

BobCAD-CAM v24 Training

Version 24 Lesson – Simple Z-Level Roughing

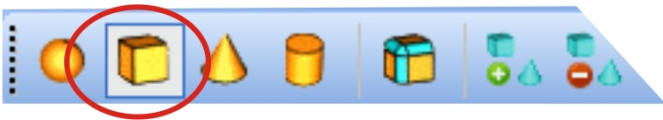
BobCAD-CAM Version 24 Training Lesson

The Version 24 offers a powerful and easy to use 3 Axis Machining Wizard to step you through the process of creating 3 Axis programs for CNC machining. This exercise has been created to teach you how to draw a simple solid model to be used for creating Z-Level Roughing toolpaths.

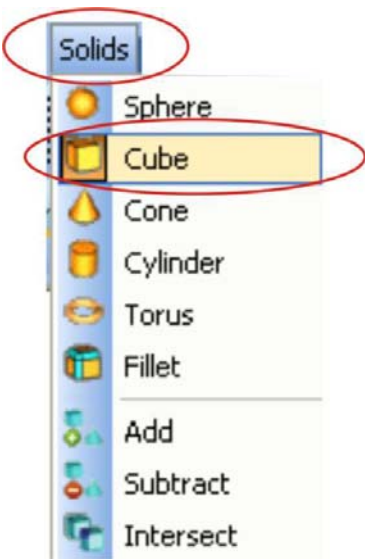
Start off with a new drawing screen and follow along doing each step.

Step 1

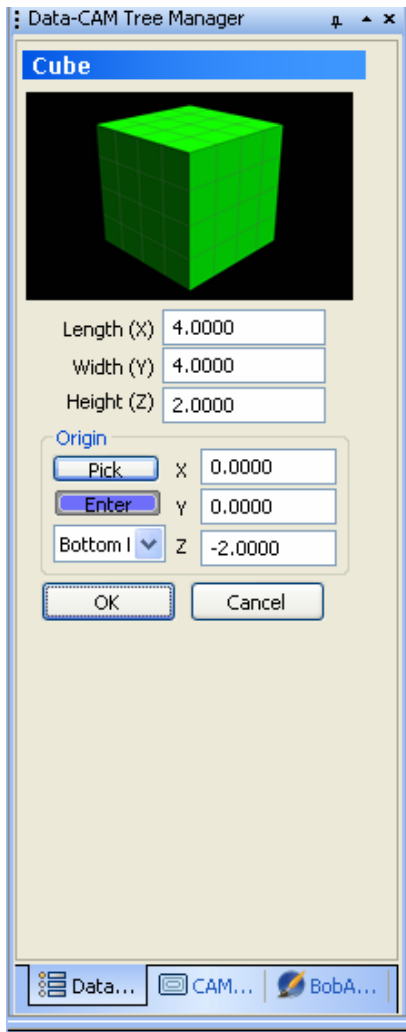
We are going to start this exercise by creating a simple 3D model. Click the Cube feature icon from the Solids Toolbar to load the feature into the Data Manager.



Or, you can do this by going to the **Solids** Main Menu and selecting Cube.



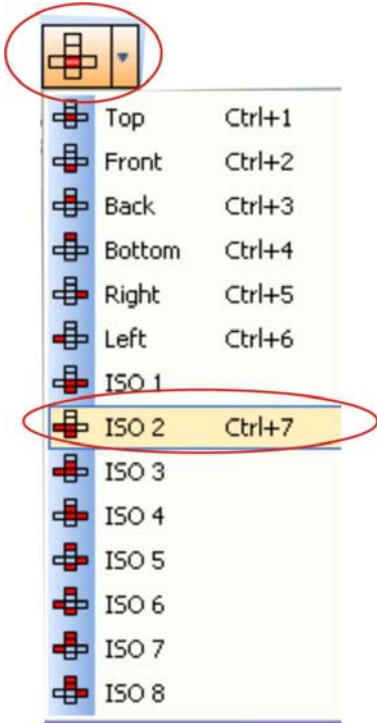
This will load the Cube feature into the Data Manager.




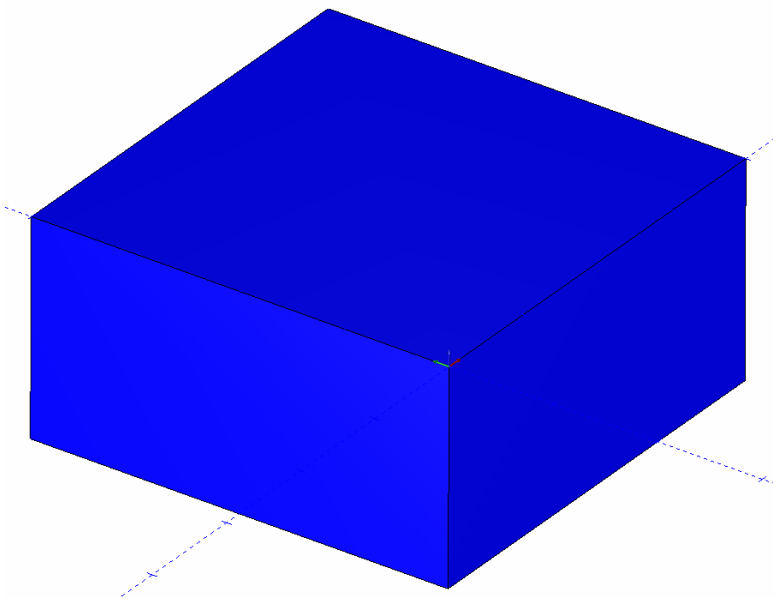
Enter 4 for the Length X
Enter 4 for the Width Y
Enter 2 for the Height Z

Choose Enter in the Origin section
Leave X at 0
Leave Y at 0
Enter -2 for Z (the top will be at Z0)
Choose **Bottom Left** from the pull down menu and click the OK button.

Now you can click the ISO 2 View icon from the View Toolbar. This will change the view to a 3D isometric view.



