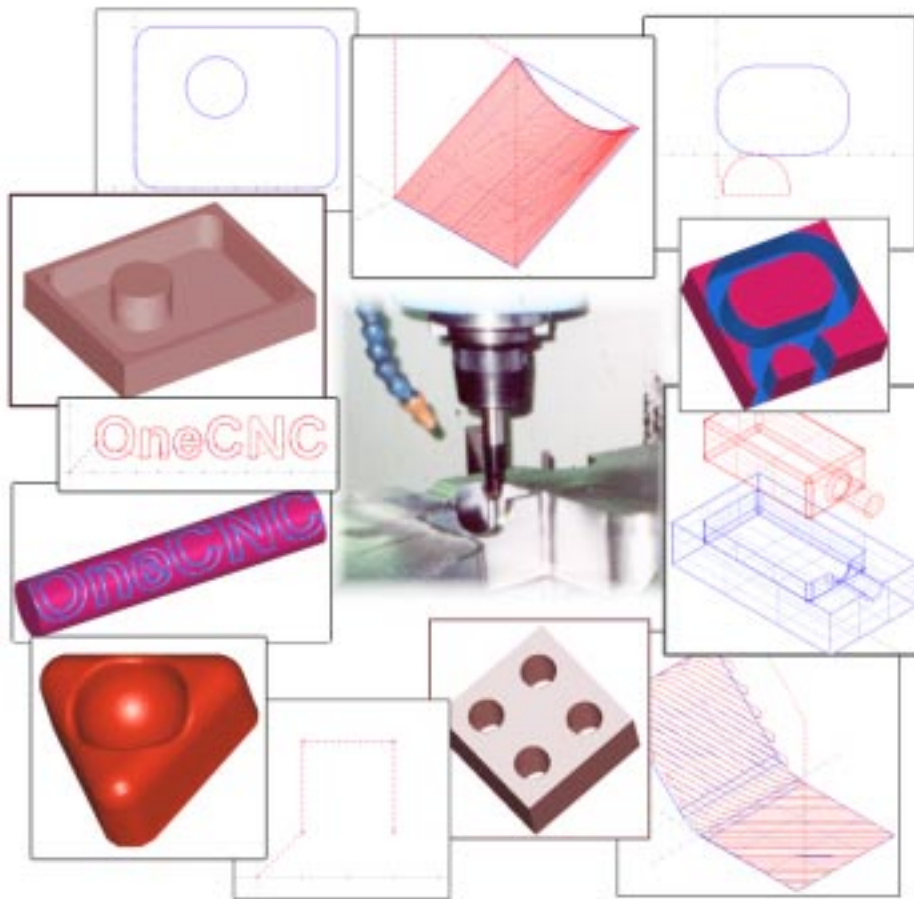


OneCNC

2000 Series

Manufacturing Software



Tutorials

For Mill Professional

Contents

<i>Tutorial</i>	<i>Page</i>
Use 2 ½ D Pocket Function	3
Draw 2D and Drill with Mill CAM	12
Draw 3D Surface and Mill Surface with Mill CAM	17
Draw 3D Surface Multi surface with Mill CAM	30
Draw Solid and Cavity Machine with Mill CAM	45
Draw 2D and Profile with Mill CAM	68
Draw Solid Z-level rough and finish with Mill CAM	73
Draw Text and use 4-Axis Wrap with Mill Cam	90

USA

OneCNC LLC

Phone: 1-727-7243988
Fax: 1-727-7240025

Email: support@onecnc.com
Internet: www.onecnc.com

Version 1.02
(rev 28-6-01)

Australia

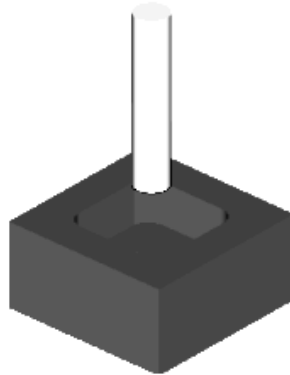
QARM Pty Ltd

Phone: 617 32862527
Fax: 617 32864992

Email: support@qarm.com.au
Internet: www.qarm.com.au

Mill Tutorial 1

Use 2 1/2 D Pocket Function



This is an example of machining a pocket in 2D with OneCNC.

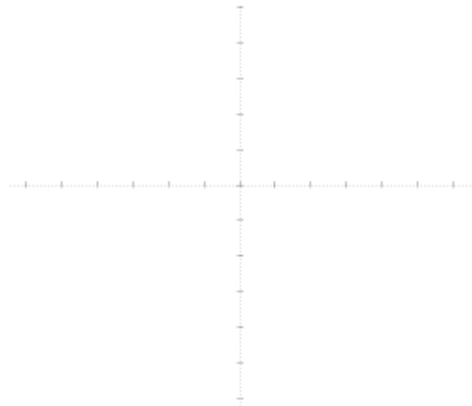
This is a tutorial to draw a 2D part using Mill then pocketing out the part using the mill pocket function.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw your pocket.



Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

Step 2. Drawing the Part

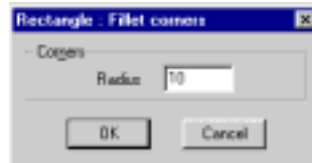


Click the **line icon** on the **menu toolbar**.



Click the **rectangle icon** on the **line menu toolbar**.

The following dialogue will appear.



Enter the value of **10** then click **OK**.

Then the position toolbar will appear.



Click the **coordinate icon** on the **position dialogue**.

Enter **X: 0, Y: 0** in the coordinate dialogue.

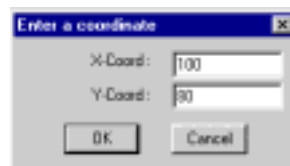


Then click **OK** and the position toolbar will again appear for the coordinate of the opposite corner of the square.



Click the **coordinate icon** on the **position dialogue**.

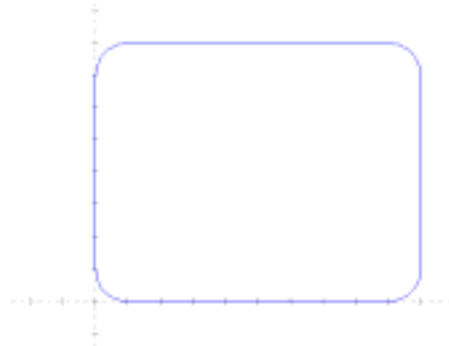
Enter **X: 100** and **Y: 80** in the dialogue and click **OK**.



The rectangle is drawn.

Then click **cancel** to terminate the rectangle function.

The drawing should look like this.



Click the **home icon** on the **menu toolbar**

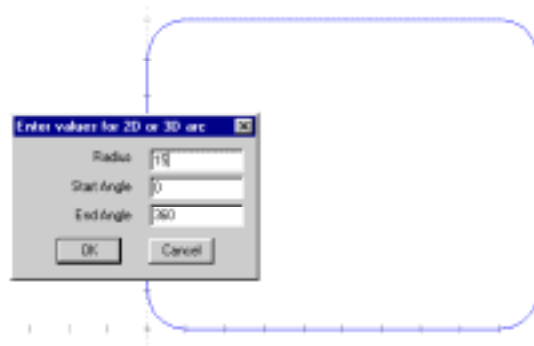


Click the **arc and spline icon** on the **menu toolbar**



Click the **circle icon** on the **arc and spline toolbar**

Enter a radius value of **15** and click **OK**.



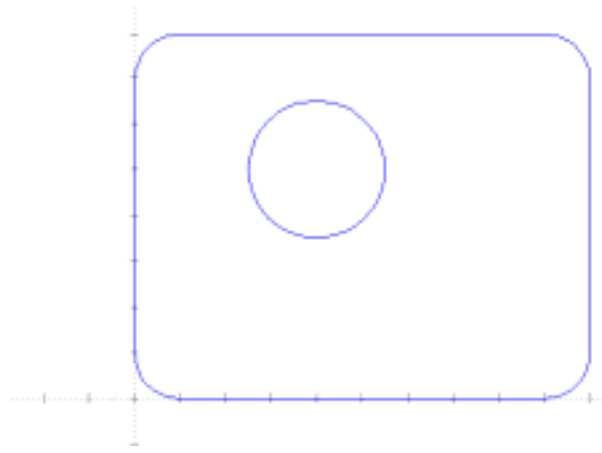
Click the **coordinate icon** on the **position dialogue**.

Enter **X: 40** and **Y: 50** and click **OK**.



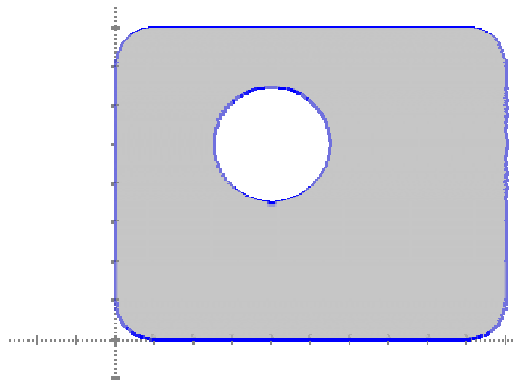
Then click **cancel** to terminate the circle function.

The drawing is now ready for pocketing.



Step 3. Pocketing the Part

The greyed area below is the area that we want to pocket to a depth of -20 mm with 2 cuts of 10 mm.



Click the **pocket icon** on the **cam toolbar**

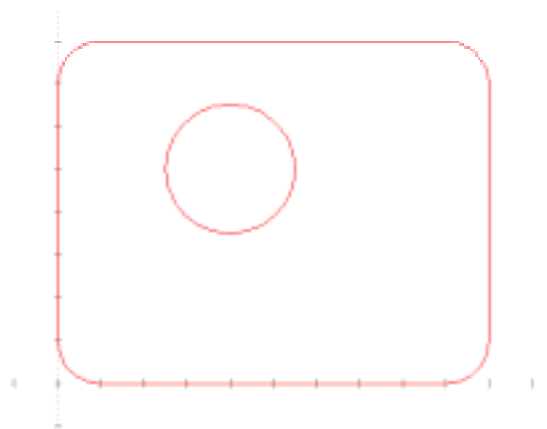
The following dialogue will appear.

Select **pick by boundaries** and click **OK**



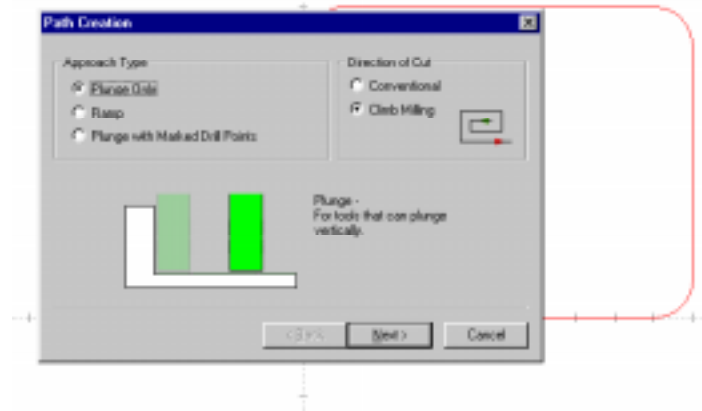
Then select the border loops (boundary) and they will turn red.

When they are all red click the **right mouse button**.



Select the **approach type** and the **direction of cut**.

Then click **next**.

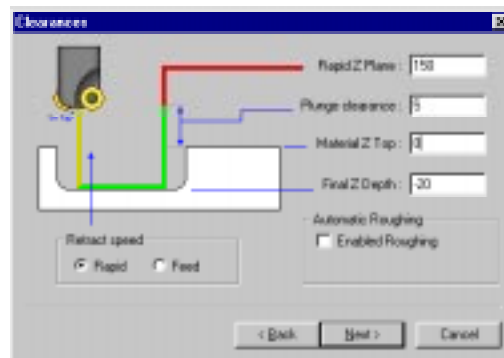


Select the **tool** and **material type** and then click **next**.

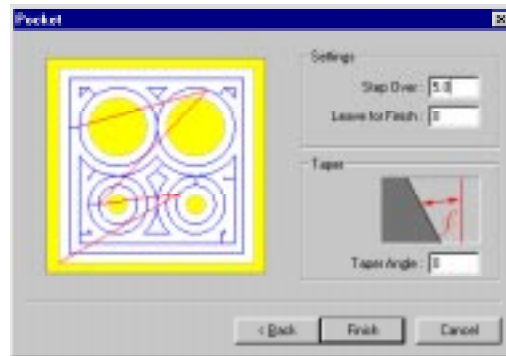


Set the **clearance plane**, **material top**, **final Z depth** and **enable automatic roughing**.

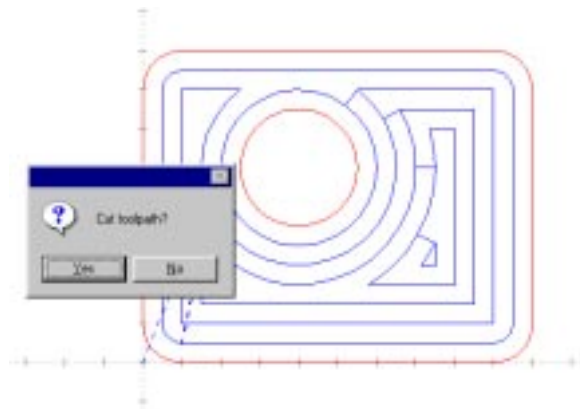
Then click **next**.




Enter the **step over value** and click **finish**.

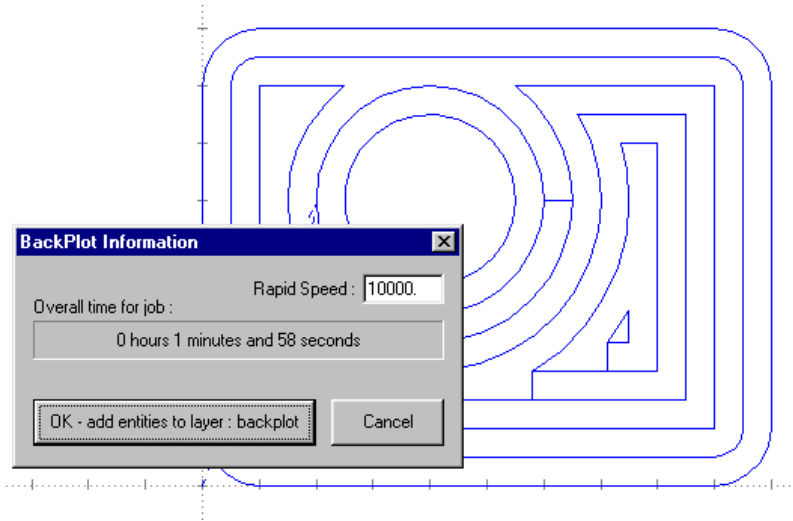


The toolpath is calculated and if it is to your satisfaction click **yes** when your asked to cut the tool path and the code will be created and placed in the NC editor.



Step 4. Obtaining a Machine Time

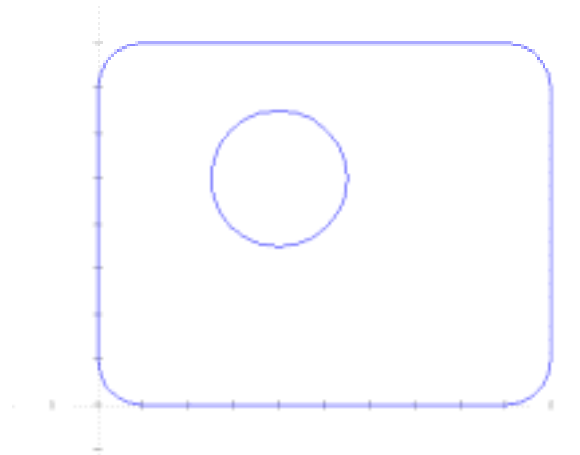
 Click the **backplot icon** on the **cam toolbar**



Then click **ok**

To remove the toolpath drawing.

 Click the **undo icon** on the **standard toolbar**



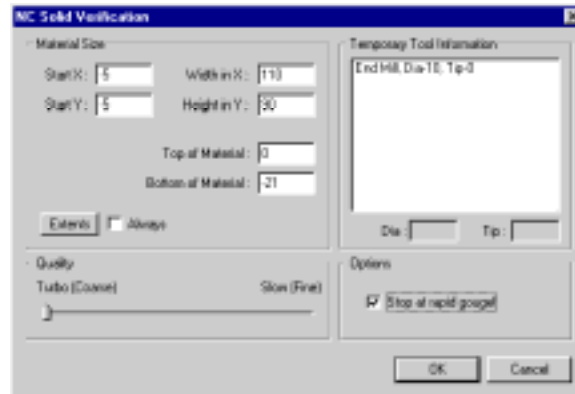
Step 5. Simulating the NC code



Click the **simulate NC** icon on the **NC CAM** toolbar.

The following dialogue will appear.

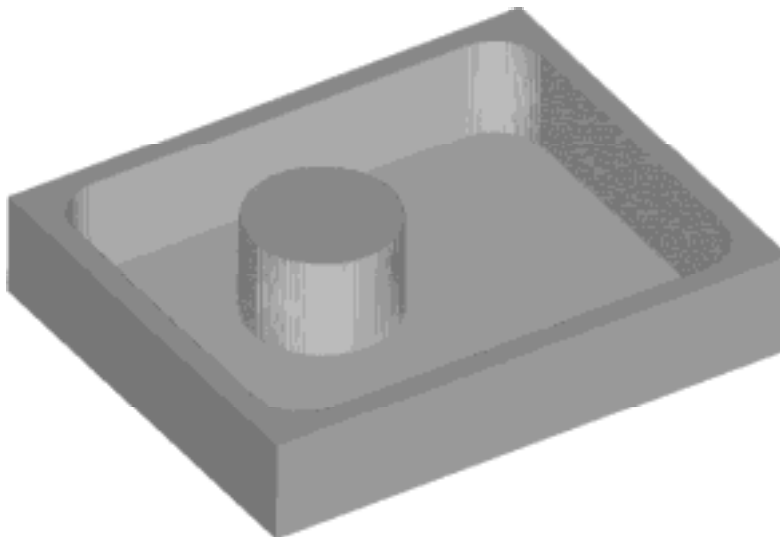
Enter the following details and click **OK**.

A screenshot of the 'NC Solid Verification' dialog box. The dialog is titled 'NC Solid Verification' and has a standard Windows-style border. It is divided into several sections: 'Material Size' with input fields for Start X (5), Width X (110), Start Y (5), Height in Y (30), Top of Material (0), and Bottom of Material (-21); 'Temporary Tool Information' with a text area containing 'End Mill, Dia-10, Tip-0' and fields for Dia and Tip; 'Quality' with 'Turbo (Coarse)' and 'Slow (Fine)' options; and 'Options' with a checked checkbox for 'Stop at rapid goseal'. At the bottom are 'OK' and 'Cancel' buttons.

Section	Field	Value
Material Size	Start X	5
	Width X	110
	Start Y	5
	Height in Y	30
	Top of Material	0
	Bottom of Material	-21
Temporary Tool Information	End Mill, Dia-10, Tip-0	
	Dia	
Quality	Turbo (Coarse)	
	Slow (Fine)	
Options	Stop at rapid goseal	<input checked="" type="checkbox"/>

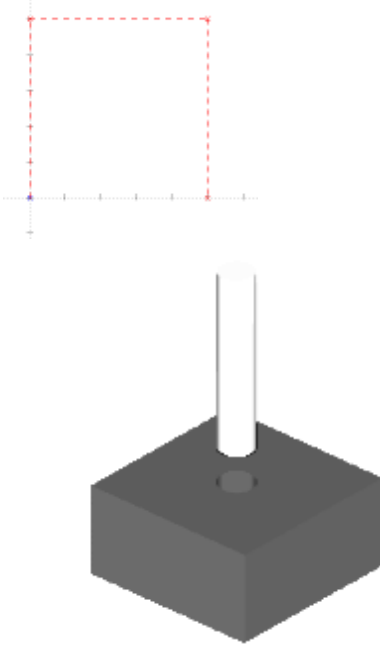
Note: Extents always box is not tag to help show the modeled toolpath for demonstration purposes extents should usually be tagged.

The toolpath is then modeled.



Mill Tutorial 2

Use Drill Function



Note: This help uses mm values.

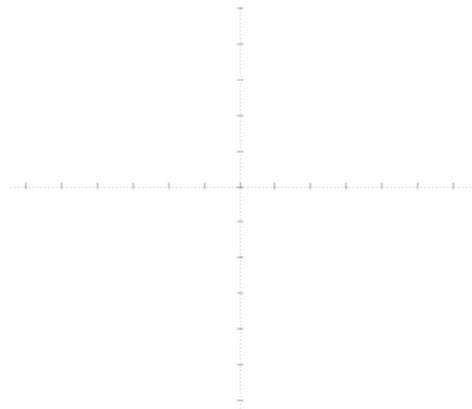
This is a tutorial to draw points using Mill then to drill those points.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw.




Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

Step 2. Create our Four Points

 Click the **point** menu icon on the **menu toolbar**.

 Click the **point by coordinate** icon on the **point menu toolbar**.
Enter **X: 10 Y: 10** and click **OK**.




Enter coordinates dialog box showing X-Coord: 10 and Y-Coord: 10. Buttons: OK, Cancel.

Then enter **X: 10 Y: 30** and click **OK**.



Enter coordinates dialog box showing X-Coord: 10 and Y-Coord: 30. Buttons: OK, Cancel.

Then enter **X: 30 Y: 10** and click **OK**.



Enter coordinates dialog box showing X-Coord: 30 and Y-Coord: 10. Buttons: OK, Cancel.

Then enter **X: 30 Y: 30** and click **OK**.



Enter coordinates dialog box showing X-Coord: 30 and Y-Coord: 30. Buttons: OK, Cancel.

Then click **cancel** to complete the drawing of points.



Step 3. Drilling the Points

We will now use these points to drill at Z -20.0 at these points.



Click the **drilling cycle icon** on the **cam toolbar**.

Select your **drill** and **material** then click **next**.

The 'Select Tool' dialog box contains the following fields and controls:

- Station Number: 5
- Length Offset: 5
- Diameter Offset: 5
- Feed rate: 1431
- Select a Tool
- Type: Drill
- Diameter: 10mm
- Tip Radius: 0mm, 2 flutes
- Material: ALUMINUM WITH HSS CUTTER
- Chipload: Material - 2.0, Tooth - 0.125
- Spindle Speed: 2305
- Comments to be added: (empty text box)
- Buttons: Formal, Edit Tools, Edit Materials, < Back, Next >, Cancel

Set your **depth** and **clearances** and click **next**.

The 'Clearances' dialog box contains the following fields and controls:

- Diagram showing: rapid plane, plunge clearance, initial plane, retract, final Z depth.
- Rapid Z Plane: 5
- Plunge clearance: 2
- Initial Plane: 1
- Final Z Depth: 20
- Retract speed: Rapid Feed
- Buttons: < Back, Next >, Cancel

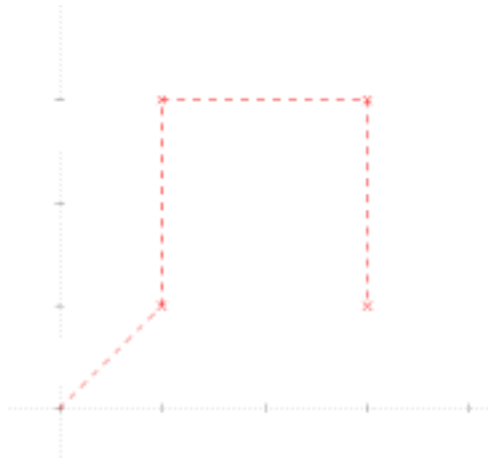
Select the **drilling cycle** and click **finish**.

The 'Drilling Parameters' dialog box contains the following fields and controls:

- Settings:
 - Retract R: 0
 - Peck Q: 11
- Retract Modes:
 - G98 (Return to initial point)
 - G99 (Return to point R)
 - None (machine defaults)
- NOTE: For automatic drilling - Select the points before using this function.
- Buttons: < Back, Finish, Cancel

Then select the **points** in the order that you wish to drill them, the home point is already selected.

Then click the **right mouse button** when they have all been selected.



