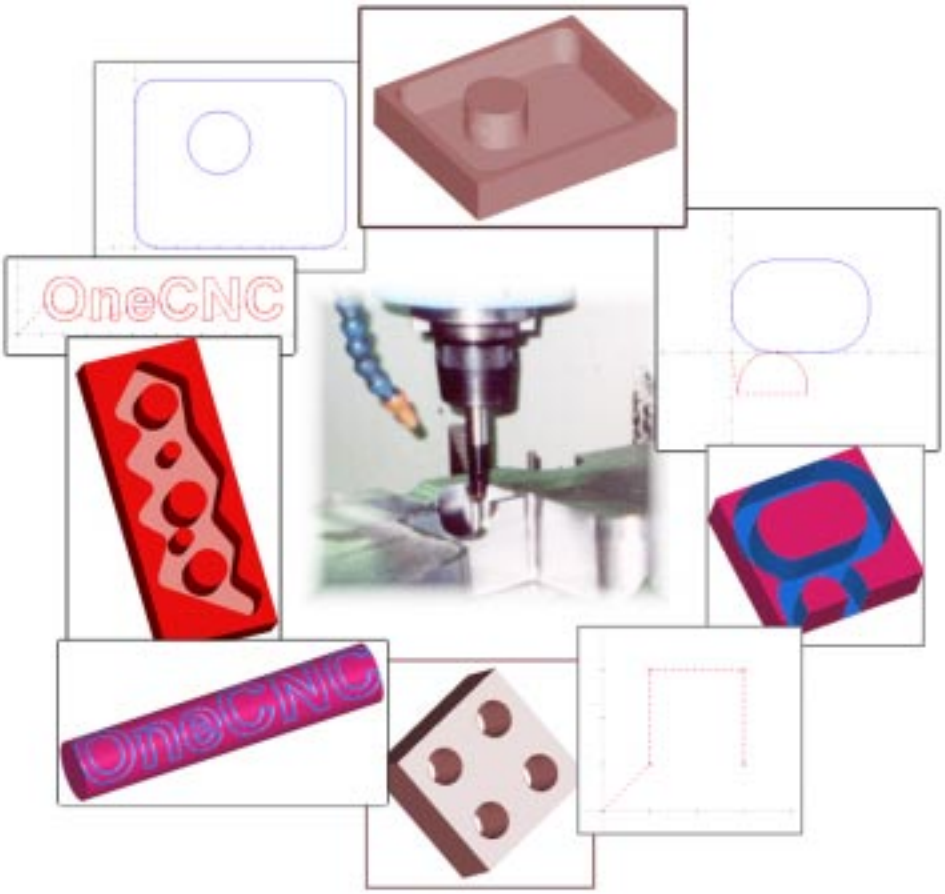
OneCNC

2000 Series

Manufacturing Software



Tutorials

For Mill Production

Contents

USA

OneCNC LLC

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

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

Australia

QARM Pty Ltd

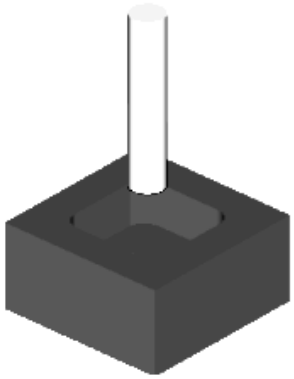
Phone: 617 32862527
Fax: 617 32864992

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

Version 1.02
(rev 28-6-01)

Mill Tutorial 1

Use 2 1/2 D Pocket Function



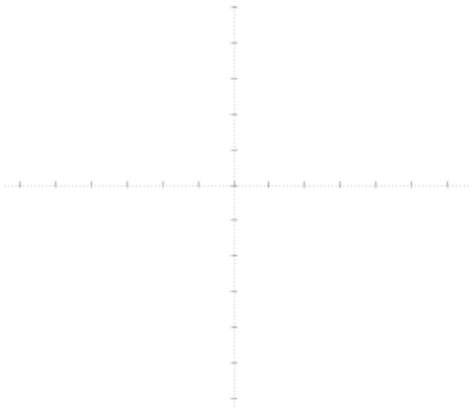
This is an example of machining a pocket in 2D with OneCNC.

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.

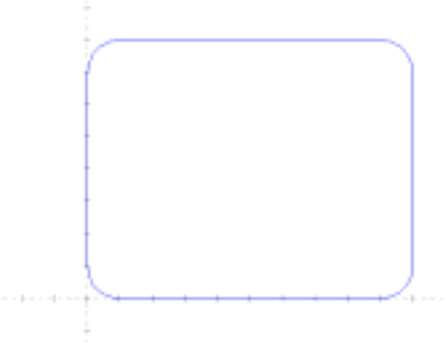
 Click the **coordinate icon** on the **position dialogue**.

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.

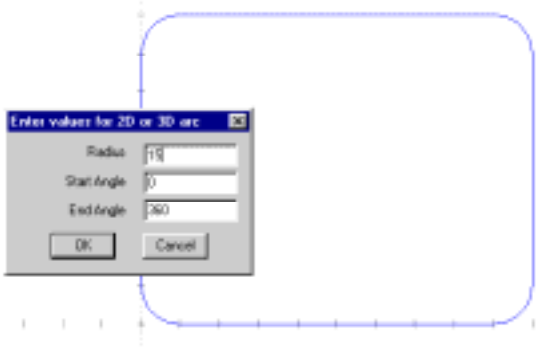


Click the **home icon** on the **menu toolbar**


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**.

