Delcam Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureTURN

FeatureCAM

Acknowledgements

FeatureCAM Version: 2012 Date: 03 August 2011 11:10

Contents

FeatureCAM overview

FeatureCAM is a CAD/CAM software suite that automates machining and minimizes programming times for parts on mills, lathes, and wire EDM. Unlike operations-based CAM systems, FeatureCAM generates toolpaths based on the features of the part, and automatically selects appropriate tools, determines roughing and finishing passes, and calculates feeds and speeds. The selections made can be based on the built-in machining knowledge that Delcam supplies 'out-of-the-box' with FeatureCAM, or from experience captured from your company, project or individual users' preferences.

FeatureCAM includes five stand-alone modules:

- **2.5D Milling** - 2.5D design and toolpath generation for 2- and 3-axis mills.
- **3D Milling** - 3D surface modeling and 3-axis toolpath generation.
- **3D Lite** - a limited version of **3D milling**.

  ***3D Lite** lets you mill only one surface per feature, but you can create multiple features. The strategies available in **3D Lite** are Z-level rough, Parallel rough, Parallel finish, Isoline, and 2D spiral.*
- **Turning** - 2-axis design and toolpath generation for 2-axis lathes.
- **Turn/Mill** - Supports lathes with C and Y-axis milling capabilities.
- **Wire EDM** - 2- and 4-axis wire EDM toolpath creation.

The following add-on modules are also available:

- **RECOGNITION** - 3D surface and solid import and the recognition of 2.5D features from solid models. Accelerates making 2.5D and turned parts from solid models.
- **Tombstone** - Multiple part manufacturing for horizontal or vertical milling machines with indexers.

- **Solid Modeling** - Solid modeling and tools for creating molds from solid models.
- **5-Axis Positioning** - Manufacture 2.5D features from 5-axis orientations.
- **Native Import Modules** - Native data can be read directly from SolidWorks, SolidWorks Assemblies, Autodesk Inventor, SolidEdge, Catia, NX, Pro-Engineer, and Step files.
- **Machine Simulation** - Modeling and simulation of a CNC machine.
- **Advanced Turn/Mill (MTT)** - Includes support for Turn/Mill in addition to support for B-axis (5-axis positioning) and multiple turret synchronization.
- **Network Database and Licensing** - Flexible product licensing allows sharing FeatureCAM licenses across a network.
- **5-Axis Simultaneous** - Manufacture 3D features while changing the tool axis.

Why creating toolpaths is so fast

FeatureCAM has the unique ability to generate toolpaths and create NC code to run the machines with a minimum amount of user input.

Traditional CAM systems are *operations-based* and require you to program every operation, one at a time, to create your part. FeatureCAM is *feature-based*; this means the part is created using features that describe that part, from simple holes, to complex pockets, to turned grooves. Machinable features contain information and rules describing how and where material removal should occur, cutting depths, whether to use climb cutting, whether to spot drill or center drill, and preferred machining strategies for roughing and finishing. This means that after you import or draw the part and identify its features, FeatureCAM automatically:

- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

You can customize this built-in 'intelligence' to your own style of cutting.

FeatureTURN - Features from Dimensions

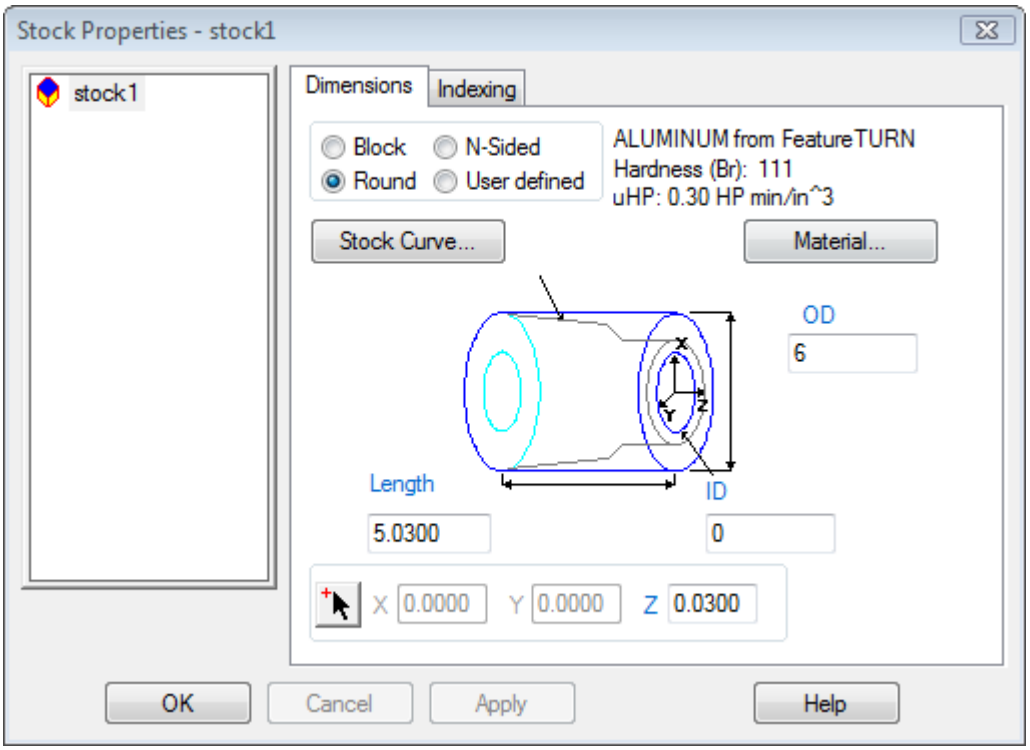
Introduction

One way features can be created is by importing or drawing the shape with geometry, chaining the geometry with curves, and then creating Features from Curves. Another way is to create Features from Dimensions, which do not require geometry or curves.

Features from Dimensions

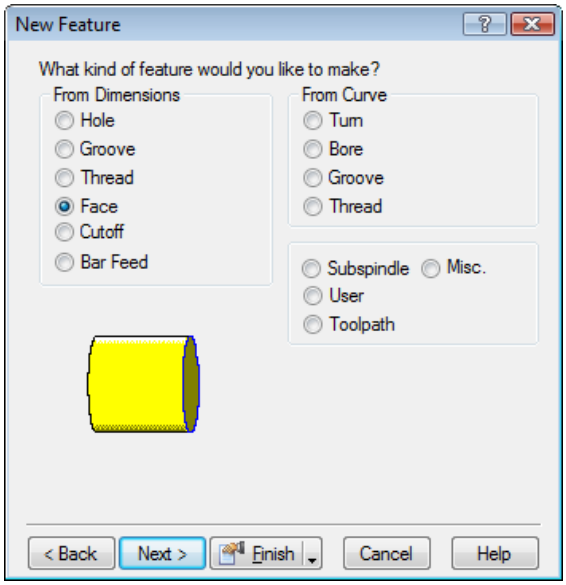
In the New Feature wizard, there is a group of features called features **From Dimensions.** These features **do not** require drawing geometry, nor do they require curves. All of the necessary criteria are included in the wizard pages as you create the feature.

- Open a **new Turn** file
- Click **Finish** to complete the Stock Wizard, and edit its properties
- Define the stock as shown below (Notice the Z value of .03. This gives us extra stock to remove during the facing process.)

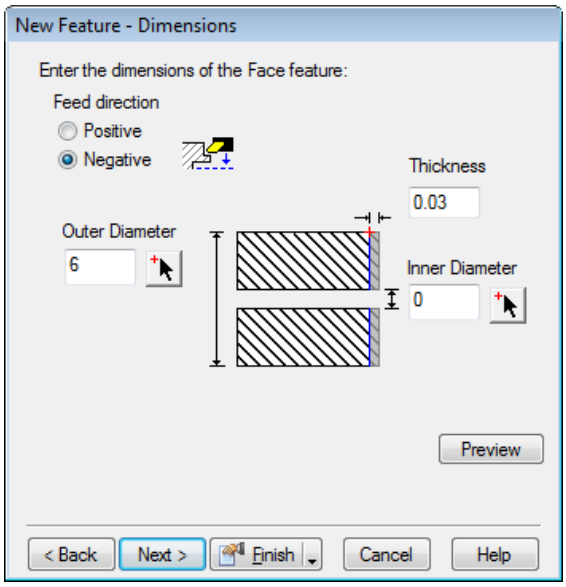


Face from Dimensions

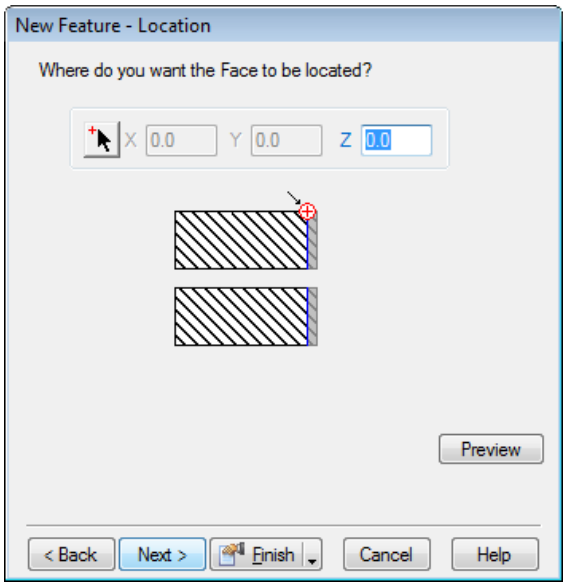
- Open New Feature and select **Face** from **Dimensions**



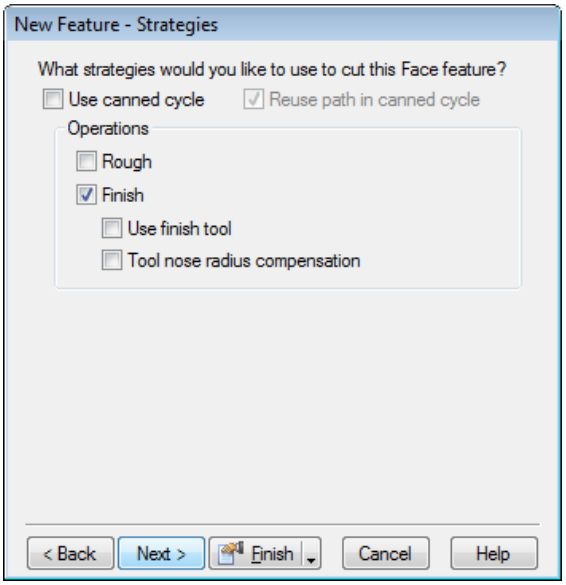
- Verify that all the default values are as shown, and click **Next**



- The location of a face feature is the Z position at which the feature will be finished. Verify that it is set to **0** and click **Next**



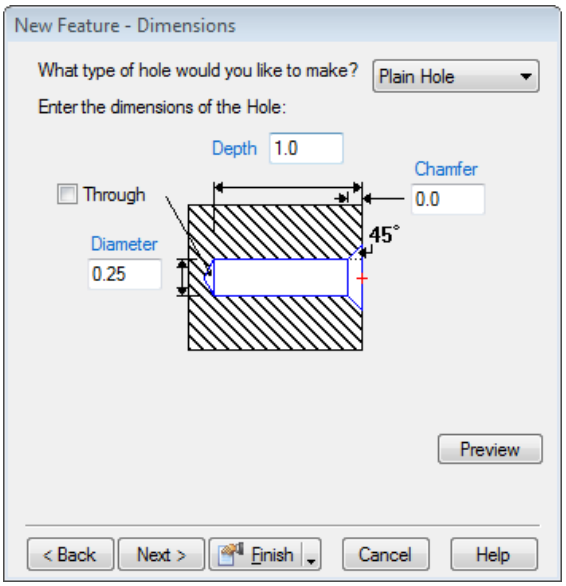
Canned cycles can be generated in the NC code for nearly every turned feature. To generate these cycles, your post processor must support them, you must turn this function on for the post, and for some features, you must also activate the canned cycles within the feature. **Reuse path in canned cycle** outputs the path geometry only once for both roughing and finishing when canned cycles are enabled. **Operations** include the ability to select Rough and/or finish passes.



- Click **Next** and **Finish**
- Click **OK** to exit the Face Properties

Hole from Dimensions

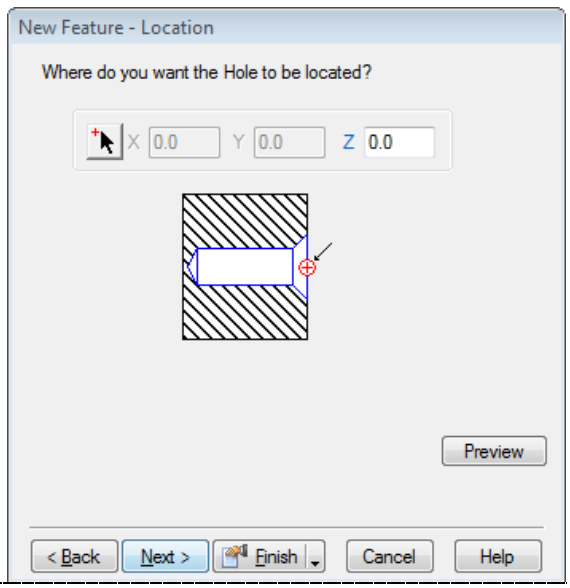
On a two axis lathe without live tooling, the only hole that can be machined is in the center on the end of the part. There are 6 types of holes that can be created by using Features from Dimensions.



- Open New Feature and select **Hole** from **Dimensions**

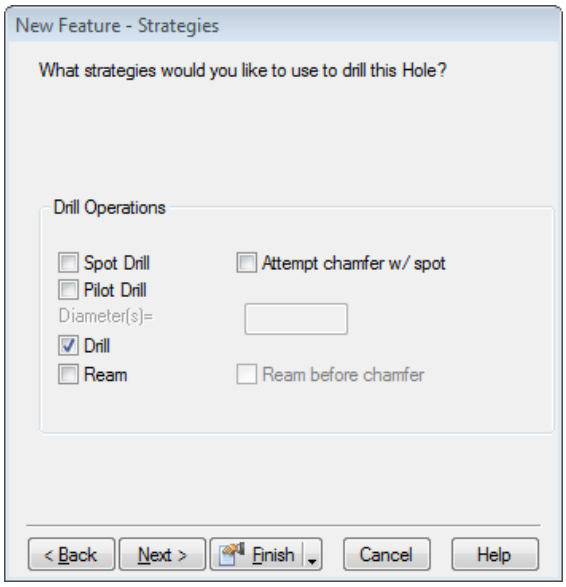
There is a drop-down menu for the type of hole that you want to create. These types include tapped, counterbored, and plain, among others. When you select a hole type, the parameters shown in this box will change to those related to the hole's construction. For this example, we will just use a plain hole.

- Select **Plain Hole** and accept all the default dimensions
- Click **Next**



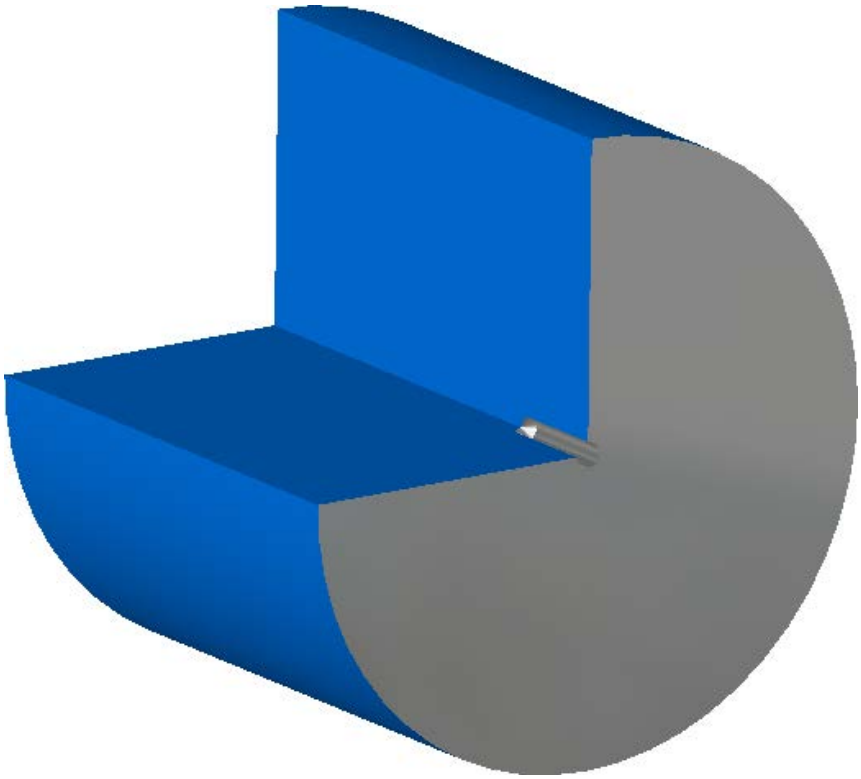
- The location of the hole is the Z position of the top of the hole. Set this value to **0** and click **Next**

- The Strategy dictates how the hole will be machined. Accept the defaults and click **Finish**



- Run a 3D simulation to see what has been programmed so far

One simulation that is available in Turn is to have the part cut away to a ¾ view whenever any work is done to the inside diameter. This option can be found under **Options > Simulation > 2D/3D Shaded**



- **Eject** the simulation
- **Close** the file without saving

Groove from Dimensions

Several types of grooves can be created from dimensions and require no geometry or curves. Simply select the type of groove you want to create and fill in the parameters. If you require a groove that cannot be described in this fashion, you must draw a curve to describe its shape.

The types of grooves that can be created from dimensions are as follows:

X axis – these are grooves positioned on the Inside Diameter or the Outside Diameter
Face – grooves that are cut from the positive Z direction
Backface – grooves that are cut from the negative Z direction
Simple – any of the above groove types can also be classified as "simple", meaning they are cut with a single tool that is the same width as the groove

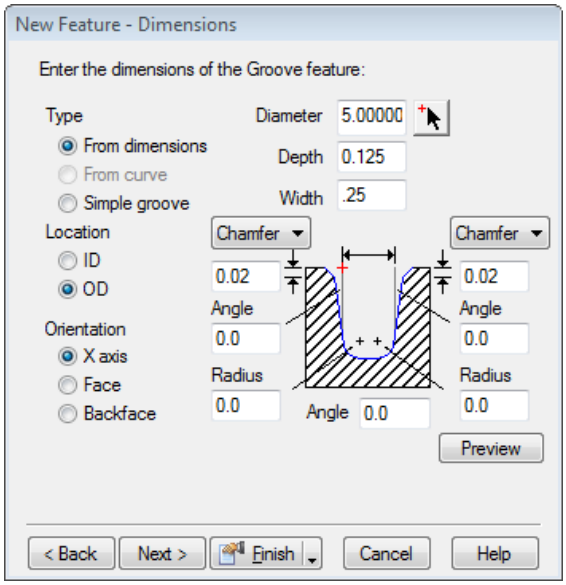
X axis Groove

- Open **FeatureTURN Basic.fm**
- Make sure that the tools toolcrib is active
- Open **New Feature** and select **Groove from Dimensions**
- Click **Next**

You will see that all of the parameters required for defining the feature are contained on this screen (the actual location of the groove will be entered on a later screen.) Keep in mind that if a groove cannot be defined by these items, it must be drawn and created from a curve.

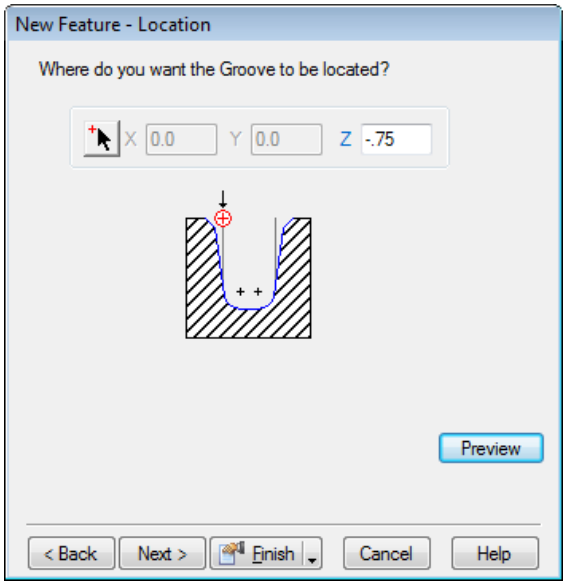
The first groove type that we are going to create is an X axis groove located on the OD of our part.

- Fill out the form as follows:
    - **From Dimensions**
    - **OD**
    - **X axis**
    - Diameter = **5.0**
    - Depth = **.125**
    - Width = **.25**
    - Chamfers = **.02**
    - All others = **0**
- Click **Next**



- Fill in the position of the groove in Z (-.75)
-

Notice how the groove is positioned. The Z value you enter will be the position of the back edge of the groove.

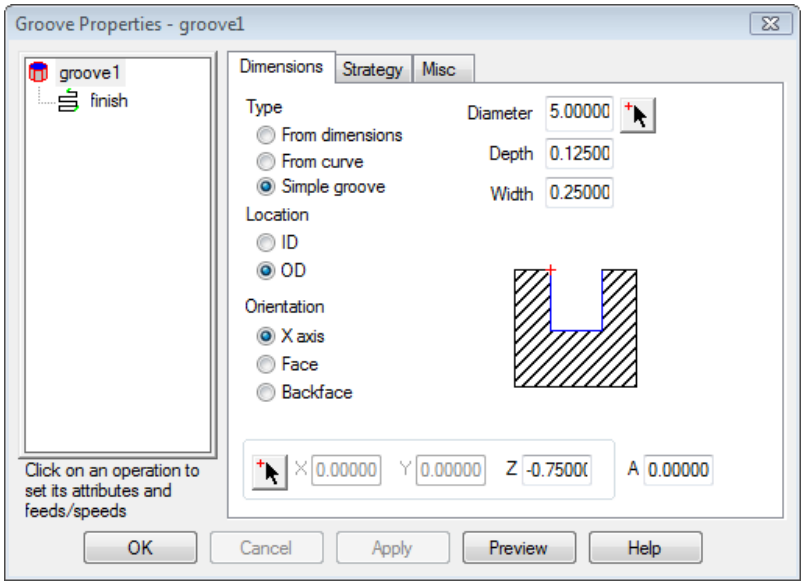


- Click **Finish**
- Click **OK** to exit the Groove Properties
- Run a simulation to see how the feature cuts the groove

The groove has been created with a rough and finish operations using a tool that is narrower than the groove width. Let's now study how he groove will behave as a "Simple" groove.

- Double-click the groove feature to access its Properties
- On the Dimensions tab, click the **Simple groove** radio button
- Click **Apply**

The first thing that you will notice is that several of the dimension parameters disappear. This is because the Simple groove is a single tool operation. Also, the rough operation is removed from the feature and a tool that is the same width of the groove is selected.

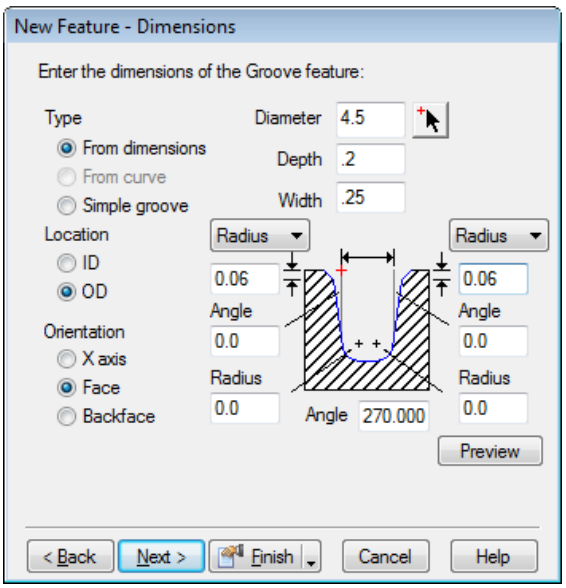


- Click **OK** to exit the Properties and run another simulation to see how the simple groove behaves
- After examining the simulation, go back into the feature and change the type back to **From dimensions** to restore the original dimensions; you will have to restore the value of **.02** in both **Chamfer** fields

Face Groove

Next, we will create a Face Groove.

