You can use either of the following methods to adjust the amount of white space on either side of a character in a block of vector text:

- Define the distance you want to place between each character using the **Character** box.
- Click and drag on the **% of space** slider to adjust the value in the **Character** box.

To adjust amount of white space between lines in a block of vector text:

- Define the distance you want to place between each line using the **Line** box.

Kerning, like **Spacing**, applies white space on either side of a character in a block of vector text. Kerning, however, considers the shape of the previous character in relation to the character that is being typed, and adjusts the distance between them so that their fit looks good.

To adjust the kerning between all the characters in a block of vector text:

1. In the **Kerning** area, click on the **Mode** list box:
   - If you want ArtCAM Pro to automatically apply kerning to newly created or edited text, click on the **Auto** option.
   - If you want to manually apply kerning to the selected block of vector text, click on the **Manual** option.
2. Define the amount of kerning that you want to apply to the selected block of vector text. To do so, you can either:
   - Define the amount of kerning using in the **Kerning** box.
   - Click and drag on the **% of space** slider to adjust the value in the **Kerning** box..

3. If you want to apply ArtCAM Pro's default kerning settings to the selected block of vector text, click on the **Default Kerning** button.

Wrapping Text Round a Curve

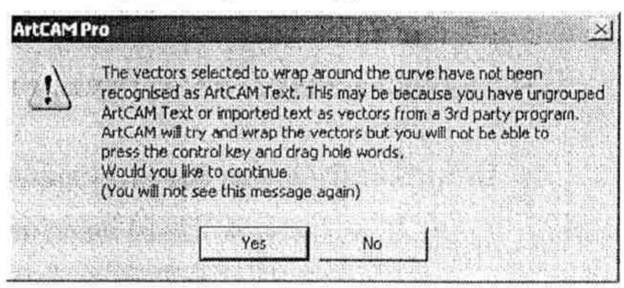
ArtCAM Pro allows you to place vector text on or around any open or closed curved vector object.

Before you can wrap a block of vector text around a curve, you must draw both the curved vector object and some vector text.

To wrap vector text around a curved vector:

1. Click to select the curved vector object. For details, see "Selecting Vectors" on page 138.
2. Hold the **Shift** key down on your keyboard, and then click to select the block of vector text you want to wrap. A bounding box surrounds the selected vector objects.
3. Click on the **Wrap Text Round a Curve** button in the **Position Size Align Vectors** area to display the **Text on a Curve** page.

   If you are attempting to wrap imported vector text created in another package, or vector text that is ungrouped, the following message box appears:



   If you want ArtCAM Pro to attempt to wrap the vector text, click on the **Yes** button. The message box closes.

Warning: Although you can wrap imported vector text created in another package, or an ungrouped block of vector text created in ArtCAM, around a curved vector object, you cannot reposition whole words using the click and drag method detailed in this section. You can drag individual letters into position, and use the **Text Settings** options on the **Text on a Curve** page, but the results may not always be accurate and unwanted copies of the text may remain in your model.

If you want to abort the wrapping of the text around the curve, click on the **No** button. The message box closes.

If the vector text was created in ArtCAM Pro, you can now wrap the vector text around the curved vector object.

4. Click on the **Select** button to wrap the vector text around the selected vector object. You can reposition the block of vector text using either of the following methods:
    - Click and drag the vector text along the curve.
    - Use the **Text Settings** on the **Text on a Curve** page. For details, see "Using the Text Settings" on page 187.

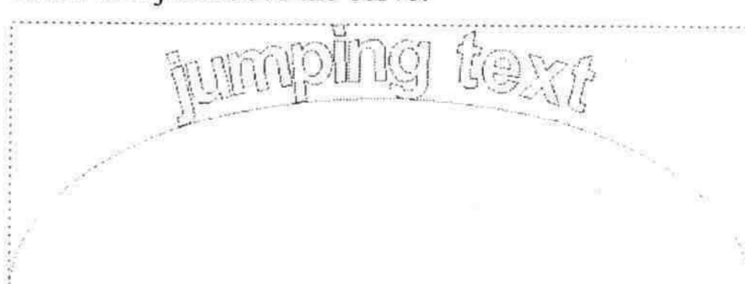
Using the Text Settings

You can select the text's position, alignment and spacing using the options in the **Text Settings** area of the **Text on a Curve** page.

Text Position

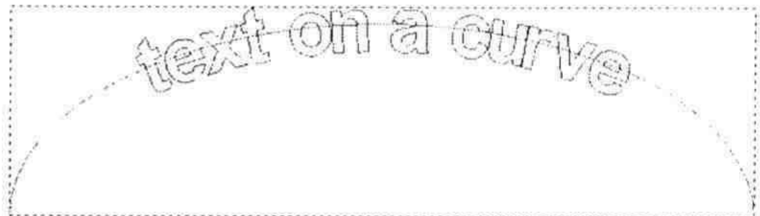
You can position the vector text on a curve in several ways using the **Text Position** options as follows:

- **Above Line** - Click on this radio button to position the vector text just above the curve:

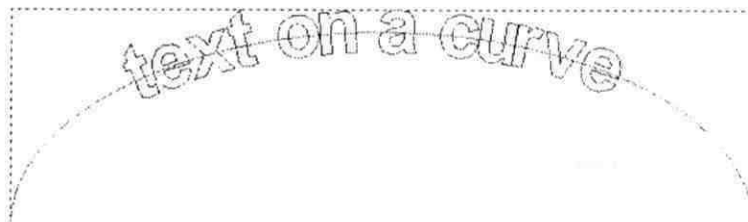


- **Base Line** - Click on this radio button to position the base of the vector text directly on the curve:

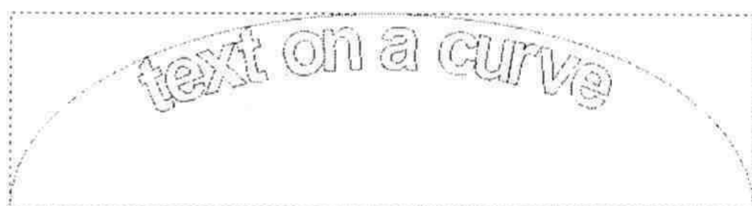
- **Centre On Ascent** - Click on this radio button to position the ascending part of the characters in the vector text on the curve:



- **Centre On Strikeout** - Click on this radio button to position the vector text so that the curve is used as strikeout through the middle of the vector text:



- **Below Line** - Click on this radio button to position the vector text below the curve:



- **Specify** - Click on this radio button , and then set the distance of the vector text from the curve using the adjacent box. To view the vector text at the specified distance, click on the **Update** button.

  For example, typing a value of -10 mm in the box produces the following result:

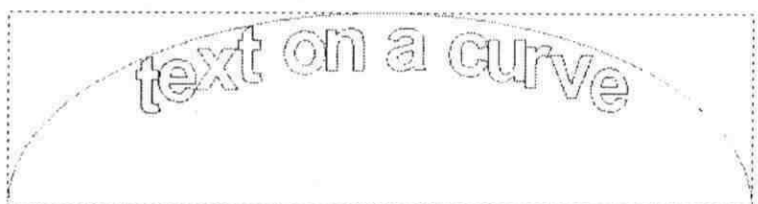
To reverse the direction of the curved vector object, click to select the **Text on other side** option.

Text on other side *Off...*	***Text on other side*** *On...*
	

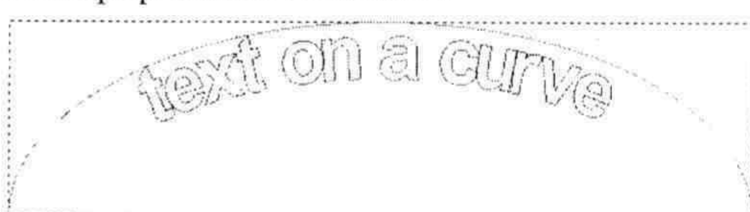
Text Alignment

You can align the vector text on a curve using either of the **Text Alignment** options as follows:

- **Vertical** - Click on this radio button to align the letters vertical to the curve:



- **Align To Curve** - Click on this radio button to align the letters perpendicular to the curve:



Text Spacing

You can set the distance between characters within a block of vector text using the **Text Spacing** slider.

Click the slider and drag to set the spacing. Drag to the right if you want to increase the spacing. Drag to the left to reduce it.

When you are satisfied with the spacing between the characters in the block of vector text, click on the **OK** button.

You can select what vector text you want to edit using the **Editing Style** options:

- **Whole Sentence** - Click on the radio button to move all text into position on the curve.
- **Single Words** - Click on the radio button to move a specific words into position on the curve.

Note: Alternatively, you can hold down the **Ctrl** key and then click and drag a specific word into place.

- **Single Letters** - Click on the radio button to move individual characters into position on the curve.

Note: Alternatively, you can hold down the **Alt** key and then click and drag a specific character into place.

Measuring Vector Objects

You can find out the size, position and angle of any vector object as follows:

1. Click on the **Measure Tool** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Measure** page.
2. Click on the position in the model that you want to measure from. This position is known as the **Anchor Point**.
3. Drag the mouse to the position you want to measure to and then click.

   Both the **Distance** and **Angle** between the current and anchor points are calculated and displayed on the **Measure** page.

ArtCAM Pro calculates the measurement details in real time and displays them on the **Measure** page as you drag the cursor into position.

By default, the cursor snaps to specific positions on the vector objects or any guidelines that have been drawn. For further information, see "Snapping to Objects" in the ArtCAM Pro Layout chapter.

Note: To temporarily disable snapping, hold down the **Shift** key on your keyboard whilst dragging the cursor.

Transforming Vector Objects

You can transform a vector object directly from the **2D View** window, or when the **Transform Vector(s)** page is displayed in the **Assistant** window.

Using Transform Vectors Mode

To transform a vector object directly from the **2D View** window, you must work in Transform Vectors mode.

To transform a vector object directly from the **2D View** window:

1. Select the vector object that you want to transform. For details, see "Selecting Vectors" on page 138.
2. Use any of the following methods to enter Transform Vectors mode:

   - Click on the **Transform Vectors** button in the **Vector Editing** area.

Note: If the **Transform Vectors Mode** button is shown as in the **Vector Editing** area, you are already in Transform Vectors mode.

   - Press the **T** key on your keyboard.
   - Hold the **Ctrl** key down on your keyboard, and then click until transform handles appear on the bounding box that surrounds the selected vector object.

   The **Transform Vectors Mode** button changes to . Resizing handles, a rotation handle and shearing control points are displayed about the selected vector object.

Note: If you click outside of the bounding box that surrounds the selected vector object once, ArtCAM Pro remains in Transform Vectors mode. If you click more than once, ArtCAM Pro returns to Select Vectors mode.

When working in Transform Vectors mode, you can transform the selected vector object in the following ways:

- To move the vector object, move the cursor over its centre, one of its spans or its bounding box, and then click and drag it into position.

Note: Hold the **Ctrl** key down to keep a copy of the selected vector object in its original position when moving it.

- To adjust the size and shape of the vector object, move the cursor over any of its resizing handles, and then click and drag.

Note: Hold the **Shift** key down when scaling to preserve the ratio between the vector object's width and height.

Note: Hold the **Alt** key down when scaling to scale the selected vector object about the centre of its bounding box.

- To adjust the angle of the vector object, click and drag the rotation handle joined to its centre of gravity.
- To shear the vector object vertically, move the cursor over the control point outside of the bottom edge of the bounding box, and then click and drag.
- To shear the vector object horizontally, move the cursor over the control point outside of the right edge of the bounding box, and then click and drag.

Note: If you want to transform the selected vector object about a defined origin and/or using specific values, press the **T** key on your keyboard again to display the **Transform Vector(s)** page in the **Assistant** window. For details, see

Using the Transform Vector(s) Page

You can use the **Transform Vector(s)** page to transform a selected vector object in a number of ways.

To display the **Transform Vector(s)** page:

1. Select the vector object that you want to transform. For details, see "Selecting Vectors" on page 138.

2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area of the **Assistant**'s Home page.

Note: Alternatively, you can press the **T** key on your keyboard twice to display the **Transform Vector(s)** page.

Note: You can also display the **Transform Vector(s)** page from the Main menu bar, by clicking on the **Vectors** menu and then the **Transform...** option.

Note: You can also display the **Transform Vector(s)** page if you right-click on the selected vector object to display the Vector Editing menu, and then click on the **Transform Vector(s)...** option

The **Transform Vector(s)** page is divided into the following areas:

- **Position and Origin** - This allows you to define the origin about which the selected vector object is transformed. For details, see "Defining the Transform Origin" on page 193.
- **Size** - This allows you to change the size of the selected vector object. For details, see "Scaling Vectors" on page 194.
- **Move** – This allows you to move a selected vector object along the X and Y axis. For details, see "Moving Vectors" on page 198.
- **Rotate** – This allows you to spin a selected vector object around the model's X and Y axes. For details, see "Rotating Vectors" on page 197.
- **Shear** - This enables you to shear a selected vector object. For details, see "Shearing Vectors" on page 200.

Defining the Transform Origin

You can transform a selected vector object relative to a defined origin using the **Position and Origin** area of the **Transform Vector(s)** page.

To define the origin by which you want to reposition, resize or rotate a selected vector object:

1. Select the vector object that you want to transform. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. The values in the **X** and **Y** boxes show the current position of the transform origin. Define the new origin about which you want to transform the selected vector object in any of the following ways:
   - Click on any of the five radio buttons on the rectangle diagram to set the origin to one of five pre-set positions. The co-ordinates of the selected position are displayed in the **X** and Y boxes.
   - Type the co-ordinate you want to set as the X and Y origins in the **X** and **Y** boxes.
   - Click to select the **Transform about (x,y)** option . The default **X** and **Y** boxes are greyed-out and new **X** and **Y** boxes are displayed on the page. The + icon appears in the **2D View** window, marking the current position of the transform origin.

     To define the origin you can either type its co-ordinates in the **X** and **Y** boxes or move the + cursor to the position in the **2D View** window that you want to use, and then click to select. In the latter instance, the co-ordinates of the selected position are displayed in the **X** and **Y** boxes.
4. Click on the **Apply** button to define the transform origin. You can now transform the selected vector object about this origin.

Scaling Vectors

You can change the size of a vector object using the **Size** area of the **Transform Vector(s)** page in the following ways:

To change the size of a vector object using specific measurements:

1. Select the vector object that you want to resize. For details, see "Selecting Vectors" on page 138.

2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to resize the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. If you want to preserve the ratio between the width and height of the selected vector object, make sure that the **Link Width and Height** option is selected. This option is selected by default.
5. Define the width of the selected vector object using the **New Width** box. If the **Link Width and Height** option is selected, the current value in the **New Height** box is adjusted. If the **Link Width and Height** option is deselected, you must define its height using the **New Height** box.

Note: If you want to keep a copy of the selected vector object in its original size, click to select the **Copy on Apply (Ctrl)** option or hold down the **Ctrl** key when scaling.

Note: Hold the **Shift** key down when scaling to preserve the ratio between the vector object's width and height.

Note: Hold the **Alt** key down when scaling to scale the selected vector object about the centre of its bounding box.

6. Click on the **Apply** button to resize the selected vector object. The values shown in the **New Width** and **New Height** boxes are adjusted.
7. Click on the **Close** button to return to the **Assistant**'s Home page.

To change the size of a vector object using the slider:

1. Select the vector object that you want to resize. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.

3. Make sure that the origin by which you want to resize the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. If you want to preserve the ratio between the width and height of the selected vector object, make sure that the **Link Width and Height** option is selected ☑. This option is selected by default.

Note: If the **Link Width and Height** option is deselected ☐, only the width of the selected vector object can be adjusted using the slider.

Note: If you want to keep a copy of the selected vector object in its original size, click to select the **Copy on Apply (Ctrl)** option ☑ or hold down the **Ctrl** key when scaling.

---

5. Click and drag the slider in the appropriate direction:
   - Drag to the right if you want to increase the size of the selected vector object.
   - Drag to the left if you want to reduce the size of the selected vector object.
6. Click on the **Apply** button to resize of the selected vector object.
7. Click on the **Close** button to return to the **Assistant**'s Home page.

To scale a vector object as a percentage of its original size:

1. Select the vector object that you want to resize. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to resize the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. Define the percentage of the vector object's original size to which you want it to be scaled in the **Scale** box.

Note: If you want to keep a copy of the selected vector object in its original size, click to select the **Copy on Apply (Ctrl)** option ☑ or hold down the **Ctrl** key when scaling.

5. Click on the **Apply** button to resize of the selected vector object. The values shown in the **New Width** and **New Height** boxes are adjusted.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Rotating Vectors

You can rotate a vector object in two ways using the **Rotate** area of the **Transform Vector(s)** page:

To rotate a vector object using a specific angle:

1. Select the vector object that you want to rotate. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to rotate the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. Type the angle by which you want to rotate the selected vector object in the **Rotate** box:
   - A positive value rotates the selected vector object clockwise.
   - A negative value rotates the selected vector object anti-clockwise.

Note: If you want to keep a copy of the selected vector object in its original position, click to select the **Copy on Apply (Ctrl)** option ☑ or hold down the **Ctrl** key when rotating.

5. Click on the **Apply** button to rotate the selected vector object by the defined angle.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

To rotate a vector object around the X and Y axes using the sliders:

1. Select the vector object that you want to rotate. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to rotate the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. Click and drag the slider in the appropriate direction:
   - Drag to the right if you want to rotate the selected vector object clockwise.
   - Drag to the left if you want to rotate the selected vector object anti-clockwise.

Note: If you want to keep a copy of the selected vector object in its original position, click to select the **Copy on Apply (Ctrl)** option or hold down the **Ctrl** key when rotating.

5. Click on the **Apply** button to rotate the selected vector object by the angle shown in the **Rotate** box.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Moving Vectors

You can reposition a vector object in two ways using the **Move** area of the **Transform Vector(s)** page:

To move a vector object to a specific position:

1. Select the vector object that you want to resize. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to move the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. Type the co-ordinate to which you want to move the selected vector object along the X-axis in the **Move X** box.

A positive value moves the selected vector to the right, while a negative value moves it to the left.

Note: If you want to keep a copy of the selected vector object in its original position, click to select the **Copy on Apply (Ctrl)** option or hold down the **Ctrl** key when moving.

5. Type the co-ordinate to which you want to move the selected vector object along the Y-axis in the **Move Y** box.

   A positive value moves the selected vector upwards, while a negative value moves it downwards.
6. Click on the **Apply** button to reposition the selected vector object.
7. Click on the **Close** button to return to the **Assistant**'s Home page.

To move a vector object along the X and Y axes using the sliders:

1. Select the vector object that you want to resize. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Make sure that the origin by which you want to move the selected vector object is correct. For details, see "Defining the Transform Origin" on page 193.
4. Click and drag the slider in the appropriate direction:
   - Drag to the right if you want to rotate the selected vector object clockwise.
   - Drag to the left if you want to rotate the selected vector object anti-clockwise.

Note: If you want to keep a copy of the selected vector object in its original position, click to select the **Copy on Apply (Ctrl)** option or hold down the **Ctrl** key when moving.

5. Click on the **Apply** button to reposition the selected vector object.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Shearing Vectors

Shearing can be visualised by thinking of an image superimposed onto a flexible rubber sheet. If you hold the sides of the sheet and move them up and down in opposite directions, the image undergoes a spatial stretching known as shearing:

You can shear a vector object in two ways using the **Shear** area of the **Transform Vector(s)** page:

To shear a vector object using specific angles:

1. Select the vector object that you want to shear. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Type the angle by which you want to shear the selected vector object vertically in the **Shear X** box.

Note: If you want to keep a copy of the selected vector object in its original shape, click to select the **Copy on Apply (Ctrl)** option or hold down the **Ctrl** key when shearing.

4. Type the angle by which you want to shear the selected vector object horizontally in the **Shear Y** box.
5. Click on the **Apply** button to shear the selected vector object.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

To shear a vector object along the X and Y axes using the sliders:

1. Select the vector object that you want to shear. For details, see "Selecting Vectors" on page 138.
2. Click on the **Transform Vector(s)** button in the **Position Size Align Vectors** area to display the **Transform Vector(s)** page.
3. Click and drag the **Shear X** slider to shear the vector object vertically.

Note: If you want to keep a copy of the selected vector object in its original shape, click to select the **Copy on Apply (Ctrl)** option ☑ or hold down the **Ctrl** key when shearing.

4. Click and drag the **Shear Y** slider to shear the vector object horizontally.
5. Click on the **Apply** button to shear the selected vector object.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Manipulating Vector Objects

There is a range of tools in both the **Position Size Align Vectors** and **Group Merge Join Vectors** areas of the **Assistant**'s Home page designed to assist you in manipulating vector objects.

Mirroring Vectors

Using the **Mirror Vectors** page, you can reverse the direction of a vector object from its original position.

To mirror a selection of vector objects:

1. Select the vector object that you want to mirror. For details, see "Selecting Vectors" on page 138.
2. Click on the **Mirror Vectors** button in the **Vector Editing** area of the **Assistant**'s Home page to display the **Mirror Vectors** page.

Note: You can also display the **Mirror Vectors** page from the Main menu bar, by clicking on the **Vectors** menu and then the **Mirror...** option.

3. If you want to produce mirrored copies and keep the original position of the selected vector objects, click to select the **Copy the Original Vectors (Ctrl)** option ☑.
4. Click on any of the options listed in the **Mirror Types** area of the page to mirror the selected vector object. For details, see "Using the Mirroring Options" on page 202.

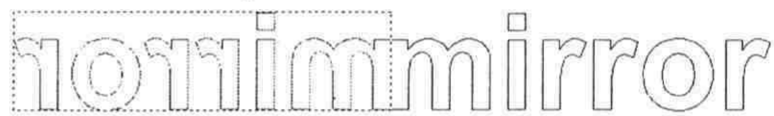
5. Click on the **Close** button to return to the **Assistant**'s Home page.

Using the Mirroring Options

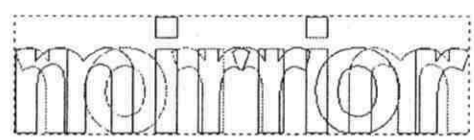
You can mirror a selected vector object horizontally, vertically or about a line.

You can use the options in the **Mirror Types** area of the **Mirror Vectors** page to mirror a vector object in the **Horizontal** plane in the following ways:

- **Left** - This option mirrors the selected vector objects in the horizontal plane about a point on the left edge of the selected vector objects:



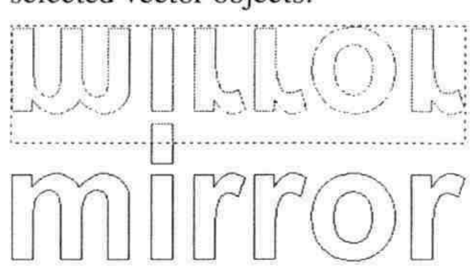
- **Centre** - This option mirrors the selected vector objects in the horizontal plane about a point in the centre of the selected vector objects:



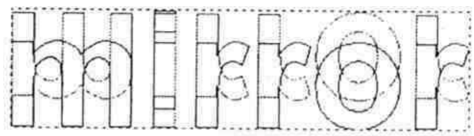
- **Right** - This option mirrors all your selected vector objects in the horizontal plane about a point on the right edge of the selected vector objects:

You can use the options in the **Mirror Types** area of the page to mirror a vector object in the **Vertical** plane in the following ways:

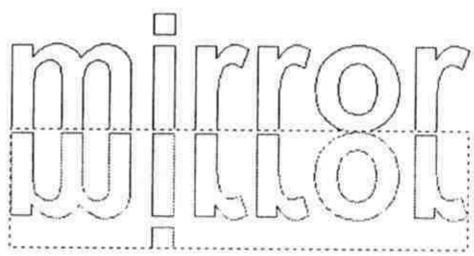
- **Top** - This option mirrors all your selected vector objects in the horizontal plane about a point on the top edge of the selected vector objects:



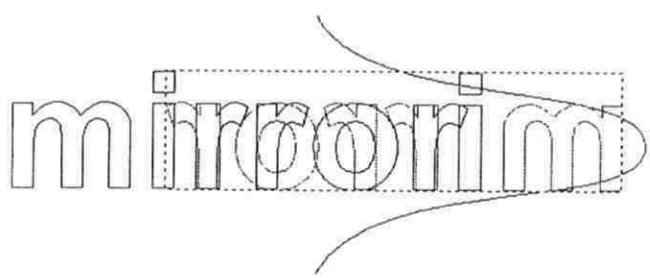
- **Centre** - This option mirrors the selected vector objects in the vertical plane about a point in the centre of the selected vector objects:



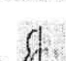
- **Bottom** - This option mirrors the selected vector objects in the horizontal plane about a point on the bottom edge of the selected vector objects:



You can also mirror a selected vector object about a selected open vector object, such as a polyline, using the **About Line** option on the page:



To mirror a vector object about an open vector object:

1. Select the open vector object about which you want to mirror a vector object. For details, see "Selecting Vectors" on page 138.
2. Hold the **Shift** key down on your keyboard, and then click to select the vector object that you want to mirror.
3. Click on the **Mirror Vectors** button  in the **Vector Editing** area to display the **Mirror Vectors** page.
4. Click on the **About Line** option on the **Mirror Vectors** page.

Note: If you want to mirror the selected vector object about the selected open vector object and keep a copy in its original position, click to select the **Copy the Original Vectors (Ctrl)** option ☑.

5. Click on the **Close** button to return to the **Assistant**'s Home page.

Aligning Vectors

You can position two or more vector objects relative to one another in different ways using the align buttons in the **Position Size Align Vectors** area of the **Assistant**'s Home page.

When you select the vector objects that you want to position relative to one another, it is important to remember that all of the vector objects you select are to be aligned to the vector object selected last. This is referred to as the base vector object.

To align two or more vector objects:

1. Select two or more vector objects that you want to position relative to one another. For details, see "Selecting Vectors" on page 138.
2. Click on any of the align buttons in the **Position Size Align Vectors** area to align all of the selected vector objects relative to the base vector object:

   - Click on the **Align Left** button to align the left edge of all other selected vector objects to the left edge of the base vector object.
   - Click on the **Align Right** button to align the right edge of all other selected vector objects to the right edge of the base vector object.
   - Click on the **Align Top** button to align the top edge of all other selected vector objects to the top edge of the base vector object.
   - Click on the **Align Bottom** button to align the bottom edge of all other selected vector objects to the bottom edge of the base vector object.

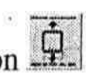
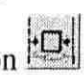
Note: You can also align the selected vectors relative to the base vector object using the Main menu bar. Click on the **Vectors** menu, and then on the **Align Vectors** option followed by the alignment option you want to use: **Left**, **Right**, **Top** or **Bottom**.

Centring Vectors

You can centre one or more vector objects inside of another in different ways using the centring buttons in the **Position Size Align Vectors** area of the **Assistant**'s Home page.

When selecting the vector objects that you want to centre inside of another, it is important to remember that all of the vector objects you select are to be centred inside of the vector object selected last. This is referred to as the base vector object.

To centre one or more vector objects inside of another:

1. Select two or more vector objects that you want to position relative to one another. For details, see "Selecting Vectors" on page 138.
2. Click on any of the centring buttons in the **Position Size Align Vectors** area to centre all of the selected vector objects inside of the base vector object:
   - Click on the **Centre Vertically** button  to align the centre of all other selected vector objects with the centre in the X-axis of the base vector object.
   - Click on the **Centre Horizontally** button  to align the centre of all other selected vector objects with the centre in the Y-axis of the base vector object.
   - Click on the **Centre Vector** button  to align the centre of all other selected vector objects with the centre of the base vector object.

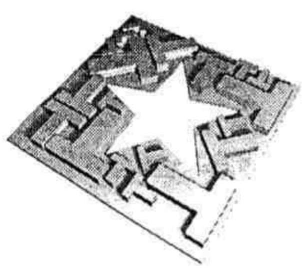
There is one additional centring button in the **Position Size Align Vectors** area, although it does not, unlike the other centring buttons, affect one vector object relative to another:

- Click on the **Centre In Page** button to align the centre of the selected vector objects to the centre of the model (the white area).

Note: You can also centre all of the selected vector objects inside of the base vector object using the Main menu bar. Click on the **Vectors** menu, and then on the **Align Vectors** option followed by the alignment option you want to use: **Horizontal Centre**, **Vertical Centre**, **Centre** or **Centre In Page**.

Note: You can also align the centre of the selected vector objects with the centre of the model by pressing the **F9** key or the **Ctrl + P** keys on your keyboard.

Nesting Vectors



If you have an area of material that cannot contain a block of vector text as it is normally written, or a group of vector objects, you need not treat it as waste material.

Rather than creating the space to fit the vector text or group of objects in the conventional sense, you can manipulate them to fit into what space is available.

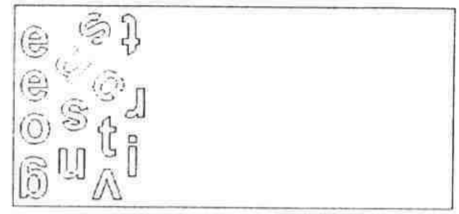
Using the **Nest Selected Vectors** button in the **Position Size Align Vectors** area of the **Assistant**'s Home page, you can be as economical as possible with your material.

You can nest vector objects within another vector object or the model area. If you are nesting vector objects within another, you must create a closed vector object that represents the shape of material that is available in which to machine. You can use the tools in the **Vector Editing** area to define its shape. For example, if you have an odd shape of material available in which you want to nest vector text, use the **Create Polyline** tool to reproduce its outline. For details, see "Creating a Polyline" on page 120.

For example, consider the vector text *Nesting Vectors* before and after it has been nested inside a rectangle:

Before...

After...



To nest vector objects:

1. Make sure that you are in **Select Vectors** mode. For details, see "Selecting Vectors" on page 138.
2. Hold the **Shift** key down on your keyboard, and then click to select the vector object(s) that you want to nest. If you want to nest a selection of vector objects within another, make sure that you select the vector object in which others will be nested first.
3. Click on the **Nest Selected Vectors** button in the **Position Size Align Vectors** area to display the **Nesting** page.

Note: You can also display the **Nesting** page from the Main menu bar, by clicking on the **Vectors** menu and then the **Nest Vectors** option.

4. Type the diameter of the tool you are using to machine the nested vector objects in the **Tool Diameter (D)** box.
5. Type the amount of extra material that you want to surround each of the nested vector objects in the **Toolpath Clearance (C)** box.
6. To allow the selected vector objects to be rotated during the nesting process, click to select the **Allow Part Rotation** option. This and the remaining options allow ArtCAM Pro flexibility when fitting the selected vector objects into the model area.

   If you have selected this option, type the increment by which you want ArtCAM Pro to rotate the vector objects during the nesting process in the **Step Angle (A)** box.

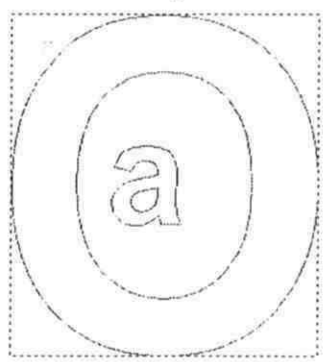
7. By default, the selected vector object(s) are nested within the vector object selected first. If you want to nest the selected vector object(s) within the model area (white area), click to select the **Model Is Sheet** option.

8. To nest a block of vector text or a group of vector objects as it is shown in the **2D View** window, click to select the **Don't Nest Inner Vectors (Preserve Groups)** option.

9. To allow vector objects to be mirrored during the nesting process, click to select the **Allow Mirrored Parts** option.

10. To allow vector objects to be nested within other vector objects that have a central cavity, click to select the **Allow Parts In Parts** option.

    In the example shown, the letter *a* is nested within the central cavity of the letter *O*:

Warning: Avoid using the **Allow Parts In Parts** option unless absolutely necessary, as this makes for complex and prolonged machining.

11. Click on the **Nest From** list box, and then on one of the following options:

    - **Bottom Left** – To nest the selected vector objects from the bottom-left corner of the model area.
    - **Bottom Right** – To nest the selected vector objects from the bottom-right corner of the model area.
    - **Top Left** – To nest the selected vector objects from the top-left corner of the model area.
    - **Top Right** – To nest the selected vector objects from the top-right corner of the model area.

12. Define how closely you want the cutting tool to follow the shape of the nested vector objects in the **Curve Tolerance** box.
13. To create a vector object in the shape of the disposable material that remains after the selected vector objects have been nested, click to select the **Create Leftover Material Vector** option.
14. To nest multiple copies of a selected vector objects, click to select the **Nest Multiple Copies** option, and then type the number of copies you want to create in the **No. of Copies** box.
15. Click on the **Nest** button to nest the selected vector objects.

    A progress bar appears beneath the **2D View** window indicating the progress made in calculating the final position of the nested vector objects.

    The calculation time depends on the **Part Rotation** options you have selected: the fewer the options selected, the faster the nesting process. You can click on the **Cancel** button at any time to stop the nesting process.
16. Click on the **Close** button to return to the **Assistant**'s Home page.

Merging Vectors

You can merge two or more shapes of vector objects in order to create a new shape of vector object using the merging buttons in the **Group Merge Join Vectors** area of the **Assistant**'s Home page. The newly merged vector object is drawn on the currently selected layer.

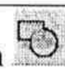
Warning: You can only merge two or more vector objects if they are closed, ungrouped and overlapping. For further information, see "Ungrouped Vectors" on page 219.

To merge two or more ungrouped vector objects:

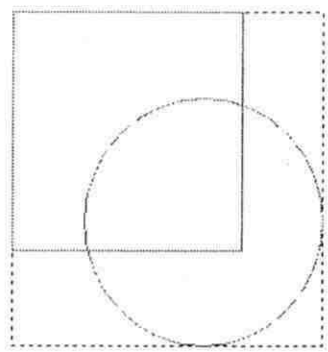
1. Select the vector objects that you want to merge. For details, see "Selecting Vectors" on page 138.
2. Click on any of the merging buttons in the **Group Merge Join Vectors** area to merge the selected vector objects:

Note: You can also merge the selected vectors using the Main menu bar. Click on the **Vectors** menu, and then on the **Merge Vectors** option followed by the merging option you want to use: **Weld**, **Intersect**, **Subtract** or **Trim**.

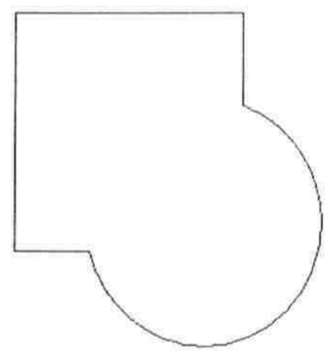
- Click on the **Weld Vectors** button  to take two or more vector objects and create a new vector object that is the outline of them when fused together:

Warning: The **Weld Vectors** button  is the only merging button in the **Group Merge Join Vectors** area that you can use to merge more than two overlapping vector objects.

Before...

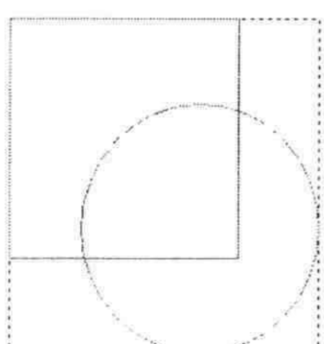


.After...

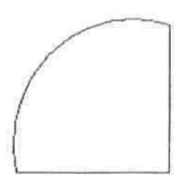


- Click on the **Intersect Vectors** button  to take two vector objects and create a new vector object that is the shape of the area where they overlap:

Before...

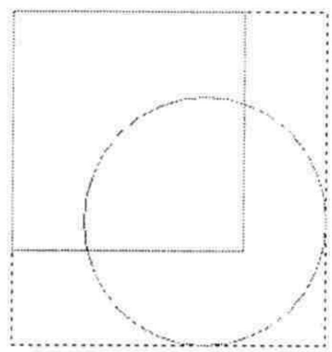


After...

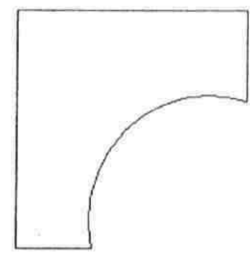


- Click on the **Subtract Vectors** button  to produce a vector object that is the shape of the area remaining when the vector object selected last is subtracted from that which was selected first:

Before...

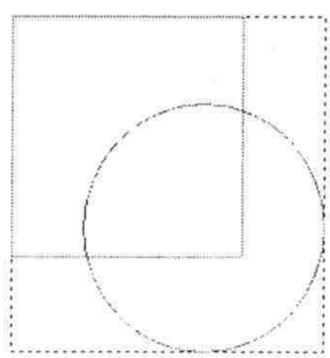


After...

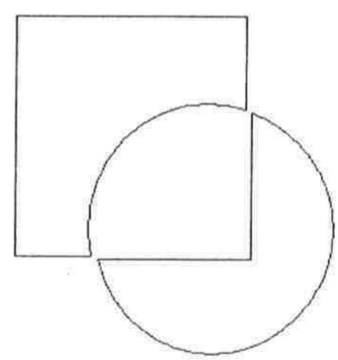


- Click on the **Trim Vectors** button  to combine two selected vector objects in such a way that all that remains are the areas of the vector objects which are not overlapping:

Before...



After...



Joining Vectors

You can join two vector objects to create one of four types of new shape using the joining buttons in the **Group Merge Join Vectors** area of the **Assistant**'s Home page. The newly joined vector object is drawn on the currently active layer.

Note: The vector objects must be open and ungrouped. For further information, see "Selecting Vectors" on page 138 and "Ungrouped Vectors" on page 219.

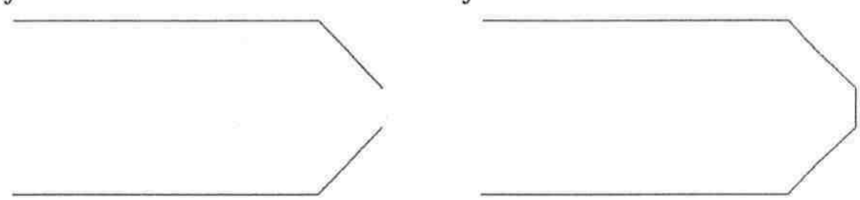
To join together open, ungrouped vector objects:

1. Select the vector objects that you want to join with another. For details, see "Selecting Vectors" on page 138.
2. Click on any of the joining buttons in the **Group Merge Join Vectors** area to join the selected vector objects:

Note: You can also join the selected vectors using the Main menu bar. Click on the **Vectors** menu, and then on the **Join Vectors** option followed by the joining option you want to use: **with a smooth curve**, **with a line**, **move end points** or **with coincident points**.

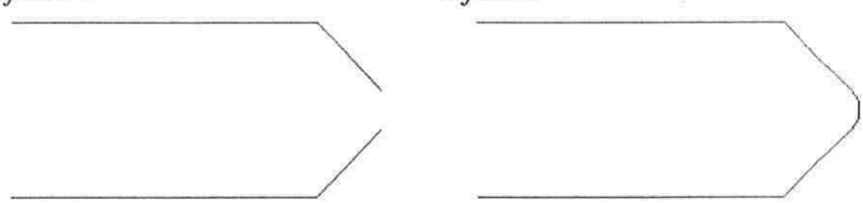
- Click on the **Join Vectors With A Line** button  to link the nearest points in the two vector objects with a linear span:

Before... *After...*



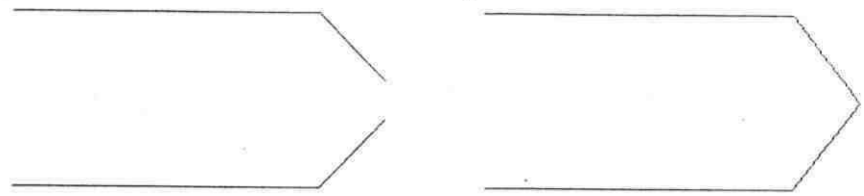
- Click on the **Join Vectors With A Curve** button  to link the nearest points in the two vector objects with a bezier curve span:

Before... *After...*



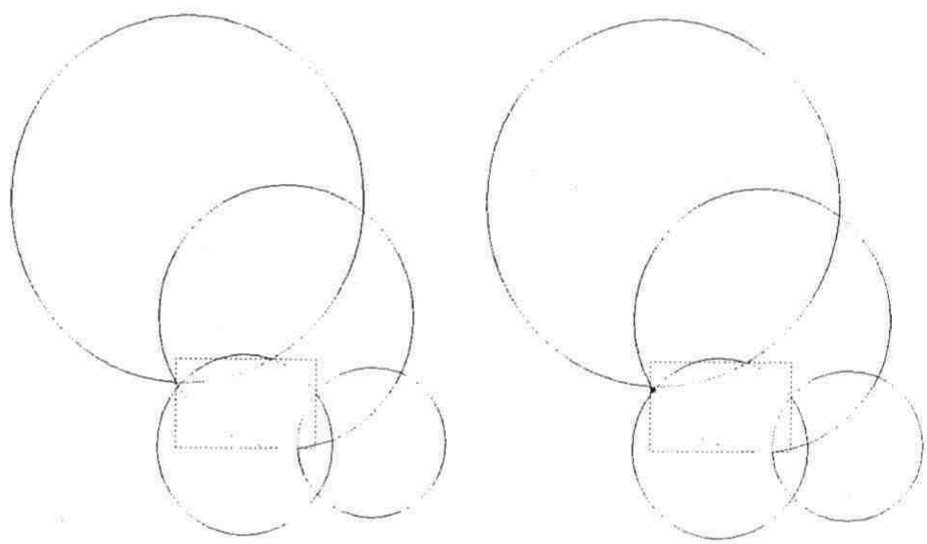
- Click on the **Join Vectors By Moving Ends** button  to link the nearest points in the two vector objects by moving each of their end points to a central position:

Before... *After...*



- Click on the **Join Vectors With Coincident Start or End Points** button  to display the **Join Multiple Vectors** page. For details, see "Joining Vectors with Coincident Points" on page 213.

Selected Coincident Vectors... *Joined Coincident Vectors...*



Joining Vectors with Coincident Points

To join together vector objects with coincident start or end points:

1. Hold the **Shift** key down on your keyboard, and then click to select the vector objects that you want to join together. The vector objects turn magenta. For details, see "Selecting Vectors" on page 138.

Note: The selected vector objects must have coincident start or end points.

---

2. Click on the **Join Vectors with Coincident Start or End Points** button in the **Group Merge Join Vectors** area to display the **Join Multiple Vectors** page.
3. If you only want ArtCAM Pro to join coincident points in the selected vector objects within a specific distance of one another, type the distance in the **Tolerance** box.
4. To calculate the number of vector objects that will remain after joining the coincident points in the selected vector objects within tolerance, click on the **Calc Info** button.
5. Click on the **Join** button to join the coincident points in the selected vector objects within tolerance.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Closing Vectors

You can close an open vector object to create one of three types of closed vector object using the closing buttons in the **Group Merge Join Vectors** area of the **Assistant**'s Home page.

Note: The vector objects must be ungrouped. For further information, see "Selecting Vectors" on page 138 and "Ungrouped Vectors" on page 219.

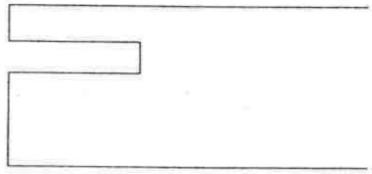
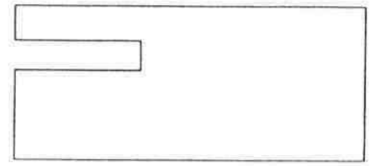
To close an open vector object:

1. Select the vector object that you want to close. For details, see "Selecting Vectors" on page 138.
2. Click on any of the closing buttons in the **Group Merge Join Vectors** area to join the selected vector objects:

Note: You can also close the selected vectors using the Main menu bar. Click on the **Vectors** menu, and then on the **Close Vector** option followed by the closing option you want to use: **with a smooth curve**, **with a line**, or **move end point**.

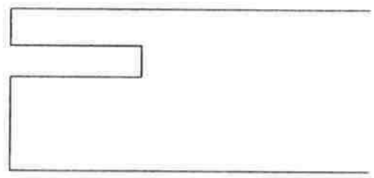
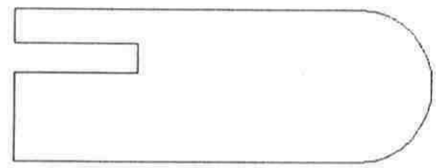
- Click on the **Close Vector With A Line** button to join the Start Point and end point with a linear span, as shown below:

Before... *After...*

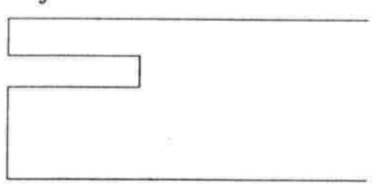
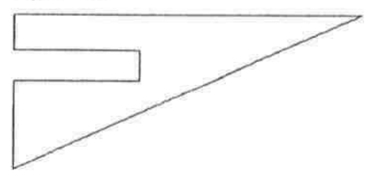
- Click on the **Close Vector With A Curve** button  to join the Start Point and end point with a bezier curve span, as shown below:

Before... *After...*

- Click on the **Close Vector – Move End Points** button  to join the end point with the Start Point, as shown below:

Before... *After...*

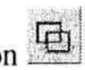
Grouping Vector Objects

Grouping combines all of the individual spans, points (nodes) and control points that make up a vector object as a single entity.

You can group an individual vector object or a selection of them. This process allows you to simultaneously machine or engrave each of the vector objects that you define as a group in the same way. For details, see "Selecting Vectors" on page 138.

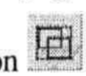
Grouping allows you to machine or engrave an open vector object along with an individual or group of closed vector objects when using certain toolpaths. These toolpaths are **Engraving** and **Drilling**. For details, see "Using Toolpath Strategies" in the Machining Models chapter.

To group an open vector object:

1. Select the vector object that you want to group. For details, see "Selecting Vectors" on page 138.
2. Click on the **Group** button  in the **Group Join Vectors** area of the **Assistant**'s Home page. The vector object turns purple. When deselected, the vector object is blue, regardless of the colour assigned to the layer on which it is drawn.

Note: You can also group the selected vector object(s) using the Main menu bar. Click on the **Vectors** menu, and then on the **Group Vector(s)** option.

To group a closed vector object:

1. Select the vector object that you want to group. For details, see "Selecting Vectors" on page 138.
2. Click on the **Group** button  in the **Group Merge Join Vectors** area of the **Assistant**'s Home page. The vector object turns purple. When deselected, the vector object appears in the colour assigned to the layer on which it is drawn.

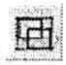
   If you group a closed vector object containing self-intersections, when deselected it turns red with the ○ icon marking the position of all intersections.

   If you select a combination of open and closed vector objects with the intention of grouping them, all of them turn magenta. However, when they have been grouped and deselected, the closed vector objects turn purple by default, while the open vector objects turn blue.

Note: You can also right-click on the selected vector object to display the Vector Editing menu, and then click on the **Group Vector(s)** option.

You can also ungroup vector objects. To ungroup any vector object:

1. Select the vector object that you want to ungroup. For details, see "Selecting Vectors" on page 138.

2. Click on the **Ungroup** button  in the **Group Merge Join Vectors** area of the **Assistant**'s Home page. The vector object is deselected. If the selected vector object is open, the vector object turns blue, regardless of the colour assigned to the layer on which it is drawn. If the selected vector object is closed, it appears in the colour assigned to the layer on which it is drawn.

Note: You can also ungroup the selected vector object(s) using the Main menu bar. Click on the **Vectors** menu, and then on the **Ungroup Vector(s)** option.

Note: You can also right-click on the selected vector object to display the Vector Editing menu, and then click on the **Ungroup Vector(s)** option.

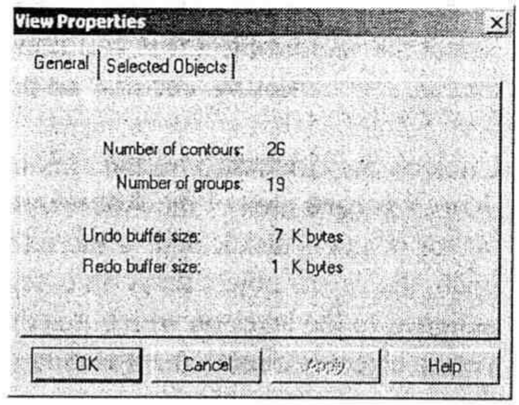
Viewing the Properties of Vector Objects

You can find out the number of points (nodes) and control points within any grouped vector object, or selection of two or more vector objects, using the **Vector Properties** dialog box.

To view the properties of a grouped vector object or selection of two or more vector objects:

1. Select a grouped vector object or two or more vector objects. For details, see "Selecting Vectors" on page 138.

2. Right-click to display the Vector Editing menu, and then click on the **Properties...** option to display the **View Properties** dialog box. Its **General** page is displayed by default:



The **Number of contours** area shows the total number of ungrouped vector objects currently selected.

The **Number of groups** area shows the total number of grouped vector objects currently selected.

3. Click on the **Selected Objects** tab to display its page.

   The **Number of selected vectors** area shows the total number of ungrouped vector objects currently selected.

   The **Number of selected groups** area shows the total number of grouped vector objects currently selected.

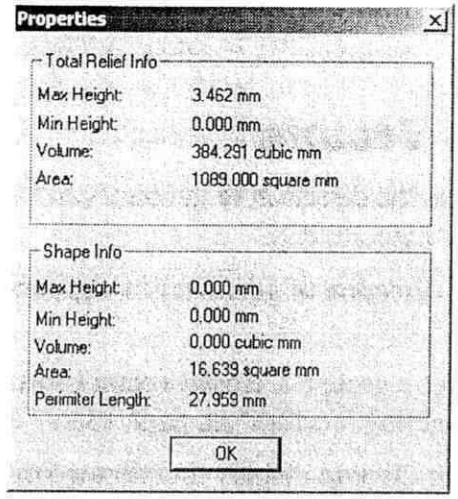
   The **Total number of points** area shows the number of points and control points in the selected vector object(s). The number of points and control points are not counted separately. For details, see "Editing Vector Nodes" on page 148.

   The **Average number of points** area shows the total number of points divided by the total number of vector objects currently selected.

4. Click on the **OK** button to close the **Vector Properties** dialog box.

To view the properties of an ungrouped vector object:

1. Select an ungrouped vector object. For details, see "Selecting Vectors" on page 138.
2. Right click to display the Vector Editing menu, and then click on the **Properties** option to display the **Properties** dialog box:



You can see the area and the perimeter length of the selected vector object in the **Shape Info** area. The perimeter length is equal to the total length of all spans within the selected vector object.

If a shape has been attributed to the selected vector object, you can also see its volume and its maximum and/or minimum height.

3. Click on the **OK** button to close the **Properties** dialog box.

Reversing a Vector Object's Direction

You can reverse the current direction of any closed vector object. The direction of a vector object determines the cutting direction of the toolpath used to machine it.

Ungrouped Vectors

You can reverse the direction of an ungrouped, closed vector object that you have created.

To reverse the direction of an ungrouped, closed vector object:

1. Select an ungrouped, closed vector object. For details, see "Selecting Vectors" on page 138.
2. Right-click to display the Vector Editing menu, and then click on the available direction change option:
   - **Make Clockwise** – To change the direction of geometry to clockwise.

- **Make Anti-Clockwise** – To change the direction of geometry to anti-clockwise.

Grouped Vectors

You can reverse the direction of geometry in a grouped, closed vector object that you have created.

To reverse the direction of geometry in a grouped, closed vector object:

1. Select a grouped, closed vector object. For details, see "Selecting Vectors" on page 138.
2. Right-click to display the Vector Editing menu, and then click on the **Reverse Vector(s)** option to reverse the direction of geometry.

You can confirm the direction of geometry in a grouped, closed vector object by ungrouping it after you click on the **Reverse Vector(s)** option, and then regrouping it again. For further details, see "Grouping Vector Objects" on page 215.

Creating Bitmaps from Vectors

You can create a bitmap image from any vector object. ArtCAM Pro creates bitmap images in the current Primary Colour around the outline of all selected vector objects. The new bitmap image is one pixel wide. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.

To convert a vector object into a bitmap image:

1. Click on the colour in the Colour Palette that you want to select as the Primary Colour.

   If you want to select a colour from a bitmap image as the Primary Colour, click on the **Pick Colour** button in the **Bitmap** toolbar, move the dropper over the colour you want to use and then click.
2. Select the vector objects that you want to create bitmap images from. For details, see "Selecting Vectors" on page 138.

3. Click on the **Vector to Bitmap** button in the **Vector – Bitmap** area of the **Assistant**'s Home page to create bitmap images from the selected vector objects.
4. Click on the **Vectors On/Off** button in the **2D View** toolbar to hide the original vector objects, allowing you to see the new bitmap images clearly.

Flood Filling Vector Objects

You can colour within the outline of any vector object in the current Primary Colour using the **Flood Fill Vectors** tool. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.

To flood fill a vector object:

1. Click on the colour in the Colour Palette that you want to select as the Primary Colour.

   If you want to select a colour from a bitmap image as the Primary Colour, click on the **Pick Colour** button in the **Bitmap** toolbar, then move the dropper over the colour you want to use and click.
2. Select the vector objects that you want to flood fill. For details, see "Selecting Vectors" on page 138.
3. Click on the **Flood Fill Vectors** button in the **Vector – Bitmap** area of the **Assistant**'s Home page to colour within the outline of the selected vector objects.
4. Click on the **Vectors On/Off** button in the **2D View** toolbar to hide the original vector objects, allowing you to see the new bitmap images clearly.

Using the Vector Doctor

ArtCAM Pro's **Vector Doctor** tool allows you to find and resolve a range of common problems found within vector artwork, often detrimental to the machining process. You can avoid problems during the manufacturing process by:

- Identifying all coincident points (nodes) in the vector objects that make up the artwork, within a specified tolerance, so that they can be joined using the **Join Vectors with Coincident Start or End Points** tool. For details, see "Identifying Coincident Points" on page 222 and "Joining Vectors with Coincident Points" on page 213.

- Identifying all loops in self-intersecting vector objects within the artwork less than a defined size, and then removing them. For details, see "Loop Detection and Removal" on page 222.

Identifying Coincident Points

You can instruct ArtCAM Pro to check for concurrent start or end points (nodes) in a selection of open vector objects, within a defined distance of one another:

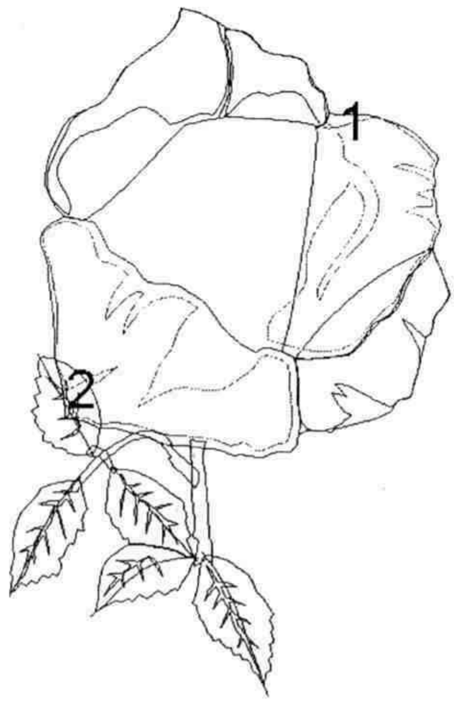
1. Select the vector artwork that you want to check for coincident points. For details, see "Selecting Vectors" on page 138.
2. Click on the **Vector Doctor** button in the **Vector Editing** area to display the **Vector Doctor** page.
3. In the **Identify Problems** area, make sure that the **Coincident Points** option is selected.
4. If you only want ArtCAM Pro to recognise points in the selected vector object within a specific distance of one another as coincident, type the distance in the **Tolerance** box.
5. Click on the **Identify** button. ArtCAM Pro marks all coincident points in the selected vector object with the icon. If you want to join together the coincident points, see "Joining Vectors with Coincident Points" on page 213.
6. Click on the **Close** button to return to the **Assistant**'s Home page.

Loop Detection and Removal

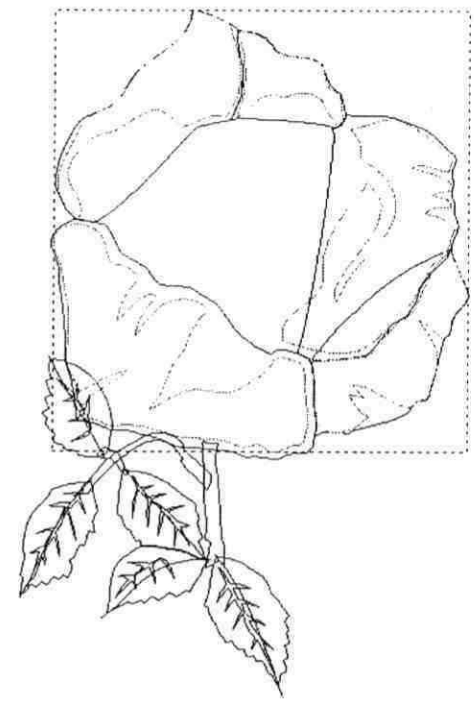
You can instruct ArtCAM Pro to identify all loops within a selection of vector objects less than a defined size, inserts a point (node) at the bottom of the loop where the vector objects self-intersect, and then trim away the loop.

For example, the rose artwork shown below contains two self-intersecting vector objects with loops. You can see the difference in their shape after the loops have been removed:

Looped Vector Objects Before...



Looped Vector Objects After...



To remove the loop in self-intersecting vector objects:

1. Select all self-intersecting vector objects in the vector artwork shown in the **2D View** window. For details, see "Selecting Vectors" on page 138.
2. Click on the **Vector Doctor** button in the **Vector Editing** area to display the **Vector Doctor** page
3. In the **Identify Problems** area, make sure that the **Vector Intersections** option is checked on.
4. Click on the **Identify** button. ArtCAM Pro marks all intersections in the selected vector object(s) with the icon.

   If you want to clear all of these icons from the **2D View** window, click on the **Clear Markers** button.
5. In the **Remove Vector Loops** area, you can define the size of loops that you want ArtCAM Pro to ignore as a percentage of the overall size of the selected vector in the **% Of Vector Area To Keep Loop** box.

   For example, if your vector artwork contains a figure 8, it is likely that you will want to keep both of its loops. If the original vector object has an area of 100 mm^2 and contains a loop of 1.5 mm^2, a value of 1% would mean that the loop would be deleted. A value of 2%, however, would mean that the loop would be ignored. ArtCAM Pro would also convert the loop into an independent vector object.

6. Type the name of the layer onto which you want to create the corrected vector object(s). If the name of the layer does not already exist, ArtCAM Pro creates a new layer for the corrected vector object(s) named *Fixed Loops*. For further details, see
7. If you want to keep a copy of the looped, self-intersecting vector object(s), make sure that the **Keep Originals** option is checked on ☑.
8. Click on the **Remove Loops** button to remove all loops in the selected vector object(s).

Note: You can also press the **Ctrl + Alt + Shift + R** keys on your keyboard to remove all loops in the selected vector object(s).

