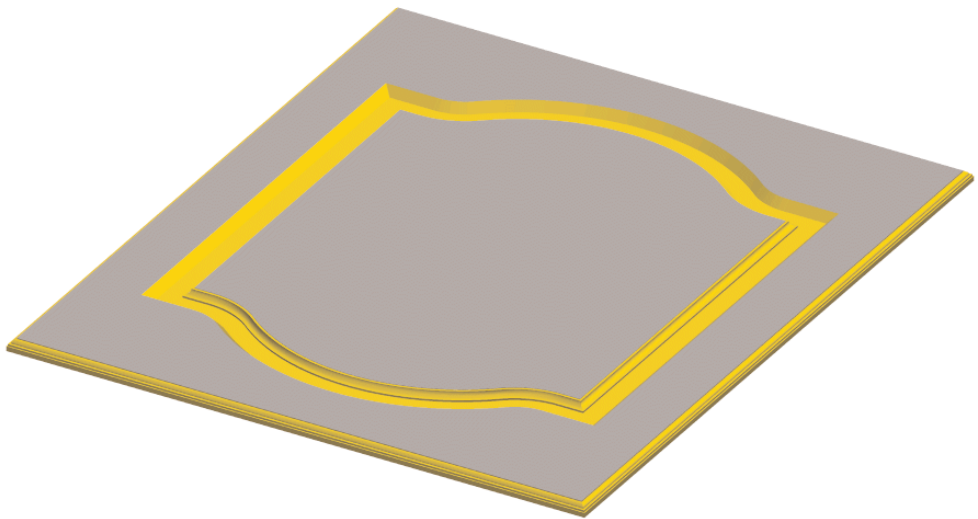


LICOM AlphaCAM

CAD/CAM system for Windows™

SIMPLE ROUTER TUTORIAL



Conventions When Using The Tutorial	4
Introduction	4
The Part Geometry	6
The Process Plan	8
Creating The Geometry	9
Draw the Outer Rectangle	9
Use APS Fast Geometry to create the raised panel profile.	9
Y axis Mirror	12
X axis Mirror	13
Offset the geometries	14
Offset the rectangle	14
Offset the profile for the bottom of the chamfer edge.	15
Offset the profile for the inner panel edge.	16
Preparing For Machining	17
Set The Machine Type	17
Set The Cutting Conditions	17
Tool Directions	18
Start Point	20
Tool Selection	21
Cutting Speed	21
Select Tool	21

The Machining Operations	22
Operation 1	22
Profile the inner geometry.	22
Display Options	24
Tool Selection	25
Select Tool	25
Operation 2	26
Profile to clean bottom of relief before chamfer	26
Operation 3	28
Engrave the Chamfer to produce sharp corner.	28
Operation 4	30
Profile outer geometry with shaped cutter	30
Operation 5	32
Profile outer geometry with straight cutter	32
Lead In/Out	34
VISUALISING THE COMPLETE PROGRAM	35
Display Options	35
Solid Simulation	36
Operation 6	37
Array the toolpaths to produce 12 doors.	37
Save and Output.	38

Conventions When Using The Tutorial

Notes and comments are in *Italics* to separate them from the main text. If there are different ways of performing the same command or option, these are also in *Italics* and can be ignored the first time you work through the tutorial.

The symbol  indicates a new command for you to action.

The symbol  indicates a sub part of command for you to action

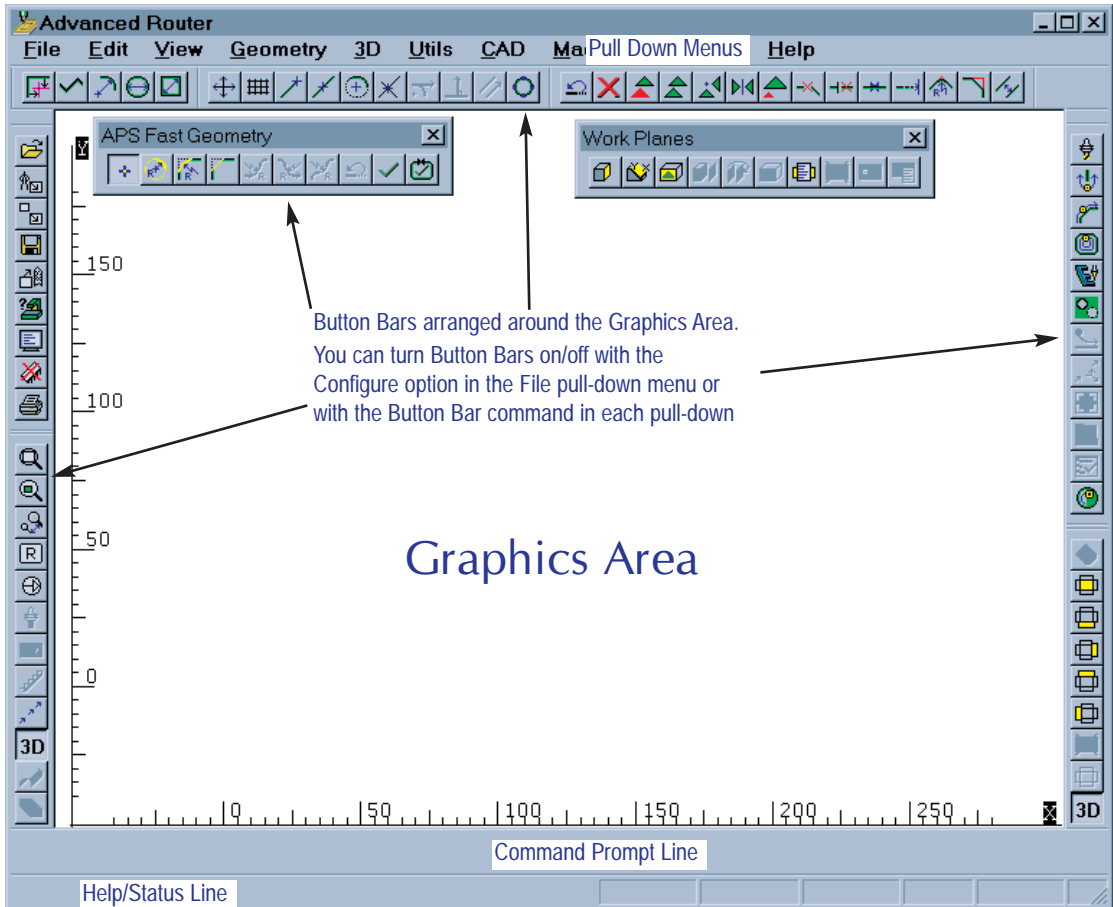
The HELP file is very comprehensive. When you have finished the tutorial, please take time to have a look at it. **Select HELP | Contents.**

Introduction

This **AlphaCAM** tutorial has been designed to give you a flavour of how **AlphaCAM** works, how interactive it is, how easy it is to learn and how easy it is to use. We assume that you are familiar with the concepts involved in CNC programming and have a reasonable understanding of your computer and the Windows operating system. **AlphaCAM** has been developed as a true 32-bit Windows 95 or NT application, so if you use other Windows programs you will be familiar with features such as floating button bars, tabbed dialog boxes, etc. If not, you should look at **HELP | Contents | Screen Layout.**

AlphaCAM systems are available for all machining disciplines. Each one is specifically designed for the machine type, but the look and feel of all the systems is standard. This tutorial can be used with both Standard and Advanced **AlphaCAM**, Mill and Router, modules. The tutorial describes the construction and machining of the geometry for the following drawing.

If you have not already done so, start the '**Advanced Router**' module. This will take you into the graphical portion of the system. Your screen will look similar to the one following.



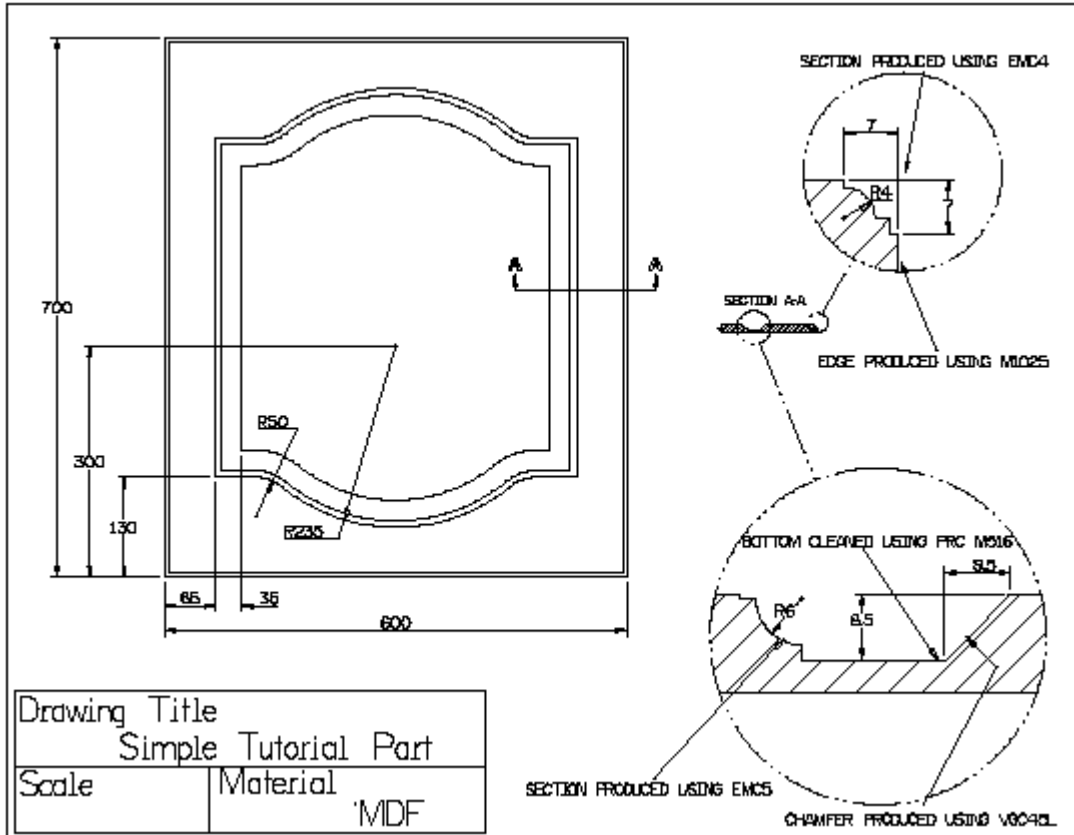
During this tutorial, we tell you where to find commands on the pull-down menus. If there is a button for the command, this will also be shown.

Take the trouble to locate the buttons on your screen. You can speed up your work by avoiding having to pull down menus and side menus to click on commands, when one click on a button will suffice.

