AUTODESK UNIVERSITY

MP22244

Programming All Your CNC Machines - FeatureCAM

Aaron Williford
Autodesk

Learning Objectives

- Learn how to use FeatureCAM software for a range of CNC machines – including 3-axis to 5-axis mills, simple lathes to multitasking turn-mill centers, wire EDM, and more
- Learn how to program like features consistent across a wide range of machine types
- Learn how to easily adapt existing programs for use on other machines
- Learn how to optimize your programs to your machines

Description

Learn about a wide variety of machines available today, from simple 3-axis mills to complex 5-axis machining centers, and from simple lathes to multitasking turn-mill centers, wire electrical discharge machining (EDM), and more. See how the world's leading feature-based CAM software handles all of these machine types. This session features Autodesk FeatureCAM.

Your AU Expert(s)

I am a Technical Consulting Manager for North America, I currently look after all training and support for North America for PowerMill, FeatureCAM, PartMaker, PowerShape, PowerInspect, and ArtCAM. I joined Autodesk through the Delcam acquisition in 2014 having joined Delcam as an Applications Engineer in 2012. I have a history that includes work in manufacturing as a machinist, programmer, and a technical instructor, as well as a business management background. My product expertise is FeatureCAM and PowerMill with practical experience programing Aerospace and industrial application products for CNC machining.

CNC machine types available

There are a wide range of CNC machine types available today to help production, job, mold, and specialty shops complete their programming and machining in a timely efficient manner. Machines available today include:

3-axis mills

3-axis mills are one of the most common machine tools found in industry. Most 3-axis mills are vertical mills with 3 axes that can move all 3 axes simultaneously – commonly X, Y, and Z axis.

Type of Work

You will find a wind range of industries using 3-axis mills, from aerospace machining, to mold making, to automotive machining, and even general machine work.

4-axis mills

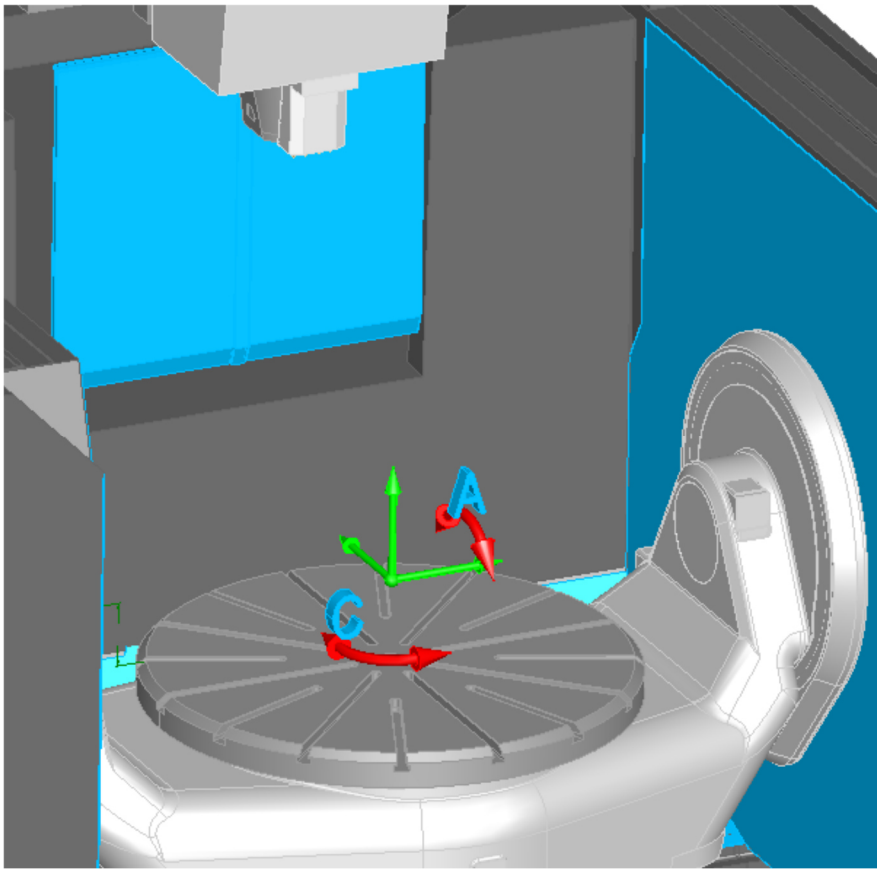
4-axis mills can be either purpose-built as 4-axis, or have an add-on/bolt-on 4-axis indexing attachment. Indexing can be done about one axis; often times vertical mills with 4-axis will either index around X or Z while horizontal milling machines will typically index around Y axis. Many 4 axis machines are capable of simultaneous indexing movement as well.

5-axis mills

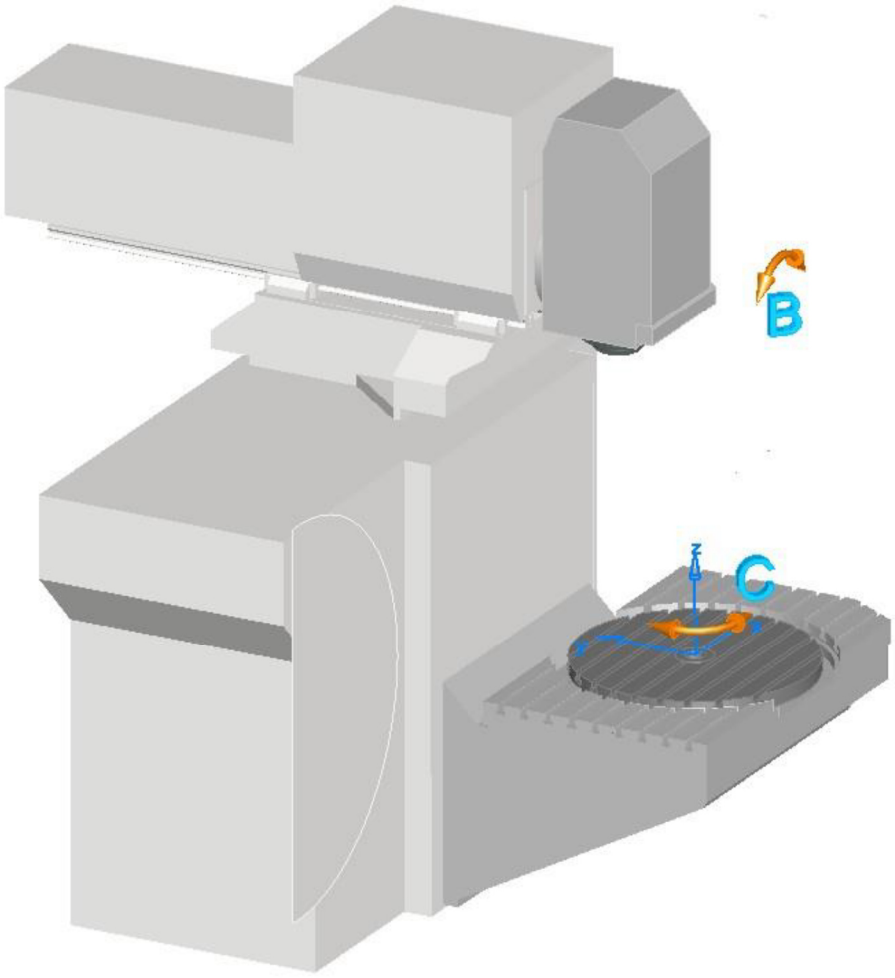
5-axis milling is one of the most complex milling types available, simply because the machines have 5 axes that can move simultaneously. These axes are typically X, Y, and Z (similar to a 3-axis) but then they add two rotary axes. There are several configuration of 5-axis machines, as such the naming convention for the two additional rotary axes changes with each configuration.

5-axis mill configurations

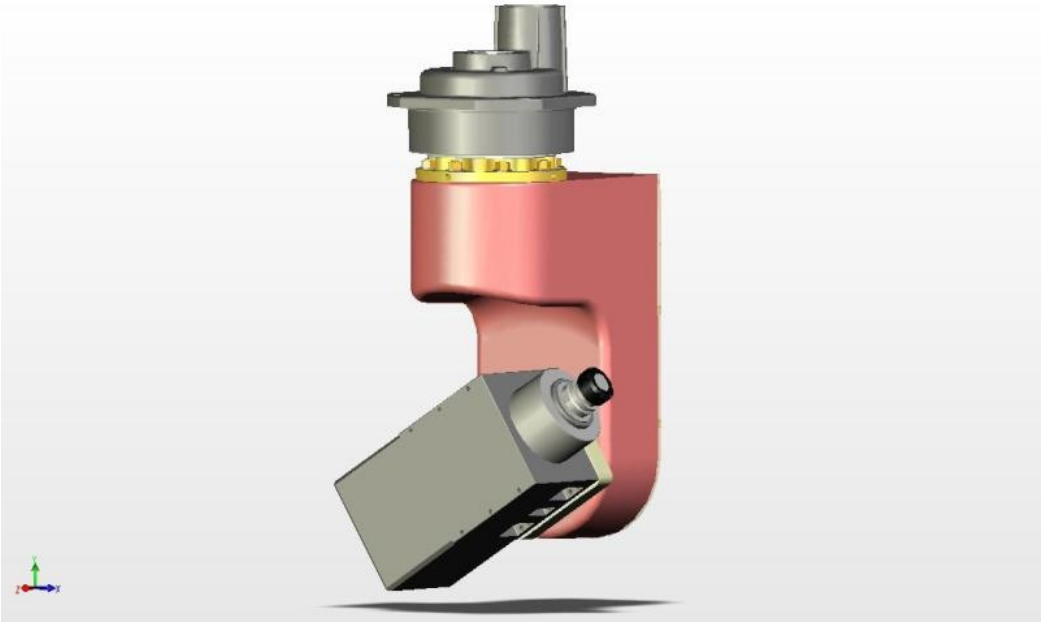
- Table-Table configurations are designed so that a single rotary table is stacked on top of another rotary table (often on top of a trunnion).



- Head-Table configurations are designed so that the table of the milling machine has a single rotary table (typically a C axis) while the head (spindle) has the ability to rotate to represent either vertical milling, horizontal milling, or in many cases anywhere in between.



- Head-Head configurations are designed so that the table is a standard milling table while the head has a double knuckle giving it the ability to rotate and tilt.



Turning centers

Turning centers are typically referred to as lathes. Lathes, in their simplest form, use a stationary (non-rotating) cutter while spinning the workpiece to remove material. Typically you will see profile turning (both inside and outside), drilling on center, tapping on center, threading (both inside and outside), and grooving operations performed on turning centers. It is important to note that the workpiece is usually cylindrical in nature, although not always. Turning centers operate on 2 axes, X and Z.

Turn/Mill machines

A turn/mill machine is still a lathe, however it has the added benefit of a live tool, allowing for other milling type work to be completed on the machine. A turn/mill machine can have an added axis over a standard turning center, the Y axis can allow for even more flexibility since it allows for some off-center milling to be completed. In addition to these elements some turn/mill machines have a B axis head instead of a standard turning turret, allowing for 5-axis turning and milling work to be completed on these machines.

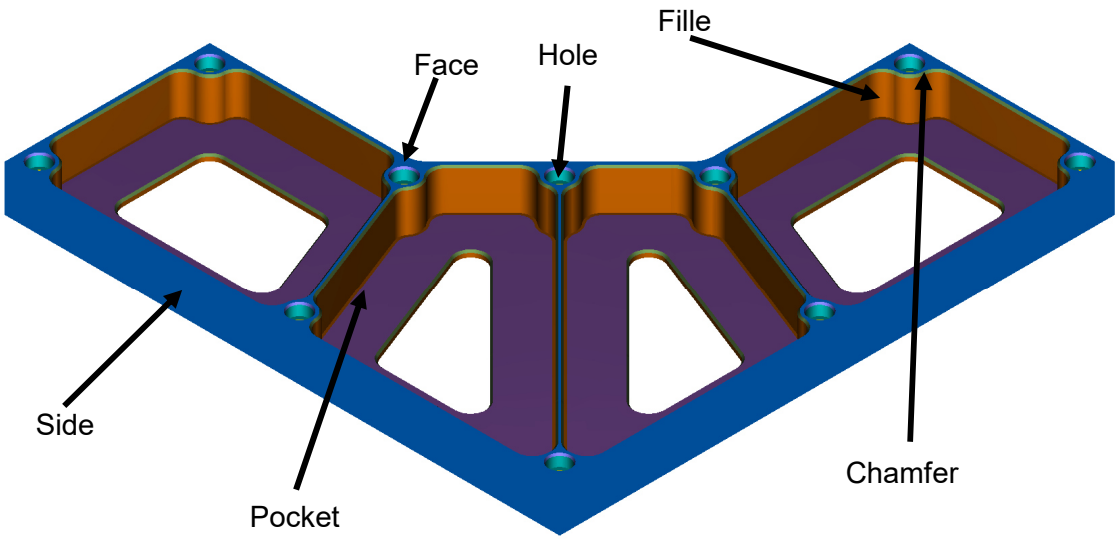
Wire Electrical Discharge Machining machines

Wire Electrical Discharge Machining is typically referred to as wire EDM. The basic idea is that a thin wire with an electric current running through it is used to ‘machine’. The process of wire EDM is typically slower than traditional machining techniques, but it does allow for near zero radius’ on internal cuts (which typical milling and turning machines cannot do) and Wire EDM is capable of cutting very hard metals that would be near impossible to cut using traditional machining techniques.

What is a Feature and why use them in CAM

If we look at a part such as the one shown below and consider how we would manufacture it we can see that it can be broken down into a series of Machining Operations. For example, we

would probably want to start by Facing off the top of the part, then machine around the two raised areas down to the next flat level. The outside area of the part would need to be roughed out and then the Side of the part finished. There are then a series of Pockets which would need to be rough and finish machined and finally we would drill the Holes and Chamfers to the part. Each of these areas that we talked about on this part, Face, Side, Pocket, Hole, Chamfer, and so on are a Feature of the part.



How does FeatureCAM use features?

FeatureCAM defines a part in the terms of Features, using common shop terms; for instance, an area that looks like a pocket can be machined using a *Pocket Feature*. When you define an area as a pocket, FeatureCAM will then automatically decide what operations to use to machine that area. All you have to do is say “This is a Pocket” and FeatureCAM will select suitable roughing and finishing operations, tooling, speeds and feeds. You are then free to modify these default values to suit your particular situation. By automating the generation of operations, the overall time taken to program a part can be drastically reduced.

In addition to using features as its base programming style, FeatureCAM is also wizard based so it will guide you through the process of programming parts using wizards that ask you questions.

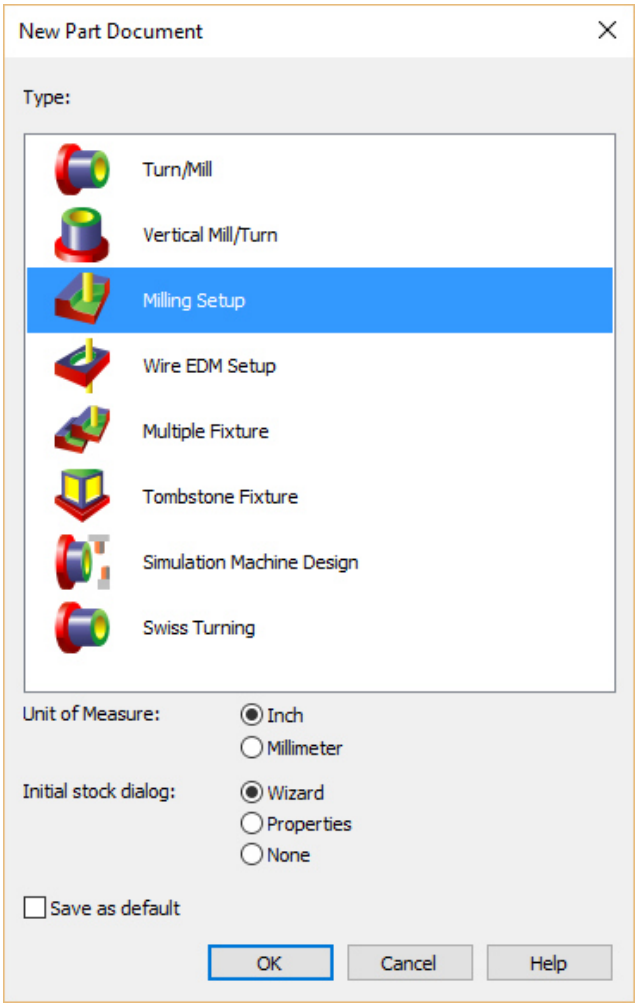
Programming on different machine types

Because FeatureCAM uses a Feature-based machining approach in which you define the end result (a feature) and the customizable settings determine everything else you need to get to that end result. This approach makes programming across a wide variety of machines easier. You will notice that interface and programming style stays consistent no matter what type of machine is being programmed.

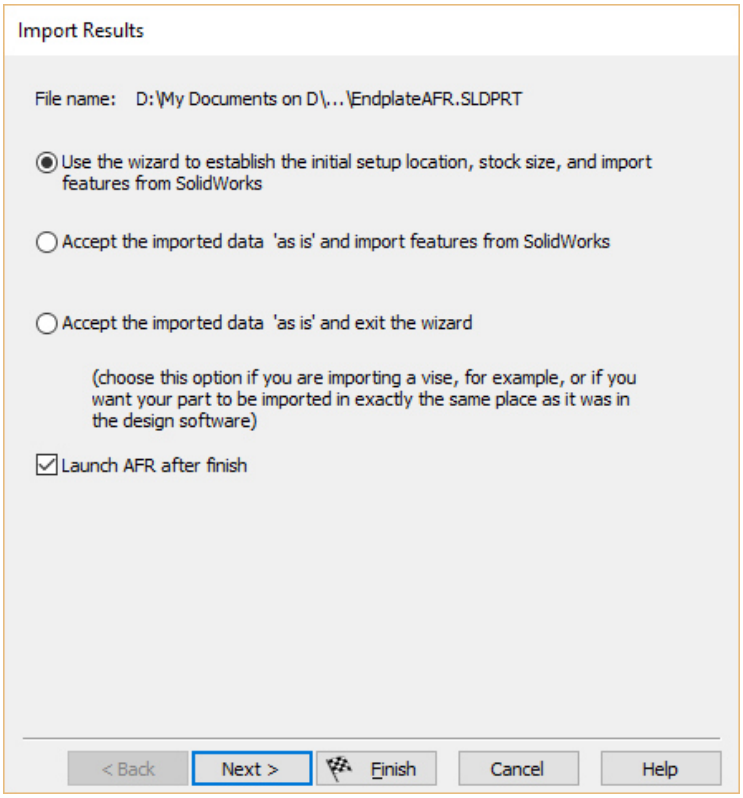
3 axis Mills

Automatic Feature recognition

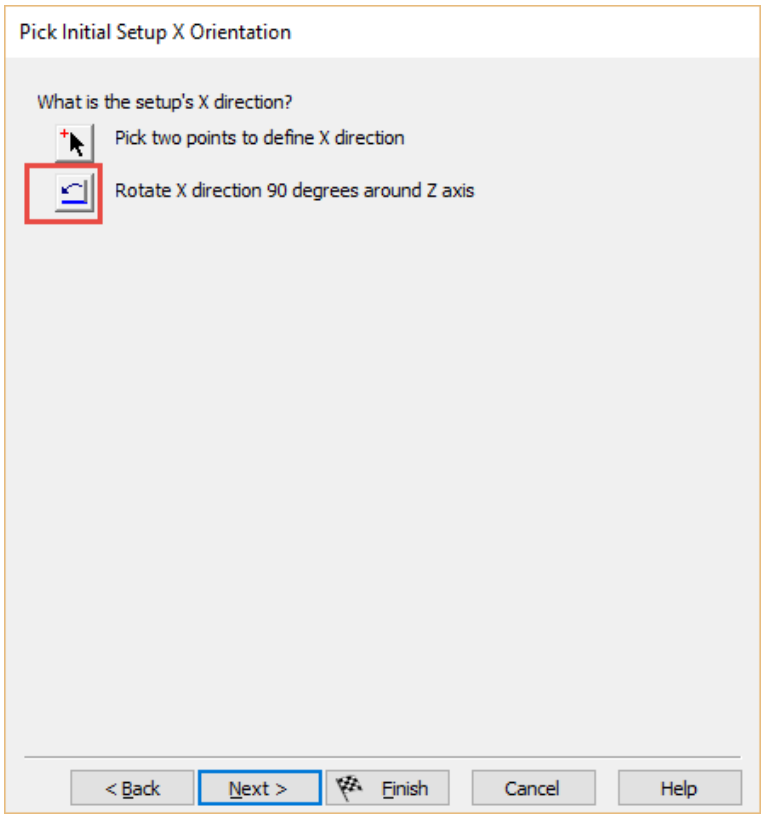
1. Import the part **AFR** into a new milling document



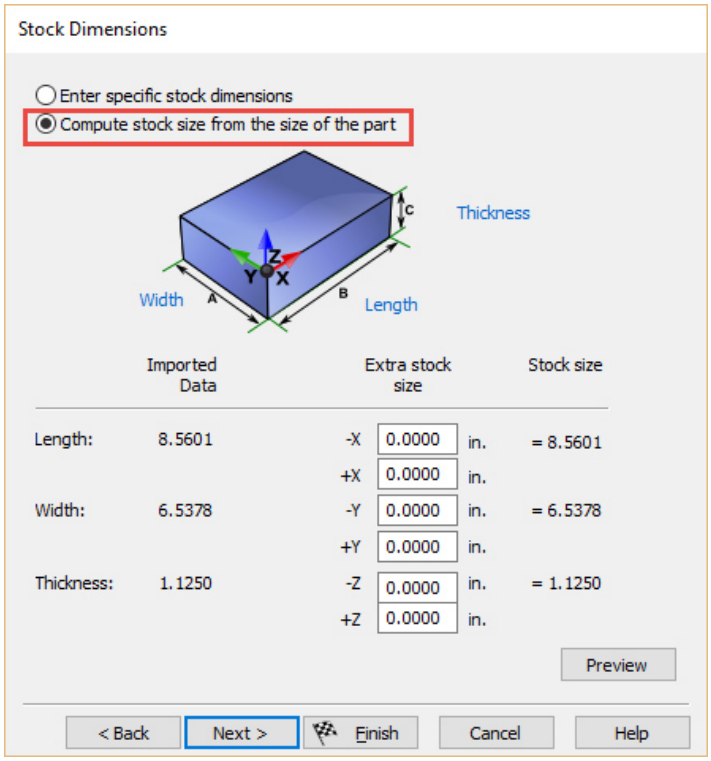
2. Use the Wizard to establish the initial setup location, and stock size. And select the checkbox **Launch AFR after finish**



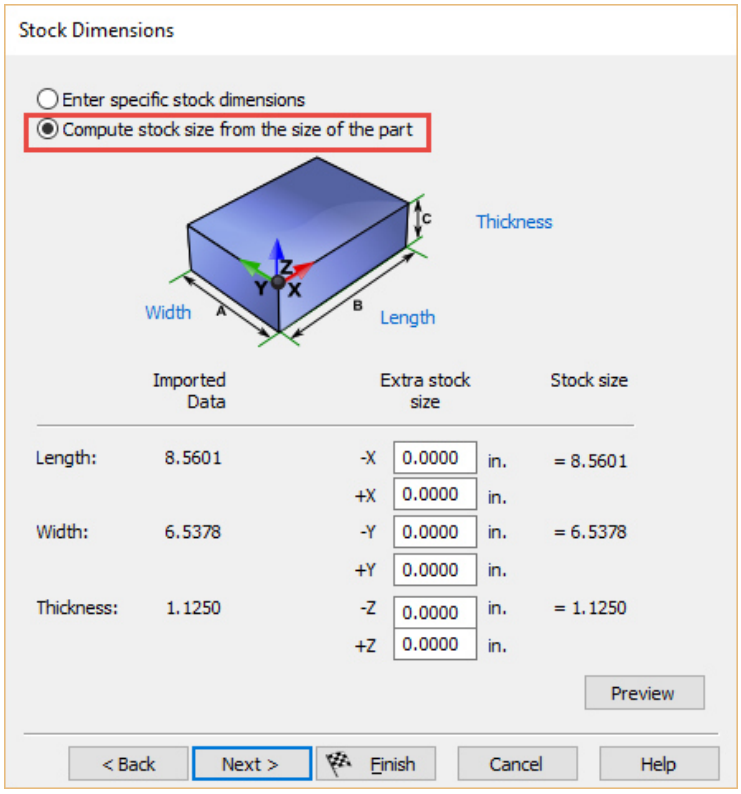
3. Press **Next** twice
4. Select **Rotate X direction 90 degrees around Z axis** 3 times, then press **Next**



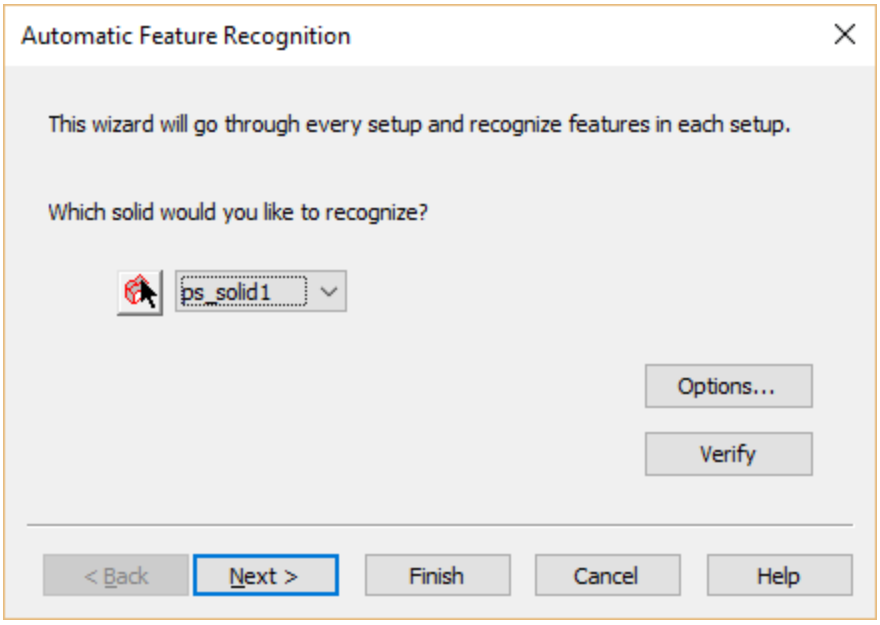
5. Press **Next** again, and select **Computer stock size from the size of the part,** then press **Next**



6. Select **UL** to set your work offset (G54) to the Upper Left corner of the top of the stock, then press **Next**



7. **No** to multi-axis positioning, **Next, Finish**
8. On the Automatic Feature Recognition wizard, select **Next**



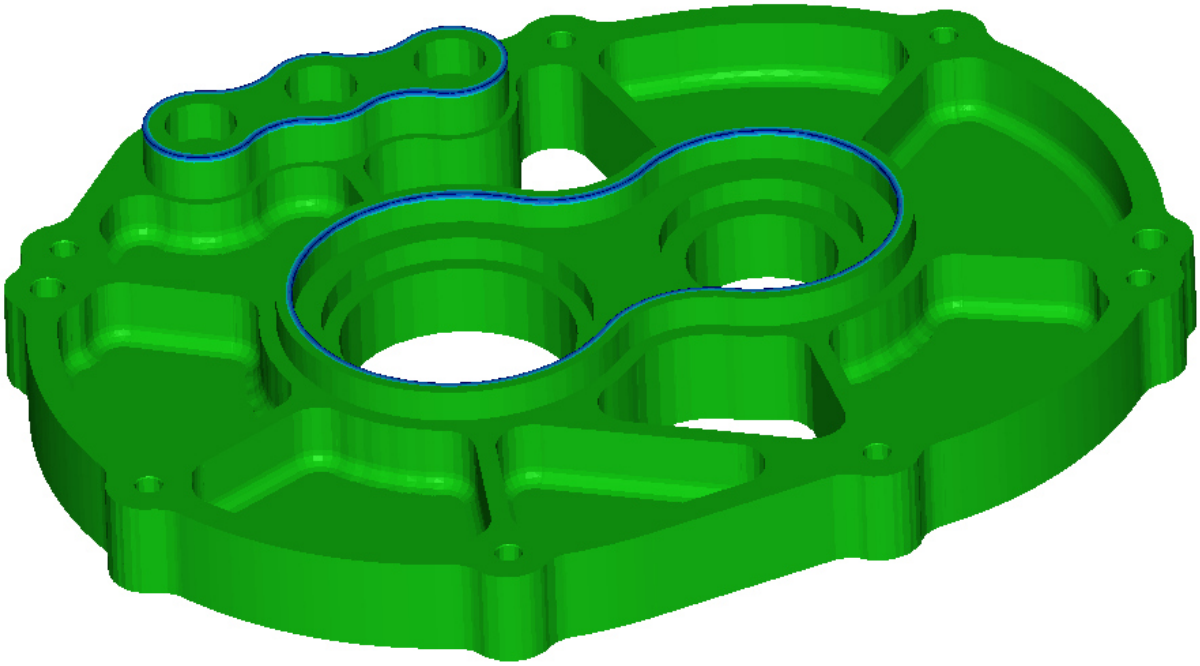
9. **Next,** then **Finish**
This part is programmed and ready to simulation

10. Select **3D Simulation** and **Play** from the Simulation toolbar

Simulation in FeatureCAM is what actually produces the toolpath and generates the NC code, so while the part was programmed we had not actually produced the toolpath until that was simulated. As you simulate this, you will notice that FeatureCAM has defined all of the necessary machining parameters for us, based on the definitions of the features that the solid model gave it. The program is safe and you could cut a good part with it

Optimizing for 3 Axis Mills

As you watch the simulation from the part that we ran Automatic Feature Recognition on, you will see that there are some things done that you may want to do different. Automatic Feature Recognition is not an be-all-end-all solution, it will program parts like this well enough that you could use the program to cut them. To begin with if you use **Part Compare** you will see that AFR did not program the chamfers around the boss with the three holes in it, or around the inside of the figure 8 pocket on the top.



In addition, when we are facing the part, we are not removing any material, this is the case because we did not leave any material on the part when we defined the stock size yet we still had AFR create a face feature. You may have also noticed that many of the pockets are done in

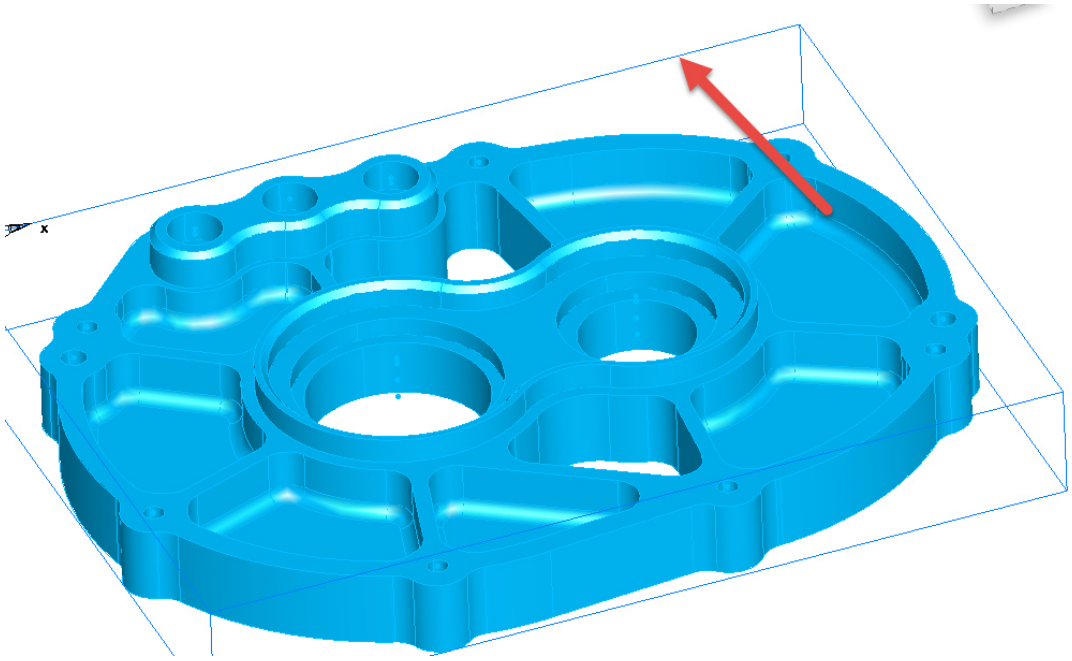
multiple depths, rather than just cutting the pocket all the way to the desire depth before moving on to the next pocket. All of these items can be modified by programming this part in a different way – using Interactive Feature Recognition.

Automatic Feature Recognition Vs. Interactive Feature Recognition

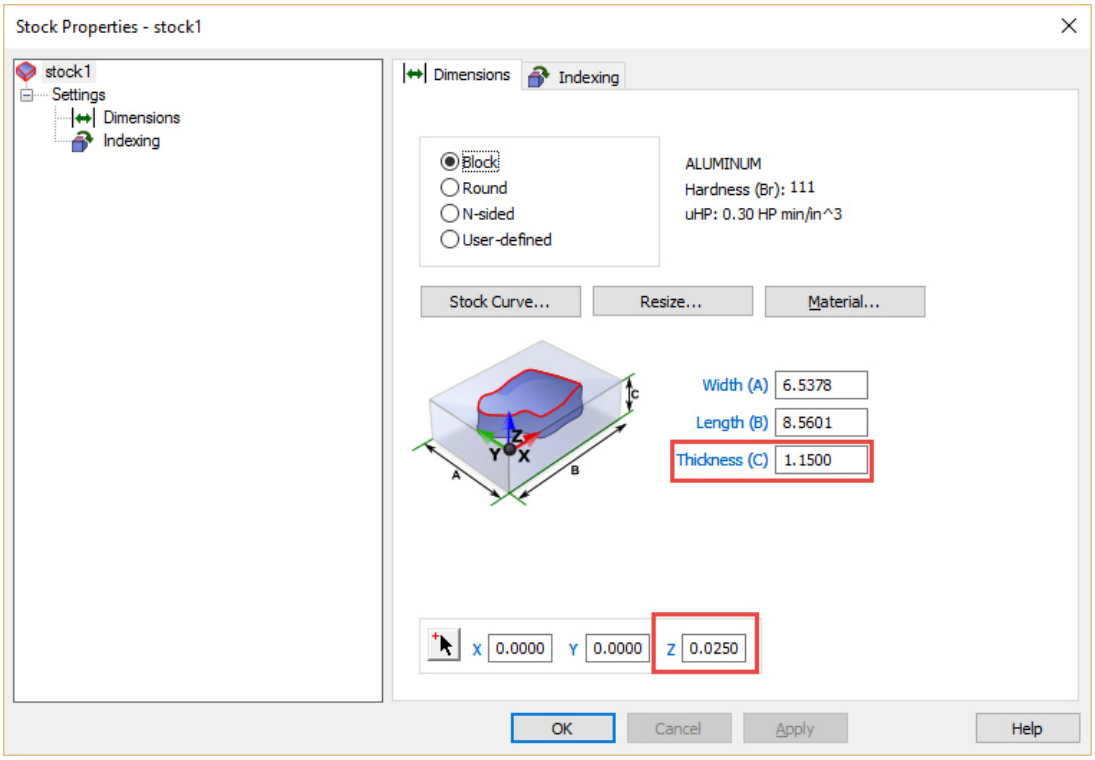
1. If you are still in simulation, Eject  the simulation
2. Press **Undo**  on the Main toolbar

You are now working from a clean slate again, no programming has been done.