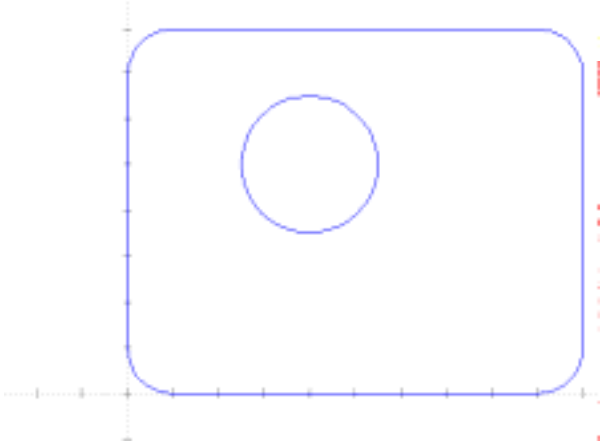



Click the **coordinate icon** on the **position dialogue**.

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

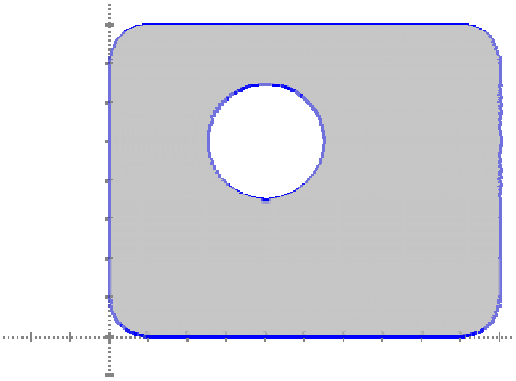
Then click **cancel** to terminate the circle function.

The drawing is now ready for pocketing.



Step 3. Pocketing the Part

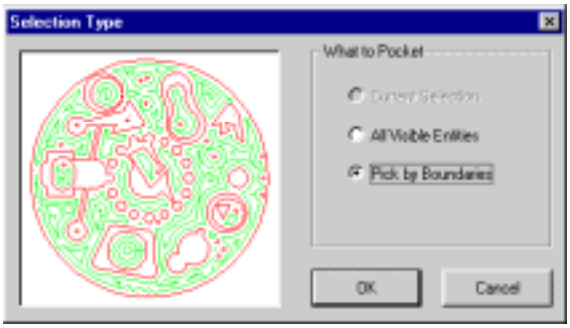
The greyed area below is the area that we want to pocket to a depth of -20 mm with 2 cuts of 10 mm.



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

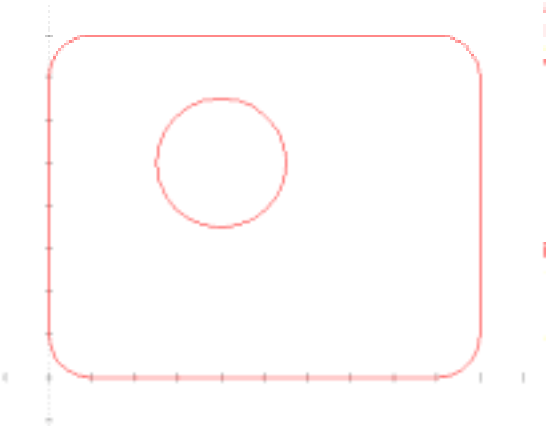
The following dialogue will appear.

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



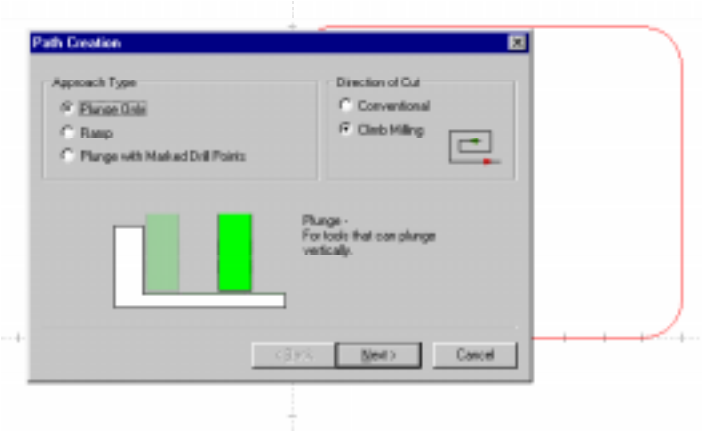
Then select the border loops (boundary) and they will turn red.

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

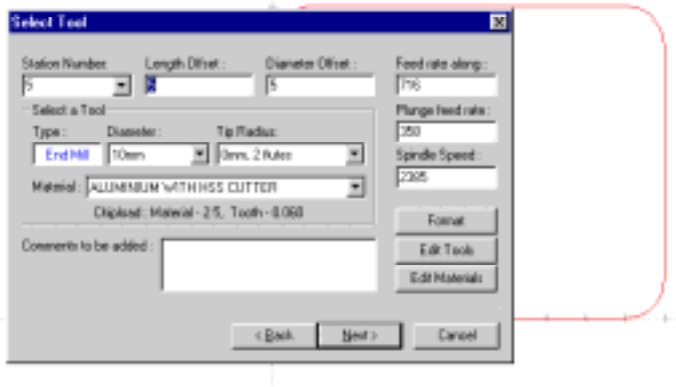


Select the **approach type** and the **direction of cut**.

Then click **next**.

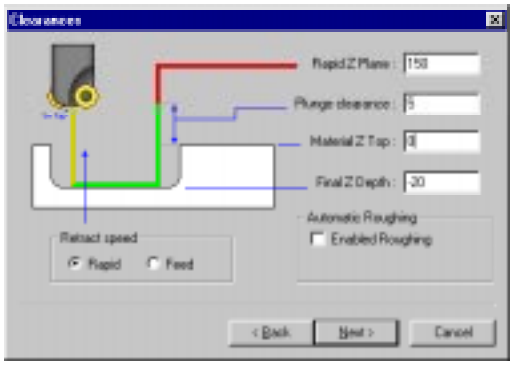


Select the **tool** and **material type** and then click **next**.