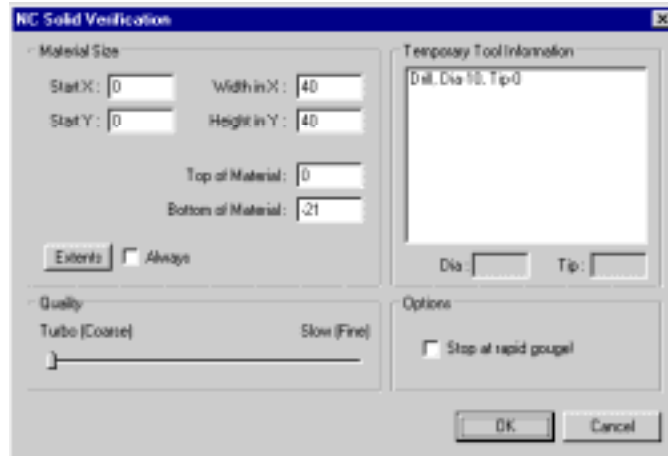
The NC code is now in the editor completed.

Step 4. Simulating the NC code

 Click the **simulate NC** icon on the **NC CAM** toolbar.

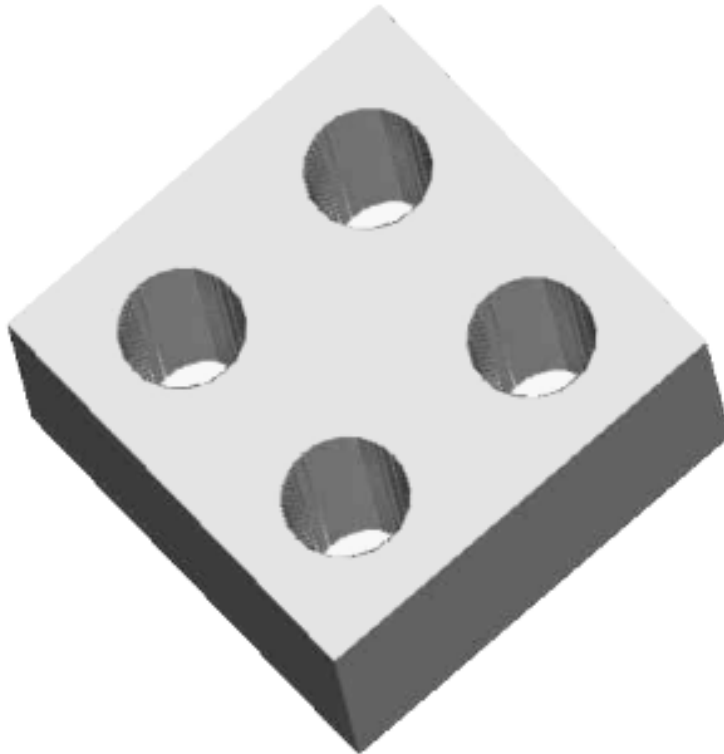
The following dialogue will appear.

Enter the following details and click **OK**.



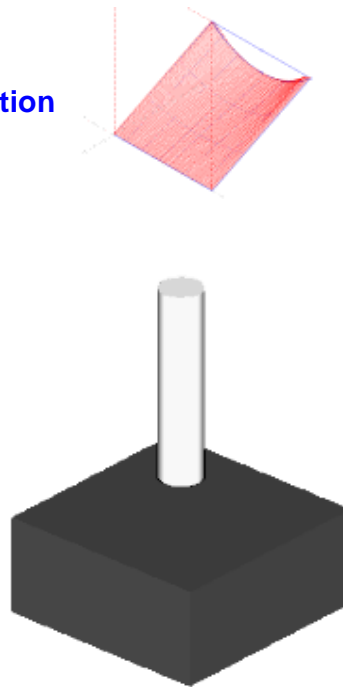
Note: Extents always box is not tag to help show the modeled toolpath for demonstration purposes extents should usually be tagged.

Your toolpath is then modeled.



Mill Tutorial 3

Use Mill 3D Surface Function



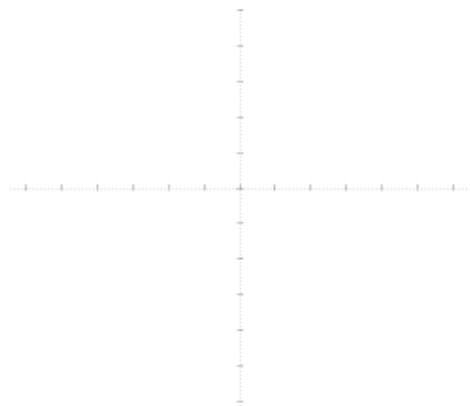
This is a tutorial to draw a 3D surface part and machine it using Mill.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw your wireframe and then surface it.



Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

Step 2. Creating the Wire Frame



Click the **change 3D views icon** on the **menu toolbar**.



Click the **Isometric view icon** on the **3D views dialogue**.

We now see the isometric View.

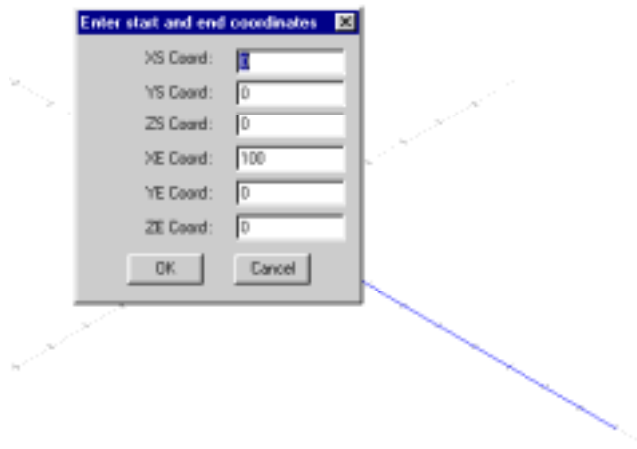


Click the **line icon** on the **menu toolbar**.

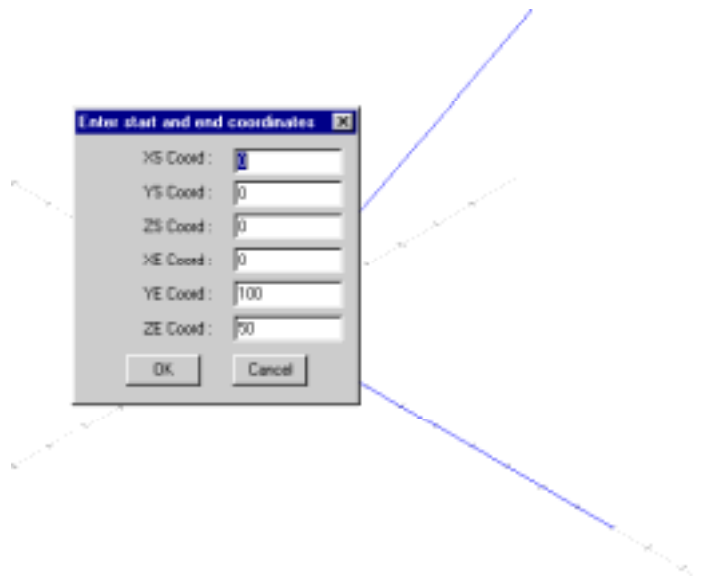


Click the **line by coordinates icon** on the **line menu toolbar**.

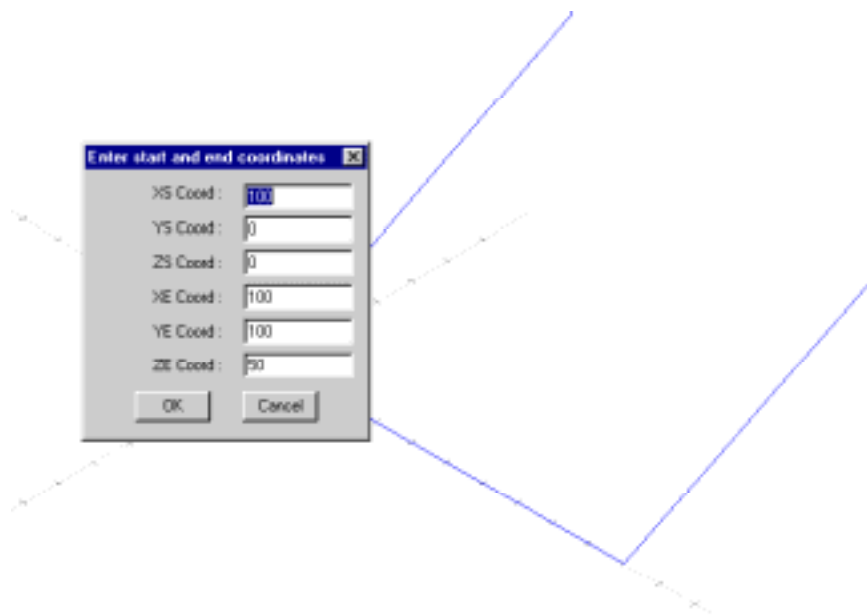
Enter the following values and click **OK**.



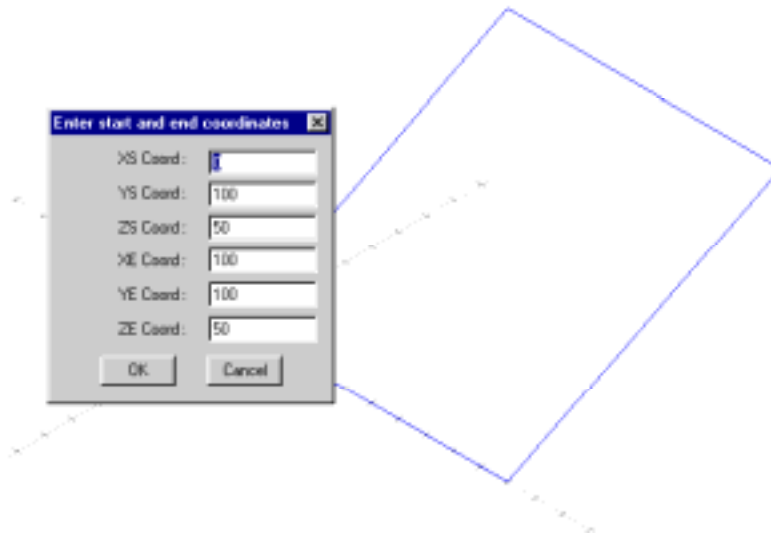
Then enter the following values and click **OK**.



Then enter the following values and click **OK**.



Then enter the following values and click **OK**.



Now click **cancel** to terminate drawing lines.



Click the **home icon** on the **menu toolbar**.

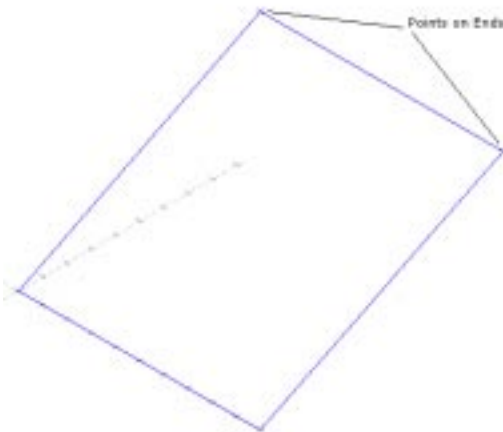


Click the **point** icon on the **menu toolbar**.



Click the **end of entity** icon on the **line menu toolbar**.

Then select the **two top corners** to place points on them.



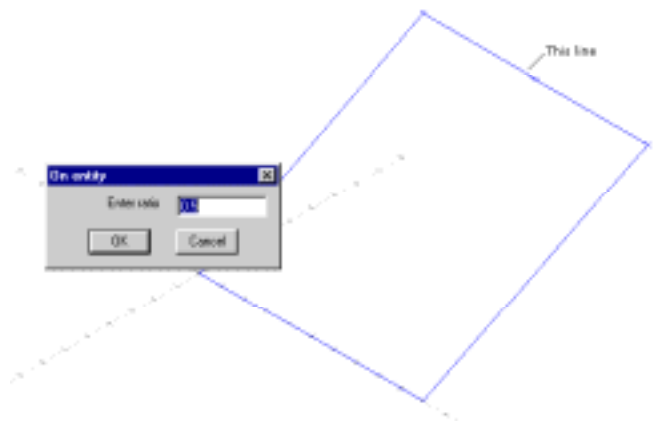
Then click the **right mouse button** to cancel points on ends.



Click the **point on entity** icon on the **point menu toolbar**.

The following dialogue will appear enter **0.5** and click **OK**

Select the **top line** between the two points.



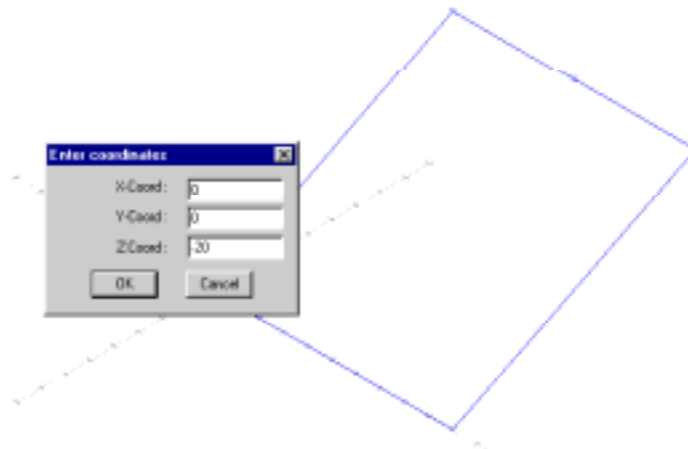
Then click **cancel**.



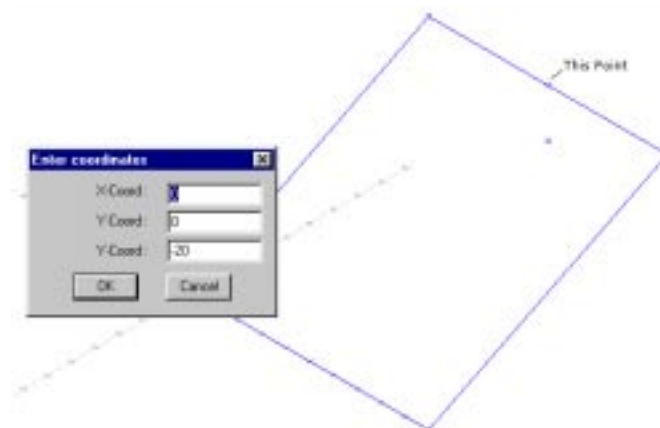
Click the **point incremental** icon on the **point menu toolbar**.

Select the point that you created on that line.

The following dialogue will appear, enter **Z: -20** and click **OK**



Then click **cancel** to cancel point incremental tool.





Click the **home icon** on the **menu toolbar**.

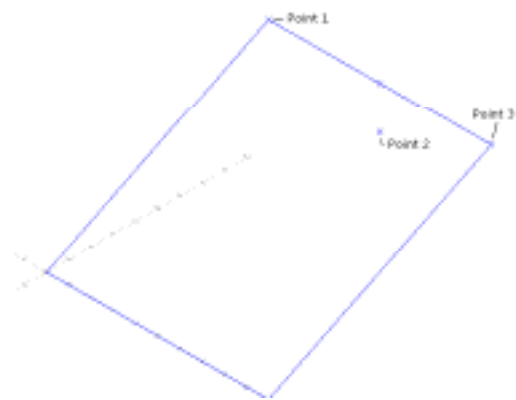


Click the **arc and spline icon** on the **menu toolbar**.

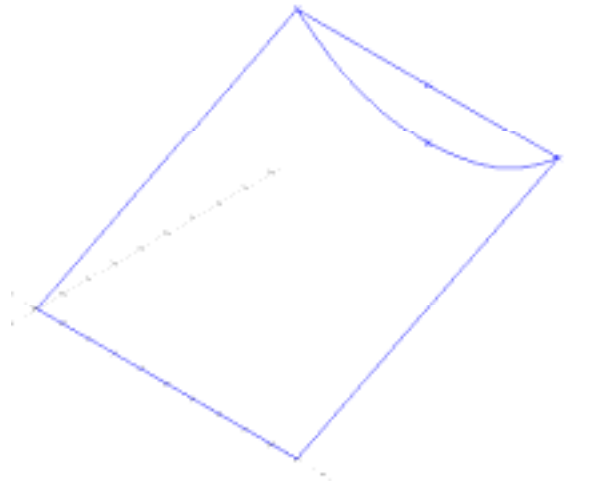


Click the **arc on 3 entities icon** on the **arc and spline menu toolbar**.

Then select these **three points**.



Instead of clicking the left mouse button to select each point put the cursor near the point and press P on your keyboard and it will select the point automatically for you.



Then click the **right mouse button** to cancel drawing arc on 3 entities.

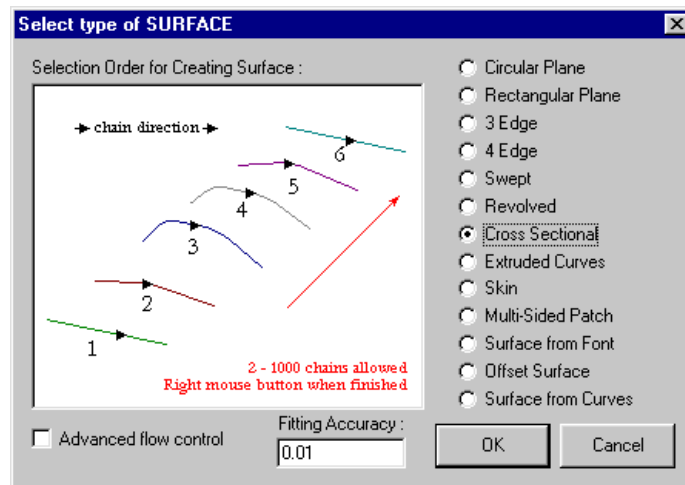
Step 3. Surfacing the Wireframe

We will now create a surface on the wire frame.



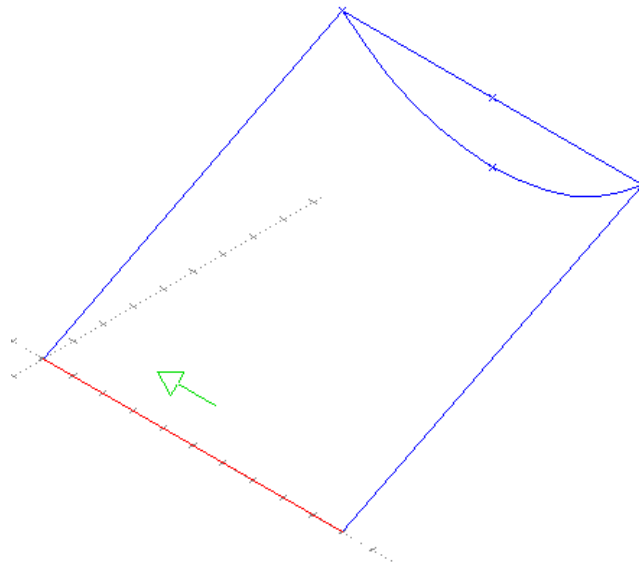
Click the **create a surface icon** on the **menu toolbar**.

Then select **cross sectional** and click **OK**.

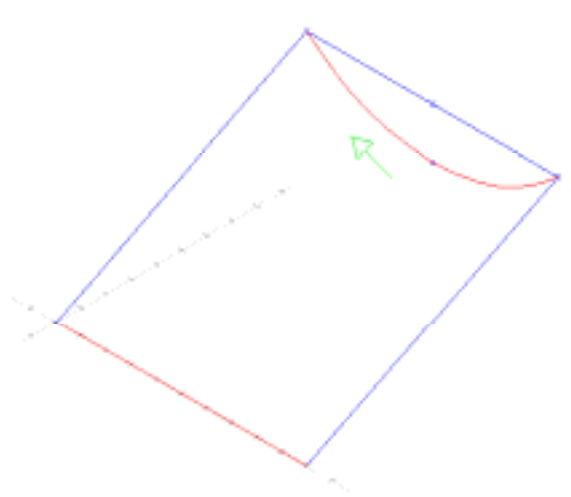


Then select the **bottom line** click the **left mouse button** and point the **arrow in this direction** and click the **left mouse button** .

Then click the **right mouse button** to accept this first line.

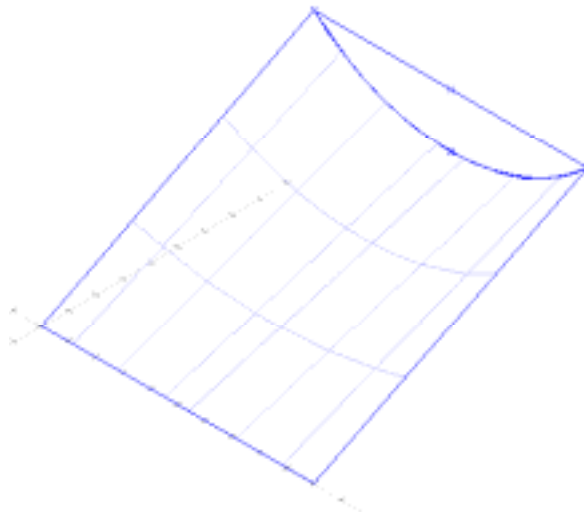


Now select the **arc** and point the arrow in the **same direction**.



Then click the **right mouse button** to accept the arc.

Then click the **right mouse button** again to terminate cross section selections and the surface will be created.



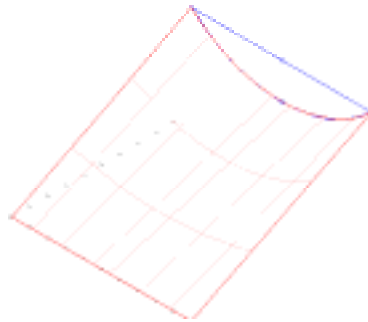
Step 4. Machining the Surface

To machine this surface with 20 mm ball mill.



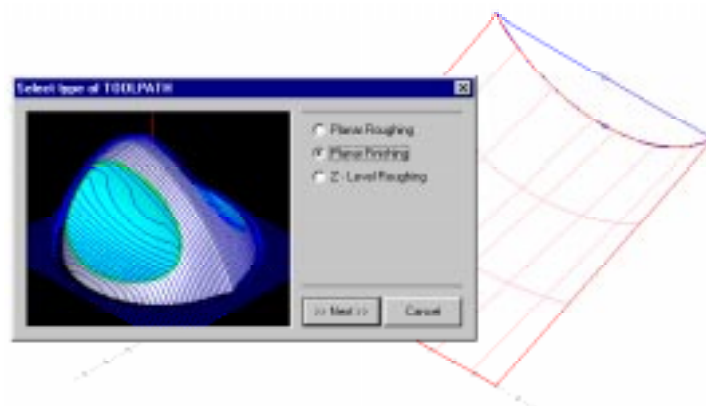
Click the **select single entity** icon on the **selections toolbar**.

Select the surface and it will turn red.



Click the **surface machine** icon on the **NC CAM toolbar**.

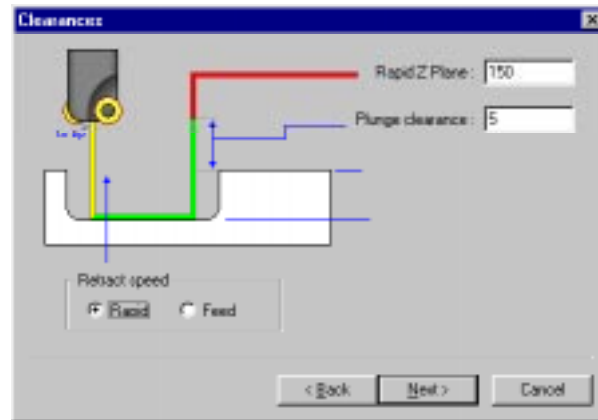
Then select **planar finishing** and click **next**.



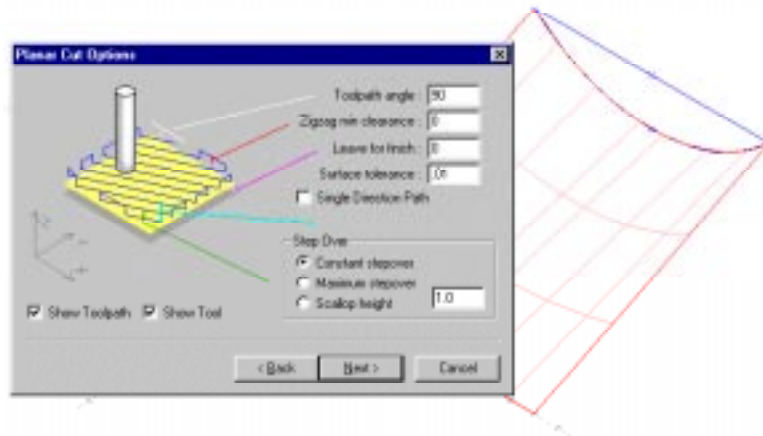
Select your **cutter** and **material** and click **next**.



