AlphaCAM

CAD/CAM system for Windows™

3D MACHINING TUTORIAL

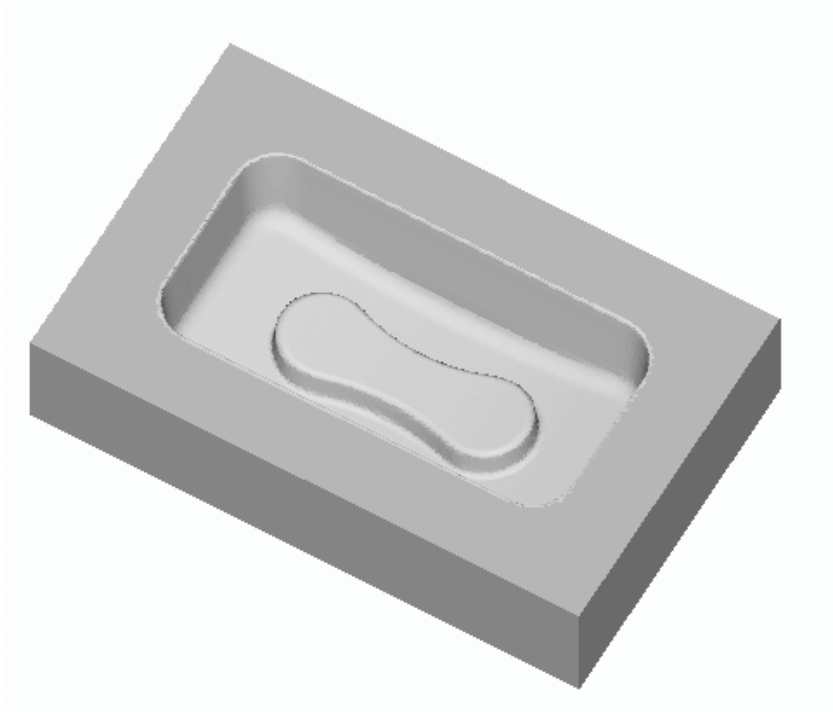


Table Of Contents.

Conventions When Using The Tutorial

1. 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.
2. The symbol  indicates a new command for you to action.
3. The symbol  indicates a sub part of command for you to action

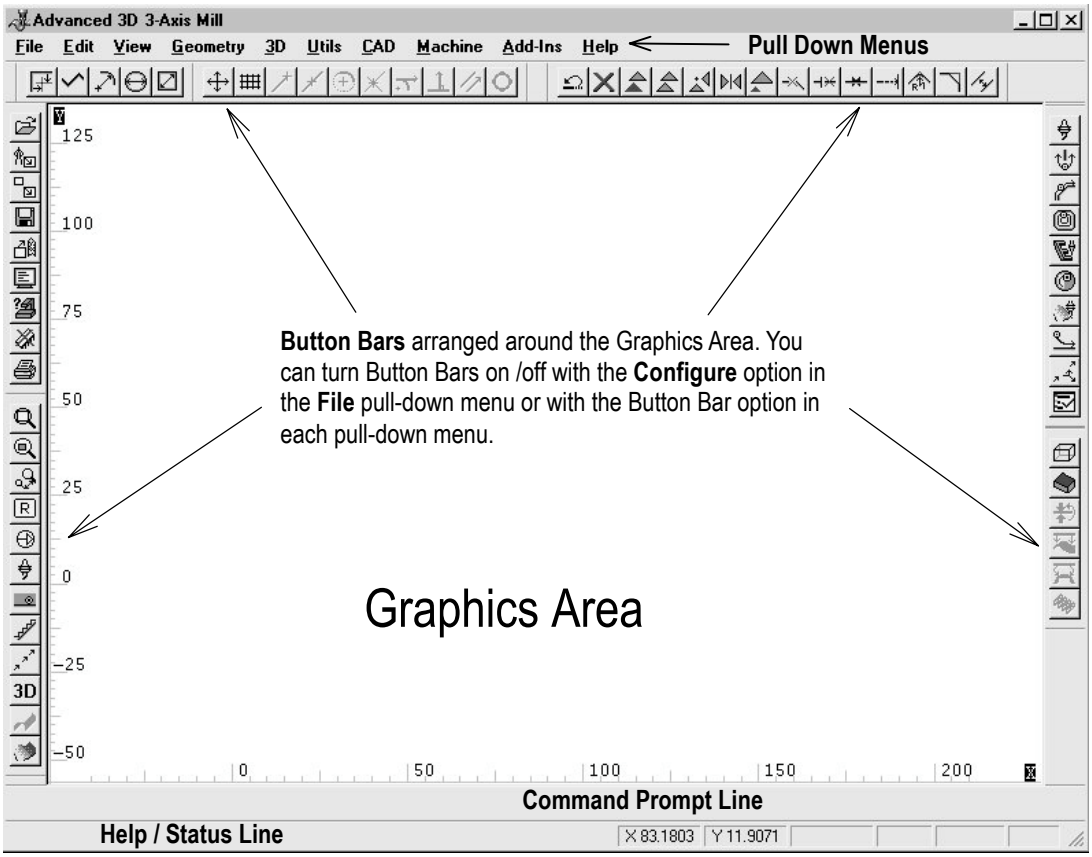
The HELP file is very comprehensive. When you have finished the tutorial, please take time to have a look at it. Select **HELP** | **Contents**.

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 3D 3 or 5-Axis Mill / Router**'. This will take you into the graphical portion of the system. Your screen will look similar to the one below.



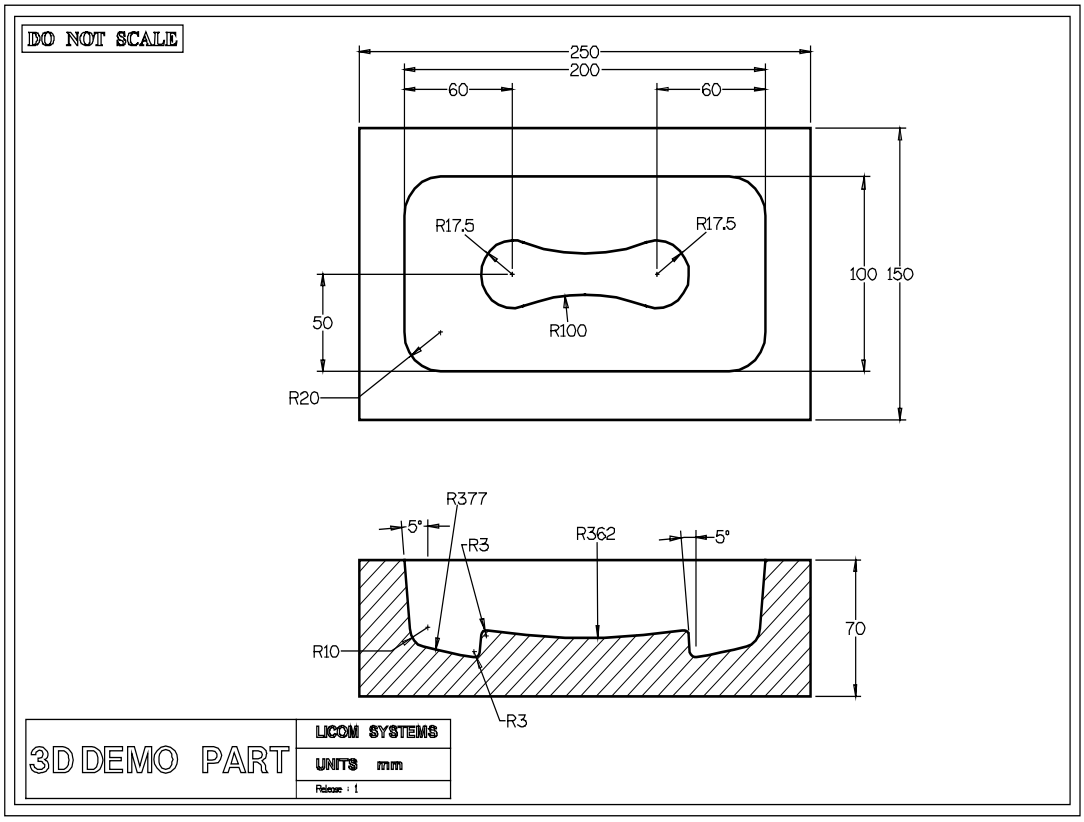
During this tutorial, we tell you where to find commands on the pull-down menus. If there is a button for the command, this will also be shown.