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

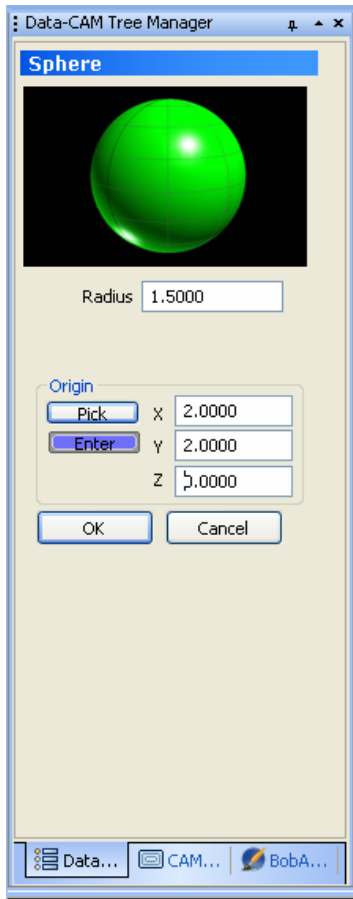


Step 2

Now draw a sphere by clicking on the Sphere feature icon on the Solids toolbar.



This will load the Sphere feature into the Data Manager.



Enter a Radius of 1.5

Select Enter in the Origin section.

Enter 2 for X

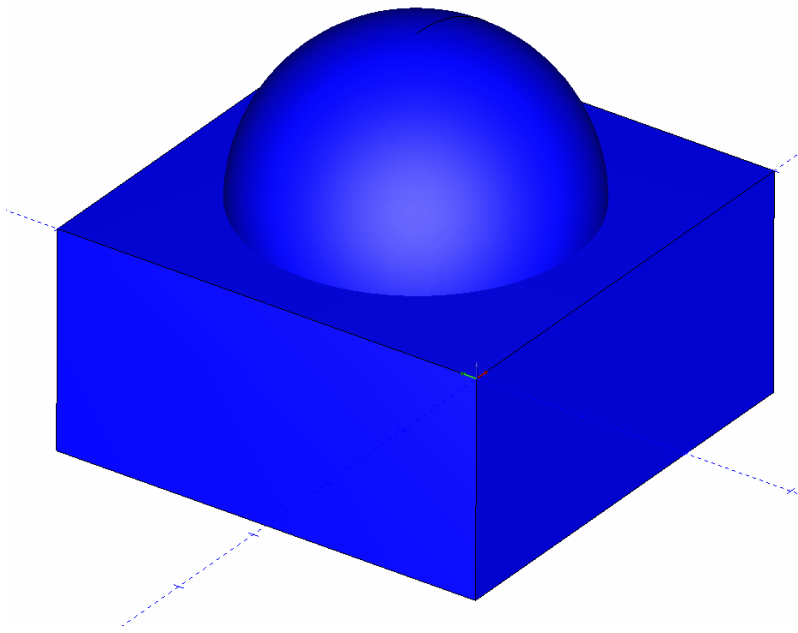
Enter 2 for Y

Leave Z at 0

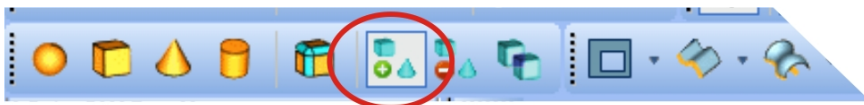
Click the OK button to draw the shape.

Now click the Cancel button.

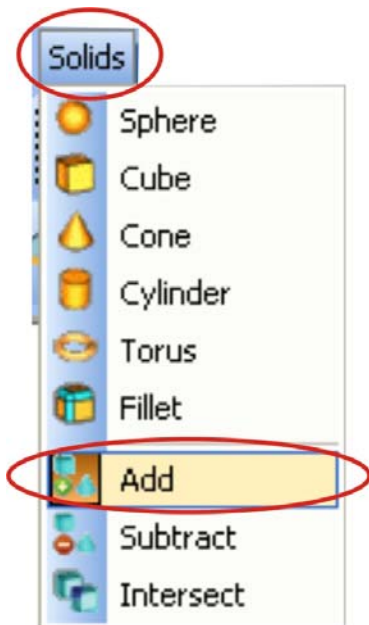
The result:



Now go to the **Solids toolbar** and select **Add**.



You can also click this feature by selecting the Solids main menu and then by clicking **Add**.



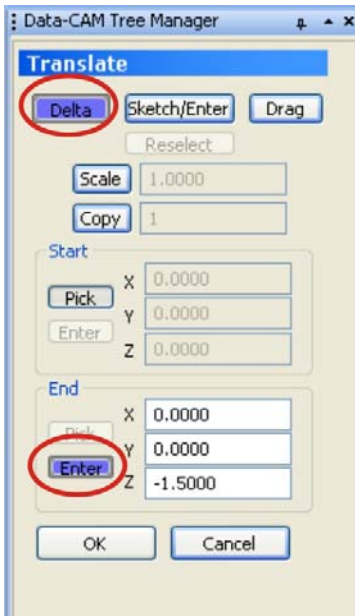
Simply click on the sphere and then click on the cube. With both selected hit the Spacebar keyboard or click the OK button in the Data Manager. Then click the Cancel button to exit the feature.

At this stage we are ready to begin creating toolpath. However, the top of this part is above Z0. This means we need to use the **Translate** feature to drop the part 1.5-inches in the Z axis. To do this click the Translate feature icon from the Utilities Toolbar.



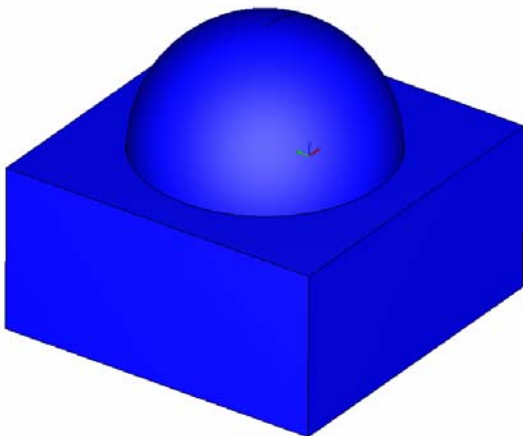
You can also access this feature by going to the main Utilities Menu.

