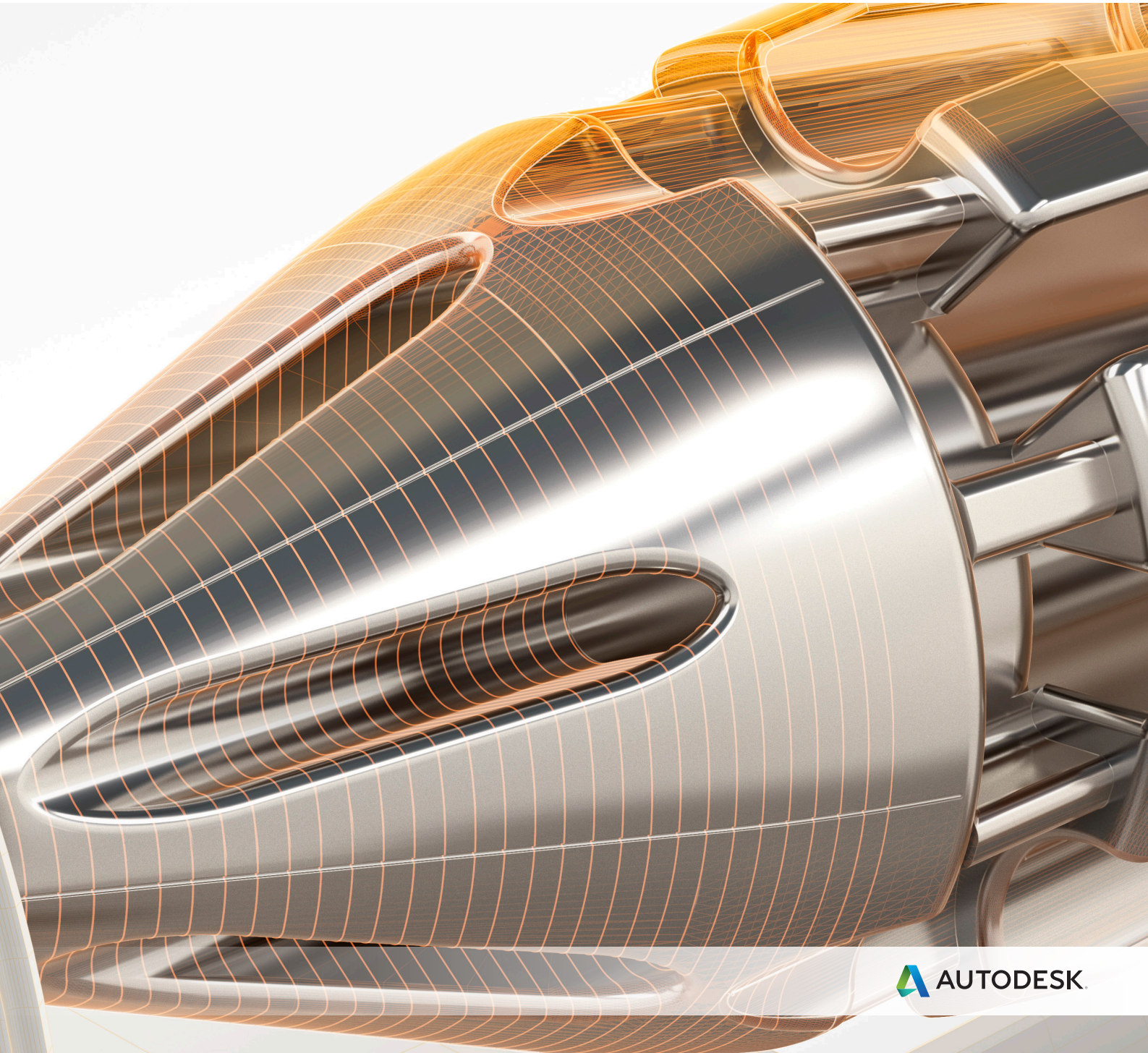
AUTODESK® **FEATURECAM®**

Automated to make parts faster

Training Course



AUTODESK.

Contents

Introduction to FeatureCAM Turning 3
CAM Software Strategies 4
Feature-Based Machining 5
Built in intelligence 6
Starting FeatureCAM 7
Evaluation Options 7
Desktop Subscription 12
Access options 12
Flexible term lengths 12
Access to the latest software 12
More about updates and releases 12
Use your software in more places with access to both current and preceding versions. 12
Autodesk support 13
Starting a new FeatureCAM part document. 13
User Interface 13
Screen Layout 14
Steps Menu 17
Steps to take to create a part in FeatureCAM 17
Importing a file into FeatureCAM 18
Stock 18
Steps (See menu in PartView 19
ViewCube 22
Manipulating orthogonal views 22
Mouse buttons 23
Left mouse button Picking and selecting. 23
Middle mouse button Dynamics. 24
Right mouse button Special Menus & FeatureCAM Options. 24
Viewing Options 25
Reset to PowerMILL Settings 26
Viewing Options General 26
Viewing Options Machine 27
Display options 28
Principle views 28
Keyboard shortcuts 30
Commonly used Ctrl Keys 32
User interface: Keyboard shortcuts 32
Blanking model entities 33
Show menu 33
Hide Menu 34
Setting file location options 35
What is a Feature? 37
How is a Feature Created? 39
From Dimensions 39