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.

Creating 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.

Open the Demo Drawing.

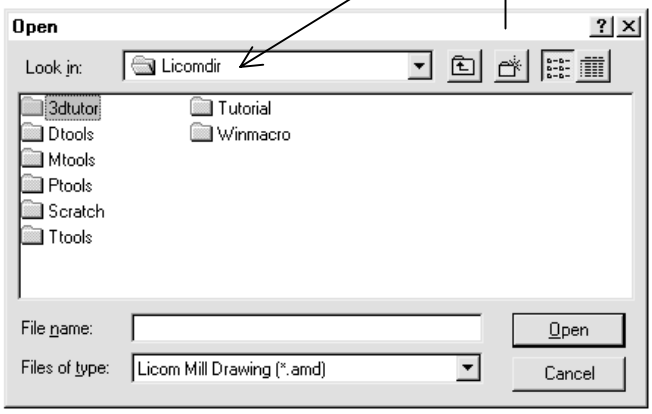
For this demonstration tutorial it is not a requirement to create the geometric surfaces. These have already been created in the sample drawing provided with the system.

Select **File** | **Open** .

The Open dialog box is displayed.

Click Here to Change the Current Drive and Directory.

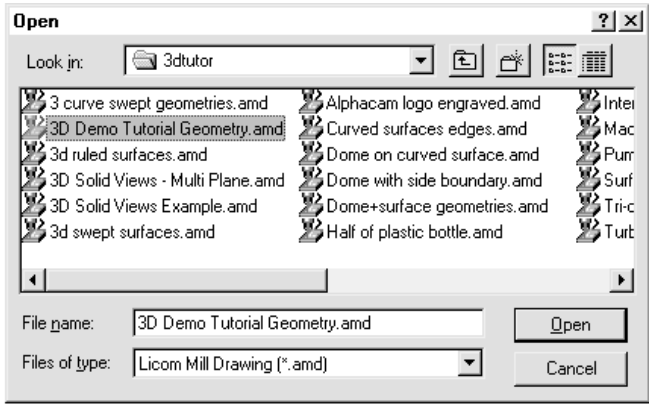
Click here to select the correct directory



Click on **Open** to change to the selected directory.

Click on **3D Demo Tutorial Geometry.amd** to select the drawing.
If the drawing is not on your system it can be downloaded from the training page of the web site. **www.licom.com**

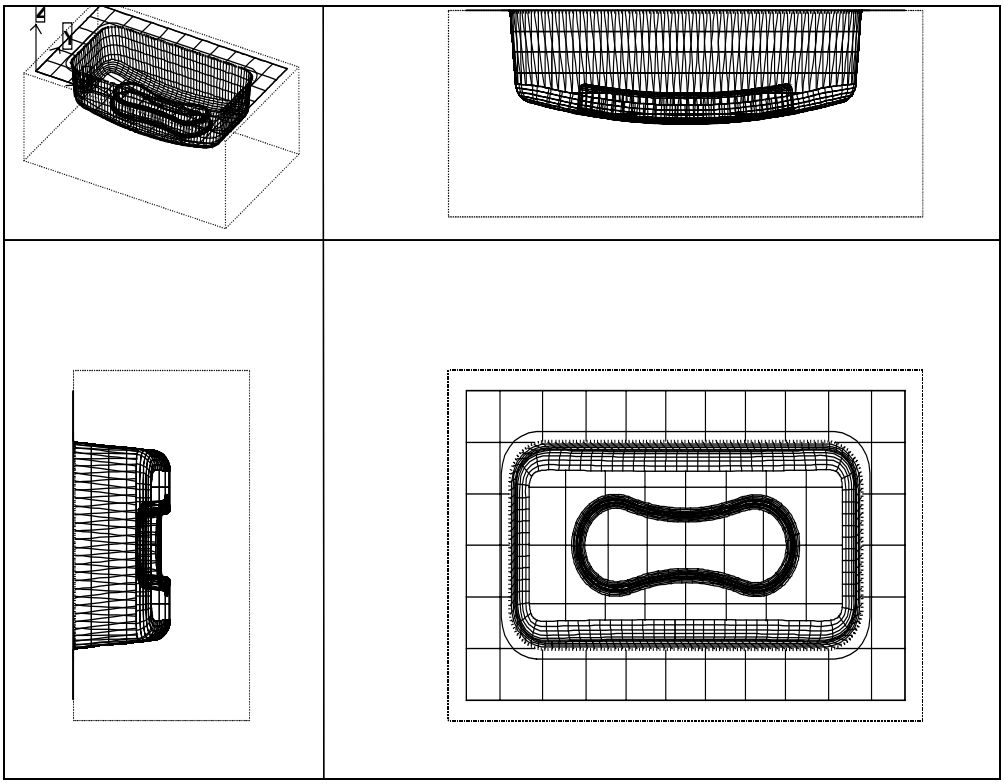
Select the demo drawing here



Click on **Open** to load the drawing into memory.

A warning box is displayed, as some layers have been turned off to avoid confusion.

Click on **OK** and the drawing is displayed.



Machining Operations

With 3D machining, components may be cut in a variety of ways in order to achieve the results required.

The following is a sample list of items that affect the method of machining -

The nature of the component being cut.
Component material (free cutting, brittle, work hardened).
Cutting tool type and material.
Method of work holding.
The coolant delivery.
The surface finish required.
The surface finish achievable.
The cutting time (cost of machining compared with cost of other finishing processes).
The machine tool rigidity and repeatable accuracy.
The part accuracy required (tolerance achievable).

Some parts require all the surfaces to be machined together, as in this example. Others will require the individual surfaces to be machined separately.

For the demonstration example, the cutting is produced in 3 operations, machining all surfaces.

The tooling and number of cuts will obviously depend on the nature of the material being cut, its usage and any work holding. The function of this tutorial is to show how different cutting strategies are applied to 3D surfaces.

For some jobs, the machining will require 2D and 3D strategies to be applied prior to the 3D machining toolpaths.

Operation 1

Z Contour Roughing Flat 20mm 2F EC SCB. End cutting roughing tool

The Z Contour roughing operation is used to remove the bulk of the material quickly.

Operation 2

Rough Machine Surface Flat 10mm 2F EC SCB. Ball End roughing tool

The rough machine surface operation is used to remove the excess material left by the Z contour roughing and leave a uniform surface for the finishing operation.

Operation 3

Finish Machine Surface Flat 5mm 2F EC SCB. Ball End finishing tool.

The finish machining operation is used to produce the component to the required finish and size.

Machining Times and Equipment

3D machining and computer-generated solid view displays require large amounts of memory and disc space.

The time taken to produce your work will depend largely on the specification of the system on which you are running AlphaCAM.

The version of Windows you are using will also have an effect. For example, 3D toolpaths and solid views are created much quicker on a computer running the Windows NT operating system than on a similar system running the Windows 95/98 systems.

