

OneCNC

2000 Series

Manufacturing Software



Tutorials

For Mill Production

Contents

<i>Tutorial</i>	<i>Page</i>
Use 2 ½ D Pocket Function	3
Draw 2D and Drill with Mill CAM	12
Draw 2D and Profile with Mill CAM	17
Draw text and use 4-Axis Wrap	22

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

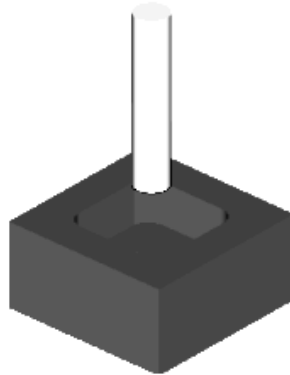
QARM Pty Ltd

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.

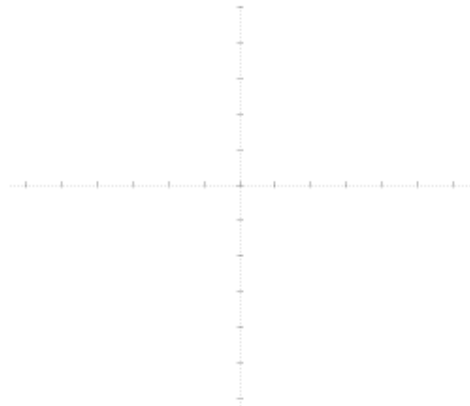
This is a tutorial to draw a 2D part using Mill then pocketing out the part using the mill pocket function.

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**.

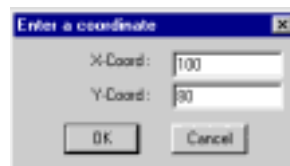
Enter **X: 0, Y: 0** in the coordinate dialogue.



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**.

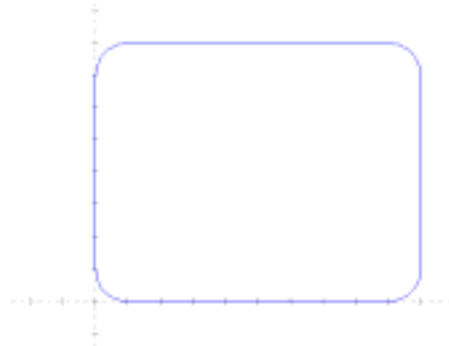
Enter **X: 100** and **Y: 80** in the dialogue and click **OK**.



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**

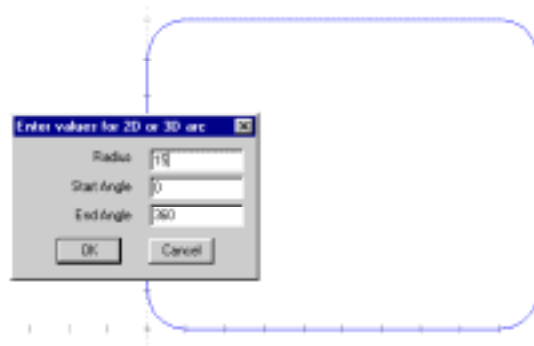


Click the **arc and spline icon** on the **menu toolbar**



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

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



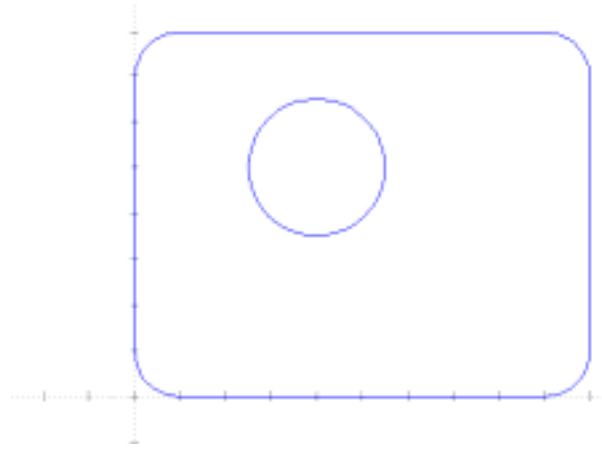
Click the **coordinate icon** on the **position dialogue**.

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



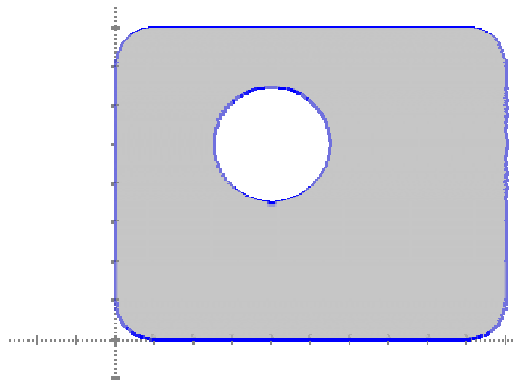
Then click **cancel** to terminate the circle function.