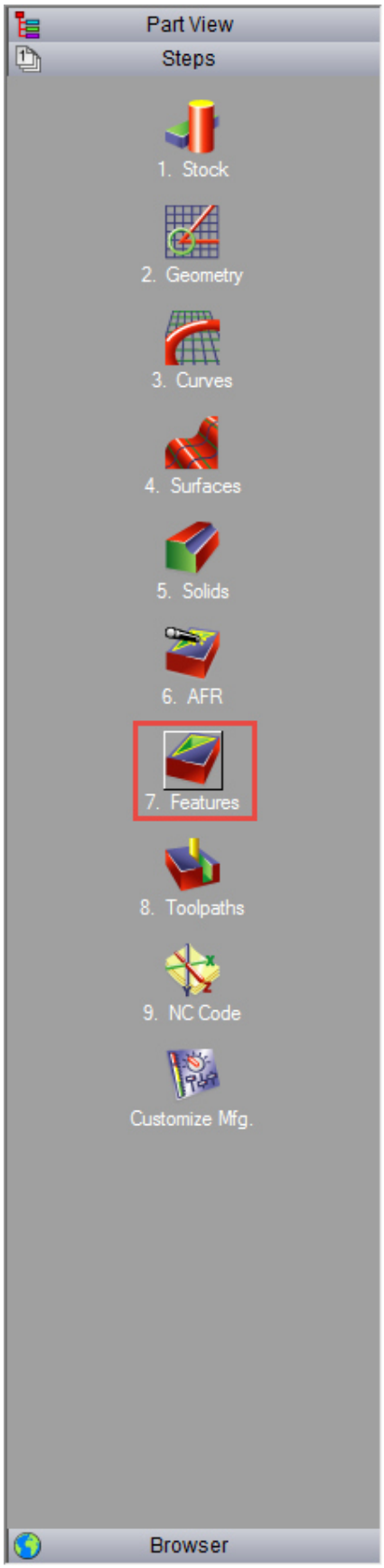
3. To add material to the top of the part for our facing operation, Double-click on the **Stock** in the graphics window



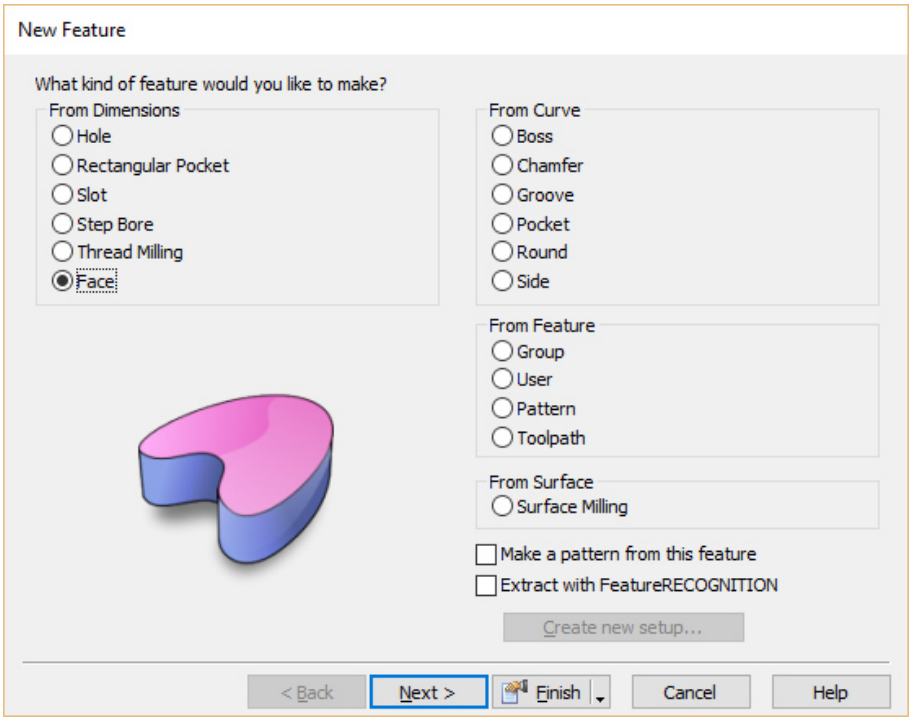
4. Fill in the **Stock Properties** as shown, and press **OK**



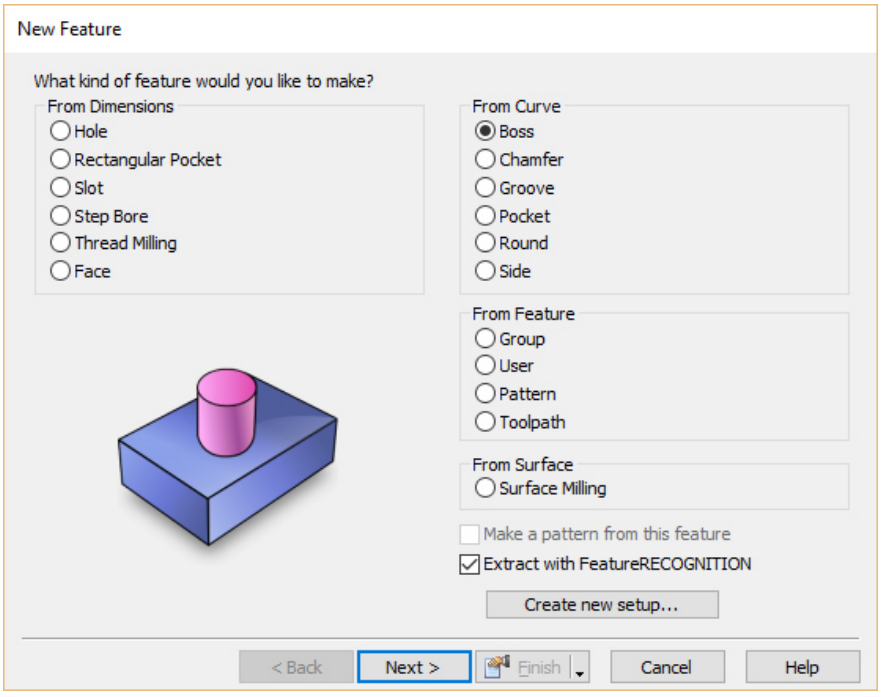
5. From the **Steps** menu select **Features**


Part View
Steps
1. Stock
2. Geometry
3. Curves
4. Surfaces
5. Solids
6. AFR
7. Features
8. Toolpaths
9. NC Code
Customize Mfg.
Browser

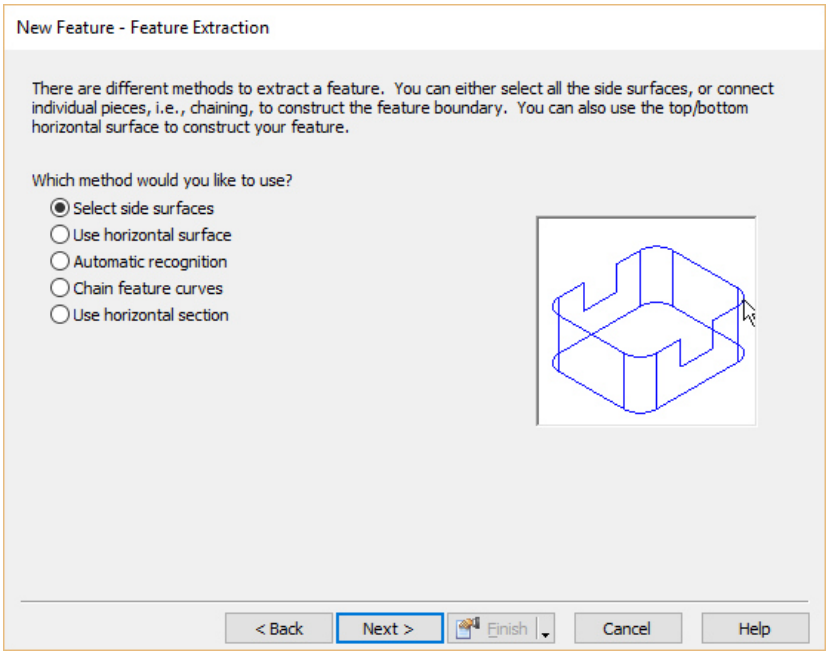
6. Select **Face,** then **Finish.** Press **OK**



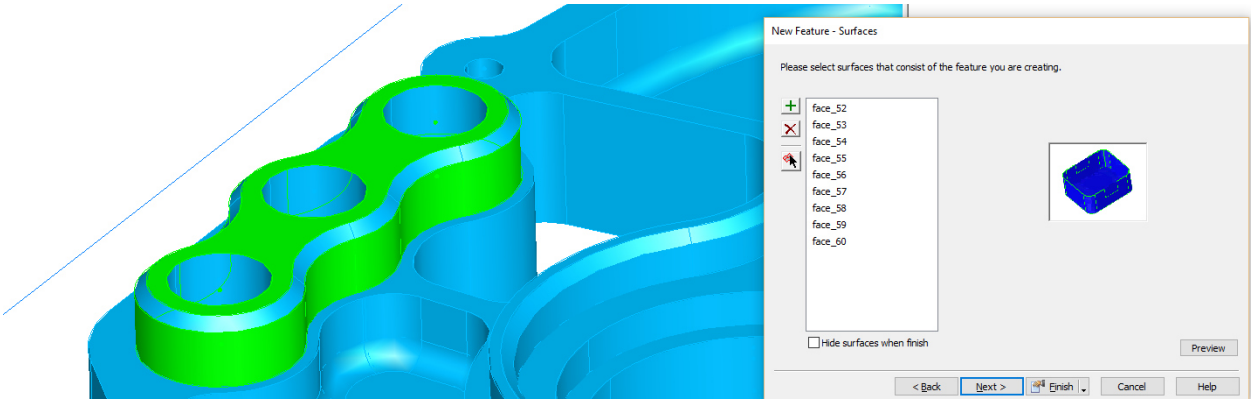
7. Select **Features** again from the steps menu
8. Select **Boss** and check the box named "**Extract with FeatureRECOGNITION",** then **Next**



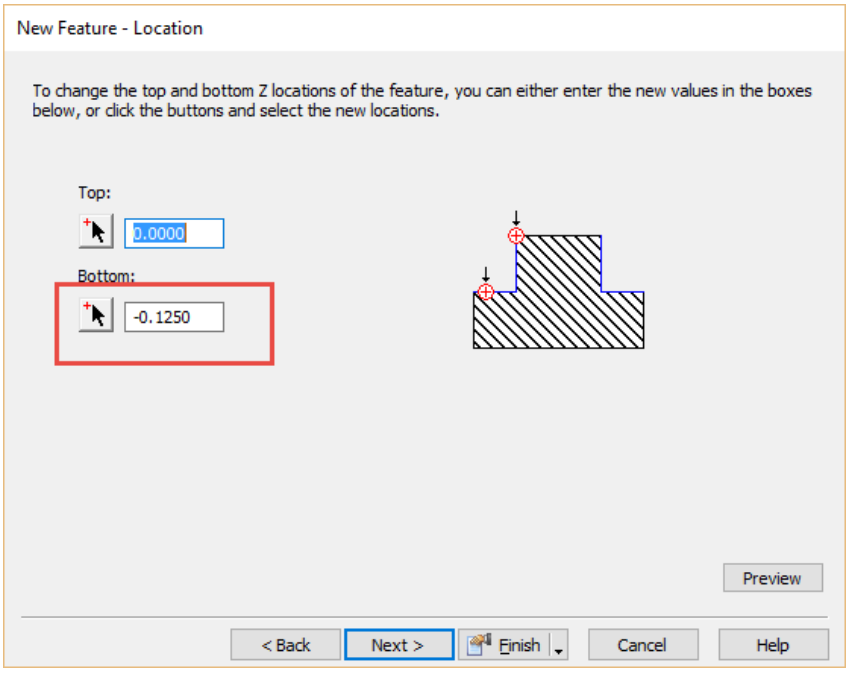
9. Choose **Select side surfaces**



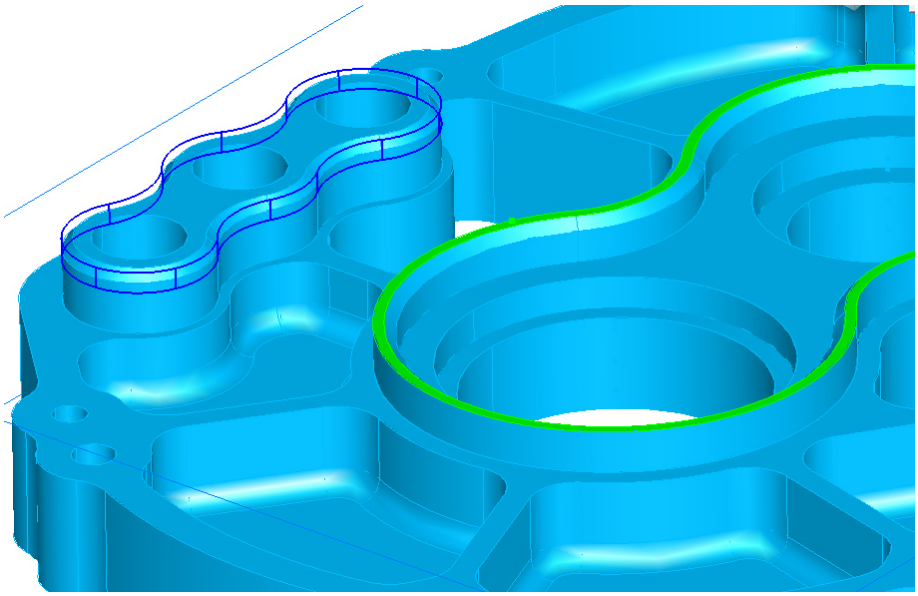
10. Once you are on the next page, then click in the graphics window to select the sides that make up the **boss** with the three holes in it, then click the green  to add the sides into the list



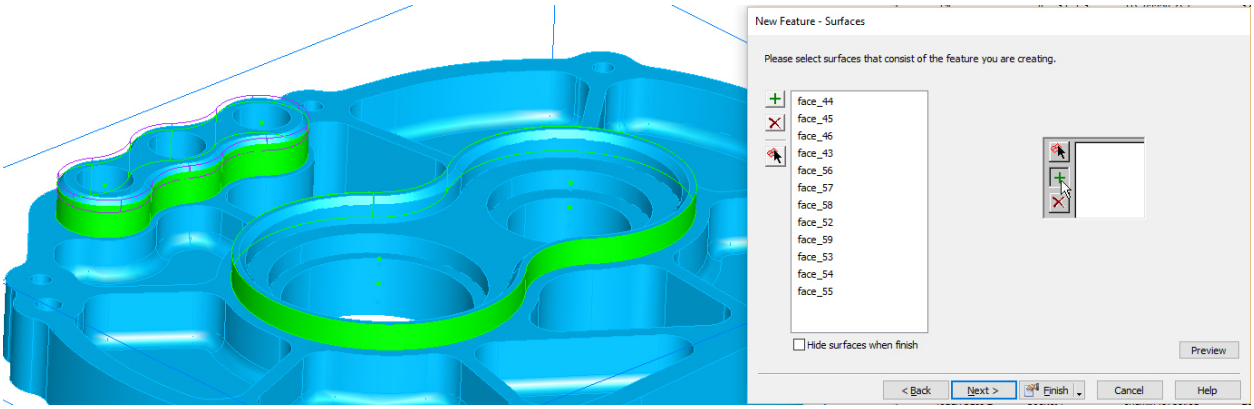
11. Click **Next**
12. **Pick Z location** for the bottom of the feature



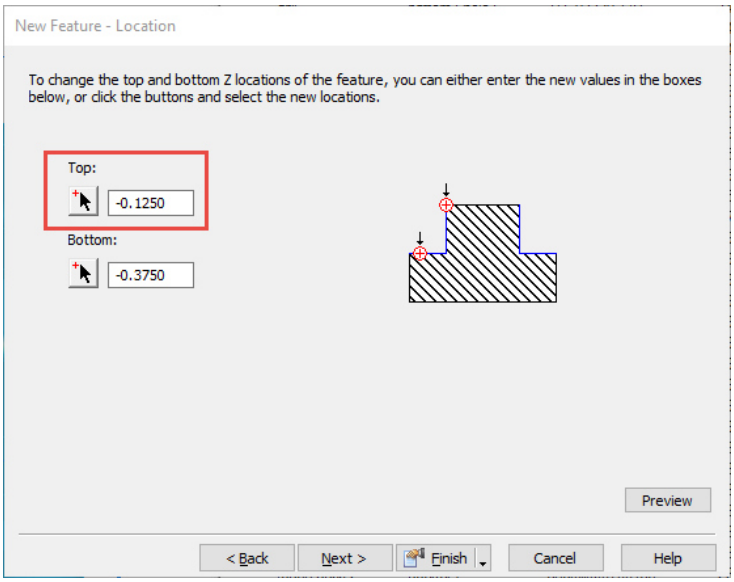
13. Select the top face of the “figure 8” pocket/boss as shown



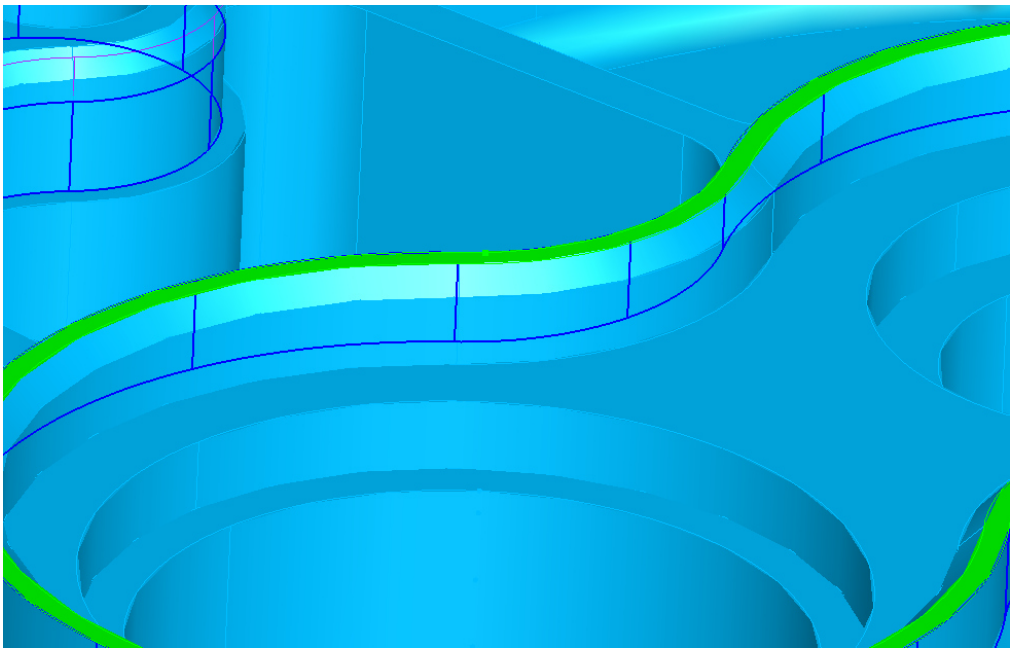
14. Click **Finish** and **OK**
15. Repeat the process above from step 7, but this time select the sides of the figure 8 boss shape as well as the sides of the boss with the three holes in it



16. Select **Next** then **Pick Z location** for the top



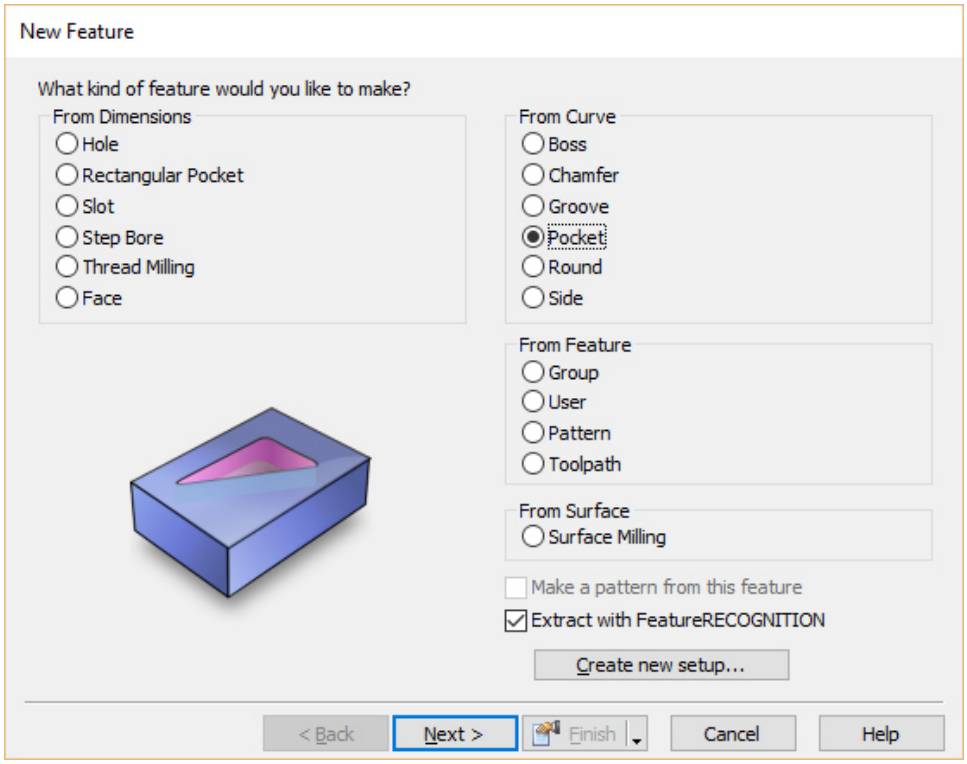
17. Select the top of the figure 8 boss/bore as shown



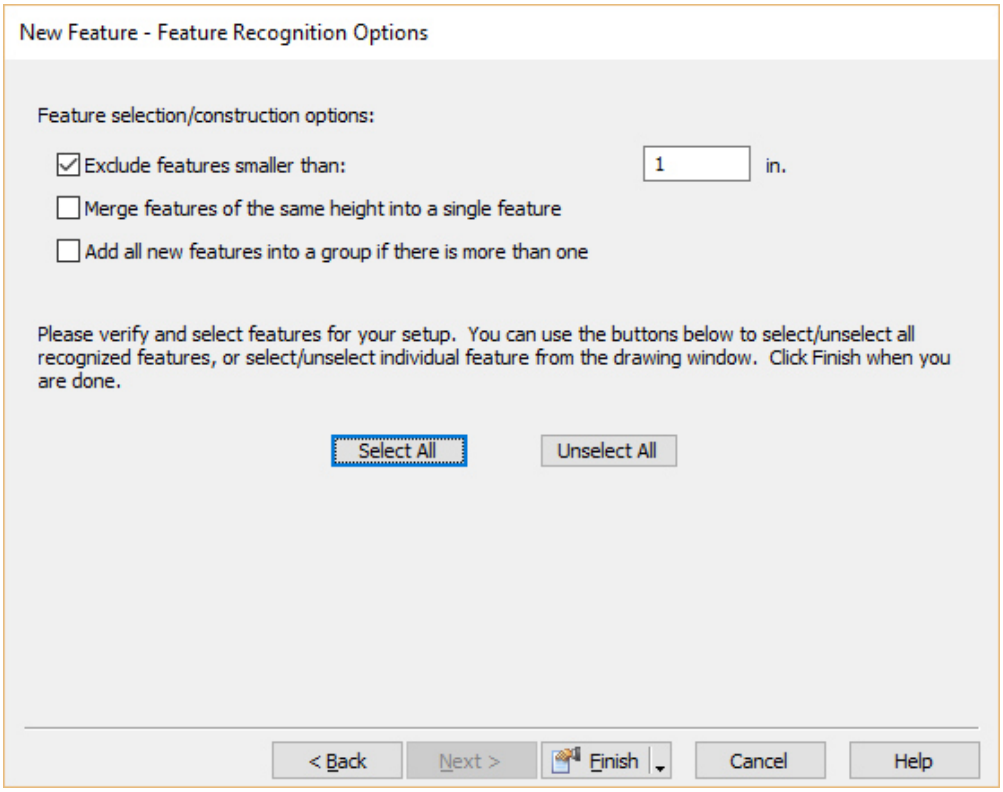
18. Select **Finish** and **OK**

Run a 3D simulation if you would like to see the results of your programming.

19. Select **Features** from the steps menu again, then select **Pocket** and ensure **Extract with FeatureRECOGNITION** is still checked



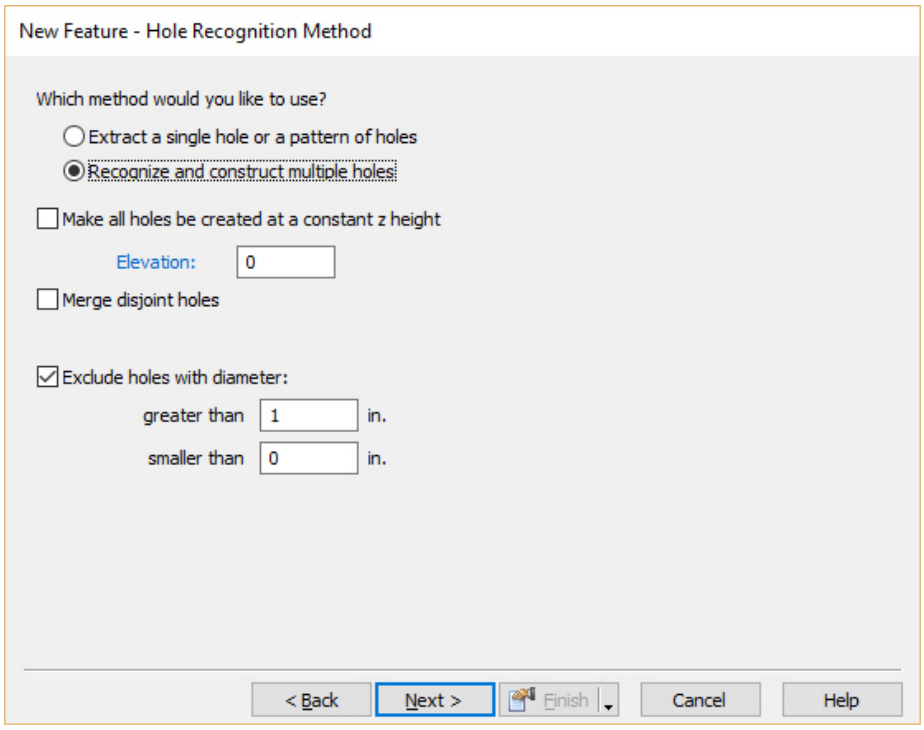
20. Select **Automatic,** then **Next**
21. Fill out the **Feature Recognition Options** dialog as shown and press **Select All**



22. **Finish**

The majority of the part is already programmed, we have just a few features left. 3D simulate if you wish to see the progress.

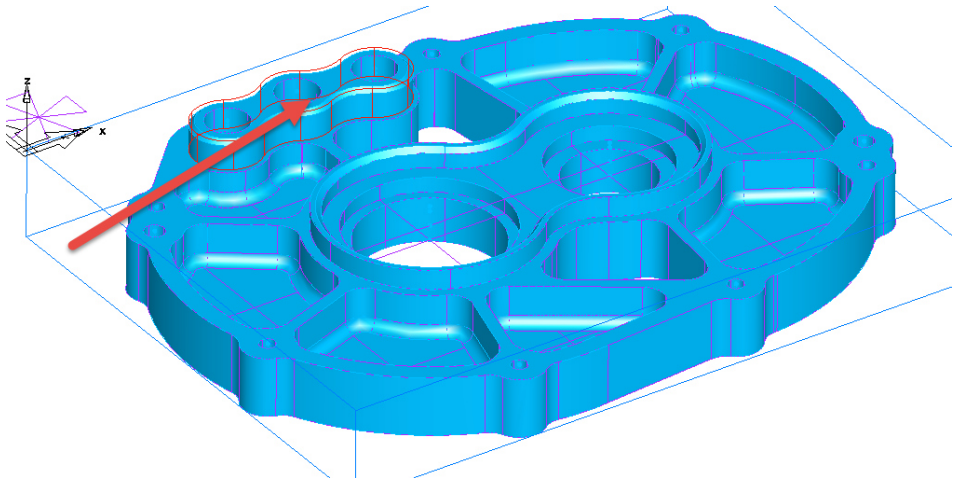
23. Create a new **Side** feature, extracting it with FeatureRECOGNITION
24. Select the **Automatic** method
25. **Select All**, and **Finish**
26. Create a new **Hole** feature, extracting it with FeatureRECOGNITION
27. Select **Recognize and construct multiple holes,** and check the box **Exclude holes with diameter:** and fill in the **greater than** box with **1**



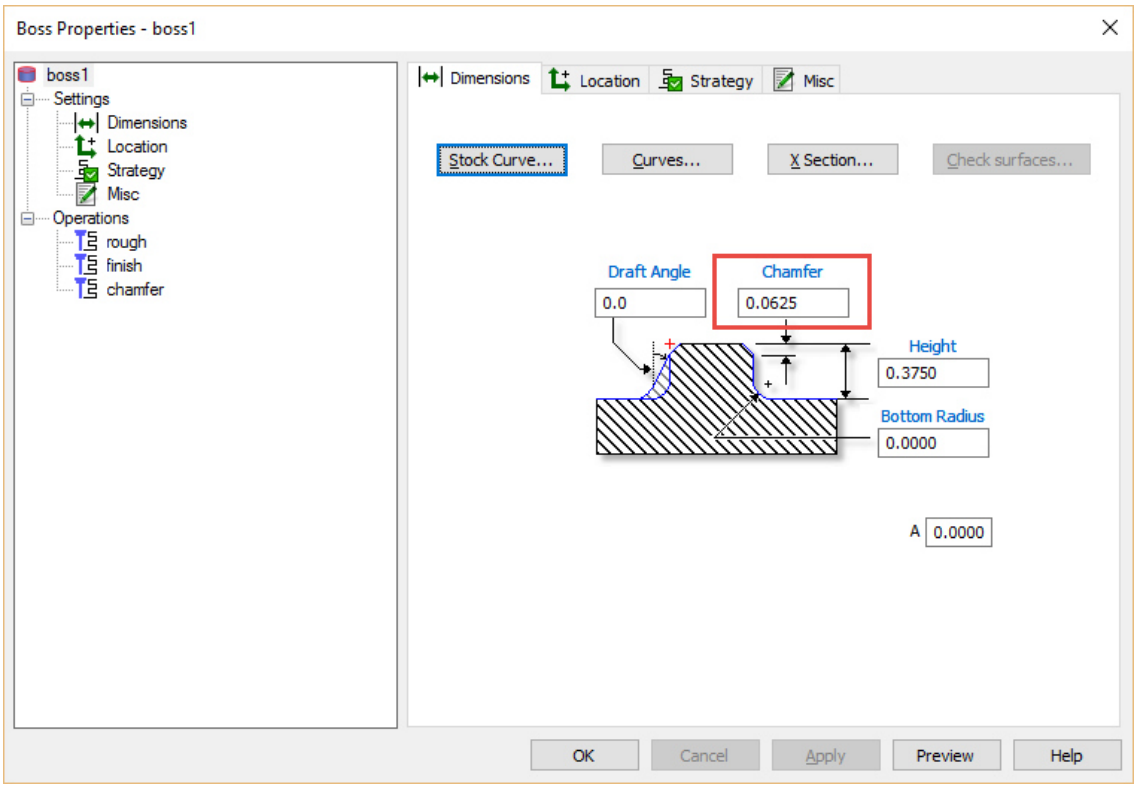
28. **Select All,** and **Finish**

We now need to make a minor modification to two features to include the chamfers that AFR missed

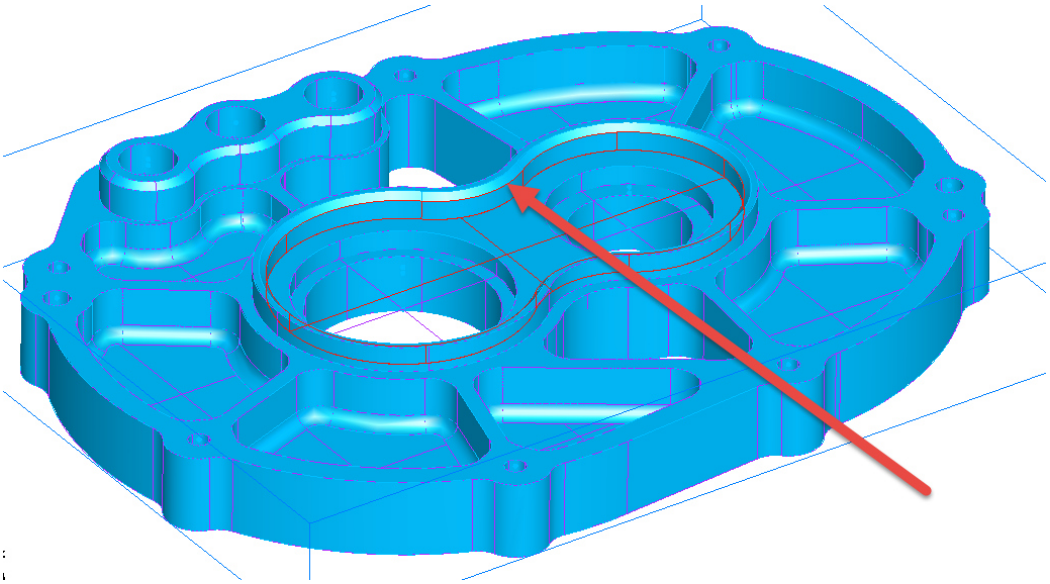
29. Double-click on the boss feature that has the 3 holes in it from the graphics window



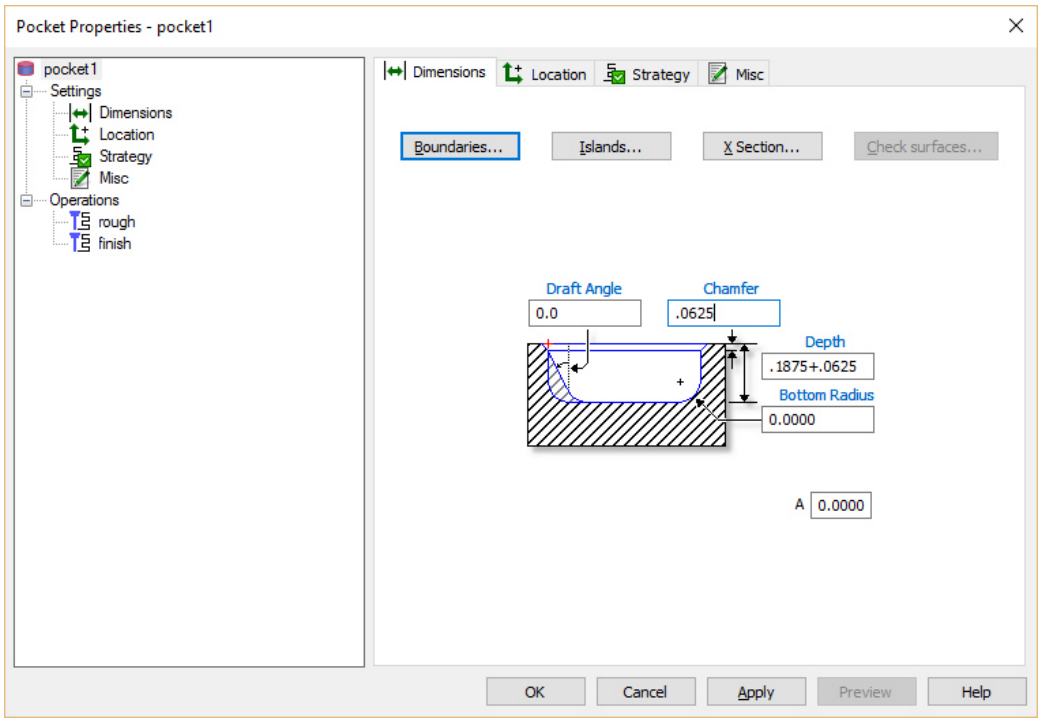
30. Add a chamfer value of .0625, then press **OK**



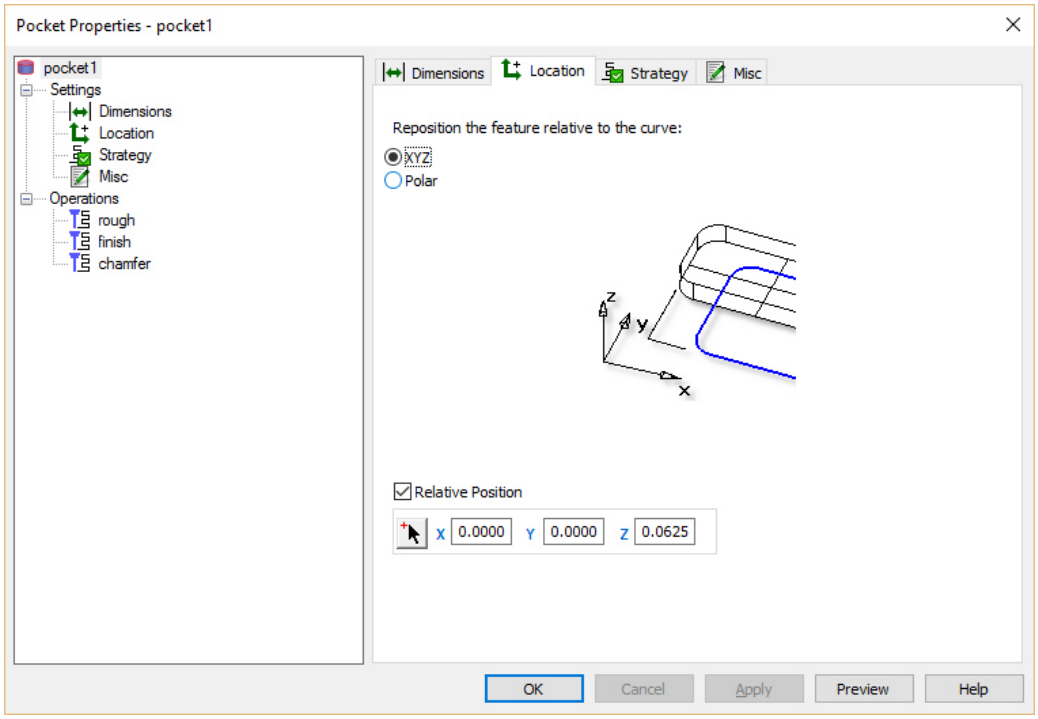
31. Double-click on the figure 8 pocket feature from the graphics window



32. Add the value .0625 to the depth of the feature
33. Enter .0625 value in for the **Chamfer**



34. Select the **Location** tab and change the Z location to .0625



35. Click **OK**
36. Run a **3D simulation**

Notice that the part is programmed in a much more efficient way now and we are fully cutting all of the features.

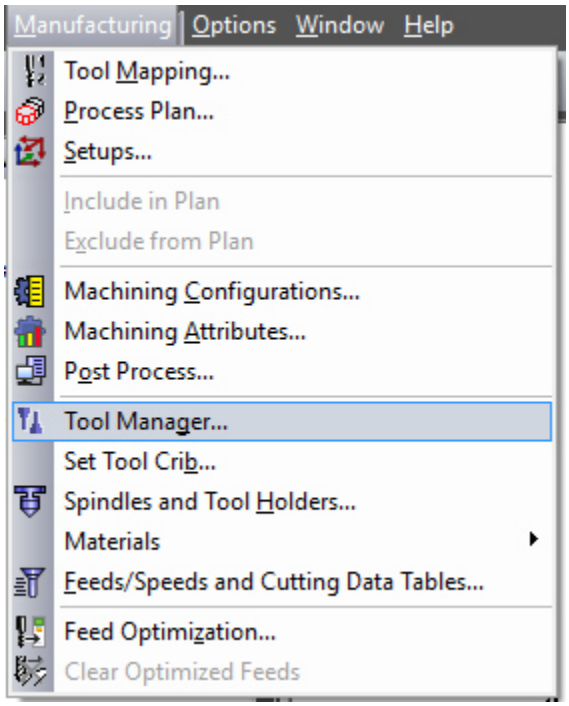
Using tools that you have in your machine

Tools in FeatureCAM are automatically chosen for you from available tools in the tool database. FeatureCAM installs a number of commonly used tools, but you have the ability to customize the tool database to suit the needs of your shop. The ways that you can customize your tool database are seemingly endless, but I would recommend that you set up a tool crib of general use tools that you have available throughout your shop for general programming. Further to this I would (and have) set up a tool crib for each machine that you have. Below is a small list of the pros of defining your tool database in this way:

- Easily switch your programs from one machine to another, FeatureCAM will automatically update to tools that you have available on that machine
- Define tools in your tool crib that stay in each machine. If you are not using that tool on a program, then FeatureCAM will not try to put another tool in that tool slot in the machine
- Having a single general tool crib allows you to see what tools FeatureCAM recommends you use in cutting certain features, which can be helpful if you are not sure what tool to use.

To work with your tool database in FeatureCAM:

1. Select the **Manufacturing** from the menu and then select **Tool Manager**



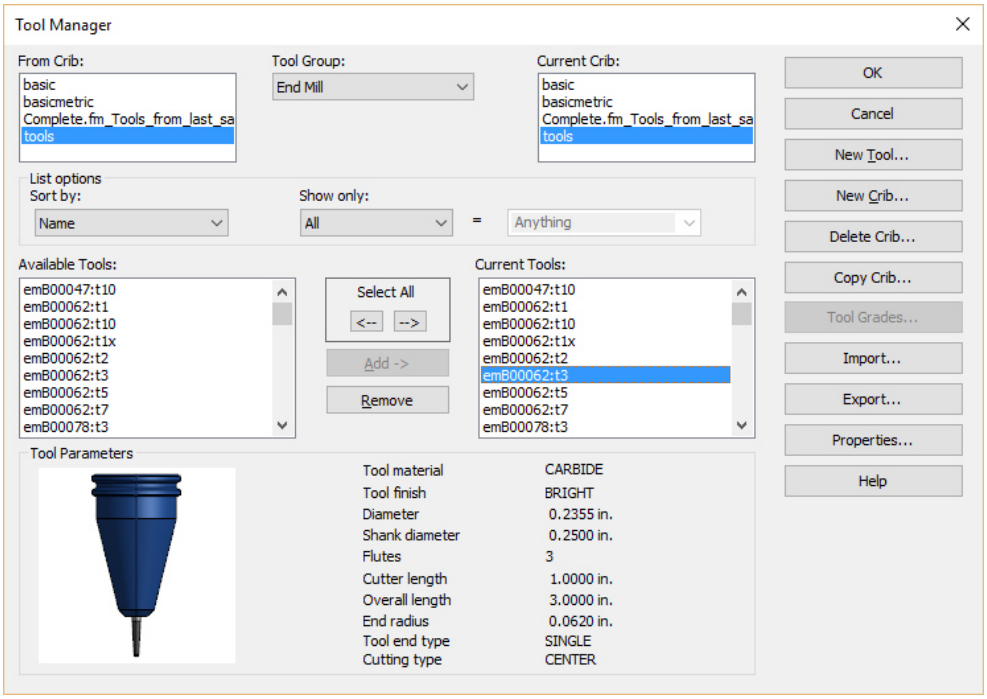
2. The Tool Manager is really separated into two sections
    a. On the left you have **From Crib**: - which allows you to get tools from other cribs you have created to populate the **Current Crib**

b. On the right you have **Current Crib** – all of the work you do will be in the Current Crib.