Set-Up Options

Select **FILE** | **Select Post**

AlphaCAM displays a dialog box showing the available post processors. Select the post processor for your machine. The post processor determines the format of the output code. AlphaCAM displays a warning if values entered exceed the maximums set in the post processor.

Select **MACHINE** | **Select Material**

AlphaCAM displays the dialog box with all the materials currently specified in the material library. Feeds and speeds is determined automatically based on the material and tools used.

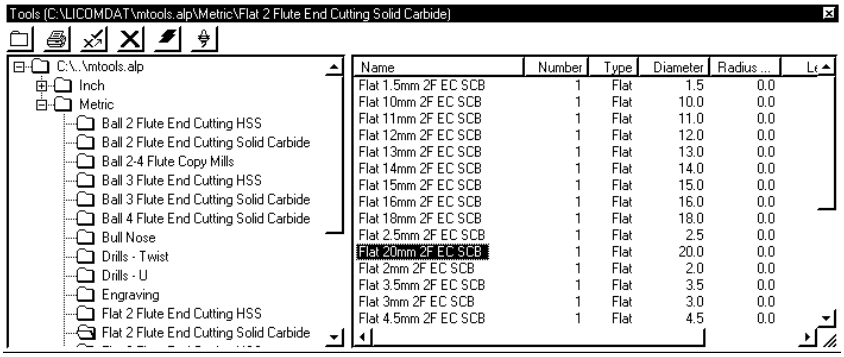
Select a suitable material and click on **OK**

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

AlphaCAM displays the Select Tool dialog box, listing all the tools currently defined on the system. Open the Metric Tool folder by clicking on the + symbol next to the folder. This displays all the folders within the metric tool library.

Note: selecting the title box of each column modifies the tool library order. For example to sort the tool library into diameter and type order, click on the Diameter title and then on the Type title.

Open the "Flat 2 Flute End Cutting Solid Carbide" folder, scroll down the tools displayed and click on the **Flat - 20MM 2F EC SCB.**



(Alternatively, you may select a suitable tool from your own library).

Whichever you choose, click on in the Select Tool dialog box and the tool is displayed on your screen.

AlphaCAM asks. "**This Tool? <ENTER> = ACCEPT, <ESC> = ABORT**"

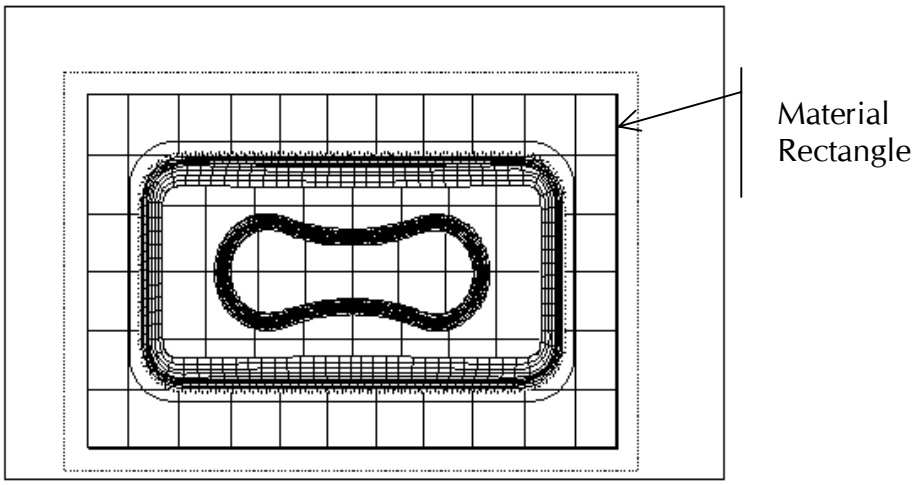
This is to ensure that you have selected the correct tool from the library.

The tool diameter and cross-section are shown, so that you can verify that you have chosen the correct tool. Press Enter to accept the tool.

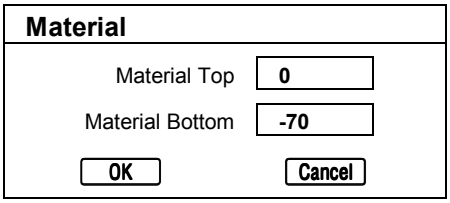
Z Contour Roughing is a strategy used to quickly remove the bulk of material from the workpiece. For this reason it is necessary to create a volume which represents the material outline. This must be done before the Z Contour Roughing strategy is selected.

Select **3D** | **Set Material Size**

AlphaCAM prompts "**MATERIAL: Pick Outline of Material**". Click on the outer edge of the rectangle surface as indicated below.



The Material dialog box is displayed.



After entering the values shown, click on OK. The material cube is displayed in white.

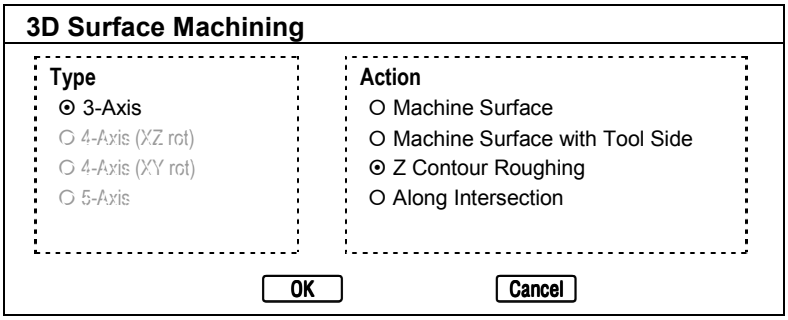
Operation 1

Z Contour Roughing.

For this operation the tool must be end cutting.

Select **MACHINE** | **3D Machining**

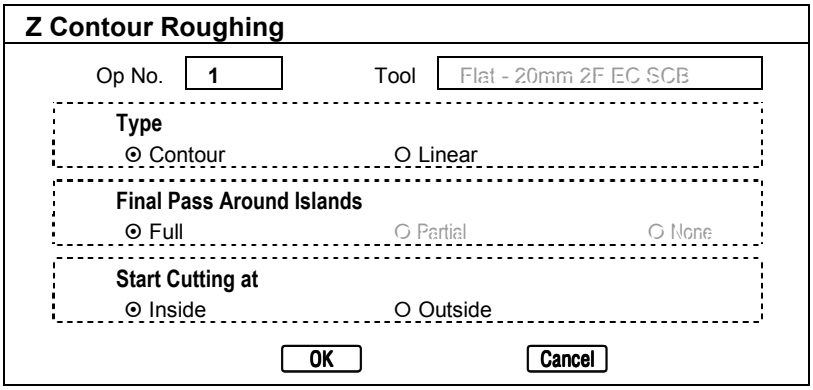
AlphaCAM displays the first 3D Surface Machining dialog box.



Note: If the module you are working on is not 5 axis, the "**Type**" section of the dialog box is not applicable and is not displayed.

After setting the options as shown, click on OK.

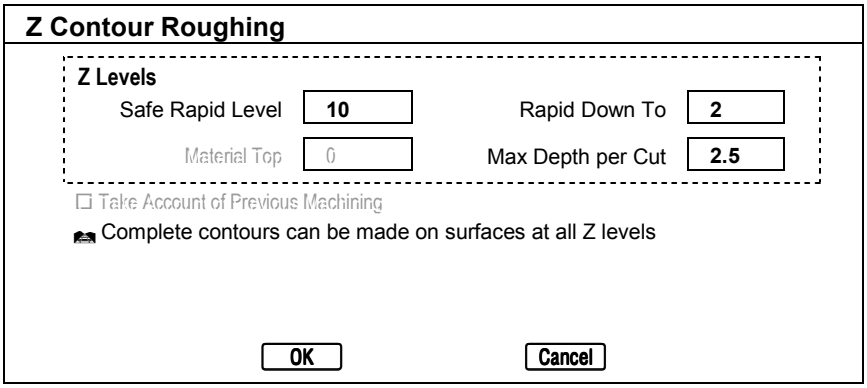
AlphaCAM displays the first Z Contour Roughing dialog box.