- Open **New Feature** and select **Groove from Dimensions**
- Click **Next**
- Fill out the form as follows:
    - **From Dimensions**
    - **OD**
    - **Face**
    - Diameter = **4.5**
    - Depth = **.2**
    - Width = **.25**
    - Radius = **.06**
    - All others = **0**
- Click **Next**

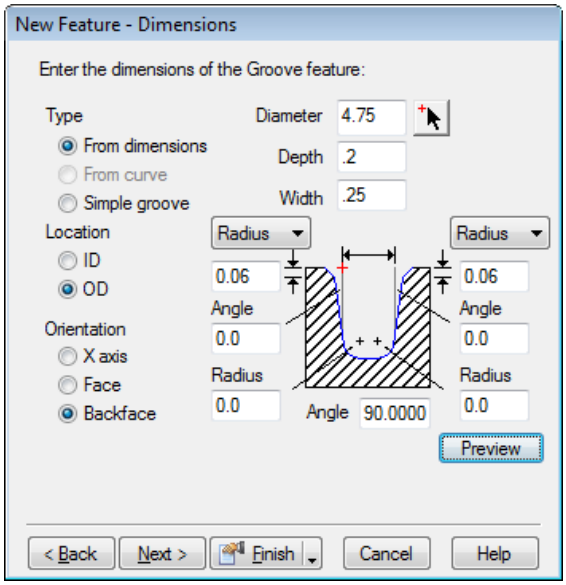


- Set the Z location at **0**
- Click **Finish** and **OK** to exit the groove properties
- Run a simulation

Back Face Groove

Next, we will create a Backface Groove.

- Open **New Feature** and select **Groove from Dimensions**
- Click **Next**
- Fill out the form as follows:
    - **From Dimensions**
    - **OD**
    - **Backface**
    - Diameter = **4.75**
    - Depth = **.2**
    - Width = **.25**
    - Radius = **.06**
    - All others = **0**
- Click **Next**



- Set the Z location at **-1**
- Click **Finish** and **OK** to exit the groove properties
- Run a simulation

Threading from Dimensions

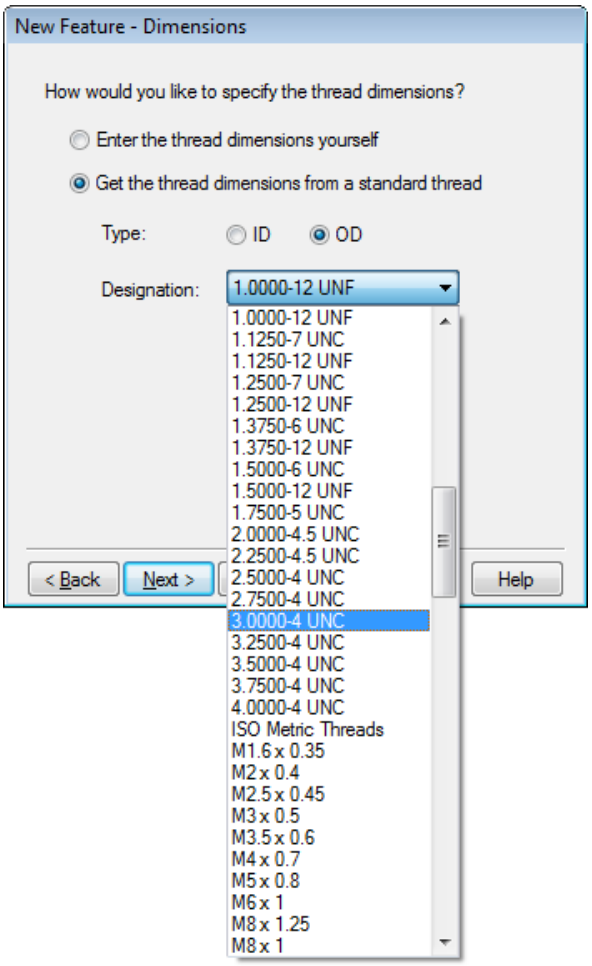
Threading is another type of feature that can be described completely with dimensions.

- Open a **new** Turning document
- Click **Finish** to exit the Stock Wizard and edit the Stock Properties

- Enter the Stock dimensions as follows:
    - OD = **2**
    - Length = **6**
    - ID = **0**
    - (Verify that the Z location of the stock is **0**)
- Click **OK** to accept the stock dimensions
- Click the New Feature wizard and select **Thread From Dimensions**

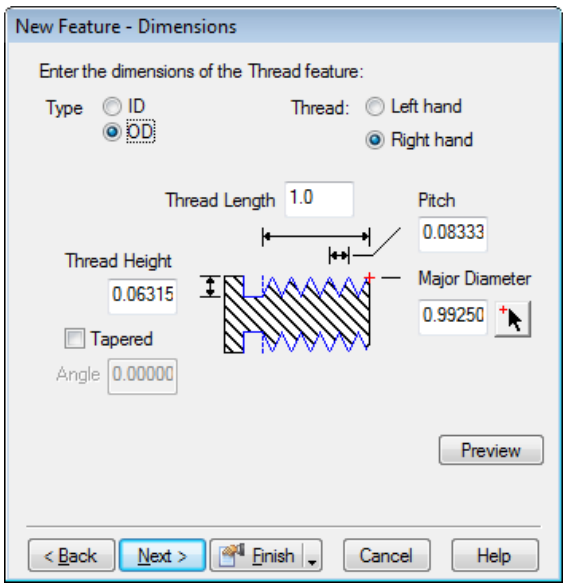
FeatureCAM gives you the choice of selecting from a list of standard threads, or entering custom dimensions.

- Select **Get the thread dimensions from a standard thread**
- Click on the drop-down menu and select **1.0000-12UNF** from the list of Standard Threads
- Click **Next**



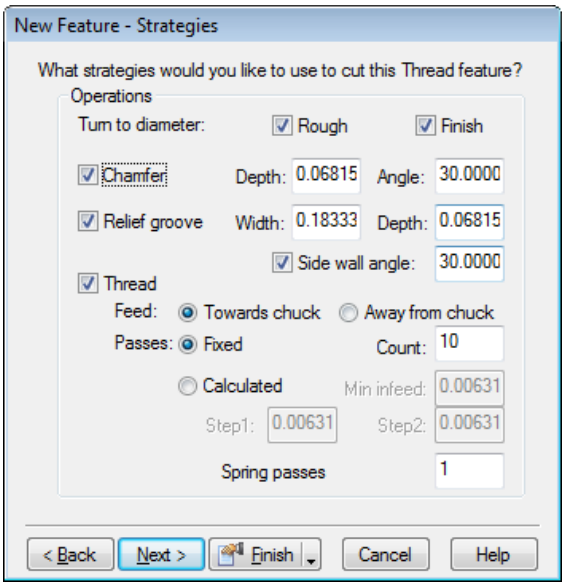
Most of the values on the Dimensions page are already entered as a result of selecting the Standard Thread. In this case, the only values that the user has to enter are the Type, the Hand, and the Thread Length.

- Enter 1 for the **Thread Length**, and verify that **OD** and **Right hand** are selected
- Click **Preview** to see how the thread is defined so far
- Click **Next**

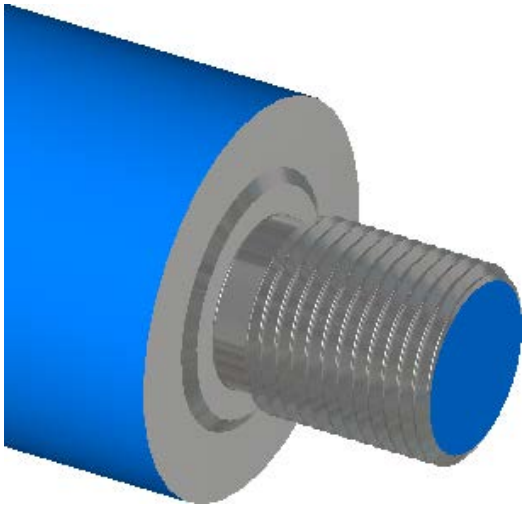


- Enter **0** for the Z location of the threads
- Click **Next**

The Strategy page of the Thread feature has several options that compliment the threading feature. If the part has not been machined to the proper diameter, you can add rough, finish and chamfer operations to the feature. You can also select a relief groove, and choose threading styles.

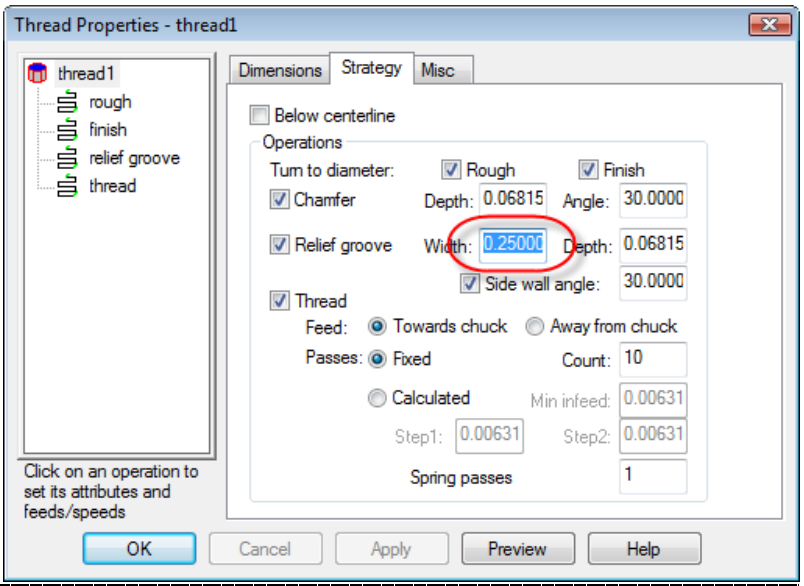


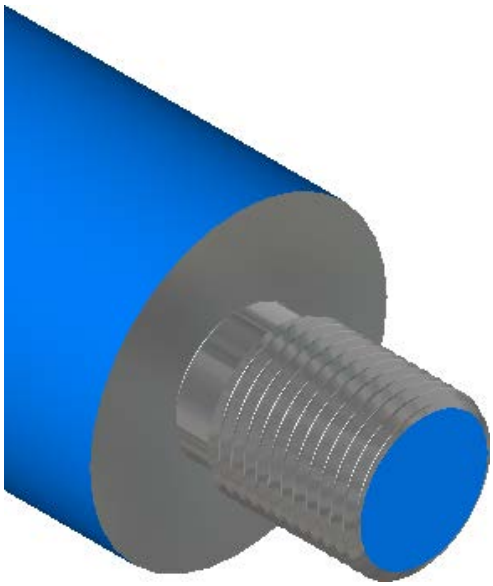
- Since the part has not been turned yet, check the **Rough** and **Finish** checkboxes
- Verify that the other settings are as shown
- Click Finish and OK
- Run a simulation



You will notice that the threading tool gouges the shoulder of the part. Follow these steps to correct the problem:

- Double-click the **Thread** feature to access it's Properties
- Select the **Strategy** tab
- Change the **Width** of the **Relief Groove** to **.25**
- Click **OK**
- Run another simulation



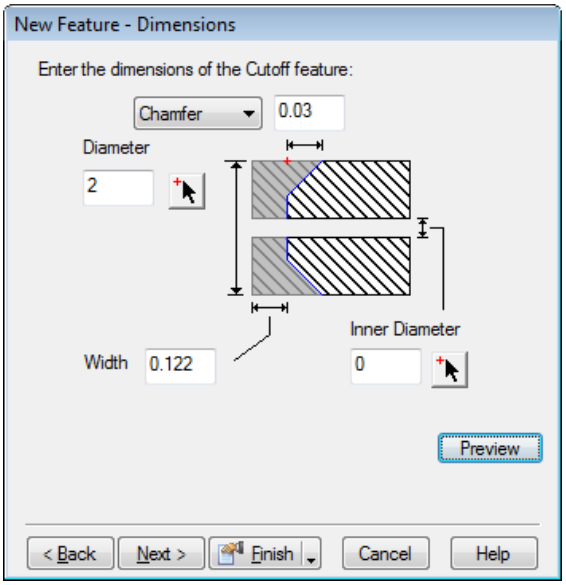


- Keep this file open for the next section

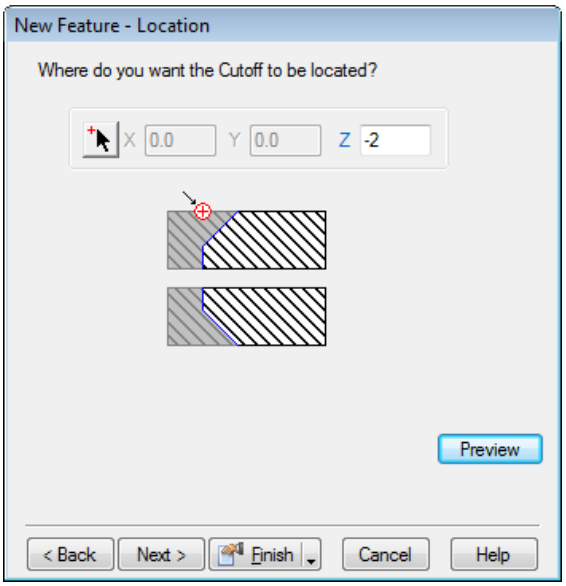
Cutoff from Dimensions

The **Cutoff** feature is used with barstock and usually a bar feeder or puller, and allows the machining process to be automated. FeatureTURN also supports a part catcher, which swings into position under the part and catches it as it is cut off. Some machines have a sub spindle instead of a tailstock that can be programmed to move in and grab the part during the cutoff procedure and then return home allowing the other side to be machined.

- Click the New Feature wizard and select **Cutoff** from Dimensions
- Click **Next**
- Fill out the form as follows
    - Chamfer = **.03**
    - Diameter = **2**
    - Width = **.122**
    - ID = **0**
- Click **Next**

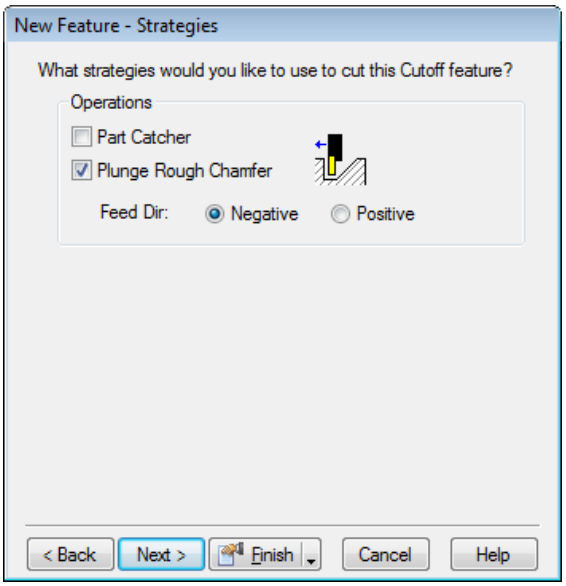


- On the **Location** tab, enter the location of Z -2.0 (Notice that the dimension value is to the back of the part)

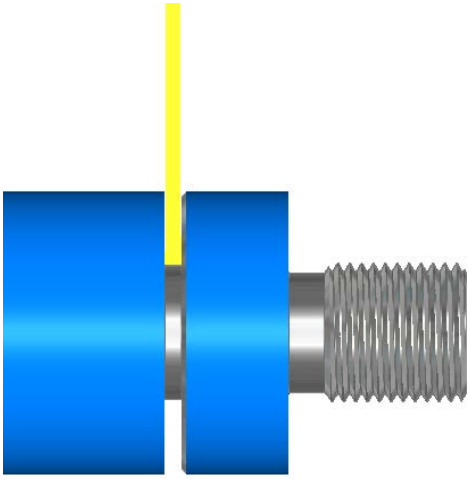


As with the Threading Strategies, the Cutoff Strategies offer more options for the Cutoff feature. Here is where you can enable the Parts Catcher and set the cutting direction.

- Make sure Plunge Rough Chamfer is selected and click **Finish**
- Click OK to exit the Cutoff properties

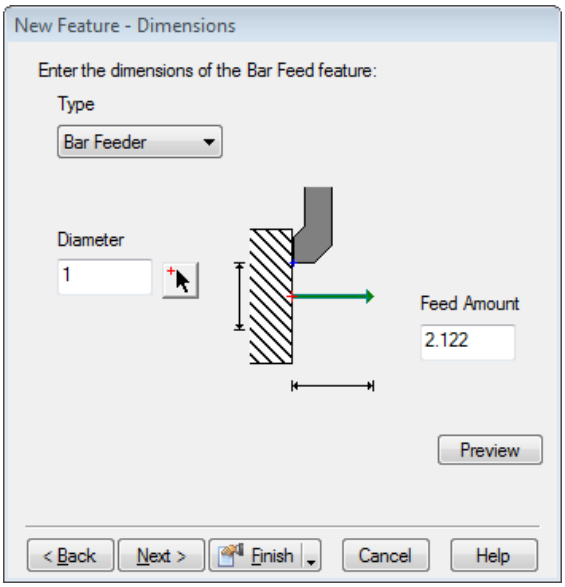


The Cutoff blade makes a straight cut to create a relief for the chamfer, and then it cuts the chamfer, and then cuts to the center.



Barfeed and Bar Puller

The **Barfeed** and **Bar Puller** Feature from **Dimensions** are designed to automate turning when using a long bar. A cycle may loop continuously until the bar stock is finished by using a **Parts Catcher** or if the machine has a sub spindle, the sub spindle can grab the part prior to the cut off and return home to machine the other side after the cutoff.



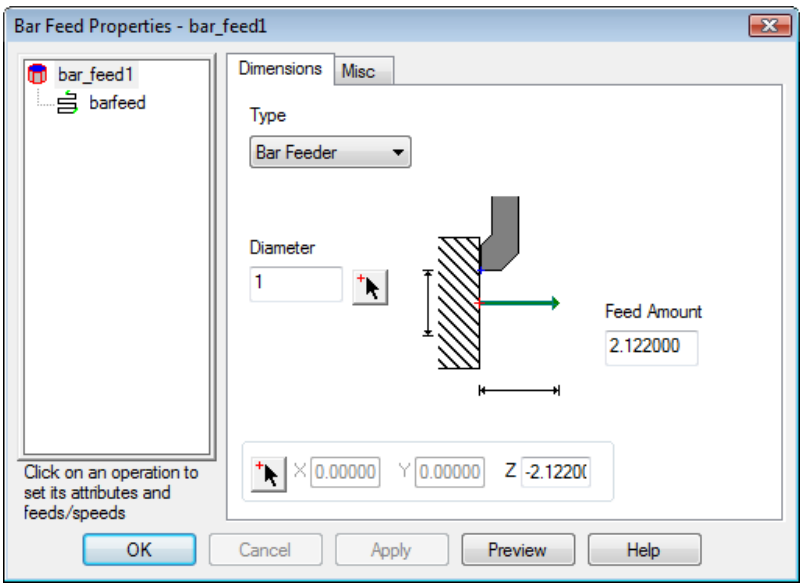
- Click the New Feature wizard and select **Bar Feed** from Dimensions
- Click **Next**
- Select **Bar Feeder** as the Type
- Set the **Diameter** to **1**

This is the diameter to which the bar stop will move.

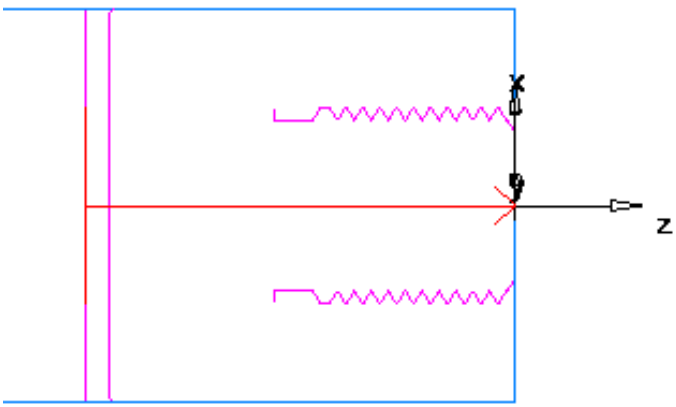
- Set the **Length** to **2.122**

The length is calculated by adding the cutoff width to the length, plus any extra material for facing.

- Click **Finish**



- In the **Bar Feed Properties** form, enter Z **-2.122**. This tells the bar stop in the turret how far to move in past Z 0 to touch the bar stock and move in Z positive to position the bar for the next part.

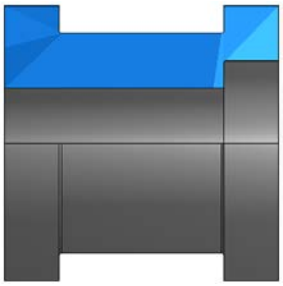


- In the Graphics Window, on the part, there is an Arrow which represents where the bar stop locates on the left end against the bar stock after cut off and the Arrow head on the right end shows where the end of the bar will stop for the next part

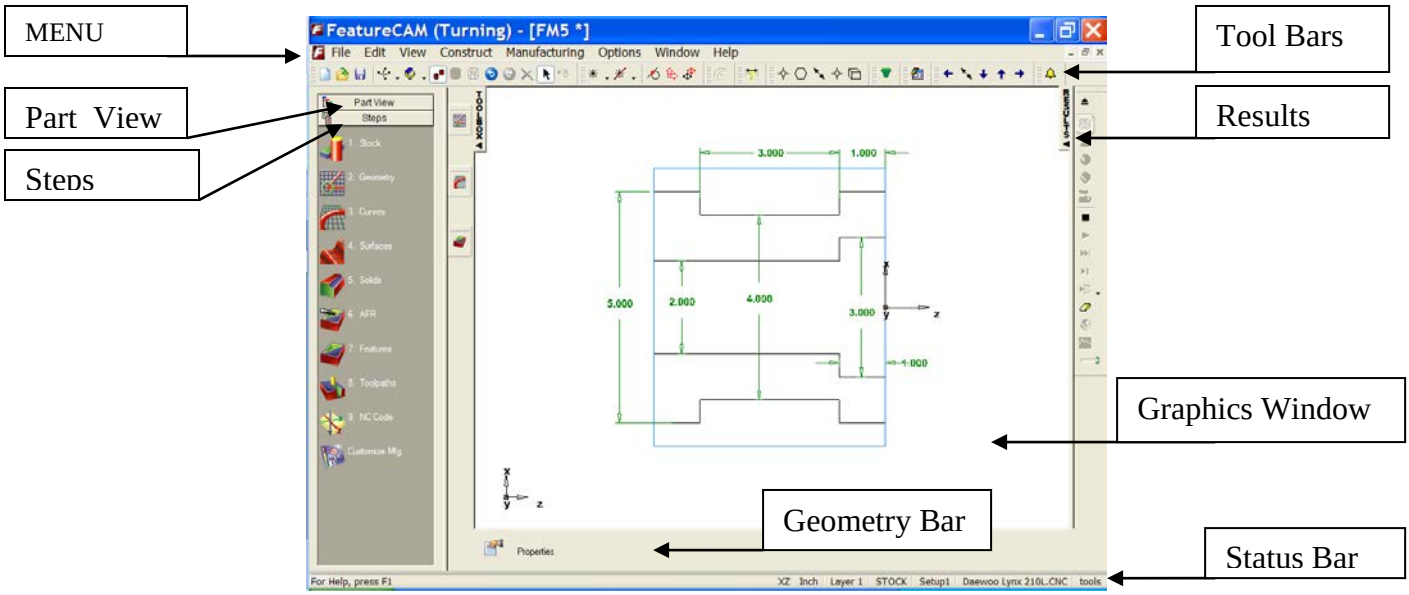
FeatureTURN Basic

Introduction

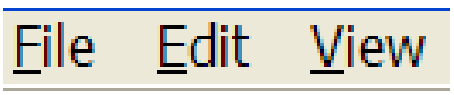
In this FeatureTURN module the user will learn basic 2-axis (X and Z) turning, boring, and grooving along with the basic user interface and different stock types. At the end of the module the user will be able to generate a program for a simple part



The user Interface is the entire viewing area of FeatureCAM and is designed to be intuitive, easy to use, and provides the many functions to assist you in creating programs accurately and in the fastest possible time.



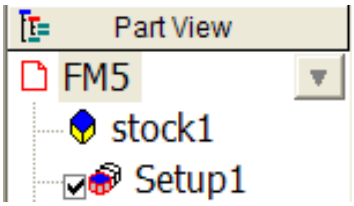
Menu Bar Located at the top. Like many other Windows based software, FeatureCAM includes a series of drop down menus. These menus are unique to FeatureCAM and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.



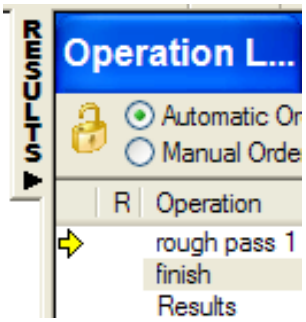
Tool Bars Usually located at the top but, may be docked in other locations. Can consist of one or more buttons and used to create objects in the Graphics Window such as the Geometry constructors for drawing the shape of your part.

Steps Located on the left side. A systematic sequence to guide, especially the beginner, as to the order of which needs to be completed first, second, etc.