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**

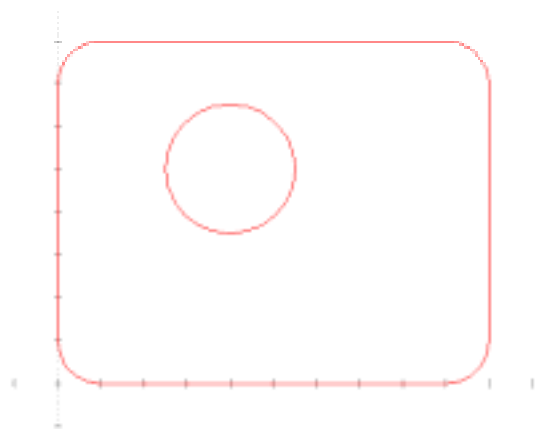
The following dialogue will appear.

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



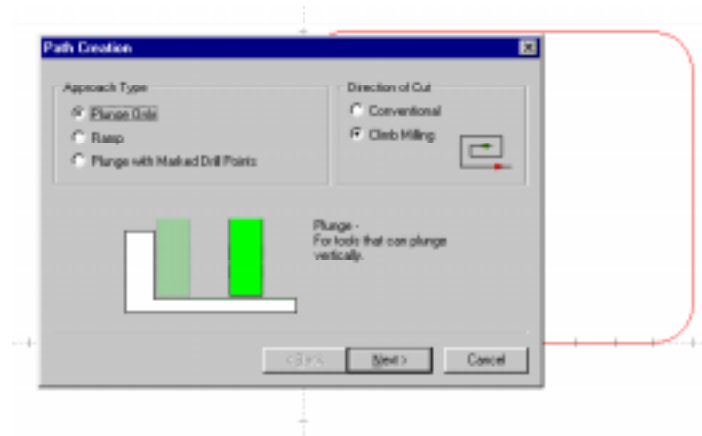
Then select the border loops (boundary) and they will turn red.

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

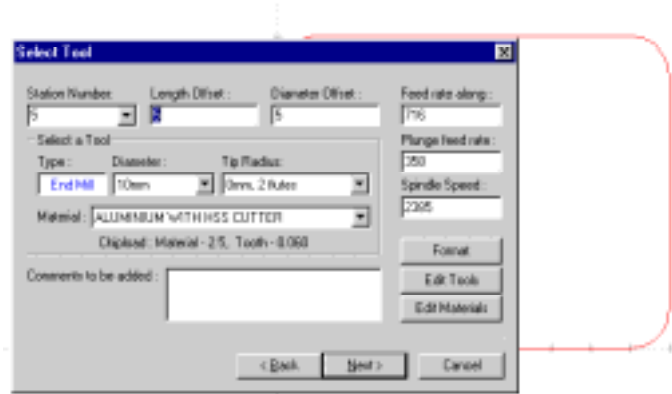


Select the **approach type** and the **direction of cut**.

Then click **next**.

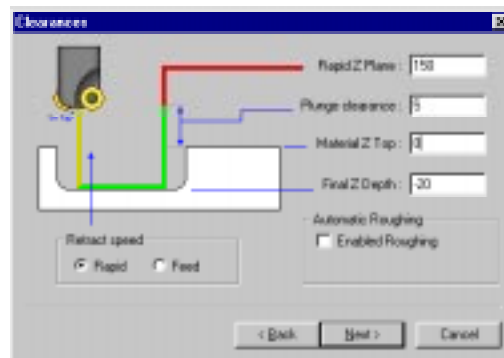


Select the **tool** and **material type** and then click **next**.

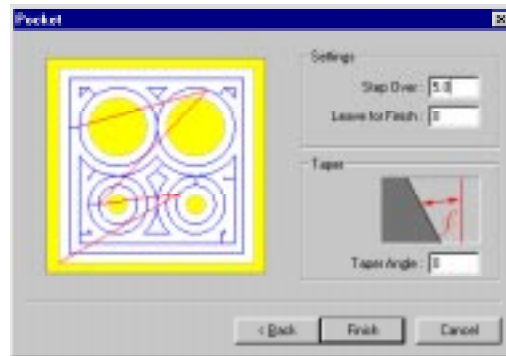


Set the **clearance plane**, **material top**, **final Z depth** and **enable automatic roughing**.

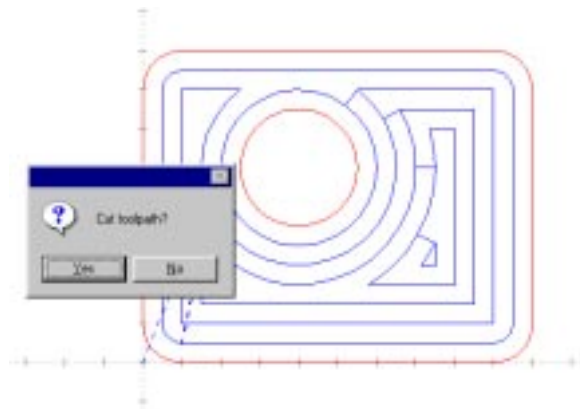
Then click **next**.



