

BobCAD-CAM v24 Training

Using The Hole Wizard

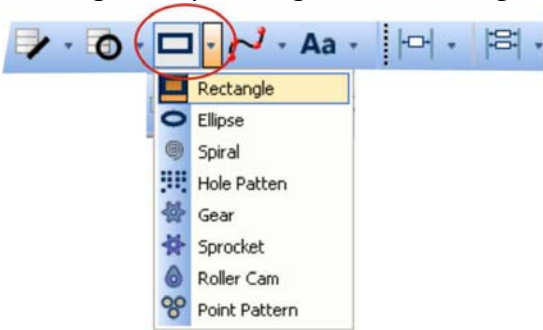
BobCAD-CAM Version 24 Training Lesson

The Version 24 offers a simple hole drilling wizard to step you through the process of creating hole programs for CNC machining.

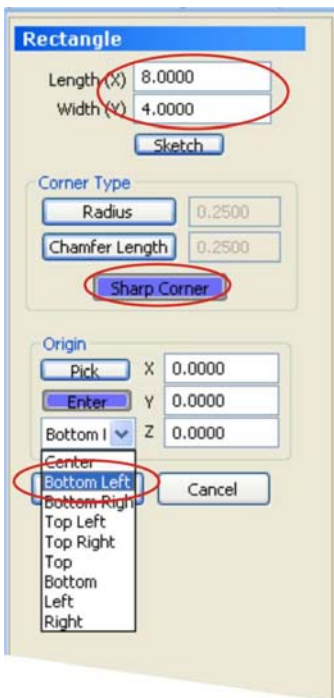
Start off with a new drawing screen and follow along doing each step.

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.



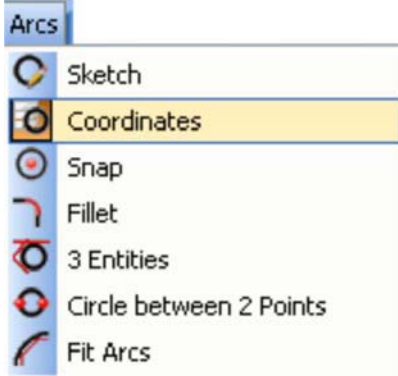
Enter a Length of 8 and a Width of 4.
Choose Sharp Corner as the corner option.
Choose Bottom Left as the origin.
Click OK. This draws the shape in the workspace.

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

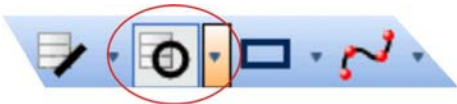
Step 2

This shape will represent the stock. Now we will create the holes to be machined.

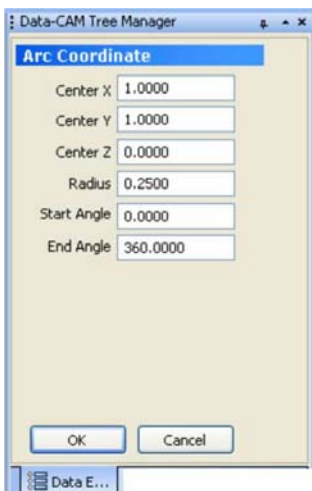
Go to the **Arcs** Main Menu and select Sketch.



Or you can simply click on the ARC Coordinate function icon.



This will load the Arc Coordinate feature into the Data Manager.



Enter 1 for Center X

Enter 1 for Center Y

Leave Center Z at 0

Enter .25 for the Radius

Hit the Enter Key or click OK to draw the first circle.

Now continue.

Leave Center X at 1

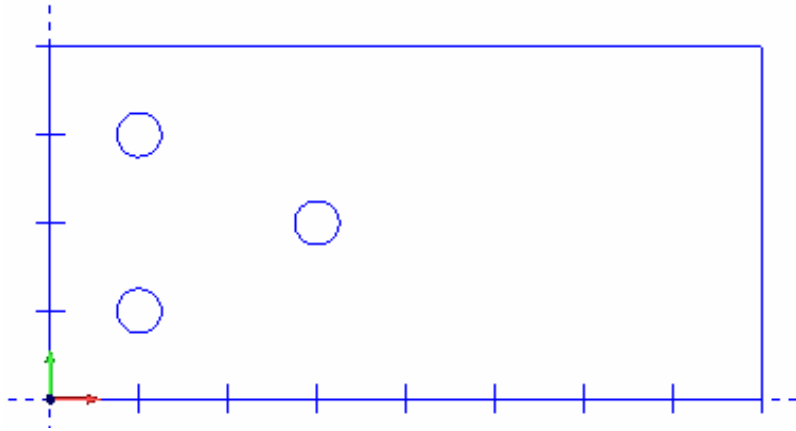
Enter 3 for Center Y

Hit the Enter key on your keyboard or click OK to draw the next circle.

Change Center X to 3

Change Center Y to 2

Hit the Enter key on your keyboard or click OK to draw the 3rd .25 inch circle.



Now change the Radius value to .5 in the Data Manager

Change Center X to 7

Change Center Y to 3

Hit the Enter key or click OK.

We are going to draw 2 more circles to complete this part.