9. Click on the **Close** button to return to the **Assistant**'s Home page.

Importing a Vector into a Model

In ArtCAM Pro, you can import vector artwork saved as files of type ***.dxf**, ***.eps**, ***.dwg**, ***.ai**, ***.wmf** or ***.pic** into a model. For more details, see "Importing Vector Artwork" in the Working with Models chapter.

Exporting Vectors

In ArtCAM Pro, you can export a vector object as data. This data can then be used in other drawing packages supporting ***.eps**, ***.dxf** and ***.pic** file formats. For details, see "Exporting Vector Data" in the Working with Models chapter.

Creating a Shape from a Vector

You can generate three-dimensional shapes from vector objects using the **Shape Editor** tool. You can also create complex freeform three-dimensional shapes from vector objects. For more details, see "Creating a Shape from a Vector" and "Creating a Shape Using Vectors" in the Working with Reliefs chapter.

Creating a Feature from a Vector

You can create three different types of features from a vector object:

A depth or height is attributed to the vector object from which the feature is created. The feature is either added to or subtracted from the surface of the existing relief in your model. The contour of the existing relief is preserved in the feature that is combined with it.

A feature is machined using a **Feature Machining** toolpath. For information on toolpaths, see "Overview" in the Machining Models chapter. For details on creating a **Feature Machining** toolpath, see "Feature Machining" in the Machining Models chapter.

Creating a Raised Feature

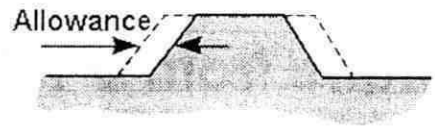
You can create a raised (male) feature from a selected vector object, usually vector text, which you can then machine using the **Feature Machining** tool. For details, see "Feature Machining" in the Machining Models chapter.

Using the **Raised Feature** page, you can:

- Control whether any central cavity in the vector object is removed or not during the feature machining process.
- Control what edge of the vector object the tool meets when cutting it from the block of material.

To create a raised feature:

1. Select the vector object from which you want to create a raised feature. For details, see "Selecting Vectors" on page 138.
2. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
3. Click on the **Raised Feature** button in the **3D Toolpaths** area of the **Toolpaths** Home page to display the **Create Raised Feature** page.
4. Type a value in the **Feature Height** box to set the height of the raised feature you want to create.
5. If you want to add or remove extra material around the selected vector object, you can type a value in the **Feature Allowance** box.



The value you enter sets the distance between the boundary of the selected vector object and the **Feature Tool** that you select on the **Feature Machining** page. Type a positive value to add material or a negative value to remove it.

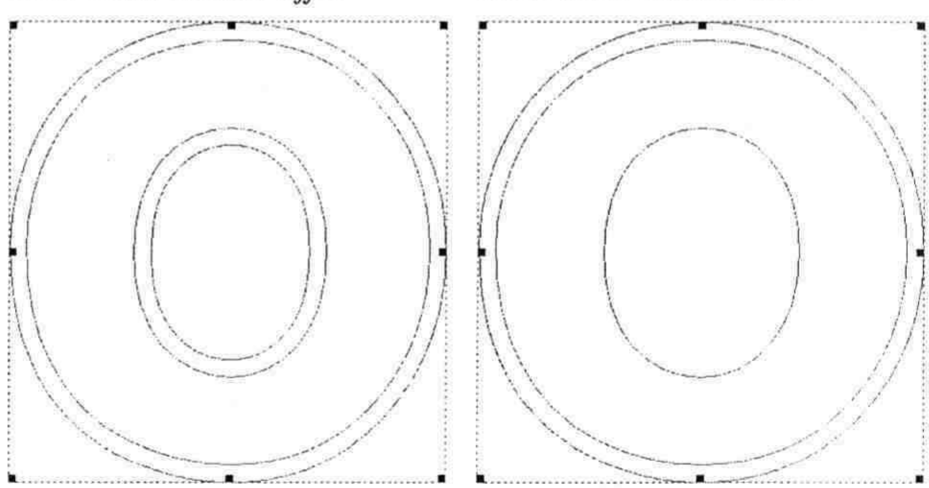
6. If the selected vector object contains a central cavity that you want to remove with the **Feature Tool** that you select on the **Feature Machining** page, click to select the **Clear Inner Islands** option ☑.

   If you do not select this option, the **Feature Tool** can only remove material inside of a central cavity as far as the **Feature Allowance** and/or **Overcut Distance** allow.

   If you select this option, any **Feature Allowance** you have already defined and/or **Overcut Distance** that you define on the **Feature Machining** page is ignored by the **Feature Tool** you use to machine the raised feature.

   In the following example, you can see that the **Feature Allowance** around the central cavity of the letter 'O' is ignored when the **Clear Inner Islands** option is selected ☑:

Clear Inner Islands Off... *Clear Inner Islands On...*



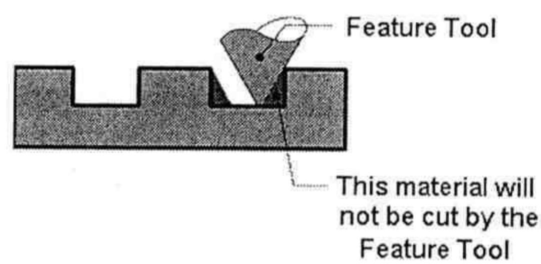
7. Click on either of the radio buttons ◉ to set the limitations for machining the selected vector object:

   - If you want to leave an allowance around the boundary of the selected vector object, including any central cavity that it may contain, select the **Use constant allowance around vector** option.

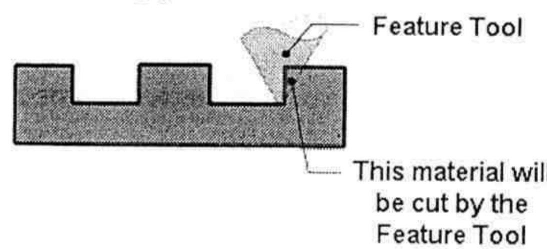
- If you want to machine around the boundary of the selected vector object, select the **Outer vector defines machining limit** option. If the selected vector object contains a central cavity, the cavity is not removed using this option. The **Feature Allowance** box is also greyed-out.

8. Click on either of the radio buttons to select how the **Feature Tool** machines the selected vector object:

   - **Vector at Top Edge** – This option places the selected **Feature Tool** in contact with the top edge of the selected vector object during the machining process.



   - **Vector at Bottom Edge** – This option places the selected **Feature Tool** in contact with the bottom edge of the selected vector object during the machining process.



9. Type a name for the raised feature in the **Name** box.
10. Click on the **Create** button.
11. Click on the **Close** button to return to the **Toolpaths** Home page.

You are now ready to create the **Feature Machining** toolpath that is used to machine the raised feature you have created. For details, see "Feature Machining" in the Machining Models chapter.

Creating a Recessed Feature

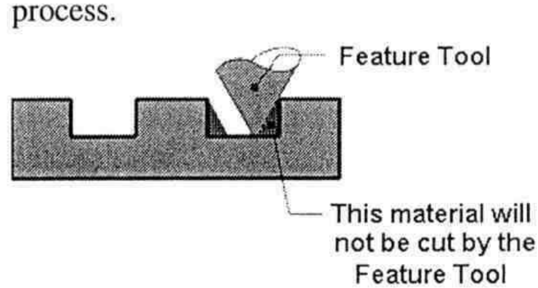
You can create a recessed (female) from a selected vector object, usually vector text, which you can then machine using the **Feature Machining** tool. For details, see "Feature Machining" in the Machining Models chapter.

Using the **Recessed Feature** page, you can:

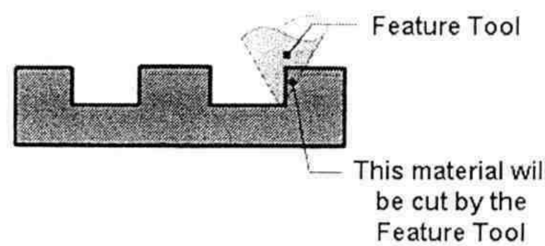
- Control what edge of the vector object the tool meets when cutting it from the block of material.

To create a recessed feature:

1. Select the vector object from which you want to create a recessed feature. For details, see "Selecting Vectors" on page 138.
2. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
3. Click on the **Recessed Feature** button in the **3D Toolpaths** area to display the **Create Raised Feature** page.
4. Type a value in the **Feature Depth** box to set the depth of the recessed feature you want to create.
5. Click on either of the radio buttons to select how the **Feature Tool** you select on the **Feature Machining** page machines the selected vector object:
    - **Vector at Top Edge** – This option places the selected **Feature Tool** in contact with the top edge of the selected vector object during the machining process.



    - **Vector at Bottom Edge** – This option places the selected **Feature Tool** in contact with the bottom edge of the selected vector object during the machining process.



6. Type a name for the recessed feature in the **Name** box.
7. Click on the **Create** button.
8. Click on the **Close** button to return to the **Toolpaths** Home page.

You are now ready to create the **Feature Machining** toolpath that is used to machine the recessed feature you have created. For details, see "Feature Machining" in the Machining Models chapter.

Creating a Centreline Engraved Feature

You can create a centreline engraved feature from a selected vector object, usually vector text, which you can then machine using the **Feature Machining** tool. For details, see "Feature Machining" in the Machining Models chapter.

The boundary of a selected vector object represents the centreline of the cutting tool. The engraved feature has no diameter other than that of the tip of the cutting tool.

To create a centreline engraved feature:

1. Select the vector object from which you want to create a centreline engraved feature. For details, see "Selecting Vectors" on page 138.
2. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
3. Click on the **Centreline Engraved Feature** button in the **3D Toolpaths** area to display the **Create Centreline Feature** page.
4. Type a value in the **Feature Depth** box to set the depth of the centreline engraved feature you want to create.
5. Type a name for the centreline engraved feature in the **Name** box.
6. Click on the **Create** button.

7. Click on the **Close** button to return to the **Toolpaths** Home page.

You are now ready to create the **Feature Machining** toolpath that is used to machine the centreline engraved feature you have created. For details, see "Feature Machining" in the Machining Models chapter.

Returning a Feature to a Vector

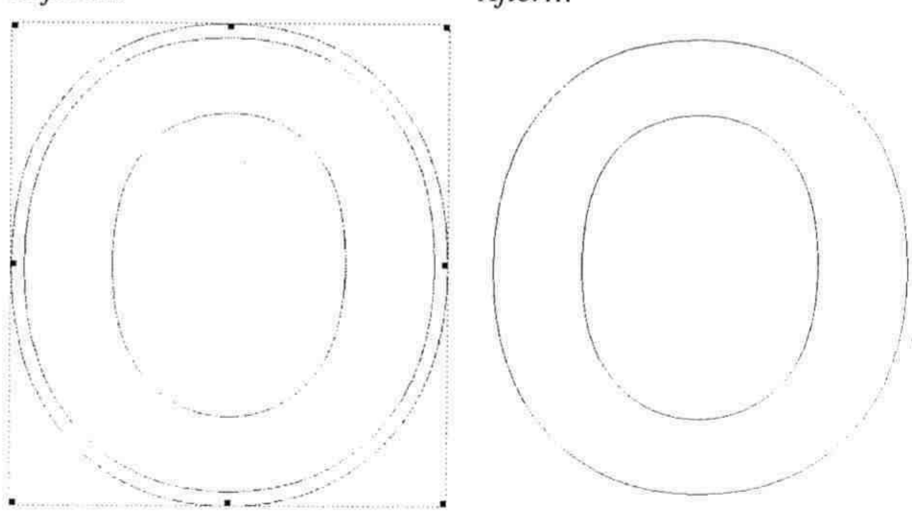
You can return a raised, recessed or centreline engraved feature to the vector object from which it was created.

To return a raised, recessed or centreline engraved feature to its original vector object:

1. Click to select the feature in the **2D View** window that you want to return to a vector object. A bounding box surrounds it.
2. Click on the **Toolpaths** tab  to display the **Toolpaths** Home page.
3. Click on the **Return Feature To Normal Vectors** button  in the **3D Toolpaths** area.

In the following example, you can see a raised feature that has been returned to a vector object:

Before... *After...*



Notice that the pass around the boundary of the letter 'O' marking the **Feature Allowance** has also been removed.

The return to a vector object is noted by a change in colour. It is now shown in the colour assigned to the layer on which it is drawn.

Working with Reliefs

Overview

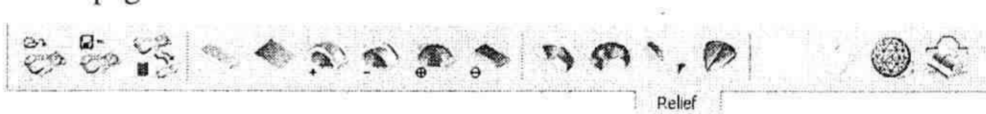
A relief is the three-dimensional model in ArtCAM Pro. When three-dimensional shapes are created using any of the relief creation tools they are combined with any current relief to create a new relief. For further information, see "Calculating a Relief" on page 272.

In ArtCAM Pro, a relief can be imported or created from three-dimensional shapes generated from two-dimensional artwork. This two-dimensional artwork could be:

- A defined area. For details, see "Creating a Dome" on page 269.
- A colour within a bitmap image. For details, see "Creating a Shape from a Bitmap" on page 232.
- A closed vector object. For details, see "Creating a Shape from a Closed Vector" on page 237.
- A combination of open and closed vector objects. For details, see "Creating a Shape Using Vectors" on page 242.

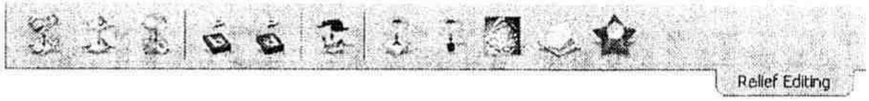
To import a relief, see "Loading a Relief" on page 304.

The relief creation tools are found in the **Relief Operations** and **Vector Based Relief Creation** areas of the **Assistant**'s Home page and on the **Relief** toolbar:



There are additional relief creation tools in the **Relief** menu located in the Main menu bar.

You can also find tools in the **Relief Editing** area of the **Assistant**'s Home page and on the **Relief Editing** toolbar that assist you in transforming, manipulating, managing and editing a relief:



For details, see "Transforming and Manipulating Reliefs" on page 280 and "Managing and Editing Reliefs" on page 302.

Using the Shape Editor

You can create simple shapes from either a colour in a bitmap image or a closed vector object using the **Shape Editor** tool and combine them with the existing relief. Using the **Shape Editor** tool, you can:

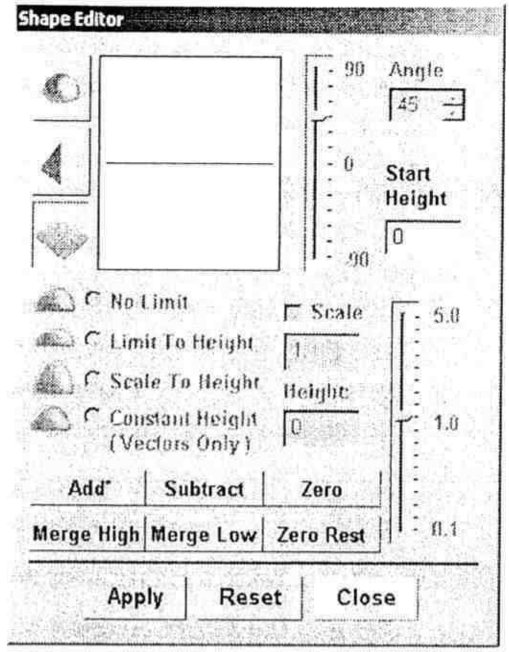
- Control the profile of the shape.
- Control the angle of the shape.
- Control the height of the shape.
- Control how the shape is combined with any existing relief.

Creating a Shape from a Bitmap

You can create simple shapes from any colour in a bitmap image and combine the shape with the existing relief in your model using the **Shape Editor** tool.

To create a shape from a bitmap colour:

1. Double-click on the bitmap colour from which you want to create a shape.to display the **Shape Editor** dialog box:



Note: You can also display the **Shape Editor** dialog box using the Main menu bar. Click on the **Colour** menu, and then on the **Shape Editor** option. Alternatively, you can double-click on the colour in the Colour Palette beneath the **2D View** window from which you want to create a shape.

2. From the **Shape Editor** dialog box, select the profile of the shape that you want to create from the selected colour:

   - Click on the **Round** button  to create a rounded shape from the selected colour.

   - Click on the **Angular** button  to create an angled shape from the selected colour.

   - Click on the **Plane** button  to create a flat shape from the selected colour.

Note: You can restore the default settings in the **Shape Editor** dialog box by clicking on the **Reset** button at any time.

3. If you want to create a rounded or an angled shape, you must define an angle for it using any of the following methods:

- Type a value in the **Angle** box. Type a positive value to produce a convex shape and a negative value to produce a concave shape. Type a value of 0 to produce a plane.

Note: If you have selected the **Plane** option, the **Angle** box is greyed-out. If so, go straight to the next step.

- Click and drag on the slider to the left of the **Angle** box. Drag the slider upward to increase the angle of the shape. Drag the slider downward to decrease it. The angle appears in the **Angle** box.
- Click on the up or down arrow on the right of the **Angle** box. The angle appears in the **Angle** box.

The default angle for a rounded or an angled shape is 45°.

4. Type a value in the **Start Height** box to define the height at which the shape starts in the Z direction. This creates a vertical side-wall beneath the shape. If you are creating a flat shape, this value controls the height of the plane.
5. Select the height option that you want to use:
   - To allow the shape to grow to the height that it would naturally reach, click on the **No Limit** radio button.

     If you want to apply a scaling factor to the Z-axis of the shape, click to select the **Scale** option, and then type a value in the **Scale** box, or click and drag on the slider that is activated.
   - To allow the shape to grow to a specific height and then plateau, click on the **Limit To Height** radio button, and then type a value in the **Height** box that is activated.

     If the natural height of the shape exceeds that of the value you type in the **Height** box, a flat top appears on the shape.

     If you want to apply a scaling factor to the shape in its Z-axis, click to select the **Scale** option, and then type a value in the **Scale** box or click and drag on the slider that is activated.

- To allow the shape to grow to a specific height by applying a scaling factor in its Z-axis, click on the **Scale To Height** radio button , and then type a value in the **Height** box that is activated. This option does not produce a flat top on the shape.

Note: If you select the **Plane** option, all of the height options are greyed-out. If so, go straight to the next step.

---

6. Click on the **Apply** button to apply the settings you have made to the selected colour. The shape that you have chosen to create is shown in the selected colour within the Colour Palette beneath the **2D View** window:

Selected Colour To Which Relief Is Applied

7. Select the relief combination method that you want to use:
    - Click on the **Add** button to add the points in the shape to the current relief. For more details, see "Adding to the Relief" on page 273.
    - Click on the **Subtract** button to subtract the points in the shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
    - Click on the **Zero** button to reset all areas of the current relief under the current Primary Colour to zero.
    - Click on the **Merge High** button to merge the shape with the current relief, so that only the highest points of the two show. For more details, see "Merging with the Relief" on page 277.
    - Click on the **Merge Low** button to merge the shape with the current relief, so that only the lowest points of the two show. For more details, see "Merging with the Relief" on page 277.
    - Click on the **Zero Rest** button to reset all areas of the current relief other than those under the current Primary Colour to zero.

The **Calculating Relief** progress bar appears beneath the **2D View** window. You can stop the relief calculation process at any time by clicking on the button.

8. Click on the **Close** button to close the **Shape Editor** dialog box.

9. Click on the **3D View** button 3D in the **2D View** toolbar to view the relief.

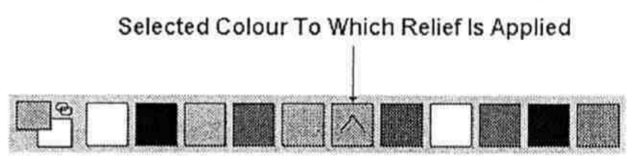
Editing the Shape Attributes

You can edit any of the shape attributes that you have applied to a colour in a bitmap image. Once you have adjusted the shape attributes you can then combine the new shape with the existing relief. It is important to remember that the new shape applied to the colour does not replace any part of the relief that was created from a shape previously applied to the same colour. The new shape can only be combined with the existing relief.

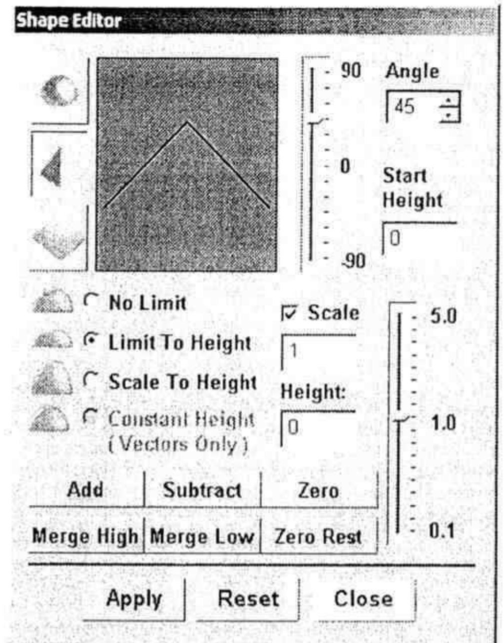
To edit the shape attributes:

1. Double-click on the colour in the Colour Palette for which you want to edit the shape attributes.

   You can see the profile of the shape in the selected colour within the Colour Palette:

   The **Shape Editor** dialog box appears showing the shape attributes currently applied to the selected colour:



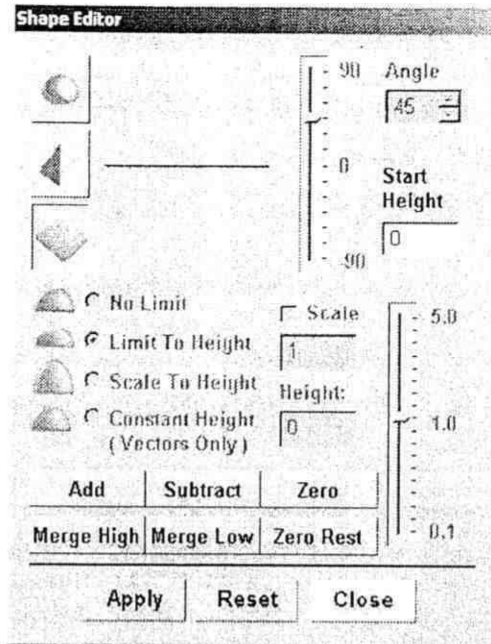
2. Follow the relevant steps in "Creating a Shape from a Bitmap" on page 232. By adjusting the current settings in the **Shape Editor** dialog box, you can change:
   - The profile of the shape.
   - The height of the shape.
   - The angle of the shape.
   - The start height of the shape.
   - The scale of the shape.
   - How the shape is combined with any existing relief.

Creating a Shape from a Closed Vector

You can create simple shapes from any closed vector object and combine them with the existing relief using the **Shape Editor** tool.

To create a shape from a closed vector object:

1. Select the vector object from which you want to create a shape. For details, see "Selecting Vectors" in the Working with Vectors chapter.
2. Right-click on the selected vector object to display the Vector Editing menu, and then click on the **Shape Editor...** option to display the **Shape Editor** dialog box:



Note: You can also display the **Shape Editor** dialog box by double-click on the vector object from which you want to create a shape. If you use this method, be careful not to move the vector object.

3. From the **Shape Editor** dialog box, select the profile of shape that you want to create from the selected vector object:

   - Click on the **Round** button  to create a rounded shape from the selected vector object.
   - Click on the **Angular** button  to create an angled shape from the selected vector object.
   - Click on the **Plane** button  to create a flat shape from the selected vector object.

Note: You can restore the default settings in the **Shape Editor** dialog box by clicking on the **Reset** button at any time.

4. If you want to create a rounded or an angled shape, you must define an angle for it using any of the following methods:

- Type a value in the **Angle** box. Type a positive value to produce a convex shape and a negative value to produce a concave shape. Type a value of 0 to produce a plane.

Note: If you have selected the **Plane** option, the **Angle** box is greyed-out. If so, go straight to the next step.

- Click and drag on the slider to the left of the **Angle** box. Drag the slider upward to increase the angle of the shape. Drag the slider downward to decrease it. The angle appears in the **Angle** box.
- Click on the up or down arrow on the right of the **Angle** box. The angle appears in the **Angle** box.

The default angle for a rounded or an angled shape is 45°.

5. Type a value in the **Start Height** box to define the height at which the shape starts in the Z direction. This creates a vertical side-wall beneath the shape. If you are creating a flat shape, this value controls the height of the plane.
6. Select the height option that you want to use:
   - To allow the shape to grow to the height that it would naturally reach, click on the **No Limit** radio button.

     If you want to apply a scaling factor to the Z-axis of the shape, click to select the **Scale** option, and then type a value in the **Scale** box, or click and drag on the slider that is activated.
   - To allow the shape to grow to a specific height and then plateau, click on the **Limit To Height** radio button, and then type a value in the **Height** box that is activated.

     If the natural height of the shape exceeds that of the value you type in the **Height** box, a flat top appears on the shape.

     If you want to apply a scaling factor to the shape in its Z-axis, click to select the **Scale** option, and then type a value in the **Scale** box or click and drag on the slider that is activated.

- To allow the shape to grow to a specific height by applying a scaling factor in its Z-axis, click on the **Scale To Height** radio button, and then type a value in the **Height** box that is activated. This option does not produce a flat top on the shape.
- To allow the shape to grow to a defined height, click on the **Constant Height (Vectors Only)** radio button, and then type a value in the **Height** box that is activated. This creates a shape where the angle or curvature changes to maintain a constant height, even where its width changes.

Note: If you have selected the **Plane** option, all of the height options are greyed-out. If so, go straight to the next step.

---

7. Click on the **Apply** button to apply the settings you have made to the selected vector object.
8. Select the relief combination method that you want to use:
    - Click on the **Add** button to add the points in the shape to the current relief. For more details, see "Adding to the Relief" on page 273.
    - Click on the **Subtract** button to subtract the points in the shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
    - Click on the **Zero** button to reset the relief within the area defined by the selected vector object to zero.
    - Click on the **Merge High** button to merge the shape with the current relief, so that only the highest points of the two show. For more details, see "Merging with the Relief" on page 277.
    - Click on the **Merge Low** button to merge the shape with the current relief, so that only the lowest points of the two show. For more details, see "Merging with the Relief" on page 277.
    - Click on the **Zero Rest** button to reset the current relief outside of the area defined by the selected vector object to zero.

The **Calculating Relief** progress bar appears beneath the **2D View** window. You can stop the relief calculation process at any time by clicking on the button.

9. Click on the **Close** button to close the **Shape Editor** dialog box.

10. Click on the **3D View** button 3D in the **2D View** toolbar to view the relief.

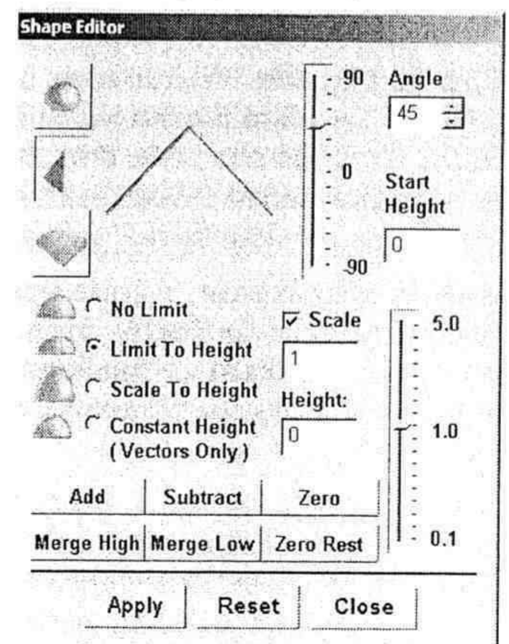
Editing the Shape Attributes

You can edit any of the shape attributes that you have applied to a vector object. Once you have adjusted the shape attributes you can then combine the new shape with the existing relief. It is important to remember that the new shape applied to the vector object does not replace any part of the relief that was created from a shape previously applied to the same vector object. The new shape can only be combined with the existing relief.

To edit the shape attributes:

1. Select the vector object for which you want to edit its shape attributes. For details, see "Selecting Vectors" in the Working with Vectors chapter.

2. Right-click on the selected vector object to display the Vector Editing menu, and then click on the **Shape Editor** option to display the **Shape Editor** dialog box:



3. Follow the relevant steps in "Creating a Shape from a Closed Vector" on page 237. By adjusting the current settings in the **Shape Editor** dialog box, you can change:
    - The profile of the shape.
    - The height of the shape.
    - The angle of the shape.
    - The start height of the shape.
    - The scale of the shape.
    - How the shape is combined with any existing relief.

Creating a Shape Using Vectors

You can create freeform shapes using vector objects and combine them with any existing relief in your model.

The freeform shapes that you can create using vector objects are as follows:

- **Swept Profile**. You can create three different types of swept profile shape using the **Swept Profiles Wizard** or any one of the three buttons in the **Vector Based Relief Creation** area of the **Assistant**'s Home page. You can **Extrude, Spin** or **Turn** a swept profile. For details, see "Creating a Swept Profile Shape" on page 243.
- **Two-Rail Sweep**. You can create a two-rail swept shape using the **Two Rail Sweep** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page and between three and five vector objects. For details, see "Creating a Two Rail Sweep" on page 258.
- **Weave**. You can create a weave shape using the **Weave Wizard** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page and two or more vector objects. For details, see "Creating a Weave Shape" on page 263.
- **ISO-Form Letters**. You can create ISO-Form letters using the **ISO-Form Letter** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page and at least one vector object. For details, see "Creating ISO-FORM Letters" on page 267.

Creating a Swept Profile Shape

A swept profile shape can be created using vector objects. You can create a swept profile shape in three different ways:

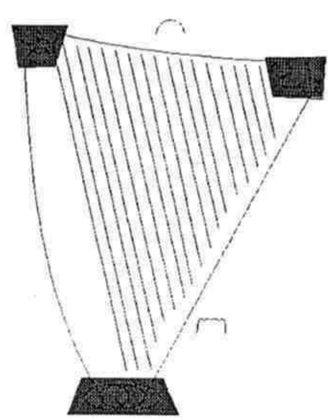
- **Extrude**. For details, see "Extruding a Shape" on page 243.
- **Spin**. For details, see "Spinning a Shape" on page 249.
- **Turn**. For details, see "Turning a Shape" on page 255.

Extruding a Shape

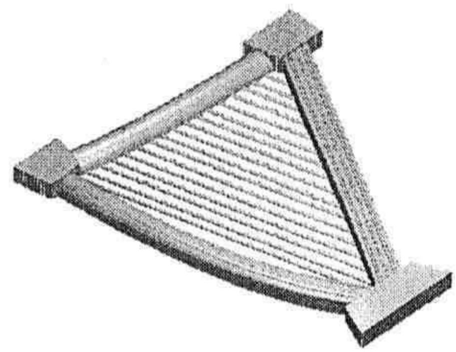
You can extrude a shape using up to four vector objects. The first vector object defines the line along which the shape is extruded. This is referred to as the drive curve. The second defines the start cross-section of the shape, while the third defines the end cross-section. The vector object used to define the end cross-section can be the same as that used to define the start cross-section. The optional fourth vector object determines the height of the shape in the Z direction.

In the following example, you can see how shapes can be extruded from vector objects to form a harp:

Before...

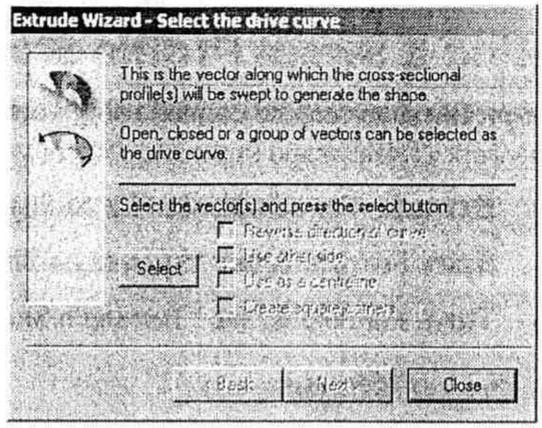


After...



To extrude a shape:

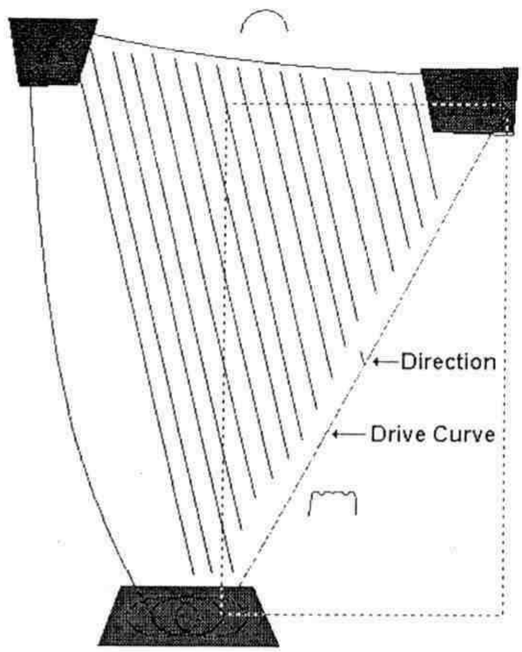
1. Click on the **Extrude** button  in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Extrude Wizard**:



Note: You can also display the **Extrude Wizard** from the Main menu bar. Click on the **Relief** menu, and then on the **Swept Profiles Wizard...** option to display the **Swept Profiles Wizard**. Click on the **Extrude** radio button, and then on the **Next** button to display the **Extrude Wizard**.

2. Click to select the vector object in the **2D View** window along which you want the cross-section to be swept. This is referred to as the drive curve. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **Select** button. Arrows are displayed along the selected vector object to show the direction of the drive curve and on which side the cross-sections are to be attached.

   In order to begin creating the frame of the harp in our example, the following polyline is selected as the drive curve:



4. If you want to change the properties of the drive curve, click on the options that you want to use:

   - To reverse the direction of the drive curve, click to select the **Reverse direction of curve** option ☑. The arrows along the selected vector object change direction.
   - To change the side of the drive curve along which the cross-section is extruded, click to select the **Use other side** option ☑. The arrows along the selected vector object change side.

Note: The **Use other side** option is greyed-out if you have the **Use as a centreline** option selected ☑.

   - If you want to use the drive curve as the centreline for the extrusion, click to select the **Use as a centreline** option ☑. Arrows are dawn along the centreline of the selected vector object.
   - If you want to produce a drive curve with sharp-edged corners, click to select the **Create square corners** option ☑.

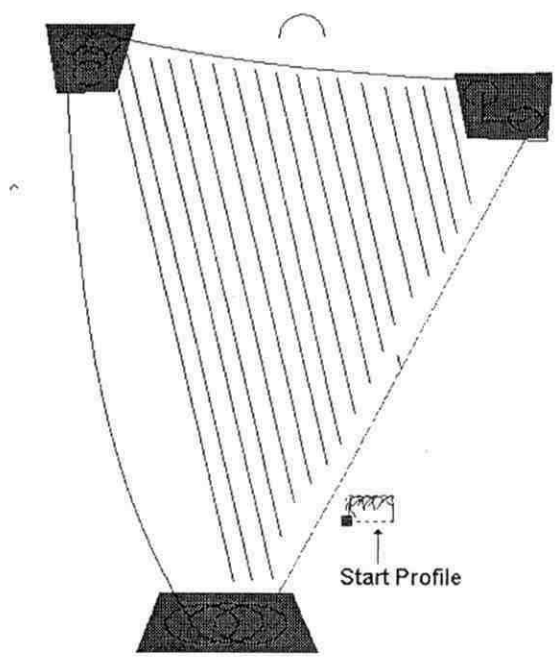
5. Click on the **Next** button. The drive curve turns red.

Tip: If you want to change any of the settings after you have clicked on the **Next** button, click on the **Back** button to return to the previous page in the **Extrude Wizard**.

6. Click to select the open, ungrouped vector object in the **2D View** window that you want to use as the cross-section at the start of the extruded shape. This is referred to as the start profile.

7. Click on the **Select** button. The arrows on the start profile show the current Z-axis direction. The side of the start profile on which the arrows are positioned determine the side on which the shape is extruded.

   In our example, the following vector object is selected as the start profile:



8. If you want to change the properties of the start profile, click on the options that you want to use:

   - To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option. The start point (node) is green and determines which end of the start profile is attached to the drive curve.

- To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option ☑.

9. Click on the **Next** button. The start profile turns blue.
10. Click to select the open, ungrouped vector object in the **2D View** window that you want to use as the profile at the end of the extruded shape:

    - If you want to use the same vector object that you had selected as the profile at the start of the extruded shape, make sure that the **End profile is the same as the start profile** option is selected ☑, and then go straight to the next step.

      In our example, the vector object used for the end profile is the same as that which was selected as the start profile.

Note: The **Select** button and the end profile options are greyed-out if the **End profile is the same as the start profile** option is selected ☑.

- If you want to use another vector object in the model, make sure that the **End profile is the same as the start profile** option is deselected ☐, click on the vector object that you want to use, and then click on the **Select** button.

If you want to change the properties of the end profile, click on the options that you want to use.

- To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option ☑. The start point (node) is green and determines which end of the end profile is attached to the drive curve.
- To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option ☑.

11. Click on the **Next** button. The end profile turns blue.
12. To add a contour and scale the swept profile in the Z-axis along its length:

- First, click to select the **Use a z modulation vector** option ☑.
- Next, click to select the vector object that you want to use as the z modulation vector.
- Next, click on the **Select** button.
- To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option ☑. The start point (node) is green and determines which end of the swept profile is attached to the drive curve.
- To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option ☑.

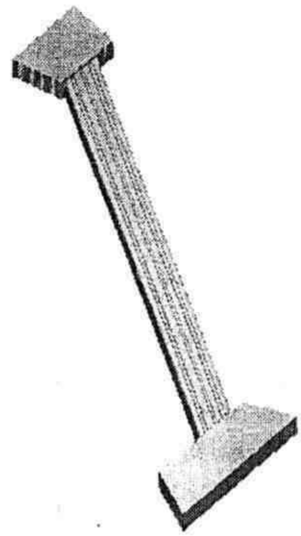
If you do not want to scale the swept profile in the Z-axis along its length, go straight to the next step.

13. Click on the **Next** button. The z modulation vector turns green.
14. Select the relief combination method that you want to use:
    - Click on the **Add** radio button ◉ to add the extruded shape to the current relief. For more details, see "Adding to the Relief" on page 273.
    - Click on the **Subtract** radio button ◉ to subtract the extruded shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
    - Click on the **Merge Highest** radio button ◉ to merge the extruded shape with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
    - Click on the **Merge Lowest** radio button ◉ to merge the extruded shape with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.

    In our example, the **Merge Highest** option is used.
15. Click on the **Extrude** button to combine the extruded shape with any existing relief in your model.

16. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. You can now see the relief.

    In our example, the relief appears in the **3D View** window as follows:



    The blocks at the top and bottom of the harp frame in our example are created by applying a flat shape with a start height of 3.5 mm (0.14") to a bitmap colour marking the shape of the blocks, and then using the **Replace Relief** button to add the shapes to the relief. For more information, see "Creating a Shape from a Bitmap" on page 232 and "Replacing the Relief" on page 272.

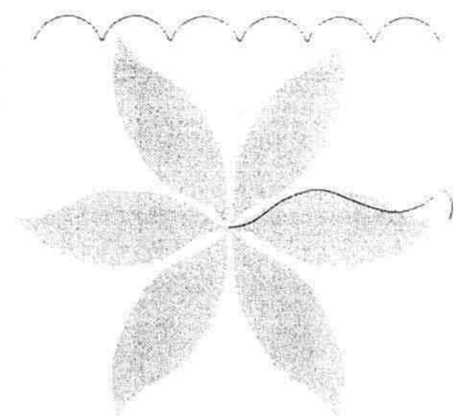
17. Click on the **2D View** button 2D in the **3D View** toolbar to return to the **2D View** window.
18. Click on the **Close** button to close the **Extrude Wizard**.

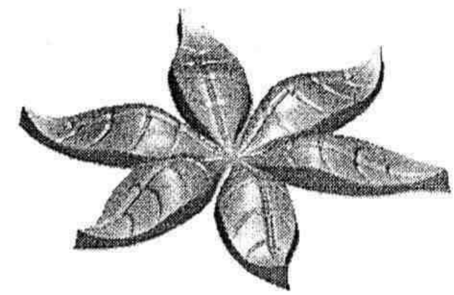
Spinning a Shape

You can spin a shape using up to three vector objects. The first of these defines the start cross-section and the second defines the end cross-section of the shape. The vector object used to define the end cross-section can be the same as that used to define the start cross-section. The optional third vector object determines the height of the shape in the Z direction.

In the following example, you can see how a shape can be spun from vector objects, along with other relief editing techniques, to form a group of leaves:

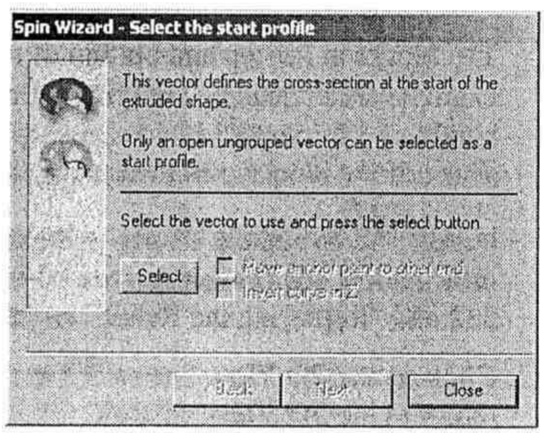
Before... *After...*





To create a spun shape:

1. Click on the **Spin** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Spin Wizard**:

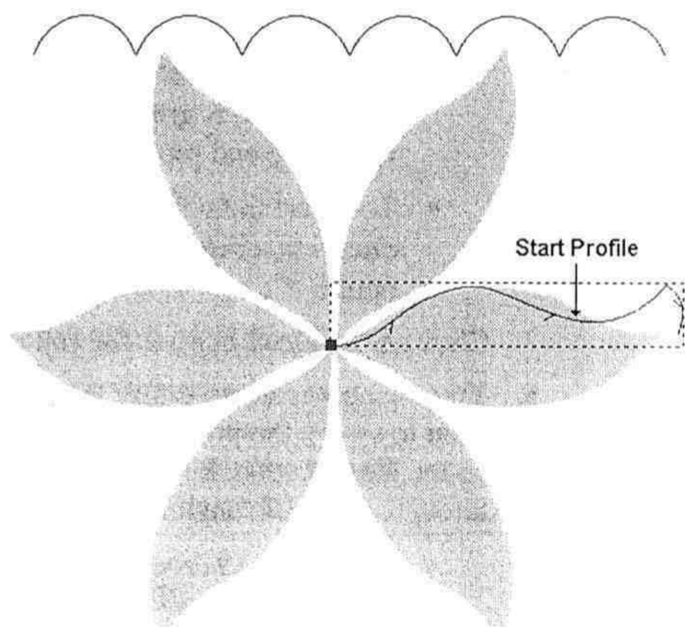


Note: You can also display the **Spin Wizard** from the Main menu bar. Click on the **Relief** menu, and then on the **Swept Profiles Wizard...** option to display the **Swept Profiles Wizard**. Click on the **Spin** radio button, and then on the **Next** button to display the **Spin Wizard**.

2. Click to select the open, ungrouped vector object in the **2D View** window that you want to use as the cross-section at the start of the spun shape. This is referred to as the start profile. For details, see "Selecting Vectors" in the Working with Vectors chapter.

3. Click on the **Select** button. Arrows are displayed about the selected vector object to show the side on which the cross-sections are to be attached.
4. If you want to change the properties of the start profile, select the options that you want to use:
    - To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option. The start point (node) is green and determines the point around which the start profile is spun.
    - To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option.
5. Click on the **Next** button. The start profile turns blue.

    In order to begin creating the leaf design in our example, the following polyline is selected as the start profile:



Tip: If you want to change any of the settings after you have clicked on the **Next** button, click on the **Back** button to return to the previous page in the **Spin Wizard**.

6. Click to select the open vector object that you want to use as the cross-section at the end of the spun shape. This is referred to as the end profile.

- To use the same vector object that you have already used, make sure that the **End profile is the same as the start profile** option is selected ☑, and then go straight to the next step.

In our example, the vector object used for the end profile is the same as that which was selected as the start profile.

Note: The **Select** button and the end profile options are greyed-out if the **End profile is the same as the start profile** option is selected ☑.

- To use another vector object, make sure that the **End profile is the same as the start profile** option is deselected ☐, click to select the vector object that you want to use, and then click on the **Select** button.

If you want to change the properties of the end profile, select the options that you want to use:

- To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option ☑. The start point (node) is green and determines the point around which the end profile is spun.
- To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option ☑.

7. Click on the **Next** button. The end profile turns blue.
8. You are now ready to define how the profile is swept. If you want to sweep the profile through 360°, starting at 0°, click on the **Next** button. If you want to sweep the profile through a specific angle:
   - First, click to deselect the **Sweep through 360 degrees** option ☐.
   - Next, type the start angle in the **Start Angle** box.
   - Now, click on the spin direction radio button ◉ that you want to use. Click to select **Clockwise** if you want to sweep the profile in a clockwise direction about the start point (node). Click to select **Anti-Clockwise** if you want to sweep the profile in an anti-clockwise direction about the start point (node).

- Next, type the finish angle in the **Finish Angle** box.
- Finally, click on the **Next** button.

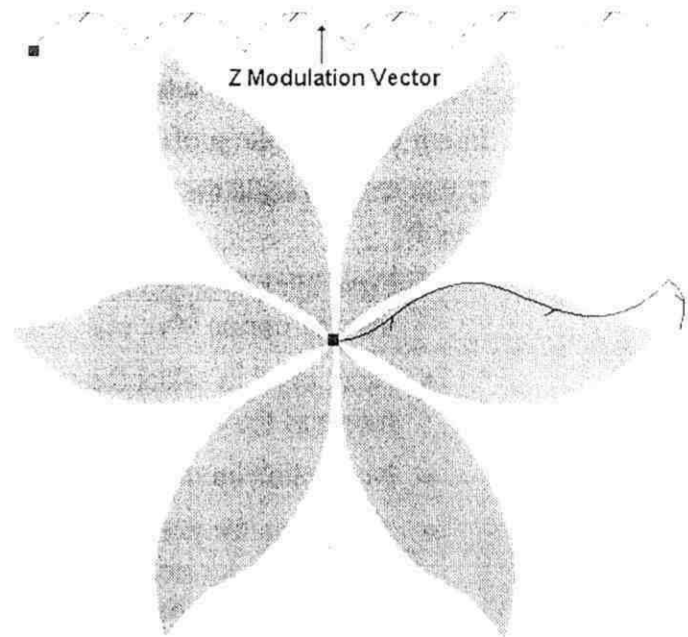
In our example, the profile is swept through 360°.

9. If you want to scale the spun profile in the Z-axis along its length:

    - First, click to select the **Use a z modulation vector** option.
    - Next, click to select the vector object that you want to use as the z modulation vector.
    - Next, click on the **Select** button.
    - To change the position of the start point (node) in the selected vector object, click to select the **Move anchor point to other end** option. The start point (node) is green and determines the point around which the end profile is spun.
    - To invert the selected vector object in the Z-axis direction, click to select the **Invert curve in Z** option.

    If you do not want to scale the spun profile in the Z-axis along its length, go straight to the next step.

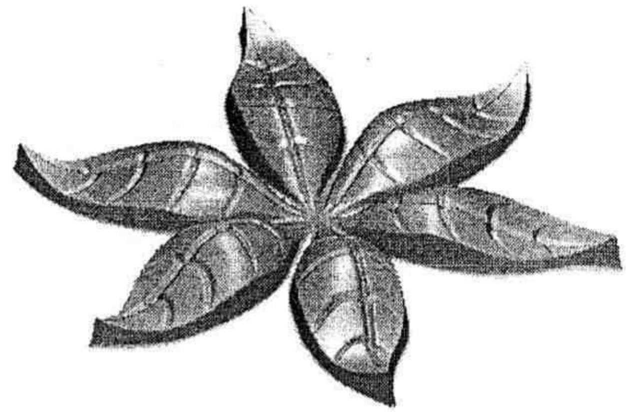
    In our example, the following polyline is selected as the z modulation vector:



10. Click on the **Next** button. The z modulation vector turns green.
11. Select the relief combination method that you want to use:
    - Click on the **Add** radio button to add the spun shape to the current relief. For more details, see "Adding to the Relief" on page 273.
    - Click on the **Subtract** radio button to subtract the spun shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
    - Click on the **Merge Highest** radio button to merge the spun shape with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
    - Click on the **Merge Lowest** radio button to merge the spun shape with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.

    In our example, the **Add** option is used.
12. Click on the **Spin** button to combine the spun shape with any existing relief in your model.
13. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. You can now see the relief.

In our example, the relief appears in the **3D View** window as follows:



The wave in the spun relief is achieved by using the z modulation vector.

The shape of the leaves in the relief is created by selecting green as the Primary Colour from the **2D View** window and then using the **Keep Under Colour** option from the **Relief** menu. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter and "Resetting the Relief Height" on page 301.

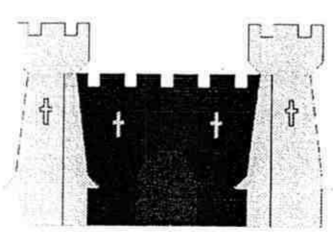
14. Click on the **2D View** button 2D in the **3D View** toolbar to return to the **2D View** window.
15. Click on the **Close** button to close the **Spin Wizard**.

Turning a Shape

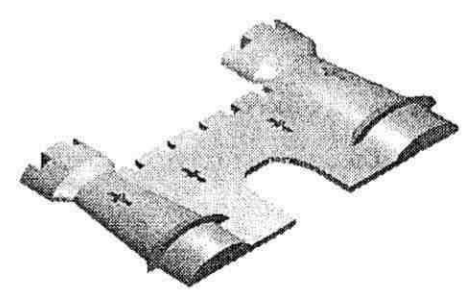
You can turn a shape using a vector object. An imaginary line between the start point (node) and end point (node) in the selected vector object acts as an axis about which it is turned to create the cross-section of the shape.

In the following example, you can see how a shape can be turned using vector objects to form castle turrets:

Before...

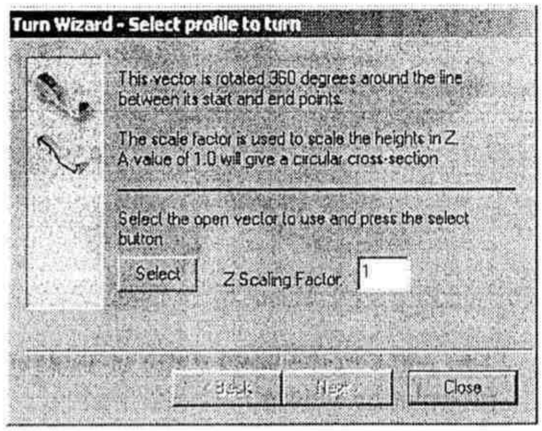


After...



To create a turned shape:

1. Click on the **Turn** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Turn Wizard**:

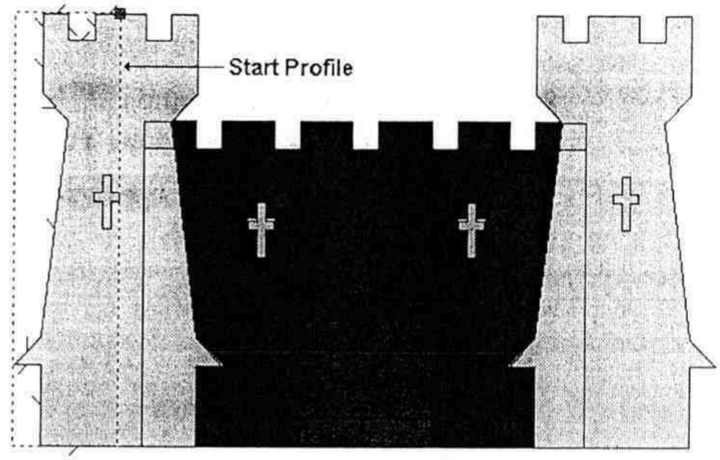


Note: You can also display the **Turn Wizard** from the Main menu bar. Click on the **Relief** menu, and then on the **Swept Profiles Wizard...** option to display the **Swept Profiles Wizard**. Click on the **Turn** radio button, and then on the **Next** button to display the **Turn Wizard**.

2. Click to select the open vector object that you want to use to create the cross-section of the shape. This is referred to as the profile. For details, see "Selecting Vectors" in the Working with Vectors chapter.

3. Click on the **Select** button. Arrows are displayed about the selected vector object to show the side on which the cross-section is to be attached.

   In order to begin creating castle turrets in our example, the following polyline is selected as the start profile:



4. Type a value in the **Z Scaling Factor** box to scale 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 start profile.

5. Click on the **Next** button. The profile turns blue.

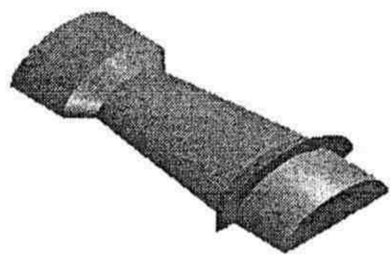
Tip: If you want to change any of the settings after you have clicked on the **Next** button, click on the **Back** button to return to the previous page in the **Turn Wizard**.

6. Select the relief combination method that you want to use:

   - Click on the **Add** radio button to add the turned shape to the current relief. For more details, see "Adding to the Relief" on page 273.
   - Click on the **Subtract** radio button to subtract the turned shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
   - Click on the **Merge Highest** radio button to merge the turned shape with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
   - Click on the **Merge Lowest** radio button to merge the turned shape with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.

7. Click on the **Turn** button to combine the turned shape with any existing relief in your model.
8. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can now see the relief.

   In our example, the relief appears in the **3D View** window as follows:

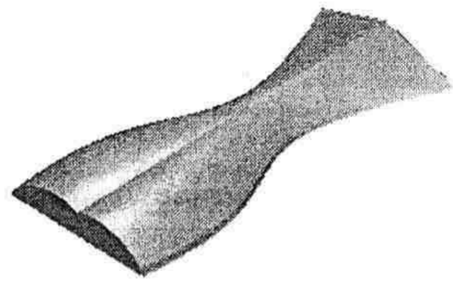
   The crenellations within the castle turrets are created by selecting white as the Primary Colour from the **2D View** window, and then using the **Zero Relief Under Colour** button in the **Relief Editing** area. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter and "Resetting the Relief Height" on page 301.
9. Click on the **2D View** button in the **3D View** toolbar to return to the **2D View** window.
10. Click on the **Close** button to close the **Turn Wizard**.

Creating a Two Rail Sweep

You can create a two-rail swept shape using up to five vector objects. The first two vector objects define the lines along which the shape is extruded. They are referred to as the drive rails. The third defines the start cross-section of the shape, while the fourth defines the end cross-section. The vector object used to define the end cross-section can be the same as that used to define the start cross-section. The optional fifth vector object determines the height of the shape in the Z direction.

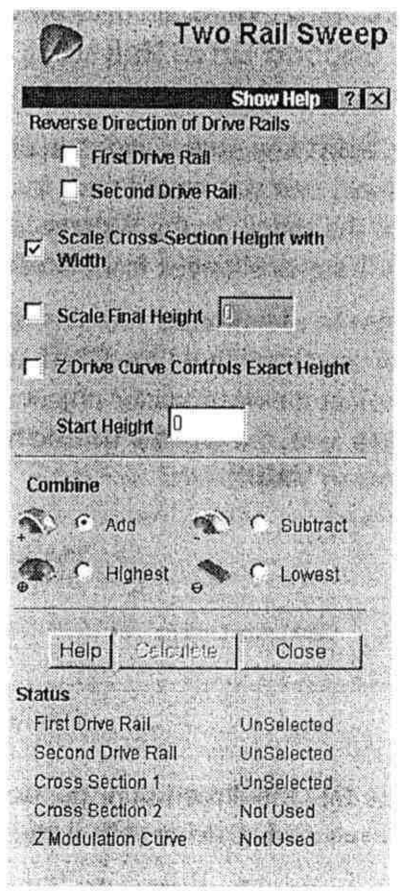
In the following example, you can see how a shape can be swept along vector objects to form a vase:

Before... *After...*



To create a two-rail swept shape:

1. Click on the **Two Rail Sweep** button  in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Two Rail Sweep** page:

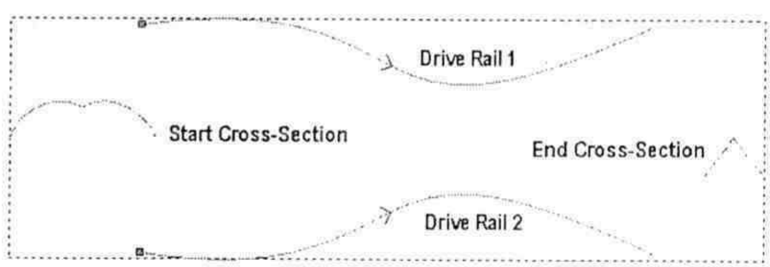


Note: You can also display the **Two Rail Sweep** page rom the Main menu bar. Click on the **Relief** menu, and then on the **Two Rail Sweep...** option.

2. Click to select the vector object along which you want the cross-section to be swept. For details, see "Selecting Vectors" in the Working with Vectors chapter. This is referred to as the first drive rail. In the **Status** area, the **First Drive Rail** status changes from **UnSelected** to **Valid**.

Warning: If you select a closed vector object for the drive rail you can only select one vector object for the cross-section. This affects the appearance of the two-rail swept shape considerably.

3. Hold the **Shift** key down on your keyboard, and then click to select the vector object along which you want the second cross-section to be swept. This is referred to as the second drive rail. Arrows appear on the two vector objects selected as the drive rails to indicate their direction. In the **Status** area, the **Second Drive Rail** status changes from **UnSelected** to **Valid**.
4. Hold the **Shift** key down, and then click to select the open vector object that you want to use to create the start cross-section of the shape. In the **Status** area, the **Cross Section 1** status changes from **UnSelected** to **Valid**.
5. If you want to use another vector object for the end cross-section of the shape, hold the **Shift** key down, and then click to select the open vector object that you want to use. In the **Status** area, the **Cross Section 2** status changes from **Not Used** to **Valid**.



6. Make sure that the direction of the two vector objects that you have selected as drive rails is the same:
   - If you need to reverse the direction of the first drive rail, click to select the **First Drive Rail** option ☑.
   - If you need to reverse the direction of the second drive rail, click to select the **Second Drive Rail** option ☑.

7. You are now ready to decide how you want to scale the swept shape. If you want to scale the two-rail swept profile in the Z-axis along its length:

    - First, click to select the **Z Drive Curve Controls Exact Height** option ☑.
    - Next, click to select the vector object that you want to use as the z modulation curve. The z modulation vector turns magenta. In the **Status** area, the **Z Modulation Curve** status changes from **Not Used** to **Valid**.