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

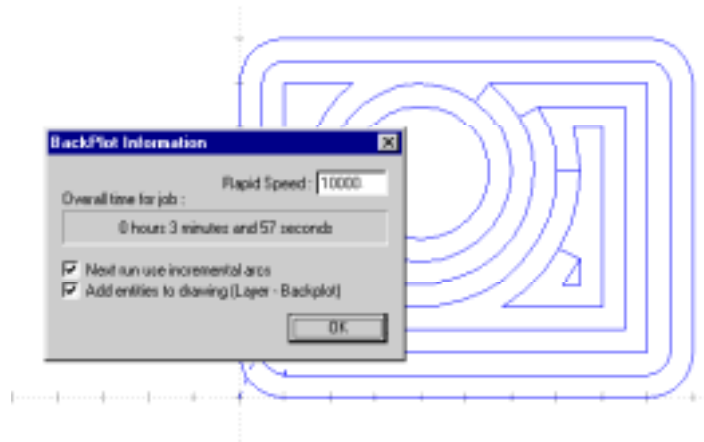


The toolpath is calculated and if it is to your satisfaction click **yes** when your asked to cut the tool path and the code will be created and placed in the NC editor.




Step 4. Obtaining a Machine Time

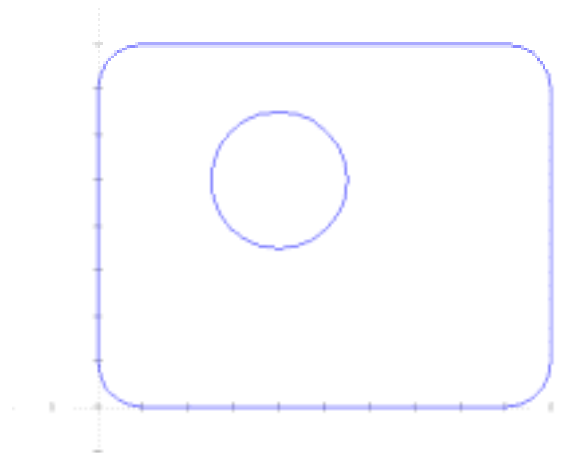
 Click the **backplot icon** on the **cam toolbar**



Then click **OK**

To remove the toolpath drawing.

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



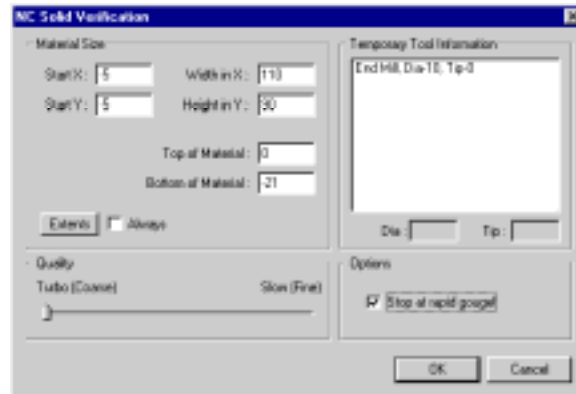
Step 5. Simulating the NC code



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

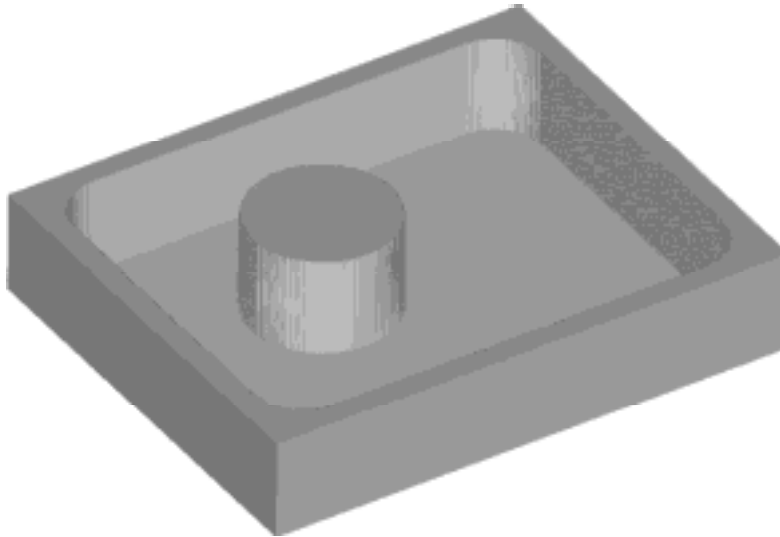
The following dialogue will appear.

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



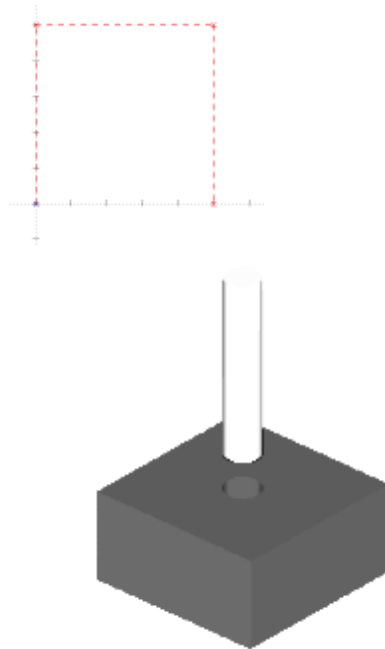
Note: the extents always tags is untagged to help demonstrate, extents always is best left on.