Leave Center X at 7

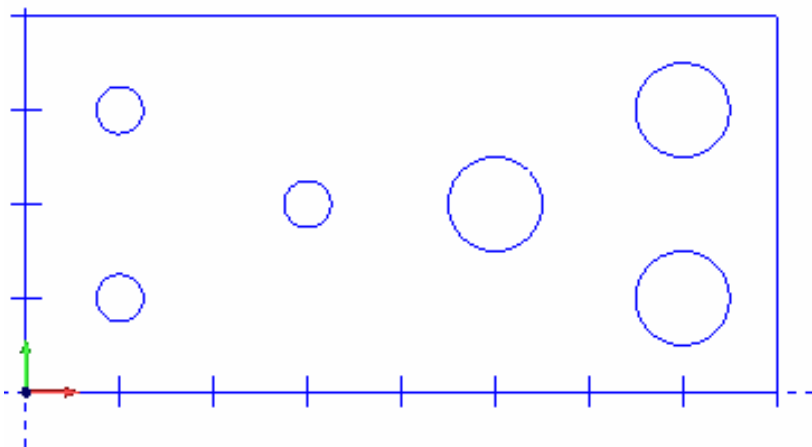
Change Center Y to 1

Hit the Enter key or click OK.

Change center X to 5

Change Center Y to 2

Click OK and then click the Cancel button to exit the feature altogether. Your part should look like the following:



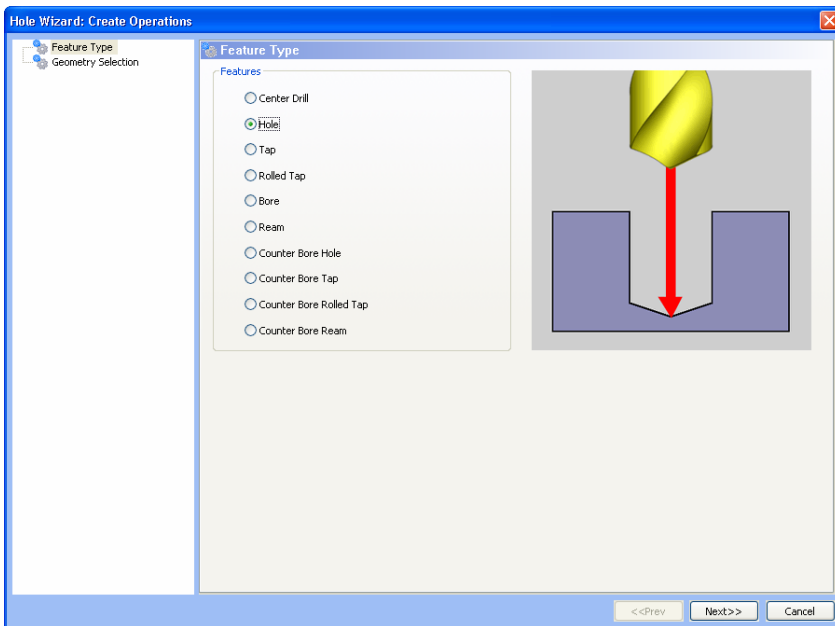
Step 2

Now we want to begin using the Drill Wizard to create the program.

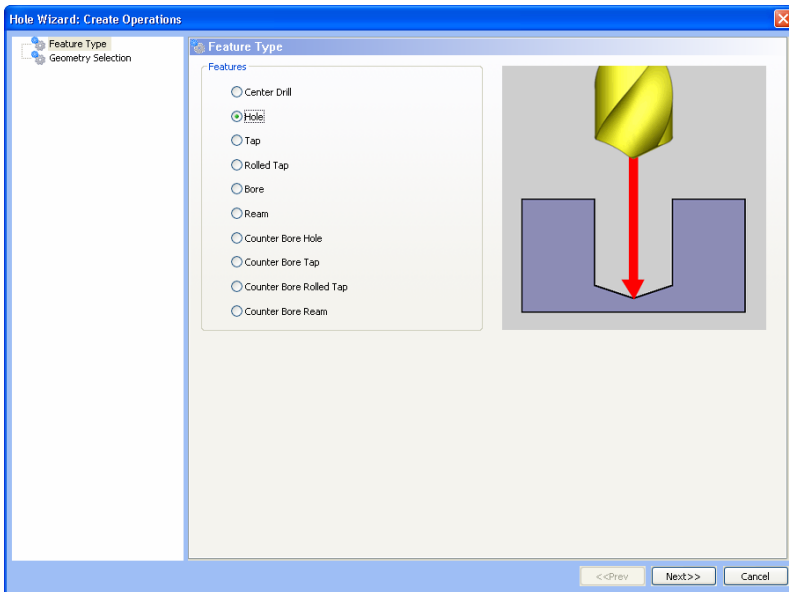


The CAM Tree lists the various modules that BobCAD-CAM has to offer. In this case we are mostly concerned with the CAM Part item and Milling Stock.

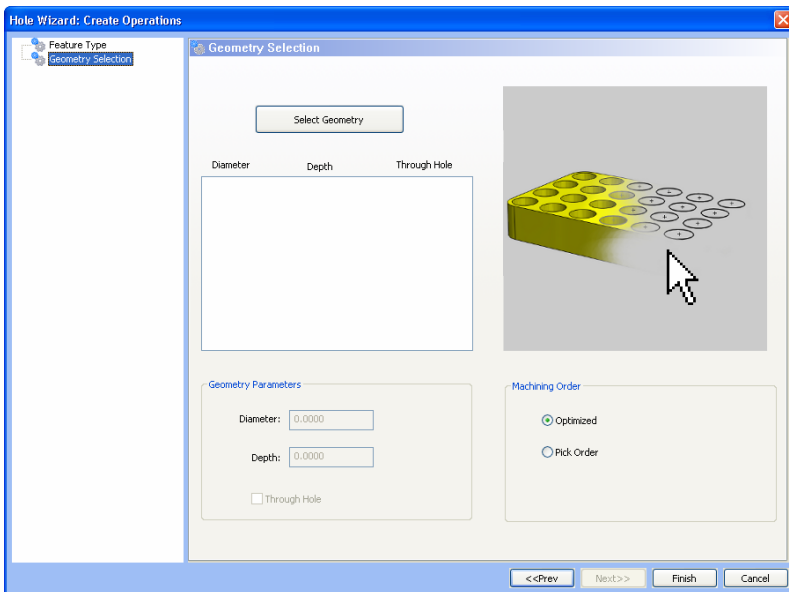
By resting/placing your cursor on Milling Stock and right-clicking your mouse you can select Drill. Do this and launch the Hole Wizard. We will be using the **Hole** feature.



