

BobCAD-CAM v24 Training

Using The Profile Wizard

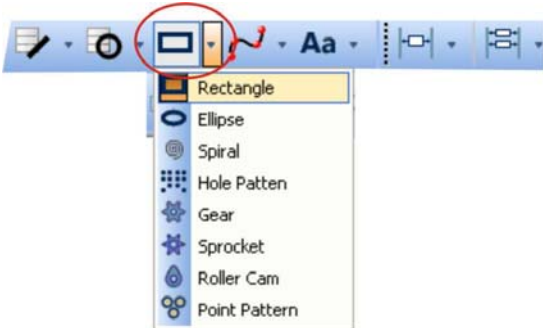
BobCAD-CAM Version 24 Training Lesson

The Version 24 offers a 2 Axis Machining Wizard to step you through the process of creating Machine programs for any of the 2 Axis Operations. In this lesson we will be creating a program using the profile feature.

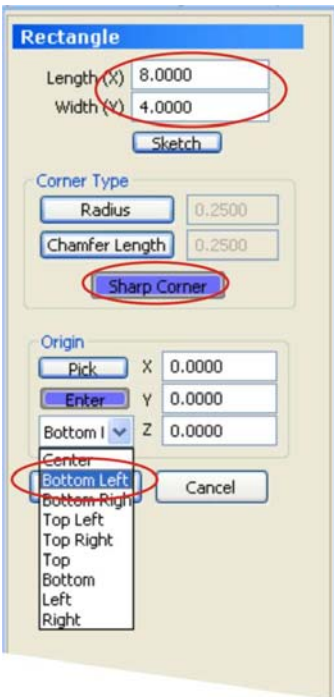
Start off with a new drawing screen and follow along doing each step.

Step 1

Select the Rectangle drawing feature. You can do this by going to the **Other** Main Menu and selecting Rectangle or by clicking on the Rectangle function icon from the main toolbars.



This will load the Rectangle feature into the Data Manager.



Enter a Length of 8 and a Width of 4.
Choose Sharp Corner as the corner option.
Choose Bottom Left as the origin.
Click OK. This draws the shape in the workspace.

Step 2

Now create a second rectangle using the following values:

Enter 4 for Length X

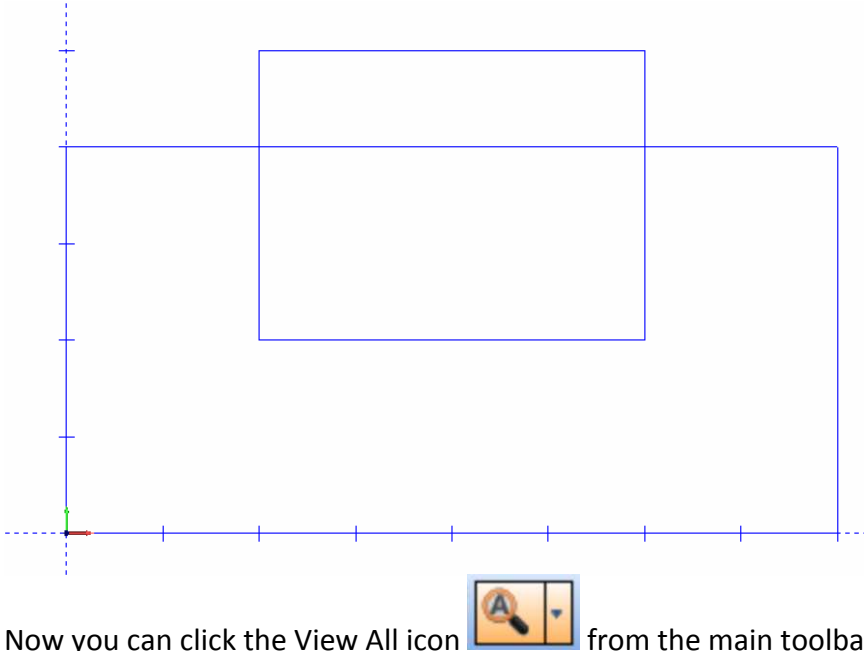
Enter 3 for Width Y


Under Origin click the Enter button.

Enter 2 for X

Enter 2 for Y

Select Bottom Left from the pull down menu and then click OK to draw the second shape. Then click the Cancel button to exit the Rectangle feature.



Now you can click the View All icon  from the main toolbar to obtain a clear view of the entire shape.

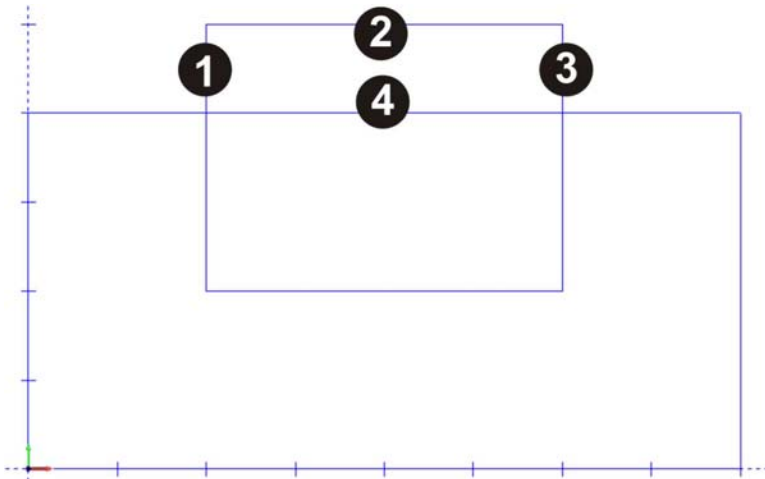
Step 3

Now use the Quick Trim feature to edit the drawing.