To see what command a button performs, place the screen pointer over the button for a couple of seconds, and a prompt will appear beside the pointer.

The Part Geometry

This drawing is the part on which you will work in this tutorial. It was drawn, dimensioned and printed using AlphaCAM. By the end of the tutorial, you will have created the NC program for the drawing.



AlphaCAM provides various ways of creating part geometry.

Conventional CAD style geometry, creation, whereby individual geometric features are created and then trimmed in order to create geometric contours. Toolpaths are then applied to the geometric contours, from which the NC program is produced;

'**APS Fast Geometry**' is a unique way of creating 'bounded' geometry. This can turn some designs into geometric contours much faster than with any conventional CAD system.

With APS Fast Geometry, you do not specify individual lines and arcs. Instead, you say how the tool should move from one element to the next. Each change in direction is called a **Turn**. APS Fast Geometry builds a 'bounded' geometry profile by automatically trimming, blending and filleting as Turn details are entered. This method is very powerful, because it allows you to answer 'Unknown' to questions about poorly specified co-ordinates and once **AlphaCAM** has enough information it will 'back-calculate' to solve the unknowns.

However, **CAD-style** geometry creation using Line, Arc and Circle commands is sometimes appropriate for simple shapes, and all the conventional CAD commands are included, plus Special Geometries, which automatically produce standard geometric shapes.

The Process Plan

Before creating the geometry to be machined, it is important to decide what geometry is required to control the machining. The geometry necessary to control the machining is defined by the process plan. In some cases, it may be necessary to create the defined geometry then use the offset command to create the centre line geometry to be machined.




OP No.	Operation Description	T No.	Tool Description
1	Profile to produce the raised panel effect	1	Router - Emc5
2	Profile to clean bottom of relief before chamfer	2	Router - M516
3	Engrave chamfer to produce sharp corner	3	Router - Vgc45l
4	Profile door edge effect	1	Router - Emc4
5	Profile around to clean door edge	2	Router - M1025
6	Array toolpaths to cut 3000x2000 sheet	4	Use subroutines and tool sorting




From the process plan, it can be seen that the geometries required to control the machining are the top edge geometries. In this example, the bottom profile of the 45-deg chamfer is created, in order to produce the flat bottom in the recess.

Creating The Geometry



Draw the Outer Rectangle

 Select **GEOMETRY | Rectangle** . This command creates a rectangle that is defined about 2 diagonal points.

 The command line prompts you to enter the co-ordinates of the first corner of the rectangle. Type **0**  **0** 

 The command line now prompts you to enter the co-ordinates of the second corner of the rectangle. Type **600**  **700** 

The rectangle is drawn on the screen.

 Select **VIEW | Zoom All**  **(Ctrl)+A** to make the geometry fit the screen

*Pressing **[Esc]** or **<RMB>** once aborts any repeating commands.*

Use APS Fast Geometry to create the raised panel profile.

As the raised panel effect is symmetrical, it is only necessary to draw one quarter and mirror about the centre horizontal and vertical axis.

The raised panel is drawn from the left middle point to the middle at the bottom.

The direction in which the geometry is drawn is NOT reflective of the method of machining

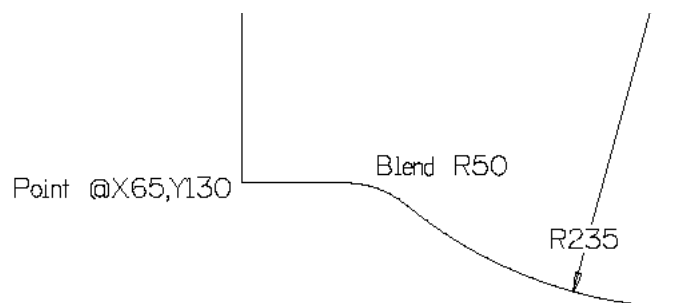
The geometry consists of four turns,

Point 1,




Point 2,

Line to Arc Blend 3,

Known Arc 4.






 Select **GEOMETRY | APS Fast Geometry | Point.** 

 The command line prompts you to enter the co-ordinates of the point.
Type **65**  **350** 

This position is selected on the screen and a phantom line will connect it to the cursor.



*The next turn is also a **Point**. The command repeats, so it is not necessary to select it again.*

 The command line prompts you to enter the co-ordinates of the next point.
Type  **130** .





Pressing  will accept the default X value 65 (from the last command).

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.




 Select **GEOMETRY | APS Fast Geometry | Line to Arc.** 

 The command line prompts you to enter the radius value of the blend.
Type **50** .

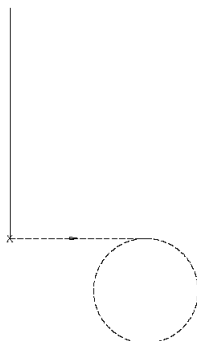
The first dialog box is displayed.

 This prompts for the direction of rotation and included angle of the blend.
The  CW direction and  < 180 angle are offered as default.
Click  on



A second dialog box is displayed.

 This prompts for the direction of the Line. Select the direction, then click  on or Type **0** .


A phantom line and blend is drawn from the last point, as these geometries can not be drawn until the next turn is entered, thereby fixing the position of the arc.






 Select **GEOMETRY** | **APS Fast Geometry** | **Known Arc.** 

 The command line prompts you to enter the radius value of the Arc.
Type **235** .

The first dialog box is displayed.

 This prompts for the direction of rotation of the Arc. Select 

 The command line prompts you to enter the co-ordinates for the circle centre. Type **300**  **300** 

A phantom circle is now drawn at this position on the screen. The arc can not be drawn until the next turn is entered. There is no other turn, as the geometry is complete. To indicate this to the system, the finish option must be selected.

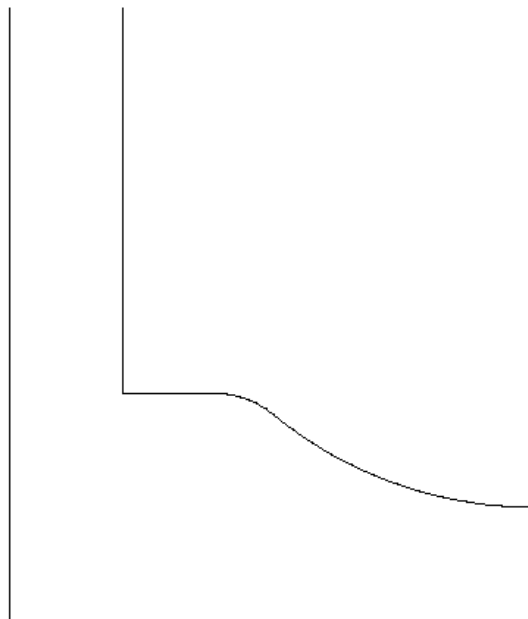
 Select **GEOMETRY** | **APS Fast Geometry** | **Finish.** 

As the geometry is being finished on an arc, it is necessary to define the end direction at which to finish. **AlphaCAM** displays a dialog box, which prompts for the End Direction.

 Select the **0** direction, then click  on **OK** or Type **0** .





The geometry changes from yellow (current geometry)
to green (finished geometry).

This completes a quarter of the part geometry.









Y axis Mirror


 Select **EDIT | Move, Copy etc. | Mirror.** 

 The command line prompts you to select the geometry to be mirrored. Click  on the $\frac{1}{4}$ geometry profile on screen and it turns blue. Then click  on or click  on any position on screen.

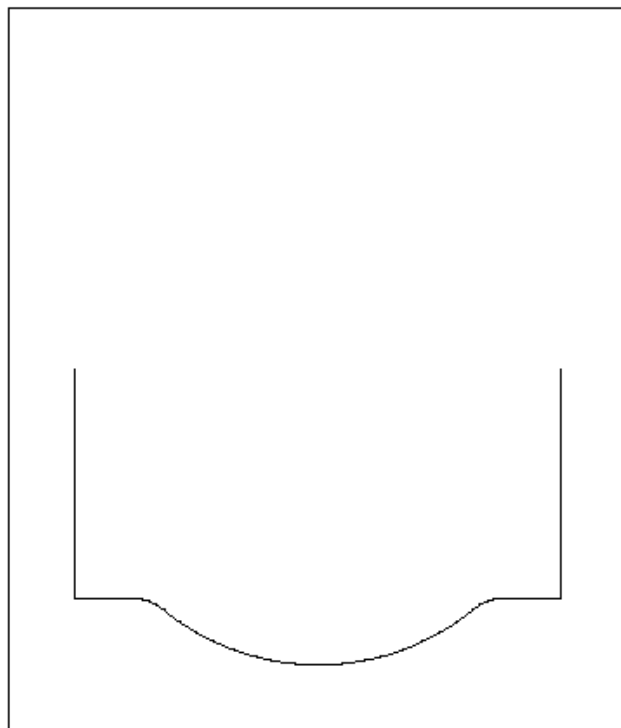
The system will now expect you to describe, using two points, the axis about which the selection is mirrored.

 The command line prompts you to enter the first point on the mirror line. Type **300**  **0** . As the axis is through the middle, any Y value will do.

 The command line prompts you for the second point on the mirror line. Type  **1** .

 A dialog box is displayed, asking if the original is to be kept. Select





The new shape is drawn and the mirror command is repeated.






X axis Mirror




 The mirror command should still be active.



If not, select **EDIT | Move, Copy etc. | Mirror.** 

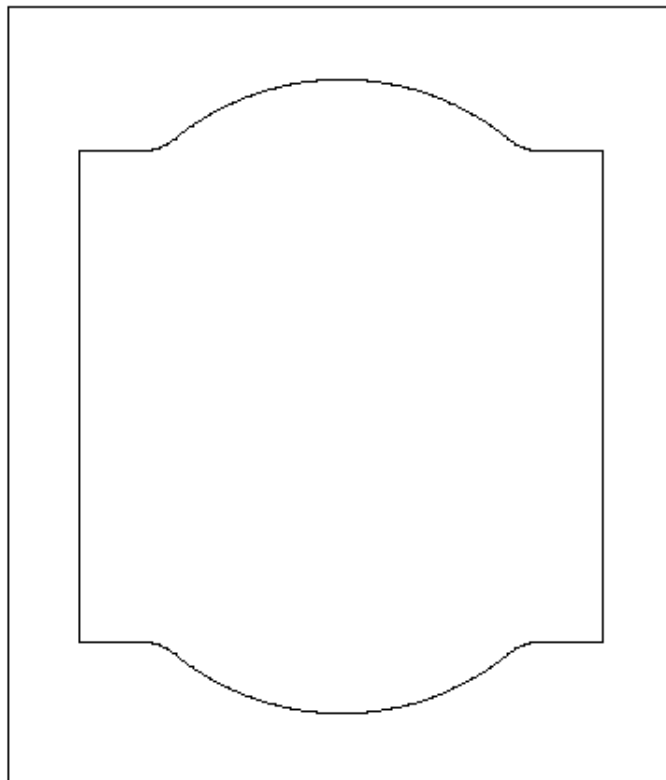
 The command line prompts you to select the geometry to be mirrored. Click  on the ½ geometry profile on screen and it turns blue, then click  on or click  on any position on screen.