The first dialog of the Drill Wizard allows you to pick the type of drilling operation you want to perform.



Select Hole and then click the NEXT button.



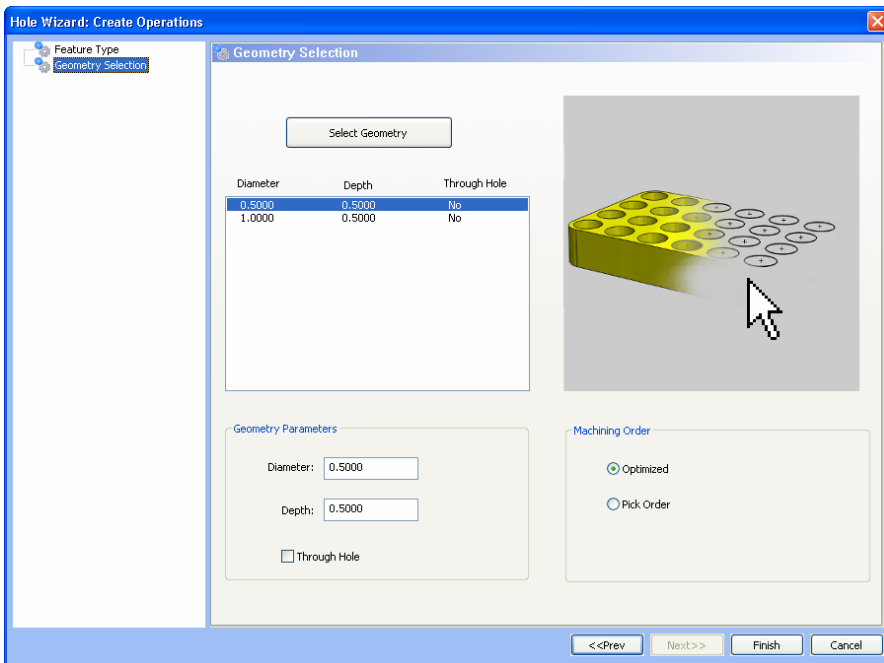
The Geometry Selection dialog allows you to select specific arc geometry for hole making as well as define a machining order. You can pick the sequence for hole drilling (by clicking on the circles in a sequence) or have the software optimize the sequence for you.

Click the Select Geometry button.

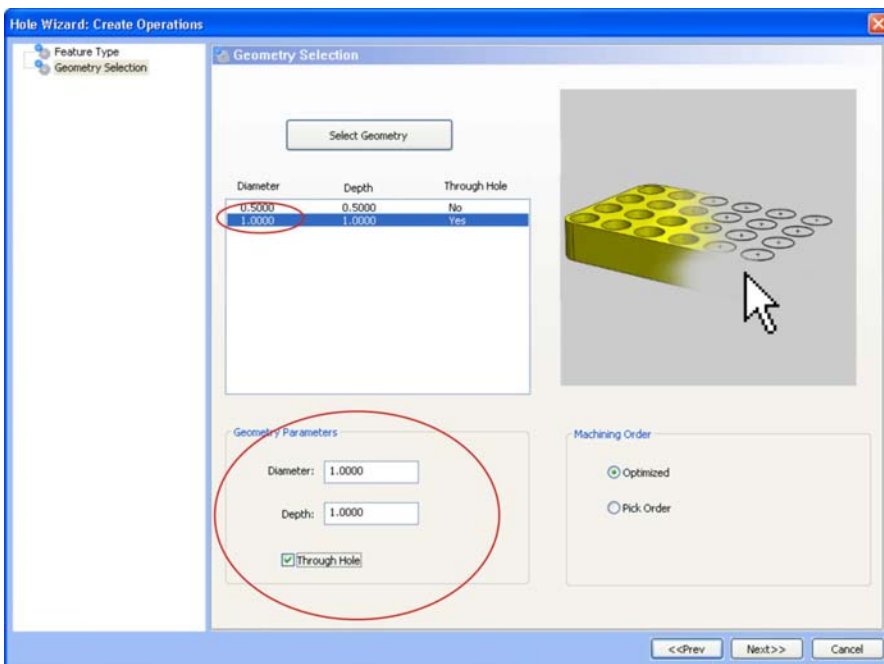


The wizard will go away and you will be in Selection Mode.

Click on each of the .25 holes (they will turn red indicating that they have been selected) and then click on each of the .5 holes. When all of the holes are selected hit the Space Bar on your keyboard to indicate the selection is complete. Now the wizard will reappear.