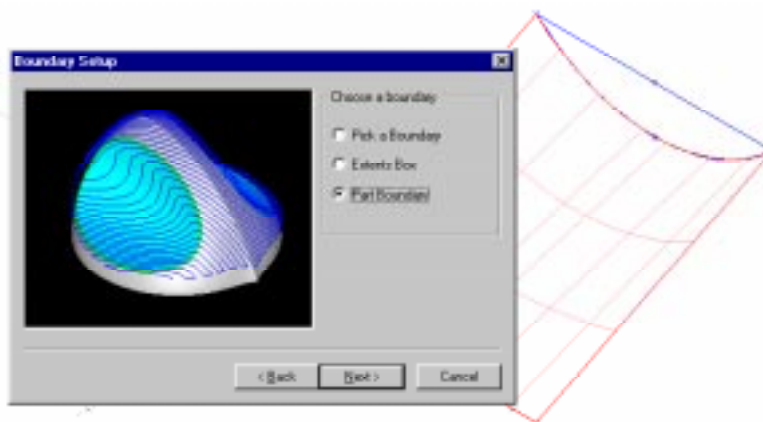
Set your **rapid plane** and **plunge clearance** and click **next**.



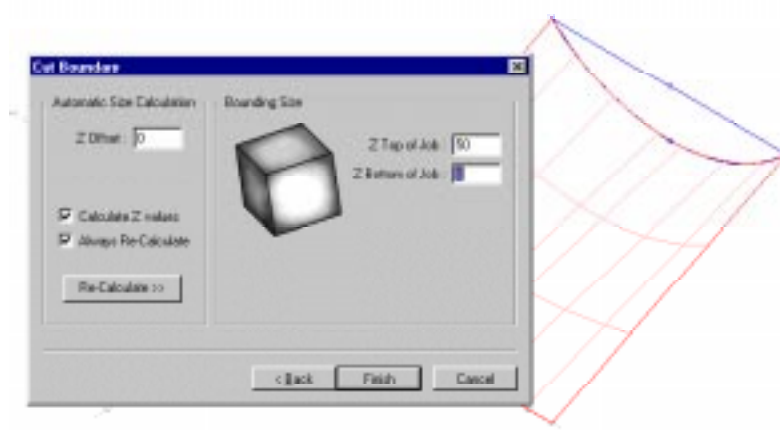
Set the **cutter options** and click **next**.



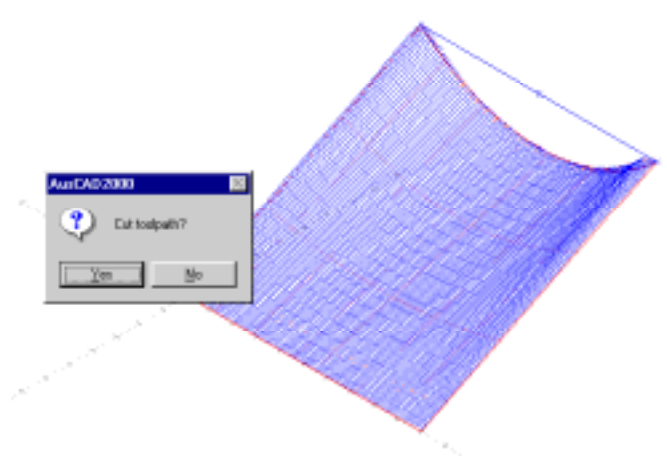
Set the **part boundary** and click **next**.



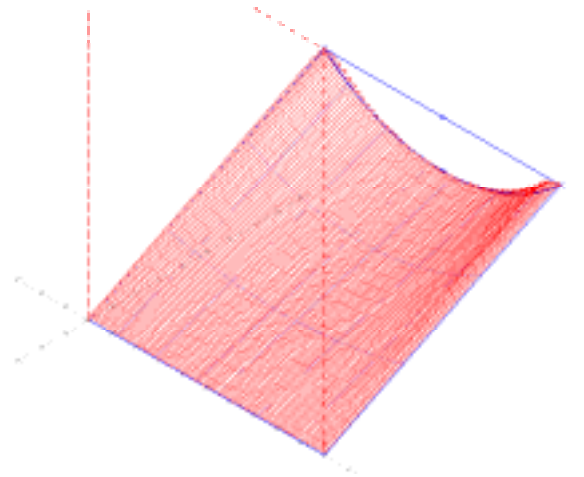
Then click **finish**.



Click **yes** to cut toolpath.



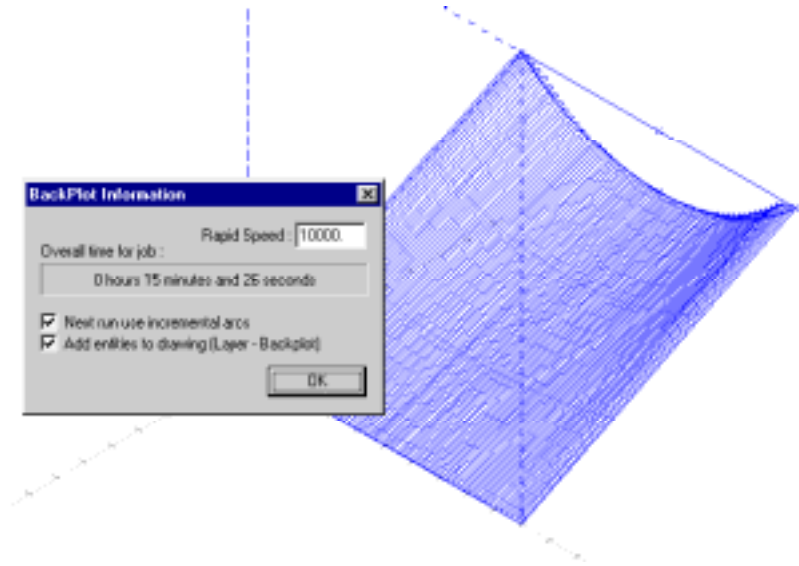
The toolpath NC code is created and is in the editor..



Step 5. Getting the Machine Time

 Click the **backplot icon** on the **NC CAM toolbar**.

The NC file is then back plotted to the screen and the time is calculated.



The dashed lines represent the rapid movements and the solid lines are the cut lines.

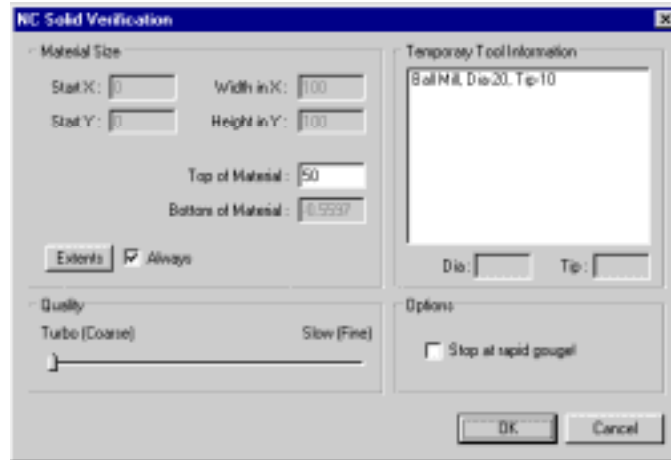
Step 6. Simulating the NC code



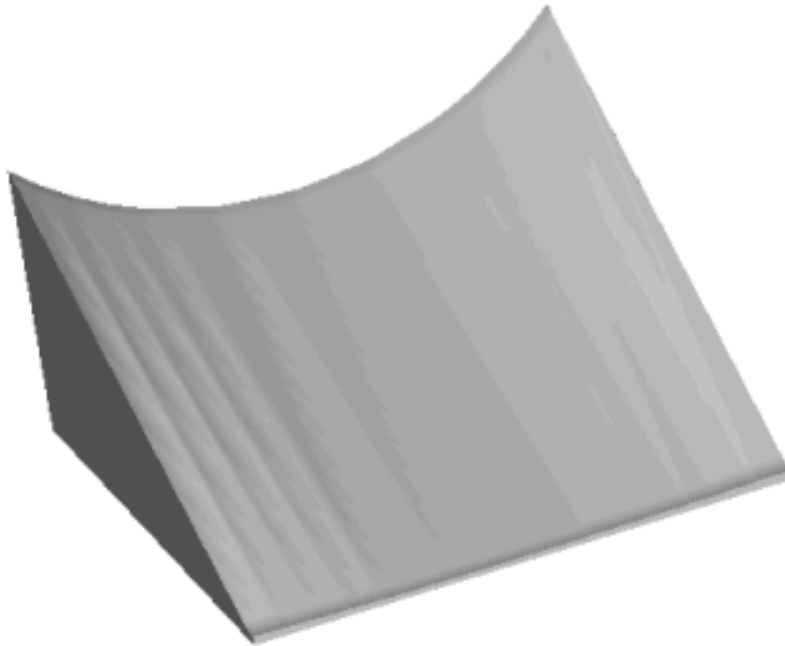
Click the **simulate NC** icon on the **NC CAM** toolbar.

The following dialogue will appear.

Enter the following details and click **OK**.

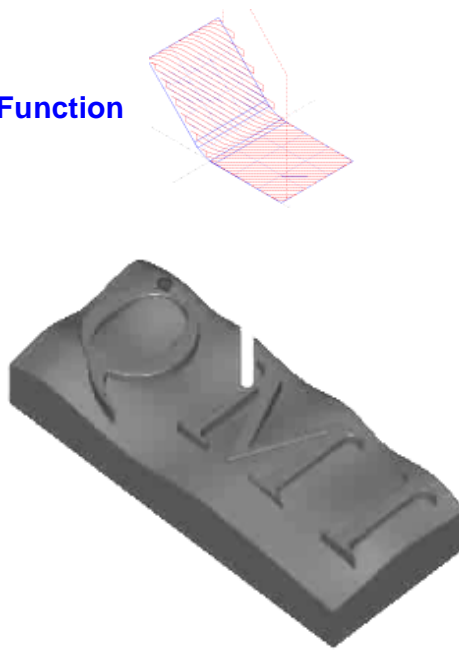


The tool path is then modeled.



Mill Tutorial 4

Use Mill Multi Surface Function



This is a tutorial to draw two 3D surfaces of a part using Mill.

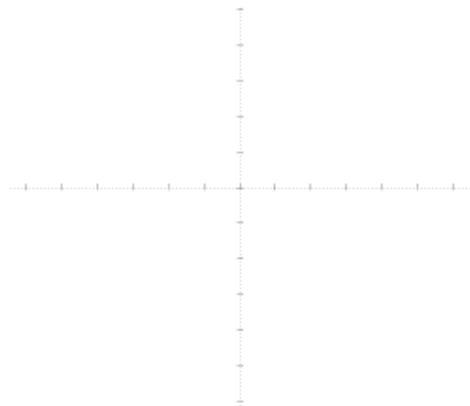
Note: This help uses mm values.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to surface a wireframe and machine.



Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

Step 2. Create Two Wireframes

 Click the **line icon** on the **menu toolbar**

 Click the **rectangle icon** on the **line menu toolbar**

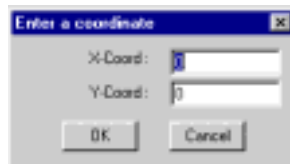
The following dialogue will appear.



Enter the value of **0** then click **OK**

 Click the **coordinate icon** on the **position dialogue**.

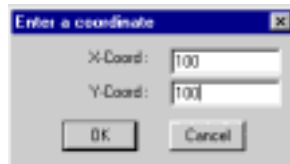
Enter **X: 0, Y: 0** in the coordinate dialogue



Then click **OK** and the position toolbar will again appear for the coordinate of the opposite corner of the square.

 Click the **coordinate icon** on the **position dialogue**.

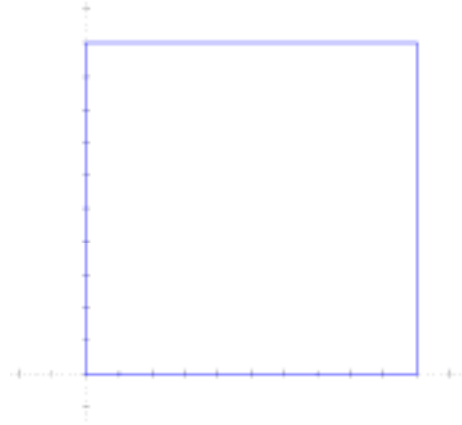
Enter **X: 100** and **Y: 100** in the dialogue and click **OK**.



Then click **cancel** to end the rectangle function.

 Click the **view all icon** on the **standard toolbar**.

 Click the **reduce half icon** on the **standard toolbar**.



Now we will turn this in to a 3D object.

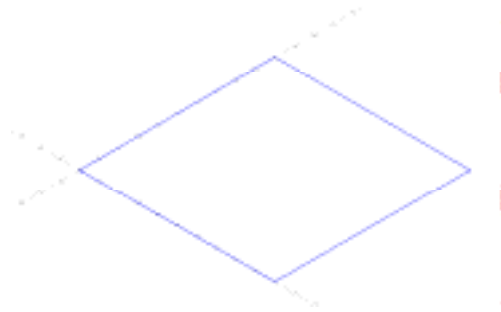


Click the **change 3D views icon** on the **menu toolbar**.



Click the **Isometric view icon** on the **3D views dialogue**.

You will now see it in isometric view.



Click the **home icon** on the **menu toolbar**.



Click the **line icon** on the **menu toolbar**.

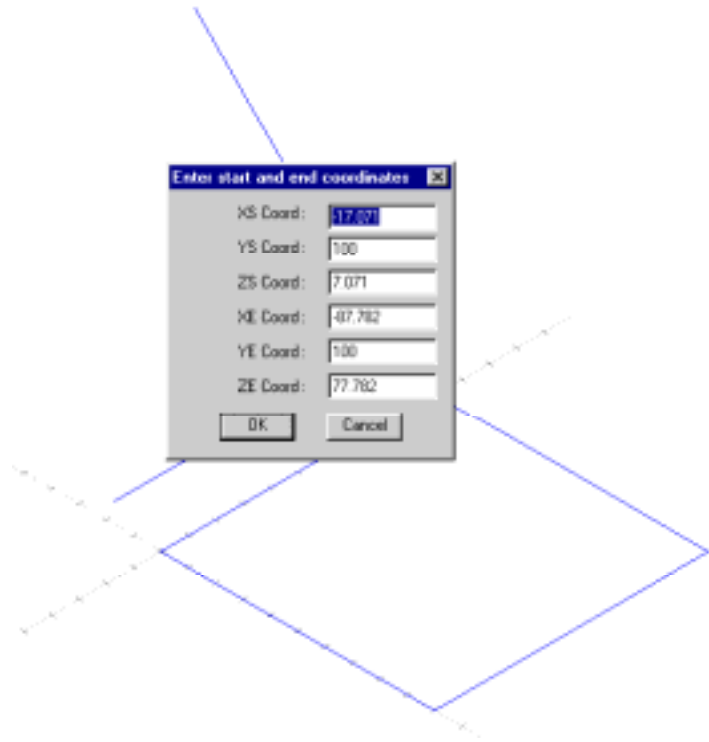


Click the **line by coordinates icon** on the **line menu toolbar**.

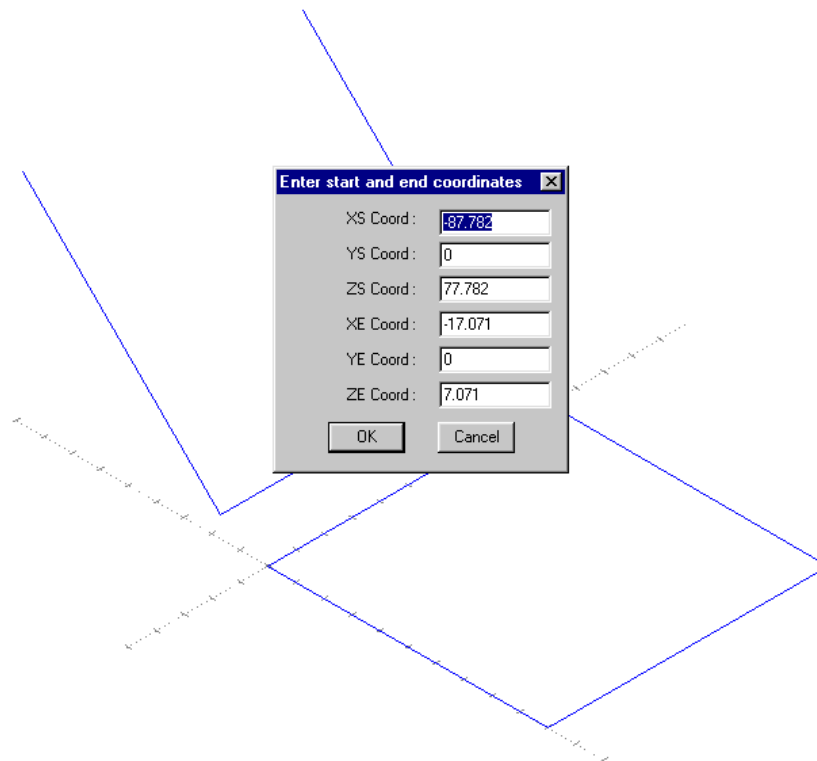
Enter the following values then click **OK**

Enter start and end coordinates	
XS Coord:	-17.071
YS Coord:	0
ZS Coord:	7.071
XE Coord:	-17.071
YE Coord:	100
ZE Coord:	7.071
OK Cancel	

Then enter the following for the next line and click **OK**

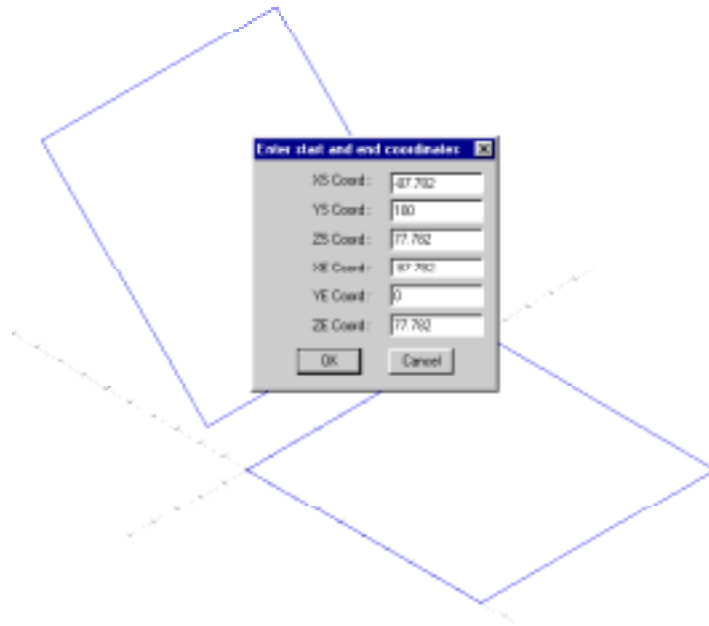


The enter the following coordinates and click **OK**



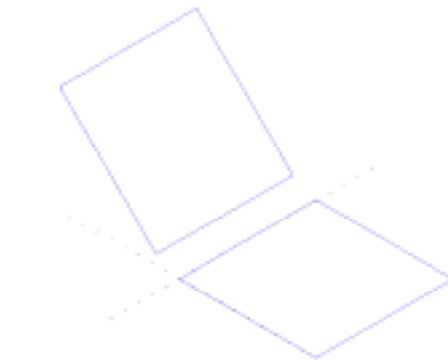
Then the coordinates for the last line and click **OK**

Click **cancel** to end the line by coordinates function.



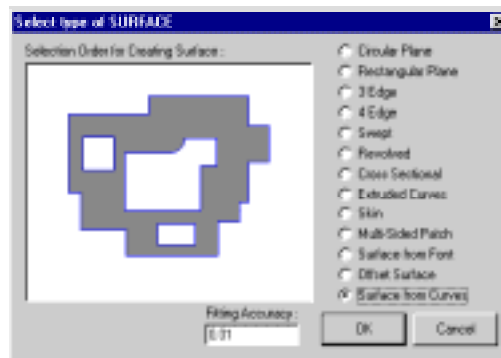
Step 3. Surfacing the Rectangles

We will now surface these rectangles.



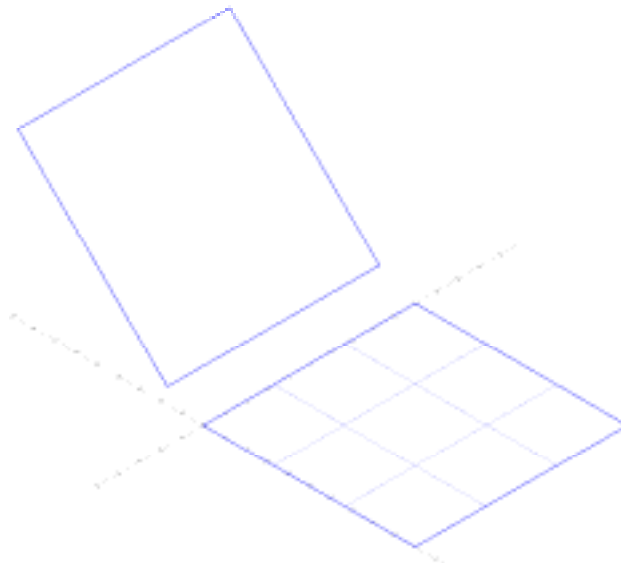
Click the **create a surface icon** on the **surfaces and solids toolbar**.

Then tag the **surface from curves** and click **OK**



Then select the **lower rectangle** with the mouse and after it turns red click the **right mouse button**

The rectangle is now surfaced

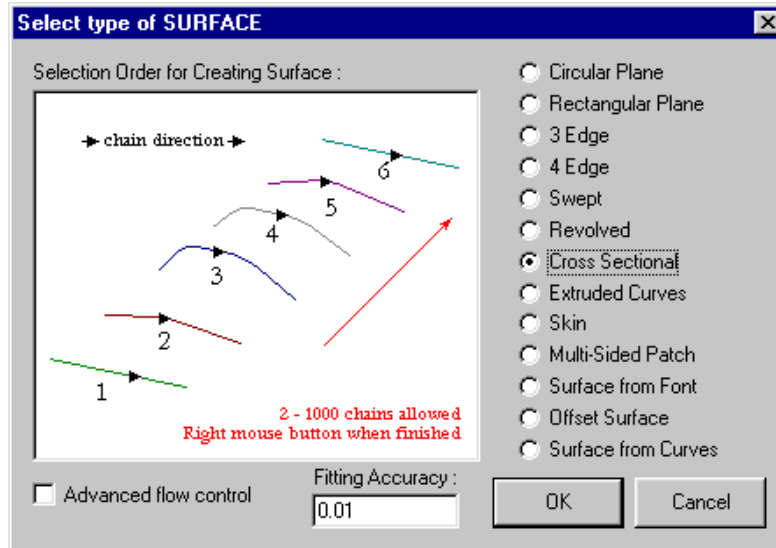


We will now surface the next one. Because the next surface is NOT on the X Y plane we will use surface cross section.

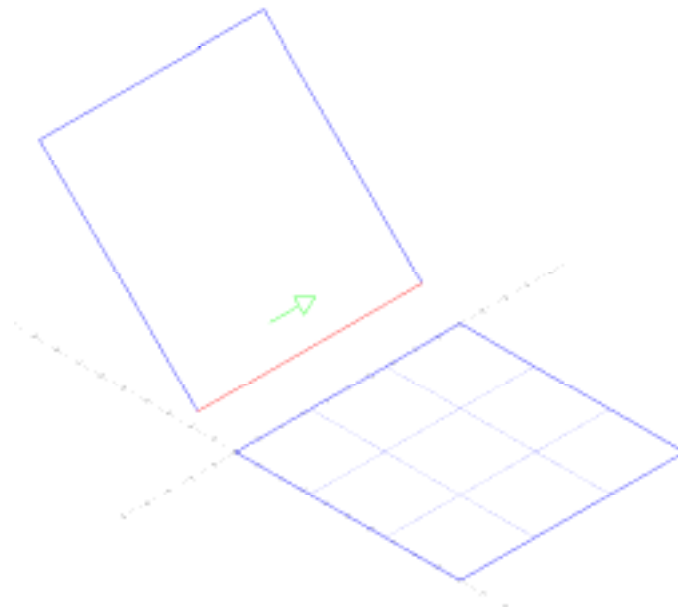


Click the **create a surface icon** on the **surfaces and solids toolbar**.