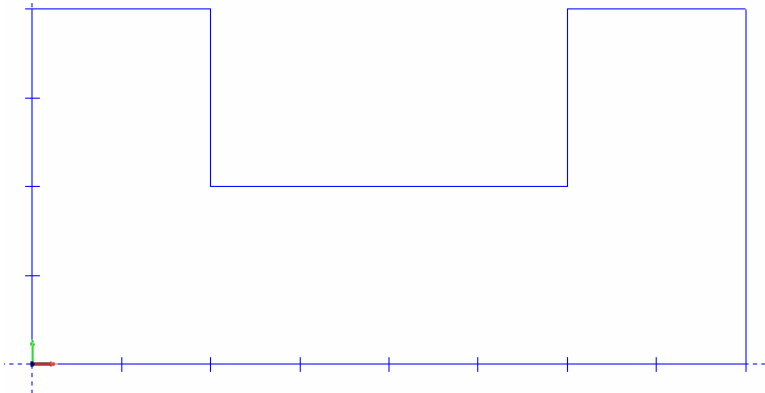


You can also go to the Main **Utilities** Menu, Trim Extend and then select Quick Trim at the bottom. We want to click on the 4 entities that are labeled in the next image.

Simply place your cursor on the geometry (numbered lines) and click your mouse to select them. As they are selected they will turn red. If you make a mistake and click on an incorrect line just click on it again to de-select it.

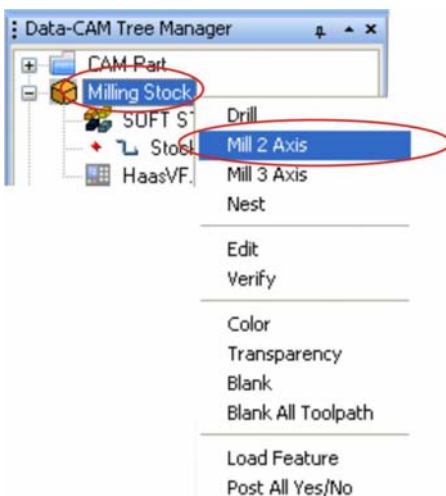


Click the Cancel button in the Data Manager when finished. You should now have the following result:

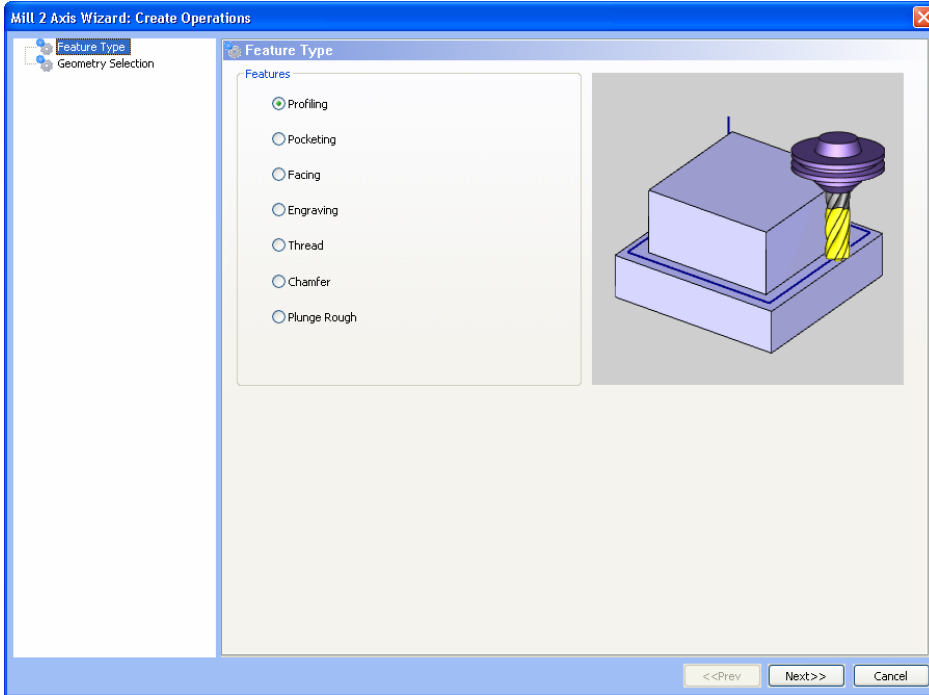


Step 4

Now we can use the 2 Axis Wizard to create the toolpath. Right-click your mouse on Milling Stock in the Data CAM Tree Manager and select Mill 2 Axis.