By clicking the feature icon it will load into the Data Manager. The **FIRST** thing to do is click on the model to select it. Once selected, hit the Spacebar key on the keyboard to indicate OK.



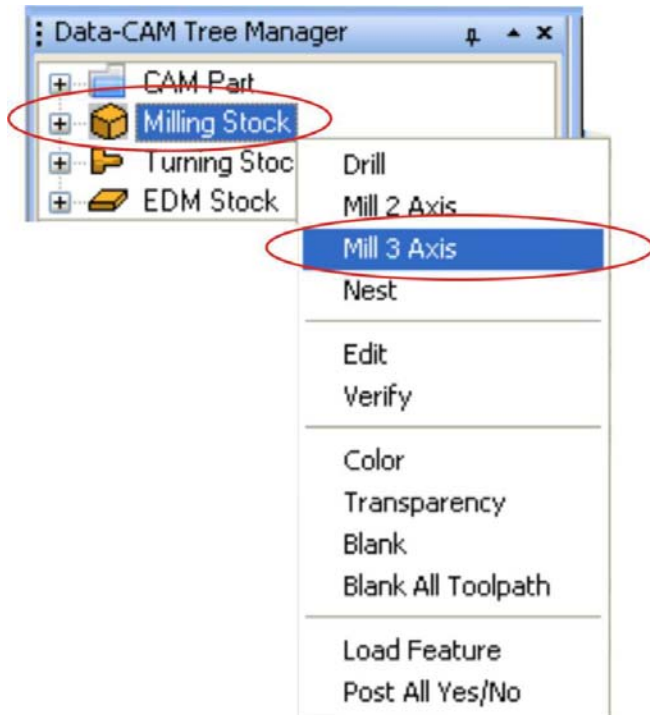
Make sure that the Delta button is selected. We are not making any copies. Click the Enter button in the End section and enter X & Y 0 and -1.5 for Z. Click the OK button to move the part and then click the Cancel button to exit the feature.

The result:

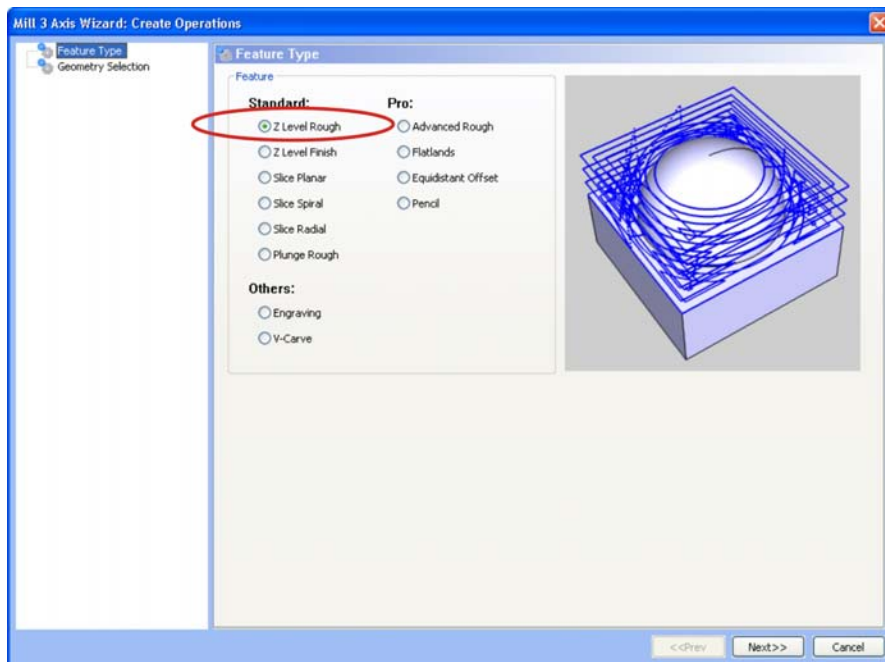


Step 3

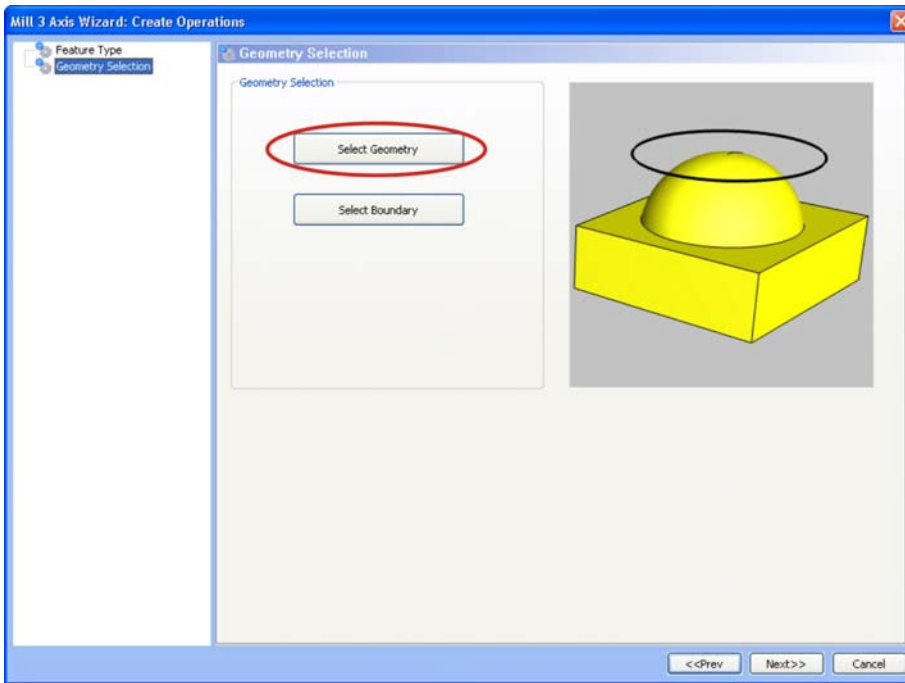
Now place your cursor on **Milling Stock** in the Tree, right click your mouse and click **Mill 3 Axis**.