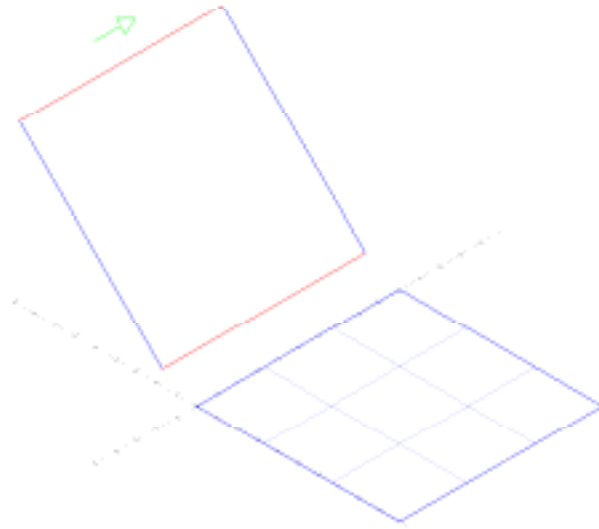
Then tag the **cross sectional surfaces** and click **OK**



Then select the **lower line** of the second rectangle and with the mouse, point the arrow in the direction like shown click the **left mouse button** and then click the **right mouse button** to select the first cross section.

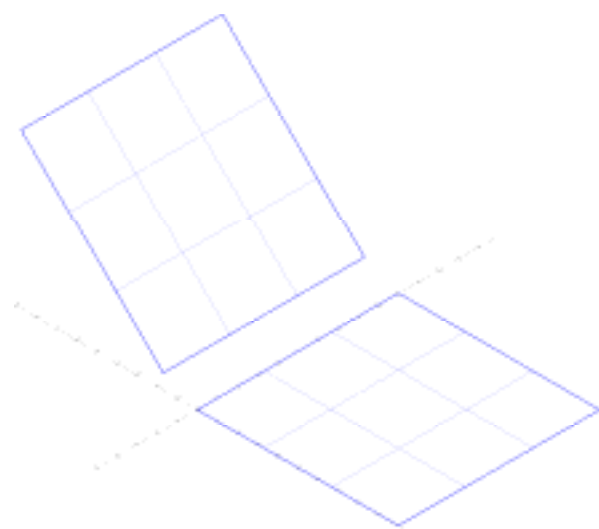


Then we will do the same with selecting the **top line of the rectangle**




Know that both lines are red click the **right mouse button** to select the second selection.

Then click the **right mouse button** AGAIN and the second surface will be created.

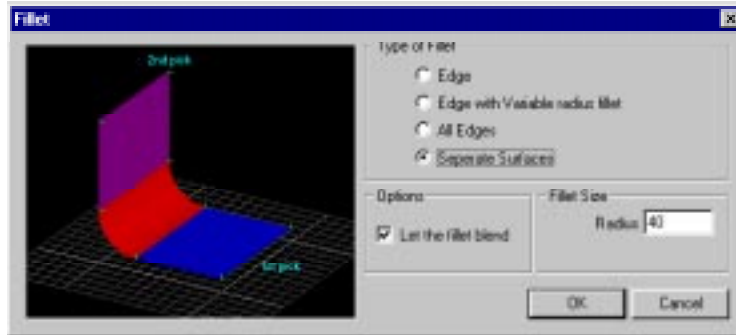


Step 4. Create a Fillet Between the Wireframes

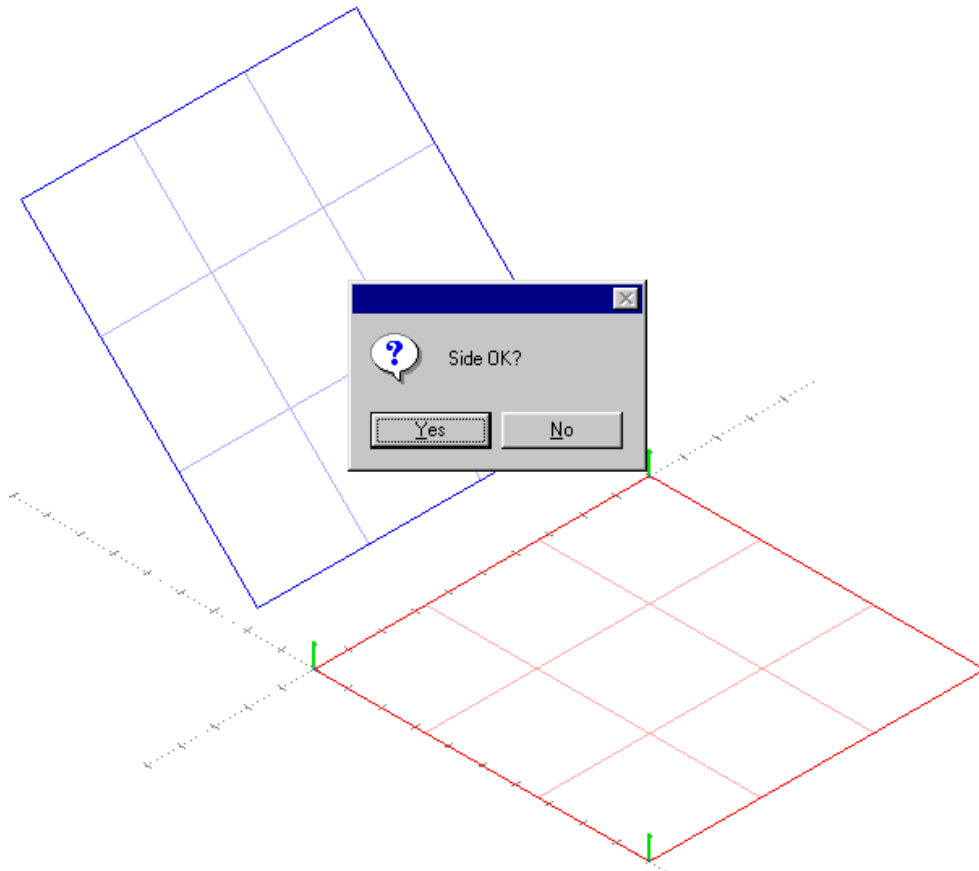
We will now fillet between these two surfaces.

 Click the **fillet solid icon** on the **surfaces and solids toolbar**.

Tag the **separate surfaces** and enter a value of **40** for the fillet size then click **OK**

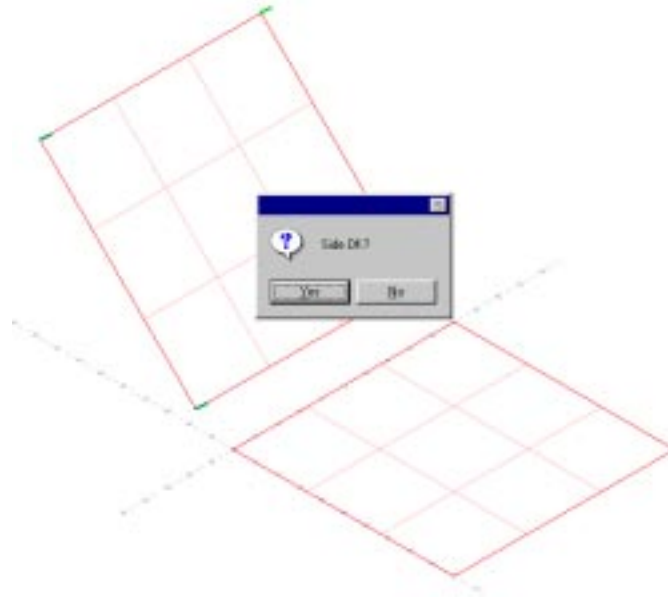


Then select the **approximate center** of the lower surface and make sure the green surface normal are pointing UP and if they are click **YES**, if are are not click **NO** and they should then be correct.

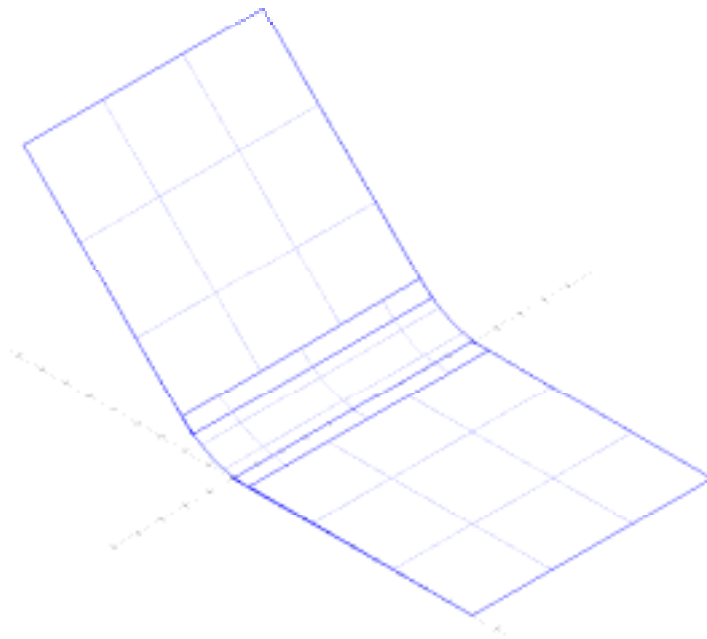


Then select the **approx center** of the other surface and also make sure the normal is pointing as shown below and if it is not click **NO** and the normal will change then when correct click **YES**

Both normals must be correctly facing for the fillet function to work.



The two surfaces are now filleted.

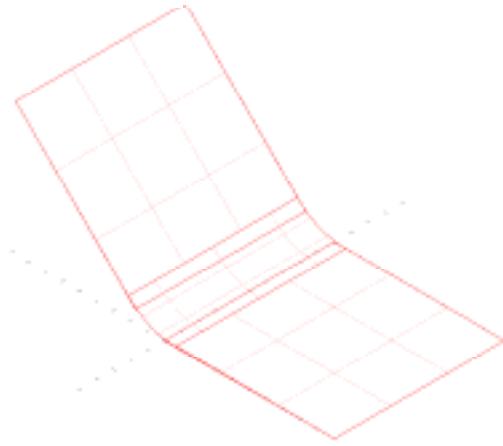


Step 5. Machining the Surfaces

We will now machine these surfaces using Mill

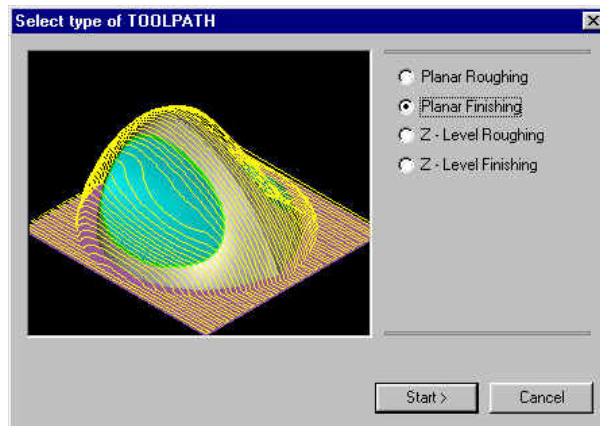


Click the **select all icon** on the **selections toolbar**.

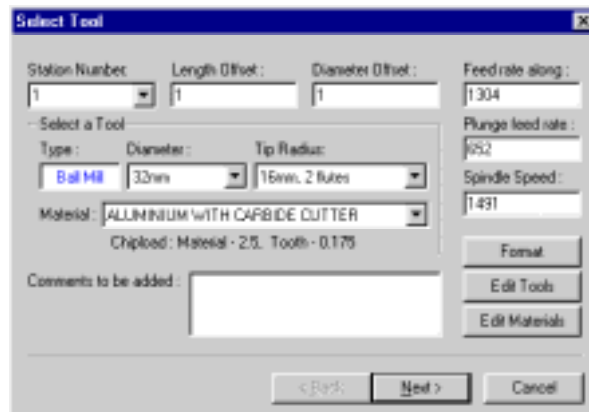


Click the **create NC code icon** on the **NC CAM toolbar**.

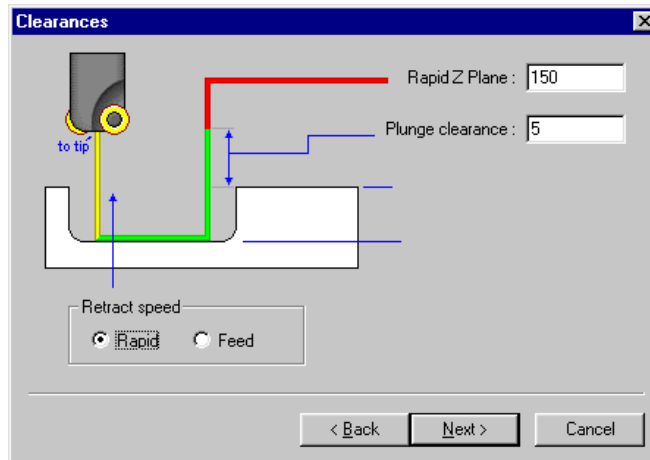
Tag planar finishing and click **start**



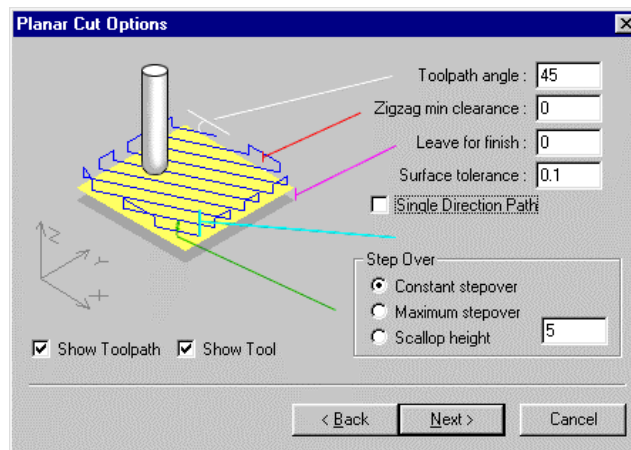
Select a tool and material and click **next**



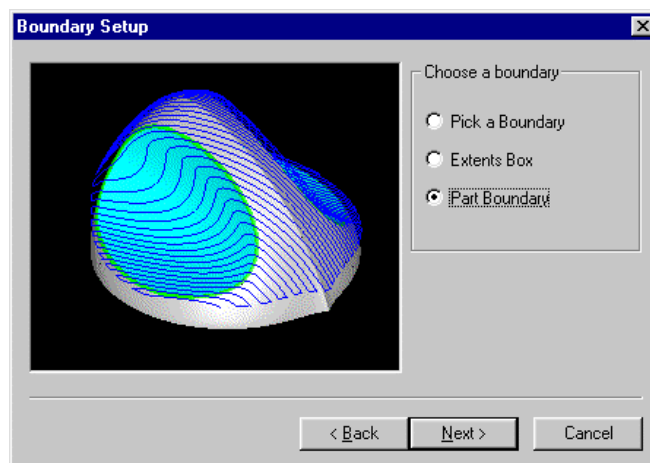
Set your **rapid clearance** and click **next**



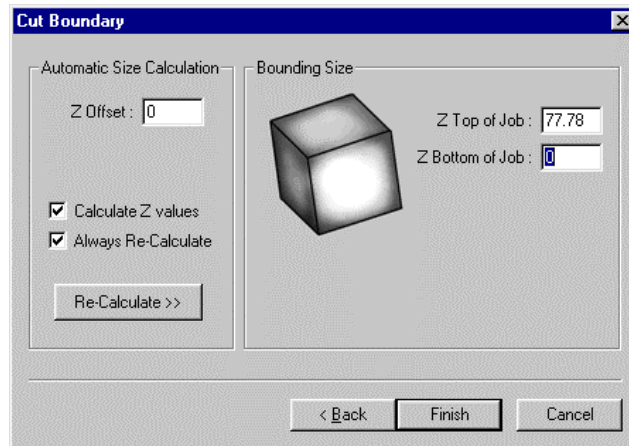
Set the following settings and then click **next**



Set your **Part Boundary** and click **next**

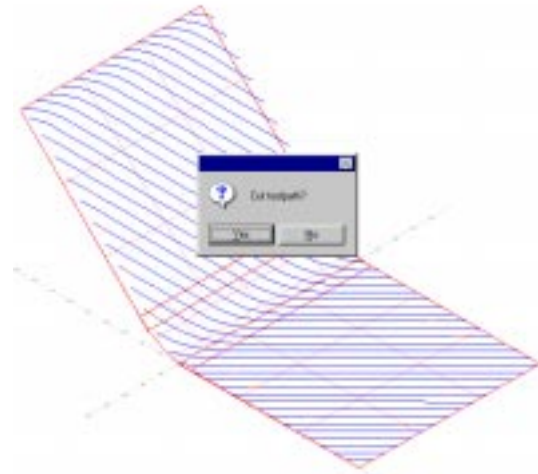


Then click **finish**

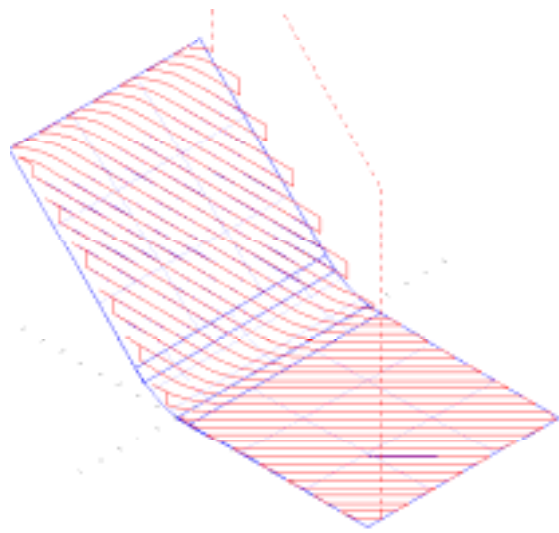


And the toolpath is created

Then click **yes** to cut the toolpath



The code is created and it is now in the NC editor.

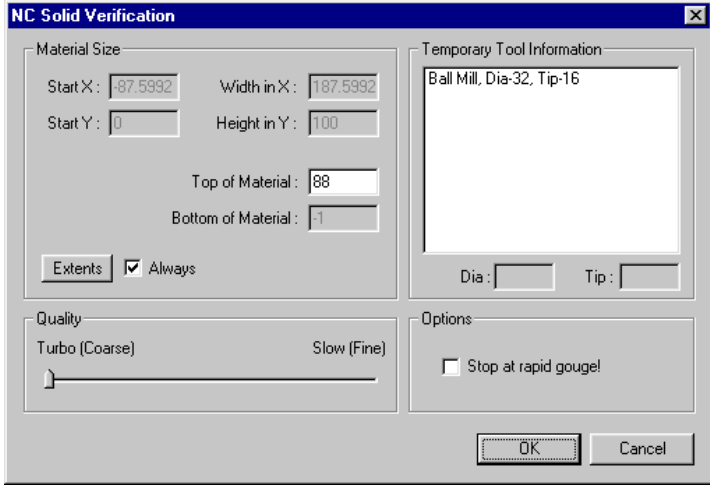


Step 6. Simulate the NC Code

 Click the **simulate NC** icon on the **NC CAM** toolbar.

The following dialogue will appear.

Enter the following details and click **OK**.

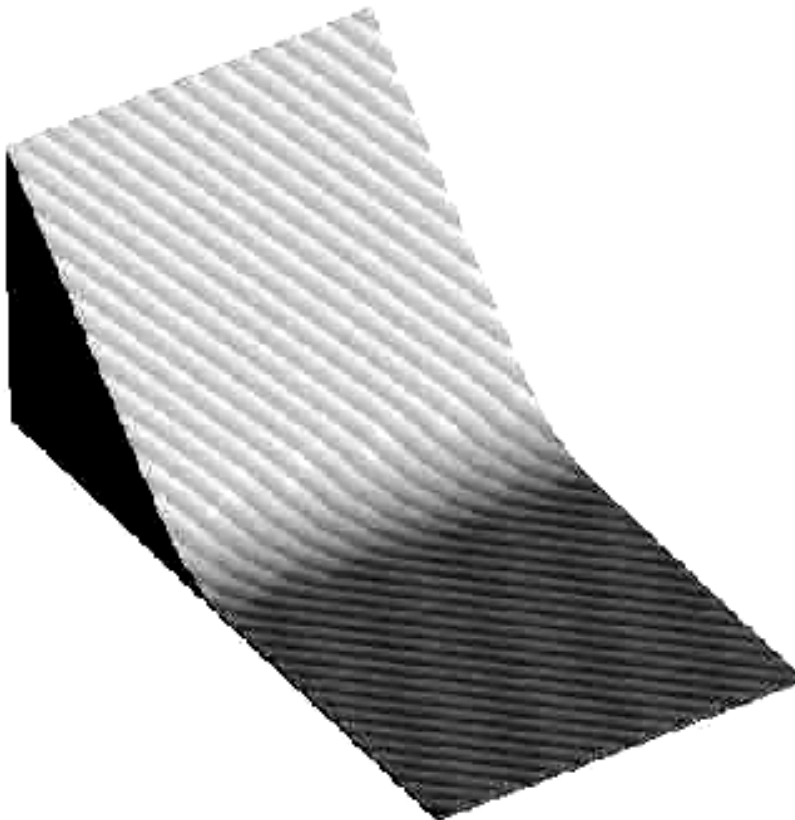


The dialog box titled "NC Solid Verification" contains the following fields and options:

- Material Size:**
 - Start X: -87.5992
 - Width in X: 187.5992
 - Start Y: 0
 - Height in Y: 100
 - Top of Material: 88
 - Bottom of Material: -1
- Extents:** Always
- Quality:** Turbo (Coarse) | Slow (Fine) with a slider.
- Options:** Stop at rapid gouge!
- Temporary Tool Information:** Ball Mill, Dia-32, Tip-16
- Buttons:** OK, Cancel

Note: Extents always box is not tag to help show the modeled toolpath for demonstration purposes extents should usually be tagged.

The toolpath will then be modeled.



Step 7. Send the Program to the machine/ Editing the Code

If you wanted to send your program to the machine:



Click the **send file icon** on the **NC Editor toolbar**.

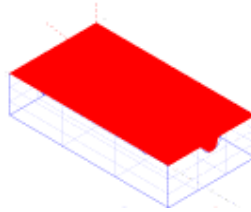
To view your code:



Click the **editor icon** on the **menu toolbar**.

Mill Tutorial 5

Use Solid Milling Function

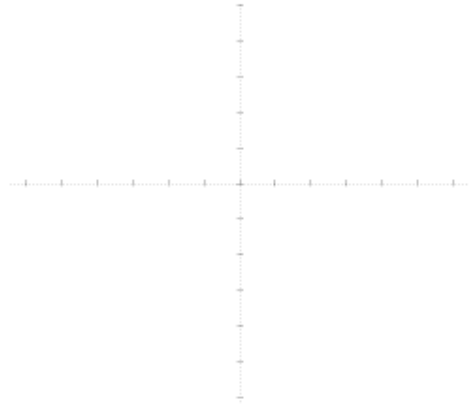


This tutorial is for creating the model of a bottle, creating a die cavity, and then machine the die block with Z level rough and planar finish.

Step 1. Create a New Drawing

 Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw your solid.



Now to make sure the NC file is clear

 Click the **new NC icon** on the **NC editor toolbar**.

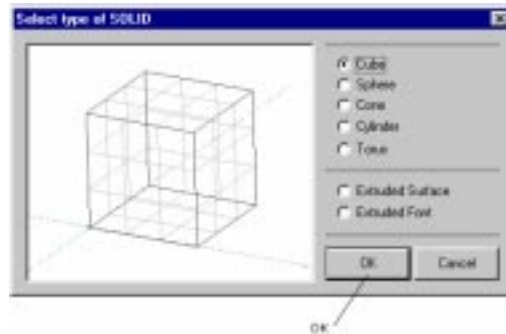
Click **Yes** or **No** to save your file.

Step 2. Creating the Body

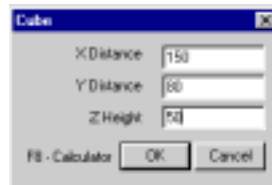


Click the **create solid icon** on the **surfaces and solids toolbar**.