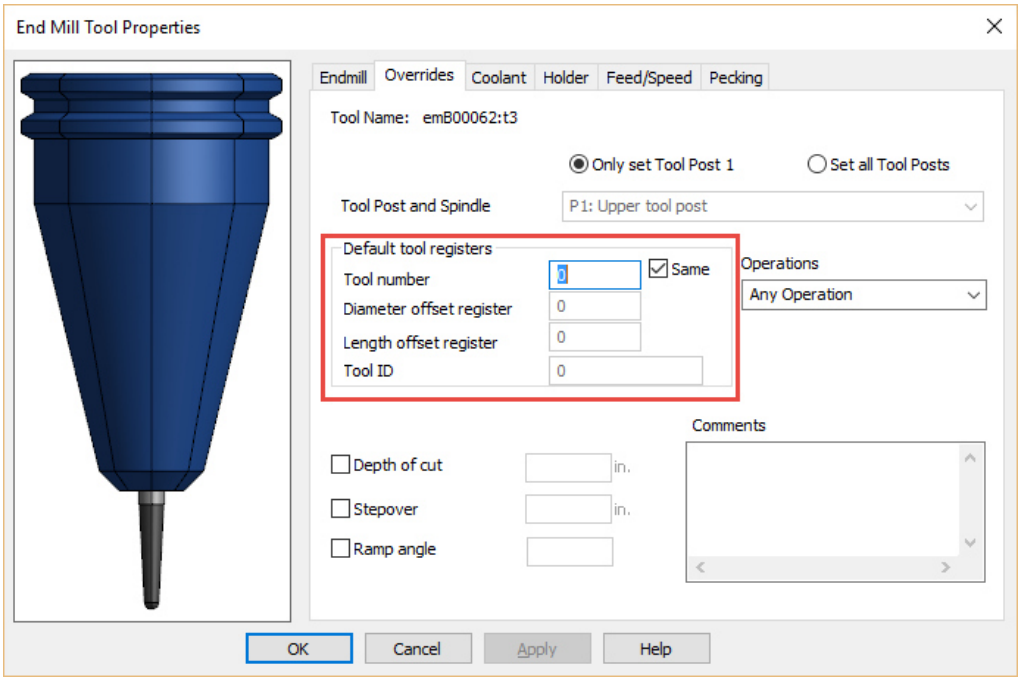
A few things to point out about the Tool Manager that are important:

- You can import an .xml file that you may have exported from FeatureCAM, maybe on another computer. This also allows you to create a backup of your tool crib(s) so you have one should anything happen to your local database or computer.
- You can create new cribs on the fly and add existing tools from other cribs to the new one to populate it from the start
- You can create new tools and fully define them as necessary

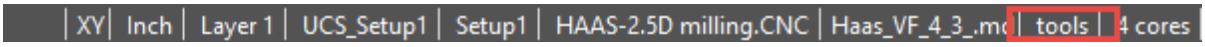
3. To set a tool in a machine's tool slot simply Double-click on the tool name in the Tool Manager



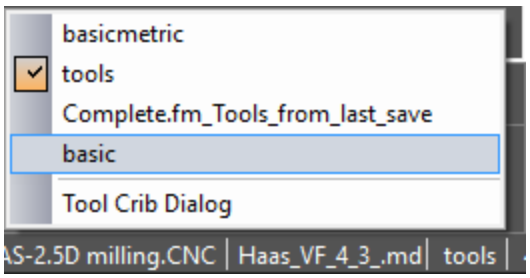
4. Select the Overrides tab and fill in the **Default tool register** sections
5. Click **OK**



6. Click **OK** on the Tool Manager and select yes to save the crib
7. You switch which tool crib you are using on your program in two ways:
    a. From the Tool Manager
        i. Navigate to the Tool Manager
        ii. In the From Crib select the crib you wish to use
        iii. Click **OK**
        iv. Click **Yes** to the dialog asking if you would like to switch from the current crib to the one you selected.
    b. On the Information bar in FeatureCAM



        i. Select your active crib name (in this case “tools”)
        ii. Select the crib you wish to use from the list

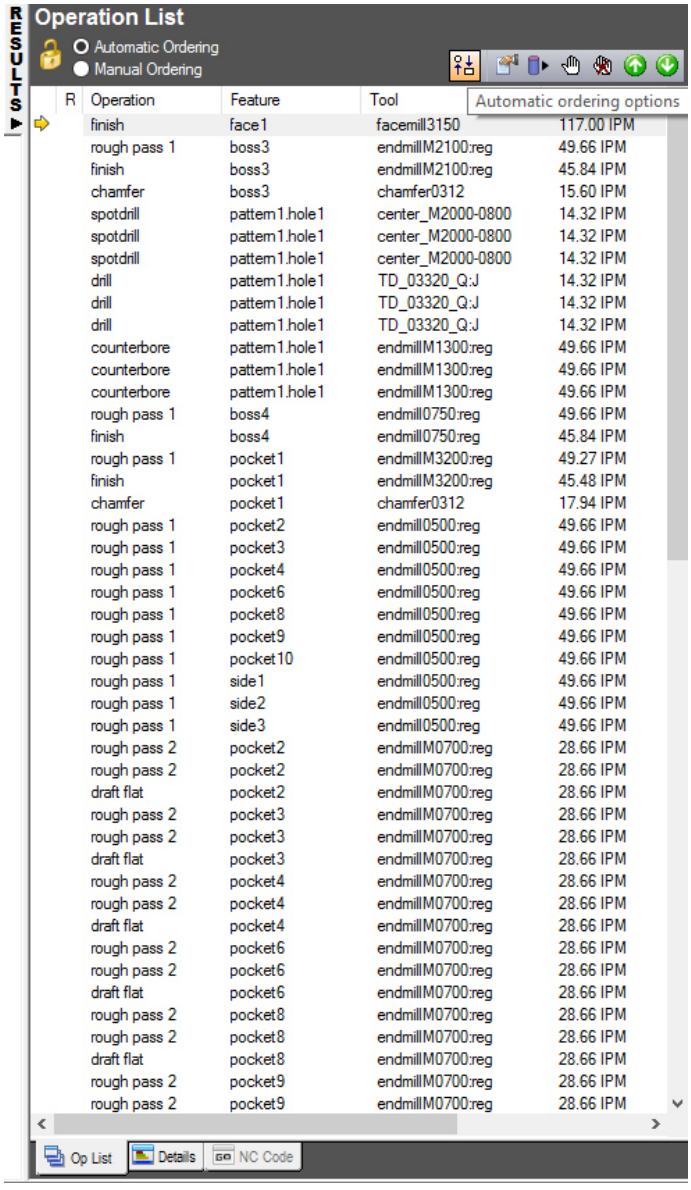


Ordering Operations

Contained within Features, are operations; an operation is one of the steps necessary in completing a feature. For example, a typical pocket will have a Rough milling **operation** and a Finish milling **operation(s)**. As you have seen, FeatureCAM creates these

operations for you based on the feature definition that you provide. If you machine your parts in the same order that you create the features that means that you will have to program with that intent to ensure the features align in the order that you want them cut. While this is the case with many Operation Based CAM system, this is not necessary with FeatureCAM because FeatureCAM can automatically order your cutting operations in a way that makes sense to machine. This ordering process is controlled through **Automatic Ordering Options**

1. Select the **Ordering Options** button in the **Results** window **– Op List**



2. You can select multiple options in this window, the options are:
    a. Minimize tool changes (default)
    b. Do finish cuts last
    c. Cut higher operations first
    d. Minimize rapid distance

4 axis mills

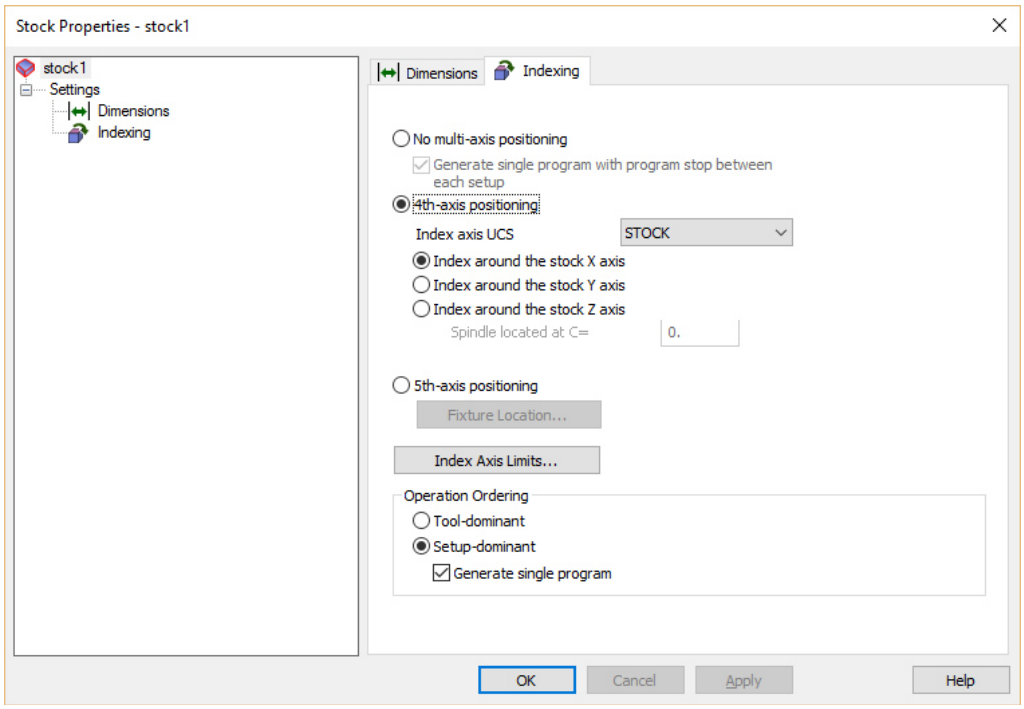
As we begin working through an example of 4-axis indexing and work into the tombstone machining module, notice that the way we program flow and general interface appearance do not change. We will continue to use the same types of features in 4-axis programming that we used in 2.5D and 3-axis programming. The only difference is that we need to specify a rotary direction now.

4-Axis Machining

1. Open the part **FM4axis_CutterBody.fm**

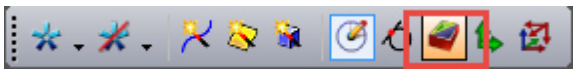
Note that we are working from a pre-turned or cast stock body in this case

2. Double-click on the Stock in the Graphics window
    a. Alternatively, you can double-click on **stock1** from the **Part View** in the **Toolbox**
3. To make a part document become a 4-axis file we just need to select the **Indexing** tab and select the options shown below. You can index around the X, Y, or Z axis.



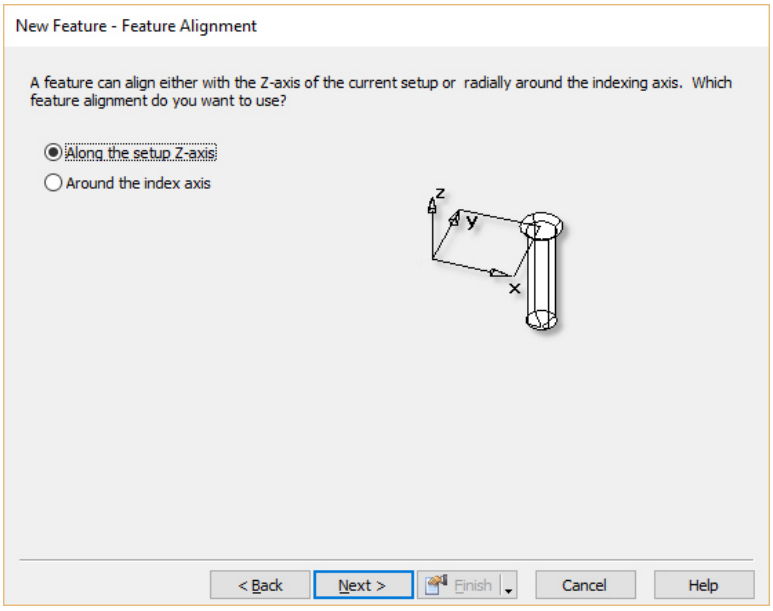
We can continue to work from the steps menu, but I am going to start using other toolbars that I have turned on.

4. From the Advanced toolbar select the **New Feature Wizard** icon

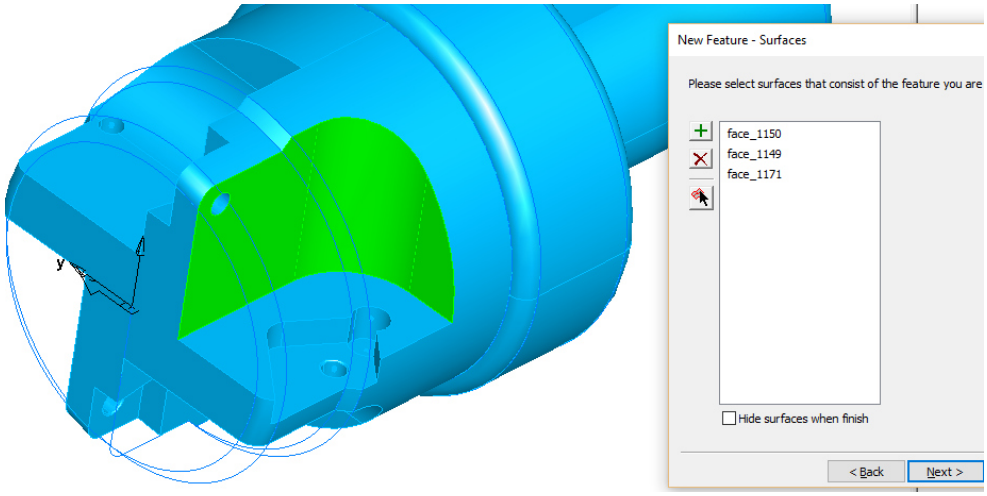


5. Select **Side** and **Extract with FeatureRECOGNITION**

6. The next dialog gives us the option of selecting where the feature reside, either along the setup z axis or somewhere else around the index axis. Select **Along the setup Z-axis**



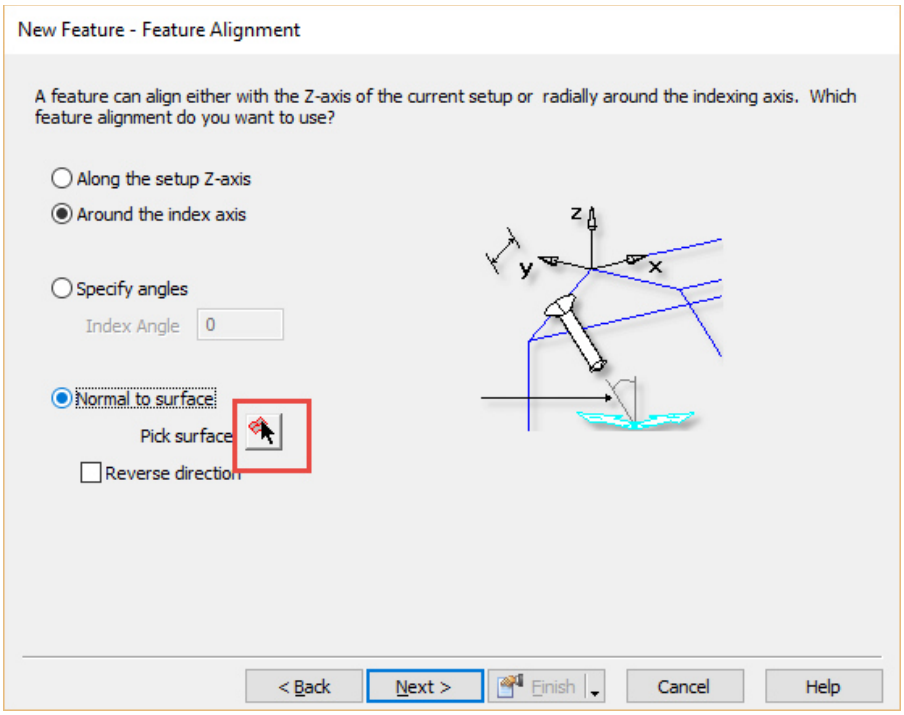
7. **Select side surfaces**
8. Pick the surfaces shown



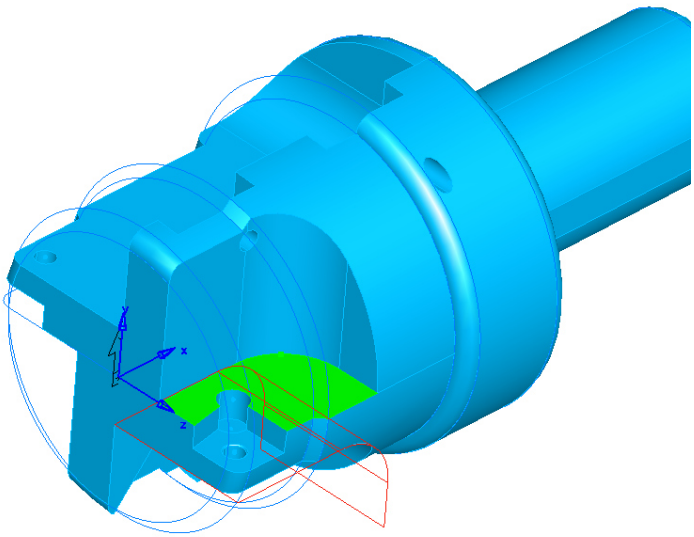
9. Click **Finish** and **OK**
10. Create another new feature using the **New Feature Wizard** again
11. Select **Side** and **Extract with FeatureRECOGNITION**
12. This time select **Around the index axis**

We now need to either define the index angle or select a surface that we want out index angle to be *normal* to

a. Select **Normal to surface** and the pick arrow

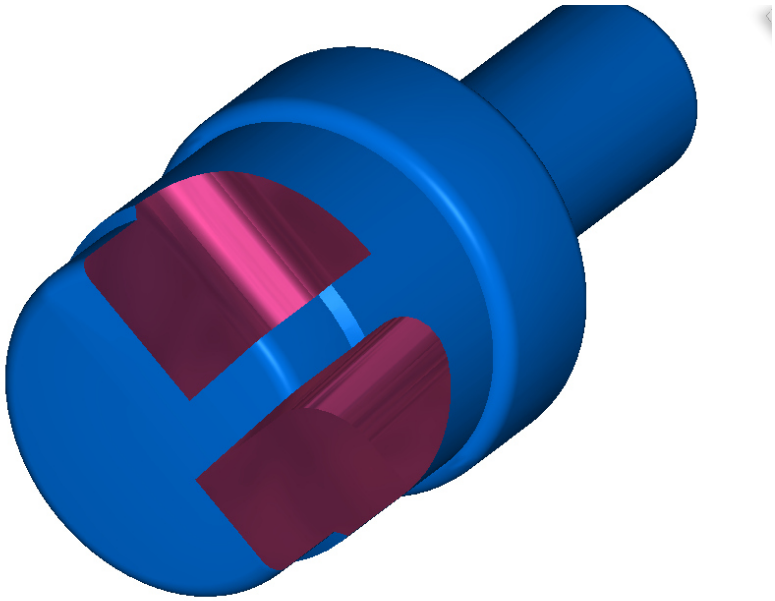


b. Pick the surface 90 degree counter clockwise to the side feature we just cut – the dark blue axis will update and show you the new "Z" direction and "XY Plane" after you select the surface

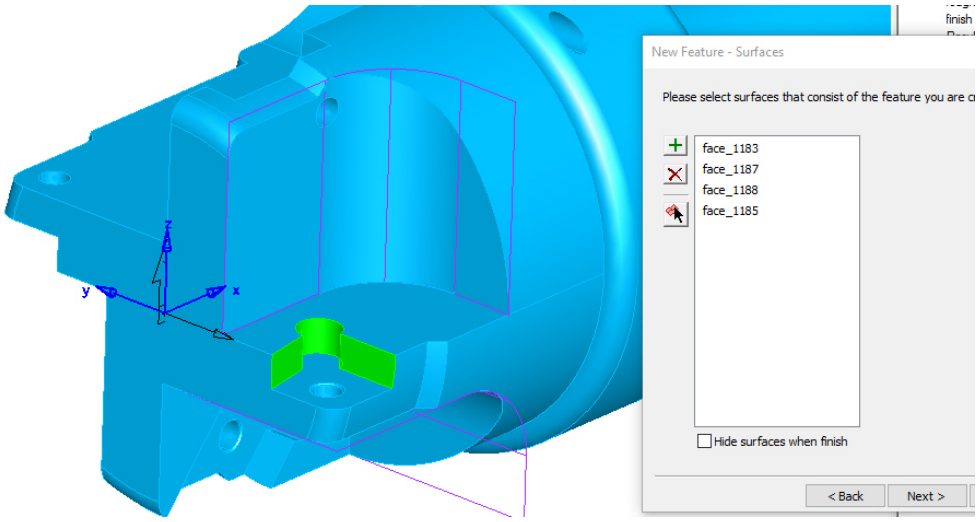


13. Select **Next** twice and select the side surface that make up that side cutout

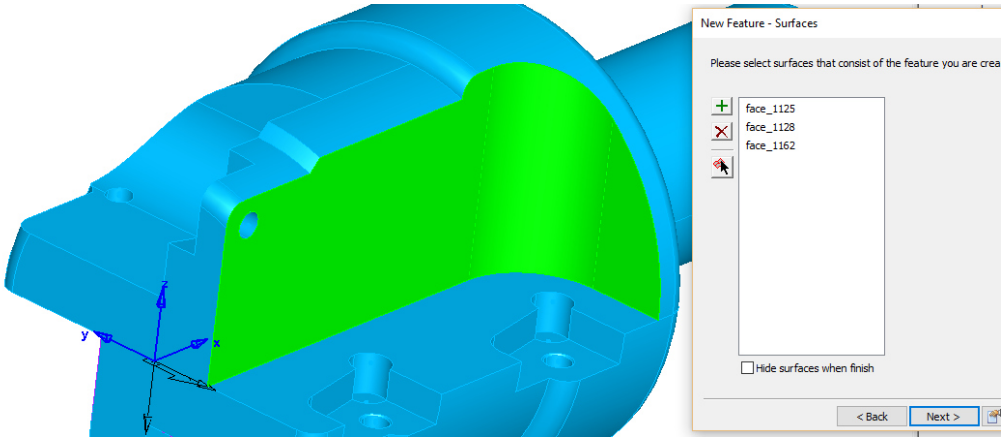
A 3-D simulation after completing the second side should look like this

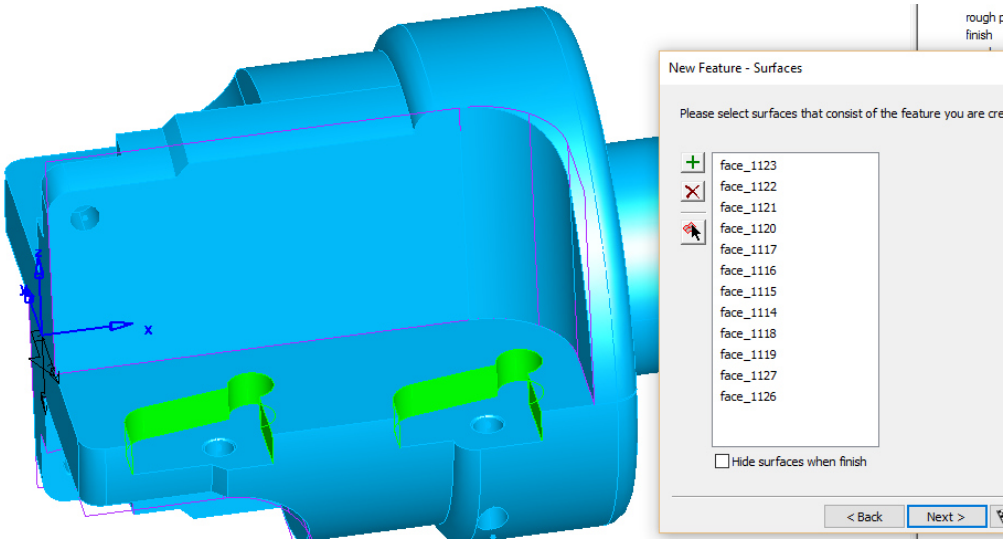


14. While we are working on this second side, we should also cut out the insert opening, Create a new side feature using the same techniques we used on the previous side feature, only this time select the side surface of the insert opening.



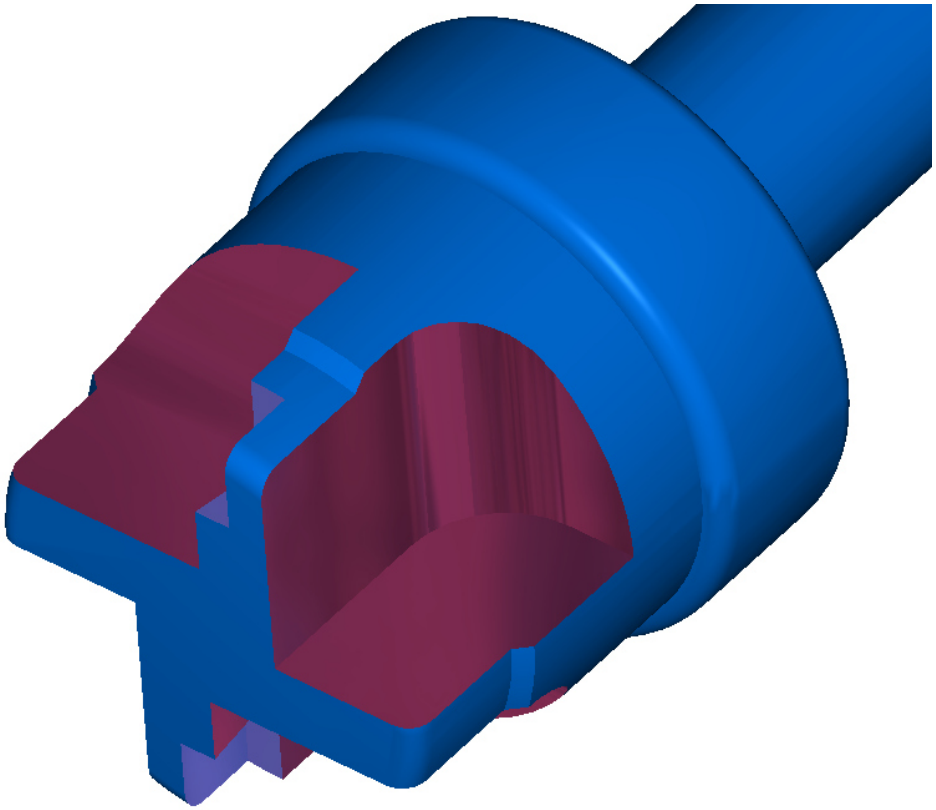
15. Rotate the part counter clockwise and repeat the steps to create a side feature for the large cut and the two insert pockets



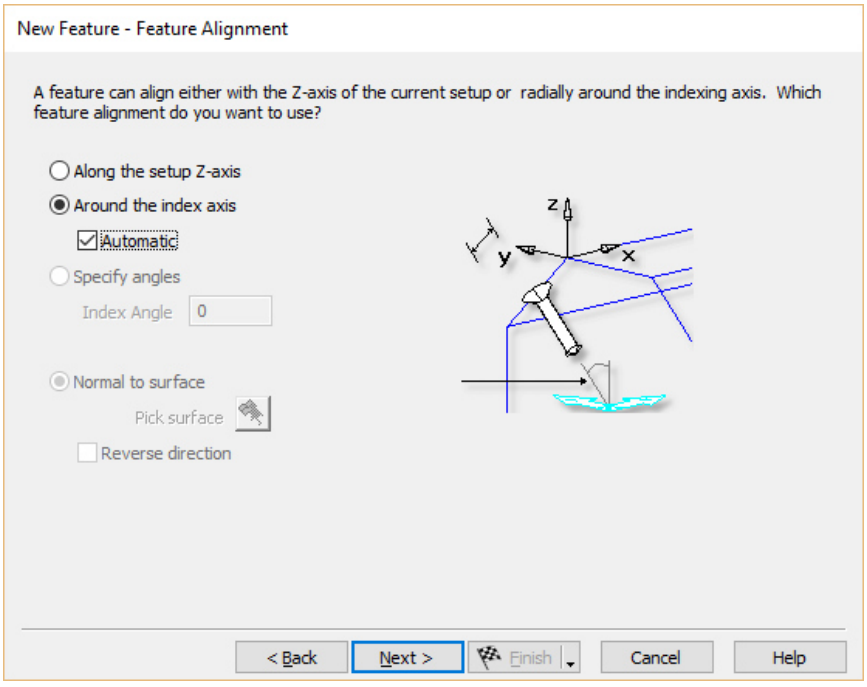


16. Follow the same steps to complete the remaining features of the part.

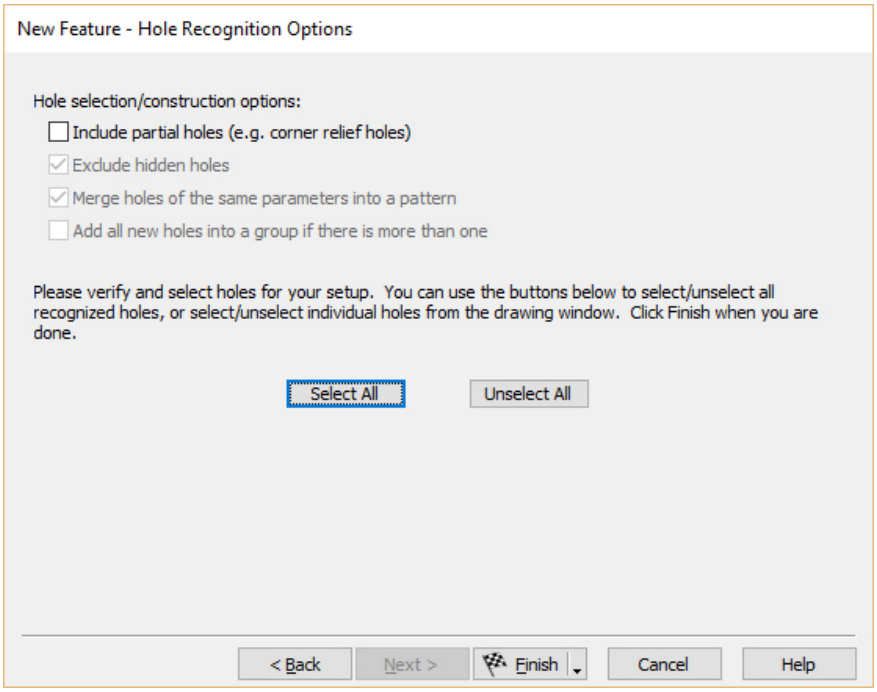
If you run a 3D simulation you will see that we forgot to cut the insert pocket on the first side that we programmed.



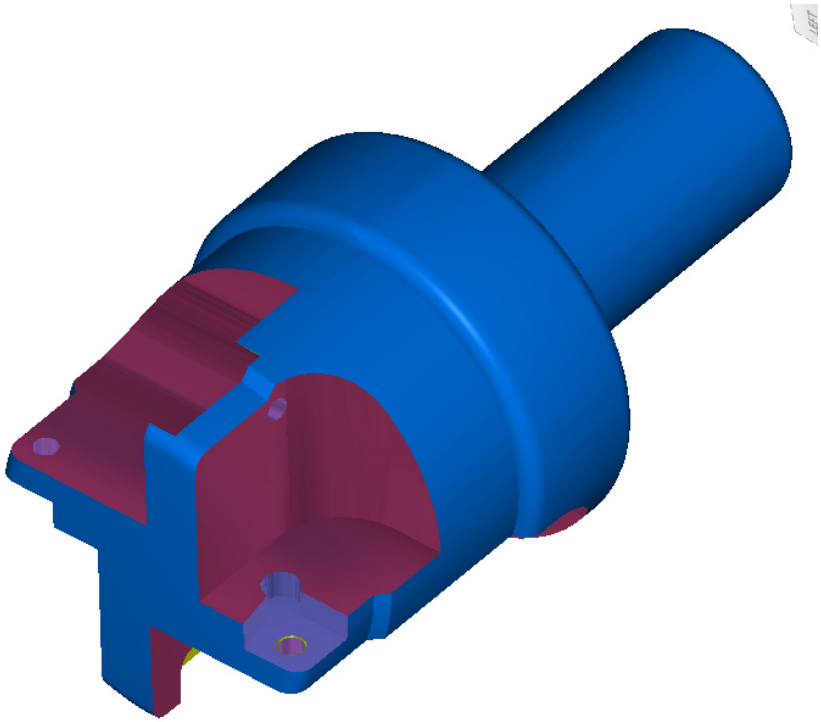
17. Program the remaining insert pocket – then we will program the holes
18. Start the New Feature Wizard again
19. Select **Hole** and **Extract with FeatureRECOGNITION**
20. Select **Around the index axis** and check the box for **Automatic**



21. Select **Next** twice
22. **Select All** and **Finish**



23. Run a **3D simulation** – the part is complete!

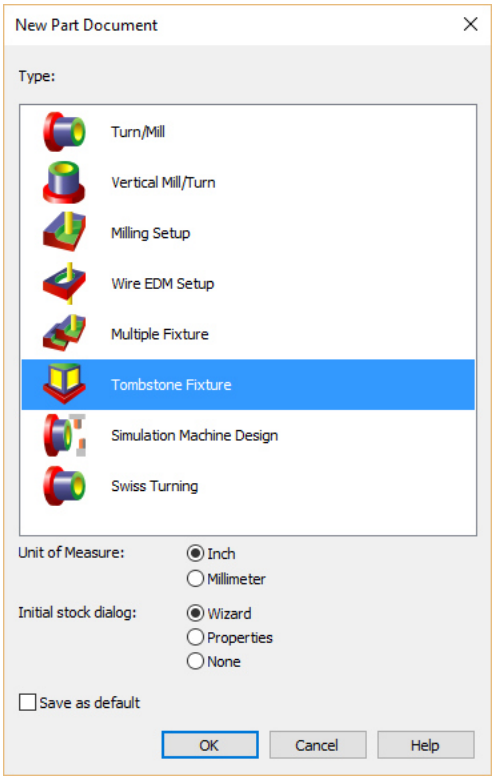


Tombstone

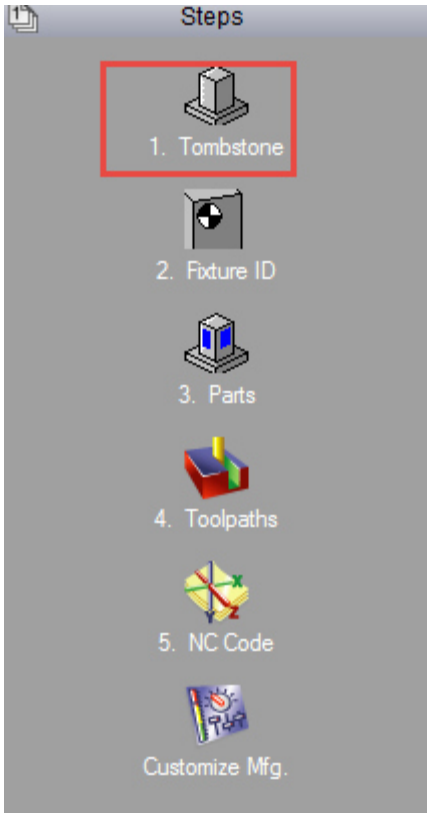
FeatureCAM makes machining multiple parts on tombstone setups easy (used in both horizontal and vertical mills). In FeatureCAM you program your part as an individual part then simply add it to a Tombstone file. FeatureCAM will re-order the operations in the tombstone file to follow the automatic ordering options that you have set in the tombstone file. You can tell FeatureCAM to cut everything on one Setup of the tombstone before moving on to the next setup, or you can tell FeatureCAM to cut everything it can on all setups in the tombstone file while it has the tool in the spindle. In other words, while the tool is in the spindle, it will use it everywhere it has been assigned to machine. This could result in more tombstone rotations however.

Let's take a look at tombstone machining on a horizontal mill.

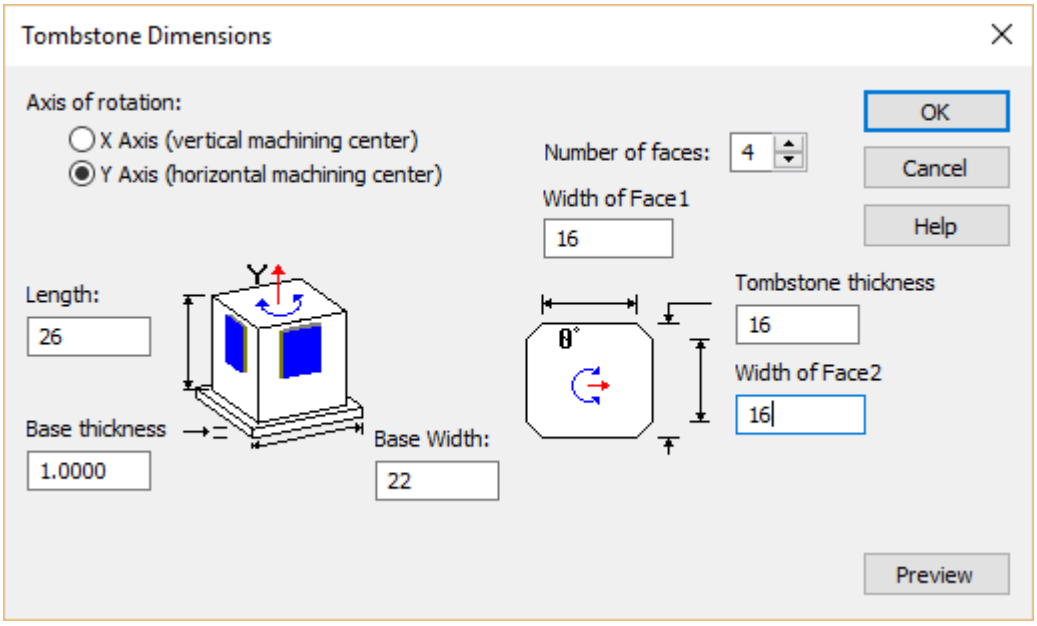
1. Open the file **Assem1.fm**
2. The .fm file contains a single programmed part, notice that there is work being done on the 3 sides of the part that are accessible (not mounted to the fixture)
3. Start a **New Part Document** and select **Tombstone Fixture**



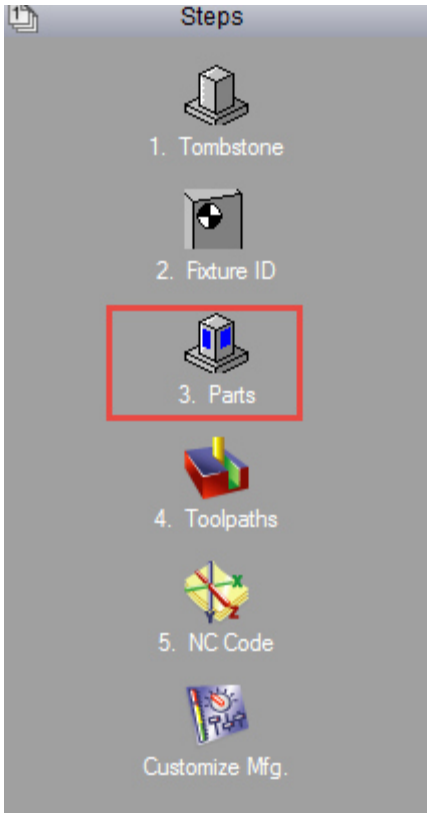
4. Click **OK** since we need to define the tombstone size before we add parts to it
5. Select **Step 1 - Tombstone**



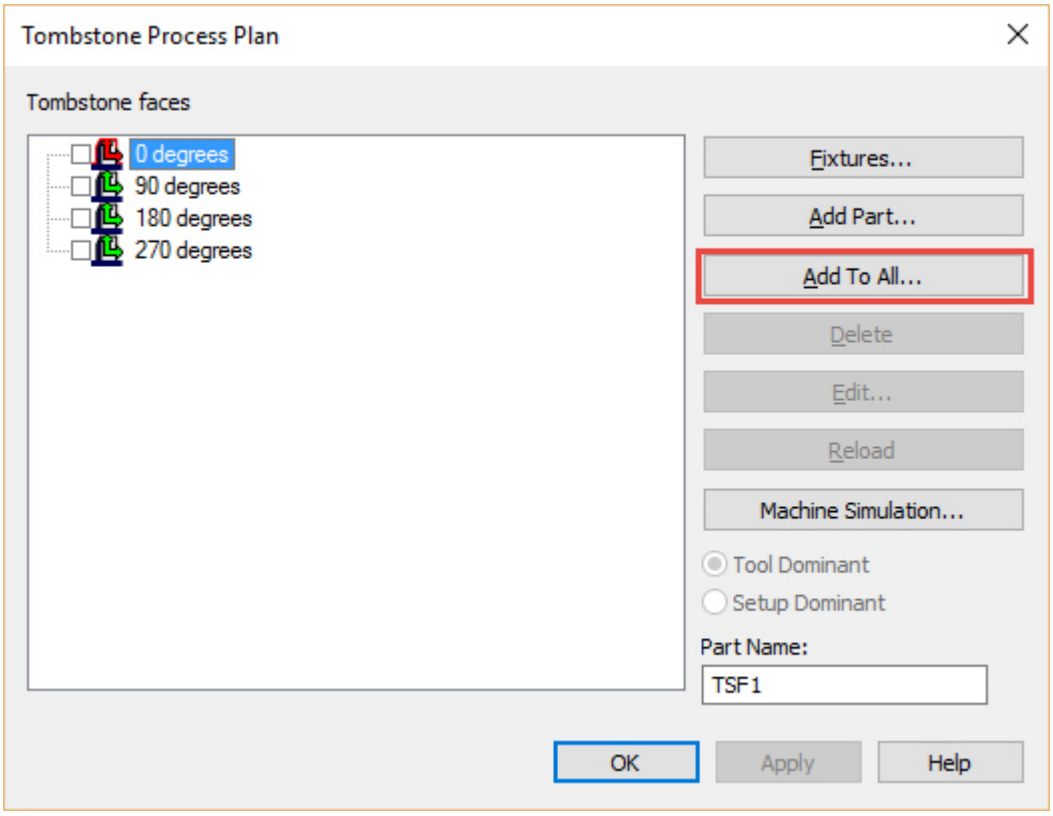
6. Fill in the Tombstone dimensions as shown to define the actual tombstone dimensions you have in your machine.