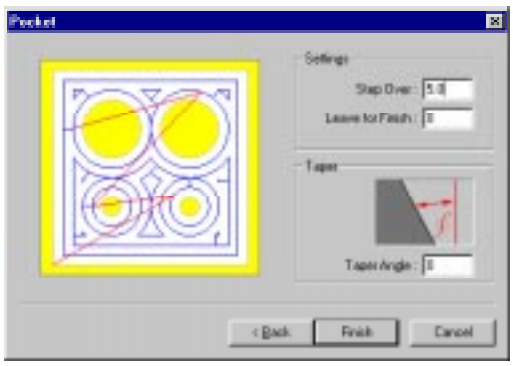


Set the **clearance plane**, **material top**, **final Z depth** and **enable automatic roughing**.

Then click **next.**



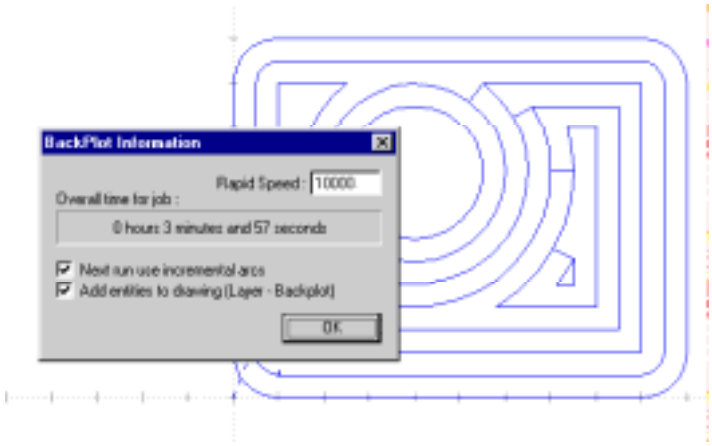
Enter the **step over value** and click **finish**.



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

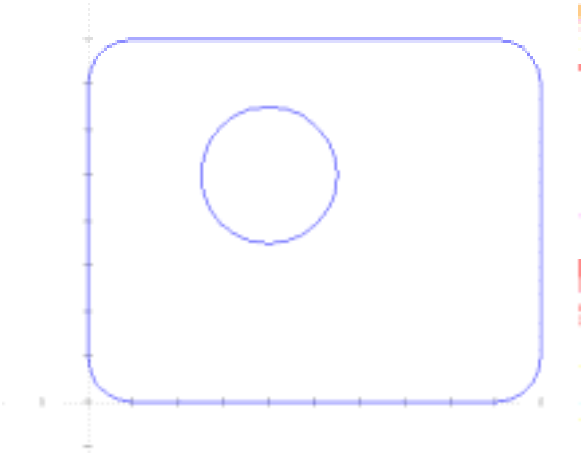
 Click the **backplot icon** on the **cam toolbar**



Then click **OK**

To remove the toolpath drawing.

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

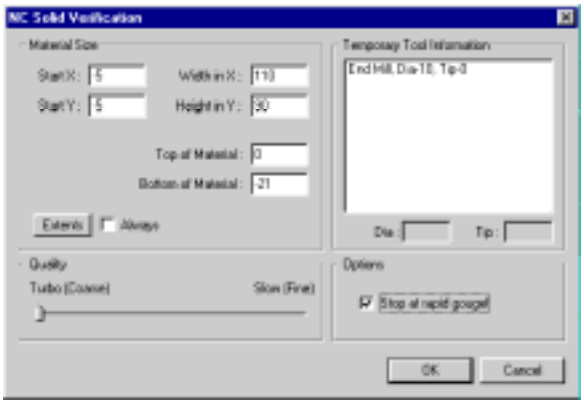


Step 5. Simulating the NC code

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

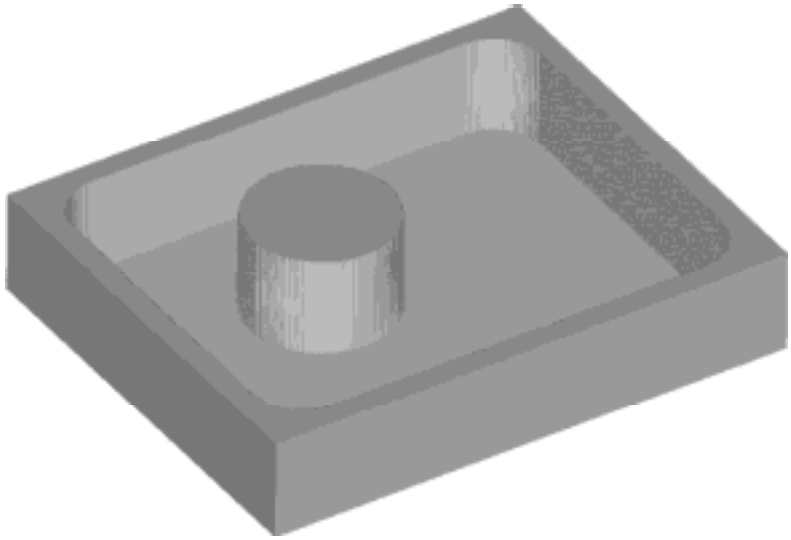
The following dialogue will appear.

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



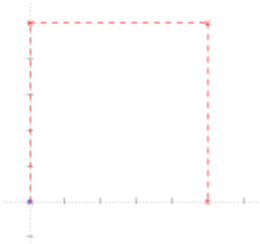
Note: the extents always tags is untagged to help demonstrate, extents always is best left on.