Select **Cube**.



Then click **OK** and enter the cube details as shown below .



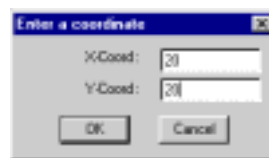
Then click **OK**.



Click the **coordinate icon** on the **position dialogue**.

The corner position dialog will show.

Enter the corner of the body coordinates **X- 20** and **Y- 20**



Then click **OK**.

The solid body for the bottle is then created.



Step 3. Changing the View



Click the **change 3D view icon** on the **menu dialogue**.

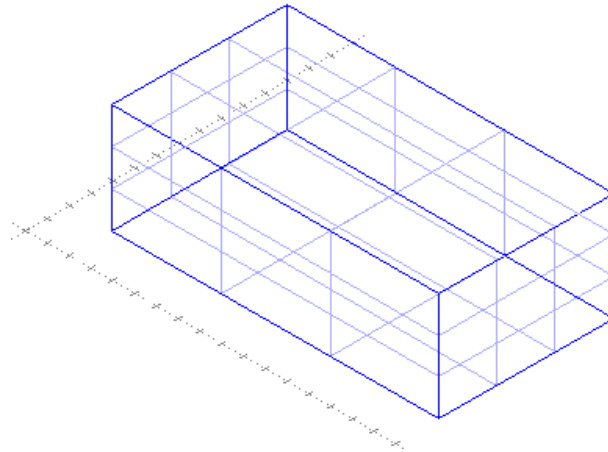


Click the **isometric view icon** on the **change 3D views dialogue**.



Click the **view all icon** on the **standard toolbar**.

The view should look like this now.



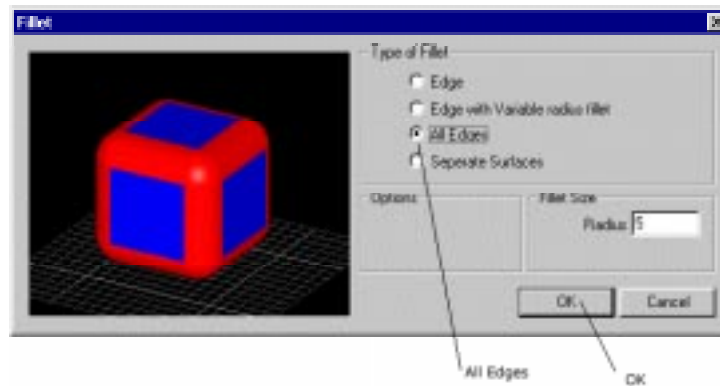
Step 4. Filletting the Body

We will now fillet the solid body with a 5 mm fillet.



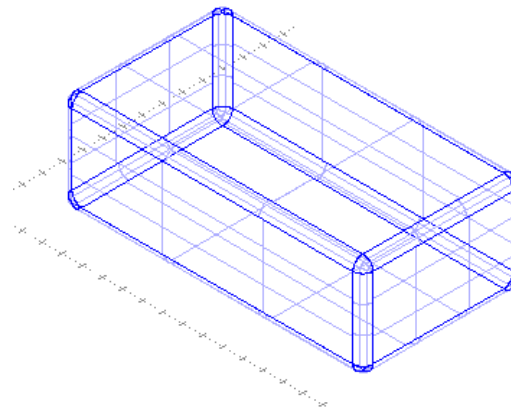
Click the **fillet surfaces or solids icon** on the **surfaces and solids toolbar**.

The following dialogue will appear.




Select **All edges** and enter the **radius 5mm**.

Then click **OK** and select the solid body.

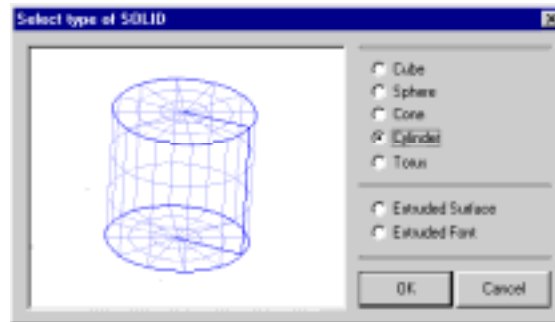


The body is then filleted on all edges.

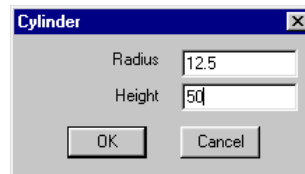
Step 5. Creating and Joining the Bottle Neck

 Click the **create solid icon** on the **surfaces and solids toolbar**.

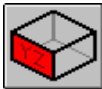
Select **cylinder**.



Then click **OK**.



Enter the values for the neck and click **OK**.

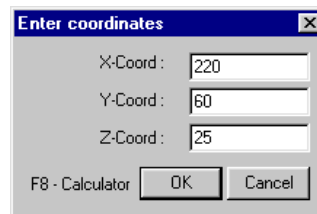


Click the **Y Z icon** on the **planes dialog**

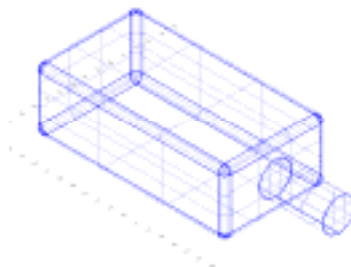


Click the **coordinate icon** on the **position dialog**.

Enter the values of the center of the body end.



Then click **OK** and the neck is created.



We will now union these two solids to form one.



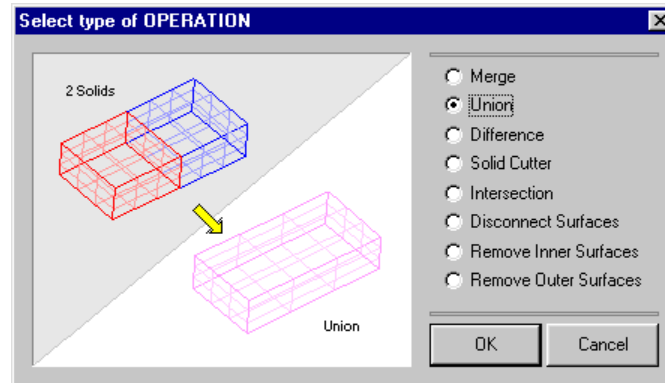
Click the **select single entity** icon on the **selections toolbar**

Select the **body** then the **neck** so that they are both red.



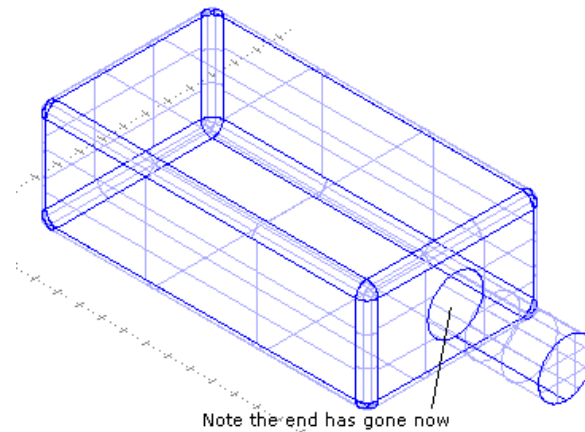
Click the **solids and surface operations** icon on the **solids and surfaces toolbar**

Select **union**.



Then click **OK**.

The two solid will then union to form one solid.



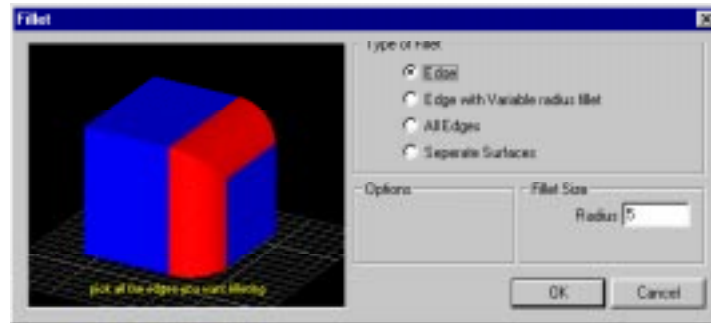
Step 6. Filletting the Neck

We will now fillet the neck of the bottle where it joins the body.

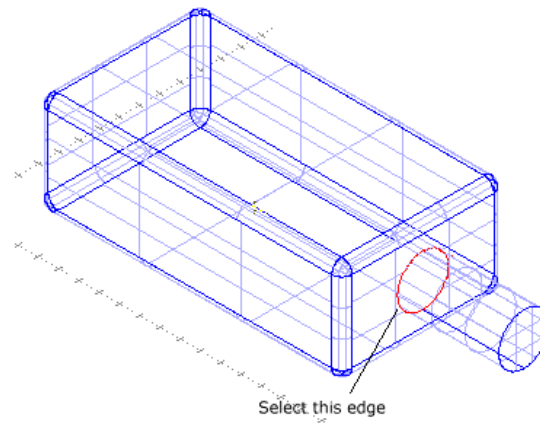


Click the **fillet surfaces or solids icon** on the **surfaces and solids toolbar**.

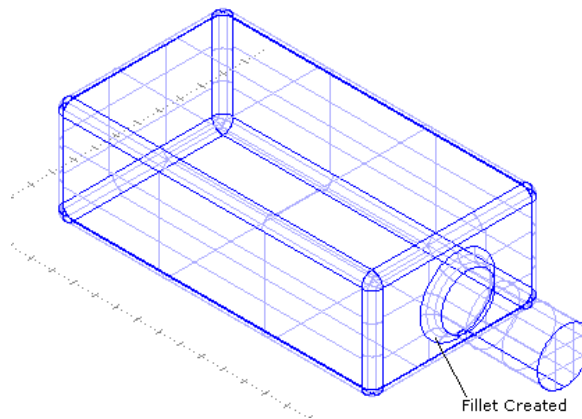
Then select **edge** and enter the **radius value** of the fillet and click **ok**.



Then select the **neck edge** and it will turn red.



Then click the **right mouse button** and the fillet will be created.



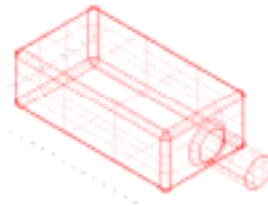
Step 7. Moving the bottle

We will now position the model where we want it to create the die cavity for it.



Click the **select single entity** icon on the **selections toolbar**

Then select the **bottle**.



The bottle will turn red after selection.



Click the **move/copy/scale** icon on the **selections toolbar**

The following dialogue will appear.

Then select **OK** as is, because we don't want to make a copy but just move it and keep the present scale.



The position dialogue will then appear.

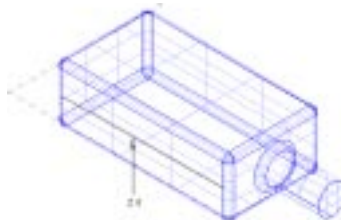


Click the **incremental** icon on the **position dialogue**.

Enter the value **- 25** in **Z** and **X Y** leave at **0**



Then click **OK**.



And the bottle is lowered in Z by 25 mm so the center of the bottle is at Z 0.

Step 8. Copying the Bottle

We will now make a copy of the bottle on another layer for use later.



Click the **select single entity** icon on the **selections toolbar**

Select the **model** it will turn red.



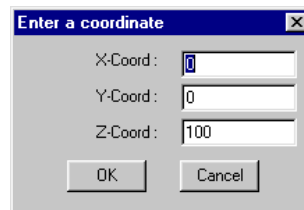
Click the **move/copy/scale** icon on the **selections toolbar**

Click **make copy** and click **OK**.

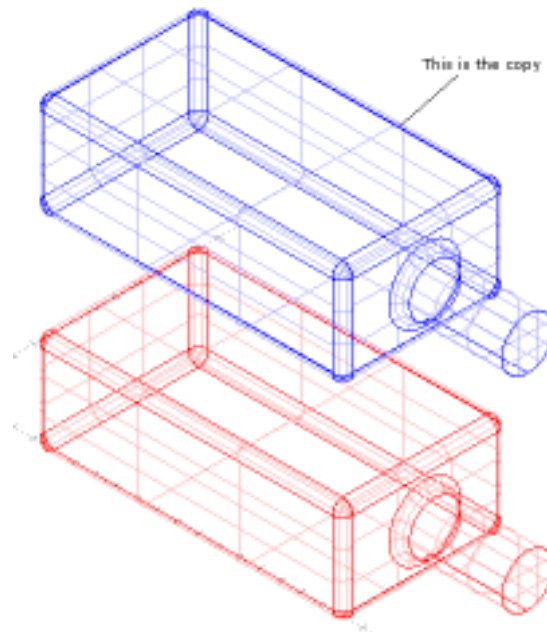


Click the **incremental** icon on the **position dialogue**.

Enter **Z: 100**.



Click OK and there will be a copy of the bottle made above the present one.



Step 9. Creating the Die Block

We made a copy as we are now going to make a die block and subtract the bottle from the die block and that copy will be lost, so that is why we have created a copy.

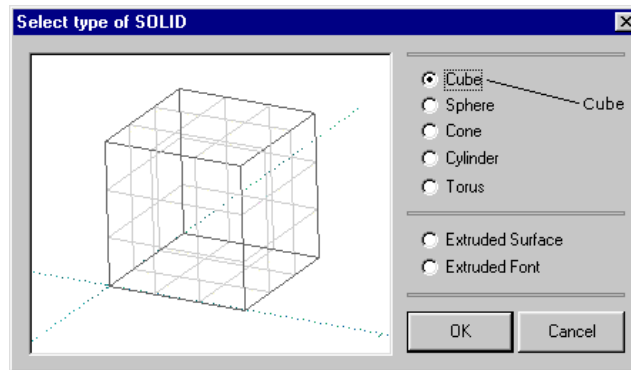
We will now create the die block.



Click the **create solid icon** on the **surfaces and solids toolbar**

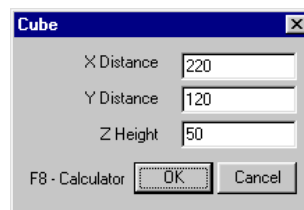
The following dialogue will appear.

Select **cube**.

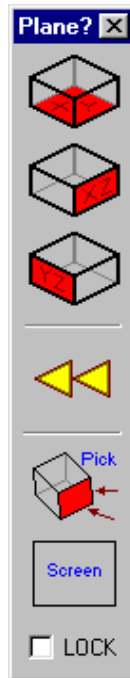


Then click **OK**.

Enter the following details.



Then select **OK**.



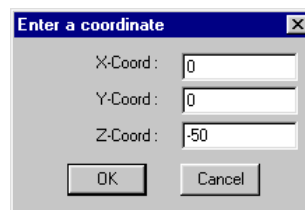
Select **XY** on the planes dialogue.



Click the **coordinate** icon on the **position** dialogue.

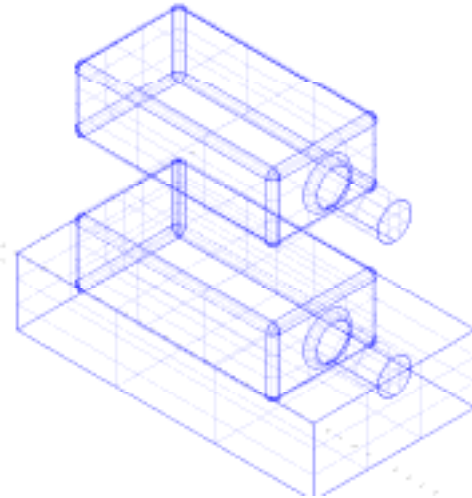
This will create the position the die block will sit at.

Enter **X: 0, Y: 0, Z: -50**.



Then click **OK**.

And the die block is created as shown below.



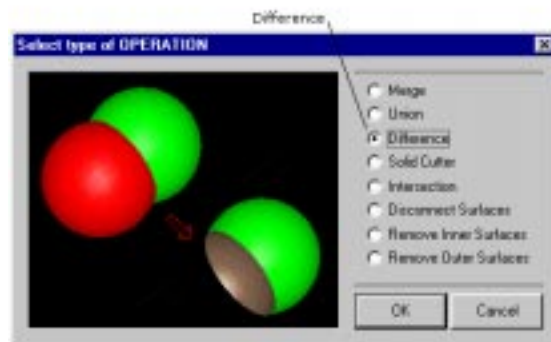
We will now subtract the model of the lower bottle away from the die block to leave the cavity for the half bottle.



Click the **solids and surface operations icon** on the **solids and surfaces toolbar**

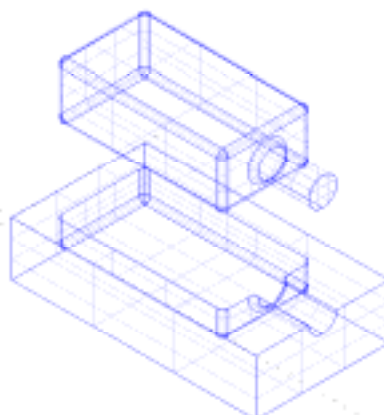
The following dialogue will appear.

Select **Difference**.



Click **OK**.

Then click and select the **die block** then the **lower bottle** and the cavity will be formed as shown below.



Step 10. Changing the Color and the Layer of the Bottle



Click the **select single entity** icon on the **selections toolbar**

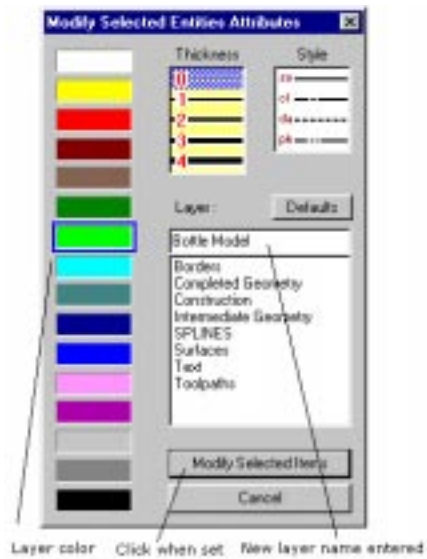
Then select the **bottle model** above the die cavity.

It will turn red.



Click the **modify selected entity** icon on the **menu toolbar**

Then select **green** and enter a **new layer name: Bottle Model**.



Then click **modify selected items**.

The layer and color will be changed.

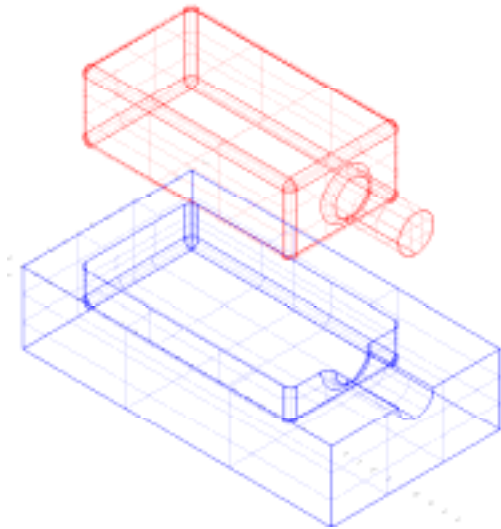
Step 11. Blanking the Green Bottle

We will blank the layer that the green bottle model is on.

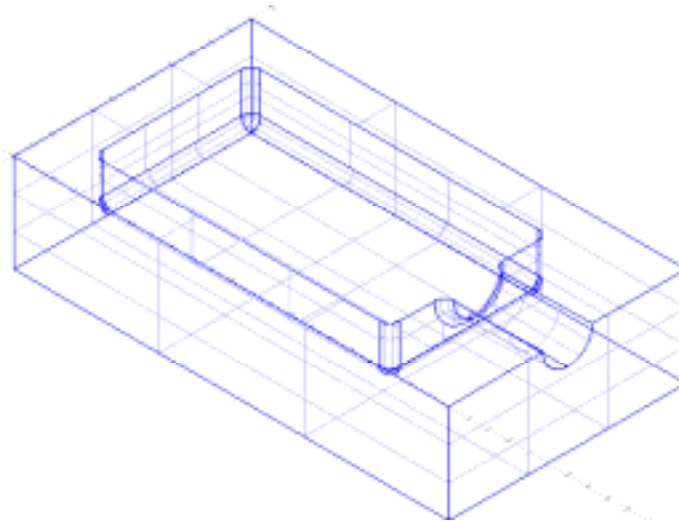


Click the **select single entity** icon on the **selections toolbar**.

Select the model and it will turn red.



Click the **blank** icon on the **selections toolbar**.



The bottle model is now not visible.

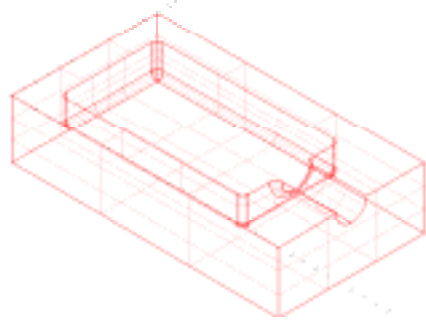
Step 12. Machining the Die

We are now ready to machine the die cavity.



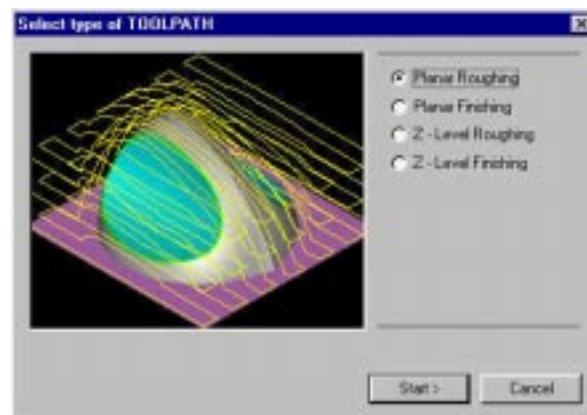
Click the **select single entity** icon on the **selections toolbar**.

Select the die cavity.

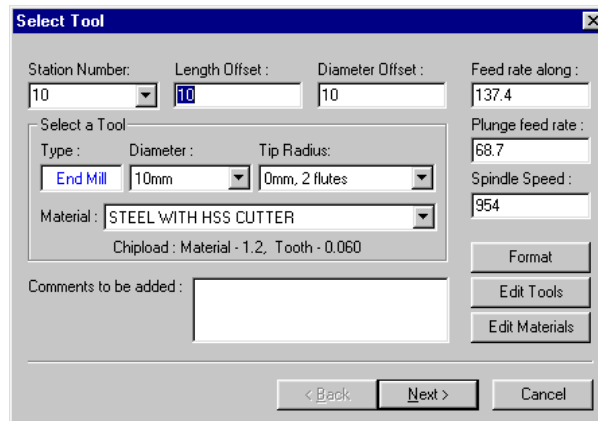


Click the **create NC code** icon on the **NC CAM toolbar**.

Tag **Planer Roughing** and click **start**.

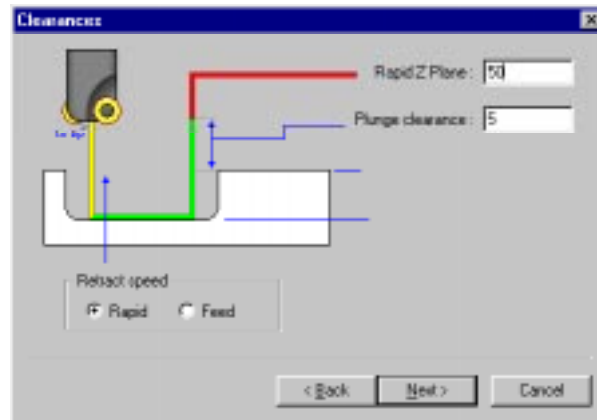


Now select the **tool type** by clicking the type box then select an **end mill** and select **10mm diameter**.

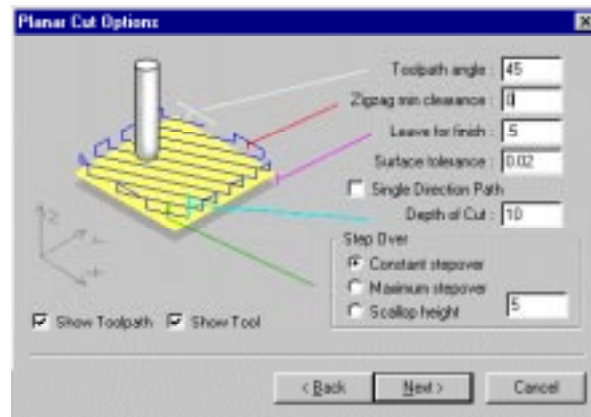


Then click **next**.

