Delcam Advanced Manufacturing Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureTURNMILL

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:

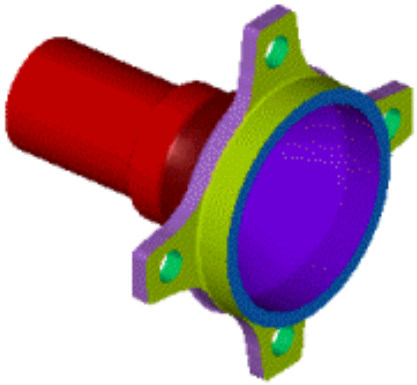
- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.

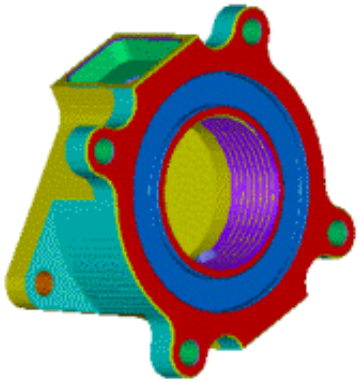
You can customize this built-in 'intelligence' to your own style of cutting.

Intro to Turnmill Features

Introduction

FeatureTURNMILL allows the combination of turning and milling features on lathes with powered rotary tools. FeatureTURNMILL supports the normal Z and X axes of turning combined with the C and optional Y-axis. All turn/mill machining centers will support C axis (which the ability to rotate about Z). Y-axis is an option which adds to the cost of the machine. You should not assume that every turn/mill machine available will possess a Y-axis.





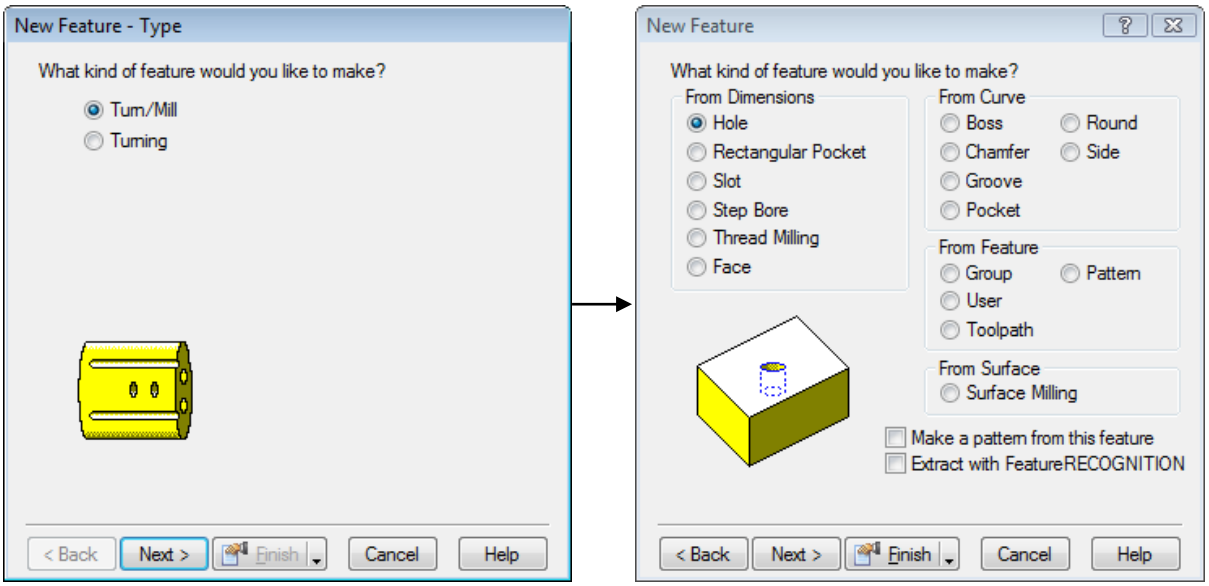
The yoke shown in the left-hand figure could be manufactured with a C axis lathe in two setups or using a C axis lathe with a subspindle. The right hand figure is a piece that requires a Y-axis due to the **flat pockets** on the top and bottom of the piece. (If these pockets were wrapped, they would have a curved bottom and then they could be manufactured without a Y-axis.)

Creating turnmill features in FeatureCAM

Turn/mill features are classified as either **Turn/Mill** or pure **Turning** features. Turn/mill features assume powered tools and turning features assume that the tool does not spin. For example, you can make a hole along the Z-axis with both feature types, but the milled one uses rotary tools and the turned hole uses a drill that does not spin, therefore the spindle will spin instead.

- **Turning** features are identical to those on a 2-axis lathe
- **Milling** features are created the same as for a 3-axis mill, except that you are given new choices for positioning and orienting the features either on the OD or on the face of the part

In the screengrabs below you can see how after selecting Turn/Mill the software will allow the user to create milling features exactly like if he would be using our basic 2.5D module.



All forms of simulation (Centerline, 2D, 3D and machine simulation) are supported. In 3D and machine simulation rotation of the part is accurately simulated, allowing the user to run toolpaths and experiment with different options using our simulation package instead of having to test different scenarios in the machine.

Tool selection for turnmill features

Turn/mill features use the same tools as the normal milling features, but they are renamed with **–rotaryX** or **–rotaryZ** appended to the name to indicate that it is a powered rotary tool. For example, if a tool called **center_4** is selected for a turnmill center drill operation, the tool is copied and the copy is called **center4-rotaryX**.

Rotary tools cannot be explicitly created, but if you manually select a tool for a turnmill operation, it is copied and the copy is designated as a rotary tool. Rotary tools cannot be used for turning operations.

Features appropriate for turnmill

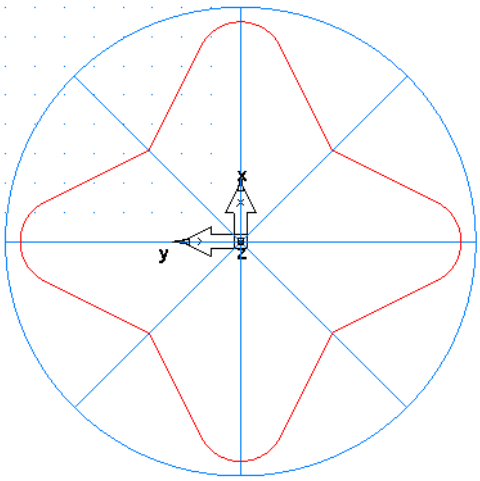
Milling features can be performed in turnmill with the following considerations:

- **Milled features on the Z face**: FeatureTURNMILL can make any feature on the face of a part by using only XZC moves, for machines that do not have a Y-axis. If you want to use Y-axis on the face of a feature, you must check the **Cut using Y-axis coordinates** check-box on the Dimensions page of the feature's Properties dialog
- **Drilled features** on the **Z face** or **OD** can be created without any restrictions
- **Unwrapped milled features on the OD**: These features are output in **X**, **Y** and **Z** moves.

- If your machine does not have a Y axis, the only features you can cut on the OD (without wrapping) is a simple slot whose length is aligned with the Z axis
- If your machine has Y-axis capabilities, you can cut the full set of milling features on the OD

- **Wrapped features** are supported (using a live tool), with known limitations - the same limitations that FeatureCAM has with wrapped 4-axis fetaures. To invoke wrapping, you must check the Wrap feature around Z-axis check-box on the Dimensions page of the feature's Properties dialog.

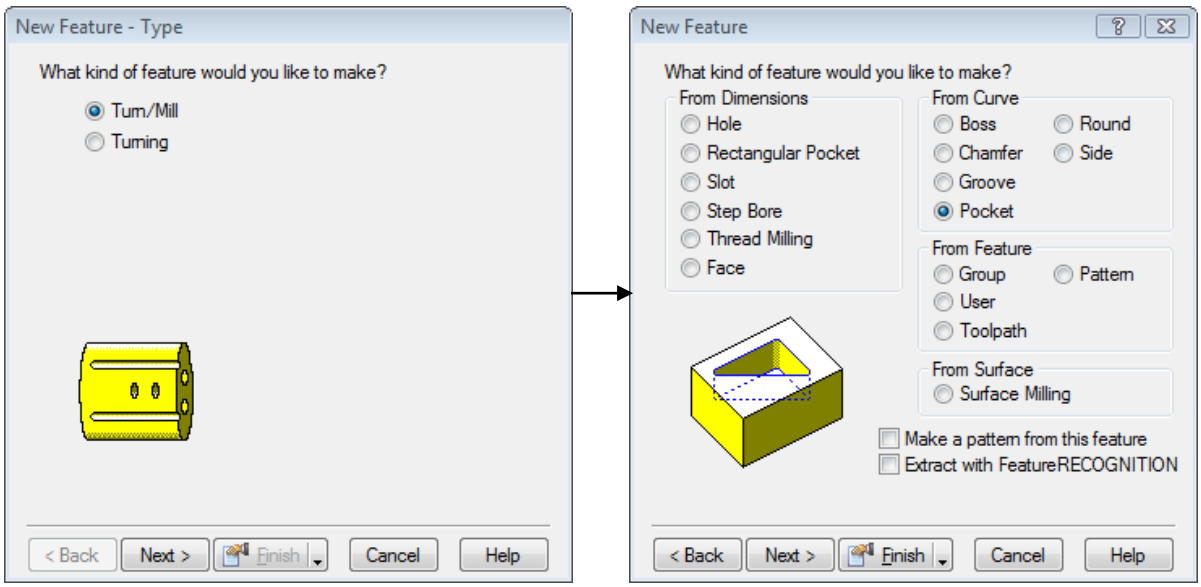
Positioning features on the face of a part



The picture to the right shows the stock like if we would be looking at it from a right-view. In the file there's a star shape curve (red) that we will machine using a pocket routine.

In our first turnmill exercise we will go through the necessary steps to create this feature and in the meantime cover basic principles that will help you understand why you would want to machine this with C or Y-axis.

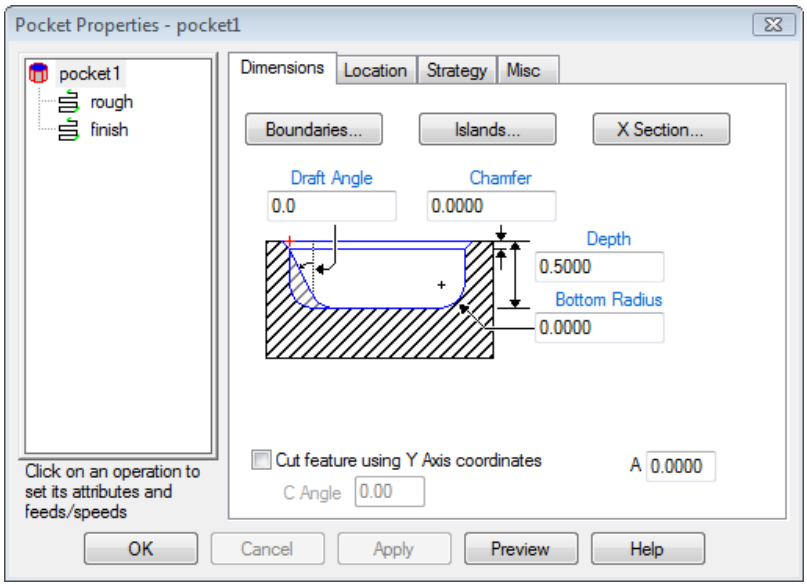
- Open the file **Starshape.fm**
- You will need the **Nakamura WT300** post for this project
- **Right-click** on the screen and view the project in **Isometric view.** Notice that we have a star shape in the front of the part. We will make a pocket out of this shape
- Go to **New Feature**, **Turn/Mill**, **Pocket From Curve**



- Click on **Next**, **pick** the curve and continue through the wizard accepting all the defaults.

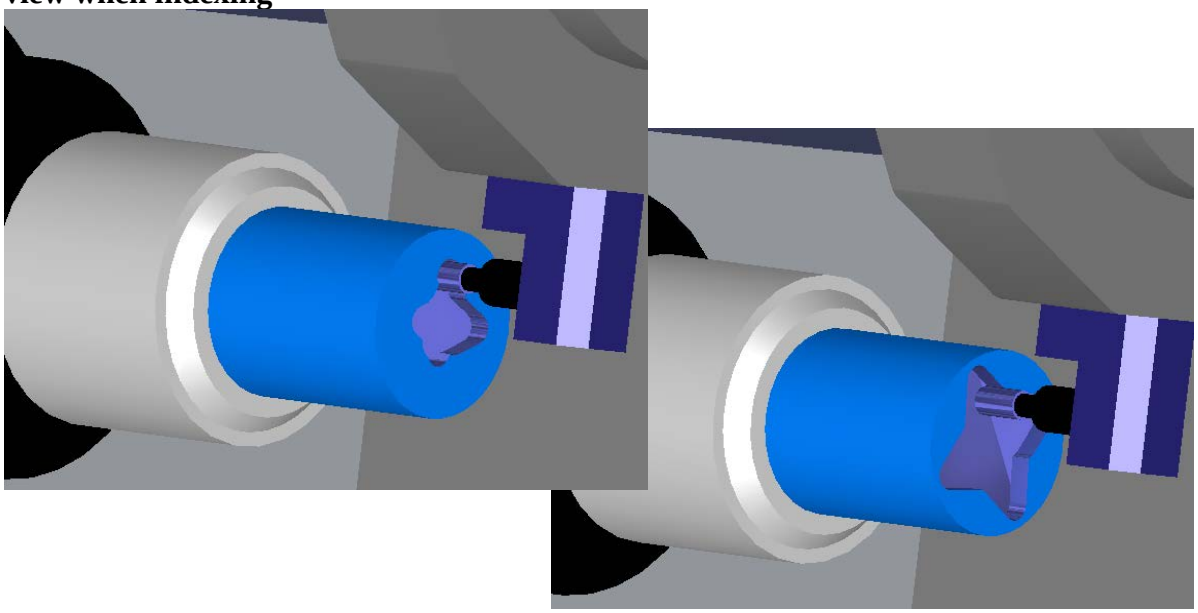
Notice that FeatureCAM will pick the machining angle based on the plane of the curve being selected. Every 2.5D feature (except simple groove) requires a planar curve. This plane determines the angle that the software will use to machine the feature.

- Once you're done with the wizard click on **Finish** and compare the values in the different field boxes are the same as the ones displayed in the screengrab below. Make sure that **Cut feature using Y axis coordinates** is **not** checked.

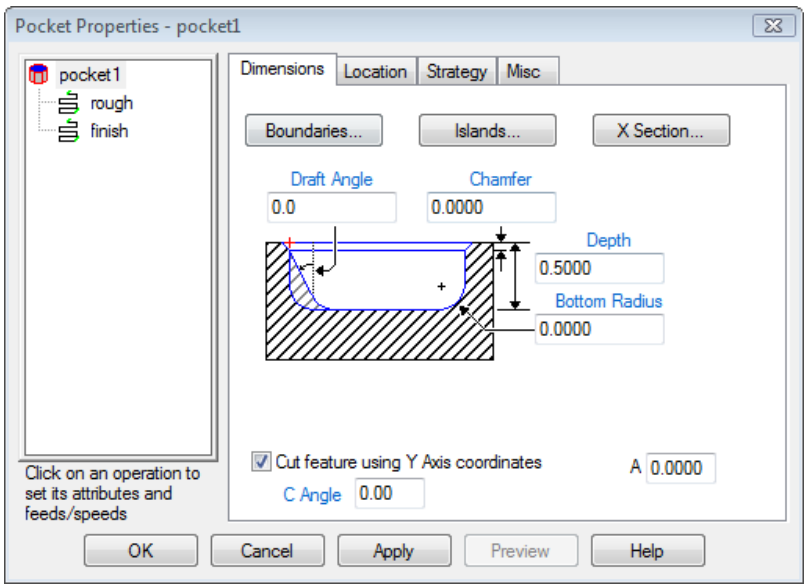


- Run **machine simulation**. Note how in order to reach all the areas inside of the pocket the tool moves **up and down** along the **X** axis and the spindle rotates in **C** to position the stock. At the moment this part is being machined with **X**,**Z** and **C**

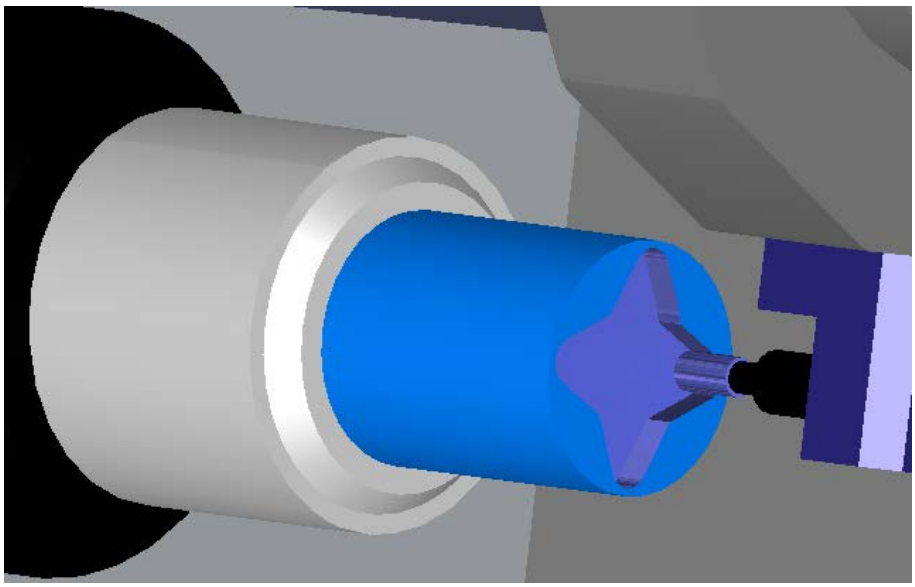
If you decide to use **3D** sim instead and you can't see the spindle rotating when the simulation is running, go to **Options**, **Simulation**, **2D/3D Shaded** and check the box **Rotate view when indexing**



- Now let's open the properties of the feature and **check** the box **Cut feature using Y axis coordinates** so that FeatureCAM outputs and simulates Y-axis moves to machine this part



- Run **machine simulation**. Note how the spindle is now **locked** and the tool moves in **X**, **Y** and **Z** to reach all areas inside of this pocket



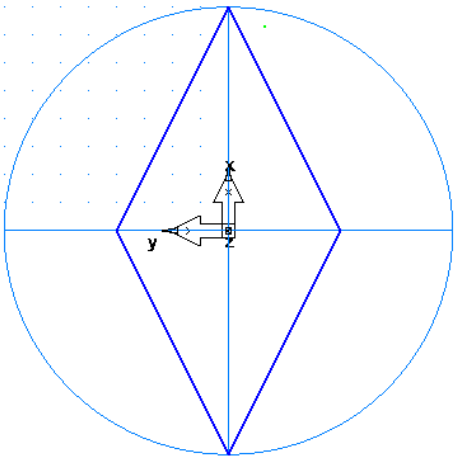
Pros and cons of Y vs. C-axis

Note: although we support both methods of machining, there are cases where the person using the software would favor one over the other. The chart below illustrates the benefits and detriments of both approaches.

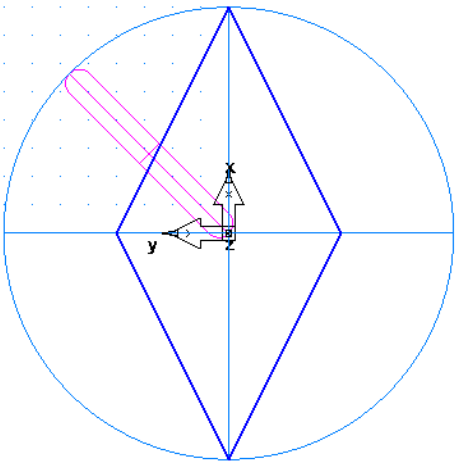
You (the user) can agree or disagree with the comments below. They are general to most machines and could not apply to your particular machine or shop background. Discussion with peer students is encouraged.

C-Axis		Y-Axis	
Pros	**Cons**	**Pros**	**Cons**
Machines are cheaper w/o Y-axis option	Surface quality suffers due to lack of Y and having to rely on C axis	Achieves best surface finish based on XY moves. No C-axis involved during machining	Expensive machines. Y-axis is usually found in upper turrets only (if multiple turret machine)
Less area limitations No constrains on Z-face features	Slower machining time. Most noticeable on older machines	Faster machining time	Limitation in the amount that the Y-axis can travel
Faster programming in some cases where repositioning would be needed with Y-axis due to limited machining area	Machining accuracy. If tool is not setup exactly on centerline of X-axis it could cause issues in features with parallel walls	Due to no C-axis involvement, features are the most accurate. If a tool is not setup on center it can be corrected using offset values in the control	Y-axis travel limits could require for a feature to be repositioned, causing possible witness marks and added programming time

Handling Y-axis limits



What is this limitation we talk about with Y-axis machines? These machines **do not** have unlimited travel for the Y-axis. The amount of travel is usually **40-50mm** in each **Y** direction, measured at **X0**. The furthest the tool travels in X, the less the Y-axis travel. The best way to understand Y-axis limits is to picture a diamond in the face of the part. You can obtain limits of your particular machine from your machine manual. Look at the picture to the right.

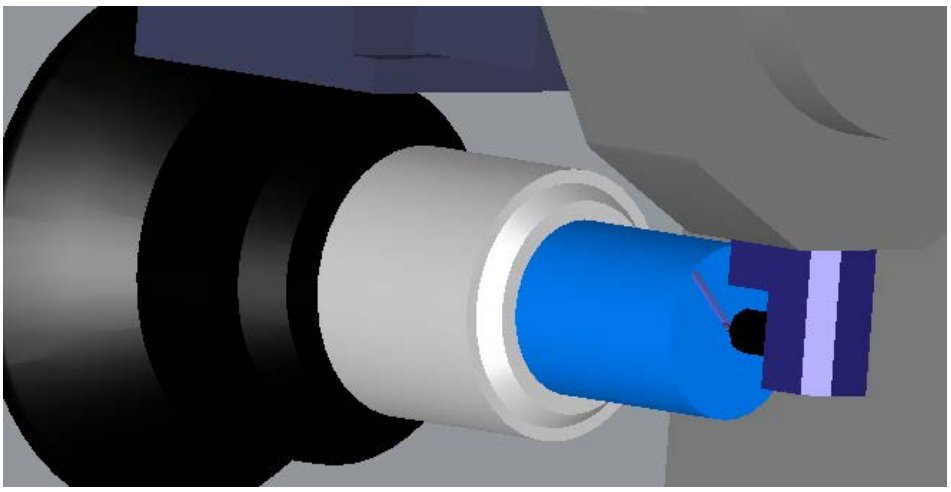


So how do we cut a feature exceeding the machine Y-axis? In the next picture you can see a 45 degree slot feature. In this case part of the slot lies outside of the Y-axis travel area. The following exercise will show us how to reposition a feature so that it can be machined within the Y-axis limit area.

- Open file **Y-axis limits.fm**
- For this project you will need to use the **Nakamura WT 300** post processor
- Run **machine simulation** and notice how the software is currently machining the slot with **C-axis**. This is the perfect scenario for when you **don't** want to use C-axis, as if the tool is not perfectly set on center of the X-axis, you could end up with a slight variation in the width of the feature

If we don't have a Y-axis machine this is the best we can do, but if we do (and our Nakamura machine does) then we want to use Y.

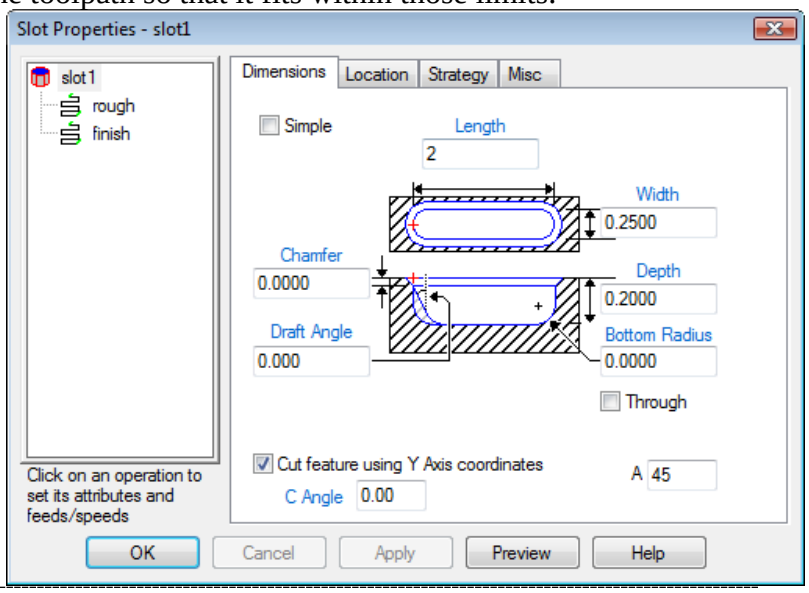
- Open the feature properties and **check** the box **Cut feature using Y axis coordinates** so that FeatureCAM outputs and simulates Y-axis moves to machine this part
- Run **machine simulation.** Notice how the slot is not being machined using **X**, **Y** and **Z** moves. The problem is that as we discussed, part of the slot area lies **outside** of the Y-axis limit boundary. How do we correct this?



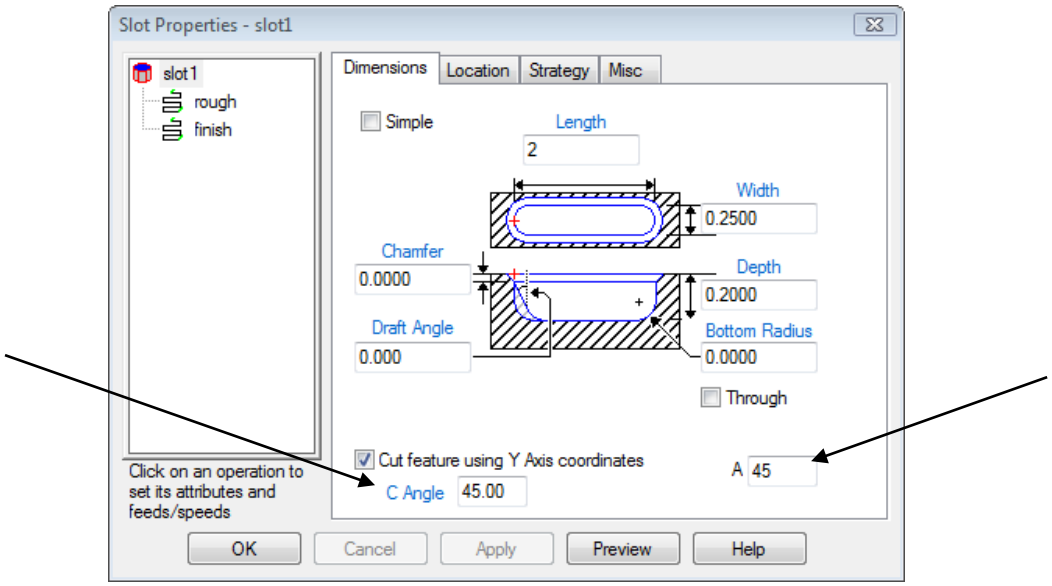
At the moment, FeatureCAM is not able to automatically detect when features are located outside of the Y-axis travel area. It is the user's responsibility to understand how big this area is in their machine and to correct the toolpath so that it fits within those limits.

- **Open** the feature and notice how the **C Angle** is set to **0**

This means that FeatureCAM will use Y-axis to machine the part, but **won't** shift the C-axis to reposition the part in a more appropriate orientation to reduce the amount of Y-axis travel. The feature will be machined in the same location where it is drawn.

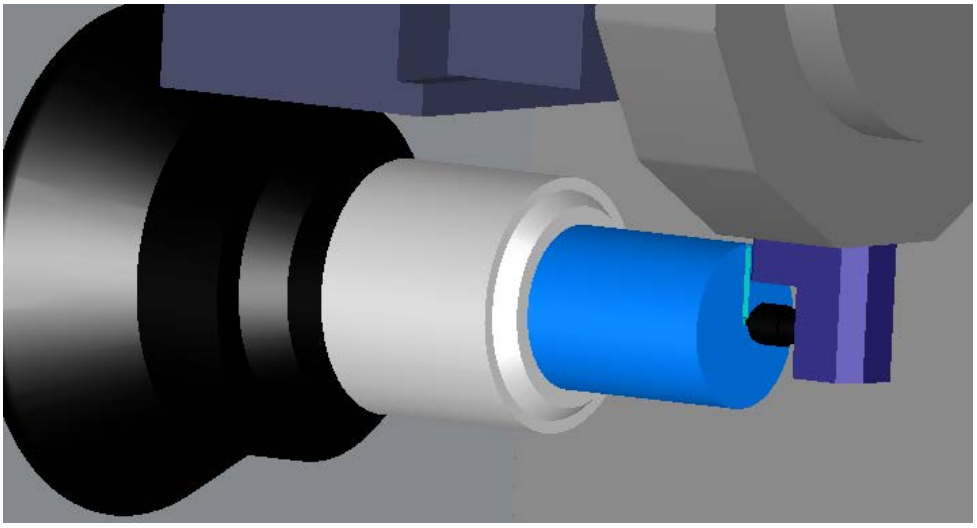


- Enter **45 degrees**, in the **C Angle** field, to make it match the angle of the slot



By entering 45 degrees in the C Angle field the **feature will be rotated 45 degrees while being machined**, which means that it will be aligned with the X axis, causing the least amount of Y-axis travel. You can also **pick** an angle if you click on the **C Angle** word and then **visually select a point** in the graphics window.

- Run **machine simulation.** Notice how the slot is not being machined using **X**, **Y** and **Z** moves but the feature is **repositioned** 45 degrees so that it fits within **Y-axis** limits



This feature (C Angle) is not only useful for limiting Y-axis moves, it can also be used if the toolpath generated exceeds the X-axis travel. For example, if the groove would be positioned with an angle of 180 degrees.

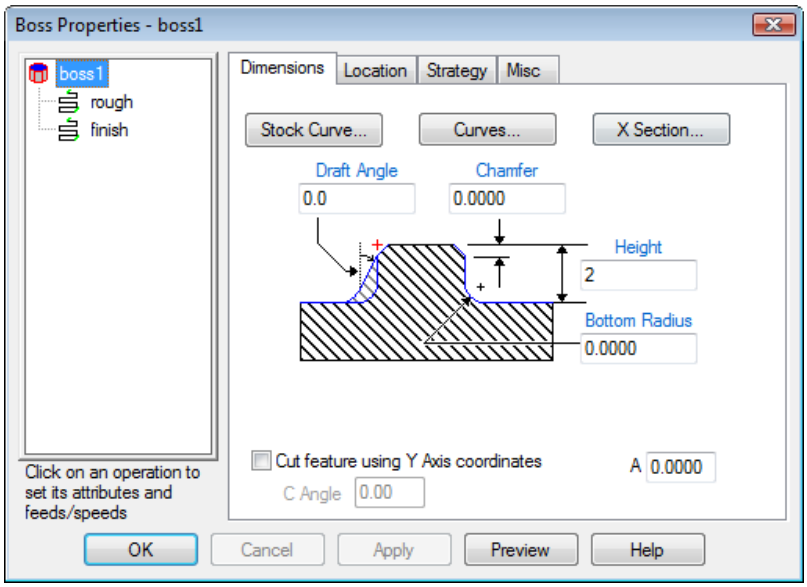
If you are ahead of the class and extra time, open file **X-axis limits.fm** and machine it using Y-axis in the X positive direction.

Positioning features about the OD of a part

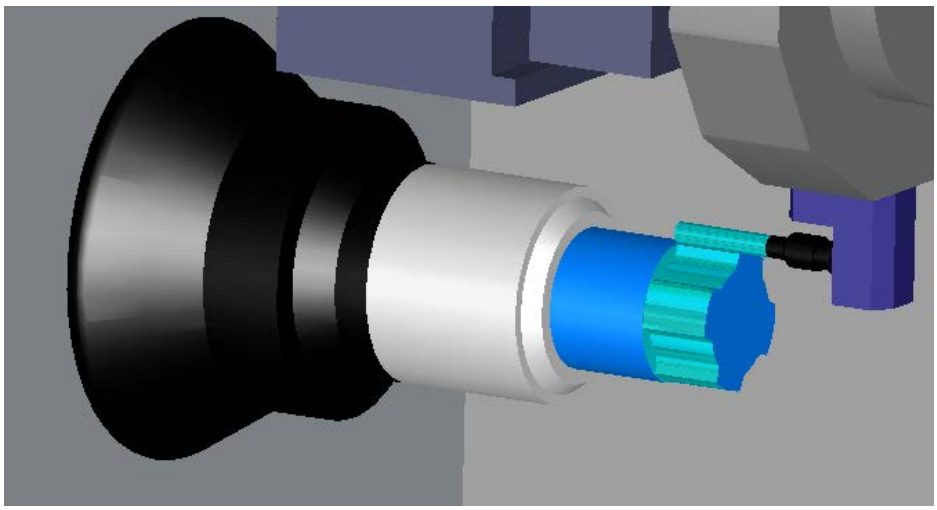
FeatureCAM also allows you to position features about the OD of a part. Just like features located on the face, they pay attention to the plane of the curve to determine the machining angle.

The following exercise will go through the necessary steps to create a feature about the OD of a part. We will first make some features from curves and one from dimensions.

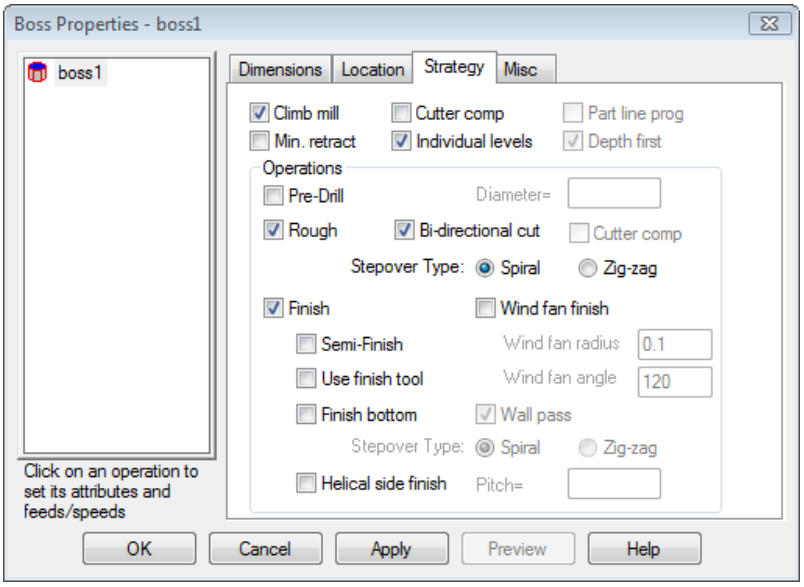
- Open file **Features about the OD.fm**
- You will need the **Nakamura WT300** post for this project
- Using the **front curve**, make a **boss** feature. Change the **height** to be **2.000**. Your feature properties should look like the screengrab below



- Run **machine simulation** and notice how at the moment FeatureCAM picks a long tool to rough and finish the part. This is not the best approach to machine this feature, as a longer tool will cause chatter, take longer time to machine and have a shorter life

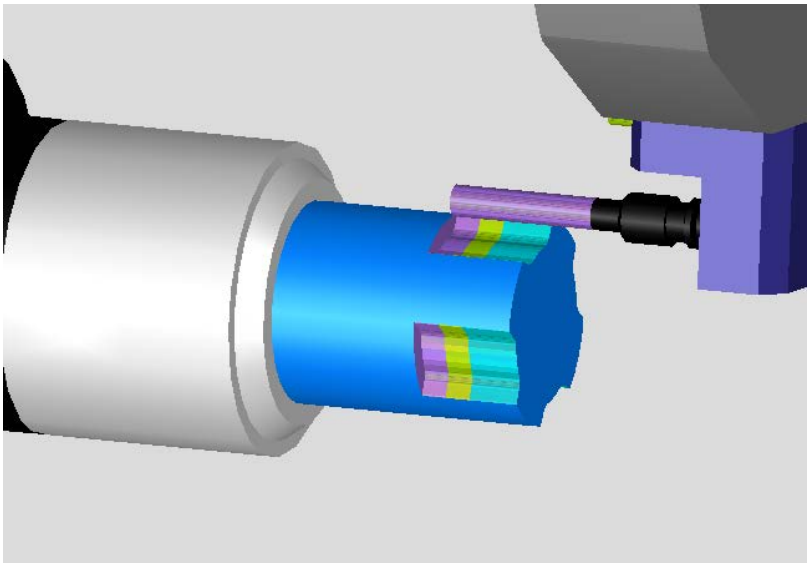


- Let's borrow a feature from our 2.5D milling module!!! Go into the **properties** of the feature, click on **Strategy** and **check** the box **Individual levels**

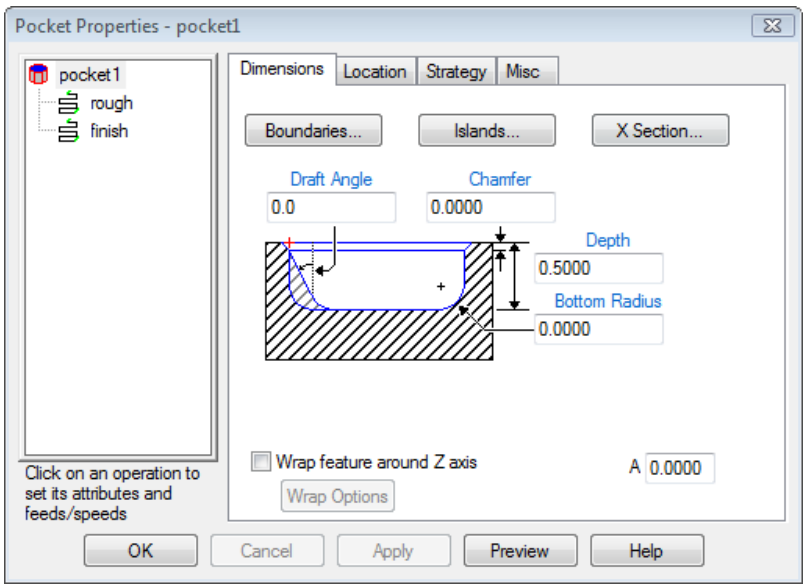


This option will treat each z-increment as a separate operation, allowing the user to select different tooling for each pass. In this case, FeatureCAM will automatically select different tooling for us.

- Run **machine simulation** once again. If you are using tool colors during simulation you should be able to distinguish fairly easily that we're using 3 different tools now to machine the same feature. Also notice how FeatureCAM picked the tools automatically depending on the z depth

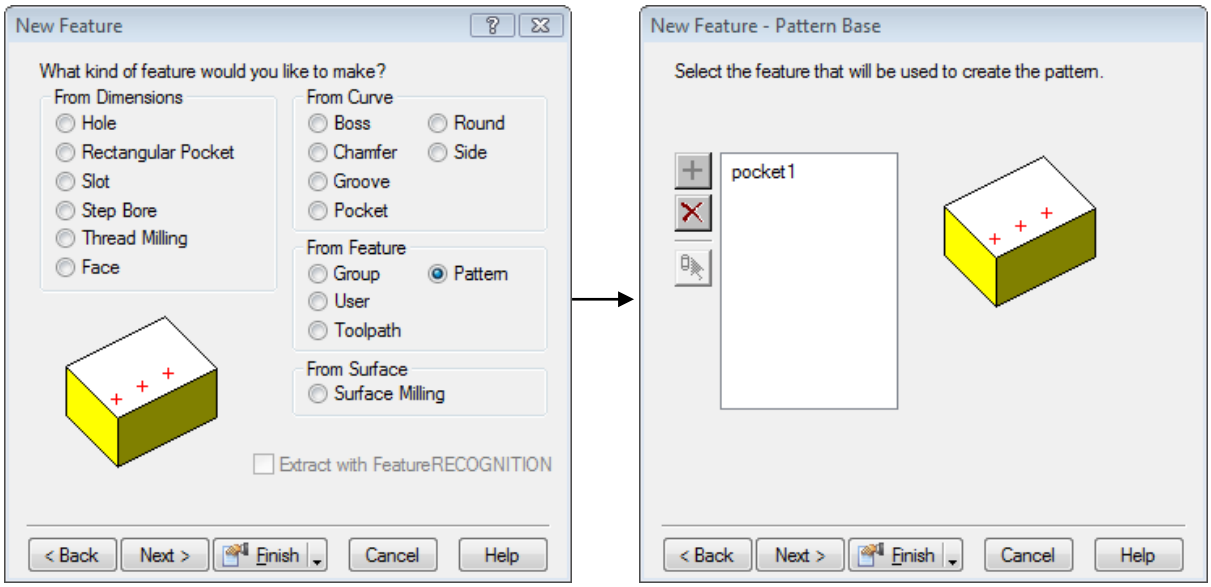


- Make a **pocket** feature out of the **circle** on the OD of the part at 90 degrees. Accept the default **depth of 0.500.** Note that FeatureCAM automatically determines the angle of the feature based on the geometry selected



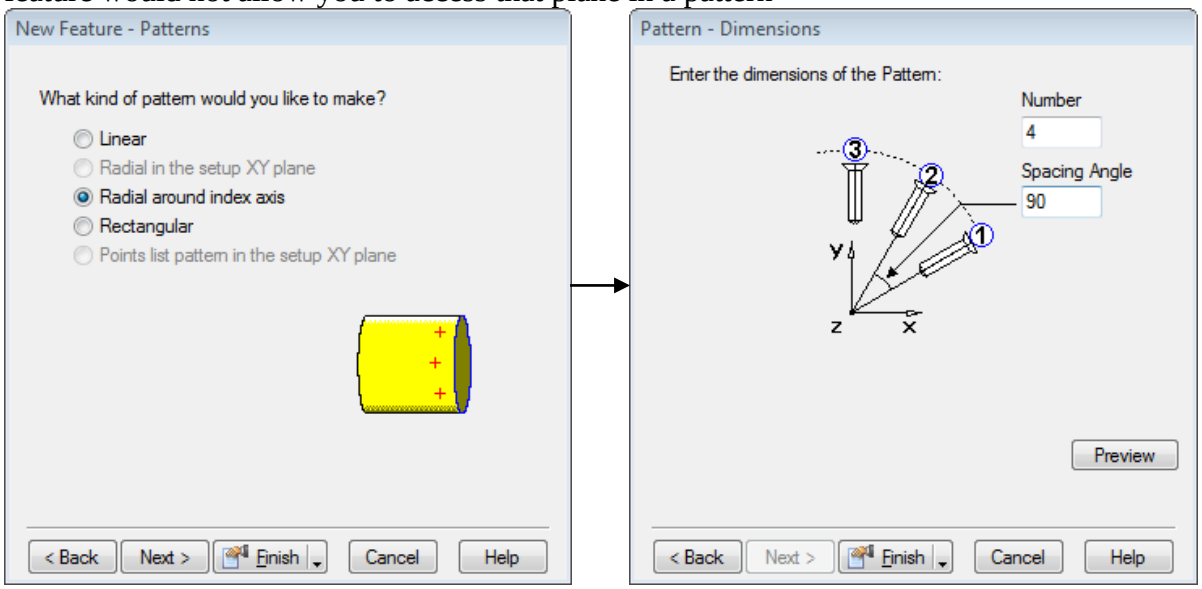
FeatureCAM will use the **plane** of the circle to determine the angle of the feature. Every plane has 2 possible Z directions. As a default, this direction will be calculated pointing **towards** the axis of rotation (Z in a turnmill document, stock axis in a 4-axis milling document). If you need to switch the direction you can do that easily in the **Location** tab of the feature using **Cut feature from other side.**

- Make a **pattern** out of the last **pocket** feature created
- Go to **New Feature**, **Turn/Mill**, **Pattern from Feature.** Make sure you select the appropriate feature you want to make a pattern out of (the circular pocket)

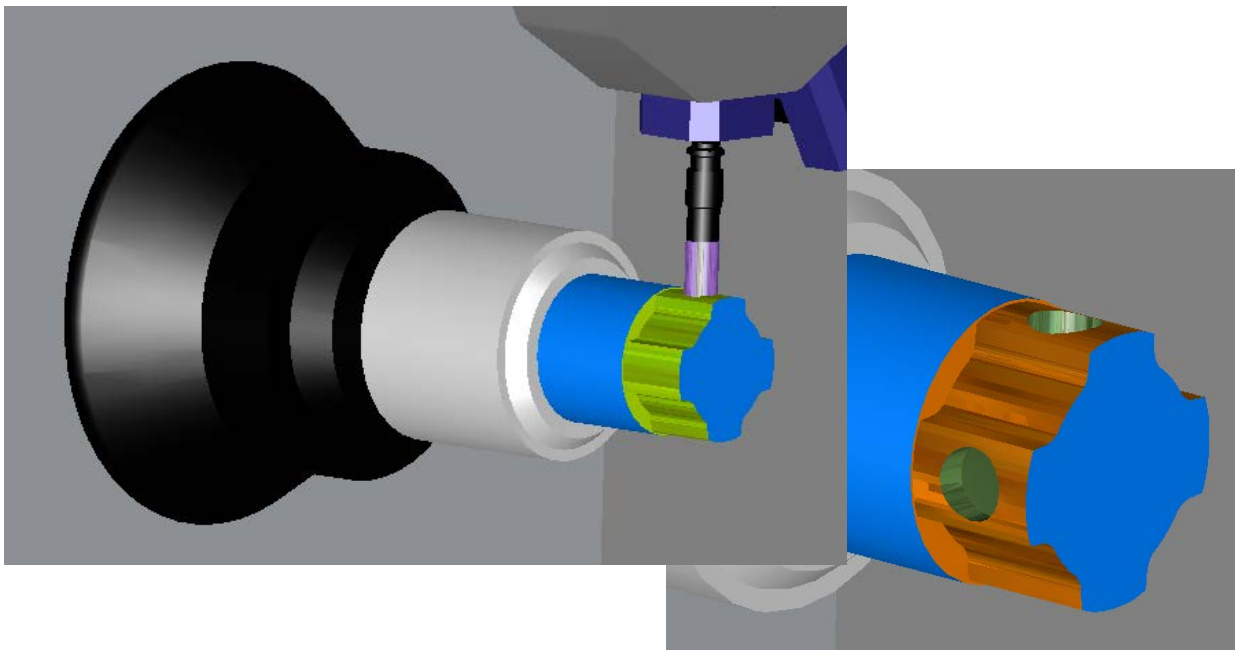


- Select **Radial around the index axis** and then enter **4** instances, **90** degrees apart

Notice how the options referring to the XY plane are disabled, as the orientation of the feature would not allow you to access that plane in a pattern

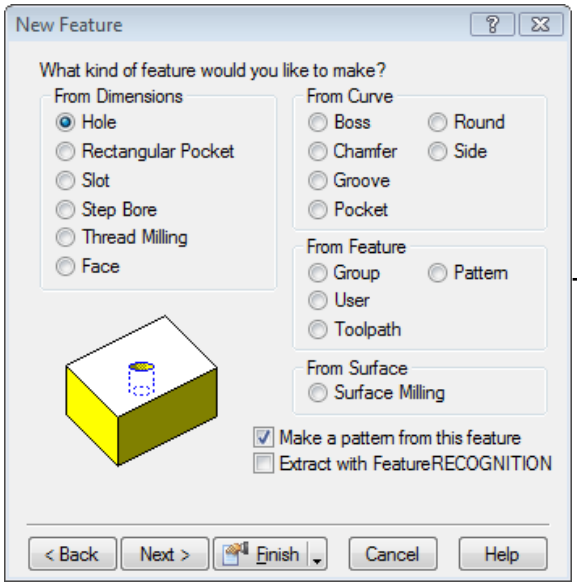


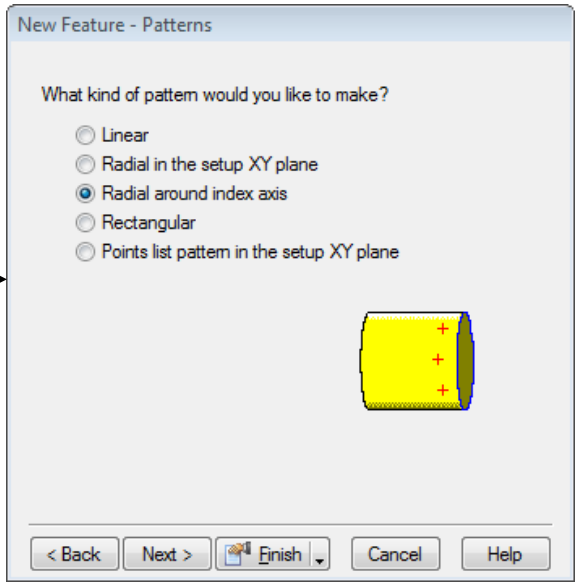
- Your part should be looking like the picture below



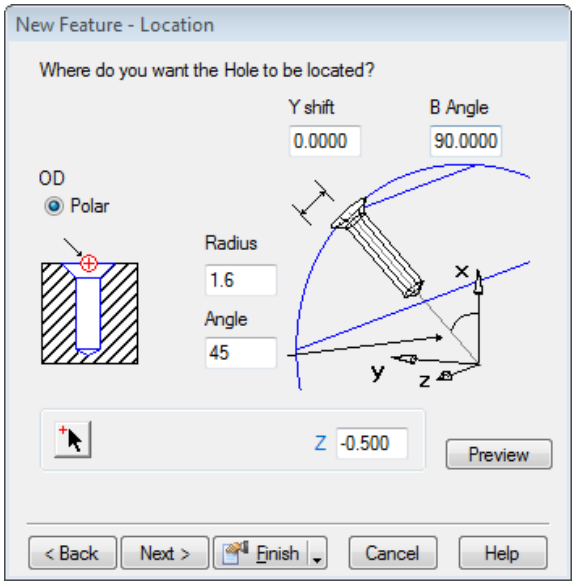
By now you get an idea of how features from curves are positioned based on the curve plane. Now let's go ahead and make some features from dimensions to see what parameters need to be changed in order to position them in the correct location.