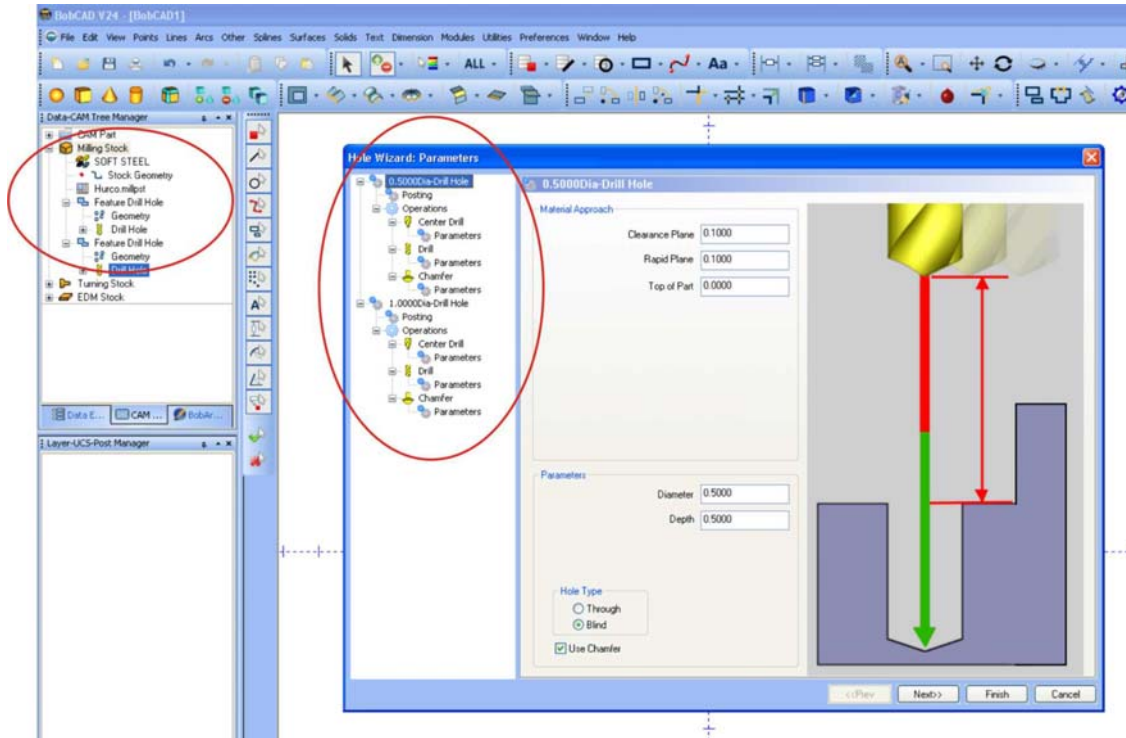


You will see that both sets of holes are listed in the dialog. By clicking on the first set of holes in the window you will see that the wizard has listed the diameter, the depth and if it is a series of through holes or not. Here you can change the hole diameter or the depth.

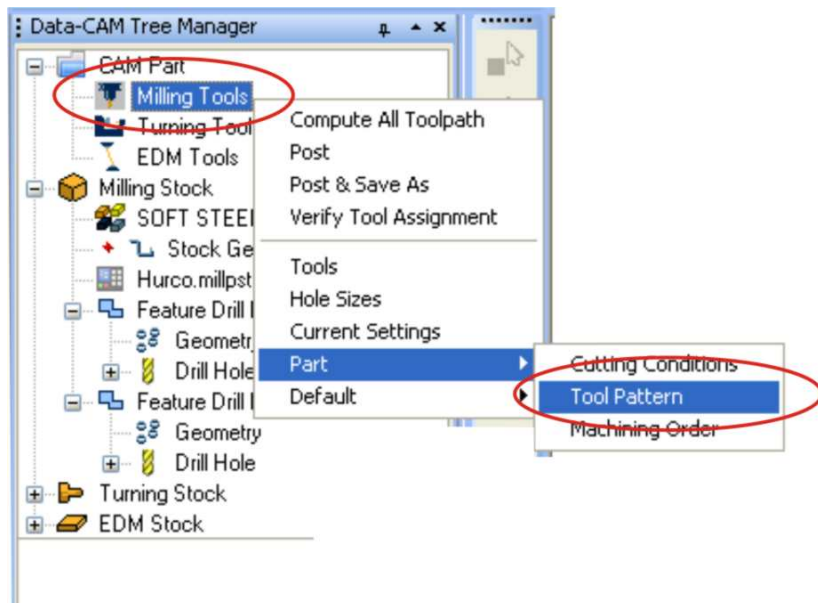


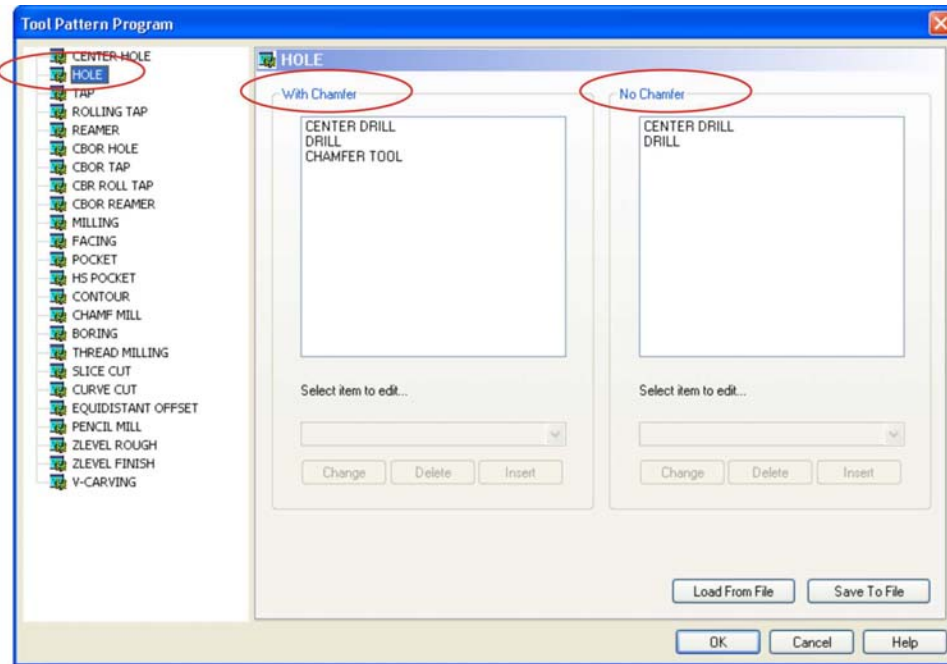
Leave the diameter the same and leave the depth the same. Now click the second listing of the larger holes. Change the Depth to 1 and click the Through Hole option. Now click the **Next** button.