    If you do not want to use a vector object to determine the height of the swept shape, you can select an alternative scaling option:

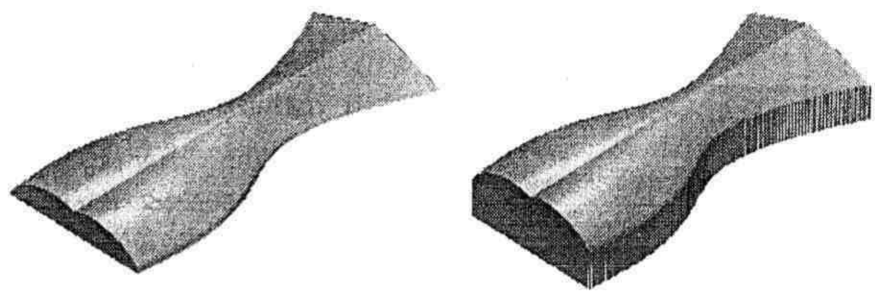
    - To scale the height of the cross-section across the two drive rails in proportion with its width, make sure that the **Scale Cross section Height with Width** option is selected ☑.

Note: Narrow sections in the swept shape appear low, while wide sections appear high.

    - If you want the height of the cross section across the two drive rails to remain constant, click to deselect the **Scale Cross section Height with Width** ☐.
    - To define the final height of the swept shape before it is combined with any existing relief, click to select the **Scale Final Height** option ☑, and then type a value in its box to set the height of the shape.

8. If you want to add a start height to the two-rail swept shape, type the height in the **Start Height** box. This produces a base that is combined with the swept shape after it has been calculated. The start height is not included in the scaled final height of the swept shape.

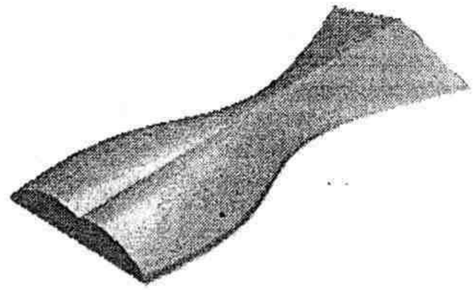
No Start Height... *2mm Start Height...*



9. In the **Combine** area of the page, select the relief combination method that you want to use:
   - Click on the **Add** radio button to add the two-rail swept shape to the current relief. For more details, see "Adding to the Relief" on page 273.
   - Click on the **Subtract** radio button to subtract the two-rail swept shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
   - Click on the **Highest** radio button to merge the two-rail swept shape with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
   - Click on the **Lowest** radio button to merge the two-rail swept shape with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.

   In our example, the **Add** option is used.
10. Click on the **Calculate** button to combine the two-rail swept shape with any existing relief in your model.
11. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. You can now see the relief.

    In our example, the relief appears in the **3D View** window as follows:



12. Click on the **2D View** button 2D in the **3D View** toolbar to return to the **2D View** window.
13. Click on the **Close** button to return to the **Assistant**'s Home page.

Help with Creating a Two Rail Sweep

You can display information on how to create a two-rail sweep if you click on Show Help at the top of the **Two Rail Sweep** page. However, if you would like more information on creating a two-rail sweep:

1. Click on the **Help** button at the bottom of the **Two Rail Sweep** page.
2. Click on the **Close** button to close the **Two Rail Sweep** Help window.

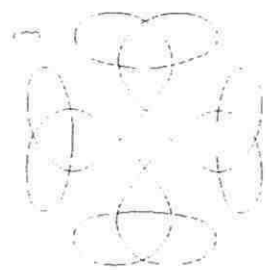
Creating a Weave Shape

A weave shape is made up of overlapping swept profiles. You can create a weave shape using at least two vector objects.

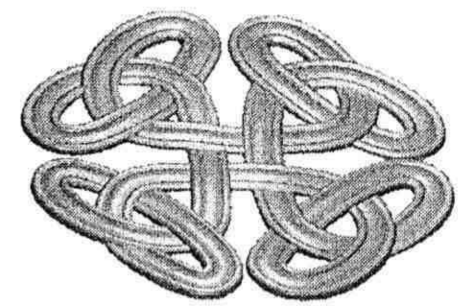
The vector objects you select for the drive rails must overlap themselves and represent the pattern in the weave shape you want to create. The last vector object you select defines the cross-section of the weave shape. ArtCAM Pro automatically extrudes the cross-section along the drive rails, scaling its height up and down at each intersection to provide a perfect inter-linked, over-and-under weave effect.

In the following example, you can see how a weave shape is created by extruding it along a vector object in the shape of a weave pattern.

Before...



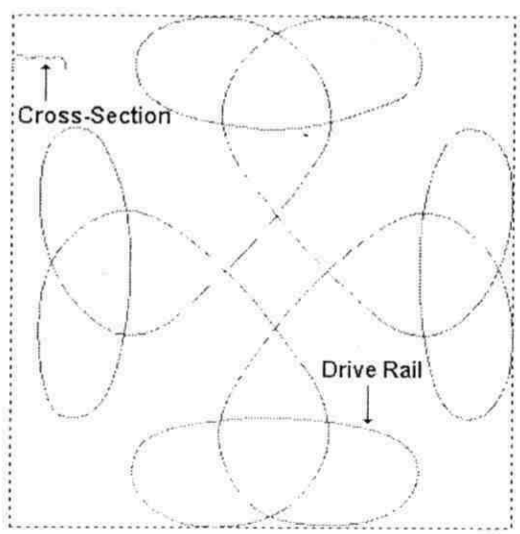
After...



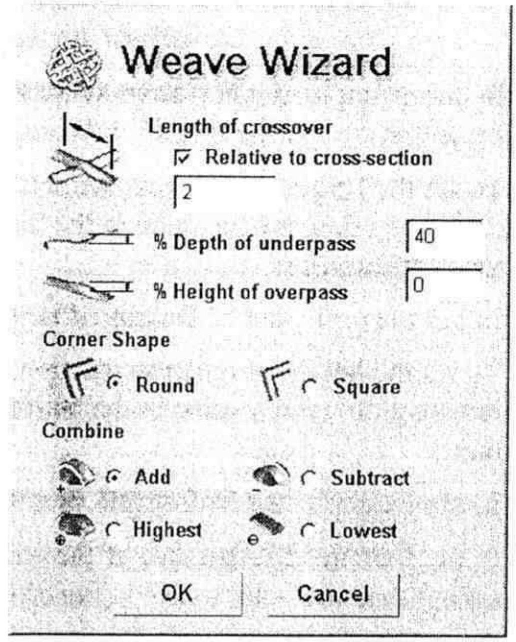
To create a weave shape:

1. Hold the **Shift** key down on your keyboard, and then click to select the vector objects along which you want the cross-section to be extruded. For details, see "Selecting Vectors" in the Working with Vectors chapter. These are referred to as the drive rails.
2. Hold the **Shift** key down, and then click to select the open vector object that you want to use for the cross-section in the weave shape.

   In order to begin creating the weave shape in our example, the following vector objects are selected:

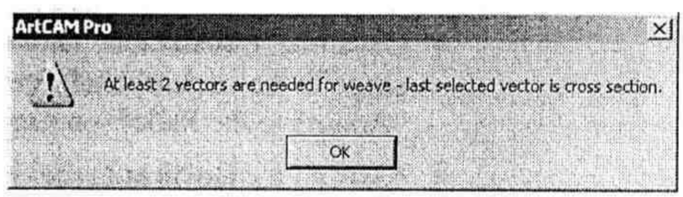
   

3. Click on the **Weave Wizard** button  in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Weave Wizard**:



Note: You can also display the **Weave Wizard** from the Main menu bar. Click on the **Relief** menu, and then on the **Weave Wizard...** option.

If you try to open the **Weave Wizard** before selecting at least two vector objects, the following error message is displayed:



Click on the **OK** button to close the message box, make sure that you have selected vector objects representing at least one drive rail and a cross-section and then repeat this step.

4. You can now set the length of the section within the drive curve that underlies or overlaps at intersections. This is referred to as the crossover.

    - To set the length of the crossover as a multiple of the cross-section's length, click to select the **Relative to cross-section** option ☑, and then type a value in the **Length of crossover** box.
    - To set the length of the crossover as an absolute distance, make sure that the **Relative to cross-**

section option is deselected ☐, and then type a value in the **Length of crossover** box.

In our example, the **Relative to cross-section** option is turned on ☑ and the length of the crossover is set to 4.

5. To set the height of the cross-section where it underlies at an intersection, type a value in the **% Depth of underpass** box.

   In our example, the **% Depth of underpass** is set to 40.

6. To set the height of the cross-section where it overlaps at an intersection, type a value in the **% Height of overpass** box.

   In our example, the **% Height of overpass** is set to 20.

7. In the **Corner Shape** area of the page, select the type of corner that you want to use in the cross-section:

   - To apply rounded corners to the cross-section, click on the **Round** radio button ◉.
   - To apply squared corners to the cross-section, click on the **Square** radio button ◉.

   In our example, **Round** corners are selected

8. In the **Combine** area of the page, select the relief combination method that you want to use:

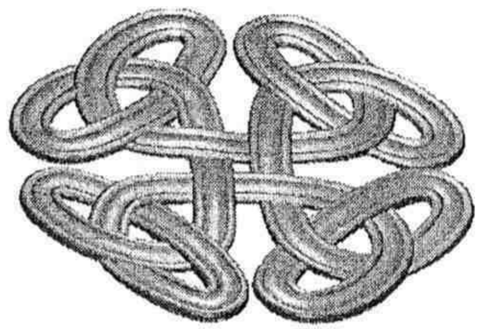
   - Click on the **Add** radio button ◉ to add the weave shape to the current relief. For more details, see "Adding to the Relief" on page 273.
   - Click on the **Subtract** radio button ◉ to subtract the weave shape from the current relief. For more details, see "Subtracting from the Relief" on page 275.
   - Click on the **Highest** radio button ◉ to merge the weave shape with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
   - Click on the **Lowest** radio button ◉ to merge the weave shape with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.

   In our example, the **Add** option is used.

9. Click on the **OK** button to close the **Weave Wizard** and combine the weave shape with any existing relief in your model.

10. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can now see the relief.

    In our example, the relief appears in the **3D View** window as follows:



11. Click on the **2D View** button in the **3D View** toolbar to return to the **2D View** window.

Creating ISO-FORM Letters

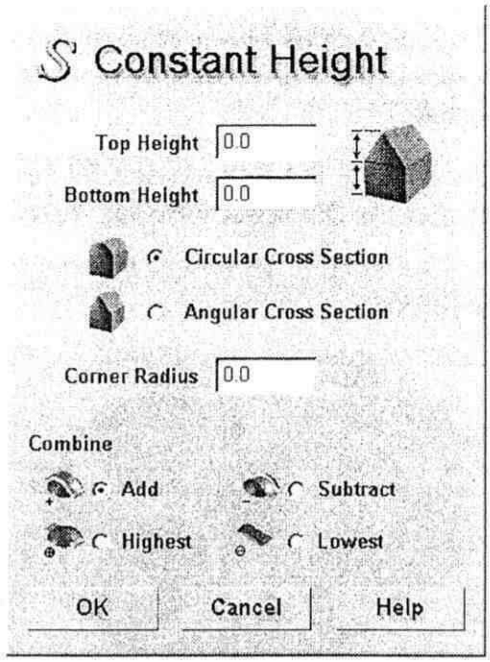
You can create ISO-FORM lettering, a shape of a constant height, from any closed vector object. You can control if the lettering has either a curved or an angled face. You can also control the radius of the corners in the lettering.

An ISO-Form letter is essentially made up of two parts, a **Bottom Height** and a **Top Height**. The former describes the vertical height (side wall) in the base of the letter, while the latter describes the height in the angled or curved cross-section added to its base to complete the letter.

To create ISO-FORM lettering:

1. Click to select the vector text from which you want to create ISO-FORM letters. For details, see "Selecting Vectors" and "Creating Vector Text" in the Working with Vectors chapter.

2. Click on the **ISO-FORM Letters** button in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Constant Height** dialog box:



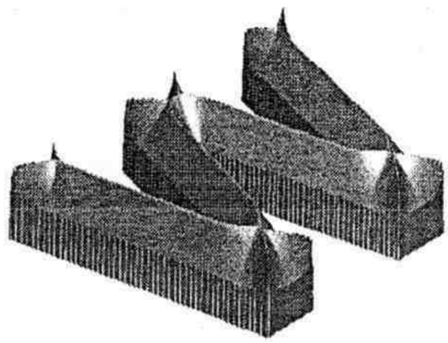
Note: You can also display the **Constant Height** dialog box from the Main menu bar. Click on the **Relief** menu, and then on the **ISO-FORM Letters...** option.

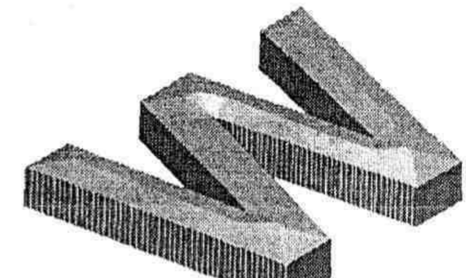
3. Type a value in the **Top Height** box to set the height of the cross-section in the ISO-FORM letters.
4. Type a value in the **Bottom Height** box to set the height of the base in the ISO-FORM letters
5. Click on either of the radio buttons to select the type of cross-section that you want to create:
   - **Circular Cross-Section** – This option allows you to create ISO-FORM letters with a curved cross-section.
   - **Angular Cross-Section** – This option allows you to create ISO-FORM letters with an angled-edged cross-section.
6. Type a value in the **Corner Radius** box to set the radius of the corners in your ISO-FORM lettering. This allows you to taper all areas of the constant height relief that are of a smaller radius than that which is defined in the **Corner Radius** box.

   In the following example, you can see the difference that defining a corner radius makes to the serifs in this ISO-

FORM letter with a constant height of 6 mm and an angular cross-section.

Corner Radius at 0.0... *Corner Radius at 12.0...*





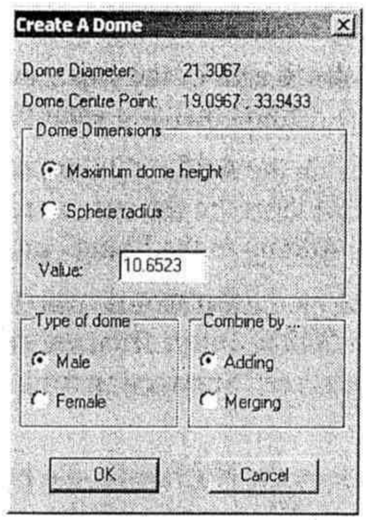
7. In the **Combine** area of the page, select the relief combination method that you want to use:
    - Click on the **Add** radio button to add the ISO-FORM letters to the current relief. For more details, see "Adding to the Relief" on page 273.
    - Click on the **Subtract** radio button to subtract the ISO-FORM letters from the current relief. For more details, see "Subtracting from the Relief" on page 275.
    - Click on the **Highest** radio button to merge the ISO-FORM letters with the current relief, so that only the highest points of the two show. For details, see "Merging with the Relief" on page 277.
    - Click on the **Lowest** radio button to merge the ISO-FORM letters with the current relief, so that only the lowest points of the two show. For details, see "Merging with the Relief" on page 277.
8. Click on the **OK** button to combine the ISO-FORM letters with any existing relief in your model and return to the **Assistant**'s Home page.

Creating a Dome

You can create a concave or convex dome shape and combine it with any existing relief in your model.

To create a dome shape:

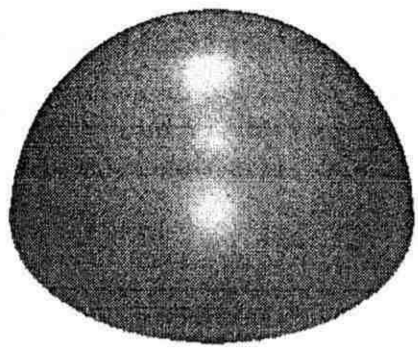
1. Click on the **Selection Rectangle** button in the **File** toolbar. For details, see "Using the Selection Rectangle" in the Working with chapter.
2. Move the cursor to the position in the model (the white area) in which you want to create the selection rectangle, and then click and drag. Release the mouse button when you have set its size.
3. From the Main menu bar, click on the **Relief** menu, and then on the **Create Dome** option to display the **Create A Dome** dialog box:



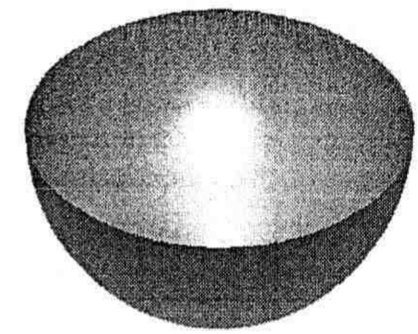
4. In the **Dome Dimensions** area, click on either of the radio buttons to define the dimensions of the dome:
   - **Maximum dome height** – This option allows you to create a dome at the maximum height relative to the diameter of the dome shown in the **Dome Diameter** area. The shape of the dome represents a complete half-sphere. If you want to reduce the height of the dome, set its maximum height using the **Value** box.
   - **Sphere radius** – This option allows you to define the spherical radius of the dome, using the **Value** box.
5. In the **Type of Dome** area, click on either of the radio buttons to define the type of dome you want to create:
   - **Male** – This option allows you to create a convex (raised) dome.

- **Female** – This option allows you to create a concave (recessed) dome.

Male Dome…



Female Dome…



6. In the **Combine By…** area, click on either of the radio buttons to select how the dome is combined with the existing relief:
    - **Adding** – This option allows you to add the dome shape to the current relief. For more details, see “Adding to the Relief” on page 273.
    - **Merging** – This option allows you to merge the the dome shape with the current relief, so that only the highest points of the two show. For more details, see “Merging with the Relief” on page 277.
7. Click on the **OK** button to combine the dome with any existing relief in your model and close the **Create A Dome** dialog box.

Creating a Feature

You can combine three different types of feature with the existing relief in a model using a vector object, usually vector text:

- Raised Feature. For details, see "Creating a Raised Feature" in the Working with Vectors chapter.
- Recessed Feature. For details, see "Creating a Recessed Feature" in the Working with Vectors chapter.
- Centreline Engraved Feature. For details, see "Creating a Centreline Engraved Feature" in the Working with Vectors chapter.

A depth or height is attributed to the vector object from which the feature is created. The height is added to, or the depth subtracted from, any existing relief in the model, and this forms the shape of the vector object, usually vector text, on the relief surface. The contour of the existing relief is preserved in the feature that is combined with it.

A feature is machined using a **Feature Machining** toolpath. For information on toolpaths, see "Overview" in the Machining Models chapter. For details on creating a **Feature Machining** toolpath, see "Feature Machining" in the Machining Models chapter.

Calculating a Relief

You can calculate a relief from the shape attributes you have assigned to a bitmap colour in the current **2D View** window using one of four different combination methods:

- **Replace** – You can replace the current relief with a shape to create a new relief. For details, see "Replacing the Relief" on page 272.
- **Add** – You can add a shape to the current relief to create a new relief. For details, see "Adding to the Relief" on page 273.
- **Subtract** – You can subtract a shape from the current relief to create a new relief. For details, see "Subtracting from the Relief" on page 275.
- **Merge** – You can merge a shape with the current relief, so that only the highest points of the two remain in a new relief. You can also merge a shape with the current relief, so that only the lowest points of the two remain in a new relief. For details, see "Merging with the Relief" on page 277.

These combination methods can be selected from the Main menu bar, by clicking on the **Relief** menu, and then the **Calculate** option.

Replacing the Relief

You can replace the current relief with a shape to create a new relief.

To replace the current relief with a shape:

1. Create the shape with which you want to replace the current relief. For details, see "Creating a Shape from a Bitmap" on page 232.

2. Click on the **Replace Relief** button in the **Relief Operations** area of the **Assistant**'s Home page to replace the current relief with the shape.

Note: You can also replace the current relief with a shape using the Main menu bar. Click on the **Relief** menu, followed by the **Calculate > Replace** option.

The **Calculating Relief** progress bar appears beneath the **2D View** window. You can cancel the relief replacement process at any time by clicking on the button.

3. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window.

You can also replace the current relief with a shape using the **Project** page:

1. Click on the **Project** tab Project to display the **Project** page.
2. Right-click on the current **2D View** listed in the **Views** area to display the context menu. Click on the **Calculate Relief** option, followed by the **Replace** option. For further details, see "Viewing Model Information" in the Working with Models chapter.

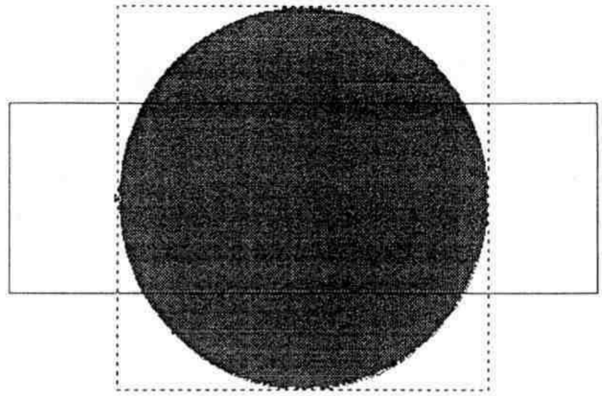
Adding to the Relief

You can add a shape to the current relief to create a new relief.

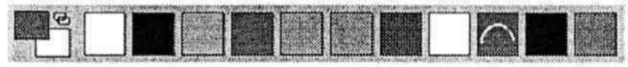
To add a shape to the current relief:

1. Create the shape that you want to add to the current relief. For details, see "Creating a Shape from a Bitmap" on page 232.

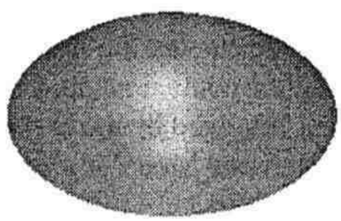
   In the following example, a blue circle is drawn in a model using the **Paint** tool:



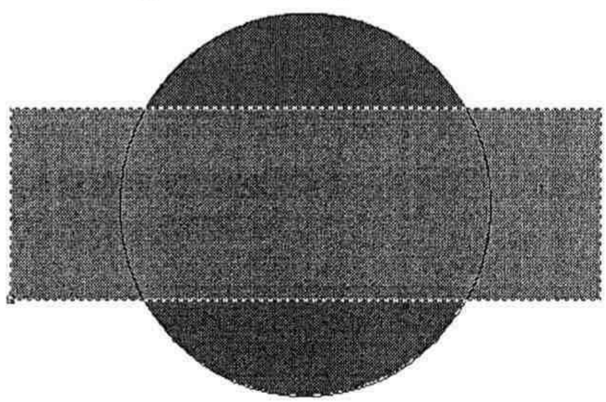
A rounded shape is then applied to the circle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:



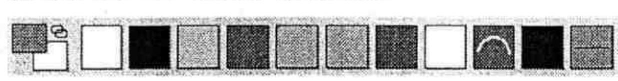
The calculated relief appears in the **3D View** window as follows:



The rectangular vector object is then flood-filled in red:



A plane is applied to the rectangle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:

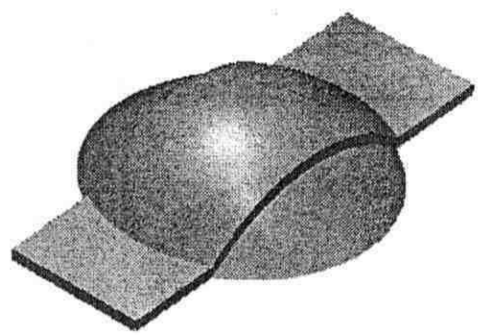


2. Click on the **Add Relief** button in the **Relief Operations** area of the **Assistant**'s Home page to add the shape to the current relief.

Note: You can also add a shape to the current relief using the Main menu bar. Click on the **Relief** menu, followed by the **Calculate > Add** option.

4. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window.

   In our example, the plane has been added to the existing relief to create the following new relief:



You can also add a shape to the current relief using the **Project** page:

1. Click on the **Project** tab to display the **Project** page.
2. Right-click on the current **2D View** listed in the **Views** area to display the context menu. Click on the **Calculate Relief** option, followed by the **Add** option. For further details, see "Viewing Model Information" in the Working with Models chapter.

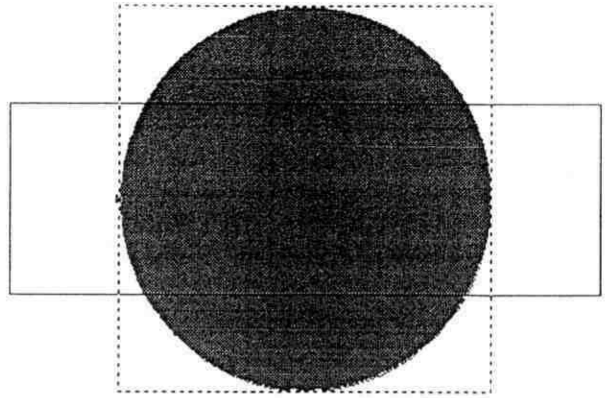
Subtracting from the Relief

You can subtract a shape from the current relief to create a new relief

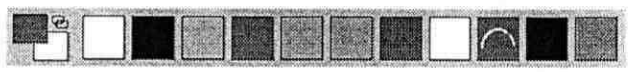
To subtract a shape from the current relief:

1. Create the shape that you want to subtract from the current relief. For details, see "Creating a Shape from a Bitmap" on page 232.

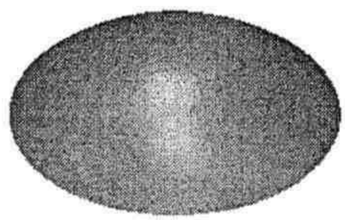
   In the following example, a blue circle is drawn in the model using the **Paint** tool:



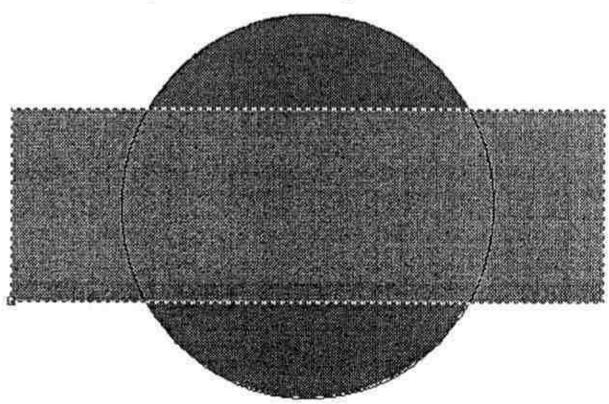
A rounded shape is then applied to the circle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:



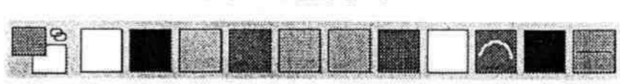
The calculated relief appears in the **3D View** window as follows:



The rectangular vector object is then flood-filled in red:



A plane is applied to the rectangle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:

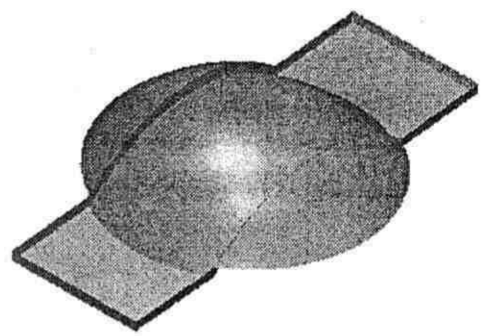


2. Click on the **Subtract Relief** button in the **Relief Operations** area of the **Assistant**'s Home page to subtract the shape from the current relief.

Note: You can also subtract a shape from the current relief using the Main menu bar. Click on the **Relief** menu, followed by the **Calculate > Subtract** option.

3. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window.

   In our example, the plane has been subtracted from the existing relief to create the following new relief:



You can also subtract a shape from the current relief using the **Project** page:

1. Click on the **Project** tab to display the **Project** page.
2. Right-click on the current **2D View** listed in the **Views** area to display the context menu. Click on the **Calculate Relief** option, followed by the **Subtract** option. For further details, see "Viewing Model Information" in the Working with Models chapter.

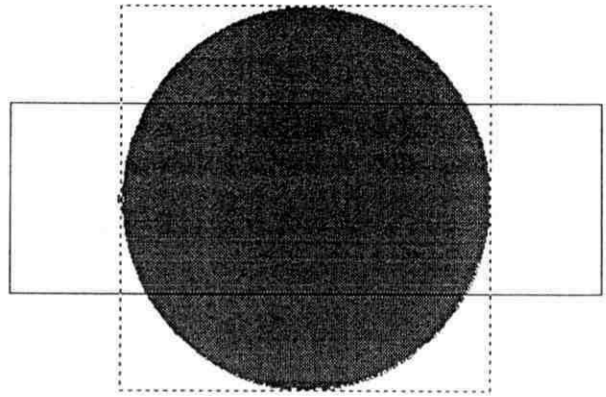
Merging with the Relief

You can merge a shape with the current relief, so that only the highest points of the two remain in a new relief. Alternatively, you can merge a shape with the current relief, so that only the lowest points of the two remain in a new relief.

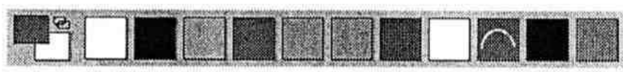
To merge a shape with the current relief:

1. Create the shape that you want to merge with the current relief. For details, see "Creating a Shape from a Bitmap" on page 232.

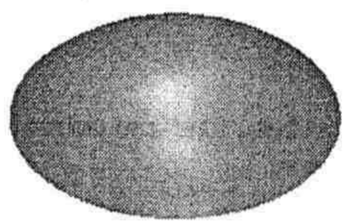
In the following example, a blue circle is drawn in the model using the **Paint** tool:



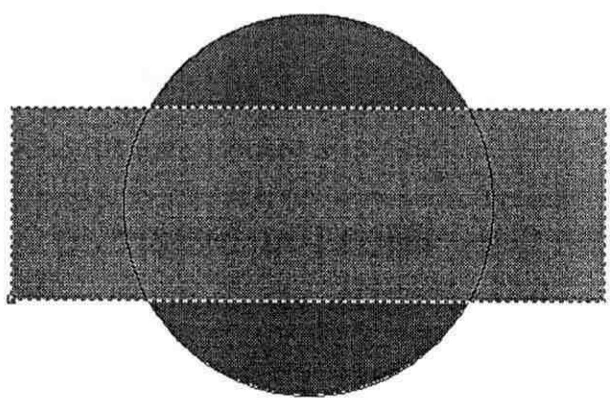
A rounded shape is then applied to the circle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:



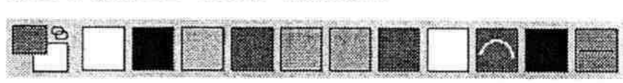
The calculated relief appears in the **3D View** window as follows:



The rectangular vector object is then flood-filled in red:



A plane is applied to the rectangle using the **Shape Editor** dialog box, as you can see by looking at the Colour Palette below the **2D View** window:



2. You can now merge the shape with the current relief:

    - Click on the **Merge High** button in the **Relief Operations** area of the **Assistant**'s Home page to merge the shape with the current relief, so that only the highest points of the two show.

    - Click on the **Merge Low** button in the **Relief Operations** area of the **Assistant**'s Home page to merge the shape with the current relief, so that only the lowest points of the two show.

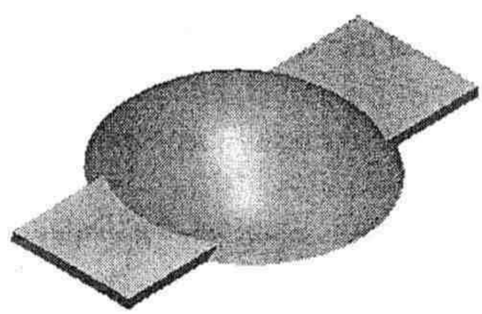
Note: You can also merge a shape with the current relief using the Main menu bar. Click on the **Relief** menu, followed by the **Calculate > Merge Highest** or **Merge Lowest** ption.

---

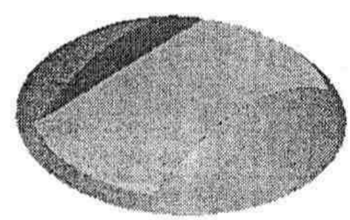
3. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window.

    The examples below show the different results achieved by using the **Merge High** and **Merge Low** options:

Merge High...



Merge Low...



You can also merge a relief using the **Project** page:

1. Click on the **Project** tab to display the **Project** page.
2. Right-click on the current **2D View** listed in the **Views** area to display the context menu. Click on the **Calculate Relief** option, followed by the **Merge Highest** or **Merge Lowest** option. For further details, see "Viewing Model Information" in the Working with Models chapter.

Transforming and Manipulating Reliefs

You can use the buttons in the **Relief Operations** and **Relief Editing** areas of the **Assistant**'s Home page, along with the **Relief** menu options in the Main menu bar, to transform and manipulate the relief that you have created or want to import.

You can transform and manipulate a relief in the following ways:

- Using 3D clipart. For details, see "Using 3D Clipart" on page 280.
- Pasting a relief along a vector. For details, see "Pasting a Relief along a Vector" on page 292.
- Inverting a relief. For details, see "Inverting a Relief" on page 294.
- Smoothing a relief. For details, see "Smoothing a Relief" on page 295.
- Scaling the relief height. For details, see "Scaling the Relief Height" on page 297.
- Scaling to volume. For details, see "Scaling to Volume" on page 299.
- Mirroring a relief. For details, see "Mirroring a Relief" on page 300.
- Offsetting a relief. For details, see "Offsetting a Relief" on page 300.
- Resetting a relief. For details, see "Resetting a Relief" on page 301.
- Resetting the relief height. For details, see "Resetting the Relief Height" on page 301.

Using 3D Clipart

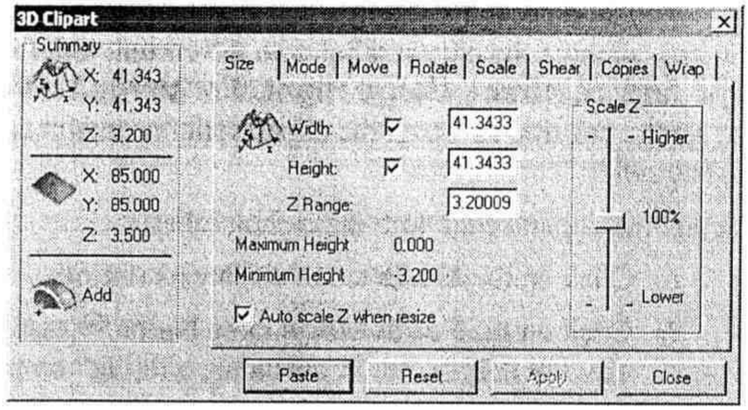
You can use the **3D Clipart** dialog box to resize and transform clipart relief, and select the way in which it is combined with the current relief.

Tip: You can find a range of clipart relief samples on the ArtCAM Pro installation CD.

The **3D Clipart** dialog box is displayed when you use either of the following options in ArtCAM Pro:

- Loading a relief. For details, see "Loading a Relief" on page 304.
- Pasting a relief along a vector. For details, see "Pasting a Relief along a Vector" on page 292.

The **3D Clipart** dialog box appears as follows:



There are eight tabs in the **3D Clipart** dialog box. Each tab contains a page of settings that allow you to edit the clipart relief and control how it is combined with the existing relief.

Resizing the Clipart Relief

You can scale the clipart relief in the Z direction and/or set its height and width.

To resize the clipart relief:

1. Click on the **Size** tab to display its settings.
2. You can scale the relief in the Z direction using any of the following methods:
   - Type its new Z height in the **Z Range** box.
   - Click and drag on the **Scale Z** slider. Click and drag upwards to increase the height of the relief in the Z direction. Click and drag downwards to reduce the height of the relief in the Z direction.
   - Click to select the **Auto scale Z when resize** option ☑. This option scales the relief in the Z direction whenever its width or height is changed.
3. To set the width and/or height of the clipart relief:

- Type its new height and/or width in the **Width** and/or **Height** box. If you want to scale the X and Y axes differentially, make sure that both the **Width** and **Height** options are selected ☑ before you type a value in each of the boxes.

4. Click on the **Apply** button to adjust the clipart relief according to your settings.

Combining the Reliefs

You can combine the clipart relief with the current relief in one of four ways: **Add**, **Subtract**, **Merge Highest** or **Merge Lowest**. You can also set the position in which the clipart relief is combined with the existing relief.

To combine clipart relief with the current relief:

1. Click on the **Mode** tab to display its settings.
2. Click on the **Paste Mode** radio button ◉ that you want to use. For details, see "Calculating a Relief" on page 272.
3. Define the height at which you want to combine the imported relief with the current relief in the **Start Height** box.
4. To calculate the height you have defined from the Z zero position in the relief, click on the **From Zero** radio button ◉. To calculate the height that you have defined from the current position in the relief, click on the **From Current** radio button ◉.
5. To paste a vector outline in the shape of the imported relief in the **2D View** window, click to select the **Copy outline** option ☑.
6. Click on the **Apply** button to adjust the clipart relief according to your settings.

Positioning the Clipart Relief

You set the position of the clipart relief.

To reposition the clipart relief:

1. Click on the **Move** tab to display its settings.
2. To move the imported relief in the Y axis direction, you can either:
    - Type a value in the **Move Y origin by** box.

- Click and drag on the vertical slider.

3. To move the imported relief in the X axis direction, you can either:
   - Type a value in the **Move X origin by** box.
   - Click and drag on the horizontal slider.
4. Click on the **Apply** button to adjust the clipart relief according to your settings.

Rotating the Clipart Relief

You can rotate and/or mirror the clipart relief along the X or Y axis around the centre of the clipart relief or a defined position in the ArtCAM model.

To rotate and/or mirror the clipart relief:

1. Click on the **Rotate** tab to display its settings.
2. To rotate the clipart relief, you can either:
   - To rotate the clipart relief about its centre, define the angle in the **Rotate by angle** box, or click and drag on the slider to set the angle.
   - To rotate the clipart relief about a specific position in the model, click to deselect the **Rotate about centre** option, and then define the new origin of rotation.

   If you have deselected the **Rotate about centre** option, you can define the origin of rotation using either of the following methods:

   - Click on the **Cursor** button, move the cursor over the position that you want to use, and then click.
   - Type the X and Y co-ordinates of the point in the **X** and **Y** boxes.
3. You can now mirror the clipart relief along the X or Y axis:
   - To mirror the clipart relief along the X-axis, click on the **Horizontal** button.
   - To mirror the imported relief along the Y-axis, click on the **Vertical** button.
4. Click on the **Apply** button to adjust the clipart relief according to your settings.

Scaling the Clipart Relief

You can scale the clipart relief to a percentage of its original size.

To scale the clipart relief:

1. Click on the **Scale** tab to display its settings.
2. To scale the clipart relief, you can either:
    - Type a percentage of the clipart relief's current size in the **New scale %** box.
    - Click and drag on the slider. Click and drag to the right to increase the size of the clipart relief. Click and drag to the left to reduce it.
3. Click on the **Apply** button to adjust the clipart relief according to your settings.

Tip: If scaling to 1% or 200% does not produce big enough or small enough results, click on the **Apply** button. This scales the relief to the new size and resets the **New Scale %** to 100. You can now repeat the scaling process to further increase or reduce its size.

Shearing the Clipart Relief

Shearing can be visualised by thinking of an image superimposed onto a flexible rubber sheet. If you hold the sides of the sheet and move them up and down in opposite directions, the image undergoes a spatial stretching known as shearing. You can shear the clipart relief both horizontally and vertically.

To shear the clipart relief:

1. Click on the **Shear** tab to display its settings.
2. To shear the clipart relief horizontally, you can either:
    - Type a value in the **Shear in X axis** box.
    - Click and drag on the horizontal slider.
3. To shear the clipart relief vertically, you can either:
    - Type a value in the **Shear in Y axis** box.
    - Click and drag on the vertical slider.
4. Click on the **Apply** button to adjust the clipart relief according to your settings.

Copying the Clipart Relief

You can make a single copy of the clipart relief, or multiple copies in either a grid or circular pattern.

To make copies of the clipart relief:

1. Click on the **Copies** tab to display its settings.
2. Select how you want to copy the clipart relief.

   If you want to create a single copy of the clipart relief, make sure that the **Single Copy** radio button is selected.

   If you want to create multiple copies of the clipart relief in a grid pattern:

   - First, click on the **Copy Linear** radio button.
   - Next, type the distance you want to set between each copy along the X-axis in the **X Distance** box.
   - Now type the distance you want to set between each copy along the Y-axis in the **Y Distance** box.
   - You are now ready to set the number of copies by typing the number of rows and columns you want to make in the **Rows** and **Columns** boxes.

   If you want to create multiple copies of the clipart relief in a circular pattern:

   - First, click on the **Copy Rotate** radio button.
   - Next, define the point around which you want to rotate the clipart relief. You can either type the X and Y co-ordinates of the origin of rotation in the **X** and **Y** boxes, or click on the **Cursor** button and then on the position in the model (the white area) that you want to use.
   - Now define the angle of rotation in the **Angle** box. Type a positive value to rotate the clipart relief clockwise. Type a negative value to rotate the clipart relief anti-clockwise.
   - You are now ready to define the number of copies you want to make in the **Number of Objects** box.
3. Click on the **Apply** button to copy the clipart relief according to your settings.

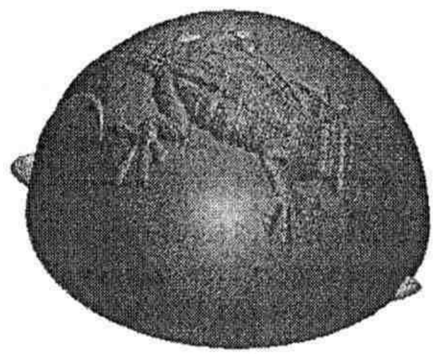
Wrapping the Clipart Relief

Clipart relief can be projected onto an existing relief in a model using the four standard relief combination options: **Add**, **Subtract**, **Merge Highest** and **Merge Lowest**. For details, see "Calculating a Relief" on page 272. Using this method of combining clipart relief with the current relief often causes distortion to the shape of the clipart relief.

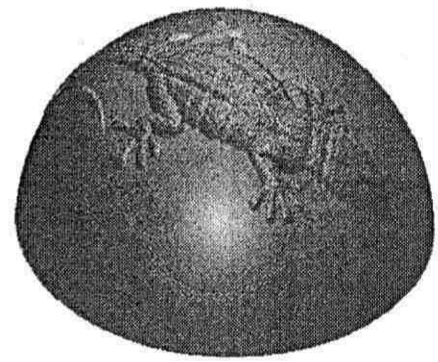
You can preserve its original shape when pasting by wrapping it around one or more of the axes in the model during the relief combination process.

In the following example, a lizard clipart relief pasted onto a dome is shown:

Projected...



Wrapped Radially...



You can see how the lizard relief is smeared across the dome when projected onto it. When pasted radially, its original size and shape is maintained.

To wrap the clipart relief:

1. Click on the **Wrap** tab to display its options.
2. In the **Wrap Type** area, click to select the wrapping option that you want to use:
    - **Project** – This option allows you to paste the clipart relief onto the relief surface.

Warning: Using the **Project** method of wrapping clipart relief onto an existing relief can cause distortion to its original shape.

- **Radial** – This option allows you to wrap the clipart relief around a spherical shape that is curved in both the X and Y-axes.

- **Wrap in X** – This option allows you to wrap the clipart relief around a cylindrical shape that is curved around the X-axis.
- **Wrap in Y** – This option allows you to wrap the clipart relief around a cylindrical shape that is curved around the Y-axis.

3. In the **Wrap Origin** area, define the origin around which the relief will be wrapped. You can use any of the following methods:

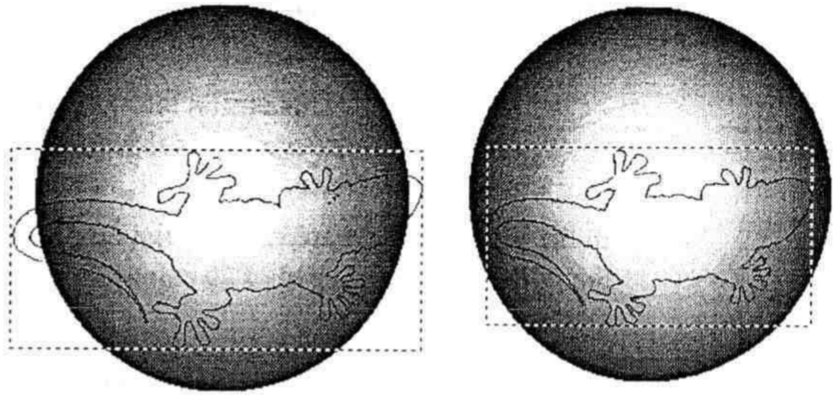
Note: If the **Project** option is selected in the **Wrap Type** area, the **Wrap Origin** area is greyed-out.

- Click on any of radio buttons on the rectangle diagram. The X and Y co-ordinates of the chosen origin appear in the **X** and **Y** boxes.
- Type the X and Y co-ordinates of the origin in the **X** and **Y** boxes.
- Click on the **Use Mouse** option, followed by the **Start** button. Move the cursor to the position in the **2D View** window that you want to use as the origin, and then click to select. The X and Y co-ordinates of the origin appear in the **X** and **Y** boxes.

4. Click on the **Apply** button to wrap the clipart relief around the existing relief. The vector outline in the **2D View** window representing the clipart relief is corrected to compensate for the distortion.

   In our example, the lizard clipart relief's vector outline appears as follows before and after it has been radially wrapped:

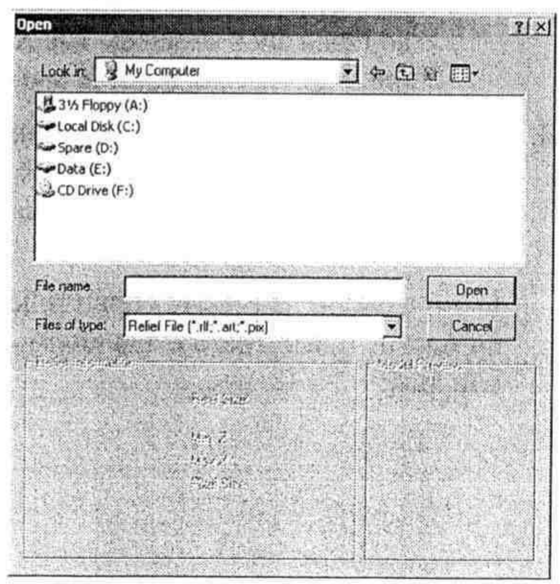
Projected... *Radially Wrapped...*



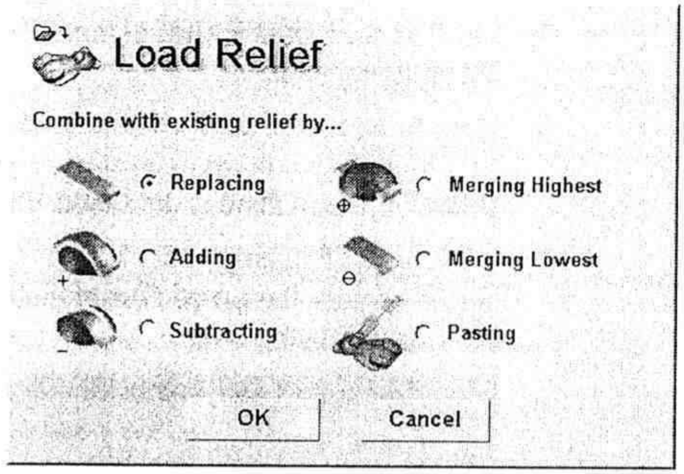
3D Clipart Example

In the following example, you can see how the **3D Clipart** dialog box is used to transform, resize and position an imported relief to create a new relief.

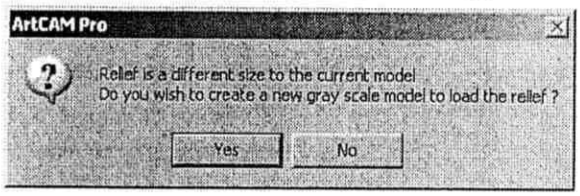
1. Click on the **Load Relief** button  in the **Relief Operations** area of the **Assistant**'s Home page to display the **Open** dialog box:



2. Click to select the **Pend_frm.rlf** file in the ArtCAM Pro 6.0\Examples\Overview directory and then click on the **Open** button to display the **Load Relief** dialog box:

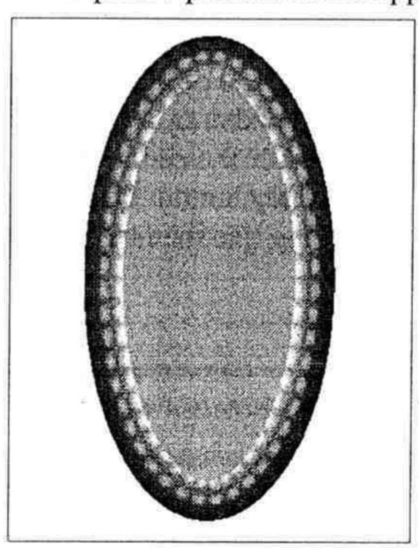


If the loaded relief is larger or smaller than the existing relief, the following message box appears:

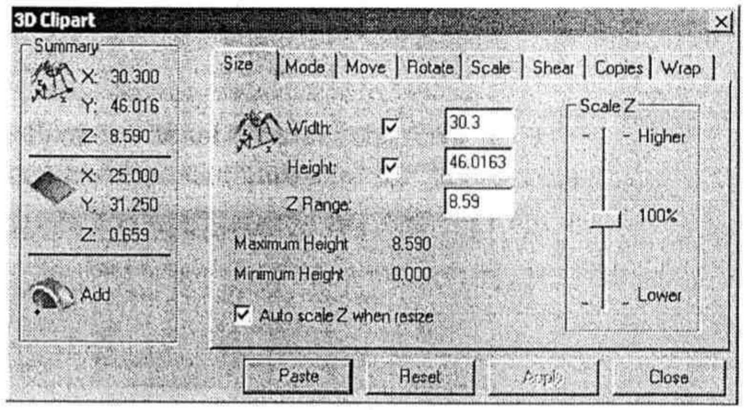


Click on the **Yes** button. The new model is created, and a greyscale image of the loaded relief is shown in the **2D View** window.

3. Click on the **Replacing** radio button, and then click on the **OK** button to close the **Load Relief** dialog box and load the relief.
4. Click to select the **2D View** window. A greyscale image of the imported pendant relief appears as follows:

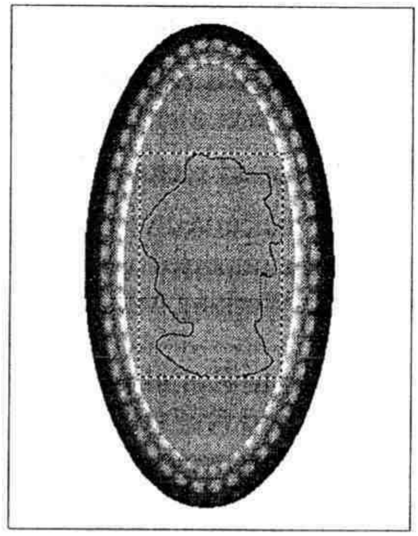


5. Click on the **Load Relief** button again to display the **Open** dialog box.
6. Click to select the **Lady.rlf** file in the ArtCAM 6.0\Examples\Overview directory, and then click on the **Open** button to display the **Load Relief** dialog box.
7. Click on the **Pasting** radio button, and then on the **OK** button to close the **Load Relief** dialog box and display the **3D Clipart** dialog box:



A vector object in the shape of a lady's head appears in the **2D View** window to represent the area of the lady relief. The imported lady relief is much larger than the pendant relief, and therefore has to be resized in order to fit within it.

8. You are now ready to scale the lady relief. In the **3D Clipart** dialog box, click on the **Scale** tab, click and drag on the sliders to reduce the size of the lady relief to fit inside the centre of the pendant relief, and then click on the **Apply** button.
9. You are now ready to position the lady relief. Click on the **Move** tab, click and drag on the sliders to position the vector object inside the centre of the pendant relief, and then click on the **Apply** button. The lady relief should now appear something like the image shown below:



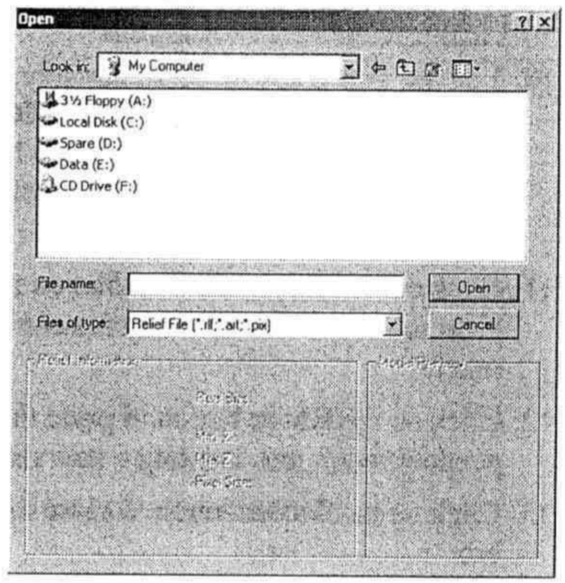
10. You are now ready to resize the lady relief. Click on the **Size** tab, click and drag on the **Scale Z** slider downwards to set the **Maximum Height** of the lady relief to approximately 0.500 mm, and then click on the **Apply** button.
11. Click on the **Mode** tab, and then on the **Add** radio button to select how the lady relief is combined with the pendant relief.
12. Click on the **Paste** button to paste the lady relief onto the pendant relief, thus creating a new relief.
13. Click on the **Close** button to close the **3D Clipart** dialog box.
14. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. The new relief should appear something like the image shown below:

Pasting a Relief along a Vector

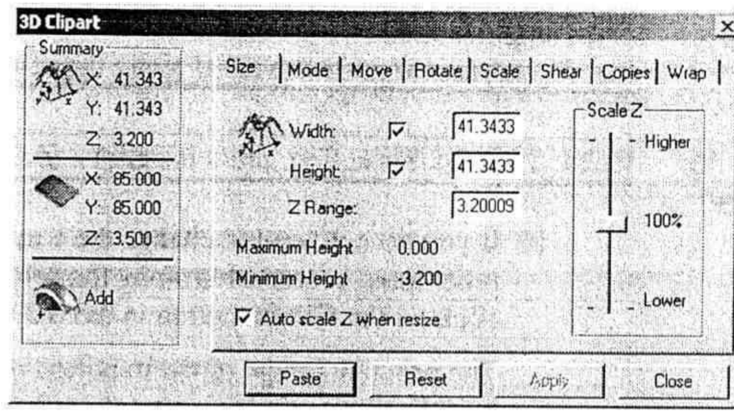
You can import a relief into ArtCAM Pro, edit the relief and then paste multiple copies of it along a selected curved vector object.

To paste a relief along a curved vector object:

1. Click to select the curved vector object along which you want to paste the imported relief. For details, see "Selecting Vectors" in the Working with Vectors chapter.
2. Click on the **Paste Relief Along Vector** button  in the **Vector Based Relief Creation** area of the **Assistant**'s Home page to display the **Open** dialog box:



3. Click on the **Look in** list box and select the directory where the relief file is stored.
4. Click to select the relief file that you want to import. The file name appears in the **File name** box and the relief dimensions appear in the **Relief Information** area.
5. Click on the **Open** button to import the selected relief. A preview image of the relief appears in the **2D View** window about the origin of your model. The **Paste Along A Curve** page is displayed in the **Assistant** window and the **3D Clipart** dialog box also appears:



6. You can use the **3D Clipart** dialog box to edit the imported relief, and control the way in which it is combined with the existing relief. For details, see "Using 3D Clipart" on page 280.
7. Click on the **Apply** button in the **3D Clipart** dialog box to adjust the relief according to your settings.
8. You can now use the **Settings** area of the **Paste Along A Curve** page to adjust the size of each consecutive copy that is pasted along the selected vector object, and the spacing between them:
   - To set the spacing between each copy as a percentage of the relief width, type a value in the **Spacing (S)** box.
   - To set the size of the first pasted copy as a percentage of the relief width, type a value in the **Start size (B)** box.
   - To set the size of the last pasted copy as a percentage of the relief width, type a value in the **Final size (E)** box.

Note: If you type a value in the **Start size (B)** and **Final size (E)** boxes, each consecutive copy of the relief pasted along the selected vector object is scaled proportionately.

   - If you want the copies of the relief to be pasted evenly along the selected vector object(s), make sure that the **Adjust to fit exactly** option is selected ☑.
9. Click on the **Paste** button on the **Paste Along A Curve** page to paste the relief along the selected vector object.

Warning: Do not click on the **Paste** button in the **3D Clipart** dialog box if you want to paste the relief along the selected vector object(s).

Note: To undo this action, press the **Ctrl + Z** keys on your keyboard.

10. If you do not want to change the way that the copies of the relief have been pasted along the selected vector object, click on the **Close** button in the **3D Clipart** dialog box.

    The preview image of the imported relief disappears from the **2D View** window about the origin of the model.

11. Click on the **Close** button on the **Paste Along A Curve** page to return to the **Assistant**'s Home page.

Inverting a Relief

You can invert the existing relief in two ways:

- In the Z-axis only, without any mirroring.
- In the Z-axis, and mirror left to right.

Either or these options allow you turn a concave shape into a convex shape, and vice versa.

To invert a relief in the Z axis only:

1. Click on the **Invert Relief Z Only** button in the **Relief Editing** area of the **Assistant**'s Home page. The relief is inverted in the Z-axis only.

Note: You can also invert the relief in the Z-axis only from the Main menu bar. Click on the **Relief** menu, and then on the **Invert > Z Axis Only** option.

2. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can view the inverted relief.

To invert a relief in the Z-axis and mirror left to right:

1. Click on the **Invert Relief – Male/Female** button in the **Relief Editing** area of the **Assistant**'s Home page. The relief is inverted in the Z-axis, and it is also mirrored left to right.

Note: You can also invert the relief in the Z-axis and mirror left to right from the Main menu bar. Click on the **Relief** menu, and then on the **Invert > Male/Female** option.

The inverted relief is mirrored to allow you to produce matching top and bottom halves when machining a model or create a mould from a male model.

Note: All engraved features on the relief remain as defined. They are added to or subtracted from the relief.

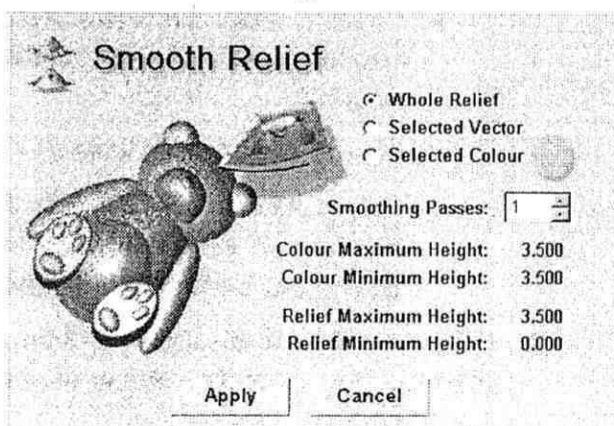
2. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. You can view the inverted relief.