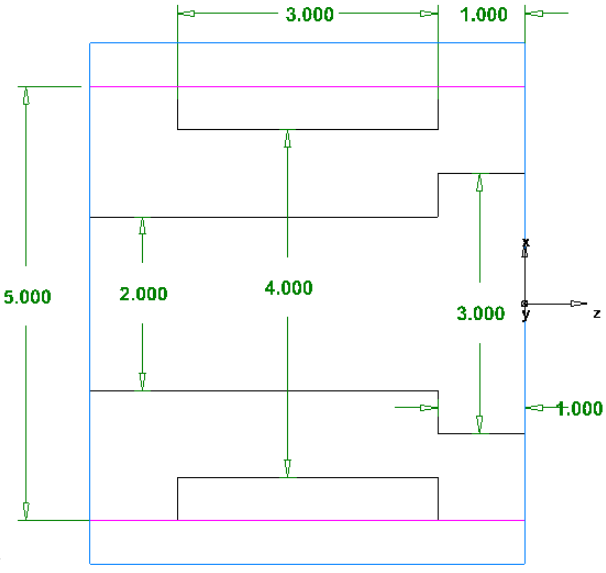
Part View Located behind the Steps. This area gives a listing and provides access to the properties of the stock, setups, features, turrets, curves, (surfaces and solids when available).



Results Located on the right side. It contains the Operation list. Whereas the Part View contains all of the features, the Operation List contains all of the hits. It also provides the details of each operation, including a tool list. The NC code is also here. All of the information residing in this area may be printed.

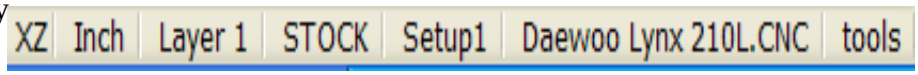


Graphics Window Located in the center. This is where most of the activity is taking place, creating stock, importing drawings, solid and surface models, constructing geometry, chaining with curves, creating features and simulating the tool path in 3D or using the centerline of the tool.



Geometry Bar Located near the bottom. This area is constantly used and is where the dimension boxes are used to locate the geometry on your stock material in the Graphics Window.

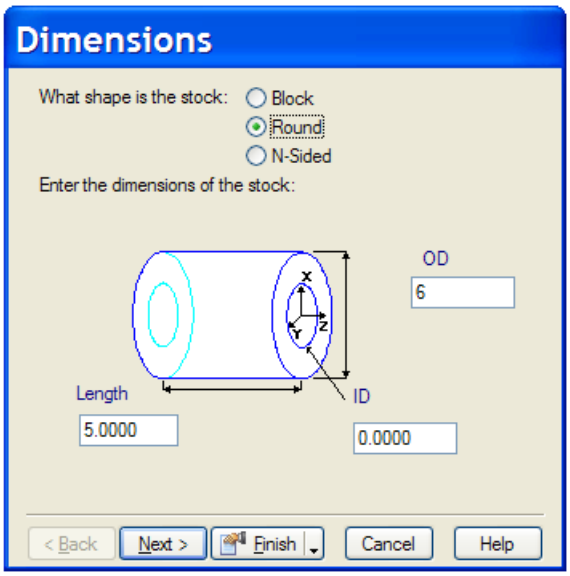
Status Bar Located at the very bottom of FeatureCAM. It contains the plane you are working in (XY–XZ-YZ), the unit of measure (inch-metric), the layer you are working in, the setup, post processor and the tool crib.



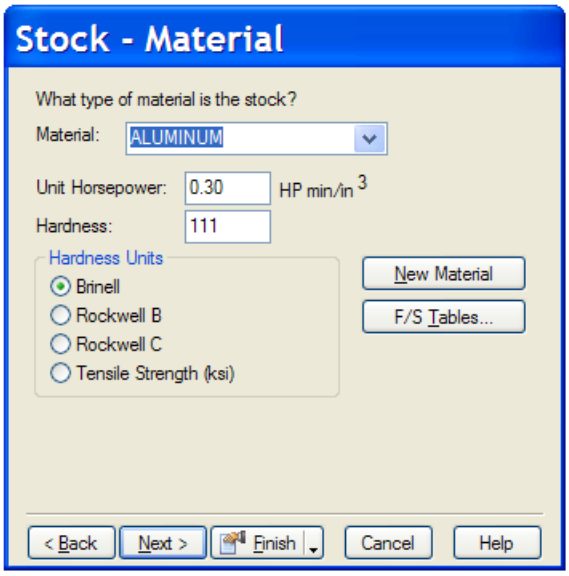
Open a new part document

- Turning Setup, Inch. Finish.

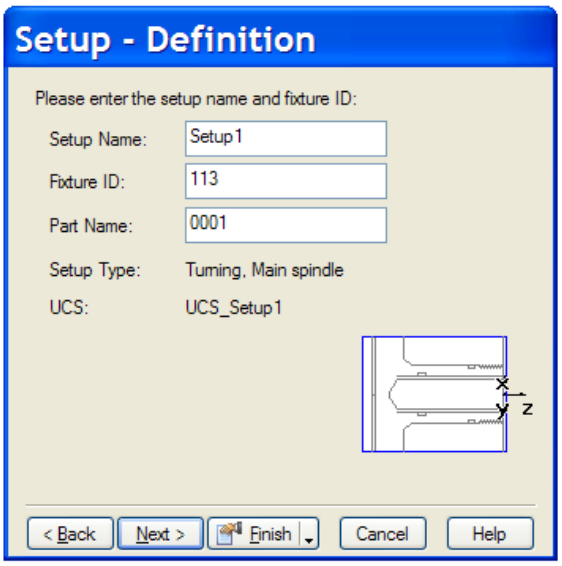
- Select Round, OD 6, Length 5, ID 0. **Next**.



- Select Stock Material to be used for the part; this is also where the feed and speed F/S Tables reside for the selected materials. **Next**.



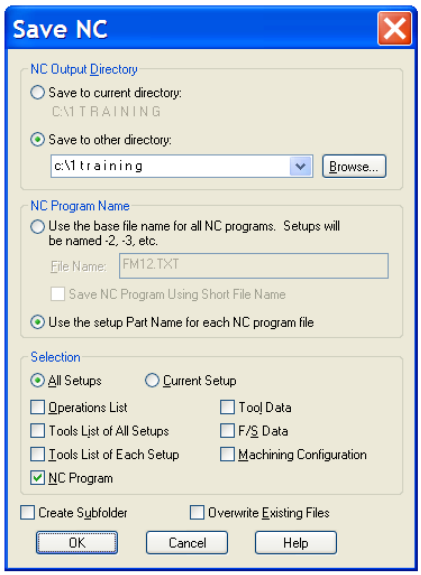
- Setup – Definition includes Setup Name, any name may be entered or use the default setting. **Next**.



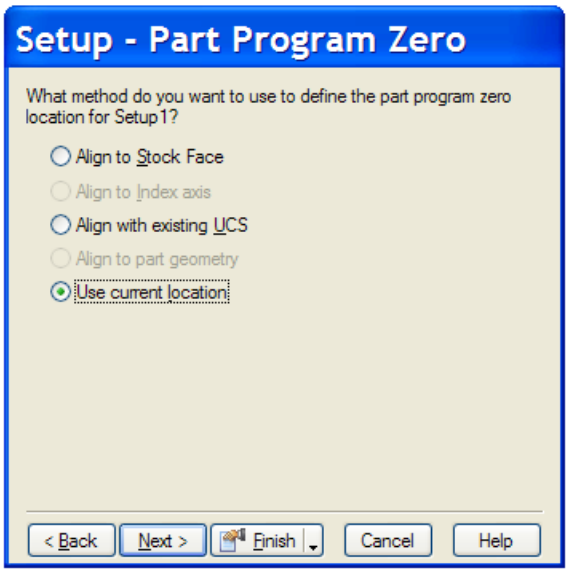
- The Part Name is inserted in and must be what is recognized in the NC code (O0001). The fixture ID comes from the post processor and the call is applied to the NC code, such as G-54.

```
%
O0001
N1
G0 G20 G40 G80 G95
G92 S3000
G96 S1200 T0101 M3
X5.67 Z0.0606 M8
G1 Z-5.0394 F0.015
X6.0
X6.0354 Z-5.0217
G0 Z0.0606
G1 X5.34 F0.015
Z-5.0394
X5.67
X5.7054 Z-5.0217
G0 Z0.0606
```

The **Part Name** is used to identify the program text file when you “Save NC” (Use Setup Part Name) under the File menu located upper left.

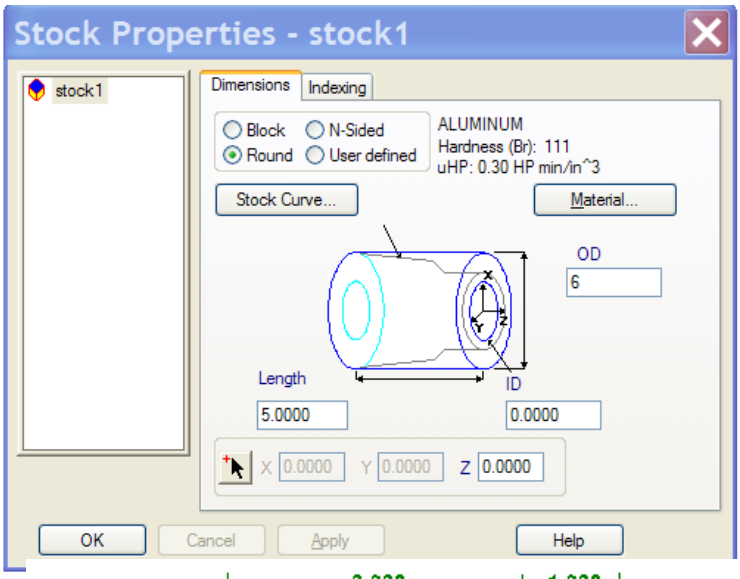


- **Setup – Part Program Zero** provides several methods to define the part program zero location. Align to the face of the material, align with another existing UCS (User Coordinate System) or accept the current location. **Next**.



- **Setup – Spindle Location** The Main Spindle will always be selected when there is no Sub Spindle being used. (Sub spindle is a second spindle at the tailstock end of the machine opposing the main spindle and used to run the second side of the part), not used in this module. **Next** and Finish.

Stock Properties page contains all of the values just entered. If you need to reopen the properties after finishing, just place your cursor on the blue wire frame of the stock in the graphics widow and double click, it will reopen it.



Drawing the Geometry

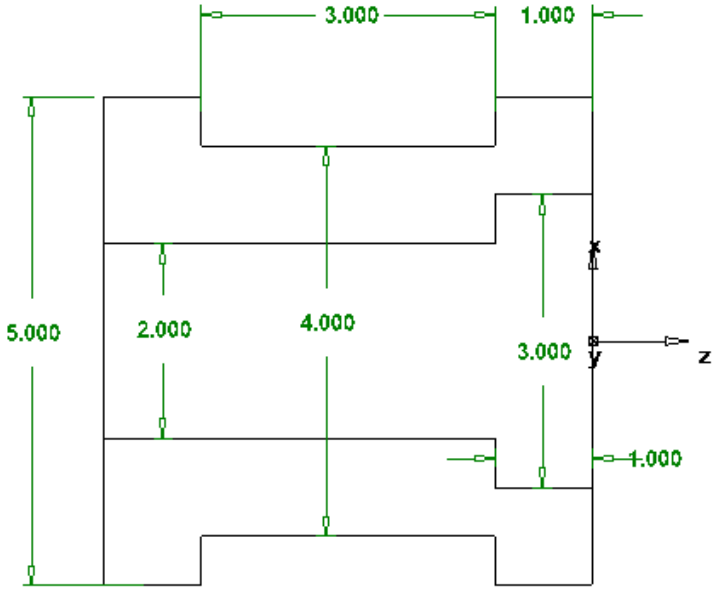
- Draw this part in the stock using the Geometry constructors under Steps and check the box Create more than one. You will then create curves and features that will include, facing, turning, back turning, zigzag turning, grooving, drilling, boring and cutoff. Only the top half of the geometry need be drawn, above the centerline.

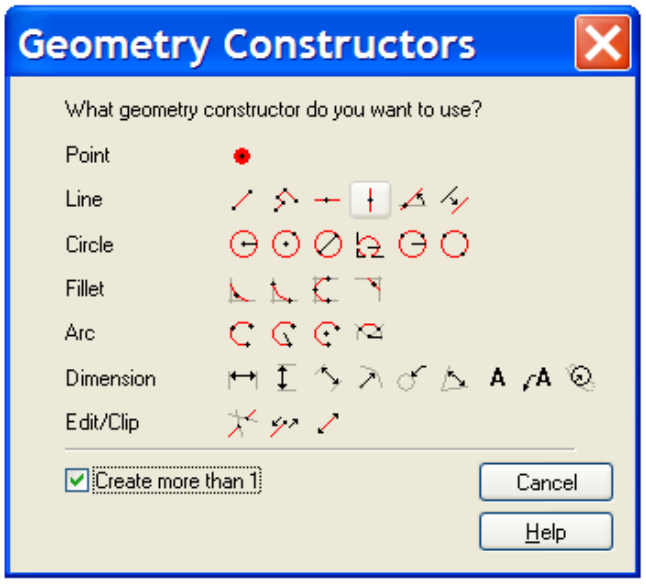
  One of the easiest ways to draw this part is to use Horizontal and Vertical lines, these lines are infinite and only require one dimension. They also create the intersections and you need only to clip the unwanted geometry.

- **Draw** Horizontal lines at diameters of 5, 4, 3 and 2.

- Draw Vertical lines at 0, -1, -4, -5.

- **Clip** the geometry to achieve the result in the drawing. Above centerline only.





Chaining the Curves

- On your drawing, the geometry to the right is the **O.D.** including the undercut and will be used to illustrate several different methods of turning. On your part, chain an open curve the full length of this geometry.

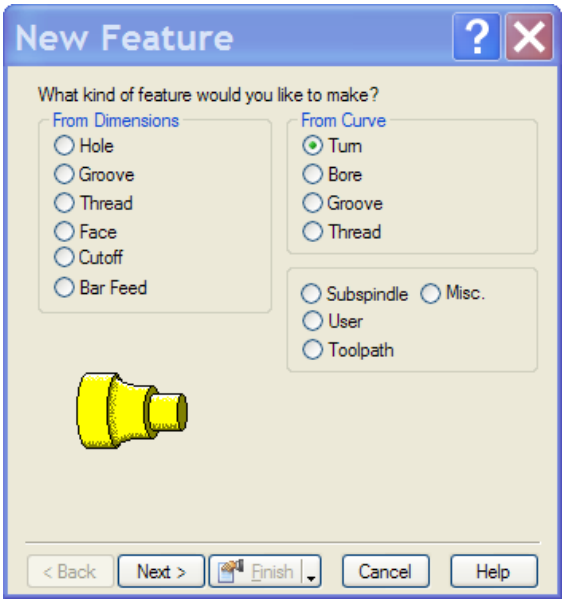
- On your drawing, the geometry to the right is the undercut and will be used for an O.D. **groove** separate from the turning and will be used to illustrate grooving strategies. On your part, chain an open curve the full length of this geometry.

- On your drawing, the geometry to the right will be used for **boring** the I.D. and will be used to illustrate boring techniques including pre-drilling. Chain an open curve on this geometry.

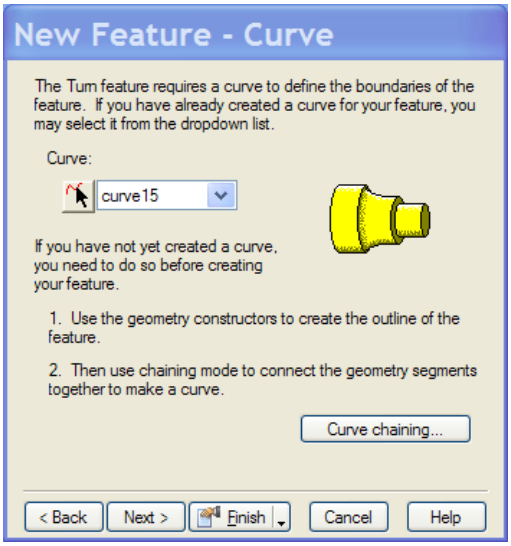
Turn Feature

- On the O.D., this curve will be used to determine the shape and area to be machined.

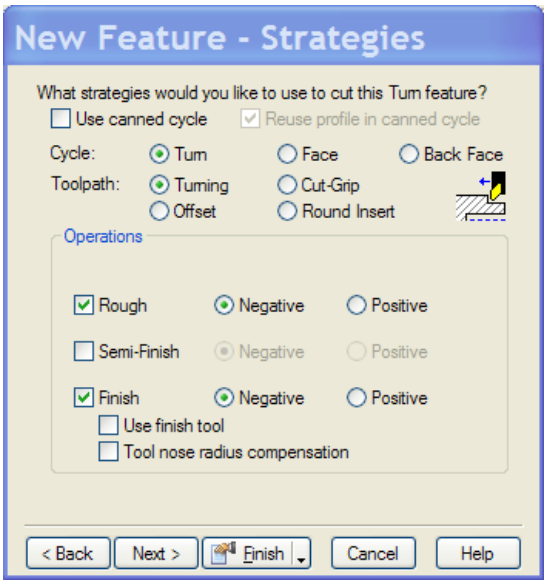
- Click on **New Feature** from Steps and select Turn from curve. Click **Next**.



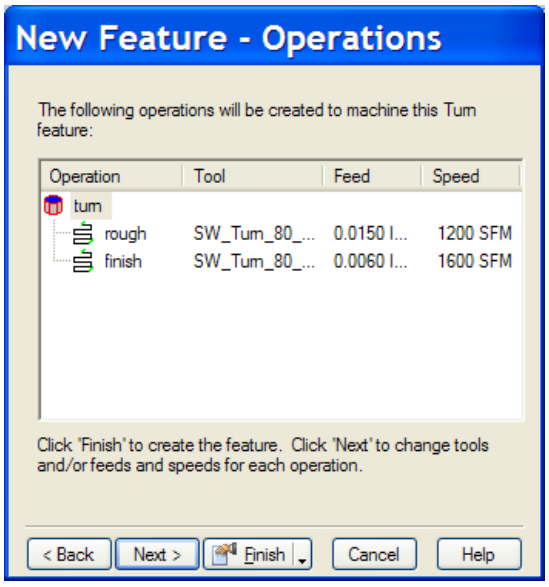
- Click on the **arrow** to select the curve from the Part View list or click on the curve in the Graphics Window. Your curve number may be different. Click **Next**. Click **Next**.



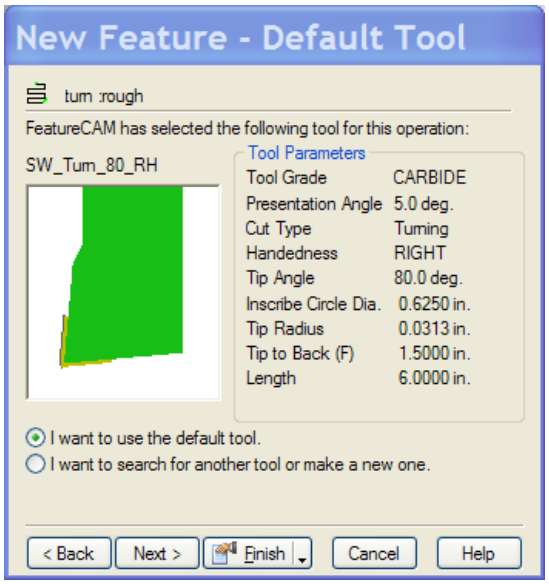
- In Strategies select:
- Cycle: Turn,
- Toolpath: Turning
- Check Rough/Negative
- Check Finish Negative. Only check Use Finish tool if you want to add another tool besides the rougher to do the finish.
- Rough and Finish will cut towards the headstock. Click **Next**.



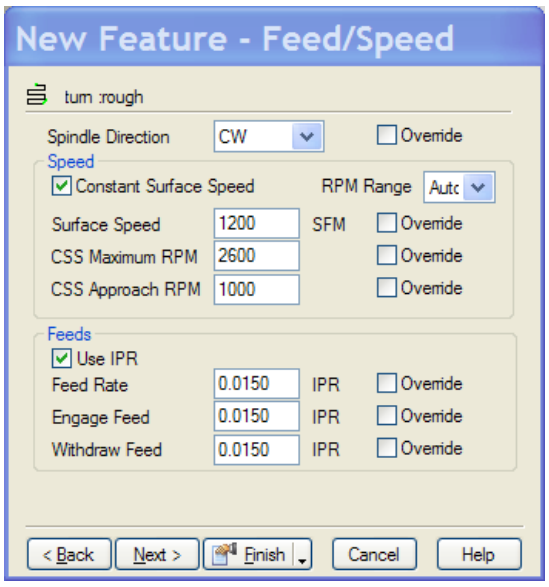
- New Feature Operations are displayed including the Operation, Tooling, Feed and Speed.
  Click **Next**.



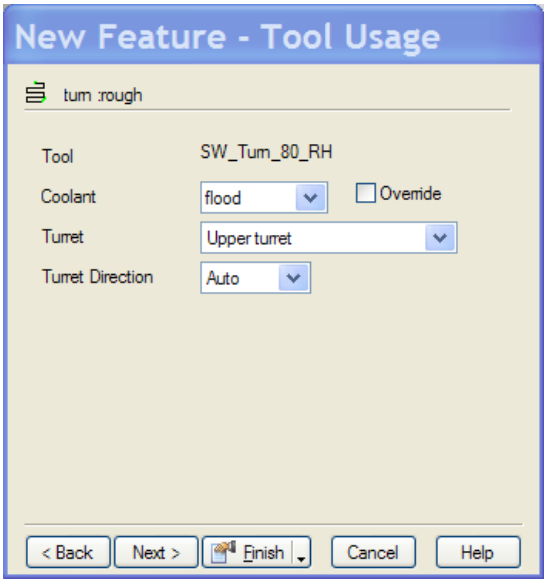
- The Default tool with description that FeatureCAM selected automatically is shown and provides the opportunity to change or modify the tool.
  Click **Next**.



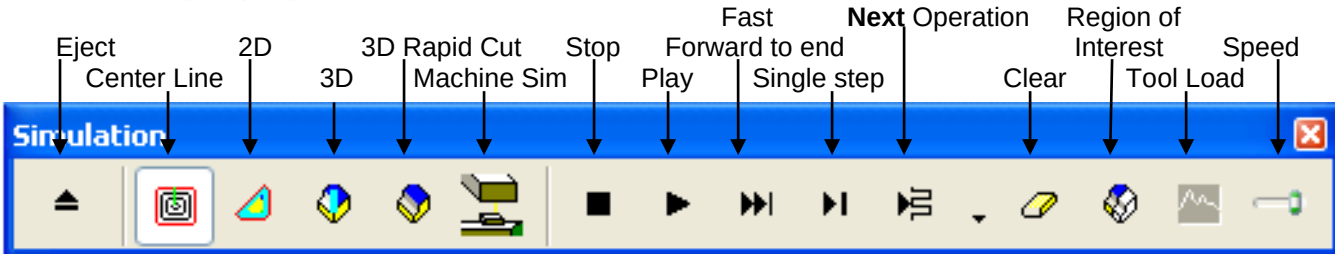
- Tooling Feeds and speeds and Spindle Direction, clockwise direction looking from the headstock to the tailstock. Click **Next**.

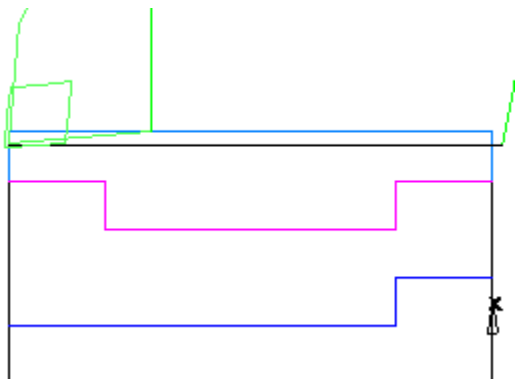


- A variety of coolant applications may be selected, default is flood but each feature may be changed to which best suits the tooling. Click Finish and Ok.

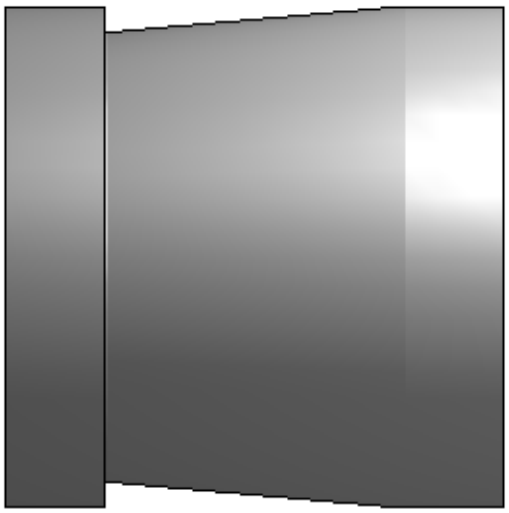


- Run Simulation. If the Simulation bar is not visible, Click on **Toolpaths** under Steps and it will fly out, click on the blue at the top of the bar, hold left mouse button down and drag to a location of your choosing. Centerline shows the center of the toolpath, green color is rapid travel and black is feed rate. 3D shows the stock in solid form and the solid tool removing the material and also reveals gouging.