This will automatically populate the CAM Tree and the wizard adding both hole drilling operations, their basic attributes and the tools that will be used based off of the V24 Tool Pattern features.



NOTE: Changing Tool Patterns is a basic and easily modifiable feature in Version 24. Out of the box, V24 is setup to add a Center Drill, Drill and a Chamfer if Chamfer is selected as an option within this wizard or just a Center drill and a Drill if the Chamfer option is not selected.





You have special Tool Patterns for the Part you are working on as well as Default Tool Pattern settings for the entire system which is global for all machining features every time you go to machine any part. When you click on one of the tools listed you have the ability to change it or delete it.

Step 3

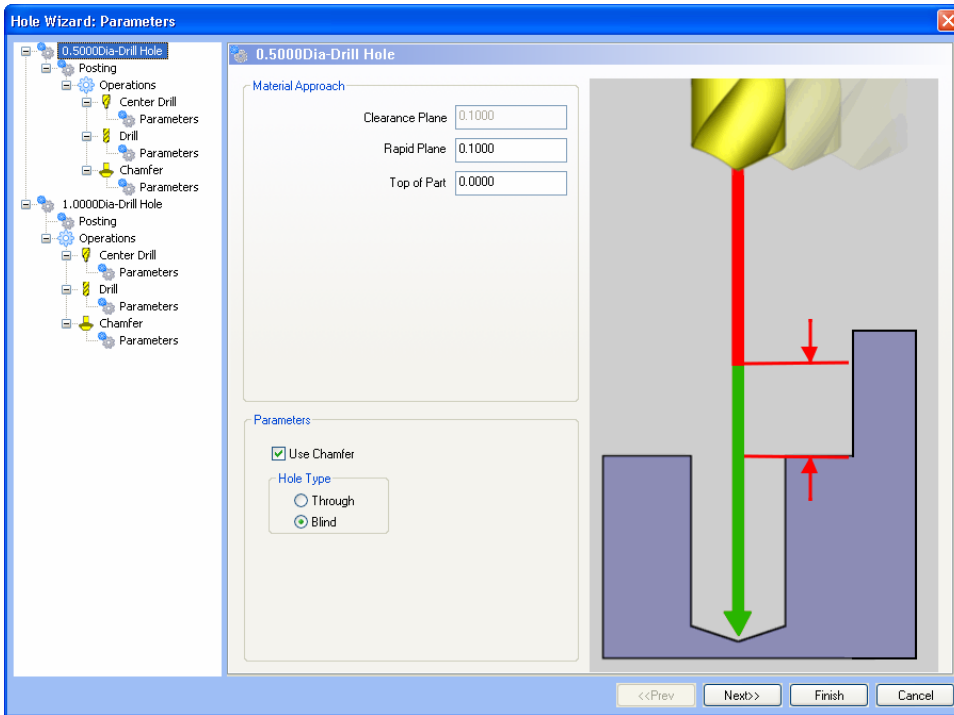
Now we can edit the operations. The wizard is setup to step you through the process of editing the chosen operations to the end and then allow you to compute the toolpath and then lastly, create the NC program.

By looking at the wizard dialog you can see a duplicate tree to the CAM Tree Manager. The wizard allows you to click on an item in the dialog (tree) and edit it. You can skip around or simply make the parameter setting you want and then click the next button all the way down the tree following the wizard.

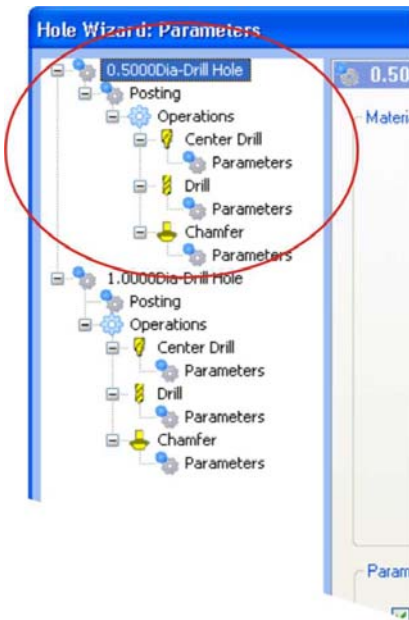
The smaller set of holes is listed in the tree first in this lesson as we selected them first. Let's go through the wizard.

The clearance plane and rapid plane are both set to .1 as the top of the part is at Z0. In the Parameters section of this box we have the diameter at .5 and the depth at .5. These are not through holes so the option will not be checked. You will notice the Use Chamfer Option. By un-checking this option you turn off the chamfer operation and chamfer tool.

The next wizard page is the Parameters Page.