The toolpath is then modeled.



Mill Tutorial 2

Use Drill Function



Note: This help uses mm values.

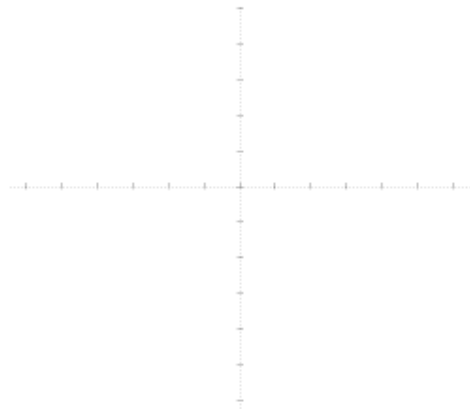
This is a tutorial to draw points using Mill then to drill those points.

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**.




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: ALUMINIUM WITH HSS CUTTER
- Chipload: Material - 2.0, Tooth - 0.125
- Spindle Speed: 2305
- Buttons: Formal, Edit Tools, Edit Materials
- Comments to be added: (empty text box)
- Navigation: < Back, Next >, Cancel

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

The 'Clearances' dialog box includes a diagram and the following settings:

- Diagram labels: rapid plane, plunge clearance, initial plane, retract, final Z depth.
- Rapid Z Plane: 150
- Plunge clearance: 5
- Retract Plane: 3
- Final Z Depth: -20
- Retract speed: Rapid, Feed
- Navigation: < Back, Next >, Cancel

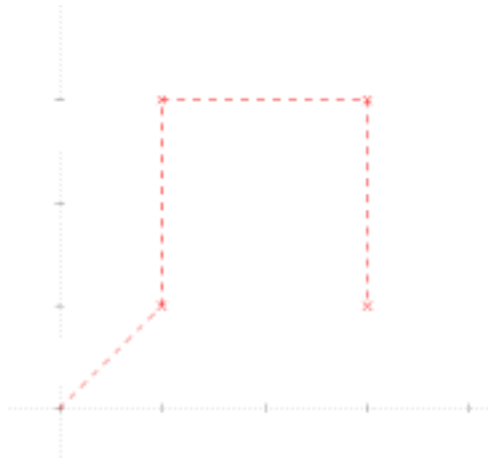
Select the **drilling cycle** and click **finish**.

The 'Drilling Parameters' dialog box features a list of cycles and the following settings:

- Settings: Retract R: 0, Peck Q: 1
- 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.
- Navigation: < 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.



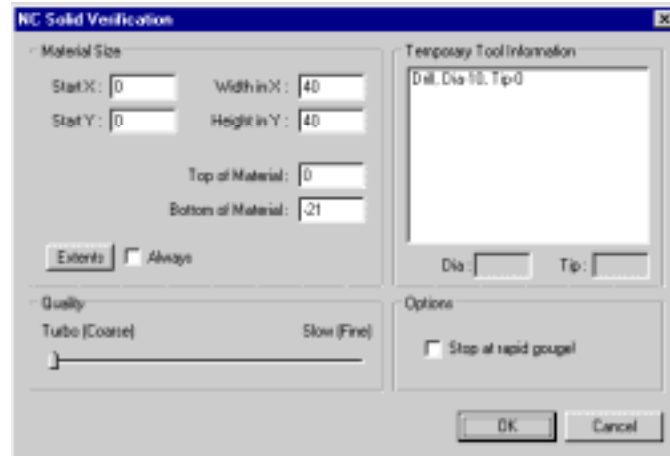
The NC code is now in the editor completed.

Step 4. Simulating the NC code

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

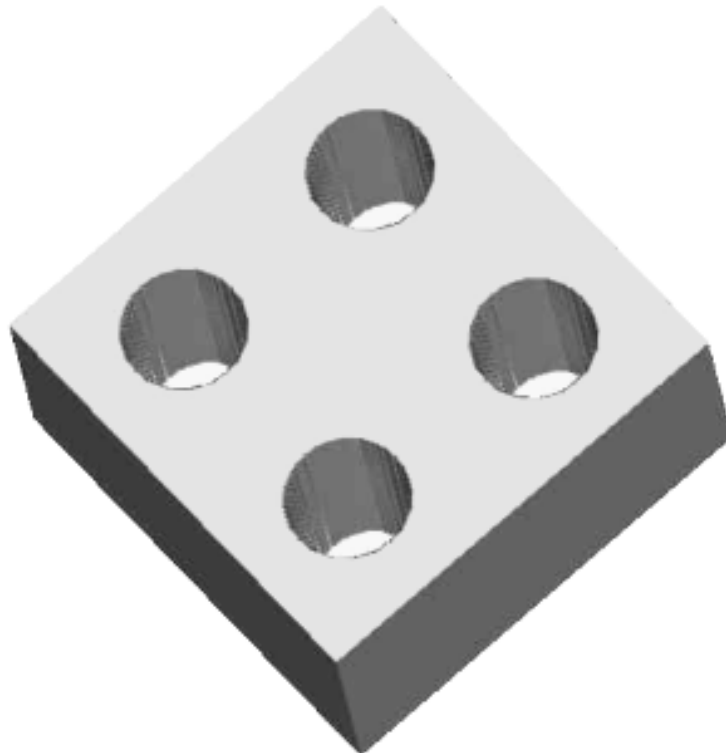
The following dialogue will appear.