The system will now expect you to describe, using two points, the axis about which the selection is mirrored.

 The command line prompts you to enter the first point on the mirror line. Type **0**  **350** . As the axis is through the middle, any X value will do.

 The command line prompts you for the second point on the mirror line. Type **1**  .

 A dialog box is displayed, asking if the original is to be kept. Select . The complete shape is drawn. The mirror command is repeated. Press or RMB 



Offset the geometries

There are three offset geometries to be created: one from the door rectangle and two from the profile geometry.

Offset the rectangle


 Select **EDIT | Break Join etc. | Offset** 




 A dialog box is displayed.



The first section prompts for the Distance: type 7.

The second section has 2 or 3 settings, which set What is to be offset.

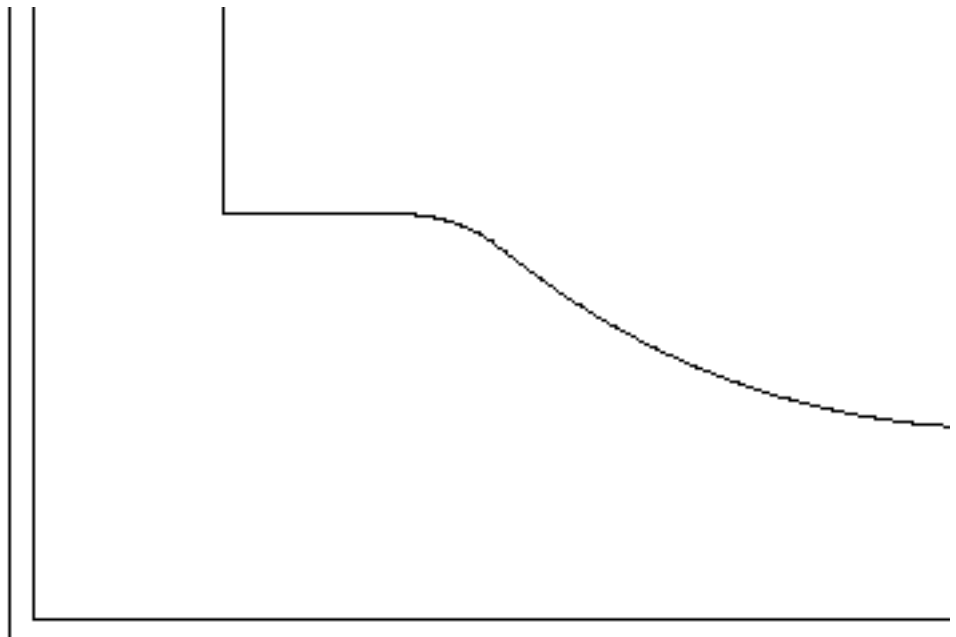
Click  on the Geometry option.

When the dialog is complete, click  on

 The command line prompts you to select the geometry that is to be offset. Click  on the rectangle on the screen. The complete rectangle should turn blue: if it does not, click  RMB on the screen and start this section again.

 The command line now prompts you to indicate on which side the geometry is to be offset. Click  on a position inside the rectangle on the screen.

The inner rectangle is drawn.






Offset the profile for the bottom of the chamfer edge.

As the whole profile is to be offset, it is necessary to ensure that it is one geometry.




 Select **EDIT | Break, Join etc. | Join.** . Click  on the raised panel profile.




*If it does not all turn blue,
select all the portions that comprise the raised panel geometry.*



 When all the raised panel geometry has turned blue,
click  on or click  RMB.

*This command joins all the selected geometries,
whose ends are the same, into one geometry.
If the geometries will not join together,
the mirror axis was probably incorrectly specified.*

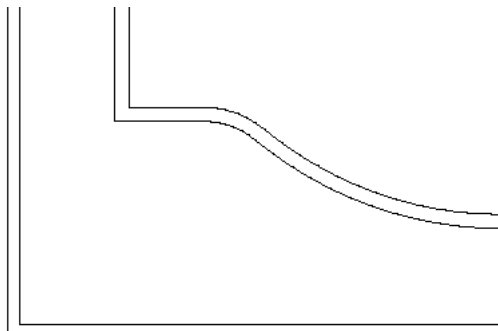
 Select **EDIT | Break Join etc. | Offset** 

 A dialog box is displayed. The first section prompts for the Distance.
Type **8.5**. The second section has 2 or 3 settings, which set What is to be offset. Click  on the Geometry option.
When the dialog is complete, click  on

 The command line prompts you to select the geometry that is to be offset.
Click  on the profile on the screen. The complete profile should turn blue: if it does not, click  RMB on the screen and start again at the previous section.

 The command line now prompts you to indicate which side the geometry is to be offset. Click  on a position inside the profile on the screen.



The chamfer bottom profile is drawn.





If the screen is looking a bit untidy, select **VIEW | Redraw** + to redraw the display.


Offset the profile for the inner panel edge.

 Select **EDIT | Break Join etc. | Offset** 

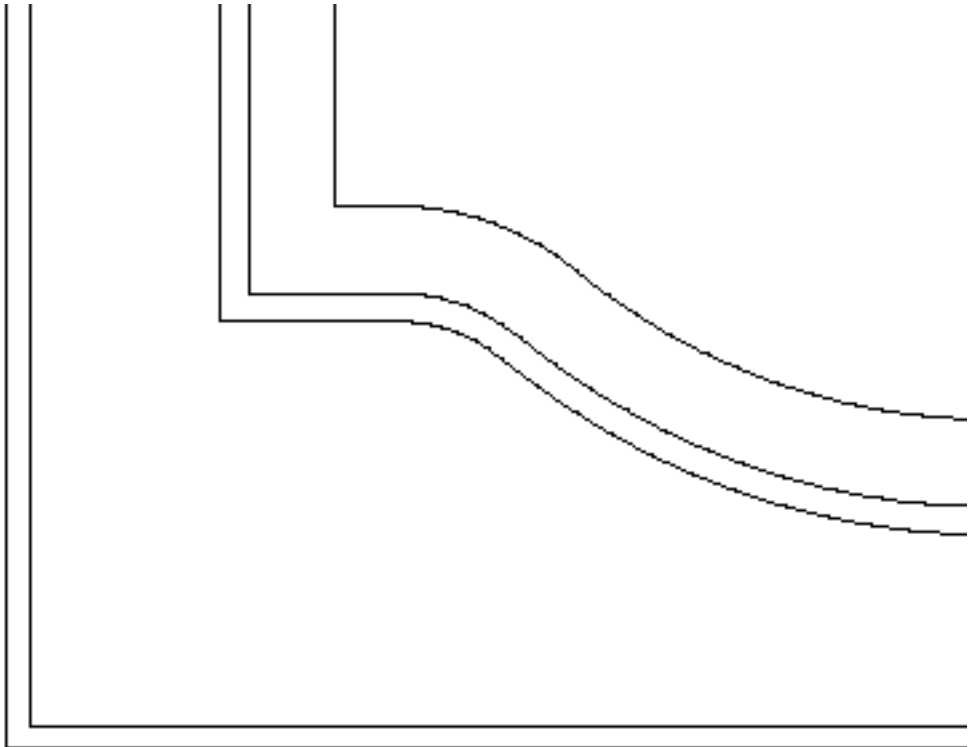
 A dialog box is displayed. The first section prompts for the Distance:
Type **35**. The second section has 2 or 3 settings, which set What is to be offset. Click  on the Geometry option.

When the dialog is complete, click  on

 The command line prompts you to select the geometry that is to be offset.
Click  on the original profile on the screen.

The command line now prompts you to indicate which side the geometry is to be offset. Click  on a position inside the profile on the screen.

The raised panel profile is drawn.

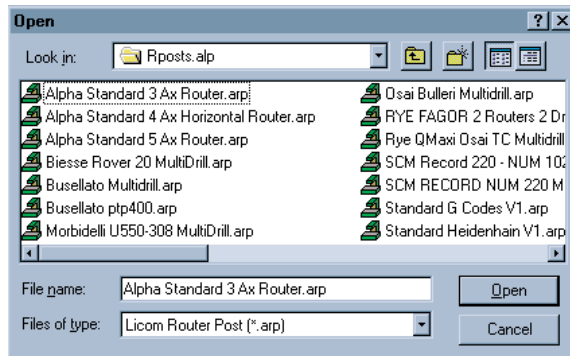


Preparing For Machining

It is necessary to set the right conditions before the machining commands can be applied to the geometry profiles.

Set The Machine Type

 Select **FILE** | **Select Post** .  The available post processors are displayed.

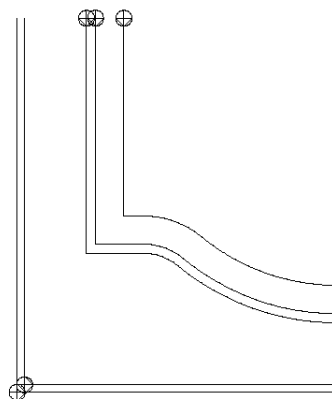


 Select a suitable post processor.

Set The Cutting Conditions

Setting the cutting conditions involves setting the way in which the tools will cut the component. The conditions only need to be set for geometries against which the tool is driven. The default tool directions are set as the direction of definition and on centre line.

 Select **VIEW** | **Display Options** | **Ghost Tools**  **(Ctrl) + (G)** to see the tool directions



In line with the operations list, it is necessary to set the following tool cutting directions.

The two outer rectangles and the innermost profile are to be cut in a CW direction with the tool positioned on the outside.

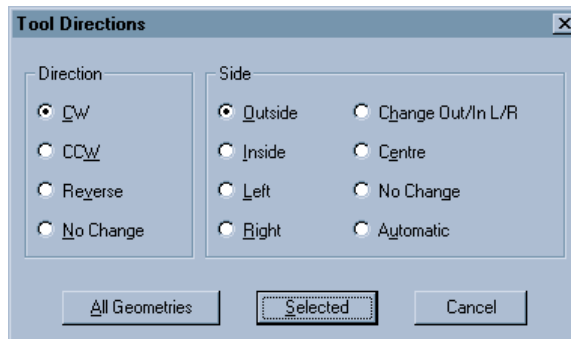
The top and bottom chamfer profiles are to be cut in a CCW direction, with the tool positioned on the inside.





In addition to setting the tool directions and side of cutting, it may also be necessary to adjust the point at which the tool starts cutting the profile.