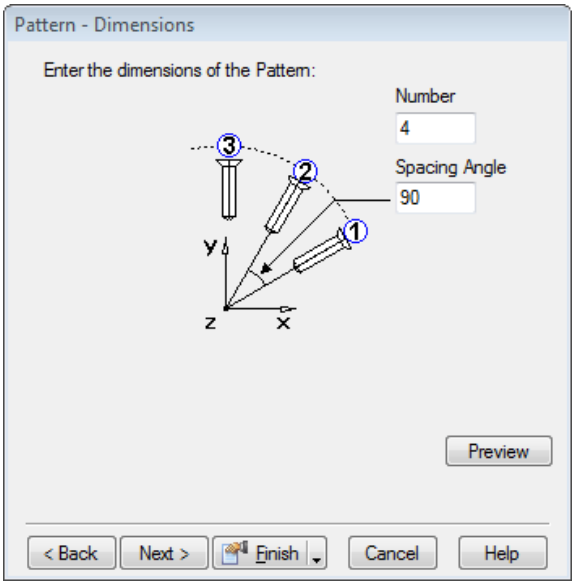
- Go to **New Feature**, **Turn/Mill**, **Hole from Dimension**. We will make some holes around the OD of the part. In order to reduce the number of steps needed to create the feature, also **check** the box **Make a pattern from this feature.**
- In the **Patterns** page, select **Radial around index axis**





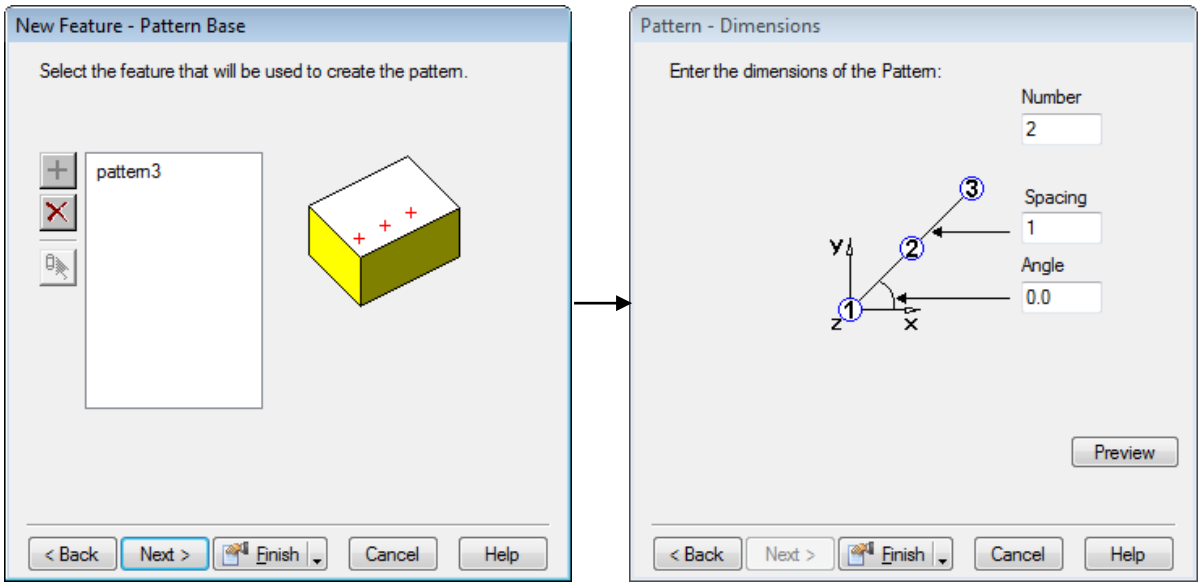
- In the **location** tab enter a **Radius of 1.6**, an **Angle of 45** and a **Z location of -0.500.**
- In the **Dimensions** page, enter **Number 4**, **Spacing Angle 90**



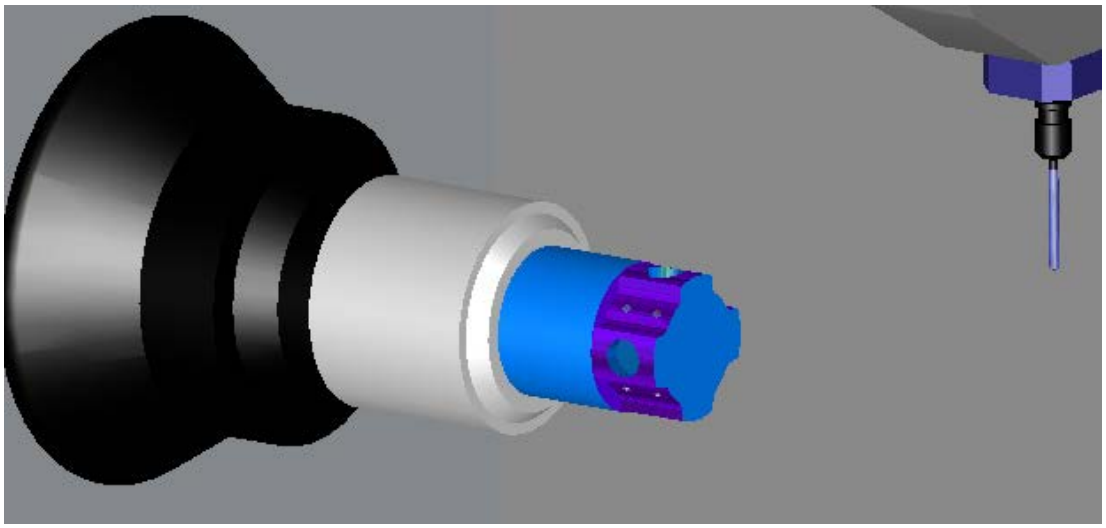


If you click on preview you should see how we're positioning 4 holes, about the index axis, with a starting angle of 45 degrees, at a Z location of -0.500, repeated 4 times at 90 degree intervals.

- We will now make a linear pattern out of the radial pattern to repeat these holes 1 inch down in the Z direction
- Go to **New Feature**, **Turn/Mill**, **Pattern from feature**
- Select the **pattern feature**, click **Next**. Then **Linear style**, **Next** and repeat this pattern **2 times** with a **Spacing of 1**



- Your finished simulated part should look like this

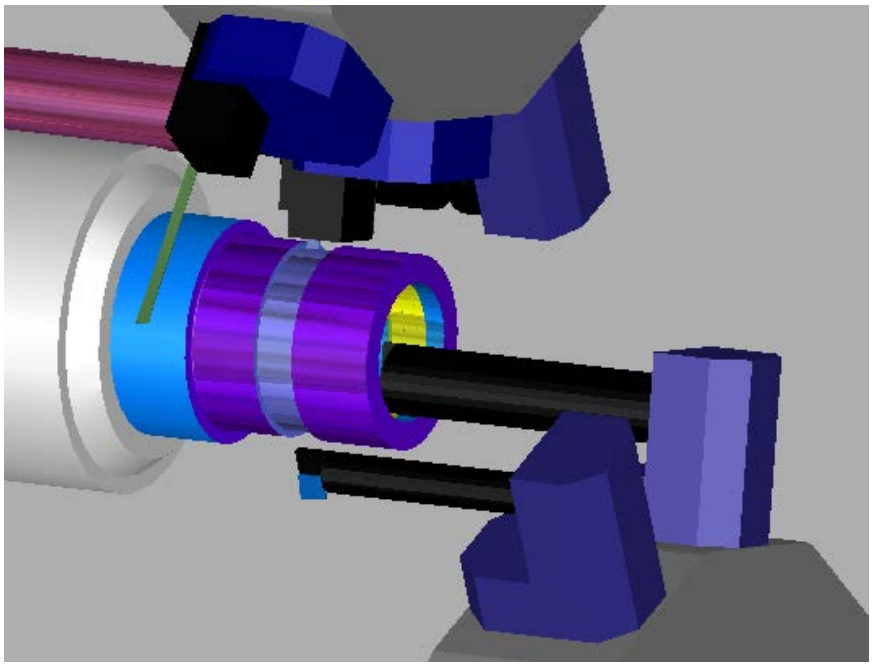


Extra challenge: if you are ahead of the class and are familiar with subspindle operations, transfer the part over to the subspindle and then create 2 flats 180 degrees apart on the other end of the stock. Cut one with Y-axis and one with C-axis

Synchronization Basics

Introduction

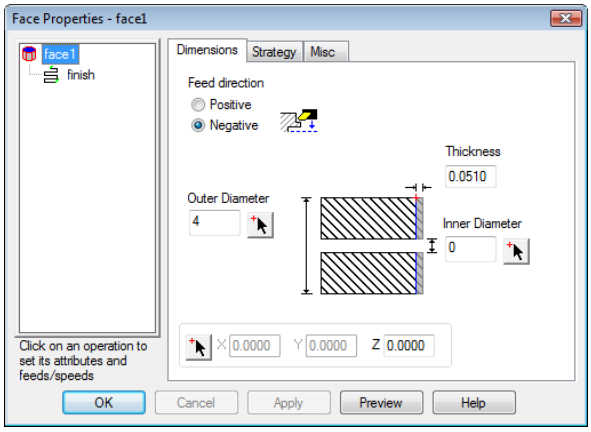
This exercise will go through the basic concepts of synchronization and how to achieve a faster running time on machines with multiple turrets. FeatureCAM can program anywhere from 1 to 4 turrets simultaneously. There is not a right or wrong answer when you do synchronization as long as simultaneous operations are capable of happening at the same time. When you are done with this file, your results should be similar to the picture below.



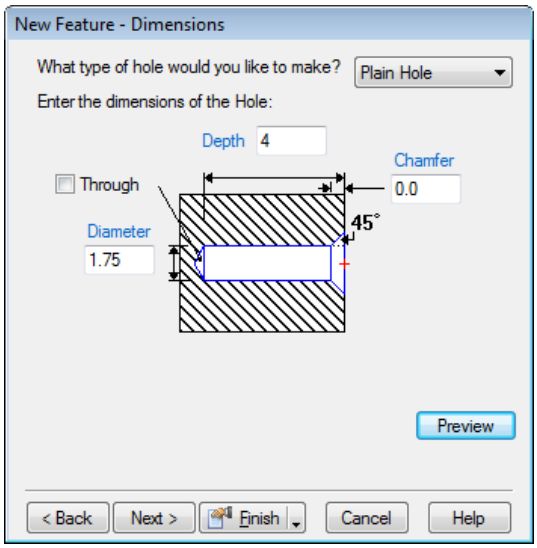
- Click on **File** then **Open**. Browse and open **Synchronization basics.fm**
- Now, in order to select the right machine for the job, we need to use the right post (which contains information like number of turrets, b axis capabilities, etc). For this job we will use **Naka WT-300 Fanuc18i w Sub & Y.CNC**

  Note that this post is available from the partner area under the TM-lbry > dual units > Nakamura – Tome WT folder. By selecting this post, you are also making sure that the correct machine design file for simulation purposes is being selected.

- We will make **all turning** features on this file. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from dimensions > face** and press **next**. Make sure the parameters in the page are correct and click on **Finish.**

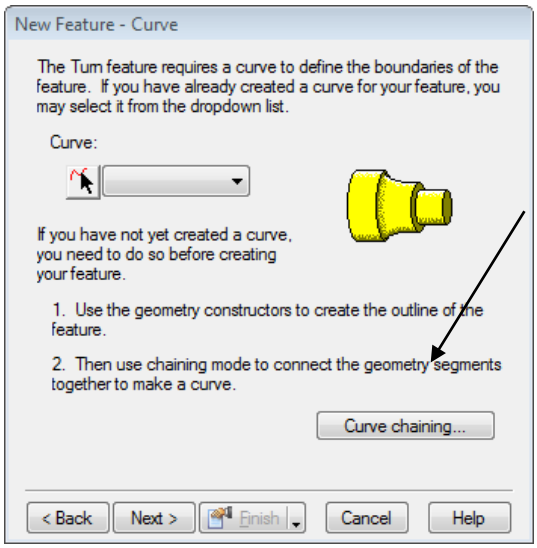


- Now let's make a hole feature on the face of the part to open up the ID. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from dimensions > hole** and press **next**. Make sure the parameters in the page are correct and click on **Finish.** You should enter a diameter of **1.75** and a depth of **4** (see screengrab below)

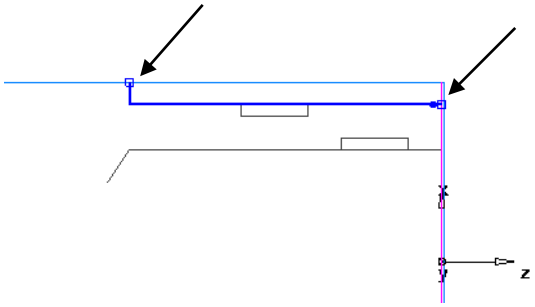


- We're ready to machine the OD now. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from curve > turn** and press **next**. From the following page select **Curve Chaining.**

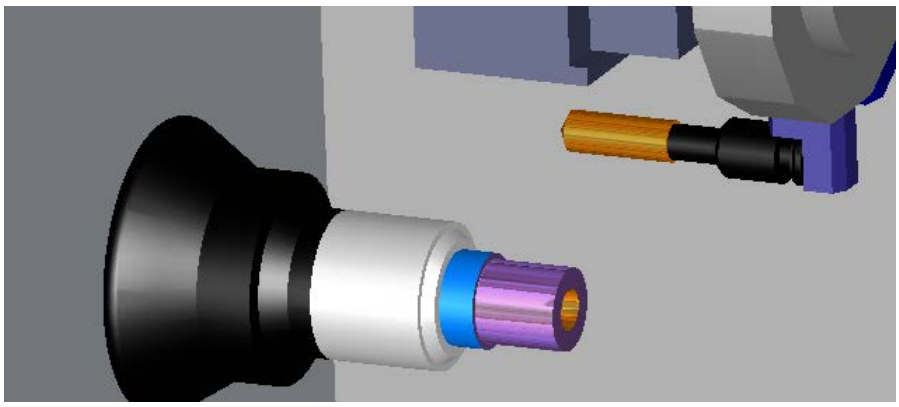
You will find that in this page we need to select a curve. Notice that we have no curves available, but FeatureCAM allows us to 'Chain on the fly'. You can click on the **Curve chaining** button and then pick the points on the screen to select the shape to be machined (this method shows extremely well when doing demos, as it allows for a continuous flow without having to stop the programming process to create curve)



- Create your curve from start to end picking on points following the geometry below



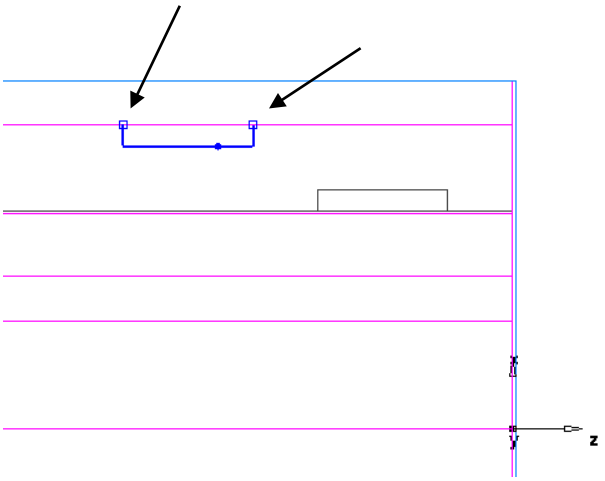
- Click **next** and notice how feature create a 3D representation of the feature to be machined. Click on **finish** and run **machine simulation** of the part to see our toolpaths so far. You should see this:



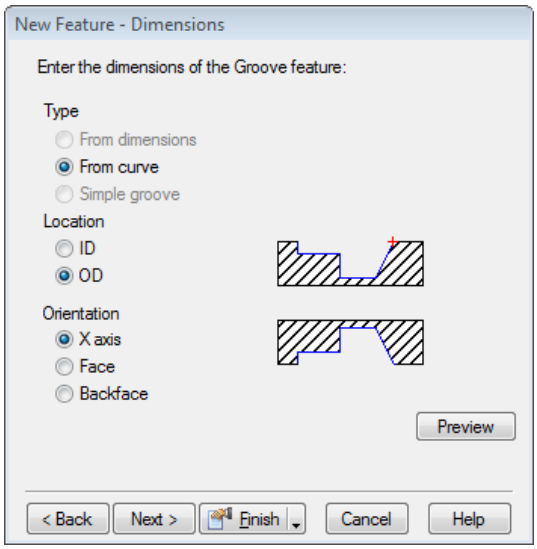
- Following the same steps, make a **Bore** feature. Pay attention **not** to select part of the internal groove when you chain on the fly
- Now we're ready to machine the 2 grooves. We could create grooves from dimensions, however for this file, we will use the geometry already

available and make grooves **from curves** selecting the curves available in the document and using once again **'chain on the fly'**

- From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from curve > groove** and press **next**. From the following page select **Curve Chaining.**

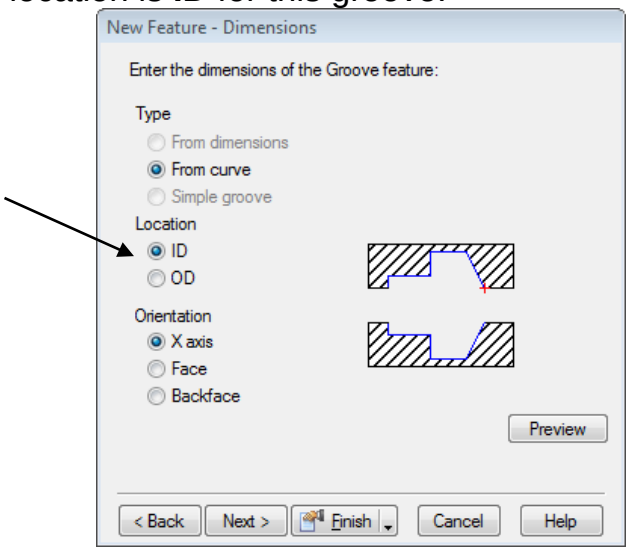


- Notice that we only need to select 3 pieces of geometry – the 2 walls and the bottom diameter of the outside groove. Once you selected the 3 pieces, continue selecting **next** until you get to the **dimensions** page. In this page it is important that you select the correct orientation of the feature. In this case this is an OD feature, so we don't need to change anything.

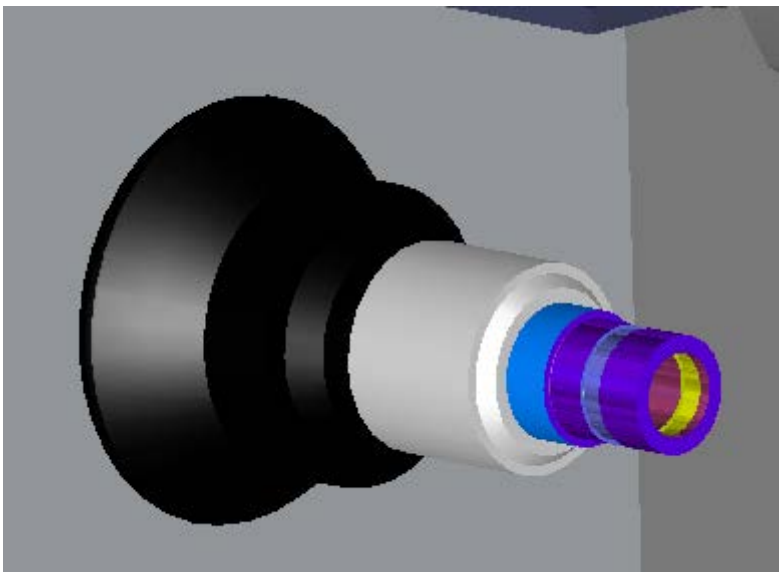


- Now let's follow the same steps and machine the **inside groove** feature. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from curve > groove** and press **next**. From the following page select **Curve Chaining.**

- Once you've chained your curve continue clicking next until you get to the **dimensions** page. Now pay attention to the **location**. You need to make sure that the location is **ID** for this groove.

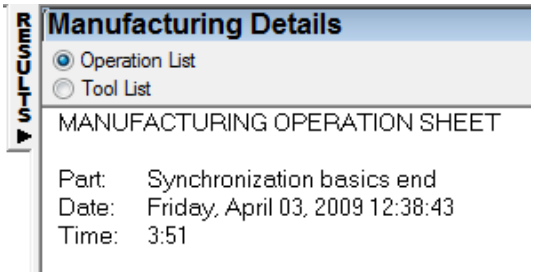


- Let's run machine simulation and take a look at the final machined part



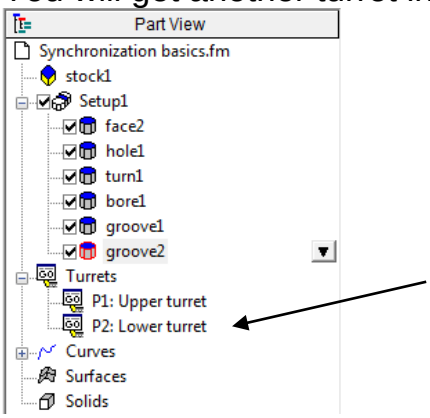
Now, why do we need synchronization?? The part is finished, we didn't gouge the part nor the machine, simulation looks good and toolpaths are fine. Well, if we look at this machine, you will notice that it has 2 turrets. At the moment, we're only using one. Basically we use synchronization to cut parts faster. It is mathematics… 2 turrets are faster than 1. So, our first step will be to look at the machining time.

- Run **centerline** sim, go to the details tab and look at the current time it takes to machine this file. My details tab looks like this (3:51). Your results could be different based on tool grade selected, material, max travel, etc.



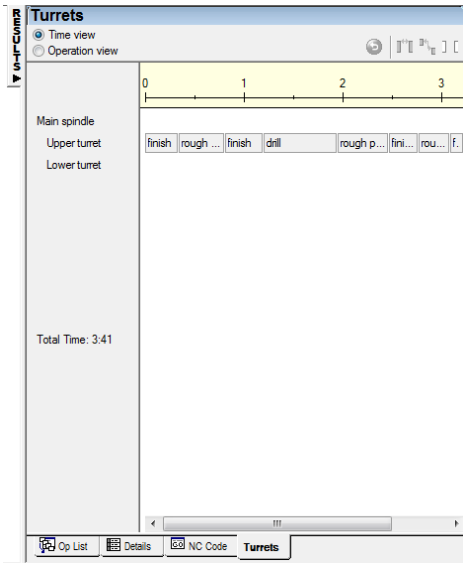
- Now let's analyze our tool usage. You will notice that whenever you select a post with multiple turrets 2 things will happen:

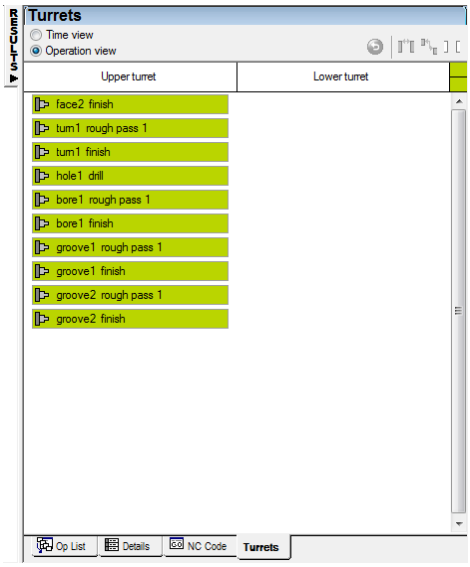
  - You will get another turret in the part view



  - You will get a new tab in the **results** window called **turrets.** Turrets is where you can see what operations are being assigned to each turret

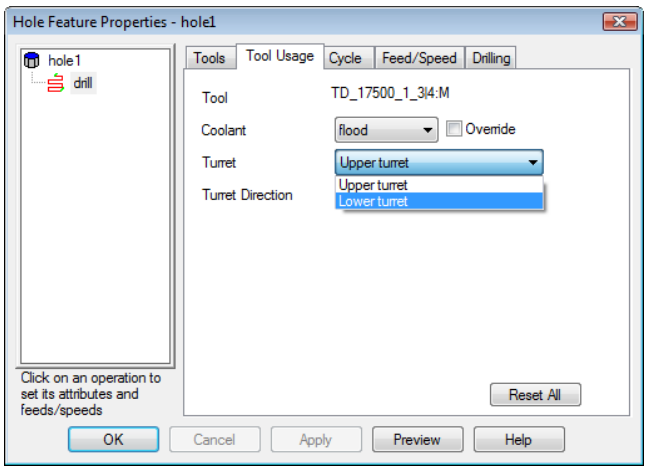
- Eject the simulation and click on **turrets**. Turrets has 2 view modes that you can select. **Time based** and **operation based**.



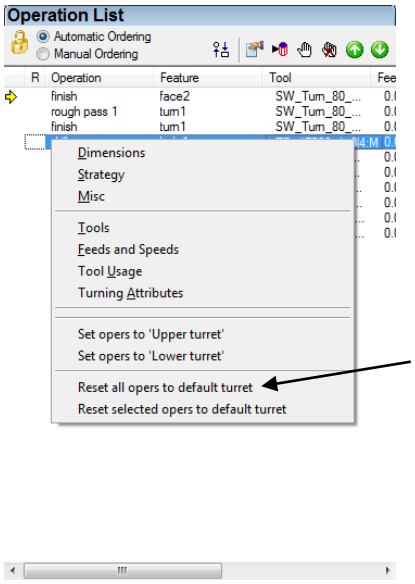


Time based offers a timeline that shows how long each operation takes. It is useful in the sense that you can visually check for operation time and adjust the order to accommodate for the shortest possible machining time. Operation based shows operations being displayed in columns. The operation based view is convenient for users, as it shows operations from top to bottom, just like NC code. It is a good idea to switch periodically while programming a part between these 2 modes in order to get the best of both worlds.

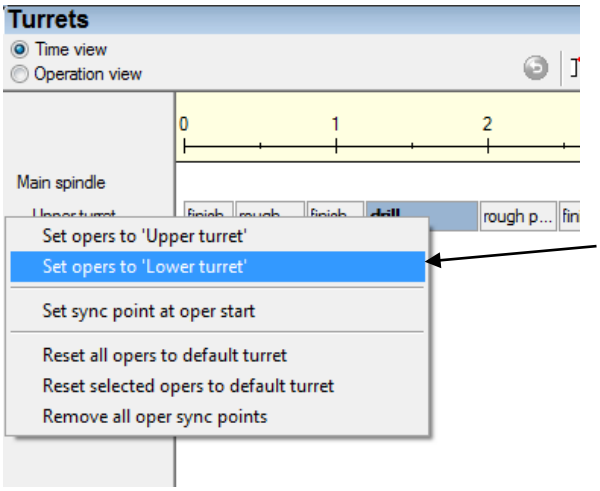
- Let's start by moving the drill operation (hole1 drill) to the lower turret. There're multiple ways to achieve this:
    - By opening the feature at the **operation level** and switching from upper to lower turret



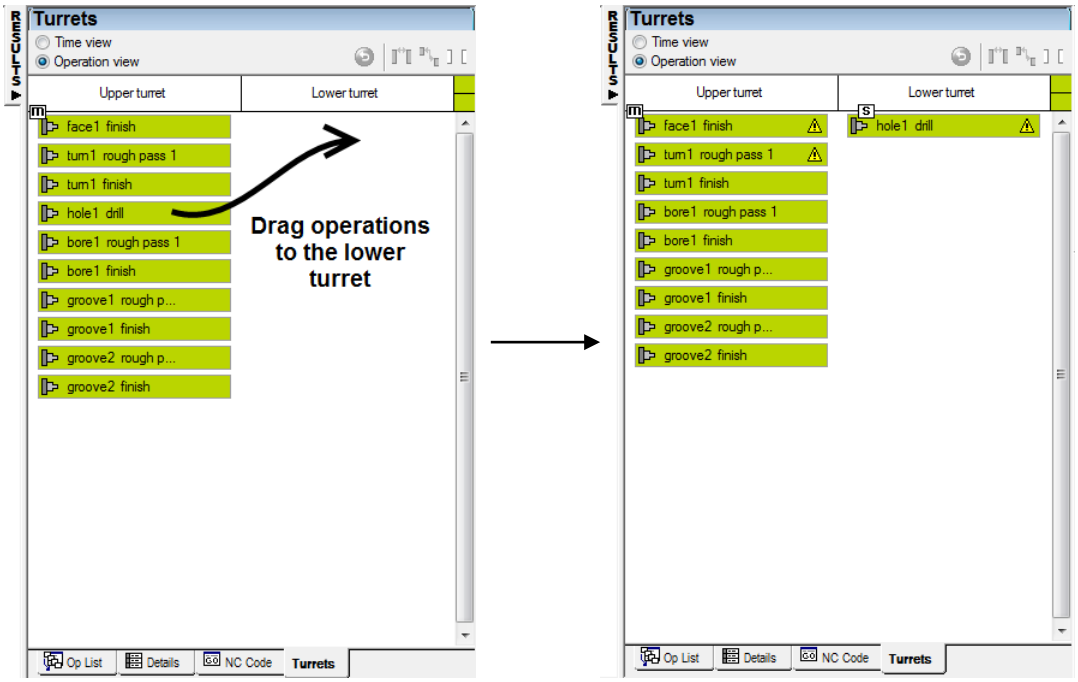
    - By right-clicking on the specific operation in the op list tab and selecting **set opers to 'Lower Turret'**. Note that you can move multiple operations with this method if you pre-select them at the same time



    - By right-clicking on the operation in the **turrets tab > Time view** and selecting the appropriate choice

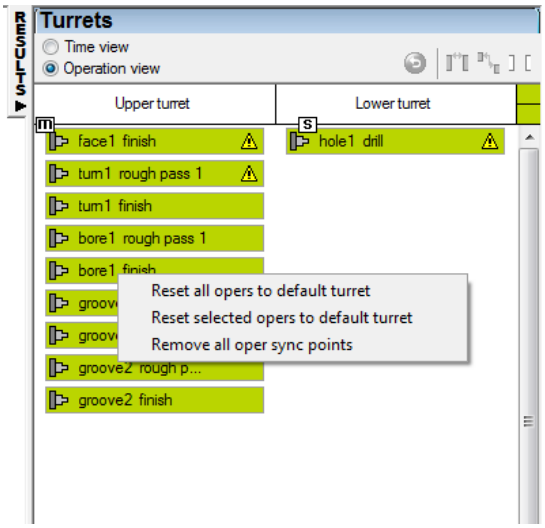


- By using the **operation view** and dragging the operation from its current turret to the desired one.

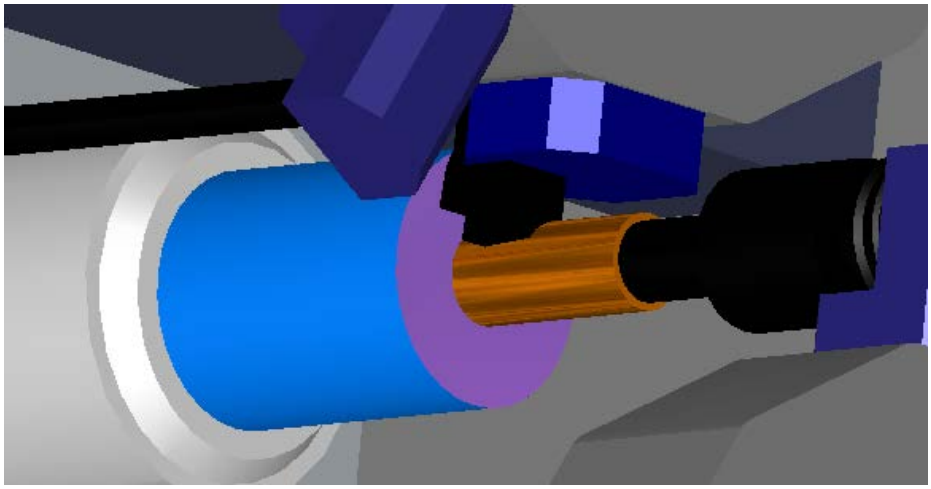


Tips:

- You can drag multiple operations in this page by holding the Ctrl key while selecting. This makes for much faster programming time as you don't have to move operations individually.
- If you make a mistake you can restore the previous ordering by moving the operation back to its original location or if you want to reset all operations you can right-click on the white area in the turrets tab and select the desired action (see below).



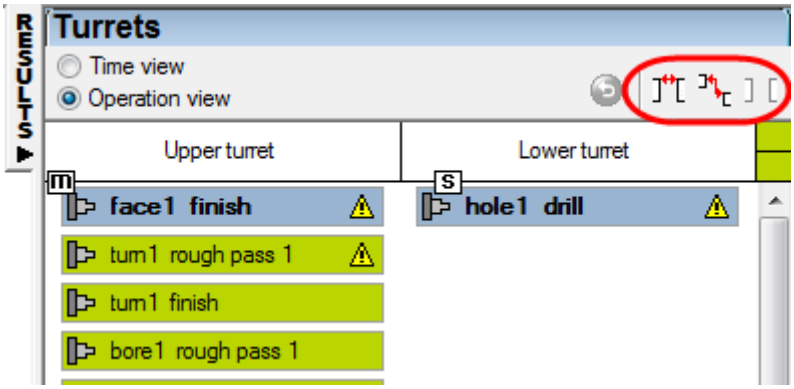
- Let's try it. **Left-click** on the operation you want to move (hole1 drill) and then **drag** it to the lower turret. You should achieve the same results as in the screengrab above.
- It is important noting that FeatureCAM will **automatically** select tooling that will apply to the current turret. For example, an operation using a **SW** tool in the upper turret will now use a **NW** tool in the lower turret
- Also notice that you now get 2 **warning** signs next to the 2 operations happening simultaneously. This is **not** an error, but just a warning. If you hover over the operation with the mouse you will get an indication of what the warning is. Another way to see what the warning refers to is by clicking on the details tab and reading the warning message.
- The warnings show because we have CSS (constant surface speed) turned on in one of the operations (face1 finish). However, the drill feature has a fixed speed. The machine would not be able to **vary** the speed for one operation and keep it **constant** for the other, so that's why it is issuing a warning. We will cover how to fix this at a later time, but for now we have a bigger problem in our hands… Let's run machine sim to see how things look so far. You should get a gouge like in the screengrab below



- The gouge happens because we have 2 operations designed to work on the face of the part. This is impossible to achieve at the same time, so we need to assign some priorities or ordering to them in order to keep the machine from crashing. **We call this ordering process Synchronization**. Synchronization is really easy to achieve in FeatureCAM. Just a matter of selecting operations and assigning ordering to them. Let's give it a go.

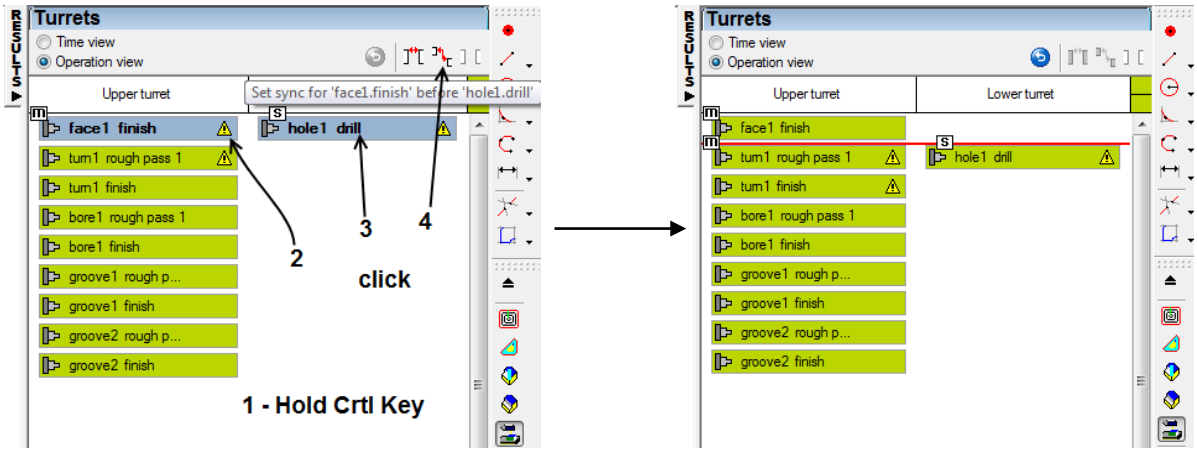
- Let's first introduce the UI objects you will need to become familiar with. They are located on the **upper right corner** of the turrets tab and become available as you select operations in the UI (either in the time view or operation view). See screengrab below. From left to right, they perform the following functions:

    - Set sync point at operation start: which means that all selected operations will start together. In this case the order how we pick the operations is **not** important
    - Set one operation before the other: which means that the first operation selected will happen **after** the operation selected after. In this case it is **very important** to select the operations in the correct order
    - Remove all sync points: will **remove** all the synchronization points set in the file except for pinch or follow operations

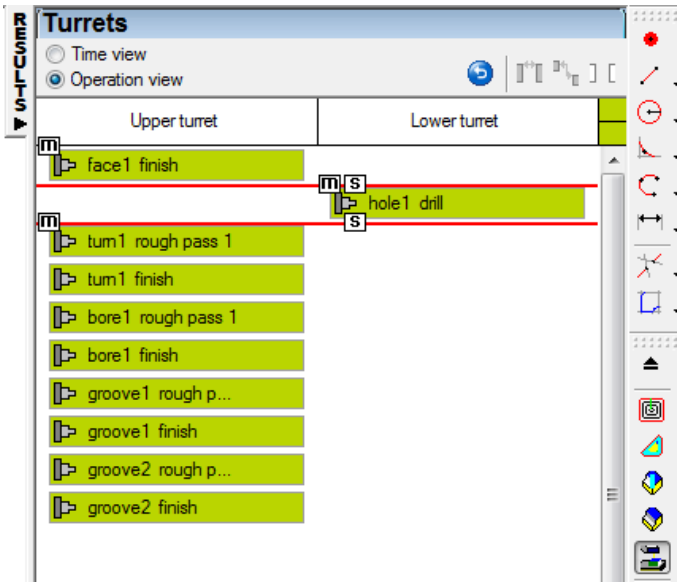


- Let's try our first synchronization exercise:
    - Hold the **Ctrl Key** – we need to synchronize 2 operations, therefore we need to select multiple operations with the Ctrl key
    - Click on the operation **face1 finish**
    - Click on the operation **hole1 drill**
    - Click on the **middle sync icon** (one operation after the other)

Another way to synchronize that will not be covered in this tutorial is to select the operations with the CTRL key and then (with the operations selected) **right-click** on them and select the appropriate synchronization option. Both methods are valid and work in the same manner. Just different ways to get to them.



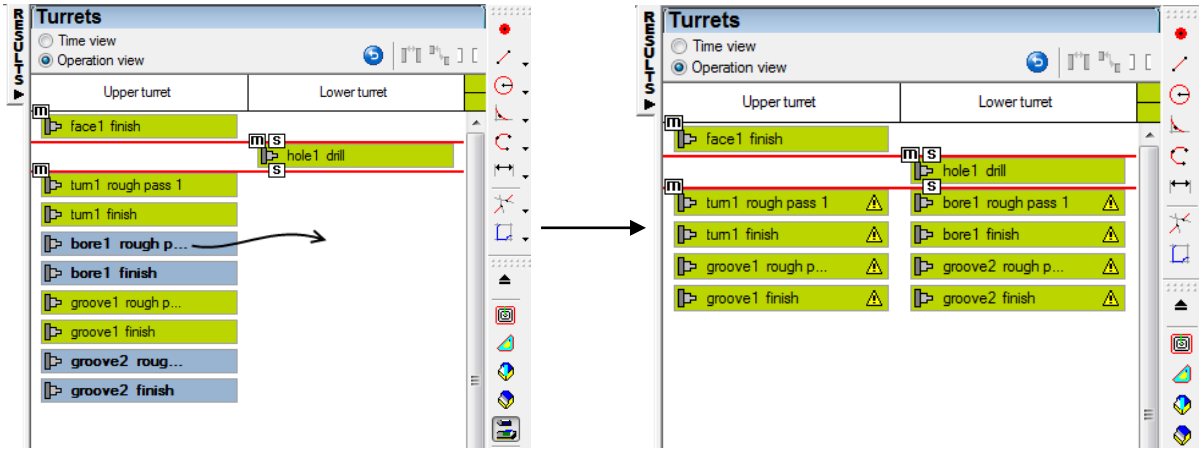
- Now we have another possible issue. We need to make sure that the drill operation will be done before machining the OD with the turn1 rough pass1 operation. Follow the steps above to make the **hole1 drill** happen before **turn1 rough pass1**. Your turrets tab should look like the picture below. Run machine simulation and see how the turrets now sit waiting for the next operation to happen.



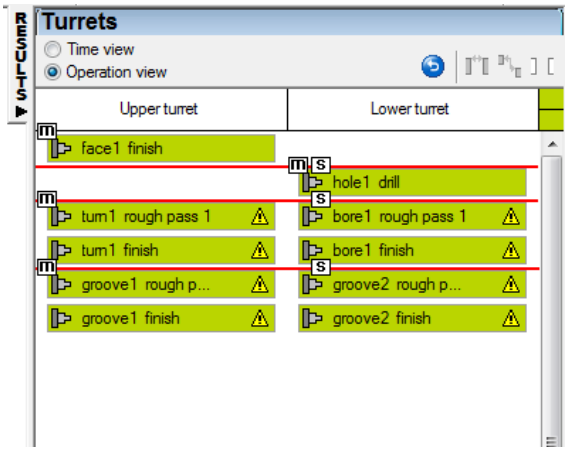
When you look at the turrets tab in the operation view mode you will notice that there're little **M** and **S** distributed along the synchronization points and at the start of the program. They stand for **main spindle** and **sub spindle** and denote which turret has control of the **spindle speed** at the time. As a default FeatureCAM will assign control of the main spindle to the upper turret and the subspindle to the lower turret. In cases where there no operations in one of the turrets, the speed control will be passed to the turret performing the operation.

If you are pinch turning or follow turning on the main spindle, the upper turret will have control of the speed. If you are pinch turning or follow turning in the subspindle, the lower turret will control the speed.

- Let's now synchronize the operations so that we turn the OD and bore the ID at the same time and also add another sync point so that we turn the OD groove and ID groove at the same time.
- First we will drag all 4 ID operations to the lower turret. To do this hold the **Ctrl Key** and select the 4 operations shown below. Then **drag** them to the lower turret

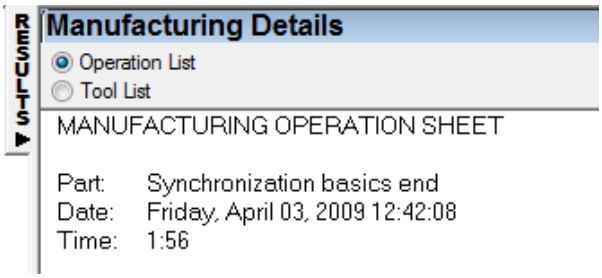


- Now turn1 and bore1 will start simultaneously. The only thing we need to do here is to make groove1 and groove2 start at the same time. To do this Hold Ctrl and click on **groove1 rough pass1** and **groove2 rough pass1** then click on the first sync icon (set sync point at operation start)

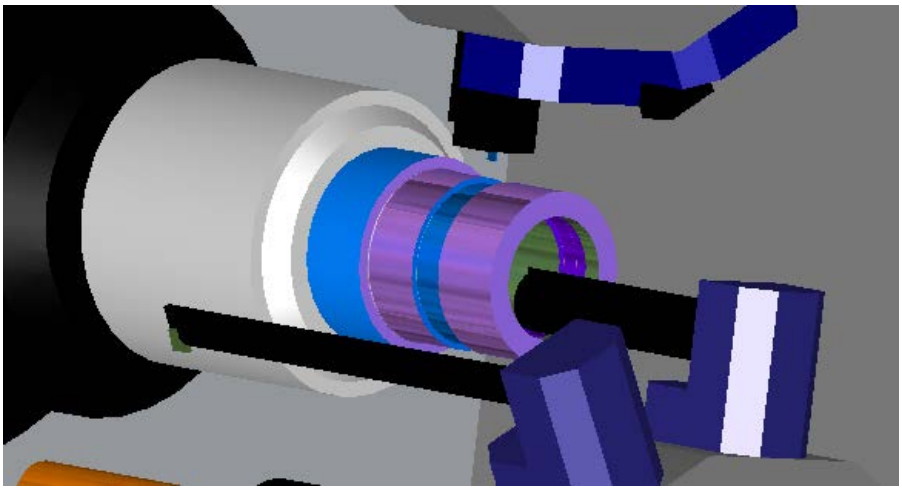


- Let's look at the time. Now… see how much time we can save by synchronizing features. Remember, previous time without synchronization was **3:51**

- Run **centerline** simulation and go to the **details** tab to look at the new time. Mine is now **1:56**. So by synchronizing features, we save close to 2 minutes per part!



You're done!!! Your simulation should look like this. There should be no gouges or collisions.



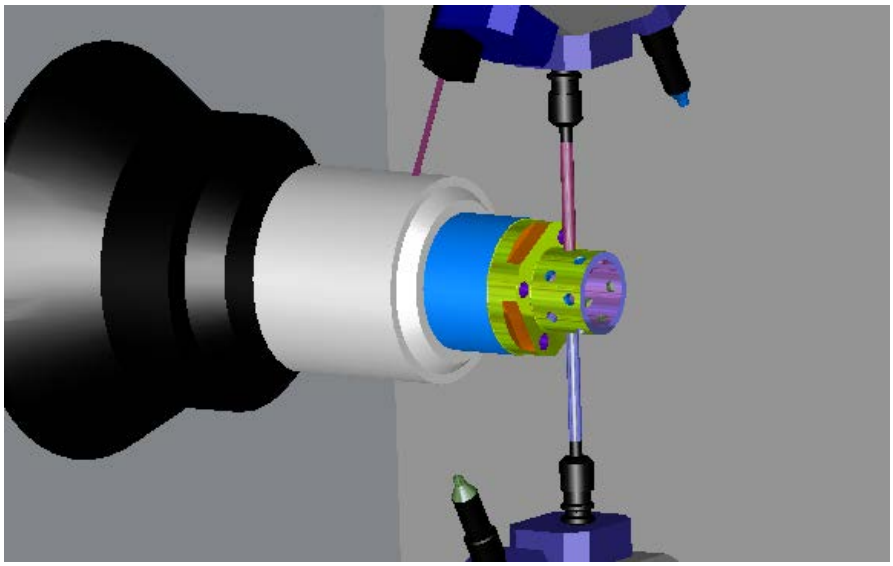
Miscellaneous notes:

- If you look at the turning operations you will notice that they have the warning signs regarding using CSS in different operations in different turrets at the same time. You can get rid of the warnings by switching to a fixed speed in every operation
- You can shorten the length of the boring bars by opening the properties of the tool being used (in the feature itself or tool mapping) and changing the length of the boring bar or the exposed length

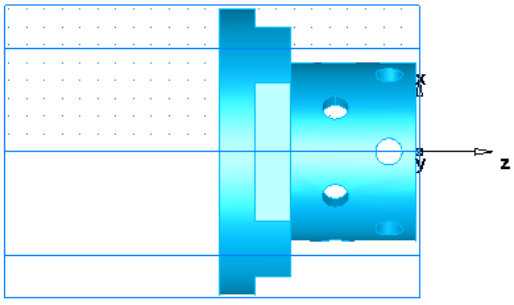
Hole synchronization

Introduction

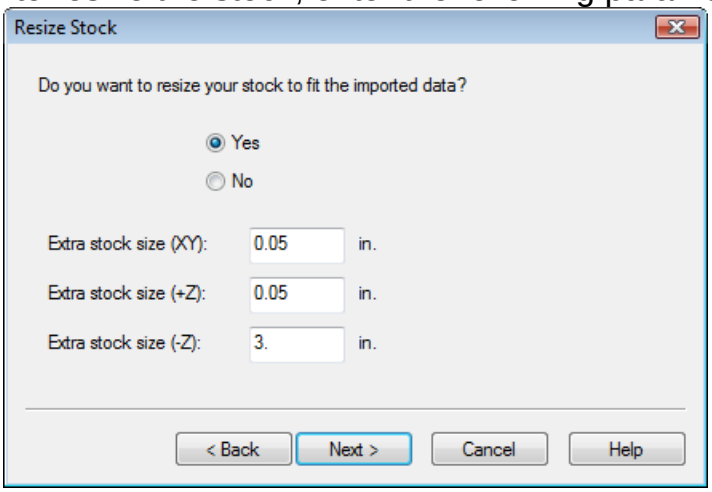
This exercise will show you how to synchronize operations to achieve much faster machining time. It will focus in the process of having 2 turrets working in the same spindle at the same time with operations that can coexist with each other.
In order to completely understand this tutorial it is critical for the user to be familiar with the previous module (101 – Synchronization basics) as this exercise will presume that you will already posses basic knowledge about some advanced turn/mill capabilities of the software.



- Click on **File** then **Open**. Since we will be importing a different file type than *.fm, go to the bottom of the open dialog and select '**Files of type: all files'.** Go to the Data folder in MTT100 and open **hole synchronization.x_t**
- Select a **Turn/Mill** document in **inches**
- Align the part using the import wizard so that Z is pointing in the same direction as the picture below

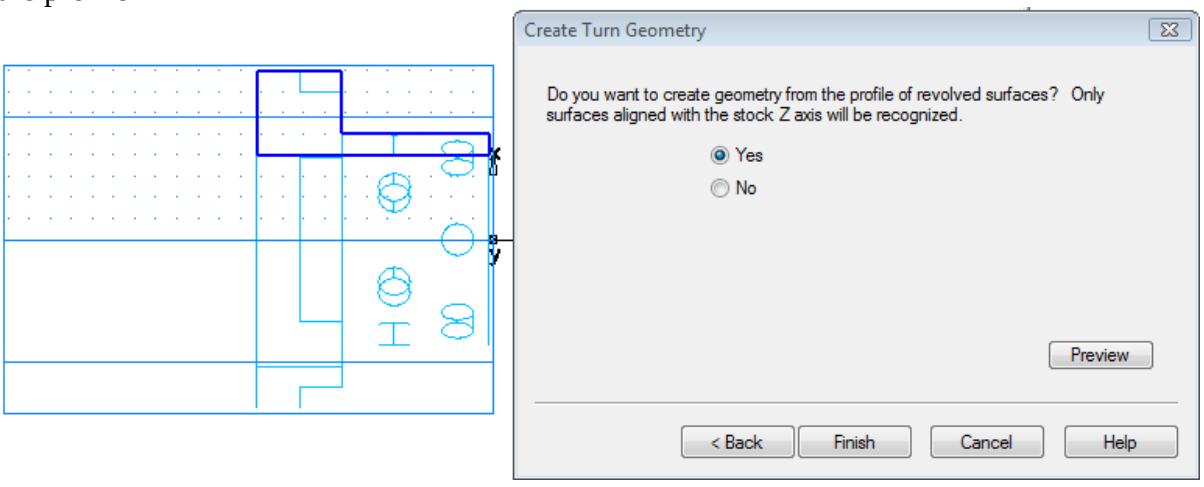


- When it is time to resize the stock, enter the following parameters



- Continue going through the wizard. **Do not** automatically recognize features when you get to that page. Although you could do this and finish the project much faster (good during a demo or when already familiar with the software), the purpose of this outline is to show you tips and tricks to improve your skills. Click **Next**
- In the final page, select **yes** so that FeatureCAM creates geometry of the profile of the solid. Click on **Finish**

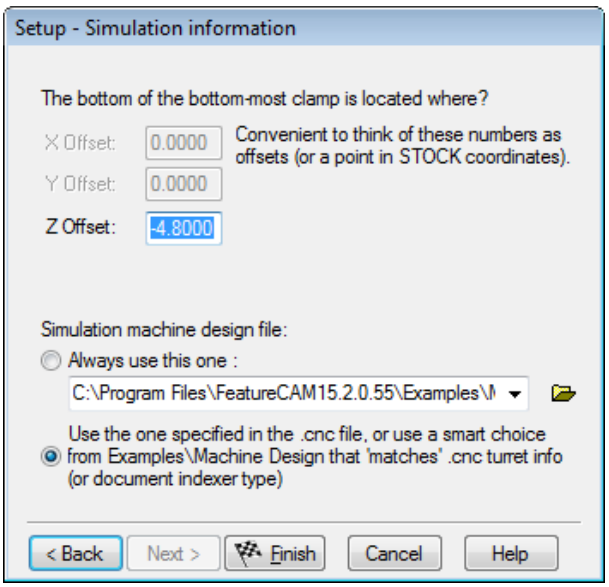
When you click on **preview** you will get a blue profile of the geometry FeatureCAM will construct for you. Notice that the solid will be revolved about the Z axis and as a result the geometry being constructed will not go into the areas with the flats. This is especially useful when it is turn to program the turned features, as we won't need to create extra geometry of the profile



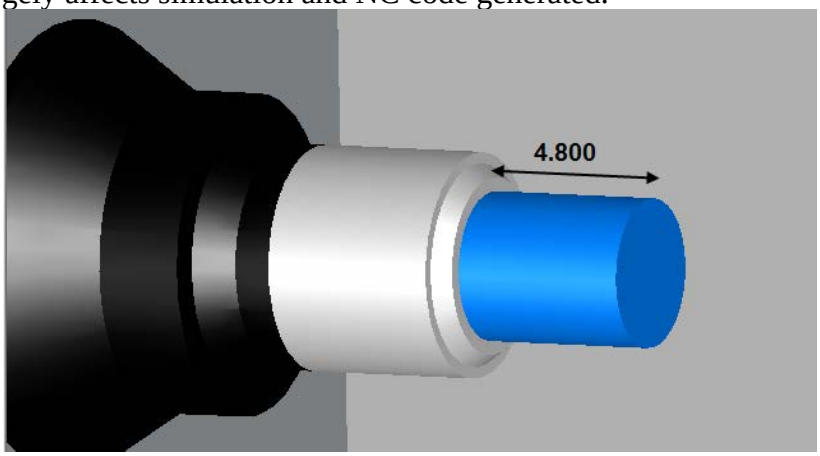
- Now, in order to select the right machine for the job, we need to use the right post (which contains information like number of turrets, b axis capabilities, etc). For this job we will use **Naka WT-300 Fanuc18i w Sub & Y.CNC**

Note that this post is available from the partner area under the TM-lbry > dual units > Nakamura – Tome WT folder. By selecting this post, you are also making sure that the correct machine design file for simulation purposes is being selected.

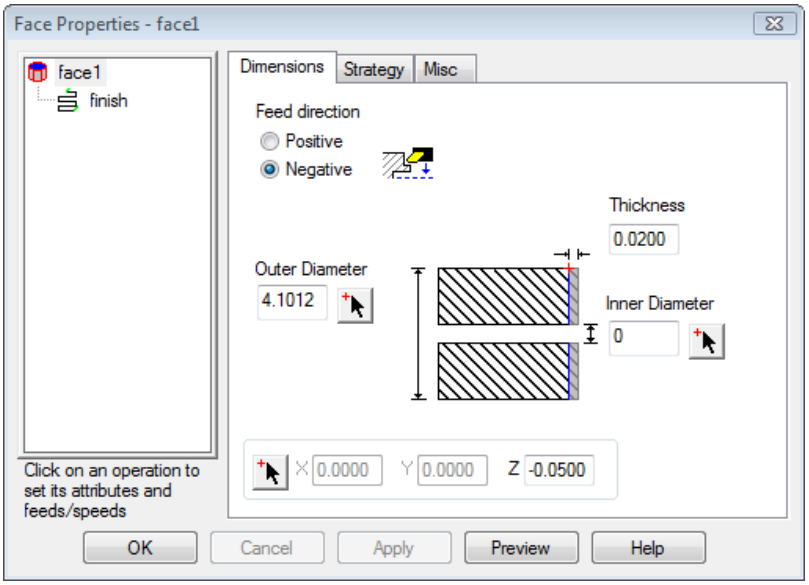
- Before we start creating features we need to position the stock appropriately in the machine. This is very important as it affects the simulation and also the NC code
- We need to tell the software how far is the **face of the jaws** (in the machine) in **relationship with the setup** we're using to program the part.
- **Double-click** on the setup go to **edit** and continue clicking **next** until you get to a page called **Setup Simulation Information**
- If we look at the properties of the stock we will notice that its length is 5.8 inches. Let's say we want to grab the part by 1 inch, so the face of the jaws would be **-4.8** from the **setup**. We need to **enter -4.8** in the Z offset in the Setup Simulation Information page.