After setting the options as shown, click on OK

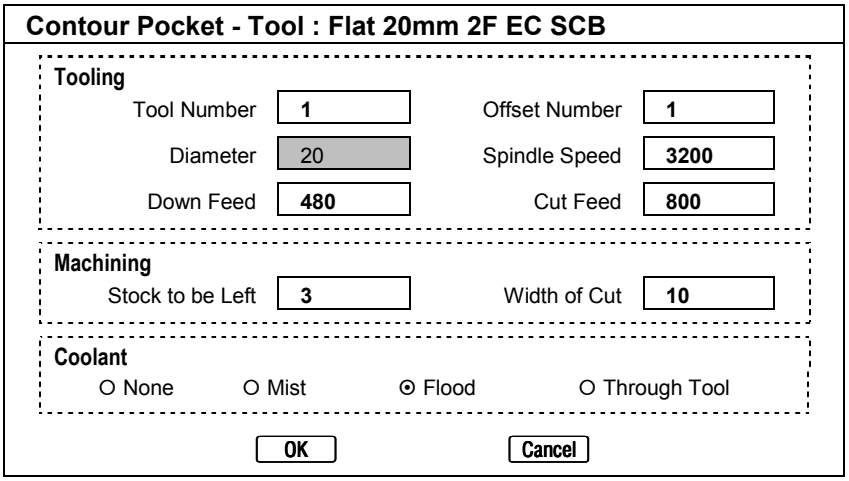
NOTE - Remember to use Tab to move between each field. Pressing Enter accepts the current values and the next dialog is displayed.
If Enter is pressed by mistake, selecting Cancel returns to the previous dialog box.

AlphaCAM displays the second Z Contour Roughing dialog box.



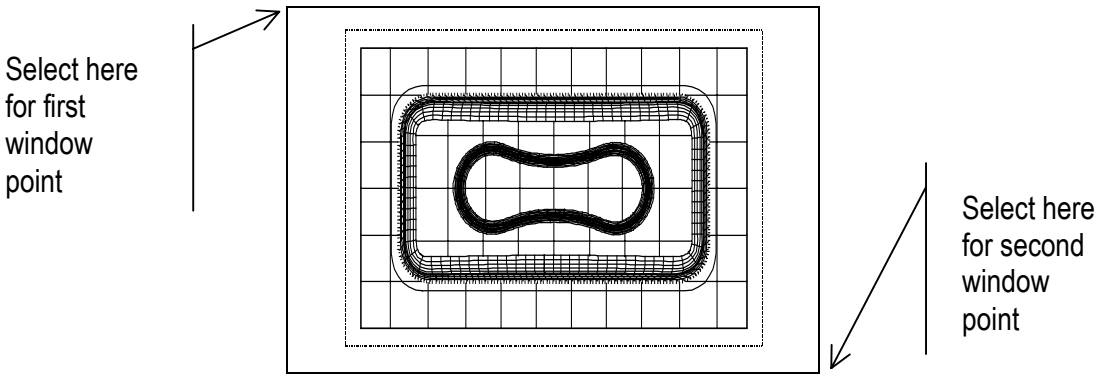
After entering the values as shown, click on [OK].

AlphaCAM displays the Contour Pocket dialog box.



AlphaCAM prompts "**ROUGH SURFACE: Pick Surfaces**".

Select all the surfaces. This can be done by windowing all the surfaces in the X, Y view (i.e. by clicking and releasing the left mouse button, moving the selection box to encompass all the surfaces, and clicking and releasing the left mouse button again). Check in the ISO view that all surfaces have been selected.



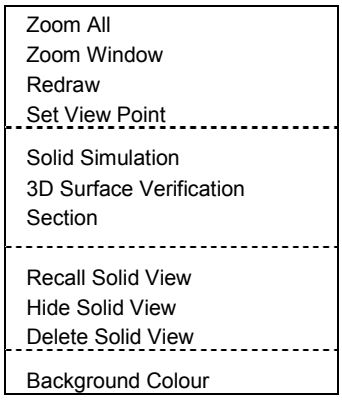
When all the surfaces have been selected (displayed in Blue), click on Finish Esc. The Z Contour Roughing toolpaths are created.

Select **VIEW** | **3D Solid Views** | **3D Surface Verification** .
Accept the default values in the dialog box displayed and click on OK. The solid view is displayed in the ISO view.

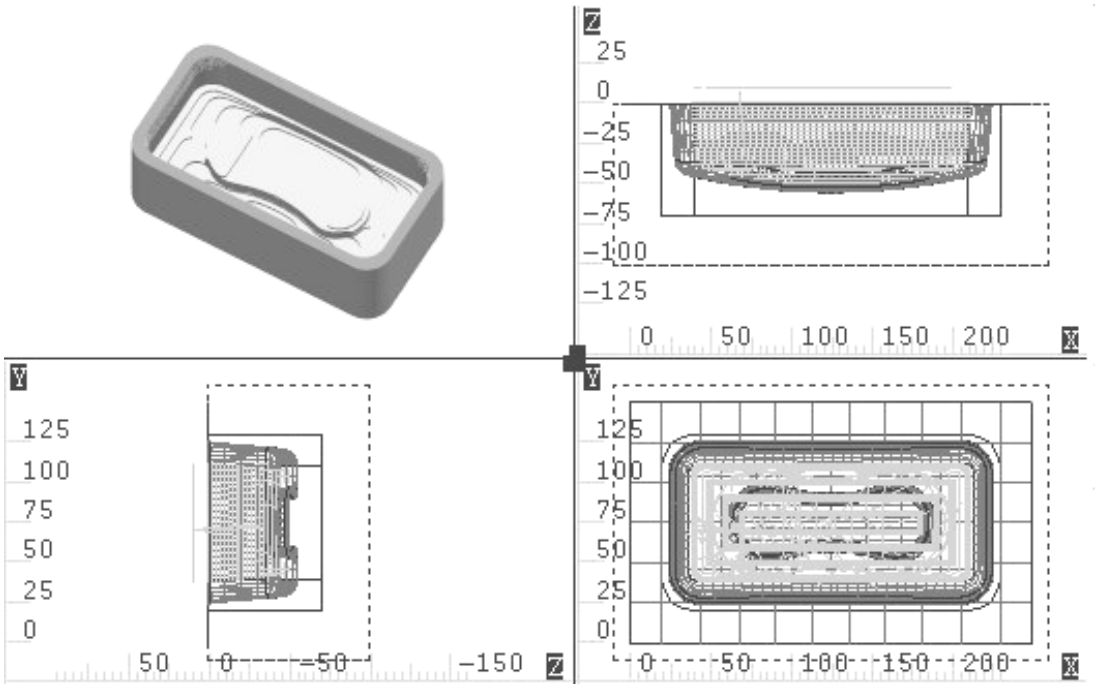
The viewing angle of the solid view can be altered using the keyboard **ARROW** keys, or by selecting **VIEW** | **Set Viewpoint** , which can also accessed from the ISO view menu.

Clicking the Page Up and PageDown keys while depressing the Ctrl key activates the Zoom Out and Zoom In commands in the ISO view.

To display the ISO view menu, position the cursor on the ISO view and click RMB. The menu is displayed.