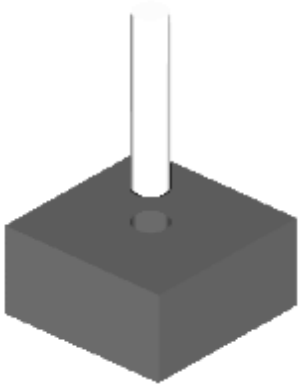
The toolpath is then modeled.



Mill Tutorial 2

Use Drill Function





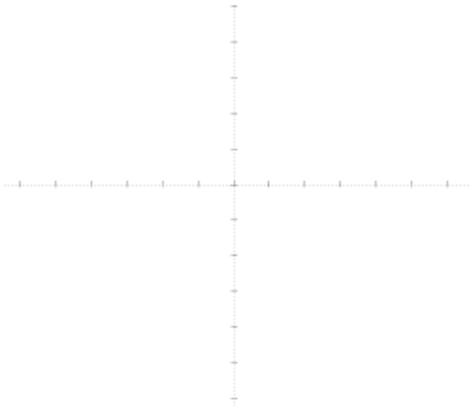
Note: This help uses mm values.

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**.

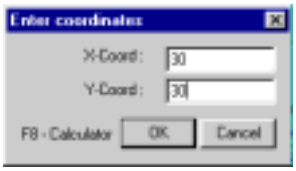
 Click the **point by coordinate icon** on the **point menu toolbar**.

Enter **X: 10 Y: 10** and click **OK**.

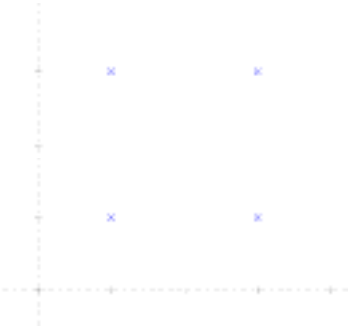
Then enter **X: 10 Y: 30** and click **OK**.

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

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



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

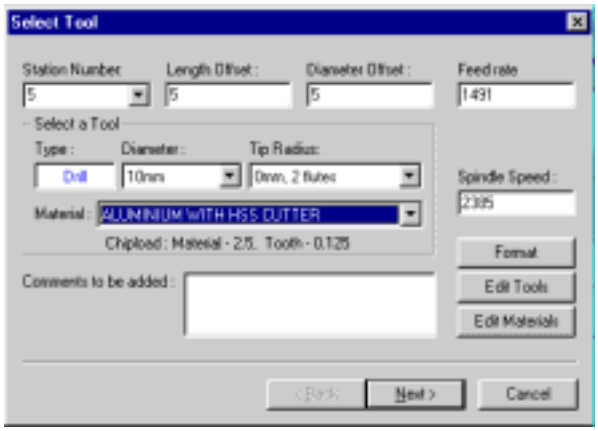


Step 3. Drilling the Points

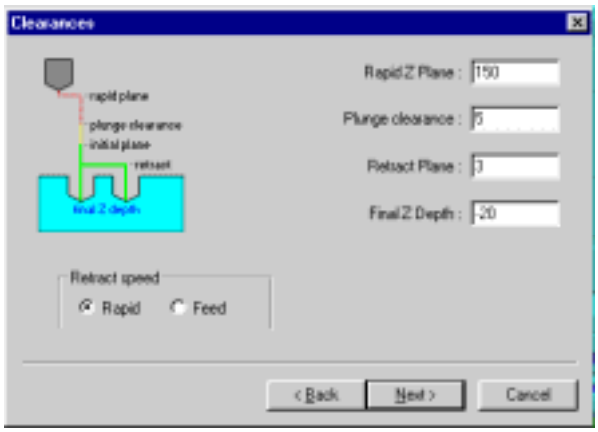
We will now use these points to drill at Z -20.0 at these points.


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

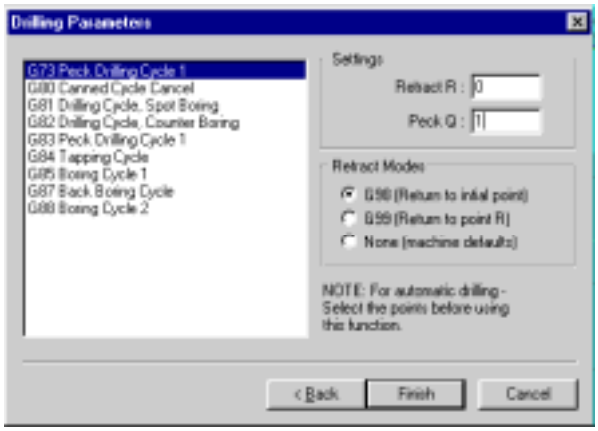
Select your **drill** and **material** then click **next**.



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

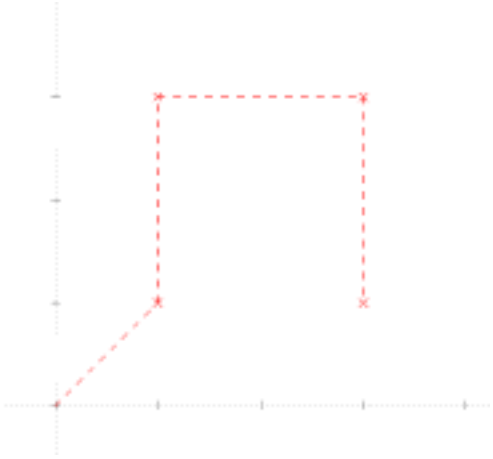


Select the **drilling cycle** and click **finish**.



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.