Tool Directions

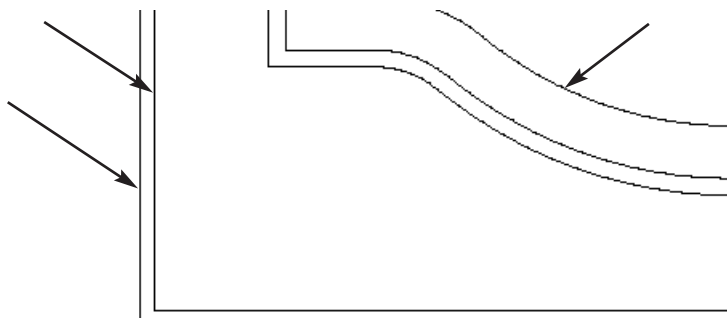
 Select **MACHINE** | **Tool Directions** 

A dialog box with 2 sections is displayed. The left section controls the direction of cutting and the right section controls the side to which the tool is positioned when cutting. The side of cutting is defined whilst looking in the cutting direction.



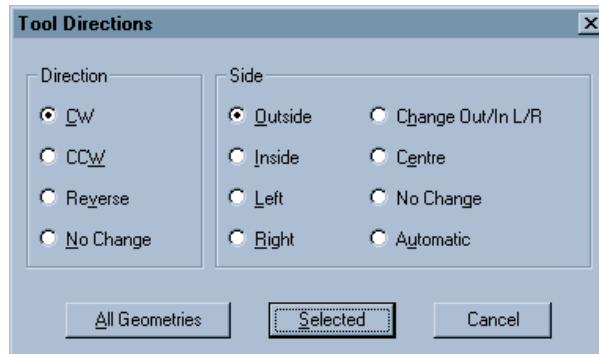
 Click  on the **CW** option to set the direction and click  on the **Outside** option. When the selection is correct, click  on the **Selected** button. The Selected button allows you to pick the geometries to which the tool directions are to be applied.





 Click  on the two outer rectangles and the innermost profile.



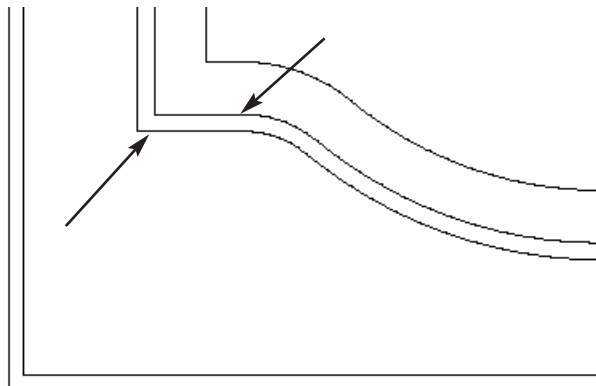
 Select **VIEW** | Redraw   +  to tidy up the display.





 Select **MACHINE** | **Tool Directions**  again.



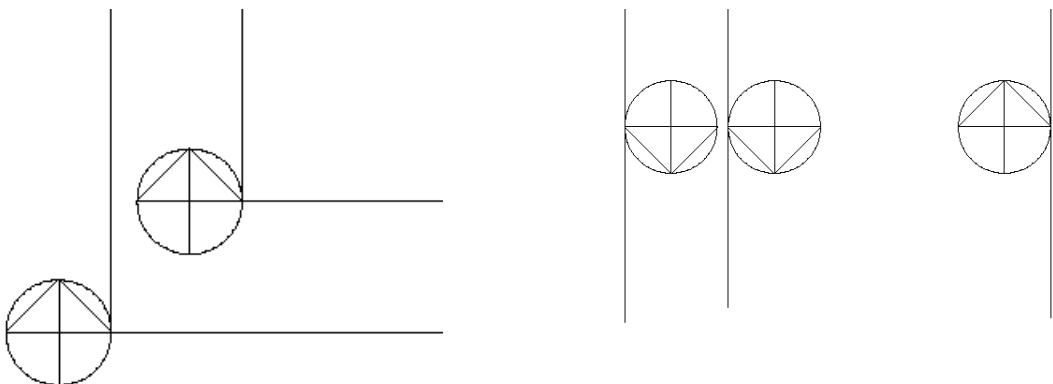
 Click  on the **CCW** option to set the direction and click  on the **Inside** option. When the selection is correct, click  on the **Selected** button.

 Click  on the top and bottom chamfer profiles.



 Select **VIEW** | **Redraw**   +  to tidy up the display.




The Ghost tools are displayed as shown.



Start Point

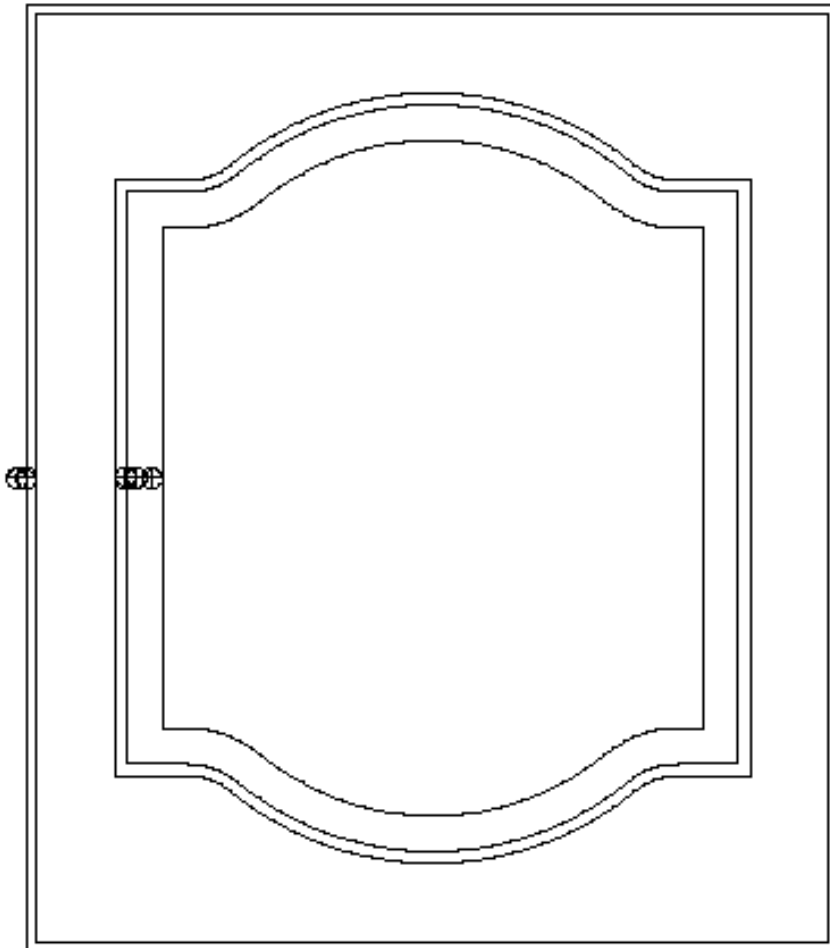
As it is not good practice to start machining on an internal corner and the part is to be arrayed, we will reposition the start point to the middle of the left vertical edge of the profiles and rectangles.

 Select **EDIT** | **Start, Order** | **Start Pt.**  **[Ctrl] + [F]**

 Select **UTILS** | **Snaps** | **middle of**  **[F7]**. Click  on the left edge of the profiles and rectangles, remembering to re-select snap to middle before selecting each geometry.

 Select **VIEW** | **Redraw**  **[Ctrl] + [R]** to tidy up the display.

The ghost tool display should resemble the following diagram.

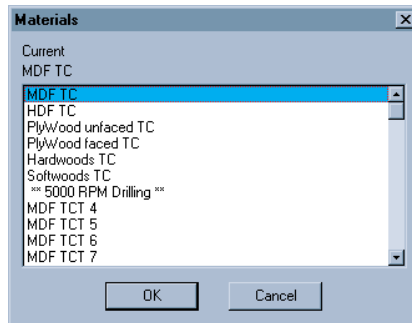





Tool Selection

Before selecting the tool for the first machining operation, it is necessary to select the cutting speed for the material and cutting operation.


Cutting Speed

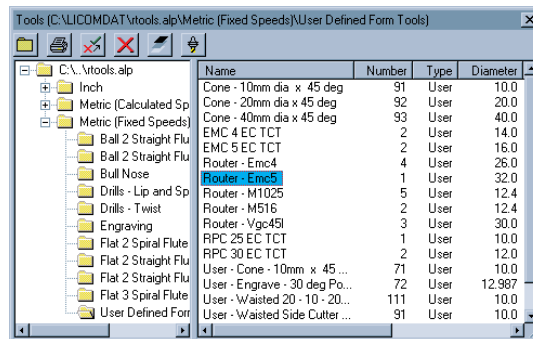
 Select **MACHINE** | **Select Material** .  The material dialog is displayed.







 Select the **MDF TC** option. To see this option, it may be necessary to use the scroll bar on the right to scroll up and down through the options. Click  on the option to highlight it, then click  on

Select Tool

 Select **MACHINE** | **Select Tool** .  The tool library dialog is displayed.



 Pick the tool name **Router – Emc5**, then click  on  (the select tool button in the tool library dialog). The tool library dialog disappears and a schematic of the tool is displayed.

 The command prompt line prompts you to press to accept the tool selection or press to return to the tool library dialog. Press .

*YOU MUST ACCEPT THE TOOL TO SELECT IT!
PICKING ANY OTHER OPTION CAUSES THE TOOL NOT TO BE SELECTED.*

The Machining Operations

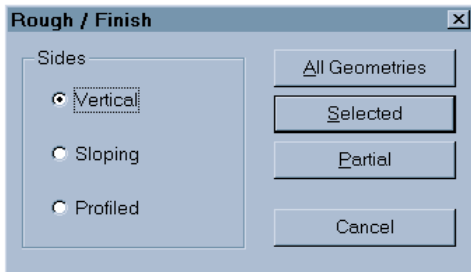
Operation 1

Profile the inner geometry.

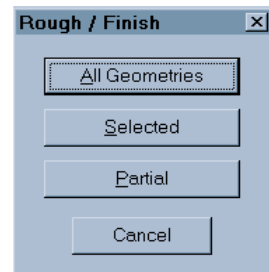
 Select MACHINE | Rough or Finish .