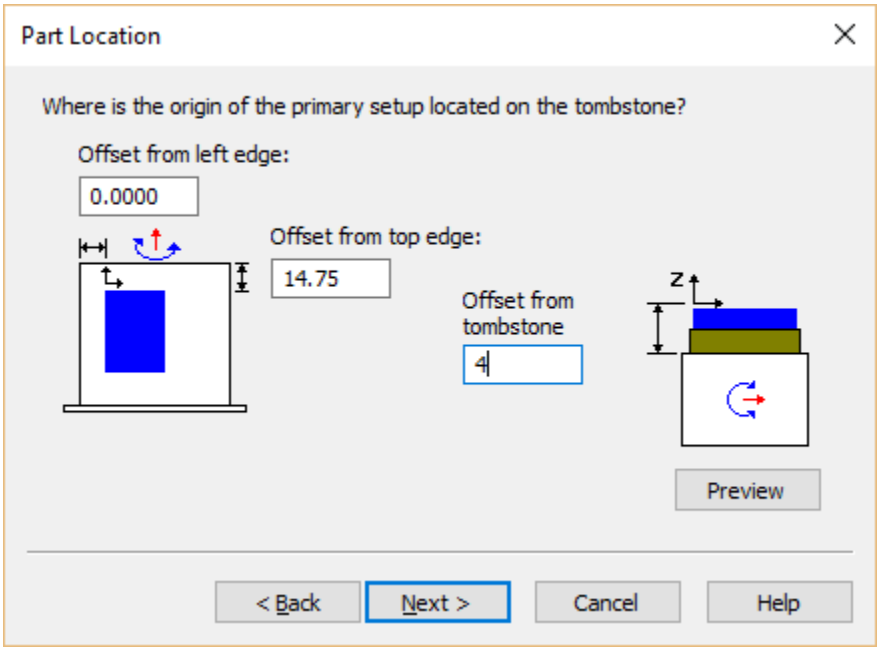
7. Click **OK**
8. Select **Step 3 - Parts**



9. Select **Add To All**



10. Click **Next** 4 times
11. Fill in the Part Location. You can offset from the Left edge, the top edge, and even from the face of the tombstone (distance from the tombstone face to the setup/stock thickness) that you defined previously.

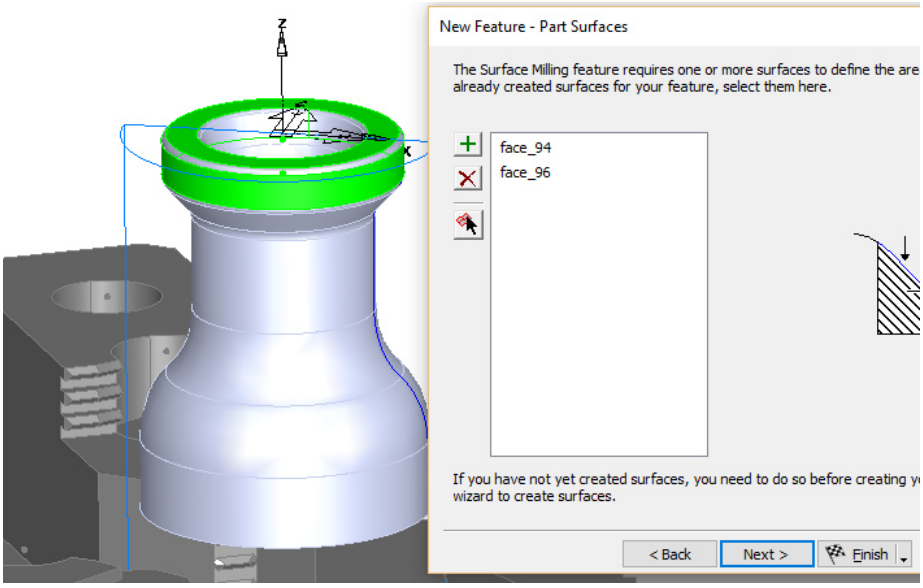


12. **Next**, then **Finish**
13. Run a **3D simulation** to see how FeatureCAM automatically re-orders the operations in a way that makes sense to machine on the tombstone.

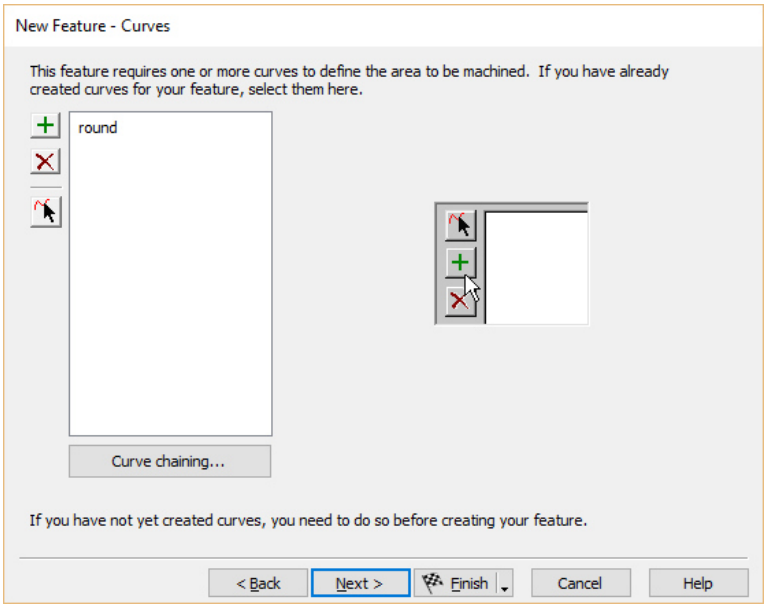
Turning Heads Machines

One of the many machine types that FeatureCAM is capable of programing is the turning heads machines. Oftentimes these machines are very large horizontal machines that have the ability to offset a turning tool from the spindle and rotate to cut simple diameters, faces, etc., but not always. In addition, there are a few turning heads attachments for vertical mills, some that even allow for multiple tools to be attached to a single turning head. We will look at an example like this.

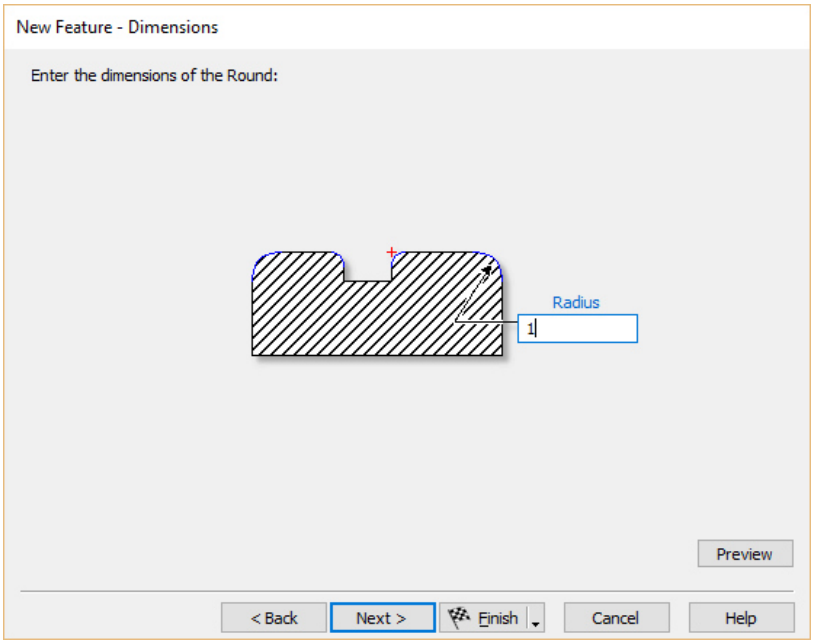
1. Open the file **TurningHead.fm**
2. Start a new side feature, extracting with FeatureRECOGNITION and using side surfaces
3. Select the small side surface shown



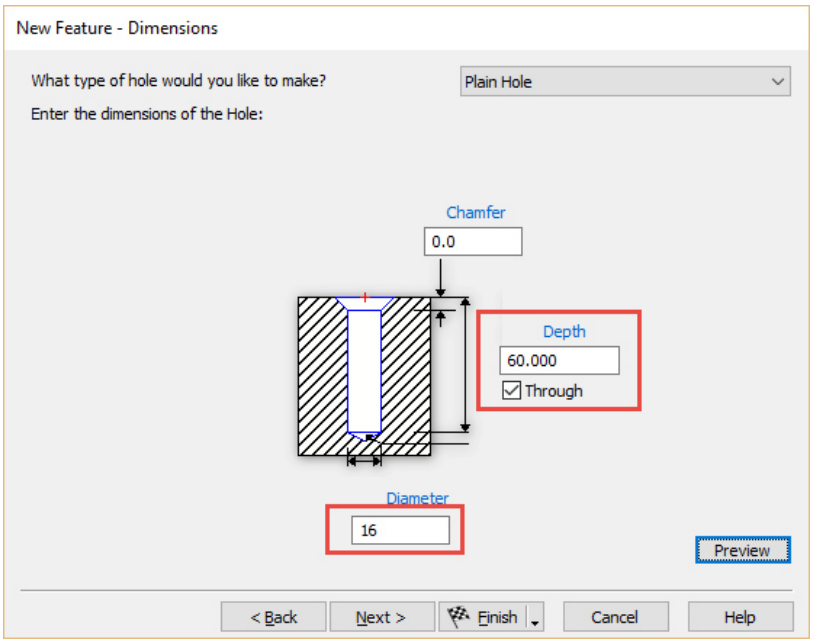
4. Accept all of the defaults for this feature by selecting **Finish**
5. Start the New Feature Wizard and select **Round** then **Next**
6. Select the curve *round* from the Part View and click the 



7. Select **Next** 3 times, enter 1 for the radius size



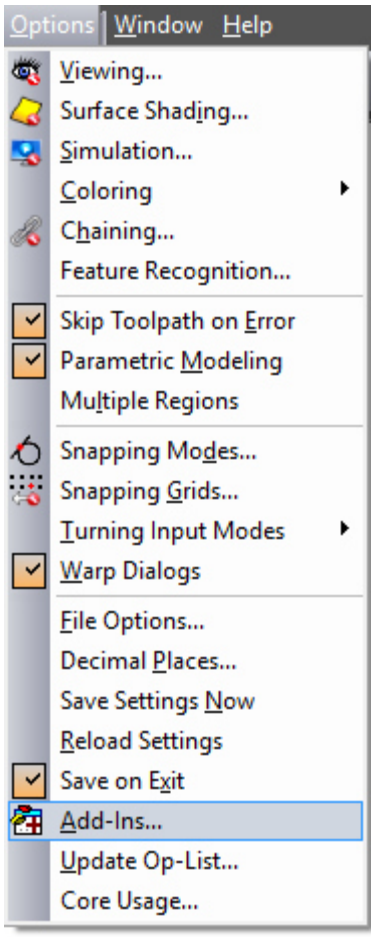
8. **Finish**
9. Start the New Feature Wizard and select **Hole,** make sure **Extract with FeatureRECOGNITION** is unchecked, then **Next**
10. Fill in the dimensions as shown



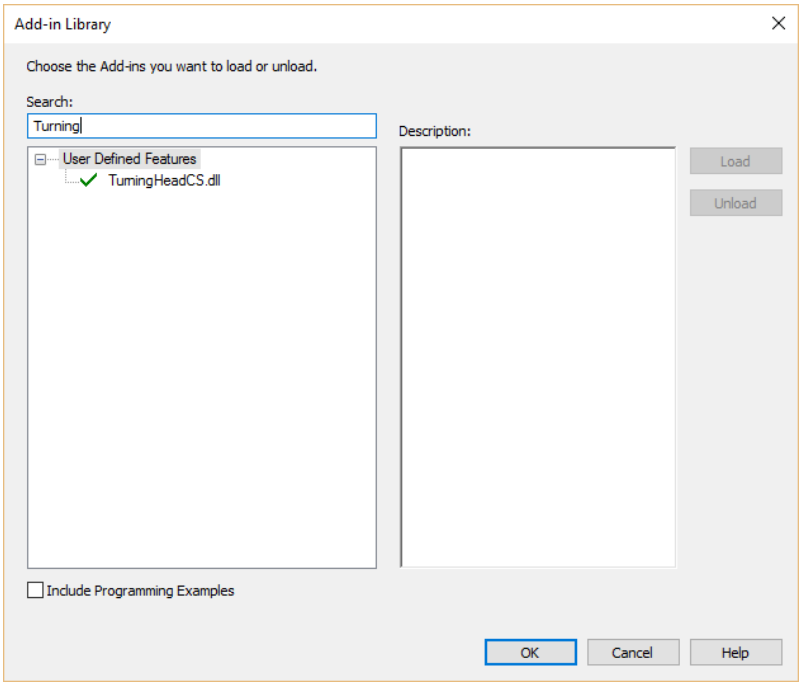
11. Finish

At this point, all we have left to program are the two profiles, one on the outside and one on the inside. To program these we will need to use FeatureCAM **Advanced Programing Interface (API)** and activate a User Defined Feature

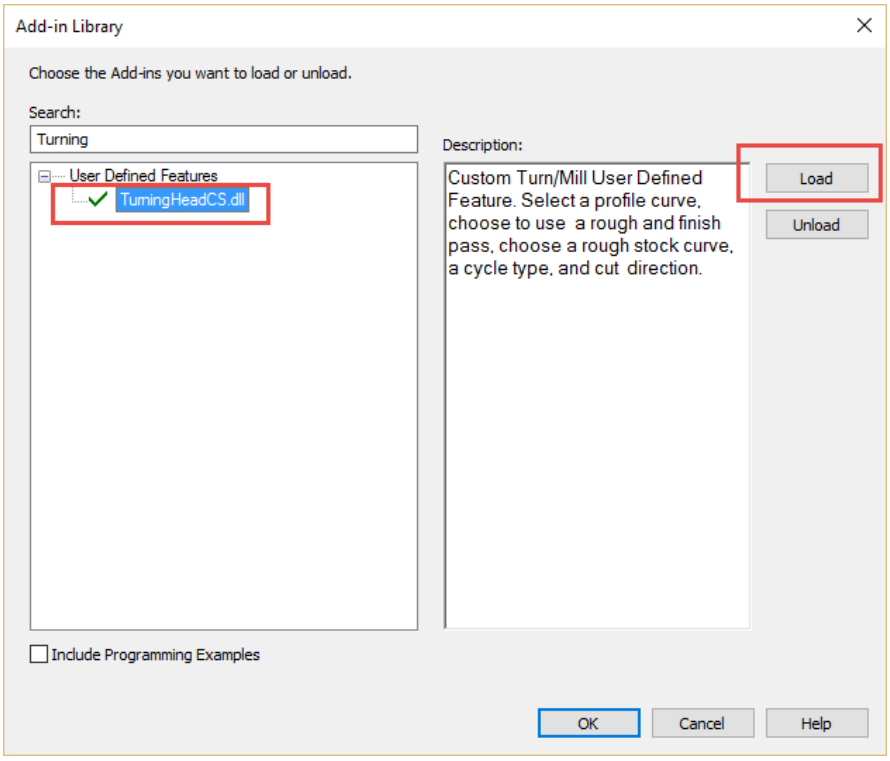
- Select **Options → Add-ins**



- Click on **Library**
- Search for *Turning*



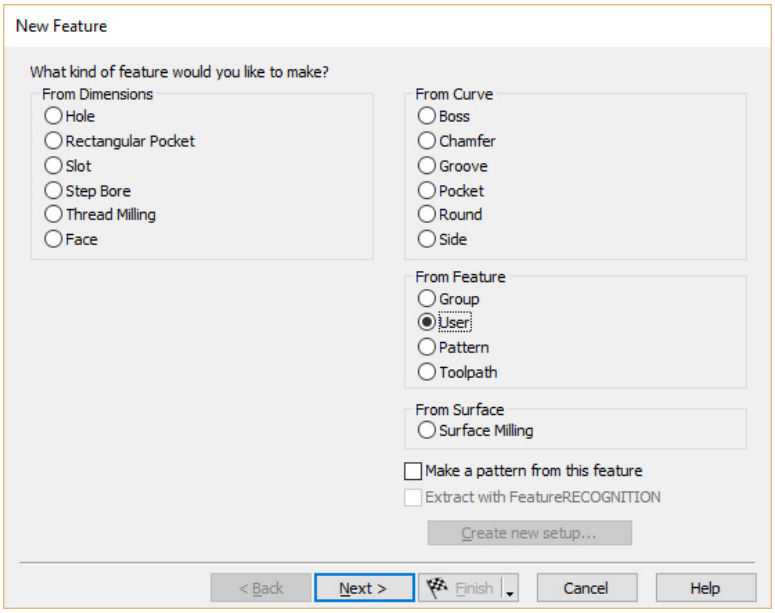
- Select **TurningHeadCS.dll** and select **Load,** then click **OK**



- Click **OK** again

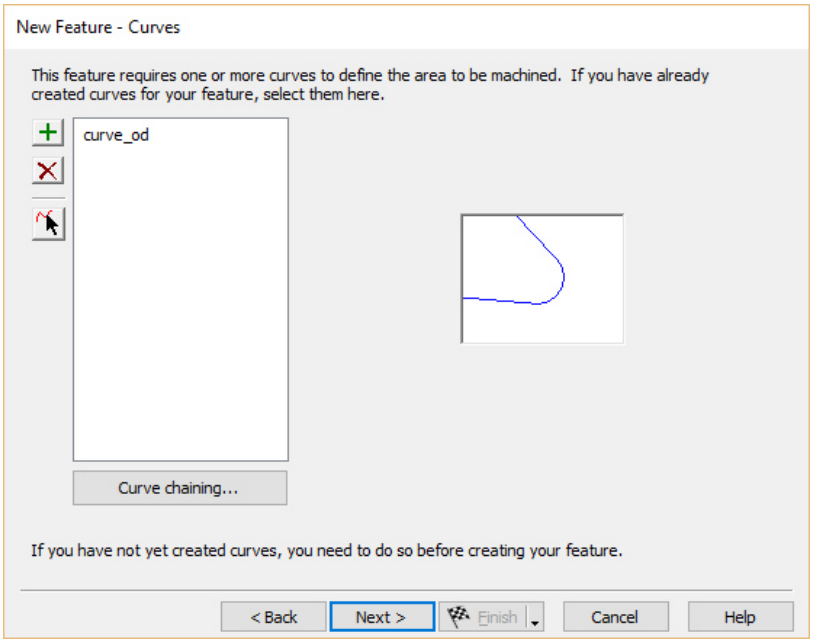
We are now ready to use the Turning Heads User Defined Feature

12. Start the **New Feature Wizard** and select **User** then **Next**



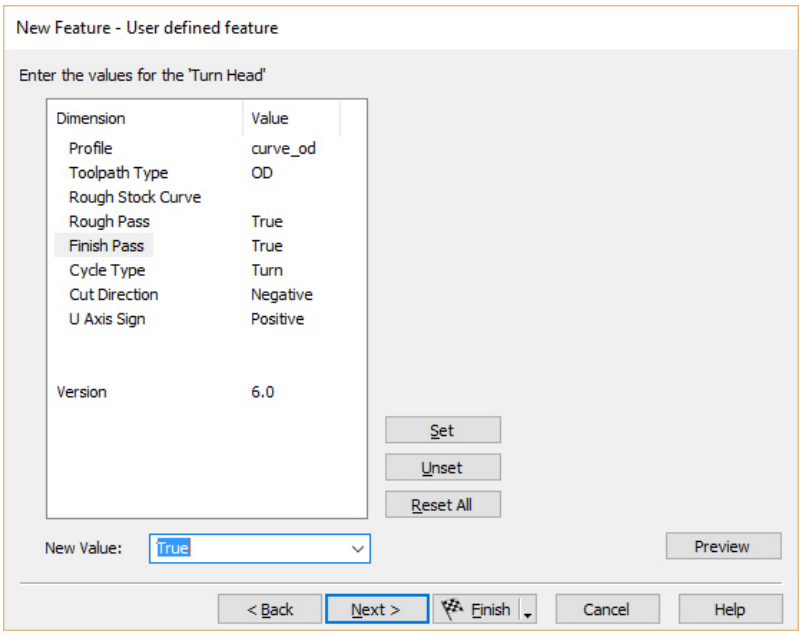
13. From the list Select **Turn Head** then **Next**

14. From the Part View select **curve_od** then click the + then click **Next** twice

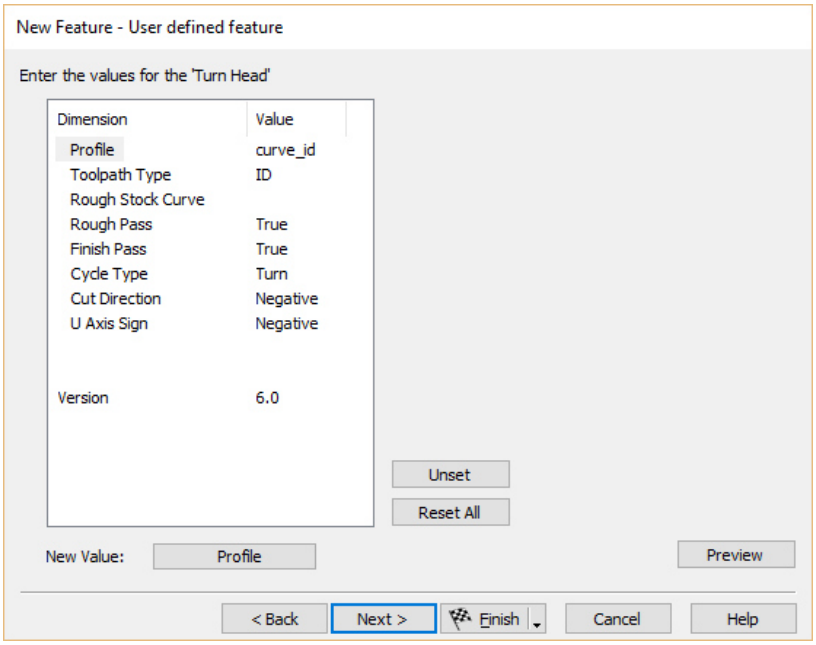


15. Fill in the user defined feature properties as shown

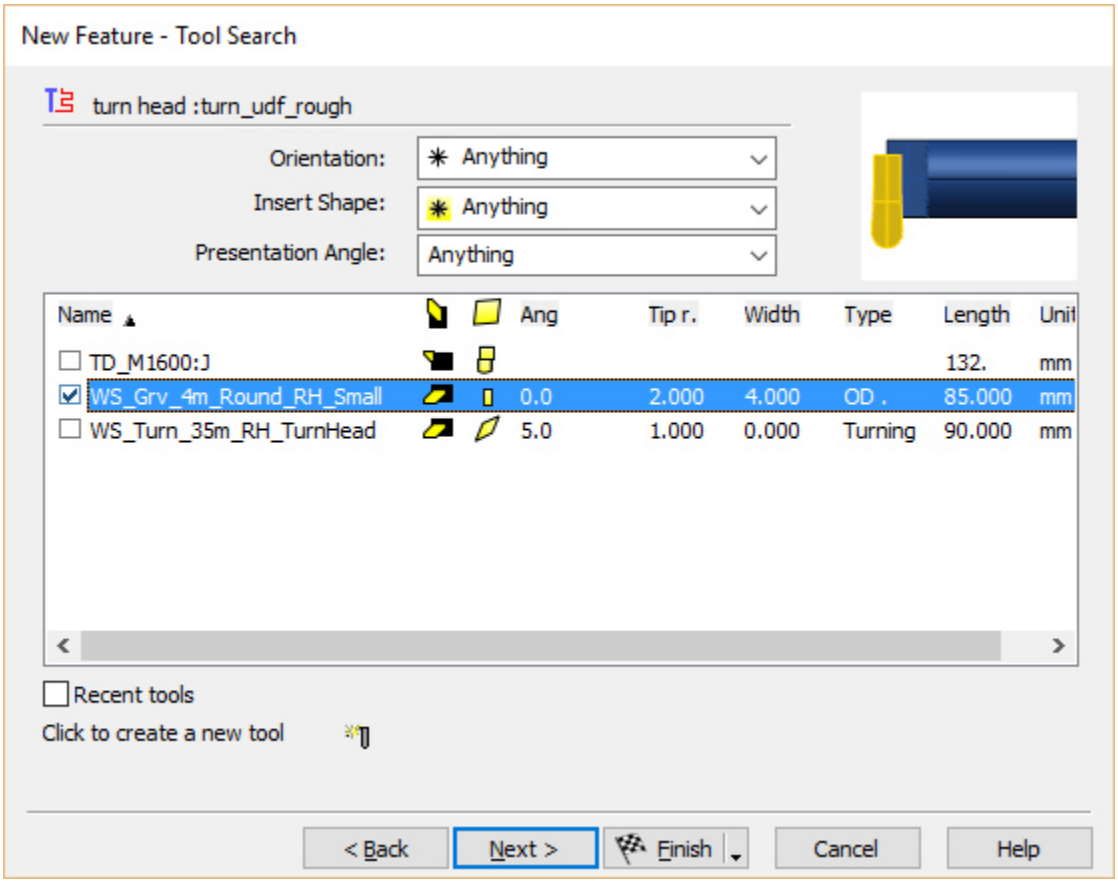
Note: Finish Pass is not turned on by default, so be sure to switch it on



16. Click **Finish**
17. Repeat the steps from step 12, this time be sure to select **curve_id** from the Part View and fill in the User Defined Feature Properties as shown



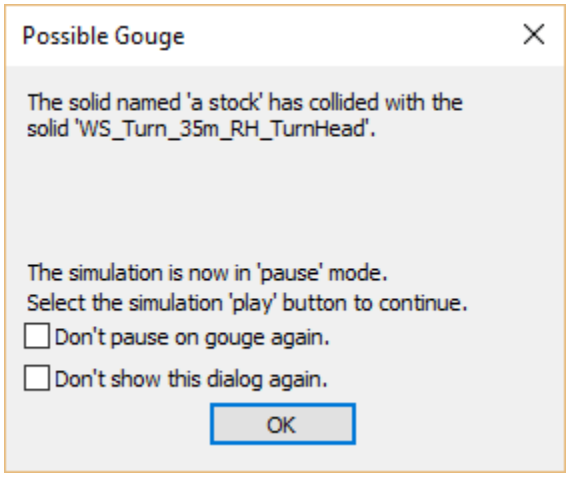
18. Select **Next** three times
19. Select the tool **WS_Grv_4m_Round_RH_Small** then click **Next** 4 times



20. Select the same tool for the Finish Operation, then click **Finish**

21. Run a Machine Simulation to see the two tools work in the turning head

22. After the rough operation of the OD profile runs you will get a gouge warning that pauses the simulation



23. Click **OK** and continue to run through the simulation

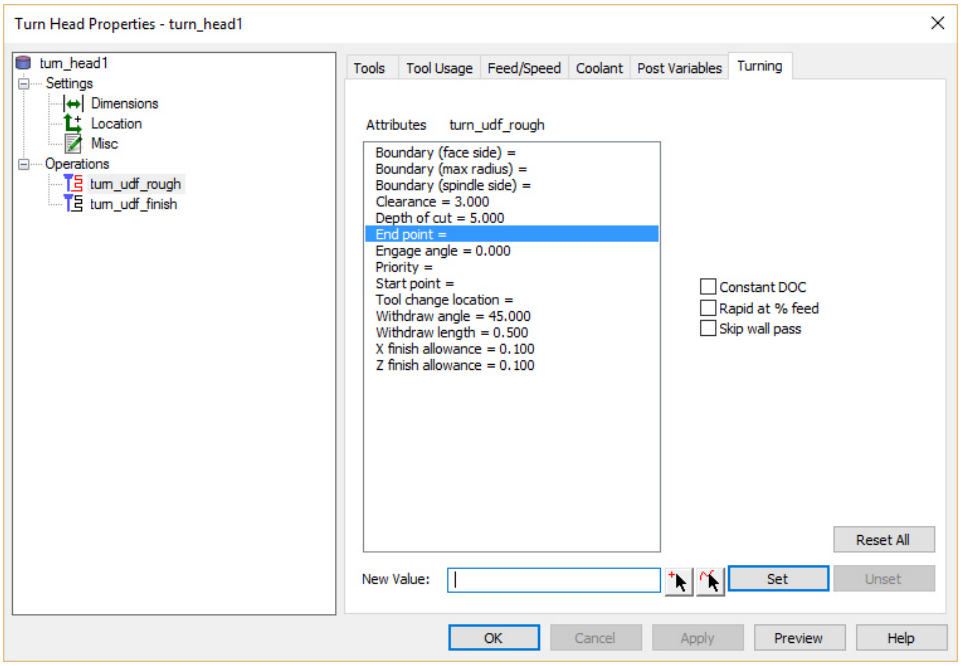
24. You will get another gouge warning after the finish operation runs on the OD. If you continue to **OK** these warning and eventually make it to the ID, you will see that

there is very little left of the part. To remedy this, we need to tell FeatureCAM where to end the toolpaths (both rough operation and finish operation) on the OD profile

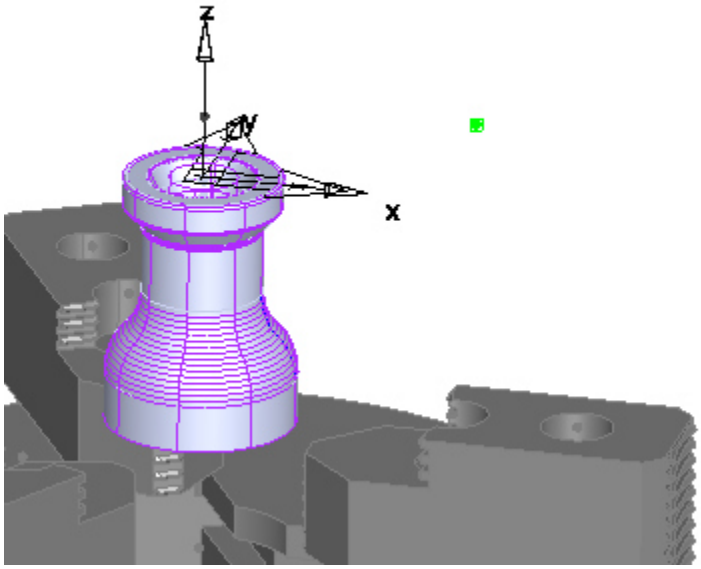
a. Stop or eject your simulation

b. Double-click on the OD profile turning head feature (either in the graphics window or in the Part View – this will open the properties of the feature

c. Select **turn_udf_rough** then select the **Turning** tab. Finally select the **End point =** attribute and select the **Pick Point**  arrow



d. You will see a point in the graphics window, select this point



e. Repeat the process for the **turn_udf_finish** then click **OK**

25. Run a Machine simulation to see the completed part using a turning head with two tools mounted in it.

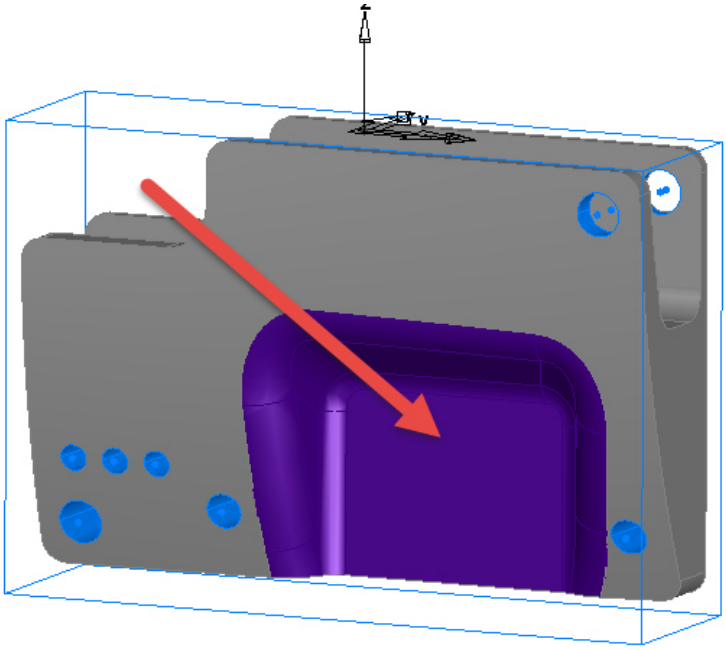
5 axis mills

FeatureCAM is capable of programming complex 5-axis milling machines, both positional and simultaneous. In this example we will see how programming using Features makes programming complex 5-axis parts simple, and how we still program using the same features that we did in 3-axis.

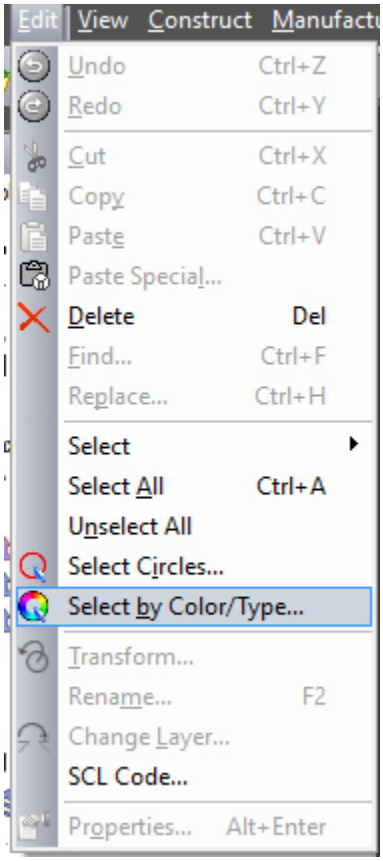
1. Open the part **5-axis start.fm**

If we look at this part, you will see that the current (finished) design state is not particularly suitable to an initial machining operation, before we begin we will clean up this part to get some better machine-able surfaces. We will utilize FeatureCAM's solid modeling capabilities to do this.

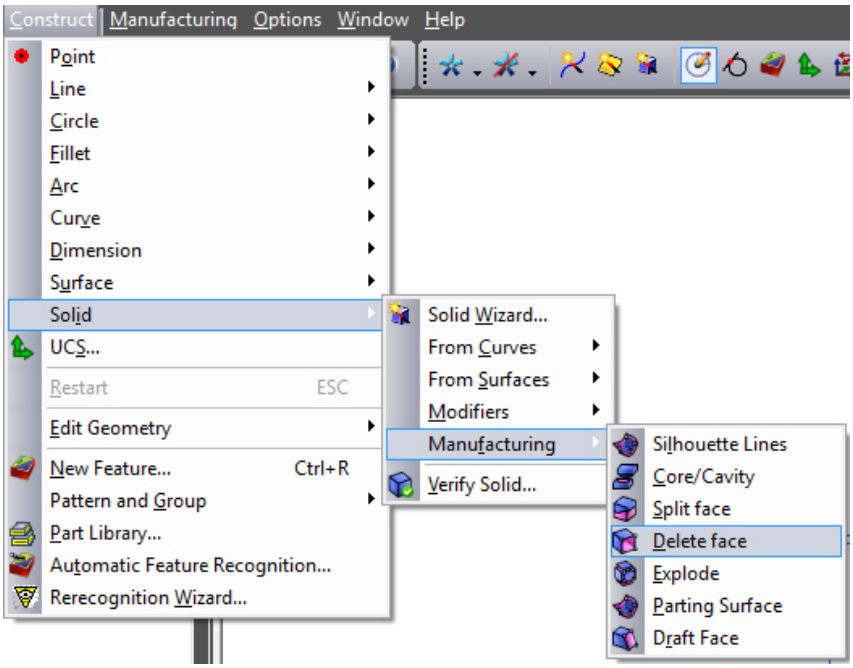
2. Pick one of the purple surfaces



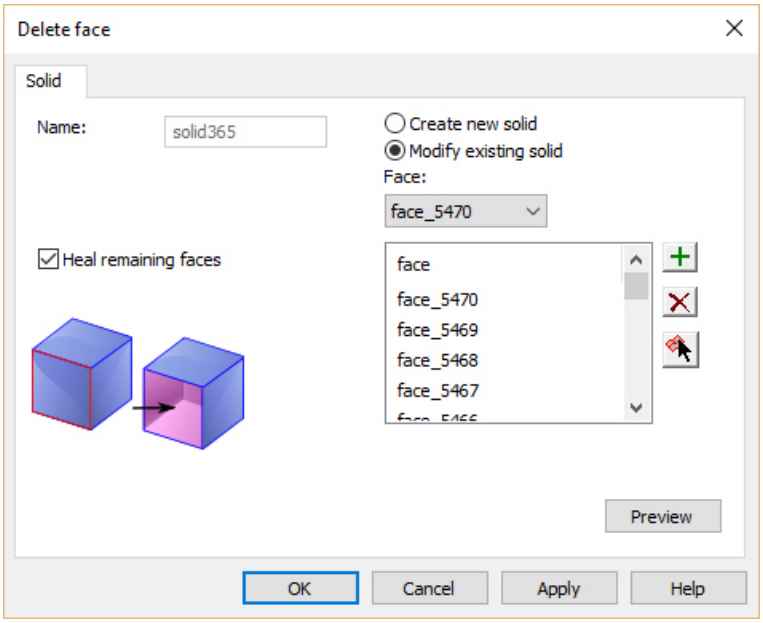
3. From the **Edit** menu choose **Select by color/type…**



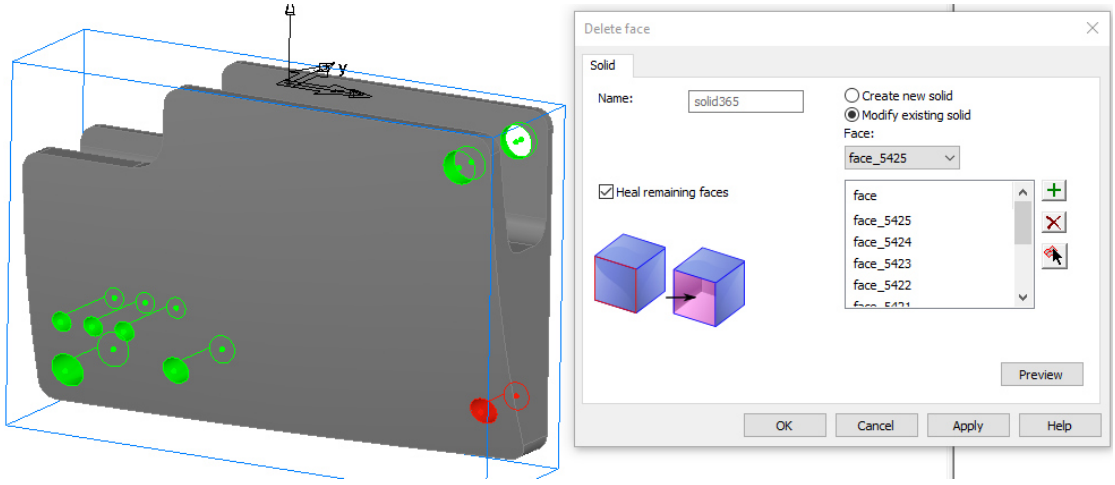
4. Click **OK** all of the purple surfaces will be selected
5. From the **Construct** menu navigate to **Solid → Manufacturing → Delete Face**



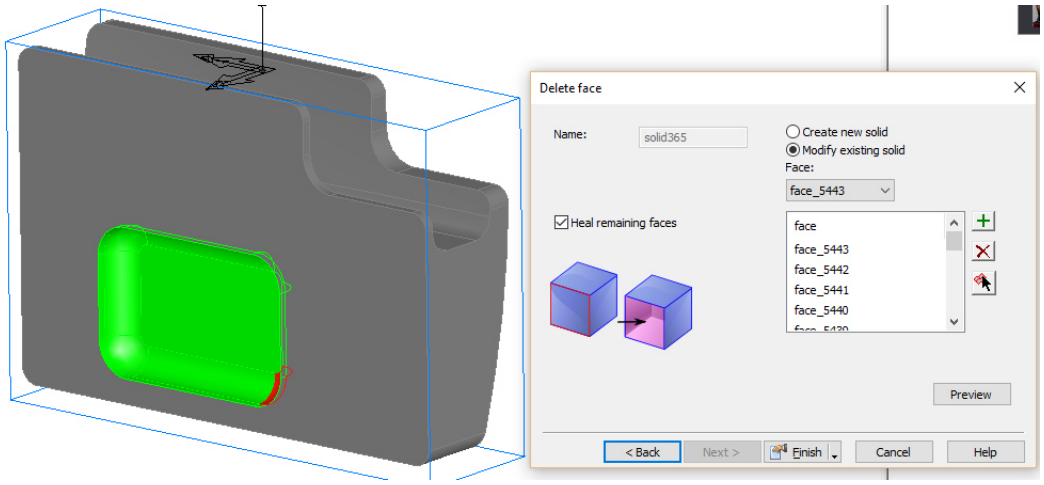
6. Select the **Modify existing solid** radio button and select the checkbox **Heal remaining faces.** Then click **OK**



7. Select one of the blue holes
8. Repeat steps 3 – 6

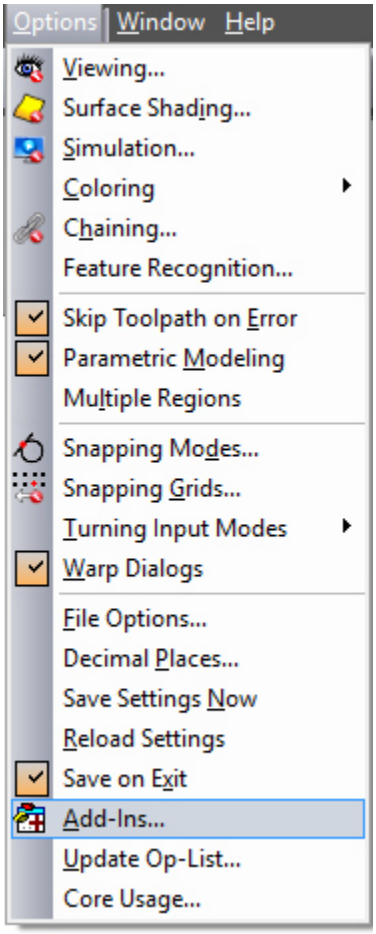


9. On the back of the part, you will notice a pink pocket. Select one of the surfaces and repeat steps 3 – 6 again.

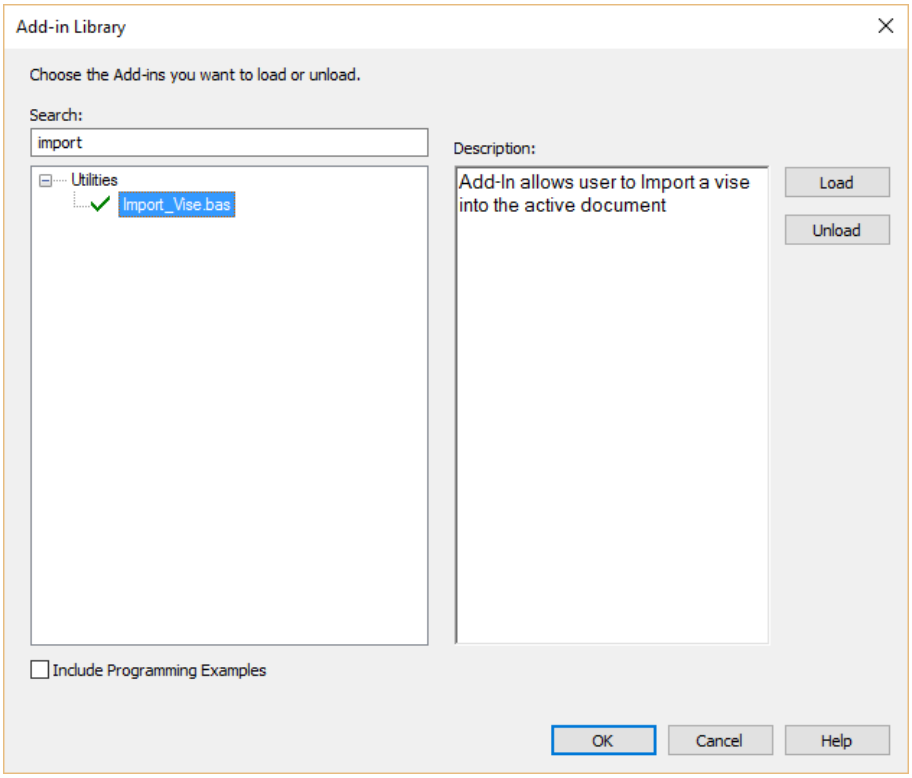


We now have a part that has much better initial machining surfaces. Before we begin machining this, we should add a vice to this part file so that we can create accurate toolpath and still be able to hold on to the part. To do this, we will utilize FeatureCAM API again, and use an add-in to import the vice and mount the part/stock in the vice.