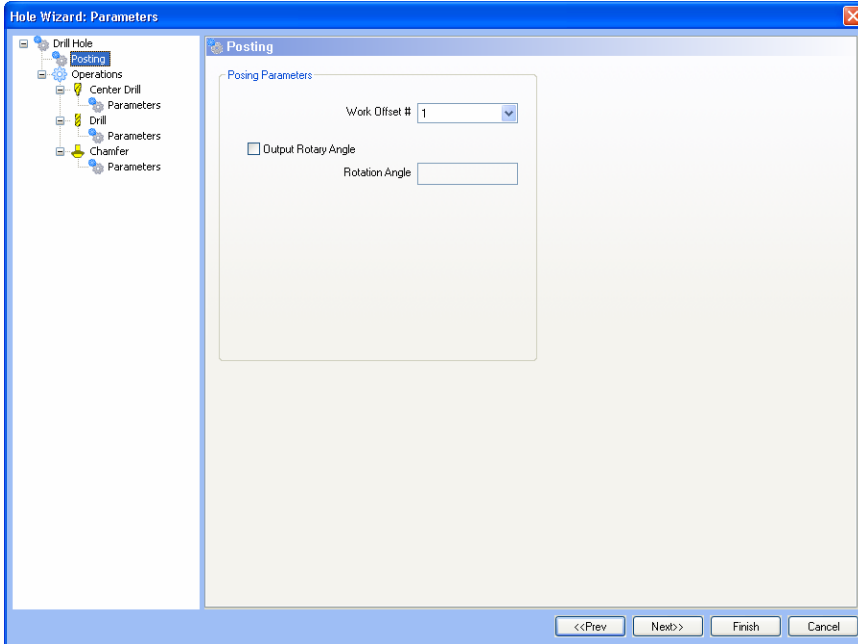
You will notice that the software sorted the holes and populated a tree within the wizard that can be used to select an item in the tree and edit that item. The first operation focuses on the smaller holes.



Here you can check the Clearance Plane, Rapid Plane and Top of Part. You can also indicate if you want to include a chamfer as a part of the operation. If you un-check Chamfer, that part of the operation will disappear from the Tree. You can also choose to create a Through Hole or a Blind Hole. When satisfied with your selections click the **Next** button to advance through the Wizard.

Step 4

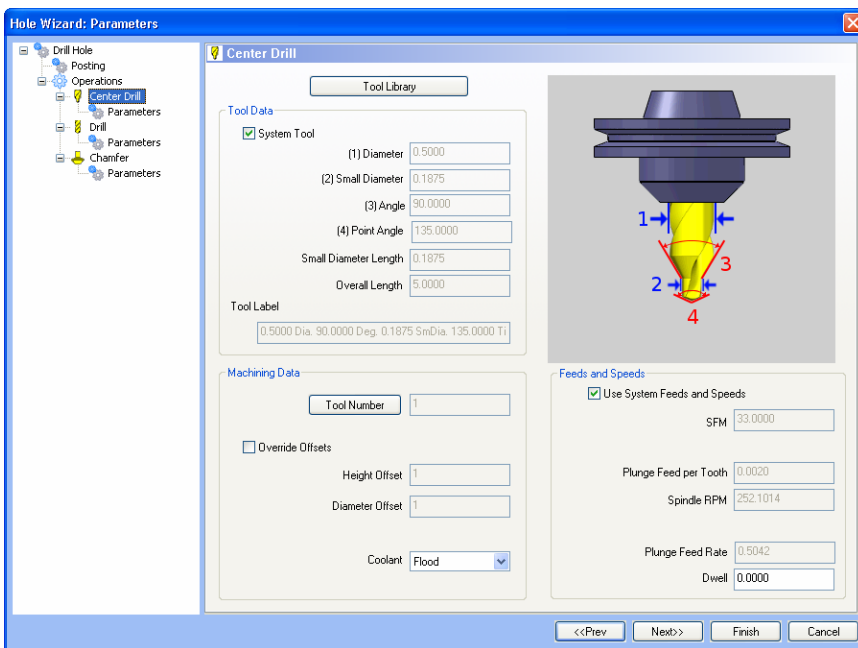
This is the Posting Page of the wizard.



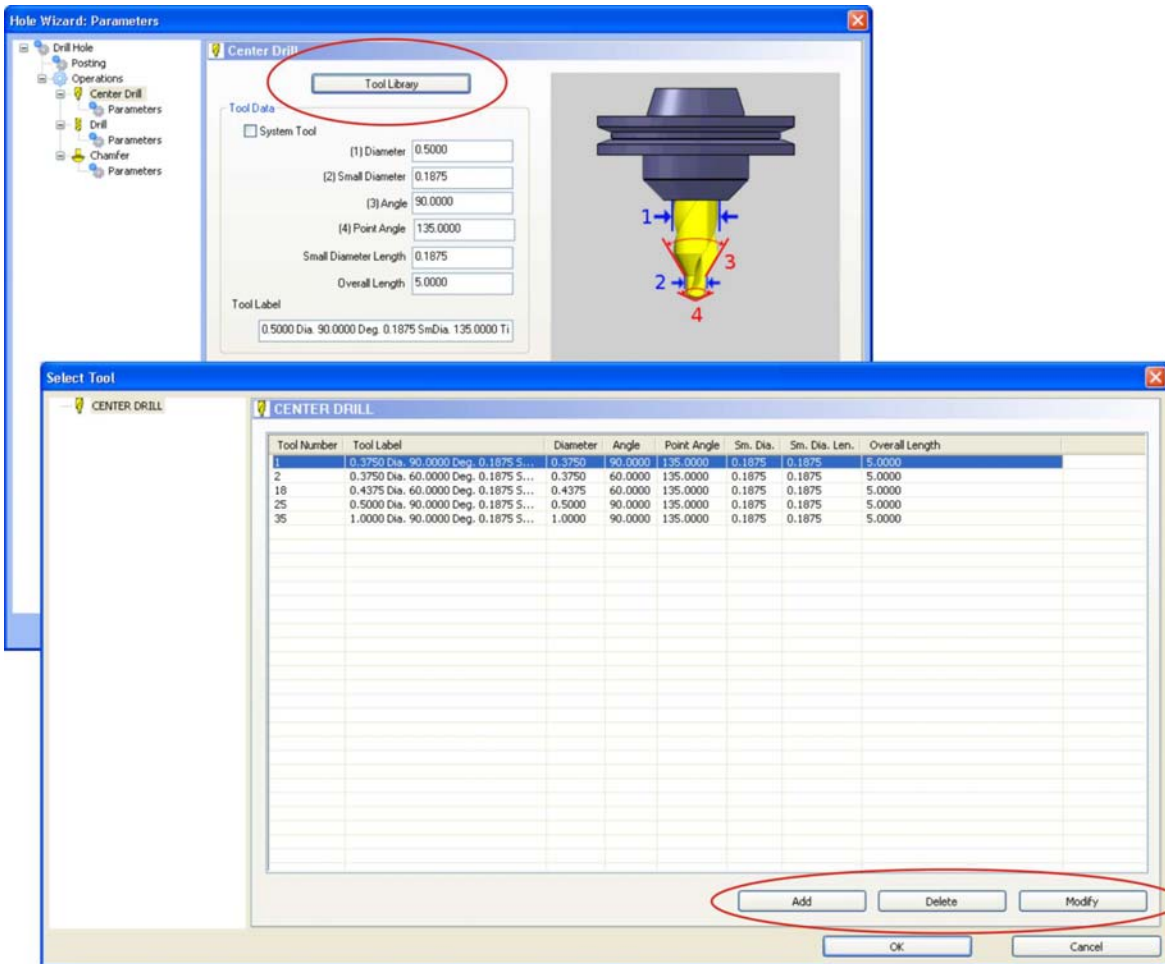
The **Work Offset** field allows the user to choose which work offset code to use for this feature in the posted code. The post processor must be configured to support the work offset chosen. By checking the **Output Rotary Angle** option allows you to input a fixed angle in this field for 4th Axis programming. Then that rotation angle will be included in the posted NC program. Because this is not a 4th Axis lesson, simply click **Next**.