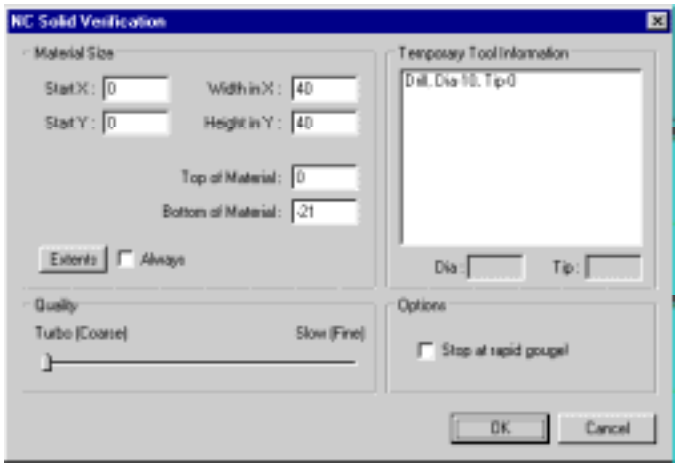
The NC code is now in the editor completed.

Step 4. Simulating the NC code

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

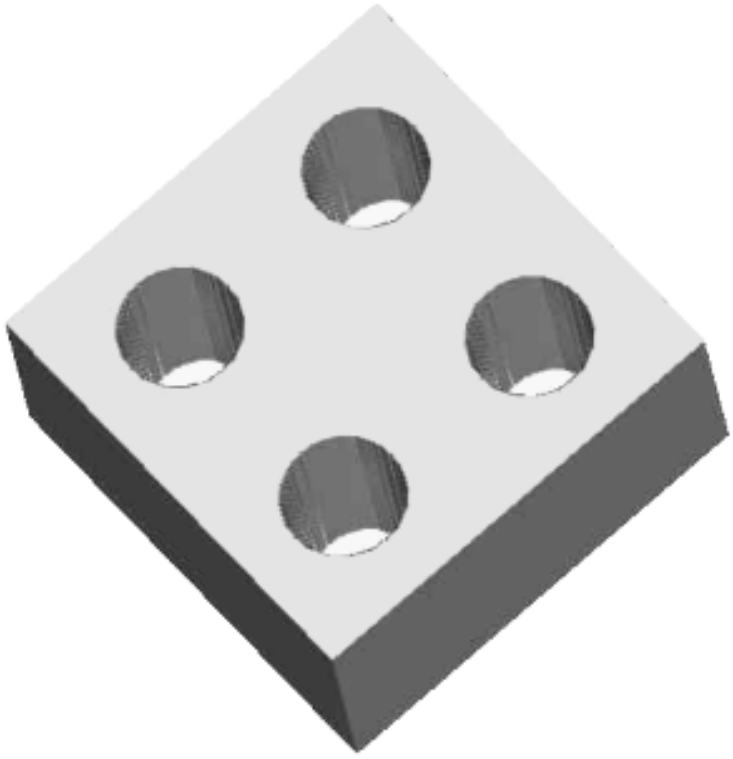
The following dialogue will appear.

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



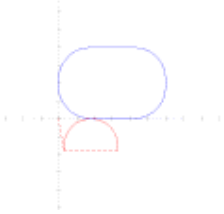
Note: the extents always tags is untagged to help demonstrate, extents always is best left on.

Your toolpath is then modeled.

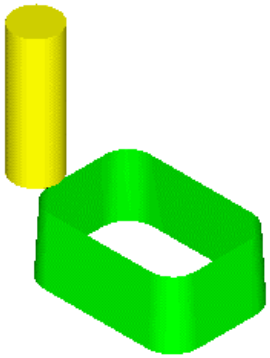


Mill Tutorial 3

Use Profile Function



Note: This help uses mm values.

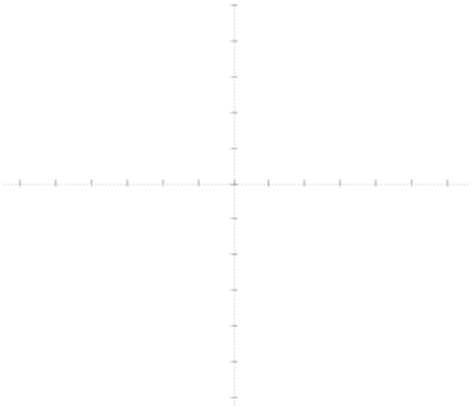


In this tutorial we use the mill profile function.

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.

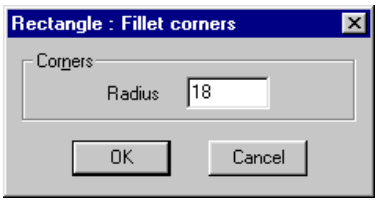
Step 2. Create a Shape to be Profiled


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


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

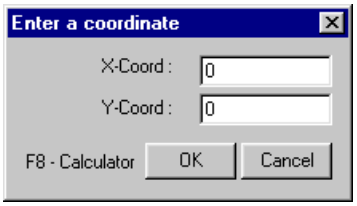
The following dialogue will appear.

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



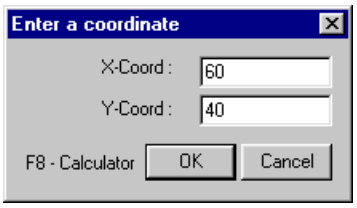

Click the **position by coordinate icon** on the **position dialogue**

Enter the following details into the coordinate dialogue.