The **Set View Point** option allows the ISO view to be rotated by selecting the viewing angle in the X, Y and X, Z view ports.
Selecting **Delete Solid View** returns the display to normal.
Selecting **Set Viewpoint, then click** on **Reset** or Spacebar returns the display to the normal ISO view.



Try using the display options to manipulate the views and display.

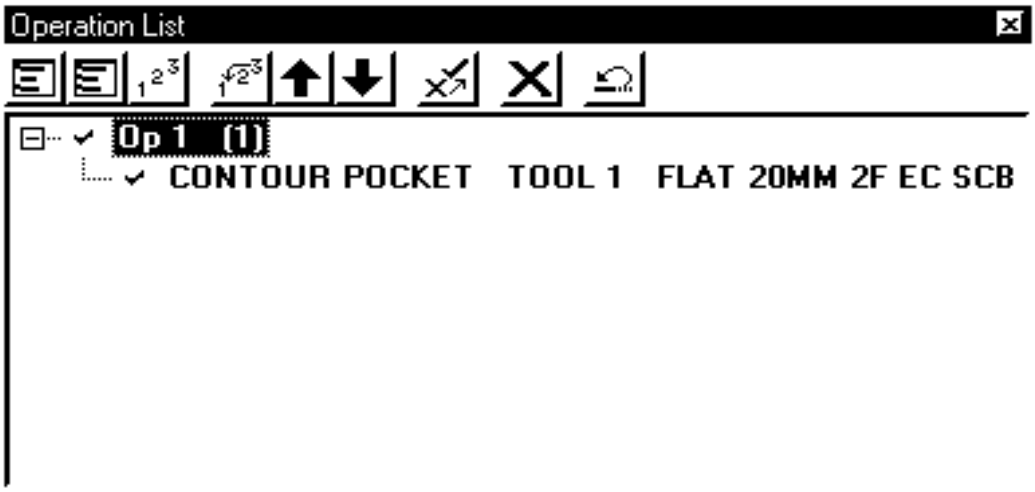
The NC Code is created at the same time as the toolpaths.

Select **FILE** | **List NC Code** to see the NC Code.

If you have altered the display, reset the view display and orientation to normal before continuing. Resetting these options is not mandatory: it is just to maintain consistency with the notes.

Before creating the toolpaths for operation 2, the toolpaths for operation 1 may be switched off to clarify the display.

To switch off toolpaths, select **MACHINE** | **Edit Operations** and click on **Hide All**.



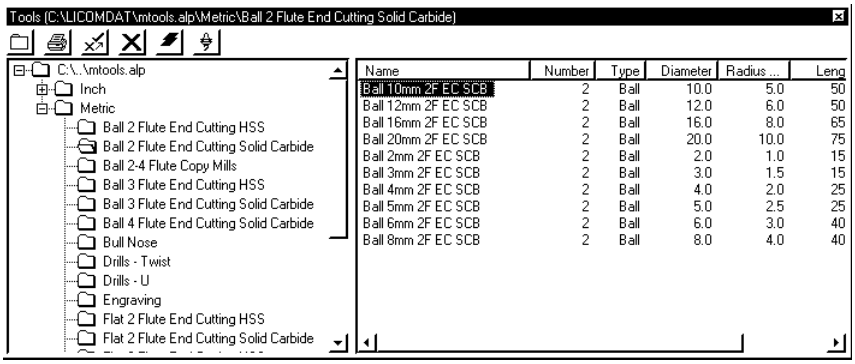
If only one view clears, click LMB in that view, so it has a thin line border and then select **VIEW** | **Redraw**.

Operation 2

Rough Machine Surface

This operation is to produce a uniform surface in order to achieve even cutting conditions for the finishing cut

Select **MACHINE** | **Select Tool** .



AlphaCAM displays the Select Tool dialog box, listing all the tools currently defined on the system. It displays the library for the current tool. Open the "Ball 2 Flute End Cutting Solid Carbide" folder, scroll down the tools displayed and click on the **Ball - 10MM 2F EC SCB.**
(Alternatively, you may select a suitable tool from your own library).

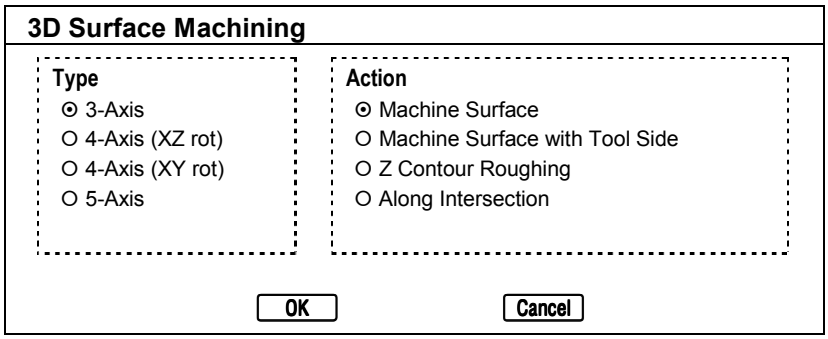
Whichever you choose, click on and the tool is displayed on your screen.

AlphaCAM asks. "**This Tool? <ENTER> =ACCEPT, <ESC> =ABORT**"

Press Enter to accept the tool.

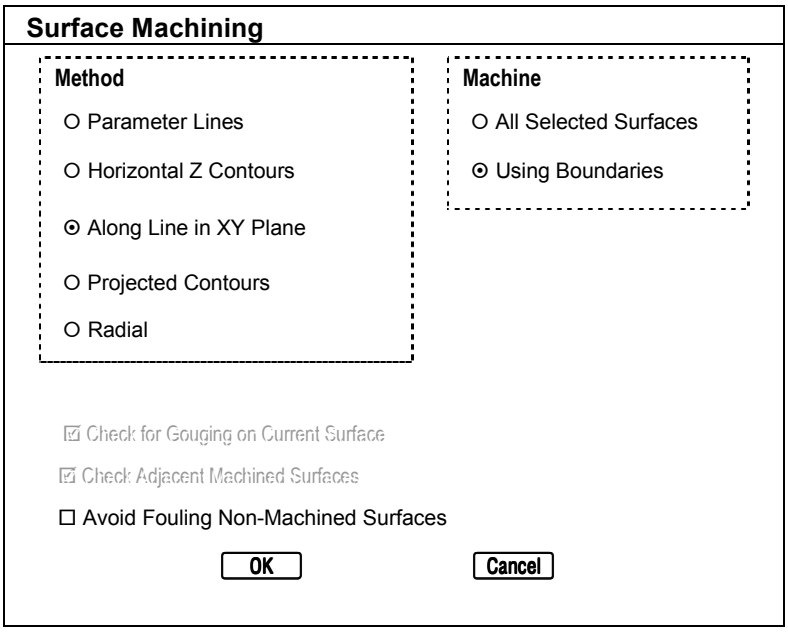
Select **MACHINE** | **3D Machining**.

AlphaCAM displays the first 3D Surface Machining dialog box.



After setting the options as shown, click on OK.

AlphaCAM displays the next Surface Machining dialog box. This allows you to choose the type of surface machining strategy you wish to apply.



Parameter lines: This machines a single surface along in the direction of the parameter lines, which define the surface. You are asked to select a point near the start of the machining. This does not have to be accurate, as AlphaCAM automatically searches from the point picked to find the nearest correct start point i.e. the corner or edge of the surface. You are then asked to pick another point to indicate the direction in which the machine is to travel and this decides which set of parameter lines the machining will follow. If the entire surface does not require machining, then a boundary may be drawn to limit the machining only to the area within the boundary. This may be a hard or soft boundary, but machining will only take place on the inside of the line.