The first dialog box shown differs between the Advanced and Standard modules

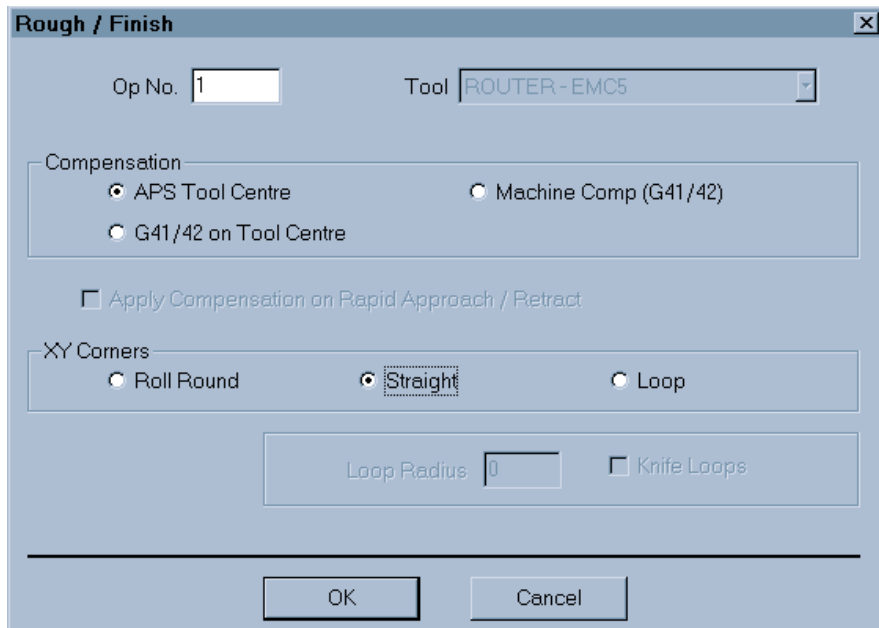
Advanced





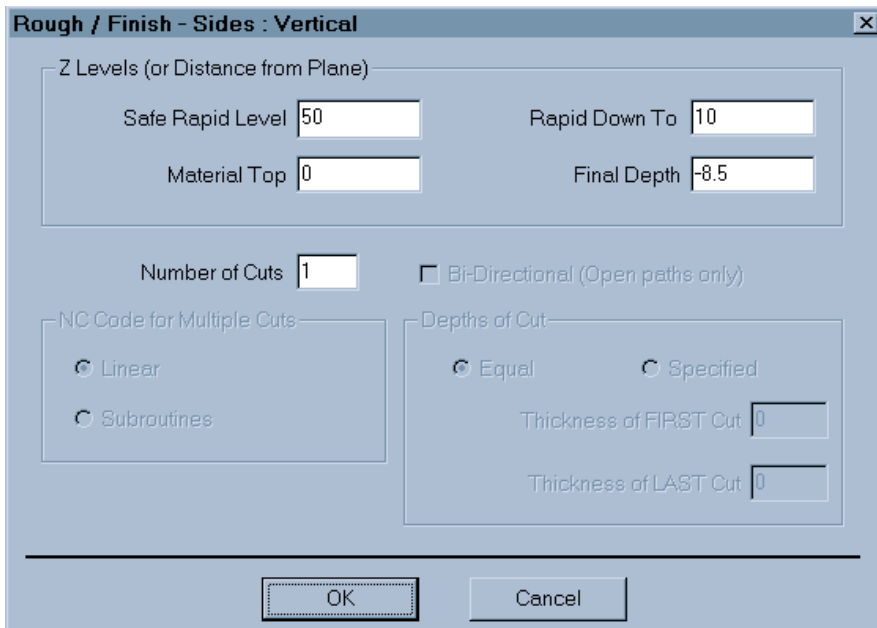
Standard



 Click  on the **Selected** button and the second dialog box is displayed.



 Set the options as shown. When complete, click  on **OK** and the third dialog box is displayed.



Rough / Finish - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level Rapid Down To

Material Top Final Depth

Number of Cuts Bi-Directional (Open paths only)


NC Code for Multiple Cuts

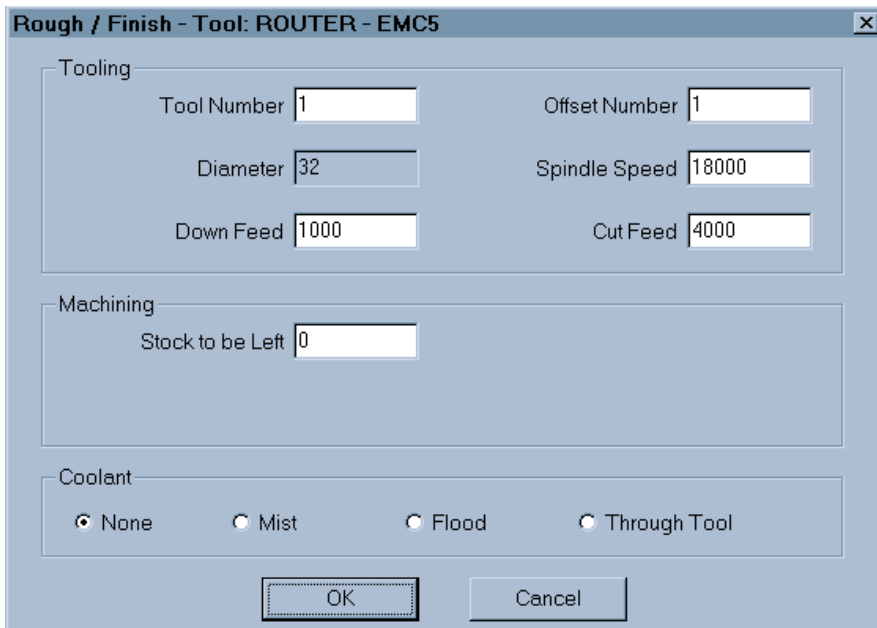
Linear Equal Specified

Subroutines

Thickness of FIRST Cut

Thickness of LAST Cut

☞ Set the options as shown. When complete, click  on and the fourth dialog box is displayed.



Rough / Finish - Tool: ROUTER - EMC5

Tooling

Tool Number Offset Number

Diameter Spindle Speed


Down Feed Cut Feed



Machining




Stock to be Left

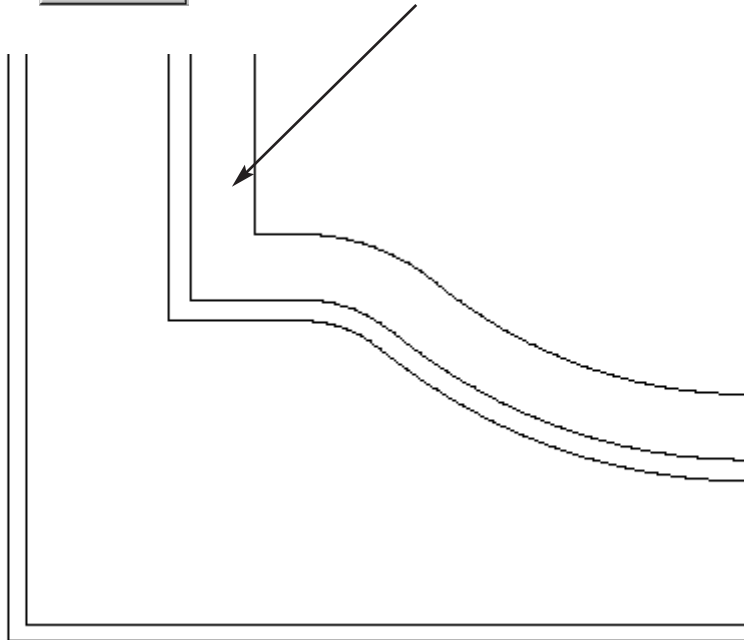
Coolant

None Mist Flood Through Tool

☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click  on

 The dialog box disappears and the command line prompts you to select the geometries. Left click  on the inner profile: it turns blue.





 Click  on 






The toolpath is displayed.

Display Options

At this stage, you may wish to utilise some of the different display options available. The commands can be found in the VIEW | Display Options menu.

To see the area of material cut by the tools, select  followed by .  To return the display to normal, select  followed by .

To see the tool in an animated display, select ,  and . **3D** The screen splits automatically into four views and an animated tool is displayed. To see the animation again, select .

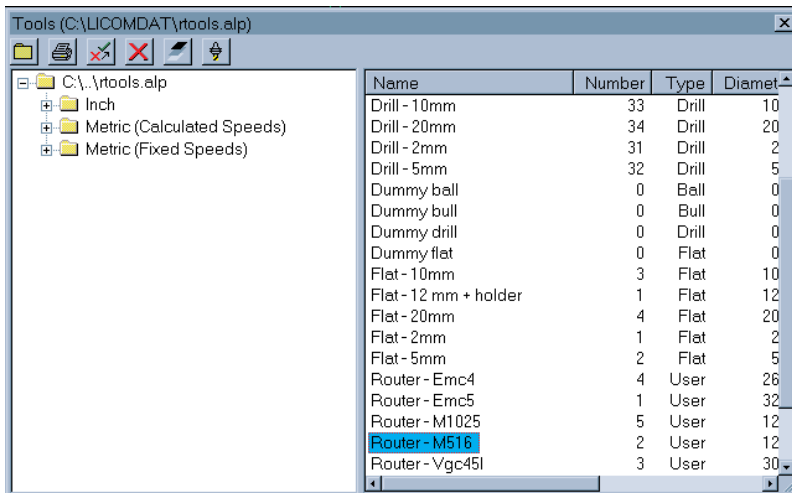
To return the display to normal, select ,  and .




Tool Selection


Before selecting the tool for the next machining operation, it may be necessary to select the cutting speed for the material and cutting operation. In this instance, this is not necessary, as they have not changed.

Select Tool

 Select **MACHINE** | **Select Tool** .  The tool library dialog is displayed.



 Pick the tool name **Router – M516**, then click  on .  The tool library dialog disappears and a schematic of the tool is displayed.

 The command prompt line prompts you to press **Enter** to accept the tool selection or press **Esc** to return to the tool library dialog. Press **Enter**.