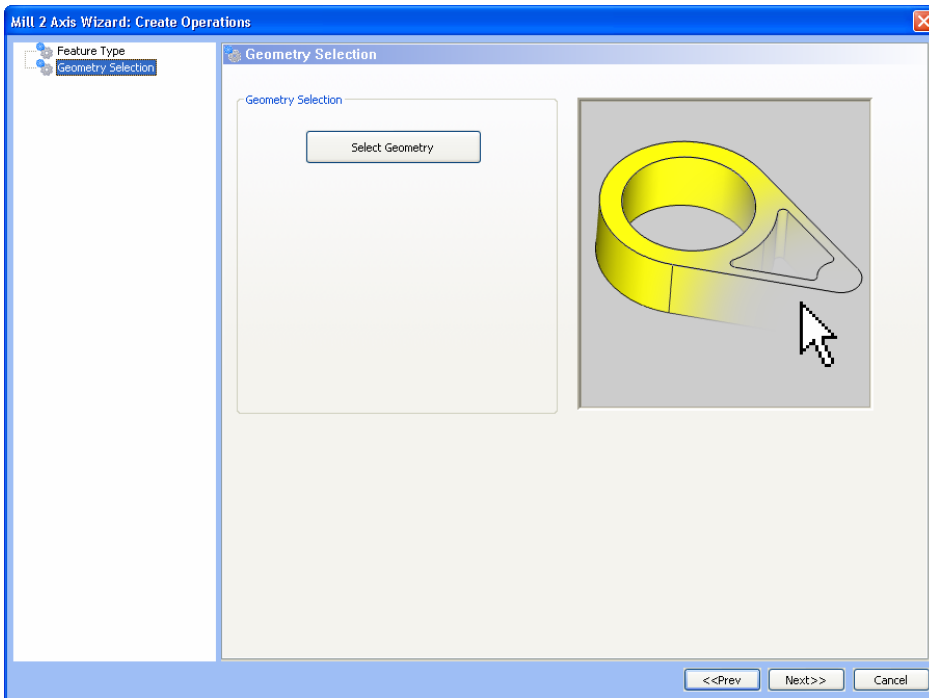


This will launch the 2 Axis Wizard and allow you to choose which 2 Axis machining operation you would like to use.



Click the Profile feature and then click the **Next** button.

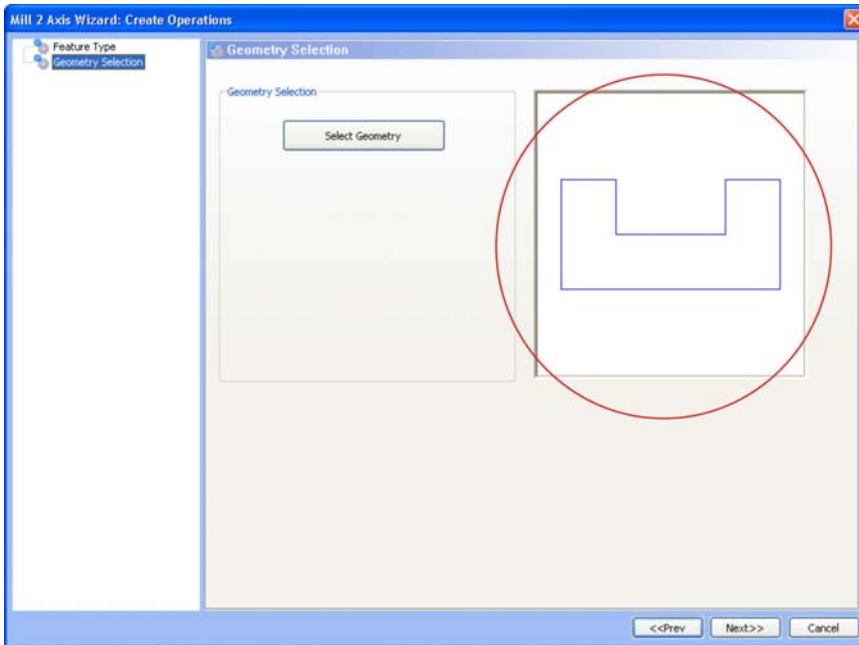
The next page is the Geometry Selection page.



Click on the **Select Geometry** button to exit the wizard and select the part in the CAD workspace.

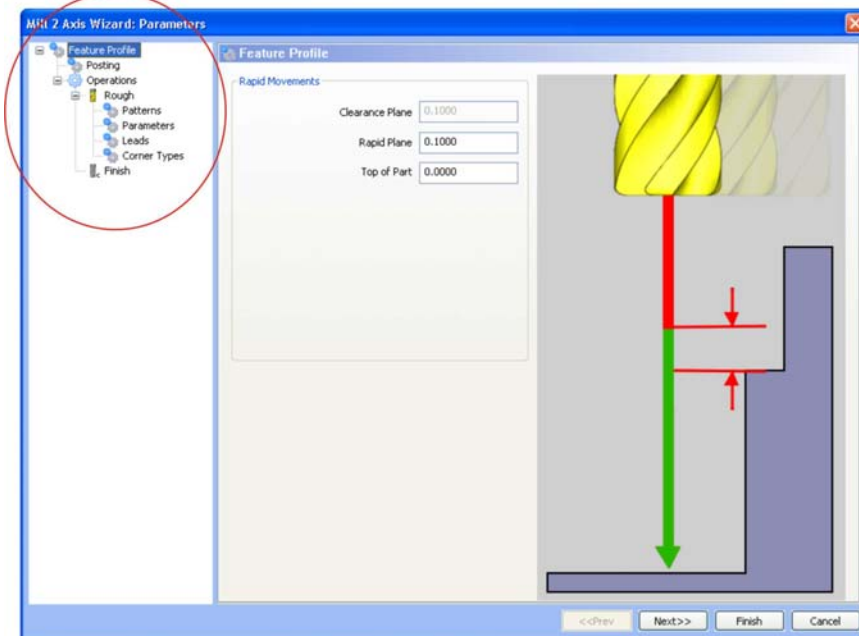
You can window pick the entire profile as this is the quickest method. To do that place your cursor in the upper left quadrant of the geometry, click and drag a selection window over the entire profile while holding down your left mouse button, then releasing it when the part has been selected.

When the entire profile is selected hit the **Space Bar** on your keyboard to indicate “OK” for the selection. Now the wizard will return.