Step 5

The next page of the wizard for this lesson is the Center Drill 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.



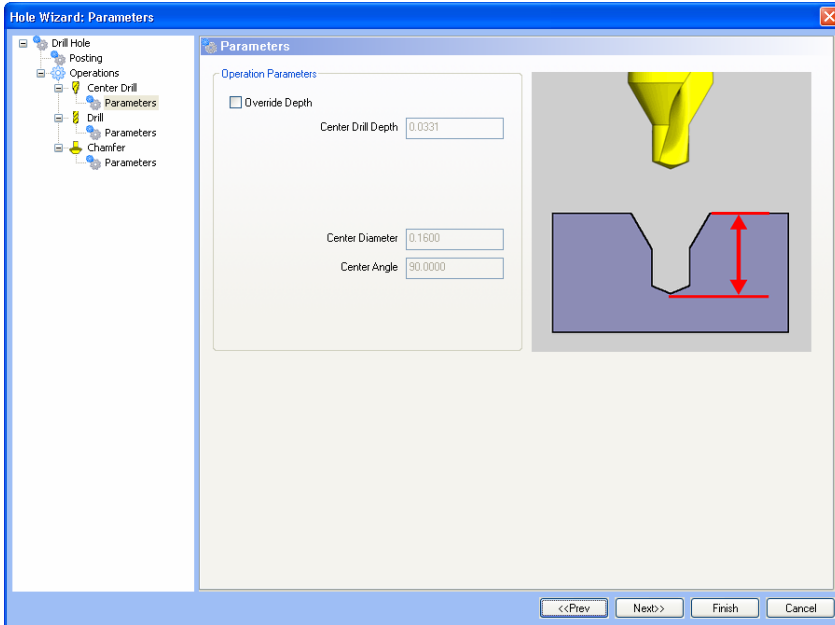
The Select Tool page for Center Drills would appear allowing you to Add, Delete or Modify an existing tool. In any case, this page is divided into 3 sections:

- **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. **NOTE:** V23 post processors will need to be updated to add the variables required for this field to be posted out correctly in V24 programs as it is a new addition to the software. 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. The Dwell value field allows you to specify a length of time for the tool to remain in the material before it retracts.

Click the **Next** button of the wizard to advance.

Step 6

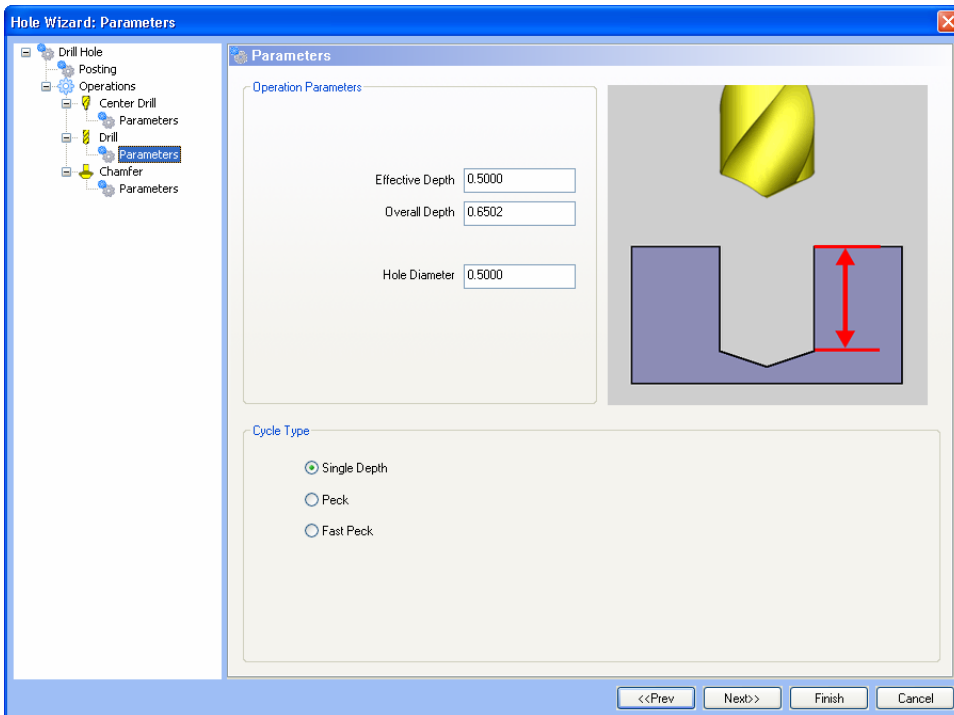
This is the Parameters page for the Center Drill operation.