This will launch the 3 axis wizard.

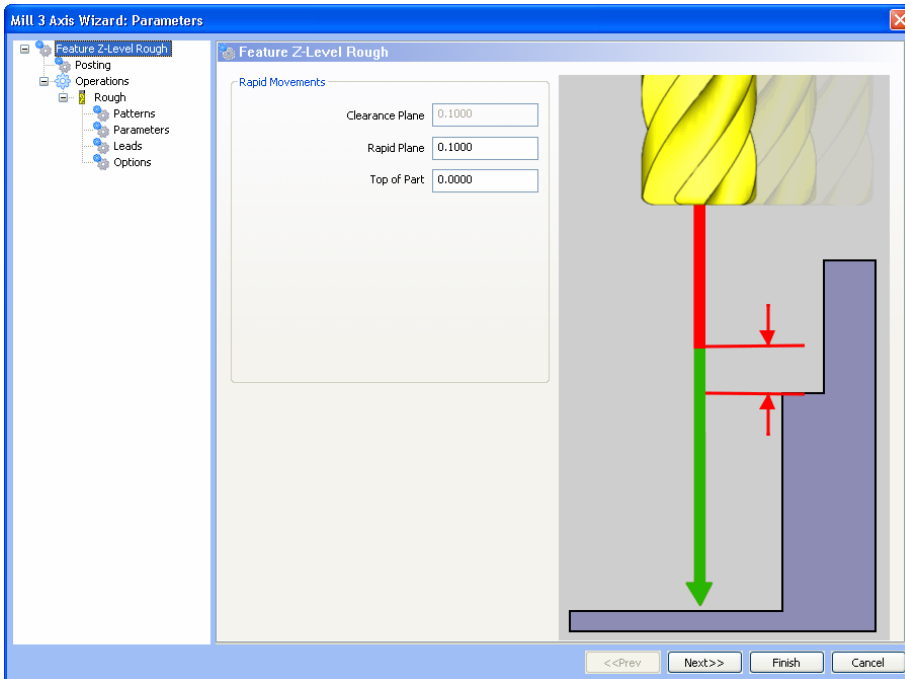


Click the **Z-Level Rough** option. Now click the **Next** button to advance to the **Select Geometry** page.



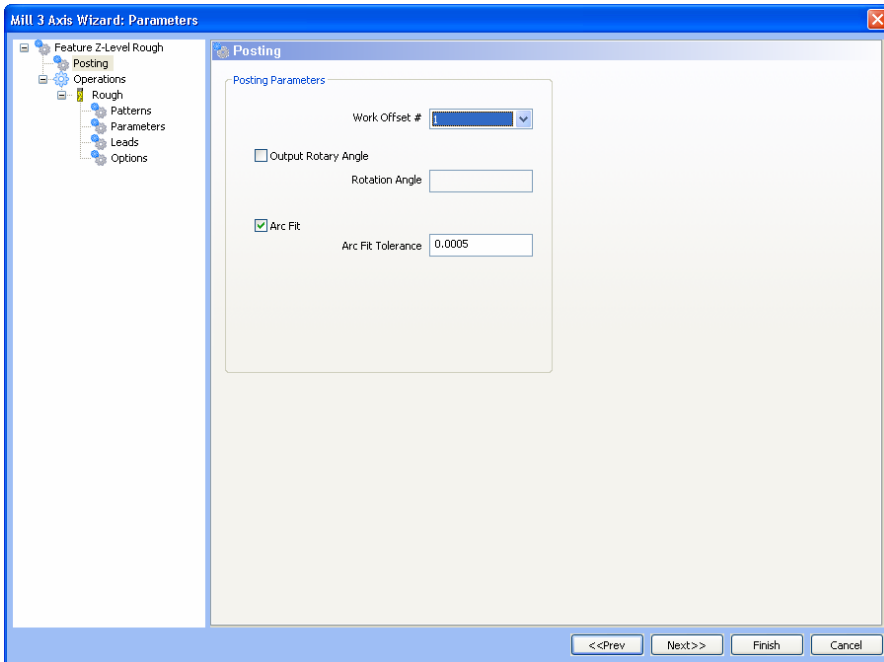
Click the Select Geometry button. The wizard will go away and you will automatically be placed in selection mode. You need to click and drag a selection window over the part. Just place your cursor in the upper left area of the part in open space, click your left mouse button and hold it down while moving your cursor to the lower right area of the workspace. Then release your mouse button to window select the part. Now hit the Spacebar on your keyboard to indicate OK. By doing this the wizard will reappear.

We have not created a boundary to contain the toolpath as it's not necessary for this part. Simply click the **Next** button to advance the wizard. This is the Feature Z-Level Rough page.

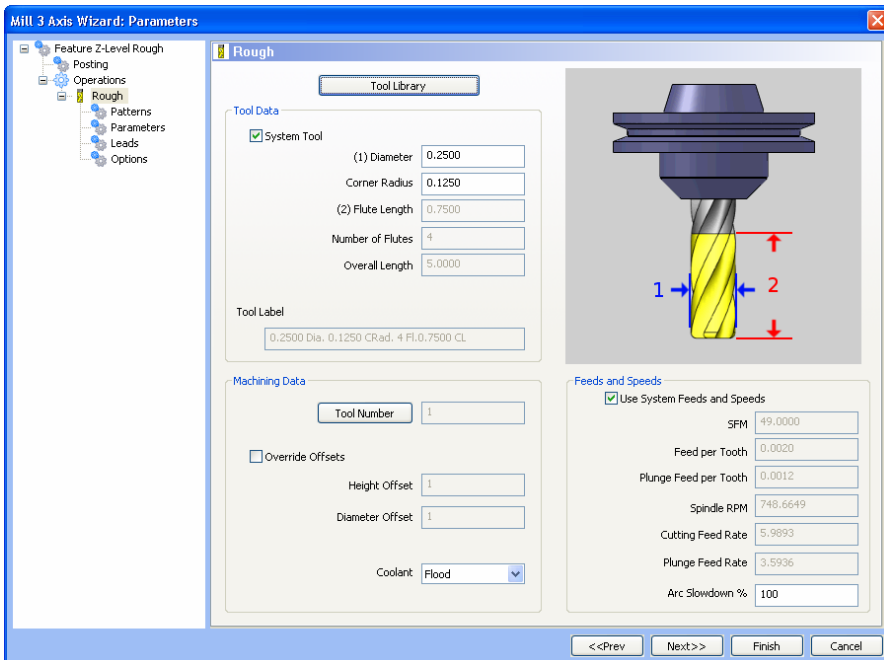


This page deals with the Clearance Plane, Rapid Plane and the Top of Part in terms of the parts position on the screen and setting up the toolpath to machine it.