Horizontal Z Contours: In this case, AlphaCAM looks for the highest point on the surface, regardless of where you picked as a start point, and starts the machining from there. This results in a series of horizontal contours of the tool tip. As boundaries can not be used with this strategy it is only possible to machine the entire surface (or surfaces) selected.

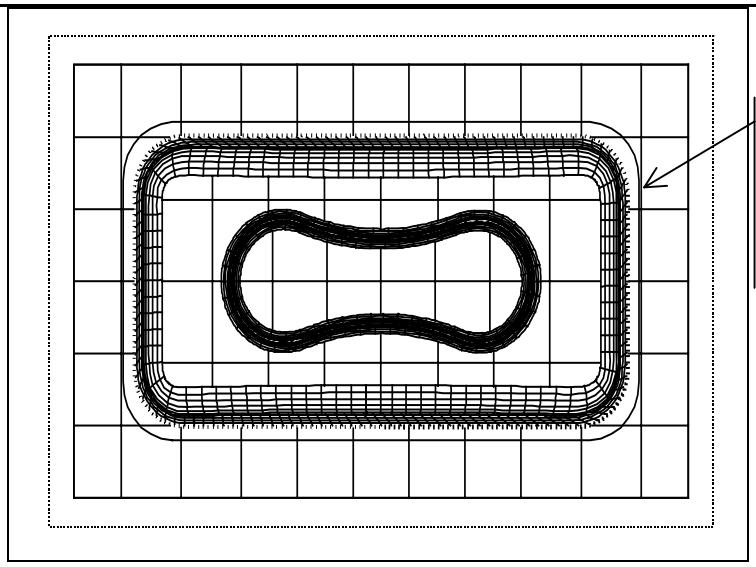
Along Line in XY Plane: With this strategy, it is possible to select the angle along which the tool is driven while the surface is machined. This applies to the tool tip as opposed to the contact point of the tool with the surface. Boundary geometries may be used to limit the extent of the machining, if required

Projected Contours: This method of surface machining must have a boundary, which will then be machined using a contour pocketing strategy i.e. with a constant stepover. The toolpaths generated are then projected down until the contact point of the tool meets the surface.

Radial: As above, this procedure must have a boundary. The toolpaths are generated in a radial fashion from a central point outwards until the boundary is met. You may determine both the angle for each stroke and the start and finish angles for the machining. The centre point for the machining is asked for after the dialog boxes have been filled in.

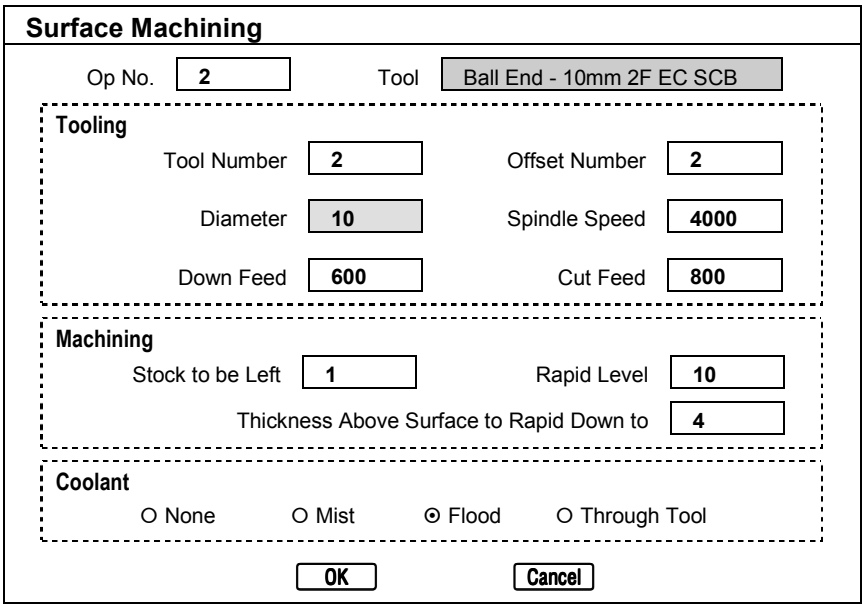
After setting the options as shown opposite, click on [OK].

AlphaCAM prompts, "**SURFACE MACHINING Select Boundaries**".



Use the LMB to pick the green rectangle with rounded corners in the XY view

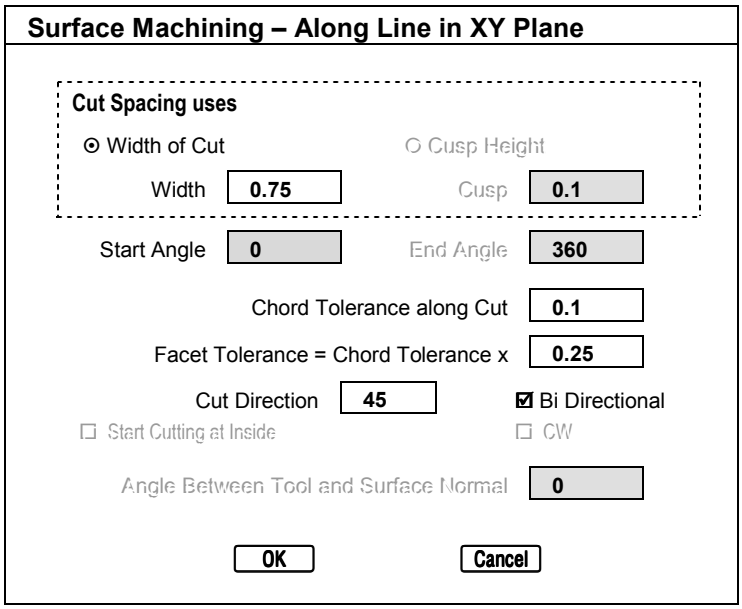
AlphaCAM displays the next Surface Machining dialog box.



It is worth noting that the figure in the "Rapid Level" field is the absolute value from the **global** datum.

After setting the options as shown, click on OK.

AlphaCAM displays the next Surface Machining dialog box.



After setting the options as shown, click on OK.

AlphaCAM prompts **FINISH SURFACE: Pick Surfaces**

Click on **Previous**: this selects the surfaces picked in the previous command.

Click on **Finish [Esc]** (or press Esc or RMB) and the toolpaths are generated.

Before creating a solid view of the results of operation 2, the toolpaths for operation 1 may be switched back on.

To switch on the toolpaths:

Select **MACHINE** | **Edit Operations** and click on **Show All**.

Select **VIEW** | **3D Solid Views** | **3D Surface Verification**. A dialog box is displayed. Click on OK. The solid view is displayed in the ISO view.

Using the display options as described on page 13, alter the viewing directions and zoom to see the result of this machining operation.

The NC Code for this operation is created` at the same time as the toolpaths.