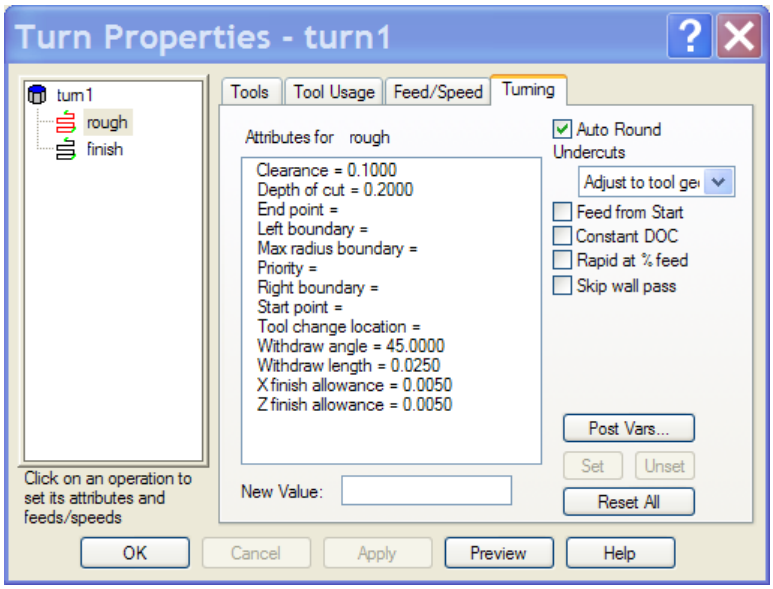




FeatureCAM defaults to an 80 degree diamond insert tool. A very common turning tool but in this case with the undercut in the O.D. you noticed that the integrity of the shape (curve) that you are cutting is protected so that the trailing side of the tool does not gouge the undercut. FeatureCAM will cut as much as it can though.



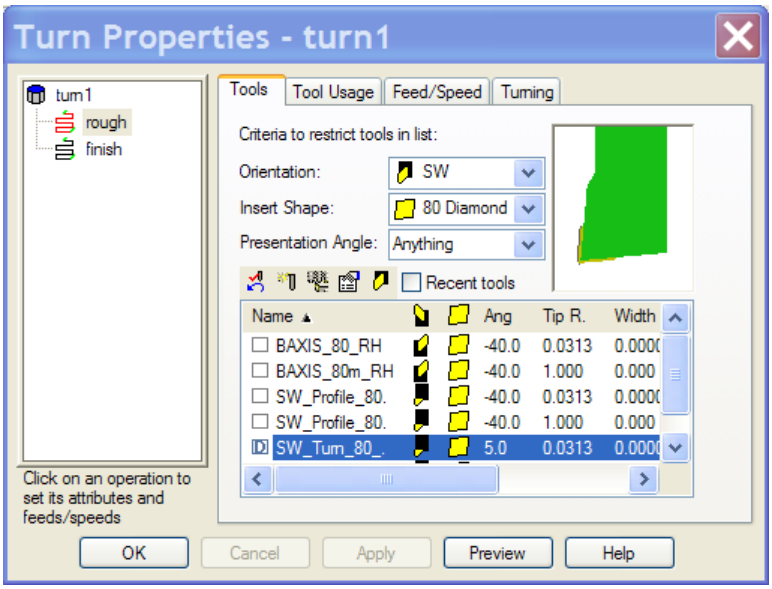
- This protection comes from the Turn Properties of this feature, Double Click on the pink of the feature in the Graphics Window to open the properties. Click on rough operation, turning tab and you see the heading Undercuts. The drop down menu reveals a setting, Adjust to tool geometry, so FeatureCAM sees the trailing edge and always prevents it from gouging into the material, saving both the part and the tool.

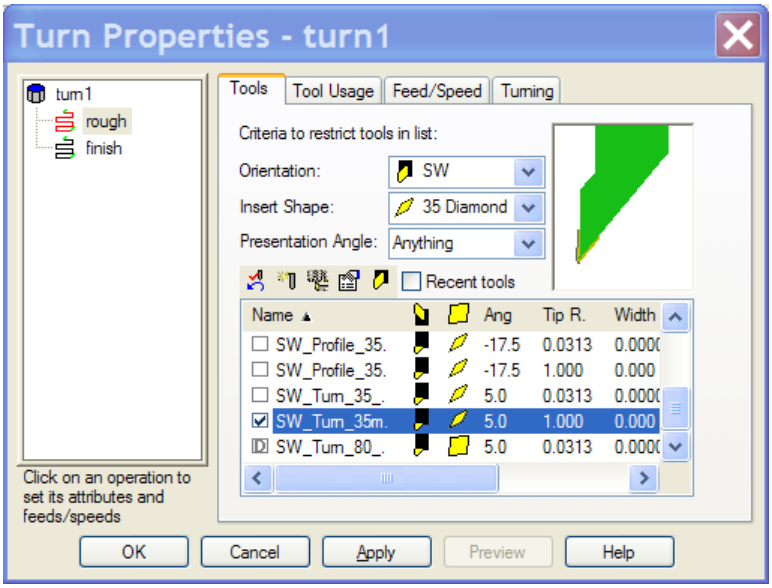


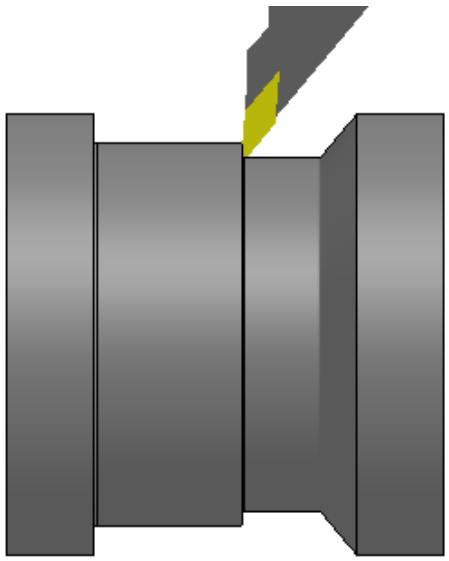
The 80 degree diamond tool may be the most used style in industry today. It is strong and allows for turning and facing with the same tool because of the 5 degree relief on two sides of the insert. In this case though by introducing the undercut in the turning operation this tool is unable to even finish one end of the undercut.

FeatureCAM allows for alternative tools to be selected from the tool crib so we can replace the 80 degree tool with a tool with more relief on the back side and this can be done on the fly as you create the feature. We will exchange this tool for a 35 degree tool with much more clearance.

- Again in the Properties, Click rough and then the Tools tab if this page is not visible. This is a listing of the tool that FeatureCAM picked as a default tool with a D in the check box and other tools that may be chosen to replace the 80 degree tool.
- First, in the Orientation box the tool is described as SW or South West. This is the orientation of the current tool and will be the same for the replacement.
- Drop the menu down in the Insert Shape box and select 35 Diamond.
- The presentation angle will remain the same.
- When you Clicked on the 35 Diamond, FeatureCAM took you to the 35 Diamond tools but does not select one, you must do that manually.
- Click on each tool description until you find the right tool displayed for the application. You must check the box to the left, highlighting it is not enough. The check in the box means the Default tool has been over ridden. FeatureCAM will change the finisher also.
- Run the 3D Simulation. It is readily visible you can cut much more of the undercut now then you could with the previous tool.
- The material in the corner on the right side of the undercut needs to be removed to finish the O.D. of the part.

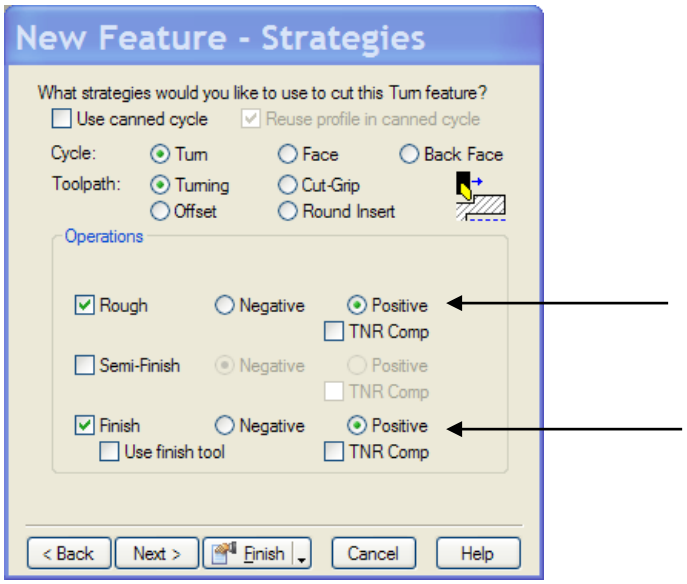




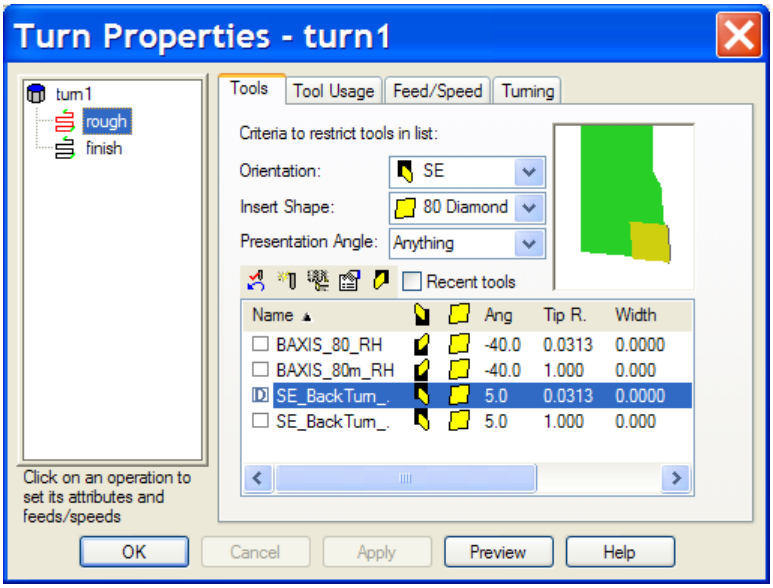


- Create another feature the same way as the first.
- Click on New Feature to create another turning feature and use the same curve.

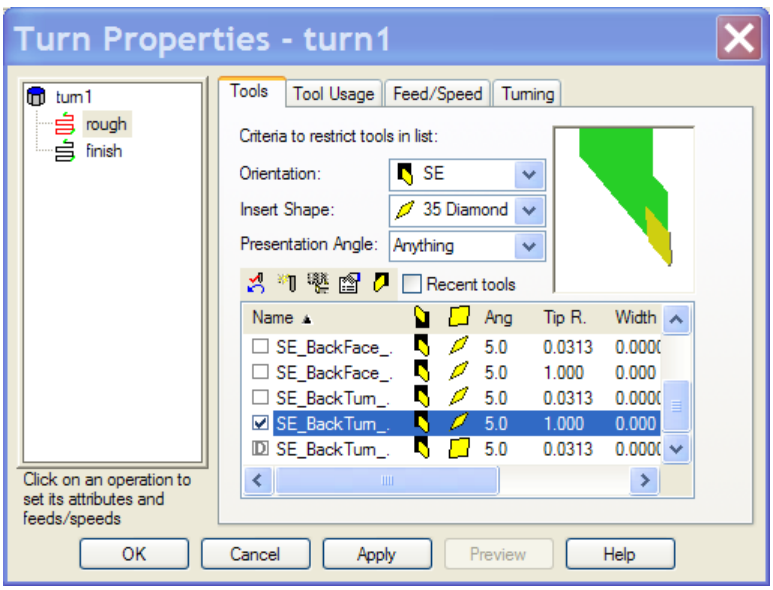
There are two differences from the first. One is the Rough and Finish will cut in a positive direction, away from the headstock.
The second is that the tool will default to 80 degree SE Back Turn.
Although the 80 degree tool may fit in the undercut now, change it to 35 degree.



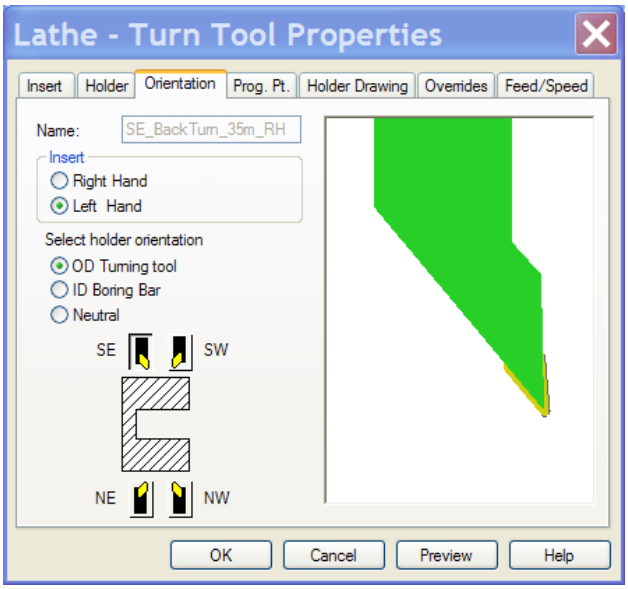
- Using the drop down for Insert Shape select 35 Diamond.
- Select a 35 degree tool that addresses the part as the 80 degree tool did. If This tool however is a right hand tool it must be changed to a left hand so it will sit in the turret with the insert on the bottom of the holder the same as the other 35 degree tool cutting negative. This puts the tool on center and the spindle continues to rotate clockwise.



- Double Click on the highlighted blue area of the tool description and this will open the properties of the tool.

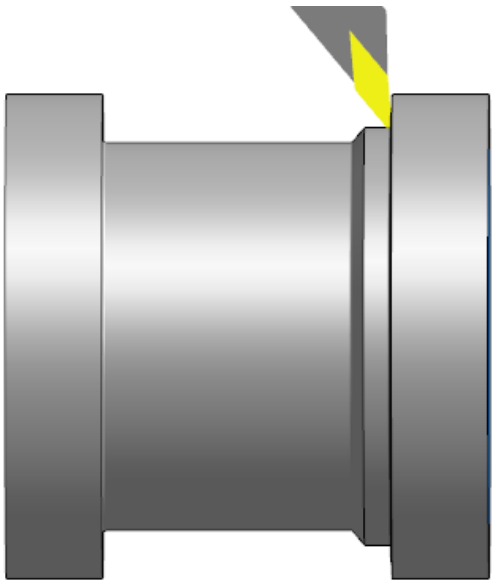


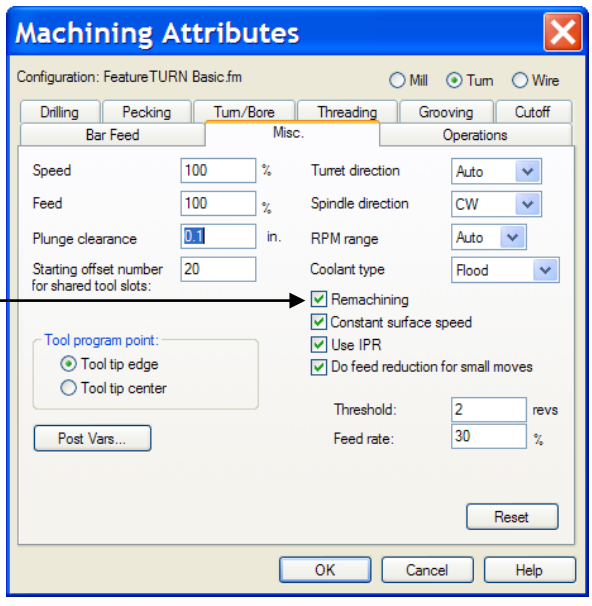
Click on the Orientation tab and select Left Hand, the insert is now located on the bottom. FeatureCAM sees the Orientation of the tool and selects the proper spindle rotation, clockwise. Click OK. Apply. OK.



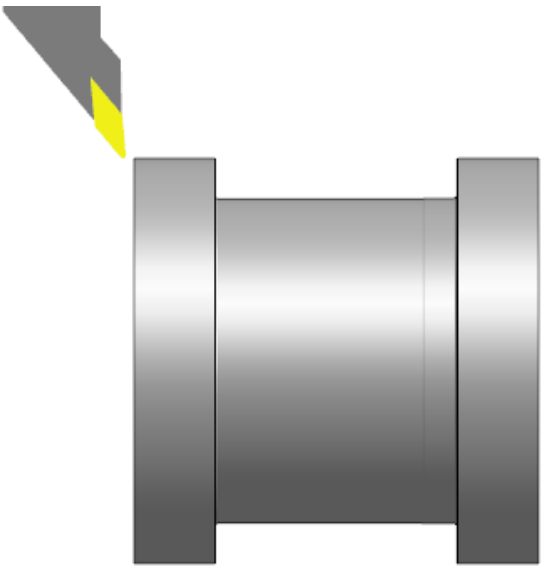
- Notice that the Back Turning tool is roughing only what the first Turning feature could not reach. This second feature sees what was left behind and only roughs this area. The reason it does this is because of a setting in the Machining Attributes, located under Manufacturing and Machining Attributes.
- Click on Manufacturing>Machining Attributes>Misc tab and you will see a selection called Remachining and when the box is checked the second tool can identify where the previous tool has removed material and machine only the remaining stock.

IMPORTANT: THE SECOND FEATURE MUST USE THE SAME CURVE AS THE FIRST FEATURE IN ORDER FOR THE SECOND TO KNOW WHAT AREA NEEDS TO BE MACHINED.

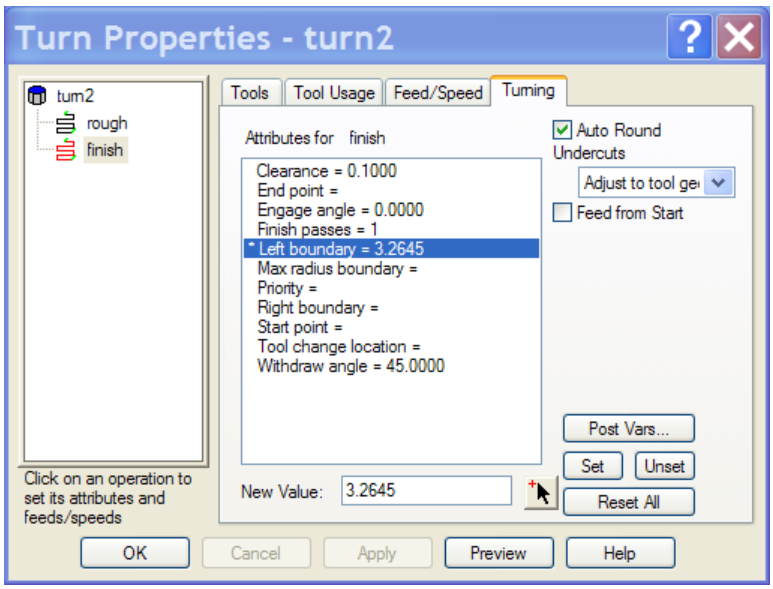


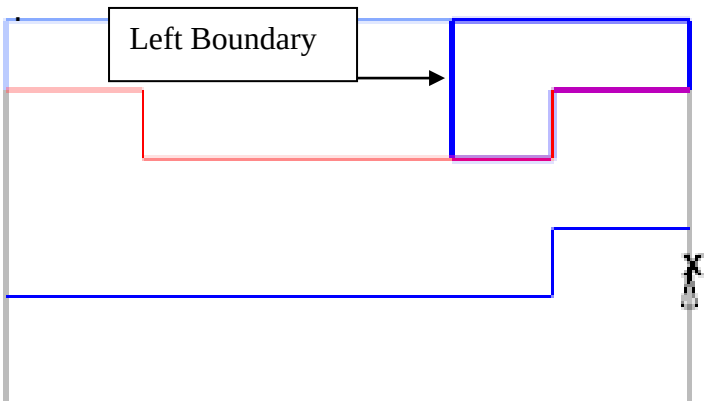


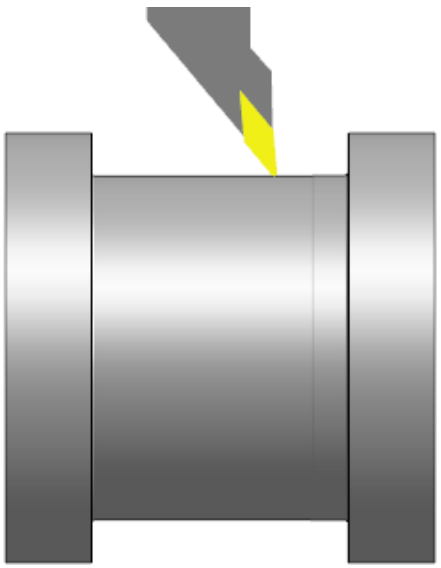
- The finisher however goes to the end of the curve to do the finish cut and needs to be controlled with the Left Boundary limit.
- In Part View, Double Click on the second turn feature and open the Properties page>Finish operation>Turning tab. Under Attributes for finish there is a selection called Left Boundary.



- Click on Left Boundary and either Click on the Arrow **Next** to the New Value box at the bottom and then click the location on the part in the Graphics Window where you want the finish cut to start **OR** enter a dimension using a positive number measuring from the Left end of the Curve. The finisher will then start in that location and blend to the previous finish cut.

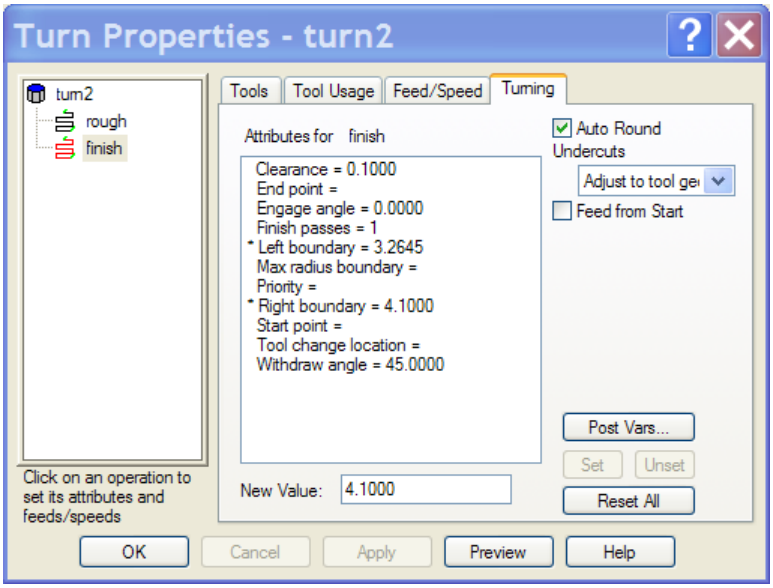


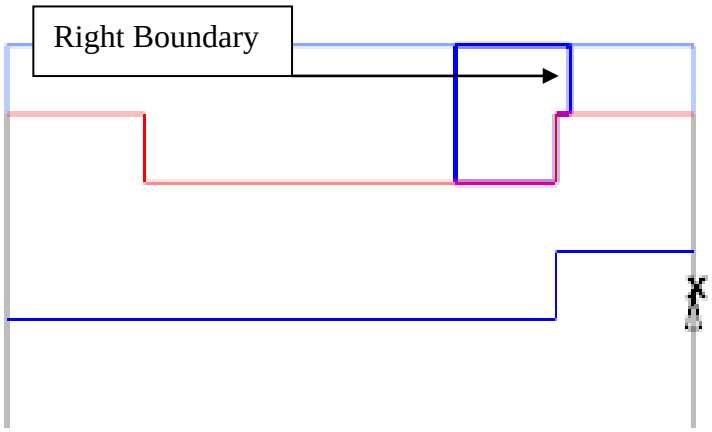


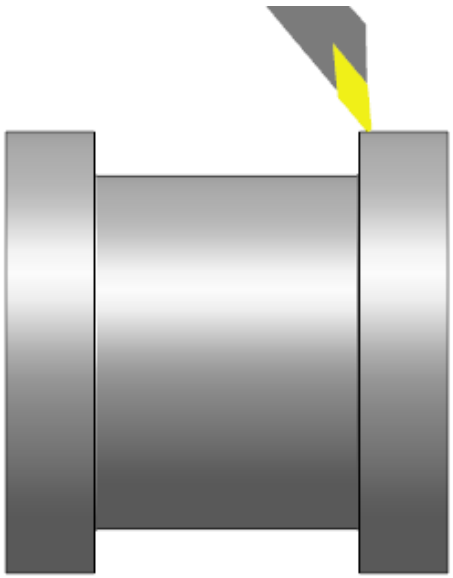


Now Set the Right Boundary.

- Click on Right Boundary and set to 4.1, this will feed the tool just around the corner on the O.D. but not feed to the end of the curve, this was already finish machined in the first feature.