Smoothing a Relief

You can remove surface irregularities that might appear on a relief after it has been calculated. You can smooth the whole surface, or a specific area of the relief.

To smooth the whole relief:

1. Click on the **Smooth Relief** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Smooth Relief** dialog box:

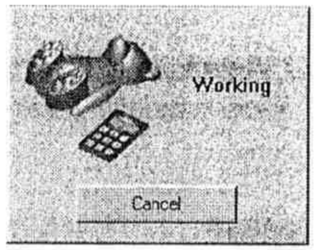


Note: You can also display the **Smooth Relief** dialog box from the Main menu bar. Click on the **Relief** menu, and then on the **Smooth...** option.

2. Click on either of the arrows joined to the **Smoothing Passes** box to set the number of smoothing passes over the selected relief.

Tip: It is better to increase and apply the number of smoothing passes gradually, checking the results of each additional smoothing pass in the **3D View** window.

3. Click on the **Apply** button to close the **Smooth Relief** dialog box and smooth the whole relief. The **Working** dialog box appears during the smoothing process:



To smooth a specific area of the relief:

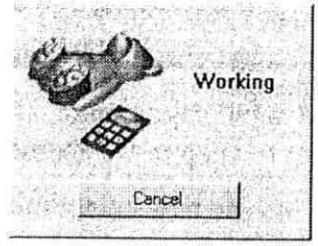
1. Click on the **Smooth Relief** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Smooth Relief** dialog box.
2. You can either use a vector object or bitmap colour to define the area of the relief that you want to smooth. If you want to use a vector object:

    - Click to select the vector object from the **2D View** window, and then on the **Selected Vector** radio button.

    If you want to use a bitmap colour:

    - Select the Primary Colour from the Colour Palette beneath the **2D View** window, and then click on the **Selected Colour** radio button.

3. Click on either of the arrows adjoined to the **Smoothing Passes** box to set the number of smoothing passes over the selected relief.
4. Click on the **Apply** button to close the **Smooth Relief** dialog box and smooth the area of the relief defined by the currently selected vector object or Primary Colour. The **Working** dialog box appears during the smoothing process:

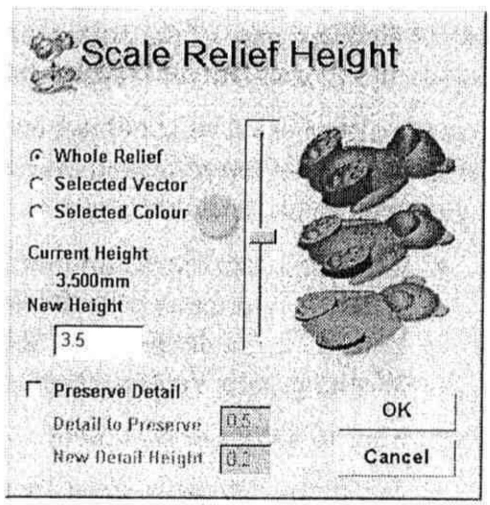


Scaling the Relief Height

You can scale the whole surface, or a specific area of the relief.

To scale the height of the whole relief:

1. Click on the **Scale Relief Height** button  in the **Relief Editing** area of the **Assistant**'s Home page to display the **Scale Relief Height** dialog box:



The current height of the relief is shown in the **Current Height** area of the dialog box.

Note: You can also display the **Scale Relief Height** dialog box from the Main menu bar. Click on the **Relief** menu, and then on the **Scale...** option.

2. To set the new height of the whole relief, you can use either of the following methods:
   - Click and drag on the slider. Click and drag upwards to increase the height of the relief. Click and drag downward to reduce it. Its new height appears in the **New Height** box.
   - Type its new height in the **New Height** box.

3. If the relief that you are scaling contains fine detail:
   - First, click to select the **Preserve Detail** option ☑.
   - Next, define the maximum depth of the detail that you want to keep in the **Detail To Preserve** box.
   - Finally, set the new height of the detail in the **New Detail Height** box.

   ArtCAM Pro scales the detail independently of the height specified in the **New Height** box and restores it to the relief after the overall scaling process is complete.
4. Click on the **OK** button to close the **Scale Relief Height** dialog box and scale the relief height.

To scale a specific area of the relief:

1. Click on the **Scale Relief Height** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Scale Relief Height** dialog box.
2. You can either use a vector object or bitmap colour to define the area of the relief that you want to scale in height. If you want to use a vector object:
   - Click to select the vector object from the **2D View** window, and then on the **Selected Vector** radio button. For details, see "Selecting Vectors" in the Working with Vectors chapter.

   If you want to use a bitmap colour:
   - Select the Primary Colour from the Colour Palette beneath the **2D View** window, and then click on the **Selected Colour** radio button. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.

Warning: If you select a vector object or bitmap colour that does not represent an area of the model with underlying relief, the **New Height** box in the **Scale Relief Height** dialog box is greyed-out. This indicates that there is no relief in which to scale the height.

3. If the area of the relief that you are scaling contains fine detail:
   - First, click to select the **Preserve Detail** option ☑.

- Next, define the maximum depth of the detail that you want to keep in the **Detail To Preserve** box.
- Finally, set the new height of the detail in the **New Detail Height** box.

ArtCAM Pro scales the detail independently of the height specified in the **New Height** box and restores it to the relief after the overall scaling process is complete.

4. Click on the **OK** button to close the **Scale Relief Height** dialog box and scale the relief height.

Scaling to Volume

You can scale an ArtCAM model to an exact volume.

To scale an open model to an exact volume:

1. From the Main menu bar, click on the **Relief** menu and then on the **Volume** option to display the **Scale To Volume** page in the **Assistant** window.

   The current size of your model is shown in the **Model Size** area. Its height in the Z direction is shown in the **Model Z Height** area. Its volume is shown in the **Current Volume** area, according to the units of measurement you are using.

2. Click on the **Volume Units** list box, followed by the units of measurement that you want to use.

   If you are using millimetres, you can select either of the following options:

   - **Cubic mm** – Click on this option to define the volume in cubic millimetres.
   - **Cubic cm** – Click on this option to define the volume in cubic centimetres.

   If you are using inches, you can select either of the following options:

   - **Cubic Inches** – Click on this option to define the volume in cubic inches.
   - **Cubic Feet** – Click on this option to define the volume in cubic feet.

3. Define the new volume for the model in the **Enter New Volume** box.

4. If you want to scale the model in the Z direction only, click to select the **Scale in Z Only** option ☑.
5. Click on the **Apply** button to scale the model.
6. Click on the **Finish** button to return to the **Assistant**'s Home page.

Mirroring a Relief

You can reverse the direction of a relief from its original position both horizontally and vertically.

To mirror the relief horizontally:

- From the Main menu bar, click on the **Relief** menu, and then on the **Mirror > Mirror Horizontal** option.

To mirror the relief vertically:

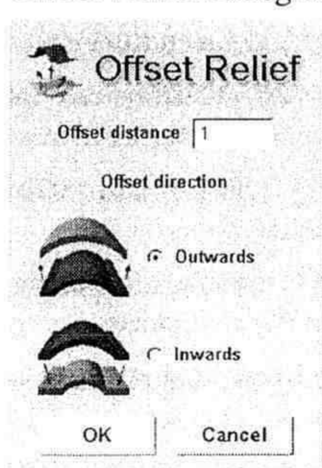
- From the Main menu bar, click on the **Relief** menu, and then on the **Mirror > Mirror Vertical** option.

Offsetting a Relief

You can create a new relief by applying an offset to the existing relief. This allows you to add or subtract material from the surface of both male and female reliefs.

To create an offset relief:

1. Click on the **Offset Relief** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Offset Relief** dialog box:



Note: You can also display the **Offset Relief** dialog box from the Main menu bar. Click on the **Relief** menu, and then on the **Offset...** option.

2. Define the distance you want to set the offset relief from the existing relief in the **Offset distance** box.
3. Click on either of the radio buttons to select the **Offset direction** you want to use:
    - If you want to thicken the material surface, click on the **Outwards** radio button.
    - If you want to thin the material surface, click on the **Inwards** radio button.
4. Click on the **OK** button to calculate the offset relief and close the **Offset Relief** dialog box.
5. Click on the **3D View** button in the **2D View** toolbar to view the new relief.

Resetting a Relief

You can reset the whole relief to zero, effectively removing it.

To reset the relief to zero:

1. Click on the **Reset Relief** button in the **Relief Operations** area of the **Assistant**'s Home page.
2. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can now see that the relief has been reset to zero.
3. Click on the **2D View** button in the **3D View** toolbar to return to the **2D View** window.

Resetting the Relief Height

You can reset the height of the relief under the current Primary Colour to zero, and vice versa.

To reset the height of the relief to zero in all areas under the current Primary Colour:

1. Select the Primary Colour from the Colour Palette beneath the **2D View** window. For details, see "Selecting the

Primary and Secondary Colours" in the Working with Bitmaps chapter. The Primary Colour represents the areas of the relief that you want to reset to zero.

2. Click on the **Zero Relief Under Colour** button in the **Relief Editing** area of the **Assistant**'s Home page.
3. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can now see that all areas of the relief under the current Primary Colour have been reset to zero.

To reset the height of the whole relief to zero, except in those areas under the current Primary Colour:

1. Select the Primary Colour from the Colour Palette beneath the **2D View** window. The Primary Colour represents the areas of the relief that you do not want to reset to zero.
2. Click on the **Keep Under Colour** button in the **Relief Editing** area of the **Assistant**'s Home page.

Note: You can also reset the height of the whole relief to zero, except in those areas under the current Primary Colour, from the Main menu bar. Click on the **Relief** menu, and then on the **Keep Under Colour** option.

3. Click on the **3D View** button in the **2D View** toolbar to display the **3D View** window. You can now see that all areas of the relief not under the current Primary Colour have been reset to zero.

Managing and Editing Reliefs

You can use the buttons in the **Relief Operations** and **Relief Editing** areas of the **Assistant**'s Home page, along with the **Relief** menu options in the Main menu bar, to manage and edit a relief that you have created.

You can manage and edit a relief in the following ways:

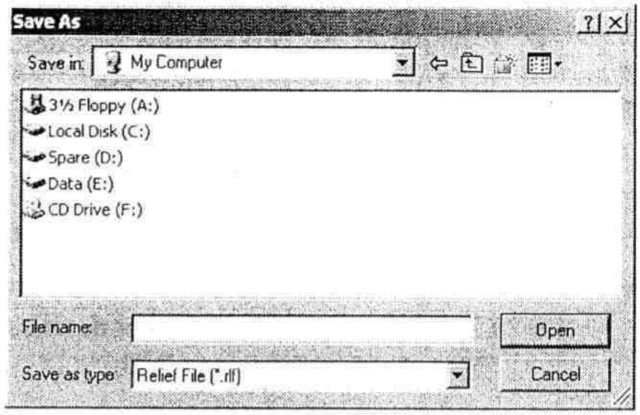
- Saving a relief. For details, see "Saving a Relief" on page 303.
- Loading a relief. For details, see "Loading a Relief" on page 304

- Calculating the surface area. For details, see "Calculating the Surface Area" on page 308.
- Displaying the calculation time. For details, see "Displaying the Calculation Time" on page 308.
- Adding a draft angle. For details, see "Adding a Draft Angle" on page 309.
- Creating a triangle mesh. For details, see "Creating a Triangle Mesh" on page 309.
- Creating a cross-section. For details, see "Creating a Cross-Section" on page 312.
- Creating an angled plane. For details, see "Creating an Angled Plane" on page 313.
- Blending 3D shapes. For details, see "Blending 3D Shapes" on page 315.
- Creating a ring. For details, see "Creating a Ring" on page 321.
- Adding texture to a relief. For details, see "Adding Texture to a Relief" on page 321.
- Sculpting a relief. For details, see "Sculpting a Relief" on page 328.
- Removing holes from the relief surface. For details, see "Removing Holes in the Relief Surface" on page 332.
- Creating a greyscale image from a relief. For details, see "Creating a Greyscale Image from a Relief" on page 332.
- Rotating the relief or the triangle mesh. For details, see "Rotating a Relief or Triangle Mesh" on page 332.

Saving a Relief

You can save a relief as an ArtCAM relief file (***.rlf**):

1. Click on the **Save Relief** button in the **Relief Operations** area of the **Assistant**'s Home page to display the **Save As** dialog box:



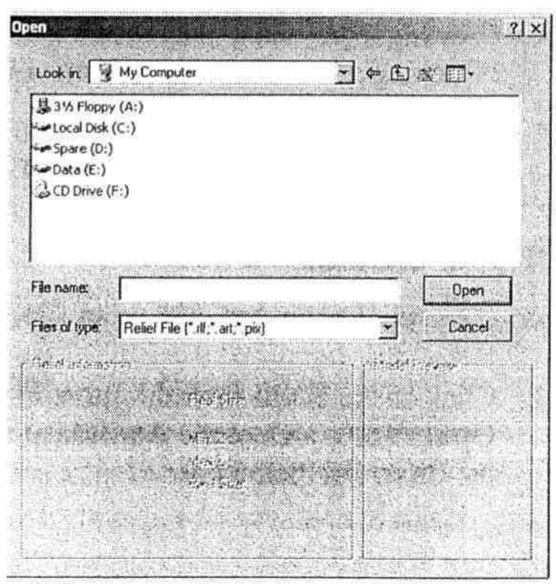
2. Click on the **Save in** list box and select the directory where you want to save the relief.
3. Type a name for the relief in the **File name** box.
4. Click on the **Save** button to save the relief.

Loading a Relief

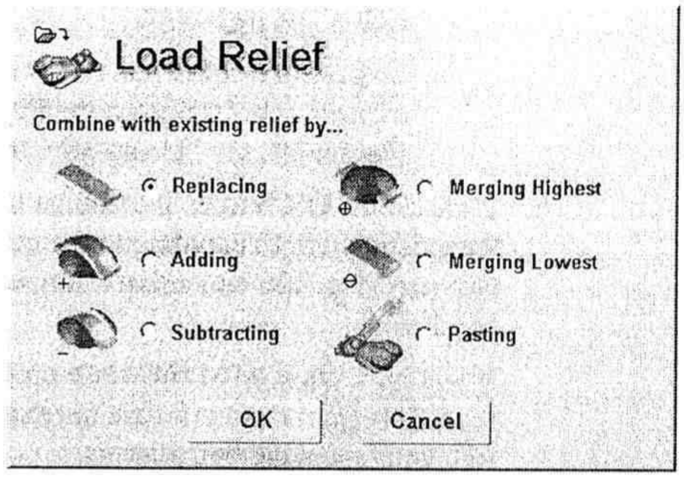
You can load a relief file (***.rlf**, ***.art**, ***.pix**), and then combine it with the existing relief in a model, or use it to replace the existing relief altogether.

To load a relief:

1. Click on the **Load Relief** button  in the **Relief Operations** area of the **Assistant**'s Home page to display the **Open** dialog box:



2. Click on the **Look in** list box and select the directory where the relief file that you want to load is stored.
3. Click to select the relief file that you want to load.
4. Click on the **Open** button to display the **Load Relief** dialog box:



5. Select how you want to use the loaded relief by clicking on one of the **Combine with existing relief by...** radio buttons :

Warning: If the pixel size of the relief that you want to load does not equal the pixel size of the existing relief, only the **Replacing** and **Pasting** options are available in the **Load Relief** dialog box.

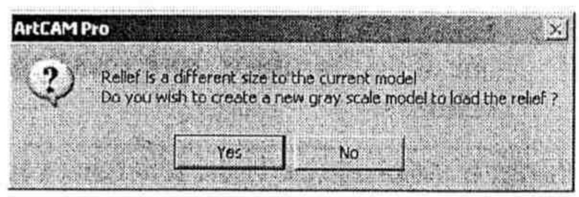
- To replace the existing relief with the relief you have chosen to load, click on the **Replacing** radio button . For more details, see "Replacing the Relief" on page 272.
- To add the relief you have chosen to load to the existing relief, click on the **Adding** radio button . For more details, see "Adding to the Relief" on page 273.
- To subtract the relief you have chosen to load from the existing relief, click on the **Subtracting** radio button . For more details, see "Subtracting from the Relief" on page 275.
- To merge the relief you want to load with the existing relief, so that only the highest points of the two remain, click on the **Merging Highest** radio

button . For more details, see "Merging with the Relief" on page 277.

- To merge the relief you want to load with the existing relief, so that only the lowest points of the two remain, click on the **Merging Lowest** radio button . For more details, see "Merging with the Relief" on page 277.
- To paste the relief you have chosen to load onto the existing relief, click on the **Pasting** radio button . For details, see "Using 3D Clipart" on page 280.

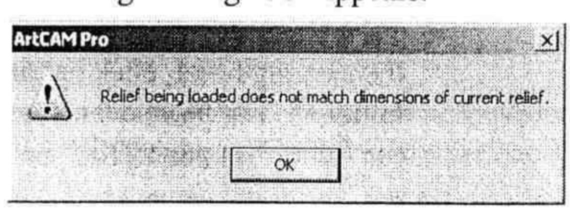
6. Click on the **OK** button to combine the loaded relief with the existing relief according to the method selected in the previous step. The new relief is shown in the **3D View** window.

   If you have chosen the **Replace** option and the loaded relief is larger or smaller than the existing relief, the following message box appears:



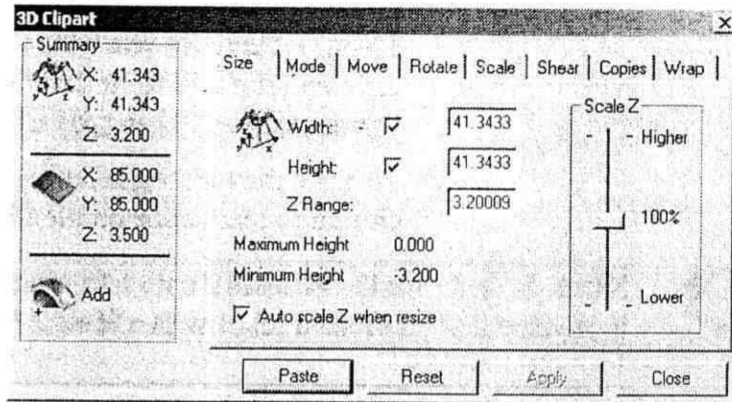
   If you want to create a new model click on the **Yes** button. If you have not recently saved the current model, a message box appears giving you the option to save any changes made to the existing model. Click on the **Yes** button to save these changes, or on the **No** button to close the model without saving them first. The new model is created, and a greyscale image of the loaded relief is shown in the **2D View** window.

   If the pixel size of the relief that you have chosen to load does not equal the pixel size of the existing relief, the following message box appears:



   Click on the **OK** button to close the message box, and then repeat these steps.

If you selected the **Pasting** option in the previous step, the **3D Clipart** dialog box is displayed:



You can use the **3D Clipart** dialog box to edit the relief you have loaded, and control the way in which it is combined with the existing relief. A preview image of the relief also appears in the **2D View** window about the origin of your model. For details on how to use the **3D Clipart** dialog box, see "Using 3D Clipart" on page 280.

You can also load a relief in the following way:

1. From the Main menu bar, click on the **Relief** menu, followed by the **Load** option, and then on the relief combination option that you want to use:
   - To replace the existing relief with the relief you have chosen to load, click on the **Replace** option. For more details, see "Replacing the Relief" on page 272.
   - To add the relief you have chosen to load to the existing relief, click on the **Add** option. For more details, see "Adding to the Relief" on page 273.
   - To subtract the relief you have chosen to load from the existing relief, click on the **Subtract** option. For more details, see "Subtracting from the Relief" on page 275.
   - To merge the relief you want to load with the existing relief, so that only the highest points of the two remain, click on the **Merge Highest** option. For more details, see "Merging with the Relief" on page 277.
   - To merge the relief you want to load with the existing relief, so that only the lowest points of the

two remain, click on the **Merge Lowest** option. For more details, see "Merging with the Relief" on page 277.

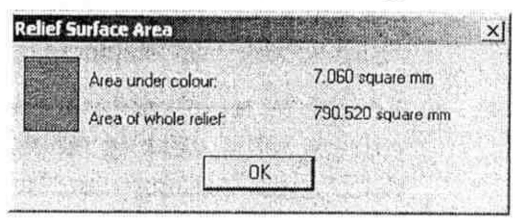
- To paste the relief you have chosen to load onto the existing relief, click on the **Pasting** radio button. For details, see "Using 3D Clipart" on page 280.
- To scale the existing relief by the relief you have chosen to load, click on the **Multiply** option.

Note: In order for the **Multiply** option to produce a feasible relief it is advised that you load a relief with a **Max Z** height of between 0 and 1.

Calculating the Surface Area

To calculate the surface area of the whole relief or the area of the relief under the current Primary Colour:

1. Make sure that the current Primary Colour represents the area of the relief of which you want to find out the surface area. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.
2. From the Main menu bar, click on the Relief menu, and then the **Surface Area Under Colour** option to display the **Relief Surface Area** dialog box:



   The surface area of the relief under the current Primary Colour and the surface area of the whole relief is shown in mm^2 or $inches^2$.

3. Click on the **OK** button to close the **Relief Surface Area** dialog box.

Displaying the Calculation Time

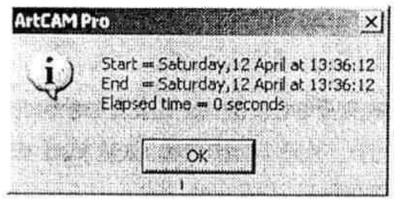
You can automatically find out how long it has taken to create a relief created using shapes generated from bitmaps and which were calculated using the relief combination buttons in the **Relief** toolbar.

For details, see "Creating a Shape from a Bitmap" on page 232 and "Calculating a Relief" on page 272.

To view information about the calculation process immediately after calculating a relief:

1. From the Main menu bar, click on the **Relief** menu, and then on the **Display Calculation Time** option.

   At the end of each relief calculation, the **ArtCAM Pro** dialog box appears showing the start time/date, end time/date and duration of the relief calculation process:

   Click on the **OK** button to close the dialog box.

If you no longer want to view this information, repeat this step to deselect the **Display Calculation Time** option in the **Relief** menu.

Adding a Draft Angle

You can remove any vertical or near vertical faces that can be seen on a relief surface by adding a constant draft angle to it.

To add a draft angle to a relief:

1. From the Main menu bar, click on the **Relief** menu, and then on the **Add Draft...** option to display the **Add Draft Angle** page in the **Assistant** window.
2. Type the draft angle you want to add to the existing relief in the **Enter Draft Angle In Degrees** box.
3. Click on the **Apply** button to add the draft angle to the relief.
4. Click on the **Finish** button to return to the **Assistant**'s Home page.

Creating a Triangle Mesh

You can create a triangle model of the existing relief, otherwise known as a **Triangle Mesh**. This is made up of a series of triangle facets that represent a closed three-dimensional shape. Three point co-ordinates and a direction describe each facet.

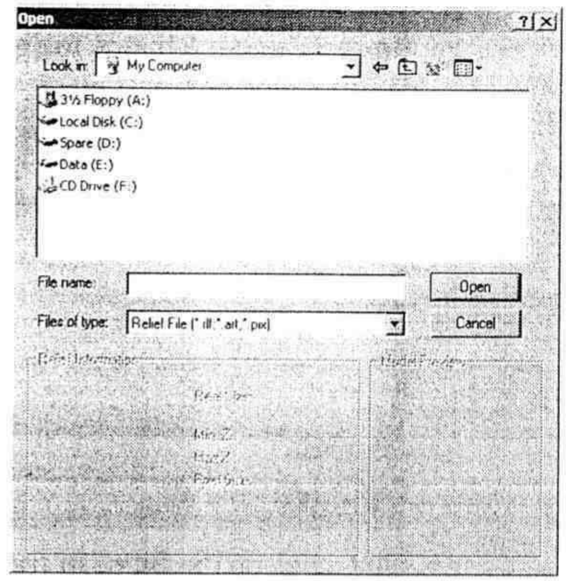
To create a triangle mesh from the existing relief:

1. Click on the **Create Triangle Mesh** button in the **Relief Operations** area of the **Assistant**'s Home page to display the **Mesh Creator** page.

Note: You can also display the **Mesh Creator** page from the Main menu bar. Click on the **Relief** menu, and then on the **Create Triangle Mesh...** option.

2. In the **Triangulation Parameters** area, type a value in the **Tolerance** box to control the number of triangles generated.
3. In the **Back Face** area, click on one of the radio buttons to select the type of shape that you want to use to close the triangle mesh:
    - **None (Open Triangulation)** – To leave the shape open, click on this radio button .
    - **Close With A Flat Plane** – To create a back face equivalent to a Z height of zero, click on this radio button .
    - **Close With Inverted Front** – To create a back face that is the inverted shape of the relief surface, click on this radio button .
    - **Use Relief From File** – To create the back face from a relief file (***.rlf**, ***.art**, or ***.pix**), providing that pixel size of the relief that you have chosen to load equals the pixel size of the existing relief, click on this radio button . If you select this option:

      First, click on the **Load** button to display the **Open** dialog box:



Next, click on the **Look in** list box and select the directory where the relief file is stored.

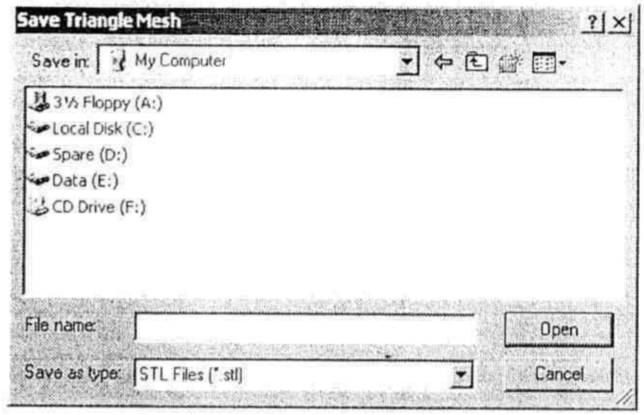
Now click to select the relief file that you want to load. The file name appears in the **File name** box. The relief dimensions appear in the **Relief Information** area of the **Open** dialog box.

Finally, click on the **Open** button to load the relief.

- **Create Offset Back Face** – To create a shape from an offset of the relief surface, click on this radio button. If you select this option, define the thickness of the offset in the **Thickness** box.

4. Click on the **Create Triangles** button to create the triangle mesh. The volume of the triangle mesh and the number of triangles created are shown in the **Result** area.
5. If you want to calculate the approximate weight of the ring component after the casting process, type a percentage in the **Shrinkage** box and then click on the **Update** button.
6. Click on the **Weight** list box, and from the list displayed click on the metal that you want to use for the cast of the ring component.
7. In the **Triangle Drawing** area, select how you want to view the triangle model in the **3D View** window, either **Shaded** or **Wireframe**.

8. You are now ready to save the triangle model. Click on the **Save Triangles** button to display the **Save Triangle Mesh** dialog box:



First, click on the **Save In** list box and select the directory in which you want to save the triangle mesh.

Next, click on the **Save as type** list box, and then on the file type you want to save the triangle mesh as (***.stl**, ***.dmt** or ***.3da**).

Finally, type the file name you want to use for the triangle model in the **File name** box, and then click on the **Save** button.

9. Click on the **Close** button to return to the **Assistant**'s Home page.

Creating a Cross-Section

You can create a vector object representing the cross-section of a relief using only the mouse and the **Create Relief Cross-Section** page.

ArtCAM Pro calculates the measurement details in real time. This means that you can view the calculated values on the **Relief Cross-Section** page as you are dragging the cursor into position.

To create a cross-section:

1. Click on the **Create Cross-Section** button  in the **Relief Operations** area of the **Assistant**'s Home page to display the **Create Relief Cross-Section** page.

Note: You can also display the **Create Relief Cross-Section** page from the Main menu bar. Click on the **Relief** menu, and then on the **Create Cross-Section...** option.

2. Move the cursor over the area in the model (the white area) that in which you want to create a cross-section, and then click on the position that you want to define as the start.

   The co-ordinates of the cross-section's start position appear in the **Anchor Position**'s **X** and **Y** area.

   By default, the + cursor snaps to points (nodes) in a vector object. This is indicated by the cursor changing to a cursor when over the points (nodes).

Note: To disable snapping, hold the **Shift** key down on your keyboard whilst dragging the cursor into position.

3. Drag the mouse over the position in the span that you want to define as the end of the cross-section, and then click.

   The distance between the start and the end of the cross-section is shown in the **Distance** area.

   The angle between the start and the end position of the cross-section is shown in the **Angle** area.

   The distance between the start and the end of the cross-section in both the X and Y axes is shown in the **X and Y distance** area.

4. If you want to align the cross-section with the X-axis, click to select the **Align Cross Section Horizontally** option ☑.

5. Click on the **Create** button to create the cross-section.

6. Click on the **Close** button to return to the **Assistant**'s Home page.

Creating an Angled Plane

You can create an angled plane using up to three defined points in a model.

To create an angled plane:

1. Click on the **Create Angled Plane** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Create Angled Plane** page in the **Assistant** window.

Note: You can also display the **Create Angled Plane** page from the Main menu bar. Click on the **Relief** menu, and then on the **Create Angled Plane...** option.

2. If you want to create an angled plane within the boundary of a vector object, click to select the vector object now. For details, see "Selecting Vectors" in the Working with Vectors chapter. If you do not select a vector object, the angled plane is created across the entire relief surface.
3. If you want to create an angled plane using three defined points, click to select the **Set Third Point** option ☑. The **Third Point on Plane** area appears on the page.

   The advantage of selecting a third point is that it adds a tilt or 'roll' to the sides of the angled plane.
4. You can use the mouse cursor to define the points on the plane:
   - Click on the **Start** button in the **Use Cursor to Select Points** area, move the + cursor over the required position in the **2D View** window, and then click. The co-ordinates for this point are displayed in the **X**, **Y** and **Z** boxes.

     The co-ordinates for this point appear in red when you click on the **Start** button, then change to black when you click in the **2D View** window.

   Alternatively, type a value in the **X**, **Y** and **Z** boxes in the **First Point on Plane** area.
5. If you had defined the first point using the mouse cursor:
   - Move the cursor over the required position in the **2D View** window, and then click to define the second point.The co-ordinates for this point are displayed in the **X**, **Y** and **Z** boxes.

     The co-ordinates for this point appear in red, then change to black when you click in the **2D View** window.

Otherwise, type a value in the **X**, **Y** and **Z** boxes in the **Second Point on Plane** area.

6. If you had defined the first and second points using the mouse cursor:

   - Move the cursor over the required position in the **2D View** window, and then click to define the third point. The co-ordinates for this point are displayed in the **X**, **Y** and **Z** boxes.

     The co-ordinates for this point appear in red, then change to black when you click in the **2D View** window.

   Alternatively, type a value in the **X**, **Y** and **Z** boxes in the **Third Point on Plane** area.

7. In the **Combine** area, click on one of the radio buttons to select how the angled plane is combined with the existing relief:

   - To add the angled plane to the existing relief, click on the **Add** option. For more details, see "Adding to the Relief" on page 273.
   - To subtract the angled plane from the existing relief, click on the **Subtract** option. For more details, see "Subtracting from the Relief" on page 275.
   - To merge the angled plane with the existing relief, so that only the highest points of the two remain, click on the **Highest** option. For more details, see "Merging with the Relief" on page 277.
   - To merge the angled plane with the existing relief, so that only the lowest points of the two remain, click on the **Lowest** option. For more details, see "Merging with the Relief" on page 277.

8. Click on the **Create** button.
9. Click on the **Close** button to return to the **Assistant**'s Home page.

Blending 3D Shapes

You can create new types of three-dimensional shapes called 'blended' shapes. A blended shape is a three-dimensional shape that can be generated from vector objects or the existing relief.

To create a blended shape:

1. Select the vector object to define the outline of the blended shape. For details, see "Selecting Vectors" in the Working with Vectors chapter.

2. Click on the **Create 3D Blend** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Create 3D Blend** page.

Note: All options on the page currently displayed in red are not currently available for the selected vector object(s).

3. In the **Profile** area, select the profile of the blended shape by clicking on the appropriate radio button :

   - **Linear** – This option instructs ArtCAM Pro to create a straight-edged shape.
   - **Convex** – This option instructs ArtCAM Pro to create a shape that curves outwards.
   - **Concave** – This option instructs ArtCAM Pro to create a shape that curves inwards.
   - **Smooth** – This option instructs ArtCAM Pro to create a shape with rounded edges.
   - **Selected Vector** – This option allows you to use a cross-section to define the profile of the shape.

     If you have selected this option, hold the **Shift** key down on your keyboard, and then click on the open vector object that you want to use as the cross-section. Both the selected vector objects turn magenta and are surrounded by a bounding box.

4. In the **Heights** area, define the height of the blended shape:

   - To define the height at the centre of the blended shape, type its value in the **Inner** box.
   - If you want to set the height at the centre of the blended shape as equal to the highest point in the existing relief, click to select the **From Relief** option .

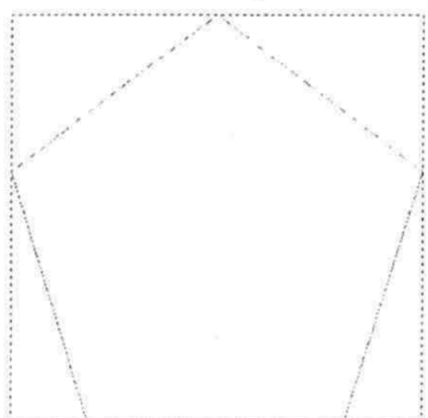
Note: If you select the **From Relief** option ☑, the **Centre** box, along with the **Filled Shape** and all four **Combine** options, are greyed-out.

- To define the height at the edge of the blended shape, type its value in the **Border** box.
- If you want to set the height at the edge of the blended shape as equal to the lowest point in the existing relief, click to select the **From Relief** option ☑.

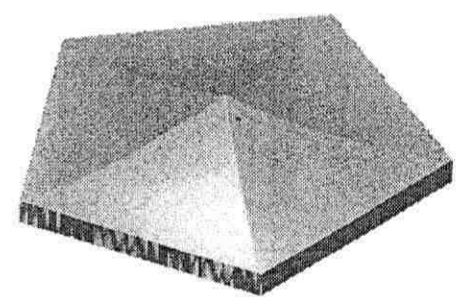
Note: If you select the **From Relief** option ☑, the **Border** box, along with the **Filled Shape** and all four **Combine** options, are greyed-out.

For example, a blended shape with a concave profile, a centre height of 15 mm and a border height of 5 mm is shown below:

Selected Vector Object...



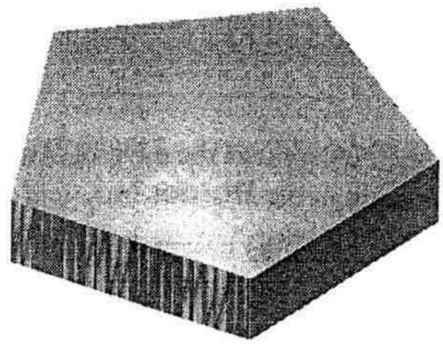
Blended Shape...



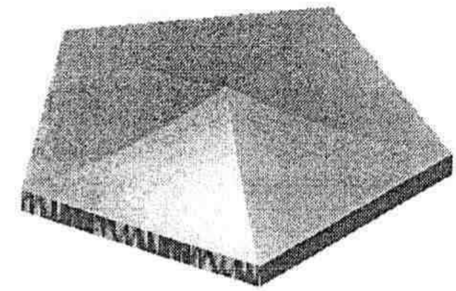
In this example, the default centre option and proportion value is used. This means that the centre of gravity in the polygon defines the centre of the blended shape and the concave profile extends across its entire surface.

Alternatively, the same blended shape could be produced from a polygonal relief with a Z height of 15 mm. In this instance, the centre height is taken from the relief and the border height is defined as 5 mm:

Relief Before...



Blended Shape...

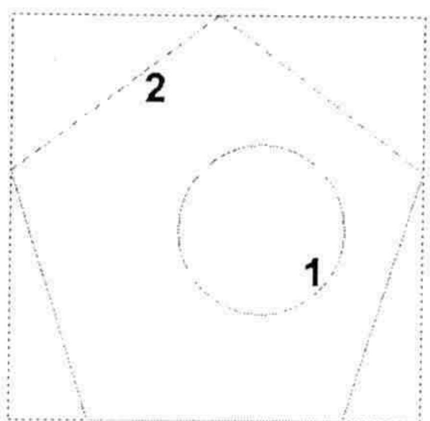


5. In the **Blend From Border To** area, click on one of the options to define the centre of the blended shape:

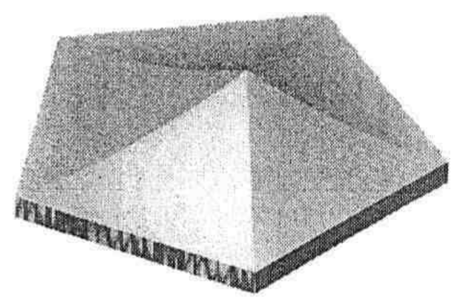
    - **Middle of Border Vector** – This option instructs ArtCAM Pro to use the centre of gravity in the vector object selected first as the centre of the blended shape.

      In our example, a circle within the polygon is used to define the centre height of the blended shape:

Selected Vector Objects...



Blended Shape...



- **Select Point with Cursor** – This option allows you to define the centre of the blended shape in either of the following ways:

  Click on the **Start** button, and then on the centre position in the **2D View** window. The X and Y co-ordinates of the centre appear in the **X** and **Y** boxes.

  Type the X and Y co-ordinates of the centre position in the **X** and **Y** boxes.

- **Inner Vector Edge** – This option instructs ArtCAM Pro to use the outline of another vector object drawn within the first as the centre of the blended shape.

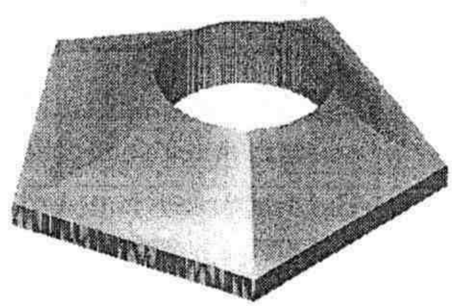
  If you have selected this option, make sure that the second vector object is positioned correctly within the first. Hold the **Shift** key down, and then click on the vector object to select it. The two vector objects turn magenta.

  If you want to create a planar cap in the shape of the second vector object at the height specified for the centre of the blended shape, make sure that the **Fill Centre** option is turned on ☑.

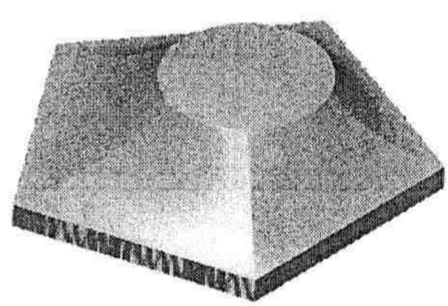
Note: If you have selected a **From Relief** option to define the blended shape's centre or border height, this option is greyed-out.

In our example, the edge of the circle is used to define the centre of the blended shape. You can see how filling the centre affects its overall shape:

Fill Centre Option OFF...



Fill Centre Option ON...



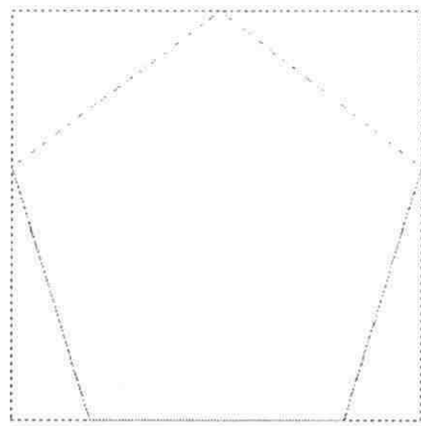
6. If you want to control the area over which the profile is extended across the surface of the blended shape, you can use either of these methods:

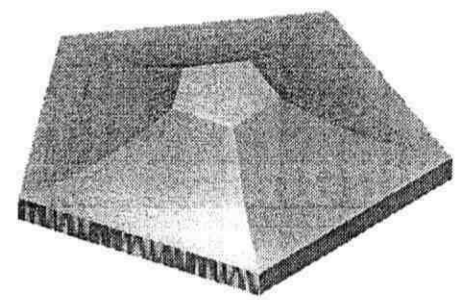
   - In the **Proportion** box, type the percentage of the area from the outside-edge of the blended shape to its centre across which you want to extend the profile.
   - Click and drag on the slider to set the area from the outside-edge to the centre of the blended shape across which you want to extend the profile.

Any value less than 100% produces a blended shape with a planar cap at the height previously specified for its centre.

In our example, if the proportion is adjusted to 75% the blended shape now appears as shown:

Selected Vector Object... *Blended Shape...*





You can see that the concave profile does not extend across the entire surface of the blended shape, and a polygonal shaped planar cap is formed over its centre.

7. In the **Combine** area, select how you want to combine the blended shape with the existing relief:

Note: If you have selected a **From Relief** option to define the blended shape's centre or border height, go straight to the next step.

- Click on the **Add** radio button to add the blended shape to the points in the existing relief.
- Click on the **Subtract** radio button to subtract the blended shape from the existing relief.
- Click on the **Highest** radio button to merge the blended shape with the existing relief, so that only the highest points of the two remain.
- Click on the **Lowest** radio button to merge the blended shape with the existing relief, so that only the lowest points of the two remain.

8. Click on the **Create Blend** button to create the blended shape. A progress bar appears beneath the **2D View** window indicating the progress ArtCAM Pro is making in calculating the blended shape.

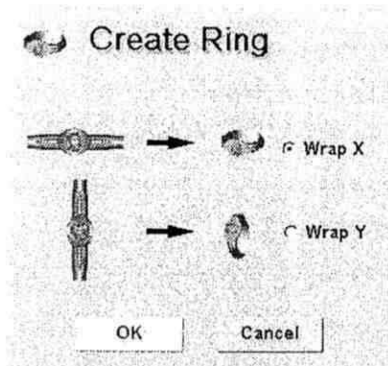
9. Click on the **Close** button to return to the **Assistant**'s Home page.

Creating a Ring

You can display a relief as it would appear when machined as a ring on a rotary axis CNC machining tool.

To create a ring:

1. From the Main menu bar, click on the **Relief** menu, and then on the **Create Ring** option to display the **Create Ring** dialog box:



2. Select the axis around which you want to wrap the relief:
   - Click on the **Wrap X** radio button if you want to wrap the relief around the X-axis.
   - Click on the **Wrap Y** radio button if you want to wrap the relief around the Y-axis.
3. Click on the **OK** button to close the **Create Ring** dialog box and create the ring.

Adding Texture to a Relief

You can apply texture to a relief in two different ways:

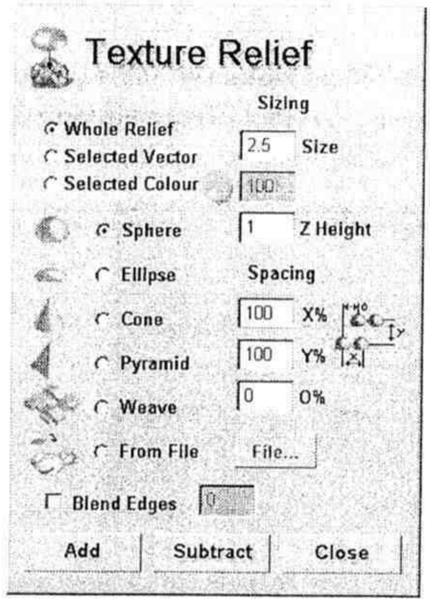
- Apply a basic pattern created in ArtCAM Pro.
- Import a relief.

Texture can either be applied to the whole relief, or to those areas of the relief under the current Primary Colour or as defined by a selected vector object.

Both of these methods of applying texture to a relief work in the same way. The pattern is selected or the relief file is imported, the size of the pattern is defined, then the repeat distance in X and Y is defined along with the pattern overlap (O). Finally, the texture is tiled onto the relief by either adding or subtracting from its surface.

To add texture to the relief:

1. Click on the **Texture Relief** button  in the **Relief Editing** area of the **Assistant**'s Home page to display the **Texture Relief** dialog box:



2. Define the area of the relief to which you want to apply a texture. By default, the **Whole Relief** option is selected. This means that the texture covers the entire relief surface.

   Alternatively, you can either use a vector object or bitmap colour to define the area(s) of the relief to which you want to apply a texture. If you want to use a vector object:

   - Click to select the vector object from the **2D View** window, and then on the **Selected Vector** radio button. For details, see "Selecting Vectors" in the Working with Vectors chapter.

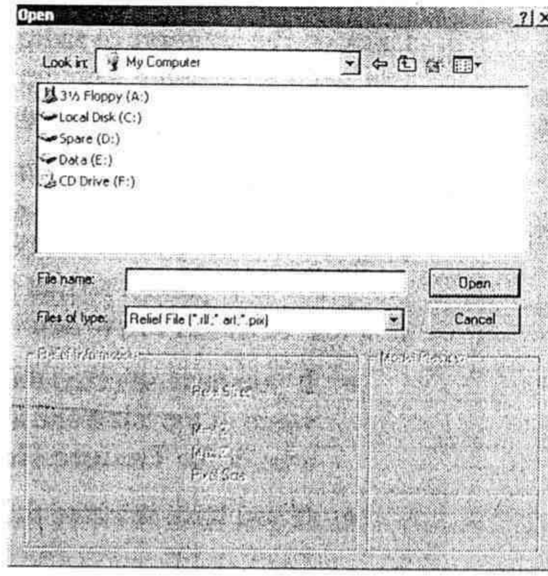
   If you want to use a bitmap colour:

   - Select the Primary Colour from the Colour Palette beneath the **2D View** window, and then click on the **Selected Colour** radio button. For details, see "Selecting the Primary and Secondary Colours" in the Working with Bitmaps chapter.

3. Click on the appropriate radio button ◉ to select the texture you want to combine with the existing relief, and then set its size and height.

   The size of the texture determines how it is applied to the relief. If the texture's current size does not match that of the area of the relief to which it is being applied, you can either compress or expand it so that it fits perfectly over the area, or allow ArtCAM Pro to automatically tile it over the area.

   - If you have selected the **Sphere** option ◉, type a value in the **Size** and **Z Height** boxes in the **Sizing** area of the **Texture Relief** dialog box.
   - If you have selected the **Ellipse** option ◉, type a value in the **Size**, **Horizontal %** and **Z Height** boxes in the **Sizing** area of the **Texture Relief** dialog box.
   - If you have selected the **Cone** option ◉, type a value in the **Size**, **Tip Radius %** and **Z Height** boxes in the **Sizing** area of the **Texture Relief** dialog box.
   - If you have selected the **Pyramid** option ◉, type a value in the **Size**, **Truncation %** and **Z Height** boxes in the **Sizing** area of the **Texture Relief** dialog box.
   - If you have selected the **Weave** option ◉, type a value in the **Size**, **Bar Width %** and **Z Height** boxes in the **Sizing** area of the **Texture Relief** dialog box.
   - If you have selected the **From Relief** option ◉, click on the **File** button to display the **Open** dialog box:



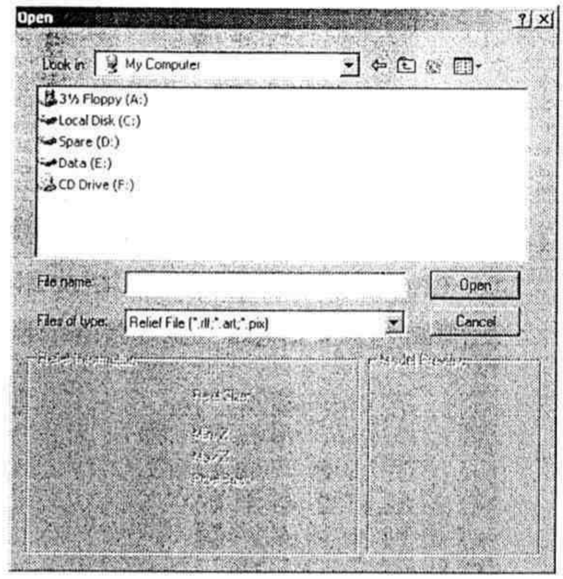
Click on the **Look in** box and select the directory where the relief file that you want to use is stored, and then click on the relief file itself. Click on the **Open** button to load the selected relief file. The relief dimensions appear in the **Sizing** and **Spacing** areas of the **Texture Relief** dialog box.

4. In the **Spacing** area, define the repeat distance along the X-axis in the **X%** box.
5. Define the repeat distance along the Y-axis in the **Y%** box.
6. Define the overlap distance in the **O%** box.
7. If you want to blend the edges of the selected texture into the relief surface, click to select the **Blend Edges** option ☑.
8. Select how you want to combine the texture with the existing relief:
   - To add the texture to the existing relief, click on the **Add** option. For more details, see "Adding to the Relief" on page 273.
   - To subtract the texture from the existing relief, click on the **Subtract** option. For more details, see "Subtracting from the Relief" on page 275.
9. Click on the **Close** button to close the **Texture Relief** dialog box.

Adding Texture Example

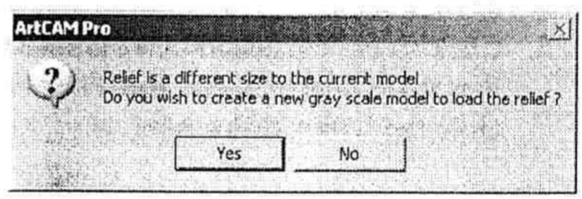
In the following example, you can see how the **Texture Relief** dialog box is used to add texture to an imported relief to create a new, textured relief.

1. From the Main menu bar, click on the **Relief** menu, and then on the **Load > Replace** option. The **Open** dialog box appears:



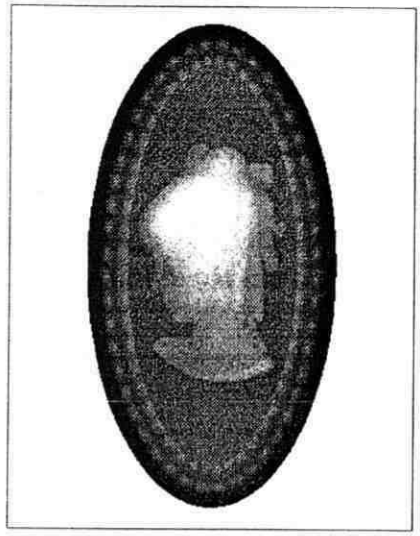
2. Click on the **Fin_pend.rlf** file in ArtCAM Pro 6.0\Examples\Overview to select it, then click on the **Open** button to import the relief.

   If the loaded relief is larger or smaller than the existing relief, the following message box appears:

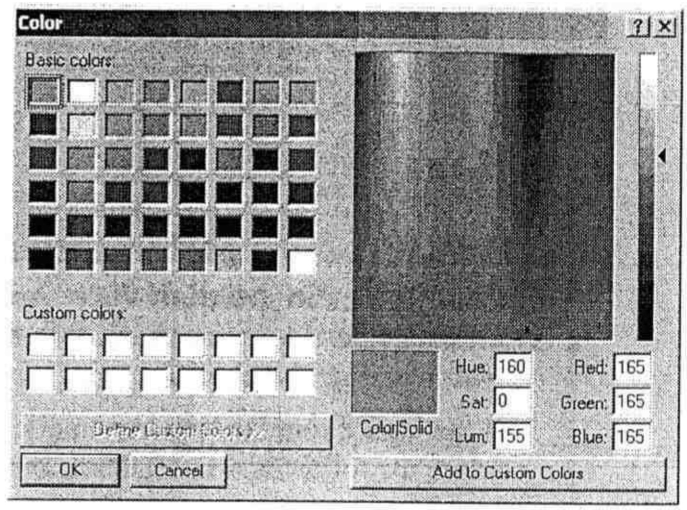


   Click on the **Yes** button. The new model is created, and a greyscale image of the loaded relief is shown in the **2D View** window.

3. Click on the **2D View** window to select it. A greyscale image of the imported relief appears in the model as follows:

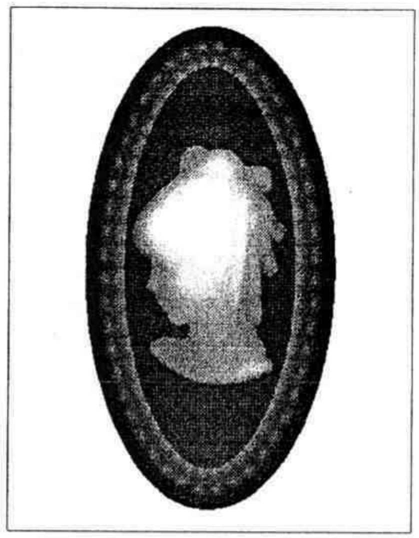


4. Click on the **Add Colour** button  in the **Bitmap Editing** area to display the **Color** dialog box:

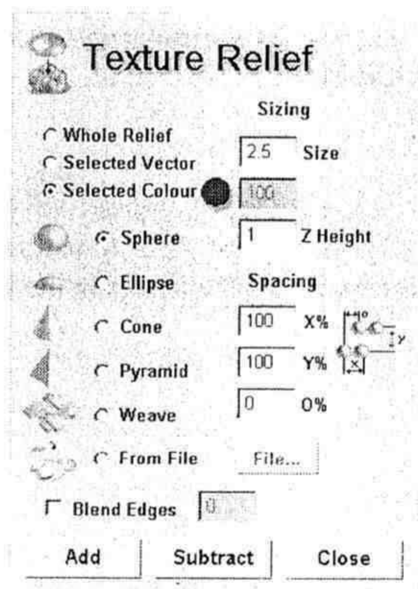


5. Click on the red colour in the **Basic Colors** area to select it as the Primary Colour, then click on the **OK** button to close the **Color** dialog box.

6. Click on the **Flood Fill** button  in the **Bitmap** toolbar, then click on the area surrounding the lady's head in the **2D View** window to flood fill it in red. The greyscale image of the imported relief appears in the model as follows:

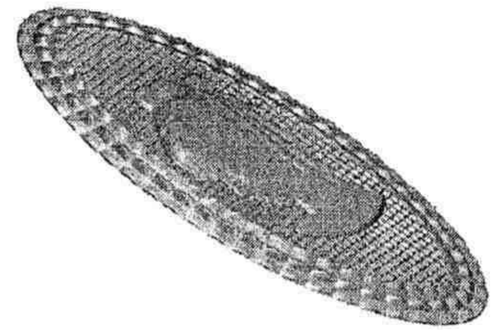


7. Click on the **Texture Relief** button in the **Relief Editing** area to display the **Texture Relief** dialog box:



8. Click to select the **Selected Colour** radio button.
9. Click to select the **Pyramid** radio button.
10. Type a value of 0.5 in the **Size** box.
11. Type a value of 50 in the **Truncation %** box.
12. Type a value of 0.1 in the **Z Height** box.
13. Click on the **Add** button to apply the texture to the area of the relief that is flood filled in red.

14. Click on the **Close** button to close the **Texture Relief** dialog box.
15. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. The new relief should appear something like the image shown below:

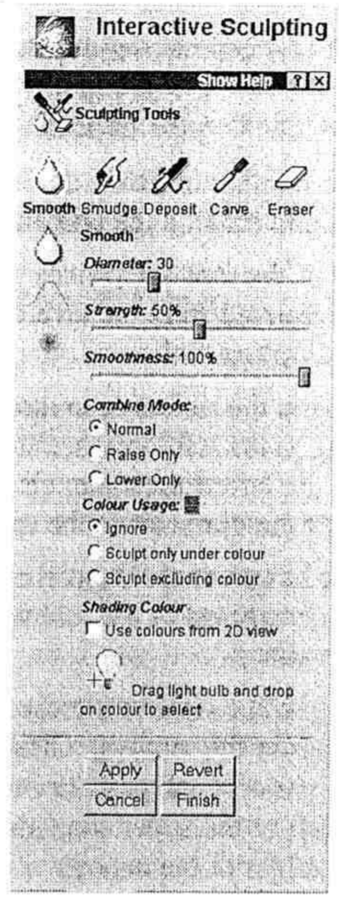


Sculpting a Relief

You can edit the shape of the existing relief using the sculpting tools available in ArtCAM Pro.

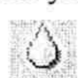
To sculpt a relief:

1. Click on the **Sculpting** button in the **Relief Editing** area of the **Assistant**'s Home page to display the **Interactive Sculpting** page in the **Assistant** window:



The **3D View** window is also displayed, showing the existing relief.

2. In the **Sculpting Tools** area, click to select which of the five tools you want to use:

   - Smooth If you want to smooth an area of the relief by blending one area with the next, click on the **Smooth** tool.

   - Smudge If you want to drag on an area of the relief to extend or reduce it, click on the **Smudge** tool.

   - Deposit If you want to add material to the relief, click on the **Deposit** tool.

   - Carve If you want to remove material from the relief, click on the **Carve** tool.

- Eraser If you want to remove the changes that you have made to the relief in the current sculpting session, click on the **Eraser** tool.

3. To change the number of pixels that make up the width of the tool, click and drag on the **Diameter** slider.

   The diameter of the tool determines how much material is deposited or removed from the area of the relief affected by the selected sculpting tool.

   Click and drag to the right to increase the width of the tool. Click and drag to the left to reduce the width of the tool. The image to the left of the slider changes as you move the slider to reflect the diameter of the tool.

4. To change the height of the deposit or the depth of the removal as a percentage of the tool width, click and drag on 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. The image to the left of the slider changes as you move the slider to reflect the height of the deposit or the depth of the removal.

5. To change the sharpness of the tool tip, click and drag on the **Smoothness** slider.

   Click and drag to the right to sharpen the tip of the tool. Click and drag to the left to soften the tip of the tool. The image to the left of the slider changes as you move the slider to reflect the smoothness of the tool.

6. If you want to sculpt the relief using a bitmap colour projected onto its surface:

   - First, click to select the **Use colours from 2D view** option in the **Shading Colour** area. The bitmap image or bitmap colours in the **2D View** window are projected onto the relief shown in the **3D View** window.

   - Next, click and drag the icon over the colour that you want to select. The colour is displayed in the **Colour Usage** area

- Finally, click on the appropriate **Colour Usage** radio button :

  Ignore – Click on this option if you do not want to sculpt underneath or around the selected colour.

  Sculpt only under colour – Click on this option if you only want to sculpt underneath the selected colour.

  Sculpt excluding colour – Click on this option if you only want to sculpt around the selected colour.

7. If you have selected the **Smooth** or **Smudge** tool, click to select a combination method from the **Relief Combine Mode** or **Combine Mode** area to use when sculpting the relief:

   - To use the average smoothing result to raise and lower the region under the tool cursor , select the **Normal** option.
   - To raise the lowest points under the tool cursor only, select the **Raise Only** option.
   - To lower the highest points under the tool cursor only, select the **Lower Only** option.

   If you have selected the **Deposit** tool, click on the appropriate **Relief Combine Mode** radio button :

   - To deposit material onto the area of the relief under the tool cursor , select the **Add** option.
   - To deposit a single layer of material onto the area of the relief under the tool cursor , regardless of how many times it passes over it, select the **Merge** option.