While the Clearance Plane is set, enter .1 for the Rapid plane and 0 for Top of Part. Click the **Next** button to advance the wizard. This is the Posting Page.

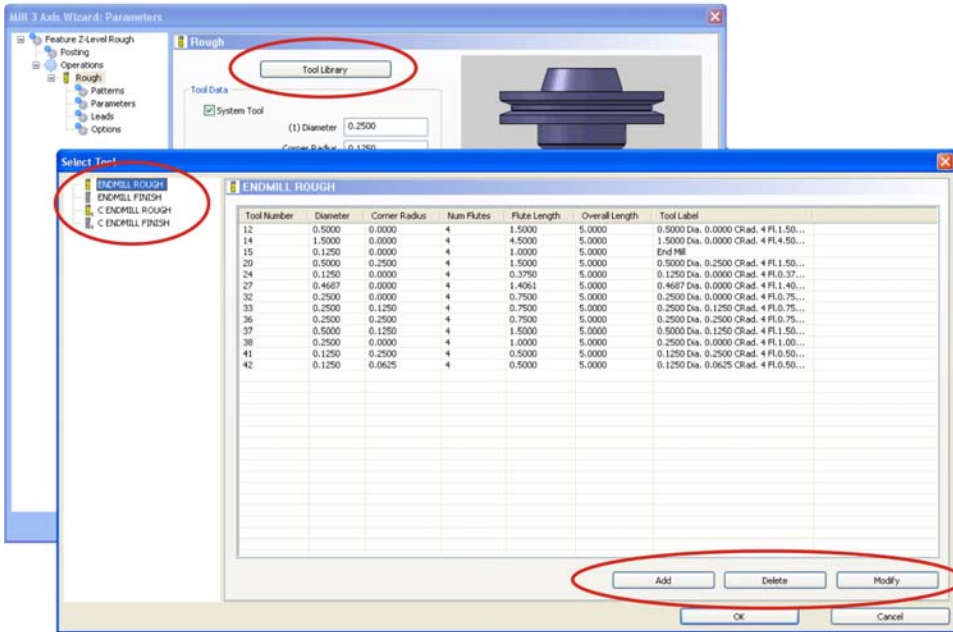


Since this is not a 4th axis program we do not need to check the Output Rotary Angle option. However, click the **Arc Fit** option. What this does is it creates toolpath using arc segments wherever it can rather than line segments. The result is a more efficient NC program. Click the **Next** button to advance the wizard.

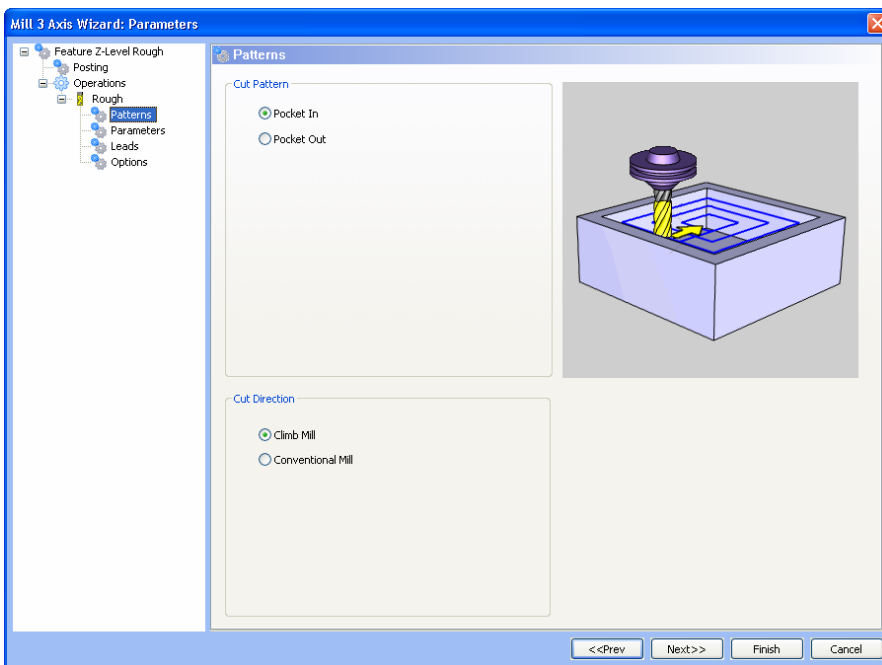


Enter .25 as the tool diameter
Enter .125 for the Corner Radius

For this exercise we will use the system tool that is already pre-loaded. The Rough Tool page allows you to add, delete or modify tools from the tool database. To access the tool database you would click the Tool Database button.

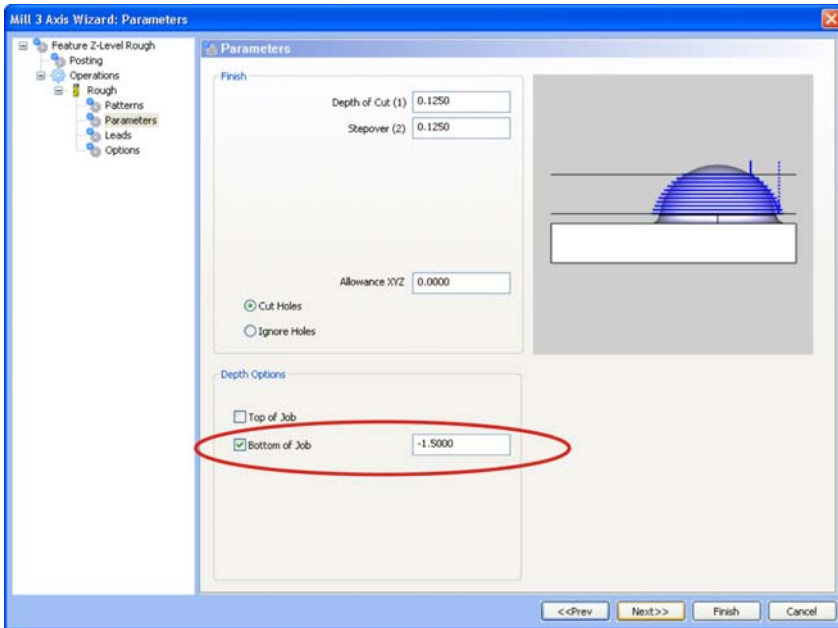


Click the **Next** button to advance the wizard. The next page is the Patterns page.