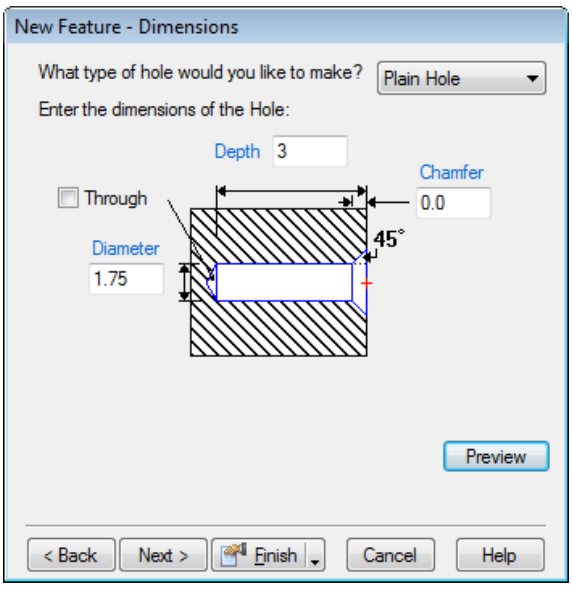
Entering -4.8 in the Setup Simulation Information page causes the following outcome when simulating. Even though the length of the actual stock is 5.8 inches, only 4.8 are hanging out of the jaws of the machine. It is very useful to understand how this works because changing this number largely affects simulation and NC code generated.



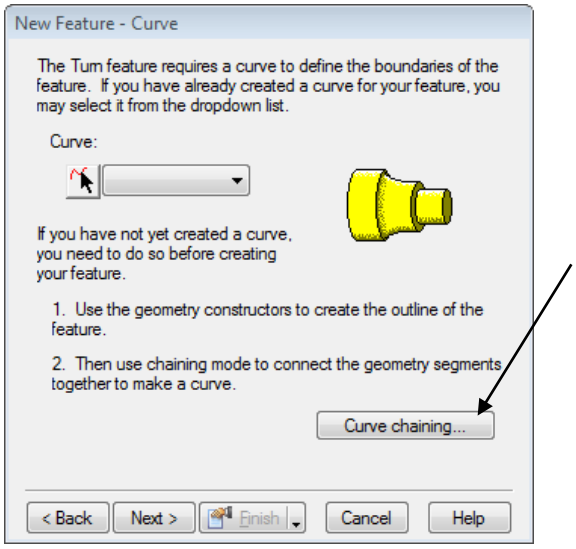
- Now we're ready to start programming features. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from dimensions > face** and press **next**. Make sure the parameters in the page are correct and click on **Finish.** Pay special attention to the **Z** location. If your setup is positioned in the face of the stock, you will need to adjust your Z location to be **-0.050** as we added 0.050 when we went through the stock wizard



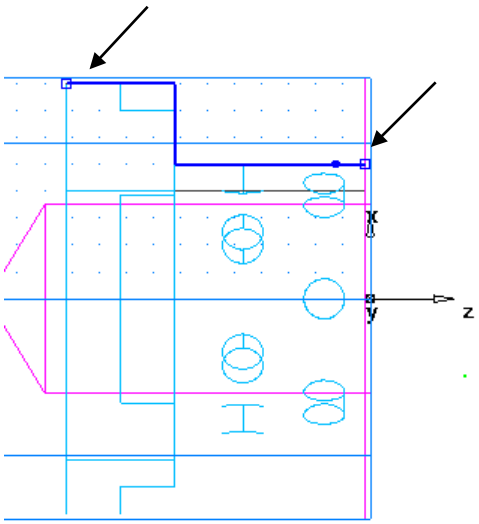
- Now let's do a hole feature on the face of the part to open up the ID. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from dimensions > hole** and press **next**. Make sure the parameters in the page are correct and click on **Finish.** You should enter a diameter of **1.75** and a depth of **3** (see screengrab below)



- We're ready to machine the OD now. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from curve > turn** and press **next**. From the following page select **Curve Chaining.**

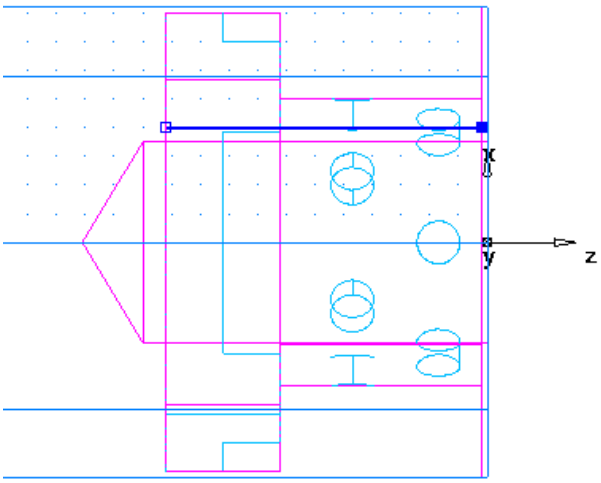


- Create your curve from start to end picking on points following the geometry below. Once selected click on **Finish**

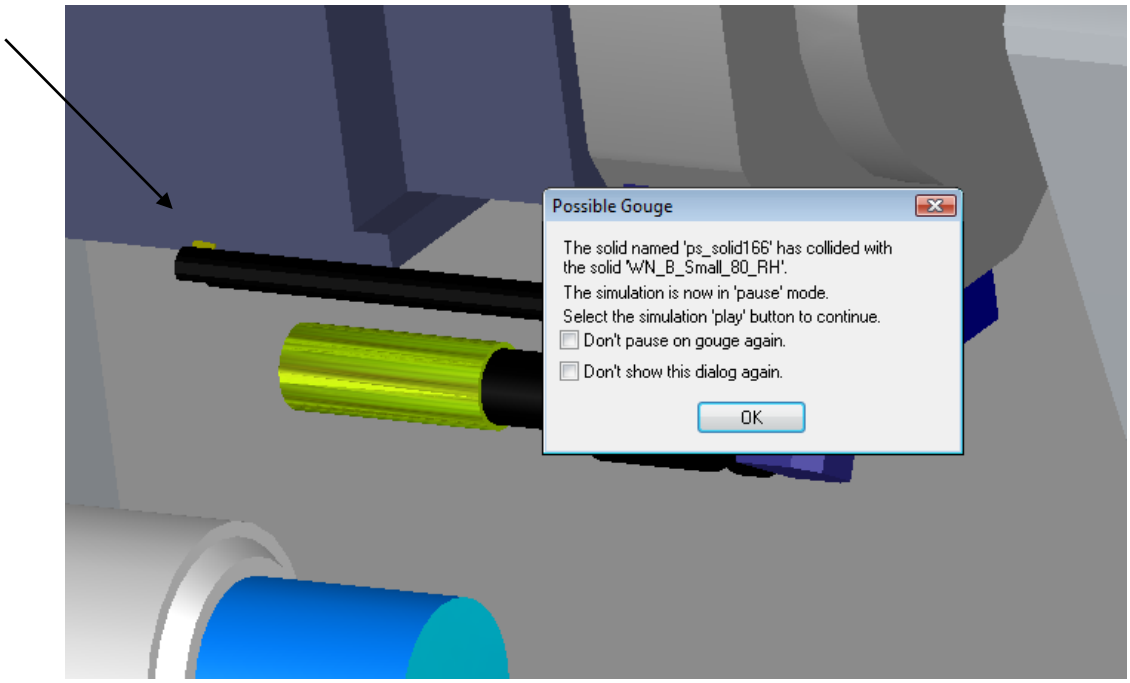


- Let's machine the ID now. From the **steps** click on the **new feature wizard**, choose **turning** and press **next.** Select **from curve > bore** and press **next**. From the following page select **Curve Chaining**

- Create your curve from start to end picking the geometry below. Once selected click on **Finish**

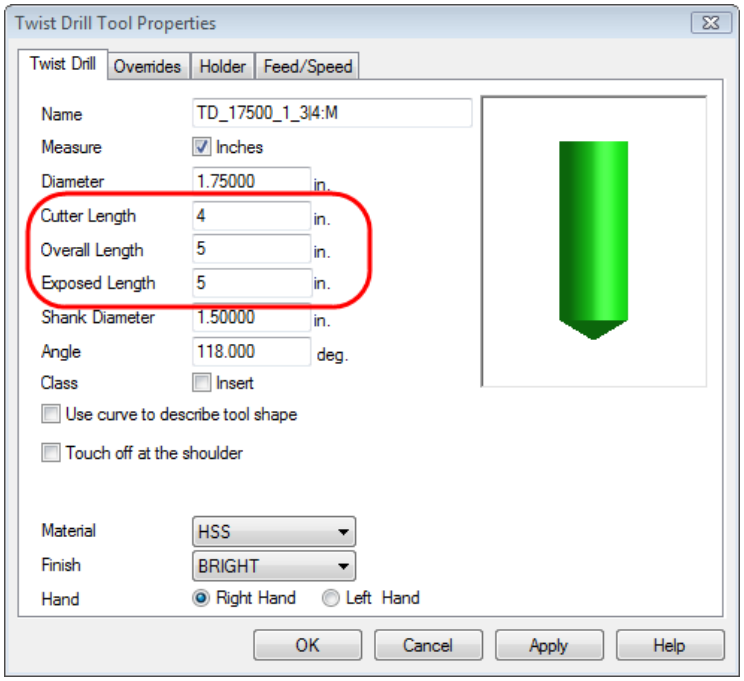


- We’re now finished with the basic turning features. Let’s run machine simulation to see the part being machined up to this point
- If you have **gouge check** turned on you will probably see a warning similar to the picture below. FeatureCAM detects a gouge when the boring bar collides with the body of the turret and pauses the simulation so that we can take action. Change the **length** of the boring bar and the drill

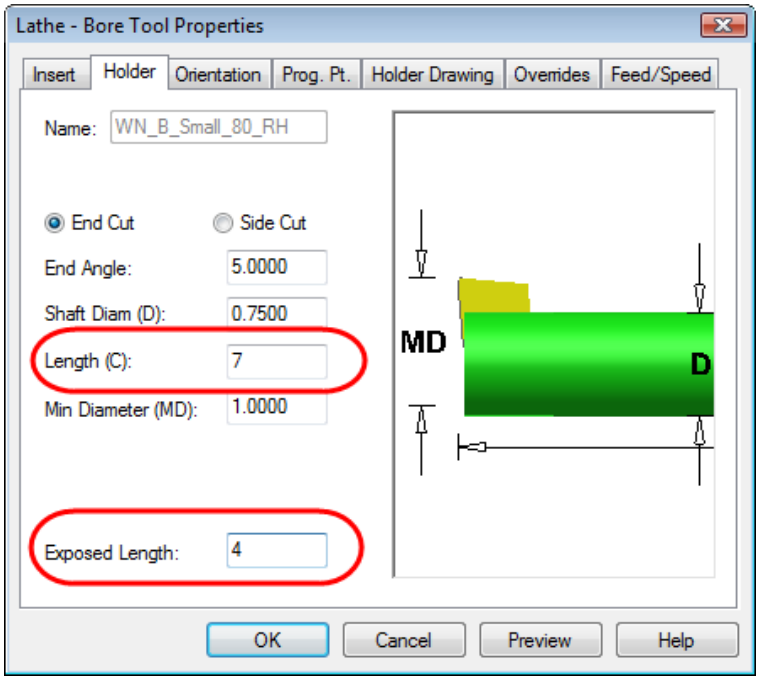


There are multiple ways to fix this. The easiest one is to **open the operations** using the tools that gouge (in this case the drill and rough/finish of the bore) and **edit** the tool properties and adjust to a realistic length that will not cause a gouge. You can also do this in the **tool manager** page, but the drawback is that you have to look for the specific tool you are using and you probably have thousands of tools in there. It is much faster and easier to go to the particular operation and make the changes in there

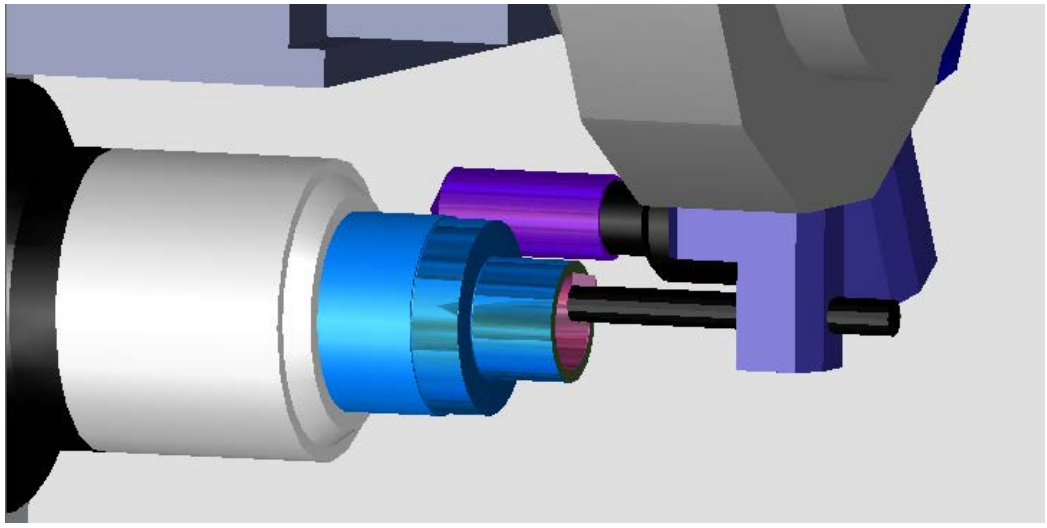
- Let's fix the drill first. **Double-click** on the tool being used and set the parameters like in the page below. Once done click **OK** and **OK**



- Now it is turn for the **bore**. Open the tool properties and change the length to **7** and the exposed length to **4**. Click **OK** and **OK**

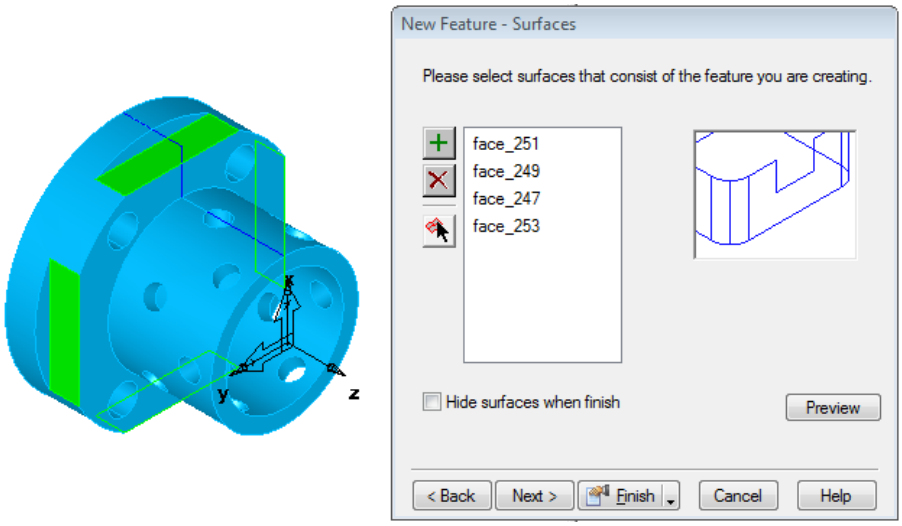


- Let's re-run machine simulation and see if we get any other collisions. You should not get any more warnings at this point. Also notice how the long boring bar now has a more realistic look to it and is not overhanging unnecessarily.



In turning it is extremely important to keep all tools as short as possible when setting up jobs. Longer tools produce bad finish, break inserts, create chattering, and cause unnecessary large clearance amounts to avoid collisions. We also want to keep the turret as close as possible to the spindles to reduce retract moves and travel as much as possible. Less retracts and shorter tools equal faster machining time which is the reason why these machines are more and more popular every day.

- Let's now move to the turn/mill feature and machine the 4 flats. There're several methods for machining the flats. We will cut the features using a combination of X and C axis moves.
- From the **steps** click on the **new feature wizard**, choose **turn/mill** and press **next.** Select **from curve > side** and check the box **Extract with Feature RECOGNITION** press **next**.
- In the next page, we want to recognize features **Along the setup Z axis.** Select that option and press **next**
- Here you have multiple methods to recognize the features. We will go through **select side surfaces** press **next.** You are welcome to select your own method if you are already familiar with other FeatureRECOGNITION options
- Select the surfaces we want to machine **(below in green)** and add them to the list. You can hold down **Ctrl** if you want to select multiple surfaces at the same time and then click on the **+** sign to add them all together then click **Next**



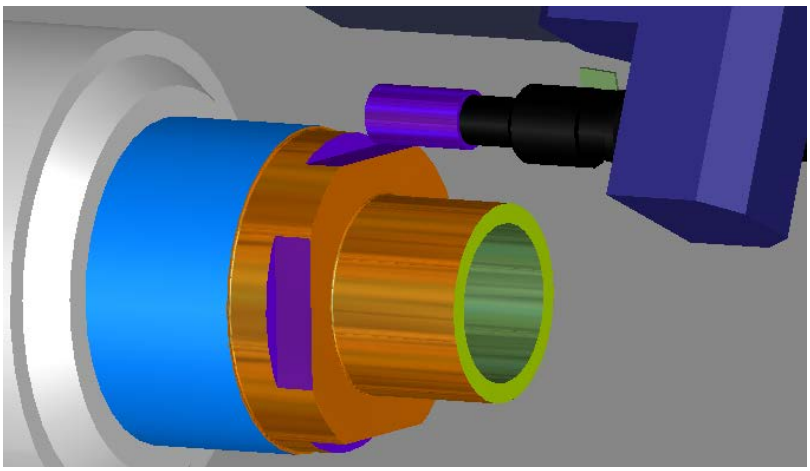
- Continue going through the wizard verifying that the settings and values in the following pages are correct. Stop when you get to the **New Feature – Strategy** page
- Since we don't have a lot of material to remove, **uncheck the rough** operations and just **check the finish**
- When it is time to select tools use a **0.750 endmill**

As a default FeatureCAM will use the biggest diameter possible when creating toolpaths. Although we could use the biggest possible tool, usually turnmill machines and live heads don't have a lot of torque, so it is better to select a smaller tool that can spin faster and provide better finish and not cause issues with the machine

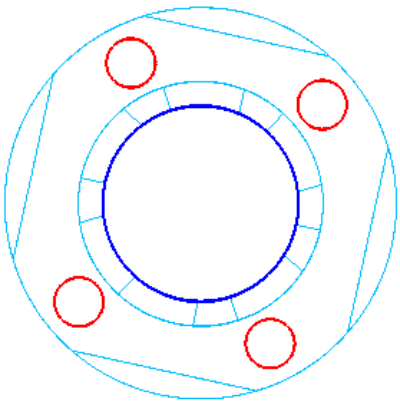
- Run **machine simulation** and see the progress so far. You will notice that the drill is probably now gouging when the end mill is machining the part. **Ignore** this warning, as we will eventually move the drill to the lower turret and the warning will disappear.

If you decide you want to solve this gouge, you can do one of the following:

- Edit the properties of the tool holder for the drill (select a shorter holder)
- Use tool mapping to assign a different tool slot for the drill so that there will be no gouges

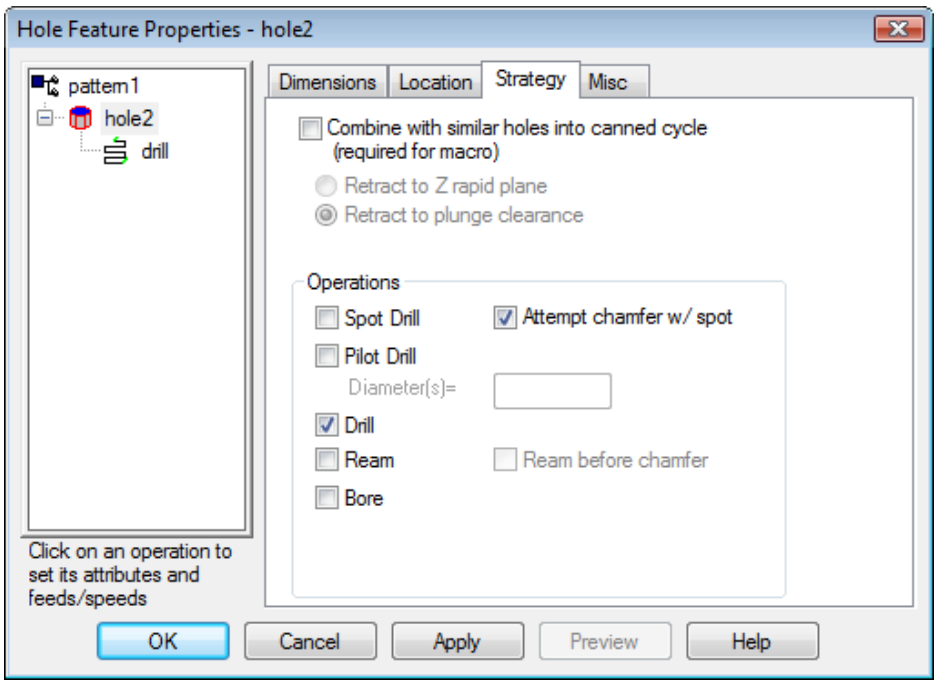


- Time to machine the 4 holes aligned with the **Z axis**. Go to the **new feature** wizard, select **turnmill**, **hole** from dimension and check the box **Extract with Feature RECOGNITION**
- Click **next**, select **Along the setup Z axis**
- **Pick** the 4 small holes in the face of the part. See screengrab below

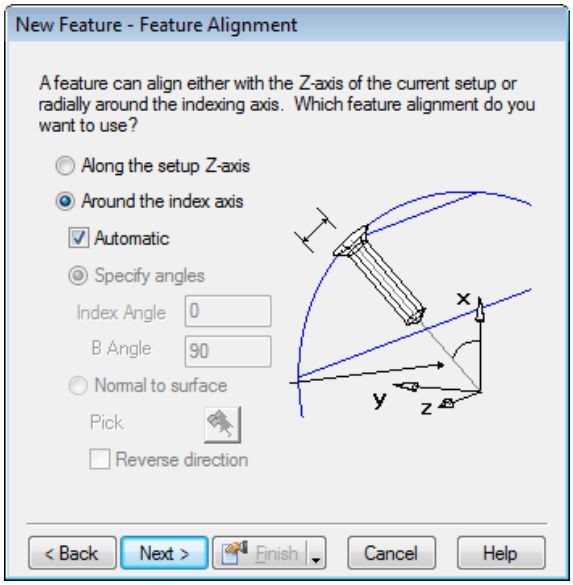


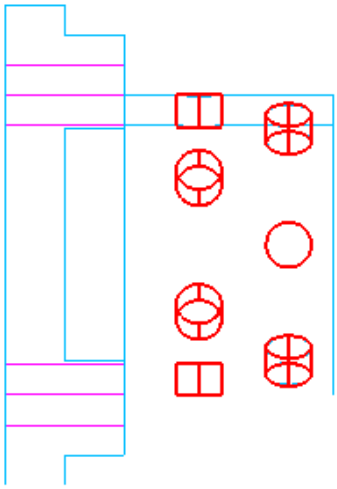
- Click on **Finish**, then open the properties of the hole feature, go to the **strategy tab** and **uncheck** the spotdrill as we are trying to save time and in this case we will not need to use a spotdrill. Click on **OK**

Note that if you decide to use a spotdrill you will need to select a smaller tool size that will not gouge with the OD of the part. Practice this if you are ahead of the class, otherwise just uncheck it from the strategy tab

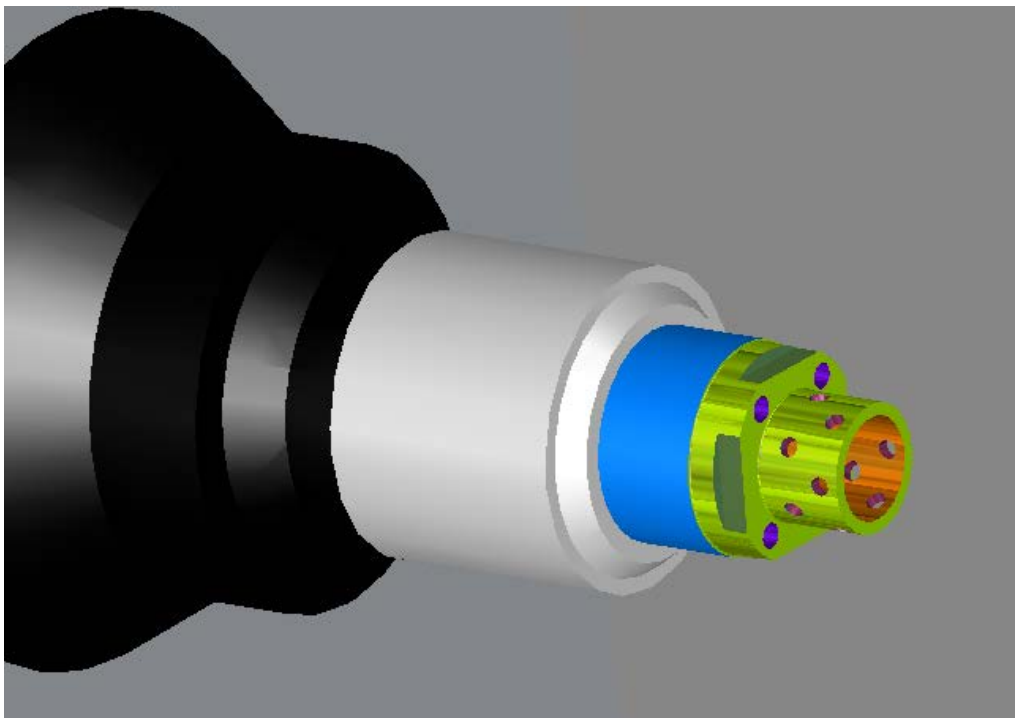


- Let's do the holes around the front diameter of the part aligned with the X axis. Go to the **new feature** wizard, select **turnmill**, **hole** from dimension and check the box **Extract with Feature RECOGNITION**
- Click **next**, select **Around the index axis,** check the **Automatic** box, **next**
- Continue through the wizard, **select all** and **finish**

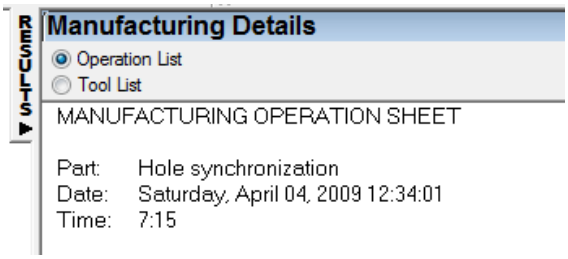




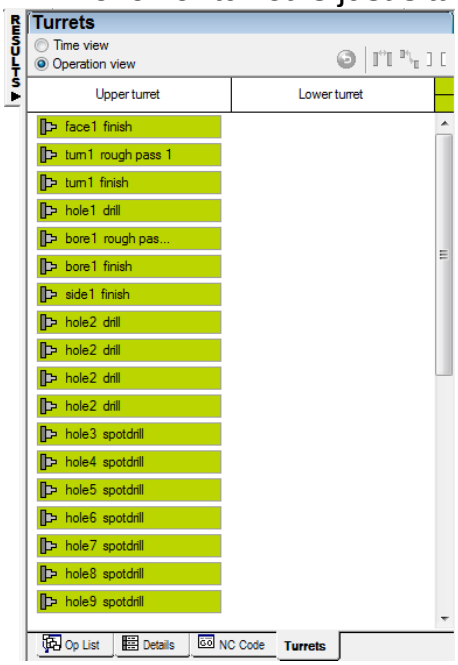
- Run **machine simulation** and look at the finished part

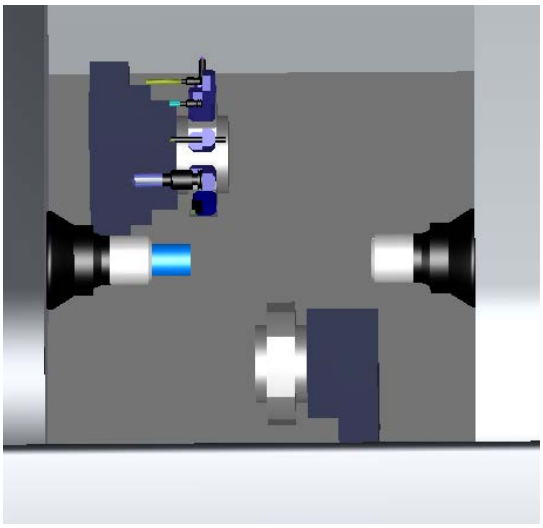


- Now **run centerline simulation** again and look at the **details** tab to see how long it takes for you to machine this part. My details tab looks like this. It takes 7:15 to machine this part. Time to play with synchronization!!!



- Notice that all operations are currently being assigned to the upper turret. The lower turret is just sitting there. Not being productive nor saving time



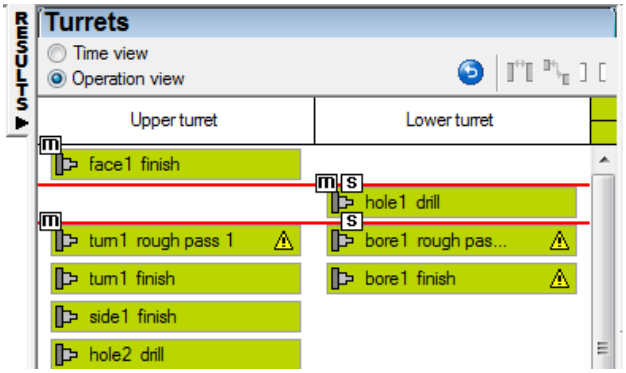


Synchronization tips and tricks:

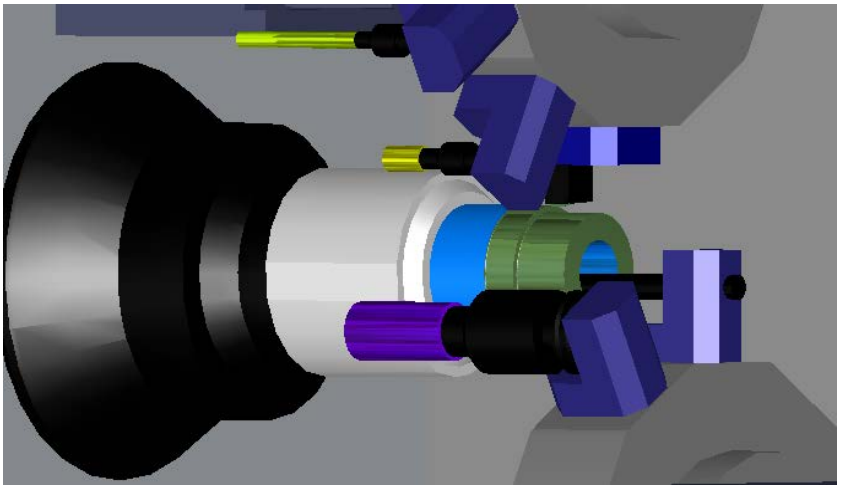
- If you make a mistake while moving operations from one turret to another and you need to start from the beginning you can do a right-click on a white area in the turret tab and select the appropriate option
  - Reset all operations to default turret
  - Reset selected operations to default turret
  - Reset all oper sync points
- Sometimes although you won't see any sync points on the screen, operations still believe they are synchronized to other operations. This usually happens when the user mistakenly syncs the same operation to 2 different operations. In order to clear these errors it is recommended that you do a right-click and **reset all operation sync points** and start synchronizing from the start
- If you select 2 operations and an option (like sync at the start or one before the other) but nothing happens when you set the sync point, this is due to the point above. **Remove all operation sync points** and you should be able to synchronize again
- You can customize the turrets view to have different colors for operations happening in the main and subspindle. Different colors aid the user identify what's happening in each spindle making simultaneous operations easier to identify

- It is a good idea to save your file before you start synching operations. The process of synchronizing operations can be quite complex and you might want to try different scenarios before you decide which one you like better

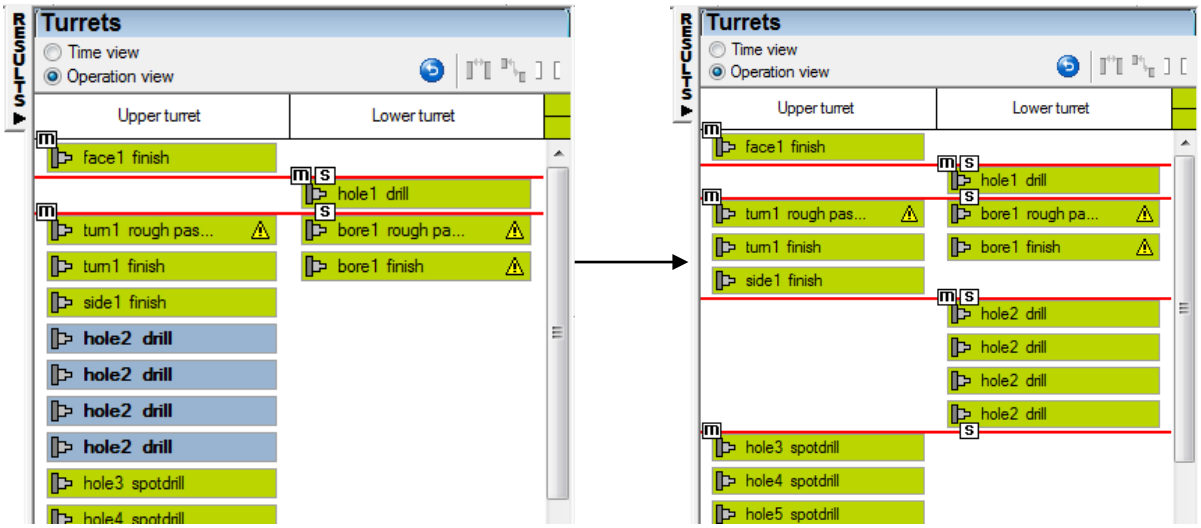
- **Move** the following operations to the **lower turret**. **If you wish**, fix the length of the new tools selected (boring bar will be a different tool orientation, so it will probably be a longer tool again)
    - Hole1 drill
    - Bore1 rough and finish
- Your turrets tab should look like the screengrab below. **Assign** the necessary **sync points** to achieve same results



- New simulation should look like the picture below. If you run **centerline** and look at the details tab you should be already saving some time. At this point, my details tab reads **6:13**, so I'm already saving a minute! Let's save even more time



- **Move** the following operations to the **lower turret**. To avoid the time it takes to do a tool change we will move the **4 holes in the Z axis** to the lower turret so that once the upper finishes the 4 flats we can start machining holes. **Assign** necessary **sync points** to achieve same results as screengrab below
    - Hole2 drill (4 times)



- Run **centerline** and look at the details tab you should be already saving even more time. At this point, my details tab reads **5:19**, so I'm already saving another minute!
- Can we save more time? **YES!**

If we look at the model you will find that the holes around the OD of the part are exactly 180 degrees apart from each other. This particular machine has its turrets 180 degrees apart as well. So what if we attempt to drill one of the holes in the upper turret and at the same time also drill the opposing hole 180 degrees apart from it? Can we do this in FeatureCAM? Yes, we can!

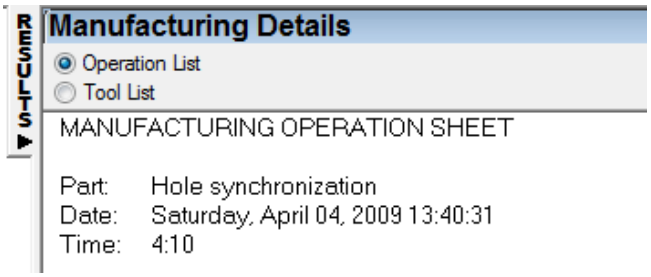
- **Move** the following operations to the **lower turret**. There're 12 holes in this part. You should have 6 spotdrill and 6 drill operations in each turret. Make sure you move consecutive holes otherwise the holes will not be 180 degrees apart.

- Move:
  - **Hole9 through 14 spotdrill** (6 times – you can select them all holding the Ctrl Key and then drag the group**)**
  - **Hole9 through 14 drill** (6 times**)**
  - **Set** the necessary **sync points** like in the screengrab below

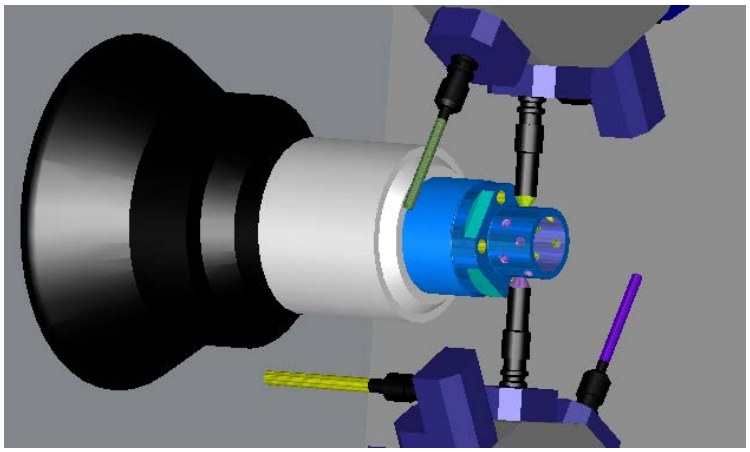
Notice that although you **could** just sync the first 2 operations (hole3 spotdrill and hole9 spotdrill) and leave the rest **without** setting synchronization points, this is **not** a safe approach, as you need to be completely sure that every matching hole is finished before the spindle turns to machine the next hole.

Not setting sync points for every set of matching holes could result in **broken tools** and **damage to the machine** and the **part**. For this exercise synchronize all matching operations at the operation start

Run **centerline** once again and look at the details tab. After the latest set of synchronization points my details tab reads **4:10**. By using FeatureCAM's synchronization capability I went from **7:15** to **4:10**, saving over 3 minutes per part.



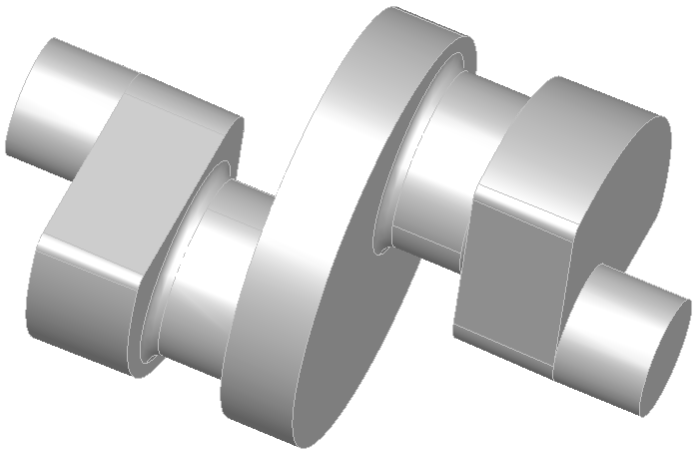
- Final simulation results



5-axis Simultaneous

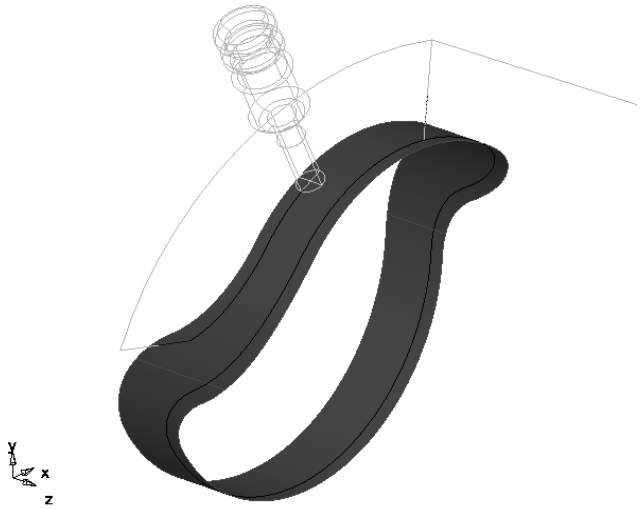
Introduction

There are two new strategies available for finishing using 5-axis simultaneous machining in FeatureCAM 2008. These two new techniques are called **5-axis Trim** and **5-axis swarf**.



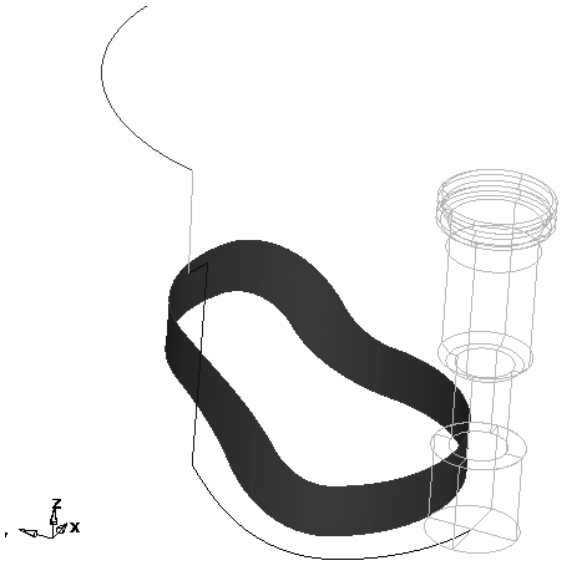
In **5-Axis Trim:**

- This strategy is used commonly in the mold industry where cast parts have a parting line edge that needs to be removed.
- Tool will cut normal to selected machining surface with selected lead and lean angles



In **5-Axis Swarf:**

- Swarf Cutting calculates toolpaths which will cut with the side of the tool.
- The side surfaces are selected and the side of the cutter will stay tangent to the surface



Crankshaft Example

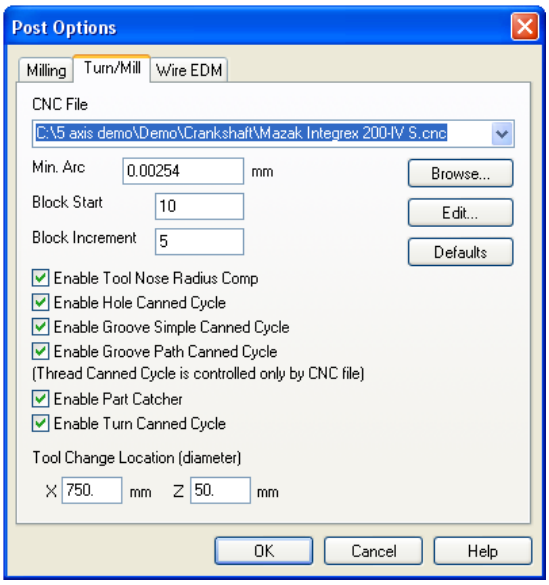
In this example part we will cut a crankshaft using the 5-axis trim and swarf strategies.

- Click on **File** then **Open**. Browse and open **CrankshaftTaper.fm**
- Click on **Manufacturing** -> **Setups** then press the edit button. Press **Next 3 times** and check "**Always use this one**" then browse for the machine design file named **integrex200.md** and press **Finish** and click **Close**.

To ensure this demo runs quickly change default graphics settings to:

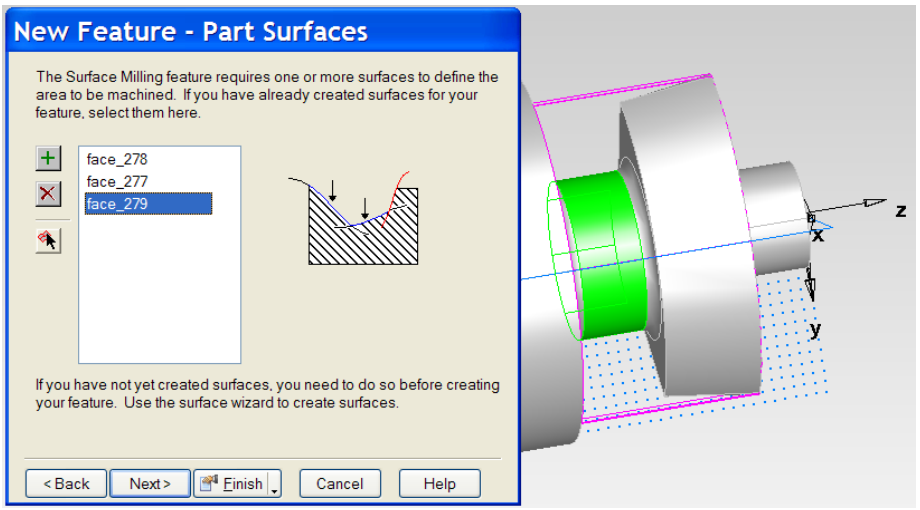
- Click on **Options** -> **Surface Shading**. Make sure use graphics hardware is checked
- Click on **Options** -> **Simulation**. On the **2D/3D shaded** tab make sure **Metallic** is **unchecked** and set **Resolution** to **4.**

- Click on **Manufacturing** menu then choose **Post Process**. Set the post processor to **Mazak Integrex 200-IV S.cnc**, located in the **Data** folder.
- Change the tool location for **X** to **750** and **50** for **Z**, then click **OK**.



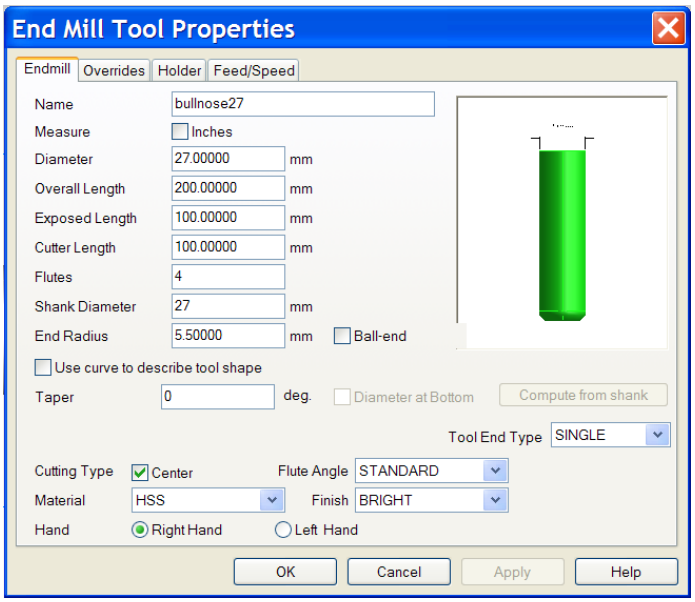
- Click on the **Manufacturing** menu and choose **Set Tool Crib**, then select **CrankshaftTaper.fm_Tools_from_last_save** and click **OK**.

- From the **Steps** click on the **New Feature** wizard, choose **Turn/Mill** and press **Next.** Select **Surface Milling** and press **Next**. Select the three part surfaces in the picture below and press **Next**.



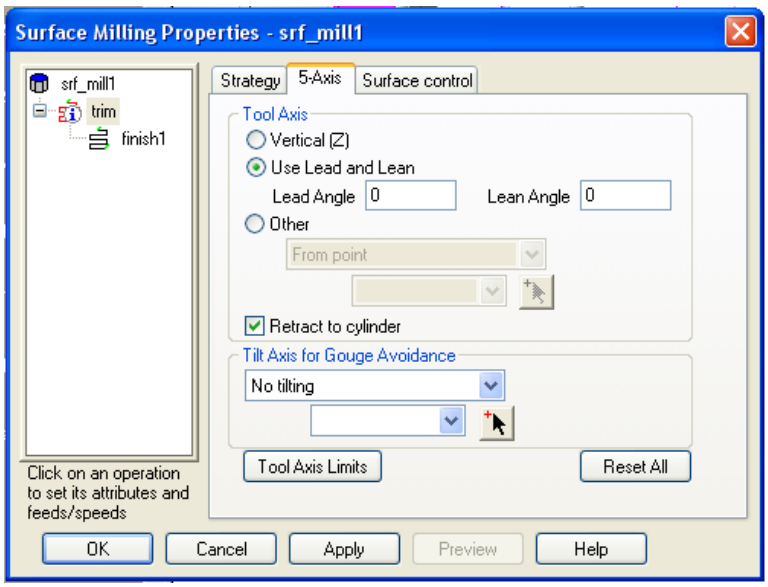
- Click **Choose a single operation** then press **Next**. Select **5-Axis Trim** and press **Next** 3 times, then select **I want to search for another tool or make a new one** and click **Next.**
- Click the **create new tool button**

- Enter the dimension of the new tool as shown below

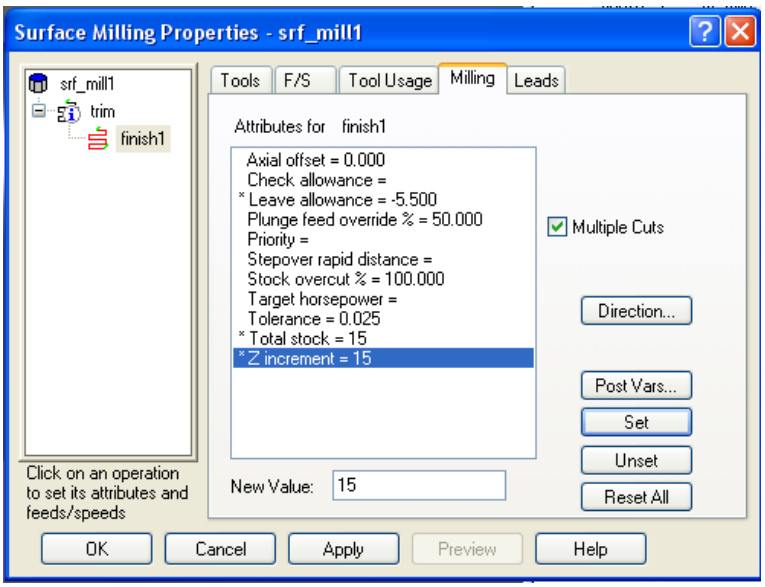


- Press **OK,** select **Create a new tool** and press **OK.** then click **Yes** to set the new tool as an override, then press **Finish**

- In the Feature properties click on the **trim** op and then click on the **5-Axis tab.** Select **Use Lead and Lean** angle and set both to **0**.