   If you have selected the **Carve** tool, click on the appropriate **Relief Combine Mode** radio button :

   - To carve material out of the area of the relief under the tool cursor , select the **Add** option.
   - To carve a single layer of material out of the area of the relief under the tool cursor , irrespective of how many times it passes over it, select the **Merge** option.

8. Click and drag the tool cursor in the **3D View** window to sculpt the relief according to the options you have selected on the **Interactive Sculpting** page.
9. Click on the **Apply** button to confirm the changes that you have made to the relief.

   If you want to undo the changes that you have made and return the relief to its original state, or as it was when the **Apply** button was last clicked in this Interactive Sculpting session, click on the **Revert** button.

   If you do not want to keep any of the changes you have made in this Interactive Sculpting session, click on the **Cancel** button.
10. Click on the **Finish** button to close the **Interactive Sculpting** page and return to the **Assistant**'s Home page.

Removing Holes in the Relief Surface

You can remove any small holes that can be seen on the relief surface.

To remove any small holes on the relief surface:

1. From the Main menu bar, click on the **Relief** menu, and then on the **Remove Small Holes** option.
2. Click on the **3D View** button 3D in the **2D View** toolbar to display the **3D View** window. Check the relief surface for any small holes that might still be present, and if necessary repeat these steps.

Creating a Greyscale Image from a Relief

A greyscale is an image in which the only colours are shades of grey. You can create a greyscale image from the existing relief. For details, see "Creating a Greyscale Image from a Relief" in the Working with Models chapter.

Rotating a Relief or Triangle Mesh

You can rotate the relief or triangle model shown in the **3D View** window. This allows you to improve the visualisation of the relief or triangle model, especially when a reflection map has been projected onto its surface. For details, see "Adjusting Light and Material Settings" in the Working with Models chapter.

To rotate a relief or triangle model:

1. Click on the **3D View** button 3D to display the **3D View** window.
2. Hold the **Ctrl** key down on your keyboard, move the cursor over the relief or triangle model, and then push the mouse in the direction in which you want to rotate it.

To stop the relief or triangle model revolving, click anywhere in the **3D View** window.

You can adjust the speed at which the relief or triangle model revolves using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter

Machining Models

Overview

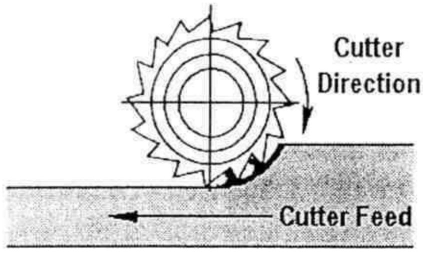
You use toolpath strategies to machine both two-dimensional and three-dimensional models. 2D toolpath strategies are used to machine all or part of a model from the vector objects you have created. 3D toolpath strategies are used to machine all or part of a model from the three-dimensional shapes you have combined to make the relief.

ArtCAM Pro provides a database containing a broad selection of predefined tools that you can use when creating your toolpath strategies. You can edit the machining parameters of these tools and add new tools to the database if necessary.

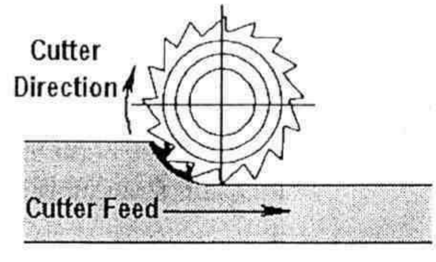
You can control the cut direction of the tools used for machining by selecting either Conventional or Climb Milling in almost all of the toolpath strategies available in ArtCAM Pro.

In Conventional Milling, the teeth of the tool meet the block of material at the bottom of the cut. The teeth of the tool slide along until sufficient pressure builds up to break through the material surface. This sliding action, under pressure, tends to abrade the edge of the tool, which results in dulling. Also, the cutting action has a tendency to lift the block of material, fixture and table from their positions.

Climb Milling...



Conventional Milling...



In Climb Milling, the teeth of the tool meet the block of material at the top of the cut, at the thickest part of the chip. This provides instant engagement of the tool with the block, providing a chip of definite thickness at the start of the cut. Climb Milling does not cause the abrasive action cause by Conventional Milling. It also permits the gradual removal of the tool from the block, so that dwell marks are largely eliminated. Climb Milling often provides a better finish, permits greater tool feed rates and prolongs the life of a tool.

You can control how the tool enters the block of material by adding ramping moves, also known as zigzag plunge moves, to almost all of the toolpath strategies available in ArtCAM Pro.

Feeding the cutter into the material surface in a straight plunge motion at full feed rate often causes pivoting around the tool in the direction of rotation. This, in turn, causes deflection. It is this deflection that causes gouging and, consequently, tool damage. Adding ramping moves, also known as zigzag plunge moves, allows the cutting tool to enter the block of material gradually, ensuring minimum tool damage and reducing the likelihood of gouging.

ArtCAM Pro allows you to create multiple toolpath strategies, each of them relating to a selected area of your model. This allows a number of roughing passes to be made, removing excess material prior to the finishing passes.

You can calculate a toolpath strategy individually or as part of a batch. Calculating toolpath strategies in a batch allows you to automatically process a collection of different toolpath strategies at once. This gives you the freedom to work continuously on creating a model, and calculate its corresponding toolpath strategies outside of working hours.

You can simulate any toolpath strategy after it has been calculated. This allows you to visualise the surface finish and the machining passes that are used to machine all or part of your model.

You can save a toolpath strategy as a template. This allows you to apply the same toolpath settings to different parts of a model without having to create a toolpath strategy over and over again. A template can be applied to a vector object or relief in a model.

In order to machine the model that you have created, you must create a toolpath file from the toolpath strategies you have calculated. A toolpath file contains one or more calculated toolpath strategies. Each toolpath strategy is made up of a sequence of commands which tell a

CNC machine tool or laser engraving system the path it is required to follow in order to produce your model.

You can save the toolpath file in a format recognised by your machine tool or engraving system. This ensures that the commands within the toolpath file are interpreted accurately by your system.

If your CNC machine has a tool changer, you can group all of the toolpath strategies for a series of tools into one machine-specific file.

If your CNC machine does not have a tool changer, you must generate separate toolpath files for each tool that you want to use.

Using Toolpath Strategies

ArtCAM Pro provides a number of toolpath strategies that you can use to machine all or part of a two-dimensional or three-dimensional model. These are available using the **Toolpaths** Home page, which is displayed when you click on the **Toolpaths** tab.

For details of the 2D toolpath strategies that are available in ArtCAM Pro, see "2D Toolpath Strategies" on page 337.

For details of the 3D toolpath strategies that are available in ArtCAM Pro, see "3D Toolpath Strategies" on page 408.

You can also find buttons on the **Toolpaths** Home page that assist you in managing and modifying the toolpath strategies you have created. For details, see "Managing and Modifying Toolpaths" on page 437.

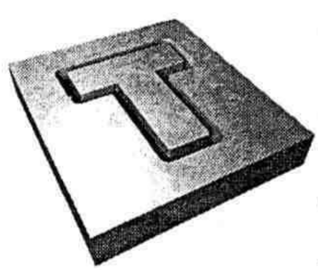
2D Toolpath Strategies

ArtCAM Pro provides a number of two-dimensional toolpath strategies that you can use to machine your model, based on the vector objects created within it. These include:

- **Profiling**. This strategy allows you to create a toolpath either inside or outside the boundary of a vector object. For details, see "Profiling" on page 338.
- **2D Area Clearance**. This strategy allows you to clear an area of material either inside or outside the boundary of a vector object. You can use a range of tools with this strategy and ArtCAM Pro automatically calculates the order in which they should be used. For details, see "2D Area Clearance" on page 346.

- **V-Bit Carving**. This strategy allows you to replicate a hand carved look for the vector object or vector text you want to machine. For details, see "V-Bit Carving" on page 352.
- **Bevel Carving**. This strategy allows you to replicate a bevelled (angle-edged) look for the vector object or vector text you want to machine. For details, see "Bevel Carving" on page 355.
- **Smart Engraving**. This strategy allows you to engrave around the boundary of a vector object or group of vector text. You can use a range of tools with this strategy and ArtCAM Pro automatically calculates the order in which they should be used. For details, see "Smart Engraving" on page 361.
- **Drilling Holes**. This strategy allows you to create drill holes using vector objects. For details, see "Drill Holes" on page 403.
- **Machine Vectors**. This strategy allows you to machine along the boundary of a vector object with the centre of a tool. For details, see "Machine Vectors" on page 367.
- **Inlay Machining**. This strategy allows you to create a selection of corresponding inlays (female) and inserts (male) from a vector object. For details, see "Inlay Wizard" on page 371.

Profiling



The **2D Profiling** button  in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that machines either inside or outside the boundary of a selected vector object, relative to the dimensions of the block of material.

A Profiling toolpath strategy is ideal for cutting out letters and shapes from a block of material.

Using the **Profiling** page, you can:

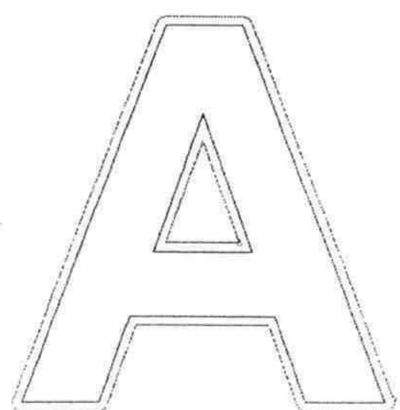
- Control where your tool cuts into and retracts from the block of material by adding lead in and lead out moves.
- Control how your tool cuts into and retracts from the block of material by adding ramping moves.

- Control the cutting direction of the tool.
- Control whether the vector object is cut or snapped out from the block of material by defining the thickness of the final machine pass and/or adding bridging.

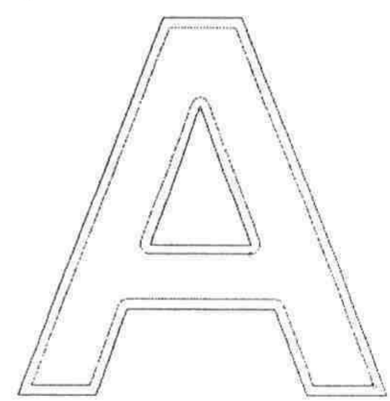
To profile a vector object:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Select the vector object that you want to profile. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **2D Profiling** button in the **2D Toolpaths** area to display the **Profiling** page.
4. In the **Profile Side** area, select how you want to profile the selected vector object:
    - Click on the **Outside** radio button to instruct the tool to profile outside of the selected vector object.
    - Click on the **Inside** radio button to instruct the tool to profile inside of the selected vector object.

Profile Side – Outside...

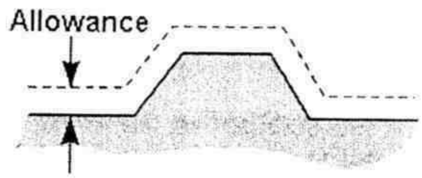


Profile Side – Inside...



5. Type the depth (Z) at which you want to cut into the surface of the material in the **Start Depth** box.
6. Type the depth (Z) for the bottom of the cut in the **Finish Depth** box.
7. If you want to add or remove extra material around the vector object, you can type a value in the **Allowance** box.

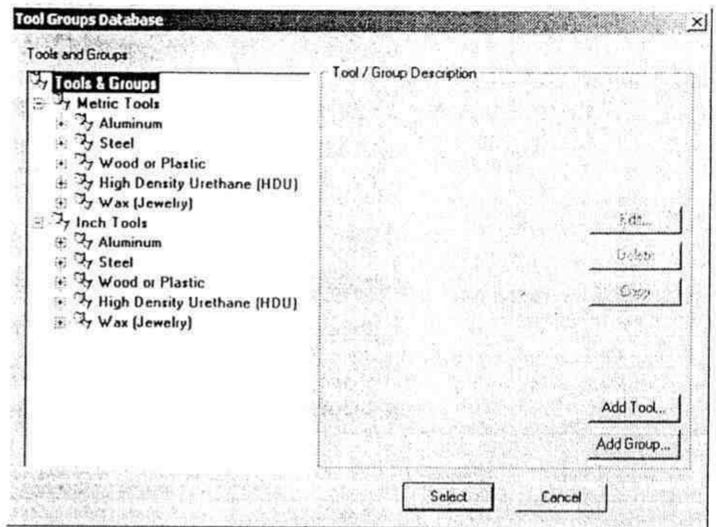
The value you enter sets the distance between the boundary of the selected vector object and the profiling tool. Type a positive value to add material or a negative value to remove it.



8. Type a value in the **Tolerance** box to specify how closely you want the cutter to follow the shape of the vector object.
9. To set the thickness of the final profile pass in the toolpath, click to select the **Final Pass Thickness** option ☑, and then define its thickness in the adjacent box.

   If your **Final Pass Thickness** is less than the thickness of the block of material, a thin 'web' of material is left around the boundary of the vector object. You can gently snap it from the block of material afterwards.
10. If you want to change the height at which the Profiling tool makes rapid moves between toolpath segments:
    - Frist, click on the ▼ arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:
    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, click on the ▲ arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
11. Click on the **Select** button in the **Profiling Tool** area of the page to open the **Tool Groups Database**:



12. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Profiling Tool** area.

    If you want to amend the machining parameters for the selected tool, click on the arrow in the **Profiling Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

    When you have finished, click on the arrow to hide the machining parameters.

13. The cutting direction defaults to **Climb Mill**. If you want to change this, click on the arrow in the **Cut Direction** area, then click on either of the **Cut Direction** radio buttons :

    - **Climb Mill** - Climb Milling rotates the cutter in the same direction as the feed motion.
    - **Conventional** – Conventional Milling rotates the cutter in the opposite direction to the feed motion.

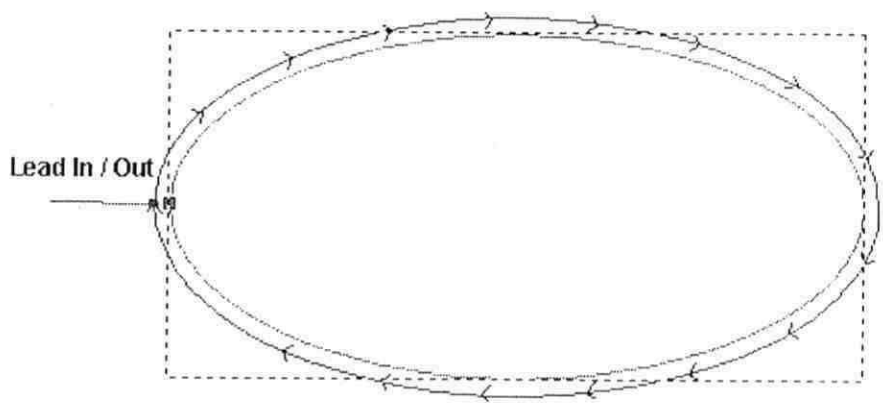
    For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

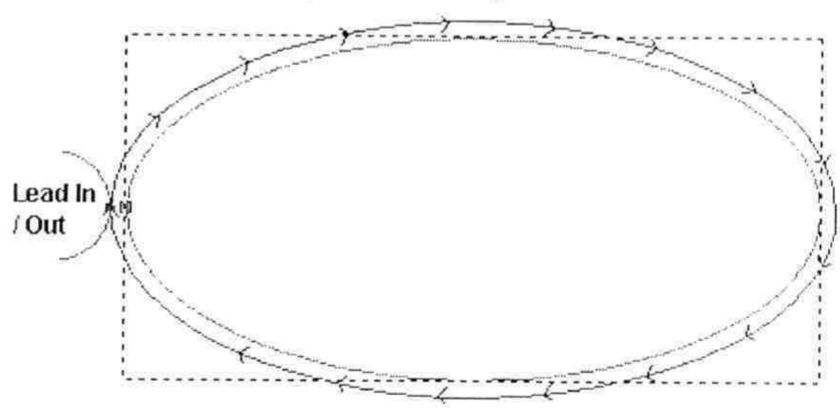
14. To add lead-in and lead-out moves to the toolpath, click to select the **Add Lead In/Out Moves** option . The lead moves settings are displayed:

- If you only want to have a lead-in move, click to select the **Do not Lead Out** option ☑.
- Type the distance from the toolpath that you want the tool to cut into and retract from the vector object's boundary in the **Distance (D)** box.
- Select how you want the tool to move using the **Add Lead In/Out Moves** radio buttons ◉:

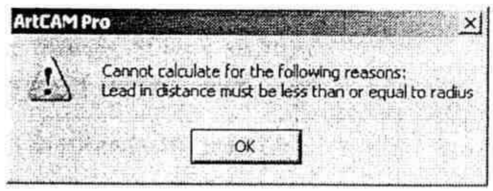
  Linear – Click on the **Linear** option ◉ to instruct the tool to lead into and out of the vector object's boundary in a straight-line motion:



Circular Arc – Click on the **Circular Arc** option ◉ to instruct the tool to lead into and out of the vector object's boundary in an arc motion:



If you select the **Circular Arc** option, define the radius of the arc in the **Radius (R)** box. The radius must be less than or equal to the value in the **Distance (D)** box, otherwise the following message box appears when you calculate the toolpath:



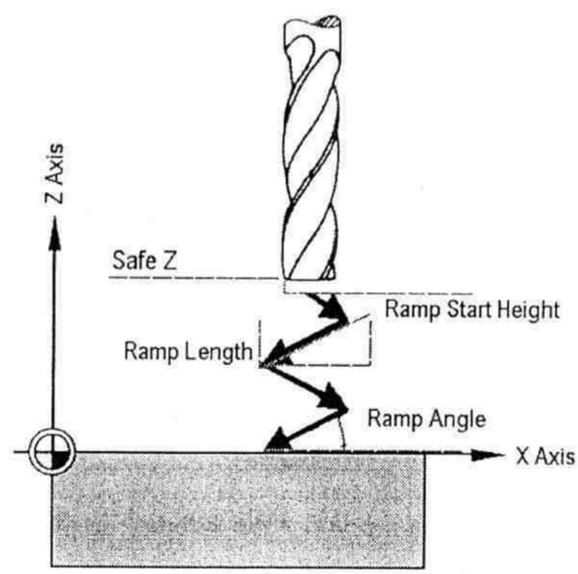
Click on the **OK** button to close the message box.

- Set the **Automatic Positioning** as follows:

  Click to select the **Automatic Positioning** option. This positions lead-in and lead-out moves at the optimum point in the vector object, which is usually within its longest linear span.

  Click to deselect the **Automatic Positioning** option. This positions lead-in and lead-out moves at the Start Point of the vector object. If you want to change the position of lead-in and lead-out moves, you can change the Start Point. See "Changing the Start Point" in the Working with Vectors chapter for details.

15. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.

- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

16. If you are machining a block of vector text, you can instruct the tool to machine according the order in which the text was created by clicking to select the **Preserve Text Order** option ☑. The cutting tool moves along the first line of text from left to right, until it locates a carriage return when it continues machining along the next line from right to left. This process is repeated until the last character in the last line of text is located.
17. Click on the **Setup** button in the **Material** area of the page to display the **Material Setup** dialog box.

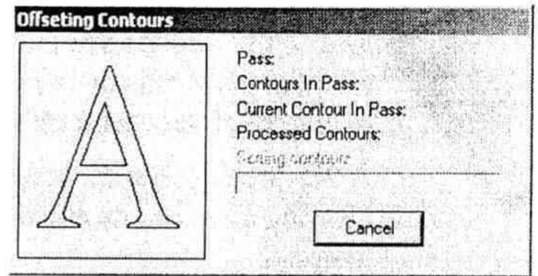
    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, and then click on the **OK** button. If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.
18. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option☐.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

---

19. Type a name for the toolpath in the **Name** box.
20. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog

box is displayed while ArtCAM Pro calculates the toolpath:



A dark red line represents the machining passes used to profile the selected vector object. It is drawn either inside or outside of its boundary, according to the **Profile Side** that you had selected. A circle marking the current start position is drawn on the toolpath preview. For details, see “Changing the Start Position” on page 454.

The toolpath preview is not shown if the **Create 2D Preview** option is deselected.

You can now simulate the toolpath. For details, see “Simulating Toolpaths” on page 485.

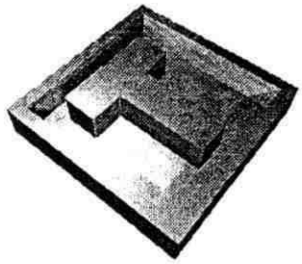
- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see “Calculating a Single Toolpath” on page 469 and “Calculating a Batch of Toolpaths” on page 469.

21. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to a profile pass in the Profiling toolpath, see “Adding Bridging” on page 450.

If you want to set the machining order of the profile passes in the Profiling toolpath, see “Setting the Machining Order” on page 458.

2D Area Clearance



The **Area Clearance** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that clears an area of material based on the boundary of the vector object you have selected.

A 2D Area Clearance toolpath strategy is ideal for clearing areas of unwanted material in a block of material.

Using the **2D Area Clearance** page, you can:

- Select multiple tools to perform *smart machining*, where ArtCAM Pro automatically uses the biggest tool first, and then the smaller tools in order of size to machine the areas which could not be reached by previous tools.
- Select a different tool clearance strategy for each of the tools you want to use. Depending on what strategy you select, you can control the angle, or where and in what direction the tool cuts into the block of material.
- Control how your tools cut into and retract from the block of material by adding ramping moves.

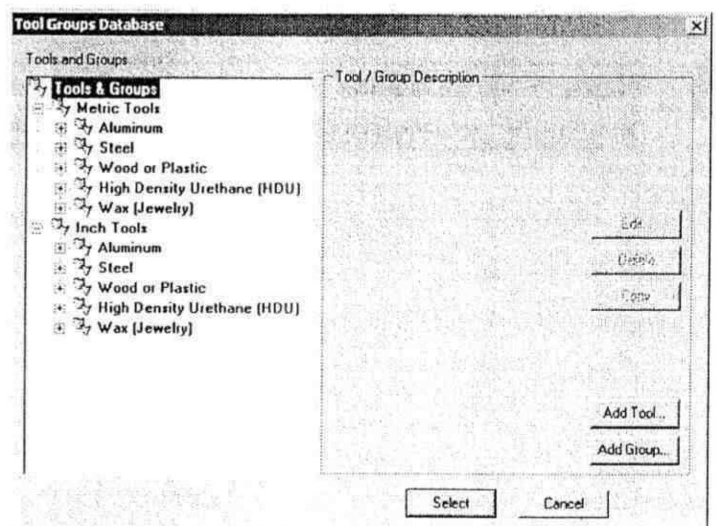
To clear an area of material:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the vector objects that represent the areas you want to clear. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **2D Area Clearance** button in the **2D Toolpaths** area to display the **2D Area Clearance** page.
4. Type the depth (Z) from the surface of the material at which you want to begin clearing the area in the **Start Depth** box.
5. Type the depth (Z) of the bottom of the area you want to clear in the **Finish Depth** box.
6. To add or remove extra material around the vector objects, type a value in the **Allowance** box. The value you enter sets the distance between the boundary of the selected vector object and the tool used. Type a positive value to add material or a negative value to remove it.

7. If you are using more than one tool, type an allowance to be left on by larger tools for the smallest tool to machine when clearing around the selected vector objects in the **Final Tool Allowance** box.
8. Type a value in the **Tolerance** box to specify how closely you want the cutting tool(s) to follow the shape of the vector object.
9. If you want to change the height at which a cutting tool makes rapid moves between toolpath segments:
    - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:

    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
10. Click on the **Add** button in the **Tools List** area to open the **Tool Groups Database**:



11. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tools List** window.

The tool's machining parameters are displayed in the area below. If you want to amend the machining parameters, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

When you have finished, click on the arrow to hide the machining parameters.

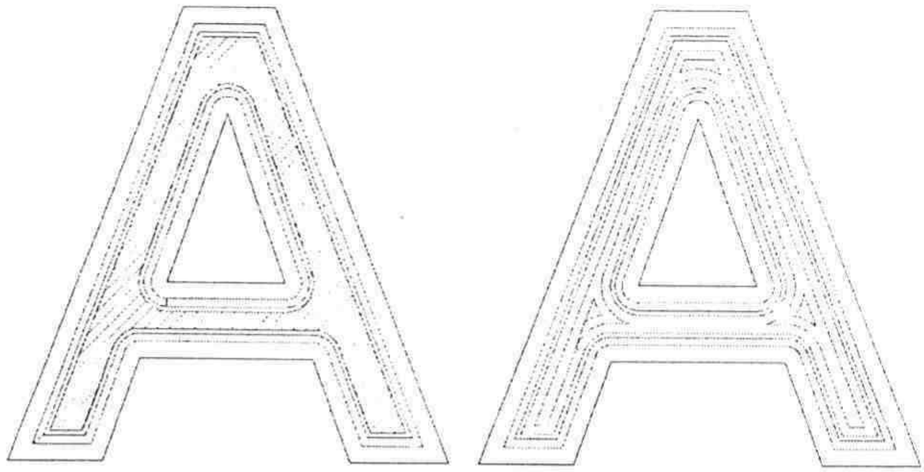
12. If you want to use more than one tool, click on the **Add** button in the **Tools List** area and select the tool from the **Tool Groups Database**.
13. Click on the tool in the **Tools List** window whose tool clearance strategy you want to select.
14. Select how you want the tool to clear the area using the **Tool Clearance Strategy** radio buttons :
    - **Raster** - This strategy machines in passes back and forth along the X-axis at a specified angle. If you select this strategy, type the angle you want the tool to move at in the **Raster Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- **Offset** - This strategy machines in repeated passes, each time moving inwards by the selected tool's **Stepover** value.

Note: You can see the stepover value of the selected tool when the machining parameters are displayed in the **Tools List** area.

Raster Tool Clearance at 45°... *Offset Tool Clearance...*



If you select this strategy, first click on either of the **Cut Direction** radio buttons :

Climb Mill – In Climb Milling, the cutter rotates in the same direction as the feed motion.

Conventional – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on either of the **Start Point** radio buttons :

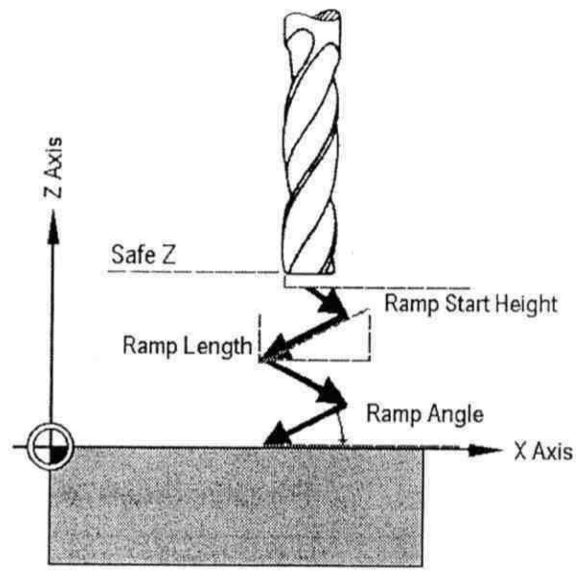
Outside - Select this option if you want the tool to cut into the material at the vector object's boundary, then machine inwards.

Inside - Select this option if you want the tool to cut into the material at the vector object's centre, then machine outwards.

15. If you want the selected tool to cut to a different depth than that displayed in the **Finish Depth** box at the top of the page, click to select the **Independent Finish Depth** option . If you have selected this option, type the relative depth (Z) of the bottom of the area you want to clear in the **Finish Depth** box below.

16. When you have set a tool clearance strategy for all of the tools in the **Tools List** area, you can choose whether or not to add ramping moves to the machining passes made by them:

    - To add ramping moves, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



    - Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
    - Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
    - Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
    - If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

17. Click on the **Setup** button in the **Material** area of the page to display the **Material Setup** dialog box.

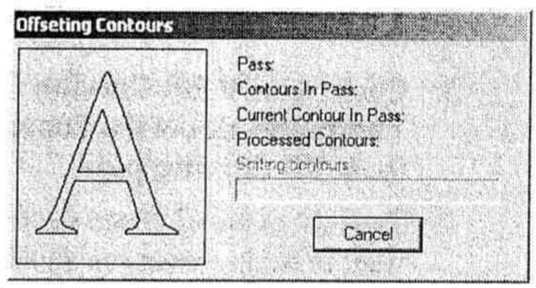
    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, and then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

18. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option .

19. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The machining passes used to clear the area about the selected vector object are represented by a sequence of dark red lines drawn either inside or outside of its boundary. The toolpath preview is not shown if the **Create 2D Preview** option is deselected .

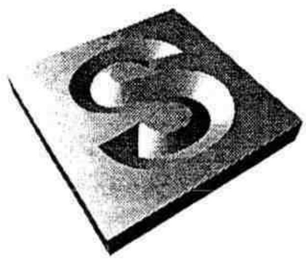
      A circle marking the position of the current Start Point is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

20. Click on the **Close** button to return to the **Toolpaths** Home page.

V-Bit Carving



The **V-Bit Carving** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that replicates a hand carved look for the vector object or text that you have selected.

The **V-Bit Carving** page allows you to control the depth of the centreline carved into the block of material.

To begin centreline carving your vector object or text:

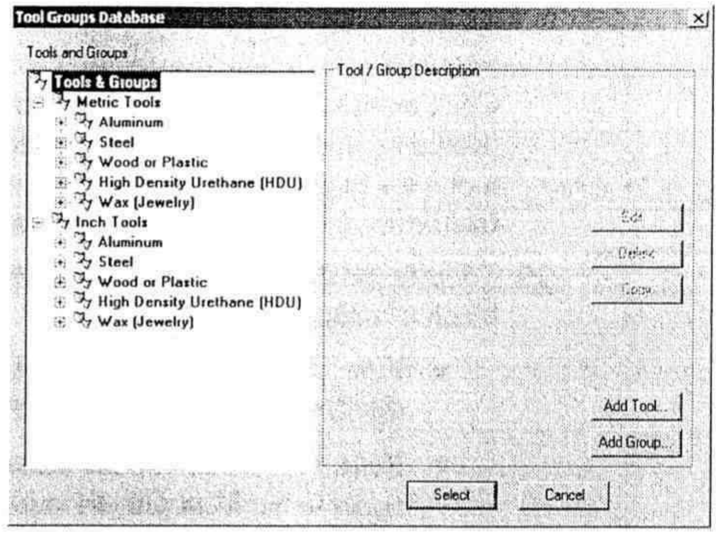
1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the vector object or text that you want to carve. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **V-Bit Carving** button in the **2D Toolpaths** area of the **Toolpaths** Home page to display the **V-Bit Carving** page.
4. Type the depth (Z) from the material surface at which you want to begin carving in the **Start Depth** box.
5. Type a value in the **Tolerance** box to specify how closely you want the Carving tool to follow the shape of the vector object.
6. If you want to change the height at which the Carving tool makes rapid moves between toolpath segments:
    - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:

    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.

- Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

7. Click on the **Select** button in the **Carving Tool** area of the page to open the **Tool Groups Database**:



8. Double-click on the V-Bit, Conical or Ball Nose tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Carving Tool** area.

   If you want to amend the machining parameters, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

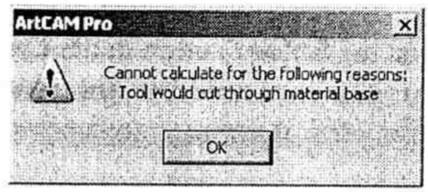
   When you have finished, click on the arrow to hide the machining parameters.

9. You can now calculate the maximum depth and width of the carving pass in the toolpath by clicking on the **Centreline** button.

   During the centreline calculation process, ArtCAM Pro displays a progress bar and a **Cancel** button beneath the design window area. If you want to stop this process, click on the **Cancel** button .

   After the centreline calculation process is complete, the maximum depth and width of the carving pass are shown below the **Carving Tool** area.

If the maximum depth of the carving pass is greater than the thickness of the material block, the following message box appears when the toolpath is calculated:



Click on the **OK** button to close the message box. To overcome this problem, you can select a different Carving tool with an angled edge, limit the maximum depth of the tool or use a thicker block of material.

10. To limit the depth at which the Carving tool cuts into the block of material:

    - First, click to select the **Limit tool maximum depth** option in the **Carving Tool** area.
    - Next, type a value in the **Maximum Depth** box equal to the **Max Depth** value shown below the **Carving Tool** area.

Warning: If you were to use a value less than the **Max Depth** calculated by ArtCAM Pro, the carving would be truncated.

11. If you are machining a block of vector text, you can instruct the tool to machine according the order in which the text was created by clicking to select the **Preserve Text Order** option . The cutting tool moves along the first line of text from left to right, until it locates a carriage return when it continues machining along the next line from right to left. This process is repeated until the last character in the last line of text is located.

12. Click on the **Setup** button in the **Material** area of the page to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

13. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option.
14. Type a name for the toolpath in the **Name** box.
15. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button.

      If you have not already calculated the carving pass in the toolpath, the selected vector object is colour-filled in cyan during the centreline calculation process.

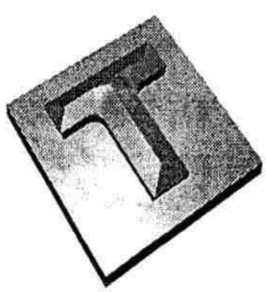
      Dark red lines drawn inside its boundary represent the machining passes used to carve the selected vector object. The toolpath preview is not shown if the **Create 2D Preview** option is deselected.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

16. Click on the **Close** button to return to the **Toolpaths** Home page.

Bevel Carving



The **Bevel Carving** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that replicates a bevelled (angled-edge) look for the vector text or object that you have selected.

To create a bevel carving, you must use two types of tool:

- An angled-edge carving tool to create the bevelled edge.
- A profiling tool to create the vertical wall below the bevelled edge.

The angle of the bevelled edge is determined by the angle of the carving tool.

Using the **3D Bevel Carving** page, you can:

- Choose whether or not you want to create a vertical wall in the bevel carving.
- Control the height of the angled edge in the bevel carving.
- Control the cutting direction of the tool.

To begin bevelling your vector text or object you must:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Select the vector object from which you want to create a bevel carving. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **Bevel Carving** button in the **2D Toolpaths** area to display the **Bevel Carving** page.
4. Type the depth (Z) from the material surface at which you want to begin carving in the **Start Depth** box. This is the apex of the bevel carved object.
5. Type the height of the wall in the block of material in the **Wall Height** box. This is the vertical wall below the bevelled edge. If you do create a vertical wall, a Profiling tool must be used to create it.
6. Type the depth (Z) for the bottom of the cut in the **Finish Depth** box.

Note: If you are cutting the vector object out of the block of material, the **Finish Depth** value is equal to the **Material Thickness** value entered in the **Material Setup** dialog box. For details, see "Adjusting the Material Setup" on page 481.

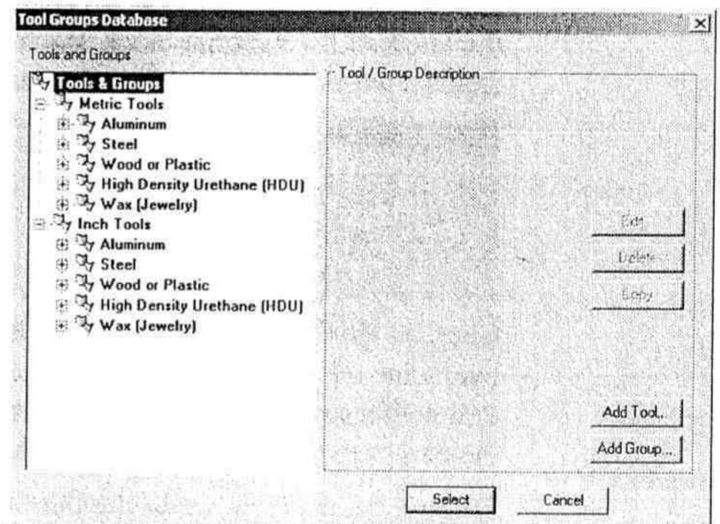
7. Type a value in the **Tolerance** box to specify how closely you want the Carving and Profiling tools to follow the shape of the vector object.
8. If you want to change the height at which the Carving and Profiling tools make rapid moves between toolpath segments:

- First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
- Next, define the height in the **Safe Z** box.

If you want to define the **Home Position** for the machining tools:

- First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
- Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

9. Click on the **Select** button in the **Carving Tool** area of the page to open the **Tool Groups Database**:



10. Double-click on the V-Bit or Conical tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Carving Tool** area.

    If you want to amend the machining parameters, click on the arrow in the **Carving Tool** area. For further details, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

    When you have finished, click on the arrow to hide the machining parameters.

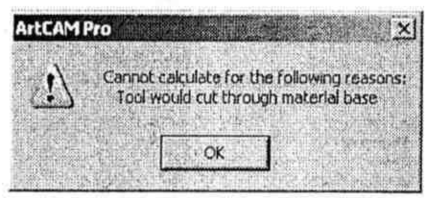
11. You can now calculate the maximum height and width of the carving pass in the toolpath by clicking on the **Centreline** button.

Note: If you have chosen a Carving tool with a smaller diameter than the maximum width of the carving pass, ArtCAM Pro makes multiple passes to clear the specified width of the carving pass.

During the centreline calculation process, ArtCAM Pro displays a progress bar and a **Cancel** button beneath the design window area. If you want to stop this process, click on the **Cancel** button.

After the centreline calculation process is complete, the maximum depth and width of the carving pass are shown below the **Carving Tool** area.

If the maximum depth of the carving pass is greater than the thickness of the material block, the following message box appears when the toolpath is calculated:



Click on the **OK** button to close the message box. To overcome this problem, you can select a different Carving tool with a smaller angled edge, limit the maximum depth of the tool or use a thicker block of material.

12. To limit the depth at which the Carving tool cuts into the block of material:
    - First, click to select the **Limit tool maximum depth** option in the **Carving Tool** area.
    - Next, type a value in the **Maximum Depth** box equal to the **Max Depth** value shown below the **Carving Tool** area.
13. Click on the **Select** button in the **Profiling Tool** area of the page to open the **Tool Groups Database** again.
14. Double-click on the tool you want to use to display its description in the **Profiling Tool** area.

15. To set the cut direction of the **Profiling Tool**, click on the arrow in the **Cut Direction** area, and then on either of the **Cut Direction** radio buttons:

    - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
    - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

    For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

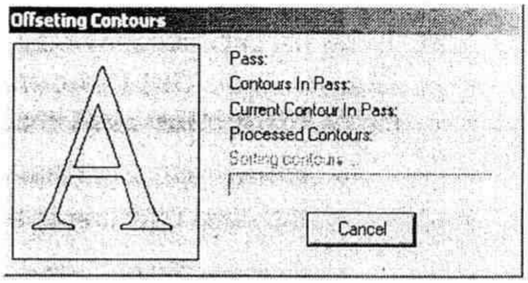
16. Click on the **Setup** button in the **Material** area of the page to display the **Material Setup** dialog box.
17. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

18. Type a name for the toolpath in the **Name** box.
19. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button. If you have not already calculated the carving pass in the toolpath, the selected vector object is colour-filled in cyan during the centreline calculation process.

      The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates each of the profile passes in the toolpath:



A dark red line represents the machining passes used to profile the selected vector object in the model. Each profile pass is drawn outside of the vector object's boundary. The toolpath preview is not shown if the **Create 2D Preview** option is deselected .

A circle marking the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

20. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to a profile pass in the Bevelled Carving toolpath, see "Adding Bridging" on page 450.

If you want to set the machining order of the profile passes in the Bevelled Carving toolpath, see "Setting the Machining Order" on page 458.

Help with creating a Bevel Carving

You can display information on how to create a Bevelled Carving toolpath if you click on Show Help at the top of the **Bevel Carving** page.

However, if you would like more information on creating a Bevelled Carving toolpath:

1. Click on the **More Help** button at the bottom of the **Bevel Carving** page to display the **Bevelled Carving Help** window.
2. Click on one of the **Step 1** radio buttons to display its corresponding **Step 2** in the window:
   - **I want to cut the bevelled carving out of the material** – Select this option if you want to remove the bevelled carving from the block of material.
   - **The bevelled carving sits within a bigger job** – Select this option if you do not want to remove the bevelled carving from the block of material.
3. Click on the **Close Window** button to close the **Bevelled Carving Help** window.

Smart Engraving

The **Engraving** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that engraves the vector text or object that you have selected.

Using the **Smart Engraving** page, you can:

- Select multiple tools to perform *smart engraving*, where ArtCAM Pro automatically selects the biggest tool first, and then the smaller tools in order of size. This provides the most efficient way of machining, as the bigger tools clear the material and the smaller ones can then machine smaller pockets to provide the engraved detail.
- Select a different tool clearance strategy for each of the tools you want to use. Depending on what strategy you select, you can control the angle, or where and in what direction the tool cuts into the block of material.
- Control the areas that the Engraving tool sharpens and/or profiles.

To engrave a vector object:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.

2. Select the vector object that you want to engrave. For details, see "Selecting Vectors" in the Working with Vectors chapter.

3. Click on the **Engraving** button  in the **2D Toolpaths** area to display the **Smart Engraving** page.

4. Type the depth (Z) from the material surface at which you want to begin engraving in the **Start Depth** box.

5. Type a value in the **Tolerance** box to specify how closely you want the Roughing and Engraving tools to follow the shape of the vector object.

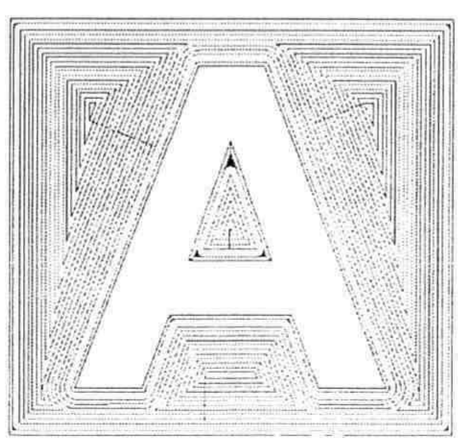
6. To place the Engraving tool in contact with the top edge of the vector object during the machining process, make sure that the **Vectors are on surface** option is selected.

   If the **Vectors are on surface** option is deselected, the Engraving tool is placed in contact with the bottom edge of the vector object during the machining process. The **Do Corner Sharpening** option is also automatically greyed out.

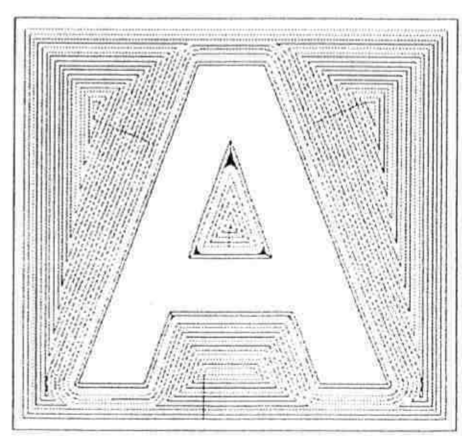
7. If you are engraving one vector object inside of another, and do not want the Engraving tool to sharpen the corners or machine the boundary of the outer vector object, click to select the **Outer vectors are boundary** option.

   In the example below, you can see that the engraving tool machines along the boundary and sharpens the corners of the outer rectangular vector object when the **Outer vectors are boundary** option is deselected:

Outer Vectors are boundary OFF...



Outer Vectors are boundary ON...



8. To automatically offset an End Mill Roughing tool in order to leave sufficient material for an angle-edged of an Engraving tool to form the side of the engraved feature, click to select the **Offset end mills for engraving tools** option.

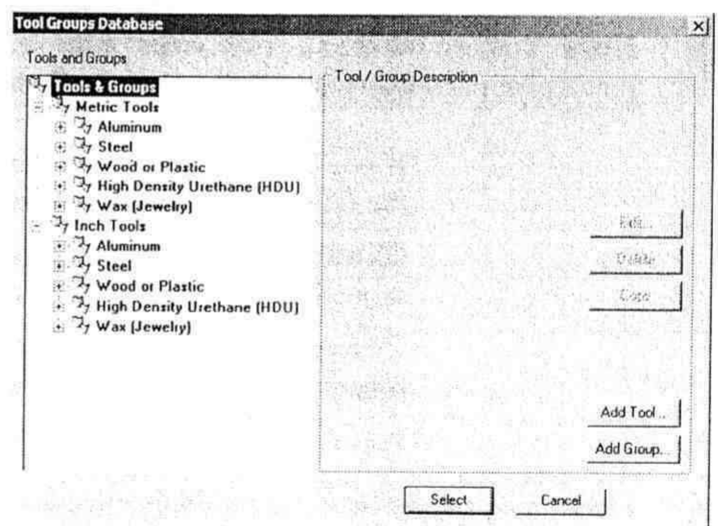
9. If you want to change the height at which the Roughing and Engraving tools make rapid moves between toolpath segments:

   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:

   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

10. Click on the **Add** button in the **Tools List** area to display the **Tool Groups Database**:



11. Double-click on the Roughing or Engraving tool you want to use. ArtCAM Pro closes the **Tool Groups Database**, highlights the selected tool in the **Tools List** window and displays its details below.

If you want to amend the machining parameters of the selected tool, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

When you have finished, click on the ▲ arrow next to the tool's description to hide its machining parameters.

12. If you want to leave extra material around the vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the Roughing or Engraving tool.

13. Select how you want the tool to clear the area using the **Tool Clearance Strategy** radio buttons ◉:

    - **Raster** – This strategy machines back and forth along the X-axis at a specified angle. If you select this strategy, type the angle you want the tool to move at in the **Raster Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- **Offset** – This strategy machines in repeated passes, each time moving inwards by the **Stepover** value of the tool you use.

Note: You can see the stepover value of the selected tool when the machining parameters are displayed in the **Tools List** area.

If you have selected this strategy, click on either of the **Cut Direction** radio buttons ◉:

Climb – In Climb Milling, the cutter rotates in the same direction as the feed motion.

Conventional – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

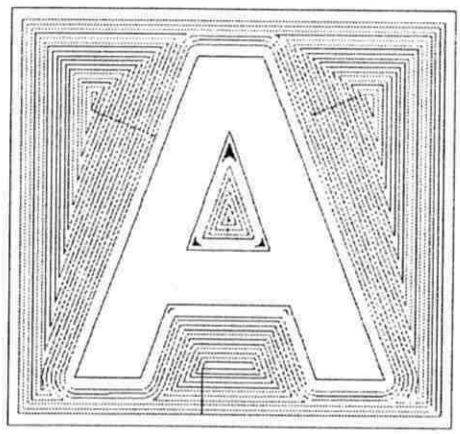
14. You can also control the movements of the selected Roughing and/or Engraving tools:

- If you want the Engraving tool to sharpen the corners of the machined vector object, click to select the **Do Corner Sharpening** option ☑.

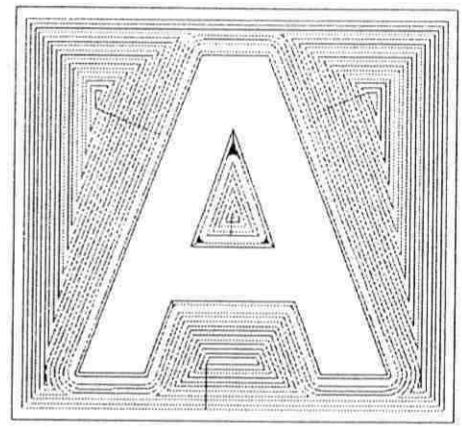
Note: If you have not selected an Engraving tool, the **Do Corner Sharpening** option is greyed out.

In the example below, you can see that the tool profiles the vector text *A* to sharpen its corners when the **Do Corner Sharpening** option is selected ☑:

Do Corner Sharpening OFF...



Do Corner Sharpening ON...



The Engraving tool is also lifted in the Z direction to minimise the curvature in the corners of the vector text *A*.

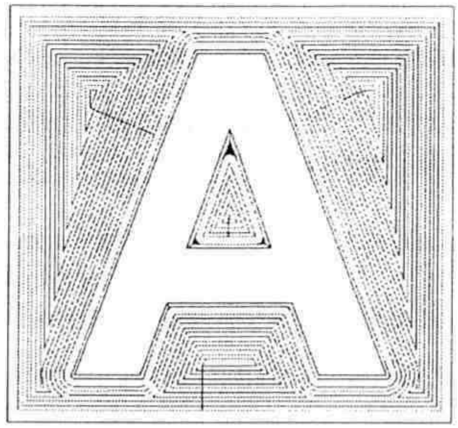
- If you want the Engraving tool to profile only the vector objects which have not already been machined with a larger tool, click to select the **Only Smart Engrave Profile** option ☑.

Note: If you have not selected an Engraving tool, the **Only Smart Engrave Profile** option is greyed out.

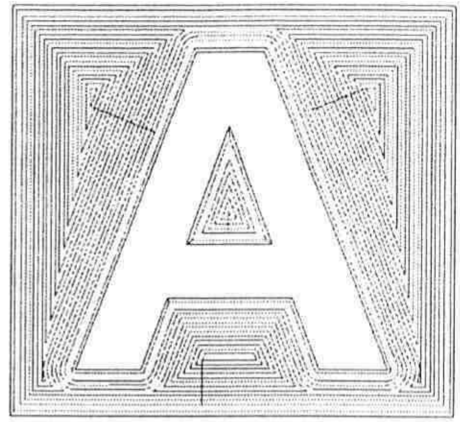
- If you want a tool to make a single profile pass around the vector object, click to select the **Profile Only** option ☑.

  In the example below, you can see that the number of machining passes is reduced, particularly in the central cavity of the vector text *A*, when the **Profile Only** option is selected ☑:

Profile Only OFF...



Profile Only ON...



- If you want a tool to cut to at a different depth than the other tools, click to select the **Independent Finish Depth** option ☑.

  Type the absolute Z zero value of the bottom of the area you want to engrave with the tool in the **Finish Depth** box below.

You can add as many tools as are needed for the job, setting the machining parameters and strategy for each tool after selecting it in the **Tools List** window.

15. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

16. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option ☐.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

17. Type a name for the toolpath in the **Name** box.

18. You can now specify when you want to calculate the toolpath:

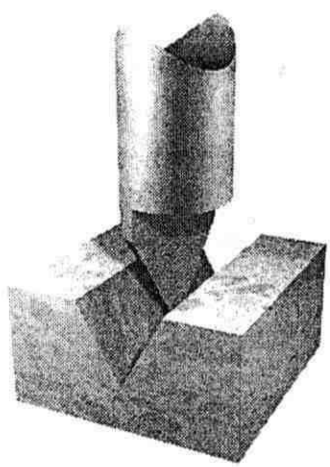
- If you want to calculate the toolpath now, click on the **Now** button.

  You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

19. Click on the **Close** button to return to the **Toolpaths** Home page.

Machine Vectors



The **Machine Along Vector** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to machine along the boundary of a vector object.

The boundary of a selected vector object represents the centreline of the Profiling tool. The machined feature has no diameter other than that of the tip of the Profiling tool.

This toolpath is mainly used to produce the effect of engraved text or to provide sharp profiling for detailed edges.

To machine a vector object:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the vector object that you want to machine. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **Machine Along Vector** button in the **2D Toolpaths** area to display the **Machine Vectors** page.
4. Type the depth (Z) from the material surface at which you want to begin machining in the **Start Depth** box.

5. Type a value in the **Tolerance** box to specify how closely you want the Profiling tool to follow the shape of the vector object.

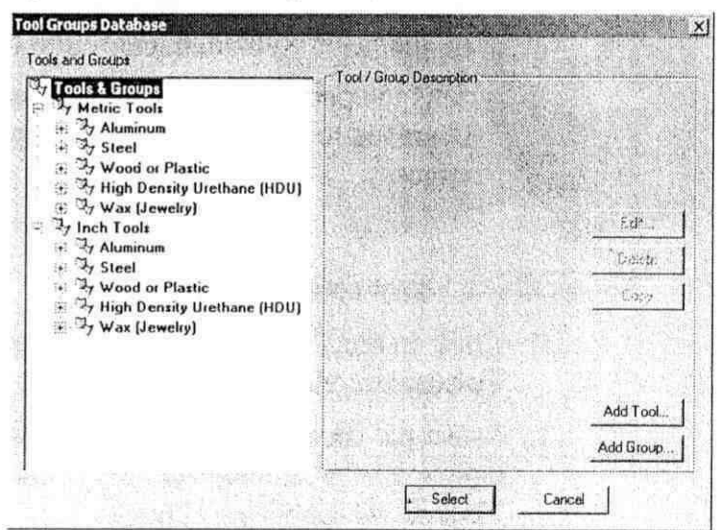
6. If you want to change the height at which the Profiling tool makes rapid moves between toolpath segments:

    - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:

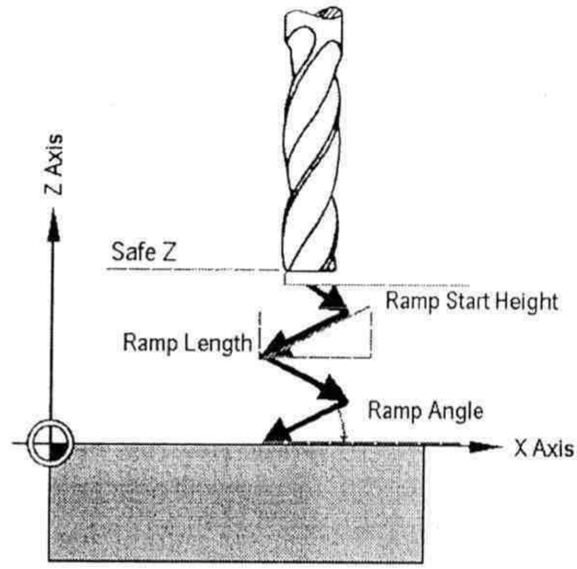
    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes

7. Click on the **Select** button in the **Profiling Tool** area to open the **Tool Groups Database**:



8. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database**, and displays the selected tool's description in the **Profiling Tool** area.

9. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option. The ramping moves settings are displayed:



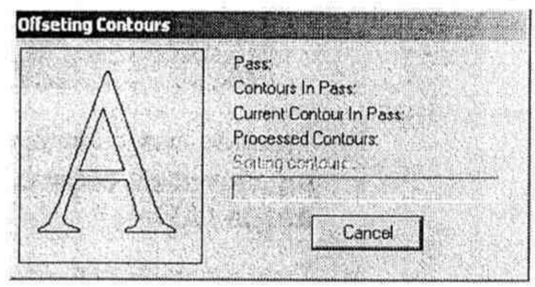
- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

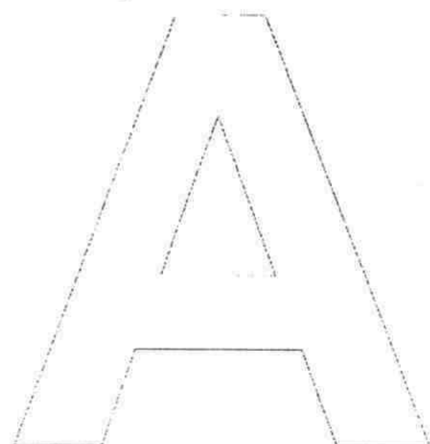
10. If you are machining a block of vector text, you can instruct the tool to machine according the order in which the text was created by clicking to select the **Preserve Text Order** option ☑. The cutting tool moves along the first line of text from left to right, until it locates a carriage return when it continues machining along the next line from right to left. This process is repeated until the last character in the last line of text is located.

11. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.
12. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option ☐.
13. Type a name for the toolpath in the **Name** box.
14. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button.

      The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The passes used to machine the vector object are represented by dark red lines drawn directly on the boundary of the selected vector object.

      A circle marking the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

      The toolpath preview is not shown if you deselected the **Create 2D Preview** option ☐.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

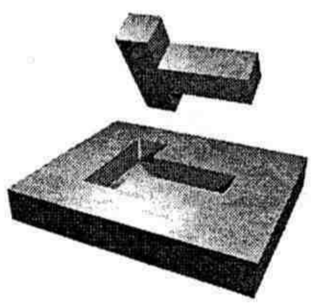
- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

15. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to a profile pass in the Machine Vectors toolpath, see "Adding Bridging" on page 450.

If you want to set the machining order of the profile passes in the Machine Vectors toolpath, see "Setting the Machining Order" on page 458.

Inlay Wizard



The **Inlay** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that can produce one of many different kinds of inlays and inserts. Essentially, you can use this strategy to machine interlocking male and female parts. ArtCAM Pro allows these interlocking parts to fit perfectly by adjusting the shape of the corners according to the tool that is used to machine them.

To produce a particular style of interlocking male and female parts:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the vector object from which you want to create an inlay or an insert. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **Inlay** button in the **2D Toolpaths** area to display the **Inlay Wizard** page.
4. Select the **Inlay Type** that you want to create:
    - **Pocket**. For details, see "Creating a Pocket" on page 372.

- **Hole**. For details, see "Creating a Hole" on page 379.
- **Stepped Pocket**. For details, see "Creating a Stepped Pocket" on page 383.
- **Stepped Hole**. For details, see "Creating a Stepped Hole" on page 390.
- **Straight Insert**. For details, see "Creating a Straight Insert" on page 394.
- **Stepped Insert**. For details, see "Creating a Stepped Insert" on page 399.

Creating a Pocket

This **Female Inlay** option displays the **Female Inlay Pocket** page in the **Assistant** window. This allows you to create a pocket, which is a hole in the material that has a bottom and does not go right through the block. You can then create a male straight insert to fit into the pocket.

Using the **Female Inlay Pocket** page, you can:

- Select both Roughing and Finishing tools.
- Select a tool clearance strategy for the Finishing or Roughing tool you want to use. Depending on the strategy you select, you can control the angle, where and in what direction the tool cuts into the block of material.
- Control how the Roughing and/or Finishing tools cut into and retract from the block of material by adding ramping moves.

To create a pocket:

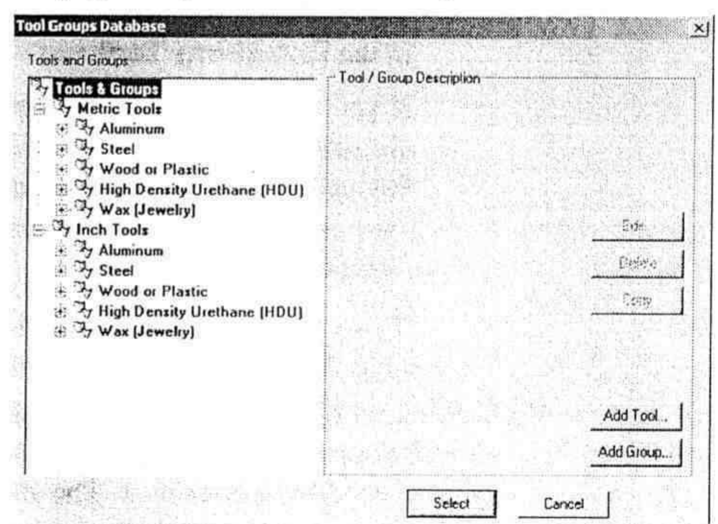
1. Click on the **Pocket** option  in the **Inlay Types** area of the **Inlay Wizard** page. ArtCAM Pro displays the **Female Inlay Pocket** page in the **Assistant** window.
2. Type the depth (Z) from the material surface at which you want to create the pocket in the **Start Depth** box.
3. Type the depth (Z) of the bottom of the pocket in the **Finish Depth** box.
4. If you want to enlarge or reduce the pocket around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between

the boundary of the selected vector object and the cutting tool(s). Type a positive value to enlarge the pocket or a negative value to reduce it.