Click the Pocket In option as well as the Climb Mill option. Click the Next button to advance the wizard.

The next stage of the wizard is the Parameters page.



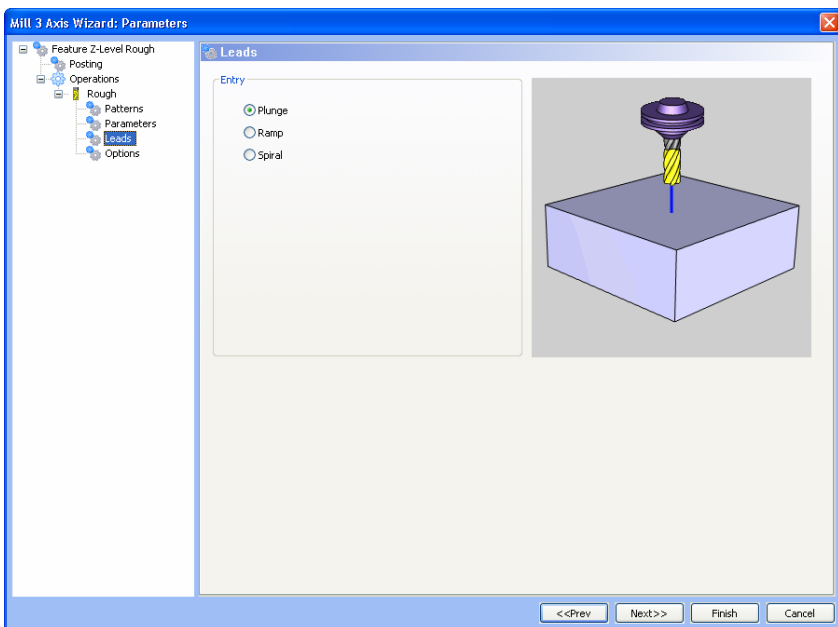
Enter .125 for the Depth of Cut

Enter .125 for the Step Over

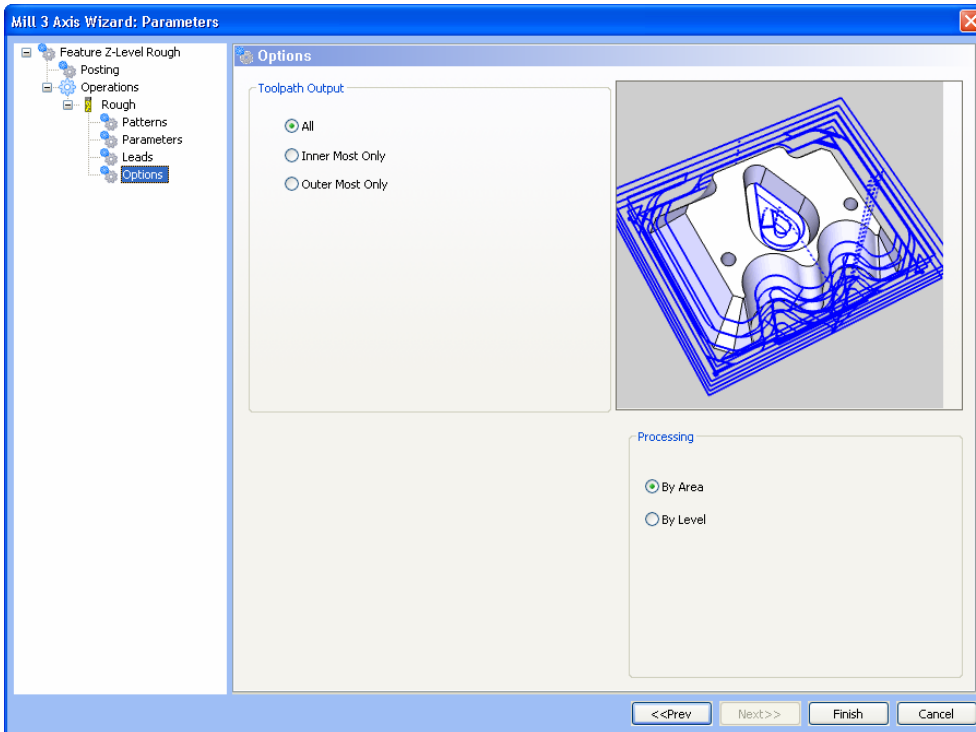
Click Bottom of Job and enter -1.5. This is as far deep as we will go in Z.

We are not going to leave a side or bottom allowance for the exercise. The part doesn't have holes so we do not need to be concerned about the cut holes or ignore holes options. If you have a model with holes and want to keep the toolpath OUT of the holes you would use this option accordingly. This page also allows you to force a specified Top of Job and/or bottom of job.

Click the **Next** button to advance the wizard. Here you will arrive at the Leads page.

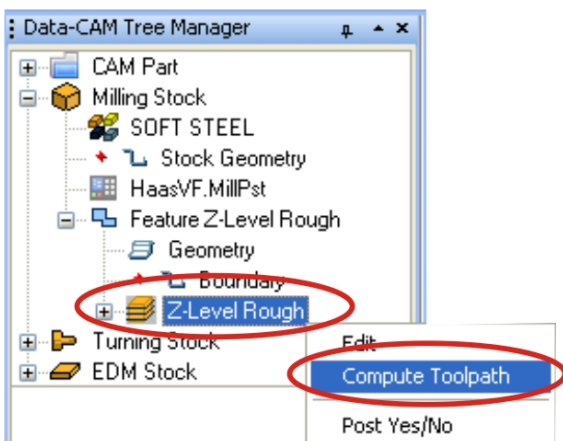


Click the **Plunge** option and then click the **Next** button to advance.