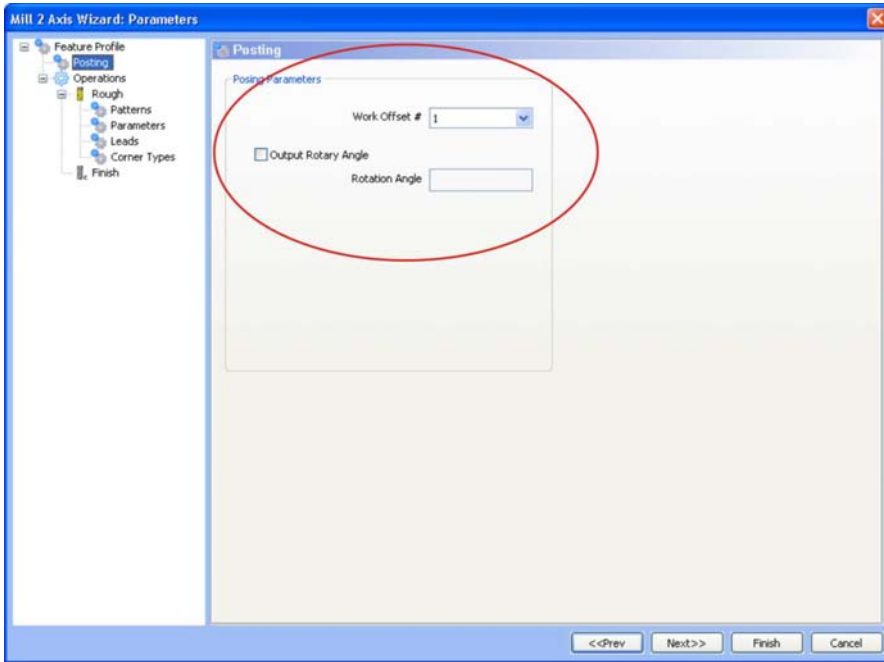
The profile geometry will be displayed in the preview window. Now click the **Next** button to advance.

The wizard will include a populated Tree inside of it with all of the items in the profile operation.

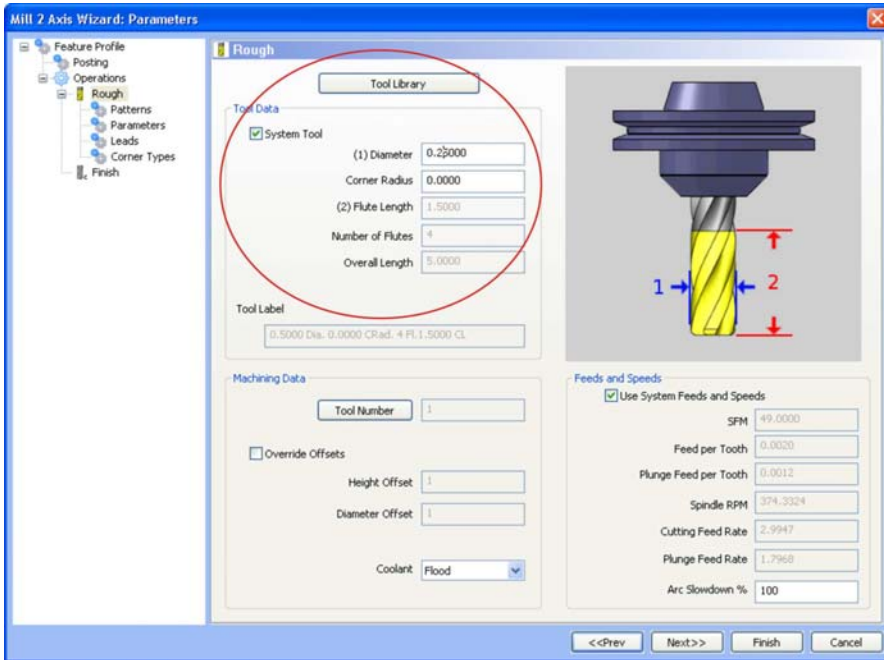


You should not have to change the Clearance Plane, Rapid Plane or Top of Part for this lesson at all. Click the **Next** button. The next page is the Posting page.