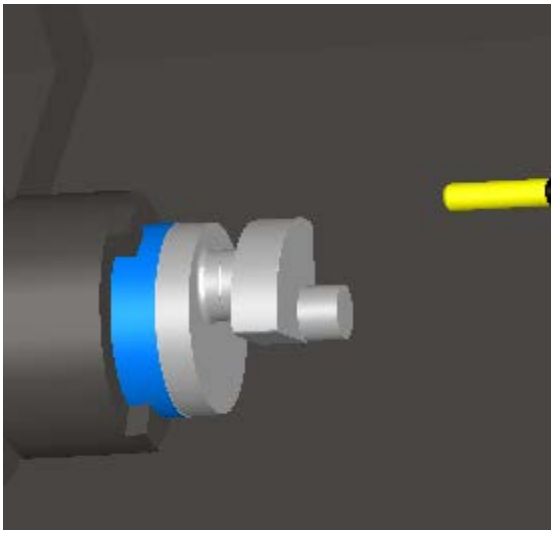


- On the **Milling** tab of the finish op click on **Multiple Cuts**, set the **Leave Allowance** to **-5.5**, **Total stock** to **15**, **Zincrement** to **15** and then press **OK**

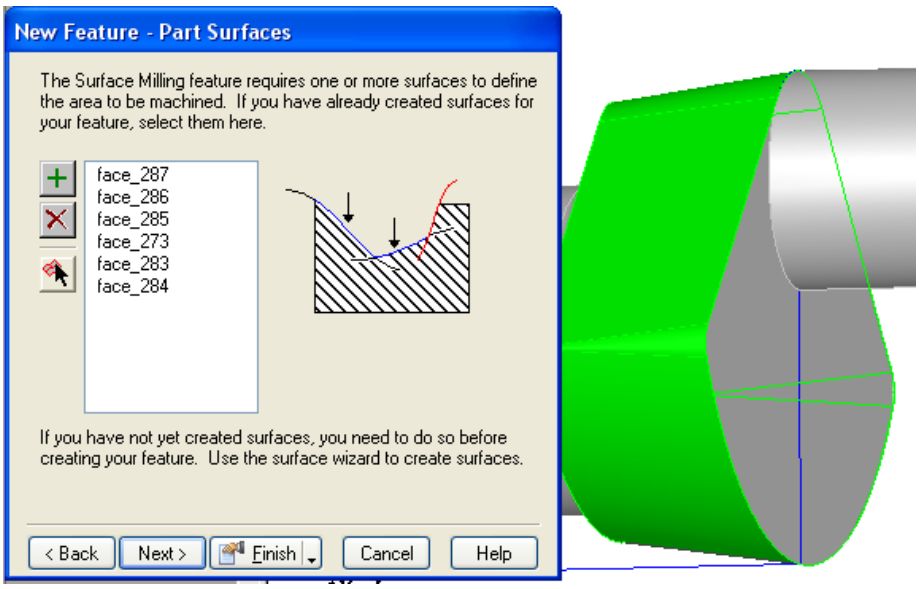


- Drag the surface milling feature just created before ss1 in the **Part View.**

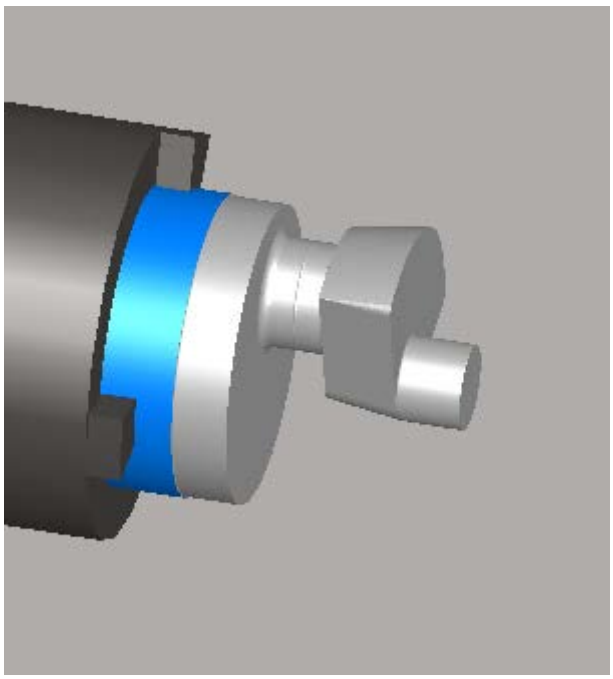
- Run **Machine Simulation** and you should see the results as shown below.



- Create a new Surface Milling Feature and select the surfaces shown below on the tapered boss surface.

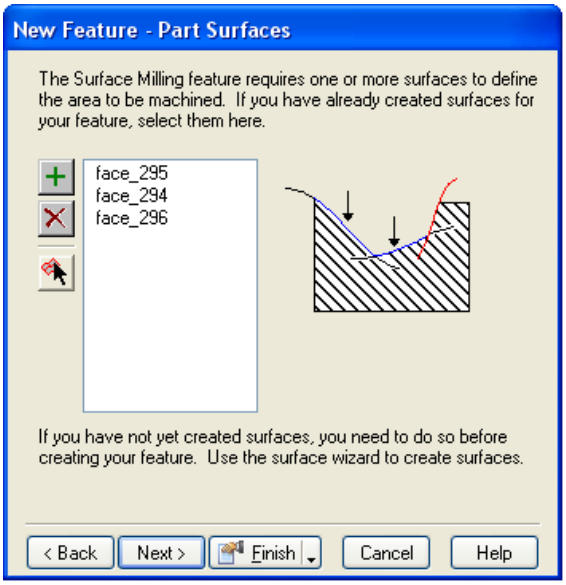


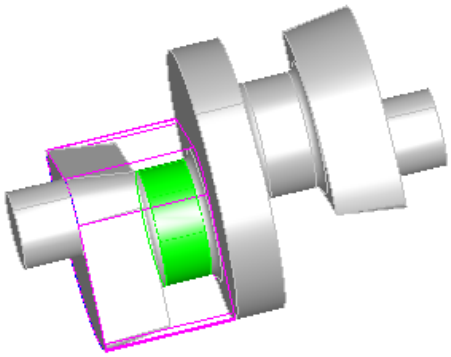
- Press **Next** then select choose **single operation** press next then select **Swarf.** Press **Next** 3 times, then select **I want to search for another tool or make a new one** and click **Next.**
- Select the bull nose tool that was created in the previous surface milling feature and then press **Finish** and click **OK**.
- Drag the surface milling feature just created before **ss1**.
- Run a Machine simulation. Results should look similar to those shown below



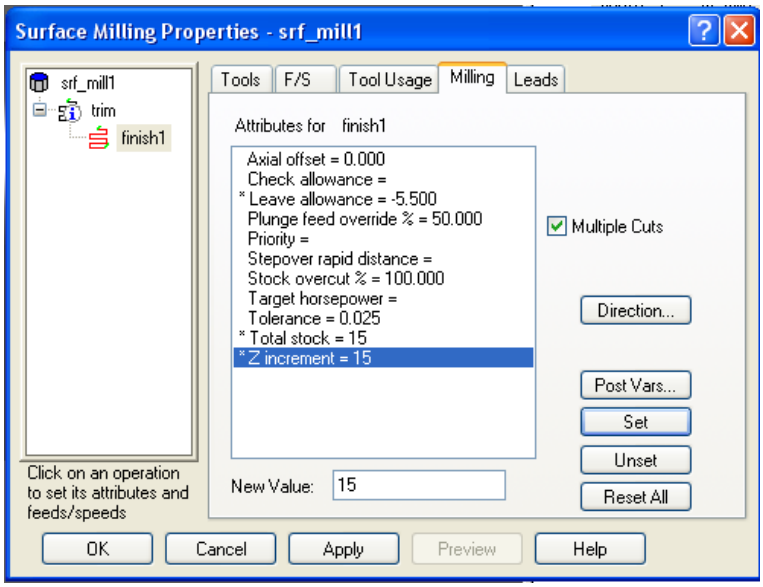
- Now the features are ready to be cut on the sub spindle. In the **Part View** click on the **Setup2** to make it active.

- Click on **Steps** then **New Feature** wizard, select **Turn/Mill** then press **Next**. Select **Surface Milling** then **Next**. Select the three part surfaces in the picture below and press **Next**.

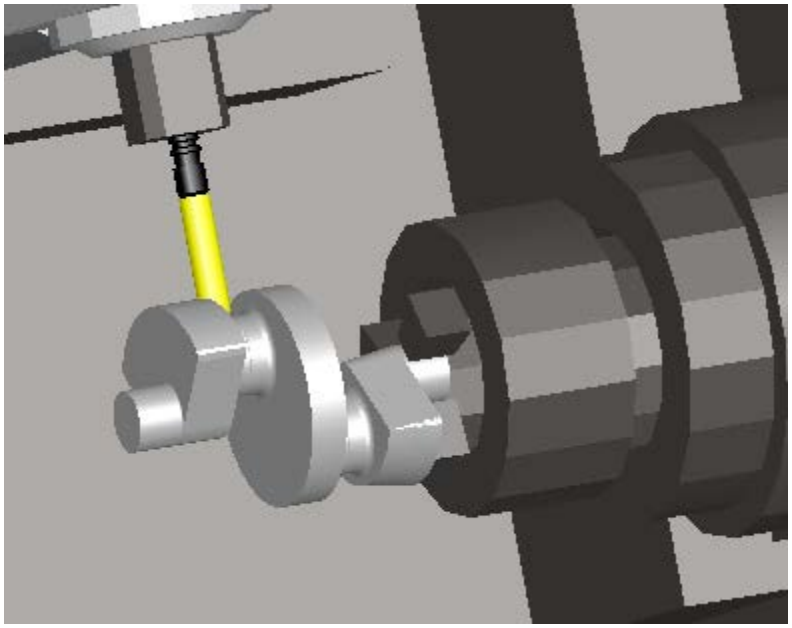




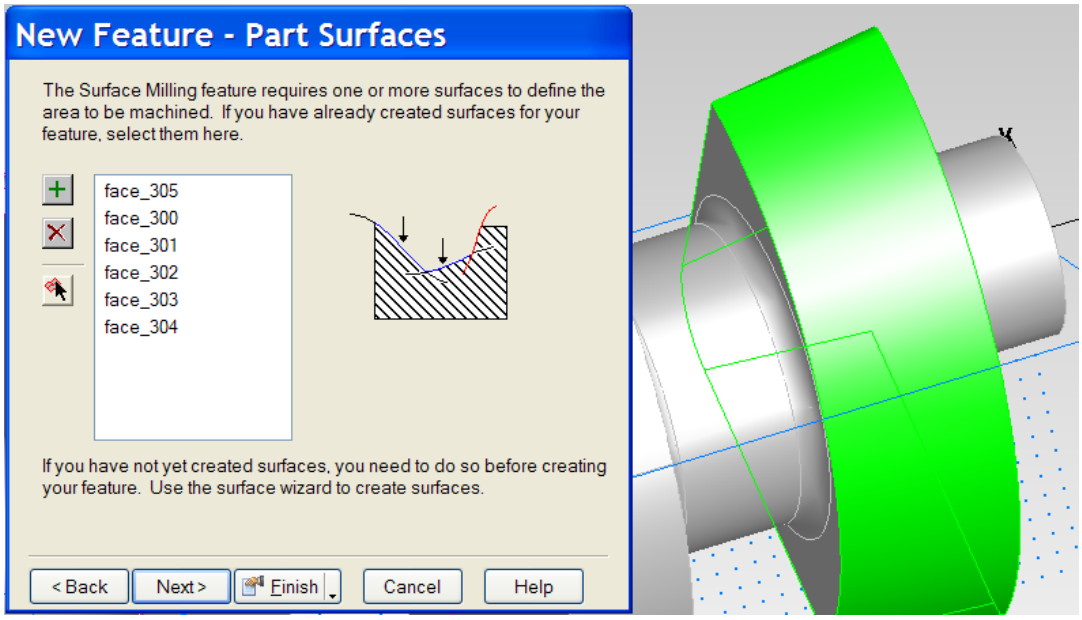
- Click **Choose single operation** press **Next**. Select **5-Axis Trim** and press **Next** 4 times. Select the 27 mm bull nose end mill created previously and press **Finish**
- In the Feature properties click on the **trim** op and then click on the **5-axis tab.** Select **Use Lead and Lean** angle and set both to **0**.
- On the **Milling** tab of the finish op click on **Multiple Cuts**, set the **Leave allowance** to **-5.5**, **Total stock** to **15**, **Z increment** to **15** and then press **OK**



- Run the machine simulation and you should see the results as shown below.

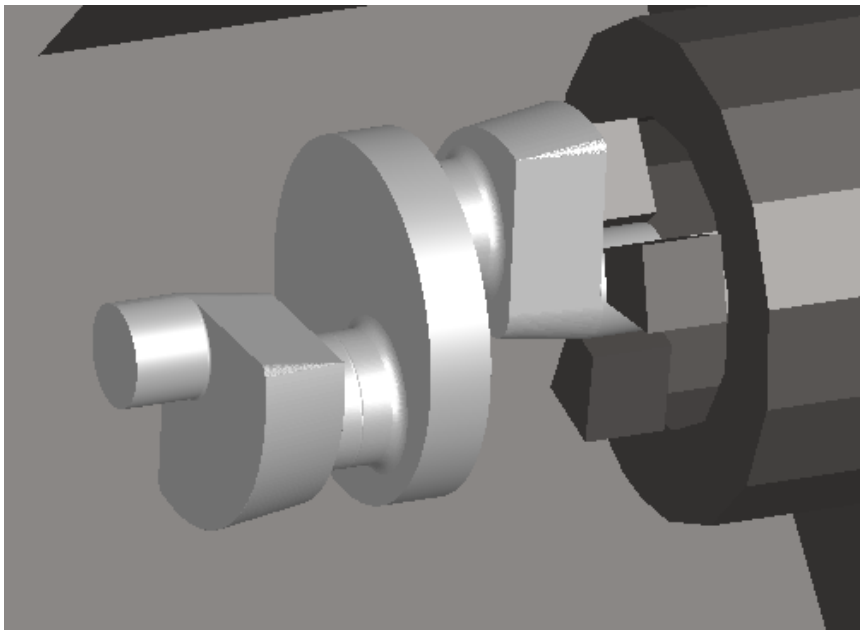


- Create a new Surface Milling feature and select the surfaces shown below on the tapered boss surface.



- Press **Next** and select **Choose single operation,** press **Next** and select **Swarf**. Press next 4 times and select the bull nose tool that was created in a previous step then press **Finish** and click **OK**.

- Run the Machine simulation. Results should look similar to below.



B-Axis Interactive Feature Recognition

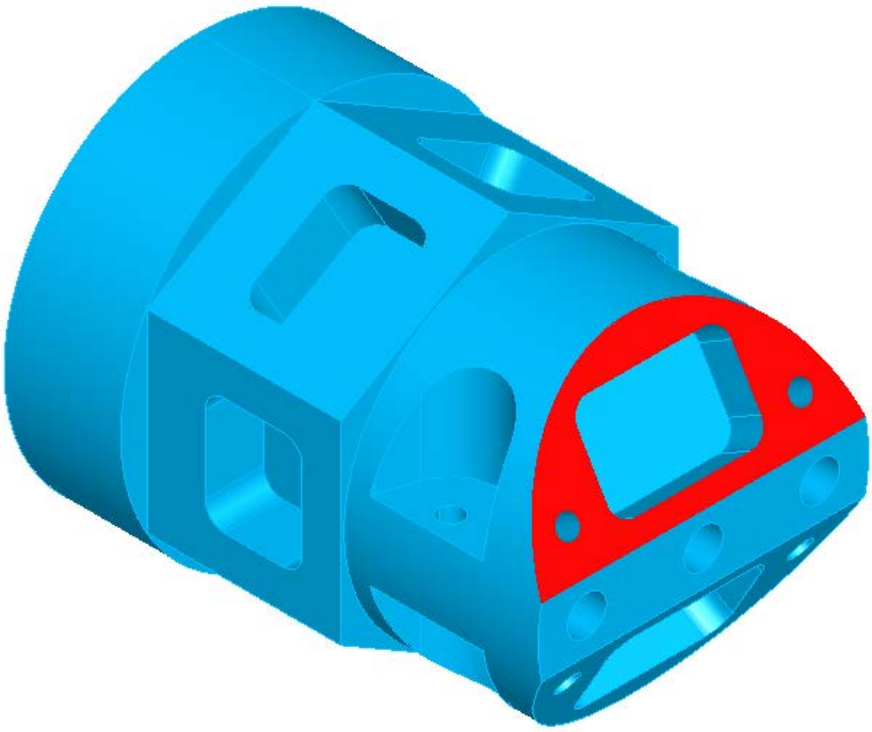
In this example we will be looking at various ways of identifying features from solids when using a B-Axis Turn/Mill. In previous versions of FeatureCAM it was necessary to create a setup with its Z axis aligned with the direction the tool would approach the feature before it could be recognized. It is now possible to identify features with multiple orientations in a single setup simply by selecting a surface whose normal is parallel to the direction that the tool would use to machine the feature.

- Open the file **B-Axis tutorial start.fm**
- Load the **Mori Seiki NT4250.CNC** file
- Ensure that the post is set to use the matching MD file
- Create a **Turn feature** from the curve **curve1** using the default values
- Create a **Face feature** using the default values

Note that the Face feature is automatically ordered so that is before the turn feature. We will now machine the two angled faces on the end of the part. We will use IFR to identify the faces using their surface normals.

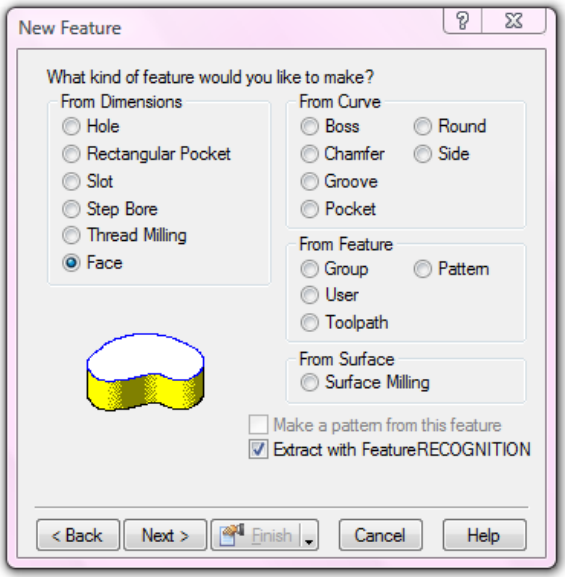
- Take an **Isometric view** and **Shade** the model

We are going to create a Face feature on the surface shown below.

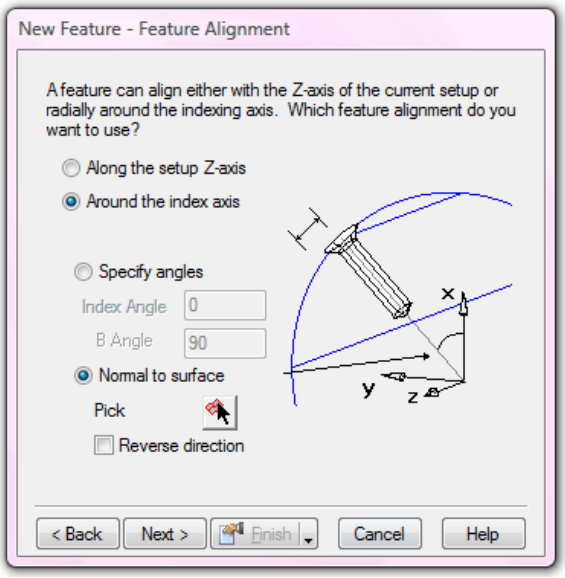


- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature

- Click **Next**



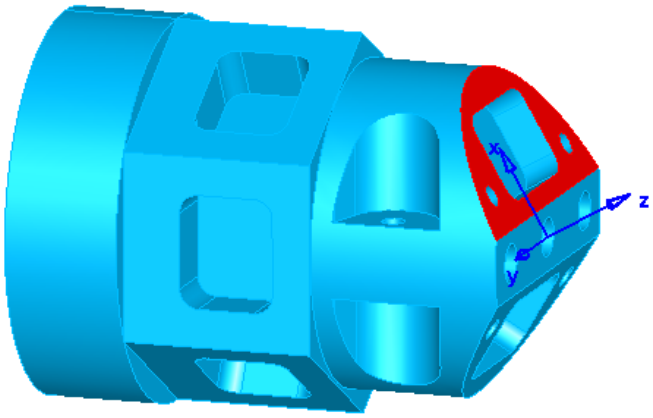
- Check **Face** and **Extract with FeatureRECOGNITION**.
- Click Next

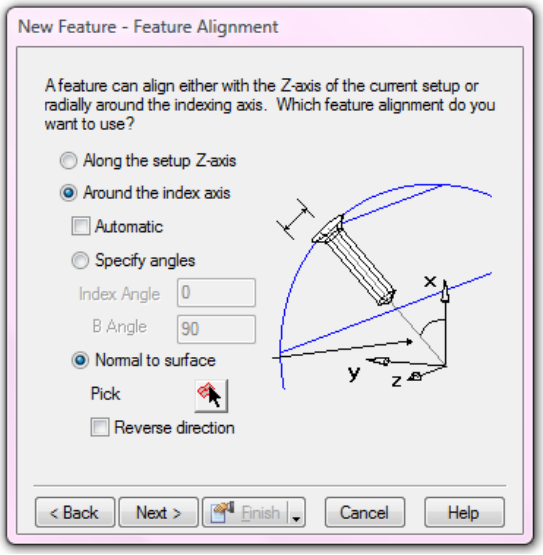


At this point we have to specify the axis along which we are going to recognize features. We could use **Specify angles** and then explicitly enter the direction normal to the face but we would need to know both the indexing C angle and also the B angle of the head. With a face with a complex orientation it can be difficult to work out the angles required. However, if we use **Normal to surface** and then pick a face FeatureCAM will automatically work out the combination of C and B angles and find the face.

- Check **Normal to surface**
- Click on **Next**

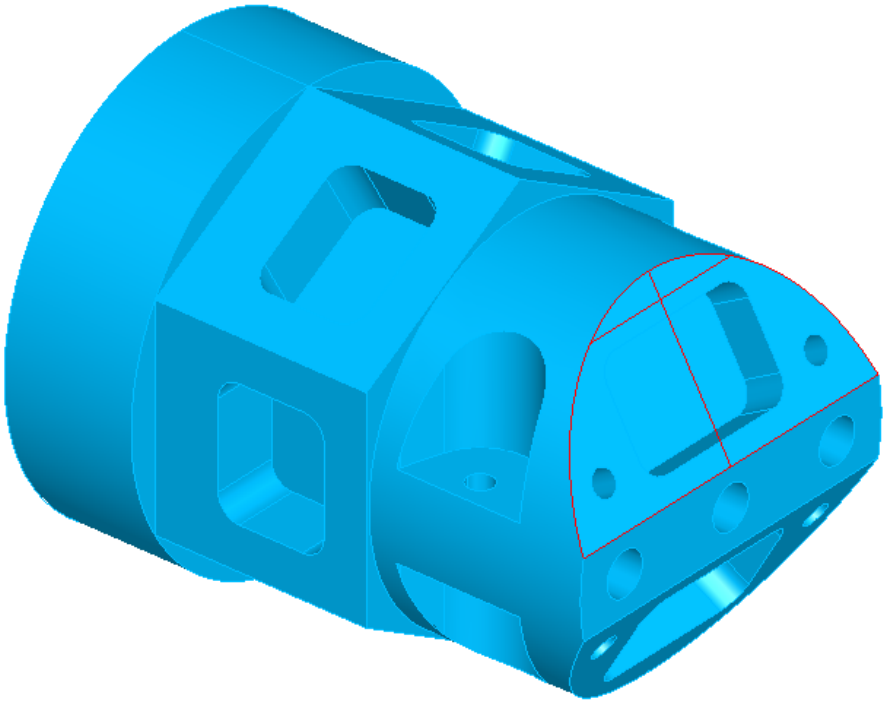
- Click on the **Pick surface** button and then select **face_510**





The Z axis for feature recognition has automatically been aligned with the normal of the surface. This is not a setup, it is a temporary axis that is just used for feature recognition and is then discarded afterwards.

- Click **Next**
- Click **Add from selected items** (the green plus on the upper left)
- Click **Next** and **Finish** and then **OK**

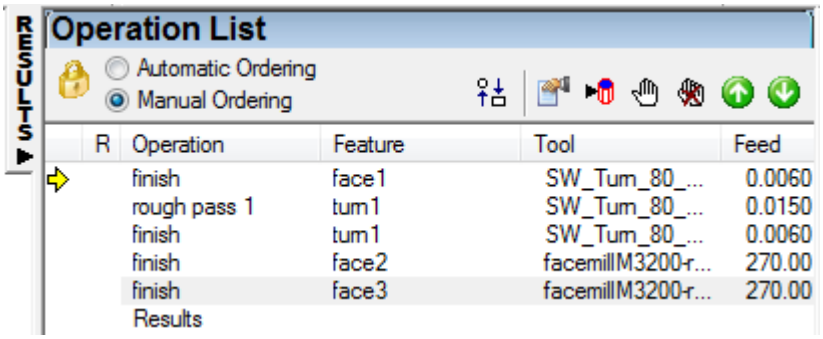


The face feature has been automatically recognised.

- **Repeat the process** to machine the face on the opposite side (**face_511**)

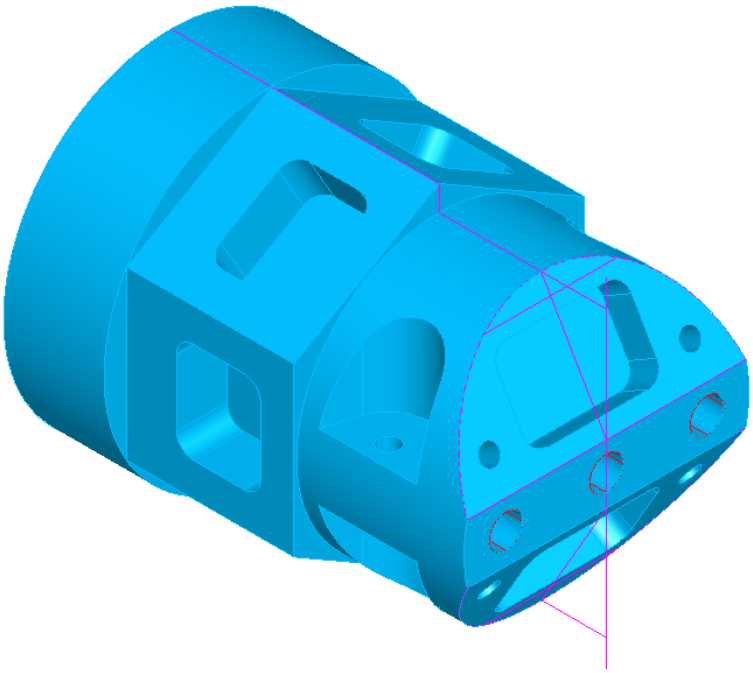
As we are creating face features they are automatically ordered so that they are machined first. We actually want to do the initial facing operation, then the turning followed by the two angled faces.

- In the **Ops view** drag the operations into the correct order as shown



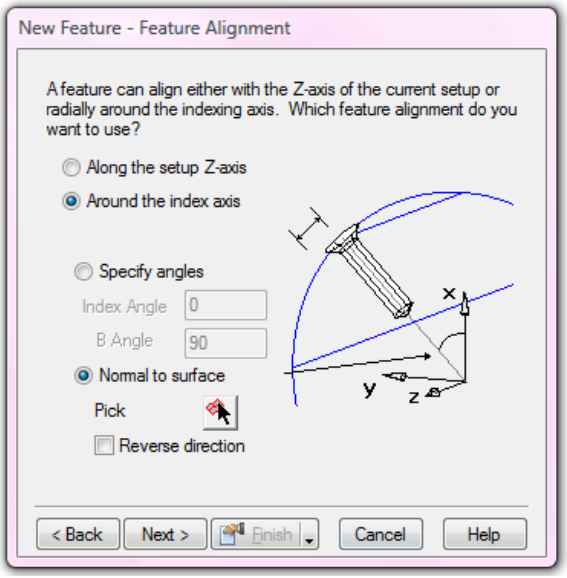
Next we will drill the holes on the face of the part; these can be identified using IFR along the setup Z axis.

- Create a new **Turn/Mill feature**
- Select **Hole** and **Extract using FeatureRECOGNITION**
- Click **Next** and select **Along the setup Z axis**
- Click **Next** twice until you reach the **Hole recognition options** page
- Click **Select all** and then **Finish**

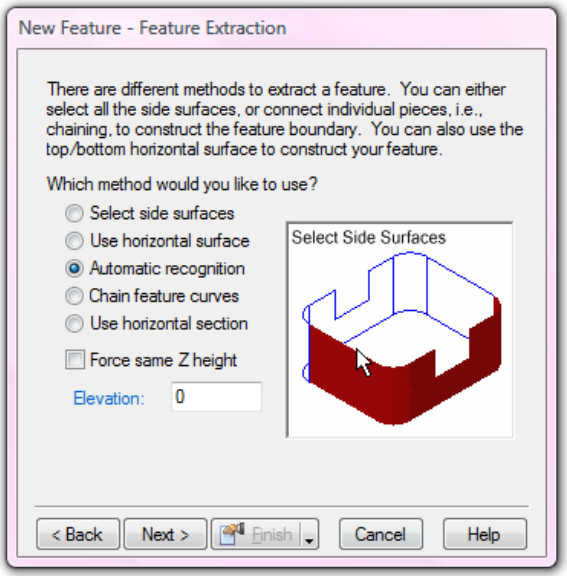


You should now have three holes on the front face of the part. We will now machine the two pockets on the angled faces.

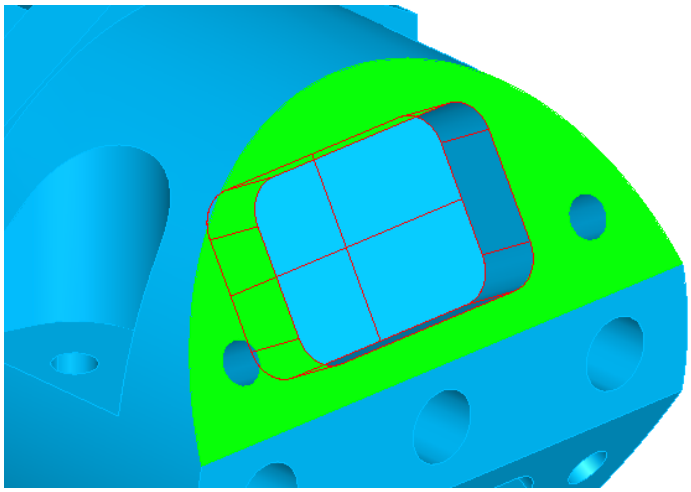
- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Check **From curve - Pocket** and **Extract with FeatureRECOGNITION**
- Click **Next**
- Check **Around index axis**
- Check **Normal to surface**



- Click on the **Pick surface** button and then select **face_510**
- Click on **Next**
- Select **Automatic recognition**



- Click on **Next**
- Click **Select all**
- Click **Finish**

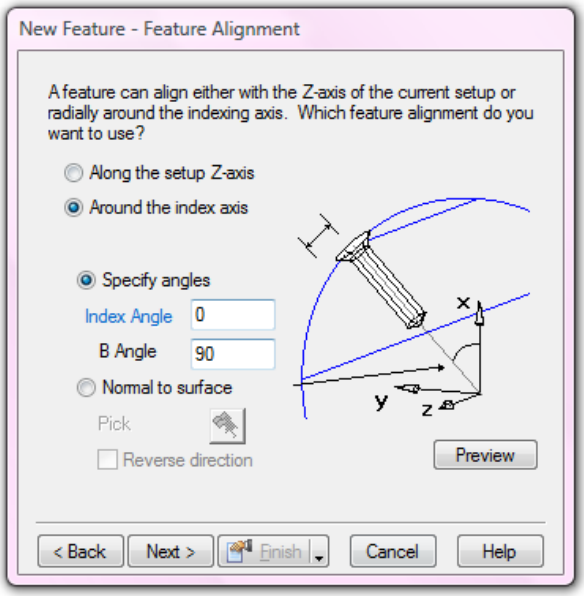


You should now see a pocket feature as shown above holes on the angled face using IFR.

- Repeat the procedure to machine the pocket on the opposite face using face_511 to identify the Z direction for feature recognition

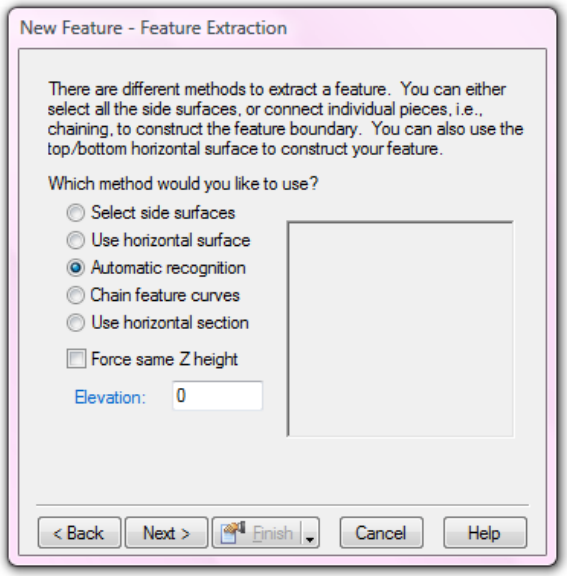
Next we will machine the 4 curved side features on the upper and lower sides of the part. As these are in a simple orientation (C 0 and C1800 we can use a specified index angle for recognition.

- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Check **From curve - Side** and **Extract with FeatureRECOGNITION**
- Click **Finish**
- Check **Around index axis**
- Check **Specify Angles** and set 0 and 90 as shown

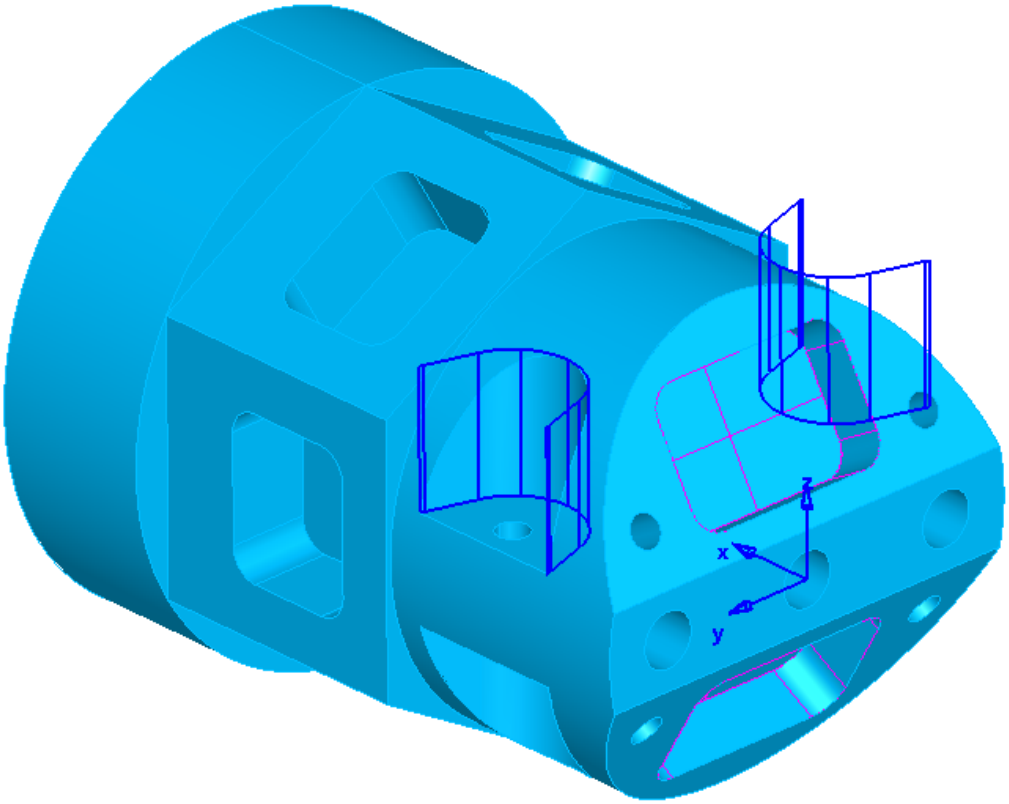


- Click **Next**

- Check **Automatic recognition**



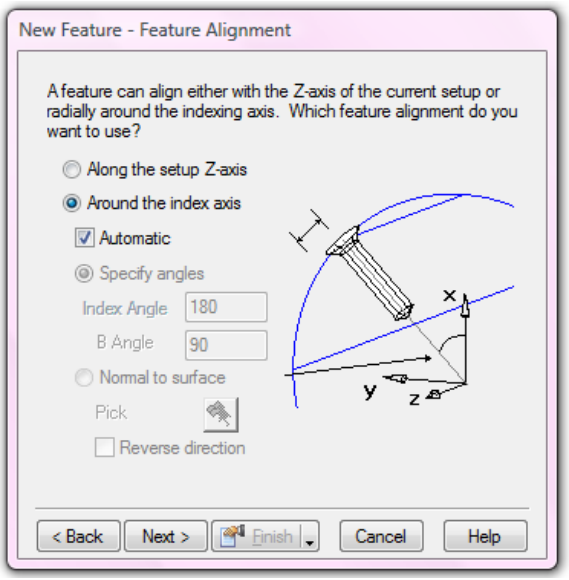
The two side features have been recognized. We just need to select them to lock them in.



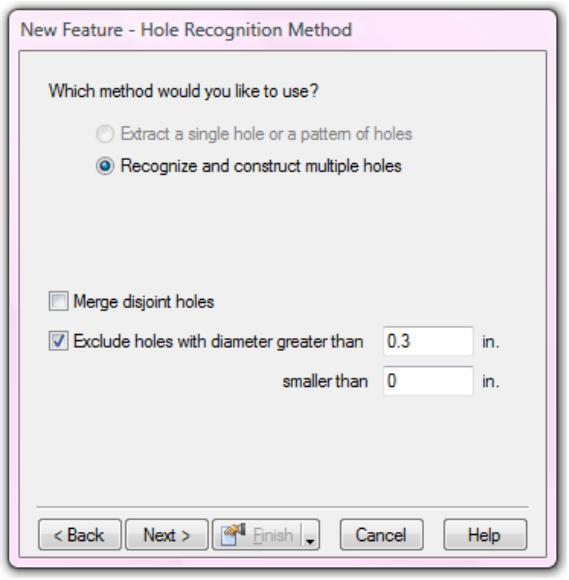
- Click **Select all** and then **Finish**
- Repeat for the opposite face (index angle 180)

We will now recognize the remaining holes which are not aligned with the setup Z axis.

- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Check **Hole** and **Extract with FeatureRECOGNITION**
- Click **Finish**.
- Check **Around index axis** and **Automatic**



- Click **Next**
- Check **Exclude holes…** and set the diameter to **0.3in**.



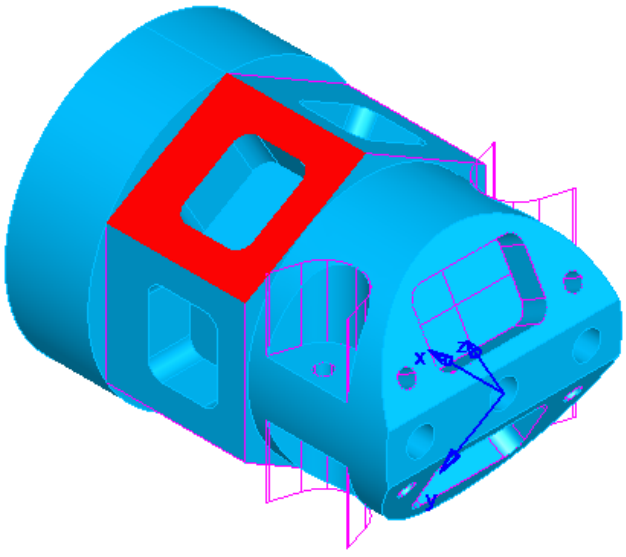
- Click **Next**
- Click **Select all**
- Click **Finish**

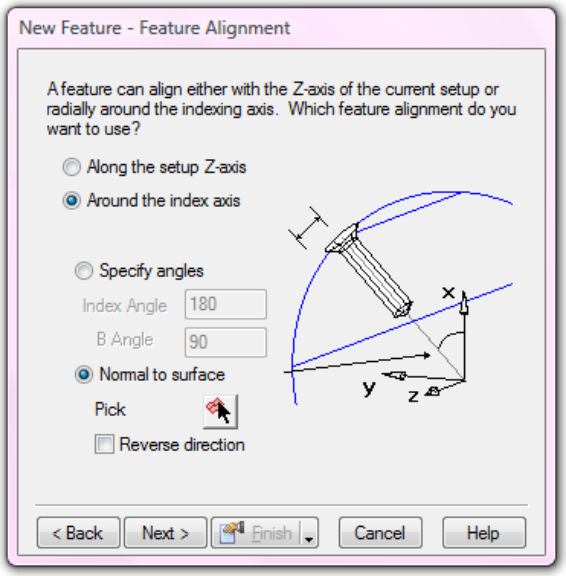
We will now make a side feature to machine the hexagon.

- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Check **Side** and **Extract with FeatureRECOGNITION**
- Click **Next**
- Check **Along setup Z axis**
- Click **Next**
- Check **Automatic recognition**
- Click **Next**
- Click **Select all** and then click on the round side feature to deselect it
- Click **Finish**

Finally we will make the pattern of pockets on the faces of the hexagon.

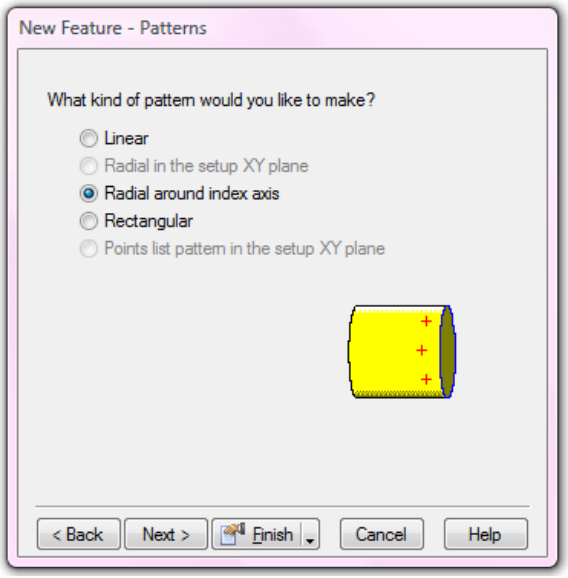
- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Check **From curve - Pocket** and **Extract with FeatureRECOGNITION**
- Click **Finish**
- Check **Around index axis**
- Check **Normal to surface**
- Click on the **Pick surface** button and then select **face_395**



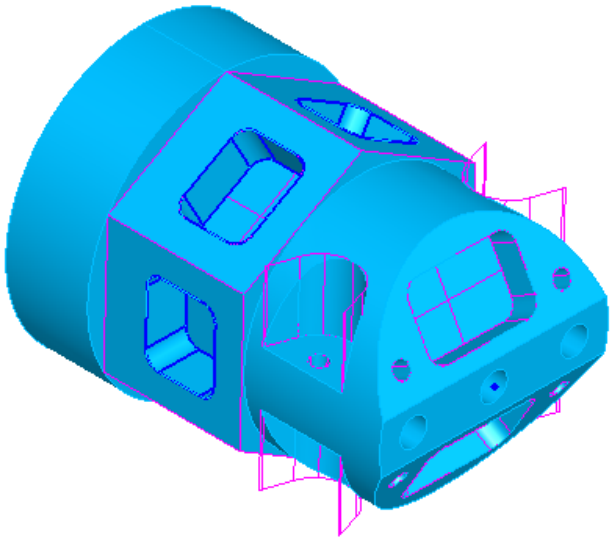


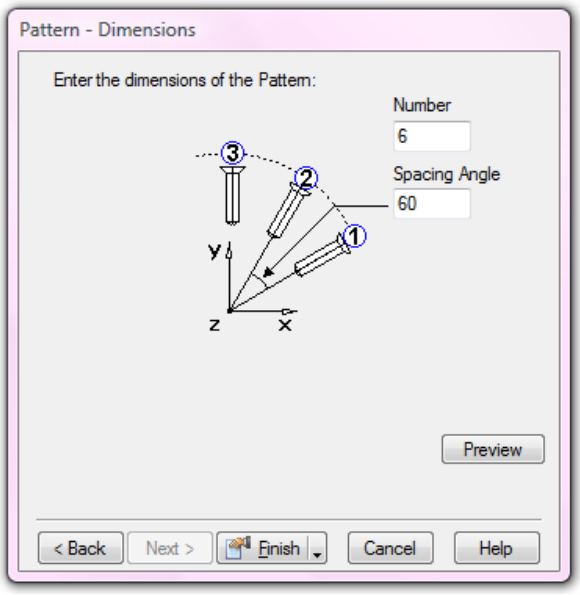
- Click **Next**
- Select **Automatic recognition**

- Click **Select all** and then **Finish**
- Open the **New feature wizard**
- Under **New Feature Type** select a new **Turn/Mill** feature
- Click **Next**
- Select **Pattern** and click **Next**
- Make sure that **pocket3** is selected then click **Next**
- Check **Radial about the index axis** and click **Next**



- Enter a **Number** of **6** and a **Spacing angle** of **60 degrees**



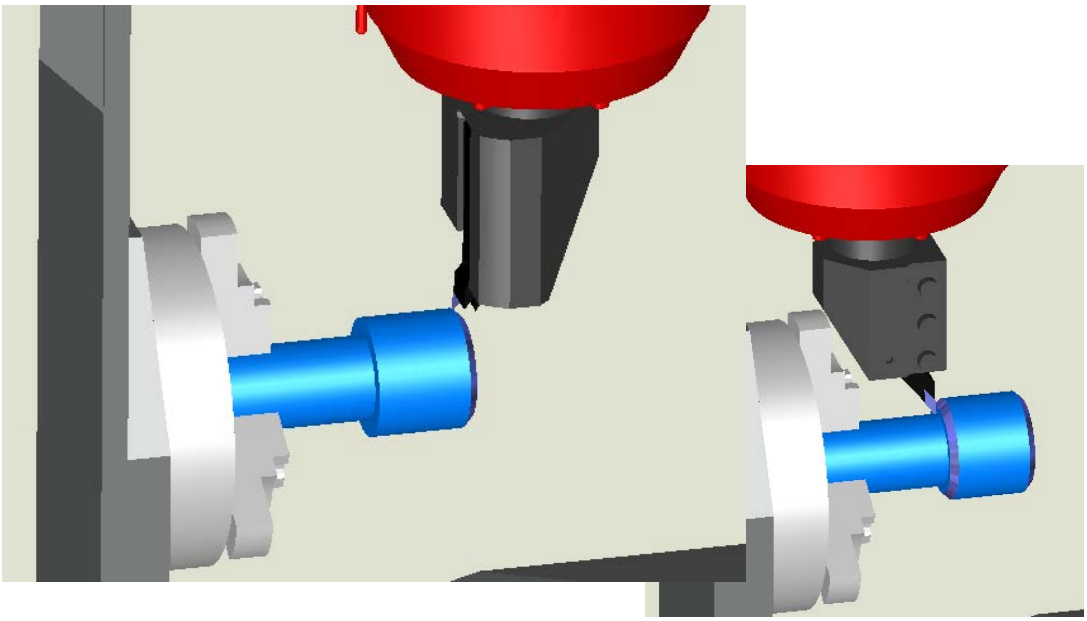


- Click **Finish**
- Run a **Machine simulation** to see the results

B-Axis Chamfers

Introduction

This chapter will take you through the necessary steps to orient a single tool so that it can cut several features that would require multiple tool changes and different tool orientations. This is especially handy for B-axis machines, since tool changes can take a long time and the programmer needs to maximize tool utilization in order to reduce cycle time.

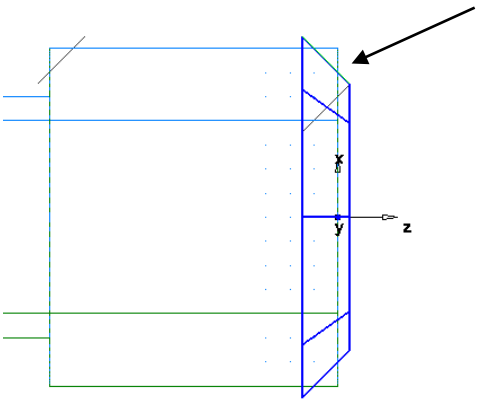


- **Open** file **B-axis chamfers.fm**
- For this tutorial we will use the **Mori NT4250.cnc** post file

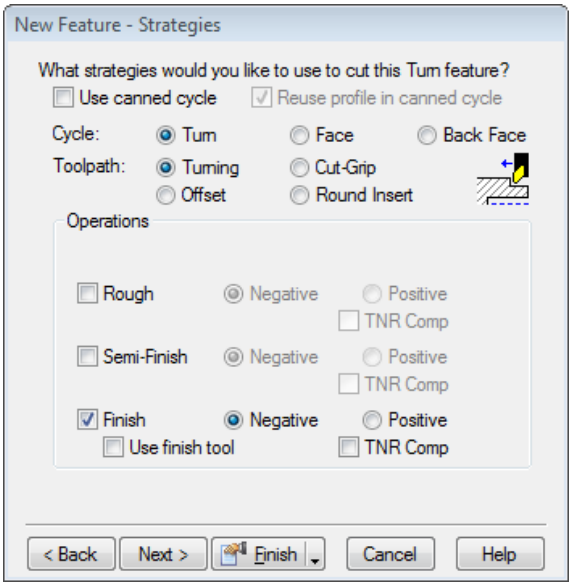
This file uses a **solid model** as a **stock**. The solid model has **no chamfers** on the corners, but we've created some geometry that we will use while creating features. In a non-B-axis machine you would have to pick **3 different tools** to machine the chamfers (they all require different tool orientations). Can we do this with just one tool? Stay tuned…

- Start by creating our first turning feature for the OD chamfer in front of the part. Go to **New Feature**, click on **Turning**, then **Turn from Curve**

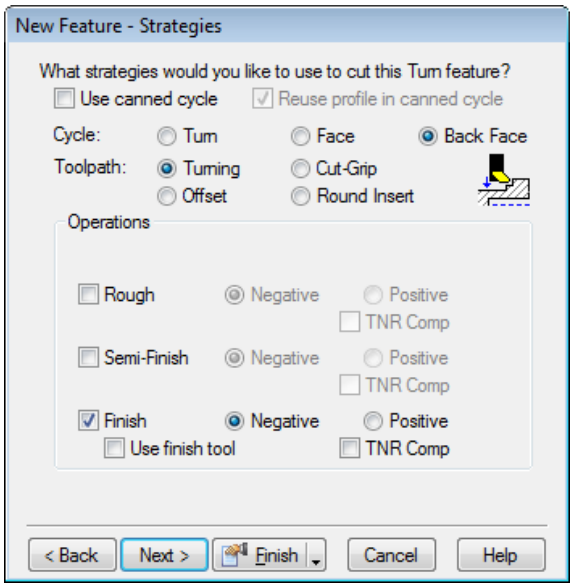
When prompted to select a curve, pick the line segment highlighted in picture to the right.

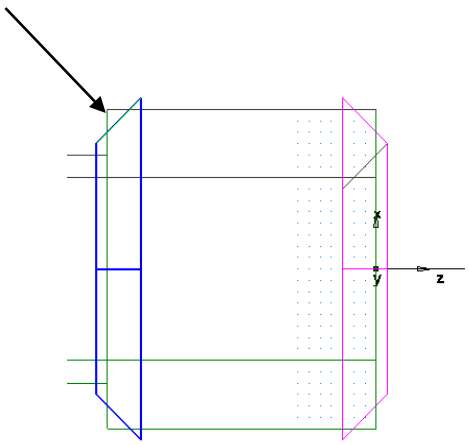


- In the **Strategy tab**, **uncheck** the **rough** operation and make sure that the Finish direction is set to **Negative**



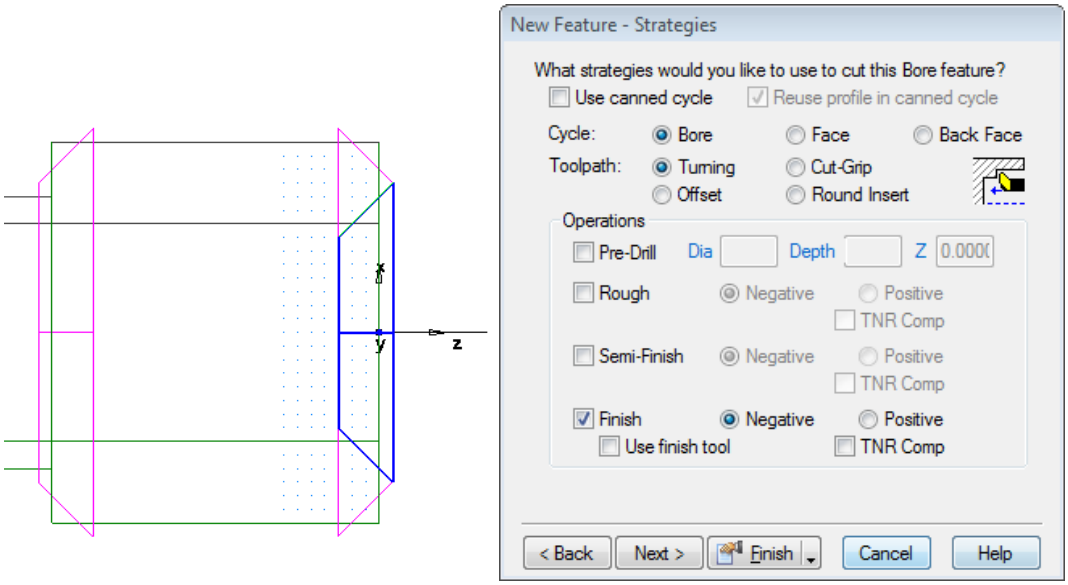
- Let's make another Turn. In this case, pick the **OD chamfer** in the **back** of the part. In the **strategy** tab, select **Back Face**. Like with the previous feature, **uncheck** the **rough**, set the **direction** to **Negative**



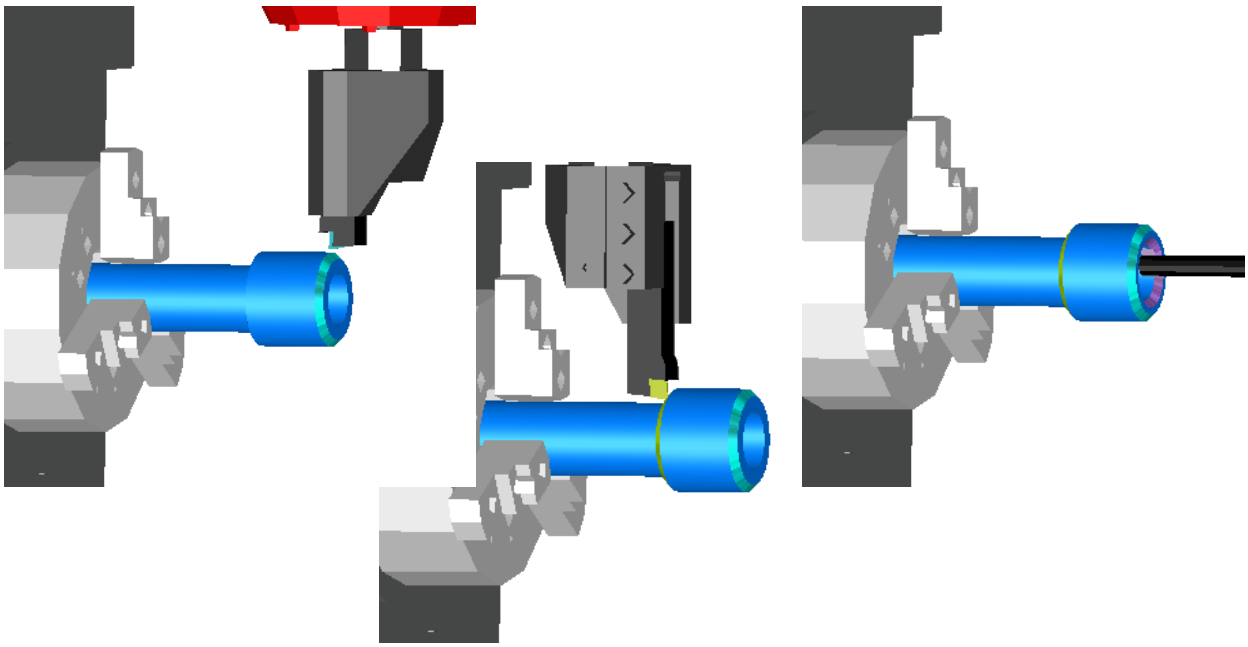


- Let's now cut the last chamfer in the ID of the part. Create a **Bore feature**, set the **cycle** to **Bore**, **uncheck** the **rough** and set the **direction** to **Negative**

For this feature you could also select a Turn > Face or a Turn > Turn approach. Some of these strategies will require you to enter additional parameters into the software (change engage angle, different tool orientations, etc). We are creating the outlined features so that we can show you benefits of using the B-axis orientation control.