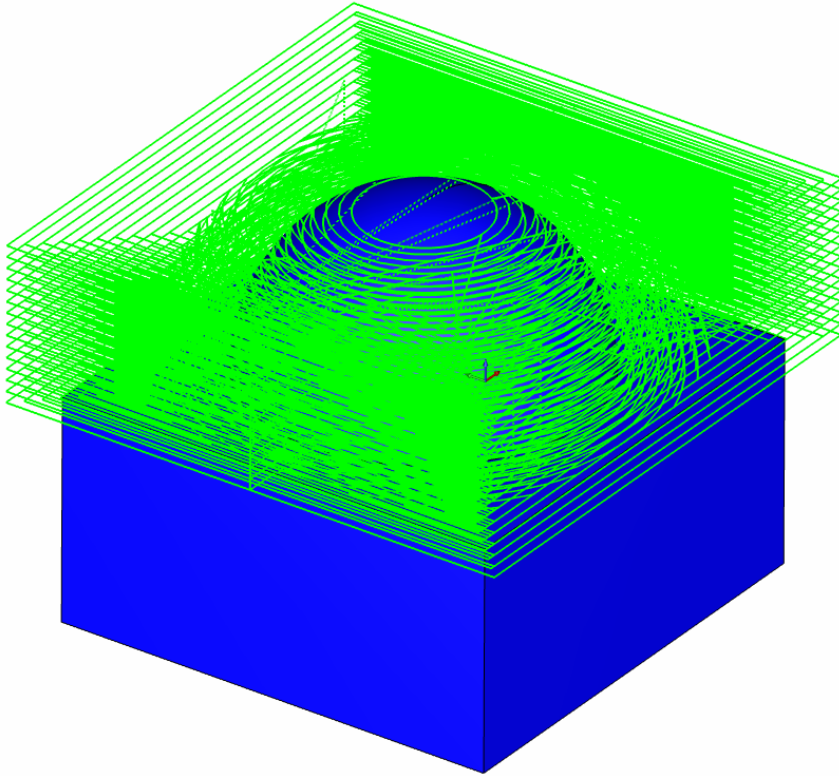


The Options page allows you to determine how to machine the part. If you have a cavity that you want to create toolpath for only the inside areas you could click the Inner Most Only option. If you want to avoid inner areas of a part file you can click the Outer Most Only option. For this exercise we will use the ALL option. Here you can also determine if you want to machine the part by areas or by levels. The By Level option instructs BobCAD-CAM to machine all areas of the part to the current pass depth before beginning the next Z pass.

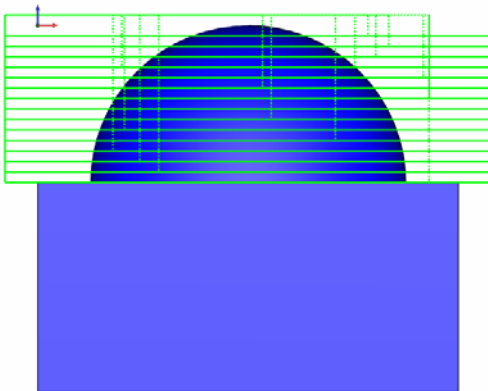
Click the **Next** button to close the wizard.



The result:



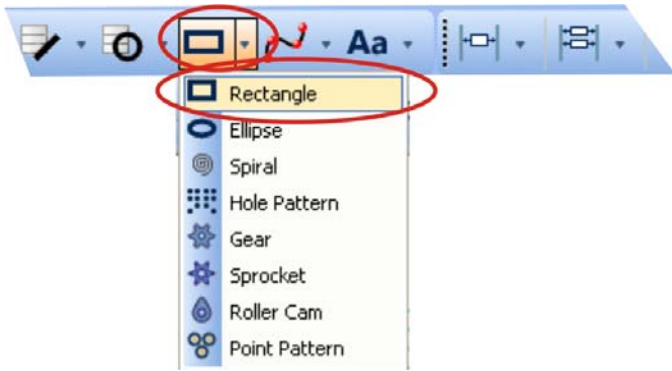
Front View:



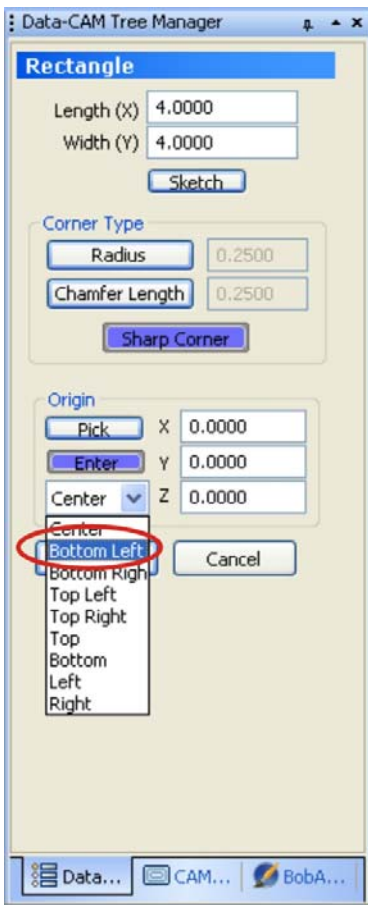
To prevent the toolpath from exceeding out beyond the edges of the block you would draw a square to represent the boundary which could also be used to define the stock. Then in the selection stage of the wizard you would click the Select Boundary option in the wizard to indicate it.

You can always add this after creating the toolpath.

For example, click the rectangle feature from the CAD Toolbar.