Make sure that there is a sufficient allowance for the corresponding male straight insert to fit into the pocket.

5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool(s) to follow the shape of the selected vector object.
6. If you want to change the height at which a cutting tool makes rapid moves between toolpath segments:
   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:
   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
7. Click on the **Select** button in the **Finishing Tool** area of the page to open the **Tool Groups Database**:



8. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Finishing Tool** area.

Important: The selected **Finishing Tool** must be the same size as the tool used to machine the male straight insert to be fitted into this pocket.

If you want to amend the machining parameters for the selected tool, click on the arrow in the **Finishing Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

When you have finished, click on the arrow to hide the machining parameters.

9. Select whether you want to use a Roughing tool or not:

   - If you want to use the Finishing tool you have already selected to create the whole pocket, leave the **Use Roughing Tool** option deselected, and go straight to the next step.

   - If you want to use a Roughing tool to create the pocket, click to select the **Use Roughing Tool** option.

     In the **Roughing Tool** area, which is now displayed, click on the **Select** button to open the **Tool Groups Database** again. Double-click on the tool you want to use. Its description is displayed in the **Roughing Tool** area.

     If you want to amend the machining parameters for the selected tool, click on the arrow in the **Roughing Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

     When you have finished, click on the arrow to hide the machining parameters.

10. If you want the Finishing tool to enlarge or reduce the pocket around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the Finishing tool. Type a positive value to enlarge the pocket or a negative value to reduce it.

Make sure that there is a sufficient allowance for the corresponding male straight insert to fit into the pocket.

11. Select how you want the tool to clear the pocket using the **Strategy** radio buttons :
    - **Raster** - This strategy machines in passes back and forth along the X-axis at a specified angle. If you select this strategy, first type the angle you want the tool to move at in the **Raster Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on one of the **Profile Pass** radio buttons :

None – Select this option if you do not want the tool to profile the selected vector object.

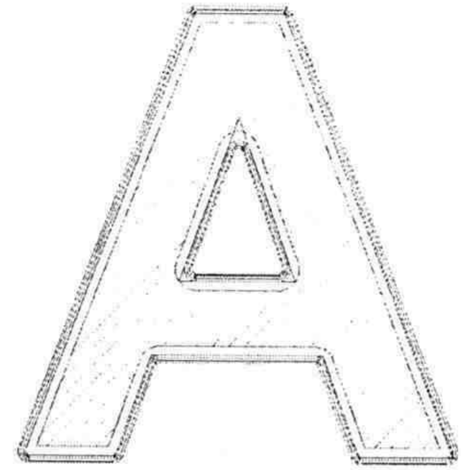
First – Select this option if you want the tool to profile the selected vector object first and then raster clear the area.

Last – Select this option if you want the tool to move outwards to raster clear the area, then profile the selected vector object.

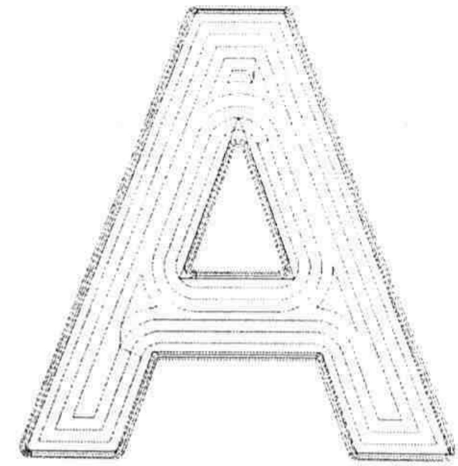
- **Offset** - This strategy machines in repeated passes, each time moving inwards by the **Stepover** value of the tool you use.

Note: You can see the stepover value of the selected tool when the machining parameters are displayed in the **Roughing Tool** and/or **Finishing Tool** areas.

Raster Strategy at 45°...



Offset Strategy at Climb...



If you select **Offset**, first click on either of the **Cut Direction** radio buttons :

Climb Mill – In Climb Milling, the cutter rotates in the same direction as the feed motion.

Conv. – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on either of the **Start Point** radio buttons :

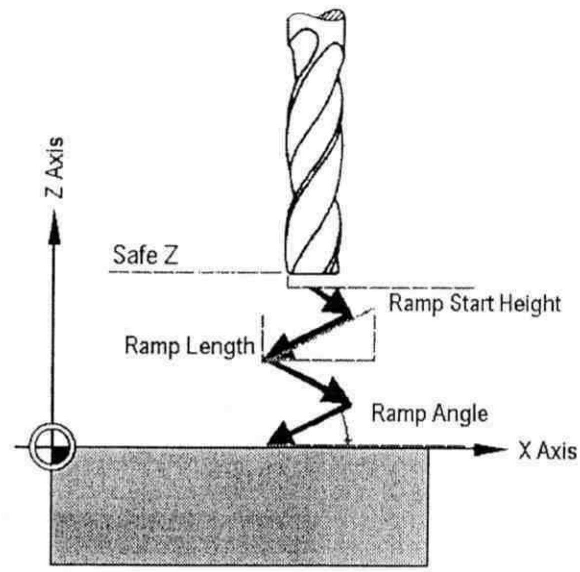
Outside - Select this option if you want the tool to cut into the material at the vector object's boundary, then machine inwards.

Inside - Select this option if you want the tool to cut into the material at the vector object's centre, then machine outwards.

Note: You can set the default offset start point using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

12. When you have set a tool clearance strategy, you can choose whether or not to add ramping moves to the machining passes made by the cutting tools:

- If you want to add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

13. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

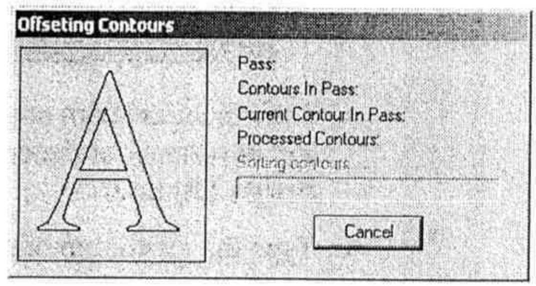
If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

14. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option ☐.

Note: You can set the default offset start point using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

---

15. Type a name for the toolpath in the **Name** box.
16. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The machining passes used to machine the pocket are represented by dark red lines drawn along the boundary and inside of the selected vector object, according to the **Strategy** selected.

      The toolpath preview is not shown if the **Create 2D Preview** option is deselected ☐.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.
    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.
17. Click on the **Close** button to return to the **Toolpaths** Home page.

Creating a Hole

This **Female Inlay** option displays the **Female Inlay** page in the **Assistant** window. This allows you to create a hole in the block of material. You can then create a male straight insert to fit into the hole.

Using the **Female Inlay** page, you can:

- Select a tool to cut the hole in the block of material.
- Control the cutting direction of the tool.

To create a hole:

1. Click on the **Hole** option in the **Inlay Types** area of the **Inlay Wizard** page to display the **Female Inlay** page in the **Assistant** window.
2. Type the depth (Z) from the material surface at which you want to create the hole in the **Start Depth** box.
3. Type the depth (Z) for the bottom of the hole in the **Finish Depth** box.
4. If you want to enlarge or reduce the hole around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the cutting tool. Type a positive value to enlarge the hole or a negative value to reduce it.

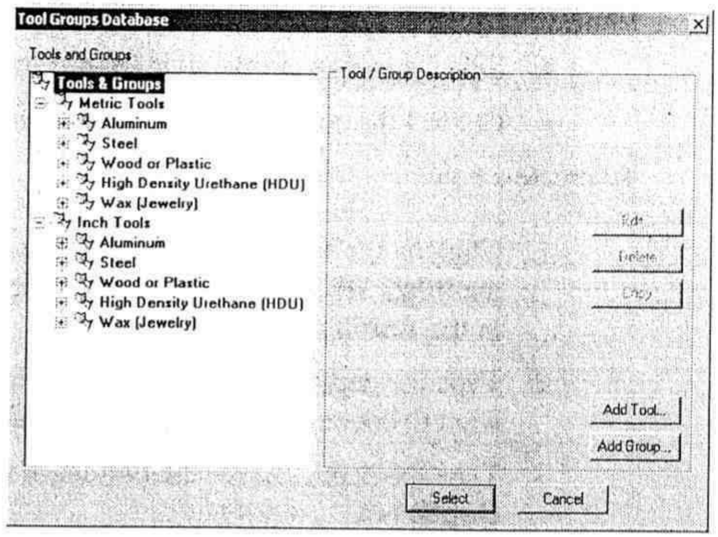
   Make sure that there is a sufficient allowance for the corresponding male straight insert to fit into the hole.
5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the vector object.
6. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:
   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:

   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.

- Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

7. Click on the **Select** button in the **Tool** area of the page to display the **Tool Groups Database**:



8. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.

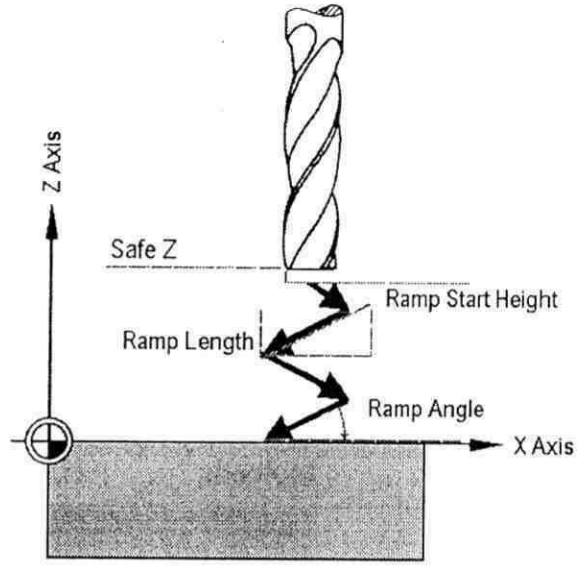
9. To set the cut direction, click on the arrow in the **Cut Direction** area of the page, then click on either of the **Cut Direction** radio buttons :

   - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
   - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

   For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

10. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option . The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

11. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

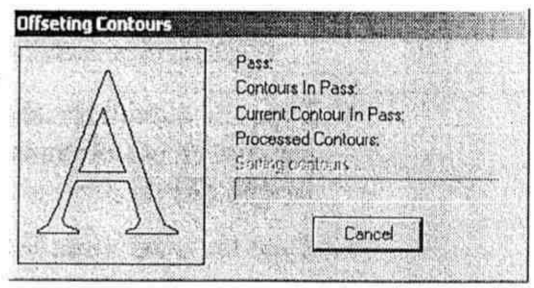
    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

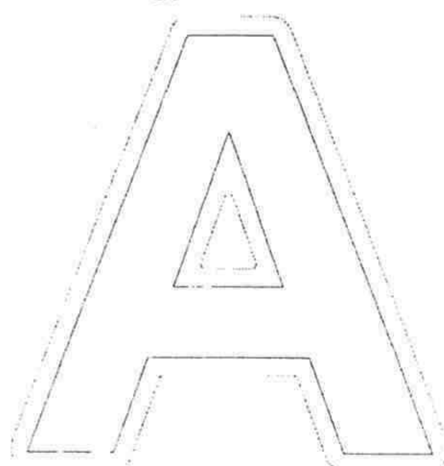
12. If you do not want to preview the toolpath in the **2D View** window, click to select the **Create 2D Preview** option.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

13. Type a name for the toolpath in the **Name** box.
14. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:



The machining passes used to machine the hole are represented by dark red lines drawn around the boundary of the selected vector object, according to the **Strategy** selected:



A circle marking the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

The toolpath preview is not shown if the **Create 2D Preview** option is deselected .

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

15. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to a profile pass in the Female Inlay toolpath, see "Adding Bridging" on page 450.

If you want to set the machining order of the profile passes in the Female Inlay toolpath, see "Setting the Machining Order" on page 458.

Creating a Stepped Pocket

This **Female Inlay** option displays the **Female Stepped Pocket Inlay** page in the **Assistant** window. This allows you to create a stepped pocket, which is a stepped hole in the material that has a bottom and does not go right through the block. You can then create a male straight insert to fit into the stepped pocket.

Using the **Female Stepped Pocket Inlay** page, you can:

- Select both Roughing and Finishing tools.
- Select a tool clearance strategy for the Finishing or Roughing tool you want to use. Depending on the strategy you select, you can control the angle, where and in what direction the tool cuts into the block of material.
- Control how the Roughing and/or Finishing tools cut into and retract from the block of material by adding ramping moves.

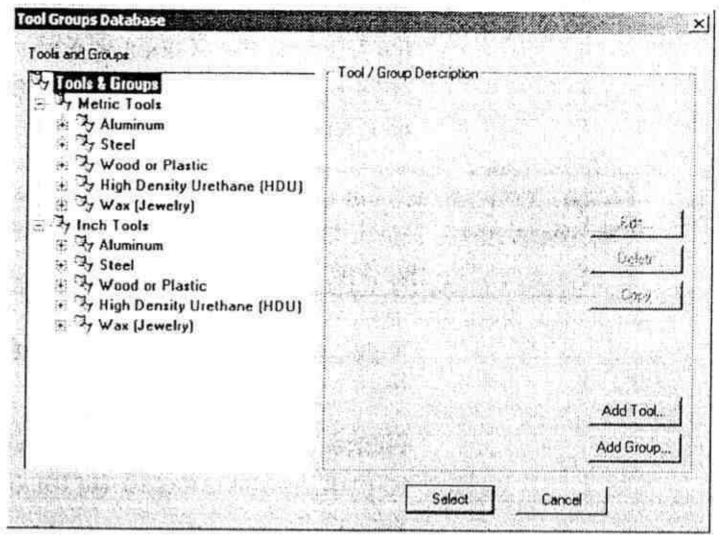
To create a stepped pocket:

1. Click on the **Stepped Pocket** option in the **Inlay Types** area of the **Inlay Wizard** page to display the **Female Stepped Pocket Inlay** page in the **Assistant** window.

2. Type the depth (Z) from the material surface at which you want to create the stepped pocket in the **Start Depth** box.
3. Type the depth (Z) of the bottom of the stepped pocket in the **Finish Depth** box.
4. If you want to enlarge or reduce the stepped pocket around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the cutting tool(s). Type a positive value to enlarge the pocket or a negative value to reduce it.

   Make sure that there is a sufficient allowance for the corresponding male stepped insert to fit into the stepped pocket.
5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool(s) to follow the shape of the selected vector object.
6. If you want to change the height at which the cutting tools used make rapid moves between toolpath segments:
   - First, click on the ▼ arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:
   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X, Y** and **Z** boxes.
   - Next, click on the ▲ arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
7. Type the relative depth of the shoulder from the **Start Depth** in the **Depth (d)** box, in the **Shoulder Dimensions** area of the page.
8. Type the width of the shoulder in the **Width (s)** box.
9. Click on the **Select** button in the **Finishing Tool** area of the page to open the **Tool Groups Database**:



10. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Finishing Tool** area.

Note: The selected **Finishing Tool** must be the same size as the tool used to machine the male stepped insert which you want to fit into this stepped pocket.

If you want to amend the machining parameters for the selected tool, click on the arrow in the **Finishing Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

When you have finished, click on the arrow to hide the machining parameters.

11. Select whether you want to use a Roughing tool or not:
12. If you want the Finishing tool to enlarge or reduce the pocket around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the Finishing tool. Type a positive value to enlarge the stepped pocket or a negative value to reduce it.

    Make sure that there is a sufficient allowance for the corresponding male straight insert to fit into the stepped pocket.
13. Select how you want the tool to clear the pocket using the **Strategy** radio buttons :

- **Raster** - This strategy machines in passes back and forth along the X-axis at a specified angle. If you select this strategy, first type the angle you want the tool to move at in the **Raster Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on one of the **Profile Pass** radio buttons :

None – Select this option if you do not want the tool to profile the selected vector object.

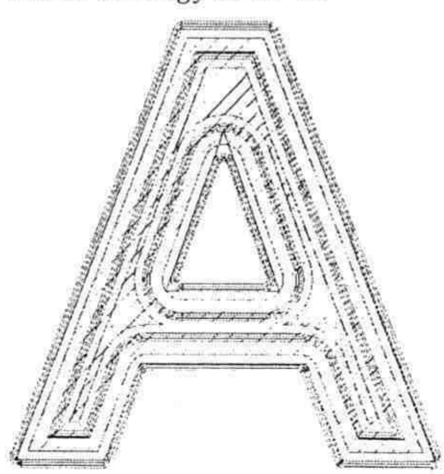
First – Select this option if you want the tool to profile the selected vector object first and then raster clear the area.

Last – Select this option if you want the tool to move outwards to raster clear the area, then profile the selected vector object.

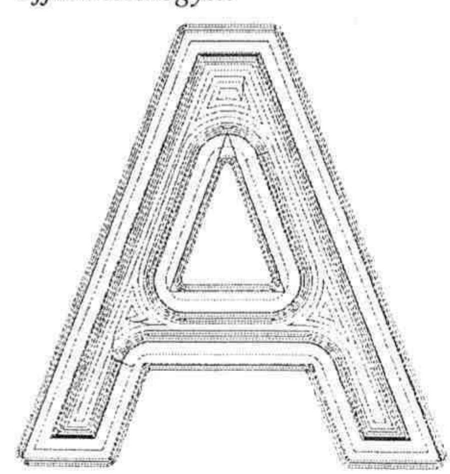
- **Offset** - This strategy machines in repeated passes, each time moving inwards by the **Stepover** value of the tool you use.

Note: You can see the stepover value of the selected tool when the machining parameters are displayed in the **Roughing Tool** and/or **Finishing Tool** areas.

Raster Strategy at 45°...



Offset Strategy...



If you select **Offset**, first click on either of the **Cut Direction** radio buttons :

Climb Mill – In Climb Milling, the cutter rotates in the same direction as the feed motion.

Conv. – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on either of the **Start Point** radio buttons :

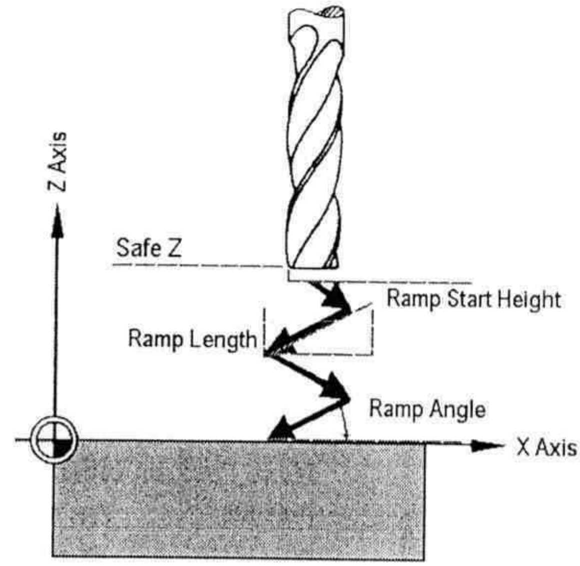
Outside - Select this option if you want the tool to cut into the material at the vector object's boundary, then machine inwards.

Inside - Select this option if you want the tool to cut into the material at the vector object's centre, then machine outwards.

Note: You can set the default offset start point using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

14. When you have set a tool clearance strategy, you can choose whether or not to add ramping moves to the machining passes made by the cutting tools:

    - If you want to add ramping moves to the toolpath, click to select the **Add Ramping Moves** option . The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

15. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

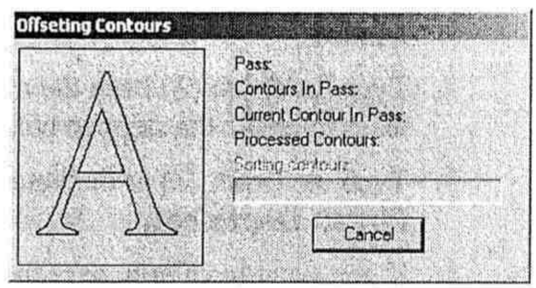
    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

16. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option .

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

17. Type a name for the toolpath in the **Name** box.
18. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The machining passes used to machine the stepped pocket are represented by dark red lines drawn along the boundary and inside of the selected vector object, according to the **Strategy** selected.

      The toolpath preview is not shown if the **Create 2D Preview** option is deselected .

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.
    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.
19. Click on the **Close** button to return to the **Toolpaths** Home page.

Creating a Stepped Hole

This **Female Inlay** option displays the **Female Stepped Hole Inlay** page in the **Assistant** window. This allows you to create a stepped hole in the block of material. You can then create a male stepped insert to fit into the stepped hole.

Using the **Female Stepped Hole Inlay** page, you can:

- Control the depth and width of the hole's shoulder.
- Select a tool to cut the hole in the block of material.
- Control the cutting direction of the tool.

To create a stepped hole:

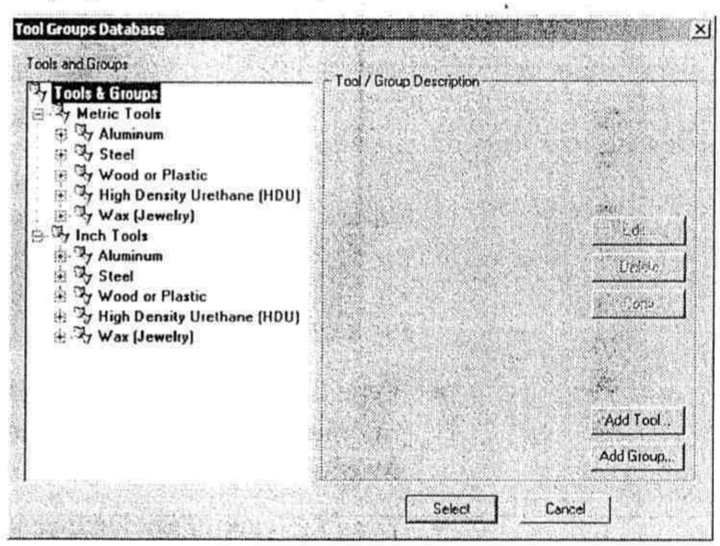
1. Click on the **Stepped Hole** option in the **Inlay Types** area of the **Inlay Wizard** page to display the **Female Stepped Hole Inlay** page in the **Assistant** window.
2. Type the depth (Z) from the material surface at which you want to create the stepped hole in the **Start Depth** box.
3. Type the depth (Z) of the bottom of the stepped hole in the **Finish Depth** box.
4. If you want to enlarge or reduce the stepped hole around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the cutting tool. Type a positive value to enlarge the stepped hole or a negative value to reduce it.

   Make sure that there is a sufficient allowance for the corresponding male stepped insert to fit into the stepped hole.
5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the vector object.
6. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:
   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

If you want to define the **Home Position** for the machining tool:

- First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
- Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

7. Type the relative depth of the shoulder from the **Start Depth** in the **Depth (d)** box, in the **Shoulder Dimensions** area of the page.
8. Type the width of the shoulder in the **Width (s)** box.
9. Click on the **Select** button in the **Tool** area of the page to display the **Tool Groups Database**:

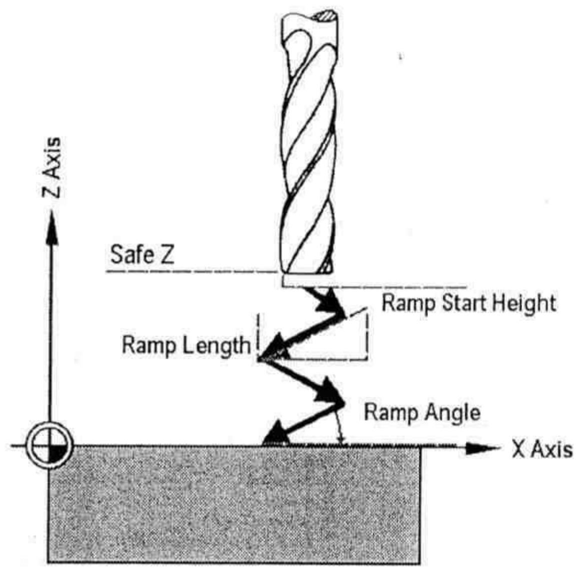


10. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.
11. To set the cut direction, click on the arrow in the **Cut Direction** area of the page, and then click on either of the **Cut Direction** radio buttons :
    - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
    - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

12. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

13. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, and then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

14. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option .

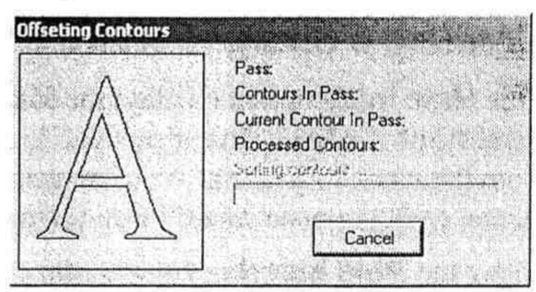
Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

---

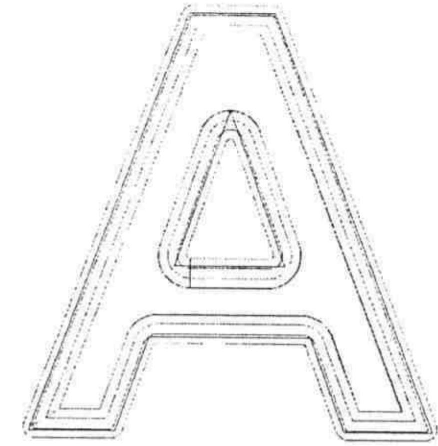
15. Type a name for the toolpath in the **Name** box.

16. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The machining passes used to machine the stepped hole are represented by dark red lines drawn around the boundary of the selected vector object:



The toolpath preview is not shown if the **Create 2D Preview** option is deselected ☐.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

17. Click on the **Close** button to return to the **Toolpaths** Home page.

Creating a Straight Insert

This **Male Inlay** option displays the **Male Insert** page in the **Assistant** window. This allows you to create a male straight insert from the block of material. You can then create a female hole or pocket to fit the male straight insert into.

Using the **Male Insert** page, you can:

- Select a tool to cut the straight insert out of the block.
- Control the cutting direction of the tool.

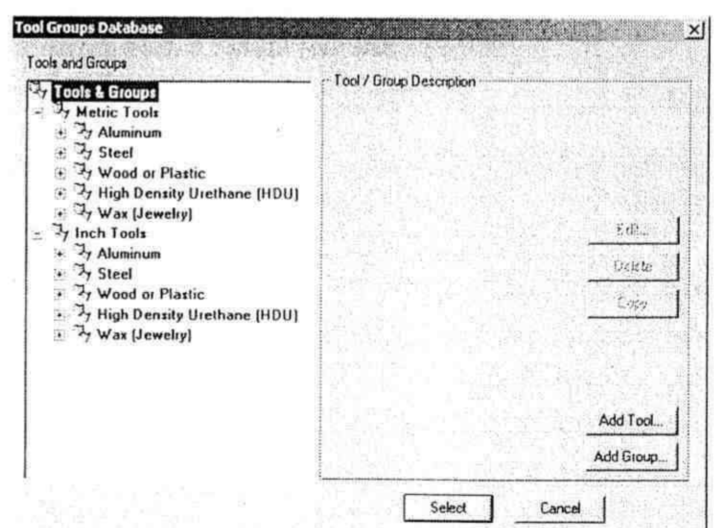
To create a straight insert:

1. Click on the **Straight** option in the **Inlay Types** area of the **Inlay Wizard** page to display the **Male Insert** page in the **Assistant** window.
2. Type the depth (Z) from the material surface at which you want to create the straight insert in the **Start Depth** box.

3. Type the depth (Z) of the bottom of the straight insert in the **Finish Depth** box.
4. If you want to enlarge or reduce the straight insert around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the cutting tool. Type a positive value to enlarge the straight insert or a negative value to reduce it.
5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the vector object.
6. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:
   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:

   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
7. Click on the **Select** button in the **Tool** area of the page to display the **Tool Groups Database**:



8. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.

Note: The selected **Tool** must be the same size as that used to machine the female hole or pocket which you want to fit this straight insert into.

   If you want to amend the machining parameters for the selected tool, click on the ▼ arrow in the **Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

   When you have finished, click on the ▲ arrow to hide the machining parameters.

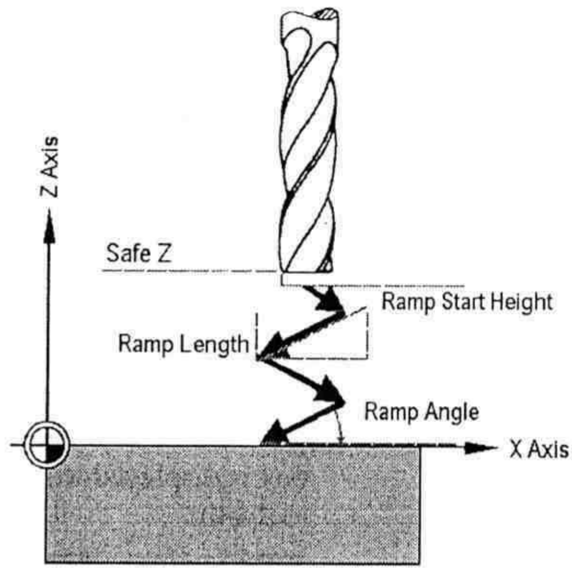
9. To set the cut direction, click on the ▼ arrow in the **Cut Direction** area of the page, and then click on either of the **Cut Direction** radio buttons:

   - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
   - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

   For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

10. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

11. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

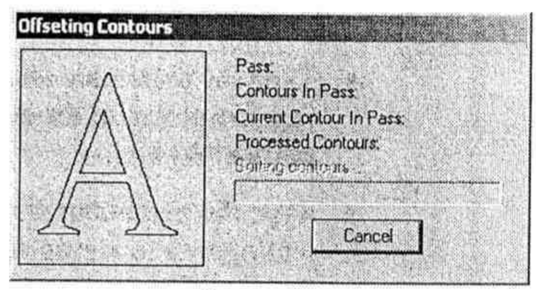
12. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

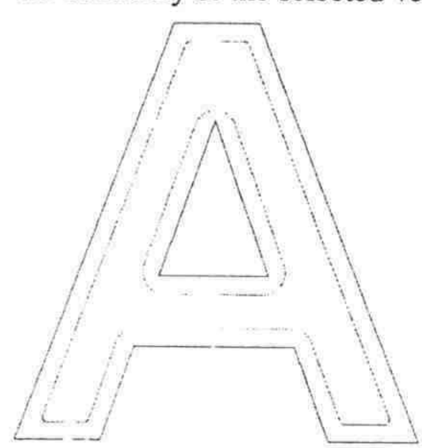
13. Type a name for the toolpath in the **Name** box.

14. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      The machining passes used to machine the straight insert are represented by dark red lines drawn around the boundary of the selected vector object:

      A circle marking the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

      The toolpath preview is not shown if the **Create 2D Preview** option is deselected.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

15. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to a profile pass in the Male Insert toolpath, see "Adding Bridging" on page 450.

If you want to set the machining order of the profile passes in the Male Insert toolpath, see "Setting the Machining Order" on page 458.

Creating a Stepped Insert

This **Male Inlay** option displays the **Male Stepped Insert** page in the **Assistant** window. This allows you to create a male stepped insert from the block of material. You can then create a female stepped hole or stepped pocket to fit the male stepped insert into.

Using the **Male Stepped Insert** page, you can:

To create a stepped insert:

1. Click on the **Stepped** option  in the **Inlay Types** area of the **Inlay Wizard** page to display the **Male Stepped Insert** page in the **Assistant** window.
2. Type the depth (Z) from the material surface at which you want to create the stepped insert in the **Start Depth** box.
3. Type the depth (Z) of the bottom of the stepped insert in the **Finish Depth** box.
4. If you want to enlarge or reduce the stepped insert around the selected vector object, you can type a value in the **Allowance** box. The allowance sets the distance between the boundary of the selected vector object and the cutting tool. Type a positive value to enlarge the stepped insert or a negative value to reduce it.

   Make sure that there is a sufficient allowance for the corresponding female stepped hole or stepped pocket.

5. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the vector object.

6. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:

    - First, click on the ▼ arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

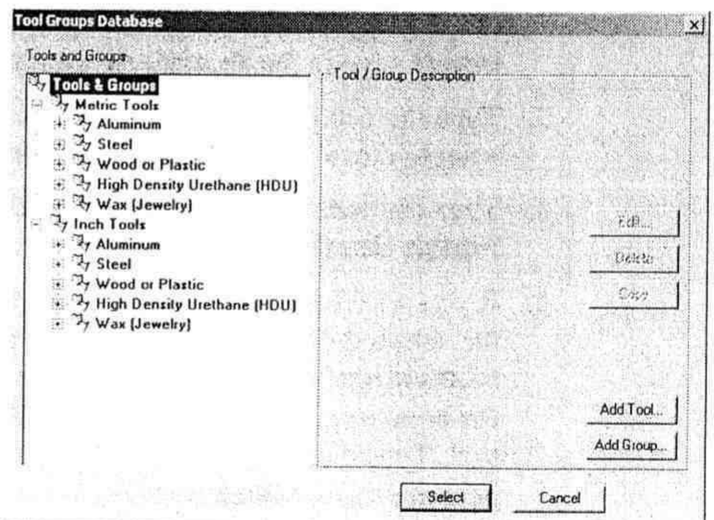
    If you want to define the **Home Position** for the machining tool:

    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, click on the ▲ arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

7. Type the relative depth of the shoulder from the **Start Depth** in the **Depth (d)** box, in the **Shoulder Dimensions** area of the page.

8. Type the width of the shoulder in the **Width (s)** box.

9. Click on the **Select** button in the **Tool** area of the page to display the **Tool Groups Database**:



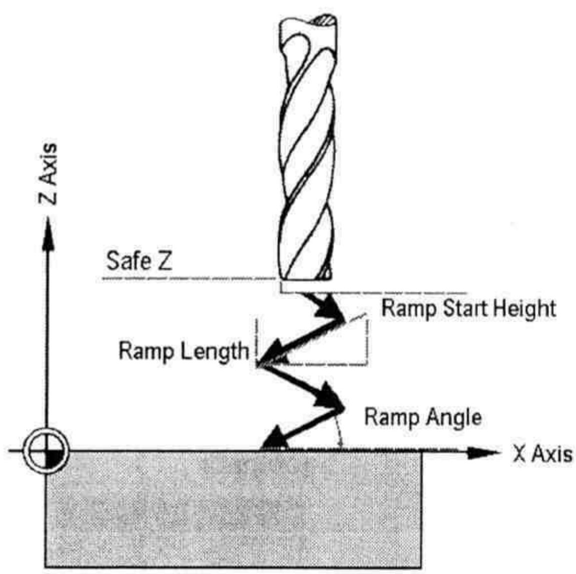
10. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.

11. If you want to amend the machining parameters for the selected tool, click on the arrow in the **Tool** area.
12. To set the cut direction, click on the arrow in the **Cut Direction** area, and then click on either of the **Cut Direction** radio buttons:
    - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
    - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

13. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the material surface in the **Max Ramp Length (L)** box.

- Type the minimum distance that you want the tool to zigzag across the material surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

14. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

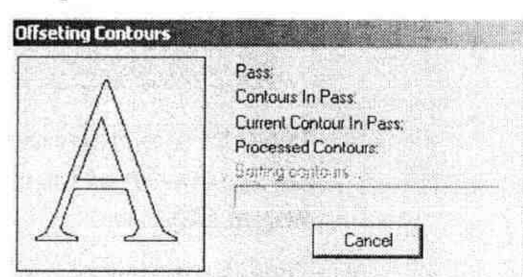
15. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option ☐.

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

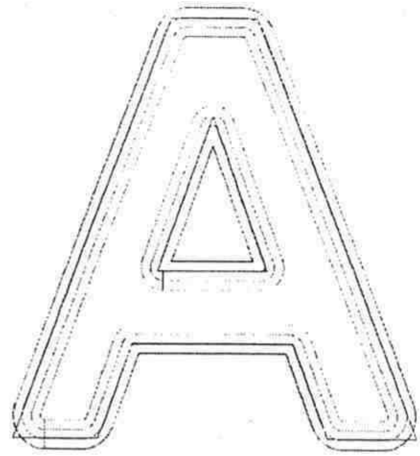
---

16. Type a name for the toolpath in the **Name** box.
17. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:



The machining passes used to machine the stepped insert are represented by dark red lines drawn around the boundary of the selected vector object:



The toolpath preview is not shown if the **Create 2D Preview** option is deselected.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

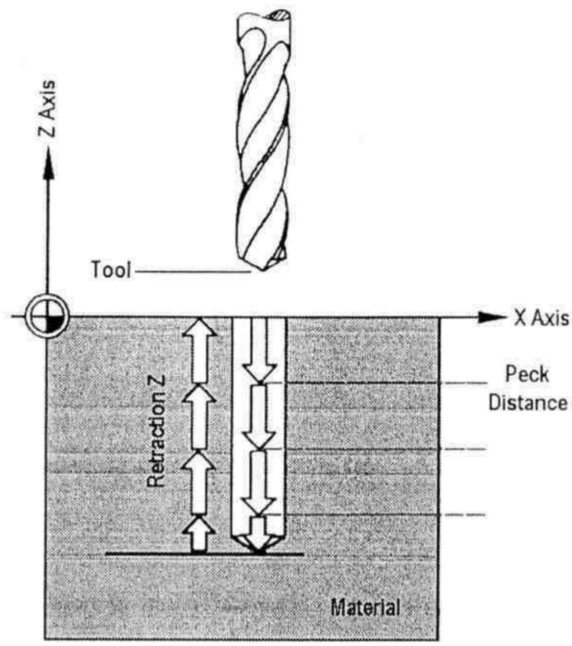
18. Click on the **Close** button to return to the **Toolpaths** Home page.

Drill Holes

The **Drilling** button in the **2D Toolpaths** area of the **Toolpaths** Home page allows you to create drill holes.

The size of the drill hole is determined by the diameter of the cutting tool that is used to machine it.

When drilling holes you can use peck drilling to reduce chip packing in the hole. The tool drills a short distance into the block of material, then withdraws. The deeper the hole, the more frequent the drill must be retracted (or pecked) to be effective:



The selected tool is retracted from the block of material to the **Retraction Z** level when it reaches the level displayed in the **Stepdown** box in the **Tool** area of the page. The **Stepdown** is also referred to as the Peck Distance. The tool then continues to cut deeper into the block of material until the **Finish Depth** is reached or the **Stepdown** is reached again. In the latter instance, the tool retracts from the block of material to the **Retraction Z** level again. This cycle continues until the **Finish Depth** is finally reached.

To create drill holes in your model:

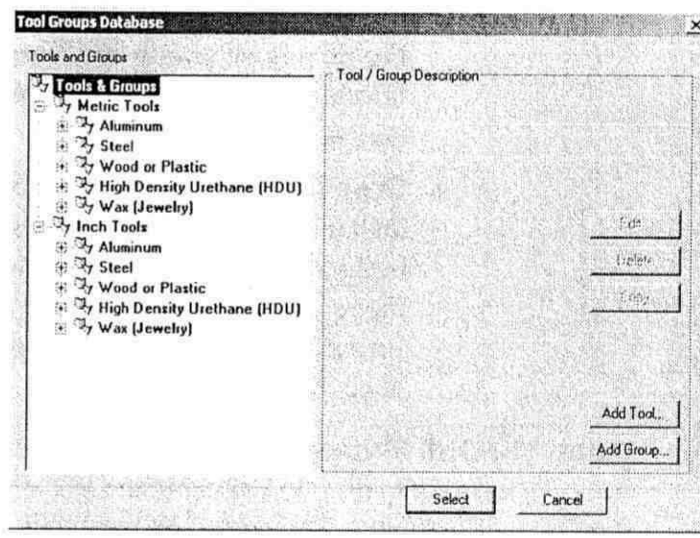
1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the vector objects through which you want to create drill holes.. For details, see "Selecting Vectors" in the Working with Vectors chapter.
3. Click on the **Drilling** button in the **2D Toolpaths** area to display the **Drilling** page.
4. Type the depth (Z) from the material surface at which you want to create the drill holes in the **Start Depth** box.
5. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:

- First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
- Next, define the height in the **Safe Z** box.

If you want to define the **Home Position** for the machining tool:

- First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
- Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

6. Click on the **Select** button in the **Tool** area of the page to open the **Tool Groups Database**:



7. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.

   If you want to amend the machining parameters for the selected tool, click on the arrow in the **Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

   When you have finished, click on the arrow to hide the machining parameters.

Note: The diameter of the selected tool is equal to the diameter of the drill holes.

8. To set where the holes are drilled in your model, click on one of the **Drill Centre of...** radio buttons :

   - **Circular Vectors** – Select this option if you have selected circles to define the position of the drill holes.
   - **All Vectors** – Select this option if you have selected any type of vector object, open or closed. The holes are drilled at the centre of the bounding box that surrounds the selected vector objects, irrespective of their shape.
   - **All Vector Nodes** – Select this option to drill holes at each point (node) in the selected vector object. For example, holes would be drilled in a square at each corner, assuming that there are no intermediate points on any of its four sides.

     For details on points, see "Inserting a Point" and "Editing Vector Nodes" in the Working with Vectors chapter.
   - **Drill Toolpath Plunges** – Select this option to drill a hole whenever the tools used in the selected toolpath preview make a plunge move. A plunge move is shown in cyan in the **3D View** window. For further details, see "Rapid and Plunge Moves" on page 489.

Note: The **Drill Toolpath Plunges** option is greyed-out if you have not selected a toolpath preview from the **2D View** window. For details, see "Selecting Toolpaths" on page 438.

Type a value in the **Offset Z** box to set the distance above or below the base of the block you want to drill to. Type a positive value if you want to set the bottom of the hole above the base of the block, and a negative value if you want to cut through it.

A preview image appears in the **2D View** window showing the position of the drill holes according to the **Drill Centre of...** option selected.

9. To drill the holes using the peck drilling method:

   - First, click to select the **Peck Drilling** option and display the **Retraction Z** box.

- Next, type the retraction level in the **Retraction Z** box.

10. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, and then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

11. If you do not want to preview the toolpath in the **2D View** window, click to deselect the **Create 2D Preview** option .

Tip: It is recommended that you create a 2D preview. You can always hide it from view by clicking on the **2D** option in the **Show in** area on the **Toolpaths** Home page.

---

12. Type a name for the toolpath in the **Name** box.

13. You can now specify when you want to calculate the toolpath:

    If you want to calculate the toolpath now, click on the **Now** button.

    The machining passes used to machine the drill holes are represented by dark red circles drawn according to the selected **Drill Centre of...** option.

    The toolpath preview is not shown if the **Create 2D Preview** option is deselected .

    You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

14. Click on the **Close** button to return to the **Toolpaths** Home page.

3D Toolpath Strategies

ArtCAM Pro provides a number of three-dimensional toolpath strategies that you can use to machine your model, based on the vector objects and/or relief created within it. These include:

- **Machining a Relief**. This strategy allows you to machine any relief that you have created in a model. For details, see "Machine Relief" on page 408.
- **Feature Machining**. This strategy is used to machine the **Raised**, **Recessed** and **Centreline Engraved** features you have created from vector objects in a model. For details, see "Feature Machining" on page 414.
- **Z Level Roughing**. This strategy allows you to quickly remove unwanted material from a relief. For details, see "Z Level Roughing" on page 419.
- **3D Laser Machining**. This strategy allows users of a laser-engraving machine to quickly remove layers of unwanted material from a relief. For details, see "Laser Machining" on page 424.
- **Cutting Out**. This strategy allows you to create a toolpath either inside or outside the boundary of a vector object. For details, see "3D Cut Out" on page 428.
- **3D Rest Machining.** Rest (as in 'rest of material') Machining allows you to find all of the areas of a relief that cannot be machined based on one tool size, and then machine only these areas with another, smaller tool. For details, see "3D Rest Machining" on page 434.

Machine Relief

The **Machine Relief** button in the **3D Toolpaths** area of the **Toolpaths** Home page allows you to:

- Machine the entire surface of a relief.
- Machine a specific area of a relief, as defined by a selected vector object.

Using the **Machine Relief** page, you can:

- Select a different tool clearance strategy for each of the tools you want to use. Depending on what strategy you select, you can control the angle, or where and in what direction the tool cuts into the block of material.

- Control how the tool cuts into and retracts from the block of material by adding ramping moves.

To machine the surface of a relief:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Machine Relief** button in the **3D Toolpaths** area to display the **Machine Relief** page.
3. Click on one of the **Area to Machine** radio buttons to select how the relief is machined:
   - Click on the **Whole Model** radio button if you want to machine the entire surface of the relief.
   - Click on the **Selected Vector** radio button if you want to machine an area of the relief as defined by a selected vector object.

     If you select this option, select the vector object that represents the area of the relief you want to machine. For details, see "Selecting Vectors" in the Working with Vectors chapter.
4. Click on the **Strategy** list box to display the strategies available and then on the strategy you want to select. If you have selected the **Whole Model** option, then you can choose from the following strategies:
   - **Raster in X** – This strategy machines back and forth along the X-axis at a specified angle.
   - **Raster in X and Y** – This strategy machines in two perpendicular directions at a specified angle. This strategy improves the surface finish but can be time consuming.
   - **Spiral** – This strategy machines in a spiral motion, stopping when the tool reaches the first edge of the relief. This strategy does not always allow you to machine the entire relief.
   - **Spiral in Box** – This strategy machines in a spiral motion. When the tool reaches the first edge of the relief it retracts, traverses along the edge of the relief

and then plunges to continue machining the next area of the relief. This strategy allows you to machine the entire relief, but can be time consuming.

If you have selected the **Selected Vector** option, then you can choose from the following strategies:

- **Raster in X** – This strategy machines back and forth along the X-axis at a specified angle.
- **Offset** – This strategy machines in repeated passes, each time moving inwards by the **Stepover** value of the tool you use. You can see the **Stepover** value of the selected tool when the machining parameters are displayed in the **Tool** area.

5. If you have selected a **Raster** strategy, type the angle at which you want the tool to move in the **Raster Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

If you have selected the **Offset** strategy:

- First, click to select either of the **Cut Direction** radio buttons ◉:

  Climb Mill – In Climb Milling, the cutter rotates in the same direction as the feed motion.

  Conventional – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion. For more details, see "Overview" on page 335.

Note: You can set the default cutting direction using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

- Next, click click to select either of the **Start Point** radio buttons ◉:

  Outside – Select this option if you want the tool to cut into the material at the boundary of the selected vector object, then machine inwards.

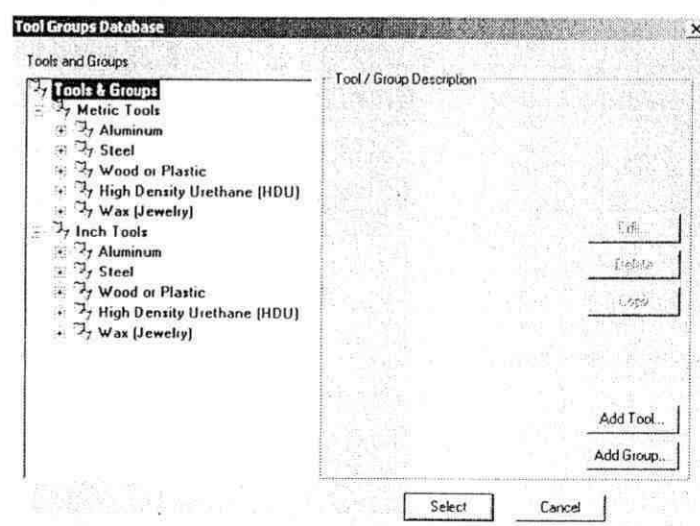
  Inside – Select this option if you want the tool to cut into the material at the centre of the selected vector object, then machine outwards.

Note: You can set the default offset start point using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

6. To leave additional material on the relief surface to allow for the final machining pass, type a value in the **Allowance** box. The allowance sets the distance between the surface of the relief and the cutting tool.
7. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the relief.
8. If you want to change the height at which the machining tool makes rapid moves between toolpath segments:
   - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:

   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
9. Click on the **Select** button in the **Tool** area to open the **Tool Groups Database**:



10. Double-click on the tool that you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Tool** area.

    If you want to amend the machining parameters for the selected tool, click on the ▼ arrow in the **Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

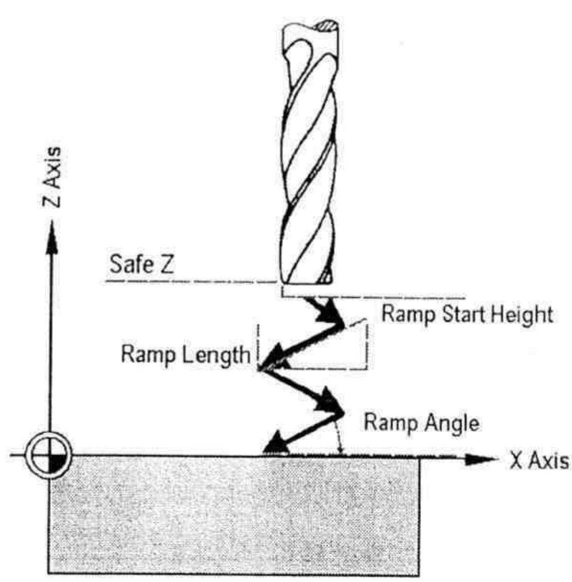
    When you have finished, click on the ▲ arrow to hide the machining parameters.

11. To define the point at which the machining tool enters and leaves the block of material:

    - First, click to select the **Do Multiple Z Passes** option ☑.
    - Next, type a value in the **Z Height of First Pass** box.
    - Finally, type a value in the **Z Height of Last Pass** box.

Note: The **Stepdown** value of the tool determines the number of passes made between the first and last pass in the toolpath.

12. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the relief surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the cutting tool to zigzag across the relief surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

13. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

14. Type a name for the toolpath in the **Name** box.
15. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.
16. Click on the **Close** button to return to the **Toolpaths** Home page.

Feature Machining

The **Feature Machining** button in the **Toolpaths** Home page allows you to machine a raised, recessed or centreline engraved feature that you have created from a vector object with a depth or height relative to the current relief. The contour of the existing relief is preserved in the feature that is combined with it. For details, see "Creating a Feature from a Vector" in the Working with Vectors chapter.

Using the **Feature Machining** page, you can:

- Select a tool clearance strategy for the machining tool you want to use. Depending on the strategy you select, you can control the angle, where and in what direction the machining tool cuts into the block of material.

To machine a feature that you have created from a vector object:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Feature Machining** button in the **3D Toolpaths** area to display the **Feature Machining** page.
3. Click on the **Choose Feature** list box to display the list of features available and then on the feature that you want to machine.

Tip: If you move the mouse cursor over a feature in the **Choose Feature** list box, a pop-up window appears detailing the attributes of the feature.

Warning: If you have not yet created a feature from a vector object, the **Choose Feature** list box is empty. For details, see "Creating a Feature from a Vector" in the Working with Vectors chapter.

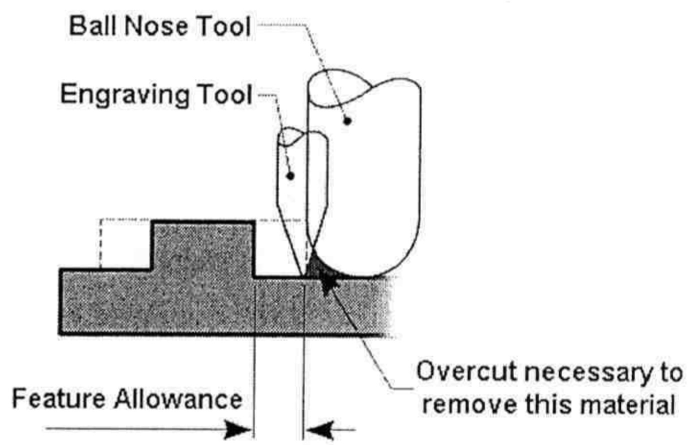
4. Click on either of the radio buttons to select the machining strategy that you want to use:
   - **Profile Only** – This strategy allows the cutting tool to make profile passes around the feature you have selected.
   - **Area Clear** – This strategy allows the cutting tool to clear an area of material based on the boundary of

the feature you have selected, then make profile passes around the feature.

Note: The selected feature is not machined using the **Area Clear** strategy. A protected area is created around the selected feature during the machining process.

If you have selected **Area Clear**, type a value in the **Overcut Distance** box to remove any ridge of unwanted material that might appear around the feature.

For example, if a Ball Nose cutting tool is used as the **Feature Tool** with an **Area Clear** strategy, a ridge of unmachined material may appear around it:



Note: The **Overcut Distance** should normally be equal to the radius of the **Feature Tool** used in an **Area Clear** strategy.

5. If you want the cutting tool to sharpen the corners in the feature as it is machined, click to select the **Corner Sharpen** option ☑.

6. To perform the machining strategy that you have selected as a series of passes in the Z direction, click to select the **Do Multiple Z Passes** option ☑.

   If you have selected this option, click on the ▼ arrow to display the **Do Multiple Z Passes** options, then establish the settings you require:

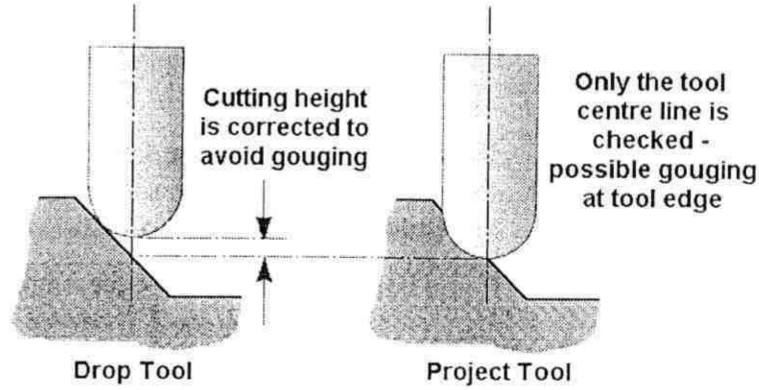
   - Type a value in the **Num Slices** box to define the number of Z passes that you want to make.

- To evenly distribute the number of Z passes through the block of material, click on the **Linear Spacing** button.
- To add another Z pass, click on the **Add** button. A new value appears in the main box and the value in the **Num Slices** box increases.
- To delete a Z pass, click to select the value from the main box, and then click on the **Delete** button. The value is removed from the main box and the value in the **Num Slices** box decreases.
- To edit any of the values shown in the main box, type the new value in the **Update** box, click to select the value from the main box, and then click on the **Update** button.
- To delete all of the current values displayed in each of the boxes, click on the **Clear All** button.

7. Click on either of the radio buttons to select how the cutting tool reaches the depth you defined when creating the feature. For details, see "Creating a Feature" in the Working with Vectors chapter.

    - If you want ArtCAM Pro to conduct a check for the probability of a collision between the tool geometry and the material surface before the tool descends to the cutting depth, click on the **Drop Tool** option.

      If a collision is anticipated, ArtCAM Pro automatically adjusts the cutting depth so as to avoid it. This can prevent the problem of gouging.



    - If you want to allow the cutting tool to descend to the cutting depth, ignoring any probability of a collision between the tool geometry and the material surface, click on the **Project Tool** option.

Warning: This option can cause gouging in the material surface around the edge of the cutting tool.

8. Click on either of the radio buttons to select the cut direction you want to use:

    - **Climb** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
    - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

    For more details, see "Overview" on page 335.

Note: You can set the default cutting direction using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

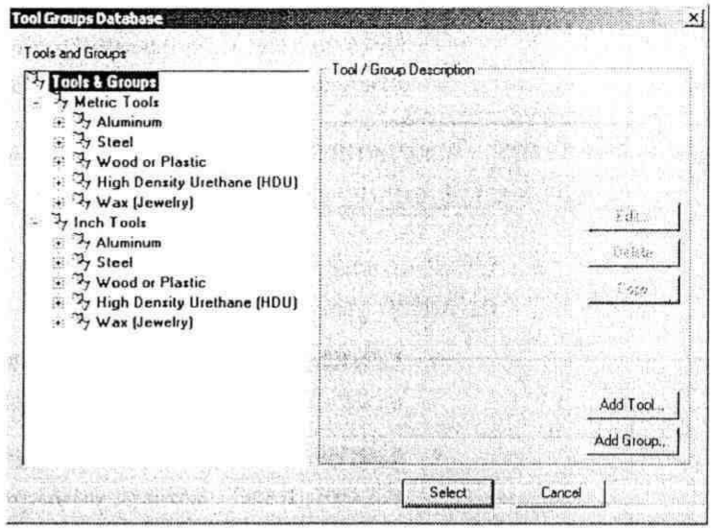
9. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the selected feature.
10. If you want to change the height at which the machining tool makes rapid moves between toolpath segments:

    - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:

    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
    - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

11. Click on the **Select** button in the **Feature Tool** area of the page to open the **Tool Groups Database**:



12. Double-click on the tool that you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Feature Tool** area.
13. If you want to amend the machining parameters for the selected tool, click on the ▼ arrow in the **Feature Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

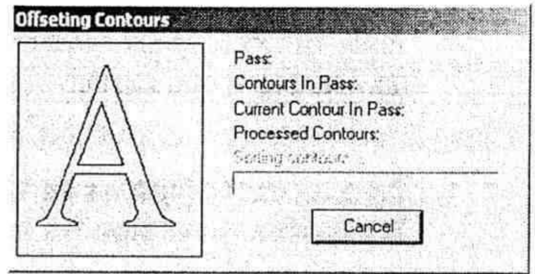
    When you have finished, click on the ▲ arrow to hide the machining parameters.
14. Click on the **Setup** button in the **Material** area of the page to display the **Material Setup** dialog box.

    Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

    If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.
15. Type a name for the toolpath in the **Name** box.
16. You can now specify when you want to calculate the toolpath:

    - If you want to calculate the toolpath now, click on the **Now** button.

      The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:



The machining passes used to create the selected feature are represented by blue lines drawn on the boundary of the vector object from which the feature was previously created. For details, see "Creating a Feature" in the Working with Vectors chapter.

You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

- If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.

17. Click on the **Close** button to return to the **Toolpaths** Home page.

Z Level Roughing

The **Z Level Roughing** button in the **3D Toolpaths** area of the **Toolpaths** Home page allows you to quickly remove unwanted material from a relief in planar slices. The machining process involves a series of passes in the Z direction at a specified depth. You can remove material across the entire surface of the relief or in selected areas, as defined by a selected vector object.

Warning: You should only use an End Mill tool with this toolpath strategy. If any other shape of tool is used, additional material is left on the relief.