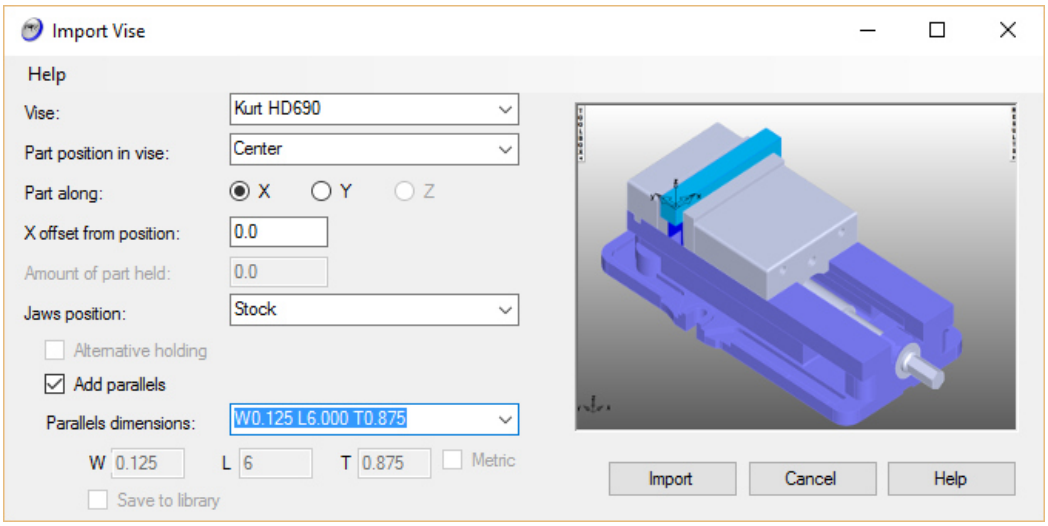
10. Navigate to **Options → Add-ins…**



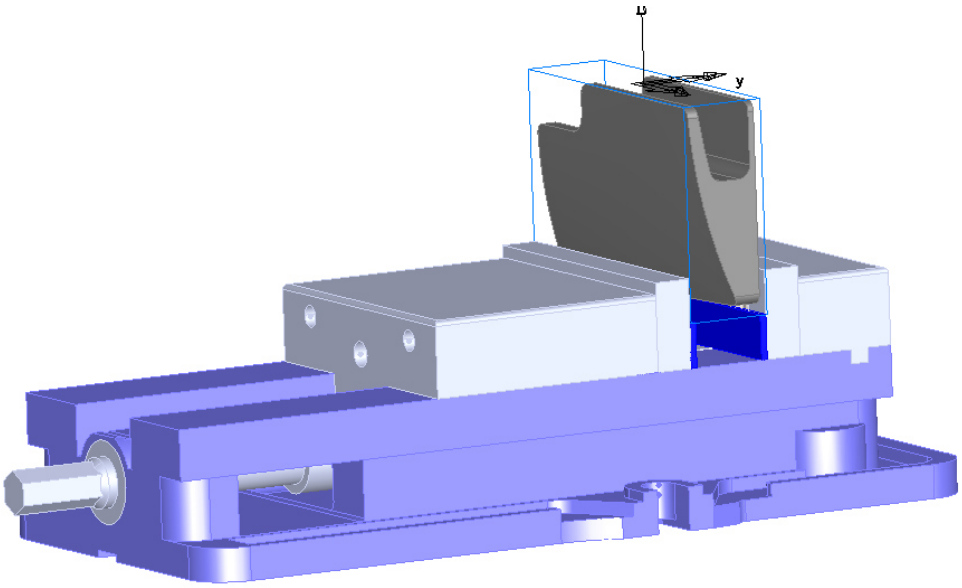
11. Click on **Library**
12. Search for *import* and select **Import_Vise.bas**
13. Click **Load,** then **OK,** then **OK** again



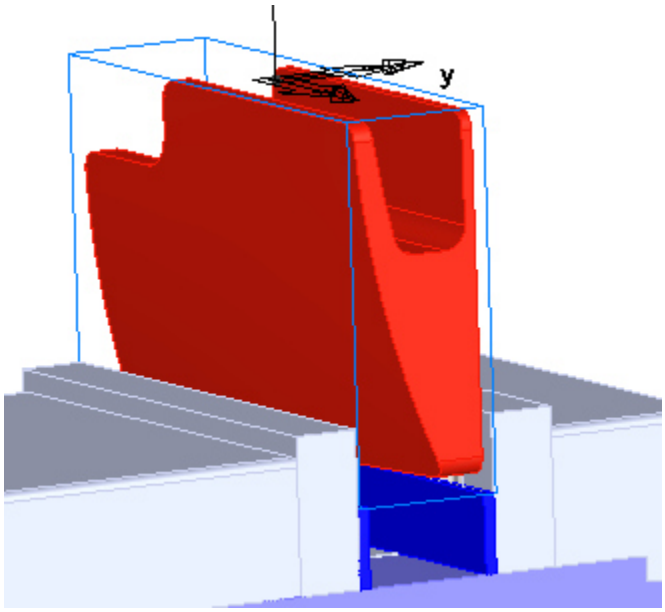
14. FeatureCAM will create a button for the import vice macro, it should be located in the top right of FeatureCAM and will look like this  click on this button
15. Fill out the form as show – note the checkbox for **Add parallels** and the size of the parallels we are using



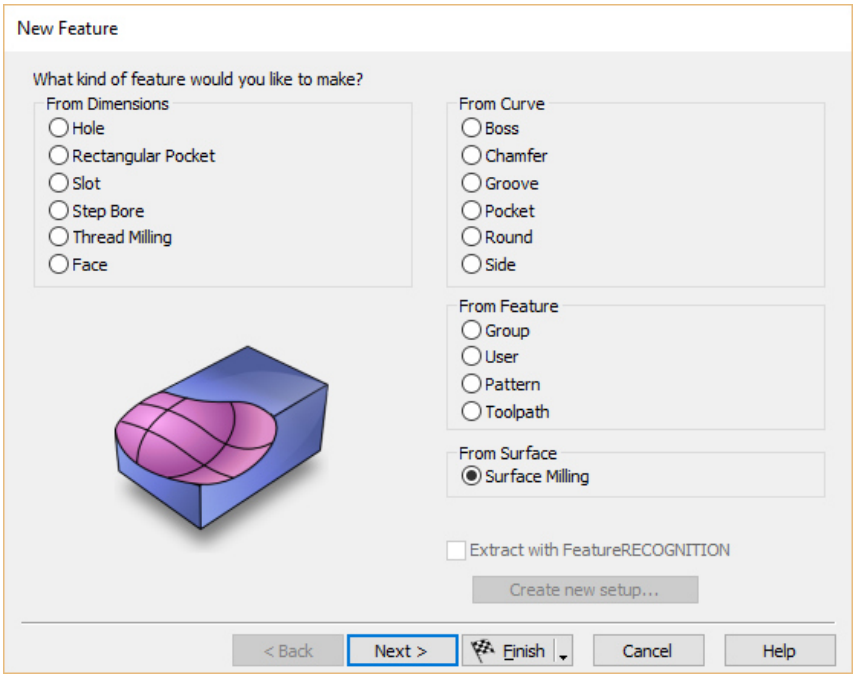
16. Click **Import**



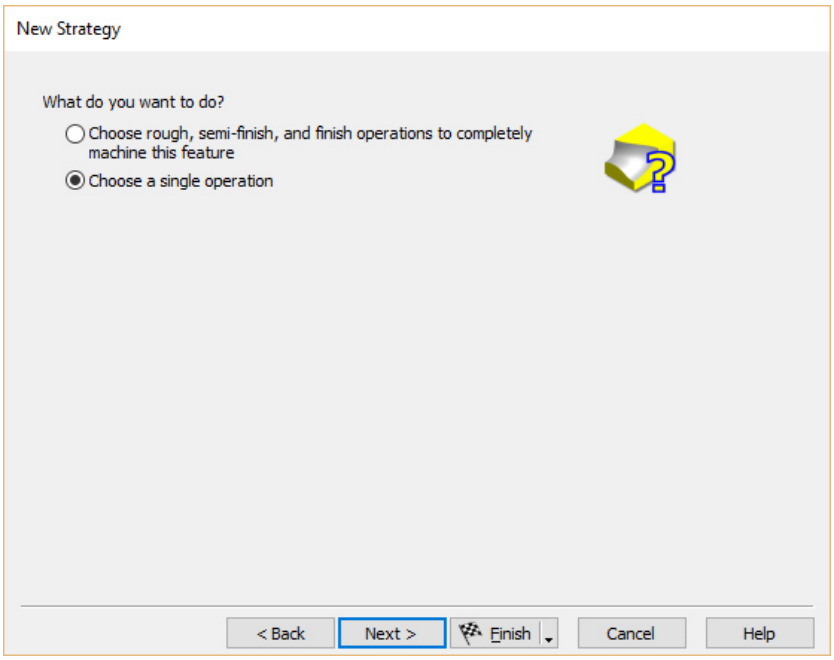
17. Select Setup2 from the Part View
18. Select all of the Vice components, right click and select **Use Solid as Clamp**
19. Select Setup3 from the Part View and repeat step 18
20. Select Setup1 from the Part View
21. Select the entire part for machining



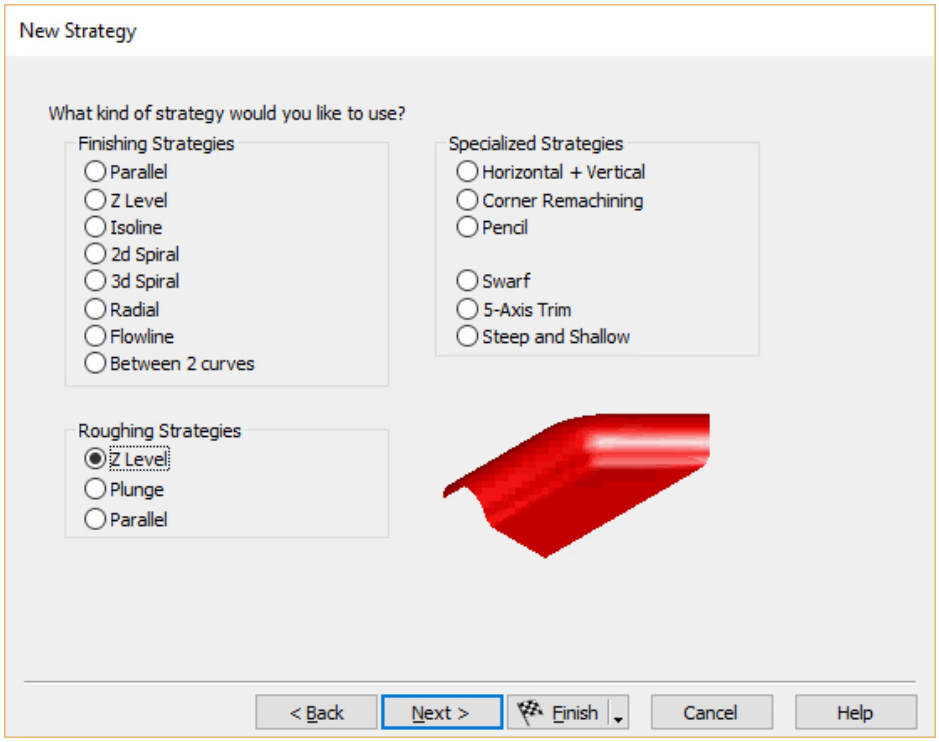
22. Start the New Feature Wizard
23. Select **Surface Milling** then **Next** twice



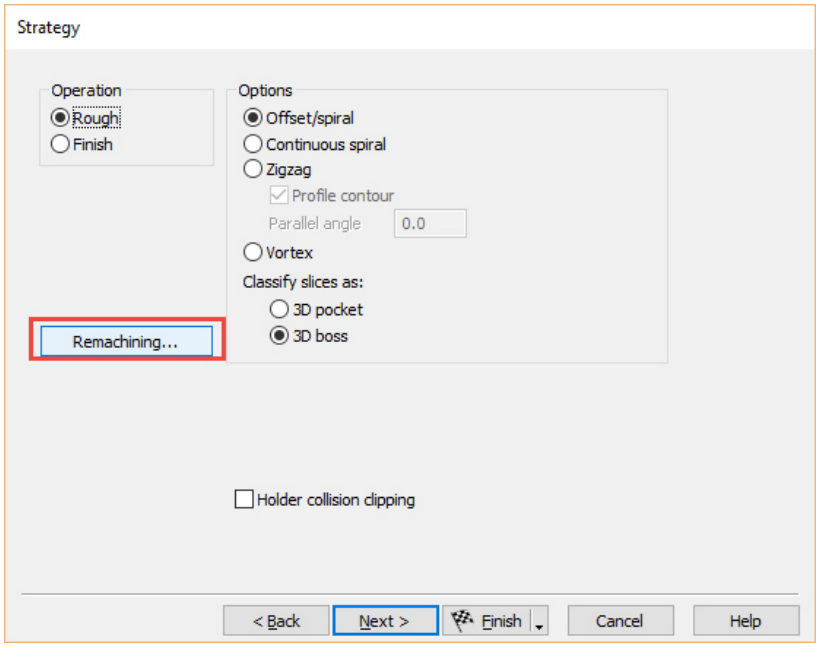
24. Select **Choose a Single Operation**



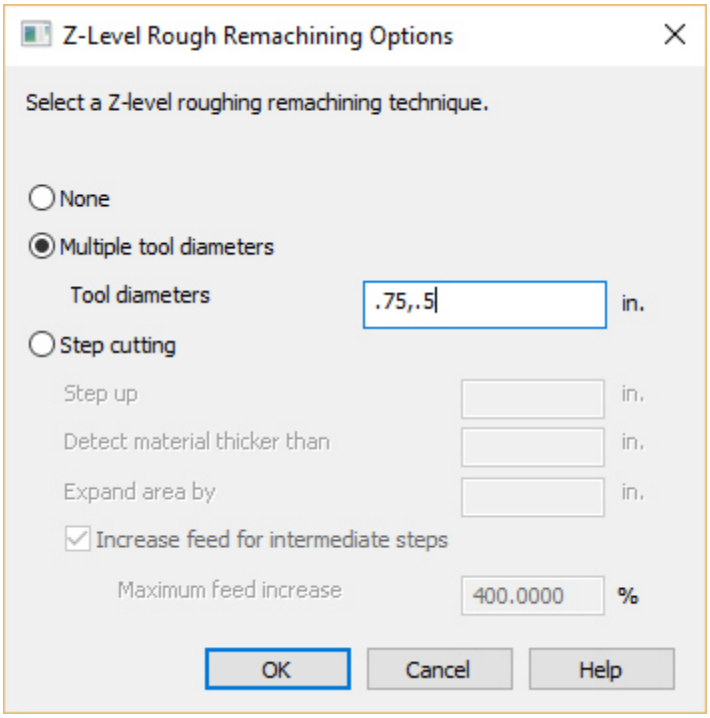
25. On the next page, select **Z level** under **Roughing Strategies,** then click **Next**



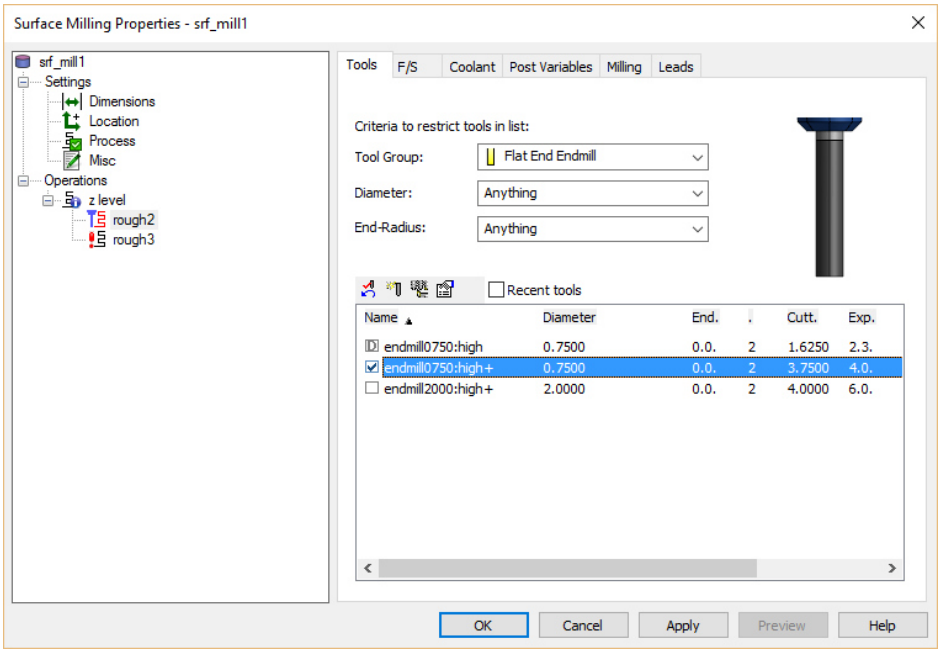
26. Select **Remachining…**



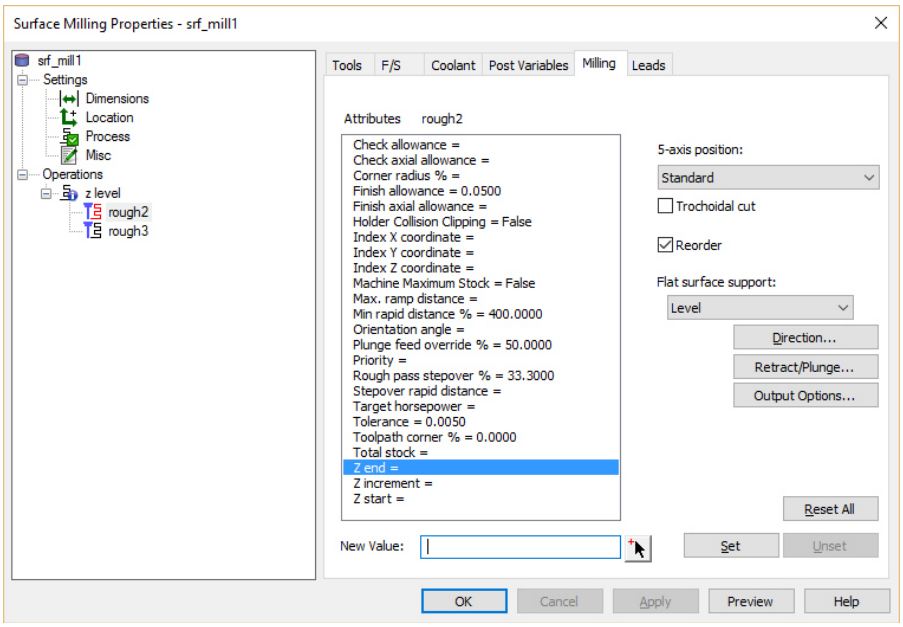
27. Select the **Multiple tool diameters** radio button, then fill in the tool diameters as ".75,.5"



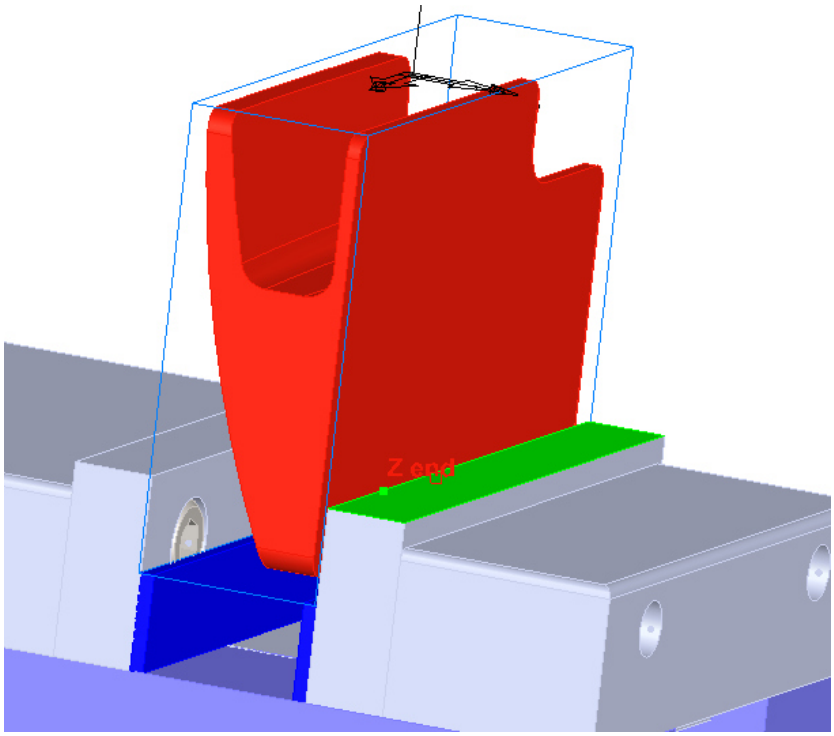
28. Click **OK**
29. Select **Finish**
30. From the Surface Milling feature properties select the first roughing operation and select the tool **endmill0750:high+**



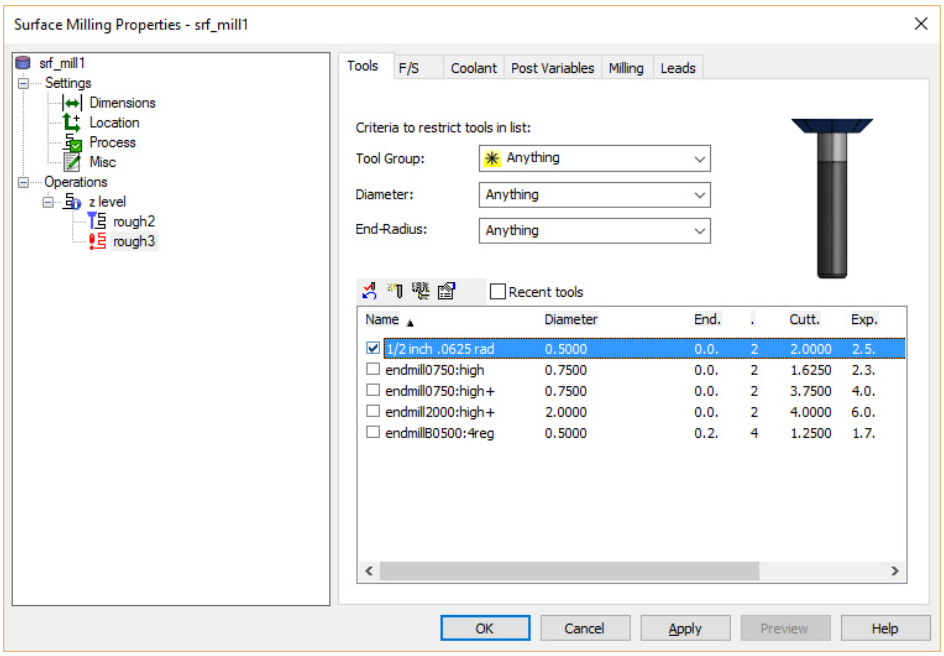
31. Select the **Milling** tab
32. Select the attribute **Z end** and select the pick arrow



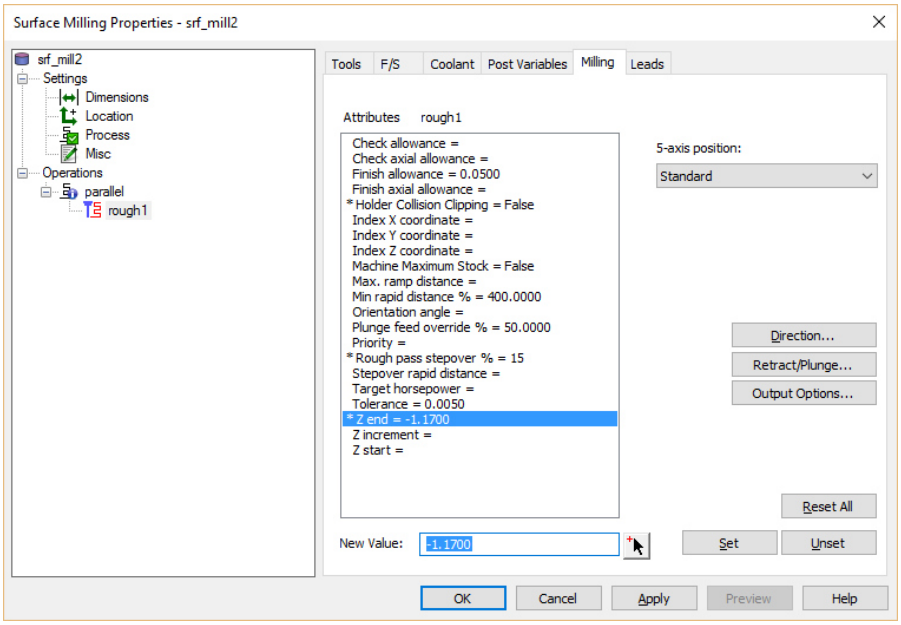
33. Pick the top of one of the jaws, since we do not need the roughing operation to go below this



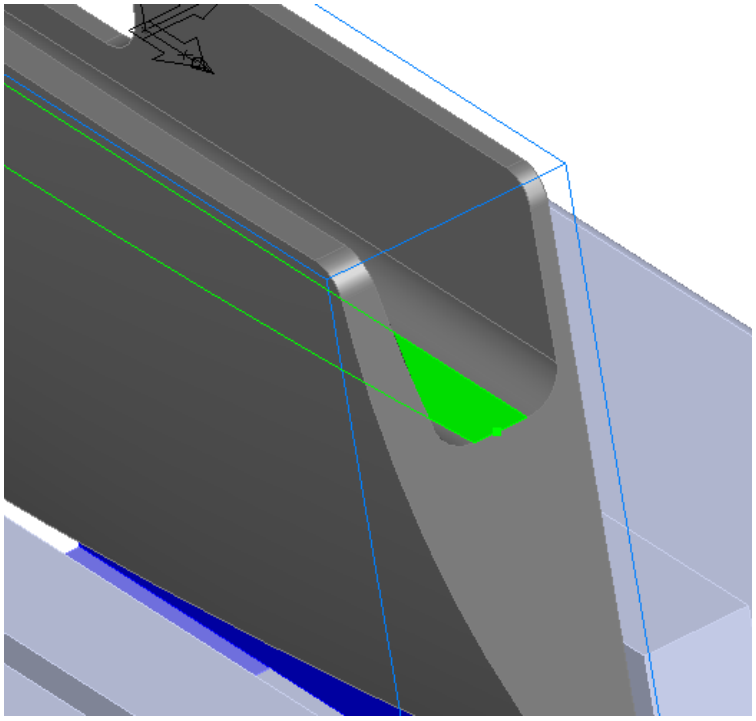
34. Select the second roughing operation and select the tool ½ **inch .0625 rad**



35. Select the **Milling** tab
36. Select the attribute **Z end** and select the pick arrow

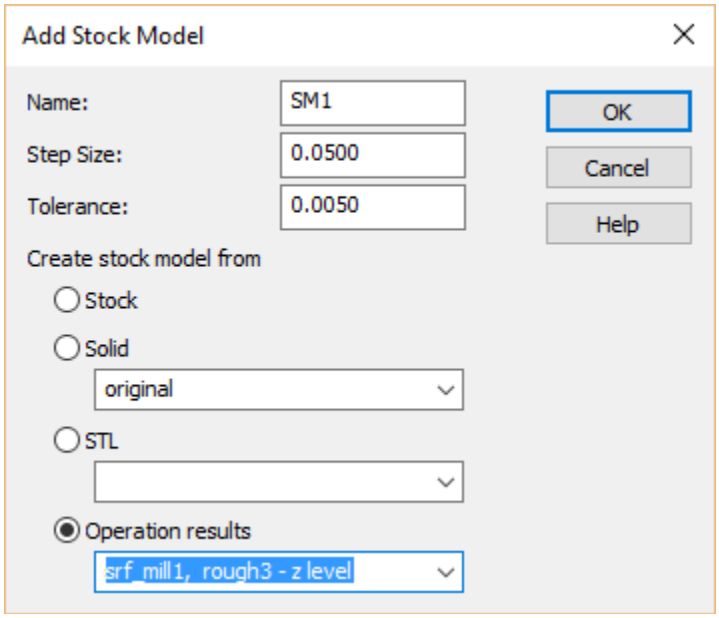


37. Pick the surface shown

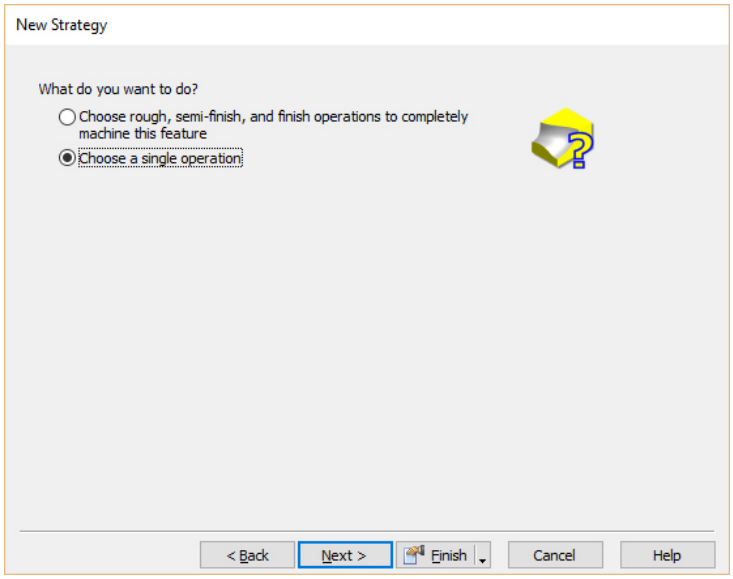


38. Click **OK**
39. Right click on **Stock Models** in the part view and select **Add Stock Model**

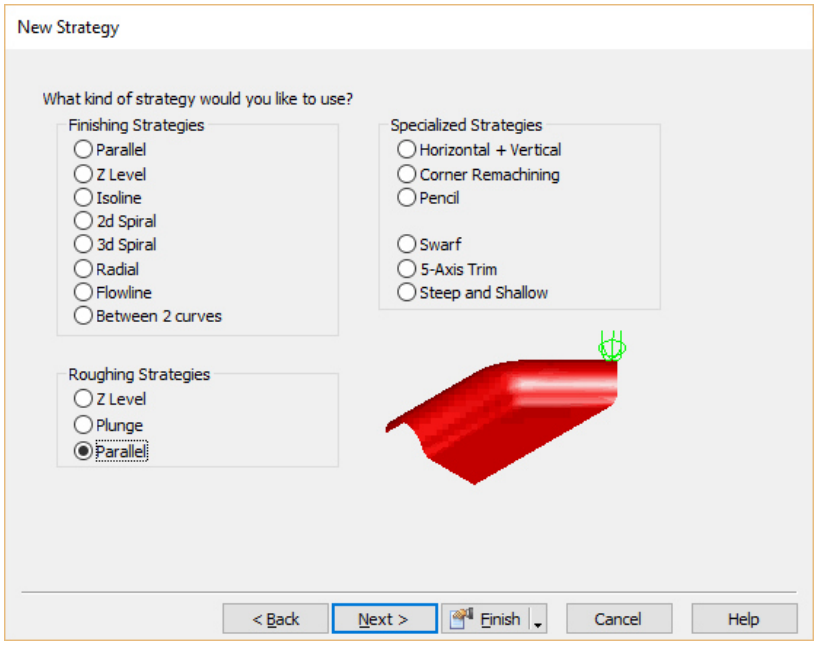
40. Fill in the form as shown and click **OK**



41. Select Setup2 from the part view
42. Select the whole part for milling
43. Start the New Feature Wizard
44. Select **Surface Milling** then **Next** twice
45. **Choose a single operation** then **Next**

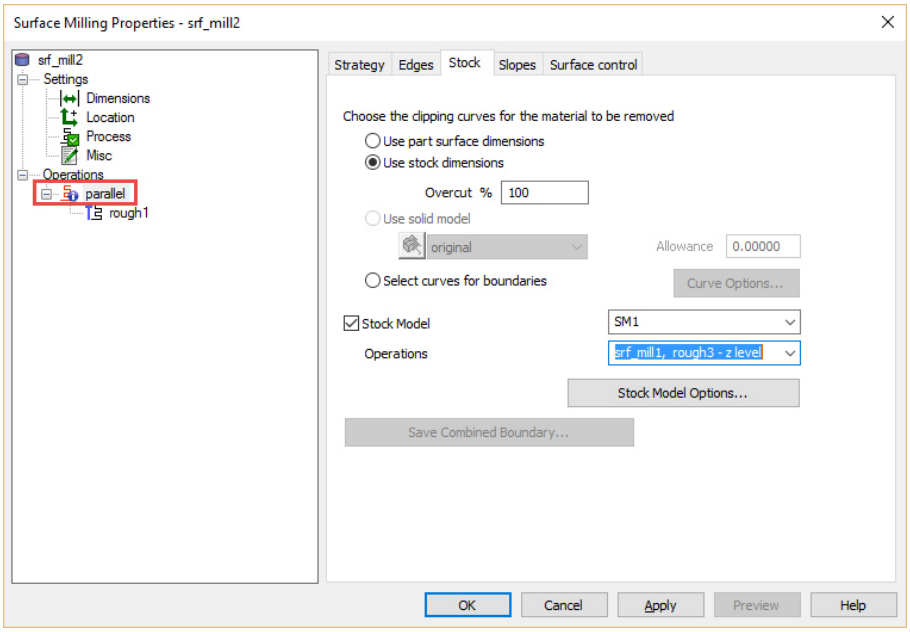


46. Select **Parallel** from **Roughing Strategies**

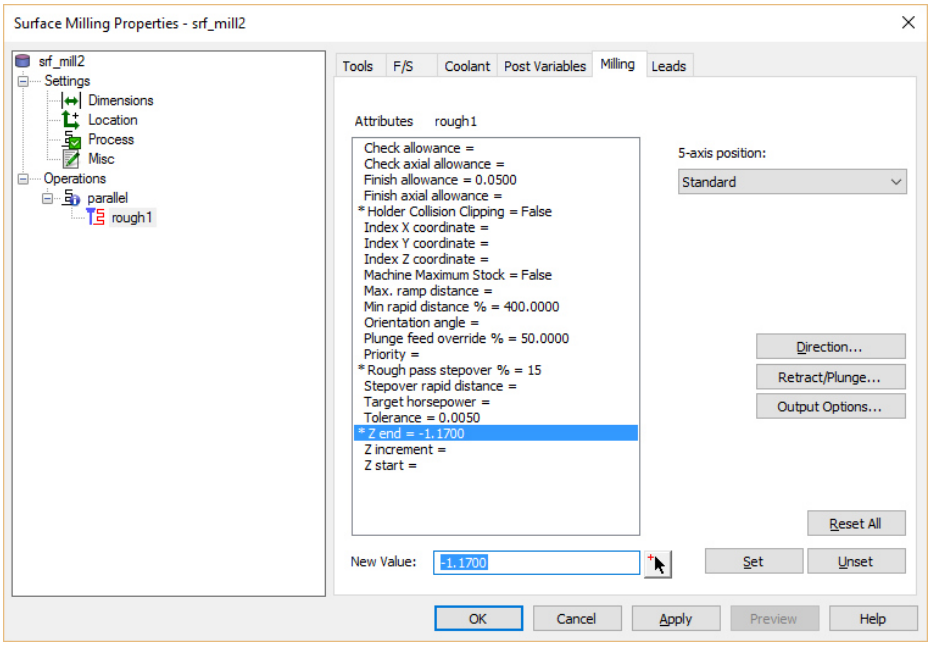


47. Select **Finish**
48. Select **Parallel** from the Feature Properties tree
49. Select the **Stock** tab
50. Fill out the **Stock** tab as shown