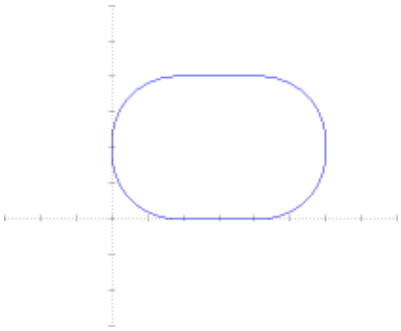



Click the **position by coordinate icon** on the **position dialogue**

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



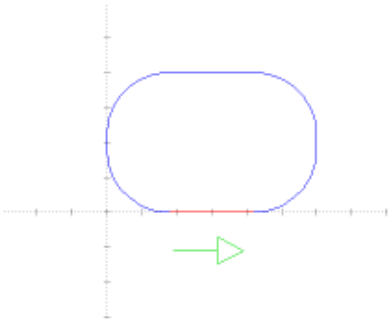
Click **Cancel** to terminate the rectangle function.



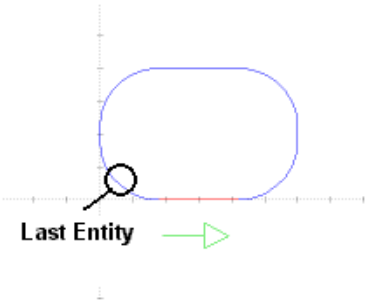
Step 3. Create NC Code


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

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

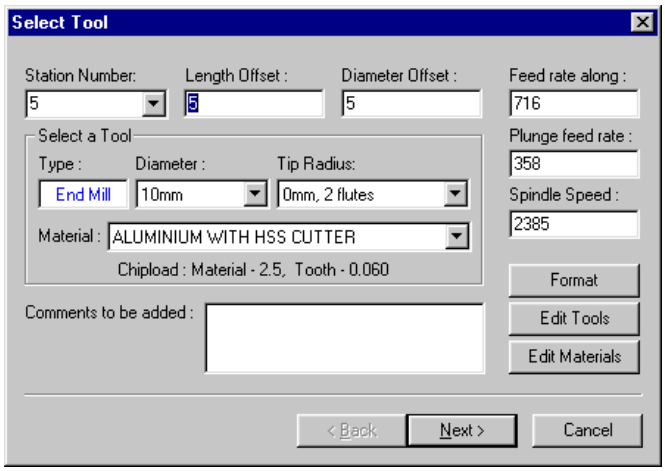


Then select the last entity

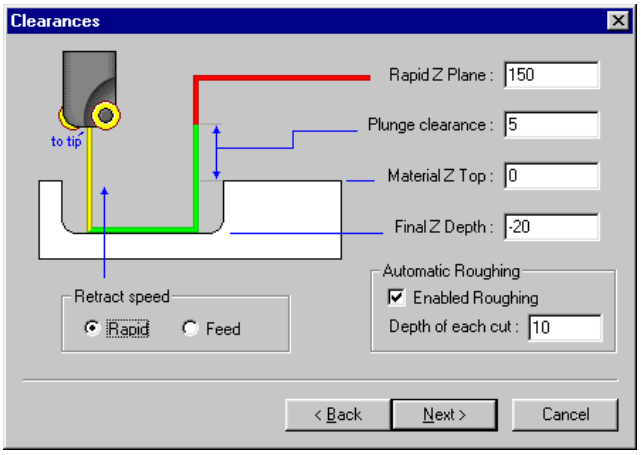


The following dialogue will appear.

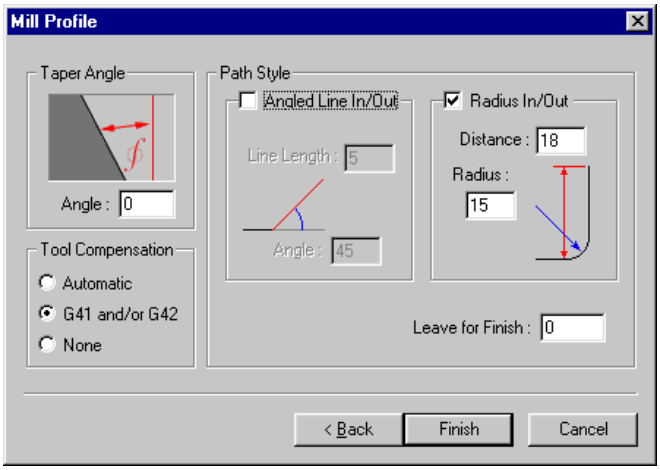
Set your **tool type**, and **material** and click **next**.



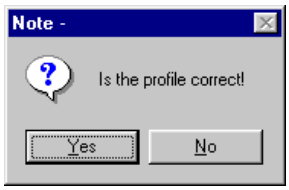
Set your **clearances** and click **next**.



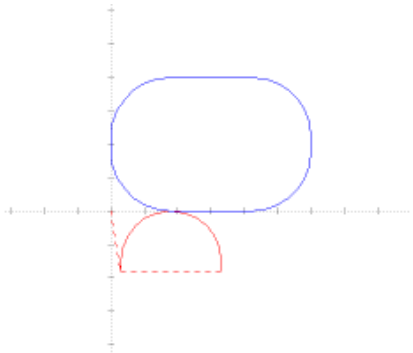
Enter the following details and click **finish.**