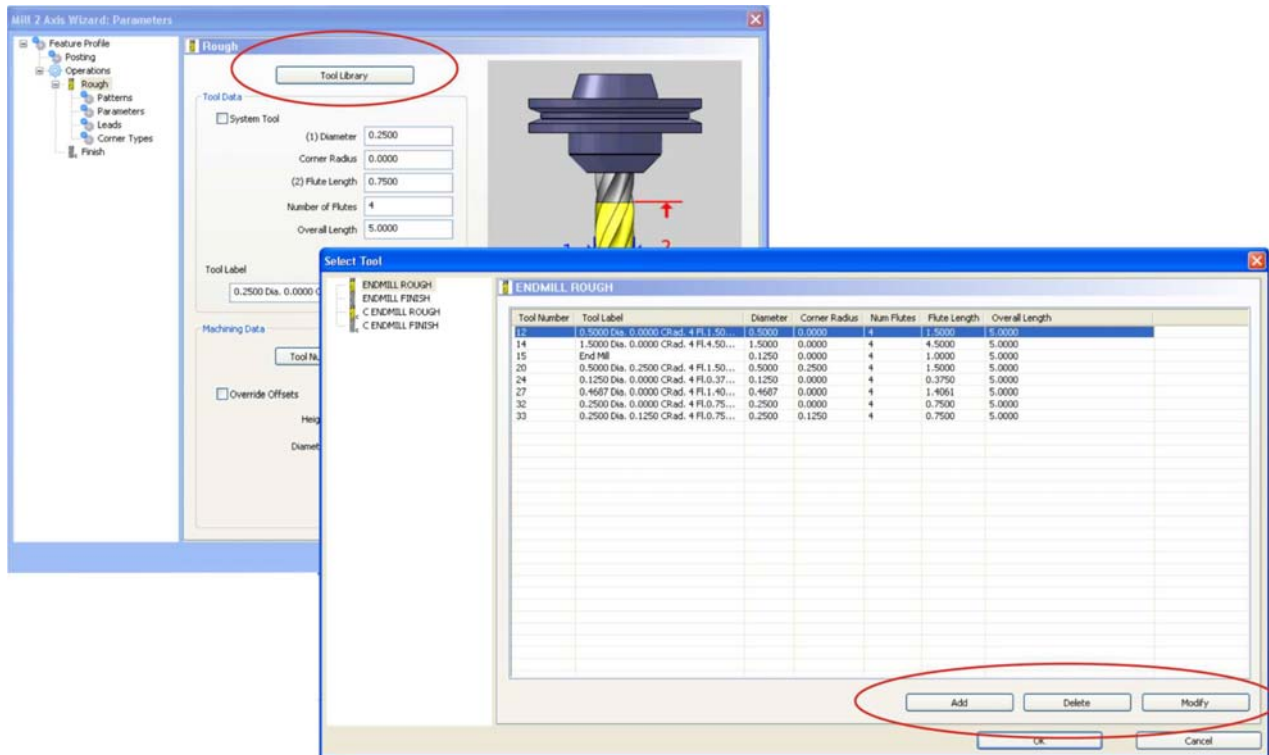
The **Work Offset** field allows the user to choose which work offset code to use for this feature in the posted code. The post processor must be configured to support the work offset chosen. By checking the **Output Rotary Angle** option allows you to input a fixed angle in this field for 4th Axis programming. Then that rotation angle will be included in the posted NC program.



Because this is not a 4th Axis lesson, simply click **Next**. This next page is the Rough Tool page.



The software will automatically load the tools for you based off of what has been setup. To access the Tool Library you can click the tool library button.



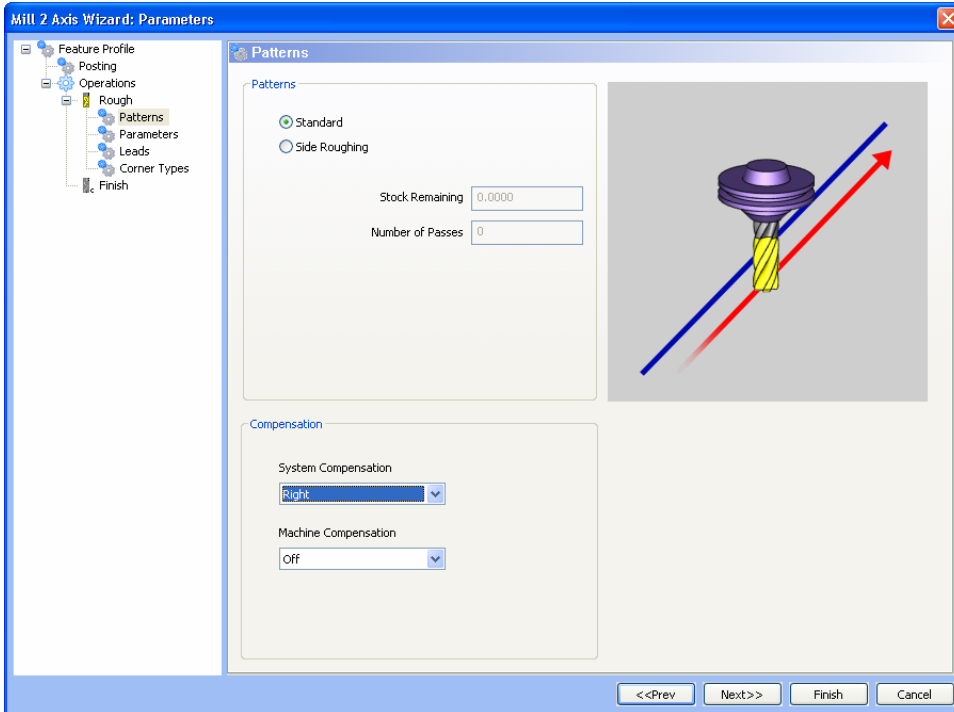
The Select Tool page for an Endmill Rough tool would appear allowing you to Add, Delete or Modify an existing tool.

This page is divided into 3 sections:

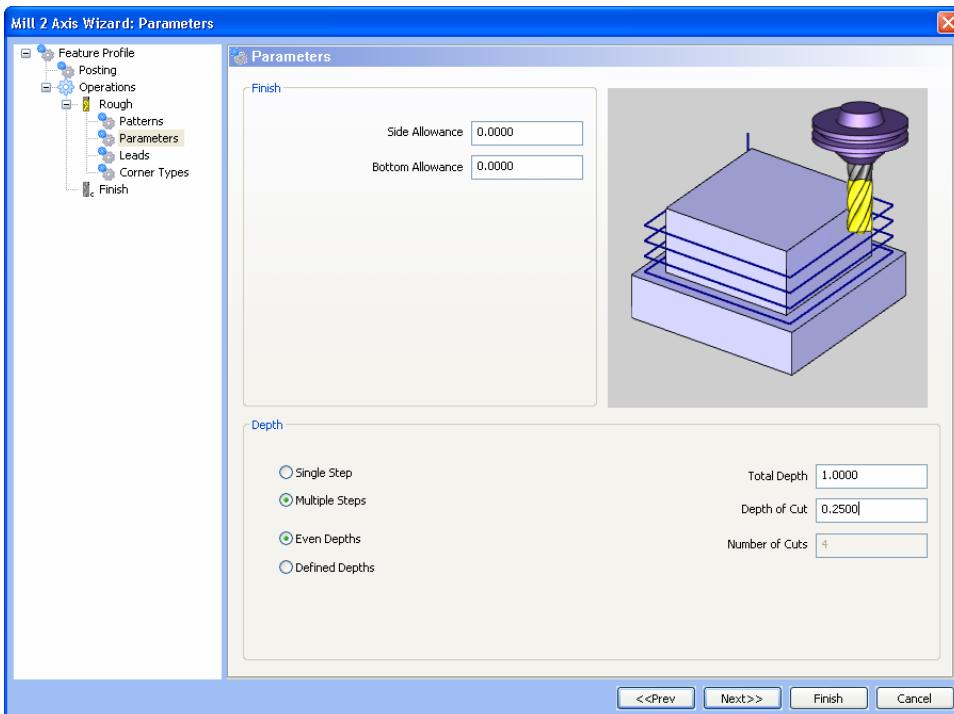
- **Tool Data.** If System Tool is checked then all of the tools data is pre-loaded into the wizard for the drilling action. By un-checking this option you will be able to manually make modifications.
- **Machine Data.** Here you can choose what type of coolant you want to be posted out in the program. By checking Override Offsets you can manually change the registry values for Height and Diameter offsets. This is also a new addition to the Version 24.
- **Feeds & Speeds.** By checking Use System Feeds & Speeds the software will calculate SFM, Spindle RPM and Plunge Feed Rate. By manually changing the Plunge Feed Rate, the Plunge Feed per Tooth will update. By changing the SFM value and using the enter key on your keyboard the appropriate values will automatically update.

Change the **Tool Diameter** to **.25** and click the **Next** button to advance the wizard.

Here we have the Patterns page. This page allows you to choose from standard profiling or side roughing. We will use standard. You will want to use System Comp Right.

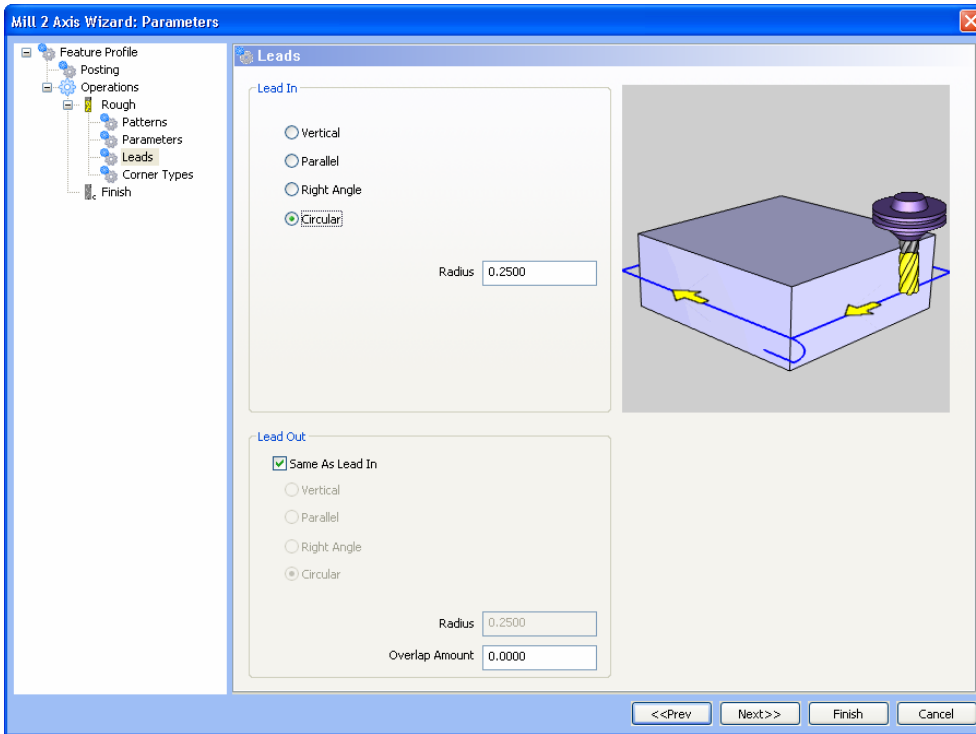


Click the **Next** button. We do not need to add side or bottom allowance. Select the Multiple Steps option, enter 1 for Total depth and enter .25 for Depth of Cut (we're doing 4 steps).