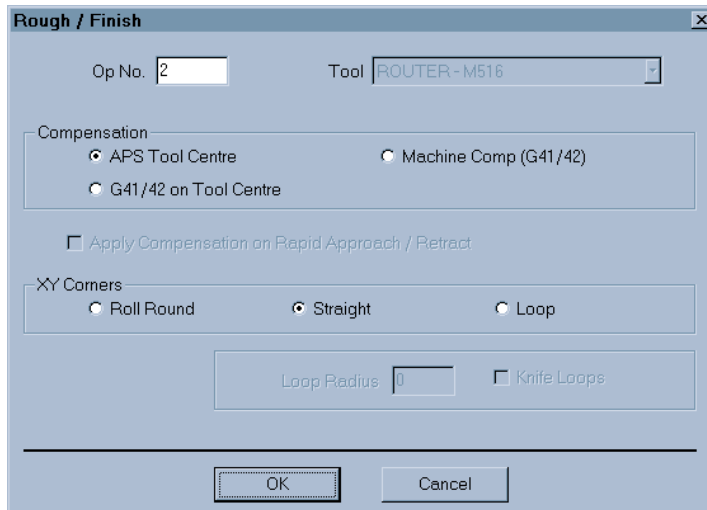
Operation 2



Profile to clean bottom of relief before chamfer

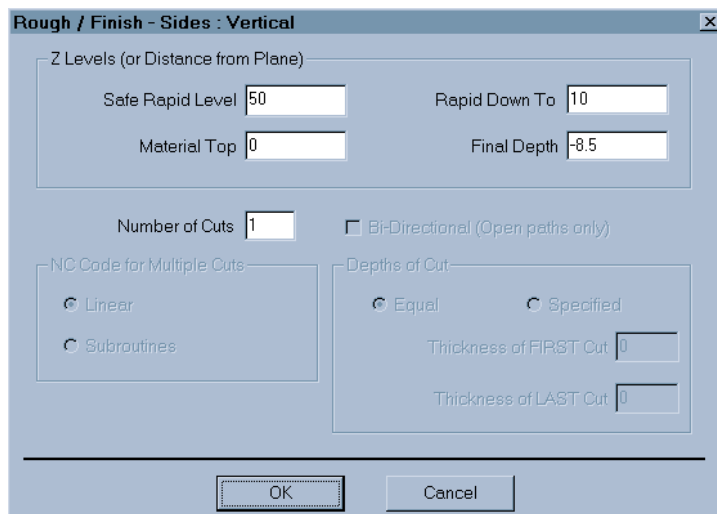
 Select **MACHINE** | **Rough or Finish** .



The first dialog box shown differs between the Advanced and Standard modules

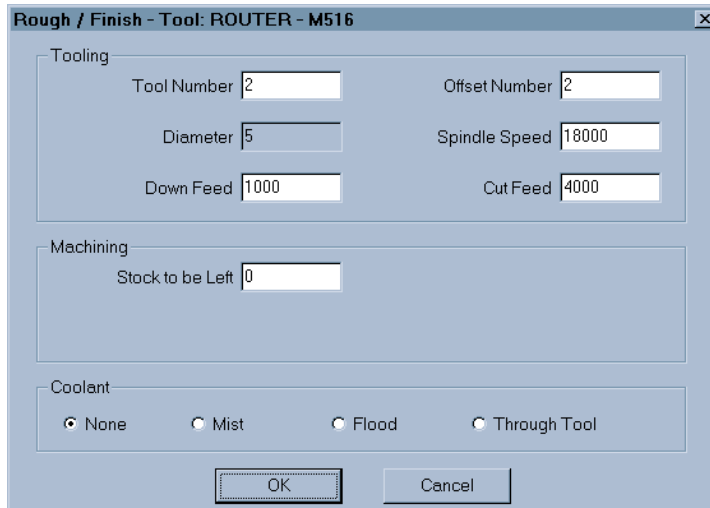
 Click  on the **Selected** button and the second dialog box is displayed.






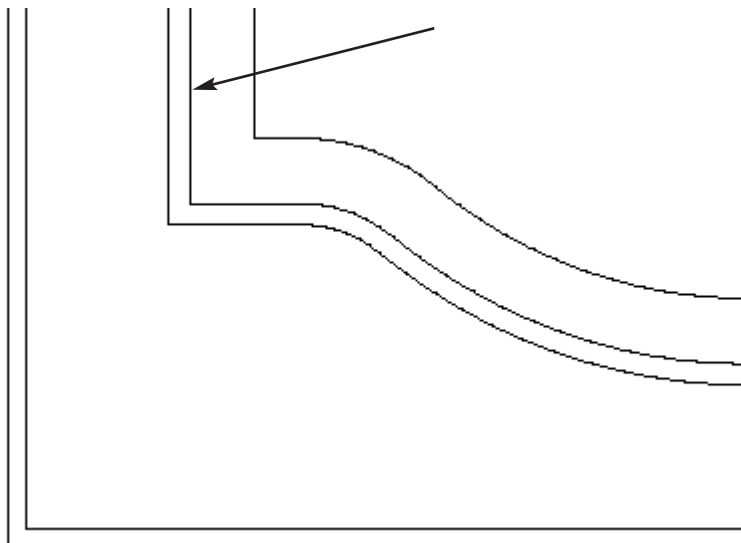
 Set the options as shown. When complete, click  on **OK** and the third dialog box is displayed.



 Set the options as shown. When complete, click  on **OK** and the fourth dialog box is displayed.



- ☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click  on
- ☞ The dialog box disappears and the command line prompts you to select the geometries. Left click  on the chamfer bottom profile: it turns blue. click  on



The toolpath is displayed.

Operation 3

Engrave the Chamfer to produce sharp corner.

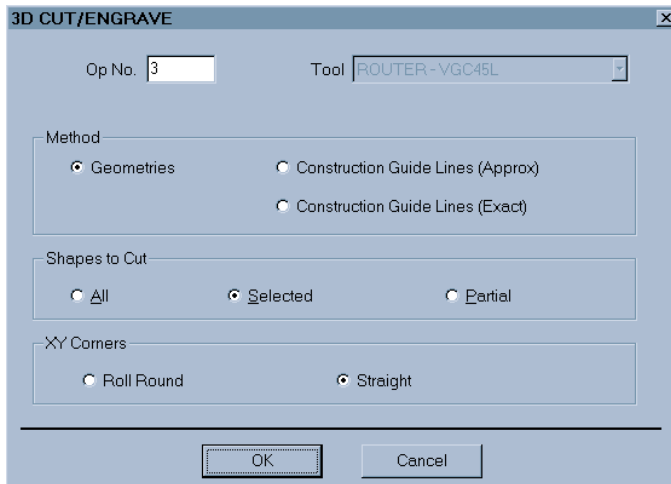
 Select **MACHINE** | **Select Tool** .  Select the tool called **Router - Vgc45l**



 Press **Enter** to accept and load the tool.

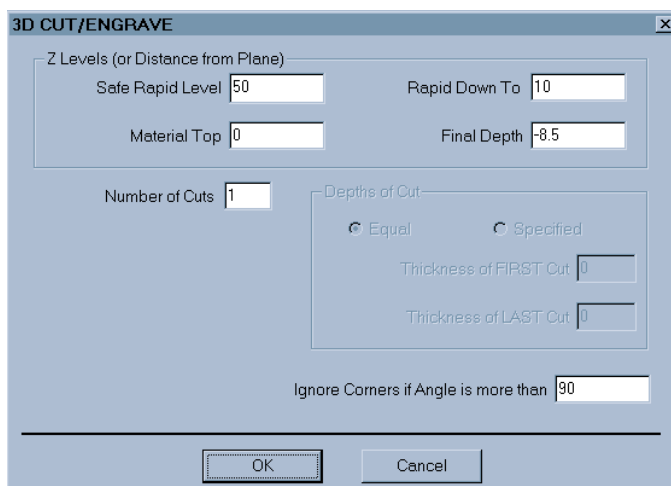
 Select **MACHINE** | **3D Engraving** . 

AlphaCAM will display three dialog boxes.

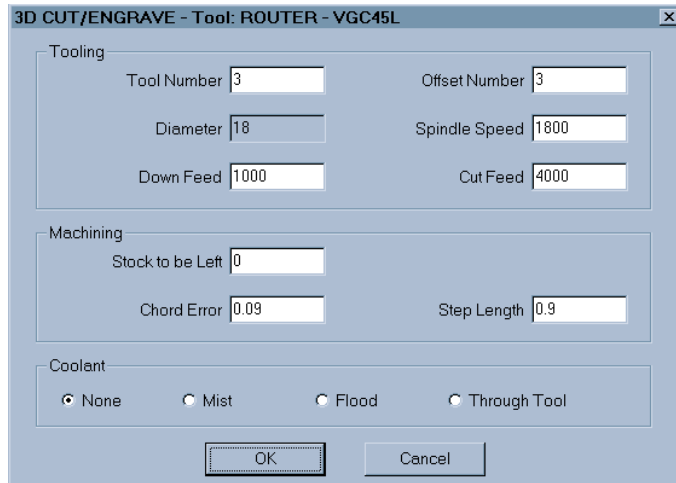
Any relevant values are retained from the previous operation.






 Set the options as shown. When complete, click  on **OK** and the second dialog box is displayed.

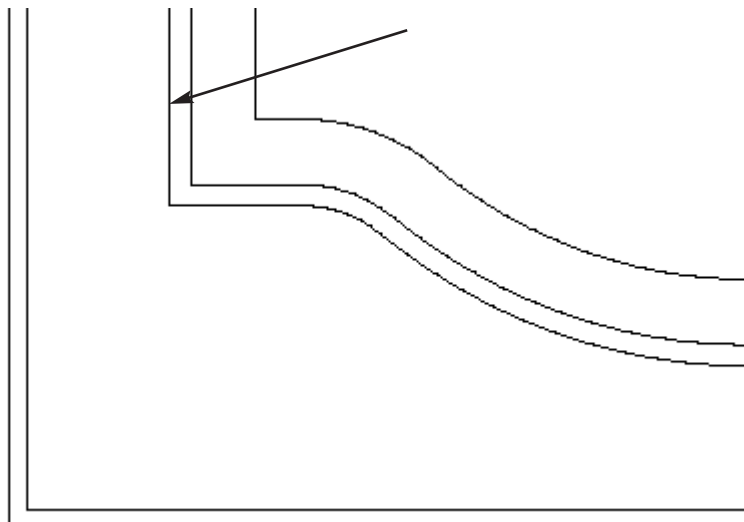


 Set the options as shown. When complete, click  on **OK** and the third dialog box is displayed.




☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click  on

☞ The dialog box disappears and the command line prompts you to select the geometries. Left click  on the chamfer top profile: it turns blue. click  on



☞ Press to finish the selection and the toolpath is drawn.

To see the NC code that has been generated, select **FILE | List NC Code** . The list screen is displayed. Select and enter any CNC code data as requested (program number, etc.). The NC program will then be displayed. To return to the graphics display, select the button.