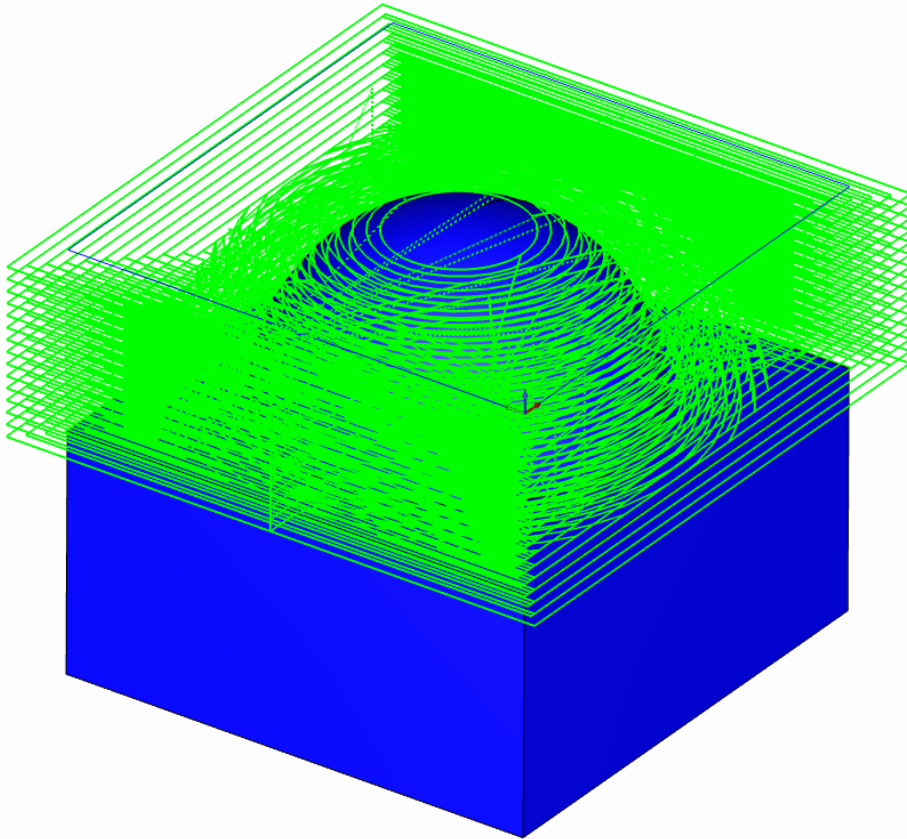


The feature will load into the Data Manager.



- Enter 4 for Length
- Enter 4 for Width
- Click the Sharp Corner button
- Choose Bottom Left
- X, Y and Z are all 0
- Click OK to draw the shape
- Click Cancel to exit the feature

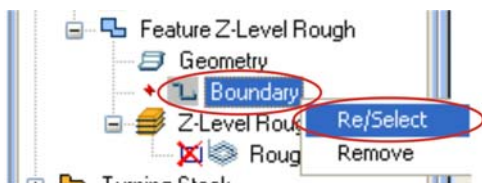
The result:



The toolpath can be blanked out by clicking the small plus (+) symbol next to Z-Level Rough in the Tree. Then right-click on Rough and click Blank.



Now click on **Boundary** in the Tree, then right-click your mouse and click **Re/Select**.

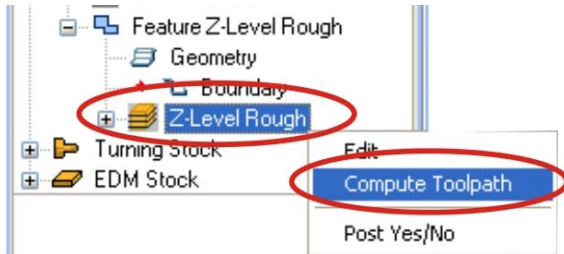


This puts you in selection mode. Hold down your SHIFT key and click on any line of the new square geometry you created. This selects the entire chain. Then hit the Spacebar on your keyboard to indicate OK.

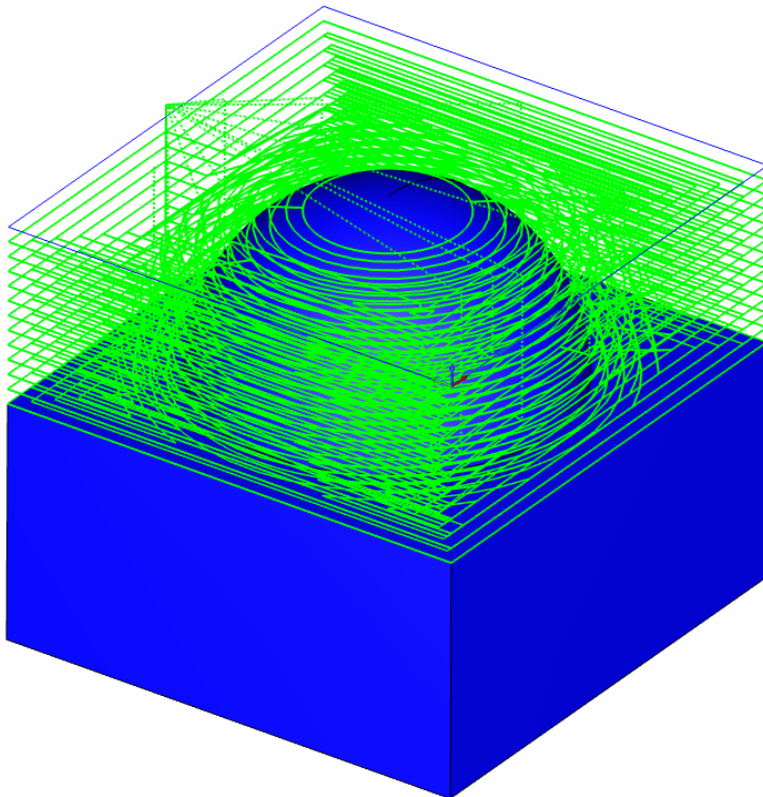
If you blanked out the toolpath, you need to un-blank it now. Right click on Rough and click Blank to **un-blank** the toolpath.



Now go back to Z-Level Rough in the Tree, right click on it and click **Compute Toolpath**.



The result:



You have completed this exercise.