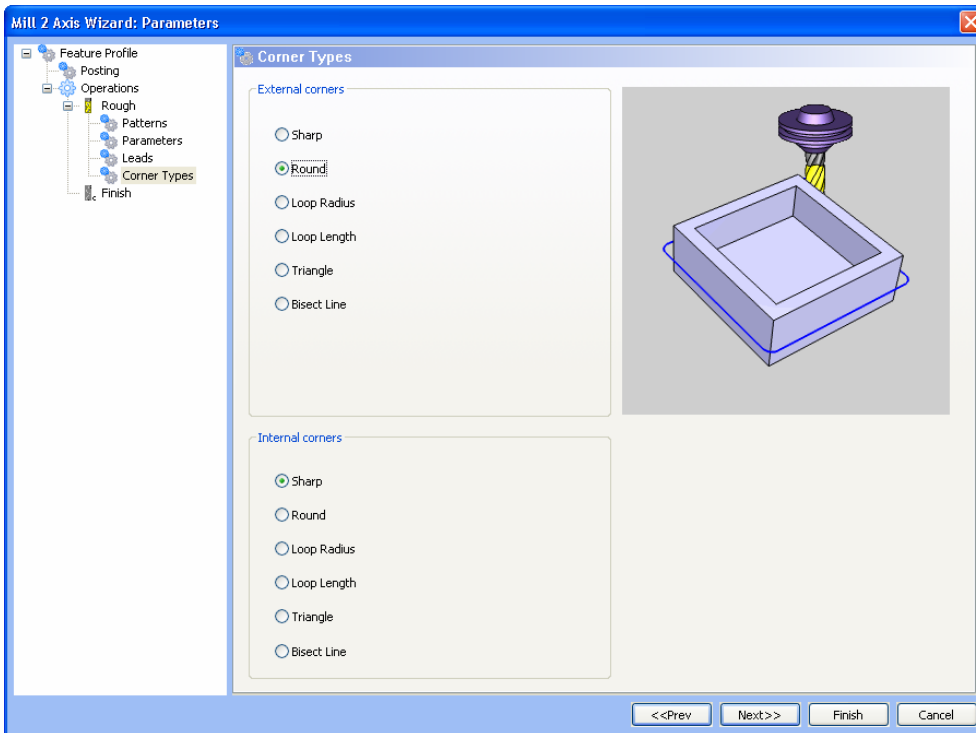


Now click the **Next** button.

Now you will see the Leads page.



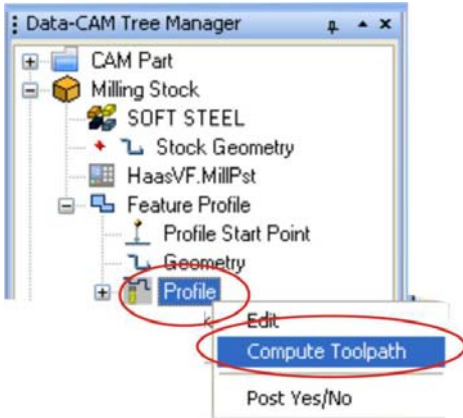
Select Circular as the lead-in type and the bottom you will see will be the same. Enter .25 for the Radius. Click the **Next** button.



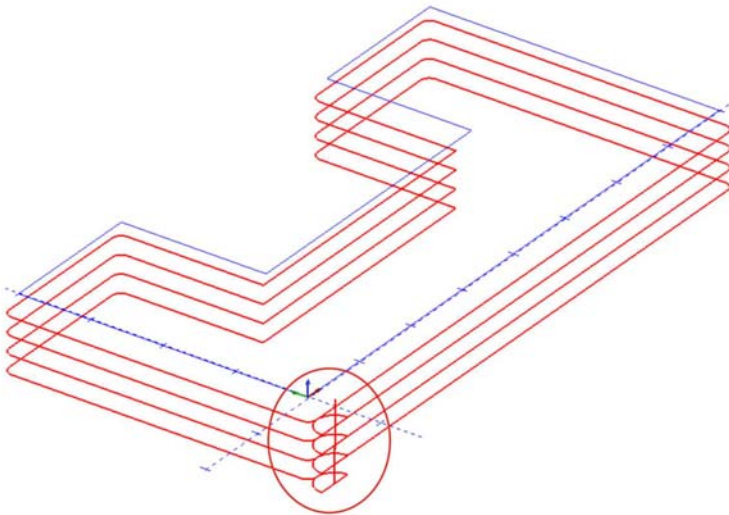
Select Round as the external corner type and click the Finish button to exit the wizard.

Step 5

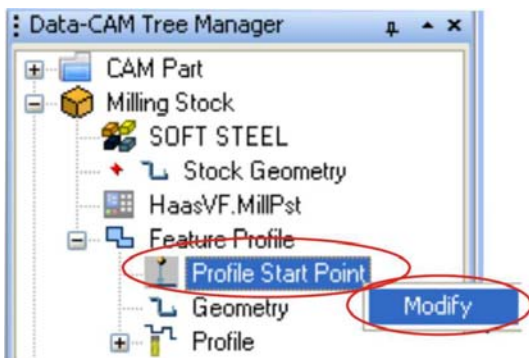
Place your cursor on Profile in the CAM Tree, right click your mouse and select Compute Toolpath.