Using the **Z Level Roughing** page, you can:

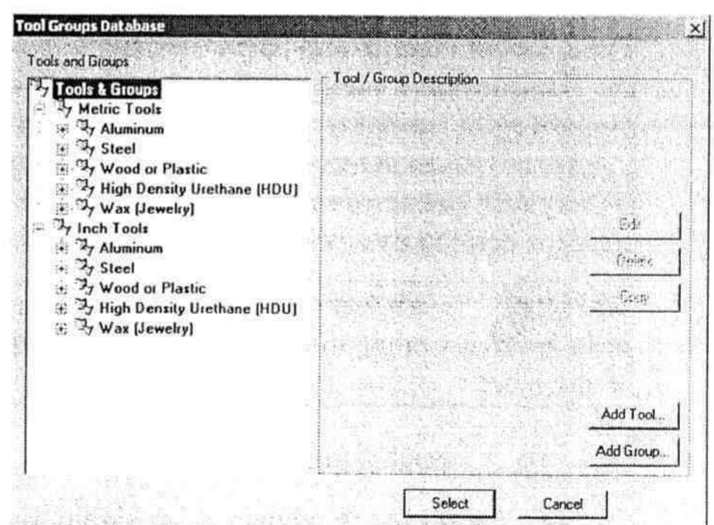
- Select the roughing tool you want to use.
- Control how the tool cuts into and retracts from the block of material by adding ramping moves.

- Depending on what strategy you select, you can control the angle, or where and in what direction the roughing tool cuts into the block of material.

To remove a cross section of a relief in the Z direction:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Z Level Roughing** button in the **3D Toolpaths** area to display the **Z Level Roughing** page.
3. Click on either of the **Area to Machine** radio buttons to select how the material is removed from the relief:
   - Click on the **Complete Relief** radio button if you want to remove material across the entire surface of the relief.
   - Click on the **Area under selected vectors** radio button if you want to remove material in an area of the relief, as defined by a selected vector object.

     If you select this option, select the vector object that represents the area of the material that you want to remove from the relief. For details, see "Selecting Vectors" in the Working with Vectors chapter.
4. Click on the **Select** button in the **Roughing Tool** area to display the **Tool Groups Database**:



5. Double-click on the tool you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Roughing Tool** area.

If you want to amend the machining parameters for the selected tool, click on the ▼ arrow in the **Roughing Tool** area. For further information, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.

Make sure that the **Stepdown** value of the tool is equal to the thickness of the planar slices you want to create. If you adjust the **Stepdown**, click on the **Apply** button in the **Z Slices** area of the page after you have done so.

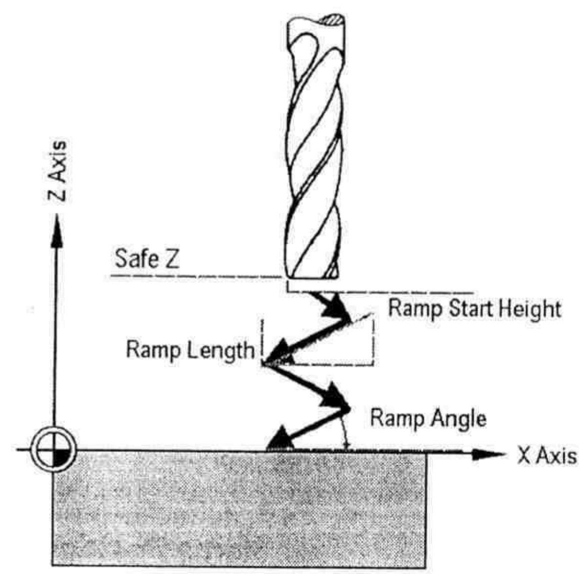
When you have finished, click on the ▲ arrow to hide the machining parameters.

6. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

   Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

   If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.

7. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:



   - Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.

- Type the maximum distance that you want the cutting tool to zigzag across the relief surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the cutting tool to zigzag across the relief surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

8. If you want to control the number of Z passes used to machine the relief:
   - First, type a value in the **Start/Surface Z** box in the **Z Slices** area of the page to set the position of the first Z pass.
   - If you want to add or remove extra material from the relief surface, type a value in the **Material Allowance** box. The allowance sets the distance between the surface of the relief and the cutting tool.
   - Next, type a value in the **Last Slice Z** box to set the position of the final Z pass.
   - Finally, click on the **Apply** button to confirm your settings.
9. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:
   - First, click on the ▼ arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, define the height in the **Safe Z** box.

   If you want to define the **Home Position** for the machining tool:
   - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.
   - Next, click on the ▲ arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

10. Type a value in the **Tolerance** box to specify how closely you want the cutting tool to follow the shape of the relief.
11. Click on either of the radio buttons to select the machining strategy that you want to use:
    - **Raster** – This strategy machines back and forth along the X-axis at a specified angle.

      If you select **Raster**, first type the angle you want the tool to move at in the **Angle** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on one of the **Profile Pass** radio buttons :

None – Select this option if you do not want the tool to profile the relief.

First – Select this option if you want the tool to profile the relief first and then raster clear the area.

Last – Select this option if you want the tool to move outwards to raster clear the area, then profile the relief.

- **Offset** – This strategy machines in repeated passes, each time moving inwards by the **Stepover** value of the tool you use.

  If you have selected **Offset**, first click on one of the **Cut Direction** radio buttons :

  Climb – In Climb Milling, the cutter rotates in the same direction as the feed motion.

  Conventional – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

  For more details, see "Overview" on page 335.

Note: You can set the default cutting direction in ArtCAM Pro using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Next, click on one of the **Start From** radio buttons :

Outside – Select this option if you want the tool to cut into the material at the boundary of the relief surface, then machine inwards.

Inside – Select this option if you want the tool to cut into the material at the centre of the relief surface, then machine outwards.

Note: You can set the default offset start point using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

---

12. Type a name for the toolpath in the **Name** box.
13. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.
    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.
14. Click on the **Close** button to return to the **Toolpaths** Home page.

Laser Machining

The **3D Laser Machining** button in the **3D Toolpaths** area of the **Toolpaths** Home page allows users of a laser-engraving machine to quickly remove layers of unwanted material from a relief. The machining process involves a laser beam making a series of passes in the Z direction at a specified depth.

Using the **Laser Machining** page, you can:

- Control whether the laser machines from the base or the surface of the relief.
- Output the toolpath in segments to allow for any refocusing of the laser that might be required between passes.

- Depending on what strategy you select, you can control the angle, or where and in what direction the laser cuts into the block of material.
- Engrave detail into a previously machined model, using two reliefs. This process is known as Rest (as in 'rest of material') Laser Engraving.

Important: The toolpath(s) need to be saved as an output file using a suitable post-processor such as the 2D HPGL (***.plt**) before being sent to a laser engraving machine. For details, see "Saving a Toolpath" on page 463.

To remove layers of a relief in the Z direction using a laser engraving machine:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **3D Laser Machining** button in the **3D Toolpaths** area to display the **Laser Machining** page.
3. Click on the **Setup** button in the **Material** area to display the **Material Setup** dialog box.

   Make sure that the thickness of the block, the material Z zero position and the position of the model in the block is correct, then click on the **OK** button.

   If you want to change any of these settings, see "Adjusting the Material Setup" on page 481.
4. If you want to control the number of Z passes used to laser machine the relief, you can:
   - Type a value in the **Start/Surface Z** box to set the position of the first Z pass.
   - To enlarge or reduce the relief, type a value in the **Material Allowance** box. The allowance sets the distance between the surface of the relief and the cutting tool. Type a positive value to enlarge the relief or a negative value to reduce it.
   - Type a value in the **Last Slice Z** box to set the position of the final Z pass.
   - To machine from the base of the relief upwards, click to select the **Reverse Tool Slice Order** option.

- Click on the **Apply** button to confirm your settings.

5. In the **Toolpath Section(s)** area, click on either of the radio buttons to select how the toolpath is calculated:
   - **Output One Toolpath** – This option allows you to laser machine the entire relief at once.
   - **Output In Sections** – This option allows you to laser machine the relief in segments. If you select this option, type a value in the **Section Thickness** box to set the depth of each toolpath segment.
6. In the **Slice Type** area, click on either of the radio buttons to set the type of strategy you want to use when laser machining the relief:
   - **Profile** – This strategy machines around the boundary of the relief surface.
   - **Raster Clearance** - This strategy machines back and forth at a specified angle.

     If you have selected **Raster Clearance**, first type a value in the **Stepover** box to set the distance between each Z pass. This value should be appropriate to the diameter of the laser beam on your laser-engraving machine.

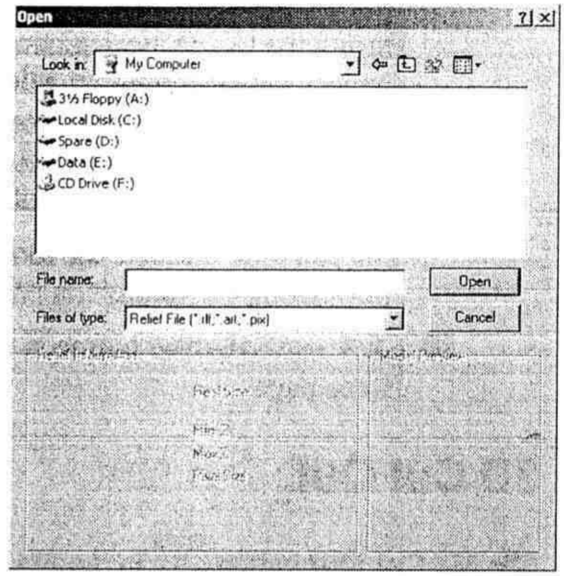
     Next, type the angle increment at which you want the laser to move during each Z pass in the **Angle Increment** box.

Note: You can set the default raster angle using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

If you want to remove the material, click to select the **Remove Material** option.

7. If you want to rest laser engrave, the current relief should contain all the engraved detail you want to include in the laser machined model. You also must have already saved the relief prior to any engraved detail being added. To rest laser engrave the relief:
   - First, click to select the **3D Rest Laser Engraving** option. The **Load** button is displayed.

- Next, load the relief representing the model prior to the addition of any engraved detail by clicking on the **Load** button to display the **Open** dialog box:



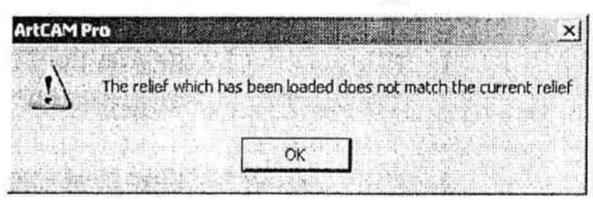
- Next, click on the **Look in** list box and select the directory where the relief file that you want to load is stored.

- Once you have found the relief file, click to select it from the main window of the **Open** dialog box. Its name appears in the **File Name** box.

Note: The size of the imported relief must match that of the current relief in the ArtCAM model.

- Finally, click on the **Open** button to load the relief.

  If the relief is not the same size as the current relief, the following message box appears:



  Click on the **OK** button to close the message box., and then repeat these steps making sure that the size of the imported relief matches that of the current relief.

8. Type a name for the toolpath in the **Name** box.

9. Click on the **Calculate** button to calculate the toolpath. If you are rest laser engraving, ArtCAM Pro generates the toolpaths needed to laser machine the differences between the two reliefs.

   During the toolpath calculation process, ArtCAM Pro displays the **Generating Slices** progress bar and a **Cancel** button beneath the design window area. If you want to stop ArtCAM Pro generating the Z slices in the toolpath, click on the **Cancel** button.

   You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.

10. Click on the **Close** button to return to the **Toolpaths** Home page.

3D Cut Out

The **3D Cut Out** button in the **3D Toolpaths** area of the **Toolpaths** Home page allows you to create a toolpath that machines either inside or outside the boundary of a selected vector object using absolute Z values.

Using the **3D Cut Out** page, you can:

- Control where your tool cuts into and retracts from the block of material by adding lead in and lead out moves.
- Control how your tool cuts into the block of material by adding ramping moves.
- Control the cutting direction of the tool.
- Control whether the vector object is cut or snapped out from the block of material by adding bridging.

To cut out a vector object from the relief:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **3D Cut Out** button in the **3D Toolpaths** area to display the **3D Cut Out** page.
3. Select the vector object that represents the area of the material that you want to cut from the relief. For details, see "Selecting Vectors" in the Working with Vectors chapter.

4. In the **Profile Side** area, select how you want to profile the selected vector object:

   - Click on the **Outside** radio button to instruct the tool to profile outside of the selected vector object.

   - Click on the **Inside** radio button to instruct the tool to profile outside of the selected vector object.

5. Type the absolute Z value from which the position of the first profile pass is to be calculated in the **Surface Z** box.

6. Type the absolute Z value from which the position of the first profile pass is to be calculated in the **Surface Z** box.

7. Type the absolute Z zero value for the bottom of the cut in the **Finish Z** box.

8. If you want to add or remove extra material around the vector object, you can type a value in the **Allowance** box. The value you enter sets the distance between the boundary of the selected vector object and the cutting tool. Type a positive value to add material or a negative value to remove it.

9. Type a value in the **Tolerance** box to specify how closely you want the cutter to follow the shape of the vector object.

10. If you want to change the height at which the cutting tool makes rapid moves between toolpath segments:

    - First, click on the arrow in the **Machine Safe Z** area to display the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

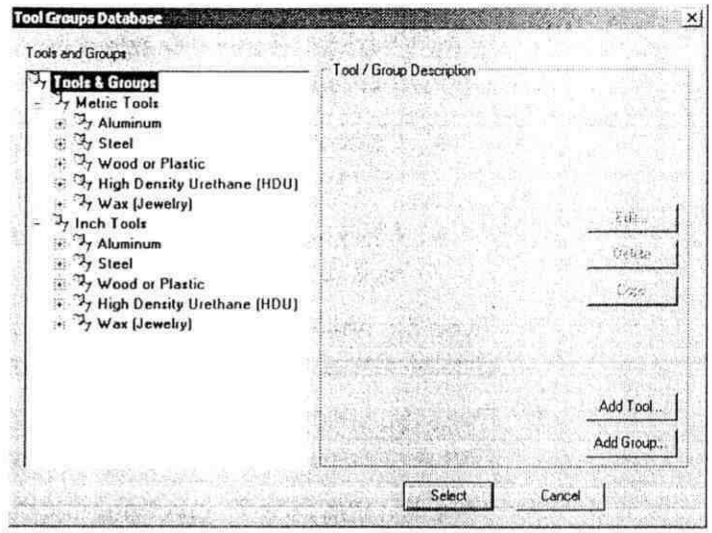
    - Next, define the height in the **Safe Z** box.

    If you want to define the **Home Position** for the machining tool:

    - First, define its position along the X, Y and Z axes in the **Home Position**'s **X**, **Y** and **Z** boxes.

    - Next, click on the arrow in the **Machine Safe Z** area to hide the **Safe Z** box and the **Home Position**'s **X**, **Y** and **Z** boxes.

11. Click on the **Select** button in the **Profiling Tool** area of the page to open the **Tool Groups Database**.



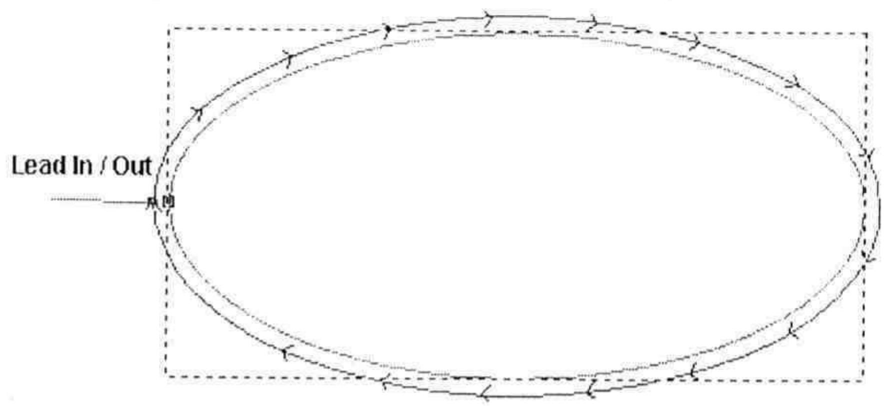
12. Double-click on the tool that you want to use. ArtCAM Pro closes the **Tool Groups Database** and displays the selected tool's description in the **Profiling Tool** area.
13. The cutting direction defaults to **Climb Mill**. If you want to change this, click on the arrow in the **Cut Direction** area of the **3D Cut Out** page, then click on one of the **Cut Direction** radio buttons :
    - **Climb Mill** – In Climb Milling, the cutter rotates in the same direction as the feed motion.
    - **Conventional** – In Conventional Milling, the cutter rotates in the opposite direction to the feed motion.

    For more details, see "Overview" on page 335.

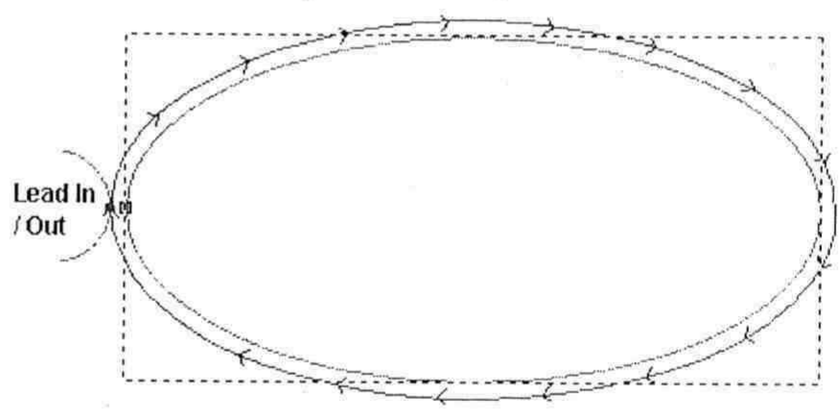
Note: You can set the default cutting direction in ArtCAM Pro using the ArtCAM Options page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

14. To add lead-in and lead-out moves to the toolpath, click to select the **Add Lead In/Out Moves** option .
    - If you only want to have a lead-in move, click to select the **Do not Lead Out** option .
    - Type the distance from the toolpath that you want the tool to cut into and retract from the block of material in the **Distance (D)** box.
    - Select how you want the tool to move using the **Add Lead In/Out Moves** radio buttons :

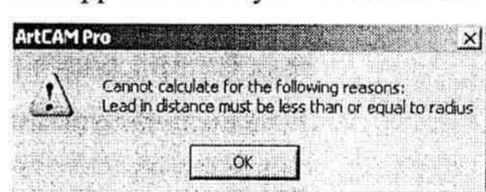
Linear – Click on the **Linear** option to instruct the tool to lead into and out of the vector object's boundary in a straight-line motion:



Circular Arc – Click on the **Circular Arc** option to instruct the tool to lead into and out of the vector object's boundary in an arc motion:



If you select the **Circular Arc** option, define the radius of the arc in the **Radius (R)** box. The radius must be less than or equal to the value in the **Distance (D)** box, otherwise the following message box appears when you calculate the toolpath:



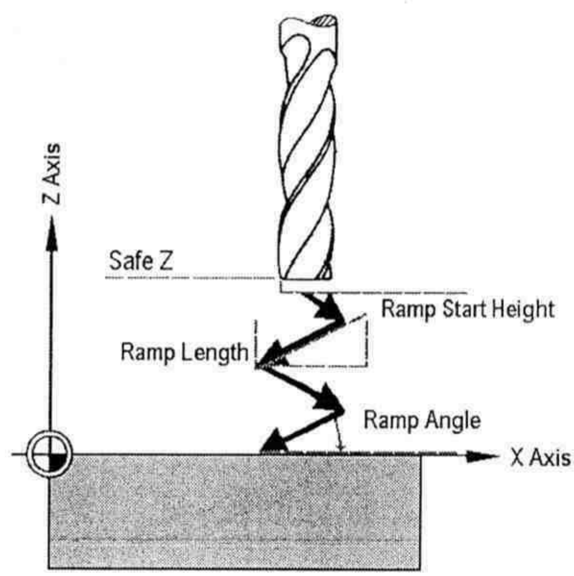
- Set the **Automatic Positioning** as follows:

  To position lead-in and lead-out moves at the optimum point in the vector object, click to select the

Automatic Positioning option ☑. This point usually within the vector object's longest span.

If the **Automatic Positioning** option is deselected ☐, the lead-in and lead-out moves are positioned at the Start Point of the vector object. If you change the Start Point, this changes the position of the lead-in and lead-out moves. For details on changing the Start Point, see "Changing the Start Point" in the Working with Vectors chapter.

15. To add ramping moves to the toolpath, click to select the **Add Ramping Moves** option ☑. The ramping moves settings are displayed:

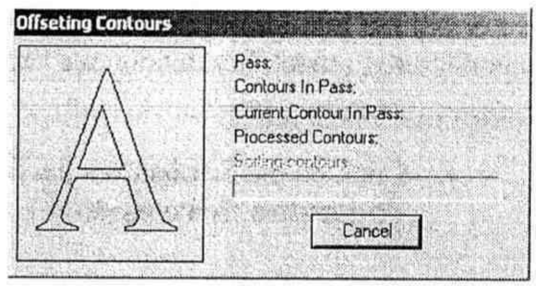


- Type the maximum angle of descent for each zig and zag movement of the cutting tool in the **Max Ramp Angle (A)** box.
- Type the maximum distance that you want the cutting tool to zigzag across the relief surface in the **Max Ramp Length (L)** box.
- Type the minimum distance that you want the cutting tool to zigzag across the relief surface in the **Min Ramp Length (Lmin)** box.
- If you want to define the height at which the ramping moves start, click to select the **Ramp Start Height (S)** option ☑, and then type a value in the box

beneath it. If you do not define the **Ramp Start Height (S)** here, the **Safe Z** level is used.

For more details, see "Overview" on page 335.

16. Type a name for the toolpath in the **Name** box.
17. You can now specify when you want to calculate the toolpath:
    - If you want to calculate the toolpath now, click on the **Now** button. The **Offsetting Contours** dialog box is displayed while ArtCAM Pro calculates the toolpath:

      A dark red line represents the machining passes used to profile the selected vector object. It is drawn either inside or outside of its boundary, according to the **Profile Side** that you had selected. A circle marking the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

      The toolpath preview is not shown if the **Create 2D Preview** option is deselected.

      You can now simulate the toolpath. For details, see "Simulating Toolpaths" on page 485.
    - If you want to calculate the toolpath as part of a batch of toolpaths at a later time, click on the **Later** button. For details, see "Creating a Single Copy" on page 443 and "Calculating a Batch of Toolpaths" on page 469.
18. Click on the **Close** button to return to the **Toolpaths** Home page.

If you want to add bridges to the 3D Cut Out toolpath, see "Adding Bridging" on page 450.

If you want to set the machining order of the 3D Cut Out toolpath, see "Setting the Machining Order" on page 458.

3D Rest Machining

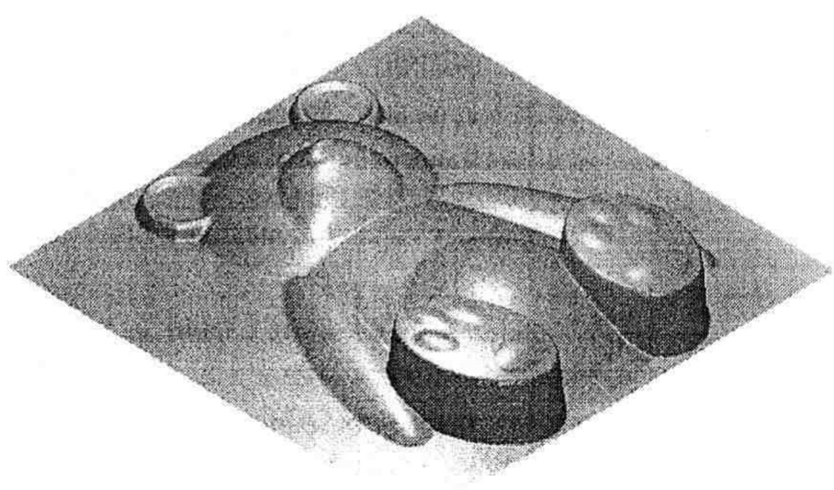
Rest (as in 'rest of material') Machining allows you to find all of the areas of a relief that cannot be machined based on one tool size, and then machine only these areas with another, smaller tool.

ArtCAM Pro compares the relief representing a finished model with simulated toolpath data, and then creates vector objects in the shape of the areas of the relief that these toolpaths fail to machine. A *Machine Relief* toolpath strategy can then be applied to some or all of these vector objects to improve the overall surface finish of the manufactured model. For details, see "Machine Relief" on page 408.

To rest machine a relief:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **3D Rest Machining** button in the **3D Toolpaths** area to display the **3D Rest Machining** page.
3. In the **Area To Rest Machine** area, select the area of the relief that you want to machine:
    - **Complete Relief** – This option instructs ArtCAM Pro to distinguish the differences between the toolpath simulation and the existing relief, and then identify them for machining purposes.

      For example, a teddy bear relief looks as follows:

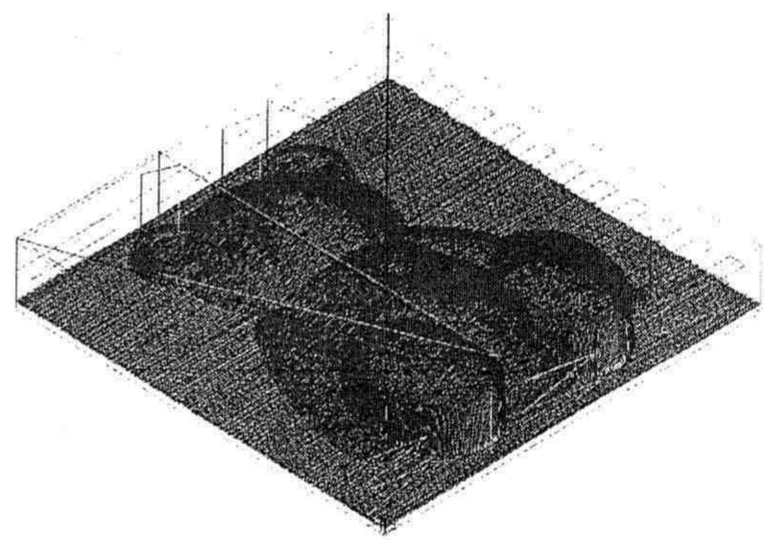


- **Area Under Selected Vectors** – This option instructs ArtCAM Pro to distinguish the differences between the toolpath simulation and the relief underlying the currently selected vector objects, and then identify them for machining purposes. For details, see "Selecting Vectors" in the Working with Vectors chapter.

4. In the **Identify Areas Already Machined By...** area, click to select the method you want to use to identify the unmachined areas of the current relief:

    - **Simulating All Toolpaths** – This option instructs ArtCAM Pro to simulate all calculated toolpaths and then compare the result against the relief to identify its unmachined areas.

      In our example, all three toolpaths used to machine the teddy bear relief are simulated. These appear in the **3D View** window as follows:

    - **Simulating the Last Toolpath** – This option instructs ArtCAM Pro to simulate the last calculated toolpath and then create a model of the unmachined areas of the relief.

    - **Using the Current Simulation** – This option instructs ArtCAM Pro to use the existing toolpath simulation shown in the **3D View** window to identify the unmachined areas of the relief.

5. After simulating a toolpath, you may notice small areas of material, or 'cusps', remain in the model as a consequence of

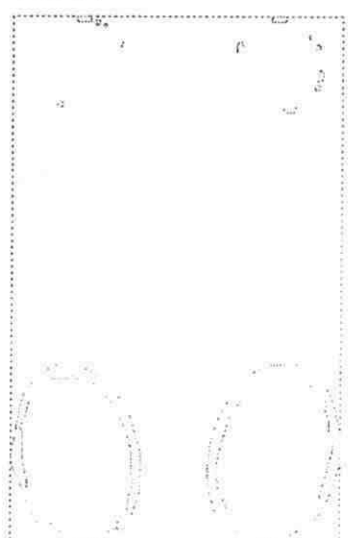
a machining tool's geometry or the stepover used between machining passes. In the **Cusp Tolerance** box, type the height at which you want ArtCAM Pro to ignore any existing 'cusps' as areas of the model that require additional machining.

6. In the **Layer for rest boundaries** box, type the name of the layer in which you want ArtCAM Pro to create the vector objects representing the unmachined areas of the relief.

Note: If the **Layer for rest boundaries** box is empty, the vector objects will be created on the currently selected layer. For details, see "Selecting a Layer" in the Working with Models chapter.

7. Click on the **Create Boundaries** button to calculate the unmachined areas of the relief from the toolpath simulation, and then create vector objects representing these areas. The simulated toolpaths are displayed in the **3D View** window.
8. Press the **F2** key to display the vector objects representing the unmachined areas of the relief in the **2D View** window.

   In our example, ArtCAM Pro identifies unmachined areas around the feet and ears in the teddy bear relief, as shown below: .

9. Click on the **Close** button to return to the **Toolpaths** Home page.

You are now ready to apply a *Machine Relief* toolpath to the vector objects representing the unmachined areas. This toolpath will improve the overall surface finish of the relief during the manufacturing process. For details, see "Machine Relief" on page 408.

Managing and Modifying Toolpaths

You can use the **Toolpaths** Home page to modify and manage the toolpath strategies that you are using.

You can modify or manage a toolpath strategy in the following ways:

- Merge a selection of calculated toolpaths into a single toolpath. For details, see "Merging Toolpaths" on page 439.
- Set the order in which the individual toolpaths selected for merging will be machined. For details, see "Creating a Single Copy" on page 443, "Creating a Block Copy" on page 446 and "Merging Toolpaths" on page 439.
- Transform one or more calculated toolpaths. For details, see "Transforming Toolpaths" on page 441.
- Make a single copy of one or more calculated toolpaths. For details, see "Creating a Single Copy" on page 443.
- Make multiple copies of one or more calculated toolpaths in a grid format. For details, see "Creating a Block Copy" on page 446.
- Set the machining order in all toolpaths that involve profile passes. For details, see "Setting the Machining Order" on page 458.
- Adjust the machining parameters of a tool both during and after the toolpath creation. For details, see "Adjusting the Machining Parameters of a Tool" on page 460.
- Save a toolpath. For details, see "Saving a Toolpath" on page 463.
- Edit toolpath settings. For details, see "Editing a Toolpath" on page 466.
- Delete a toolpath or toolpath group. For details, see "Deleting Toolpaths" on page 466.
- Calculate a toolpath by itself or as part of a batch. For details, see "Calculating a Single Toolpath" on page 469 and "Calculating a Batch of Toolpaths" on page 469.

- Create a **Toolpath Summary** and calculate the estimated machining time. For details, see "Using a Toolpath Summary" on page 471.
- Add to, edit or delete the tools within the **Tool Groups Database**. "Using the Tool Groups Database" on page 474.
- Adjust the **Material Setup**. For details, see "Adjusting the Material Setup" on page 481.
- Delete the material.. For details, see "Deleting the Material" on page 483.
- Save a toolpath as a template. For details, see "Creating a Toolpath Template" on page 483.
- Load a toolpath template. For details, see "Loading a Toolpath Template" on page 484.

Selecting Toolpaths

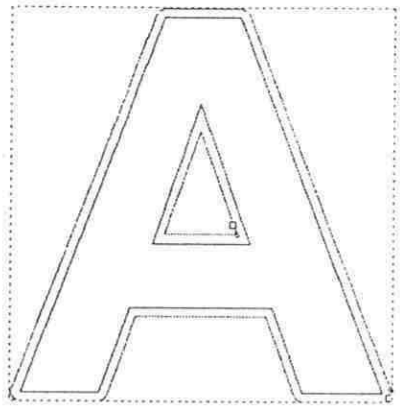
You can manipulate calculated 2D or 3D toolpaths in the same way as you would vector objects in ArtCAM Pro.

You can transform, copy or edit profile passes within a toolpath, provided that a **2D Preview** was created during the calculation process. Toolpath previews are shown in the **2D View** window only and are dark red.

To select a toolpath preview from the **2D View** window:

1. Make sure that you are in **Select Vectors** mode. For details, see "Selecting Vectors" in the Working with Vectors chapter.
2. Click to select the preview of the toolpath that you want to use. The selected toolpath preview turns blue and is surrounded by a bounding box.

   For example, the preview of a Profiling toolpath is shown below:



Note: You can set the default colour of a selected toolpath preview using the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

To select more than one toolpath preview from the **2D View** window:

1. Make sure that you are in **Select Vectors** mode . For details, see "Selecting Vectors" in the Working with Vectors chapter.
2. You can now use either of the following methods to select more than one toolpath preview:
   - Click and drag to form a bounding box around the previews of the toolpaths that you want to use. The selected toolpath previews are blue and surrounded by a bounding box.
   - Hold down the **Shift** key on your keyboard, and then click to select each of the toolpath previews that you want to use. The selected toolpath previews are blue and surrounded by a bounding box.

You can merge, edit, save, delete, calculate, create a template or adjust the machining parameters of a tool using the toolpaths listed on the **Toolpaths** Home page.

To select a toolpath listed on the **Toolpaths** Home page:

1. Click on the name of the tool. The tool name is highlighted in blue.

Merging Toolpaths

You can merge calculated 2D or 3D toolpaths using the same tool geometry and number into a single toolpath. This avoids the need for the tool to repeatedly return to the Home position when machining them. Merging toolpaths saves machining time and also reduces wear on cutting tools.

Warning: Once a selection of toolpaths has been merged into a group, they cannot be ungrouped or edited. You should save the model before merging any toolpaths or use the **Preserve original toolpaths** option on the **Toolpath Merge** page. This way, if any mistakes are made, you can return to the toolpaths in their original state.

To merge a selection of toolpaths:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Merge** button in the **Toolpath Operations** area to display the **Toolpath Merge** page. A list of all calculated toolpaths is displayed.

Note: You can also display the **Toolpath Merge** page if you right-click on a selected 2D toolpath preview to display the Toolpath Editing menu, and then click on the **Merge Toolpaths** option

3. Make sure that only the toolpaths that you want to merge are checked:
   - If you want to merge a 2D or 3D toolpath, its **Show In 3D** option must be selected.
   - If you do not want to merge a 2D or 3D toolpath, its **Show In 3D** option must be deselected.
4. In the **Merging Order** area, select the order in which you want the toolpaths to be machined:
   - **As List** - Click on this radio button if you want the toolpaths to be machined in the order in which they are listed.

     You can prioritise the toolpaths using the and arrows. Click to select the name of the toolpath in the list that you want to prioritise, and then on the appropriate arrow to set its position in the list. The toolpath at the top of the list is the first to be machined, and so on.

- **Automatic** - Click on this radio button if you want ArtCAM Pro to calculate the best order in which to machine the toolpaths, finding the shortest route between them.

5. If you want to keep a copy of the toolpaths in their original state, click to select the **Preserve original toolpaths** option.
6. Type a name for the merged toolpath in the **Name** box.
7. Click on the **Calculate** button to merge the selected toolpaths into a single toolpath.

   A progress bar appears beneath the **2D View** window indicating the progress made in merging the toolpaths.
8. Click on the **Close** button to return to the **Assistant**'s Home page.

Transforming Toolpaths

You can move and/or mirror a calculated 2D or 3D toolpath.

To transform toolpaths:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Transform** button in the **Toolpath Operations** area to display the **Toolpath Transform** page. A list of all calculated toolpaths is displayed.

Note: You can also display the **Toolpath Transform** page if you right-click on a selected 2D toolpath preview to display the Toolpath Editing menu, and then click on the **Transform Toolpath** option

3. Select which of the toolpaths you want to transform:
   - **Selected** – Click on this radio button if you want to transform a single selected toolpath only.

     To select a toolpath from the list, click on its name. It is highlighted in blue.

     If the toolpath makes use of more than one tool, click on the toolpath name to transform all of its machining passes or on an individual tool name if you want to transform a single machining pass.

- **All Visible** – Click on this radio button if you want to transform all of the checked toolpaths listed on the page.

  If you want to transform a 2D or 3D toolpath, its **Show In 3D** option must be selected.

  If you do not want to transform a 2D or 3D toolpath, its **Show In 3D** option must be deselected.

4. If you want to move the selected toolpath(s):
   - Type the distance by which you want to move the selected toolpath(s) along the X-axis in the **X** box.
   - Type the distance by which you want to move the selected toolpath(s) along the Y-axis in the **Y** box.
   - Type the distance by which you want to move the selected toolpath(s) along the Z-axis in the **Z** box.
5. If you want to keep a copy of all selected toolpaths in their original position, click to select the **Preserve Originals** option.
6. If you want to mirror the selected toolpath(s):
   - Click on the **Mirror X** button to mirror the selected toolpath(s) about the X-axis.
   - Click on the **Mirror Y** button to mirror the selected toolpath(s) about the Y-axis.
7. Click on the **Apply** button to transform the selected toolpath(s).
8. Click on the **Close** button to return to the **Assistant**'s Home page.

Copying Toolpaths

You can create a single copy or multiple copies of any calculated 2D and/or 3D toolpath using the **Copy Toolpath** page.

There are two options available for copying toolpaths on the **Copy Toolpath** page:

- **Copy** – This option allows you to make single copies of calculated 2D and/or 3D toolpaths, and paste them to a specific position in the model. For details, see "Creating a Single Copy" on page 443.

- **Block Copy** – This option allows you to make several copies of calculated 2D and/or 3D toolpaths in a grid format. For details, see "Creating a Block Copy" on page 446.

Creating a Single Copy

You can create single copies of calculated 2D and/or 3D toolpaths.

To create single copies of toolpaths:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Copy** button in the **Toolpath Operations** area to display the **Copy Toolpath** page.

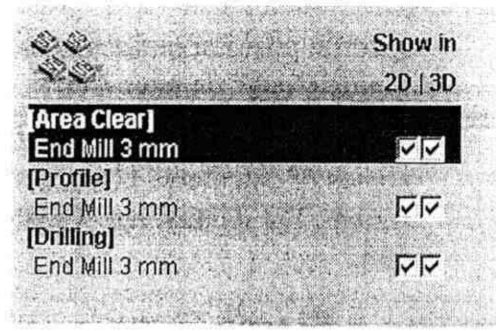
Note: You can also display the **Copy Toolpath** page if you right-click on a selected 2D toolpath preview to display the Toolpath Editing menu, and then click on the **Copy Toolpath** option

3. Click on the **Copy** radio button to display its settings.
4. Select which of the toolpaths you want to copy:
   - **Selected** – Click on this radio button if you want to copy a single toolpath only.

     To select the toolpath from the list, click on its name. It is highlighted in blue.

     If the toolpath makes use of more than one tool, click on the toolpath name to copy all of its machining passes or on an individual tool name if you want to copy a specific machining pass.

     For example, a selected 2D Area Clearance toolpath is shown below:

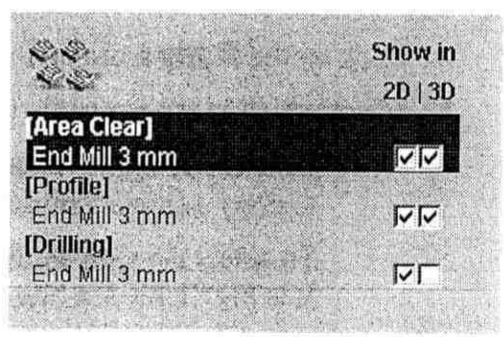


- **All Visible** – Click on this radio button if you want to copy all of the checked toolpaths in the list.

  If you want to block copy a 2D or 3D toolpath, its **Show In 3D** option must be selected.

  If you do not want to block copy a 2D or 3D toolpath, its **Show In 3D** option must be deselected.

  If we were to copy only the Profiling and 2D Area Clearance toolpaths as listed above, the Drilling toolpath's **Show In 3D** option must be deselected, as shown:



5. In the **X Offset** box, type the distance between the selected toolpath and its copy along the X-axis.

   The values you type as the X and Y offsets set the distance between the bottom left corner of the selected toolpath and its copy. You can use both positive and negative values:

   - To create a copy to the right and above the selected toolpath, type positive values in the offset boxes e.g. 90.
   - To create a copy to the left and below of the selected toolpath, type negative values in the offset boxes e.g. -90.

   In our example, the X offset is set to 45.

6. In the **Y Offset** box, type the distance you want to set between the selected toolpath and its copy along the Y-axis.

   In our example, the Y offset is set to -47.

7. If you have selected a 3D toolpath, in the **Z Offset** box type the distance you want to set between the selected toolpath and its copy along the Z-axis.

Note: If you have selected a 2D toolpath, this box is greyed-out.

8. If you want to merge the copy with its original toolpath:

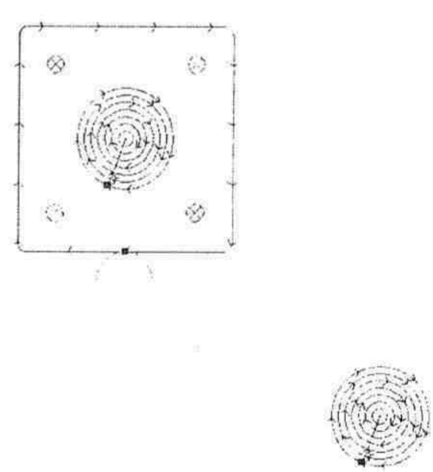
   - First, prioritise the toolpaths using the and arrows. Click to select the name of the toolpath in the list that you want to prioritise, and then on the appropriate arrow to set its position in the list. The order in which the toolpaths are listed on the page reflects the machining order of the toolpaths that make up the merged toolpath. The toolpath at the top of the list is the first to be machined, and so on.
   - Next, click on the **Merge Results** option to select it. A **Name** box appears.
   - Finally, type a name for the new toolpath in the **Name** box.

9. Click on the **Apply** button to create the new toolpath(s).

   In our example, the copied toolpath appears in the **2D View** window, as shown below:

Before...

After...



The copied toolpath is named *Area Clear 1* by default in the list of toolpaths on the page.

10. Click on the **Close** button to return to the **Assistant**'s Home page.

Creating a Block Copy

You can make copies of calculated 2D and/or 3D toolpaths in a grid format.

To create block copies of toolpaths:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Copy** button in the **Toolpath Operations** area to display the **Copy Toolpath** page. A list of all calculated toolpaths is displayed.

Note: You can also display the **Copy Toolpath** page if you right-click on a selected 2D toolpath preview to display the Toolpath Editing menu, and then click on the **Copy Toolpath** option

3. Click on the **Block Copy** radio button to display its settings.
4. Select which of the toolpaths you want to block copy:
   - **Selected** – Click on this radio button if you want to block copy a single toolpath only.

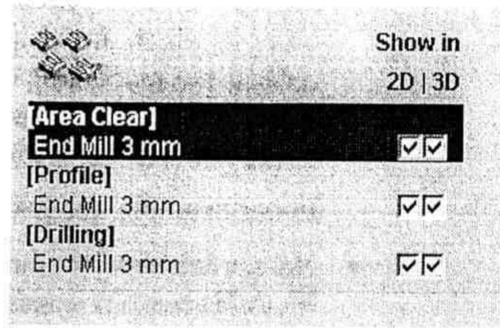
     To select the toolpath from the list, click on its name. It is highlighted in blue.

     If the toolpath makes use of more than one tool, click on the toolpath name to block copy all of its machining passes or on an individual tool name if you want to block copy a specific machining pass.
   - **All Visible** – Click on this radio button if you want to block copy all of the checked toolpaths in the list.

     If you want to block copy a 2D or 3D toolpath, its **Show In 3D** option must be selected.

     If you do not want to block copy a 2D or 3D toolpath, its **Show In 3D** option must be deselected.

For example, the **Show In 3D** option for all three toolpaths is selected, meaning that all of the listed toolpaths are visible, as shown below:



5. Type the distance you want to set between each copy along the X-axis in the **X Offset** box.

   The values you type as X and Y offsets set the distance between the bottom left corner of each copy of the selected toolpath(s).

   You can set the distance you want between each copy using a positive or negative value.

   - To create a block of copies to the right of and above the selected toolpath(s), type positive values in the offset boxes e.g. 90.
   - To create a block of copies to the left of and below of the selected toolpath(s), type negative values in the offset boxes e.g. -90.

   In our example, the X offset is set to 45.

6. Type the distance you want to set between each copy along the Y-axis in the **Y Offset** box.

   In our example, the Y offset is set to -47.

7. Type the distance you want to set between each copy along the Z-axis in the **Z Offset** box.

8. Define the number of copies by typing the number of rows (horizontal) and columns (vertical) you want in the **Number of Rows** and **Number of Columns** boxes.

   In our example, the number of rows and columns is set to 2.

9. If you want to merge all selected toolpaths and their block copies into a single toolpath:

- First, prioritise the toolpaths using the and arrows. Click to select the name of the toolpath in the list that you want to prioritise, and then on the appropriate arrow to set its position in the list. The order in which the toolpaths are listed on the page reflects the machining order of the toolpaths that make up the merged toolpath. The toolpath at the top of the list is the first to be machined, and so on.

  In our example, the list remains unchanged.

- Next, click to select the **Merge Results** option to . A **Name** box appears.
- Finally, type a name for the new toolpath in the **Name** box.

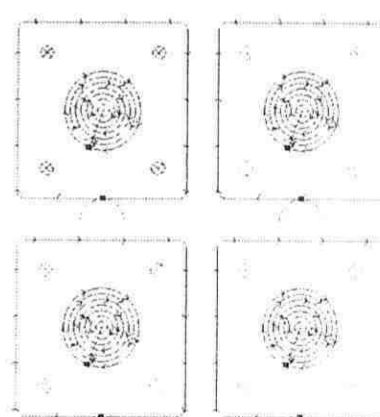
  In our example, the merged toolpath is named *Merged*.

10. Click on the **Apply** button to create the new block of toolpath copies.

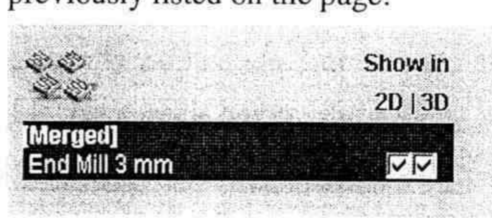
    In our example, the block of copied toolpaths appears in the **2D View** window, as shown below:

Before...

After...



The *Merged* toolpath replaces all of the individual toolpaths previously listed on the page:



11. Click on the **Close** button to return to the **Assistant**'s Home page.

Editing a Profile Pass

You can edit any profile pass within any calculated toolpath using the **Profile Options** page.

Note: You cannot edit a profile pass in a toolpath without using a toolpath preview. Make sure that the **Create 2D Preview** is always selected ☑ when you create a toolpath.

The 2D toolpaths that involve profile passes are **Profiling**, **Bevel Carving**, **Machine Vectors**, **Female Inlay** (Female – Hole) and **Male Insert** (Male – Straight). For details, see "2D Toolpath Strategies" on page 337.

The only 3D toolpaths that involves profile passes is **3D Cut Out**. For details, see "3D Toolpath Strategies" on page 408.

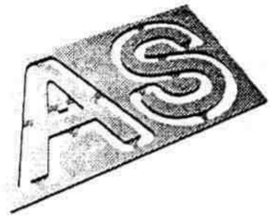
To display the **Profile Options** in the **Assistant** window:

1. Select the toolpath preview containing the profile pass that you want to edit. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button in the **2D Toolpaths** area.

There are three options available for editing a profile pass on the **Profile Options** page, but the settings for only one of these options can be displayed on the page at any one time:

- **Bridges** – This option allows you to add bridges, or tabs, to the selected profile pass. For details, see "Adding Bridging" on page 450.
- **Start Point** – This option allows you to change the position of the start point in the selected profile pass. For details, see "Changing the Start Position" on page 454.
- **Leads** – This option allows you to make several copies of a selected toolpath in a circular pattern using a user-defined centre of rotation. For details, see "Adding Lead Moves" on page 455.

Adding Bridging



Bridging is a precautionary measure to prevent a profiled vector object from shifting in the block of material as it is machined.

The cutting tool used to profile vector objects lifts slightly in the Z direction about the position of each bridge, leaving small tabs that hold the vector object in position during machining. The machined vector object can then be gently snapped out from the block of material.

You can add bridges to any profile pass within a calculated toolpath, provided that a **2D Preview** was created during the calculation process.

The toolpaths that involve profile passes are **Profiling**, **Bevel Carving**, **Machine Along Vector**, and **Inlay** (Female – Hole and Male – Straight).

To add bridging to a profile pass:

1. Select the toolpath preview containing the profile pass that you want to edit. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button in the **2D Toolpaths** area to display the **Profile Options** page.

Note: If you click on the **Profile Options** button with a toolpath preview already selected, the bridging settings are displayed by default.

---

4. Click on the **Bridges** option to display the bridging settings on the page.
5. Define the length of each bridge in the **Bridge Length** box.
6. Define the thickness of each bridge in the Z direction in the **Bridge Thickness** box.
7. To set how the bridges are added to the selected profile pass, click on either of the **Add Bridges To Profiles** radio buttons :

- **Constant Number** – Select this option to place a specific number of evenly spaced bridges on the selected profile pass. If you select this option, define the total number of bridges in the **Number** box.
- **Constant Spacing** – Select this option to place the bridges at a specific distance apart from one another on the selected profile pass.

  If you select this option, first define the distance you want between bridges, from the centre of each bridge, in the **Distance** box.

  Now type the minimum number of bridges you want on the selected profile pass, regardless of the distance you have set between them, in the **Min. Number** box.

  You can also specify the maximum number of bridges you want to create using the **Max. Number** box.

8. Click on the **Create Bridges** button to add the bridges to the selected profile pass.
9. Click on the **Close** button to return to the **Assistant**'s Home page.

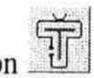
Editing Bridging

You can edit the bridging that you have added to a profile pass in several ways. You can:

- Insert new bridges. For details, see "Inserting a Bridge" on page 451.
- Delete an individual or all existing bridges. For details, see "Deleting Bridges" on page 452.
- Change the length of individual or all existing bridges. For details, see "Changing " on page 453.
- Move bridges into a new position on the profile pass. For details, see "Moving a Bridge" on page 453.

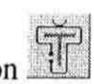
Inserting a Bridge

To add an individual bridge to any place in a selected profile pass:

1. Select the preview of the profile pass to which you want to add a bridge. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab  to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button  in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. Move the cursor over the position in the profile pass in which you want to insert the new bridge's centre-point.
5. Press the **I** key on your keyboard or double-click the mouse to insert the new bridge. The new bridge is the same length and thickness as the existing bridges on the selected profile pass.

Deleting Bridges

To delete any or all of the existing bridges on a selected profile pass:

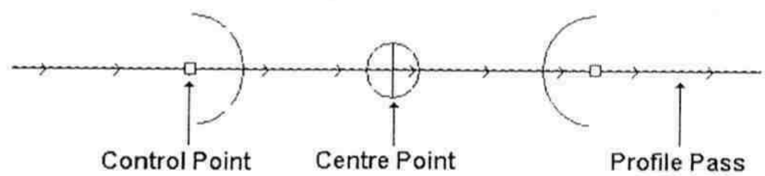
1. Select the preview of the profile pass containing the bridge(s) that you want to delete. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab  to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button  in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. If you want to delete an individual bridge, move the cursor over the bridge you are removing, then use either of these methods:
    - Press the **D** key on your keyboard. You must have clicked at least once in the **2D View** window before you can do this.
    - Double-click on the bridge itself.

   If you want to delete all of the existing bridges on the selected profile pass, click on the **Delete All Bridges** button.

Changing the Length of a Bridge

To change the length of any or all of the existing bridges on a selected profile pass:

1. Select the preview of the profile pass containing the bridge(s) that you want to extend or reduce. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. Use the **Zoom In Tool** to focus on the bridge that you want to extend or reduce, and then click to select its centre-point. For details on using the **Zoom In Tool**, see "Zoom In Tool" in the ArtCAM Pro Layout chapter. The bridge is blue with a control point at each end:



5. Click and drag either of its control points to set the bridge length.
6. If you want all existing bridges in the selected profile pass to share the new length of this bridge, click on the **Update All Bridges** button.

Moving a Bridge

To change the position of an individual bridge on a selected profile pass:

1. Select the preview of the profile pass containing the bridge(s) that you want to move. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.

3. Click on the **Profile Options** button in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. Use the **Zoom In Tool** to focus on the bridge that you want to move, and then click and drag on its centre-point to move it along the selected profile pass into its new position.. For details on using the **Zoom In Tool**, see "Zoom In Tool" in the ArtCAM Pro Layout chapter.

Changing the Start Position

The start position in a profile pass determines the place at which the cutting tool enters the block of material when you are machining a model. A circle marking the position of the current start position is drawn on the toolpath preview. For details, see "Changing the Start Position" on page 454.

To change the start position relative to a defined reference point:

1. Select the preview of the profile pass in which you want to change the start position. For details, see "Selecting Toolpaths" on page 438. The circle drawn on the toolpath preview marking the current start position turns blue.
2. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. Click on the **Start Point** option to display the Start Point settings on the page.
5. In the **Reference Point** area, click on one of the four radio buttons to define the reference point by which you want to reposition the start point:
    - **Centre of Gravity** – This option sets the reference point in the main mass of the vector object from which the selected toolpath was created.
    - **Centre of Bounding Box** – This option sets the reference point in the centre of the selected profile pass, as defined by the bounding box that now surrounds it.

- **Highest or Lowest X or Y Value** – This option sets the reference point as the position in the selected profile pass with the highest or lowest X or Y value, depending on its cutting direction.

  Closest to Corners or Mid of Bounding Box – This option sets the reference point as close as possible to one of the four corners in the bounding box that now surrounds the selected profile pass, its centre, or one of the four mid-points in its sides.

   Depending on which option you select, a number of radio buttons are displayed in the **Position** area indicating the positions available for the start point in the selected profile pass, relative to its reference point.

6. Reposition the start point in the selected profile pass. You can either:

   - Click on one of the available radio buttons in the **Position** area of the page.
   - Click and drag on the circle marking the current start position in the selected toolpath preview shown in the **2D View** window.

Note: Any lead moves added to a profile pass are also moved along with the start point, although the distance between them remains the same.

7. Click on the **Apply** button to set the new position of the Start Point.
8. Click on the **Close** button to return to the **Assistant**'s Home page.

You can set any point (node) in a selected vector object as the Start Point. For further information, see "Changing the Start Point" in the Working with Vectors chapter. This defines the Start Point position in a toolpath when it is first created.

Adding Lead Moves

Lead moves are a precautionary measure to prevent dwell marks from appearing on a model at the point at which the tool enters and leaves the block of material when machining a profile pass.

Instead of plunging into the block of material, machining the profile pass and then retracting from it, the tool enters and leaves the block at a specified distance from the start/end of the profile pass. This is either the Start Point in the vector object from which the profile pass was calculated, or in the centre of its longest span.

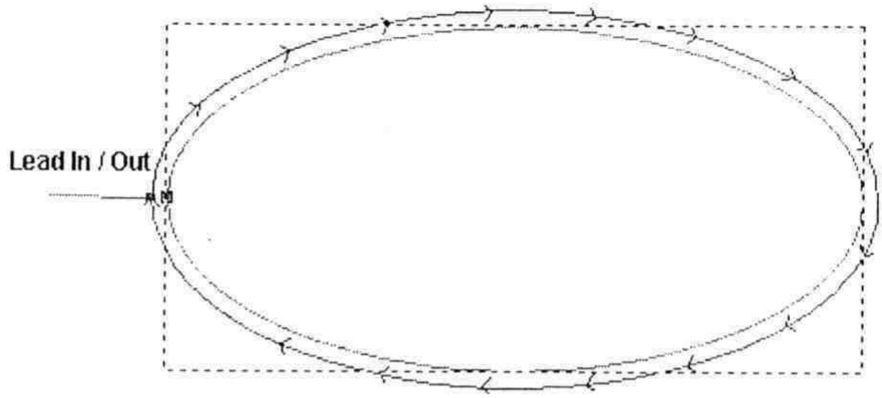
You can add lead moves to any profile pass within a calculated toolpath, provided that a **2D Preview** was created during the calculation process.

The toolpaths that involve profile passes are **Profiling**, **Bevel Carving**, **Machine Along Vector**, and **Inlay** (Female – Hole and Male – Straight).

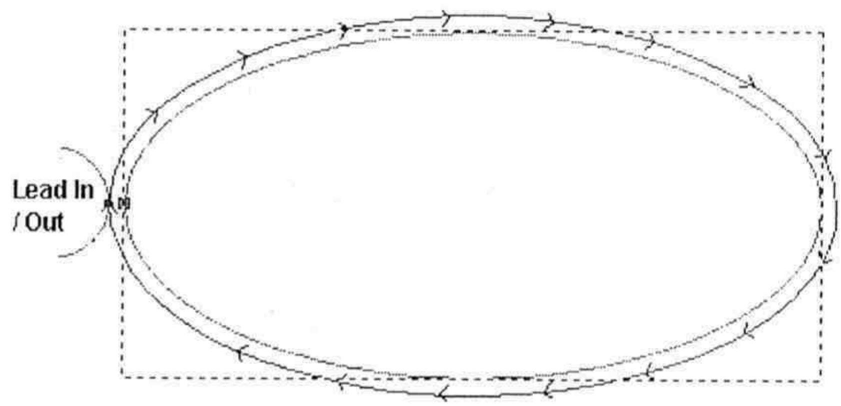
Note: You can add lead moves to a Profiling or 3D Cut Out toolpath when creating the toolpath itself. For details, see "Profiling" on page 338 and "3D Cut Out" on page 428.

To add lead moves to a profile pass:

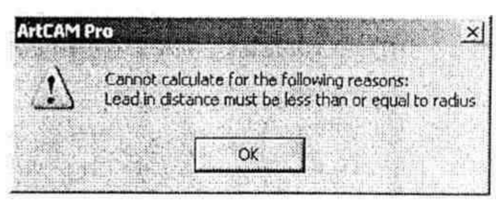
1. Select the toolpath preview containing the profile pass to which you want to add lead moves. For details, see "Selecting Toolpaths" on page 438.
2. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
3. Click on the **Profile Options** button in the **2D Toolpaths** area to display the **Profile Options** page. The bridging settings are displayed on the page.
4. Click on the **Leads** option to display the lead move settings on the page.
5. If you do want to add a lead-out move to the profile pass, click to select the **Do not Lead Out** option.
6. Type the distance from the profile pass that you want the tool to cut into and retract from the block of material in the **Distance (D)** box.
7. Type the distance from the start/end point in the profile pass that you want the tool to machine over in the **Over Cut (O)** box. This further helps to create a smooth finish.
8. Select how you want the tool to move using either of the radio buttons :
    - **Linear** – Click on the **Linear** option to instruct the tool to lead into and out of the vector object's boundary in a straight-line motion:



- **Circular Arc** – Click on the **Circular Arc** option to instruct the tool to lead into and out of the vector object's boundary in an arc motion:



If you select the **Circular Arc** option, define the radius of the arc in the **Radius (R)** box. The radius must be less than or equal to the value in the **Distance (D)** box, otherwise the following message box appears when you calculate the toolpath:



Click on the **OK** button to close the message box.

9. Click on the **Apply** button to add the lead moves.
10. Click on the **Close** button to return to the **Assistant**'s Home page.

Setting the Machining Order

When machining a toolpath that is made up of more than one profile pass, you can set the order in which each of them are machined.

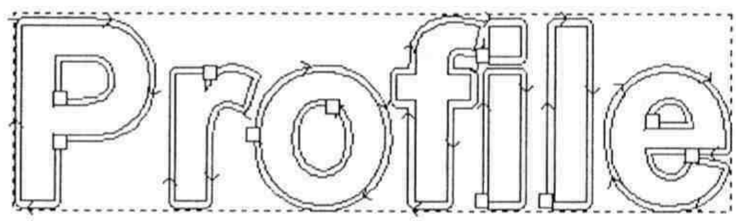
You can set the machining order in a calculated toolpath, provided that it contains more than one profile pass and a **2D Preview** was created during the calculation process.