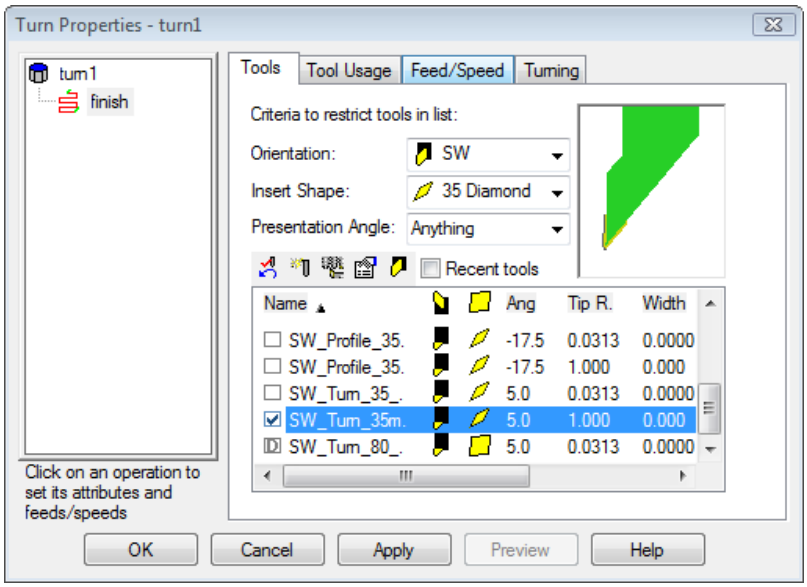


- Now run **Machine simulation** and notice how the software picks **3 tools** to machine the **3 features**, as every feature requires a different tool orientation



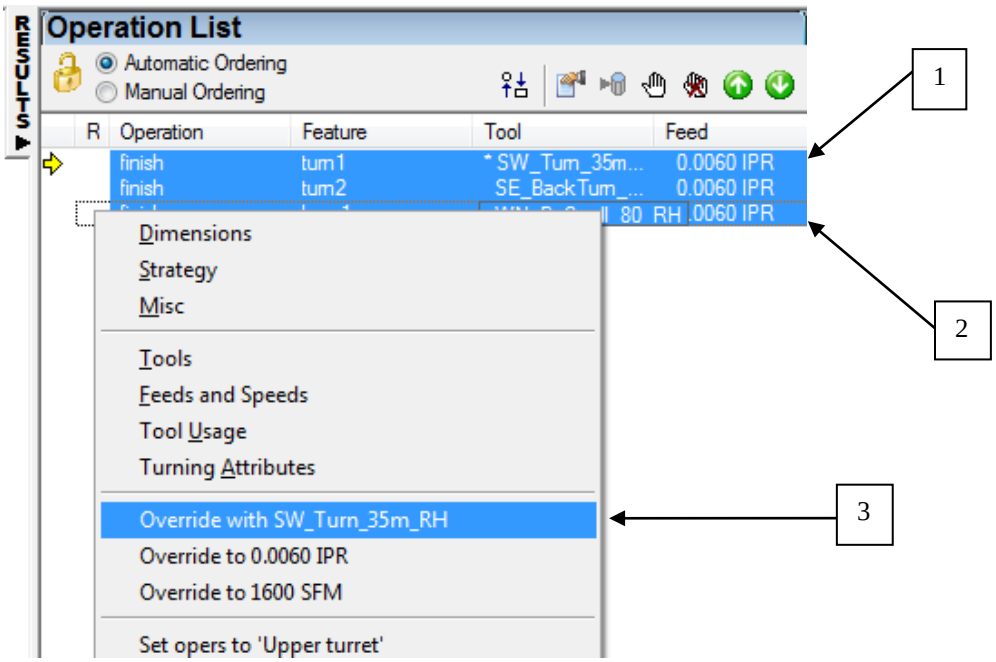
With a non-B-axis machine you would have no other choice than to use 3 tools. The problem with this is that you have to go through a tool change after every feature, adding to your total machining time. With a B-axis machine, although you would still retract after every operation, you don't have to pick a different tool (carrousel or index turret). You just re-orient the B-axis and move on.

- So, let's pick just one tool for all operations. Go to the **first** turn feature and, click on the **finish operation** and then on the **Tools tab**
- Filter tools by **Insert Shape** and select a **SW_Turn_35 degree diamond**

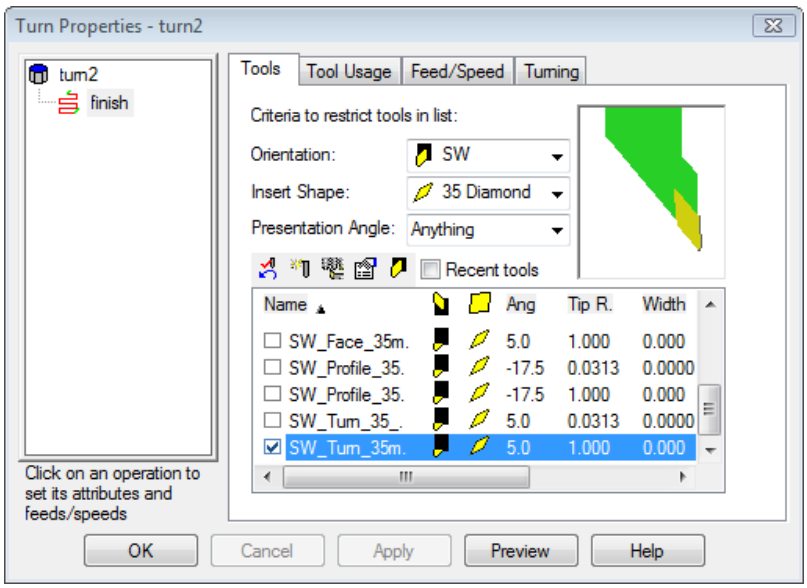


- Once we've assigned the tool we would like to use for all operations, let's override ALL operations to use the same tool. There're multiple ways to do this. The easiest one is to do it in the **Op list**

    - **Select** the first operation
    - **Hold Shift Key** and select the last operation
    - **Right-click** on the last operation
    - Select **Override with SW_Turn35**
    - Click on **Yes** when you get the warning about overriding operations with the same tool



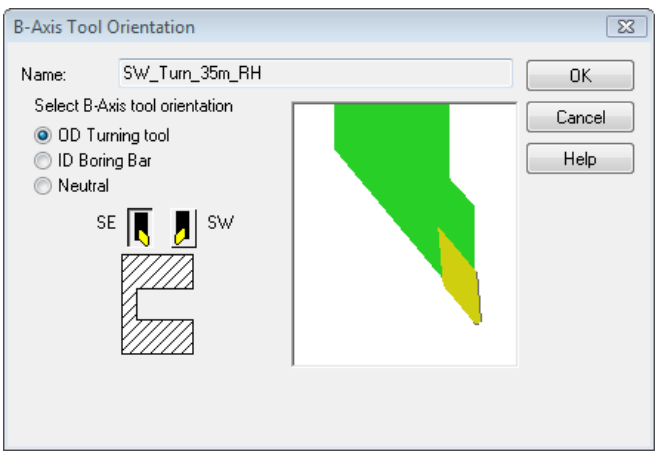
- Now go to the **tool properties** tab of the **second turn** feature and notice how FeatureCAM **automatically** rotated the tool 180 degrees about its axis so that it can achieve the necessary orientation for the feature



So how do we do this? Can we manually control this rotation if we want to? The answer is yes. When you **select a B-axis post**, FeatureCAM will activate a new icon in your Tools tab. The **Set B-axis tool orientation** icon

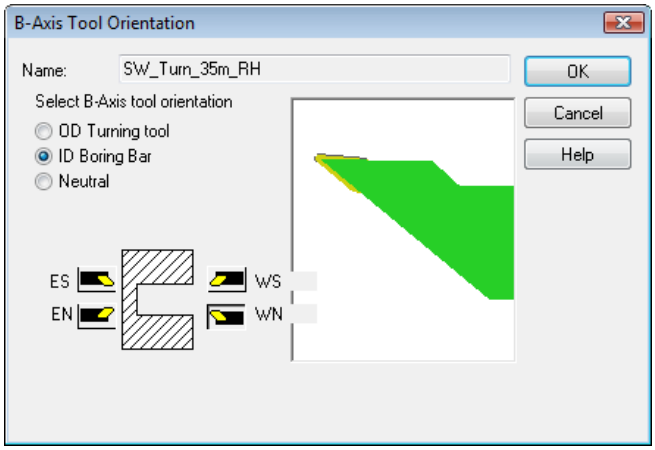
<u>There are 3 available options within the dialog:</u>

- **OD Turning tool:** allows the user to set a tool as a SE or SW (NE or NW if lower turret)
- **ID Boring Bar:** allows the user to select a boring bar approach angle (ES, EN, WS, WN)
- **Neutral:** allows the user to enter angles for the tools
  - B-axis angle: head angle
  - Tool rotation angle: to rotate the tool about the center axis (i.e. this would make a SE tool out of a SW tool)

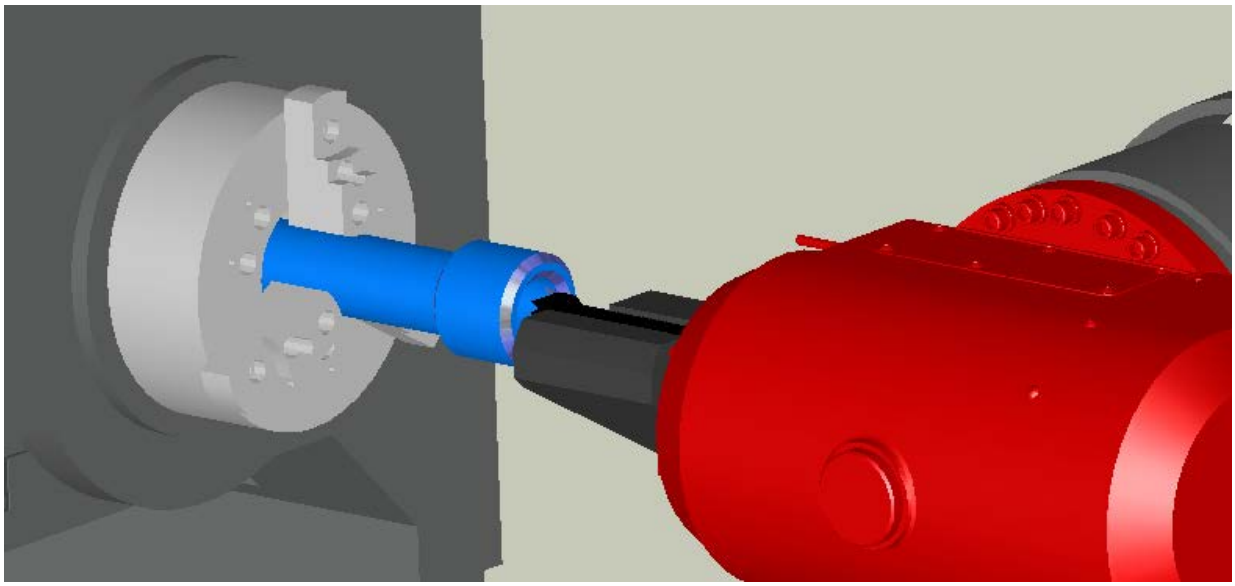


If you click on the **Set B-axis tool orientation** icon in this feature, you will see that FeatureCAM automatically **rotated** the tool from **SW** to **SE** so that it could be used for the Back Face turning feature

- Now go to the **tool properties** tab of the **last** turn feature. Click on the Set B-axis tool orientation button and set the tool as an ID Boring Bar with a WN angle



Click **OK**, **Apply** and **OK** again. Your final simulation results should be similar to the picture below



.

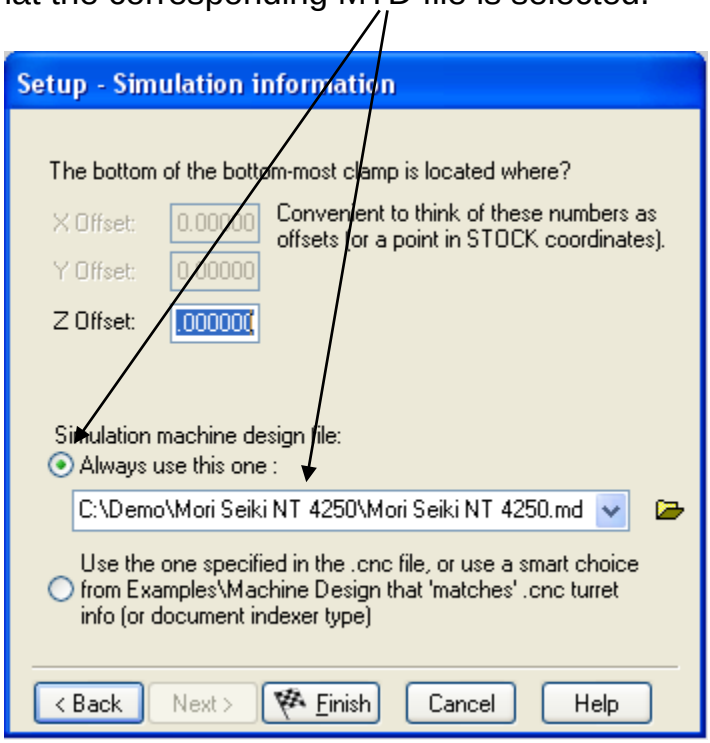
Turn/Mill Demonstration

Introduction

This demonstration is intended to give the customer an overview of FeatureCAM Turn/Mill.

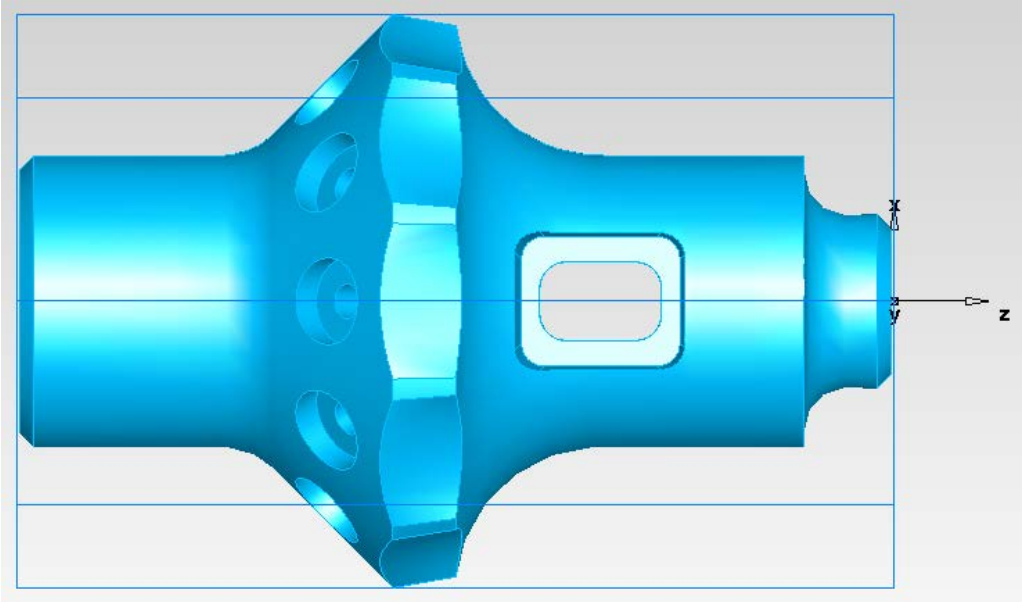
Before the Demonstration

- Open the **Turn/Mill** document **Demo Start.fm**
- Select the **Tools** tool crib.
- From **Options > Addins** select the addin **subspindle.bas**
- Make sure that the **Mori Seiki NT 4250 post** has been selected.
- **Double click Setup1**, click **Edit**.
- Click **Next** until you reach the **Simulation information page**.
- Check that the corresponding MTD file is selected.



Start of Demonstration

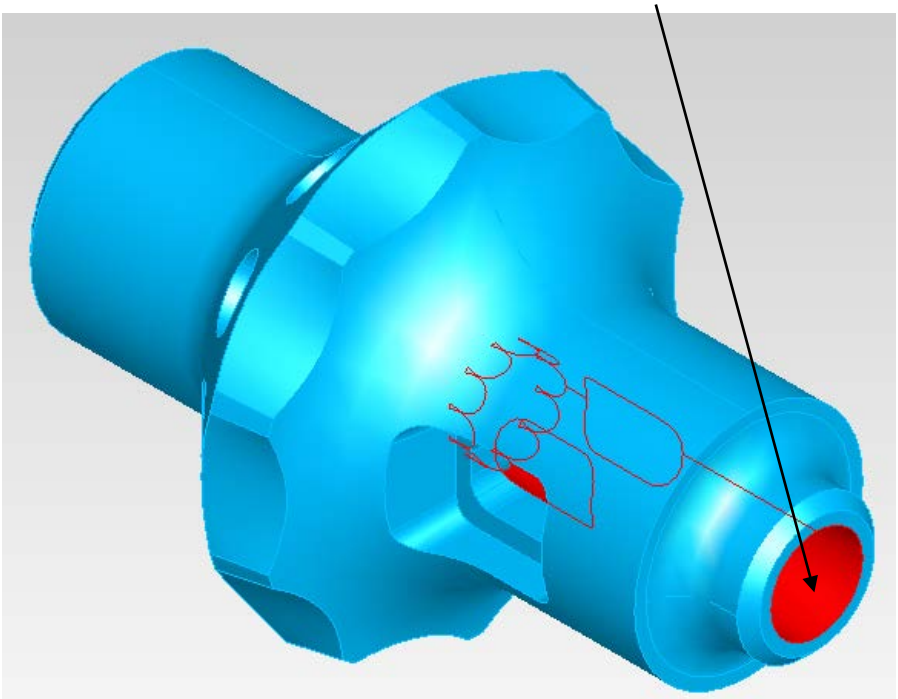
- Select a **Top view** and **shade** the part.



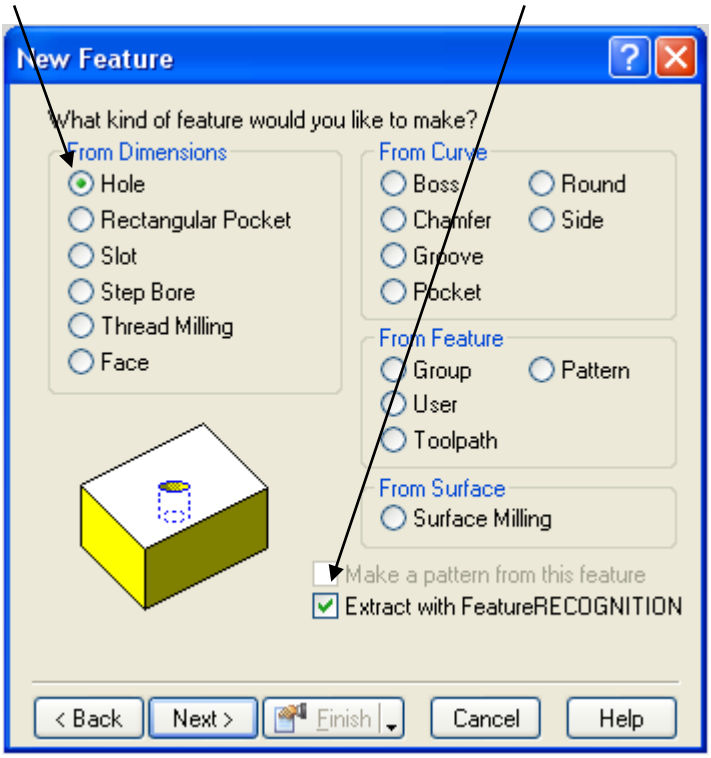
The stock, part and setup should appear as above.

The first two features are to be a facing operation followed by a drilling operation.

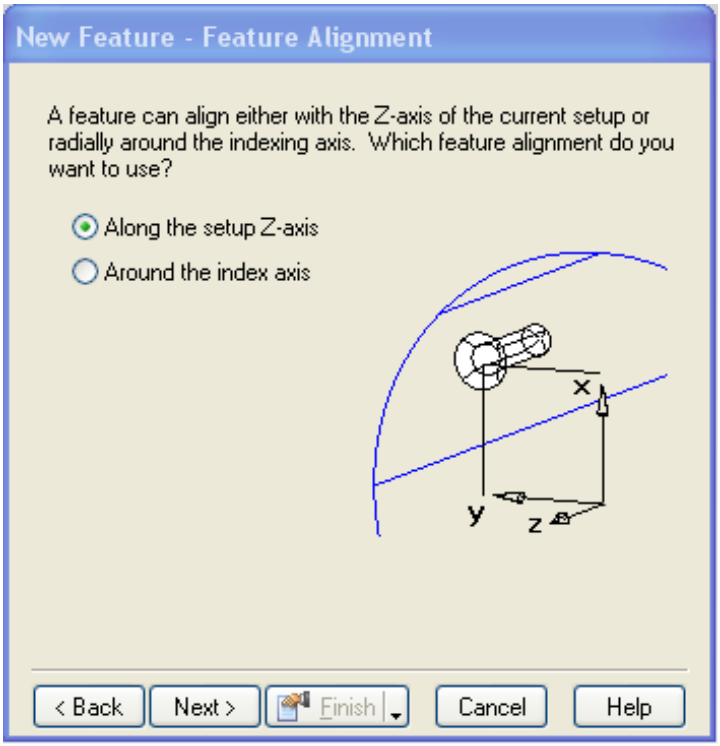
- Create a new **Turning** feature.
- Check **Face** and click next to the **Location form**.
- Set the **location** to **Z - 0.05**.
- **Hide all** then **show all solids**.
- Select and **isometric view** and **pick** the **inside surface** of the hole.



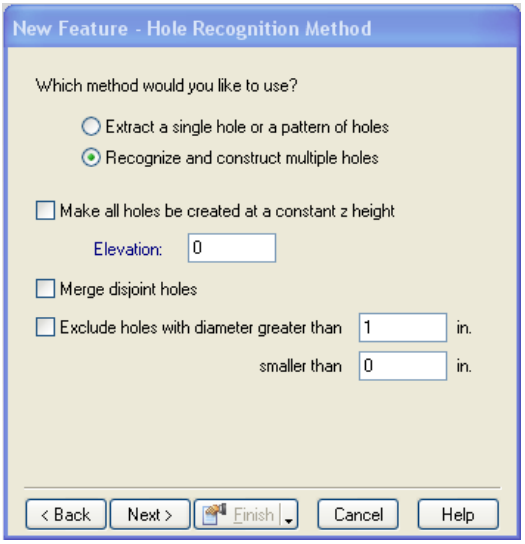
- Create a new **Turn/mill feature**.
- Check **Hole** and **Extract with feature recognition** then click **Next**.



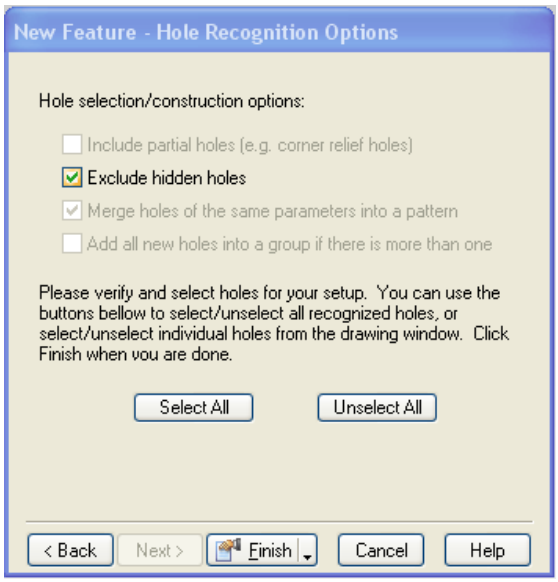
- Check **Along setup Z axis** and click **Next**.



- Fill in the form as below and click **Next**.

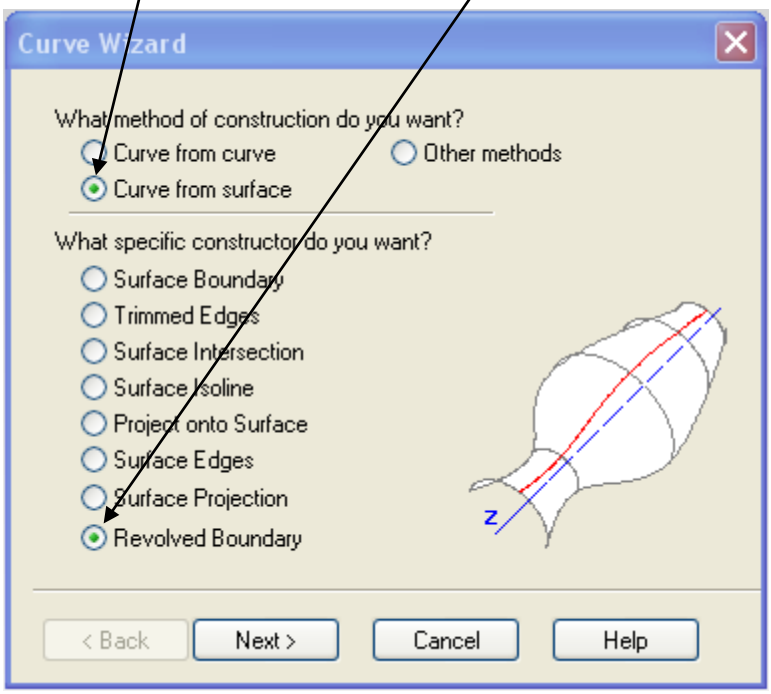


- Click **Select all** and then **Finish** and **OK**.

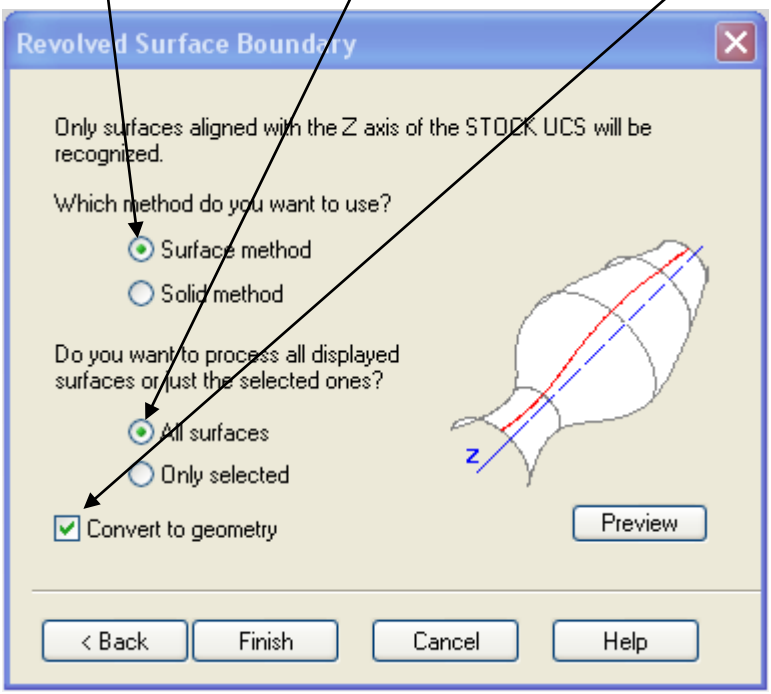


We will now extract the turning profile from the model.

- Open the **Curve wizard**.
- Check **Curve from surface** and **Revolved boundary** then click **Next**.

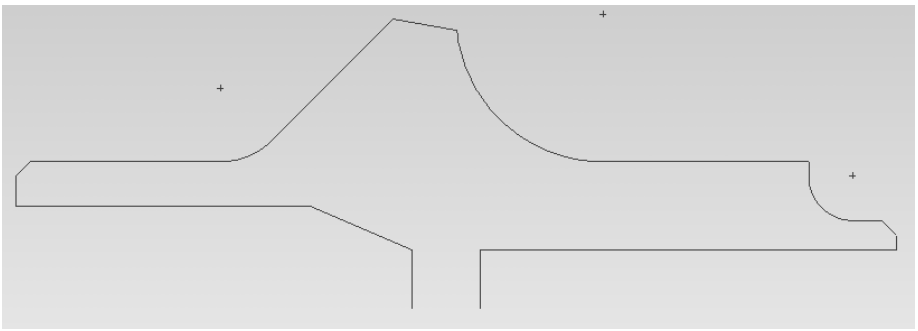


- Check **Surface method**, **All surfaces** and **Convert to geometry**.

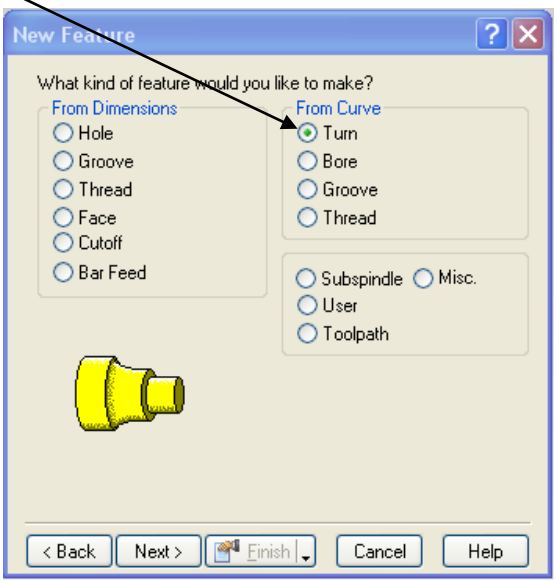


- Click **Finish**.
- **Hide all**.
- **Show all geometry**.
- Select a **Top view**.

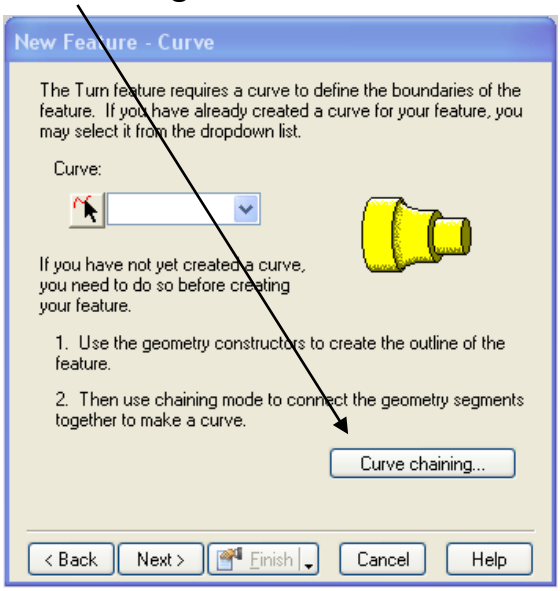
The extracted geometry should appear as below. We are now ready to make a turning feature to rough and finish the profile of the part. This operation will cut half of the profile. The other half will be cut on the sub-spindle later.



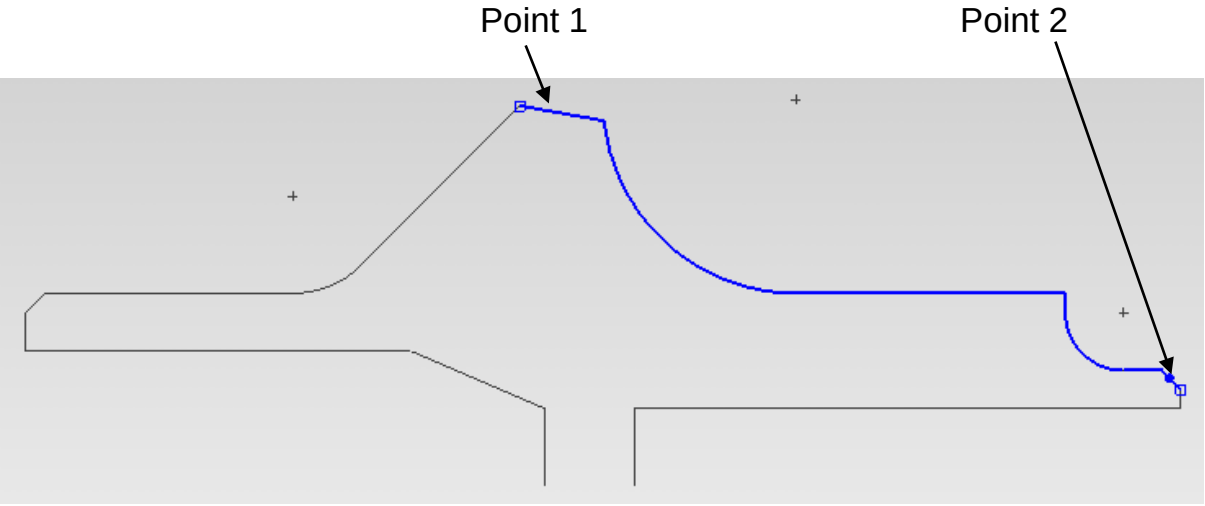
- Create a new **Turning feature**.
- Select **Turn** and click **Next**.



- Click **Curve chaining**.

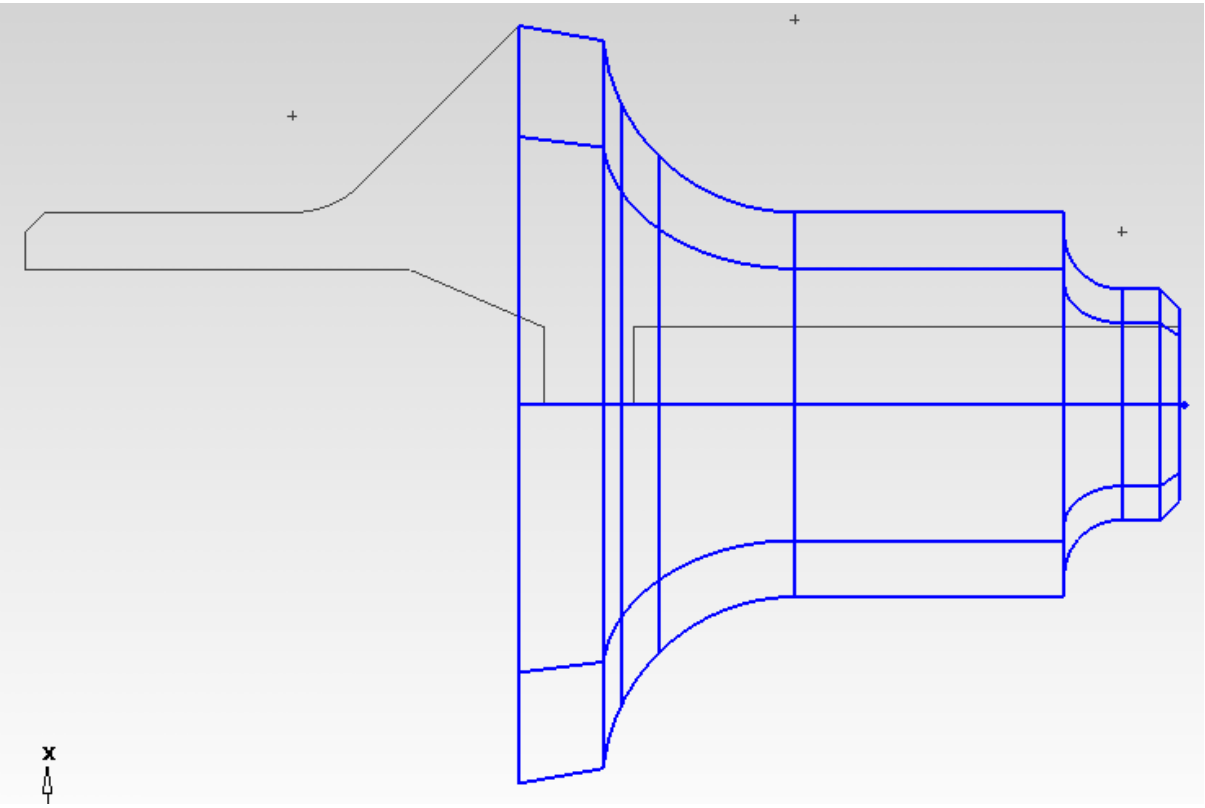


- Click on the geometry at the points indicated to generate a curve.



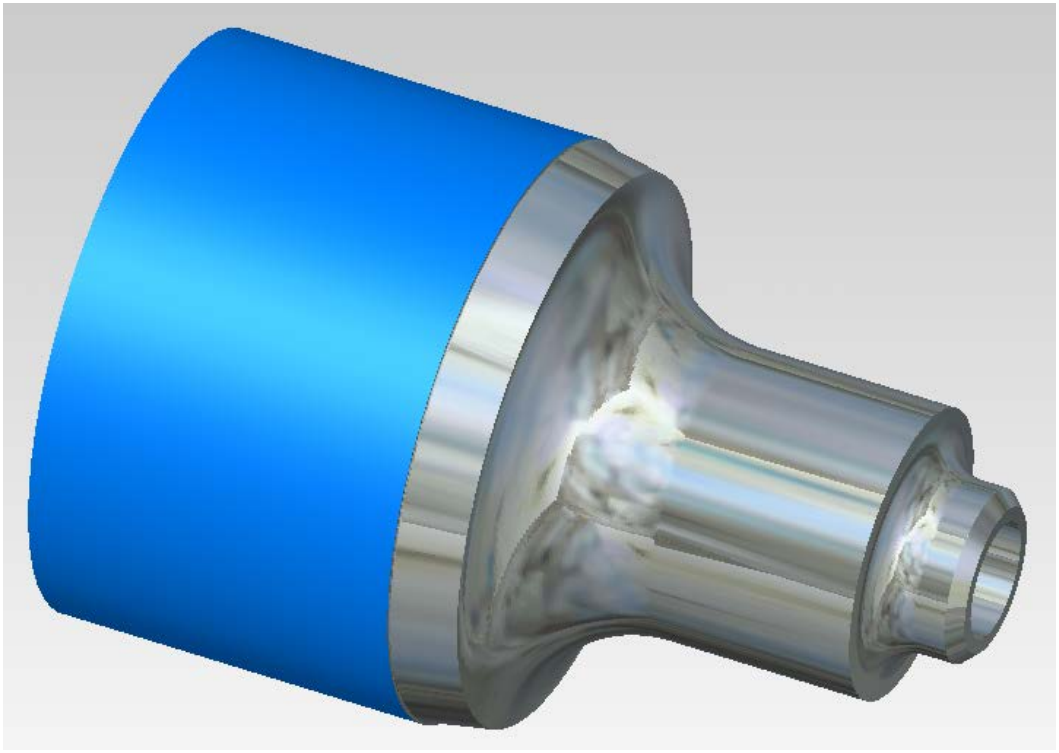
- Click **Next** then **Finish** and **OK**.

The turning feature should appear as below.



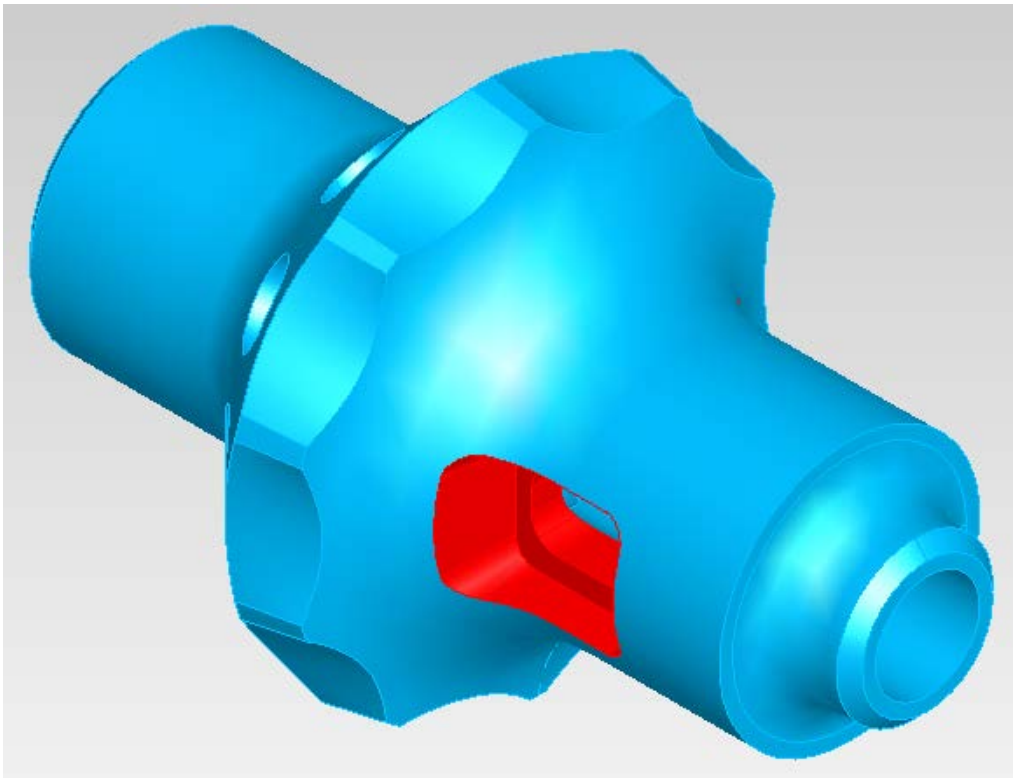
- Select an **Isometric view**.
- Run a **3D simulation**.

The turned part should now look like this.

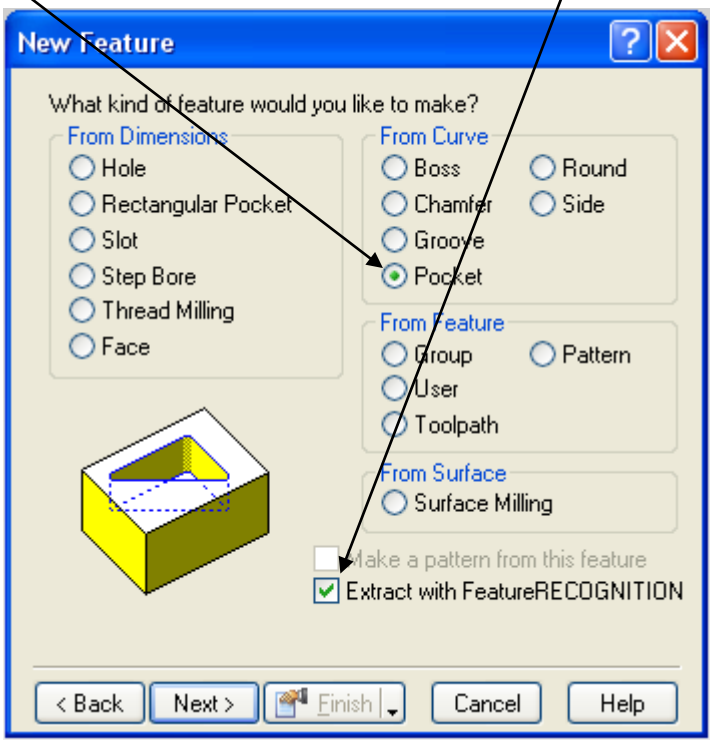


- **Eject** the simulation.
- **Hide all** and **Show all Solids**.

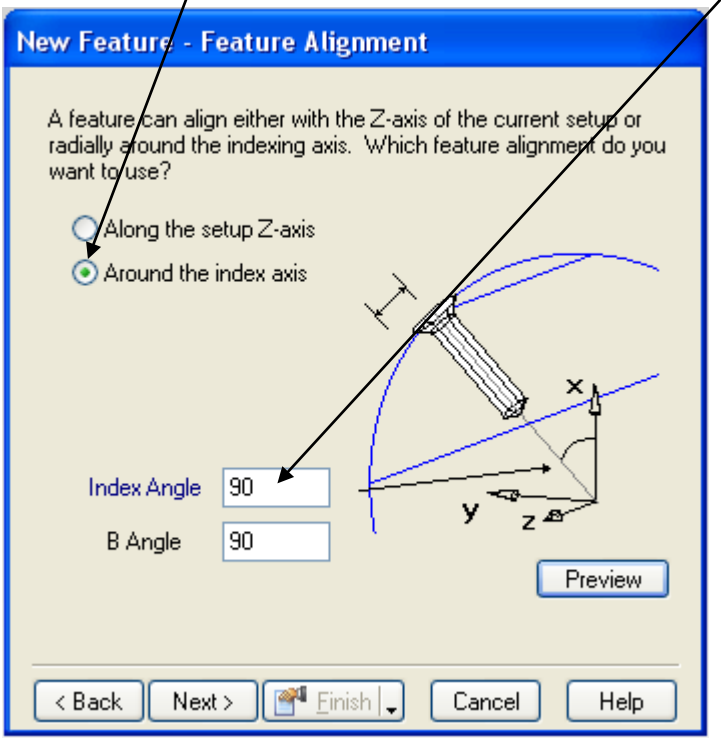
We will now mill out the pocket areas highlighted below.



- Create a new **Turn/Mill feature**.
- Check **Pocket** and **Extract with feature recognition** then click **Next**.

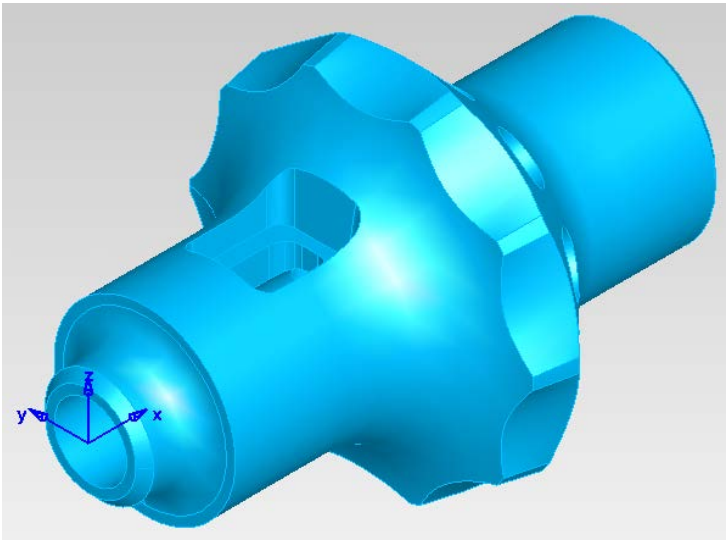


- Check **Around the index axis** and enter an **Index angle** of **90 degrees**.

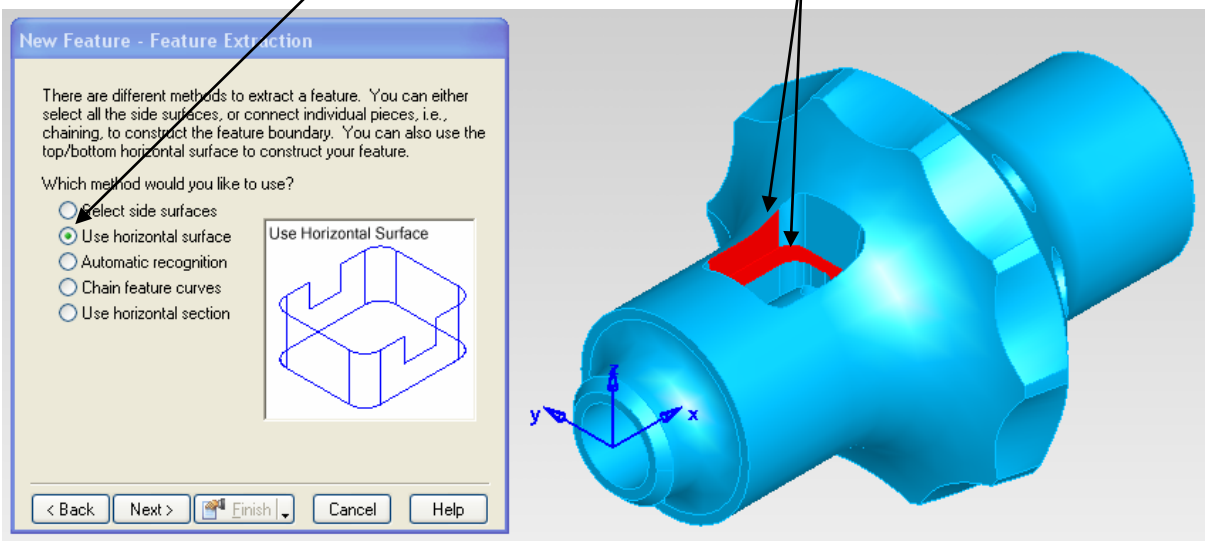


- Click **Preview**.

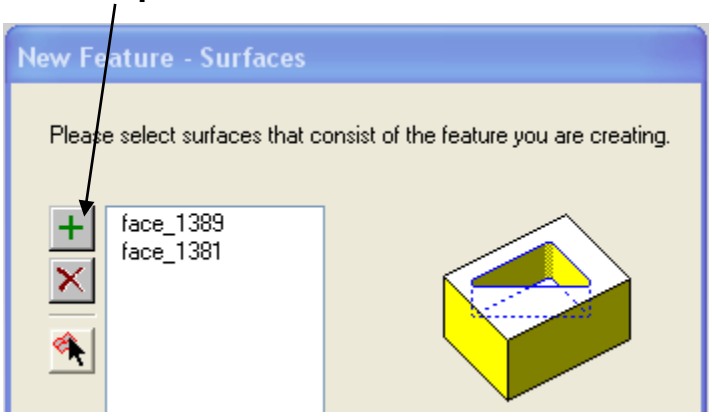
- Select an **Isometric view**. You should see the pocket pointing upwards with the Z axis aligned as shown below.



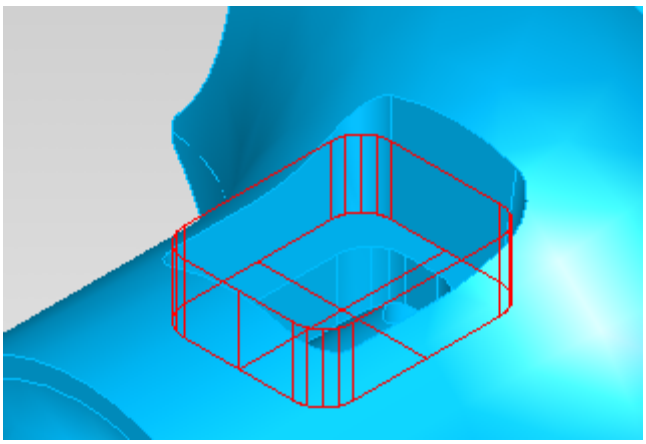
- Check **Use horizontal surface** and select the **two surfaces** indicated.