This will allow FeatureCAM to reference the current rough stock condition after the first two tools cut for this toolpath so we do not overcut the model in areas that we have already cut

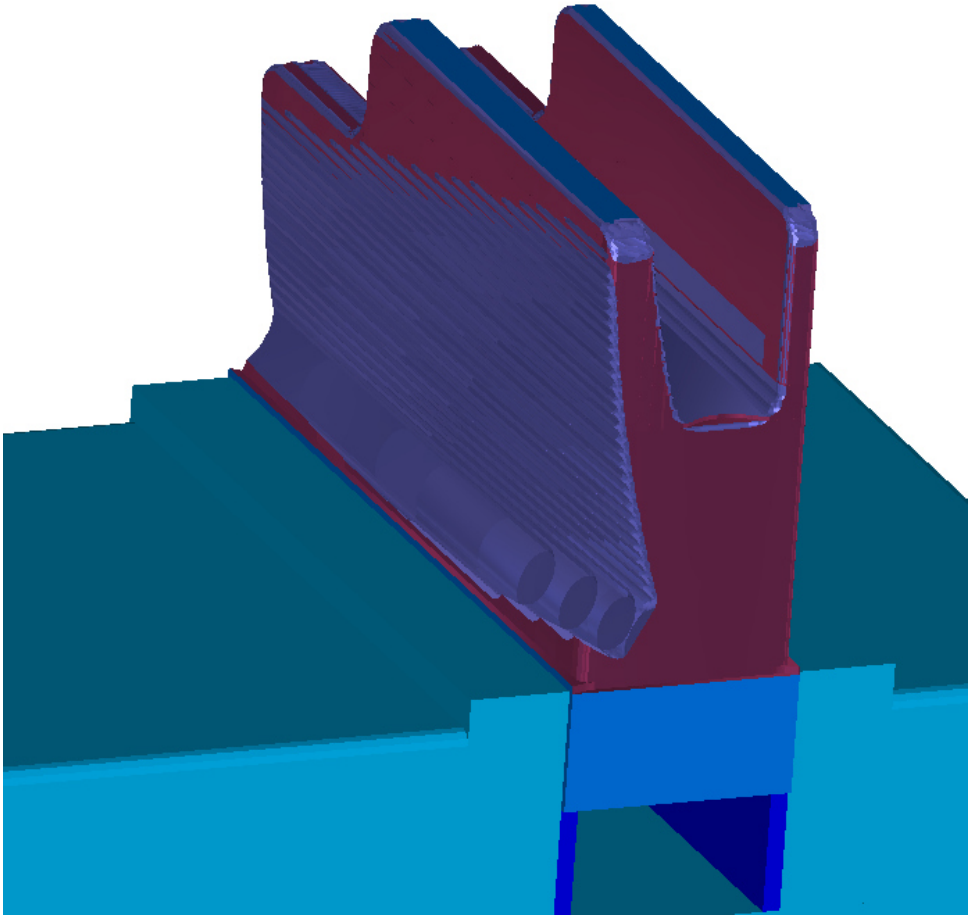


51. Select the roughing operation, then the **Milling** tab
52. Set the **Rough pass stepover %** to 15 and the **Z end** to -1.17

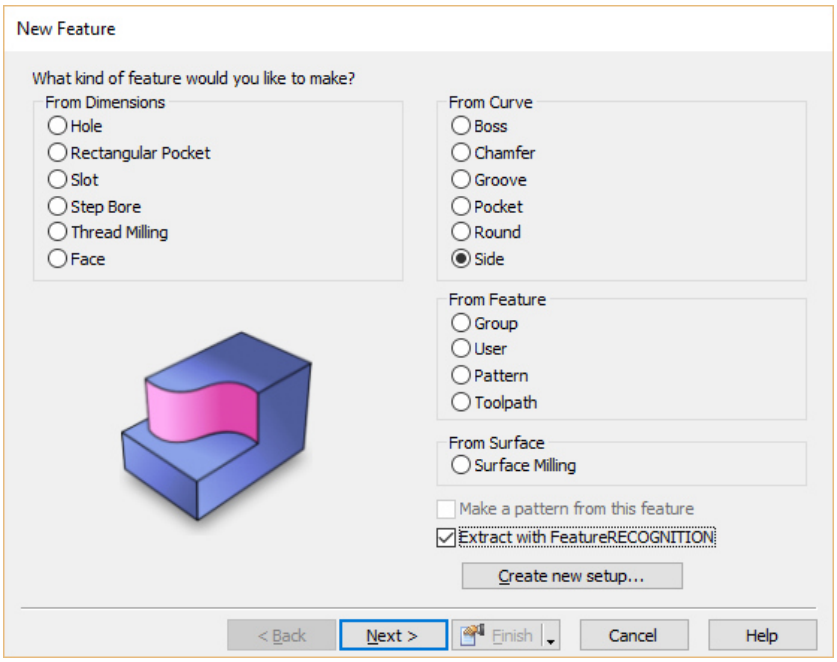


53. Click **OK**

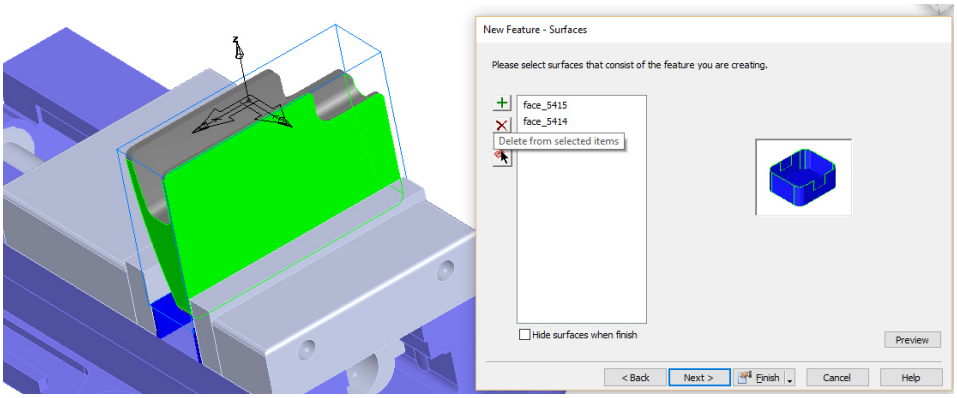
A **3D simulation** now will show you the rough state of your part



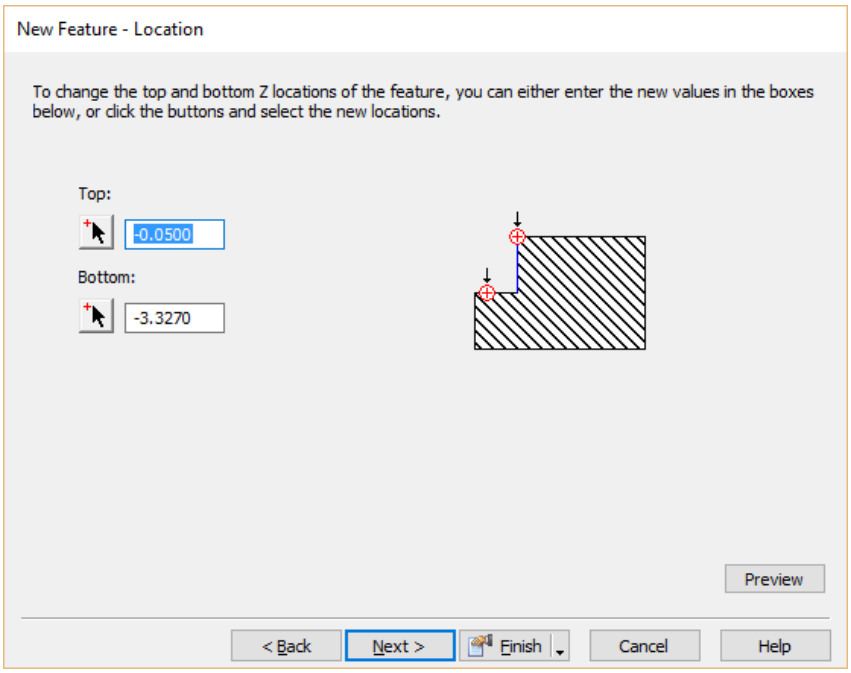
54. Select Setup1 from the part view
55. Start the New Feature Wizard
56. Select **Side** and **Extract with FeatureRECOGNITION** then **Next**



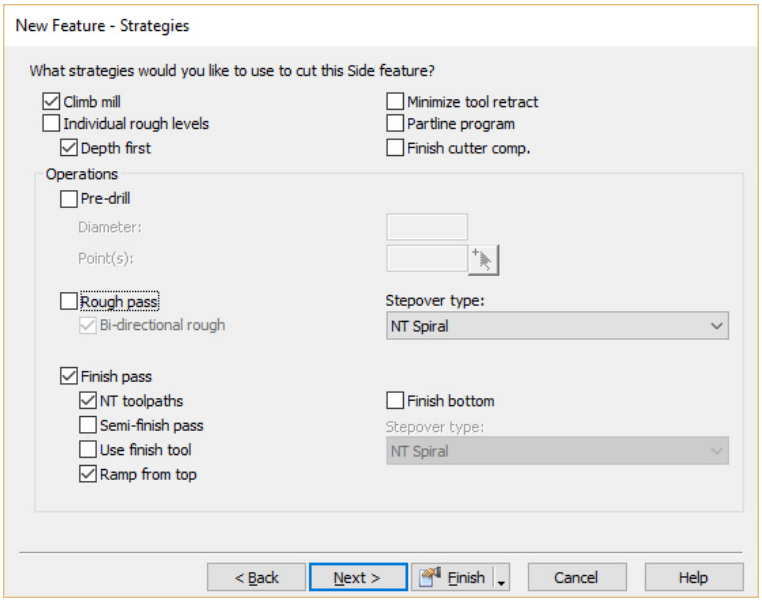
57. **Select side surfaces** then **Next**
58. Select the side surfaces shown then **Next** twice



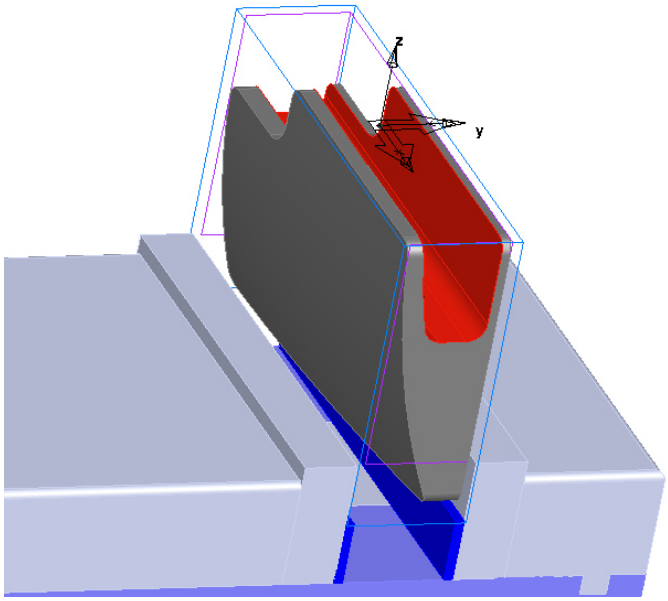
59. Pick the arrow for **Bottom** and select the top of one of the jaws and remove .05 from that value



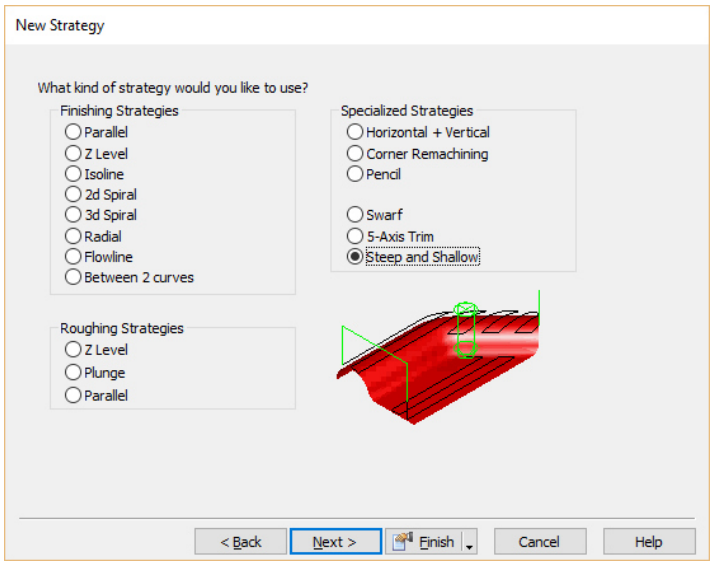
60. Click **Next** twice
61. Uncheck **Rough Pass** since we already roughed the part then click **Finish**



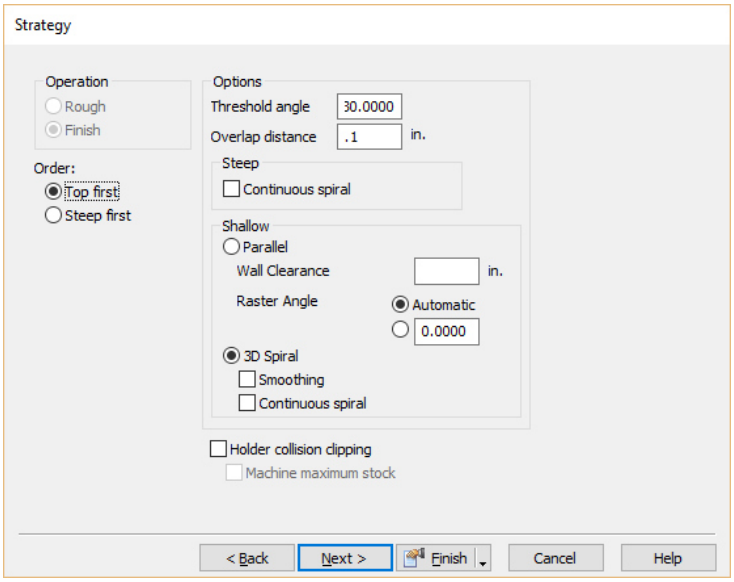
62. Select the surfaces that make up the contoured **inside** of the part



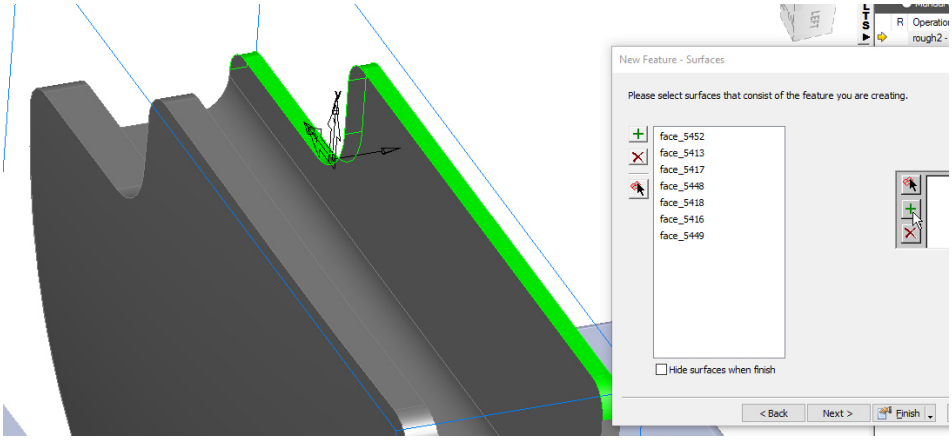
63. Start the New Feature Wizard and select **Surface Milling** then **Next** twice
64. Select **Choose a Single Operation** then **Next**
65. Select **Steep and Shallow** from the **Specialized Strategies** then **Next**



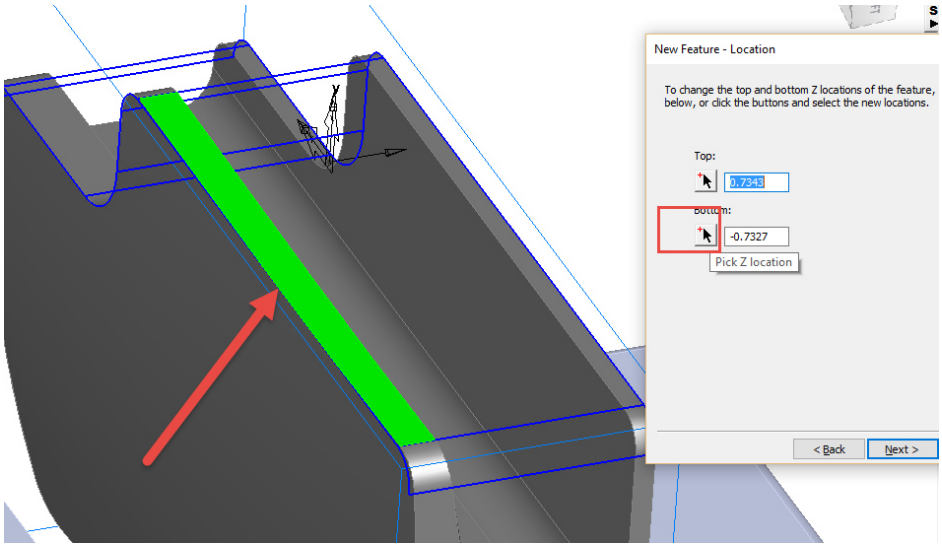
66. Fill in the Overlap distance with .1 then **Finish**



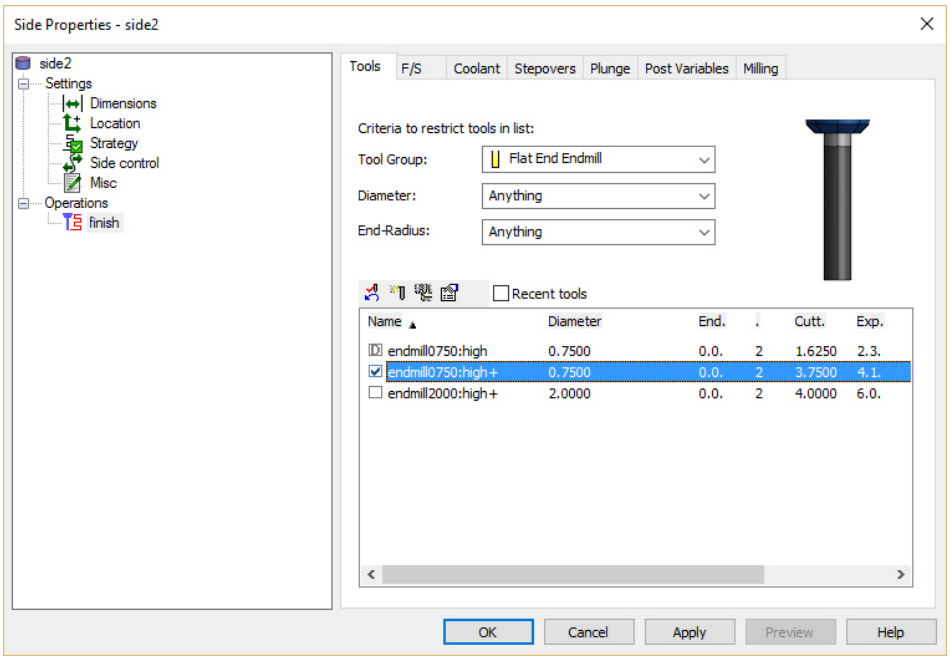
67. Select Setup3 from the Part View
68. Start the New Feature Wizard
69. Select **Side** and **Extract with FeatureRECOGNITION** then **Next**
70. **Select Side Surfaces** Then **Next**
71. Select the surfaces shown



72. Select **Next** twice
73. Pick the **Bottom** from the model as shown then click **Next** twice



74. Uncheck **Rough pass** then click **Finish**
75. Select the Finish operation from the Feature Properties and choose the tool **endmill0750:high+**



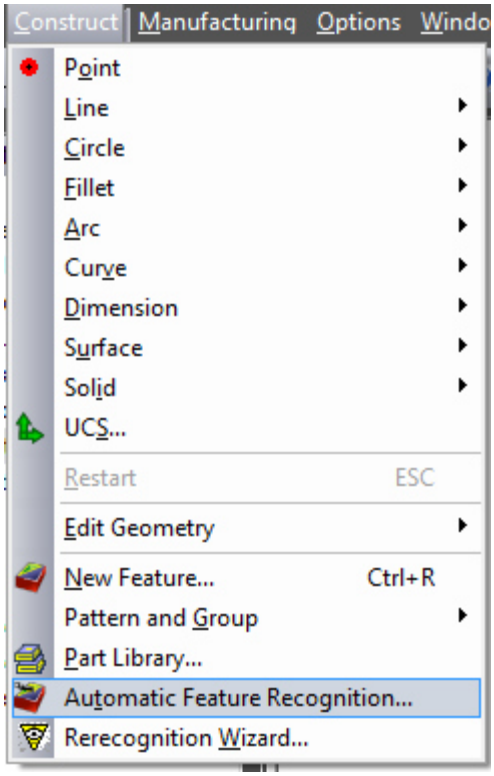
76. Run a 3D simulation. Everything that can be machined from this setup/orientation of the part is completed.

Turn and Turn/Mill

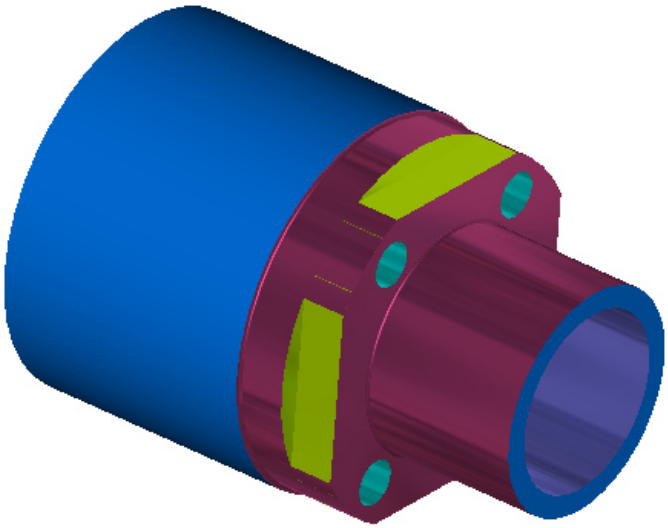
Turning and Turn/Mill in FeatureCAM is just as easy as everything else we have seen so far. FeatureCAM utilizes common terms for turning, such as *turn* and *bore* to ensure that you are able to understand the feature types. You will also see that when you are milling in turn/mill, you

will be working with the same feature types that you have been in all milling documents. FeatureCAM turn and turn/mill has support for single turret, multi-turret, single spindle, sub-spindle, tailstock, steady rest, as well as B-axis heads.

1. Open the part **Hole syncronization.fm**
2. Let's use **Automatic Feature Recognition** to program as much of this as we can
3. Navigate to **Construct → Automatic Feature Recognition**



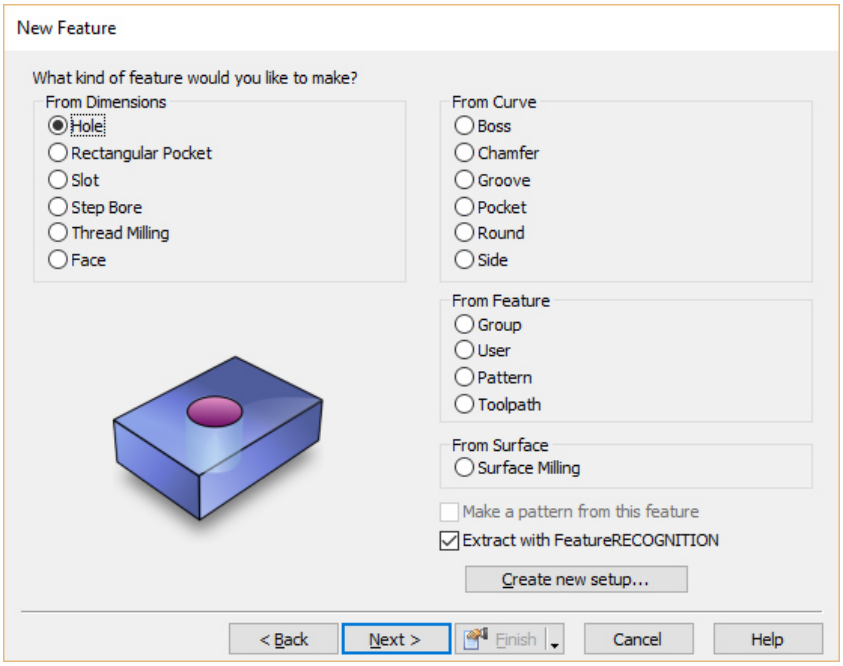
4. Select **Next** twice
5. Click **Finish**
6. FeatureCAM has created features for everything except the holes around the OD of the part



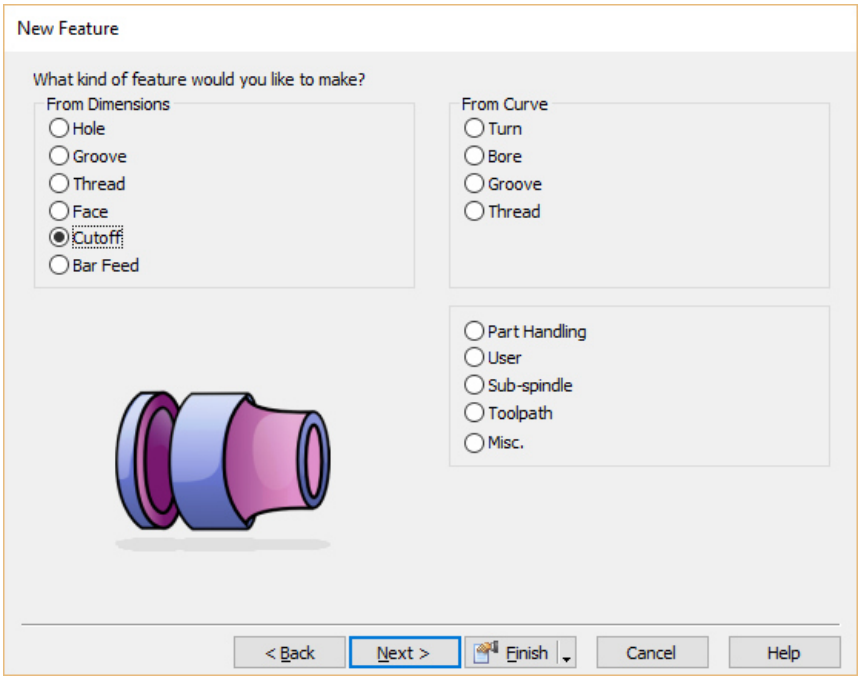
7. Start the New Feature Wizard and select **Turn/Mill** then Next

   Notice that this window is exactly the same as what is used in Milling

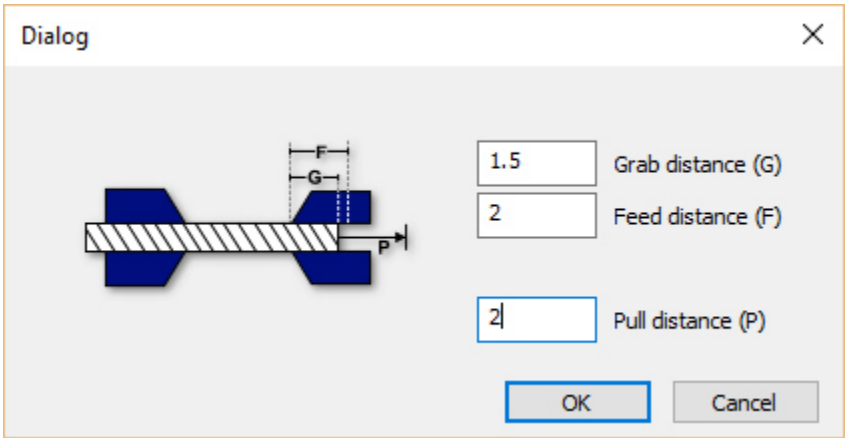
8. Select **Hole** and **Extract with FeatureRECOGNITION** Then **Next**



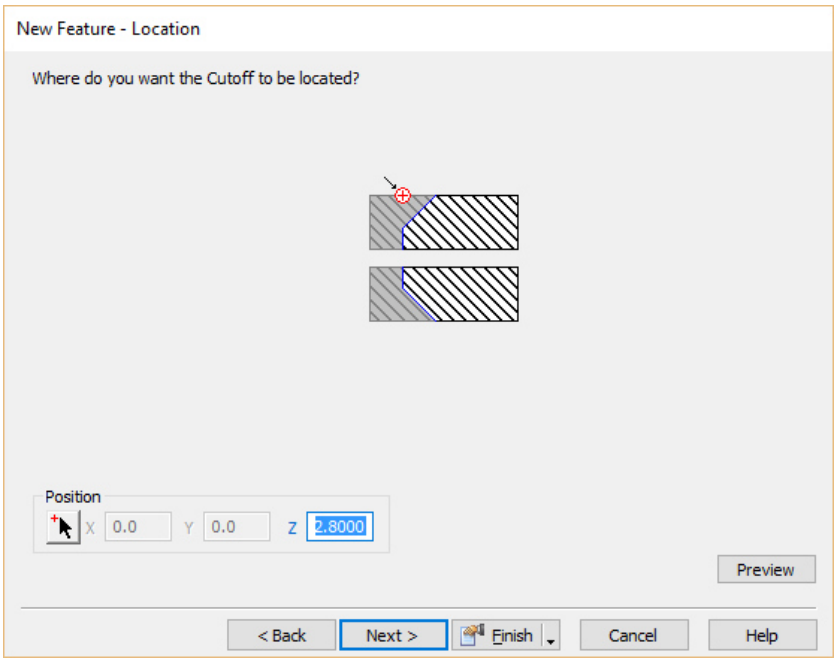
9. Select **Around the index axis** and check the box for **Automatic** then Click **Next** twice
10. Click **Select All** then **Finish**
11. Start the New Feature Wizard and select **Turning** then **Next**
12. Select **Cutoff** then **Next**



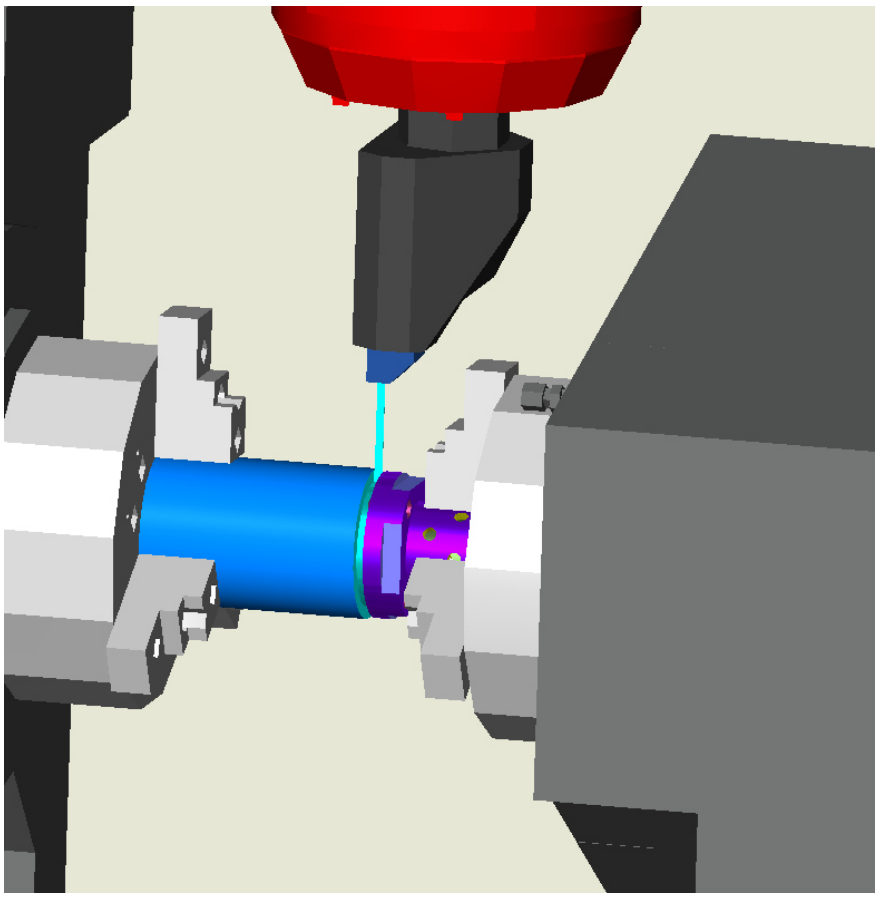
13. Select the **Transfer to sub spindle option** and click on the button **Transfer Parameters**
14. Fill in the Dialog as shown



15. Click **OK** and **Next**
16. Pick the back face of the part then **Finish**



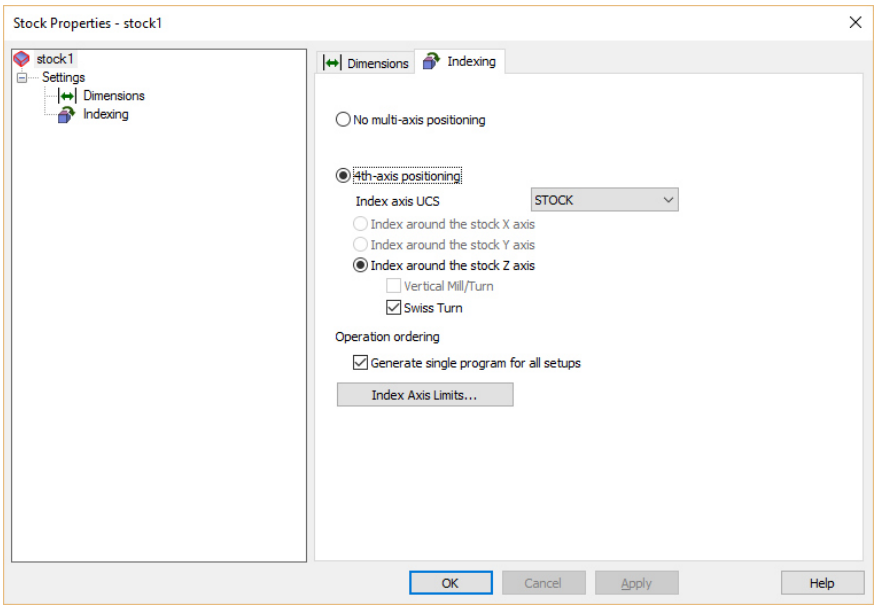
17. A machine simulation will show the part being transferred to the sub spindle while it is cut off



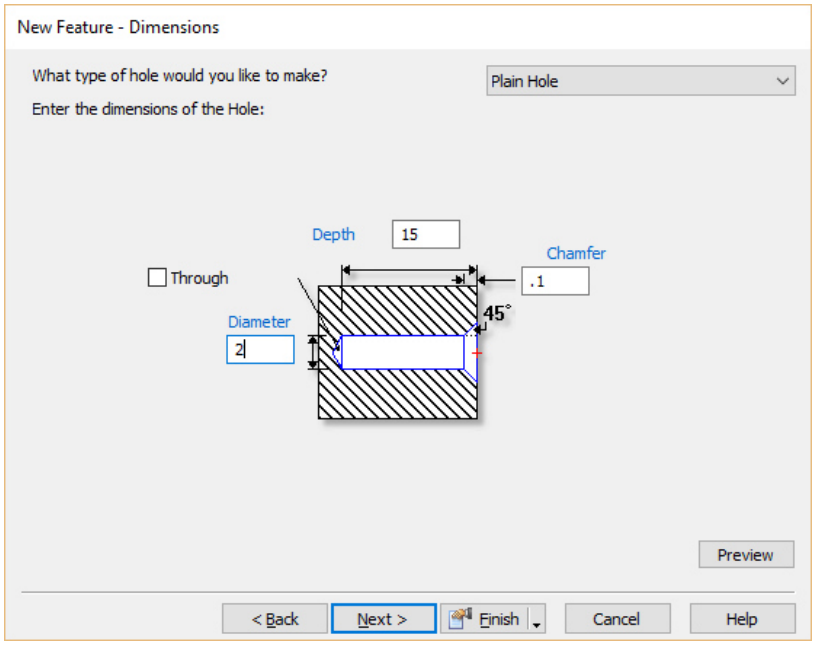
Save this file as we will use it a bit later on when talking about optimizing your turn/mill programs

Swiss Turning

1. Open the file **FCAMSwiss_Example2_Start.fm**
2. Open the **Stock Properties**. One of the things that defines Swiss style machines is the sliding headstock. In FeatureCAM, this is done with the help of the .mtd file, machine simulation, and a checkbox in the Stock Properties, **Indexing** tab. Here you will notice a checkbox for **Swiss Turn**. This is already checked in this file, so no need to do anything here.



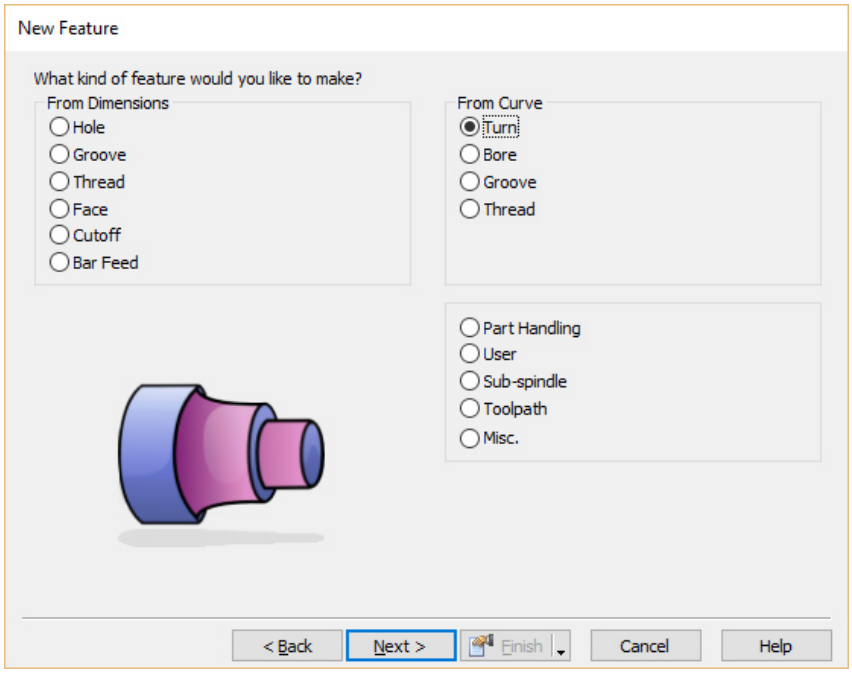
3. Start the **New Feature Wizard** and select **Turning** then **Next**
4. Select **Face** then **Finish**
5. Start another new turning feature and select **Hole**
6. Fill in the dimensions as shown



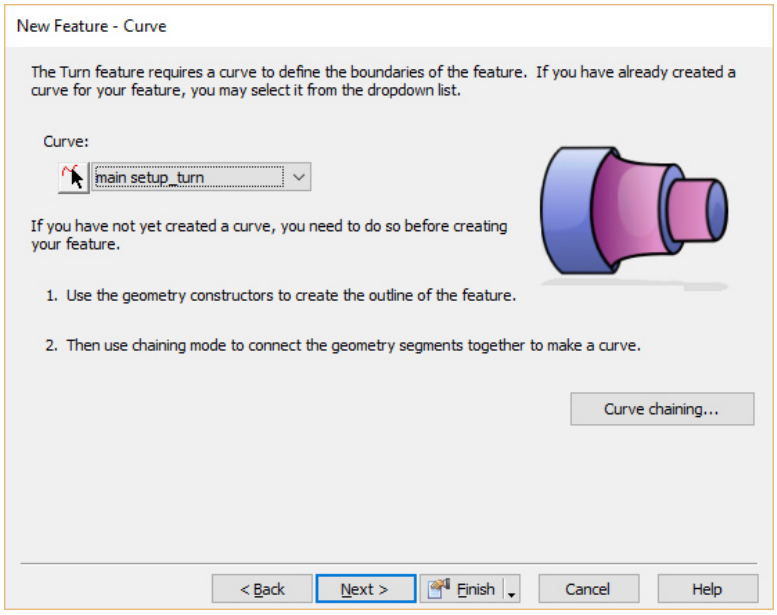
7. **Finish**
8. Notice the  next to the chamfer operation – this means that we have something that we need to fix. If you select the chamfer operation, you will see that FeatureCAM did not pick a tool for this operation. Select the tool **csink82:M0500** and click **OK**

To save time I have pre-created some curves that I extracted in various ways from the model. Some were created using the **Revolved Boundary** method, while others were extracted with **Surface Edges** then projected to flatten them out.