Enter the **clearances** and click **next**.

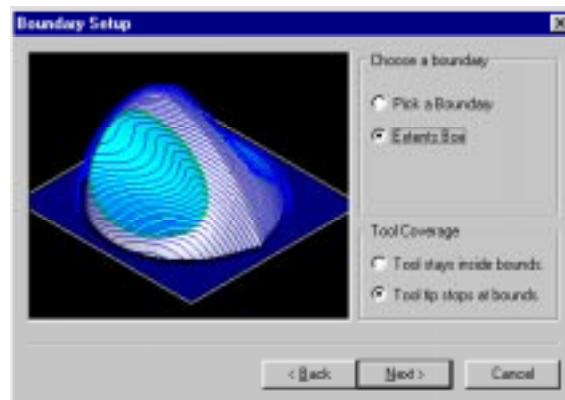


Enter the following details.



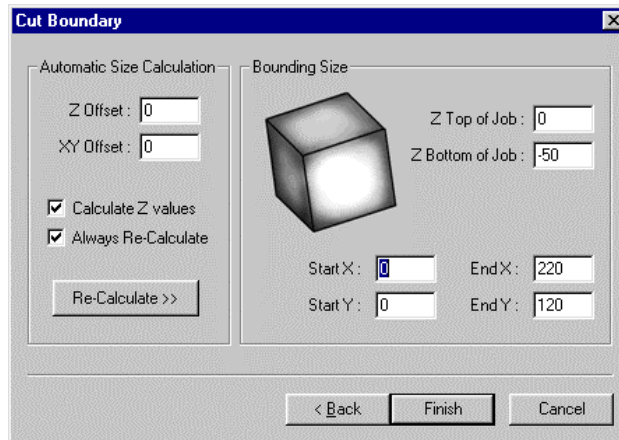
Then click **next**.

Set the following settings.

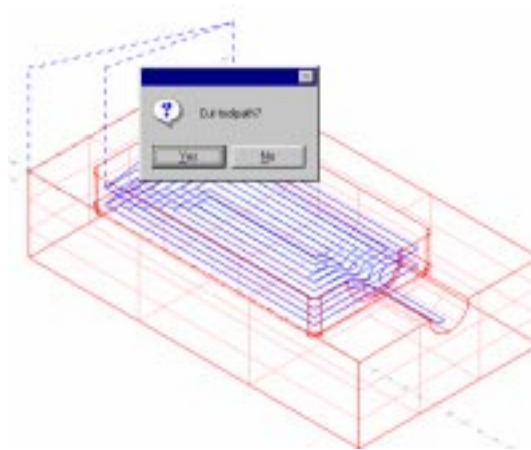


Click **next**.

Set the **settings** and click **finish**.



The Z level roughing toolpath will then be created.

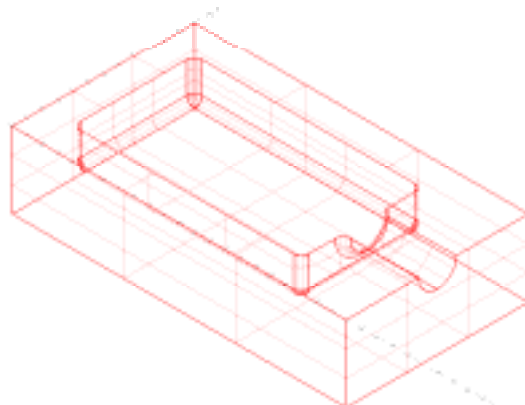


Click **yes** and the code will be saved to the NC editor.

We will now do the finish cut.

 Click the **select single entity** icon on the **selections toolbar**.

Select the cavity again and it will turn red.

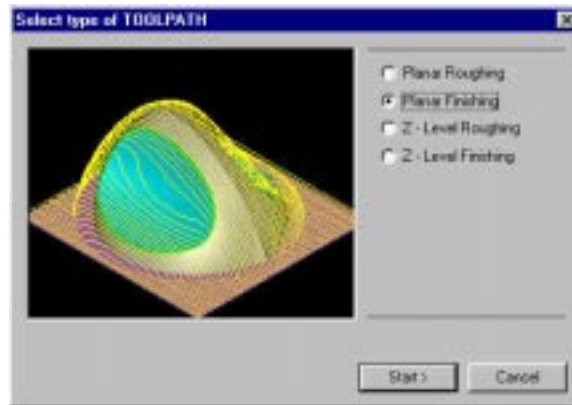


Select the machine model icon again.



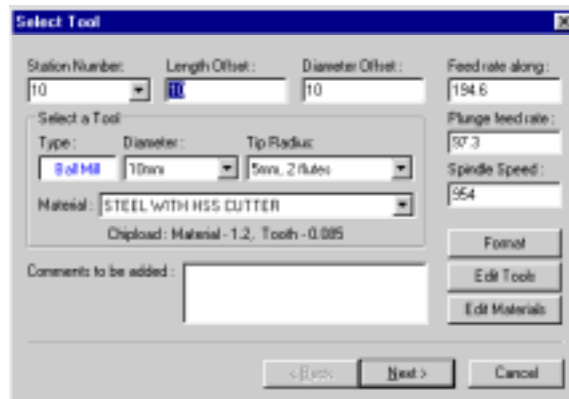
Click the **create NC code icon** on the **NC CAM toolbar**.

And select **Planar finishing**.



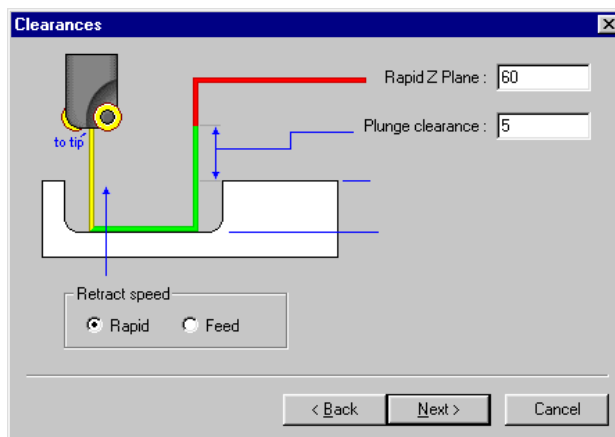
Then click **start**.

Select a **10mm ball mill**.



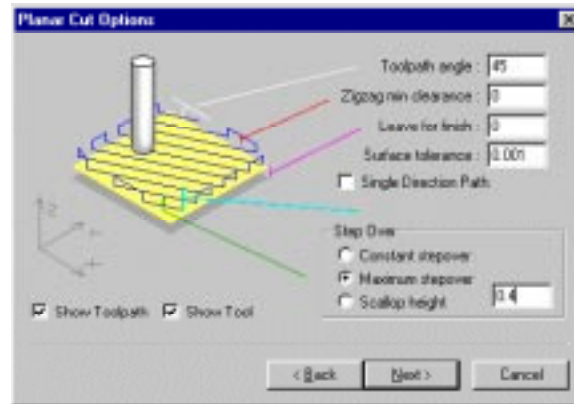
Then click **next**.

Set the **clearances**.



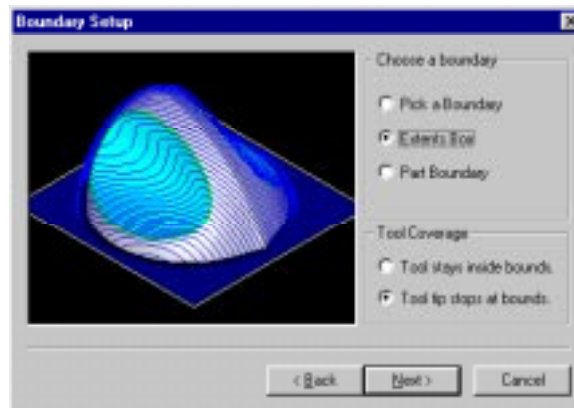
Then click **next**.

Set the **settings**.

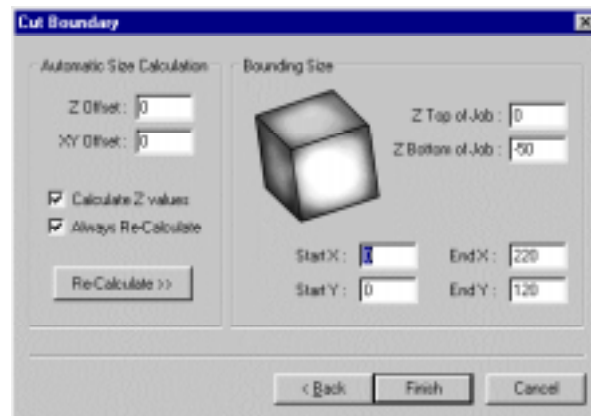


Click **next**.

Set the **settings**.



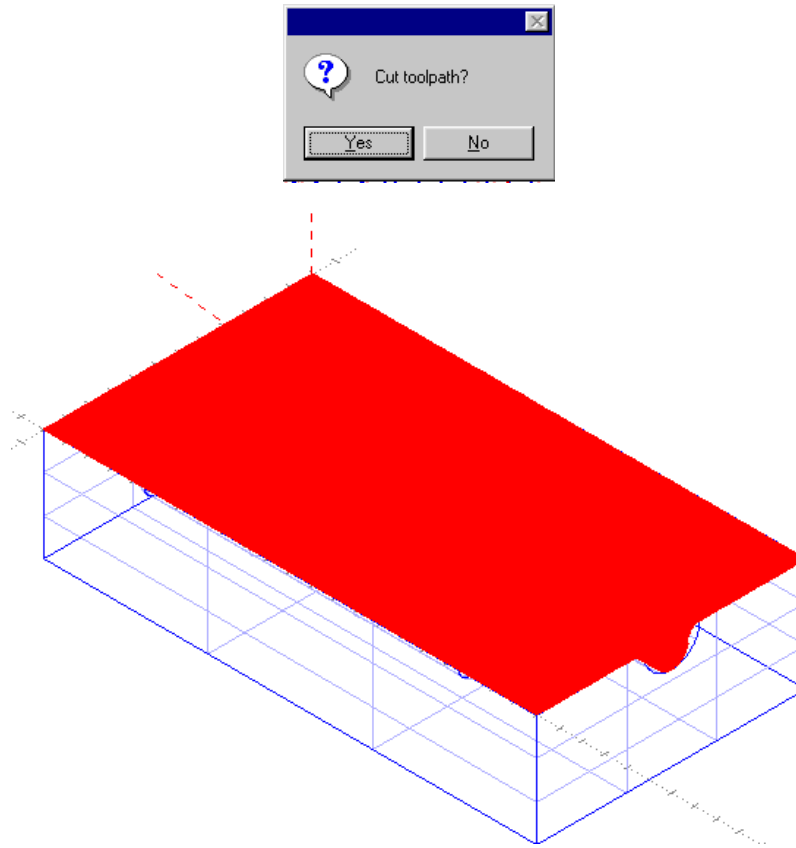
Click **Next**.



Then click **finish**.

The toolpath will be created.

Click **Yes** when asked to cut the toolpath.



Both programs are now in the NC editor ready to be machined.

Step 13. Simulating the NC Code



Click the **simulate NC** icon on the **NC CAM** toolbar.

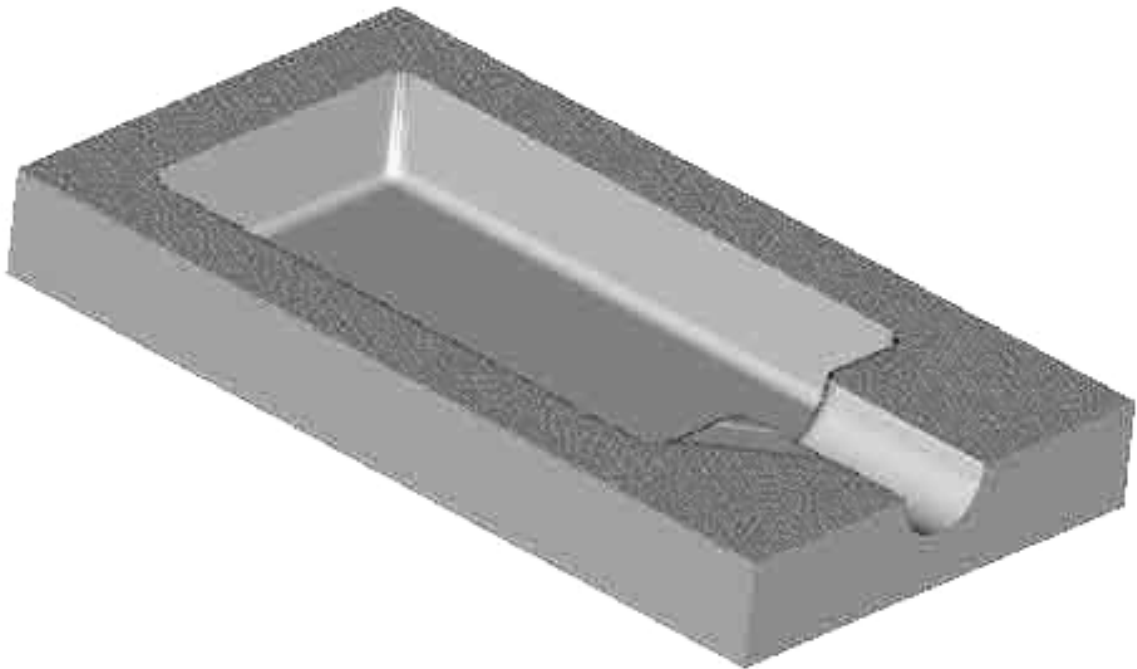
The following dialogue will appear.

Enter the following details and click **OK**.

The dialog box is titled "NC Solid Verification" and contains the following fields and options:

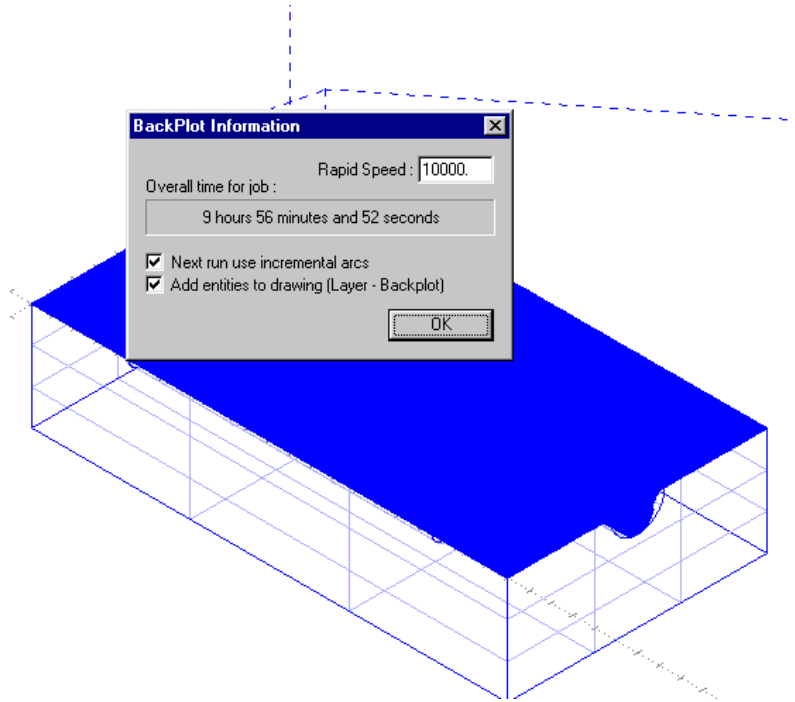
- Material Size:**
 - Start X: 0
 - Width in X: 220
 - Start Y: 0
 - Height in Y: 120
 - Top of Material: 0
 - Bottom of Material: -26
- Temporary Tool Information:**
 - End Mill, Dia-10, Tip-0
 - Ball Mill, Dia-10, Tip-5
 - Dia: []
 - Tip: []
- Quality:**
 - Turbo (Coarse)
 - Slow (Fine)
 - A slider bar is positioned between Turbo and Slow.
- Options:**
 - Stop at rapid gouge!
- Buttons:** Extents, Always (checked), OK, Cancel

The toolpath is then modeled.




Step 14. Obtaining the Machining Time

 Click the **backplot icon** on the **NC CAM toolbar**.



This has back plotted both programs and calculated the machine time.

To remove the back plot drawing.

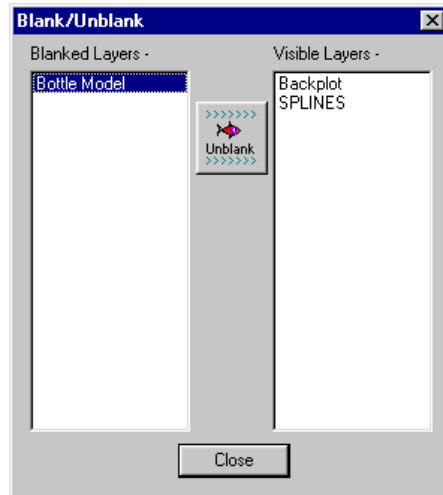
 Click the **undo icon** on the **standard toolbar**.

Step 15. Unblanking the Green Bottle



Click the **layer icon** on the **selections toolbar**.

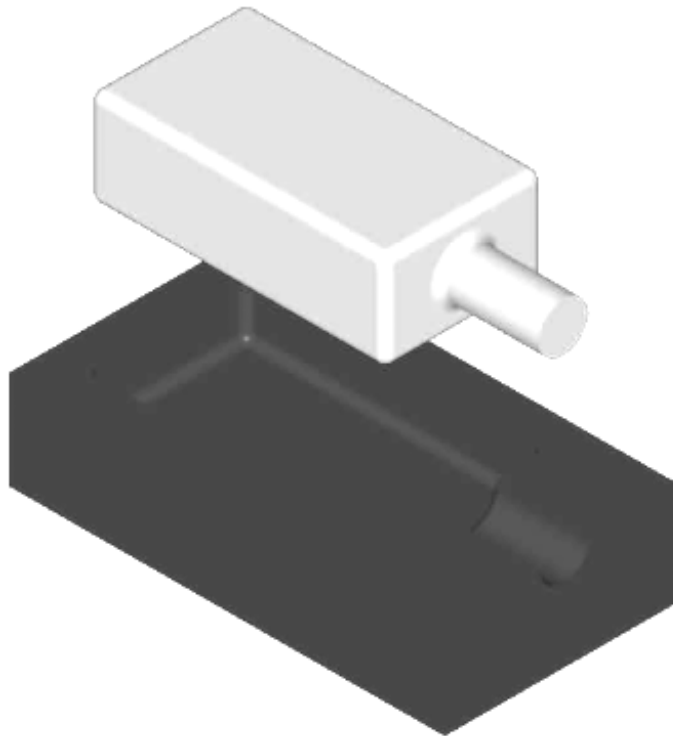
Double click **bottle model** to un blank the bottle model.



To See the completed job.



Click the **render icon** on the **standard toolbar**



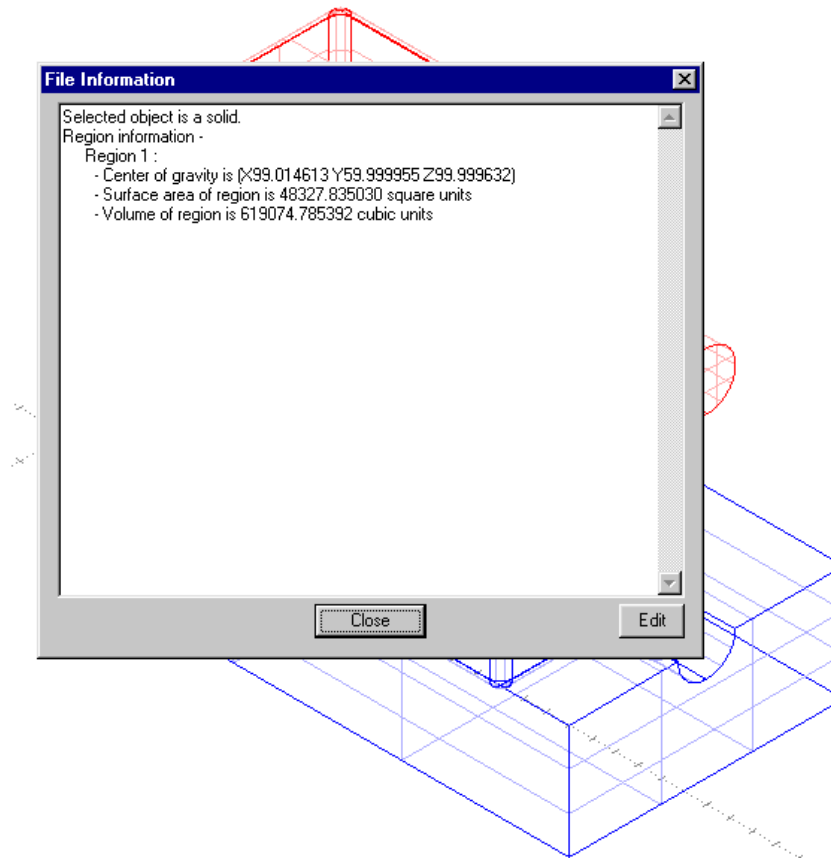
Step 16. Viewing Information on the Bottle

With a solid model you also have other valuable information about the Solid.



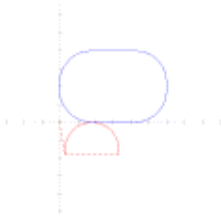
Click the **verify single entity** icon on the **standard toolbar**.

Then select the **solid model of the bottle** and the following information will be displayed.

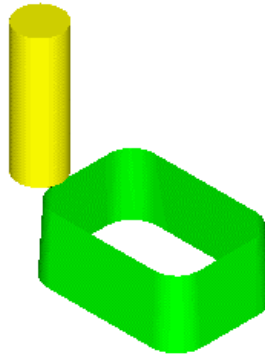


Mill Tutorial 6

Use Profile Function



Note: This help uses mm values.



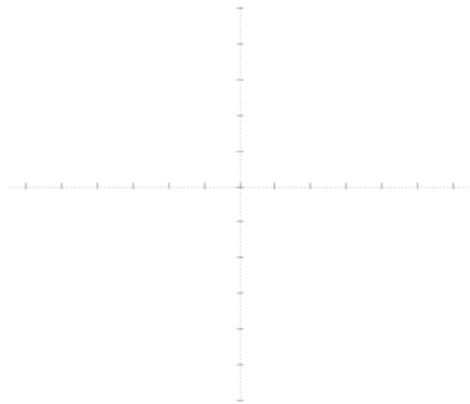
In this tutorial we use the mill profile function.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw your wireframe and then surface it.



Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

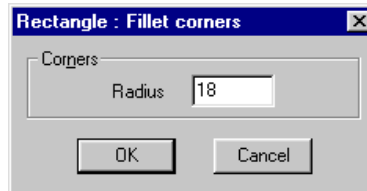
Step 2. Create a Shape to be Profiled

 Click the **line icon** on the **menu toolbar**.

 Click the **rectangle icon** on the **line menu toolbar**.

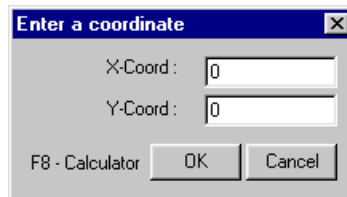
The following dialogue will appear.

Enter **radius: 18** and click **OK**



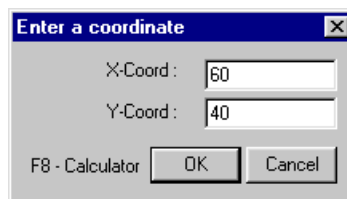
 Click the **position by coordinate icon** on the **position dialogue**

Enter the following details into the coordinate dialogue.

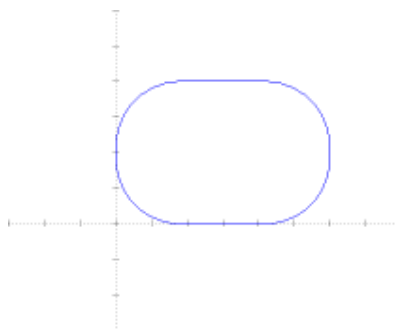


 Click the **position by coordinate icon** on the **position dialogue**

Enter the following details to finish the rectangle and click **ok**.



Click **Cancel** to terminate the rectangle function.