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



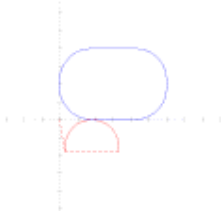
Note: the extents always tags is untagged to help demonstrate, extents always is best left on.

Your toolpath is then modeled.

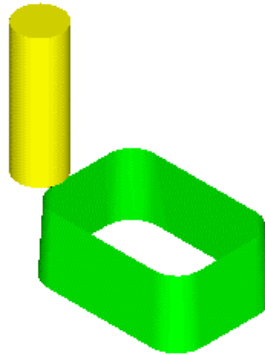


Mill Tutorial 3

Use Profile Function



Note: This help uses mm values.



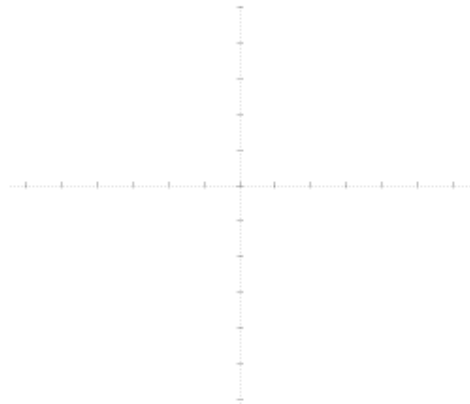
In this tutorial we use the mill profile function.

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.

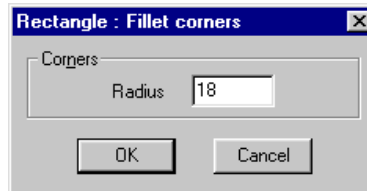
Step 2. Create a Shape to be Profiled

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

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

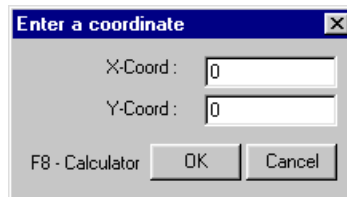
The following dialogue will appear.

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



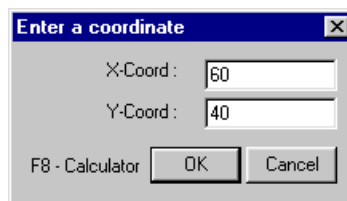
 Click the **position by coordinate** icon on the **position dialogue**

Enter the following details into the coordinate dialogue.

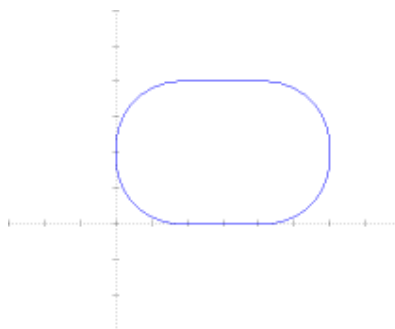


 Click the **position by coordinate** icon on the **position dialogue**

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



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

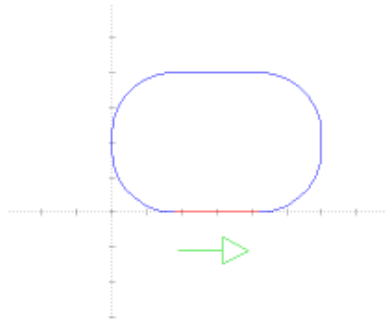


Step 3. Create NC Code

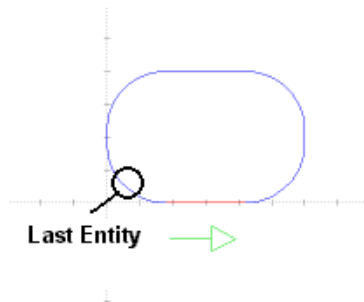


Click the **mill profile icon** on the **NC CAM dialogue**

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**.