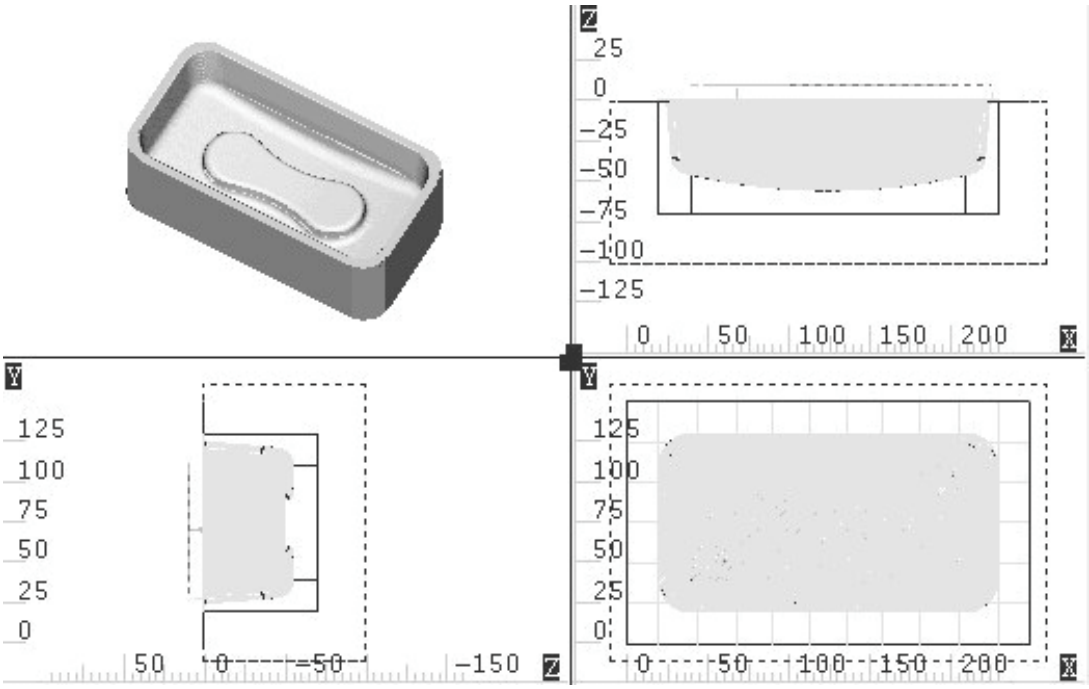
To see the NC Code, select **FILE** | **List NC Code**.

If you have altered the display then reset the view display and orientation to normal before continuing as described previously. Resetting these options is not mandatory: it is just for consistency with the notes.

Before creating the toolpaths for operation 3, the toolpaths for operations 1 and 2 may be switched off.

To switch off the toolpaths:

Select **MACHINE** | **Edit Operations** and click on **Hide All** .If only one view clears, click LMB in that view, then select **VIEW** | **Redraw**

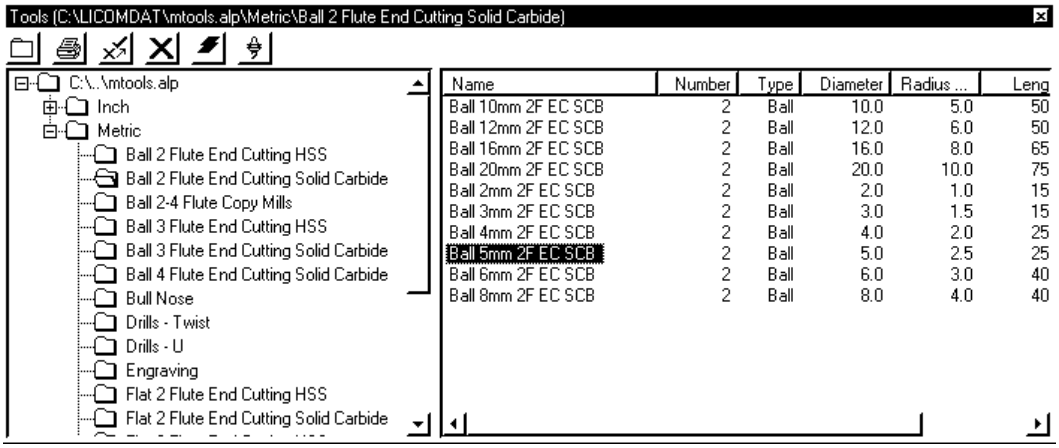


Operation 3

Finish Machine Surface

This operation is to produce the component to the required finish and size.

Select **MACHINE** | **Select Tool**.



AlphaCAM displays the Select Tool dialog box, listing all the tools currently defined on the system. It displays the library for the current tool. The next tool is from the same folder. Scroll down the tools displayed and click on the **Ball - 5MM 2F EC SCB.**
(Alternatively you may select a suitable tool from your own library).

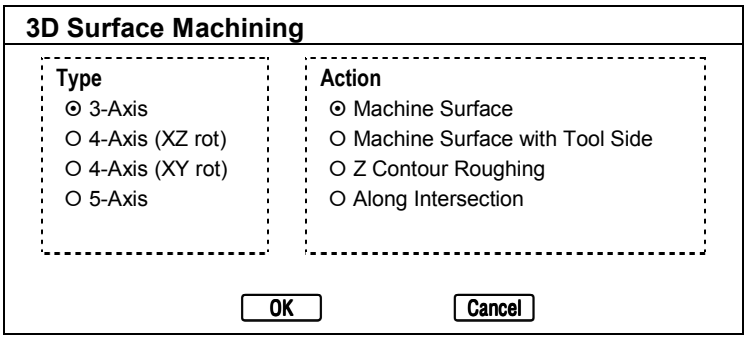
Whichever you choose, click on and the tool is displayed on your screen.

AlphaCAM asks. "**This Tool? <ENTER> =ACCEPT, <ESC> =ABORT**"

Press Enter to accept the tool.

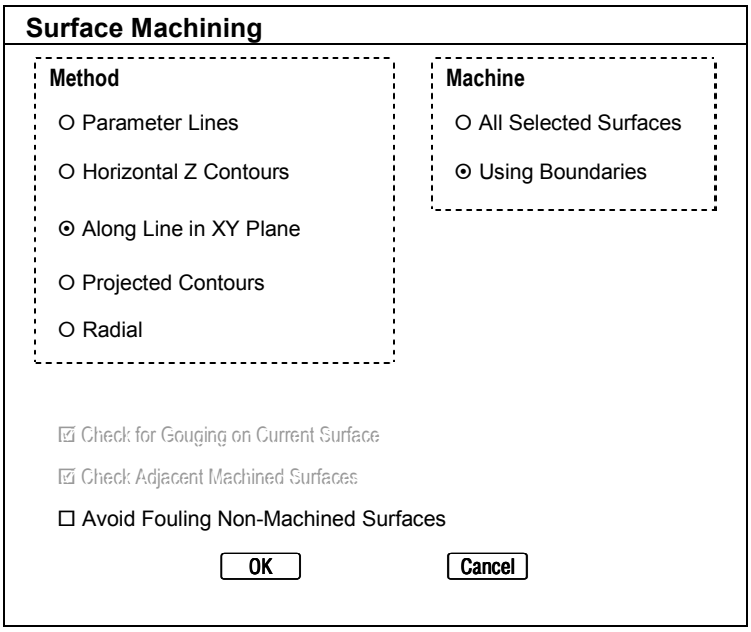
Select **MACHINE** | **3D Machining**

AlphaCAM displays the first 3D Surface Machining dialog box.



After setting the options as shown, click on OK.

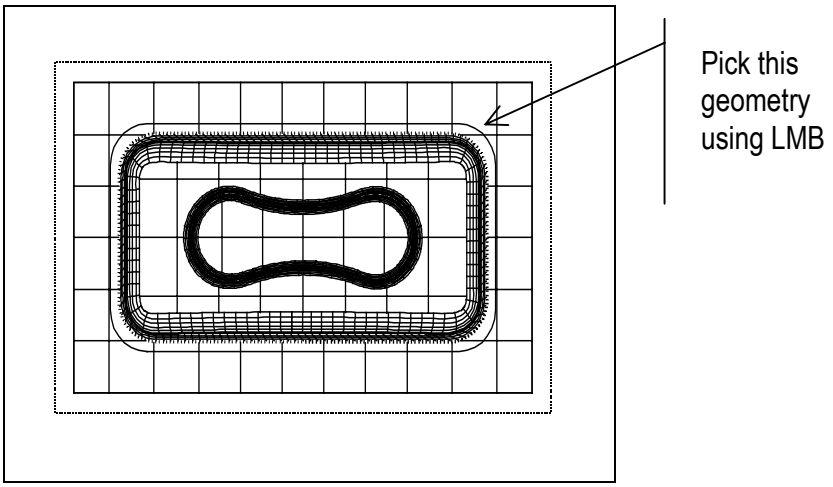
AlphaCAM displays the first Surface Machining dialog box.