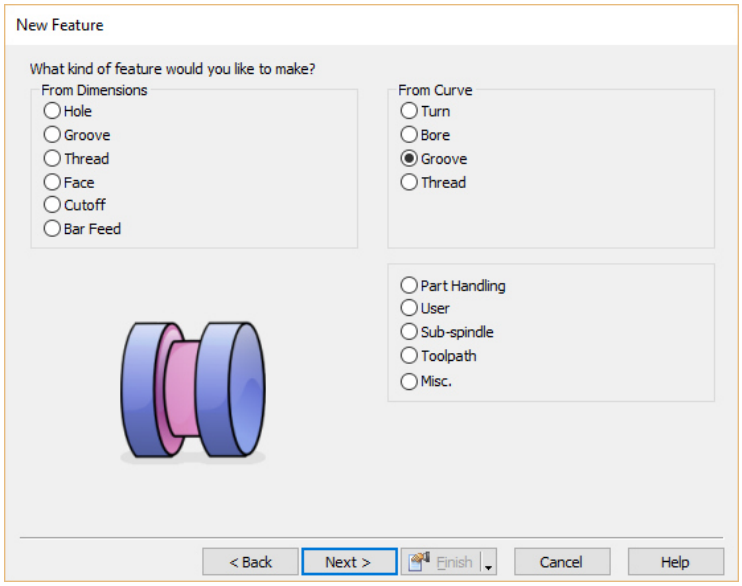
9. Start a new **Turning** Feature and select **Turn** in the From Curve section



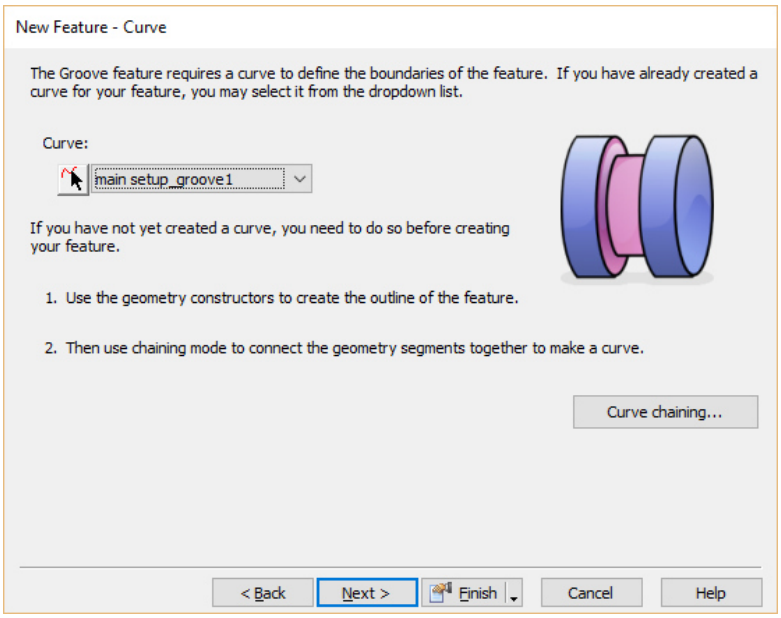
10. Select the curve **main setup_turn** from the dropdown



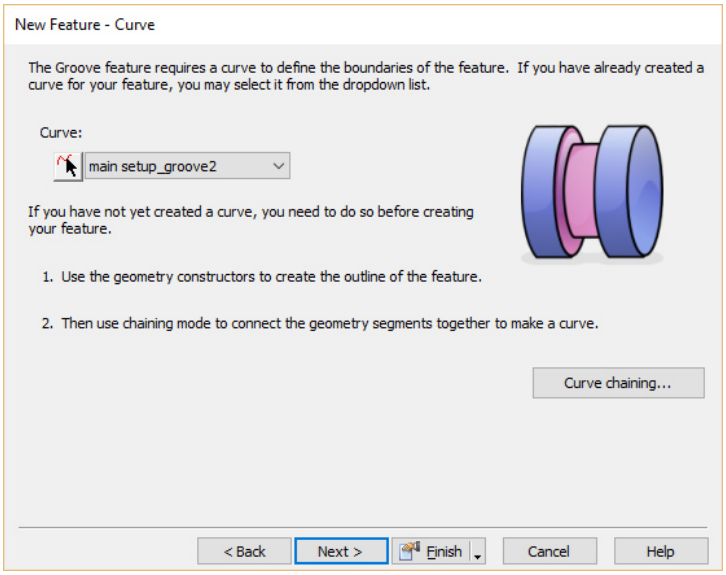
11. **Next** twice.
12. Uncheck **Rough pass** then **Finish**
13. Start the new Feature Wizard, select **Turning,** then **Groove** in the From Curves section



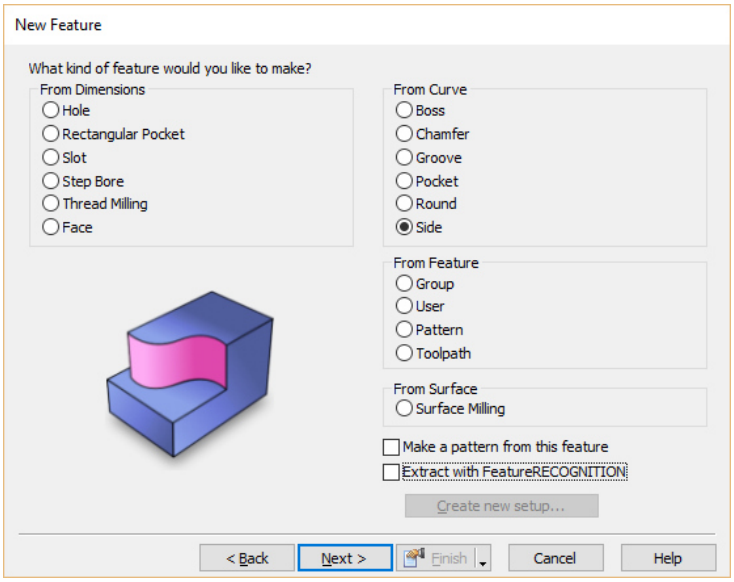
14. **Next**, then select the curve **main setup_groove1** from the dropdown



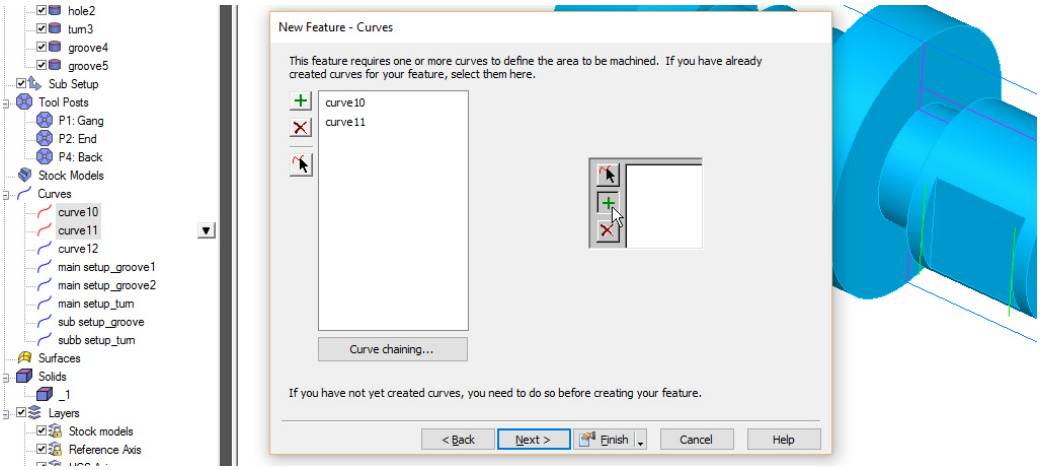
15. **Finish**
16. Repeat this process for the second groove, the curve name is **main setup_groove2**



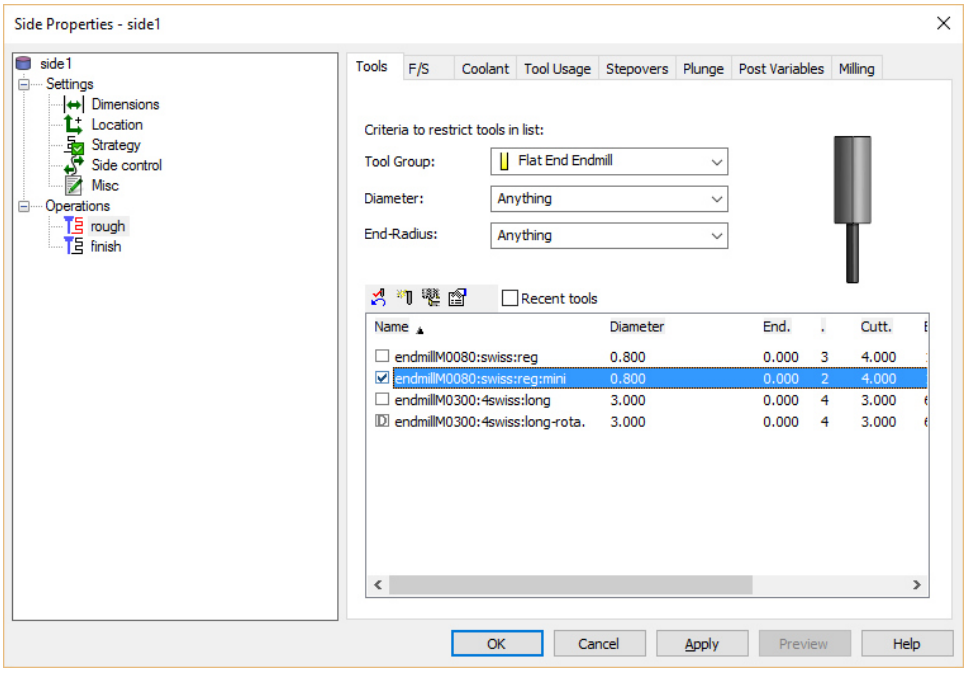
17. Start the New Feature Wizard and select **Turn/Mill** then **Next**
18. Select **Side** and **Next**



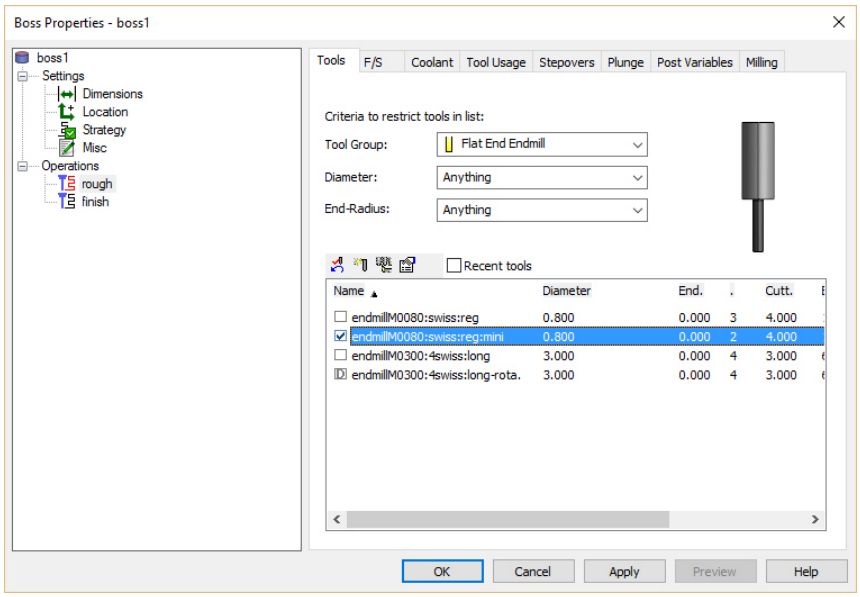
19. From the Part View select **curve10 and curve11** and click  to add them to the list



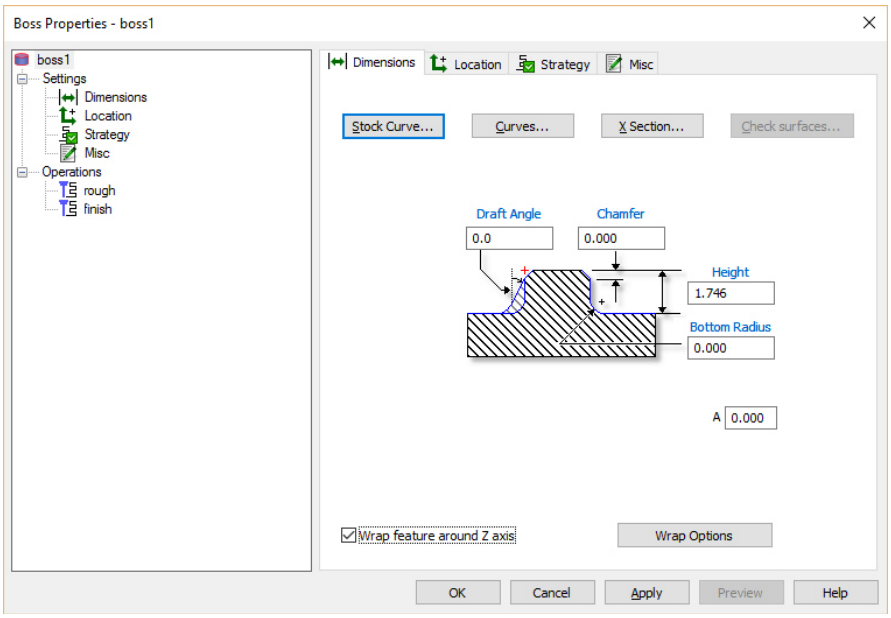
20. **Next** three times
21. Fill in the depth value at **.5mm**
22. Select the rough operation from the feature properties and select the tool **endmillM0080:swiss:reg:mini**



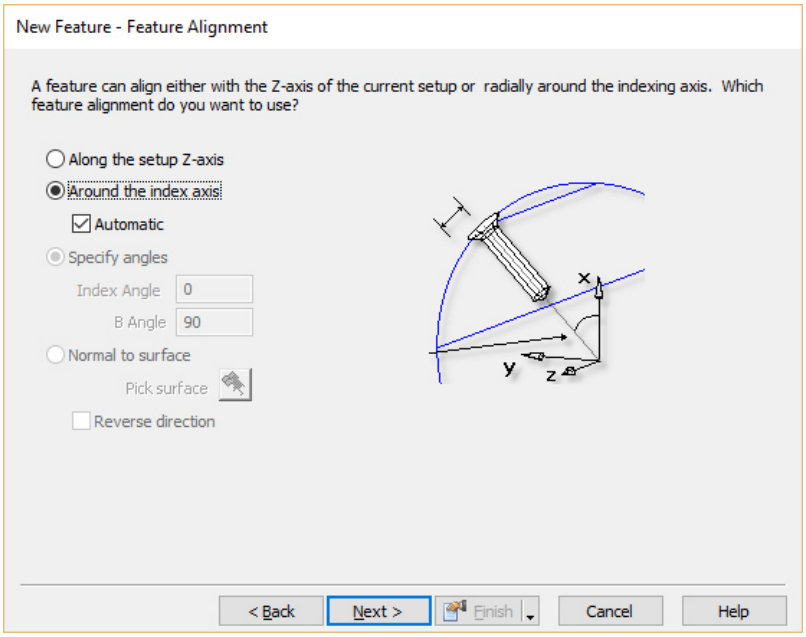
23. Click **OK**
24. Start the New Feature Wizard and select **Turn/Mill** then click **Next**
25. Select **Pocket** and **Next**
26. Select **curve12** from the part view and add it to the feature curves list
27. Click **Next** twice
28. Set the **Height** to **1.746** then **Finish**
29. Select the rough operation from the feature properties and select the tool **endmillM0080:swiss:reg:mini**



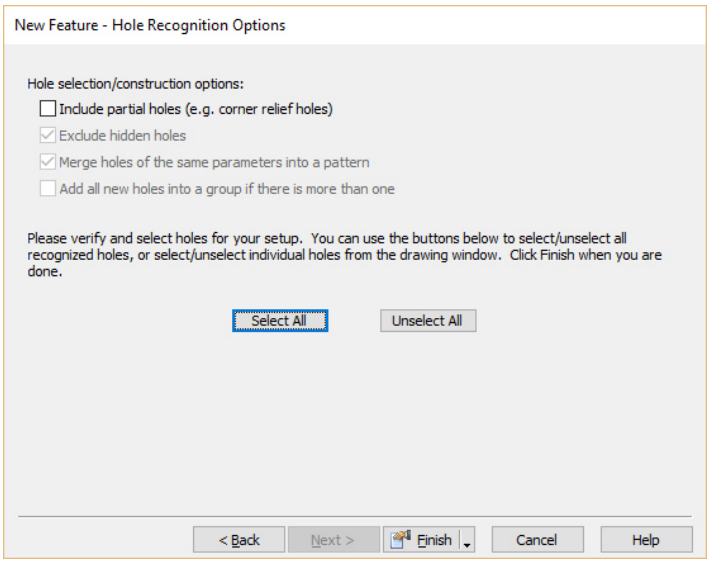
30. From the **Dimensions** tab of the feature properties select the checkbox **Wrap feature around Z axis**



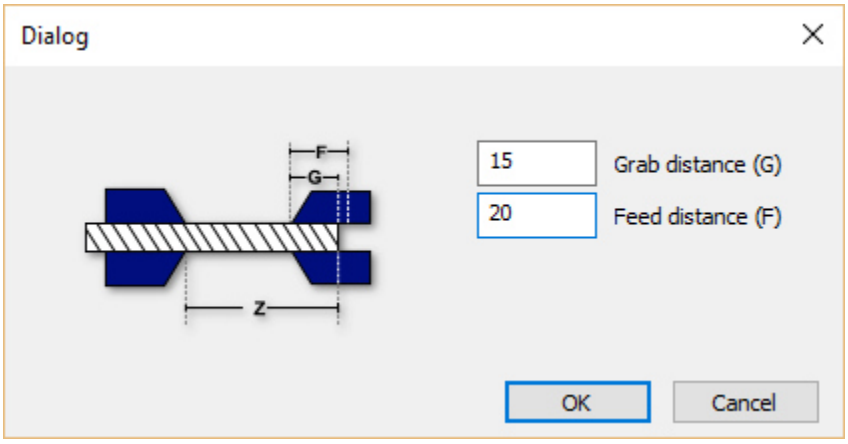
31. Click **OK**
32. Start the new Feature Wizard and select **Turn/Mill** then click **Next**
33. Select **Hole** and **Extract with FeatureRECOGNITION**
34. Select **Around the index axis** and check **Automatic**



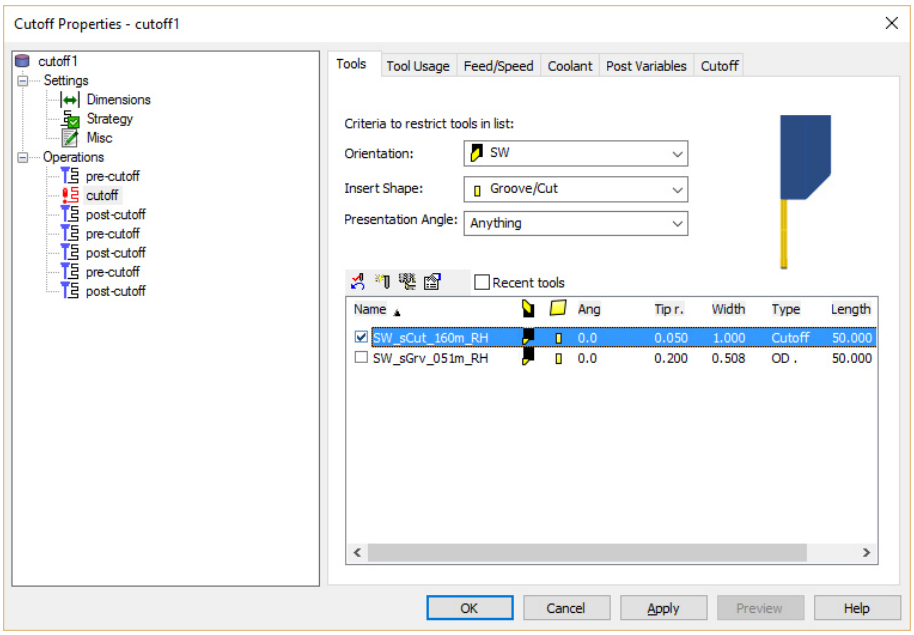
35. Select **Next** twice and click **Select All**



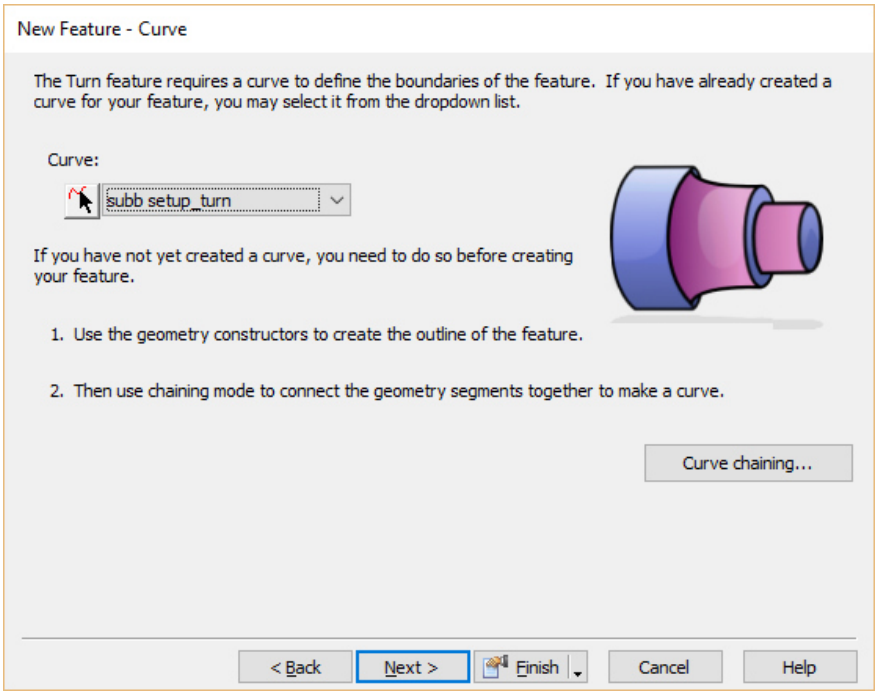
36. Click **Finish**
37. Start the New Feature Wizard and select **Turning** then **Next**
38. Select **Cutoff** then **Next**
39. Check the box **Transfer to sub spindle** and fill in the **Transfer Parameters** as shown



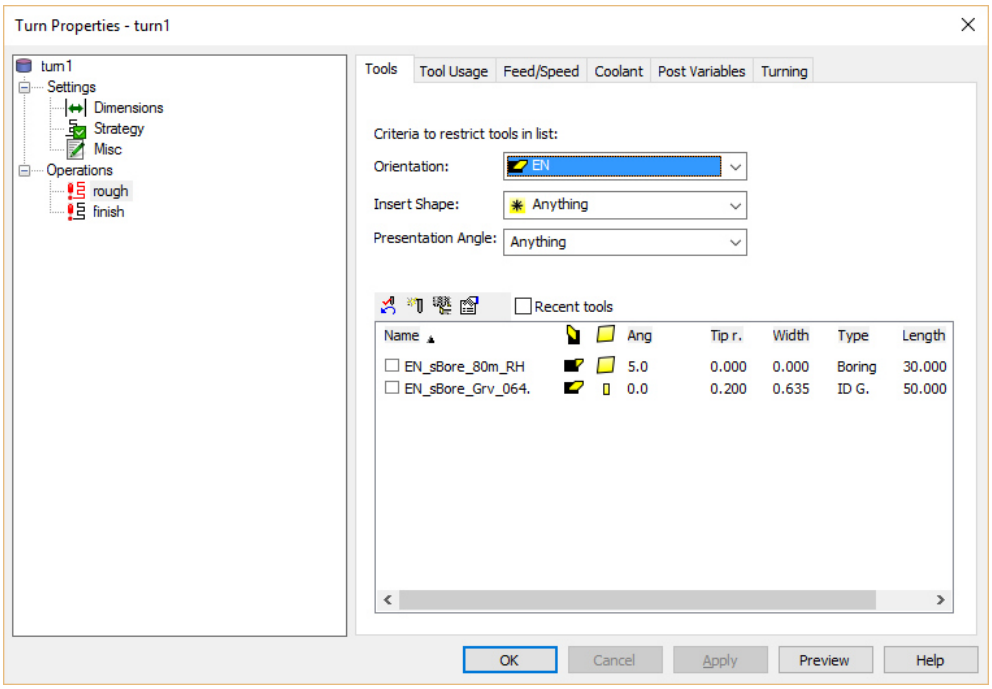
40. Click **Next** and set the **Z** Position to -25mm then click **Finish**
41. Select the cutoff operation from the feature properties and select the tool **SW_sCut_160m_RH**



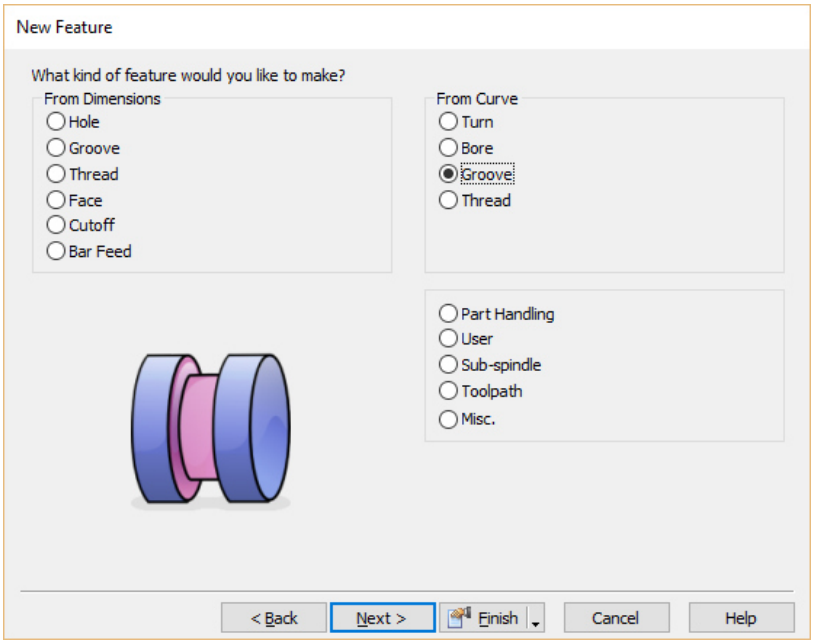
42. Click **OK**
43. Select the setup **Sub Setup** from the Part View
44. Start the New Feature Wizard and select **Turning** then click **Next**
45. Select **Turn** then **Next**
46. Select the curve **subb setup_turn** from the dropdown



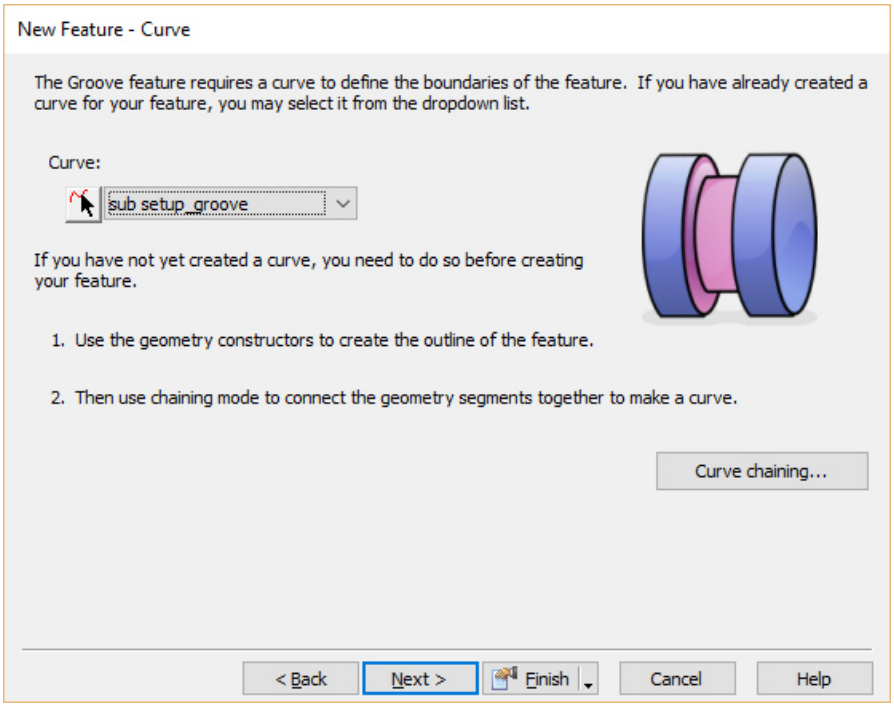
47. Click **Finish**
48. Select the rough operation from the feature properties and change the tool Orientation dropdown to **EN**
49. Select the tool **EN_sBore_80m_RH**



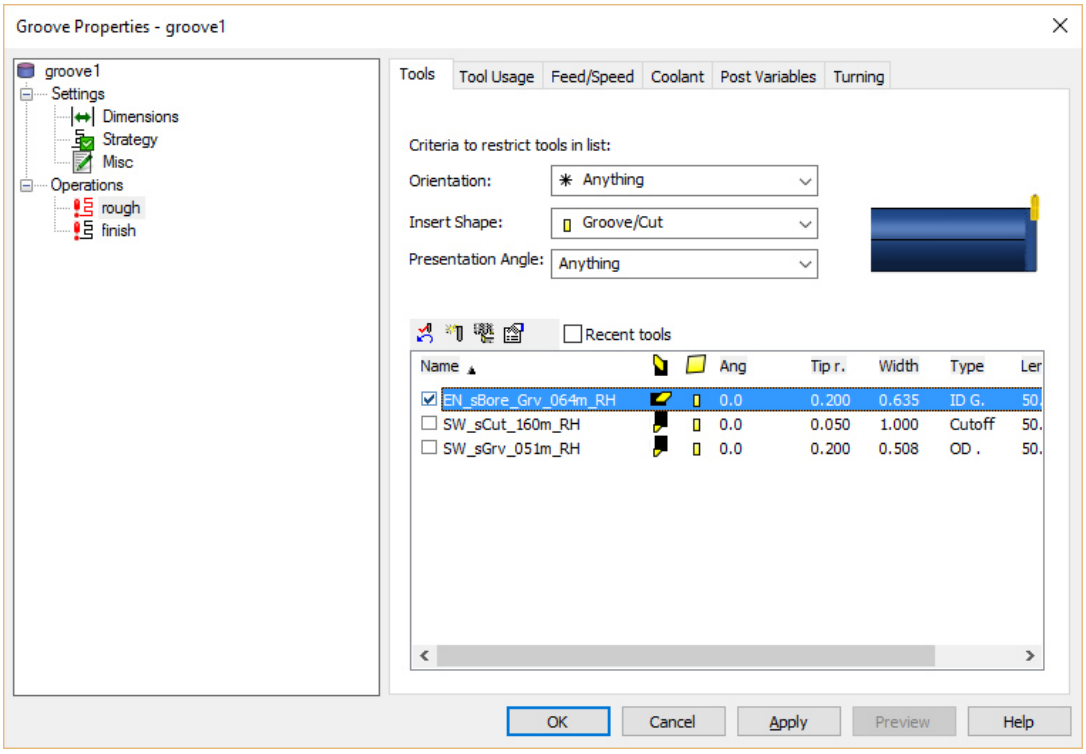
50. Click **OK**
51. Start the New Feature Wizard and select **Turn** then **Next**
52. Select **Groove** in the From Curve section then **Next**



53. Select the curve **sub setup_groove** from the dropdown then **Finish**

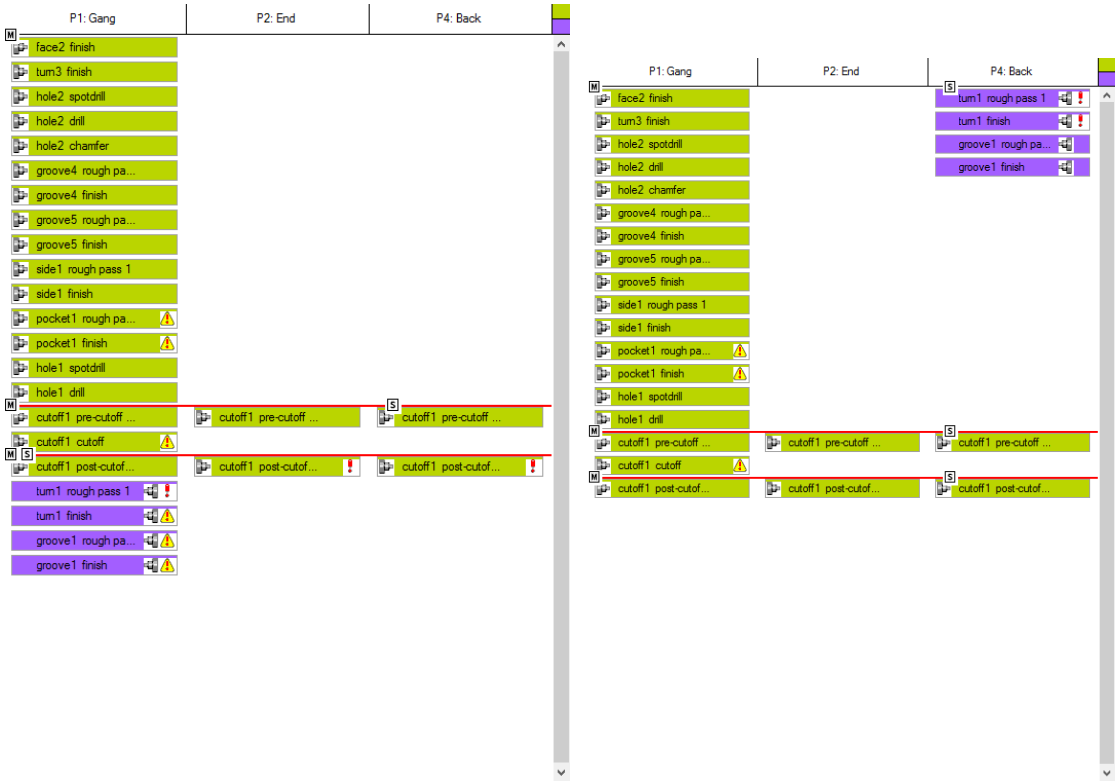


54. Select the rough operation from the feature properties and select the tool **EN_sBore_Grv_064m_RH**

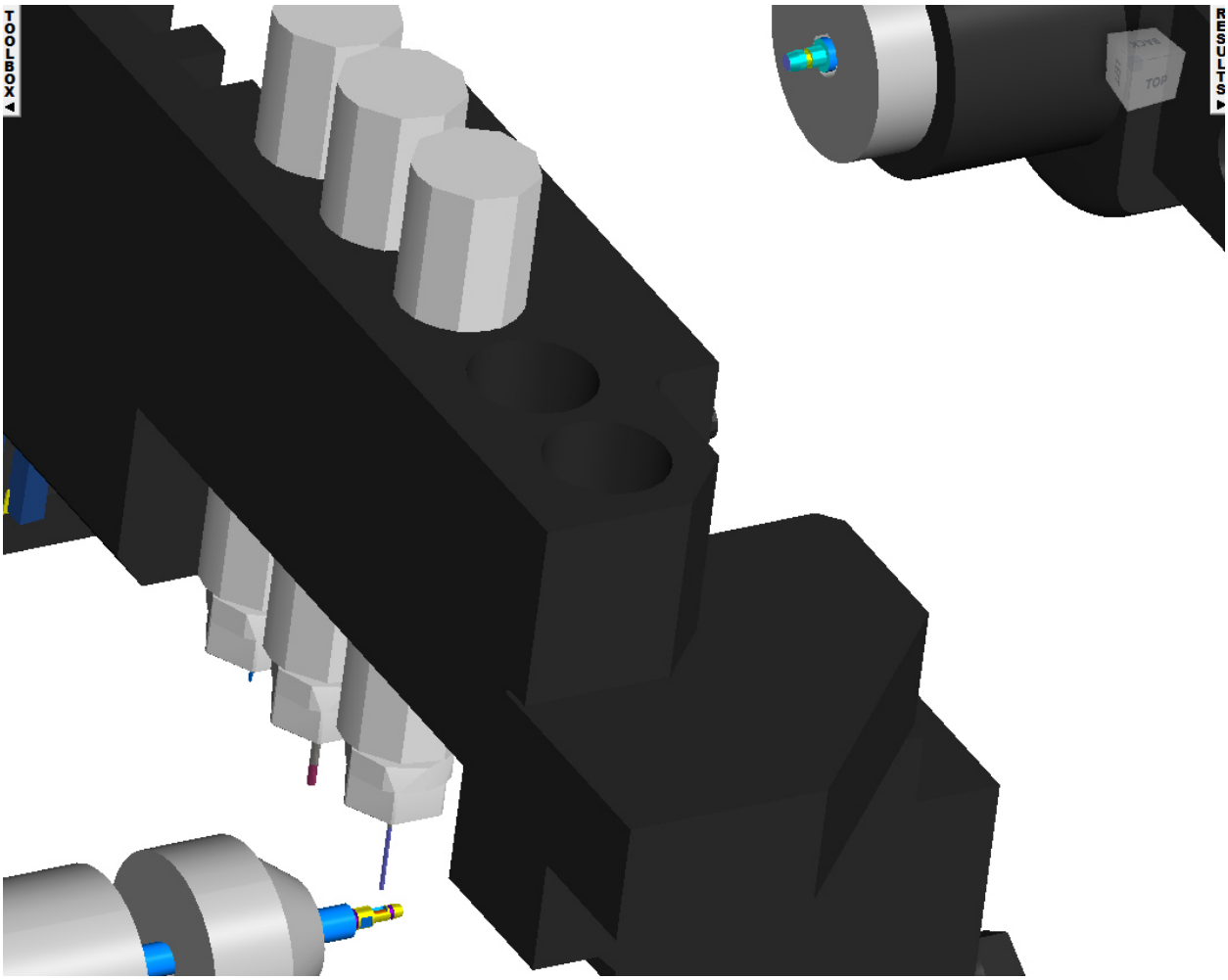


55. Click **OK**
56. Open the **Tool Posts** tab from the **Results** window on the right side of the FeatureCAM Interface.