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



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

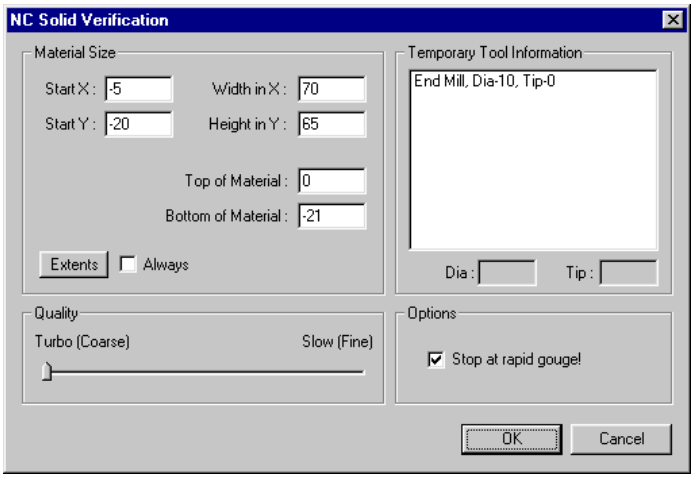


Step 4. Simulating the NC code


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

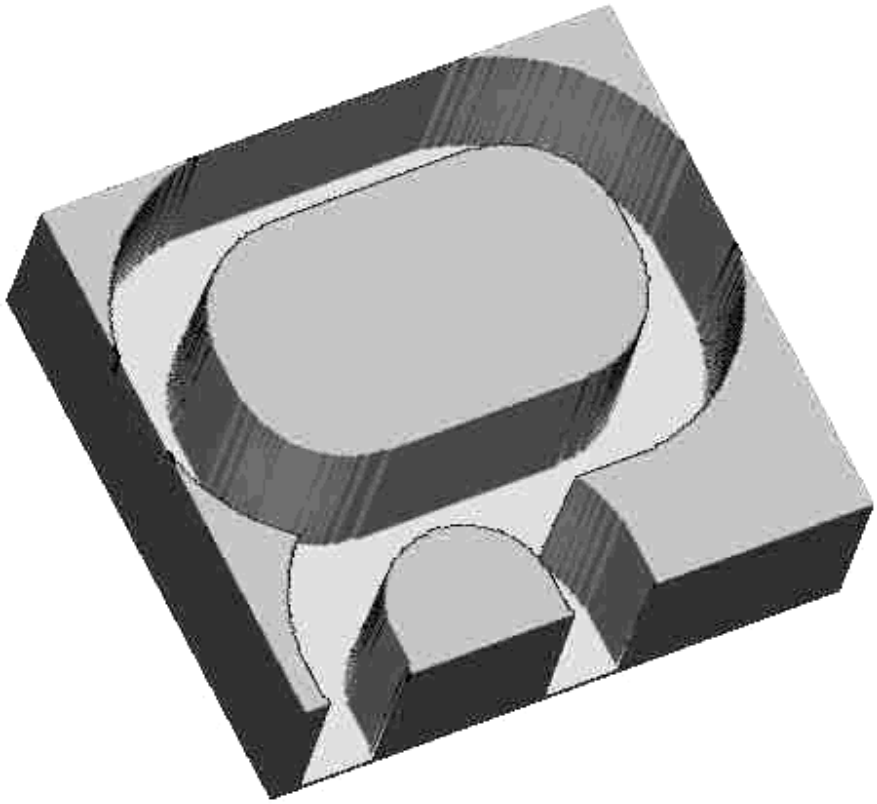
The following dialogue will appear.

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



Note: the extents always tags is untagged to help demonstrate, extents always is best left on.

The toolpath is then modeled.



Mill Tutorial 4

Draw 2D Text and Use 4-Axis Wrap

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

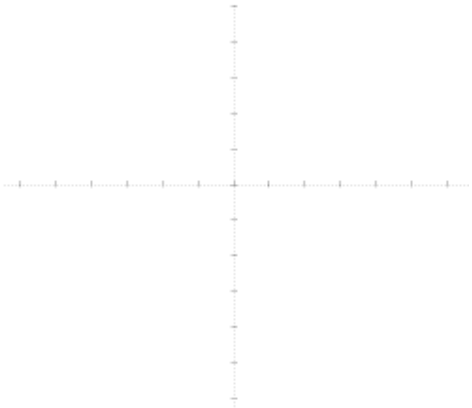
This tutorial require the 4-axis module to operate.

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**

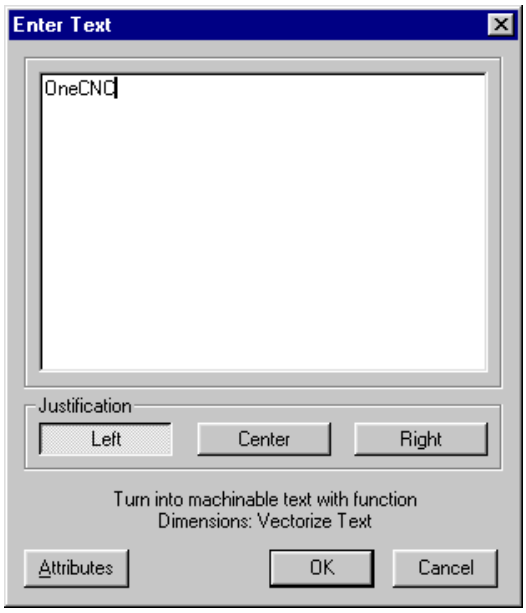
Step 3. Creating Text


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


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

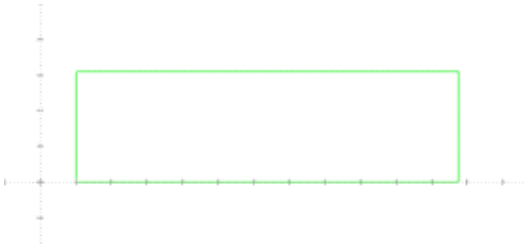
The following dialogue will appear.

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



Enter the following and click **OK**

Click OK again and the text as shown below.

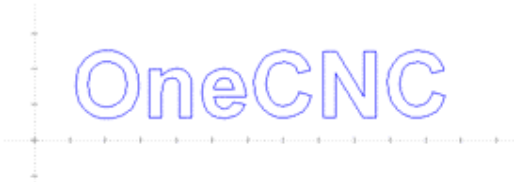


Your text is then created.