- Click **Next**.
- Click the **Green plus** to add the faces to the selection then **Finish** & **OK**.

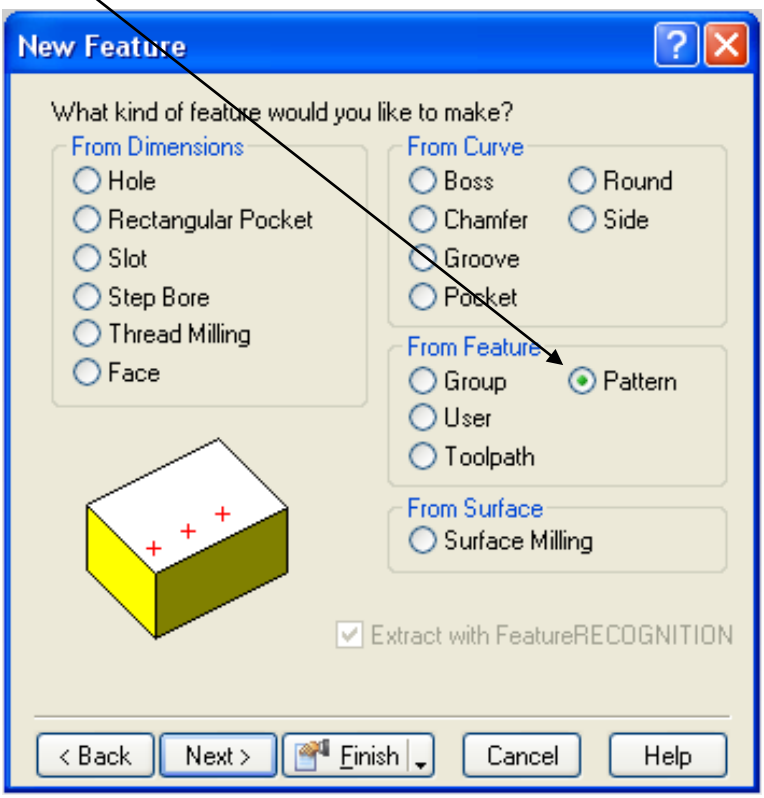


Note how the pocket depth has been set by the height of the side surface picked. In this case we will have a heavy first cut but reduced air cutting.

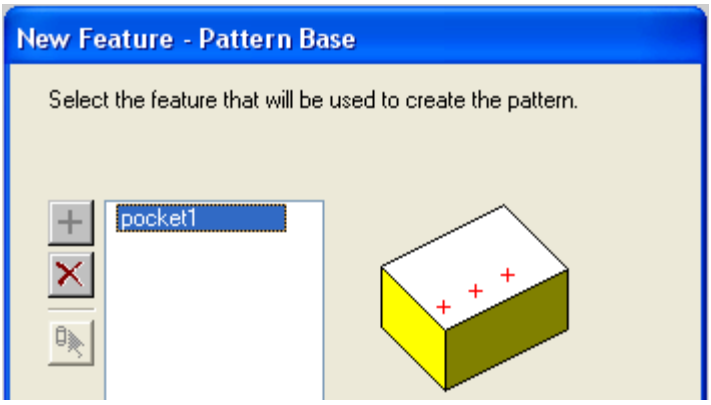


There is an identical pocket on the opposite side of the part. A pattern will be created from the existing pocket feature to cut the two pockets in a single operation.

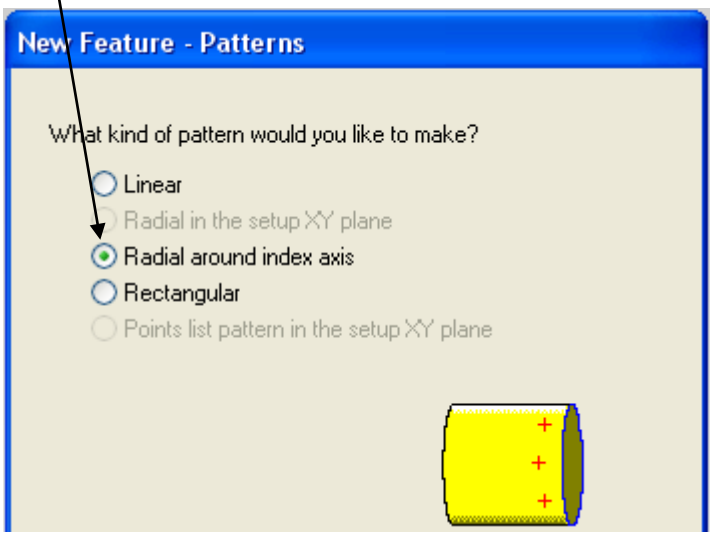
- Create a new **Turn/Mill feature**.
- Check **Pattern** and click **Next**.



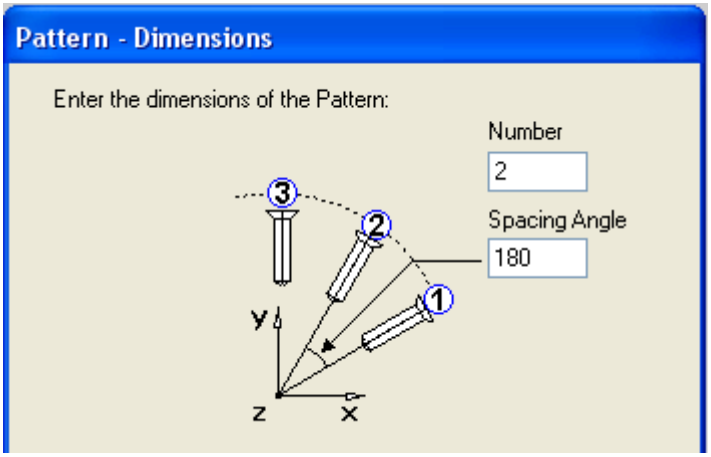
The feature pocket1 should already be selected. If not select it and click the green plus.



- Click **Next**.
- Check **Radial around index axis** and click **Next**.

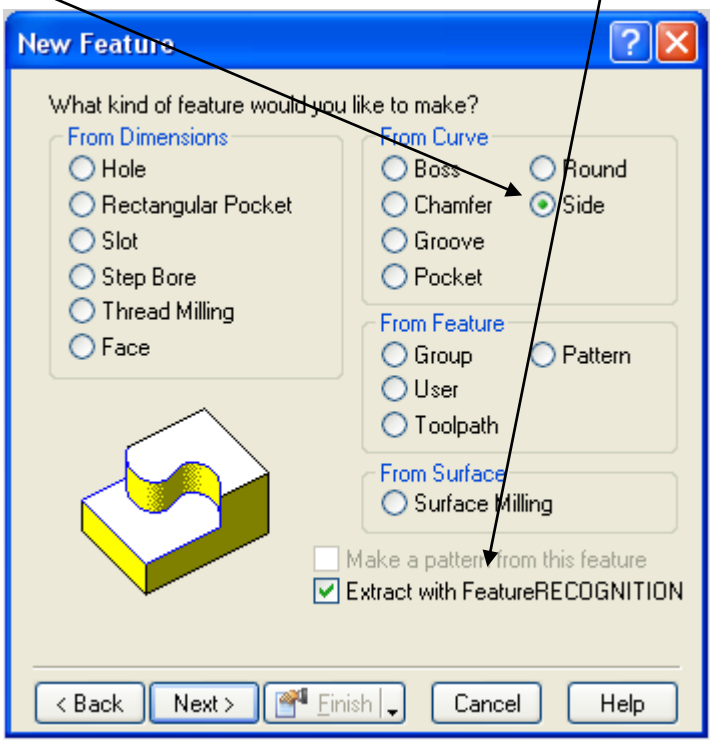


- Enter the values shown below and click **Finish** and **OK**.

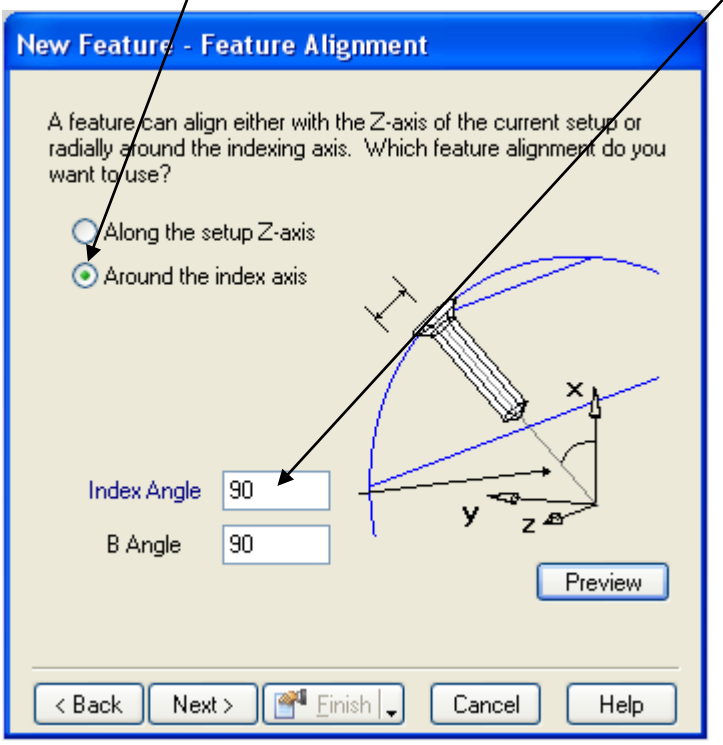


The lower part of the pocket will now be machined, as this has no bottom it will be created as a side feature using automatic recognition.

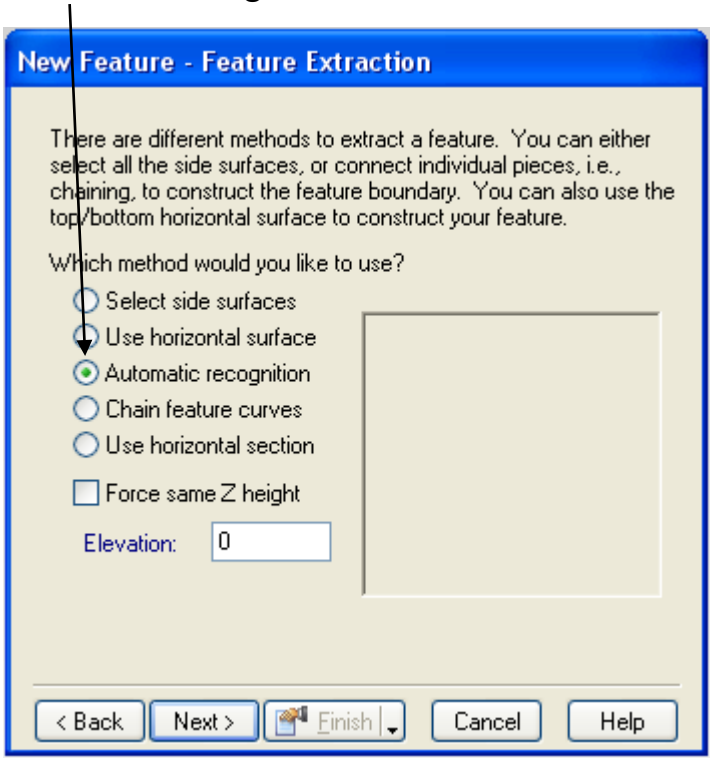
- Create a new **Turn/Mill feature**.
- Check **Side** and **Extract with feature recognition**.



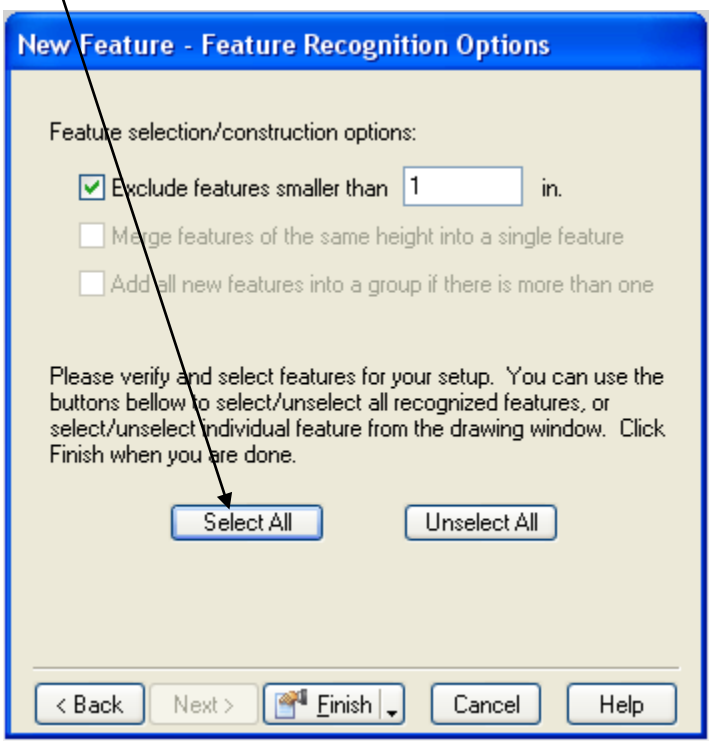
- Click **Next**.
- Check **Around the index axis** and enter an **Index angle** of **90 degrees**.



- Click **Next**.
- Check **Automatic recognition** then click **Next**.

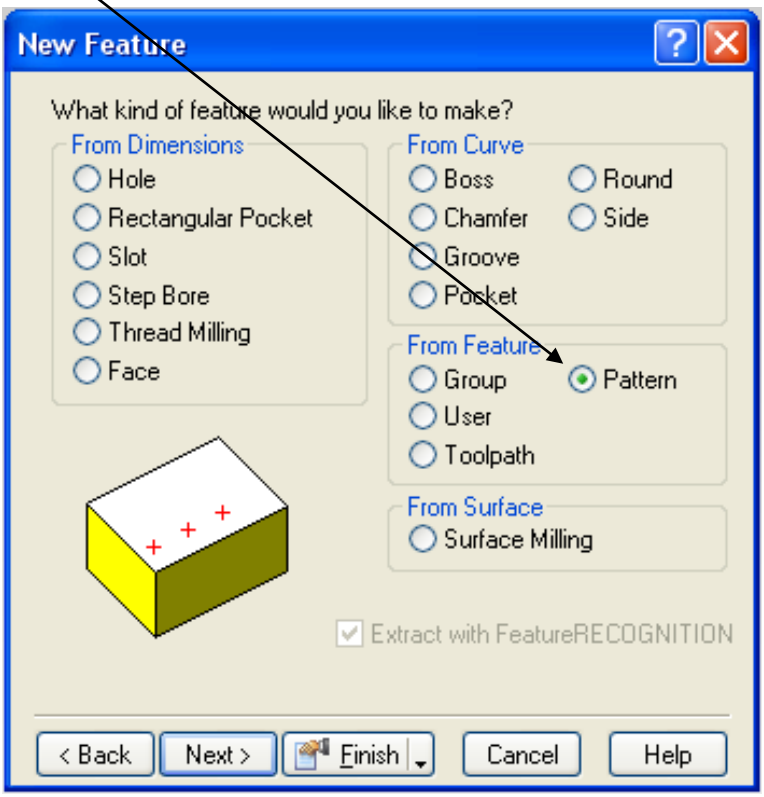


- Click **Select all** and then **Finish** and **OK**.

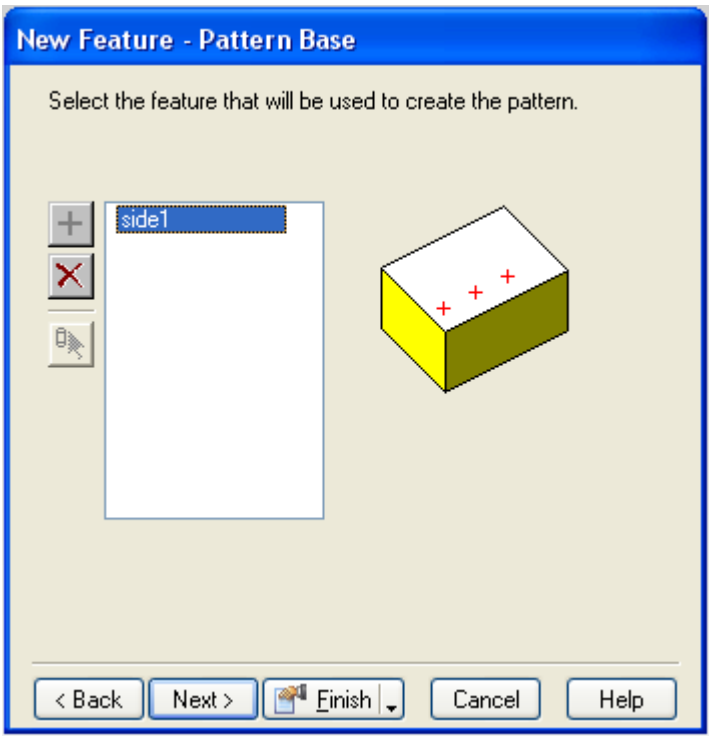


As before a pattern will be created from the side feature to machine both sides of the part in a single operation.

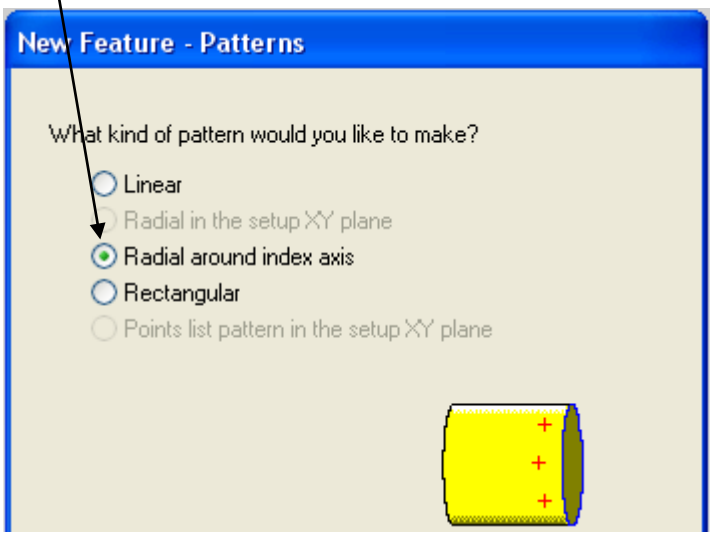
- Create a new **Turn/Mill feature**.
- Check **Pattern** and click **Next**.



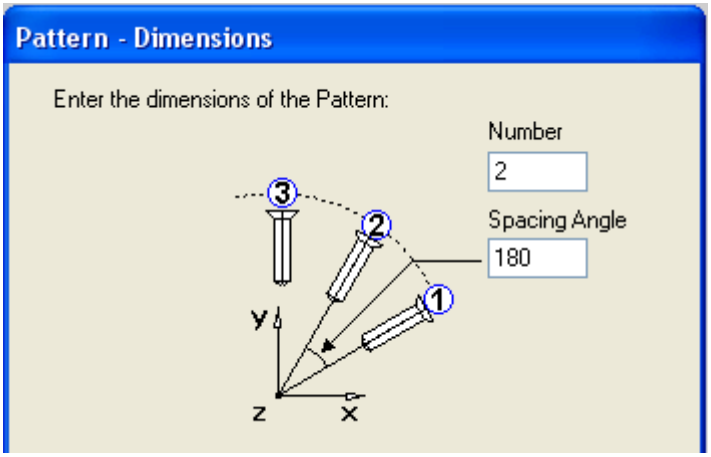
The feature side1 should already be selected. If not select it and click the green plus.



- Click **Next**.
- Check **Radial around index axis** and click **Next**.



- Enter the values shown below and click **Finish** and **OK**.

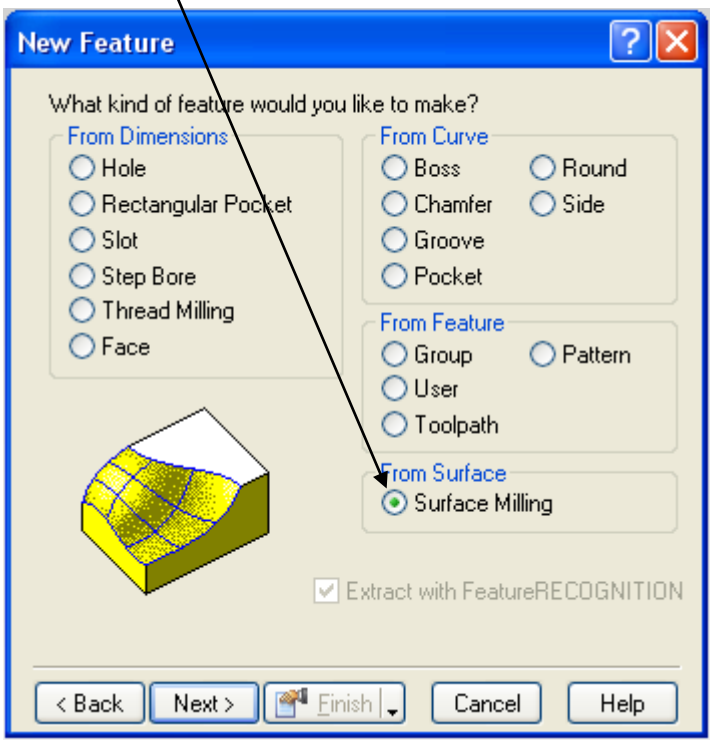


- Select an **Isometric view** and run a 3D simulation.
- Select a Top view, note the the pocket sides have been cut vertical.

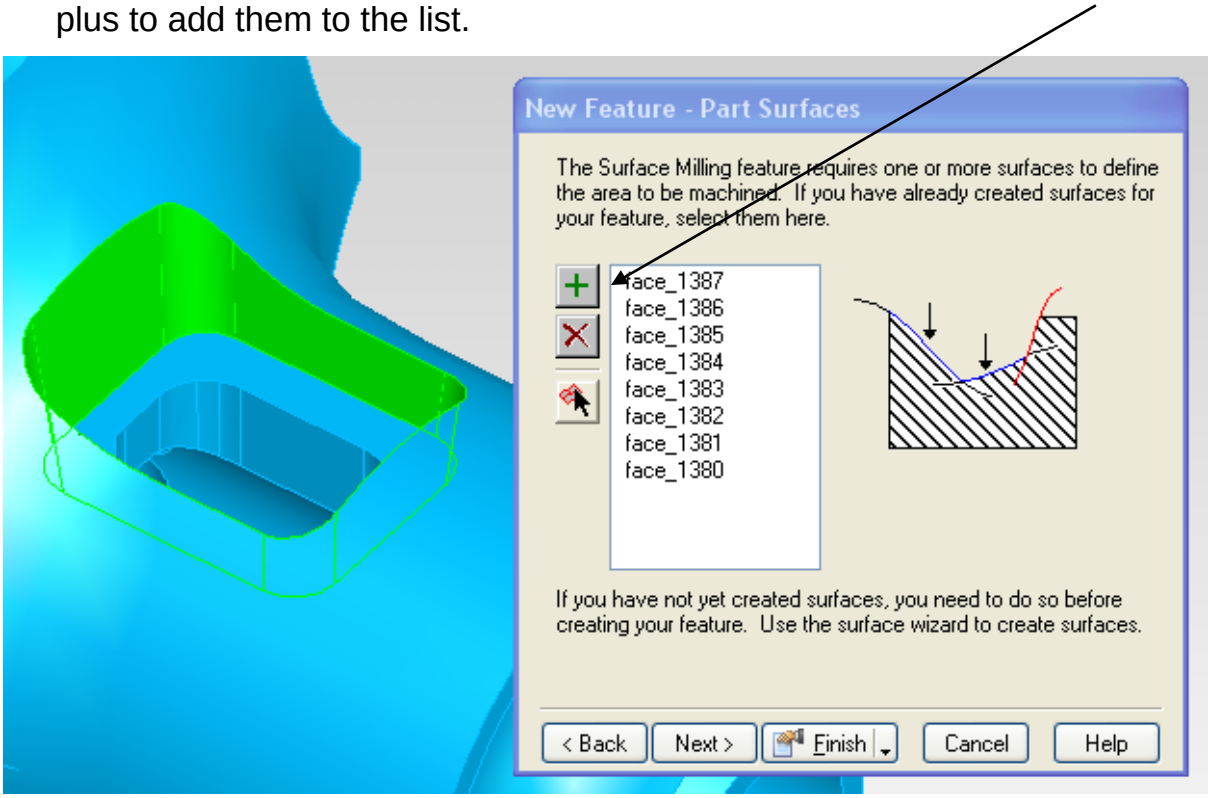
We shall now add a swarf machining operation to add the draft to the pocket sides.
A swarf milling operation is a multi-axis process that cuts a wall with the side of the cutter.
The cutter is automatically tilted to maintain contact with the wall.

- **Eject** the simulation.

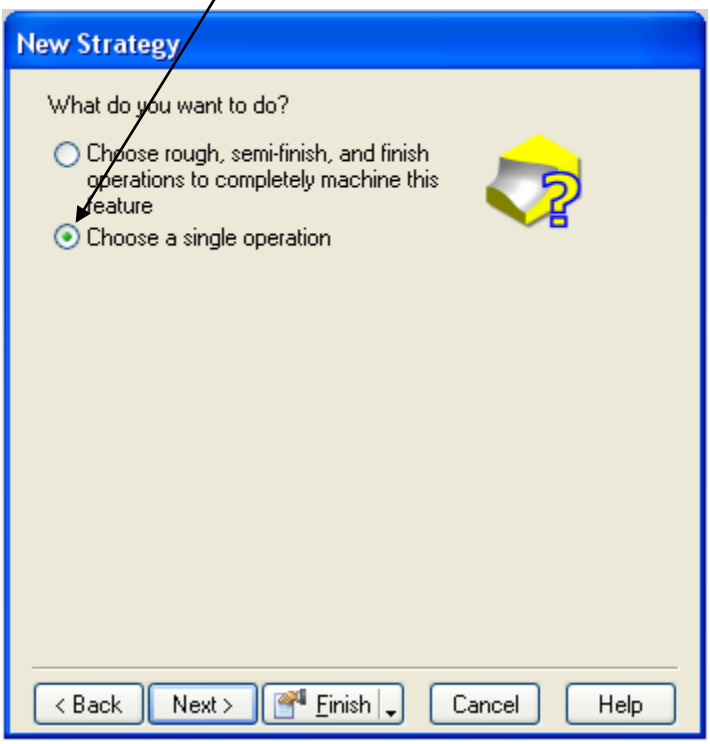
- Create a new **Turn/Mill feature**.
- Check **Surface Milling** and click **Next**.



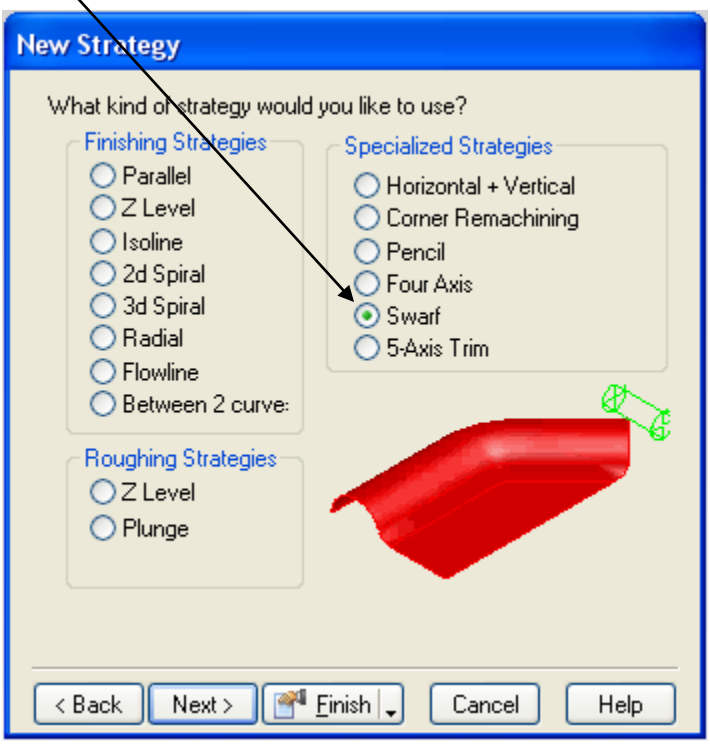
- **Shift select** the faces making the side of the pocket and click the green plus to add them to the list.



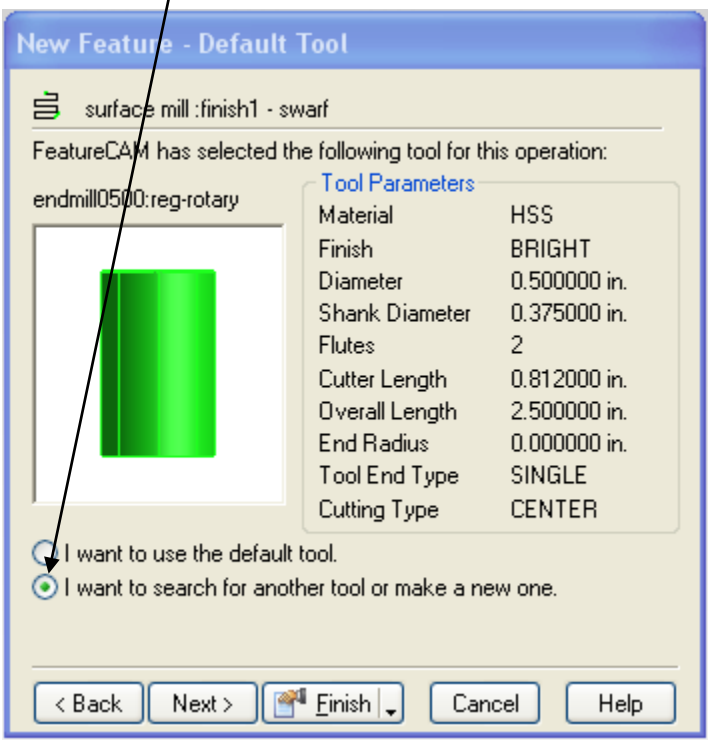
- Click **Next**.
- Check **Choose a single operation**.



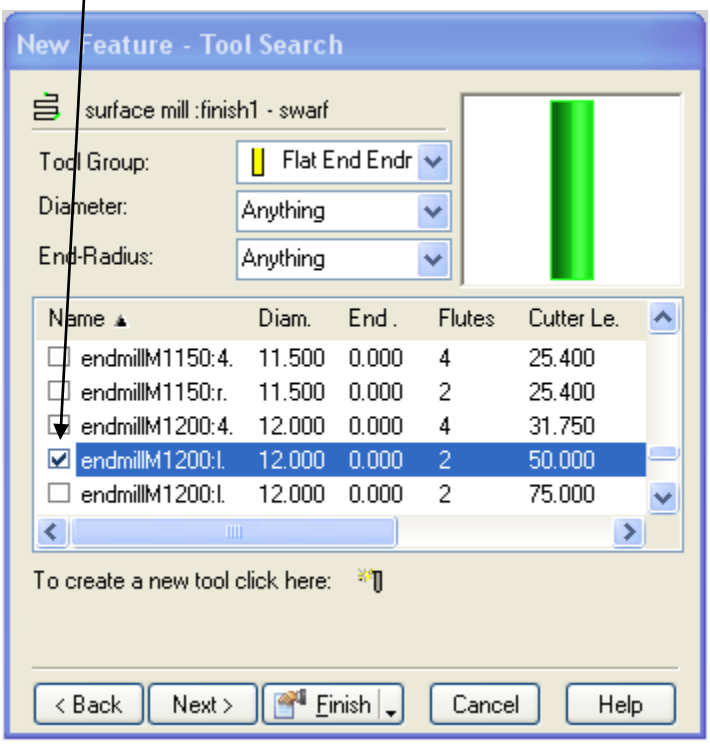
- Check **Swarf** and click **Next**.



- Click **Next** to the **Default tool** page.
- Check **I want to search for another tool** then click Next.

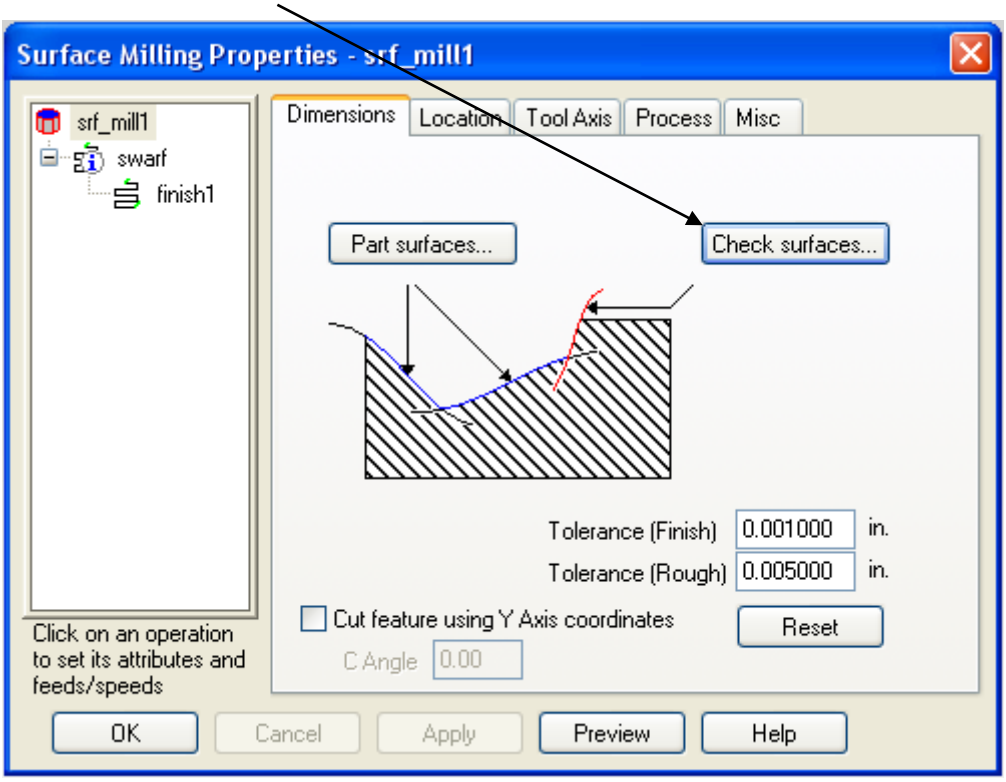


- Select a **12mm long reach flat end mill**.

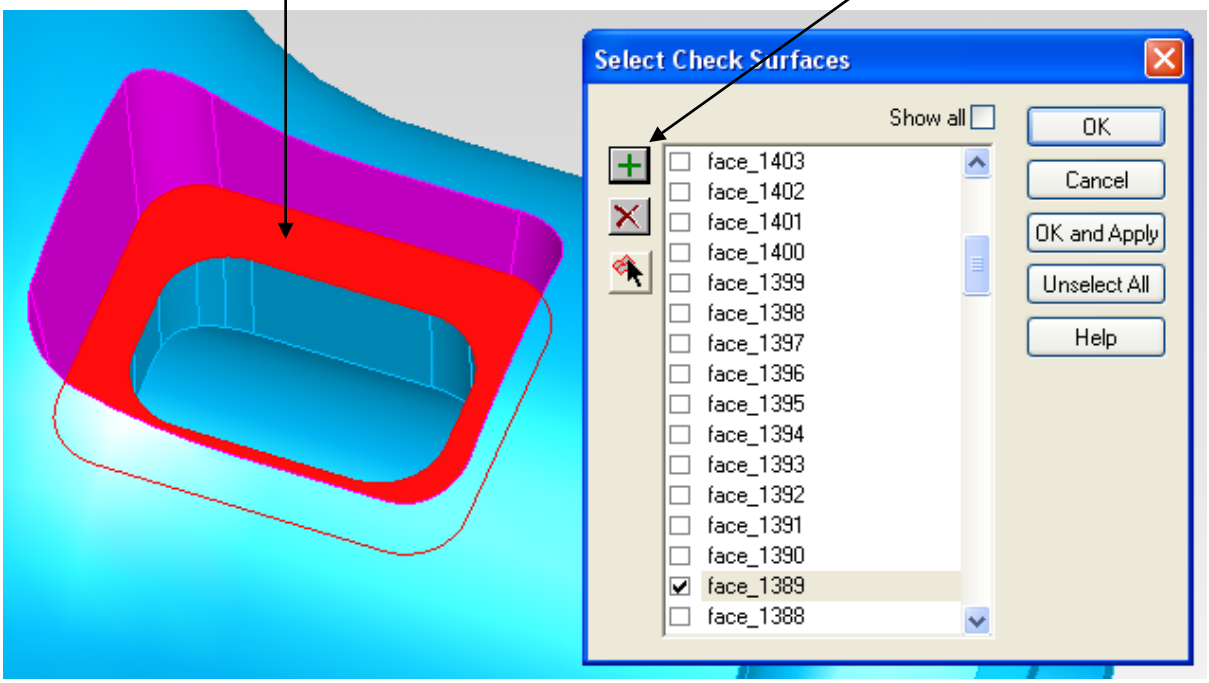


- Click **Finish**.

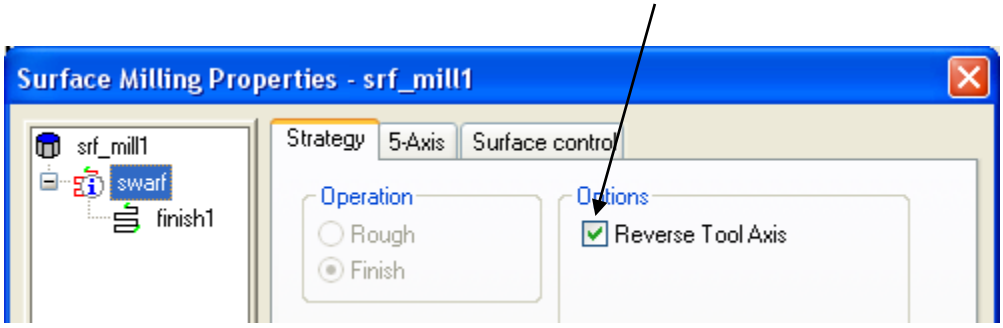
- Select **Check surfaces** from the **Dimensions tab**.



- Select the **bottom face** of the pocket and click the **green plus**.

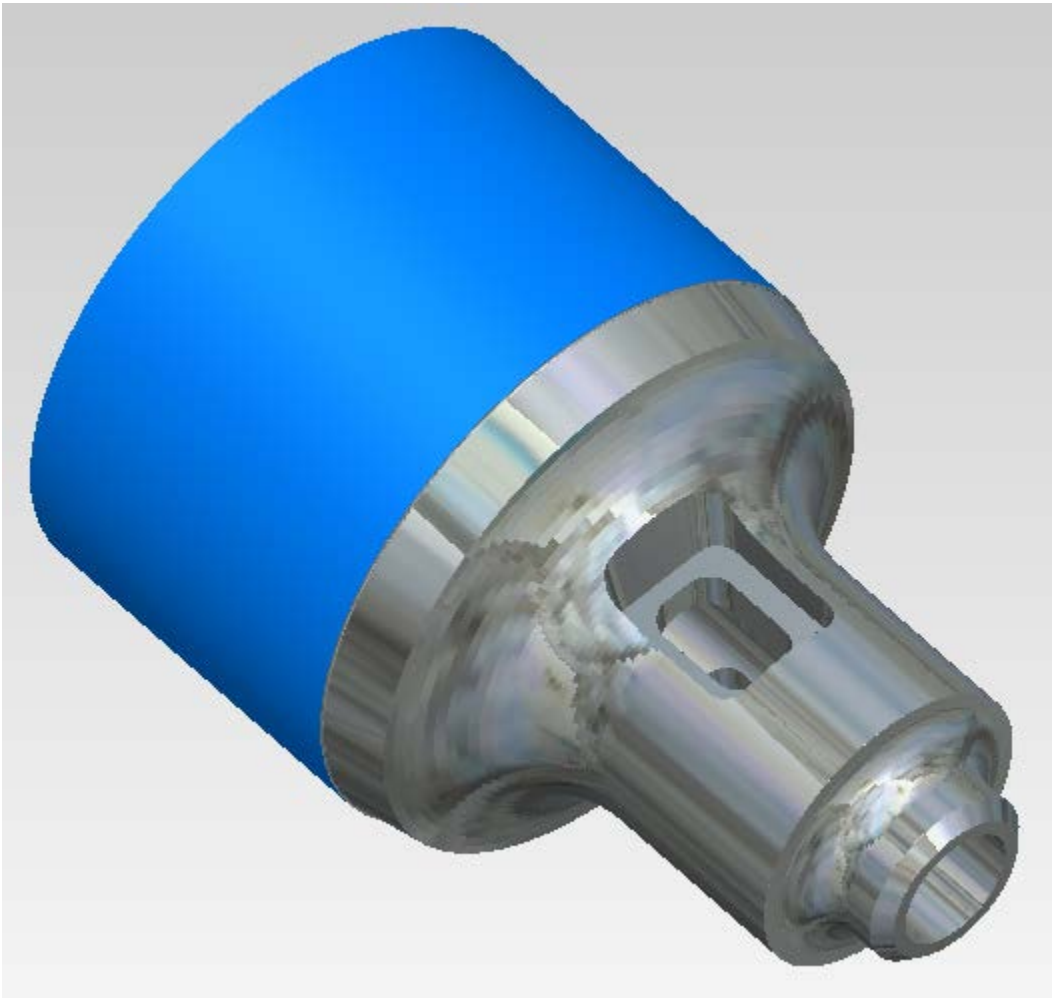


- Click **OK and Apply**.
- Select the **Swarf** operation.
- Select the **Tool axis tab** and check **Reverse tool axis**.

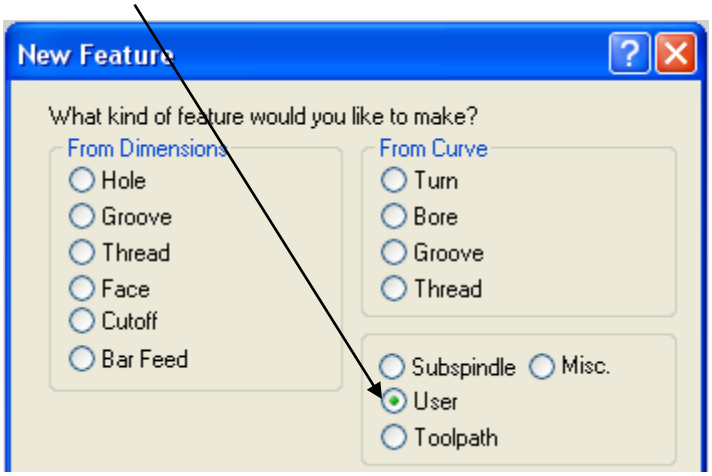


- Click Apply and then OK.
- Create a swarf feature on the other side of the part using the method described above. Run a **3D simulation** to show how the tool axis tilts during the swarf machining operation.

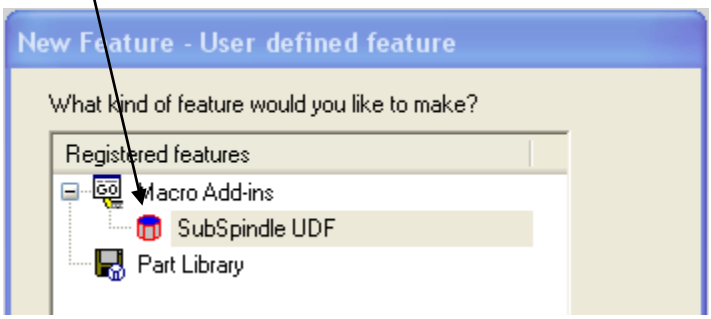
One half of the part is now completed. We need to pass the part over to the sub-spindle to machine the other end.



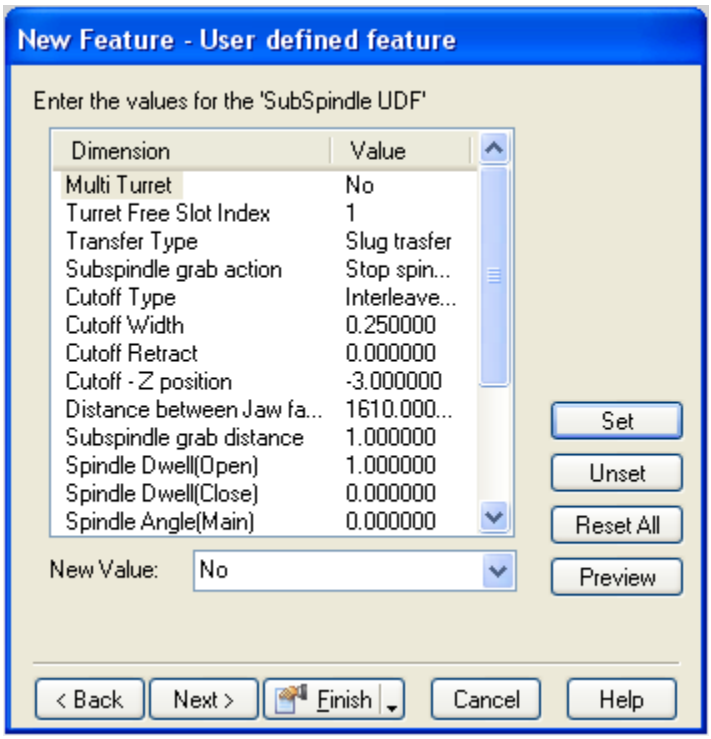
- Create a new **Turning feature**.
- Check **User** and click **Next**.



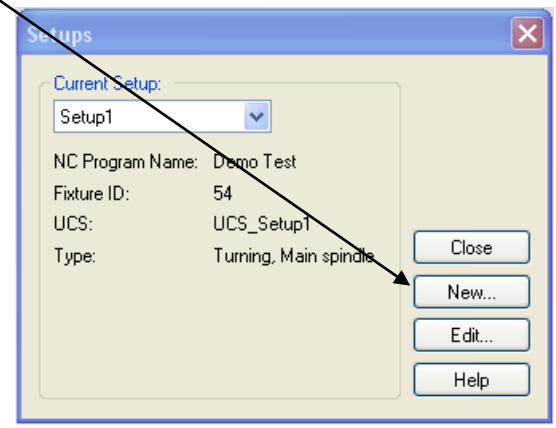
- Select **SubSpindle UDF** and click **Next**.



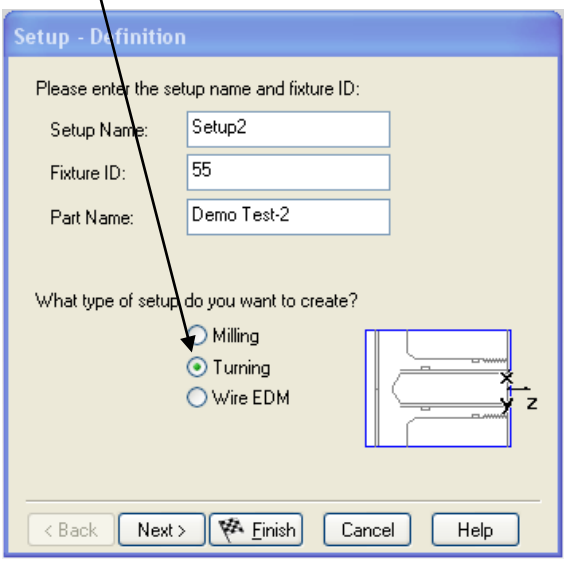
- Set **Multi Turret = No** and **Transfer type = Slug transfer**.
- Click **Finish**.



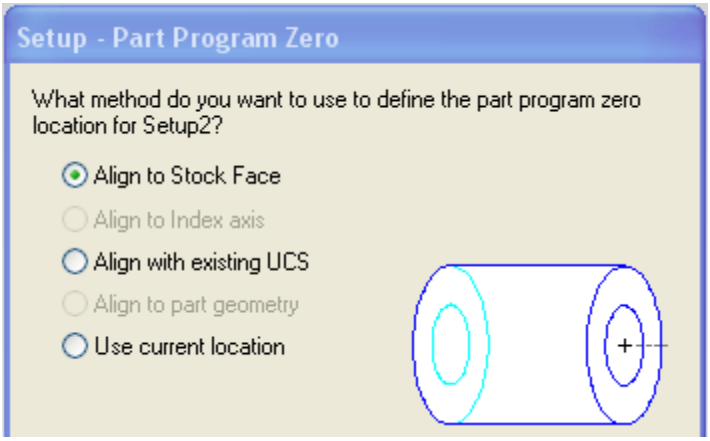
- Double click on **setup1** to open it's properties.
- Click **New**.



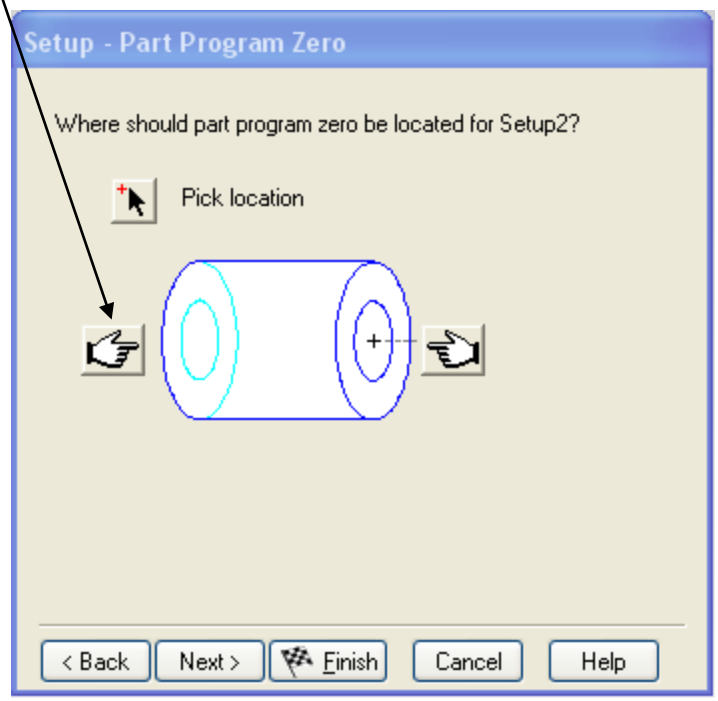
- Check **Turning** and click **Next**.



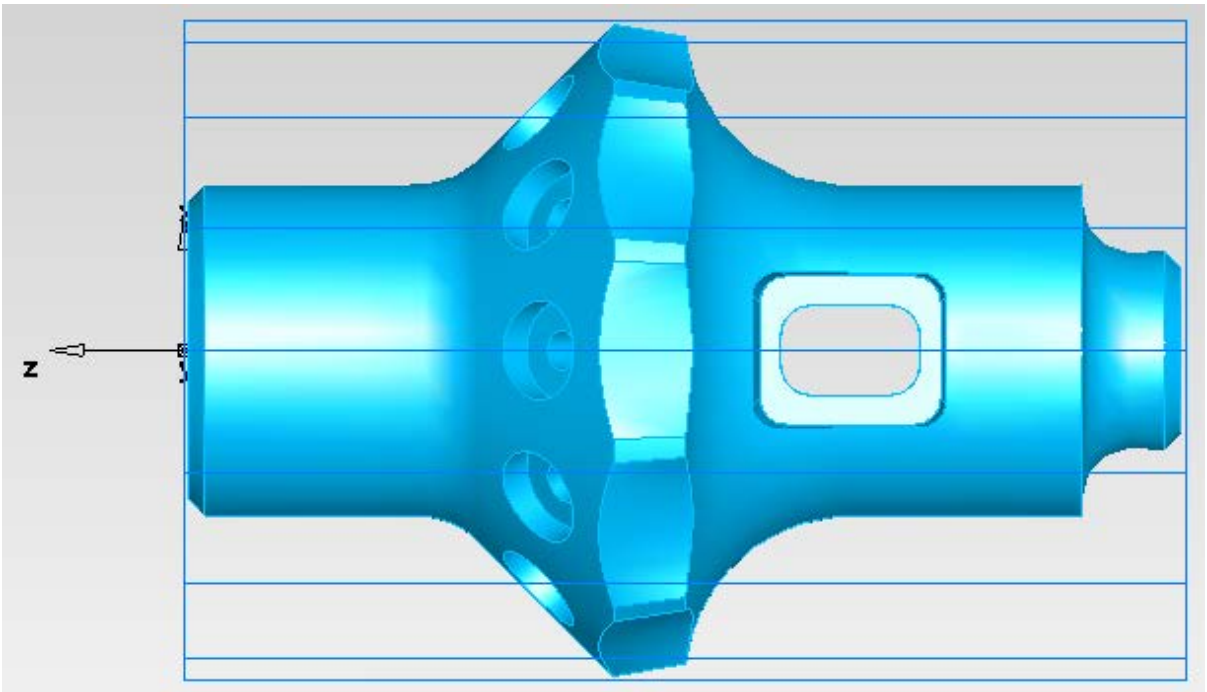
- Check **Align to stock face** then click **Next**.