57. Select the 4 sub spindle operations (colored in purple) and drag them to the back working tool post (P4)



58. Run a **3D simulation** and, if desired, a Machine Simulation

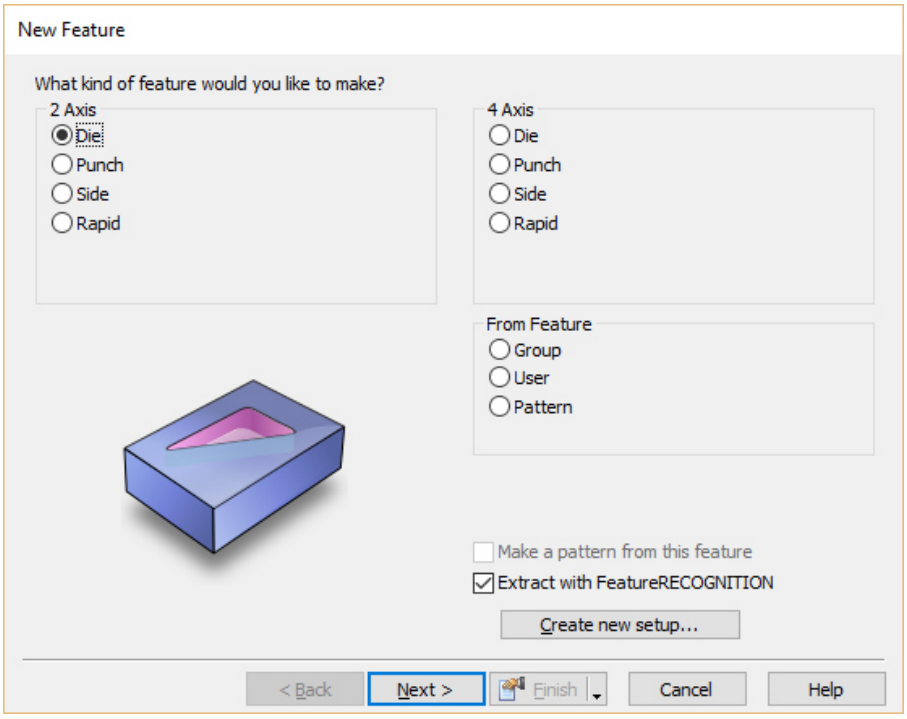


Wire Electrical Discharge Machining (EDM)

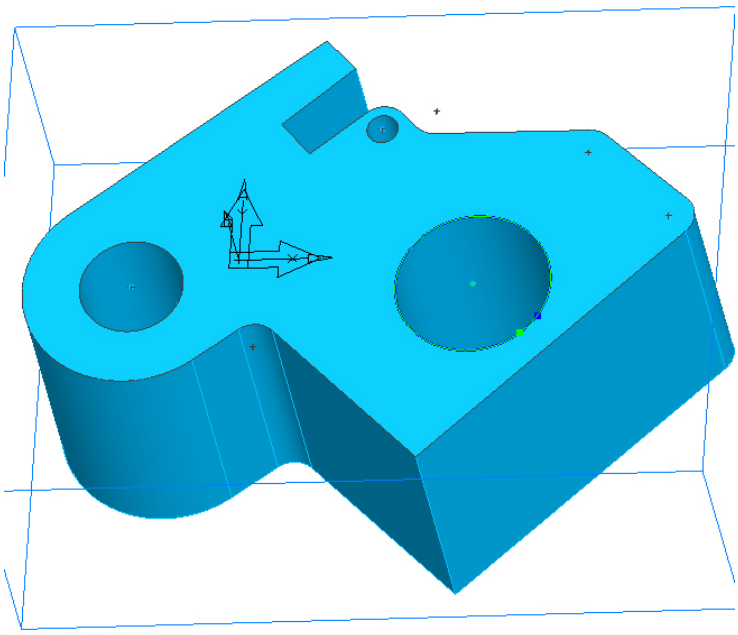
Programming wire EDM in FeatureCAM is as seamless as everything else we can done this far. Again, FeatureCAM utilizes standard industry terms such as Die, Punch, and Side for the feature types available. FeatureCAM is capable of programing strait 2 axis wire EDM features as well as tapered wall (4-axis) features.

2-axis

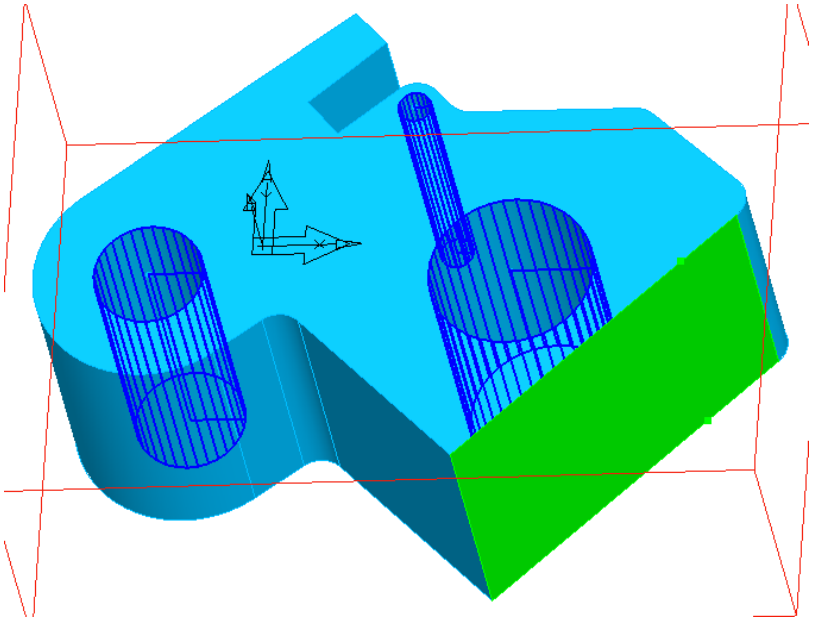
1. Open the file **FW_2Axis(FR).fm**
2. Start the New Feature Wizard and select **Die and Extract with FeatureRECOGNITION**



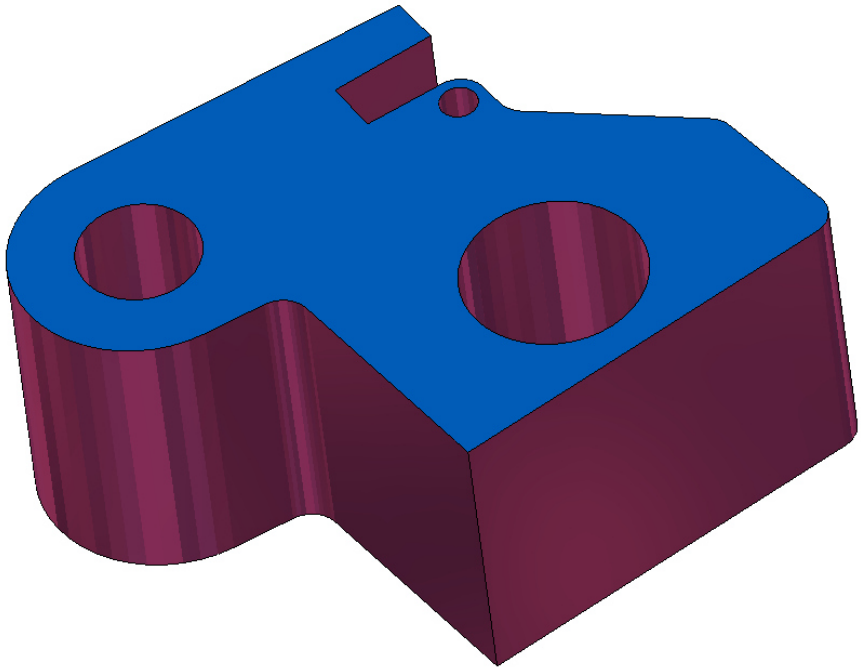
3. Select **Next** twice
4. FeatureRECOGNITION will generate geometry for us so that we can create curves to define the boundaries of our die cuts. Select the large circle then click **Create**



5. Pick the other large circle and click **Create** again
6. Finally click the small hole and click **Create** again
7. Select **Next** twice
8. Pick the blue word **Thickness**
9. Pick the top and bottom of one of the side wall or the top and bottom surface to pick the thickness of the feature

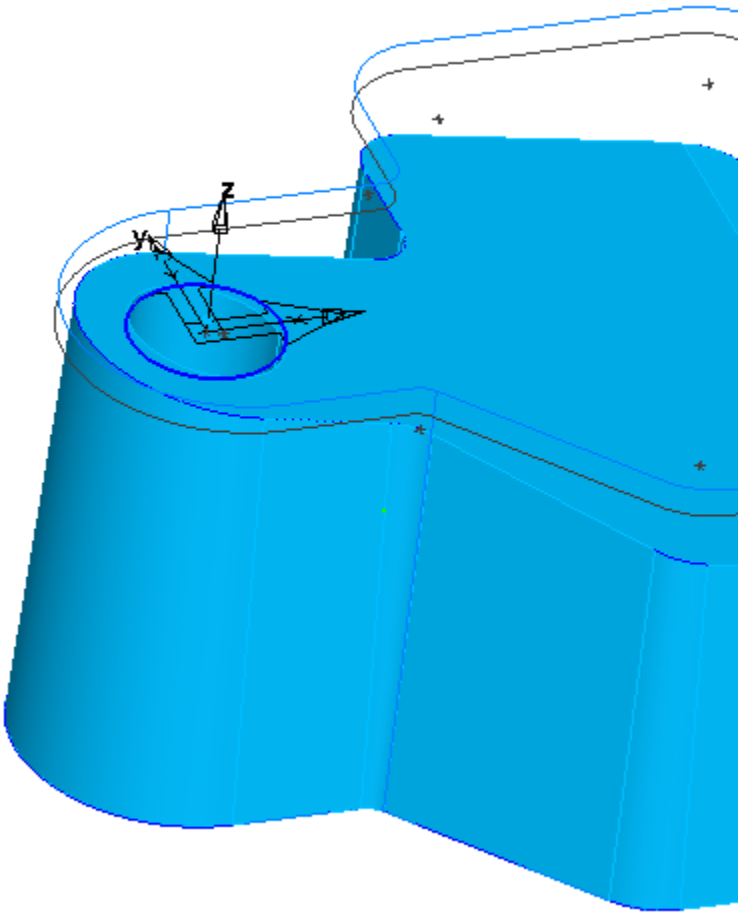


10. Click **Finish** and **OK**
11. Start the New Feature Wizard and select **Punch** and **Extract with FeatureRECOGNITION**
12. **Next** twice
13. Select the geometry that makes up the outside profile to chain the feature curve then select **Next**
14. Select **Finish**
15. Run a 3D simulation – to remove the cut pieces simply click on them

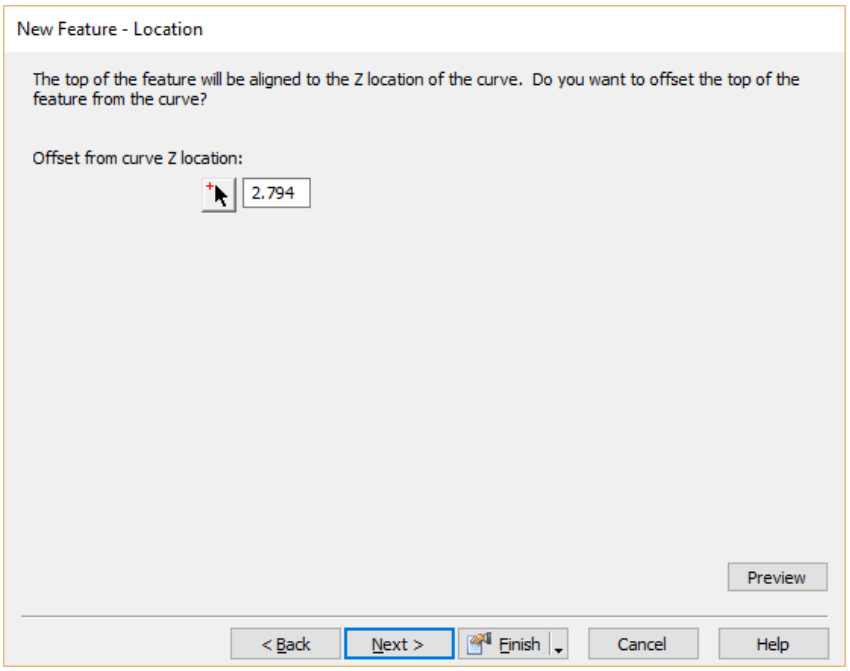


4-axis

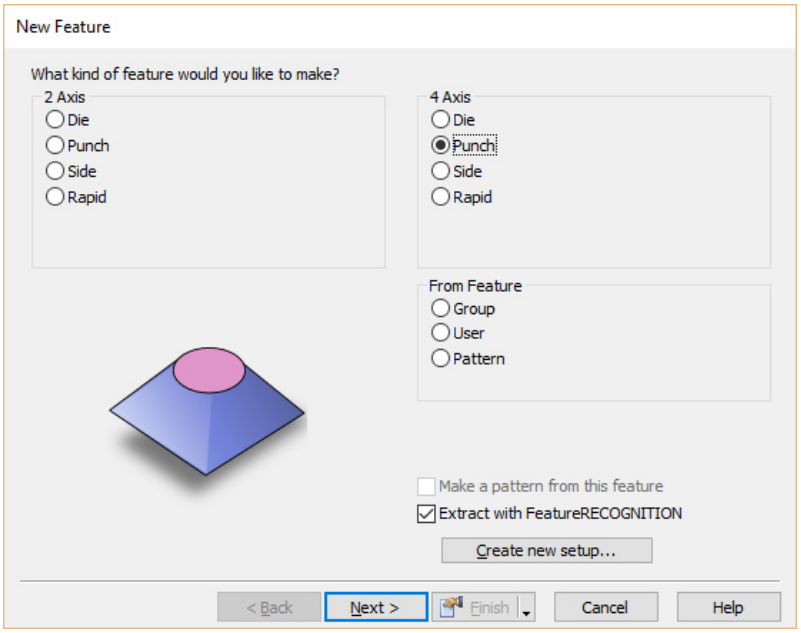
1. Open the part **FW_4Axis.fm**
2. Start the New Feature Wizard and select **Die** and check the box **Extract with FeatureRECOGNITION** then **Next** twice
3. Select the circle that makes up the hole then click **Next**



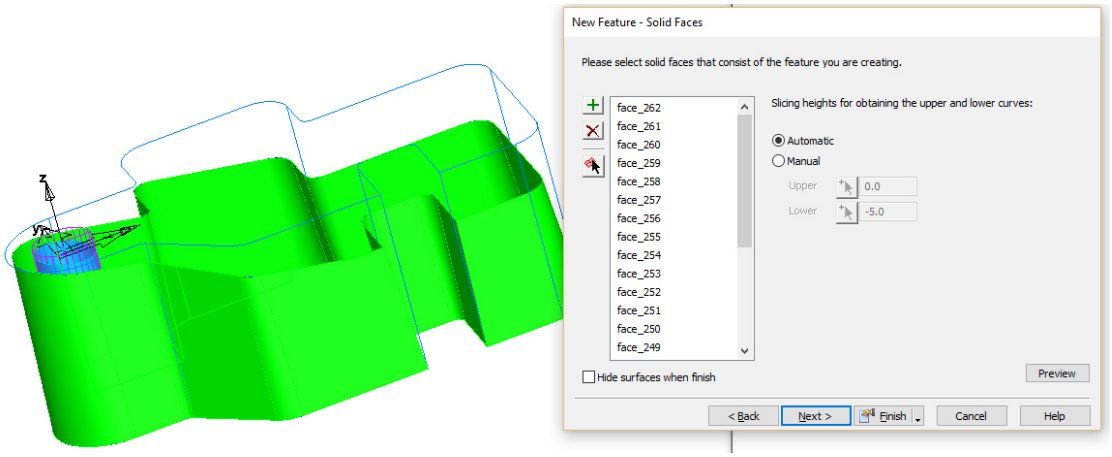
4. Pick the **Offset for curve Z location** and pick the top of the stock



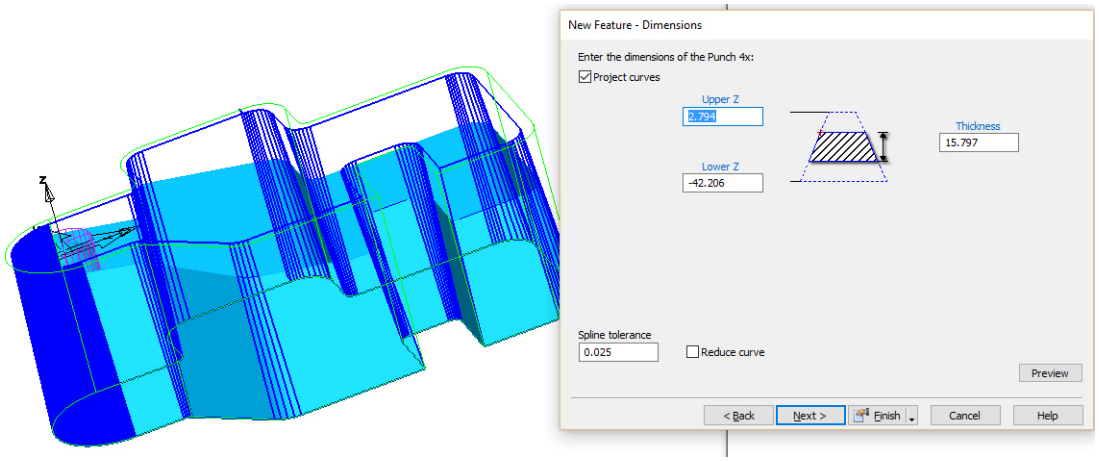
5. Click **Next**
6. Pick the word **Thickness**
7. Pick the top of the stock and the bottom of the stock
8. Click **Finish**
9. Start the New Feature Wizard and select **Punch** from **4 axis**
10. Check the box **Extract with FeatureRECOGNITION** then select **Next** twice



11. Drag select the entire visible part
12. Hold down **Ctrl** and select the surface that makes up the hole
13. Click the  to add the surfaces into the list



14. Click **Next** and set the **Upper Z** to the top of the stock

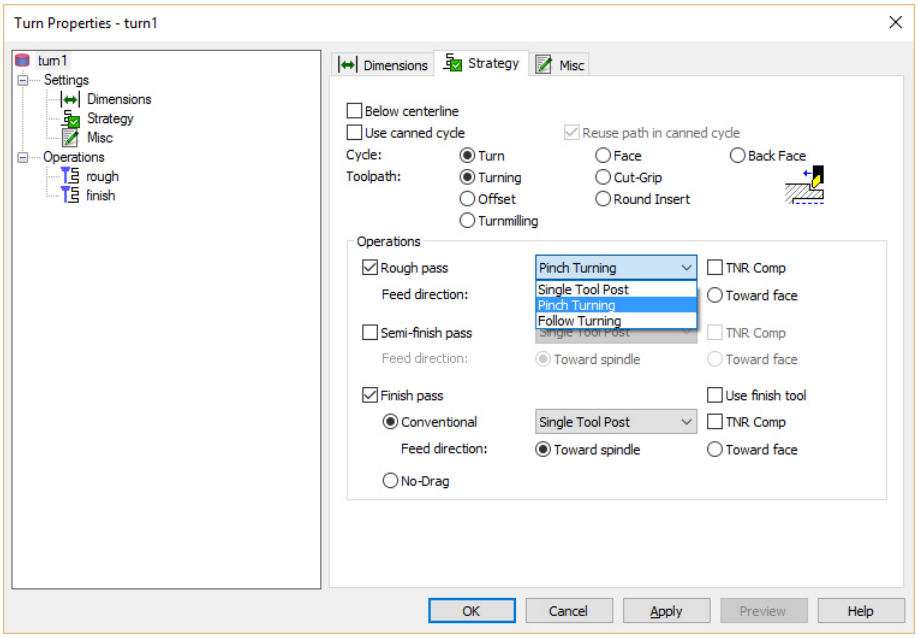


15. Click **Finish**
16. Run a 3D simulation

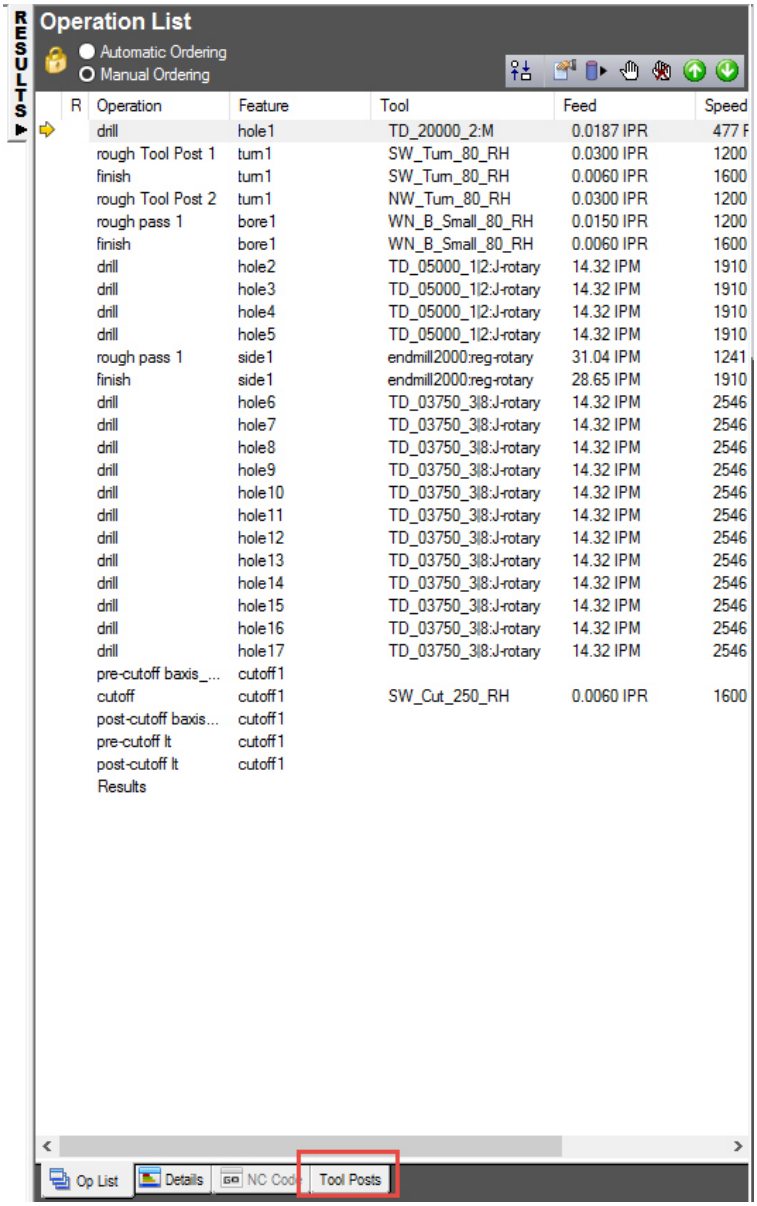
Optimizing multi-tasking programs

Think back to the turn/mill part that we programmed earlier and reference the machine simulation that we looked at. While the part did get cut complete, we were only using the upper turret to cut everything, while the lower turret simply sat there and did nothing. Let's take a look at how we can optimize this program and more efficiently utilize the machine.

1. Open the part file that you programmed, alternatively you can open the file **Hole synchronization not synchronized.fm**
2. Open the properties of turn1 feature and select the **Strategy** tab
3. In the **Operations** section pull the drop down next to **Rough Pass** and select **Pinch Turning**

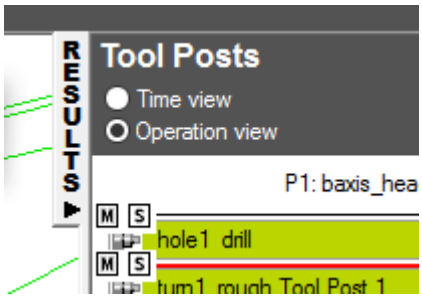


4. Click **OK**
5. In the **Results** Window select **Tool Posts**

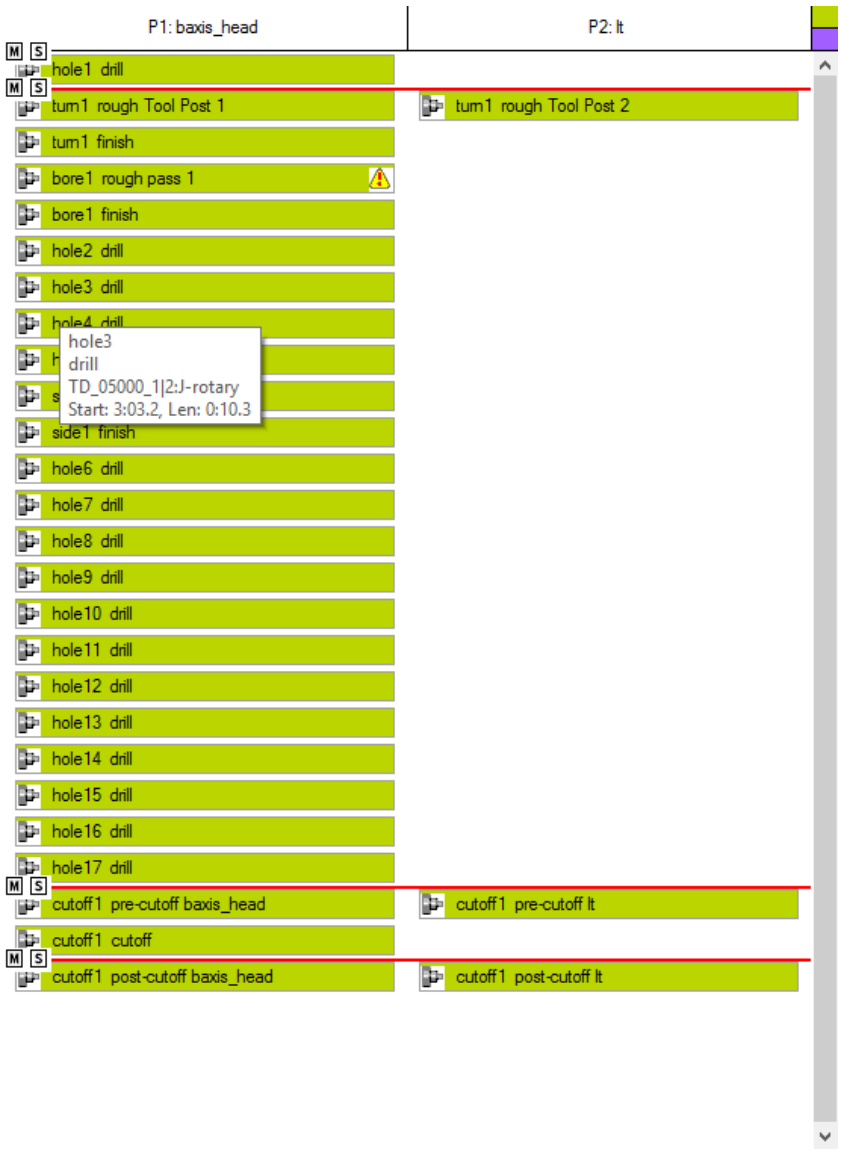


R	Operation	Feature	Tool	Feed	Speed
	drill	hole1	TD_20000_2:M	0.0187 IPR	477 F
	rough Tool Post 1	turn1	SW_Turn_80_RH	0.0300 IPR	1200
	finish	turn1	SW_Turn_80_RH	0.0060 IPR	1600
	rough Tool Post 2	turn1	NW_Turn_80_RH	0.0300 IPR	1200
	rough pass 1	bore1	WN_B_Small_80_RH	0.0150 IPR	1200
	finish	bore1	WN_B_Small_80_RH	0.0060 IPR	1600
	drill	hole2	TD_05000_1I2:J-rotary	14.32 IPM	1910
	drill	hole3	TD_05000_1I2:J-rotary	14.32 IPM	1910
	drill	hole4	TD_05000_1I2:J-rotary	14.32 IPM	1910
	drill	hole5	TD_05000_1I2:J-rotary	14.32 IPM	1910
	rough pass 1	side1	endmill2000:reg-rotary	31.04 IPM	1241
	finish	side1	endmill2000:reg-rotary	28.65 IPM	1910
	drill	hole6	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole7	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole8	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole9	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole10	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole11	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole12	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole13	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole14	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole15	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole16	TD_03750_3I8:J-rotary	14.32 IPM	2546
	drill	hole17	TD_03750_3I8:J-rotary	14.32 IPM	2546
	pre-cutoff baxis_...	cutoff1			
	cutoff	cutoff1	SW_Cut_250_RH	0.0060 IPR	1600
	post-cutoff baxis...	cutoff1			
	pre-cutoff lt	cutoff1			
	post-cutoff lt	cutoff1			
	Results				

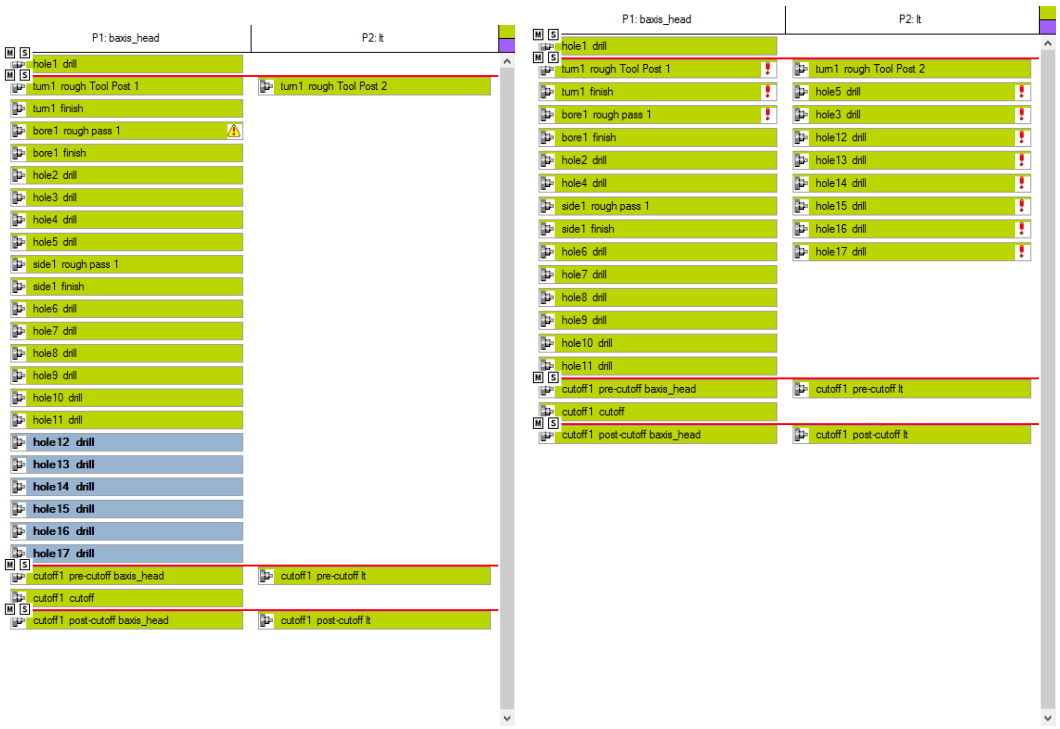
6. Select **Operations View**

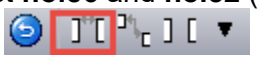


7. We can see now that we are using both the upper and lower turrets to turn the profile of the part. We could do the same thing with the bore, but it really is not large enough to fit both tools in it.

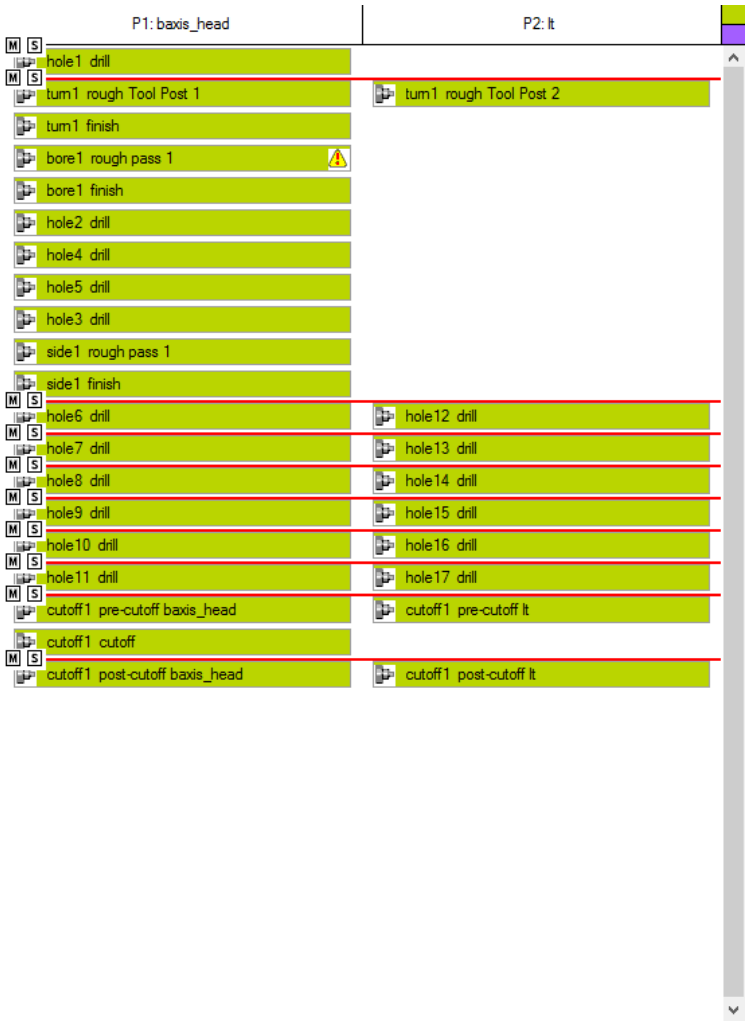


8. Holes 6-17 are all of the holes around the OD of the part (12 holes all together). If we look at the holes, in every instance there is a hole that is oppose it. In the **Tool Posts** tab select **hole12 – hole17** and drag them over to P2 (or turret 2)



9. Select **hole6** and **hole2** (use the Ctrl key) and click on the button **Set sync point at oper start** 

10. Repeat the process, **hole7** and **hole13**, **hole8** and **hole14**, **hole9** and **hole15**, **hole10** and **hole16**, **hole11** and **hole17**



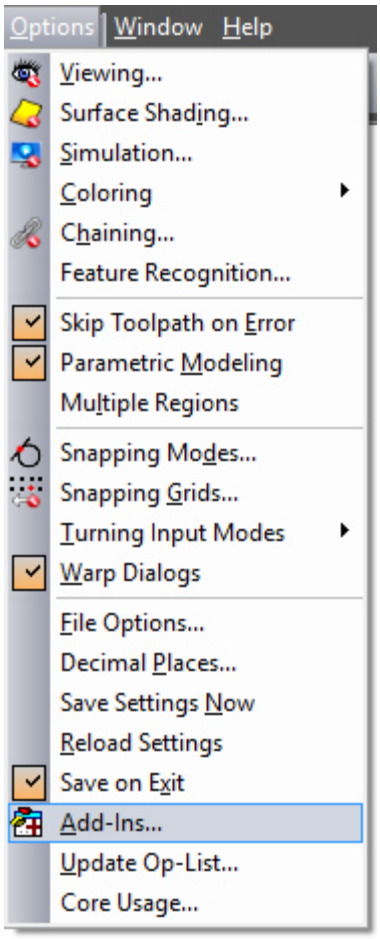
11. Run a **Machine Simulation** to see the completed part

    We are now utilizing the lower turret on the program. FeatureCAM will automatically put out the wait codes in the NC program to ensure that the machine performs the same way we see it in the machine simulation

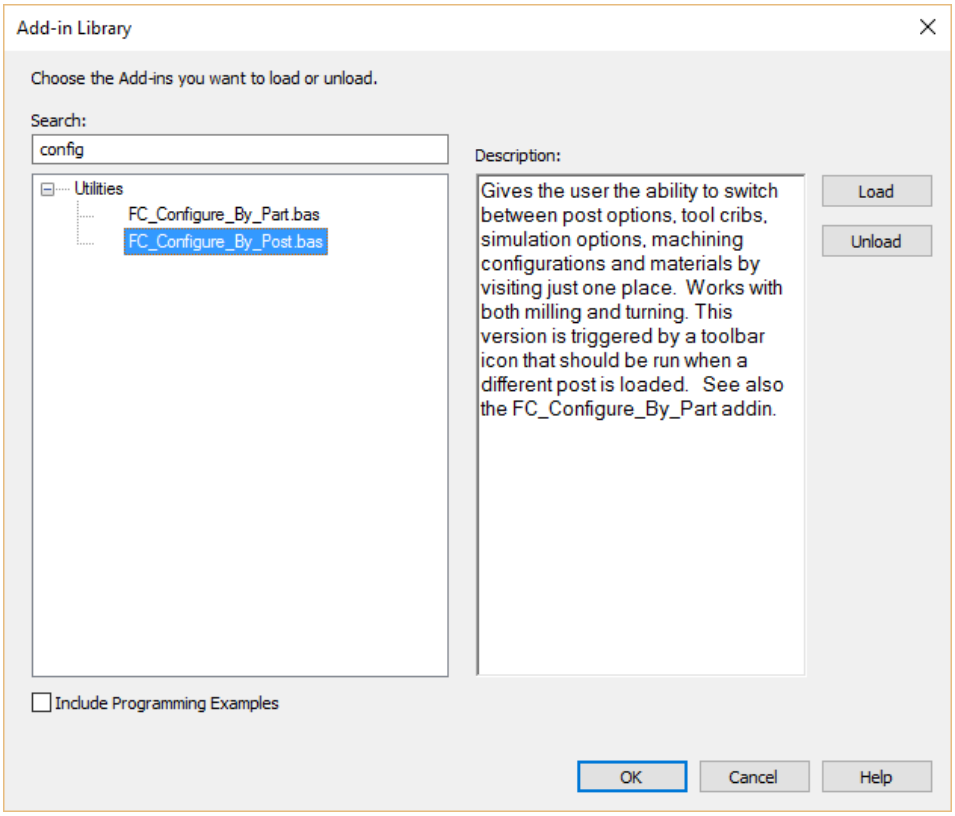
Using completed programs on different machines

FeatureCAM has the ability to quickly and easily change existing programs between like type machines. The basis of this is to load up the part file that you already have programmed, change the post, change the tool crib, and change the machining configuration then re-run simulation to post your code. There is also a user friendly button and dialog that can do this as well.

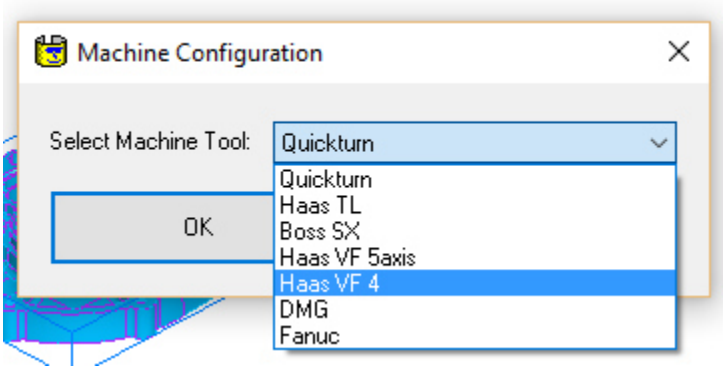
1. Open the part **Complete.fm**
2. Select **Options → Add-ins…**



3. Select **Library**
4. Search for *Config*
5. Select **FC_Configure_By_Post.bas** and click **Load**



6. Click **OK** twice
7. You will get a button that appears in the upper right of FeatureCAM that is a triangle
    Click this button
8. In the **Select the Machine Tool** dropdown select **Haas VF4**



9. Click **OK**
10. Notice that the active tool crib is **Haas**
11. Open the properties of the feature **boss3**
12. Select the rough operation and take note of the tool that is being used **endmillM2100:reg**
13. Run a 3D simulation and try to note the depths of cut of some of the milling features
14. Click the triangle button again
15. Select **DMG** from the dropdown and click **OK**
16. A number of things have changed

a. The post is now **DMC103V**
b. The active tool crib is now **DMG**
c. Open the properties of the feature **boss3**
d. Select the rough operation and take note of the tool that is now being used. With the change of the tool crib, FeatureCAM also changed the tool that is being used to a tool that is available in that tool crib (**21mm reg 2 flt EM**)
e. Run a 3D simulation and try to see the difference in the depths of cut of some of the milling features

We did not change any of the programming, we simply changed which machine we are going to run this part on, FeatureCAM took care of the rest for us. This is another great advantage of the feature based machining approach that FeatureCAM uses.

Conclusion

As you can see, the feature based CAM approach makes FeatureCAM one of the most versatile CAM packages available. You can easily program a wide range of machines from 3-axis to 5-axis mills, simple lathes to multitasking turn-mill centers, wire EDM machines and even Swiss turning machines. Programming is consistent across all of FeatureCAM making it easy to use and easy to learn. FeatureCAM makes it easy to optimize your programs and ensure you are putting out NC code that will cut your parts the way you want and save you time in machining. If the machine you intended to have the part on is not available, you can easily make the necessary changes to put it on a different machine.