Operation 4

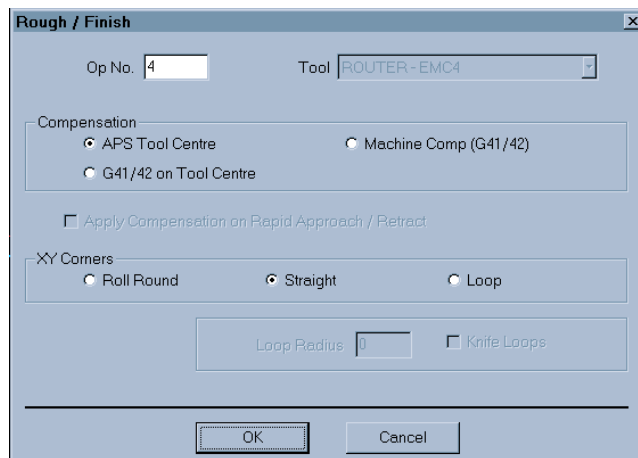
Profile outer geometry with shaped cutter

 Select **MACHINE** | **Select Tool** . Select the tool called **Router – Emc4**

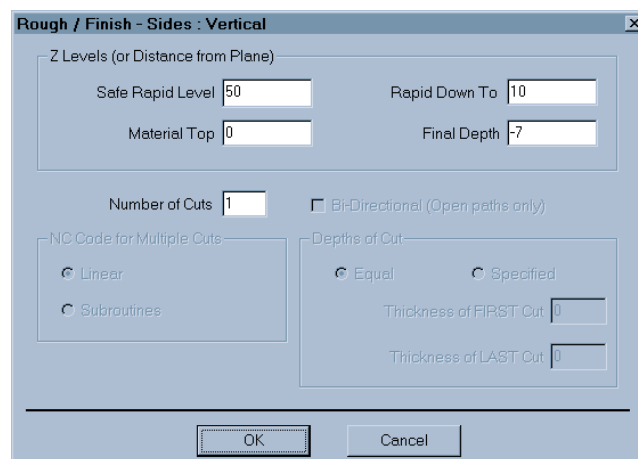
 Press **Enter**  to accept and load the tool.



 Select **MACHINE** | **Rough or Finish** .

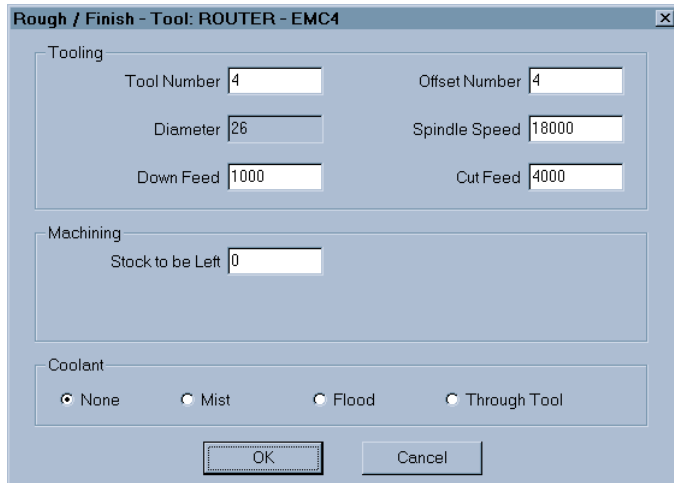
The first dialog box shown differs between the Advanced and Standard modules.





 Click  on the **Selected** button and the second dialog box is displayed.




 Set the options as shown. When complete, click  on **OK** and the third dialog box is displayed.

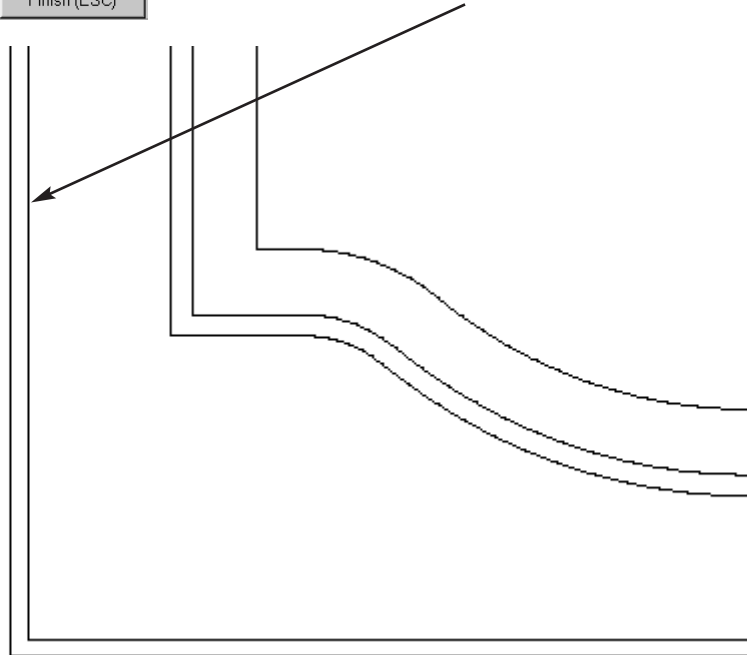


☞ Set the options as shown. When complete, click  on and the fourth dialog box is displayed.

☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click  on

☞ The dialog box disappears and the command line prompts you to select the geometries, left click  on the outer profile top: it turns blue.

Click  on



The toolpath is displayed.

Operation 5

Profile outer geometry with straight cutter

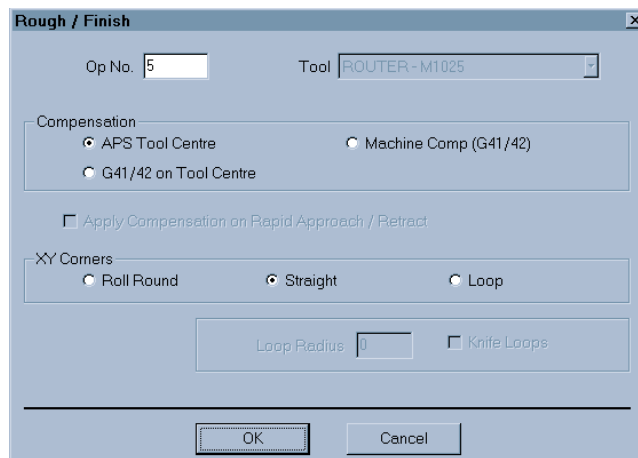
 Select **MACHINE** | **Select Tool** . Select the tool called **Router – M1025**



 Press **Enter** to accept and load the tool.

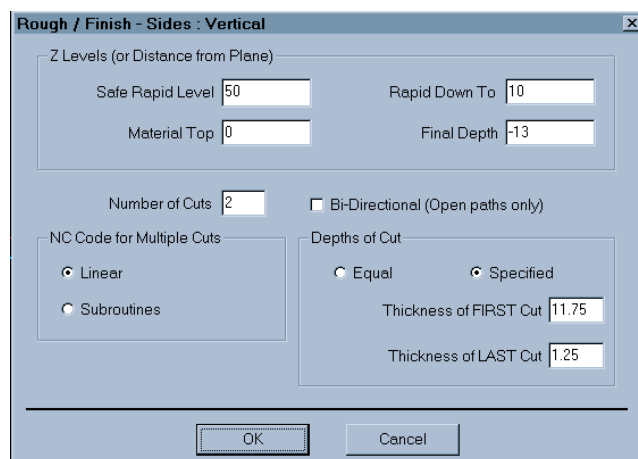
 Select **MACHINE** | **Rough or Finish** .



The first dialog box shown differs between the Advanced and Standard modules

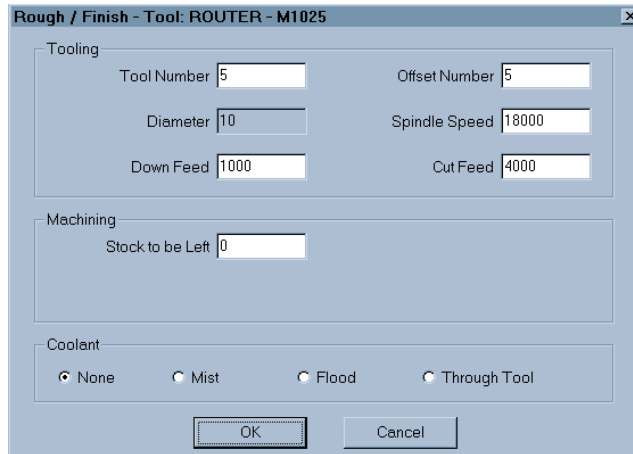
 Click  on the **Selected** button and the second dialog box is displayed.






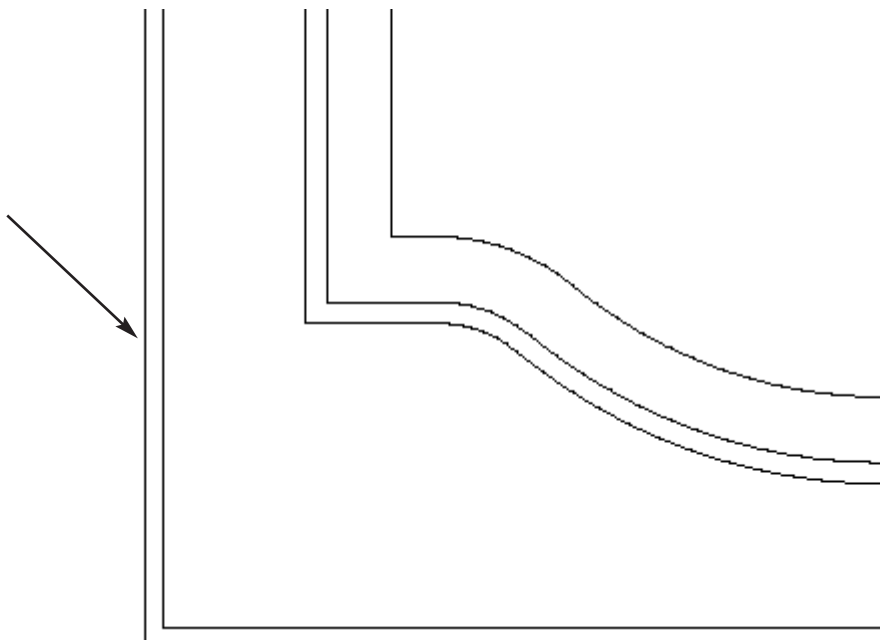
 Set the options as shown. When complete, click  on **OK** and the third dialog box is displayed.



 Set the options as shown. When complete, click  on **OK** and the fourth dialog box is displayed.



- ☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click  on
- ☞ The dialog box disappears and the command line prompts you to select the geometries, left click  on the outer profile: it turns blue.
Click  on

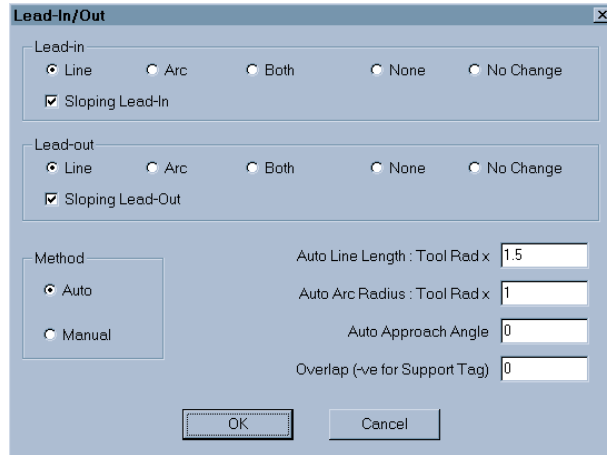


The toolpath is displayed.