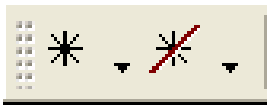






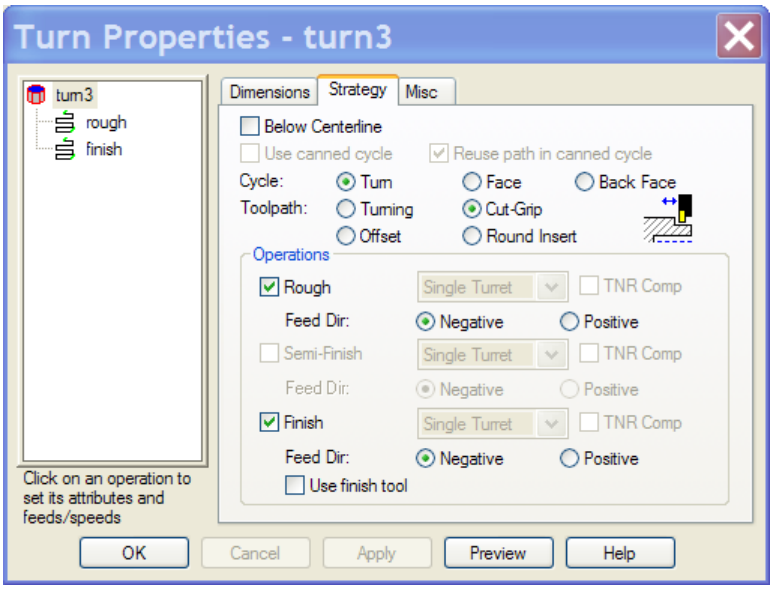
The O.D. and undercut are now complete using turning, back turning and right and left boundaries.

In Part View, Setup 1, uncheck ALL of the boxes, this deactivates the features and they will not simulate. Also using the hide button on the upper tool bar, usually an asterisk, one is hide the other is show, drop the menu and hide all features. This only hides the previous features but does not delete them; they may be recalled anytime by rechecking in Part View and then using the show button to show all features.

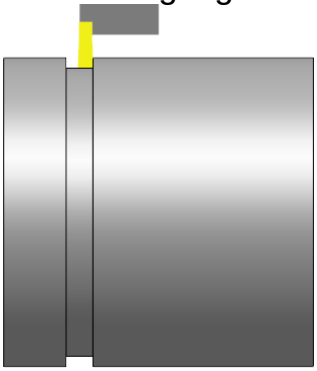


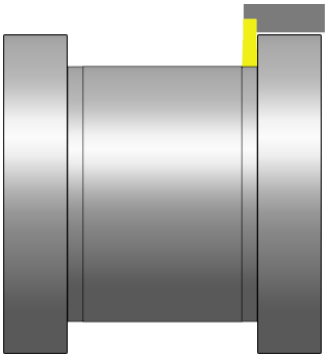
- Click on New Feature and create a turn feature the same as before using the same curve.

- This time in the Strategy Tab select:
- Cycle: Turn
- Toolpath: Cut-Grip
- Rough: Negative
- Finish: Negative



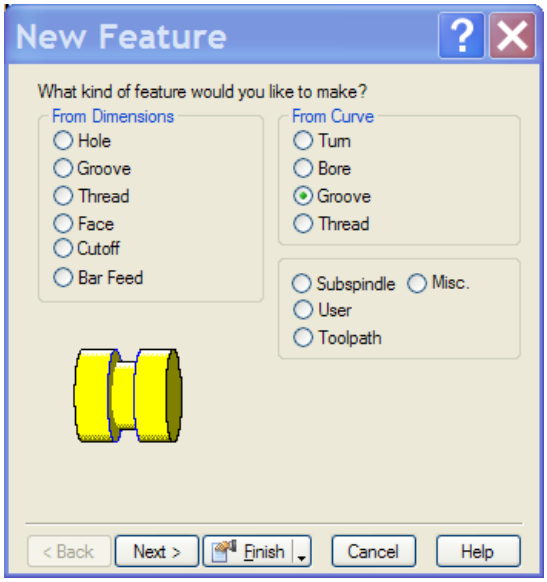
Cut-Grip is a specially designed tool for **Turning** (also grooving) and looks much like a standard grooving tool but because of the geometry of the insert and the design of the holder it can cut parallel to the Z axis almost as fast as a standard turning tool. In large quantity production machining speed is of the essence and the Cut-Grip toolpath that FeatureCAM provides is a Zigzag cut, feeding toward the chuck and the **Next** pass feeds away from the chuck without retracting and starting at the end every time. It also finishes the undercut without changing tools and having to perform a back turning feature, again saving time. A standard grooving tool may be used in a Turn feature but does not Zigzag and is designated in the Tool Properties, Insert tab.



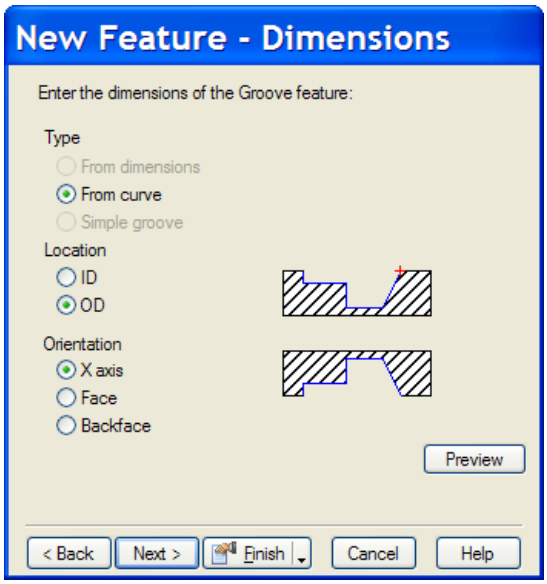


Groove Feature

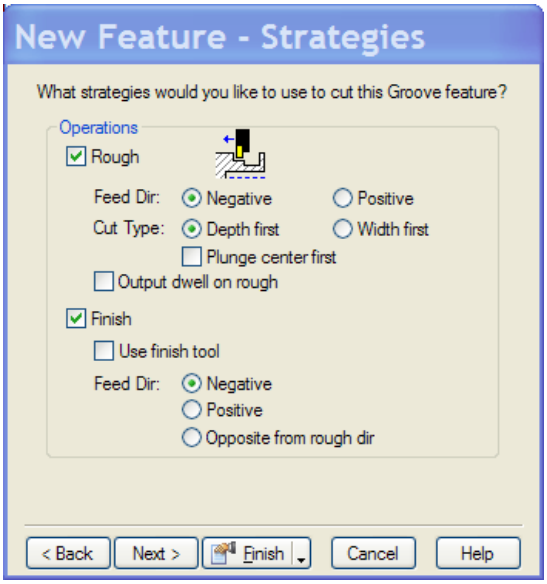
- The **Next** feature to consider is grooving. Click on New Feature and select Groove from a curve. Click **Next** or open **FeatureTURN Basic.fm**.
- Select the curve for the undercut only (U shaped), either from the Graphics Window or under Part View. Click **Next** and **Next** again.



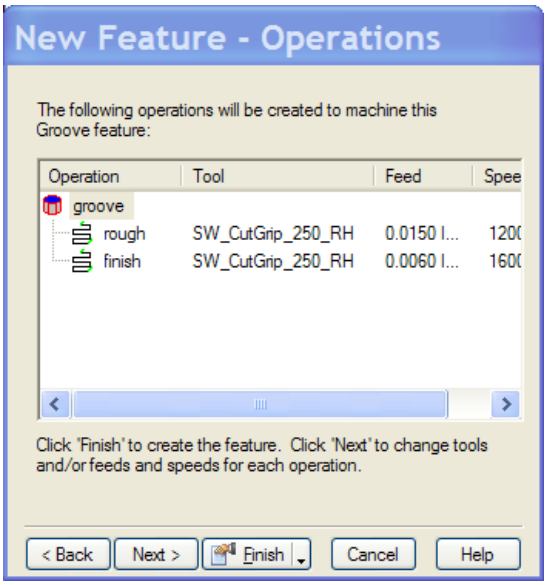
- Select the following
- Type: From Curve
- Location: OD
- Orientation: X axis



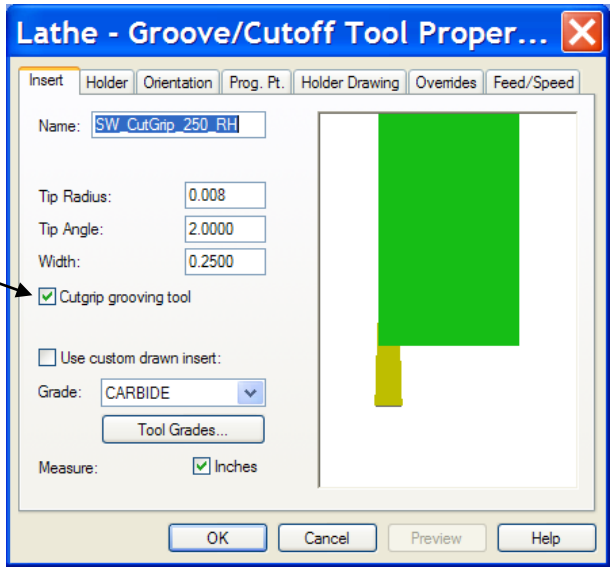
- Select the following
- Rough
- Feed Direction: Negative
- Cut Type: Depth First
- Finish
- Feed Direction: Negative.



The Cut Grip Tool may have defaulted which can also groove but the Groove feature does not have Left and Right Boundary controls like Cut Grip and does not Zigzag.

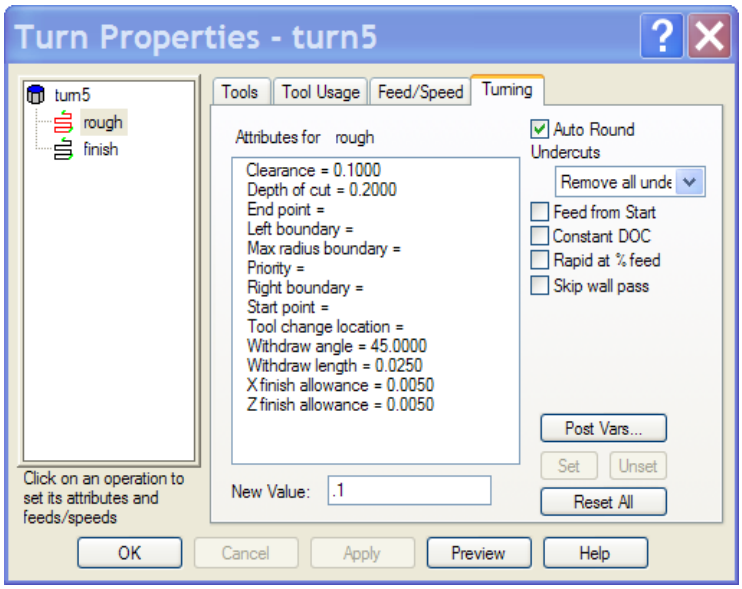


In the tool illustration the tools look the same but the Groove/Cutoff tool Properties and Insert tab contains the check box to designate if it is intended to perform the Cut Grip function. In the case of this groove either will work.

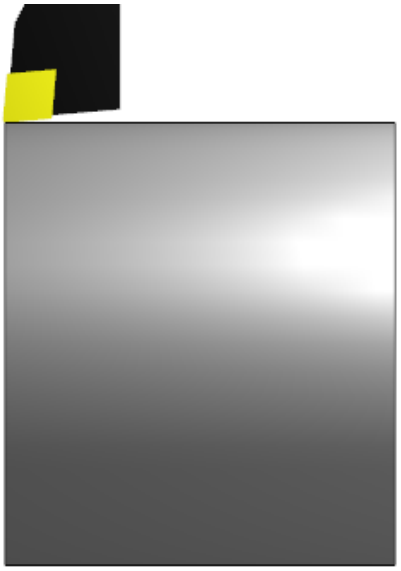


Note: In order to perform the groove function, the OD must be turned to finished size first.

- Click on new Feature and select Turn from curve and create a Turn feature just like the first, using the same curve. An 80 degree tool is ok for this feature. The tool will want to feed down into the undercut again but there is a setting to prevent this under the rough/turning tab then Undercuts/Remove all undercuts (drop down menu).
- Do the same to the finisher.

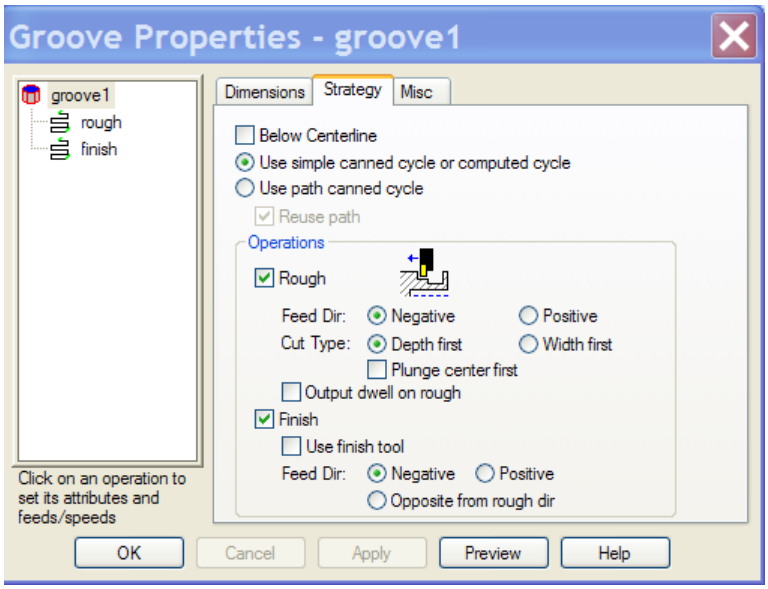


The tool stays out of the undercut and removes the material from the OD before grooving.



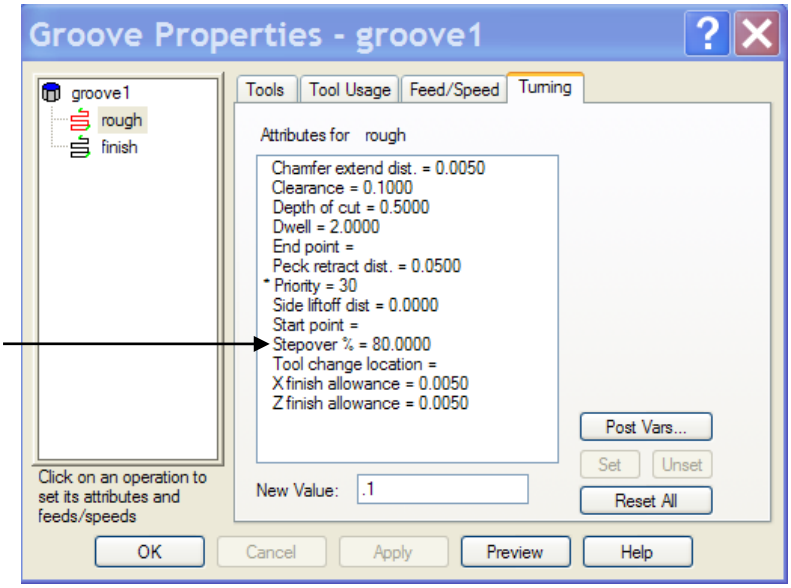
Simple Canned Cycle

- On the Strategy tab Click on simple canned cycle when on any rectangular simple groove feature (straight walls and no chamfer or bottom radius) will be posted out as a single line of code. Post and control have to support this.

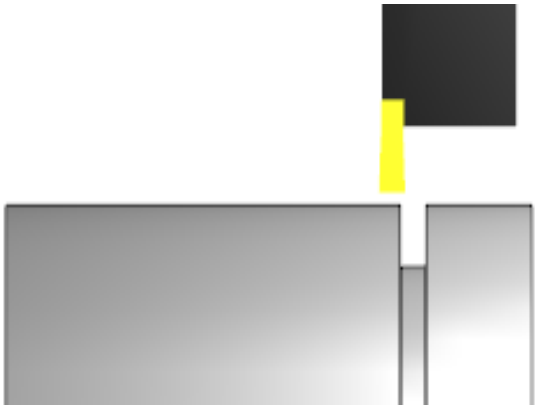


Rough/Negative/Depth first

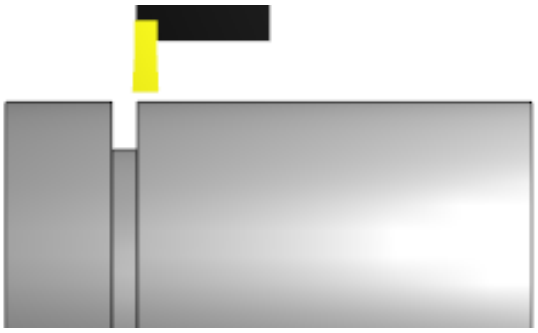
The Rough operation starts at the end of the curve farthest away from the chuck then plunges to full depth, retracts and steps over a designated distance toward the chuck. The step over is controlled by rough/turning/Stepover, a percentage of tool width.



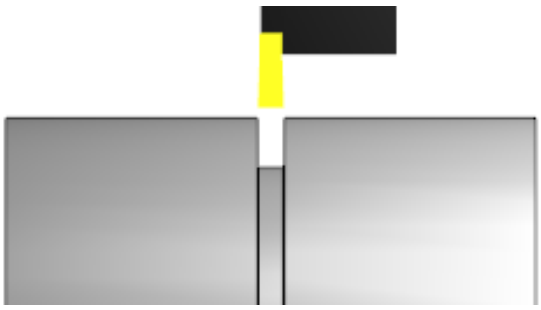
Rough/Negative/Depth first
Starts far end steps over toward chuck.



Rough/Positive/Depth first.
Starts near end and steps over away from the chuck.

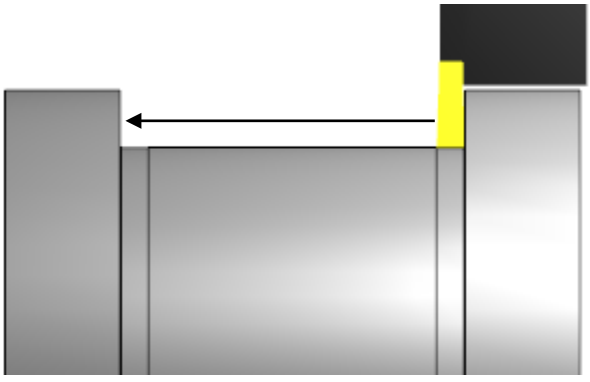


Plunge center first.
Tool plunges in the center of the curve and then alternates stepping over negative and positive until roughed out.



Rough/Negative/Width first.
Instead of plunging to full depth the tool plunges to a depth of something less, controlled by rough/turning/depth of cut then steps over the full width of the groove and starts over continuing until the groove is roughed out to full rough depth.

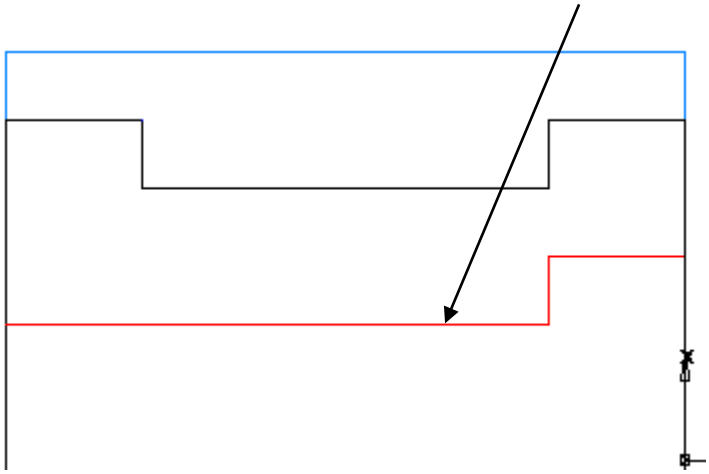
Finish/Negative
The tool cuts the vertical wall to the finished depth on the end closest to the chuck, retracts then travels to the opposite end of the groove and cuts that vertical wall to the finished depth then feeds parallel to Z axis to the other wall finishing the bottom of the groove.



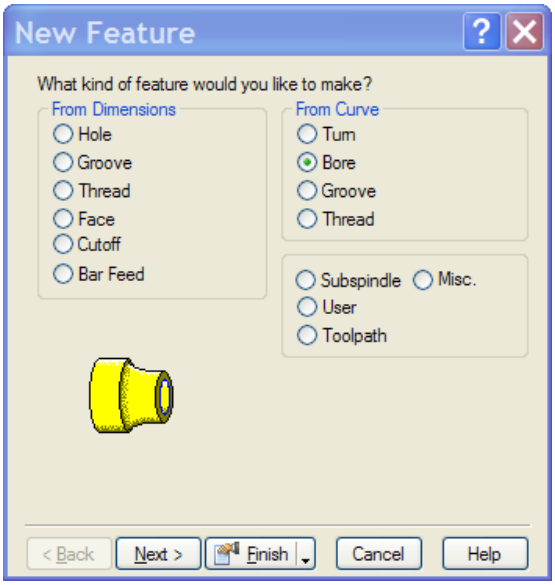
Boring

Boring operations require a curve to create the feature. These passes are parallel to Z.

- Open **FeatureTURN Basic Bore.fm** in the Data folder. Select this **curve**.

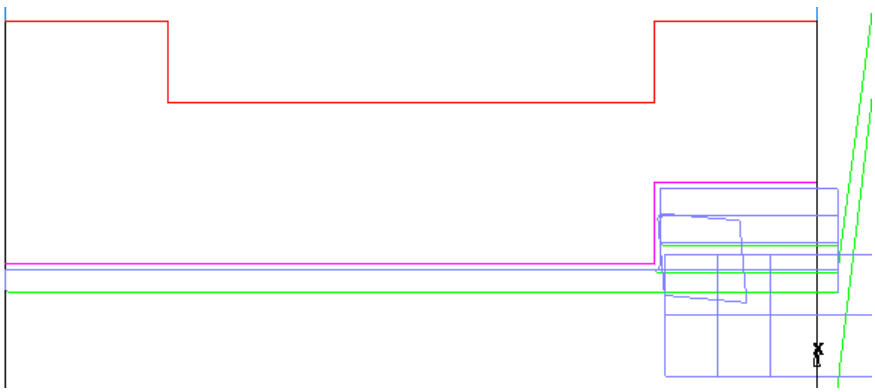


- Click on **New Feature** and select **Bore**. **Next**.
- Select curve. **Next** and **Next**.
- In Strategies Select:
- Cycle: Bore
- Toolpath: Turning
- Operation: Pre-Drill Dia1.750, Depth 5.0, Z 0
- Operations: Rough/Negative
- Operations: Finish/Negative



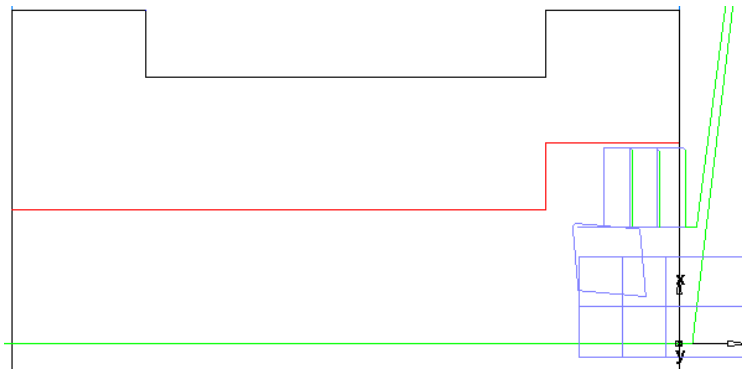
If the material has a hole already drilled through the center then of course you would not need to use pre-drill.

- Run centerline simulation.



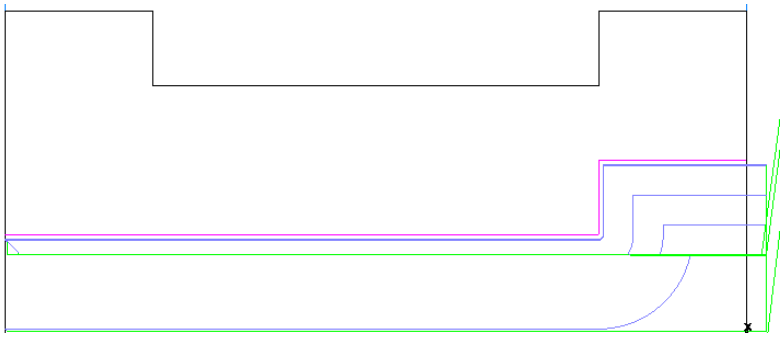
Face Boring

Face Boring is a series of facing moves from inside the drilled hole cutting parallel to X in a positive direction.