The toolpaths that involve profile passes are **2D Profiling**, **Bevel Carving**, **Machine Along Vector** and **Inlay** (Female – Hole and Male – Straight).

To set the order in which the profile passes in a toolpath are machined:

1. Select the preview(s) of the profile pass(es) for which you want to set the machining order. For details, see "Selecting Toolpaths" on page 438.

   For example, selecting the Profiling toolpath around *Profile* vector text appears as follows:

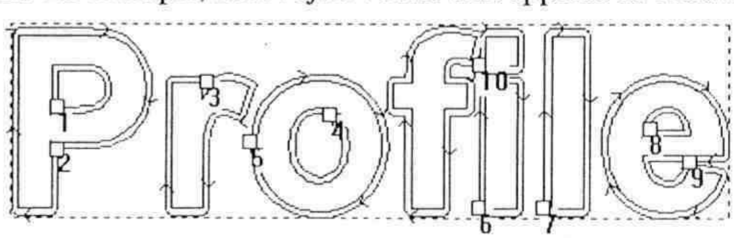
   

2. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.

3. Click on the **Toolpath Ordering** button in the **2D Toolpaths** area to display the **Toolpath Ordering** page.

   Each of the profile passes within the selected toolpath is automatically numbered at its Start Point.

   In our example, the *Profile* vector text appears as follows:

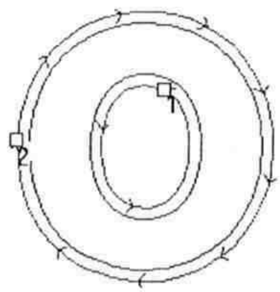
   

4. Click on each of the profile passes in the toolpath in turn to set the order in which they are machined.

The cursor shows the number that is to be given to the next profile pass that you click on.

The position of each profile pass in the overall machining order is also displayed in the **Segment Position** box as you click on it.

In our example, if we click on the profile pass inside of the central cavity of the letter *o* in the *Profile* vector text first, and then on the profile pass around the outside of the letter, they are numbered as 1 and 2 respectively:



Note: It is recommended that you machine the profile pass inside of a central cavity before the profile pass around the outside of the selected text.

5. Click on the **Close** button to return to the **Toolpaths** Home page.

Resetting the Machining Order

You can correct any mistakes you make when setting the machining order of the profile passes in a selected toolpath:

1. Click on the **Reset Count** button.
2. Click on the profile pass that you want to set as the first to be machined.
3. Continue this process until each profile pass in the toolpath is numbered accordingly.

Manipulating the Machining Order

You can manipulate the order in which the profile passes are machined in a selected toolpath by using the options in the **Segment Ordering** area of the **Toolpath Ordering** page:

- Click on the **Order segments from start** radio button to machine the numbered profile passes in sequential order.

The cursor shows the number that is to be given to the next profile pass that you click on, counting upwards from 1.

- Click on the **Order segments from end** radio button to machine the numbered profile passes in reverse order.

  The cursor shows the number that is to be given to the next profile pass that you click on, starting with the number equal to the total number of profile passes in the toolpath and counting down.

- Click on the **Swap segments** radio button to swap the position in the machining order of the first profile pass you click on with that of the second you click on.

  The cursor shows two question marks. After you click on the profile pass that you want to swap with another, its number is shown on the cursor. For example, if you click on the first segment, the cursor is shown as. After you click on the profile pass that you want to swap with that which is already selected, the cursor is shown as.

Adjusting the Machining Parameters of a Tool

You can adjust the machining parameters for CNC machine tools in ArtCAM Pro as follows:

- When creating a toolpath, in the toolpath's page. For example, the **Profiling** page. For details, see "Adjusting Machining Parameters When Creating a Toolpath" on page 461.
- After creating a toolpath, using the **Toolpaths** Home page. For details, see "Adjusting Machining Parameters in a Created Toolpath" on page 462.

Note: You can also change the tool's default machining parameters stored in the **Tool Groups Database**. This method should only be used before selecting a toolpath. For details, see "Editing a Tool" on page 476.

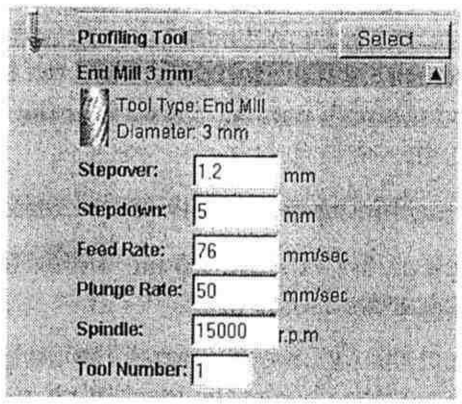
Adjusting Machining Parameters When Creating a Toolpath

You can change the machining parameters of the CNC machine tool used to machine a model, when creating a toolpath.

To change the machining parameters of a CNC machine tool:

1. With the toolpath page displayed in the **Assistant** window, display the machining parameters of the selected tool by clicking on the arrow in the tool area of the page.

   For example, if you have selected a 3mm End Mill tool in a Profiling toolpath, the tool area of the **Profiling** page looks like this:

2. You can now set the new machining parameters:

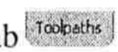
   - To change the stepover value of the selected tool, type a value in the **Stepover** box. The **Stepover** value defines the distance between adjacent machining passes.
   - To change the maximum cutting depth of the selected tool, type a value in the **Stepdown** box. The **Stepdown** value generates multiple machining passes.
   - To change the feed rate of the selected tool, type a value in the **Feed Rate** box. The **Feed Rate** value defines the rate at which the tool moves in relation to the block of material.
   - To change the plunge rate of the selected tool, type a value in the **Plunge Rate** box. The **Plunge Rate** value defines the rate at which the tool moves in the Z direction and plunges into the block of material.

- To change the rotational speed of the spindle, type a value in the **Spindle** box. The spindle is the part of the machine tool that rotates during operation. On a mill it holds the tool in position. On a lathe it holds the block of material.
- To give the selected tool a number, click on the **Tool Number** list box and then on the number you want to give to the tool. This number should correspond with the position of the tool in the CNC machine's tool changer.

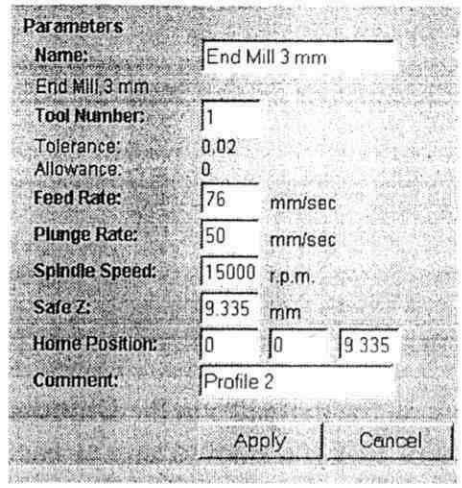
Adjusting Machining Parameters in a Created Toolpath

When you have created a toolpath, you can adjust the machining parameters, except for the **Stepdown** and the **Stepover**. In addition, you can amend toolpath parameters such as the **Home Position** and the **Safe Z** level.

To adjust the machining or toolpath parameters:

1. Click on the **Toolpaths** tab  to display the **Toolpaths** Home page.
2. Select the tool listed beneath the toolpath name for which you want to change the machining parameters. For details, see "Selecting Toolpaths" on page 438.
3. Click on the **Edit Parameters** button to display the machining parameters for the selected tool.

   For example, if you selected a 3mm End Mill tool when creating a Profiling toolpath, the **Parameters** area looks like this:



4. You can change the machining parameters in this area as described in "Adjusting Machining Parameters When Creating a Toolpath" on page 461. In addition, you can amend:
   - The tool's name.
   - The height, or **Safe Z** level, at which the cutting tool makes rapid moves between toolpath segments.
   - The **Home Position** (XYZ) of the tool.
   - The comment about the toolpath.
5. Click on the **Apply** button.

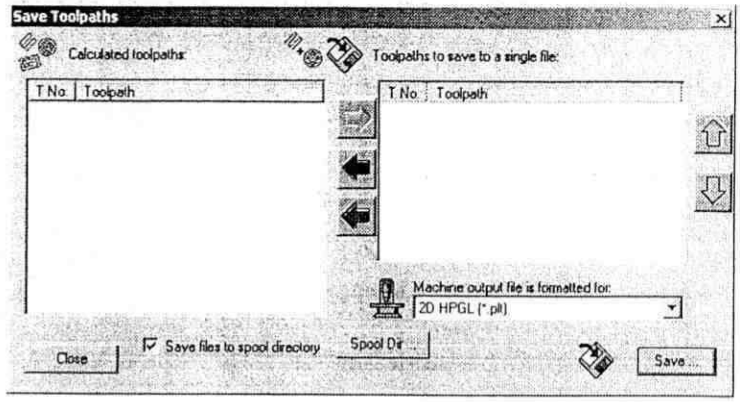
Saving a Toolpath

When you have created a toolpath, it can be saved in two formats:

- You can save the toolpaths along with the model you are currently working on as a model file (***.art**). For details, see "Saving a Model" in the Working with Models chapter.
- You can export the toolpath data as a file specific to your CNC machine tool, as explained below.

To save the toolpath data as a machine-specific file:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Save Toolpaths** button in the **Toolpath Operations** area to open the **Save Toolpaths** dialog box:



3. Click on the toolpaths in the **Calculated toolpaths** window that you want to save as one machine-specific toolpath file. To select more than one toolpath, hold the **Shift** key down on your keyboard as you click on each of the toolpaths.

4. Click on the right button to transfer your selected toolpaths to the **Toolpaths to save to a single file** window.

5. Make sure that the toolpaths are in the correct order.

   You can use the up and down buttons in the right of the dialog box to set the order of the toolpaths. Each click, up or down, moves the selected toolpath one position in the list.

   If you want to alter the selection, you can:

   - Click on the black left button to transfer files back to the **Calculated toolpaths** window.
   - Click on the red left button to transfer all files back to the **Calculated toolpaths** window.

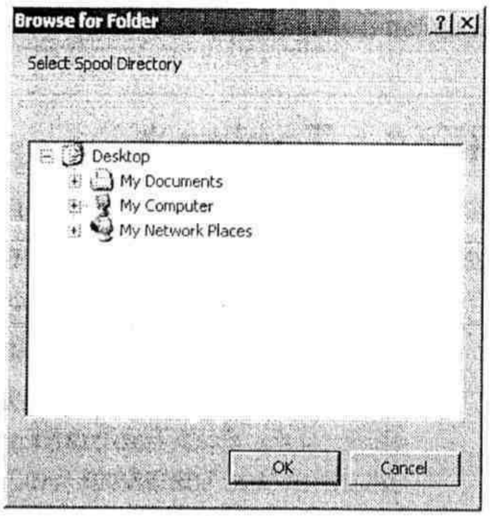
6. Click on the **Machine output file is formatted for** list box, and then on the relevant machine format to select it.

7. If you want to save the machine-specific toolpath file directly to the spool directory:

   - Make sure that the **Save Files to Spool Directory** option is selected.

Note: If you have already selected the spool directory using ArtSpool, the path of the spool directory is shown.

- Click on the **Spool Dir...** button to display the **Browse For Folder** dialog box:



- Click on the folder to which you want to spool your toolpath files.
- Click on the **OK** button to select the spool directory and close the **Browse for Folder** dialog box.

8. Click on the **Save** button.

   If the **Save Files to Spool Directory** option is selected ☑, the **Save As** dialog box appears with the current spool directory selected.

   If the **Save Files to Spool Directory** option is deselected ☐, the **Save As** dialog box is displayed:

   - First, click on the **Save In** list box and select the directory you want to save the machine-specific toolpath file in.
   - Next, type the file name for the machine-specific toolpath in the **File name** box.
   - Now click on the **Save as type** list box, and then on the file type in which you want to save your machine-specific toolpath.
   - Finally, click on the **Save** button to close the **Save As** dialog box.

9. Click on the **Close** button in the **Save Toolpaths** dialog box.

Note: You can also save a toolpath from either the **2D View** or the **3D View** in the same way. From the Main menu bar, click on the **Toolpaths** menu, followed by the **Save Toolpath** option to display the **Save Toolpaths** dialog box. For details on using the **Save Toolpaths** dialog box, return to the start of this section.

Editing a Toolpath

You can change the settings in any toolpath that you have created.

To edit the settings in a toolpath using the **2D View** window:

1. Select the preview of the toolpath that you want to edit. For details, see "Selecting Toolpaths" on page 438.
2. Right-click to display the Toolpath Editing menu, and then click on the **Edit Toolpath** option to display the toolpath's details in the **Assistant** window.

For example, if you select a Profiling toolpath and then the **Edit Toolpath** option from the Toolpath Editing menu, the **Profiling** page is displayed in the **Assistant** window.

Alternatively, you can edit toolpath settings using the **Toolpaths** Home page:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the toolpath listed that you want to edit. For details, see "Selecting Toolpaths" on page 438.
3. Click on the **Edit Toolpaths** button in the **Toolpath Operations** area to display the page for the toolpath in the **Assistant** window.

Deleting Toolpaths

You can delete a toolpath or toolpath group that you have created.

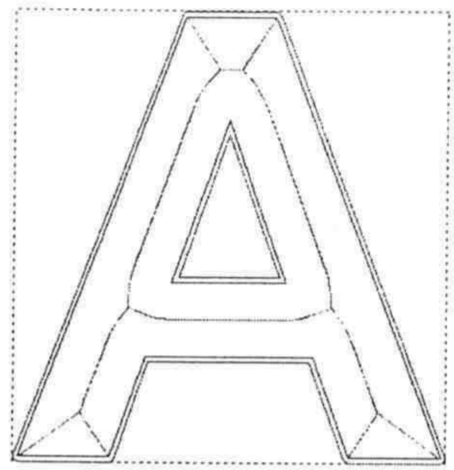
A toolpath group is a toolpath that requires the use of more than one machining tool. For example, a Bevel Carving toolpath uses both a Carving tool and a Profiling tool.

Deleting a Toolpath

To delete a toolpath from the **2D View** window:

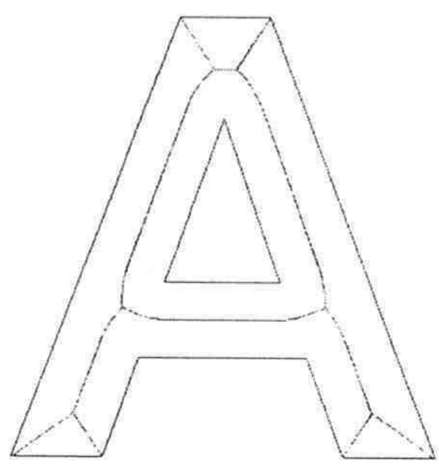
1. Select the preview of the toolpath that you want to delete to select it. For details, see "Selecting Toolpaths" on page 438.

   For example, if you want to delete the profile pass in a Bevelled Carving toolpath it would appear something like this when selected:

2. Right-click to display the Toolpath Editing menu, and then click on the **Delete Toolpath** option.

   In our example, you can see that the profile pass in the Bevelled Carving toolpath is now deleted:

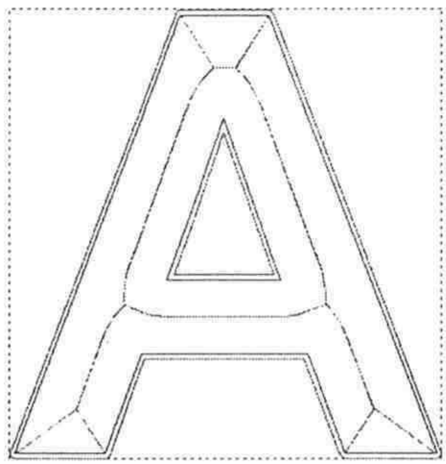
Deleting a Toolpath Group

To delete a toolpath group using the **2D View** window:

1. Select a preview toolpath that is part of a toolpath group to select it. For details, see "Selecting Toolpaths" on page 438.

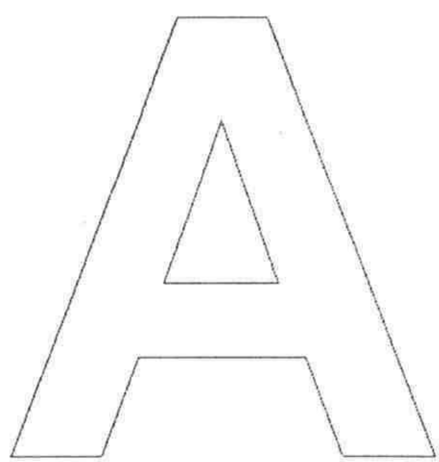
The selected part of the toolpath group is blue and surrounded by a bounding box.

For example, if you want to delete a Bevel Carving toolpath, and select the profile pass in the toolpath, the toolpath looks like this when selected:



2. Right-click to display the Toolpath Editing menu, and then click on the **Delete Toolpath Group** option.

   In our example, you can see that both the centreline and the profile pass in the Bevelled Carving toolpath group are now deleted:



Note: You can also delete a toolpath using the **Toolpaths** Home page. Click to select the toolpath group listed, for example Bevelled Carving, and then on the **Delete Toolpath** button in the **Toolpath Operations** area.

Calculating a Single Toolpath

You can calculate a single toolpath during its creation using the **Now** button on the page corresponding to it. For example, when the **Profiling** page is shown in the **Assistant** window. If you had chosen to calculate a toolpath **Later**, you can calculate it at a later time using the **Toolpaths** Home page. For details, see any of the toolpaths listed in “2D Toolpath Strategies” on page 337 and “3D Toolpath Strategies” on page 408.

All toolpaths listed on the **Toolpaths** Home page in red have not yet been calculated. All toolpaths listed in black have already been calculated.

To calculate a single toolpath:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Select the toolpath that you want to calculate. For details, see "Selecting Toolpaths" on page 438.
3. Click on the **Calculate Selected Toolpath** button in the **Toolpath Operations** area.

   The page relating to the selected toolpath appears in the **Assistant** window and the calculation process begins. The page shown in the **Assistant** window closes when the calculation process is complete.

Calculating a Batch of Toolpaths

You can calculate a sequence of toolpaths at once using ArtCAM Pro’s batch calculation process. This gives you the freedom to work continuously on creating a model during the day, and calculate all of its corresponding toolpaths during the night.

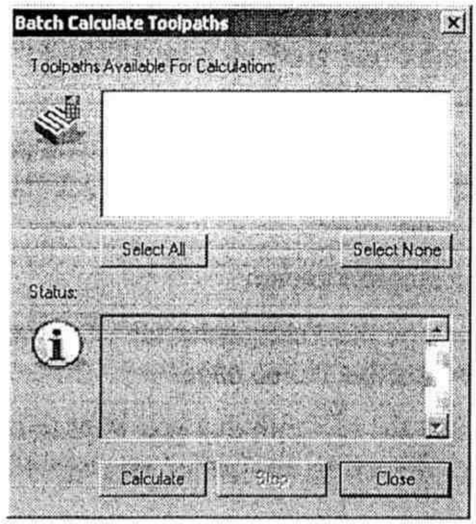
If you had chosen to calculate several toolpaths **Later** during their creation, you can calculate them in a batch sequence using the **Toolpaths** Home page. For details, see any of the toolpaths listed in “2D Toolpath Strategies” on page 337 and “3D Toolpath Strategies” on page 408.

All toolpaths listed on the **Toolpaths** Home page in red have not yet been calculated. All toolpaths listed in black have already been calculated.

To calculate a batch of toolpaths:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.

2. Click on the **Batch Calculate Toolpaths** button in the **Toolpath Operations** area to display the **Batch Calculate Toolpaths** dialog box:



3. From the **Toolpaths Available For Calculation** window, make sure that all of the toolpaths that you want to calculate as part of this batch are selected. All of the available toolpaths are selected by default.

    - If you click on a selected toolpath it is then deselected, and vice versa.
    - You can deselect all of the available toolpaths by clicking on the **Select None** button.
    - To select all of the available toolpaths, click on the **Select All** button.

4. Click on the **Calculate** button to begin the batch calculation process. The following details are displayed in the **Status** window during the batch calculation process:

    - The time and date on which the batch calculation process commences.
    - The elapsed time for the calculation of each of the toolpaths within the batch.
    - The elapsed time for the entire batch calculation process.

5. Click on the **Close** button to close the **Batch Calculate Toolpaths** dialog box.

Using a Toolpath Summary

You can review, print or save details of all of the toolpaths that you have created in order to machine a model.

The **Toolpath Summary Information** dialog box displays the following details:

- Job dimensions used.
- Toolpaths used.
- Machining parameters of all tools used.

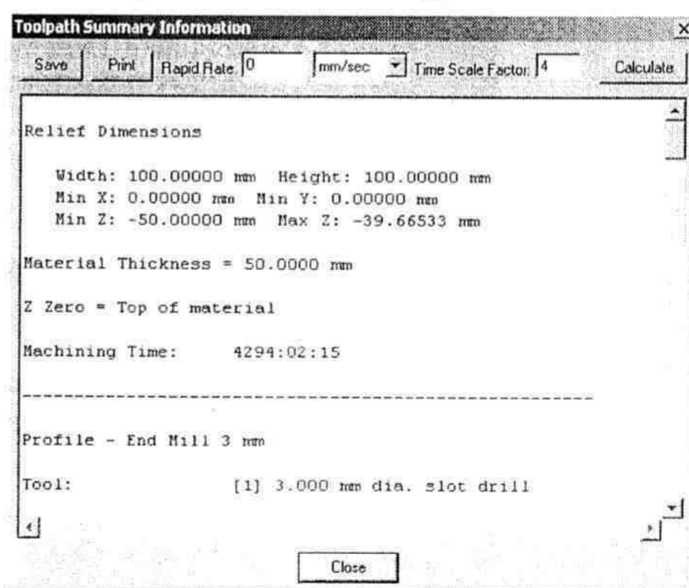
A total machining cycle time is also calculated based on the Feed Rates of the various tools that you had selected when creating a toolpath.

To display the **Toolpath Summary Information** dialog box for an open model:

1. Click on the **Toolpaths** tab 

to display the **Toolpaths** Home page.

2. Click on the **Toolpath Summary** button  in the **Toolpath Operations** area to display the **Toolpath Summary Information** dialog box:



You can use the scrollbar at the right of the **Toolpath Summary Information** dialog box to view all of the toolpath information.

Calculating the Estimated Machining Time

You can amend the machining parameters to estimate the actual machining time for a job.

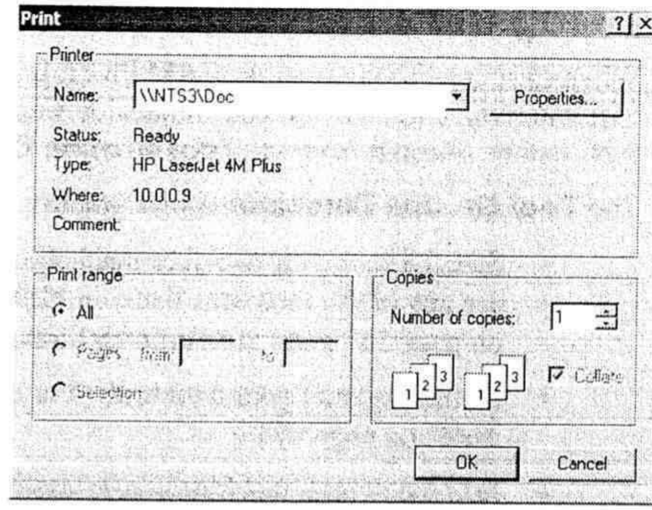
To calculate the estimated machining time of a toolpath:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Summary** button in the **Toolpath Operations** area to display the **Toolpath Summary Information** dialog box.
3. Type a value in the **Rapid Rate** box. You should confirm the maximum Rapid Rate of your CNC machine by consulting the manufacturer's manual. If the manual contains no reference to the maximum Rapid Rate, use the maximum Feed Rate value.
4. Click on the list box and select the appropriate unit of speed.
5. Type a value in the **Time Scale Factor** box. The default time scale factor is 4. You should experiment with finding the appropriate value for your machining.
6. Click on the **Calculate** button.
7. Click on the **Close** button to close the **Toolpath Summary Information** dialog box.

Printing a Toolpath Summary

To print a toolpath summary:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Toolpath Summary** button in the **Toolpath Operations** area to display the **Toolpath Summary Information** dialog box.
3. Click on the **Print** button to display the **Print** dialog box:



4. Click on the **Name** list box and click on the printer you want to use.
5. Click on the **OK** button to print the toolpath summary at the selected printer.
6. Click on the **Close** button to close the **Toolpath Summary Information** dialog box.

Saving a Toolpath Summary

To save a toolpath summary:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Toolpath Summary** button in the **Toolpath Operations** area to display the **Toolpath Summary Information** dialog box:
3. Click on the **Save** button to display the **Save Toolpath Summary** dialog box.
4. Click on the **Save In** list box and select the directory where you want to save the toolpath summary.
5. Type the file name you want to use for the toolpath summary in the **File name** box.
6. Click on the **Save** button. The file is saved as a text file (***.txt**).
7. Click on the **Close** button to close the **Toolpath Summary Information** dialog box.

Using the Tool Groups Database

When creating a toolpath in ArtCAM Pro, it is necessary to select a tool with which to machine your model. A broad range of pre-defined tools can be selected from the **Tool Groups Database**.

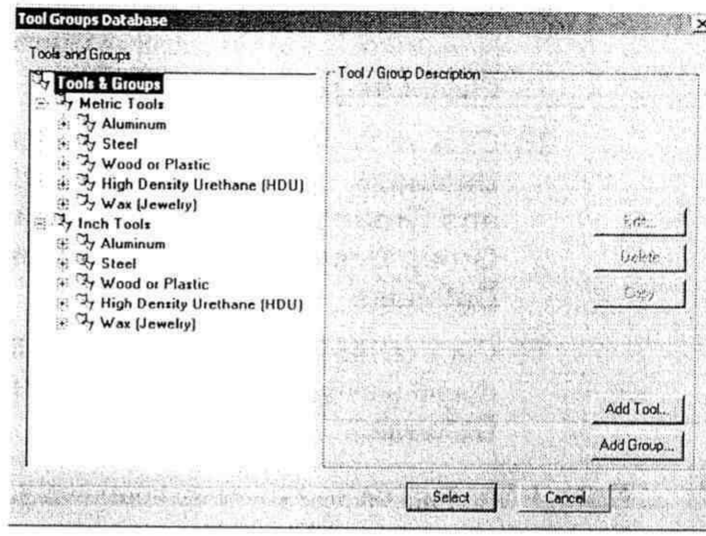
The **Tool Groups Database** allows you to:

- Select a tool with which to machine a toolpath. For details, see any of the toolpaths listed in "2D Toolpath Strategies" on page 337 and "3D Toolpath Strategies" on page 408.
- Define and add your own tools. For details, see "Adding a Tool" on page 474.
- Edit the default machining parameters of any selected tool. For details, see "Editing a Tool" on page 476.
- Delete any of the listed tools. For details, see "Deleting a Tool" on page 479.
- Group tools together, as you prefer. For details, see "Adding a Tool Group" on page 479.
- Edit the name and notes for a tool group. For details, see "Editing a Tool Group" on page 480.

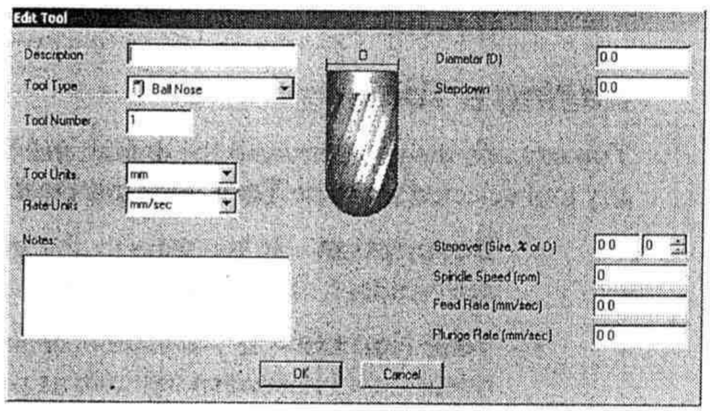
Adding a Tool

To define and add a new tool to the **Tool Groups Database**:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Tool Database** button in the **Toolpath Operations** area to open the **Tool Groups Database**:



3. Click on the **Add Tool** button to open the **Edit Tool** dialog box:



4. Type a name for the tool in the **Description** box.
5. Click on the **Tool Type** list box, and then click to select one of the tool types listed. An image representing the selected **Tool Type** appears in the middle of the dialog box. Boxes to type in the necessary machining parameters of the tool appear in the right side of the dialog box.
6. Click on the **Tool Units** list box, and then click to select the appropriate unit of measurement.
7. Click on the **Rate Units** list box, and then click to select the appropriate unit of speed.
8. Type any relevant information concerning the practical use of the tool you are adding to the **Tool Groups Database** in the **Notes** box.

9. Type a value in all of the boxes relating to the machining parameters of the tool, such as **Diameter**, **Stepover** and **Flute Length**.
10. Click on the **OK** button to add the tool to the **Tool Groups Database**. The tool you have added appears in the **Tools and Groups** window, and its details appear in the **Tool / Group Description** area of the **Tool Groups Database**.
11. Click on the tool in the **Tools and Groups** window and drag it into the appropriate group in the **Tool Groups Database**.

Note: For details on how to set up your own group, see "Adding a Tool Group" on page 479.

---

12. Click on the **OK** button to close the **Tool Groups Database**.

Editing a Tool

You can edit the geometry and the default machining parameters of any tool selected from the **Tool Groups Database**:

- **Description** – If you want to change the name of the selected tool, type it in this box.
- **Tool Number** – If you want to change the number of the selected tool to correspond with its position on a tool changer, type it in this box.
- **Tool Units** – If you want to change the unit of measurement for the selected tool, click on the **Tool Units** list box and then on the unit of measurement you want to use.
- **Rate Units** – If you want to change the unit of speed for the selected tool, click on the **Rate Units** list box and then on the unit of speed you want to use.
- **Notes** – If you want to make any notes about the selected tool, type them in this box.
- **Diameter** – If you want to change the diameter of the selected tool, type it in this box.
- **Stepdown** – If you want to change the maximum cut depth of the selected tool, type it in this box.

- **Stepover** – If you want to change the distance between adjacent machining passes made by the selected tool, type it in this box.

  You can also adjust the stepover of the selected tool using the **% of** box and buttons:

  - Each click on ▲ increases the stepover of the selected tool by 1% of the value shown in the **Diameter** box.
  - Each click on ▼ decreases the stepover of the selected tool by 1% of the value shown in the **Diameter** box.
  - Type a value in the **% of** box to define the **Stepover** as a percentage of the tool **Diameter**.
- **Spindle Speed** – If you want to change the rotational speed of the spindle, type it in this box.
- **Feed Rate** – If you want to change the rate at which the tool moves in relation to the block of material, type it in this box.
- **Plunge Rate** – If you want to change the rate at which the tool moves in the Z direction and plunges into the block of material, type it in this box.

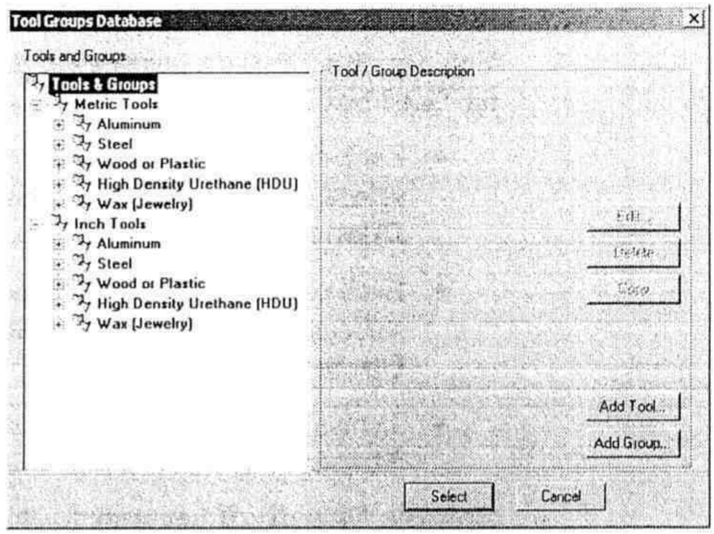
Editing the geometry of certain types of tools in the **Tool Groups Database** requires more data than with other tools. The additional information required includes:

- **Included Angle** – To change the angle of a V-Bit tool, type it in this box.
- **Half Angle** – To change the angle of a Conical tool, type it in this box.
- **Flat Radius** – To change the radius of a flat Conical tool, type it in this box.
- **Tip Radius** – To change the radius of a rounded Conical tool

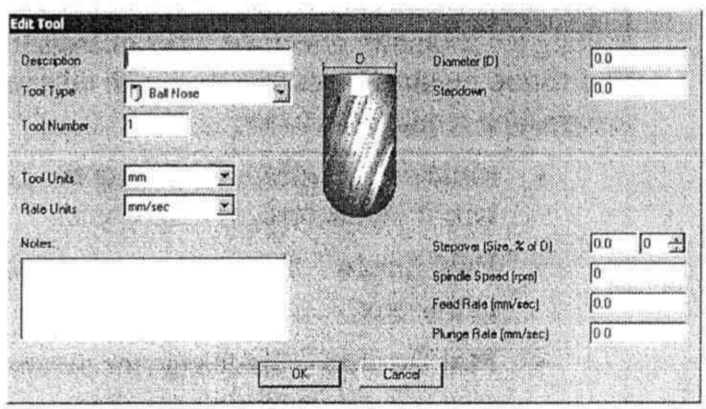
To edit the machining parameters or the geometry of a tool in the **Tool Groups Database**:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.

2. Click on the **Tool Database** button in the **Toolpath Operations** area to open the **Tool Groups Database**:



3. Click on the tool group in the **Tools and Groups** window that contains the tool you want to edit, and then on the tool itself. The selected tool is highlighted in blue and its machining parameters appear in the **Tool / Group Description** area of the dialog box.
4. Click on the **Edit** button to display the **Edit Tool** dialog box:



5. Type a new value in the appropriate box, or click on the appropriate list box option, to change the geometry and/or machining parameters of the selected tool.
6. Click on the **OK** button to save these new values and to return to the **Tool Groups Database**. The tool you have edited appears in the **Tools and Groups** window, and the information that you had typed in the **Edit Tool** dialog box

appears in the **Tool / Group Description** area of the **Tool Groups Database**.

7. Click on the **OK** button to close the **Tool Groups Database**.

Deleting a Tool

You can delete any tool listed in the **Tool Groups Database**:

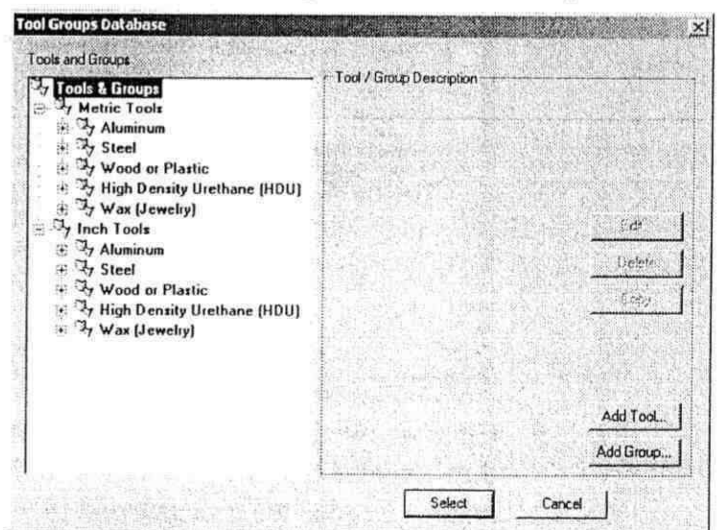
1. Click on the tool that you want to delete in the **Tools and Groups** window. The selected tool is highlighted in blue.
2. Click on the **Delete** button.

Adding a Tool Group

Within the **Tool Groups Database**, tools are organised into various groups for ease of reference. When you define and add a selection of tools to the **Tool Groups Database**, you may want to create a new group for them.

To create a new tool group:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Tool Database** button in the **Toolpath Operations** area to open the **Tool Groups Database**:



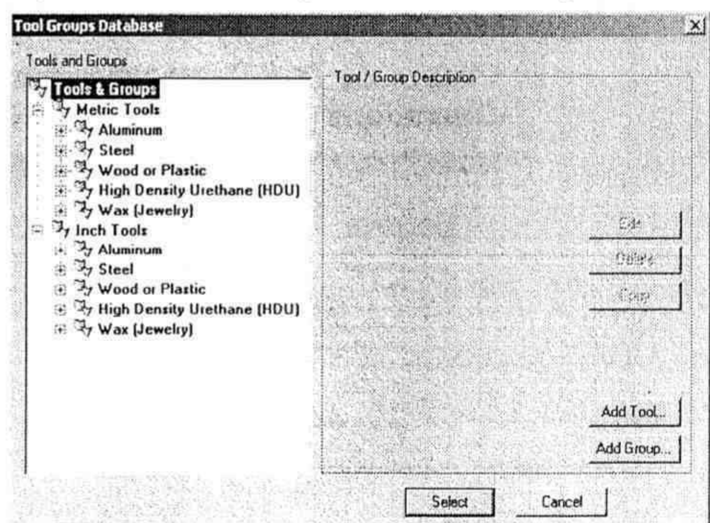
3. Click on the **Add Group** button. A folder named **New Group** appears in the **Tools and Groups** window:

4. Click on the **New Group** folder, and then type the name for the tool group you want to add to the database. You can now click and drag any tools listed in the **Tools and Groups** window into the tool group folder that you have created.
5. Click on the **OK** button to close the **Tool Groups Database**.

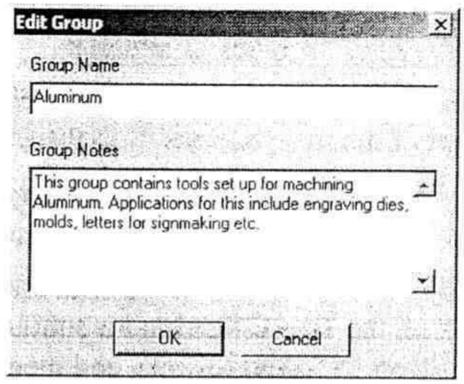
Editing a Tool Group

You can edit the geometry and the default machining parameters of any tool selected from the **Tool Groups Database**:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Tool Database** button in the **Toolpath Operations** area to open the **Tool Groups Database**:



3. Click on the tool group in the **Tools and Groups** window that contains the tool you want to edit, and then on the tool itself. The selected tool group is highlighted in blue and details about the group appear in the **Tool / Group Description** area of the dialog box.
4. Click on the **Edit** button to open the **Edit Group** dialog box:



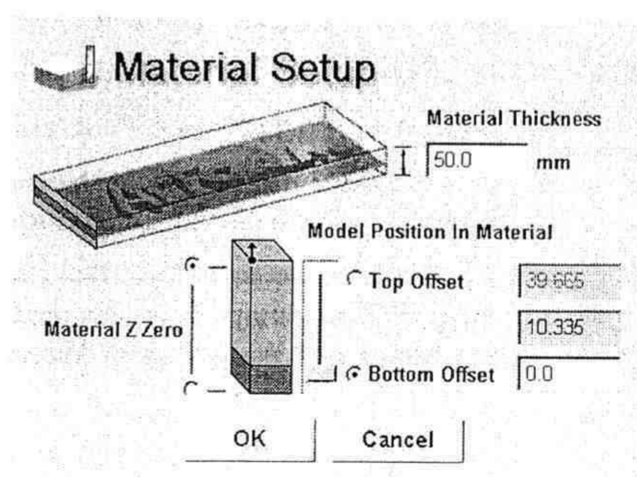
5. If you want to give the tool group a new name, type it in the **Group Name** box.
6. If you want to make any notes about the selected tool, type them in the **Notes** box.
7. Click on the **OK** button to close the **Edit Group** dialog box.

Adjusting the Material Setup

You can edit the thickness of a block of material and the position of a model within it. You can also change the origin in the block of material.

To adjust the material setup:

1. Click on the **Toolpaths** tab  to display the **Toolpaths** Home page.
2. Click on the **Material Setup** button  in the **Toolpath Operations** area to display the **Material Setup** dialog box:



3. Type a value in the **Material Thickness** box to define the depth (Z) of the block of material.
4. To set the Z zero position in the block of material, click on either of the radio buttons in the **Material Z Zero** area.

   The **Material Z Zero** icon appears on the top or the bottom of the block to mark the origin.
5. To specify the amount of material above the relief surface, click on the **Top Offset** radio button in the **Model Position In Material** area, and then either:
   - Type a value in the **Top Offset** box.
   - Click and drag on the slider to adjust the value in the **Top Offset** box.

   The value in the **Bottom Offset** box changes automatically as the value in the **Top Offset** box changes.

Note: The second box in the **Model Position In Material** area shows the current depth (Z) of the existing relief.

6. To specify the amount of material below the base of the relief, click on the **Bottom Offset** radio button in the **Model Position In Material** area then either:
   - Type a value in the **Bottom Offset** box.
   - Click and drag on the slider to adjust the value in the **Bottom Offset** box.

   The value in the **Top Offset** box changes automatically as the value in the **Bottom Offset** box changes.
7. Click on the **OK** button to close the **Material Setup** dialog box and save the settings.

You can also adjust the material set-up using the **Project** page:

1. Click on the **Project** tab Project to display the **Project** page.
2. With the **Machining** element shown, click on the material thickness value to display the **Material Setup** dialog box.

For information on how to use the **Material Setup** dialog box, return to the beginning of this section. For further details, see "Viewing Model Information" in the Working with Models chapter.

Deleting the Material

You can delete the material that you have defined using the **Material Setup** dialog box.

If you have changed the position and/or origin of a model using the **Material Setup** dialog box, neither the model nor its origin revert to their previous positions after the material has been deleted. You can reset the position of the model after the material has been deleted using either the **Set Position** dialog box or the **Material Setup** dialog box.

For details, see "Setting the Position of a Model" in the Working with Models chapter and "Adjusting the Material Setup" on page 481.

To delete the material:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Delete Material** button in the **Toolpath Operations** area.

The material is deleted from the **3D View** window and is no longer listed in the **Objects To Draw** list box. For details, see "Objects To Draw" in the ArtCAM Pro Layout chapter.

Creating a Toolpath Template

You can create a template from any toolpath that you have already created in ArtCAM Pro.

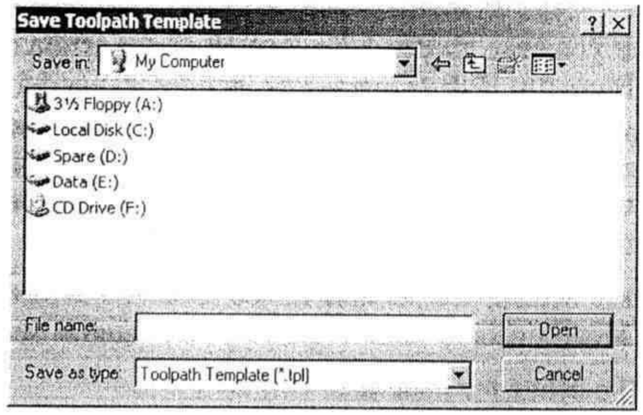
The toolpath template file (***.tpl**) contains all of the original settings that you had made when creating the toolpath.

This settings saved within the toolpath template can then be applied to a selected vector object in order to machine all or part of a model.

To create a toolpath template:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Select the toolpath listed on the **Toolpaths** Home page from which you want to create a template. For details, see "Selecting Toolpaths" on page 438.

3. Click on the **Save Toolpath As Template** button in the **Toolpath Operations** area to display the **Save Toolpath Template** dialog box:



4. Click on the **Save In** list box and select the directory you want to save the toolpath template in.
5. Type the file name you want to use for the toolpath template in the **File** name box.
6. Click on the **Save** button.

The toolpath template can now be loaded at any time to machine all or part of a model. For details, see "Loading a Toolpath Template" on page 484.

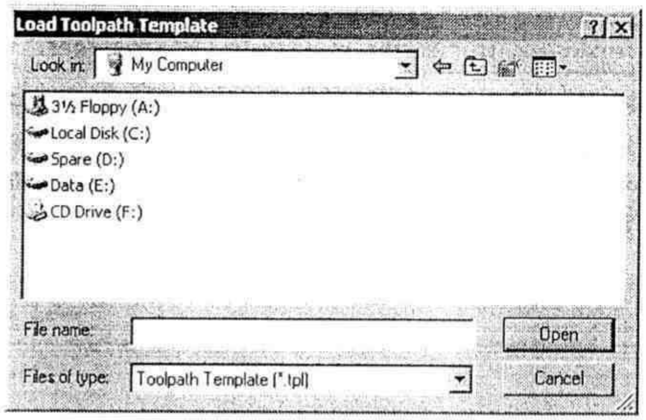
Loading a Toolpath Template

When you load a toolpath template, the toolpath settings within the file are automatically assigned to the vector object in the model from which the original toolpath that forms the template was created.

If the vector object from which the original toolpath was created has been deleted, or you want to use the template with a different model altogether, you need to select the vector object that you want to assign the toolpath template to before the toolpath can be edited or calculated.

To load a toolpath template file (***.tpl**):

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Click on the **Load Toolpath Template** button in the **Toolpath Operations** area to display the **Load Toolpath Template** dialog box:



3. Click on the **Look In** list box and select the directory that contains the toolpath template file you want to use.
4. Once you have found the toolpath template file, click on the file name listed in the main window of the **Load Toolpath Template** dialog box.

Note: You are only able to select toolpath template files (***.tpl**). You can confirm this by clicking on the **Files of Type** list box.

The name of the file you have selected appears in the **File Name** box.

5. Click on the **Open** button to list the toolpath on the **Toolpaths** Home page.

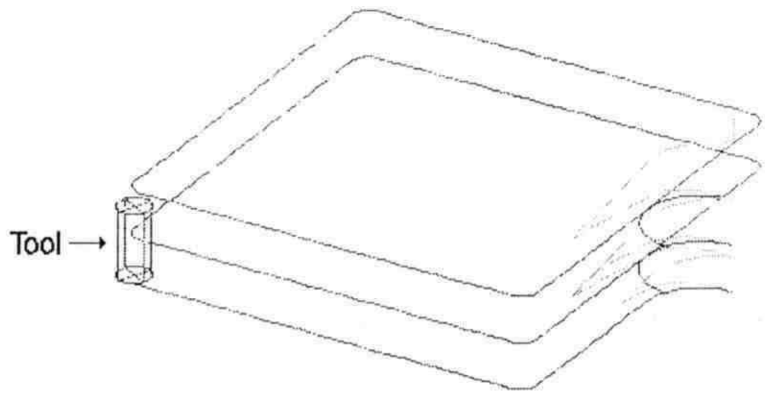
You can now edit or calculate the toolpath that has been loaded. For more details, see "Calculating a Single Toolpath" on page 469, "Calculating a Batch of Toolpaths" on page 469 and "Editing a Toolpath" on page 466.

Simulating Toolpaths

You can simulate a toolpath that has been calculated. This allows you to visualise the machining passes used to create a finished model. The name of a calculated toolpath is shown on the **Toolpaths** Home page in black. If it is shown in red, the toolpath has not yet been calculated and cannot be simulated.

A toolpath simulation is a more informative display of the toolpath than that offered by the dark red lines that make up a toolpath preview in the **2D View** window. In a toolpath simulation, you can see an image of the tool that you selected when creating the toolpath move gradually over an image of the block of material, creating a representation of the machined vector object.

As an aid to visualisation, you can see the tool used in a calculated toolpath move gradually over the block of material during a toolpath simulation. For example, a 3mm End Mill tool in a simulated profile pass is shown below:



You can simulate calculated toolpaths in three different ways:

- A single toolpath.
- A group of toolpaths. A toolpath group requires the use of more than one machining tool. For example, Bevelled Carving.
- All toolpaths.

To simulate a toolpath:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Select the toolpath listed on the **Toolpaths** Home page that you want to simulate. For details, see "Selecting Toolpaths" on page 438.

Warning: You can only simulate calculated toolpaths. These are listed on the **Toolpaths** Home page in black. Uncalculated toolpaths are listed in red.

3. Make sure that the machining parameters of the tool listed in the **Parameters** area are correct. If you want to adjust its parameters:
    - First, click on the **Edit Parameters** button to display all parameter boxes.

- Next, type the correct machining parameters in the appropriate boxes.
- Finally, click on the **Apply** button.

4. Make sure that the **Draw** option in the **Toolpath Simulation** area is selected.
5. You can now simulate the toolpath in either of the following ways:

   - Click on the **Simulate Toolpath Fast** button to display the **3D View** window and simulate the whole toolpath using the full capacity of your computer processor.
   - Click on the **Simulate Toolpath** button to display the **3D View** window and the **Simulation Control** panel:

If you click on the **Simulate Toolpath** button, use the playback buttons on the **Simulation Control** panel to simulate the toolpath:

- – Click on the **Pause Simulation** button if you want to pause the toolpath simulation.
- – Each click on the **Single Step Simulation** button simulates a successive tool move in the toolpath.
- – Click on the **Run Simulation at Normal Speed** button if you want to simulate the whole toolpath using only a percentage of your computer processor's capacity.
- – Each click on the **Run to Next Retract Move** button simulates the toolpath, pausing when the tool retracts from the material, using only a percentage of your computer processor's capacity.
- – Click on the **Run Simulation at Maximum Speed** button if you want to simulate the whole

toolpath using the full capacity of your computer processor.

- ⏭ – Each click on the **Run to Next Retract Move** button simulates the toolpath, pausing when the tool retracts from the material, using the full capacity of your computer processor.
- ⏏ – Click on the **Abort Simulation** button if you want to stop the toolpath simulation and close the **Simulation Control** panel.

Note: You can also simulate a toolpath by selecting a toolpath preview in the **2D View** window, right-clicking on the toolpath preview to display the Toolpath Editing menu and then and then clicking on the **Simulate Toolpath** option.

To simulate a group of toolpaths:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.

Warning: You can only simulate calculated toolpaths. These are listed on the **Toolpaths** Home page in black. Uncalculated toolpaths are listed in red.

2. Select any part of the calculated toolpath group that you want to simulate. For details, see "Selecting Toolpaths" on page 438.
3. From the Main menu bar, click on the **Toolpaths** menu, followed by the **Simulate Toolpath** option to display a representation of the calculated toolpath group and the machined vector object(s) in the **3D View** window.

Note: You can also simulate a toolpath group by selecting a toolpath preview in the **2D View** window that belongs to the group, right-clicking on the toolpath preview to display the Toolpath Editing menu and then and then clicking on the **Simulate Toolpath Group** option.

For information on how to use the **Material Setup** dialog box, return to the beginning of this section. For further details, see "Viewing Model Information" in the Working with Models chapter.

You can also simulate a group of toolpaths using the **Project** page:

1. Click on the **Project** tab to display the **Project** page.
2. With the **Machining** element shown, right-click on the toolpath name to display the context menu, and then click on the **Simulate Toolpath** option.

For further details, see "Viewing Model Information" in the Working with Models chapter.

To simulate all of the toolpaths you have calculated:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Simulate All Toolpaths** button to display a representation of the calculated toolpaths and the machined vector object(s) in the **3D View** window.

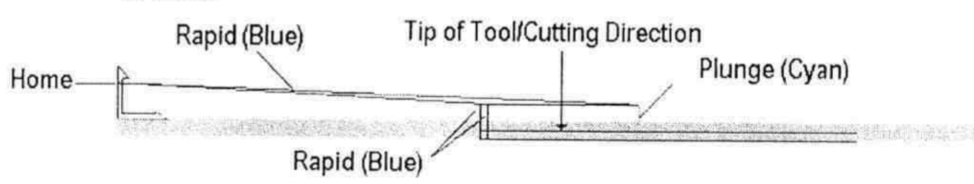
Note: You can also simulate all toolpaths by clicking on the **Toolpaths** menu in the Main menu bar, and then on the **Simulate All Toolpaths** option.

Viewing a Toolpath

You can view a toolpath you have created in both the **2D View** and **3D View** windows. A toolpath is displayed as a sequence of dark red lines in the **2D View** window, and a series of multi-coloured lines in the **3D View** window.

The colour in which a toolpath is displayed within the **3D View** window shows the movement the selected tool will make when machining the vector object.

The following example shows a toolpath simulation as seen along the Y-axis:



Rapid and Plunge Moves

The selected tool leaves the **Home** position, moves to the **Safe Z** level, then above and across the material surface until it reaches the point of entry. This movement is known as a **Rapid** move, and a blue line marks this.

The tool cuts into the material surface at its **Plunge** rate. A cyan line marks plunge moves.

Once the cutting is complete, the tool retracts from the block of material to the **Safe Z** level and then moves above and across the material surface until it reaches the **Home** position. This is another **Rapid** move marked by a blue line.

If a toolpath simulation contains several **Rapid** and **Plunge** moves, the lines representing the movement of the selected tool in the **3D View** window can become obscured. For this reason, you may want to hide them from view.

Rapid and **Plunge** moves are drawn in a toolpath simulation by default, but can be hidden in either of the following ways:

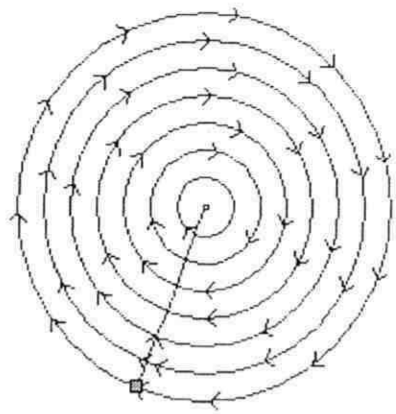
- Click on the **Toolpaths** menu in the Main menu bar, followed by the **Toolpath Drawing > Show Rapid and Plunge Moves** option.
- Deselect the **Rapid & Plunge Moves** option in the **Machining** area of the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

Cutting Direction

As an aid to visualisation, arrows indicating the cutting direction used in a 2D toolpath are used in toolpath previews drawn in the **2D View** window. The cutting direction is hidden by default, but can be shown in either of the following ways:

- Click on the **Toolpaths** menu in the Main menu bar, followed by the **Toolpath Drawing > Show Cutting Direction** option.
- Select the **Cutting Direction** option in the **Machining** area of the **ArtCAM Options** page. For details, see "Managing ArtCAM Pro's Preferences" in the ArtCAM Pro Layout chapter.

A Conventional cut direction is illustrated with anti-clockwise arrows, and a Climb Mill cut direction with clockwise arrows. For example, we can see that a Climb Mill cut direction has been used in the Area Clearance toolpath preview shown below:



Hiding a Toolpath

You can hide any toolpath shown in either the **2D View** or **3D View** window.

To hide a toolpath shown in the **2D View** window:

1. Click to select the toolpath preview that you want to hide.
2. Right-click to display the Toolpath Editing menu, and then click on the **Hide Preview** option.

To hide a toolpath shown in the **3D View** window:

1. Double right-click in the **3D View** window to display a list of toolpath viewing options, and then click on the toolpath you want to hide. The toolpath that you want to hide is deselected

Note: You can hide any of the toolpaths shown in the **3D View** window using the **Objects To Draw** button in the **3D View** toolbar. For details, see "Objects To Draw" in the ArtCAM Pro Layout chapter.

Restoring a Hidden Toolpath

You can restore a toolpath that has been hidden in either the **2D View** or **3D View** window.

To restore a toolpath hidden in the **2D View** window:

1. Click on the **Toolpaths** tab Toolpaths to display the **Toolpaths** Home page.
2. Make sure that the **2D View** window is displayed.

If the **3D View** button is in the toolbar above the design window, the **2D View** is displayed. If not, click on the **2D View** button.

3. Click to select the **Show In 2D** box next to the relevant toolpath you want to view. The toolpath appears in the **2D View** window in dark red.

To restore a toolpath hidden in the **3D View** window:

1. Double right-click in the **3D View** window to display a list of toolpath viewing options, and then click on the toolpath you want to view. The toolpath that you want to restore is now selected.

   For example, clicking on the **Profile – End Mill 3mm** option restores the toolpath to view.

Note: You can restore any of the toolpaths hidden in the **3D View** window using the **Objects To Draw** button in the **3D View** toolbar. For details, see "Objects To Draw" in the ArtCAM Pro Layout chapter.

Resetting a Simulation

You can reset a toolpath simulation that has been drawn in the **3D View** window.

To reset a toolpath simulation:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.

2. Click on the **Reset Simulation** button in the **Toolpath Simulation** area.

Deleting a Simulation

You can delete a toolpath simulation from the **3D View** window.

To delete a toolpath simulation:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.

2. Click on the **Delete Simulation** button in the **Toolpath Simulation** area.

Saving a Simulation as a Relief

You can save a toolpath simulation as a relief file (***.rlf**).

To save a toolpath simulation as a relief:

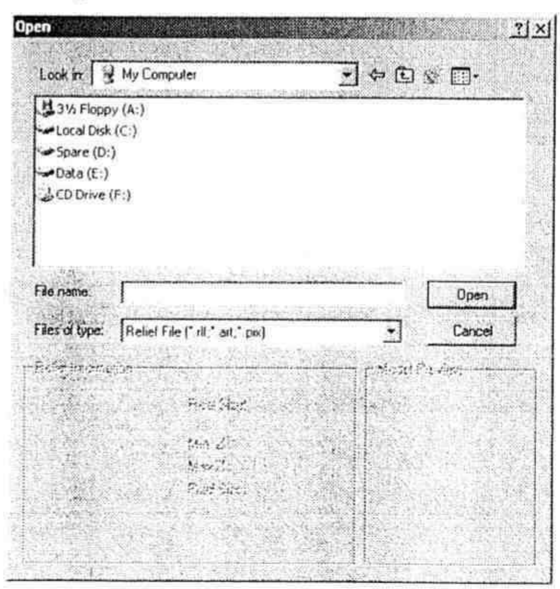
1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Save Simulation As A Relief** button in the **Toolpath Simulation** area to display the **Save As** dialog box:
3. Click on the **Save In** list box and select the directory in which you want to save the relief.
4. Type the file name that you want to use for the relief in the **File Name** box.
5. Click on the **Save** button.

Loading a Simulation from a Relief

You can load a toolpath simulation from a relief.

To load a toolpath simulation from a relief:

1. Click on the **Toolpaths** tab to display the **Toolpaths** Home page.
2. Click on the **Load Simulation From Relief** button in the **Toolpath Simulation** area to display the **Open** dialog box:



3. Click on the **Look In** list box and find the relief file (***.rlf**) that you want to open.
4. Once you have found the file, click on the file name listed in the main window of the **Open** dialog box. The name of the relief file you have selected appears in the **File Name** box.

Note: You are only able to select relief files (***.rlf**). You can confirm this by clicking on the **Files of Type** list box.

5. Click on the **Open** button to load the relief.

Index

2

3

D

E

F

G

H

I

L

M

N

O

T

U

V

W

Z