Lead In/Out


 Select **MACHINE | Tool - Lead In/Out** 

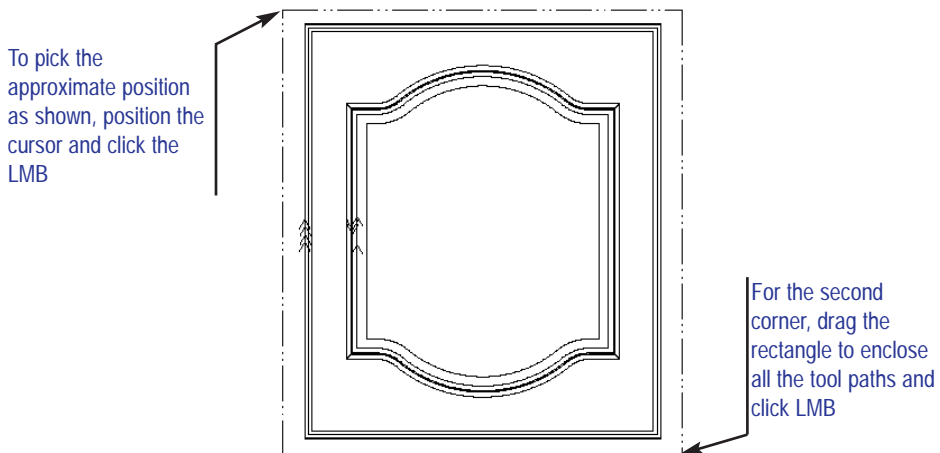
A dialog box is displayed. There are several sections, each setting an aspect of the lead in and/or out that is applied.



 Set the options as shown. When complete, click  on **OK**

These options cause the tool to ramp in and out and the lead angle to be tangential to the cut.

 Select all the geometries by dragging a selection box around the whole drawing. The selected toolpaths turn from green to blue, to indicate that they have been selected.










 Press  to finish the selection and apply tool lead in and out to the selected toolpaths.

VISUALISING THE COMPLETE PROGRAM

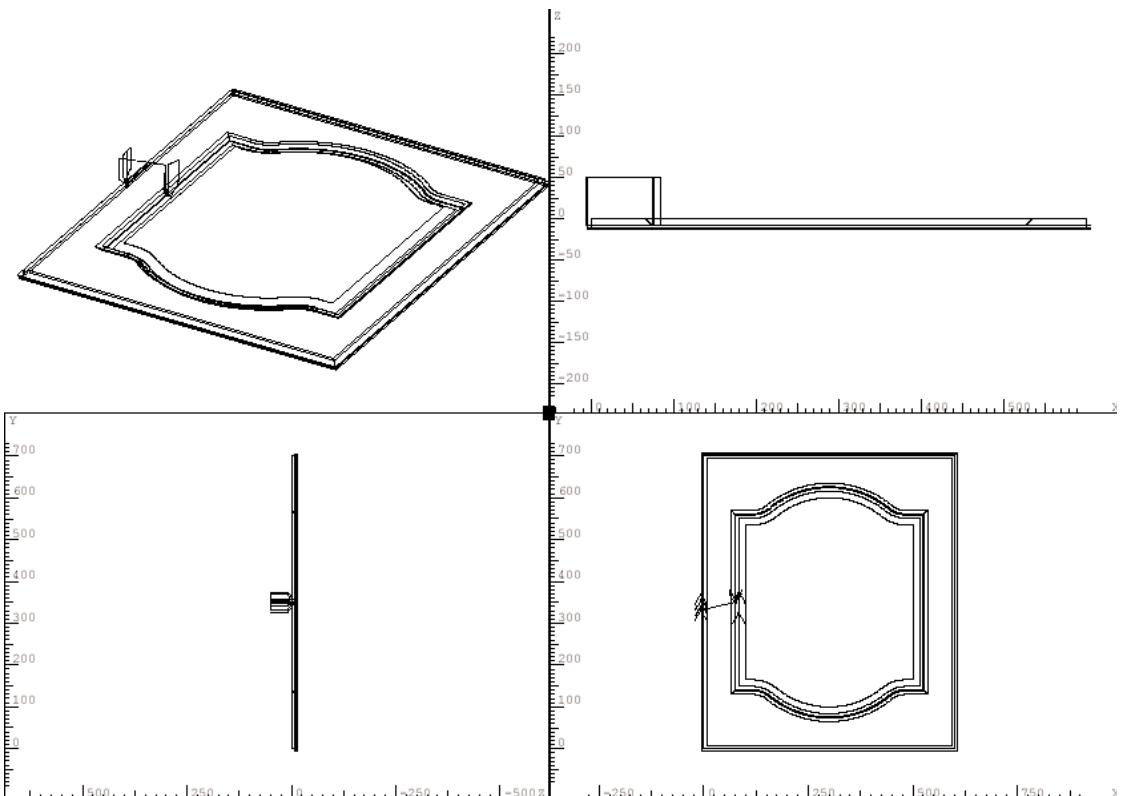
Display Options

At this stage, you may wish to utilise some of the different display options available. The commands can be found in the **VIEW | Display Options** menu.

To see the area of material cut by the tools, select  followed by . To return the display to normal, select  followed by .



To see the tool in an animated display, select ,  and . The screen splits automatically into four views and an animated tool is displayed. To see the animation again, select .

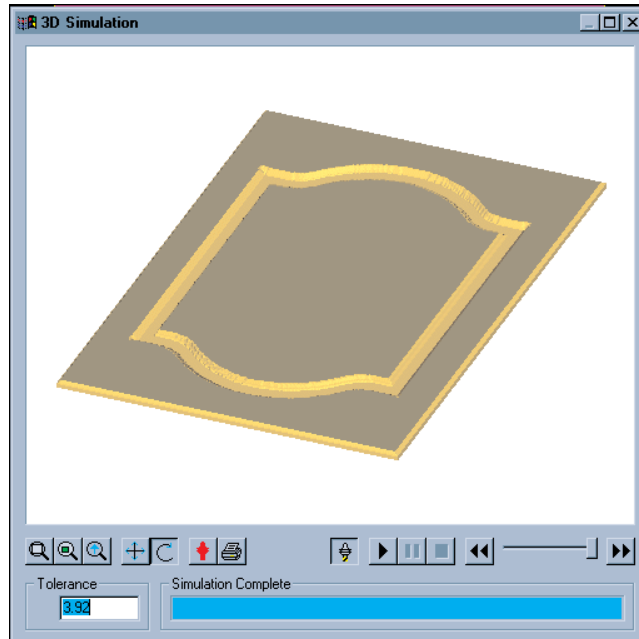
To return the display to normal, select ,  and .



Solid Simulation

 Select **VIEW | 3D Simulation**  to display tool motion removing material from a 'solid block'

 AlphaCAM displays a warning message if you haven't defined a material boundary and offers to define one for you. Click  on to accept this and the 3D simulation window is displayed.




Switching the tool display off  will speed up the simulation results.

Altering the Tolerance to a smaller value will improve the detail of the display.

NOTE:

The tolerance is related to the overall size of the machined part. Making the tolerance too small will have a dramatic effect on the speed and may even cause it to run out of virtual memory and hang the system. E.G. This part is 600mm x 700mm x 12mm setting the tolerance smaller than .5 would cause the system to run very slowly.


AlphaCAM will display the toolpaths removing material from a 'solid' block. This will give you extra confidence that the toolpaths you have created are correct.

When you are ready to conclude the machining, close the 3D simulation by clicking on the  at the top right corner of the 3D simulation window .

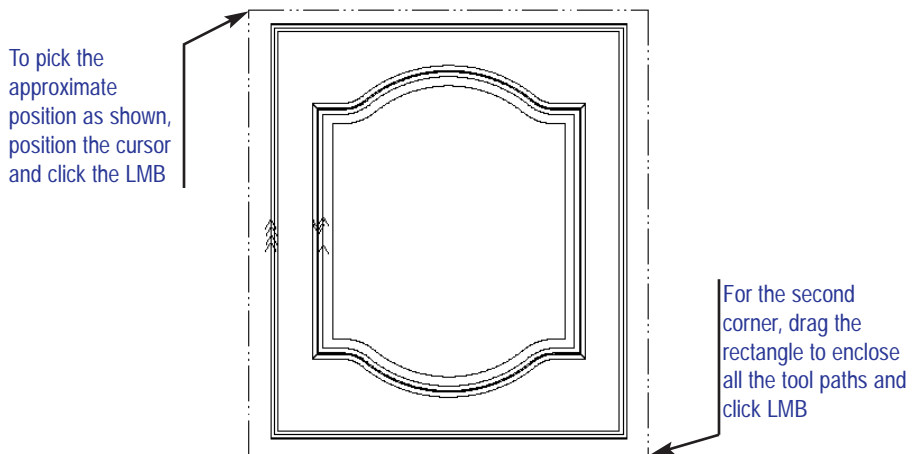
Operation 6

Array the toolpaths to produce 12 doors.

 Select **EDIT** | **Move, Copy etc.** | **Array** 

 The command line prompts you to select what is to be arrayed. Select all the toolpaths by boxing/windowing all the items: they turn blue.

 Click  on or click  on any position on screen.



A dialog box is displayed.


 Select **Toolpaths** Only array the toolpaths.

A second dialog box is displayed.

 Select **Subroutine Copy: with Tool Sorting**

Produce subroutines and cut shapes before changing tool

A third dialog box is displayed, requesting the array values






 Type **3** **625** **4** **725**, then click on or press

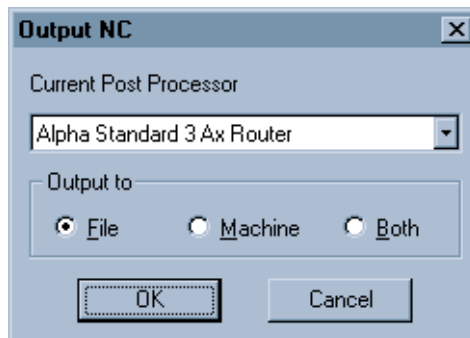
 Select **VIEW** | **Zoom All.** 





The array of toolpaths is displayed. Subroutines are displayed as dashed lines.

Save and Output.

Finally, it is necessary to save the job and output the NC-code.

-  Select **FILE | Save As**: the Save As dialog box is displayed.
-  Select a suitable save in drive and directory.
-  Enter a suitable filename for this job.
-  Select **FILE | Output NC** . The output dialog is displayed.



-  Set the options as shown. When complete, click  on and the Save As dialog box is displayed. The options are set the same as the job details. It is possible to save the NC-Code in the same folder, using the same name as the job, because **AlphaCAM** uses different extensions for drawings (.ard) and NC files (.anc).
-  Either press or click  on to save the NC-Code.

Manipulation of the NC file and dispatch of the NC file to the machine tool is done using

AlphaEdit + RS232 Comms 