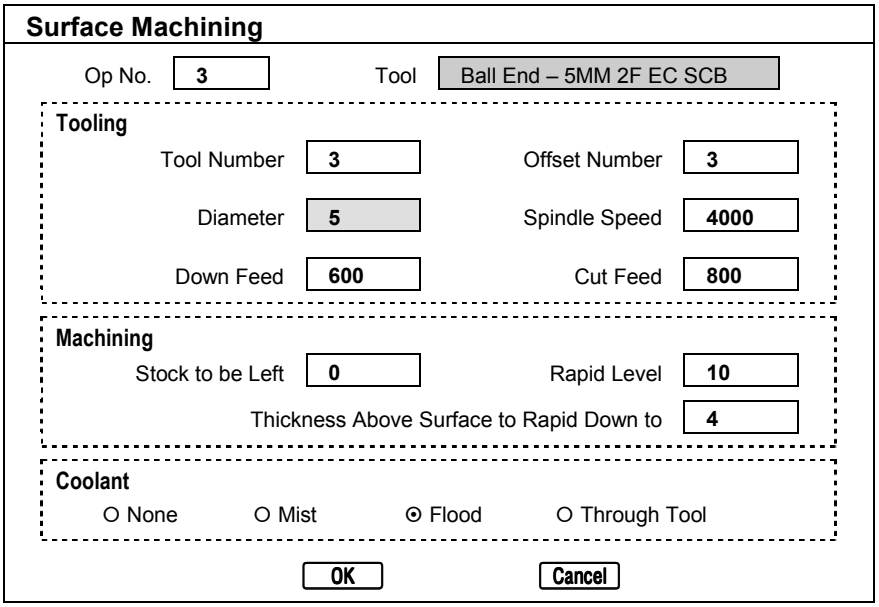
After setting the options as shown, click on OK.

AlphaCAM prompts," **SURFACE MACHINING Select Boundaries**".

Use the LMB to pick the green rectangle with radius corners in the XY view

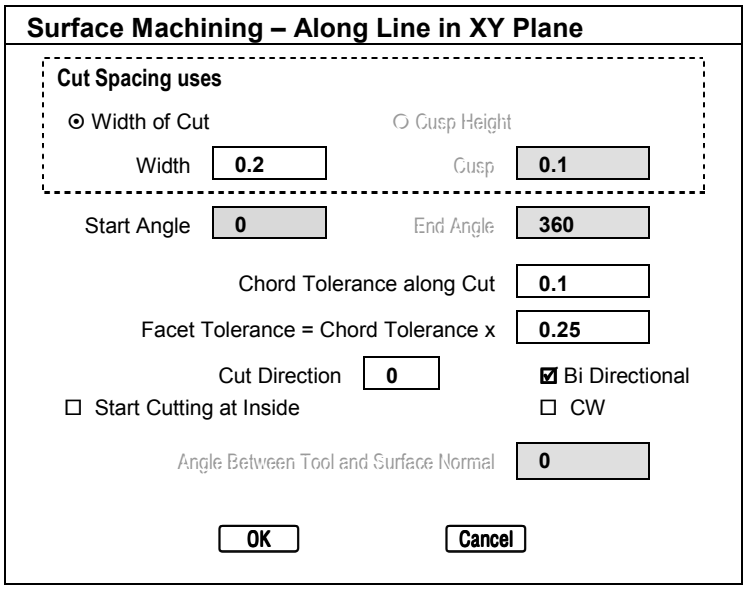


AlphaCAM displays the second Surface Machining dialog box.



After setting the options as shown, click on OK.

AlphaCAM displays the third Surface Machining dialog box.



After setting the options as shown, click on OK .

AlphaCAM prompts "**FINISH SURFACE: Pick Surfaces**".

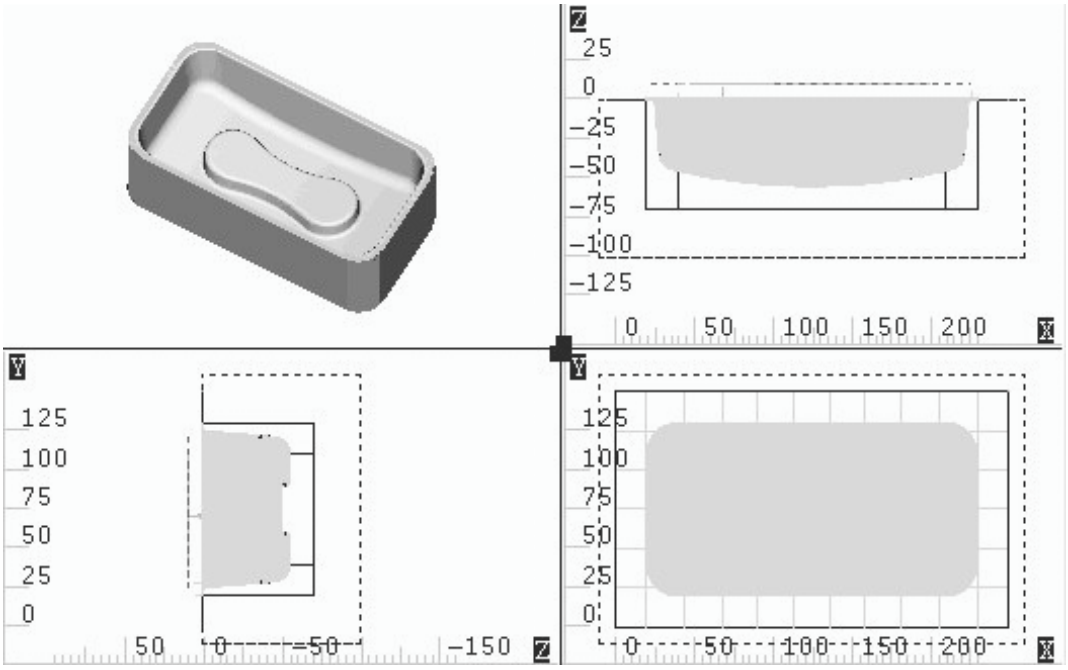
Click on **Previous**: this selects the surfaces picked in the previous command.

Click on **Finish [Esc]** (or press Esc or RMB) and the toolpaths are generated.

Switch on the toolpaths for ops 1 & 2, as described on page 19.

Select **VIEW** | **3D Solid Views** | **3D Surface Verification**. . A dialog box is displayed. Click on OK. The solid view is displayed in the ISO view.

Using the display options as described on page 12, alter the viewing directions and zoom to see the result of this machining operation.



The NC Code for this operation is created at the same time as the toolpaths.

To see the NC Code, select **FILE** | **List NC Code**

To switch on all toolpaths, select **MACHINE** | **Edit Operations** and click on **Show All**, then OK.

If only one view clears, click LMB in that view, then select **VIEW** | **Redraw**.

This concludes the tutorial. As you can see, AlphaCAM has easily guided you through the CAM process. A reasonably complex example like this part would have taken many hours to program manually, instead of just a few minutes. Moreover, you would have had to proof the program on the machine too. Feel free to experiment with the other commands within the AlphaCAM system.