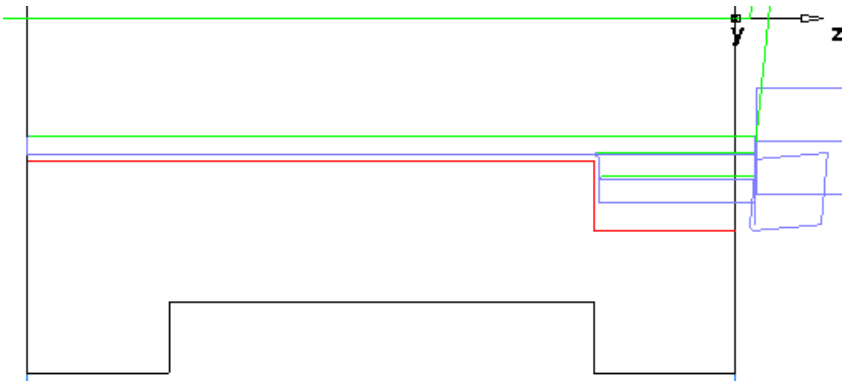
Offset Boring

Offset boring offsets the curve shape to the beginning of the cut. Mostly used for a casting that has a similar shape to the finished curve.



Boring below Centerline

When boring below centerline, though possible in FeatureCAM, the boring bar must be inverted and mounted on the turret so the machine doesn't exceed the travel limits in the X negative direction.



Boring with Canned Cycles

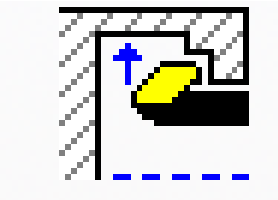
Canned cycles can be generated in the NC code for nearly every turned feature. To generate these macros, your post processor must support them, and you must turn this function on for the post and for some features you must also activate the canned cycles on the feature level. **Reuse path in canned cycle** outputs the path geometry only once for both roughing and finishing.

Tool Nose Radius Compensation

Tool nose radius compensation ignores the tool radius when generating semi-finishing and finishing passes for turn, bore and facing features. The actual part geometry is output as the toolpath. It is assumed that the tool radius compensation will be performed by the operator at the machine tool when **Tool nose radius compensation** is activated.

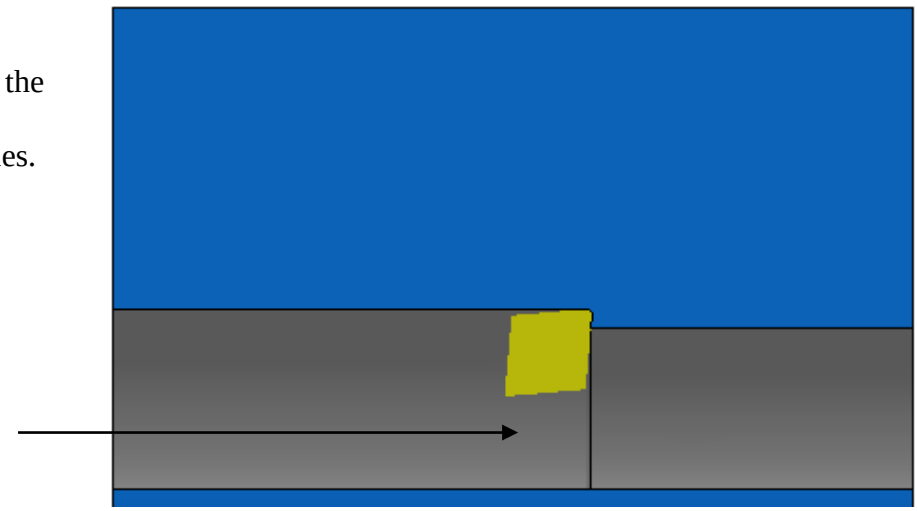
Back Face Boring

Back face boring is usually used when a previous boring operation couldn't clean up all of the material because of an undercut. A special bar must be used specifically designed to cut with relief on the face. Moves are parallel to X positive.



Back Boring

Back Boring cuts parallel to the Z and is performed using a positive direction in Strategies.

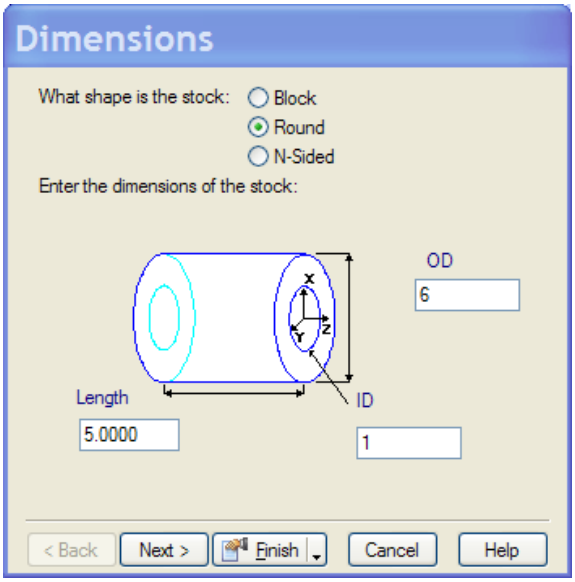


Boring with a groove tool

When there is an undercut in the bore, a groove tool can be used. If Cut-Grip is checked in the tools properties on the Insert tab the tool path will be a zigzag motion, alternating negative and positive directions parallel to the Z. If the box is not checked it will be all negative or all positive cuts according to the settings in the Strategies.

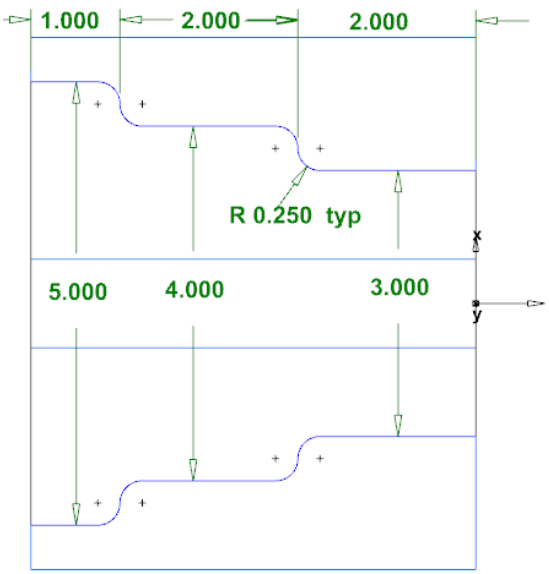
Turning with no under cuts

- Open a New Turn Document, top tool bar, extreme left.
- Round Stock. OD 6, Length 5, ID 1. **Next**.
- Aluminum Material. **Next**.
- No Multi-Axis. **Next** and **Next** again.
- Setup, Use current location. **Next**.
- Main Spindle. **Next**.
- **Finish** and **OK**.

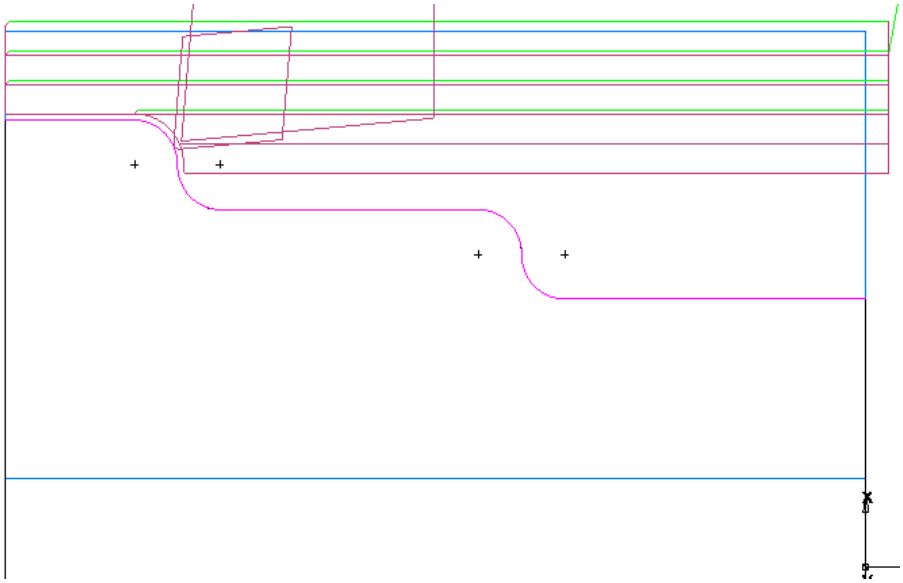


- Draw this part in the stock in the Graphics Window using geometry and then chain a curve to the geometry or open **FeatureTURN Basic2.fm** file.

Remember that the geometry only needs to be drawn above the centerline.



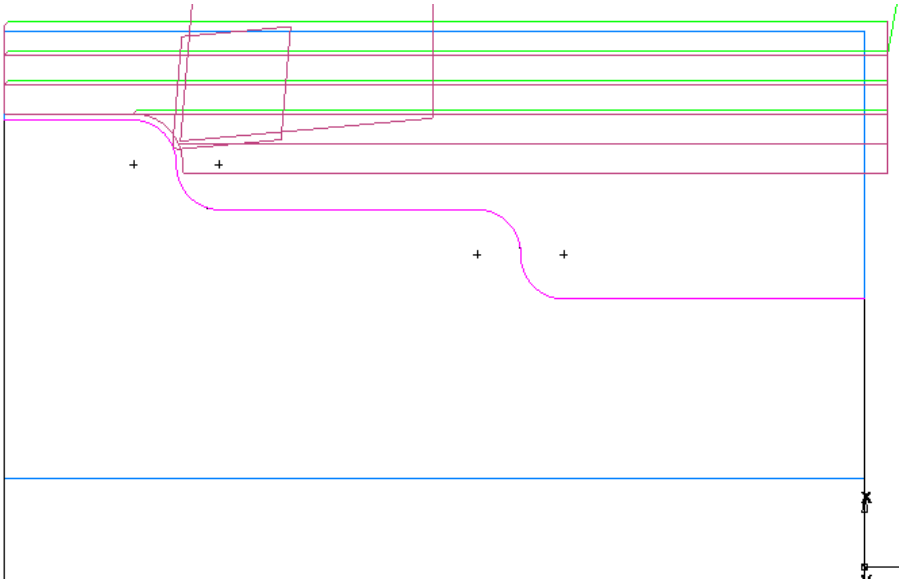
Roughing Cuts with specified depth and finish allowance.



Click on New Feature and select Turning. **Next**.

- Select Turn from curve. **Next**.
- Select the curve. **Next** and **Next**.
- From the Strategies Select:
- Cycle: Turn
- Toolpath: Turning
- Operations: Rough and Finish/Negative.

The cuts are straight but move up the face to remove the scallops left by the angle of the tool.



- **Depth of cut** can be specified as a default in the attributes or click on the feature to open the properties. Then click on **rough>turning** and select Depth of Cut. It defaults to .200 but may be changed by clicking on **Depth of Cut>New Value** at the bottom and entering the new depth and then click Set at the right.

Constant Depth of Cut (DOC) The roughing operation leaves the specified finish allowance on all diameters but sometimes that makes some passes come between the .200 depths. FeatureCAM must cut each diameter to satisfy the leave allowance but the depth of cut might be less than .200. It makes this pass but starts at the end of the part when it only needs to cut a short distance on the affected diameter. It may also take extra full length cuts.
Constant depth of cut always cuts at .200 for the full length of the part and only cuts the affected diameters surface to the finish allowance when necessary and is a shorter cut. This will reduce the number of full length passes reducing cycle time.

Wall Pass moves the tool up each face that it cuts to smooth the face and can be turned off by checking the Skip Wall Pass box in the Turn Tab located on the right side.

Clearance is the distance from the part that the tool rapids too off the material and then goes into feed rate.

End Point is a specified location (vector) that the tool can be directed to retract to when operation is completed.

Max Radius Boundary is used to restrict the tool from starting the cuts from the stock boundary in the event the material has already been removed.

Priority can be used to set the order in which feature operations are executed. You should use this sparingly because it can affect the ordering that FeatureCAM does automatically.

Start Point allows you to select a location for the tool to rapid too other than the normal clearance location it then rapids to the clearance location from the Start Point. Another selection associated with start point is Feed from Start, the tool then feeds to the clearance location. An example is when supporting with a tail stock, you might choose to rapid away from the tailstock and feed to the clearance to control its approach.

Withdraw Angle controls the **angle** at which the tool retracts at the end of the cut.

Withdraw Length controls the **length** that the tool retracts at the end of the cut.

X Finish Allowance is the amount of material left on the **diameter** by the rough tool to be removed by the finish tool.

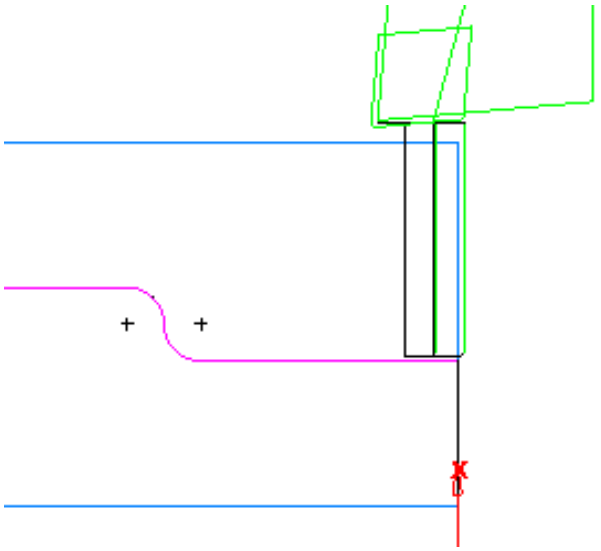
Z Finish Allowance is the amount of material left on the **face** by the rough tool to be removed by the finish tool.

Auto Round simply rolls the tool around any sharp corner to remove the sharp edge. If there is any radius at the corner then Auto Round is not active.

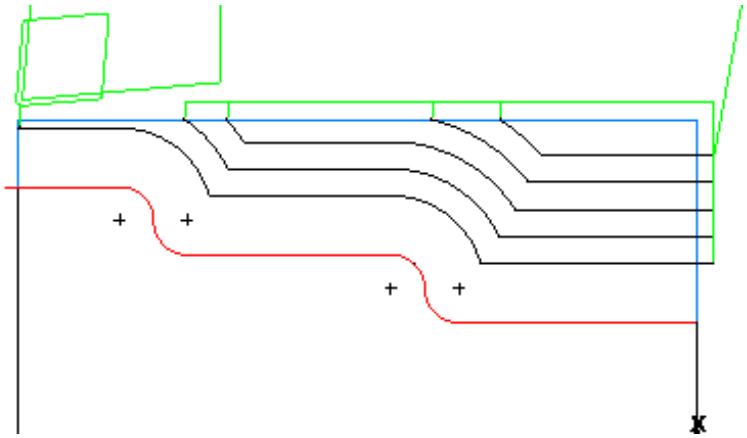
Post Variables can pass information directly to the NC code from the feature but the post processor must setup for variables.

Engage Angle is under the finish operation/turning tab and controls the approach angle of the finish tool to clearance point.

Face Turning roughs the part in facing moves starting from the outside and cuts toward the curve in the negative X axis direction instead of turning parallel to the Z axis. If the last face does not go to the stock boundary in X you may need to use the Max Radius Boundary set to the stock diameter dimension to make the tool start outside the stock.



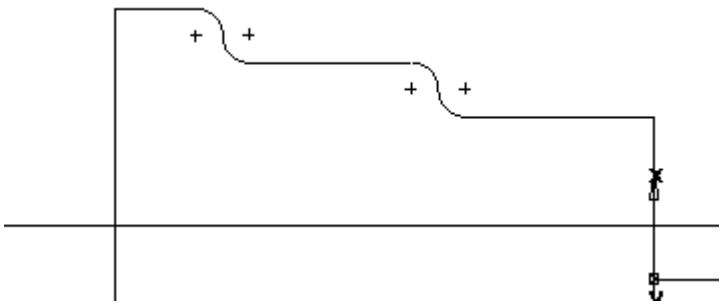
Offset Turning uses the shape of the curve and offsets it to the end of the part for the tool to follow. It begins to cut the shape of the curve on the first pass and cuts parallel until finished. Good for castings.



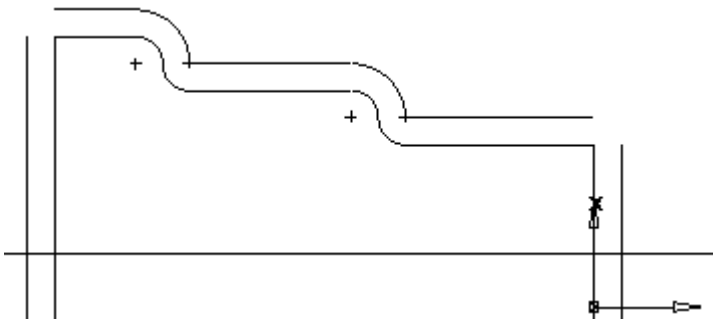
Stock Curve

A stock curve can be used to represent a specific shape of the stock rather than bar stock. An example is when a forging or casting is used for the stock. The shape of the material is near the finished part shape with enough extra material to be turned down to the finish part size and shape.

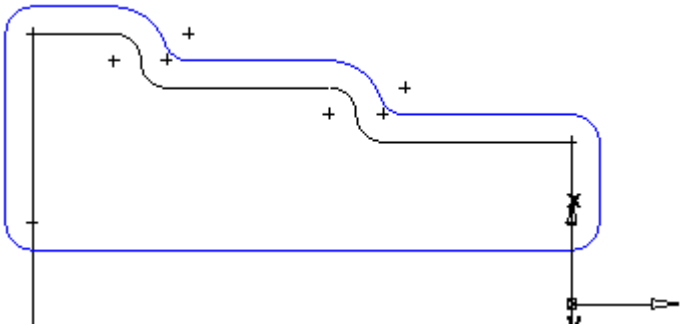
- To draw a stock shape and then chain a curve, use **FeatureTURN Basic2.fm** file and hide all features, curves and stock. Draw a horizontal line at a 1.000 diameter.



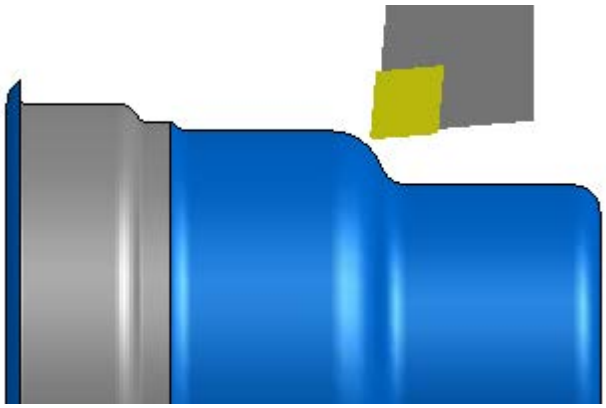
- Offset all geometry by .250 as illustrated.



- Radius the remaining corners at .250 and chain a curve around this shape.



- Click on **Stock** under Part View then on the Dimension tab click on **Stock Curve** button and select the curve. The stock now takes the shape of this curve and FeatureCAM recognizes this as the stock boundary instead of the bar stock and will rapid to the clearance distance and not cut air. Previous toolpath strategies apply.

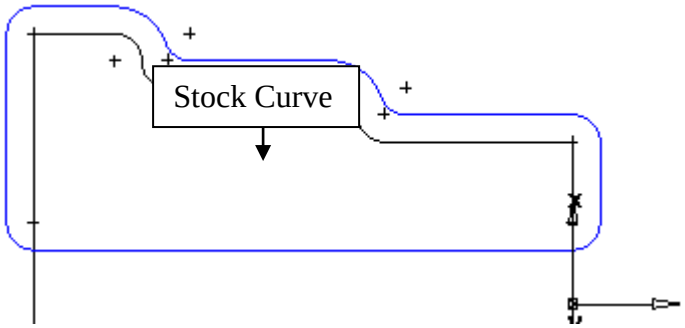


The purpose of using the following tool is to split the curve and examine the curve pieces.

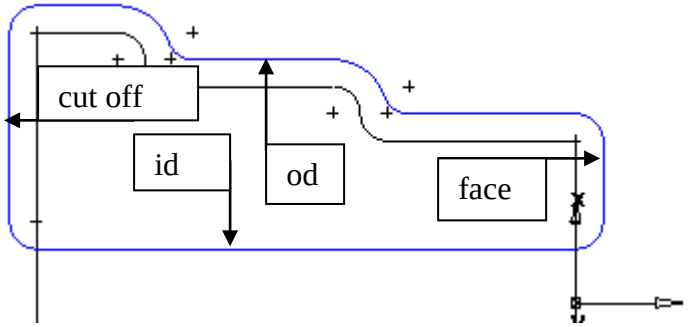
Turn Stock Boundary Tool

- Open **Turn Stock Boundary Tool.fm** in the Data folder.

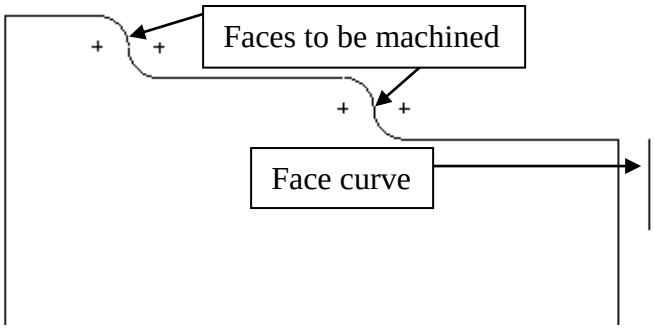
When creating a custom boundary for your stock such as a shape to represent a casting or forging you can draw the shape and use a closed curve which can also include a hole through the center and when you 3D simulate, the stock will take the shape of the curve and FeatureTURN will accept the shape and use it to calculate the material to be removed instead of from bar stock.



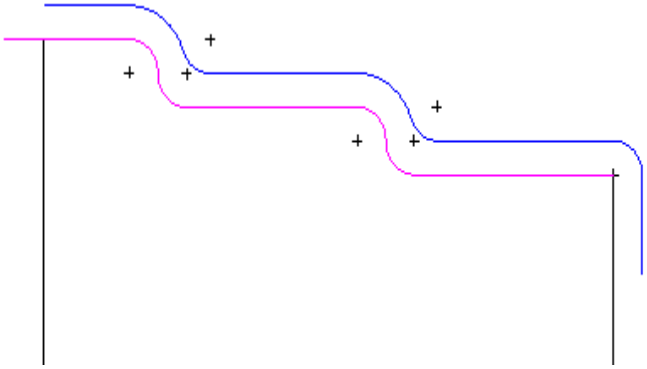
FeatureTURN has made available a development tool which can be used to**" split a selected curve or solid outline into turn curves"** and names them. This tool of the same name is located on the status bar and can be accessed by holding the control key and right clicking on the status bar and then click on the tool.



Case History example: When the turn feature called "Face – Turning" from the strategy tab was used and a closed curve for the Stock Boundary was also used an error ***"Error: TPD19: Feature curve start or end point in material"*** was displayed. The reason for this was the Stock Boundary curve must **shade** the faces to be machined in order to perform the cuts and the piece of the Boundary curve called **"face"** did not shade or cover the faces.



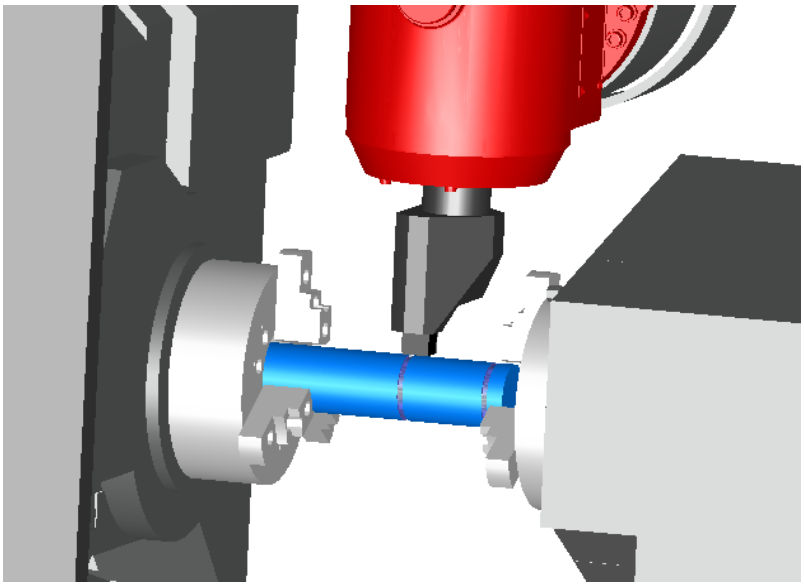
The **Feature also** contains a **stock curve** and an additional **new curve** to cover the **od** and **face** had to be created an entered in the **Feature Properties Dimension Tab** by clicking on the **Stock Curve Button** and select the new curve in the **Feature only**. The **Original Closed Curve** was still used as the **Stock Boundary** in the Stock Properties.