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

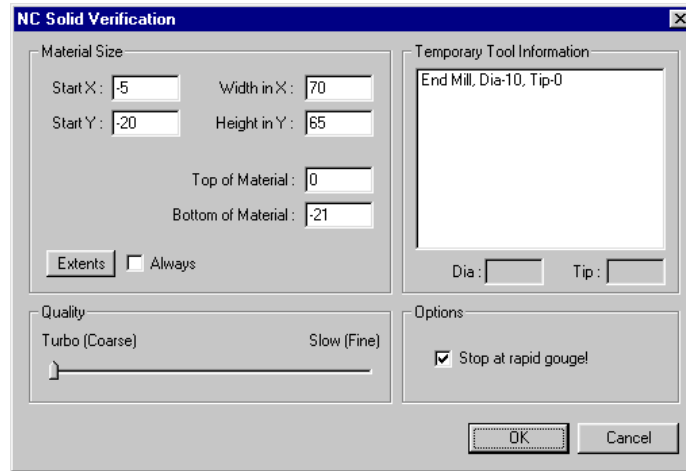
Step 4. Simulating the NC code



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

The following dialogue will appear.

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

A screenshot of the 'NC Solid Verification' dialog box. The dialog is titled 'NC Solid Verification' and has a close button (X) in the top right corner. It is divided into several sections: 'Material Size' with input fields for Start X: -5, Width in X: 70, Start Y: -20, Height in Y: 65, Top of Material: 0, and Bottom of Material: -21; 'Extents' with an unchecked checkbox for 'Always'; 'Quality' with a slider between 'Turbo (Coarse)' and 'Slow (Fine)'; 'Temporary Tool Information' with a text area containing 'End Mill, Dia-10, Tip-0' and input fields for 'Dia:' and 'Tip:'. The 'Options' section has a checked checkbox for 'Stop at rapid gouge!'. At the bottom are 'OK' and 'Cancel' buttons.

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)

Temporary Tool Information: End Mill, Dia-10, Tip-0

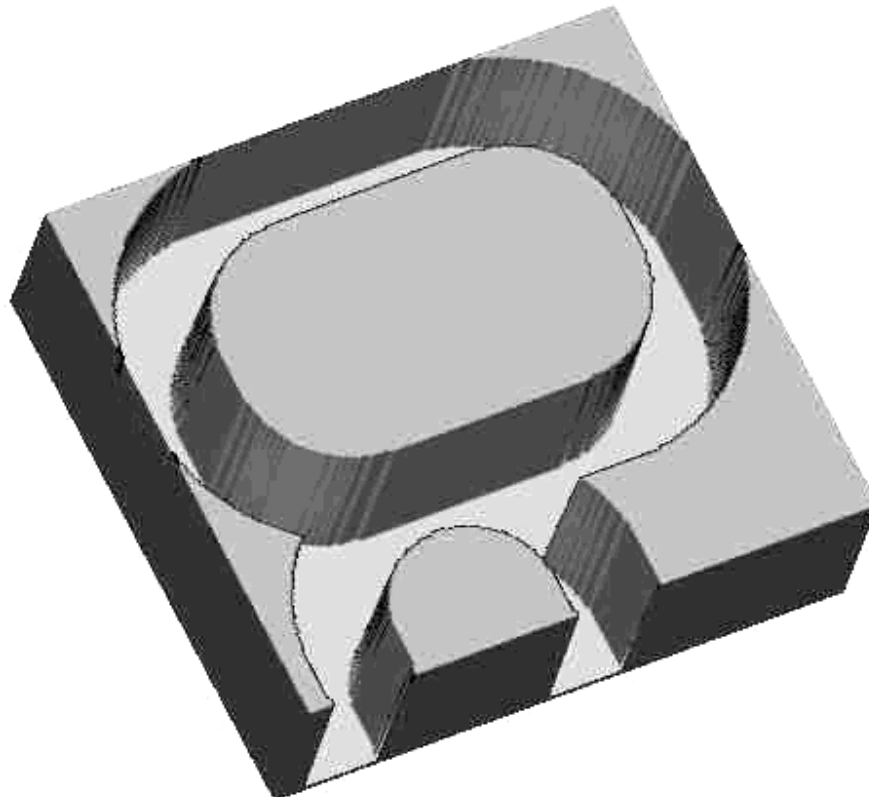
Dia: Tip:

Options: Stop at rapid gouge!

OK Cancel

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.

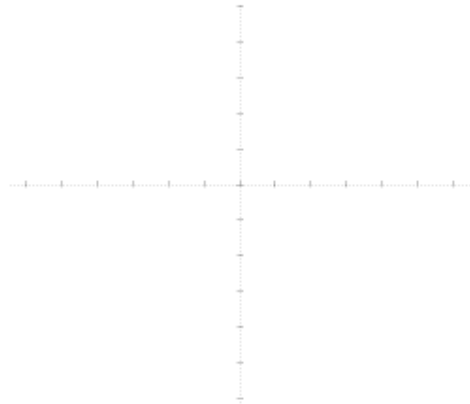
NOTE: this tutorial uses mm values.

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.