- Click the **Left hand button** and then click **Finish**.

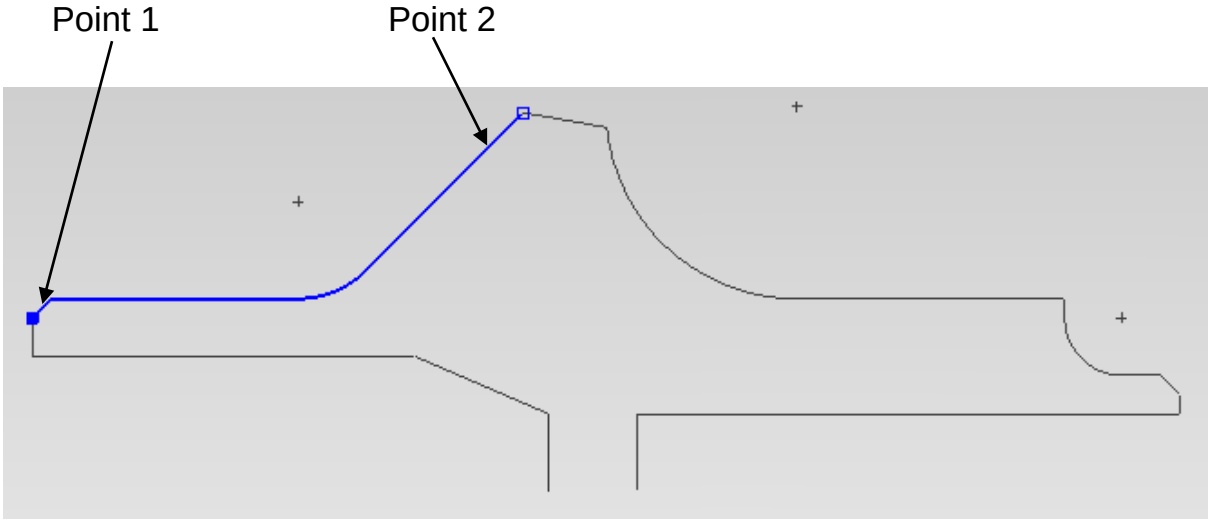


The new setup and the part should now be aligned as shown below.



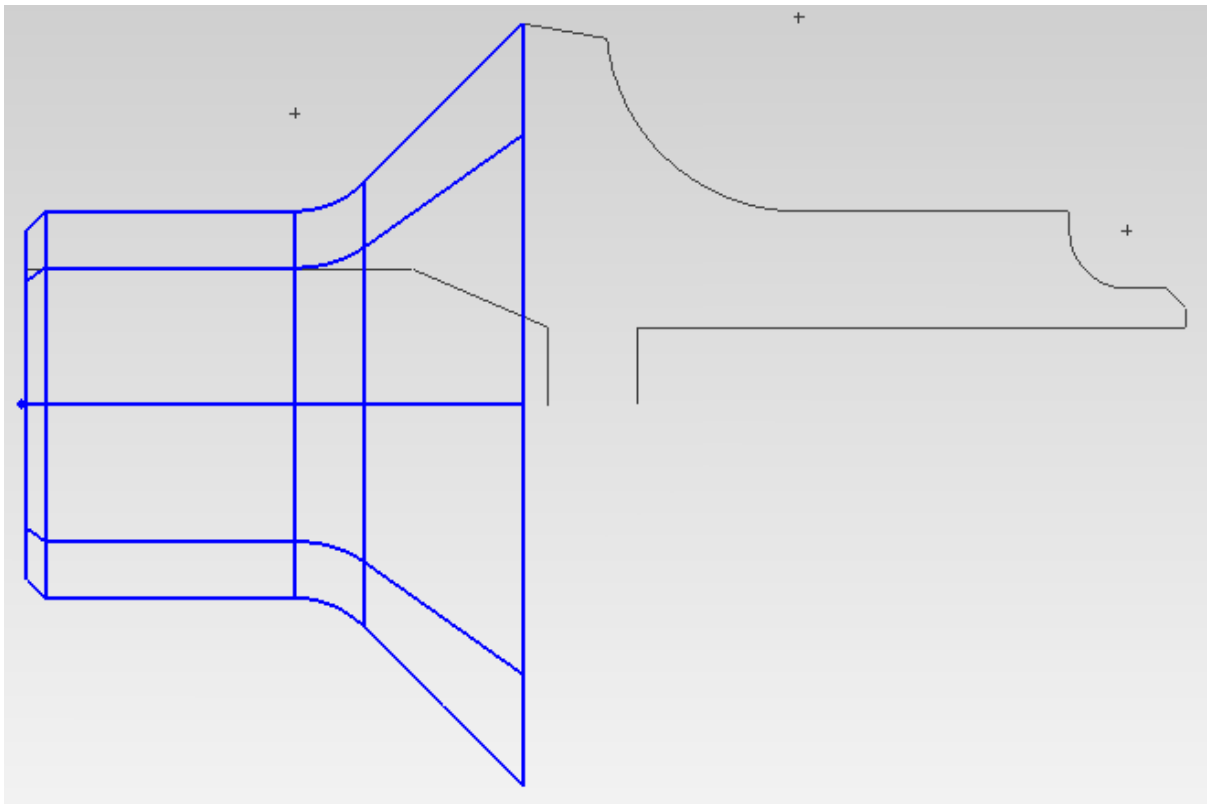
- Create a new **Turning face** feature at **Z -0.05**
- **Hide all**.
- **Show all geometry**.
- Select the **Top View**.
- Create a new **Turning Feature**.
- Check **Turn** & click **Next**.

- Click **Chain Curves**.
- Select the geometry at the points indicated to create the curve.



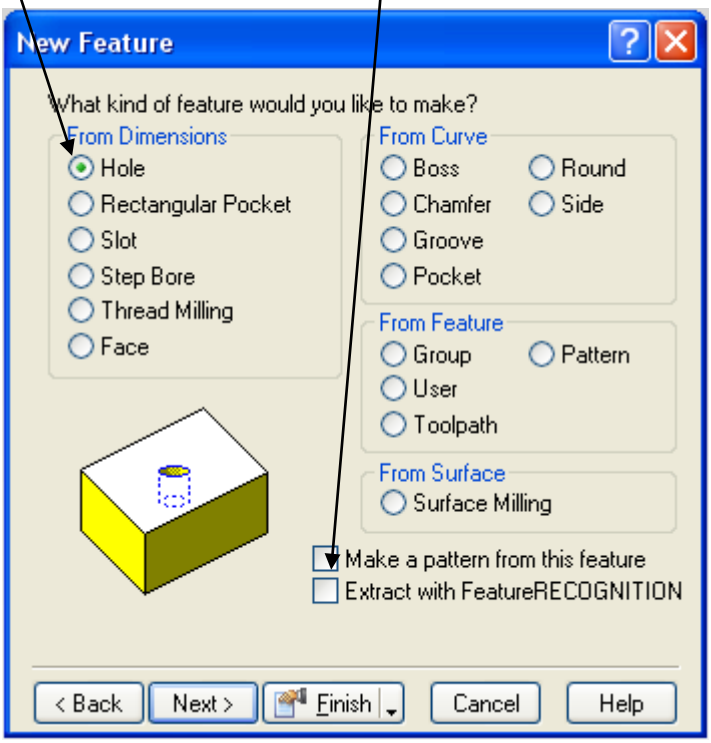
- Click Finish and then OK.

The new turning feature should now appear as below.

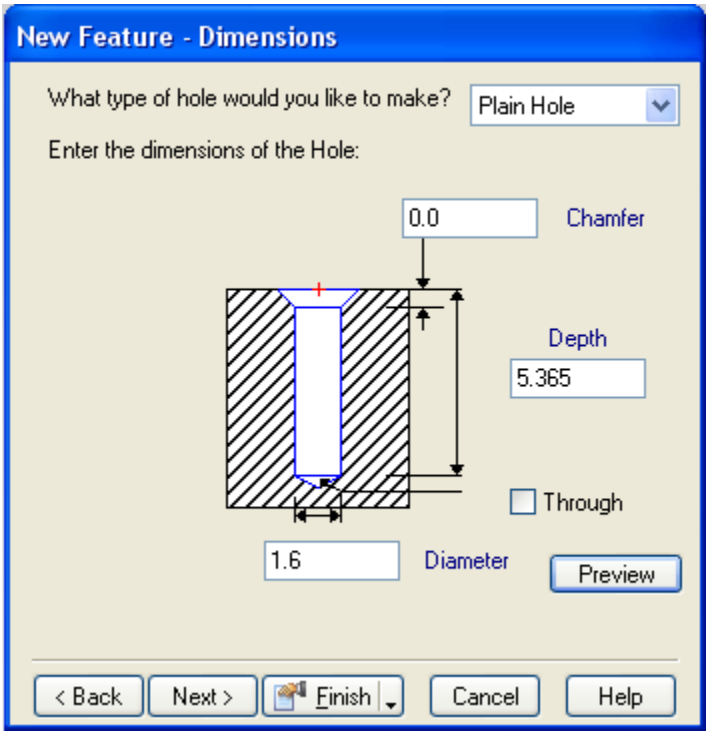


We will now create a hole as a starting point for the large bore in the left hand end of the part.

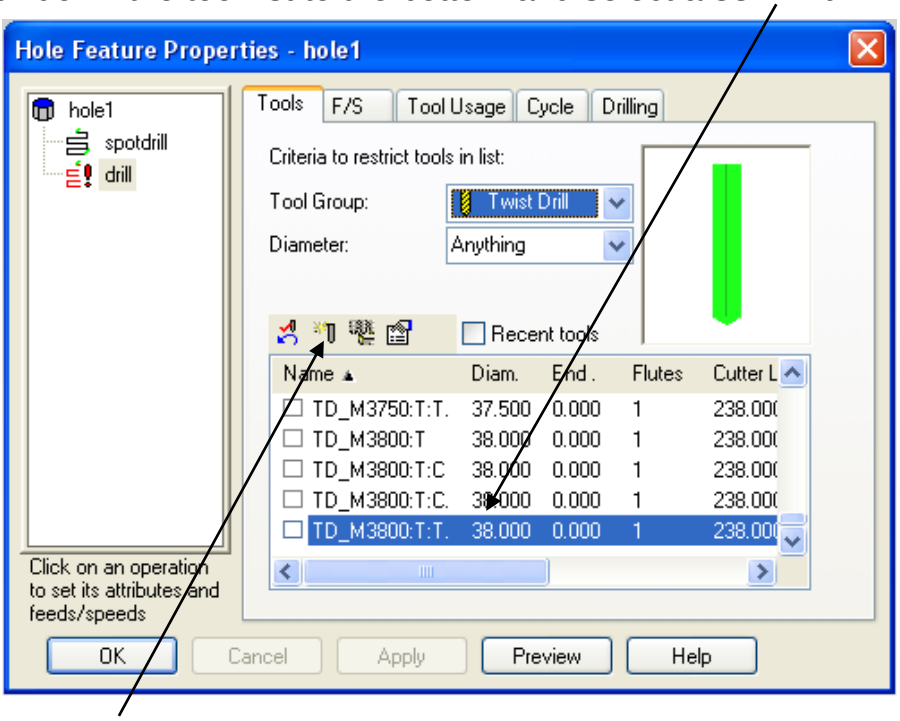
- Create a new **Turn/Mill feature**.
- Check **Hole**, uncheck **Extract with FeatureRECOGNITION** & click **Next**.



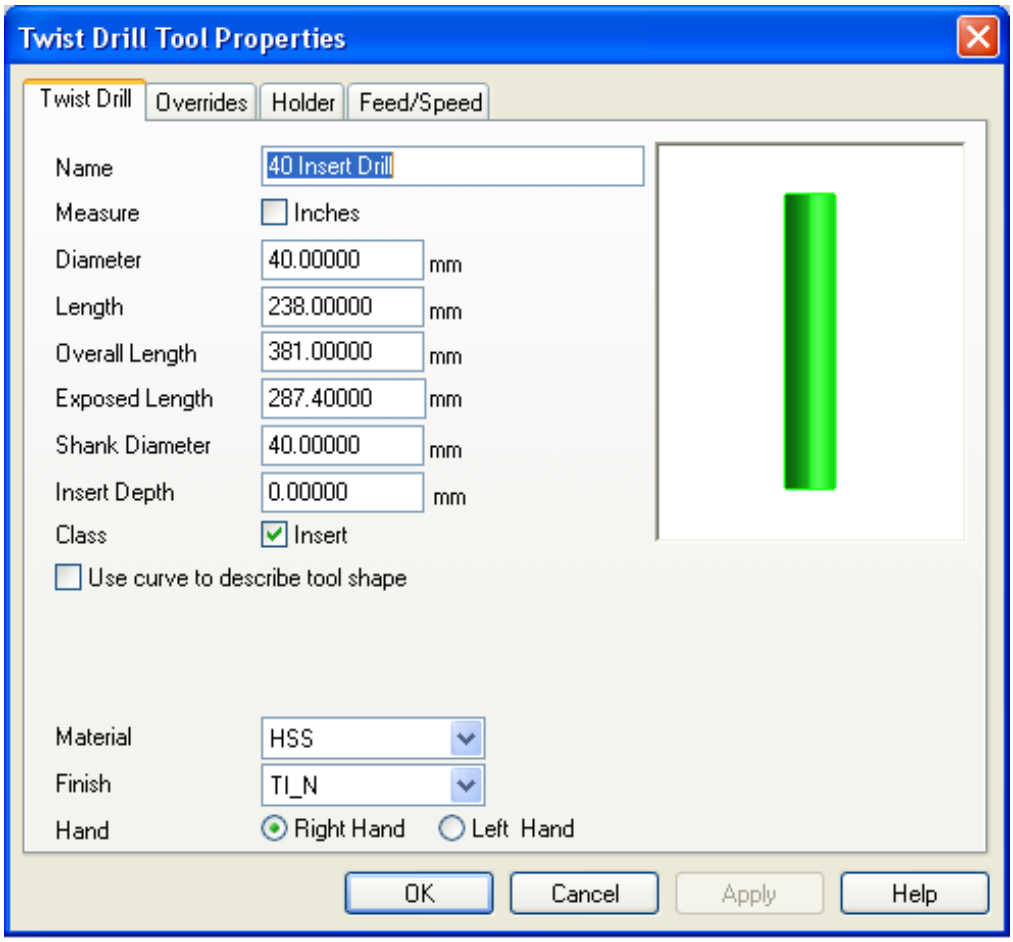
- Fill in the form as shown and click **Finish**.



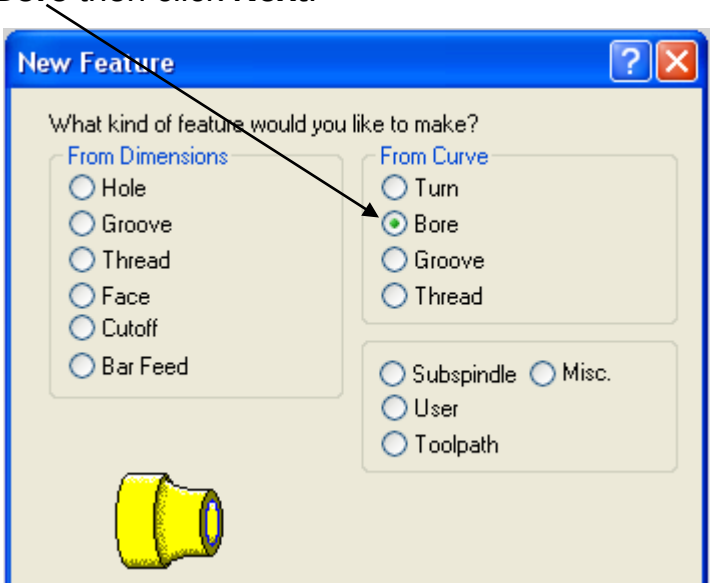
- Click on the **Drill operation** and select the **Tools tab**.
- Select **Twist drills**.
- Scroll down the tool list to the bottom and select a **38mm drill**.



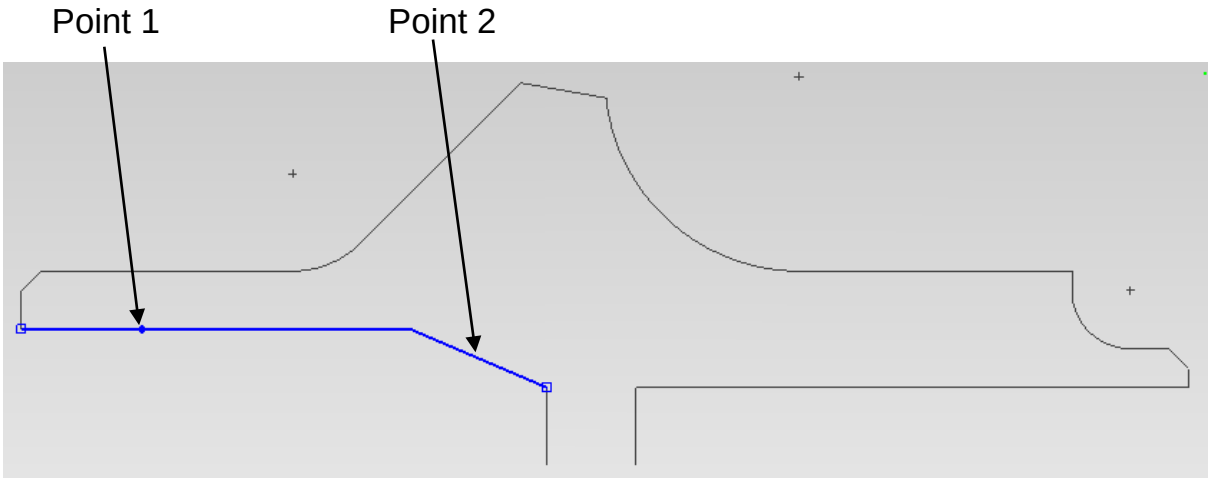
- Click **New tool**.
- Fill in the form as shown below, click **Apply** and then **OK**.



- Click **Yes** to set the new tool as the override.
- **Hide all**.
- **Show all geometry**.
- Select the **Top View**.
- Create a new **Turning feature**.
- Check **Bore** then click **Next**.



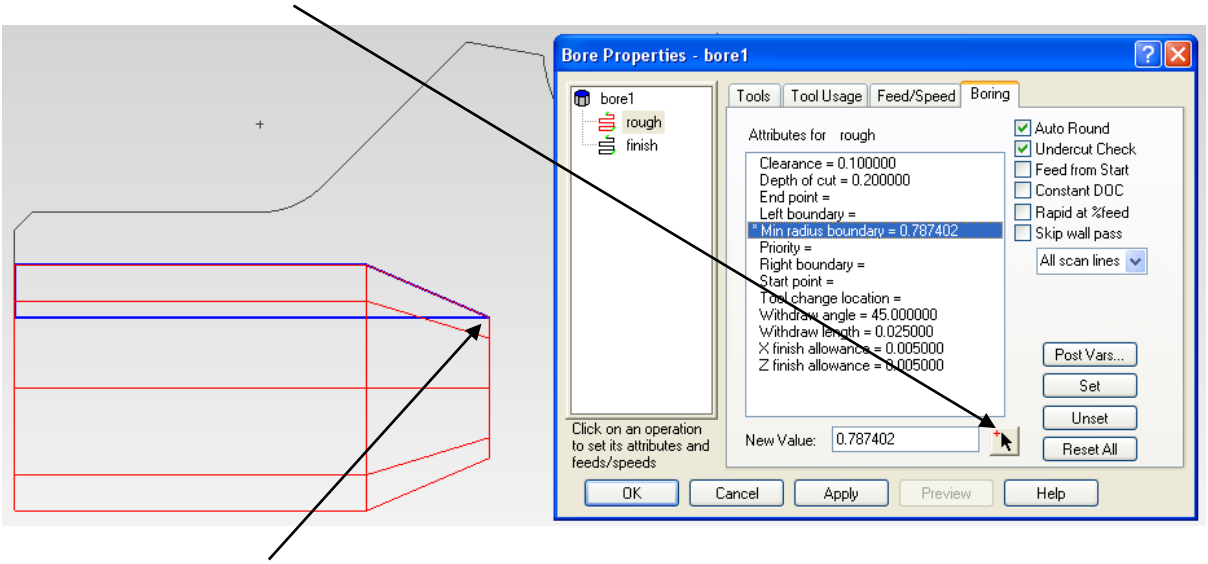
- Click **Chain Curves**.
- Select the geometry at the points indicated to create the curve.



- Click Finish and then Preview the feature

Note how the roughing starts at the centre. As the hole has been pre-drilled this is unnecessary.

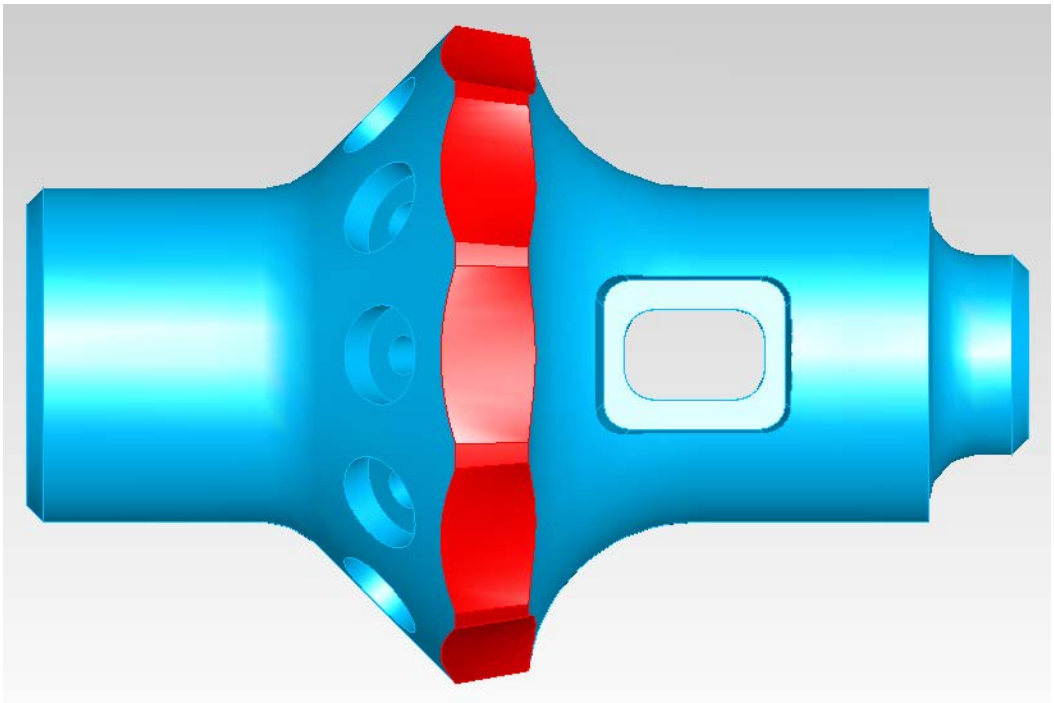
- Select the rough operation.
- Select **Min radius boundary**.
- Click on the arrow



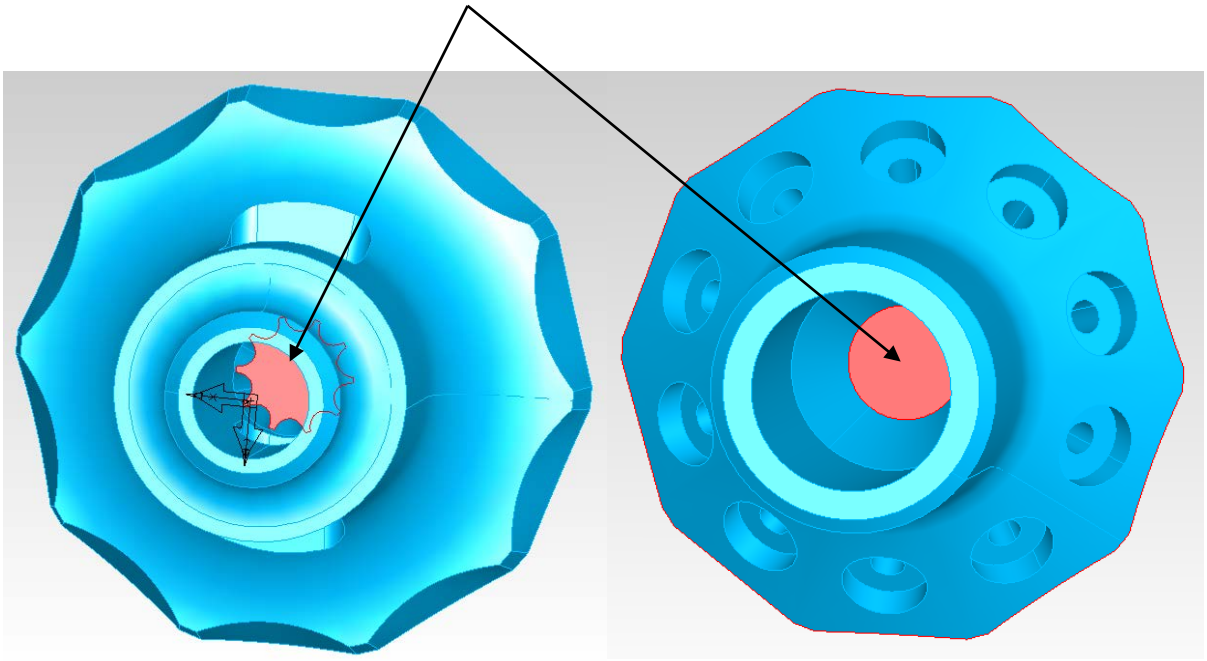
- Select this point on the bore.
- Click **Apply** and then **OK**.

We will now finish the scalloped edge of the part using a swarf machining operation.

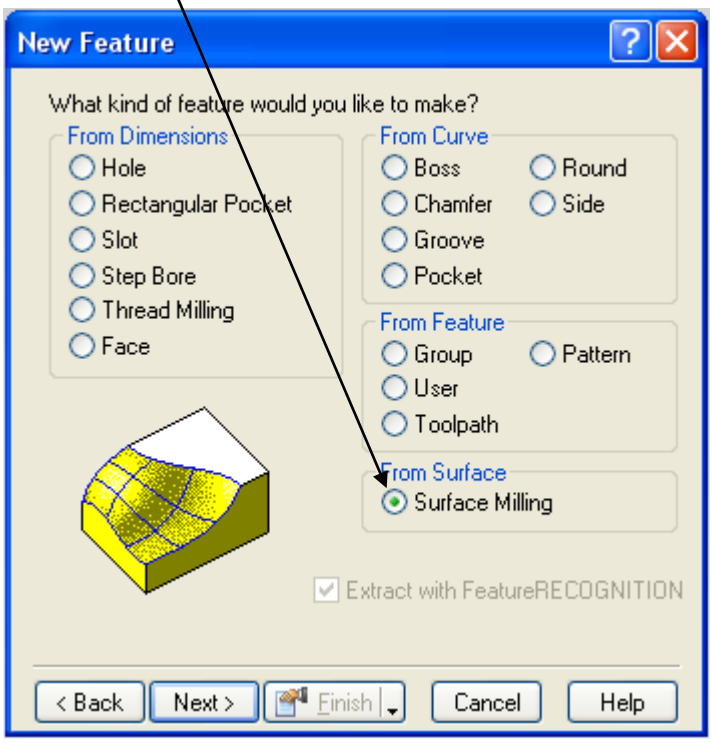
- **Hide all** and then **Show all solids**.
- Select a **Top view** and **box select** the surfaces highlighted below.



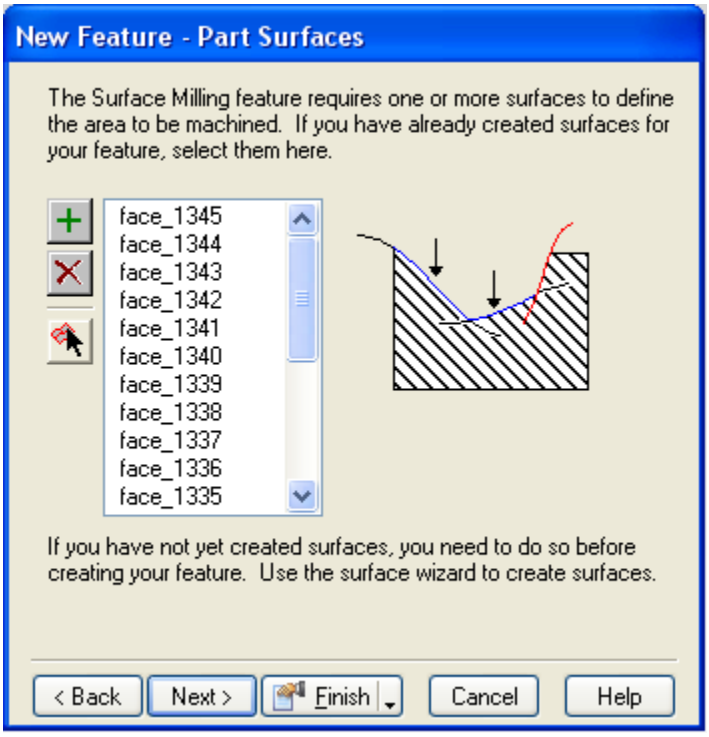
- Rotate the view and **deselect** the surfaces indicated below.



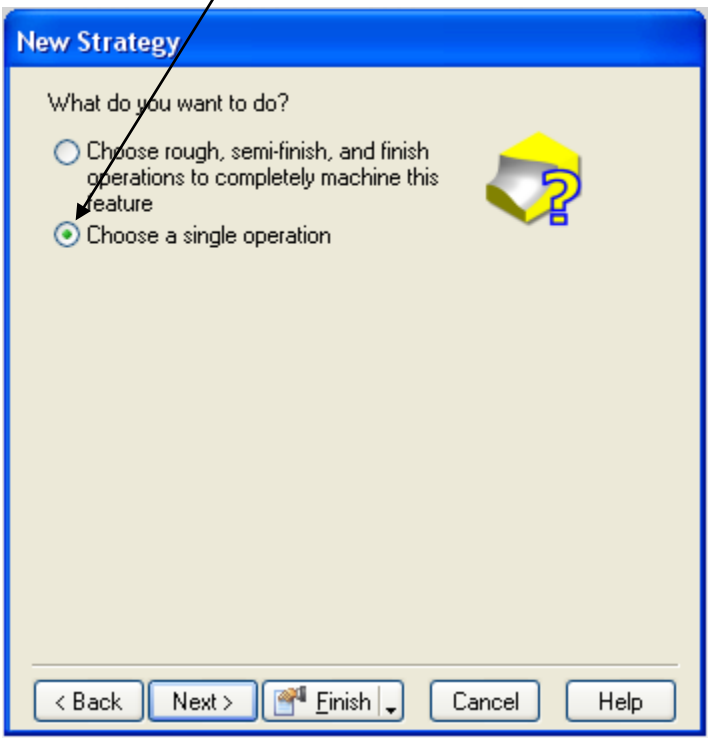
- Create a new **Turn/Mill feature**.
- Check **Surface Milling** and click **Next**.



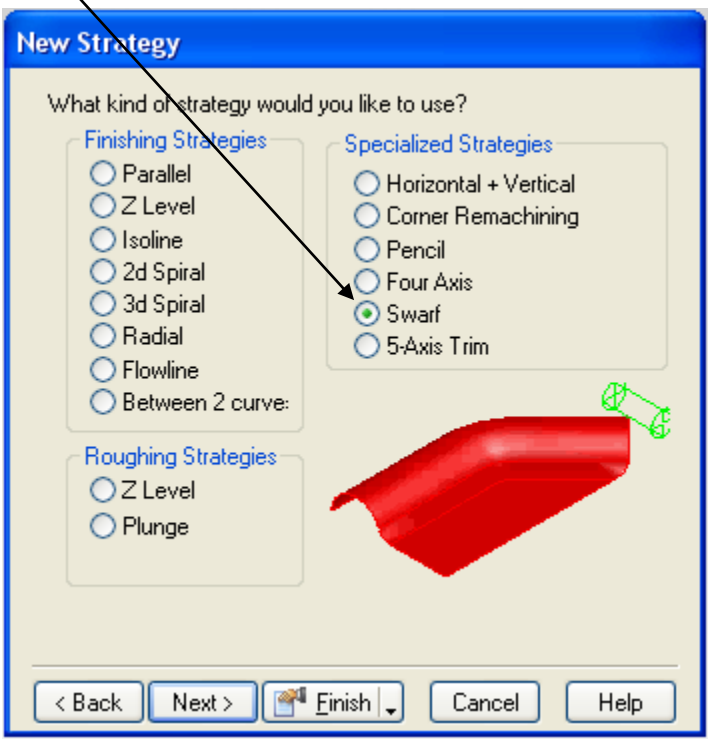
- Click on the green plus to add the selected faces to the feature.



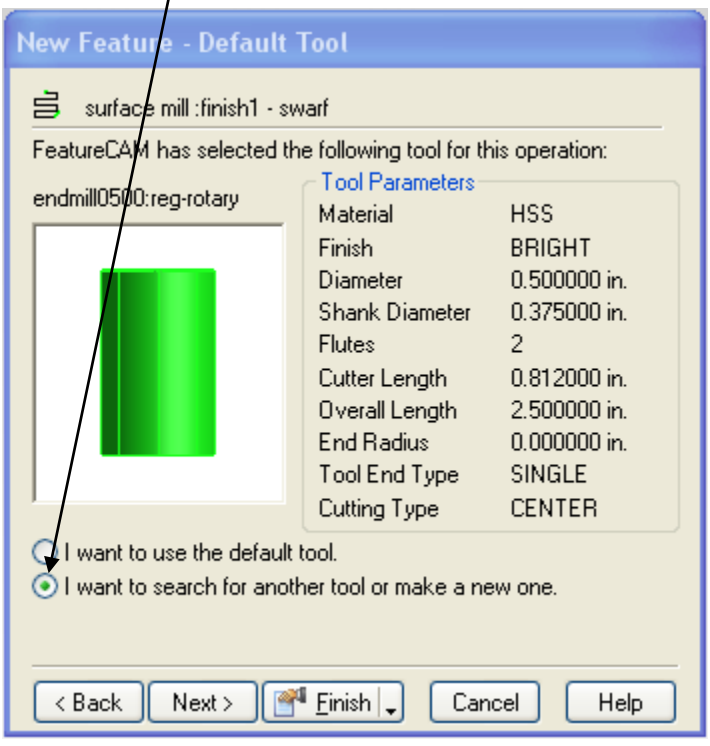
- Click **Next**.
- Check **Choose a single operation**.



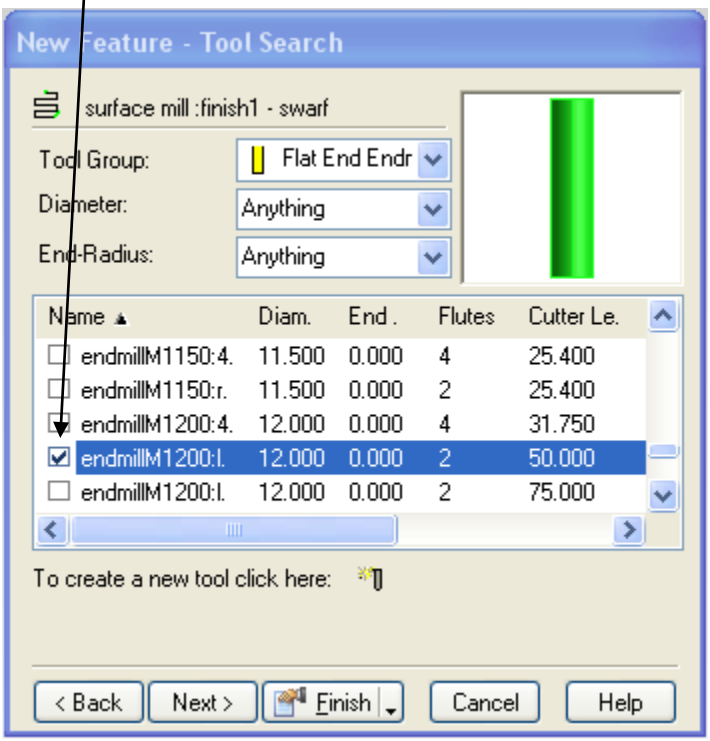
- Check **Swarf** and click **Next**.



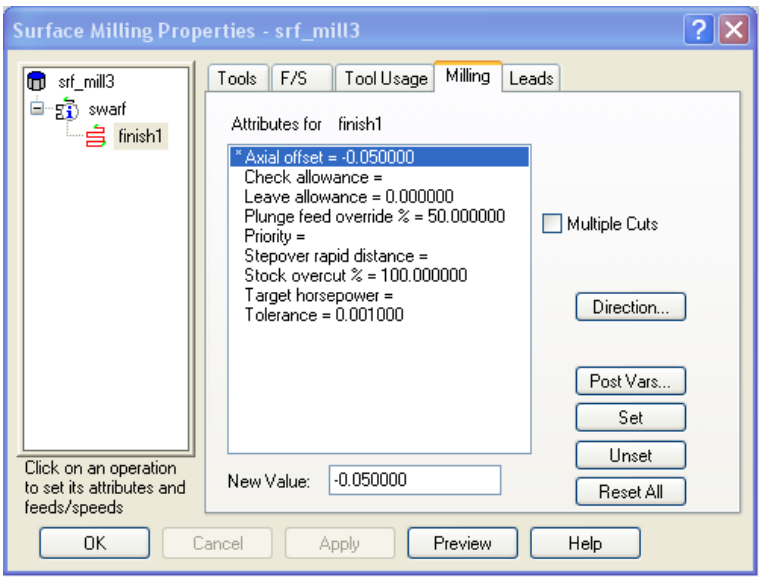
- Click **Next** to the **Default tool** page.
- Check **I want to search for another tool** then click Next.



- Select a **12mm long reach flat end mill**.



- Click **Finish**.
- Select the **finish 1** operation then the **Milling tab**.
- Set the **Axial offset** to **- 0.05**.

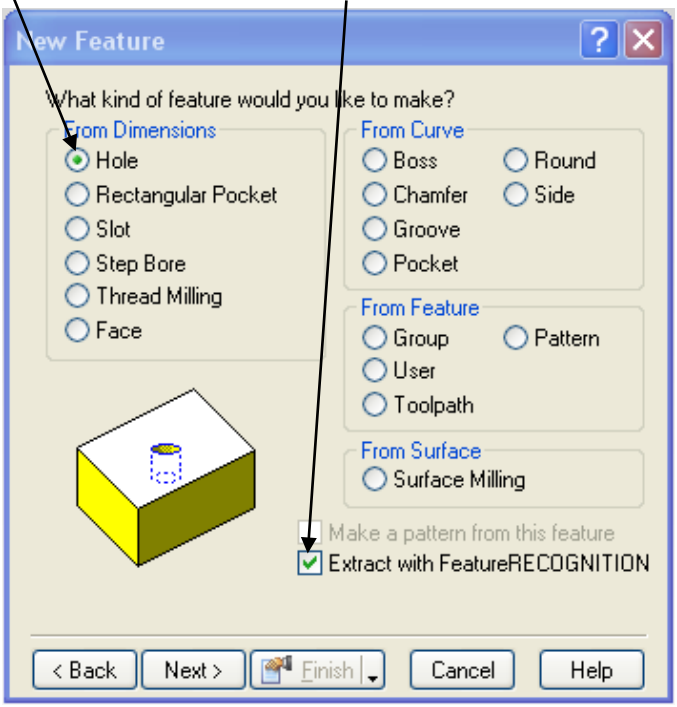


The axial offset will ensure that the end of the tool goes past the bottom of the swarf feature.

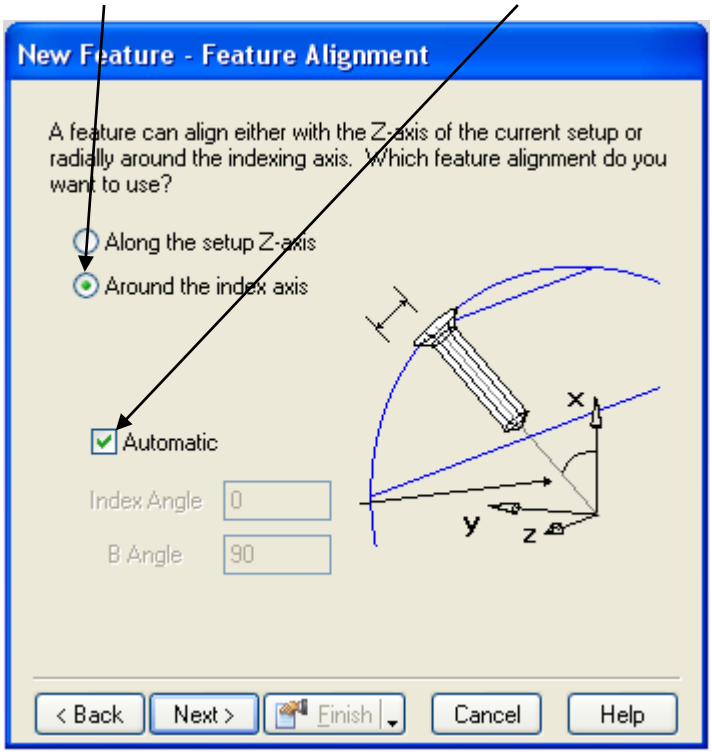
- Click **Preview**.
- **Simulate** the swarf feature.
- **Eject** the simulation.
- Click **OK.**
- Click in the graphics window to deselect all faces.

The final operation is to drill and counterbore the holes on the conical section of the part. These holes will be created using Automatic Feature Recognition.

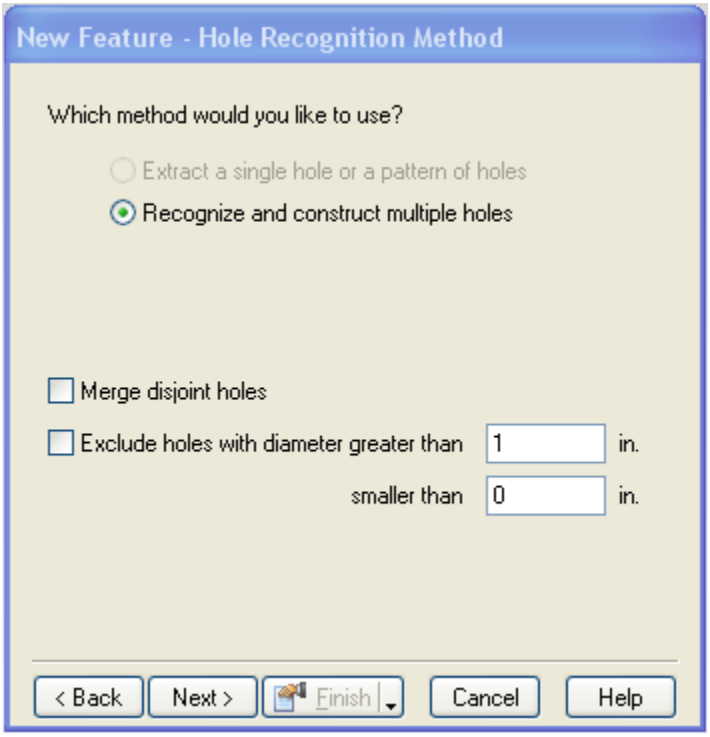
- Create a new **Turn/Mill** feature.
- Check **Hole** and **Extract with FeatureRECOGNITION**.



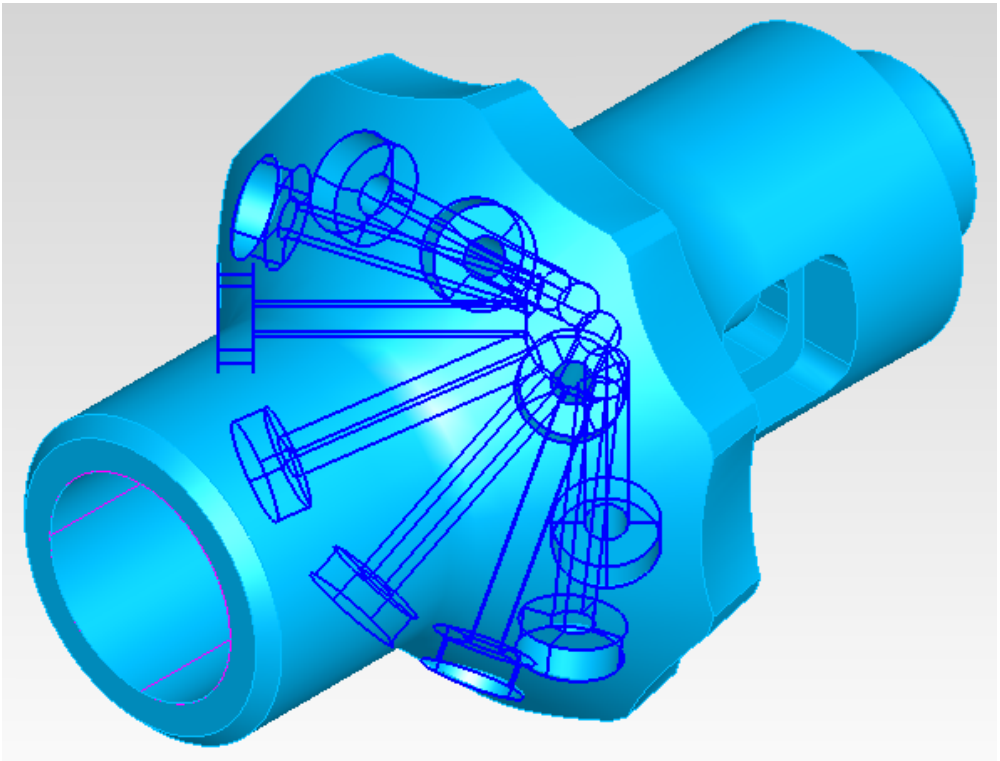
- Click **Next**.
- Check **Around the index axis** and **Automatic**.



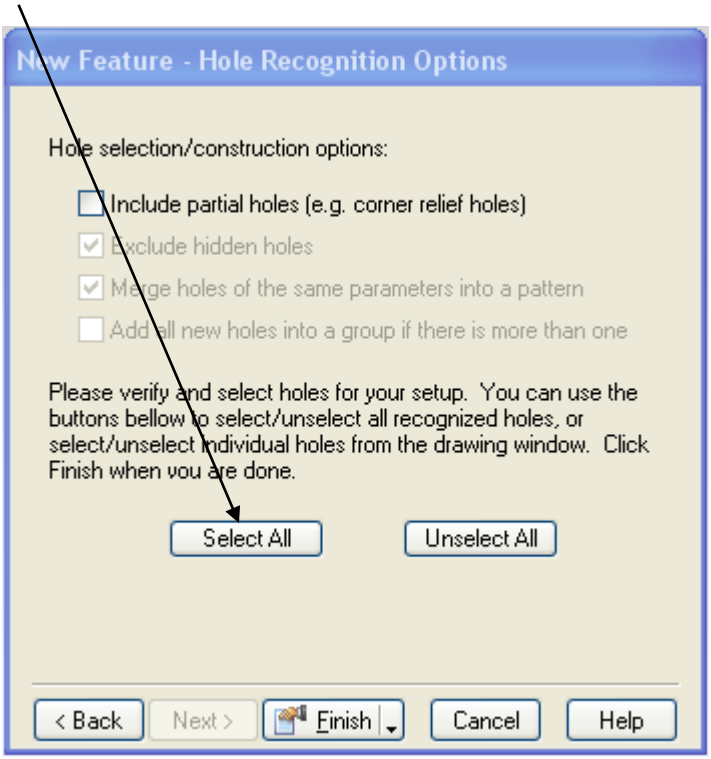
- Click **Next**.
- Check that the form is filled in as below then click **Next**.



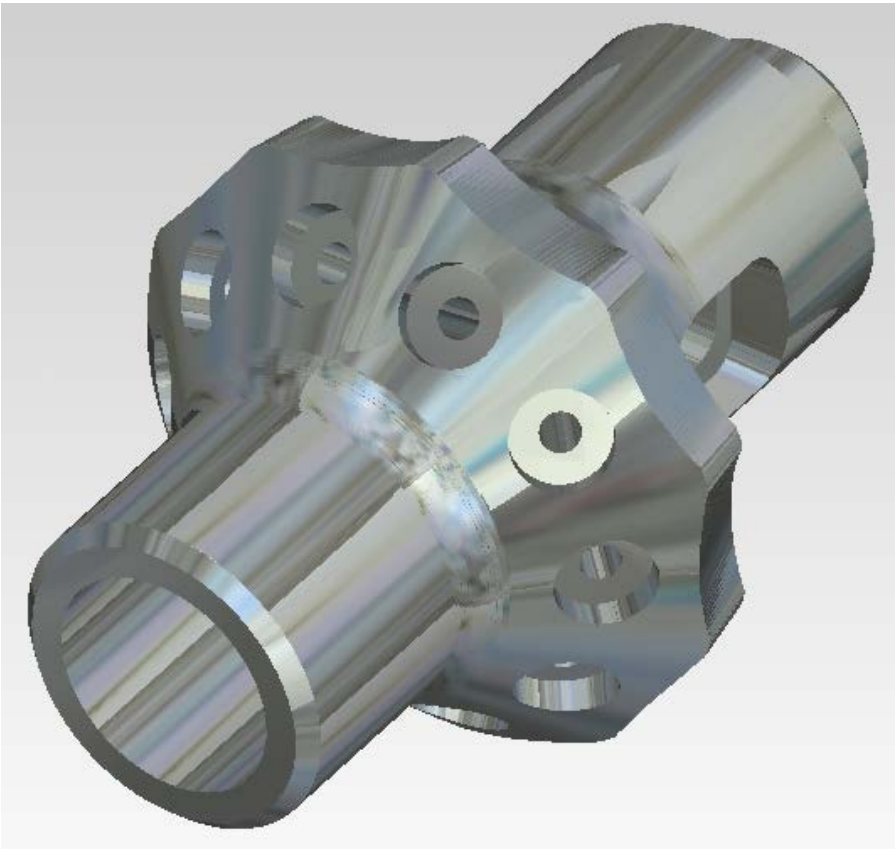
FeatureCAM will now use it's Automatic Feature Recognition to find the holes.
The holes in the part should appear as below.



- Click **Select all** and then **Finish**.



- Run a **3D simulation** using **Simulate to next operation** until the part is transferred to the **sub spindle** then **Play**.

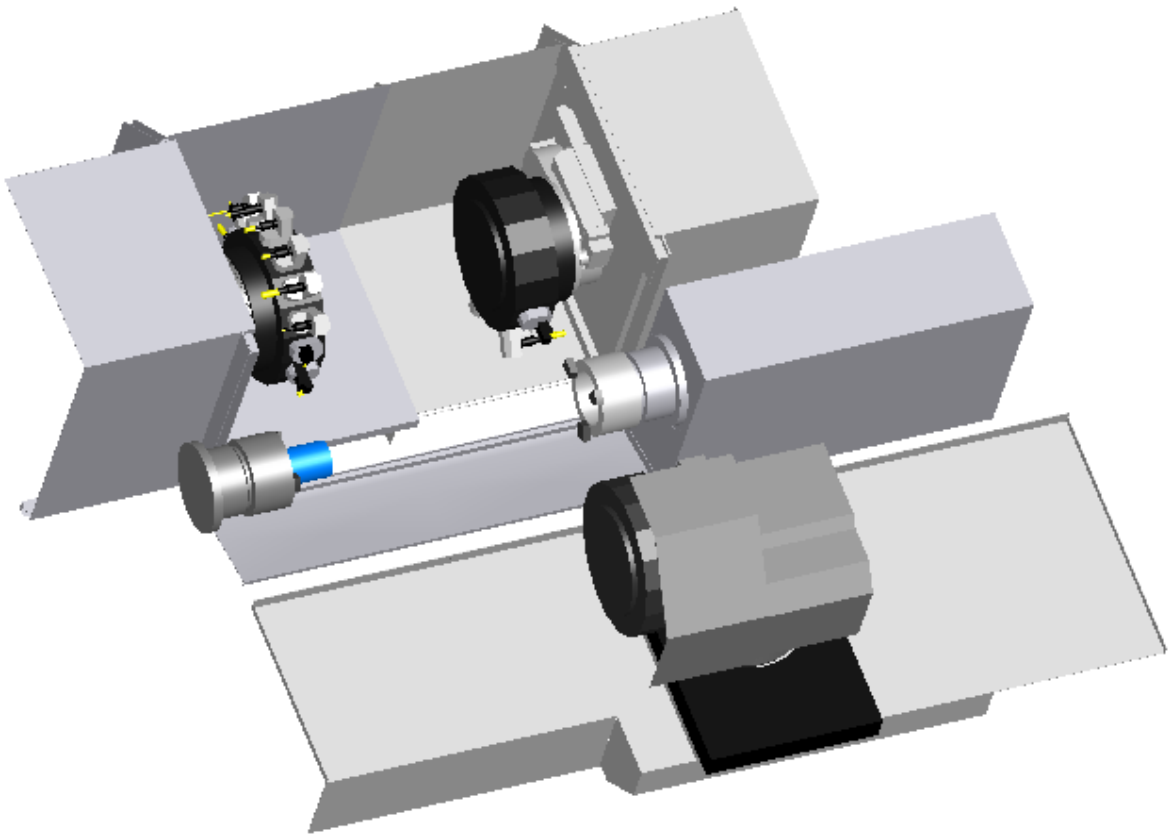


The part is now finished.