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

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

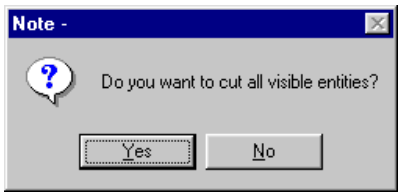
Select the text and it will become vectorized.



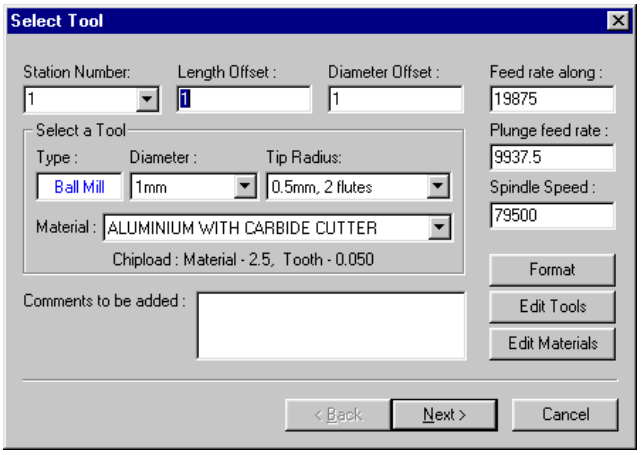
Step 4. Cutting the Toolpath


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

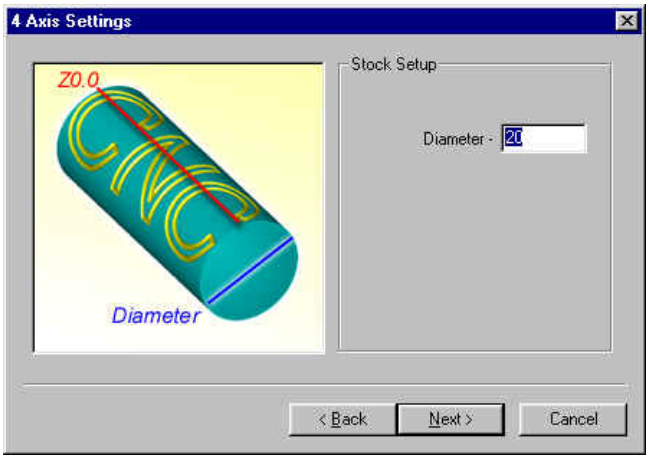
The following will appear select **Yes**.



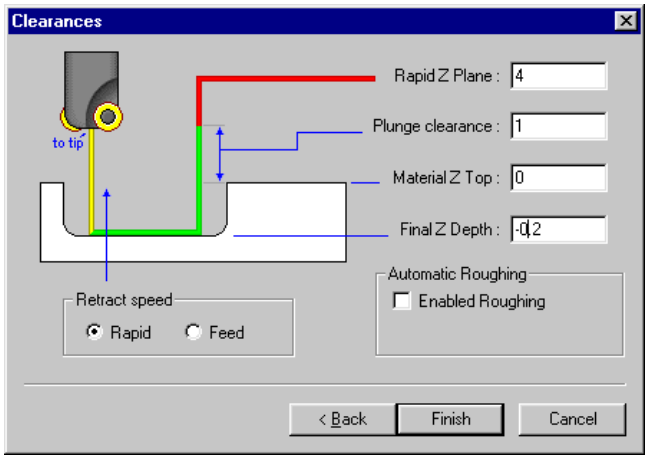
Enter the following details and click **next.**



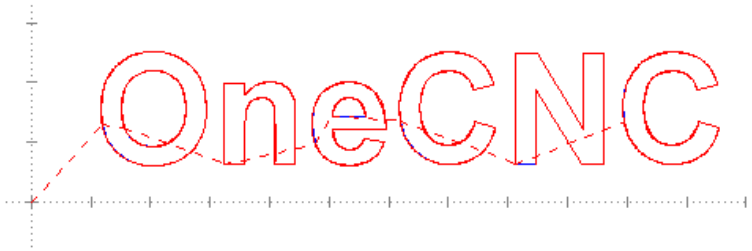
Enter a diameter of **20** and click **OK.**



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



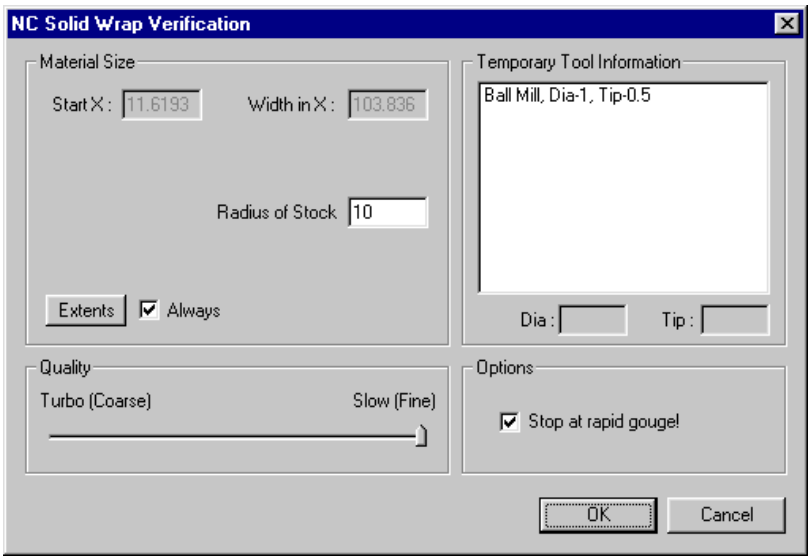
The NC Code is then created and placed in the NC editor.



Step 5. Simulating the NC Code


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

Enter the following details and click ok



The toolpath is then modeled.