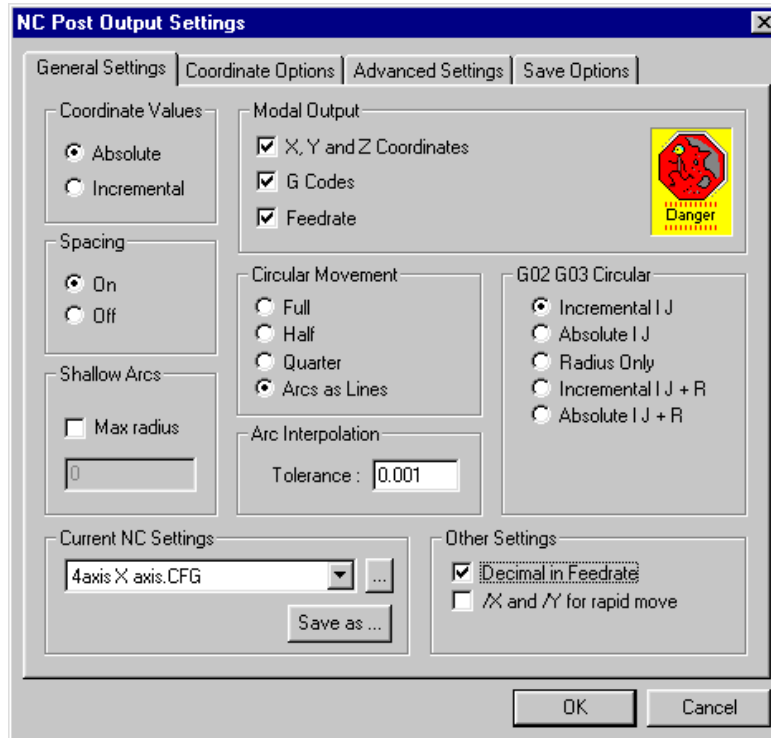
Step 2. Setting Up 4-Axis

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

NC Setup

The following dialogue will appear

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



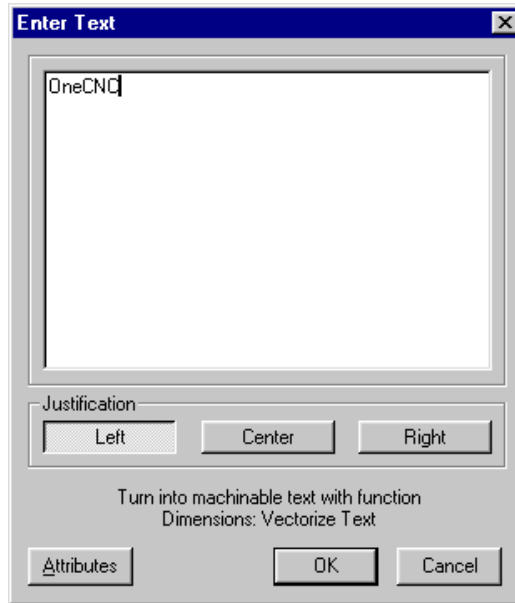
Step 3. Creating Text

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

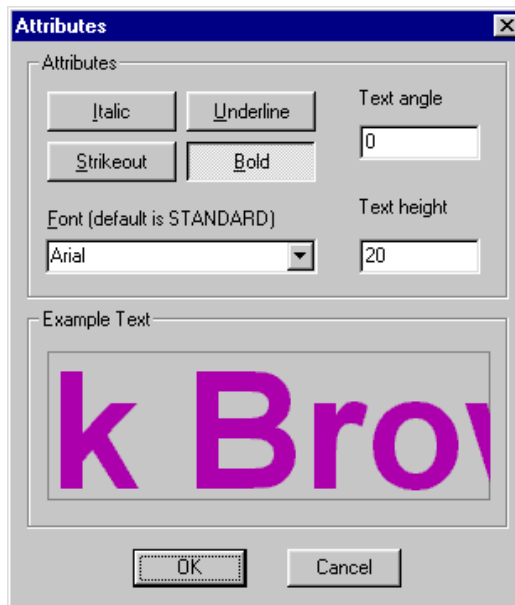
A Click the **create text icon** on the **text menu toolbar**.

The following dialogue will appear.

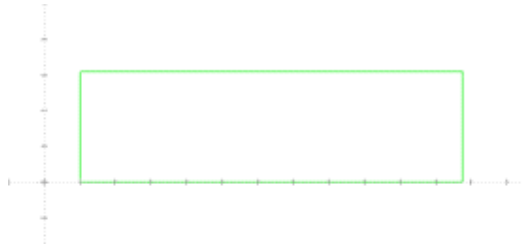
Enter the the following **details** and click **attributes**.



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.