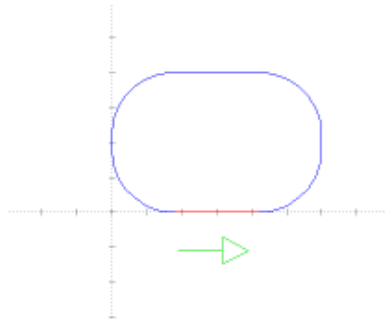


Step 3. Create NC Code

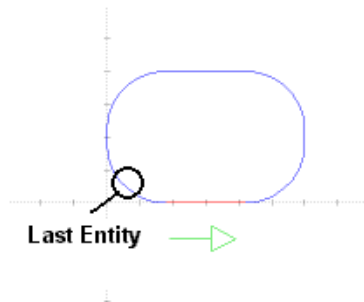


Click the **mill profile icon** on the **NC CAM dialogue**

Select the **first entity** and put the arrow in the **direction of the cut**.



Then select the last entity



The following dialogue will appear.

Set your **tool type**, and **material** and click **next**.

Select Tool

Station Number: 5 Length Offset: 5 Diameter Offset: 5 Feed rate along: 716

Select a Tool

Type: End Mill Diameter: 10mm Tip Radius: 0mm, 2 flutes

Material: ALUMINIUM WITH HSS CUTTER

Chipload: Material - 2.5, Tooth - 0.060

Plunge feed rate: 358

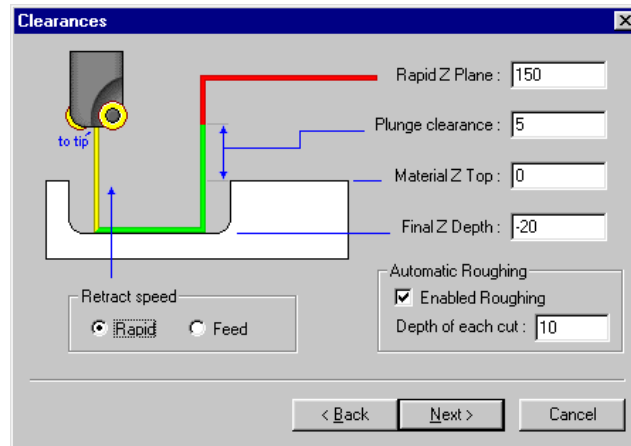
Spindle Speed: 2385

Comments to be added:

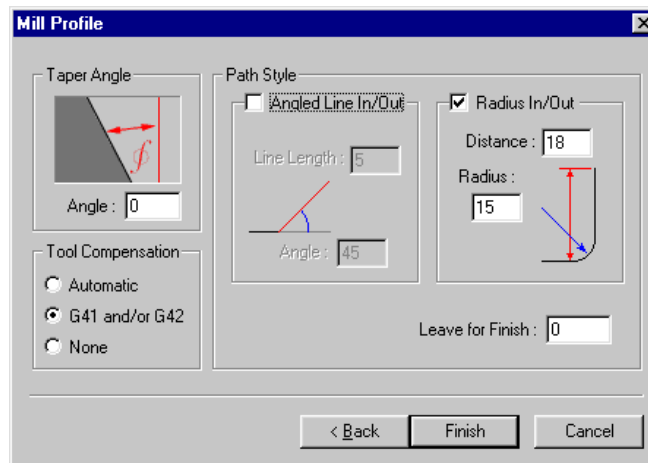
Format Edit Tools Edit Materials

< Back Next > Cancel

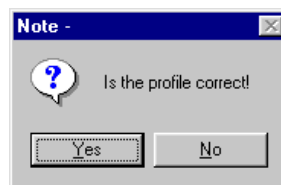
Set your **clearances** and click **next**.



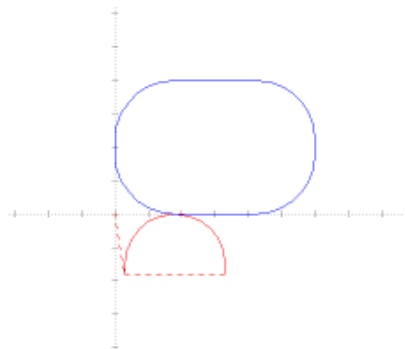
Enter the following details and click **finish**.



If the profile is correct then press **Yes**.



And the tool path is cut and code placed in the NC editor.



Step 4. Simulating the NC code



Click the **simulate NC** icon on the **NC CAM** toolbar.

The following dialogue will appear.

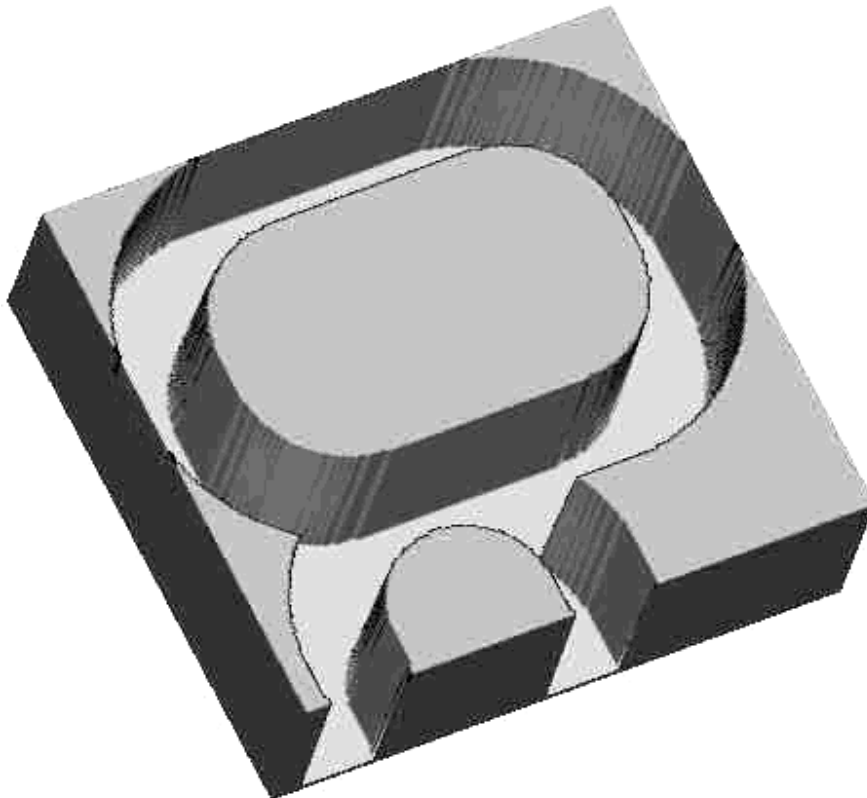
Enter the following details and click **OK**.

The dialog box titled "NC Solid Verification" contains the following fields and options:

- Material Size:**
 - Start X: -5
 - Width in X: 70
 - Start Y: -20
 - Height in Y: 65
 - Top of Material: 0
 - Bottom of Material: -21
- Extents:** Always
- Quality:** Turbo (Coarse) | Slow (Fine) with a slider.
- Options:** Stop at rapid gouge!
- Temporary Tool Information:** End Mill, Dia-10, Tip-0
- Buttons:** OK, Cancel

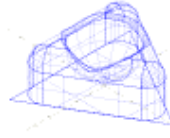
Note: Extents always box is not tag to help show the modeled toolpath for demonstration purposes extents should usually be tagged.

The toolpath is then modeled.



Mill Tutorial 7

Create Wireframe Surface and Solid



This tutorial is designed to teach you how to create a model using wireframe, surface and solid geometry.

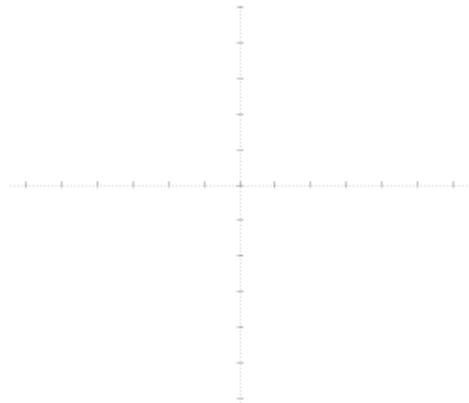
Note: The unit values shown in this help topic are inch values.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw a triangle.




Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

Step 2. Creating a Wireframe Triangle

 Click the **create polygon icon** on the **line menu toolbar**.

The following Dialogue Box will Appear:

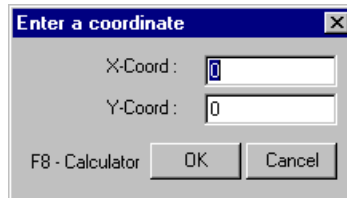


Enter the following details and click **OK**.

The Position Toolbar will now appear.

 Click the **coordinate icon** on the **position dialogue**.

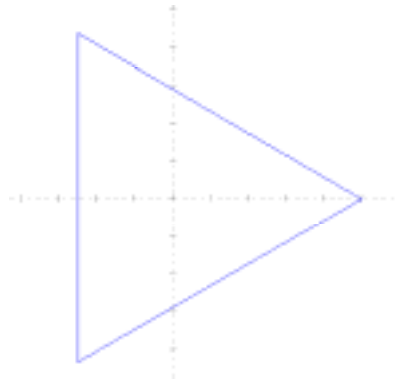
The following dialogue will appear enter **X: 0, Y: 0** and click **OK**



Click **Cancel** to end this function, when the polygon dialogue reappears.

 Click the **view all icon** on the **standard toolbar**.

 Click the **reduce half icon** on the **standard toolbar**.



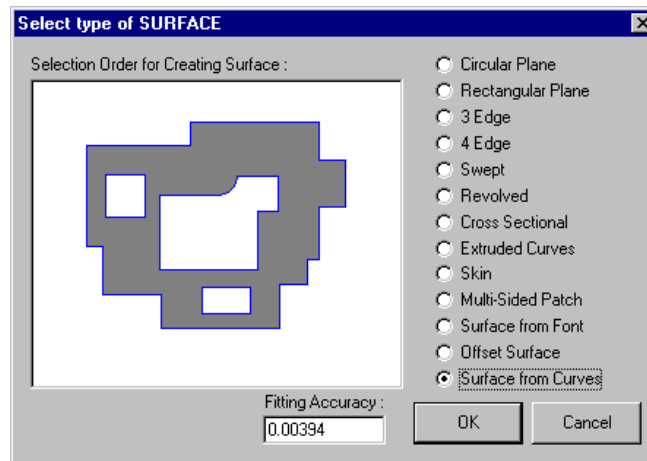
Step 3. Surfacing the Triangle

From here we will use the chain that consists of the three lines that make up the polygon to create a surface.

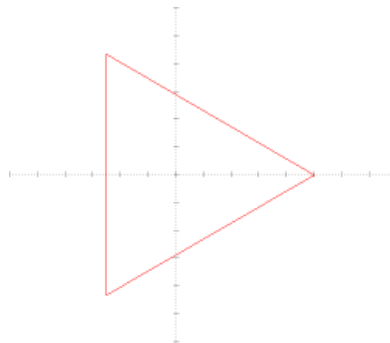


Click the **create a surface icon** on the **surfaces and solids toolbar**

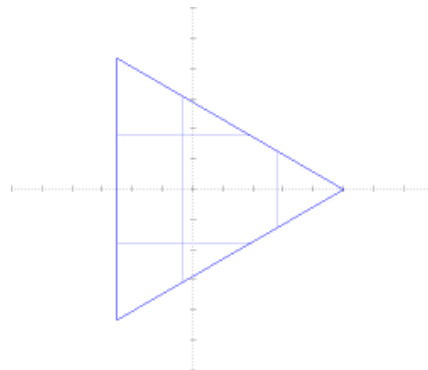
Click the **surface from curves** box and click **OK**.



Click the **left mouse button** onto **any line** on the triangle



Then click the **right mouse button** and the surface will be created.



Step 5. Creating a Solid from the Triangle

We will now create a solid by extruding the surface 3 inches up in Z.

Now we will turn the triangle in to a 3D object.

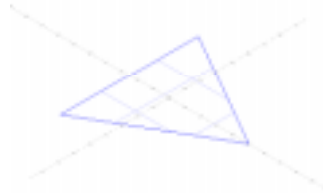


Click the **change 3D views icon** on the **menu toolbar**.



Click the **Isometric view icon** on the **3D views dialogue**.

You will now see it in isometric view.

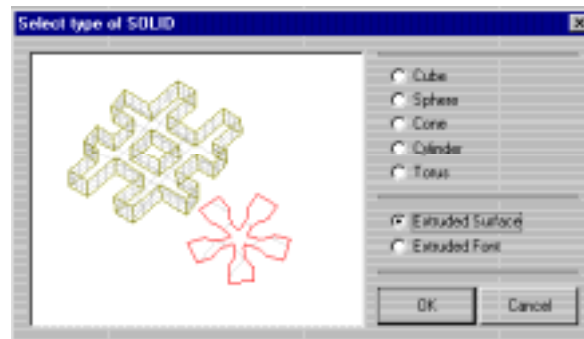


Now we will create a solid from the triangle.

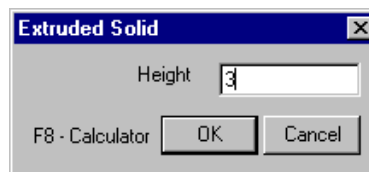


Click the **create solid icon** on the **surfaces and solids toolbar**

The following dialogue will appear:

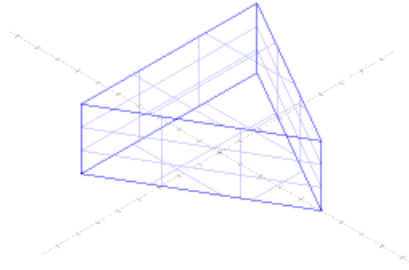


Select **Extruded Surface** and Click **OK**. The following dialogue box will appear:



Enter **height: 3** and Click **OK**.

Select the **triangle** with the **left mouse button** to create the solid.



The result is the surface has been extruded up 3" to create a solid.

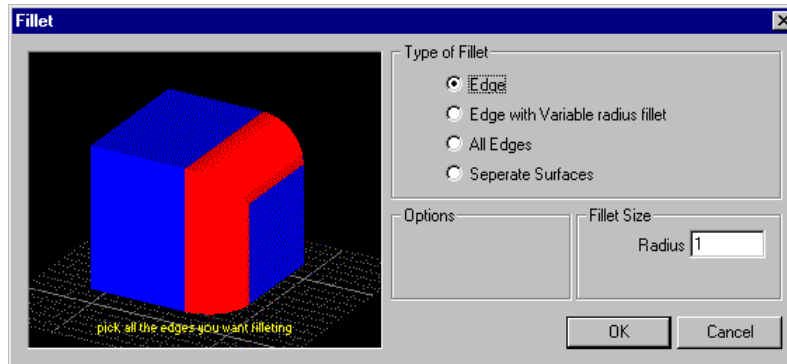
Step 6. Filletting the Solid

We will now place a 1" fillet between the top and sides and the corners of the model.



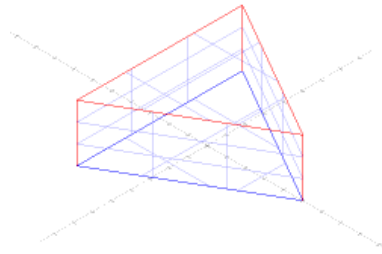
Click the **fillet surfaces or solids icon** on the **surfaces and solids toolbar**.

The following dialogue box will appear:



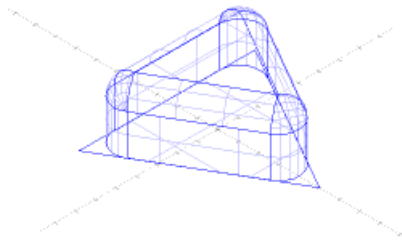
Select **Edge** for Type of Fillet and enter Radius: 1 and click **OK**.

Using the **left mouse button** select all the **edges that make up the top and the side** of the solid.



Click the **right mouse button** to create the filleting.

A 1" fillet is created between all corners and between the side and to surfaces as seen below.

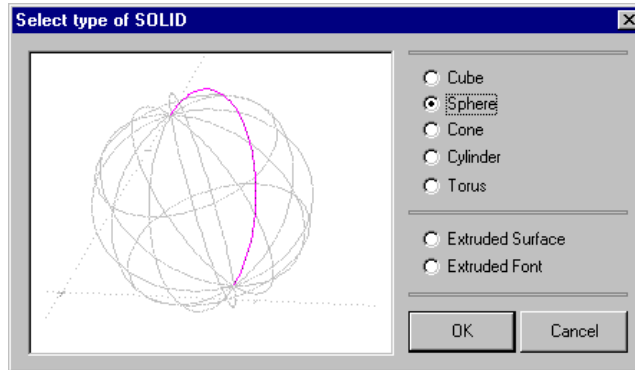


Step 7. Creating a Sphere for the Difference Function



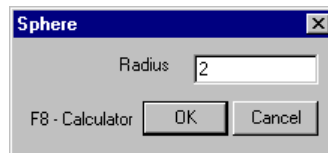
Click the **create solid icon** on the **surfaces and solids toolbar**

The following dialogue box will appear:



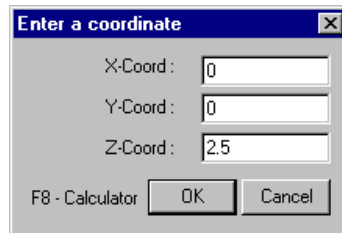
Select **Sphere** and Click **OK**.

Enter **Radius: 2** and Click **OK**

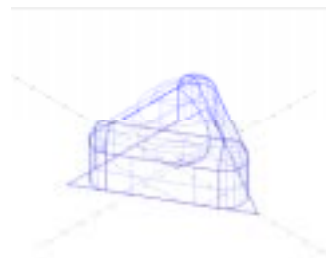


Click the **coordinate icon** on the **position dialogue**.

Enter **X: 0, Y:0, Z: 2.5** in the dialogue box, then click **OK**.



A 4" diameter sphere is drawn with a center at $X_0, Y_0, Z_{2.5}$ and intersects the triangular solid we created.



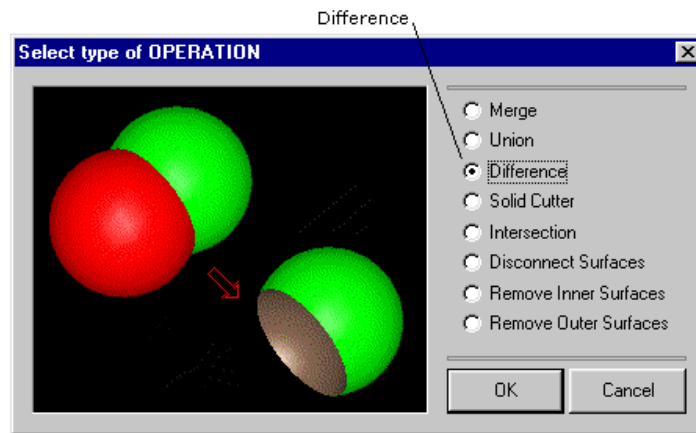
We are now going to remove the section of the sphere that intersects the triangular solid and create a cavity.



Click the **solids and surface operations icon** on the **solids and surfaces toolbar**

The following dialogue will appear.

Select **Difference**.

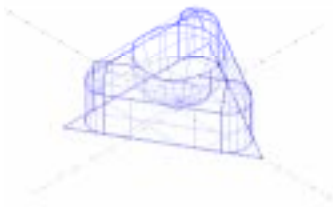


Click **OK**.

First click the **object to be modified**, which is the **triangular base**.

Secondly click the **object to be removed**, which is the **sphere**.

The sphere has been removed from the triangular solid where they intersect.



Step 8. More Filleting

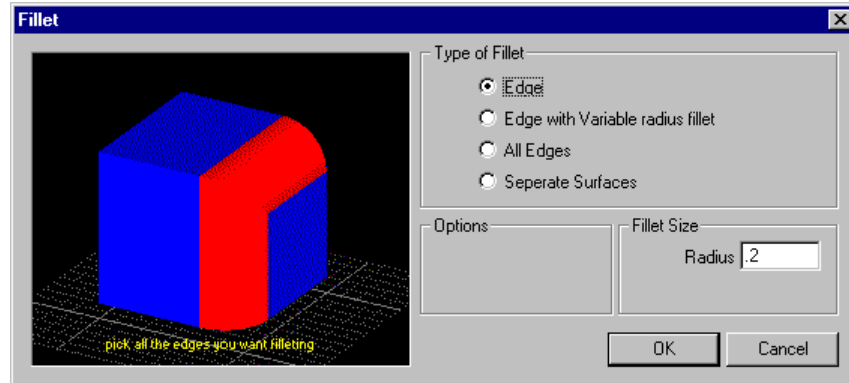
We will now put a .2' fillet along the edge where the sphere was removed from the solid.



Click the **fillet surfaces or solids icon** on the **surfaces and solids toolbar**.

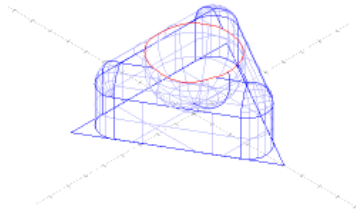
The Fillet dialogue box will appear like below:

Select **edge**.



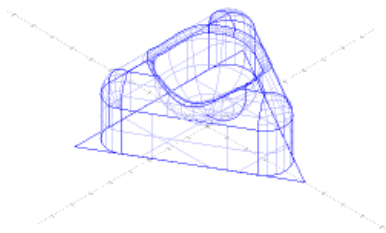
Enter **Radius: .2** and click **OK**.

With the left mouse button select **all segments** that make up the circle on top. So that your drawing should match the drawing below.



After all have been selected **right mouse click**.

A .2" radius is created around the circular opening at the top of the model.



Step 9. Rendering the Object



Click the [render](#) icon on the **standard toolbar**.

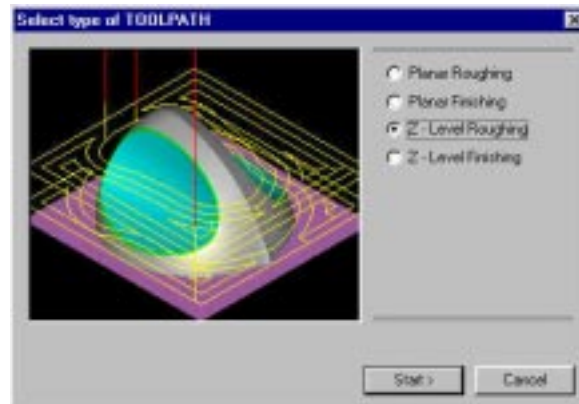


Step 10. Z-Level Roughing

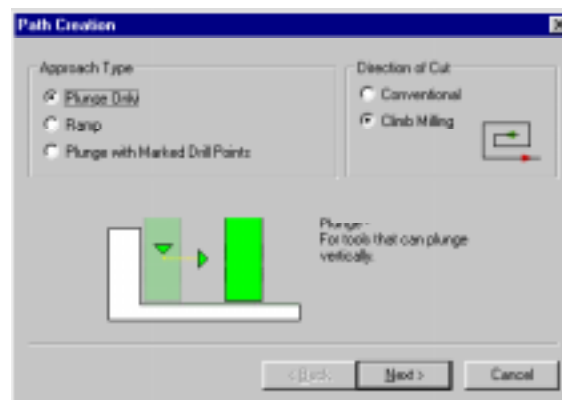
 Click the **create NC** icon on the **NC CAM** toolbar.

The toolpath wizard will appear.

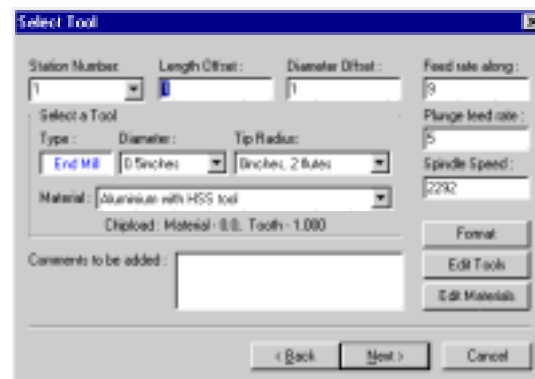
Select **Z-Level Roughing** and click **start**.