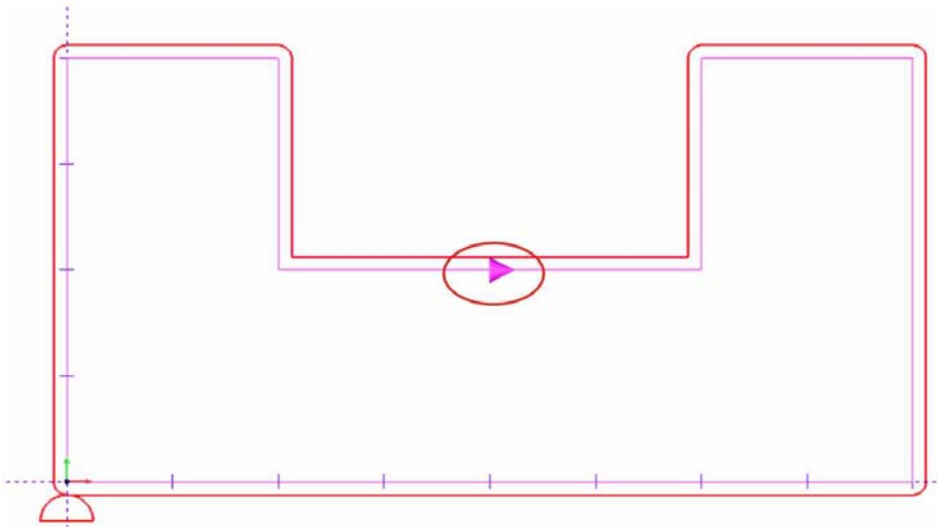
Now you will have the toolpath on the screen. In this example my lead-in and out toolpath is located in the lower left corner of my part. Yours may be different depending on how the geometry was selected.



Version 24 offers the ability to easily modify your start point. Using a **Top View** for your part, Right-click on Profile Start Point in the CAM Tree and select Modify.

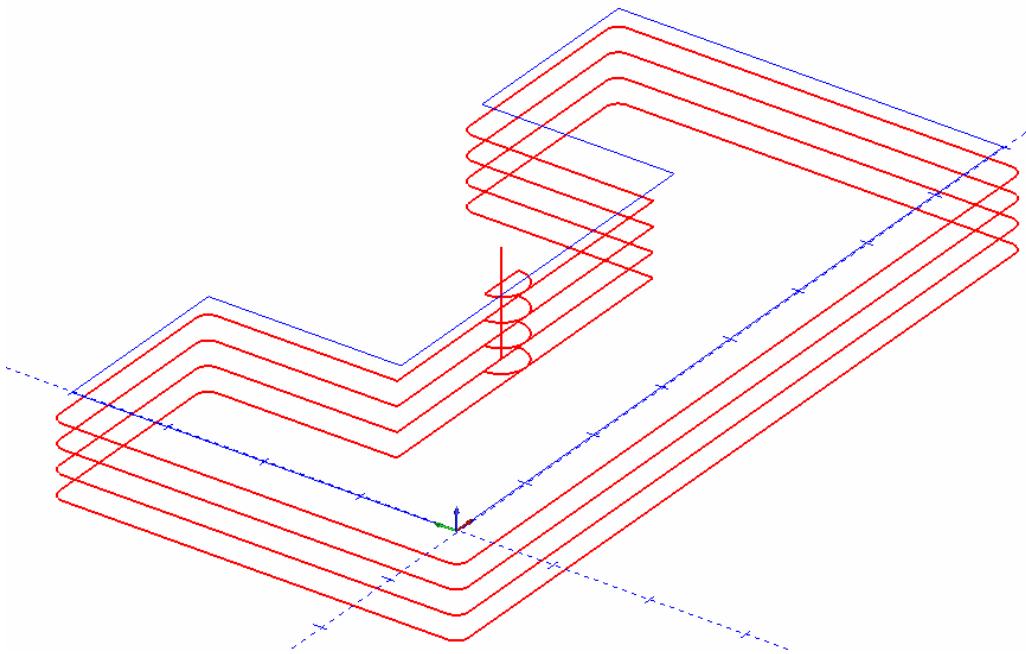


You will automatically be put in selection mode and can click on any of the entities of the profile to change your start point. In the image below you will be able to see the purple arrow that indicates the new selected start point and direction.

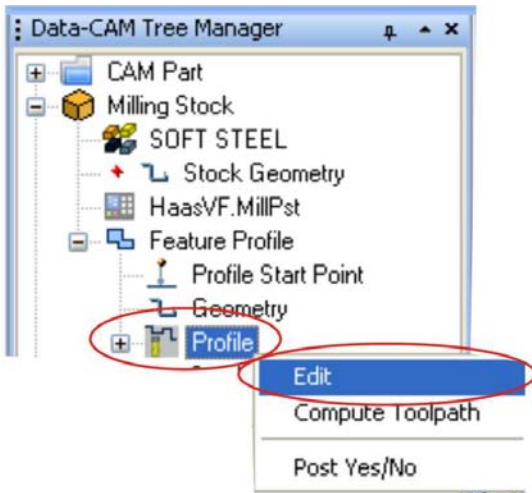


By clicking on the arrow a second time you can change the direction.

You can play around with this feature until you are comfortable using it. When finished, hit the Space Bar on your keyboard to indicate OK. Then re-compute the toolpath with the new start point.

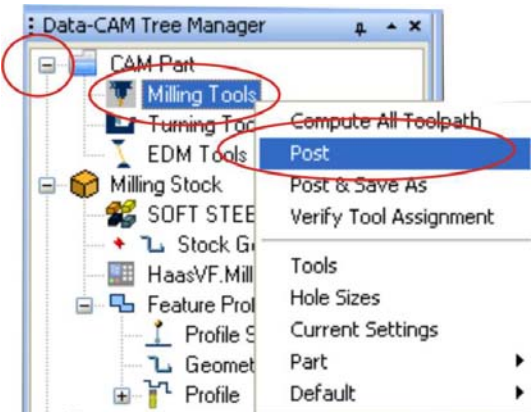


If you have made changes to your start point and direction and need to edit the compensation settings in the wizard you can do this by right-clicking on Profile in the CAM Tree and selecting Edit to access the wizard.



Step 6

To post the program click the small plus symbol next to CAM Part to expand the Tree, right-click on Milling Tools and select Post.



That concludes this lesson.