Multi-Turret Synchronization

In this document, we will be looking at reducing the machining time for a turn/mill part by synchronizing the motions of the turrets. The part we will be working on already has all of the features defined. The machine we are using will be a Mori Seiki NZ which has three turrets. What we will be doing is to use these three turrets on the machine to cut features simultaneously. By doing more than one operation at a time, we can greatly reduce the overall time taken to finish the part.

The machine has two upper turrets and one lower. The upper turrets work on individual spindles only, the lower turret can work on (address) both the main and the sub spindles.



The above image shows the machine with the main body removed. The main spindle is on the left, the sub on the right. The turret on the upper left is the main turret upper spindle. The one to the upper right is the sub spindle upper turret. These two turrets can only work on the part when it is being held by the relevant spindle. The turret at the lower right can work on either spindle.

If you want to check which turrets can address which spindle for a particular post, you will find this information by opening the post in XBUILD and going to **CNC Info > Turrets**.

First of all, we will open the FM file and load the matching post and .md file.

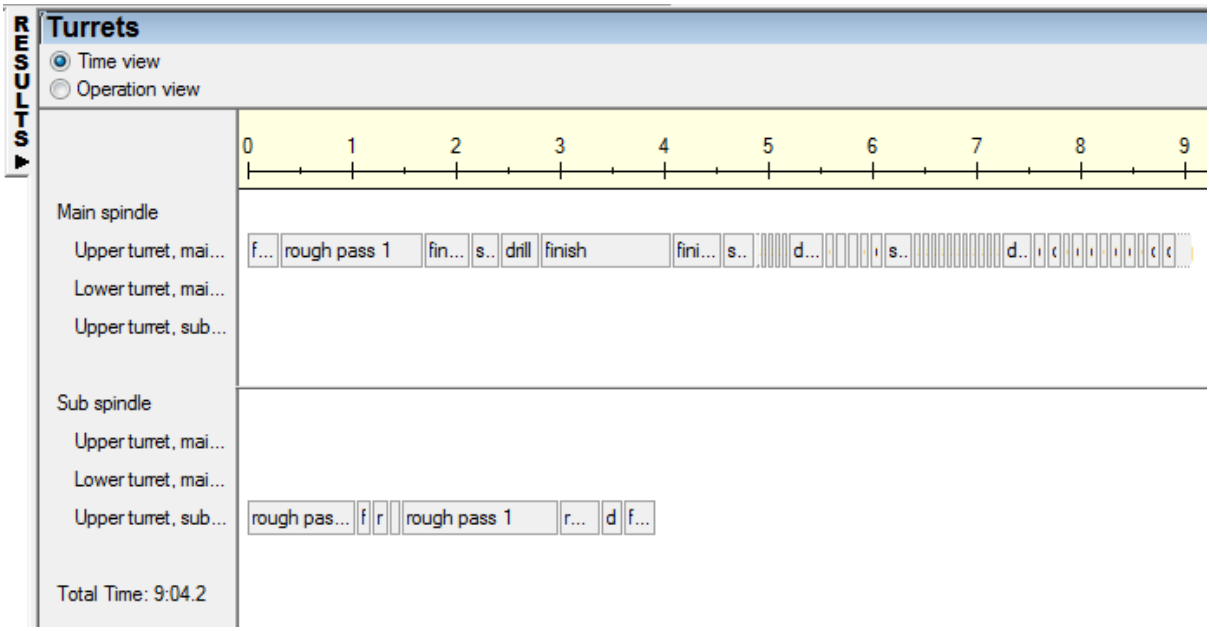
- Open the file **3 turret synchronization start.fm**
- From the same directory load the **Mori Seiki NZ.CNC** file
- Pick the crib **3 turret synchronization start.fm_Tools from last save**
- Run a **machine simulation**

Notice how the operations happen sequentially; the part is first machined on the main spindle and then handed off to the sub spindle for the second series of operations. All of the operations are happening on the upper turrets at the moment.

- **Eject** the simulation
- In the **Results** window click on the **Turrets View** (last tab lower right)

There are two options here: we can select either the **Time View** or the **Operation View**.

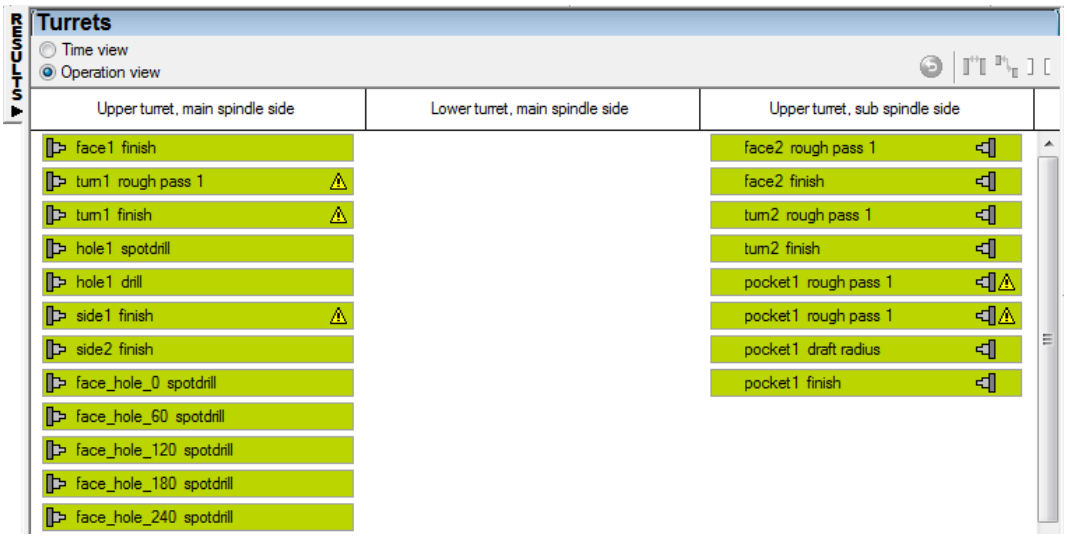
- Select **Time View**



The Time view shows each of the operations along a time line from left to right. The length of each operation is shown graphically as a distance along the time line. In the image above, you can see that the main spindle operations are shown as a group above those on the sub spindle. In each case, the operations are shown on the relevant turret. When there are many operations, it may be difficult to see the names of each operation. The time line scale can be expanded or reduced by using the middle mouse scroll key to enable the names of shorter operations to be seen. The scale along the top indicates the time in minutes taken for each operation. As the operations are shown sequentially, the overall time can be seen by going to the end of the time line or by looking at the Total Time figure which is shown at the lower left of the image above. In this case the time is 9 minutes and 4 seconds.

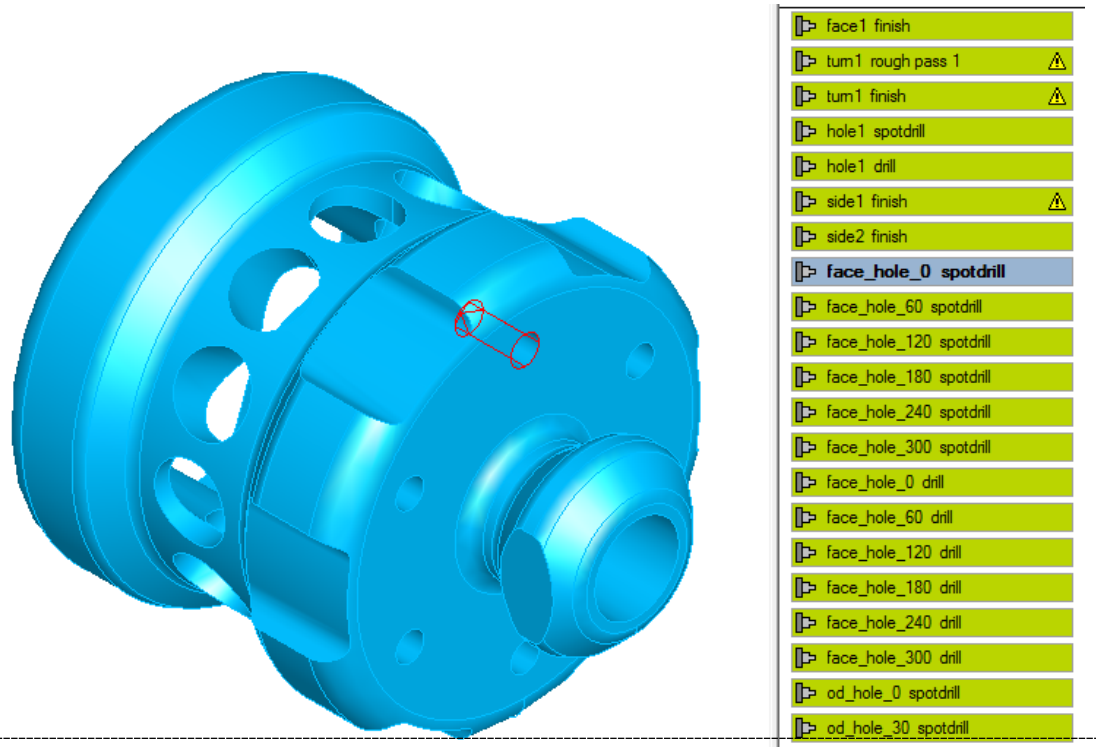
We will now look at the operation view.

- Select **Operation View**



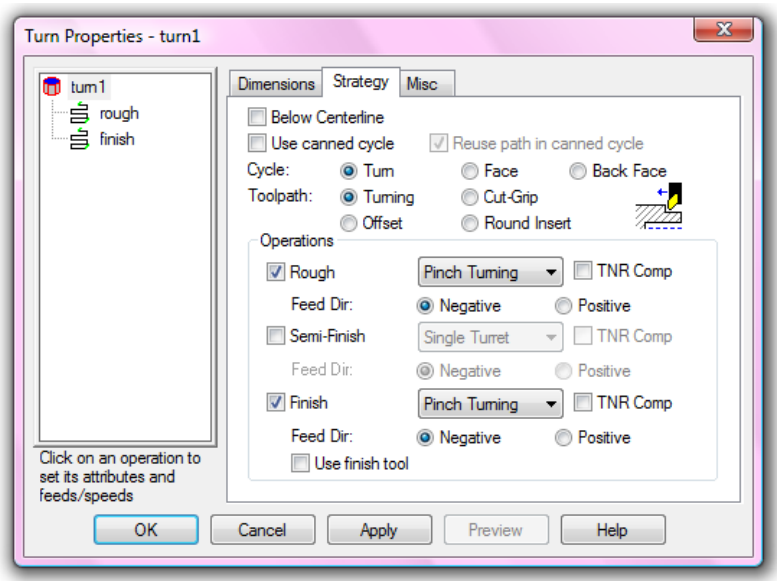
The view now shows the operations as three columns - one for each turret. In this case, the length of each operation is not represented graphically. The operations are ordered from top to bottom; that is, the first operation in each column is going to happen before the second. At the moment, there is no synchronization, so the turrets are working independently of each other. Notice that there are no operations assigned to the lower turret. We will now start to synchronize the motions of the turrets so that more than one operation happens at a time.

- From the View menu **Hide All**
- Now **Show All Solids**
- Click down each of the features in the in turn to familiarise yourself with the location of each one. You will notice that the holes have been individually named so that their positions can easily be located.



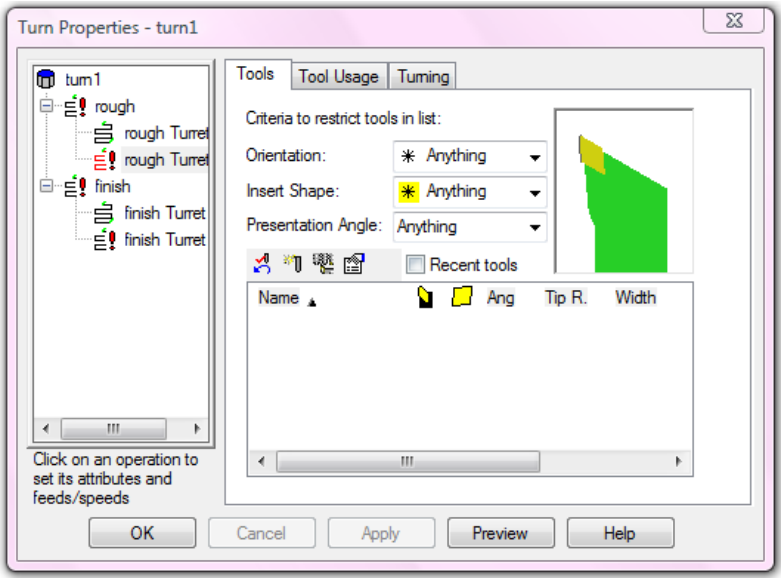
The first change that we will make will be to change the rough and finish turning operations on the main spindle to use **Pinch Turning**. What this will do is to use both the upper and lower turrets on the main spindle simultaneously to turn the OD of the part. As we will be using two diametrically opposed tools, we can effectively increase the feedrate by a factor of two and cut the time in half. This is an example of automatic synchronization; FeatureCAM will set the synchronization points without any manual intervention on our part.

- Double click on the operation Turn1 to open its properties
- Select the Strategy tab
- Set both the rough and finish operations to be pinch turning



- Click **Apply**

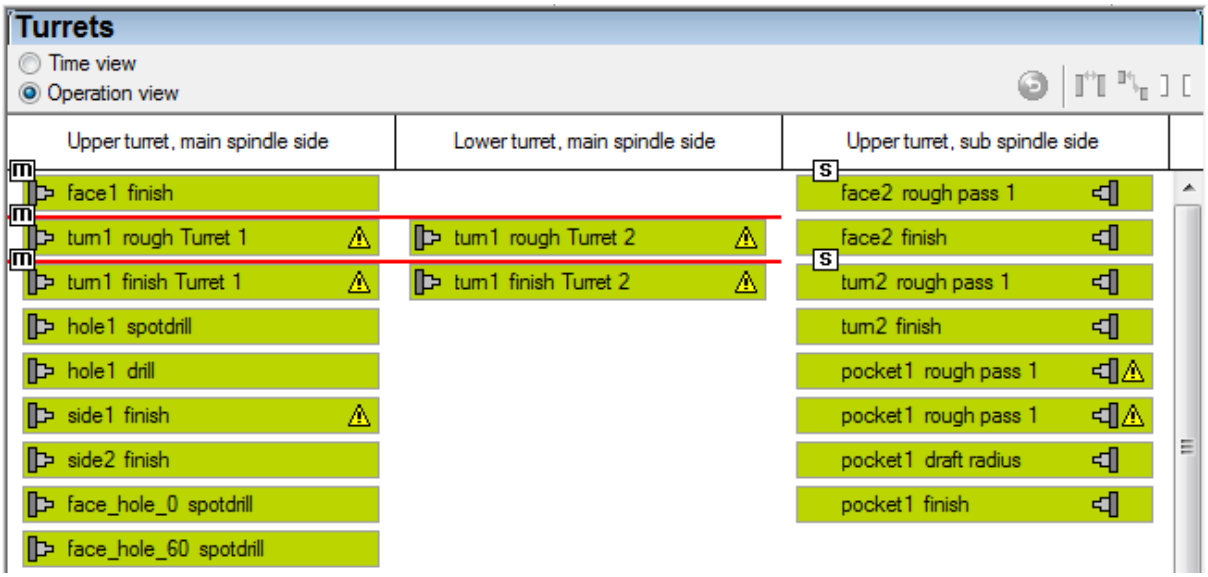
You will see that FeatureCAM has not found tools for the new operations that are on the lower turret. This is because the “Tools from last save” were all upper turret tools so there are no tools available with the necessary orientation to work off the lower turret.



- Change the **tool crib** to **Tools**

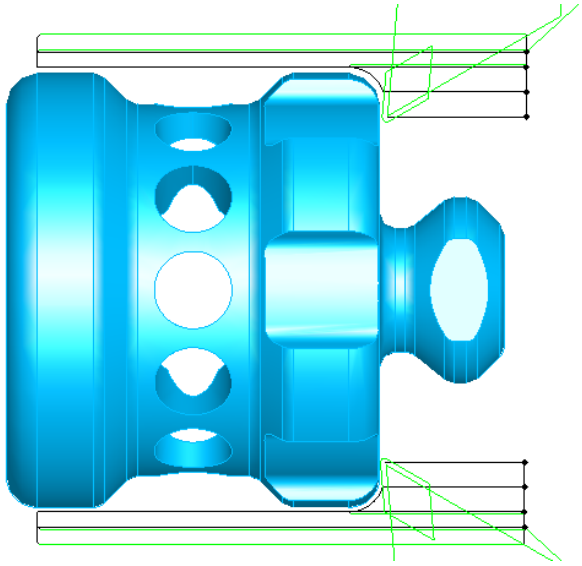
Now that there are lower turret tools available the warnings disappear. As the tools in the original crib all had overrides their selection is not changed.

- Click **OK** and check the **Turrets View**



Now you can see that there are two new operations on the lower turret on the main spindle side. The horizontal red lines mean that the operations are synchronized at their start. The small M and S icons indicate which turret is going to be controlling the feedrate for a particular operation. This is important where using constant surface speed (CSS) in a turning operation as the distance of a tool from the center of the part will be controlling how fast the spindle is turning.

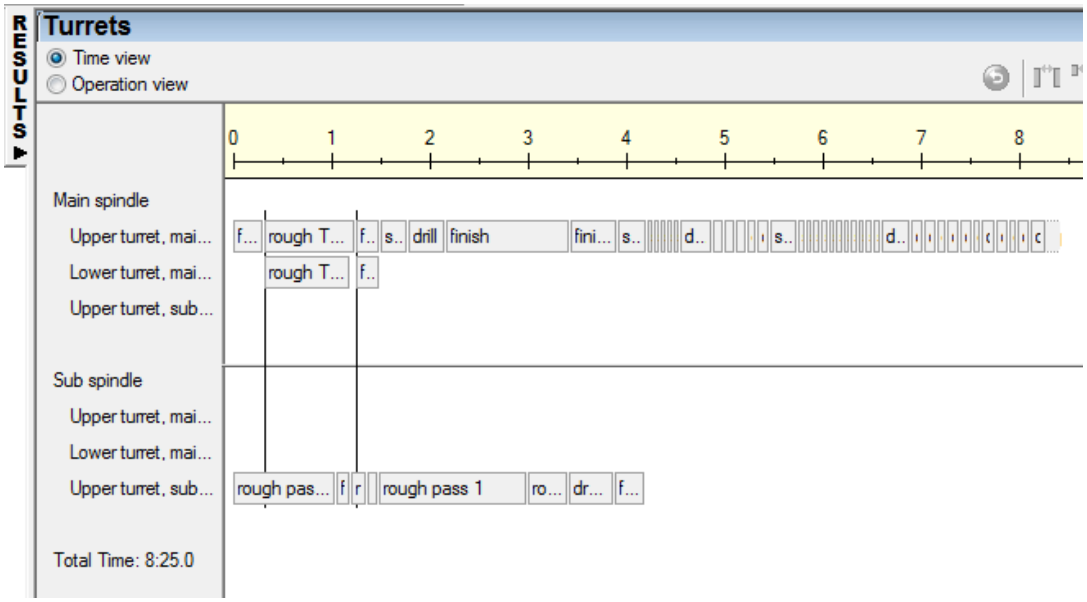
- **Preview** the operation **turn1**
- **Single step** a **centerline simulation**



Note how the two tools are now moving in synchronization with each other. Both the upper and lower turrets are working together to clear the material from the part. The black dots that appear at the beginning of each pass are **Wait Codes**. That is the post processor will issue a code that tells the machine tool controller that it should not start the next operation until the last one has finished and both turrets are ready in position to start the next pass.

- Turn all of the operations back on

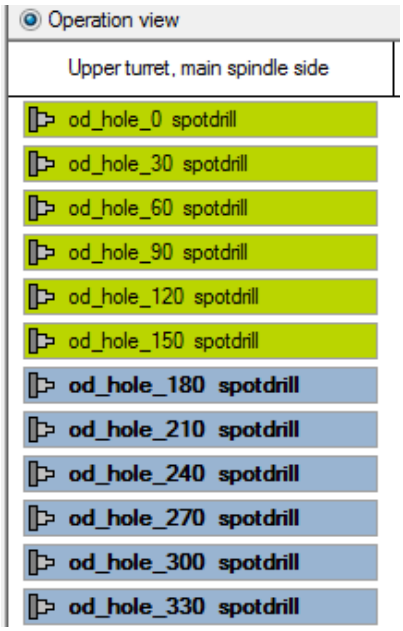
- Run a **centreline simulation**
- Select the **Time View**



Note that on the Main Spindle timeline we are now using both the upper and lower turrets for the rough and finish turn operations. The Total Time figure is now 8 minutes and 25 seconds, a saving of 39 seconds.

We will now look at manually synchronizing some of the drilling operations on the main spindle. We will use both the upper and lower turrets to drill the OD holes simultaneously. The first thing to do is to allocate which hole is to be machine from which turret. We will allocate half of the OD holes to the upper turret on the main spindle and the remainder to the lower turret.

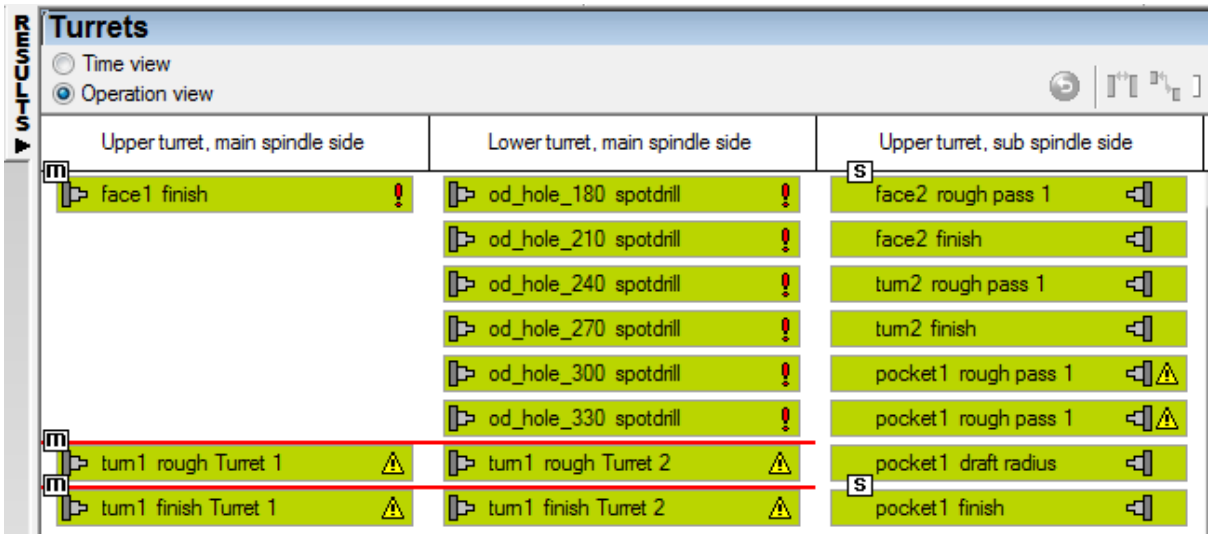
- Select the OD holes indicated below (180 through 330). Use the control key to multiple select the holes.



We are selecting these Spotdrill operation on the OD so that we can move them over on to the lower turret. We will then need to pair them up with their opposing holes on the upper spindle so that they can be drilled simultaneously.

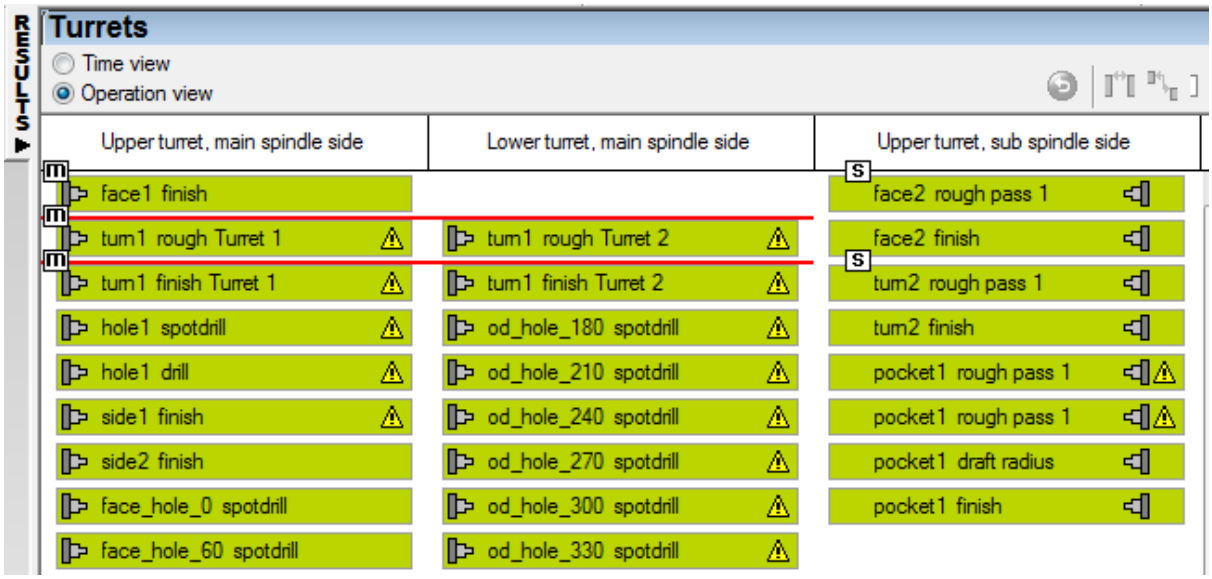
- **Holding down the control key drag and drop**

the operations into the center column, this will move the operations onto the lower turret



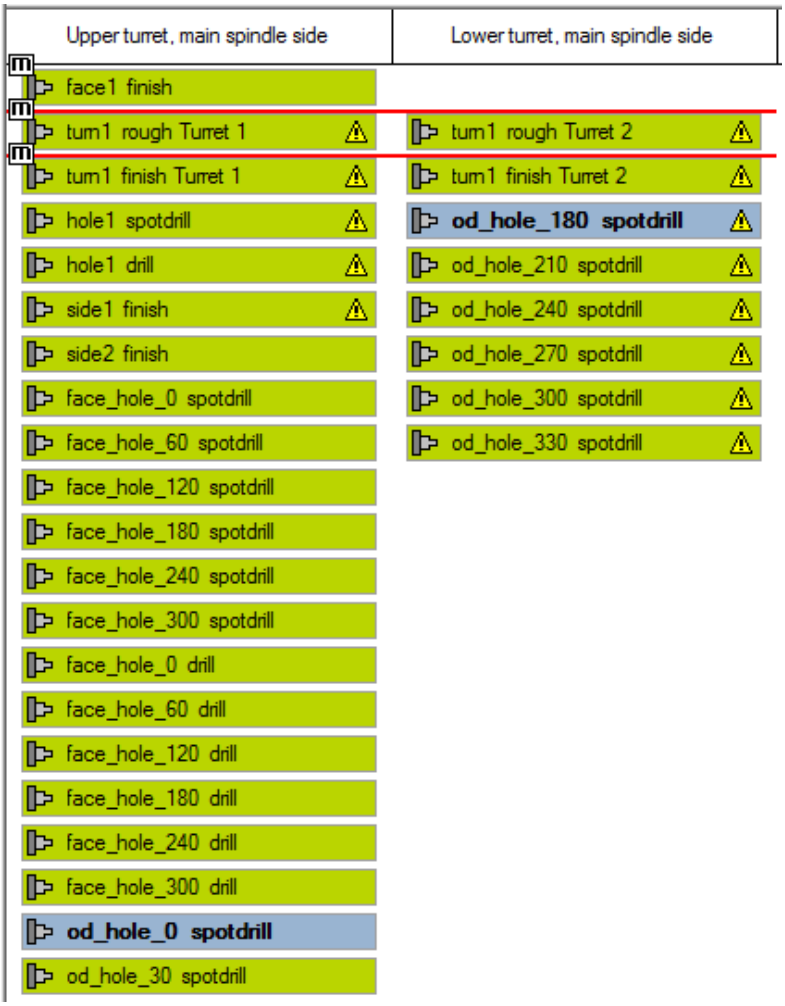
Note the red exclamation marks indicating a warning. In this case the problem is that the current timings mean that we are trying to carry out turning and milling operations at the same time, obviously this is not possible. We now need to manually reorder the operations so that they happen in a sensible sequence.

- **Control select** the OD holes on the lower turret and then drag and drop them below the turn1 finish Turret 2 operation as shown below

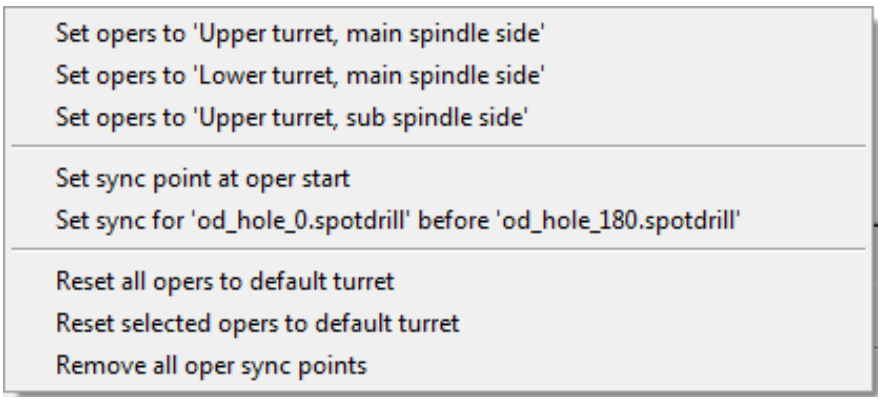


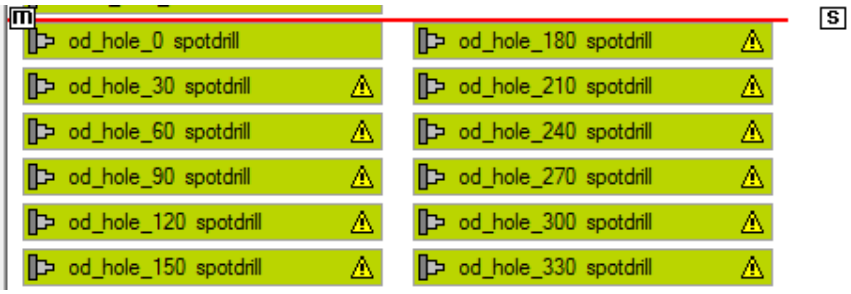
We can now start matching up the upper and lower turret motions for opposing holes. We will be first matching **od_hole_0 spotdrill** with **od_hole_180 spotdrill**.

- Click on **od_hole_0 spotdrill**
- Hold down the control key and click on **od_hole_180 spotdrill**



- Right click on **od_hole_0 spotdrill**
- Select **Set synch point at operation start**

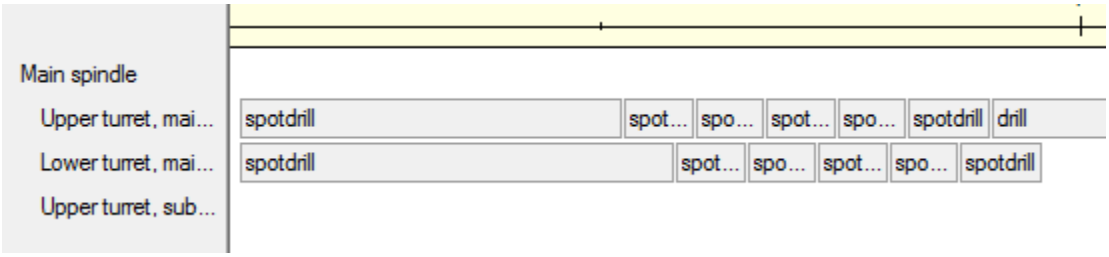




The operations **od_hole_0 spotdrill** and **od_hole_180 spotdrill** are now synchronized. Note the warnings against the other holes. This is because they are not synchronized. As the holes are different depths they will take different times to machine. This will result in the hole drilling operations getting out of synch so that when one is finished it is not ready to start the next opposing pair of holes.

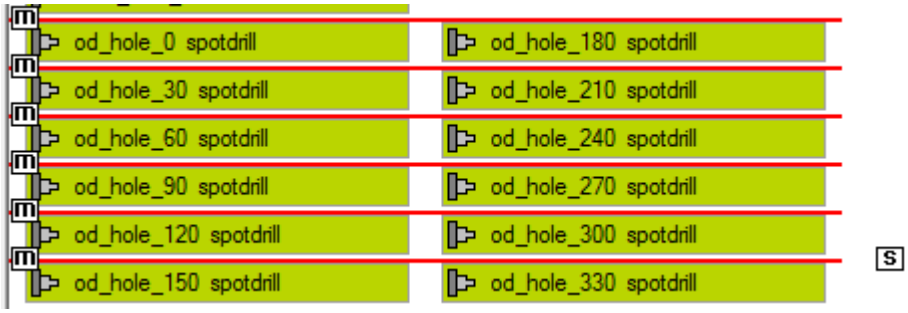
- Select the **Time View**

You can see this problem clearly in the time view. Note how the first two spotdrill operations start simultaneously but end at different times. As each operation follows on they get gradually more and more out of step.



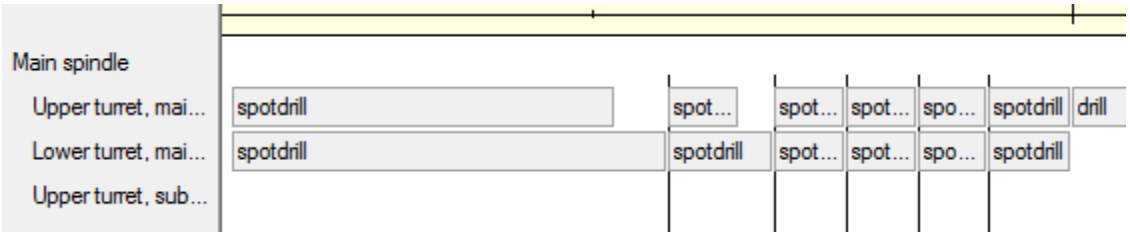
We need to match all of these spotdrill operations in pairs. This will introduce **Wait Codes** into the program. A wait code which will not allow the machine to move onto the next operation until the previous one on the opposing spindle is completely finished.

- Click on **od_hole_30 spotdrill**
- Hold down the control key and click on **od_hole_210 spotdrill**
- Right click on **od_hole_30 spotdrill**
- Select **Set synch point at operation start**
- **Repeat** for the other OD spotdrill operations until they are all matched as shown below, (60-240, 90-270, 120-300 and 150-330)



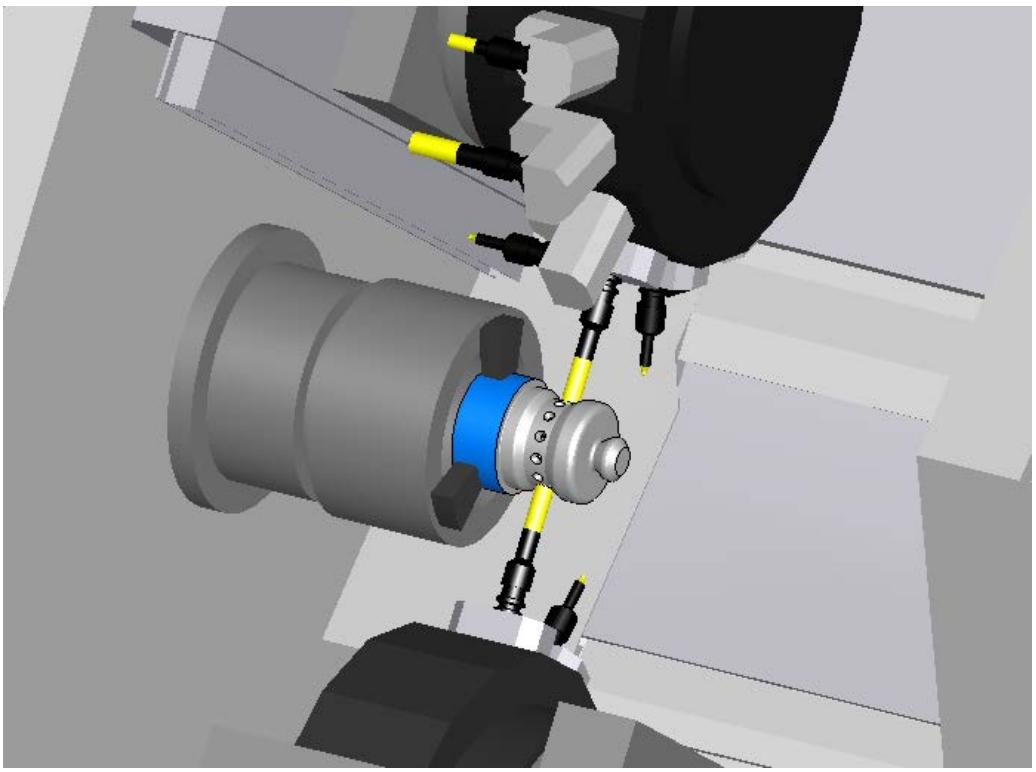
Note how all of the warnings have now disappeared.

- Switch to the **Time View**



Note how the spotdrill operation on the upper turret takes a shorter time than that on the lower. By synchronizing the operations we are forcing the upper turret to wait until the lower has finished. They then move on together to the next pair of holes.

- Repeat the synchronization process for the operations **od_hole_0 drill** through **od_hole_330 drill** matching 30-210, 60-240, 90-270, 120-300 and 150-330.



- **Run** a **Machine simulation**

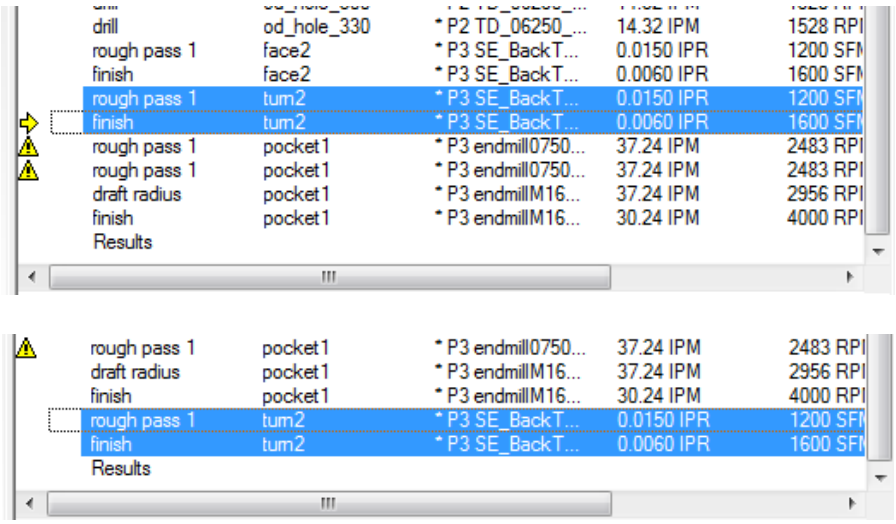
You will see that the two turrets work together, if one finishes before the other it waits and then the two drills move on simultaneously to start the next drilling operation. Note, if you get a collision it may be necessary to change the tool mapping on the lower turret.

- Select the **Time View**

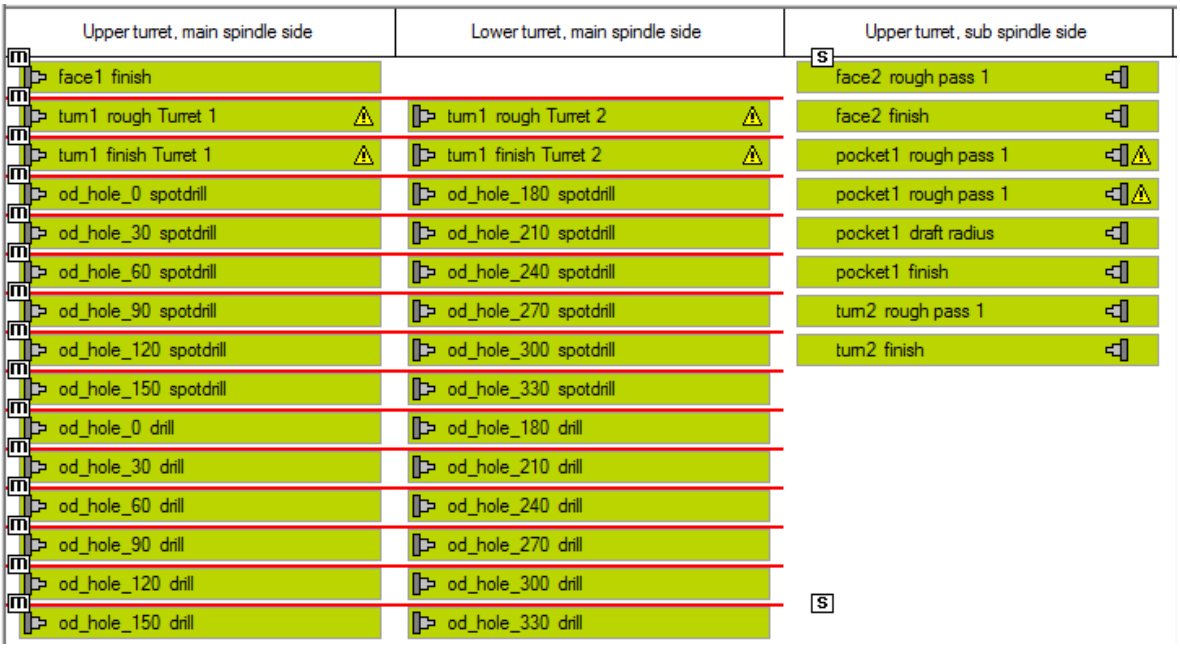
You will see that the Total Time for the part is now down to 7 minutes and 23 seconds, a saving of 1 minute and 41 seconds over the original operation sequence.

We will now work on the operations on the sub-spindle. We will change the turn operation to be Pinch so that we use the sub-spindle upper turret and the lower turret simultaneously. In order for this to work we need to move the **turn2 operation** to a position in the operation ordering where the lower turret has finished working on the main spindle operations and becomes available to work on the sub-spindle. We can change the operation order in several ways; in this case we are going to move them in the operations list.

- Go to the **Ops list tab**
- Select the two operations that make up turn2
- Drag and drop them to the end of the program

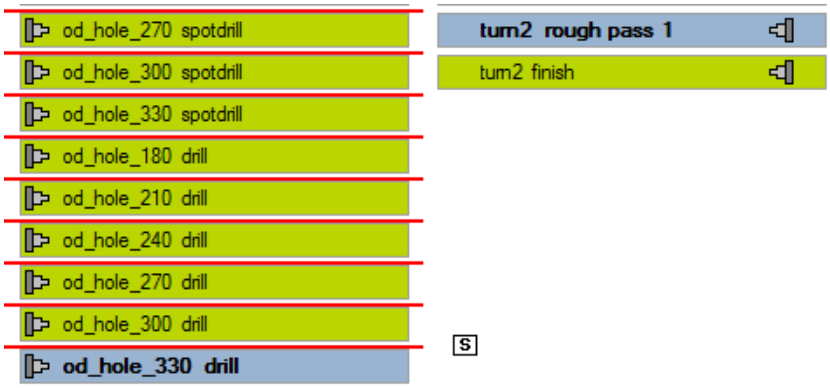


- Go back to the **Turrets tab Operations view**

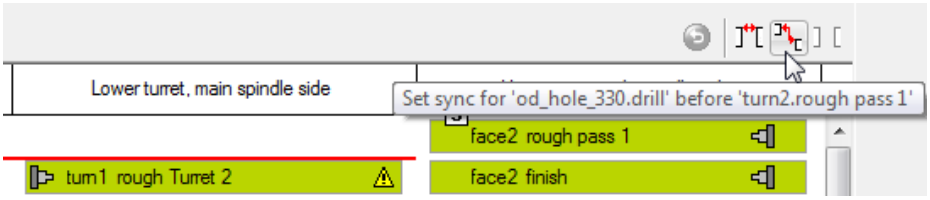


We now need to add a synchronization point so that the **turn2 rough pass 1** operation happens after the **od_hole_330 drill**. This will introduce a wait code so that the sub-spindle will machine the face & pocket operations using the sub-spindle upper turret and then stop until the lower turret has finished all of the drilling operations.

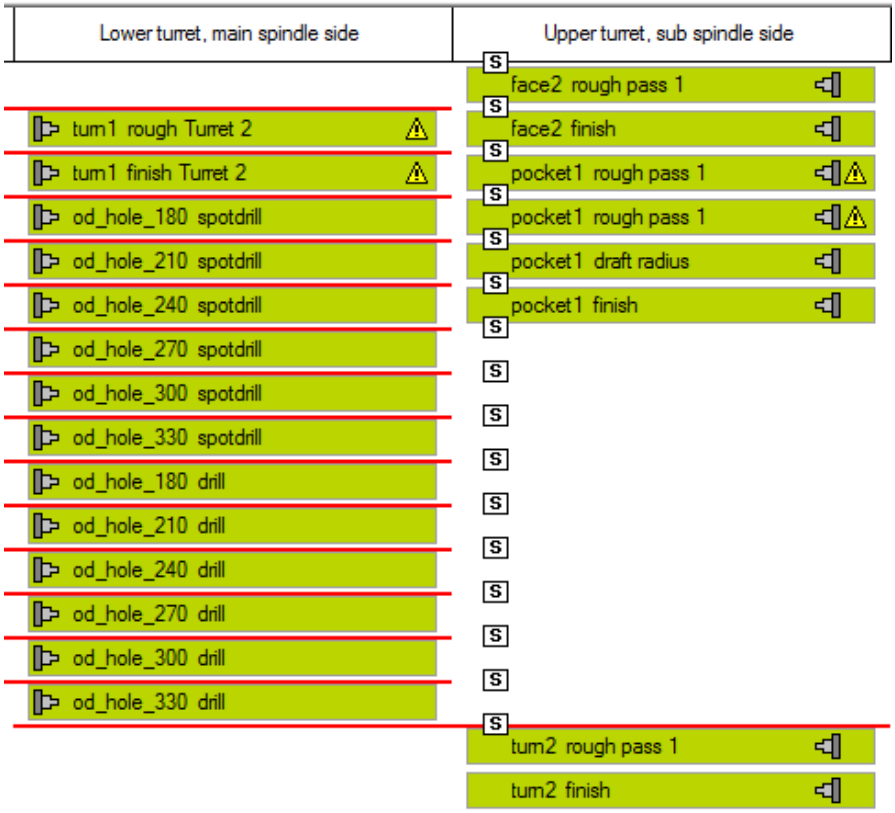
- Hold down the control key and left click on **od_hole_330 drill**
- Hold down the control key and left click on **turn2 rough pass 1**



- Click on the icon indicated to place the operations in the correct order

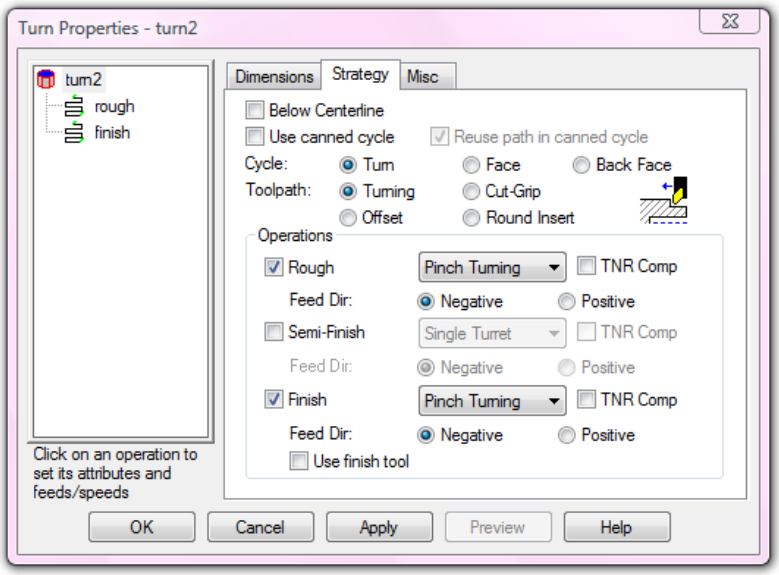


The order in which you select the operations is important. They will be reordered in the order that you selected them. This is why we selected the feature **od_hole_330 drill** before **turn2 rough pass 1**. Note that as the operations **turn2 rough pass 1** and **turn2 finish** are part of the same operation they will be moved together even though we only selected one.

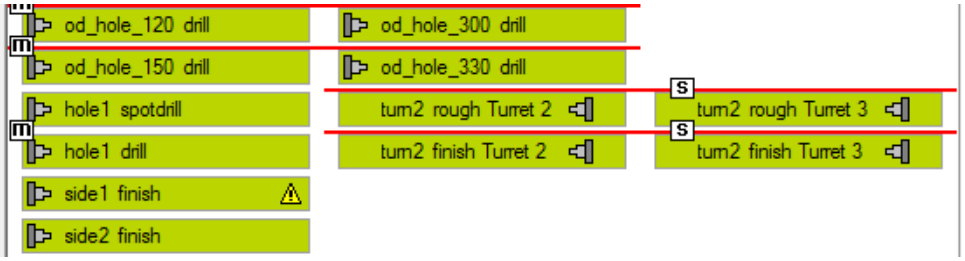


We will now change the rough and finish operations in turn2 so that pinch turning is used.

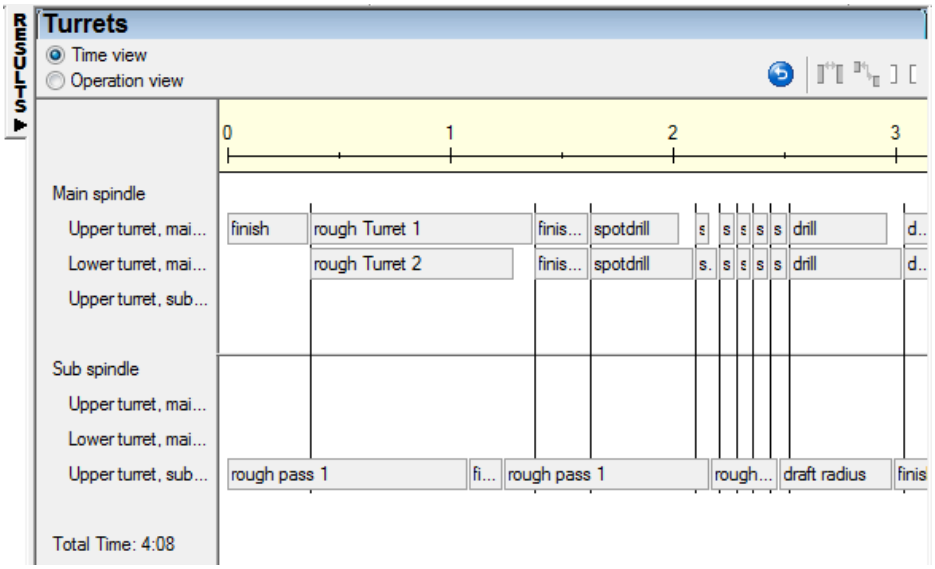
- Double click on the operation Turn1 to open its properties
- Select the Strategy tab
- Set both the rough and finish operations to be pinch turning



- Click **Apply** and then **OK**



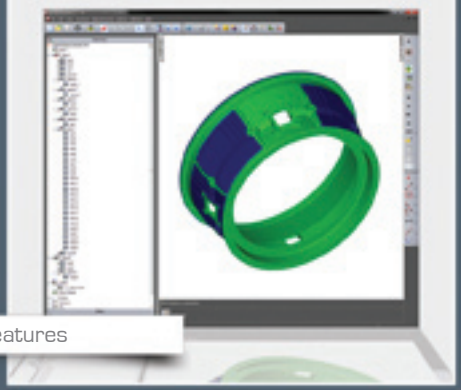
- Go to the **Time View**



The Total Time is now down to 4 minutes and 20 seconds from the original 9 minutes and 4 seconds giving a total saving of 4 minutes 44 seconds or approximately 52%.

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**