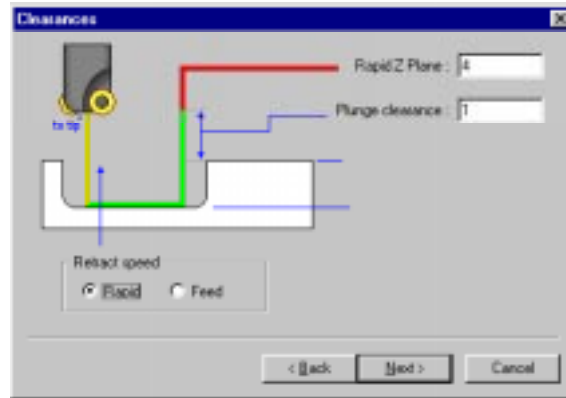
Set your **approach type** and your **direction of cut** and click **next**.



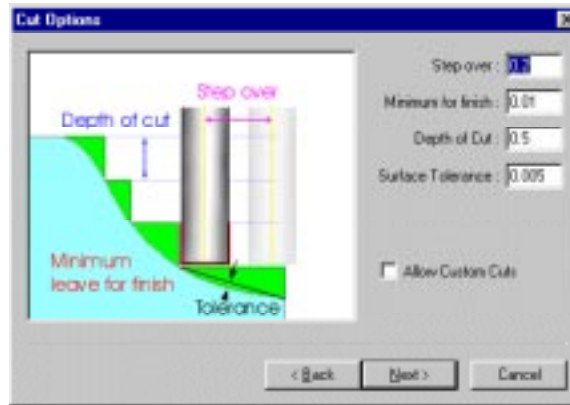
Select your **tool** and click **next**.



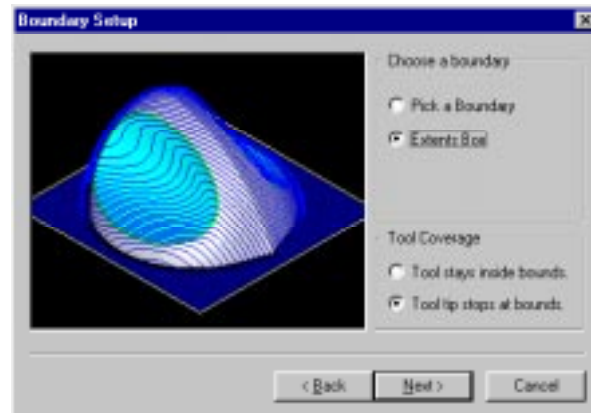
Set your **clearances** and click **next**.



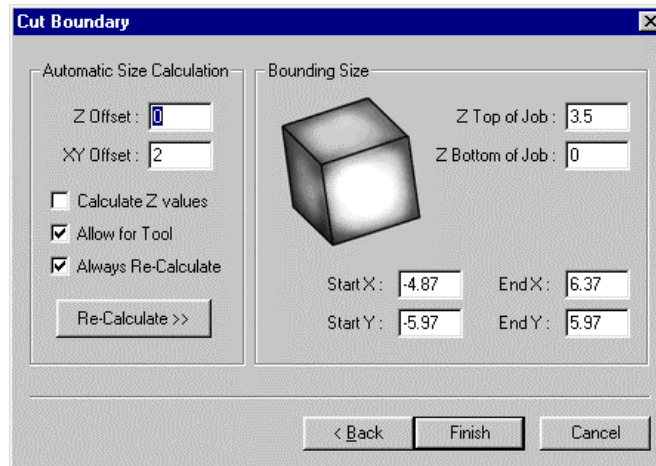
Set your **cut options** and click **next**.



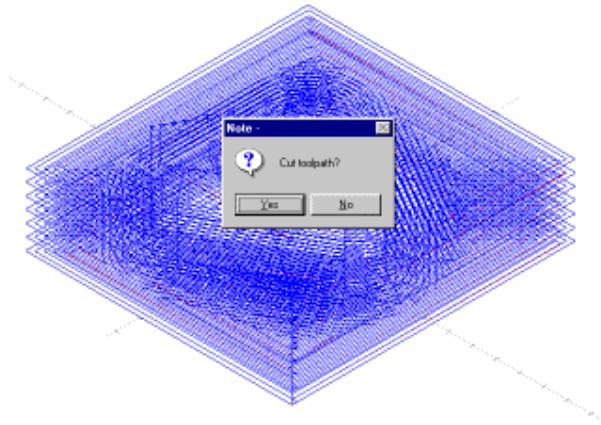
Set your **boundary** and click **next**.



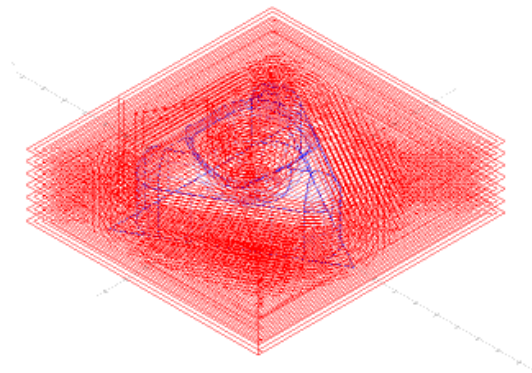
Set your **cut boundary** and click **finish**.



Your toolpath is then drawn, click **yes** to cut the tool path.



The NC Code is then created and placed in the NC editor.



Click the **simulate NC** icon on the **NC CAM** toolbar.

The following dialogue will appear.

Enter the following details and click **OK**.

Material Size

Start X: -4.8701 Width in X: 11.2403

Start Y: -5.9704 Height in Y: 11.9407

Top of Material: 3.5

Bottom of Material: -0.1

Extents Always

Temporary Tool Information

End Mill, Dia-0.5, Tip-0

Dia: Tip:

Quality

Turbo (Coarse) Slow (Fine)

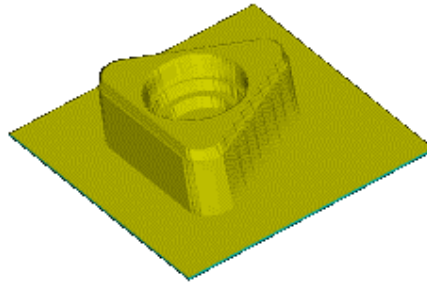
Options

Stop at rapid gouge!

OK Cancel

Note: Extents always box is not tag to help show the modeled toolpath for demonstration purposes extents should usually be tagged.

The roughing toolpath is then simulated.

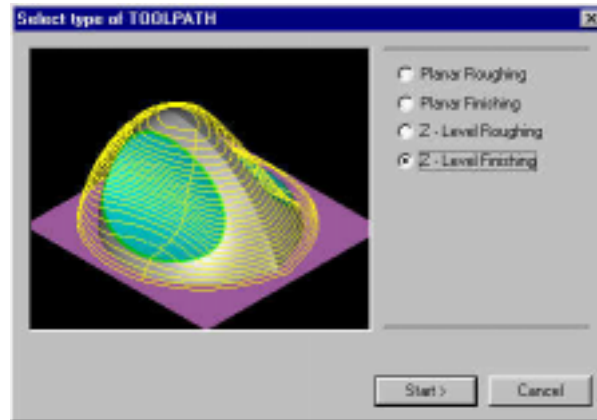


Step 11 Z-Level Finishing

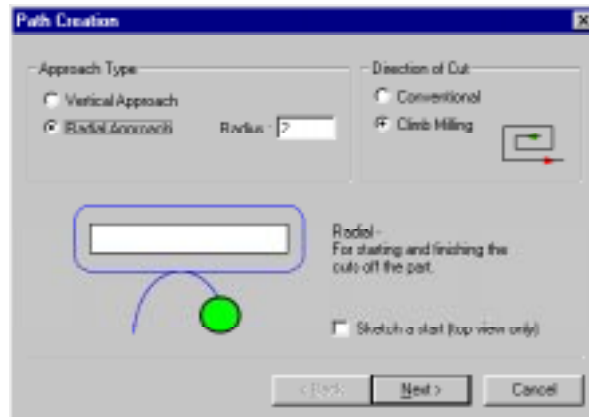
 Click the **create NC** icon on the **NC CAM** toolbar.

The toolpath wizard will appear.

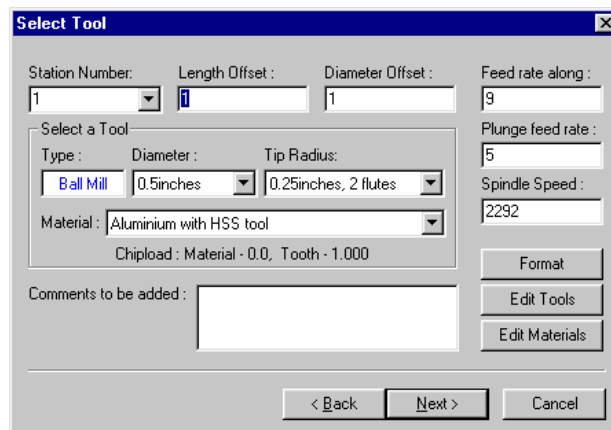
Select **Z-Level Finishing** and click **start**.



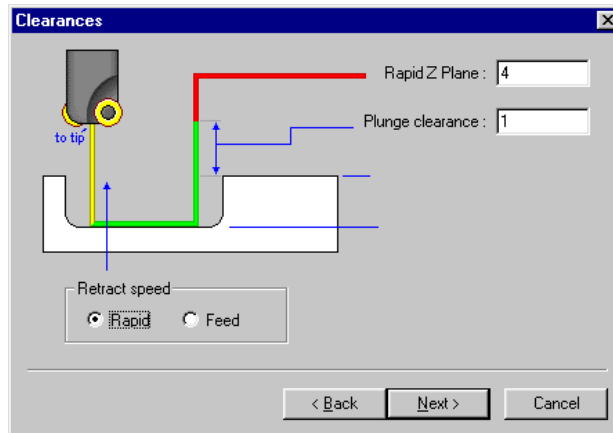
Set your **path Creation** and click **next**.



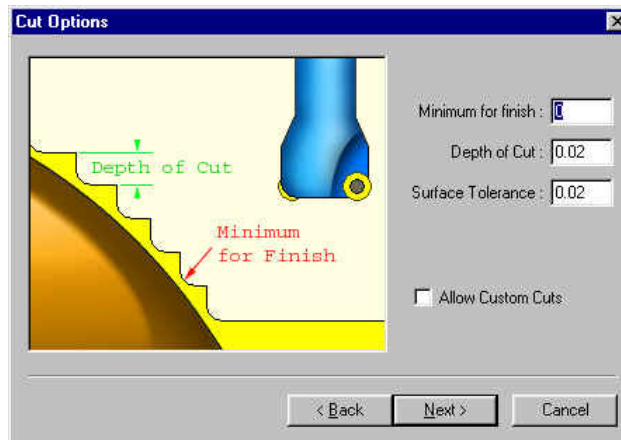
Set your **tool** and click **next**.



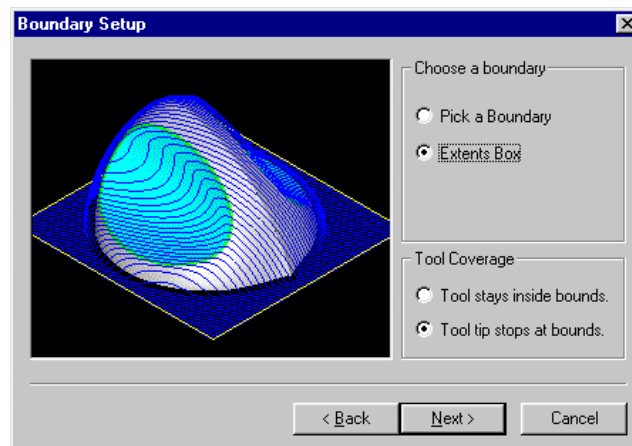
Set your **clearances** and click **next**.



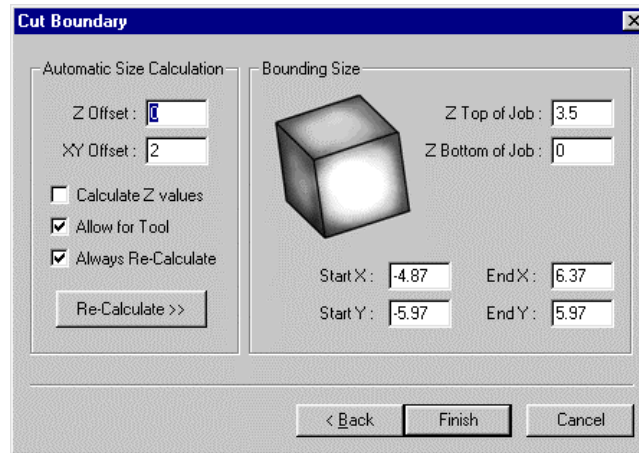
Set your **cut options** and click **next**



Set your **boundary** and click **next**.



Set your **cut boundary** and cut **finish**.



Your Toolpath will then be created.

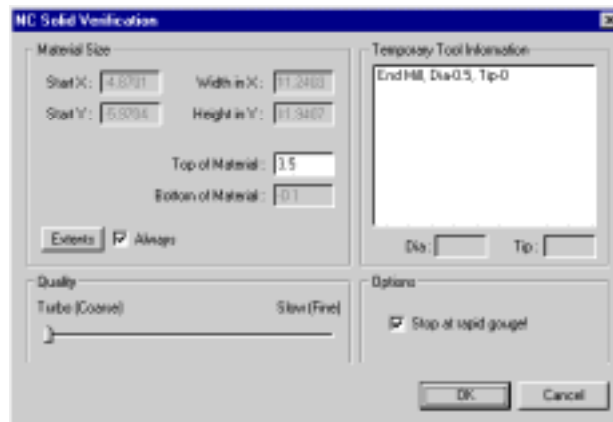
Click **yes** to cut the toolpath and the code will be created and placed in the NC editor.



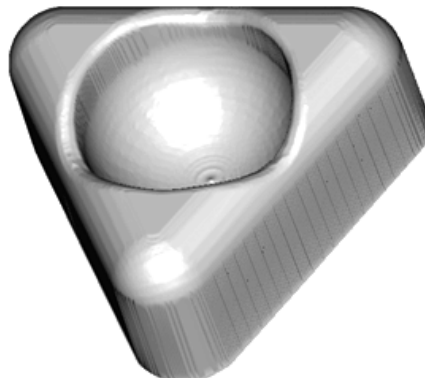
Click the **simulate NC icon** on the **NC CAM toolbar**.

The following dialogue will appear.

Enter the following details and click **OK**.



The roughing and finishing toolpath will then be created.



Mill Tutorial 8

Draw 2D Text and Use 4-Axis Wrap



In this tutorial will will draw 2D text and cut using 4-axis wrap.

This tutorial require the 4-axis module to operate.

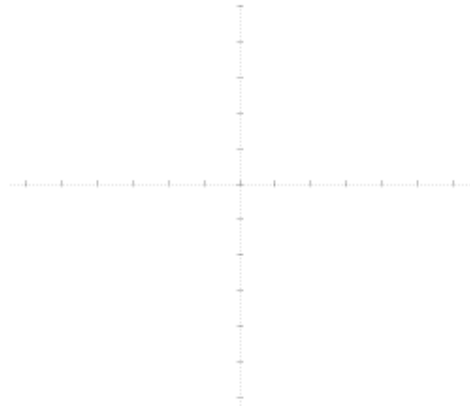
NOTE: this tutorial uses mm values.

Step 1. Create a New Drawing



Click the **new drawing icon** on the **standard toolbar**.

You should have a blank drawing ready to draw.



Now to make sure the NC file is clear



Click the **new NC icon** on the **NC editor toolbar**.

Click **Yes** or **No** to save your file.

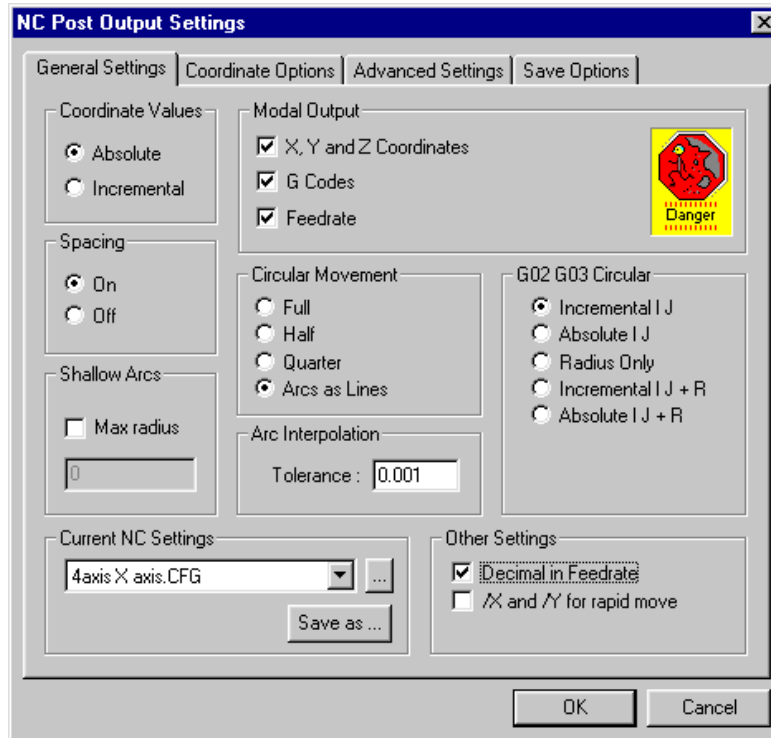
Step 2. Setting Up 4-Axis

Click on **NC Setup** on the top menu bar

NC Setup

The following dialogue will appear

Select **4axis X axis.CFG** and click **OK**



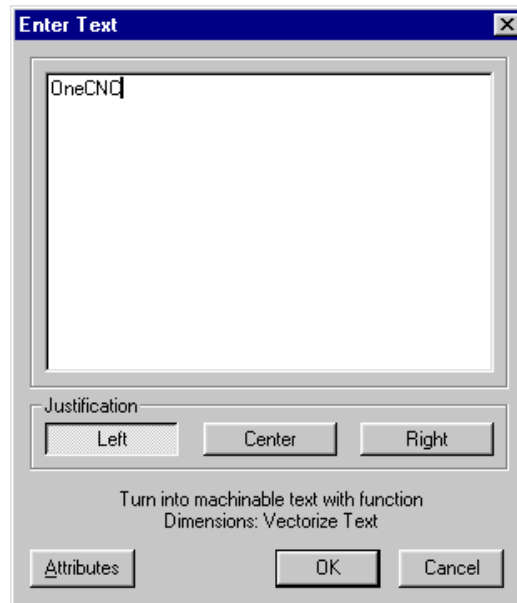
Step 3. Creating Text

A Click the **text icon** on the **menu toolbar**.

A Click the **create text icon** on the **text menu toolbar**.

The following dialogue will appear.

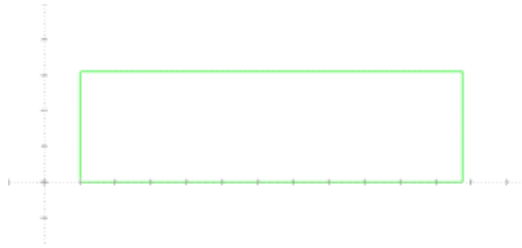
Enter the the following **details** and click **attributes**.



Enter the following and click **OK**



Click OK again and the text as shown below.



Your text is then created.

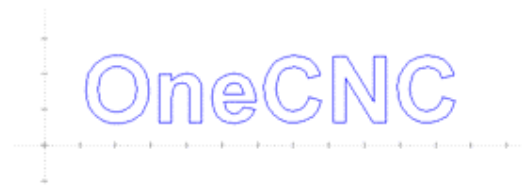


Click **cancel** to terminate the text dialogue.



Click the **vectorize text icon** on the **text menu toolbar**.

Select the text and it will become vectorized.

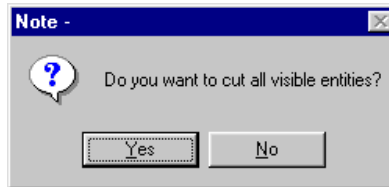


Step 4. Cutting the Toolpath

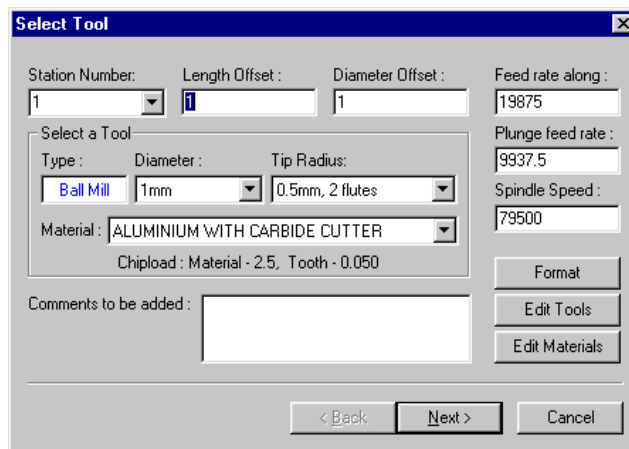


Click the **cut a visible entities in 2D** icon on the **NC CAM** toolbar.

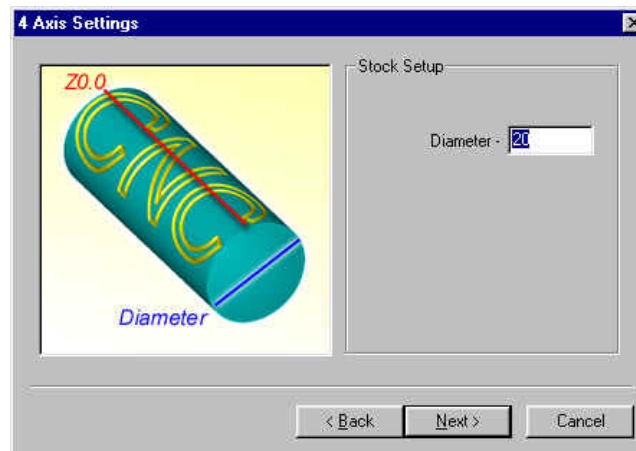
The following will appear select **Yes**.



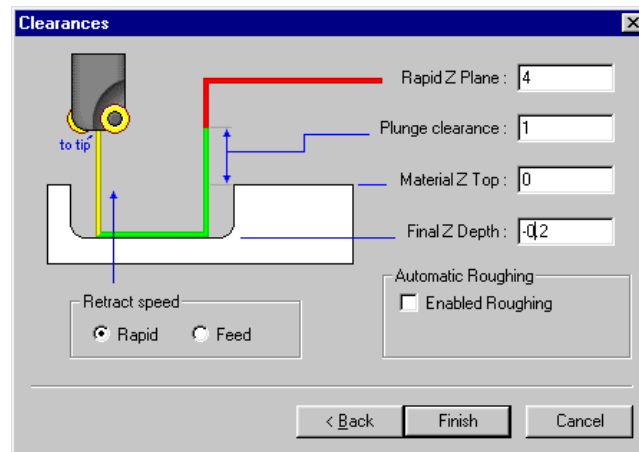
Enter the following details and click **next**.



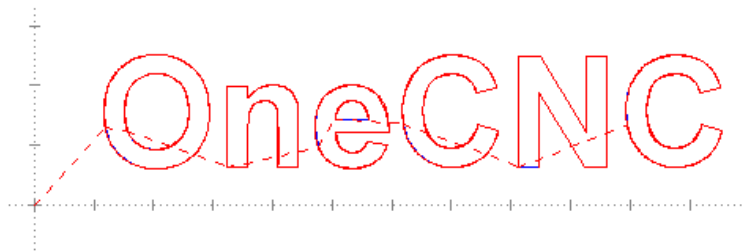
Enter a diameter of **20** and click **OK**.



Enter the following details and click **OK**.



The NC Code is then created and placed in the NC editor.



Step 5. Simulating the NC Code



Click the **simulate NC** icon on the **NC CAM** toolbar.

Enter the following details and click ok

The dialog box titled "NC Solid Wrap Verification" contains the following fields and options:

- Material Size:**
 - Start X: 11.6193
 - Width in X: 103.836
 - Radius of Stock: 10
- Temporary Tool Information:**
 - Ball Mill, Dia-1, Tip-0.5
 - Dia: []
 - Tip: []
- Extents:** Always
- Quality:** Turbo (Coarse) to Slow (Fine) slider, currently positioned towards Slow (Fine).
- Options:** Stop at rapid gouge!

Buttons: OK, Cancel

The toolpath is then modeled.