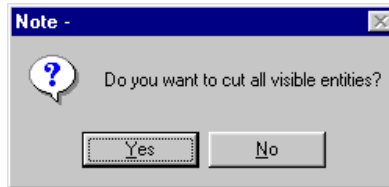


Step 4. Cutting the Toolpath

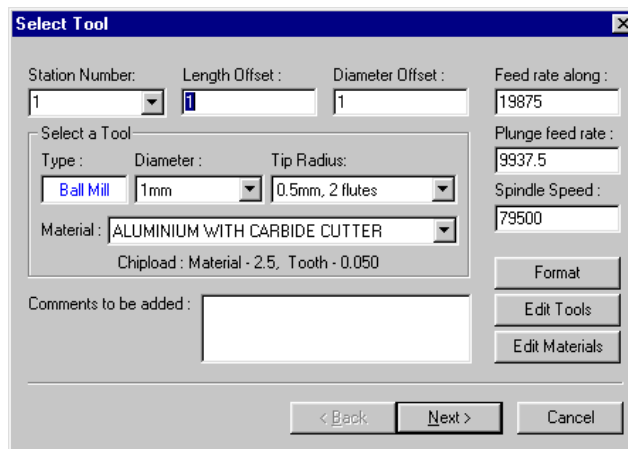


Click the **cut a visible entities in 2D** icon on the **NC CAM** toolbar.

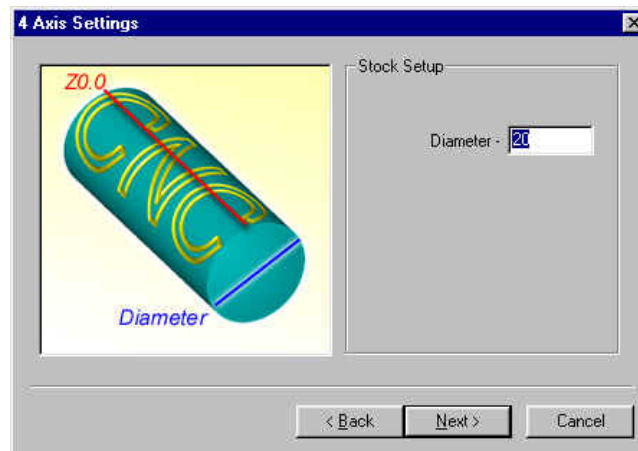
The following will appear select **Yes**.



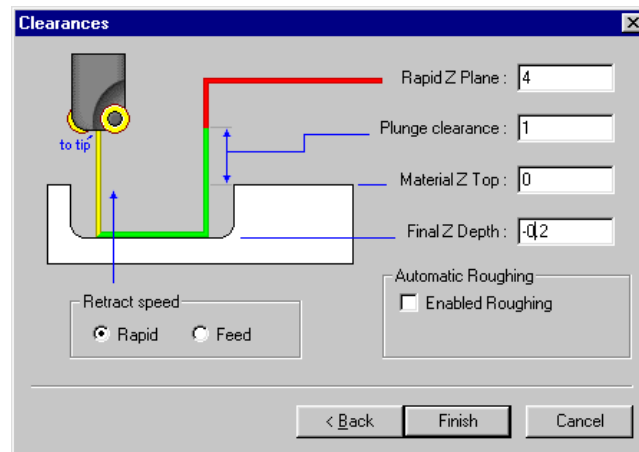
Enter the following details and click **next**.



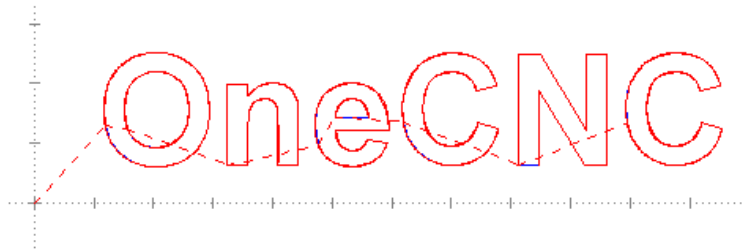
Enter a diameter of **20** and click **OK**.



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



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

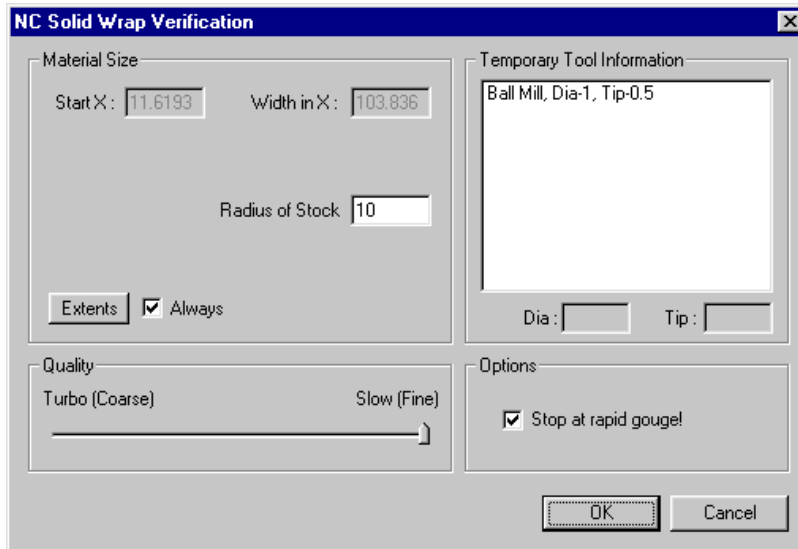


Step 5. Simulating the NC Code



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

Enter the following details and click ok



The dialog box is titled "NC Solid Wrap Verification" and 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: []
- 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.