These values will already be loaded based on the operation. By clicking the **Override Depth** option you will be able to manually change the Center Drill Depth, Center Drill Diameter and Center Angle if you would like.

By clicking the **Next** button you will move on to the Drill page. Click the Next button.

This is the Drill Parameters page.



Here you have the ability to enter an Effective Depth and an Overall depth as well as the Hole Diameter. These values can be changed as needed but will be pre-loaded into these fields based off of the tool you are using. Remember, we are creating a blind hole in this operation, not a through hole. You can also choose a cycle type here and edit the parameters for Peck and Fast Peck.

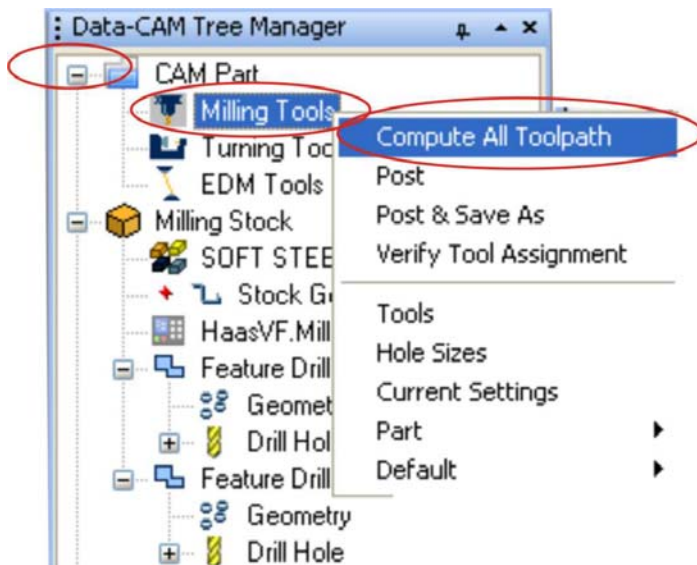
Now click the **Next** button.

Because we chose to create a chamfer operation you can edit the chamfer tool. By clicking **Next** you can modify the Chamfer Depth as needed.

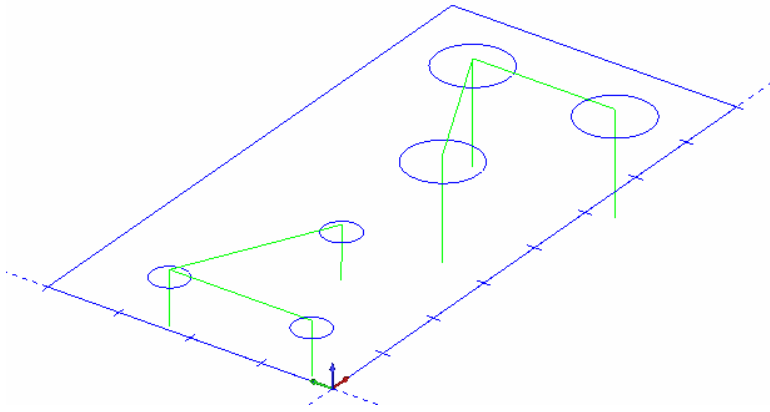
Click **Next** again to advance to the Drill Operation. Click through the wizard checking everything. Add a Chamfer and create a Through Hole this time. Use a Single Depth cycle type. When complete, click the Finish button to close the wizard.

Step 7

Go to CAM Part at the top of the CAM Tree and click the small plus symbol next to access Milling Tools. Now place your cursor on Milling Tools and right-click your mouse to select Compute Toolpaths.



Here is the result:



Step 8

Now we want to post the code. To do this, go back to Milling Tools, right-click your mouse and select Post.

```
[BEGIN PREDATOR NC HEAD  
[MCH_FILE=4AXVMILL.MCH]  
[COORD_SYS 1=X0 Y0 Z0]  
[MTOOL T1 S4 D0.5 C0.1875 J  
[MTOOL T2 S4 D0.5 C0. A118.  
[MTOOL T3 S7 D1.125 C0.125  
[MTOOL T4 S4 D1. C0.1875 A:  
[MTOOL T5 S4 D1. C0. A118. I  
[SBOX X0. Y0. Z-1. L7. W3. H  
[END PREDATOR NC HEADEF  
  
%  
0100 (PROGRAM NUMBER)  
(PROGRAM NAME - HOLE WI  
(POST - HAAS VF)  
(DATE - TUE. 11/23/2010)  
(TIME - 03:36PM)  
  
N01 G00 G17 G40 G49 G80 G  
  
(FIRST CUT - FIRST TOOL)  
(JOB 1 HOLE RANDOM POIN  
(FEATURE DRILL HOLE)  
  
(TOOL #1 0.5 CENTER DRILL)  
N02 T1 M06  
N03 G90 G54 X1. Y1. S252 M  
N04 G43 H1 D1 Z0.1 M08  
N05 G01 G00 M08 M07 G00 G00
```

You have completed this lesson.