Subspindle features

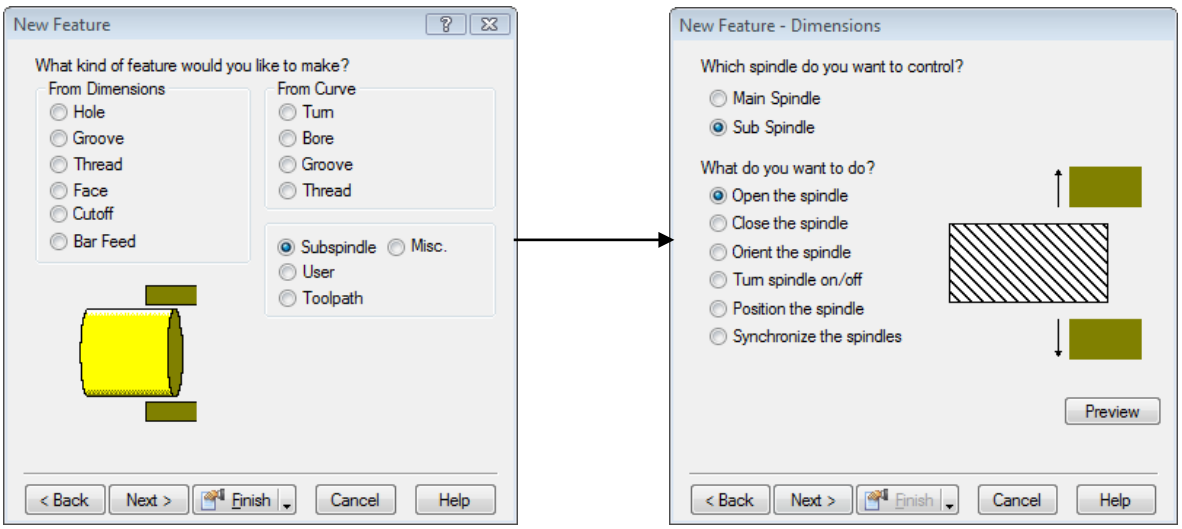
Introduction

Subspindle features are essential when manufacturing parts in turning centers with multiple spindles. Like the name implies, they can control either one of the spindles in the machine at any given time. The following chapter will explain some of the most familiar applications used in the field nowadays.



Common usage

Subspindles can be used to support the part from both ends of the stock or to change which spindle is used to hold the stock (during a part transfer for example). There is only one feature-type for sub-spindles, but it performs many different tasks. You will access the subspindle feature through **New Feature**, then **Turning**



The process to make operations is quite simple, you first specify which spindle you want to control (either the main or sub-spindle) and then select the action you want to perform. The list below offers a brief description of each command and its application:

Open the spindle - Opens the current spindle.

Close the spindle - Closes the current spindle.

Orient the spindle - Rotates the current spindle. Note that FeatureCAM orients the spindle during cutting. This feature type is only needed to orient the spindle before grabbing the part or initializing the spindle position.

Turn spindle on/off - Direct control over rotating the spindle or turning it off. FeatureCAM automatically controls the spindle, but this feature type may be necessary to provide precise control of the spindle when moving from one spindle to the other.

Position the spindle - Direct positioning of the current spindle.

Synchronize the spindles - Synchronizing spindle rotations for milling or turning.

Basic subspindle feature example

FeatureTURN provides individual control over the different subspindle functions. The user creates the individual operations and then arranges them in the right order to achieve the desired result. For example:

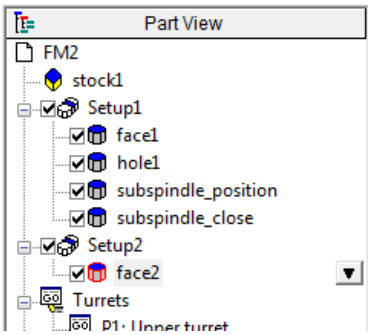
Switch from the main spindle to the subspindle

- Subspindle position
- Subspindle close
- Main spindle open
- Subspindle position

Switch from the main spindle to the subspindle using a cutoff feature from bar

- Subspindle position
- Subspindle close
- Cutoff feature
- Subspindle position

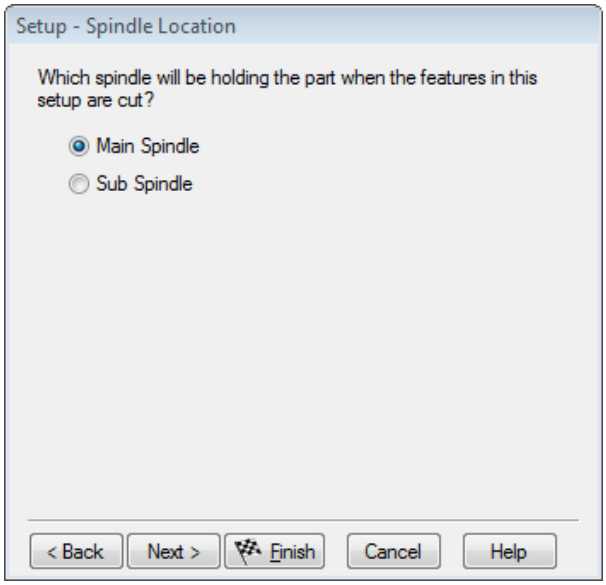
Something important is that the features must be included in the proper setups. The subspindle commands can be located at the end of the main setup or the beginning of the subspindle setup. The part view, shown below, has the subspindle commands at the end of the first setup. If you are using the subspindle to support the end of the stock, order the subspindle features so that they occur when you need the extra support.



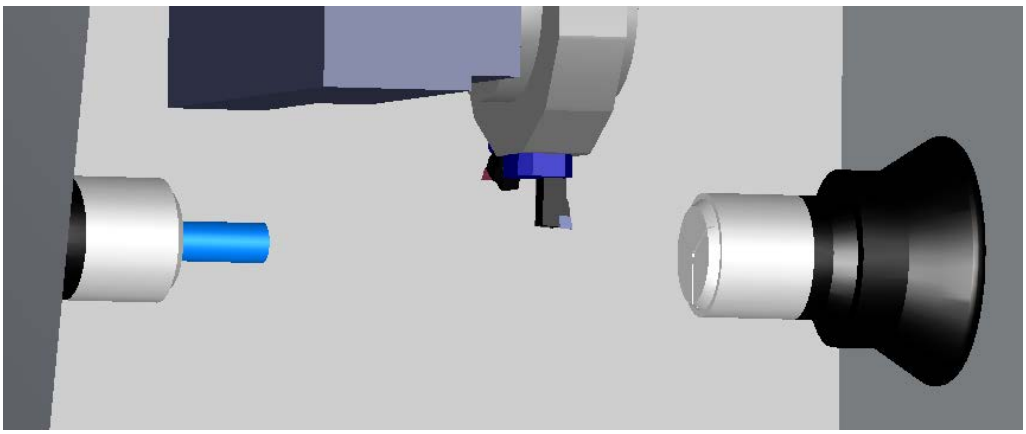
- Click on **File** then **Open**. Browse and open **Subspindle.fm**
- You will need to use the **Naka WT300** post for this exercise
- Run **3D** simulation and notice how FeatureCAM machines both ends of the part

We achieved these results by creating 2 setups and assigning one as a **main** spindle setup and one as a **subspindle** setup. You can set a setup as a Main or Sub Spindle by **editing** the properties.

Taking the time to properly assign a setup will allow the software to make more automatic and intelligent decision for you (like tool selection, outputting NC code for transfers, inserting M00 between setups happening on the same spindle, etc) and to show the proper simulation results



- Now run **Machine simulation** and notice how FeatureCAM machines both ends of the part, but as we can see in the picture below, the part is still being held by the main spindle, so the subspindle only cuts air. It is time for us to do our first transfer and move the part from the main to the sub spindle



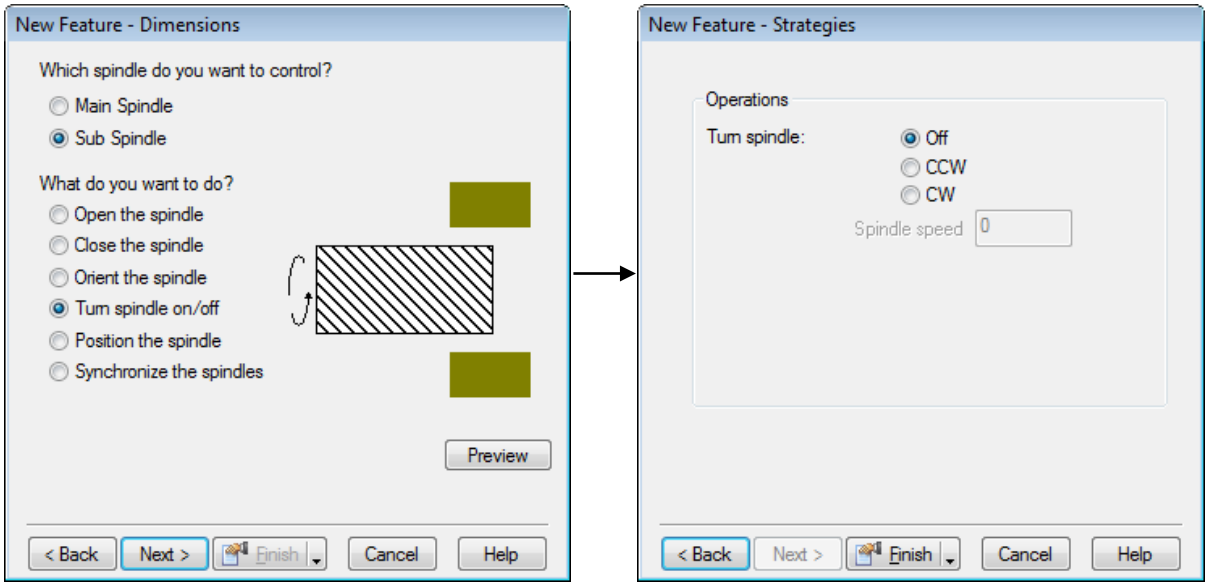
There're 2 methods to create subspindle features in FeatureCAM at the moment. One is to create individual subspindle features (one at a time) until the transfer is complete. This has the benefit that we have ultimate control over the operations performed, however it can be a time consuming task. The other is to use an Addin API feature included with the software. We will first cover the individual method and then do a quick overview of the addin.

Creating individual subspindle features

- To start let's mentally go through what needs to happen in order to successfully transfer the part. Probably in this order:
    - **Stop** both spindles (let's call this user preference)
    - **Orient** the spindles (so that milling features will be positioned correctly between the main and sub)
    - **Open** the **sub spindle**
    - **Position** the **sub spindle** (position to grab the part)
    - **Close** the **sub spindle**
    - **Open** the **main spindle**
    - **Position** the **sub spindle** (send it back home)

Now let's do one operation at a time. It is important to be on the right setup when we make the features as this will affect the result if you are using automatic ordering as your method to organize operations

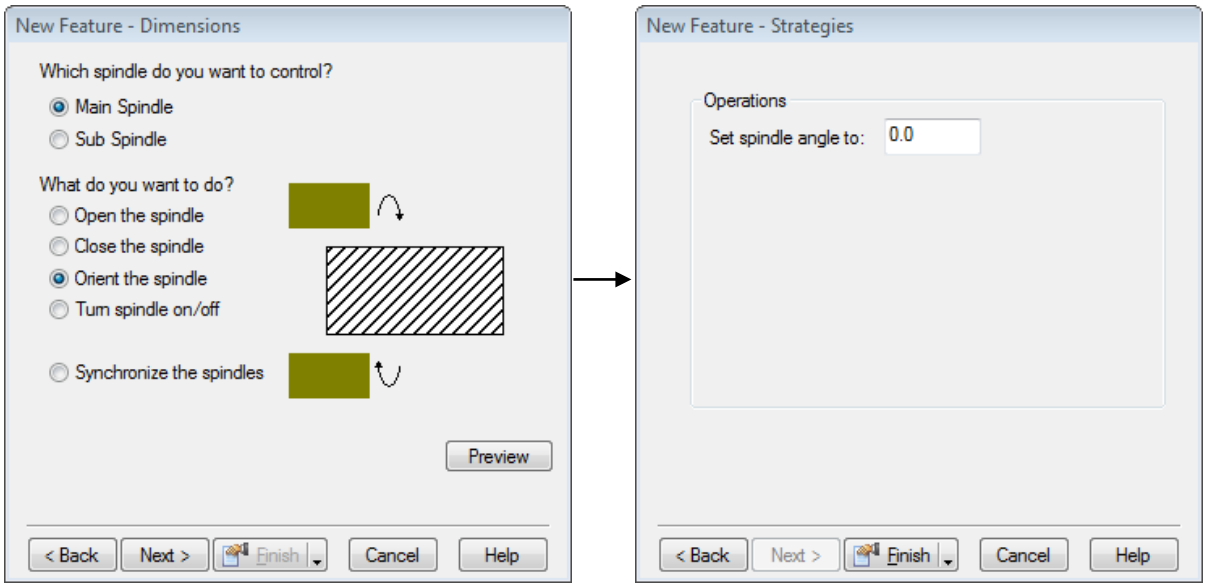
- Start by stopping both spindles. Go to **Features,** then **Turning**, then **Subspindle** then select **Main Spindle** and **Turn spindle On/Off**. In the next page set it to **Off.** Repeat steps for the subspindle, making sure you select **Sub Spindle** when in the appropriate page



It is important to note that a transfer could also happen while the spindles are still rotating. This depends largely in the capabilities of the machine and the preferences of the programmer or operator running the toolpaths. For purposes of this exercise, we will stop both spindles.

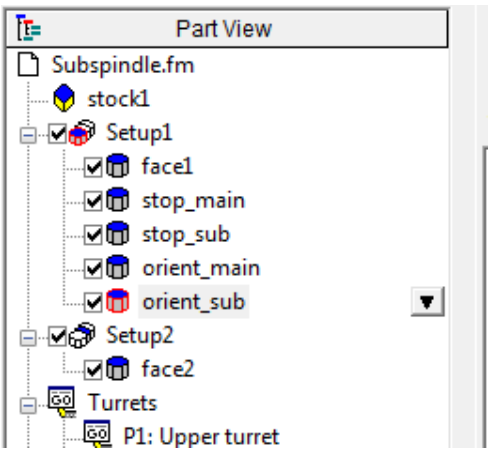
For purposes of keeping your file organized and understand what we're doing let's also rename the subspindle features to something more appropriate. You will notice that when FeatureCAM creates a new subspindle feature we name it ss, ss1, ss2, etc. Rename the feature with a more sensible name (like stop_main or stop_sub).

- Let's now orient both spindles to make sure they are both stopped at the same C angle. Go to **Features,** then **Turning**, then **Subspindle** and select **Main Spindle** > **Orient the spindle**. In the next page leave the default angle as 0. Do the same for the sub spindle

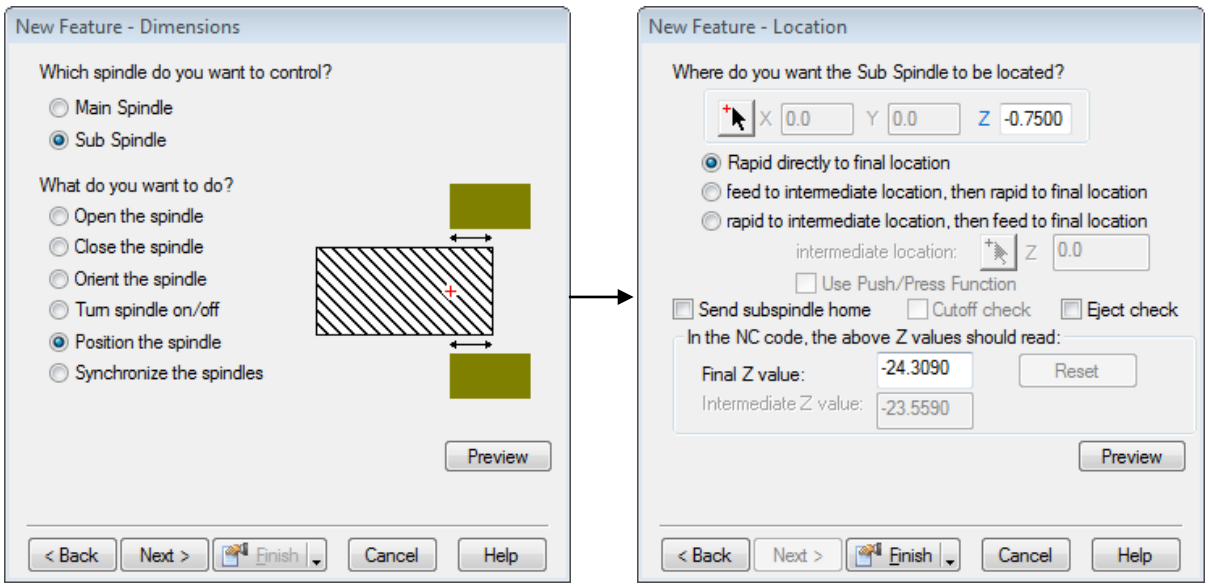


Orienting the spindle is essential if you've created milling features in the first setup that need to line up with the second setup. Not orienting both spindles with the same C angle could cause mismatching features in the machine center, even though the program will look fine in FeatureCAM. By orienting the spindles you make sure that both spindles are at the same angle and the part will come out of the machine looking like it does in FeatureCAM.

By now your Part-view should look like the screengrab below



- Open the subspindle so that we can move it over the stock. Go to **Features,** then **Turning**, then **Subspindle** and select **Sub Spindle** > **Open**. Click **Finish**
- It is now time to **position** the **Subspindle** over the stock. Go to **Features,** then **Turning**, then **Subspindle** and select **Sub Spindle** > **Position the spindle**. In the location tab pick a point where you want to grab the part (I picked -.75) and **Finish**



A great way to understand the location window is to divide it in 2 sections. The upper part will be used for simulation purposes (how much we will be grabbing the workpiece). The lower one will be used for NC code calculation. Notice that when you pick a point in the graphics window, FeatureCAM will automatically calculate and update the final Z value to be output in the NC code. On the other hand, if the operator already knows what the value is, he can manually enter it in the **Final Z value** field.

How do we calculate this number? As a default the final Z value is calculated:

Distance between spindles – (stock length – sim setup information Z) + grab amt = final Z

27.559 – (5.200 - 1.200) + 0.750 = 24.309

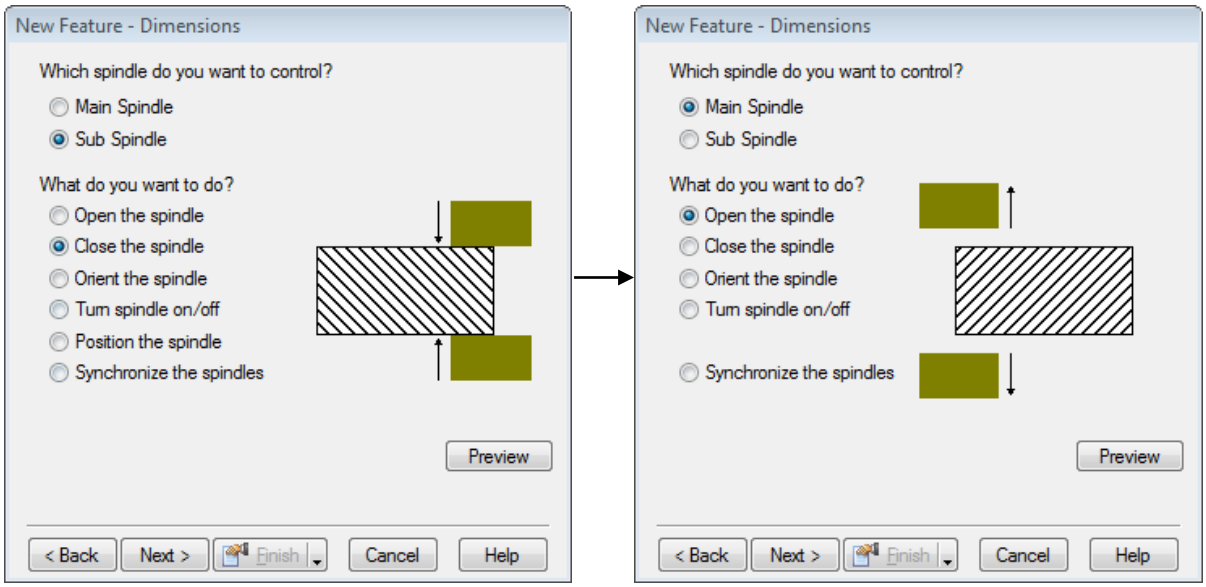
More information on the Sub spindle location tab (skip if already familiar)

- First decide how you want the subspindle to arrive at the final location. Select one of the following:
  - **Rapid directly** - The spindle makes a single rapid move from its current location to the final subspindle location.
  - **Feed to intermediate location, then rapid to final location** - The spindle feeds to the intermediate location and then rapid to the final location. This is useful for gradually removing the support of one of the spindles.
  - **Rapid to intermediate location, then feed to final location** - The spindle will rapid to the intermediate location and then feed the rest of the way. This can be applied to approach the part.
- Next determine the final location specified in the coordinates of the main spindle user coordinate system. This can be done by any of the following methods:

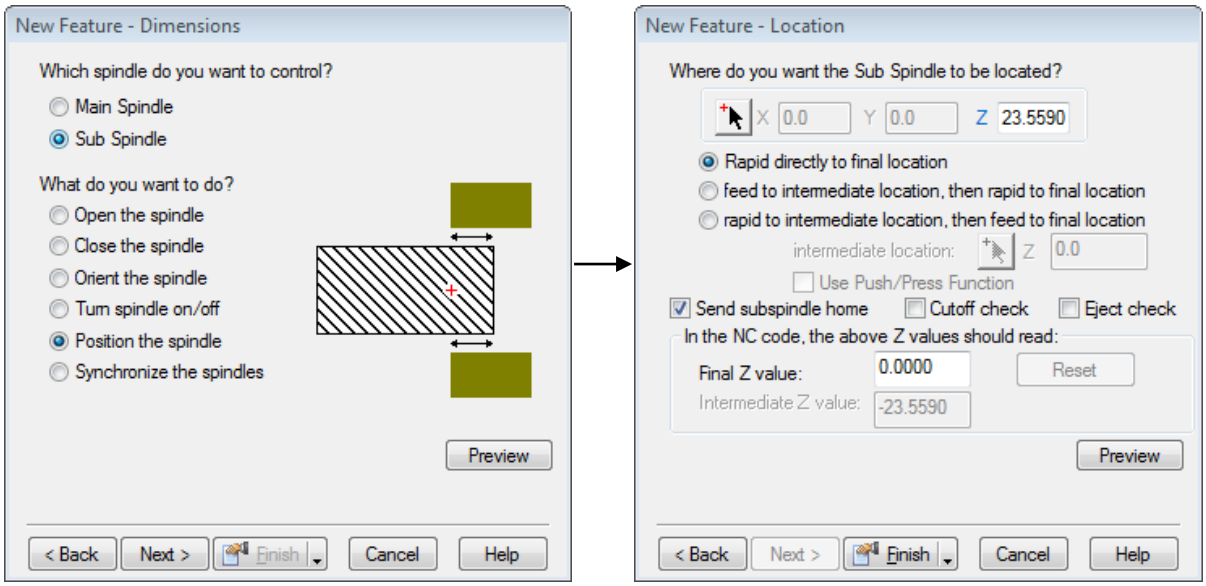
- **Click** the Pick point button and select the point on the screen.
- **Type** in the coordinates directly. Remember that the coordinates are relative to the main spindle user coordinate system.
- If you have selected a strategy that requires an intermediate point, specify the intermediate location by either picking the point or entering the coordinates relative to the main spindle user coordinate system.

- At the bottom of the dialog, FeatureCAM will automatically show you the Z coordinates that it will output in the NC code. If these are not the coordinates you want, you can type the Z values for the Final Z value and Intermediate Z value directly into the bottom text boxes. These values are used in the NC code, but the values shown at the top of the dialog are used for the toolpath simulation

- **Close** the **sub spindle** to hold the part, then **open** the **main spindle** to release it and allow the sub spindle to take it back home.

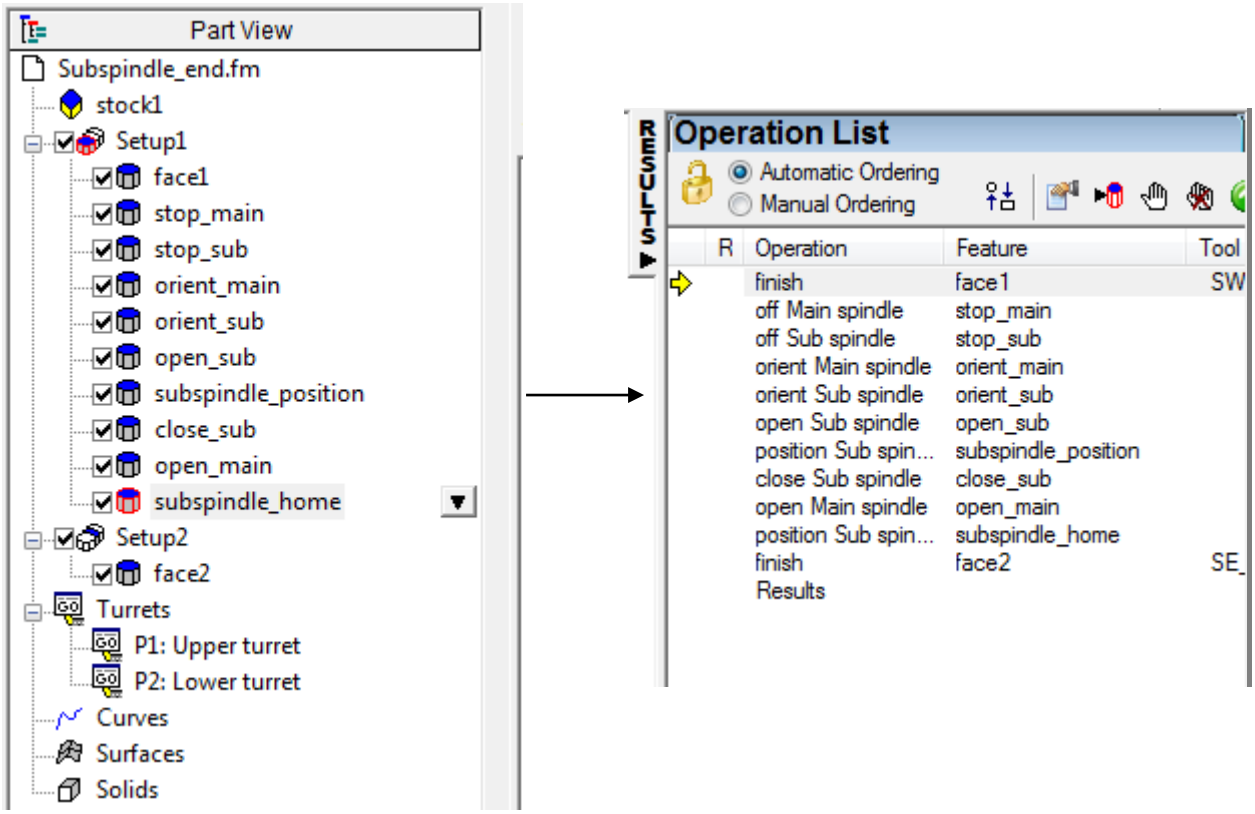


- Now make a **sub spindle > position the spindle** feature, and check the box **Send subspindle home**

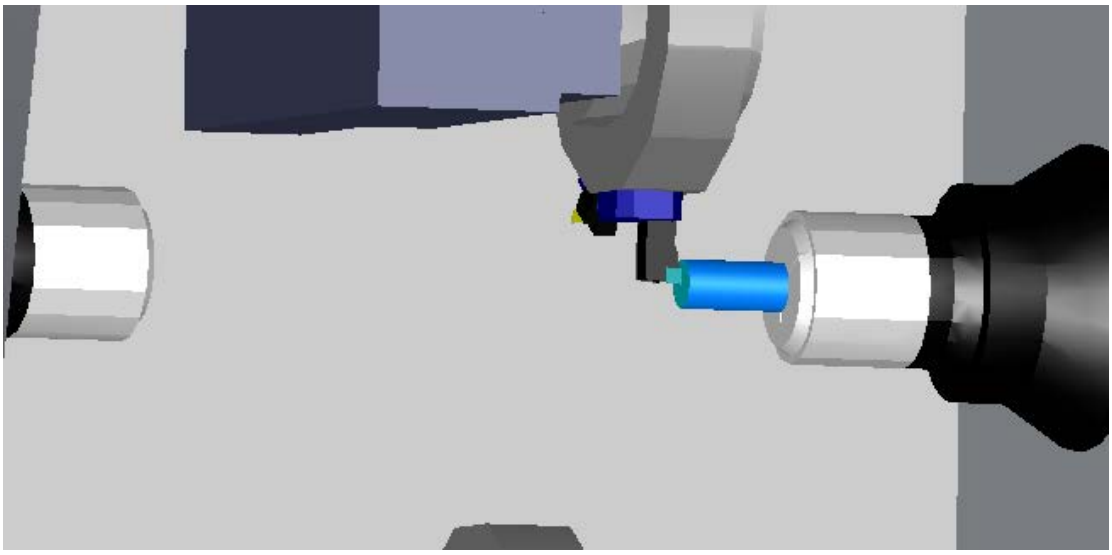


When you check the box **Send the subspindle hom**e feature will automatically set the **Final Z value to 0** and at the same time simulate the spindle going back home

- Your Part-view should look like this. Your **operation list** should have all the operations from **setup1** happening first, followed by **setup2**



- Now go ahead and run **machine simulation** and you should see how the sub spindle grabs the part, takes it home and the upper turret machines the face feature



Subspindle strategy options

This page changes based on the type of subspindle operation you are creating

- If you are **opening the spindle**, you can then specify to extend the part catcher before opening and an optional dwell after spindle is opened
- If you are **closing the spindle**, you can specify an optional dwell after spindle is opened
- If you are **orienting** the spindle, specify the orientation angle on the strategy tab
- If you are **turning the spindle on/off**, you are presented with the options of Off, CW (Clock-wise) and CCW (Counter Clock-wise)
    - If you are turning on the spindle, you must specify the speed
    - If you have multiple spindles, you are also presented with some synchronization options. You can leave the synchronization mode unchanged by selecting No change
    - Select On to begin synchronization or Off to end synchronization
- If you are **synchronizing the spindles**, you are presented with the option of synchronizing for milling or turning
    - For turning, you can choose:
        - Off (no synchronization)
        - Speed Synchronization to rotate both spindles at the same speed
        - Phase synchronization to maintain relative spindle orientation. Phase synchronization is useful when using Hex stock for example.
    - For milling, you can choose:
        - Off (no synchronization)
        - C axis synchronization
        - You can also select to output the turret sync code into the NC code by checking Output turret sync code after synchronizing spindles.

Using the latest version of the macro addin

The subspindle macro addin is an option that was added to the software to simplify subspindle transfer features and make the software easier to use. As you can see from the exercise above, to transfer a part from the main spindle to the sub spindle takes a fair amount of individual operations. It is crucial for each operation to be in the right order, have the right parameters, etc.

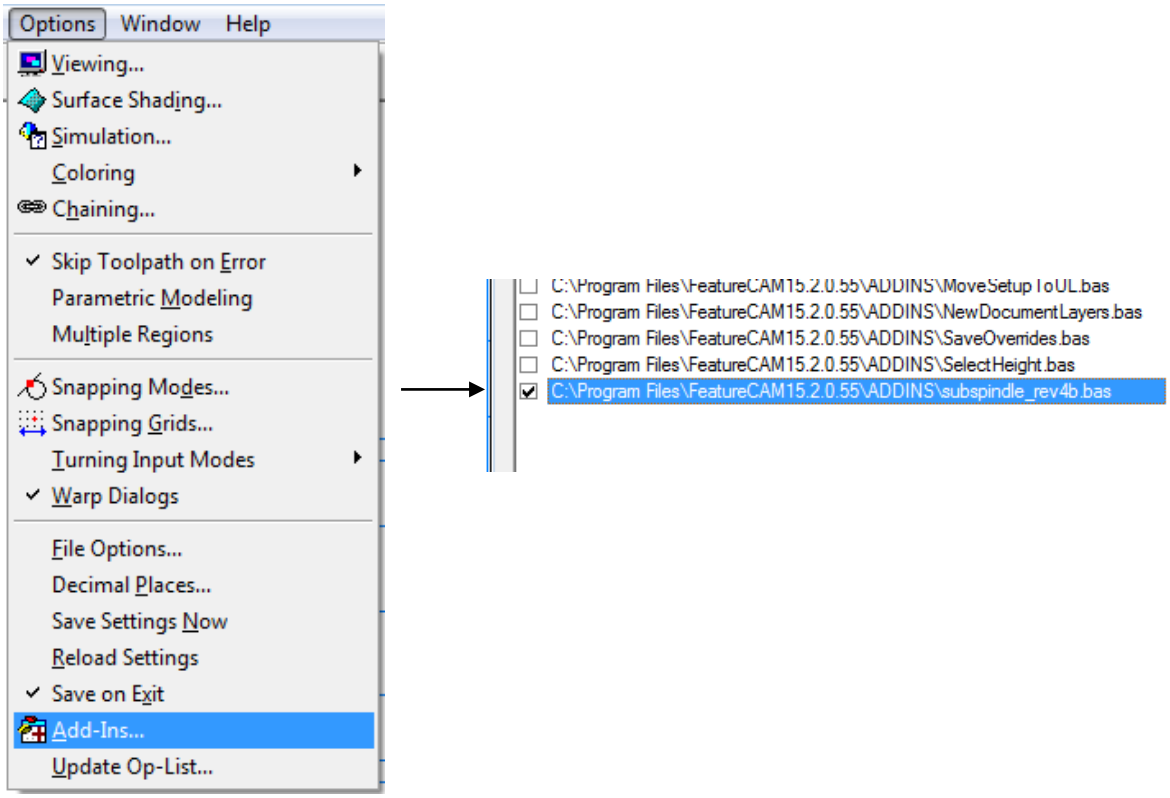
We analyzed the options carefully and created an adding that lets the user enter a few parameters and then FeatureCAM takes care of creating the necessary operations automatically. Currently, (April 2009) you can create up to 11 different transfer operations with this macro, anywhere from a slug transfer to one involving a bar cutoff, from 1 to 2 turrets machines, and lots more.

Understanding this tool is essential if you are required to do transfer in daily basis. Let’s see how easy it is to use it with the same example file used above.

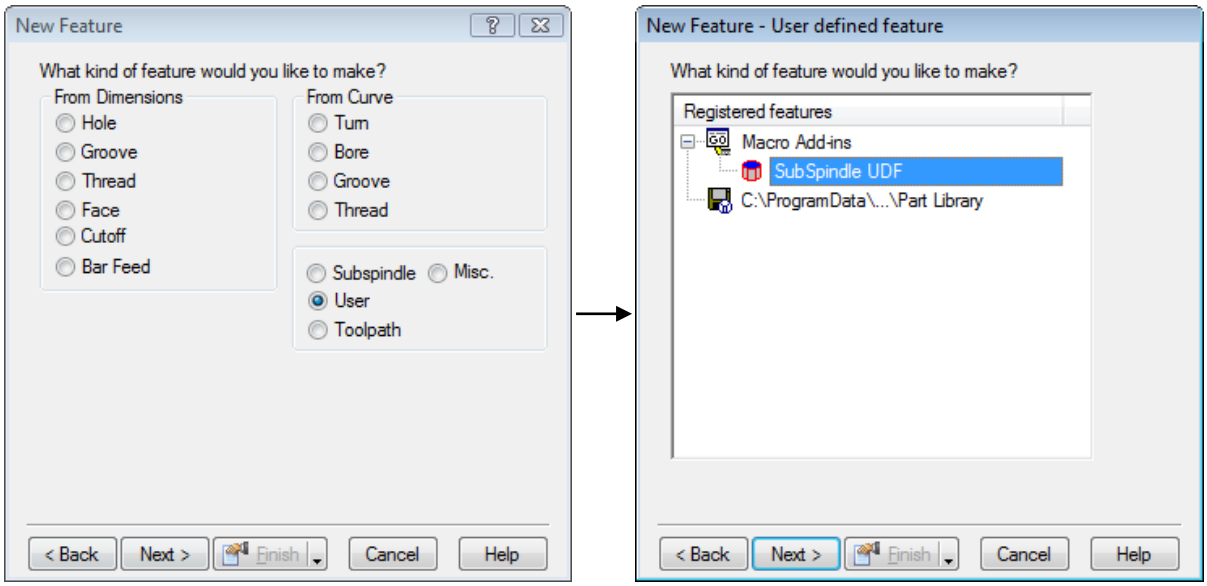
- Click on **File** then **Open**. Browse and re-open **Subspindle.fm**
- You will need to use the **Naka WT300** post for this exercise

- Now let's make sure that we **activate** the subspindle transfer addin. Go to **Options**, then **Add-ins** and make sure there's a checkmark next to **subspindle_rev4b** (the revision could change as we make further changes to the addin), click **OK**

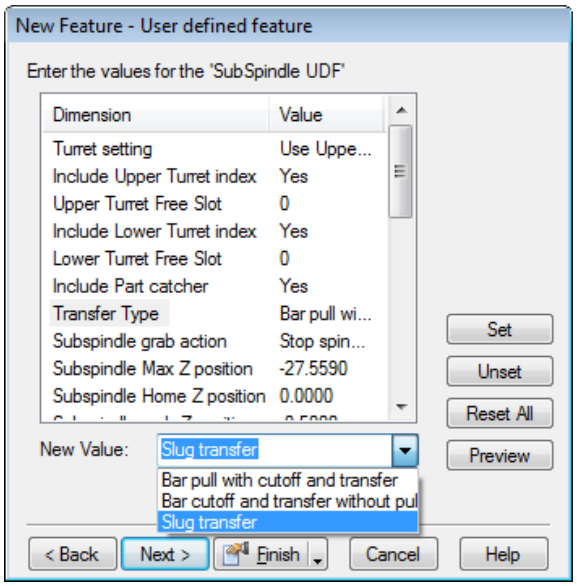
Note that you only need to activate the addin once. After you activate it, it will show in your list of active addins, regardless of whether you restart the software or shut your computer down.

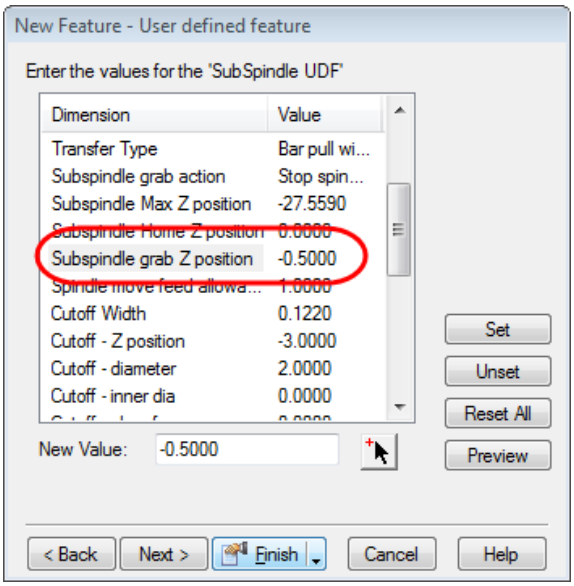


- Now go to **New feature**, **Turning**, **User** select **Subspindle UDF** go **Next**

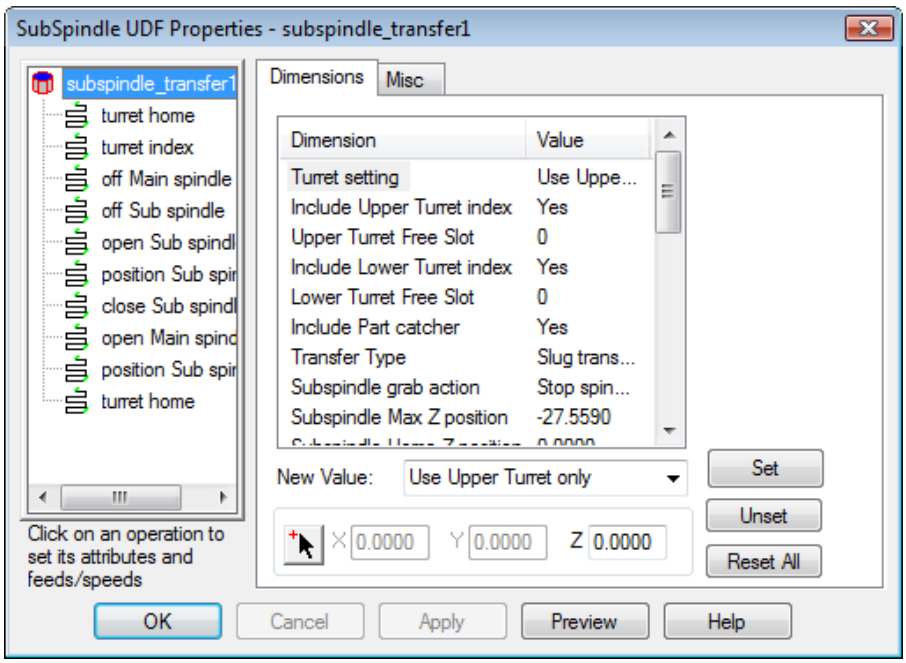


- In the next page we're going to change 2 attributes:
  - **Transfer Type:** set this to **slug transfer**
  - **Subspindle grab Z position:** The default is **0.500**. We could enter a different number of we can pick a point in the graphics window by selecting the attribute and then using the pick arrow at the bottom of the window. Let's leave it at 0.500

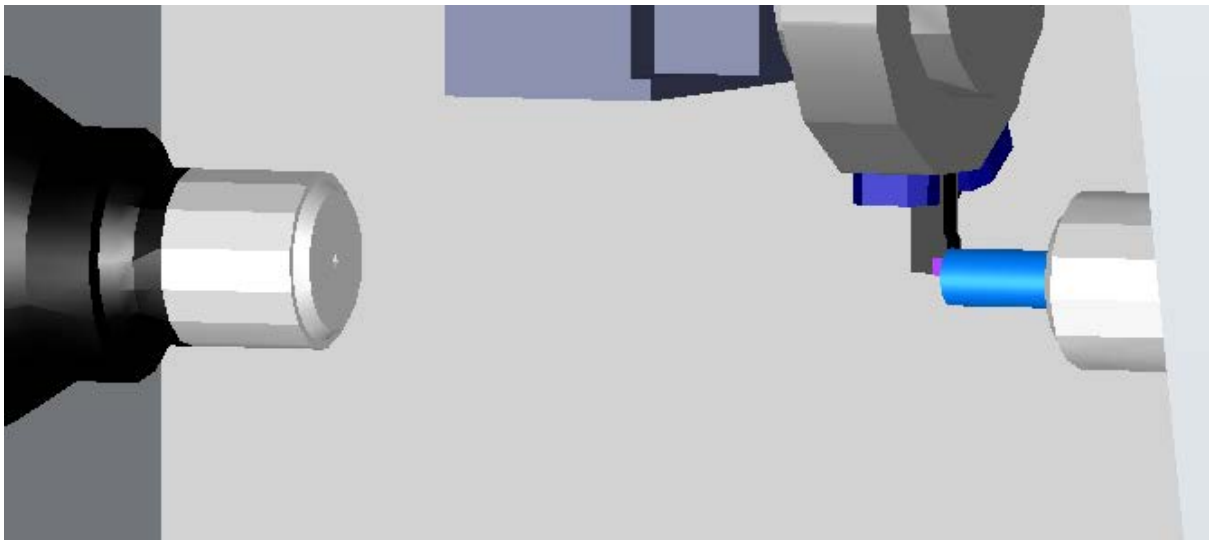




- Click **Finish.** Notice how FeatureCAM created all necessary operations in the correct order. In fact, we created 10 subspindle operations to transfer the part by changing 2 parameters in the addin



- Now run machine simulation to see the part being transferred from the main to the sub spindle



What do the different attributes mean?

The chart below explains the purpose of every value that you can change in the subspindle UDF. It is recommended that you keep this chart as a reference next to your computer

Attribute	Default value	Description
Turret setting	Use Upper Turret only	Options are **Multi turret with sync points**, **Use Upper Turret only**, and **Use Lower Turret only**.
Include Upper Turret index	Yes	Whether or not to index the upper turret.
Upper Turret Free Slot	0	Index of the free slot in the upper turret.
Include Lower Turret index	Yes	Whether or not to index the lower turret.
Lower Turret Free Slot	0	Index of the free slot in the lower turret.
Include part catcher	Yes	Whether or not to extend part catcher.
Transfer Type	Bar pull with cutoff and transfer	Options are **Bar pull with cutoff and transfer**, **Bar cutoff and transfer without pull**, and **Slug transfer**.
Subspindle grab action	Stop spindles	Options are **Stop spindles**, **Orient spindles**, **Keep spindles rotating**, and **Subspindle already grabbing**.
Subspindle Max Z position	From CNC file	Taken from the CNC file. You can override it.
Subspindle Home Z position	From CNC file	Taken from the CNC file. You can override it.
Subspindle grab Z position	-0.5000	Z location where the subspindle grabs the part.

Spindle move feed allowance	1.0000	Feed distance of the subspindle moves.
Cutoff Width	0.1220	Width of the cutoff operation.
Cutoff - Z position	-3.0000	Z position of the cutoff.
Cutoff - diameter	Stock diameter	Diameter of the cutoff operation.
Cutoff - inner dia	0.0000	Inner diameter of the cutoff operation.
Cutoff - chamfer	0.0000	Cutoff chamfer width.
Cutoff - turret	Upper Turret	Options are **Upper Turret** and **Lower Turret** for the cutoff.
Use Push/Press function	Yes	Whether to use Push/Press for the spindle position.
Spindle Dwell (Open)	1.0000	Spindle dwell value for spindle open operation.
Spindle Dwell (Close)	0.0000	Spindle dwell value for spindle close operation.
Spindle Angle (Main)	0.0000	Spindle angle value for main spindle orient operation.
Spindle Angle (Sub)	0.0000	Spindle angle value for subspindle orient operation.
Spindle On/Off Type	CW	Options are **Off**, **CCW**, and **CW**.
Spindle Speed	0.0000	Spindle speed value for CW/CCW subspindle-on operation.
NC Z override	No	If Yes, the NC code Z value is be overriden.
NC Z override value	0.0000	NC code Z override value.
Version	Version number	Version number of the macro.

Transfer combination and order of operations created

You can create one of 11 different types of subspindle transfers using a combination of the **Transfer Type** and **Subspindle grab action** attribute values. These are the operations for each transfer type

Transfer Type = Bar pull with cutoff and transfer ***Subspindle grab action = Stop spindles***	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	

	position Sub spindle (to cutoff location)
close Main spindle	
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar pull with cutoff and transfer *Subspindle grab action = Orient spindles*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
orient Main spindle	**orient Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	
	position Sub spindle (to cutoff location)
close Main spindle	
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar pull with cutoff and transfer *Subspindle grab action = Keep spindles rotating*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	

start Main spindle	starts Sub spindle automatically
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	
	position Sub spindle (to cutoff location)
close Main spindle	
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar pull with cutoff and transfer *Subspindle grab action = Subspindle already grabbing*	
Main spindle	Sub spindle
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar cutoff and transfer without pull *Subspindle grab action = Stop spindles*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar cutoff and transfer without pull *Subspindle grab action = Orient spindles*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**

orient Main spindle	**orient Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar cutoff and transfer without pull *Subspindle grab action = Keep spindles rotating*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
	position Sub spindle (grab part)
	close Sub spindle
cutoff (part off)	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Slug transfer *Subspindle grab action = Stop spindles*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	
	position Sub spindle (go home)

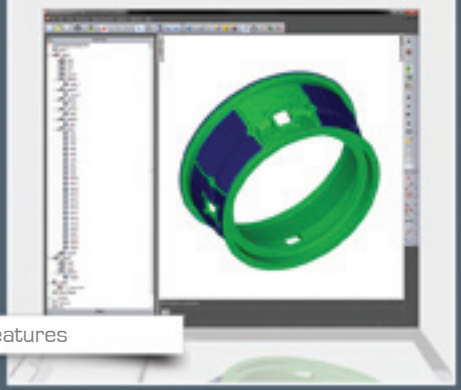
Transfer Type = Slug transfer *Subspindle grab action = Orient spindles*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
orient Main spindle	**orient Sub spindle**
	open Sub spindle
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	
	position Sub spindle (go home)

Transfer Type = Slug transfer *Subspindle grab action = Keep spindles rotating*	
Main spindle	Sub spindle
upper **turret home**	lower **turret home**
upper **turret index** (optional)	lower **turret index** (optional)
off Main spindle	**off Sub spindle**
	open Sub spindle
on-turn speed spindle sync - Start Spindle Synchronization Mode (Main and Sub spindles)	
start Main spindle	starts Sub spindle automatically
	position Sub spindle (grab part)
	close Sub spindle
open Main spindle	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

Transfer Type = Bar pull with cutoff and transfer *Subspindle grab action = Subspindle already grabbing*	
Main spindle	Sub spindle
open Main spindle	
	position Sub spindle (go home)
off-turn spindle sync - Stop Synchronization Mode (Main and Sub spindles)	
off Main spindle	**off Sub spindle**

FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV

www.delcam.tv

FeatureCAM Learning Zone

www.delcam.tv/lz

FeatureCAM Website

www.featurecam.com

THE QUEEN'S AWARDS FOR ENTERPRISE: INNOVATION 2010

THE QUEEN'S AWARDS FOR ENTERPRISE: INTERNATIONAL TRADE 2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

To contact your local reseller, visit **www.delcam.com/resellers**