© Copyright 2016 Autodesk, Inc. All rights reserved

From Curve 40
Saving your work 44
Save Options 44
Send Part Files dialog 45
FeatureCAM file types 46
Part Documentation 47
Saving your settings 48
Import/Export 49
Importing Files 51

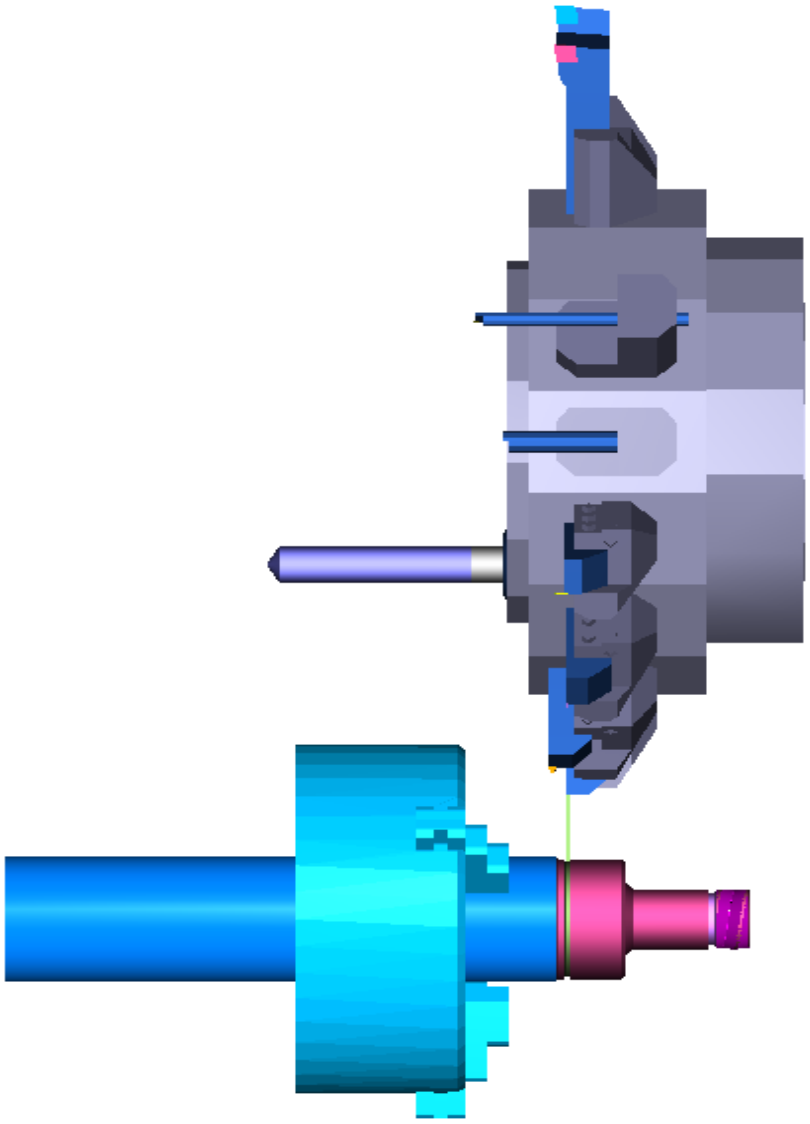
© Copyright 2016 Autodesk, Inc. All rights reserved

AUTODESK® FEATURECAM®

Introduction to FeatureCAM Turning

*This module will familiarize you with **Autodesk FeatureCAM's** history, describe what **FeatureCAM** is and show how it is advantageous over other CAM systems.*

*Increased shop productivity is the primary goal of all **FeatureCAM** products. Traditional CAM systems are **Operations Based** and require you to program every operation, one at a time, to create your part. All the manufacturing details are left to the machinist. That means that you must specify the details for every Spot Drill, Drill, Ream and Roughing pass of your part.*



© Copyright 2016 Autodesk, Inc. All rights reserved

*With **FeatureCAM** you generate your part using **Features** such as a **Tapped Hole, Boss,** or a **Turned Groove** and the operations are automatically created. **FeatureCAM** also manages the details of the manufacturing process such as Tool selection, Speed and Feed Rates, and Toolpaths. To modify any element of the part program, just change a few settings on a feature and a whole new set of operations are generated to reflect your changes.*

***Ease of use** is another guiding principal of **FeatureCAM** products. Our system was originally developed as a graphical Windows-based program. **FeatureCAM** programs are full of graphical feedback, **Step-by-Step Wizards** and tutorial-style animations that run directly in the program dialog boxes. Ease of use has always been a **FeatureCAM** advantage and we continue to study ways to make our system even easier to learn and use.*

CAM Software Strategies

*There are three main different types of **CAM** software strategies for creating toolpaths in today's modern **CAM** software. These strategies are **Operation based**, **Processed based** and **Feature Based**. Some of these strategies use **Knowledge Based Machining (KBM). KBM** is when machining intelligence is built right into the **CAM** software. **KBM** is used at various levels and degrees in different **CAM** systems.*

***Operations-based CAM** systems require the user to go through multiple steps to machine each part. The user must select which type of operation to use, select the machining boundary and the type of toolpath, then manually select tools, feeds and speeds and multiple machining options such as stepover distance and the incremental step in Z. These multiple steps must be repeated one at a time for every operation on a part, making it a time-consuming process that is prone to error. These systems offer little or no **KBM**.*

***Process-based CAM systems** attempt to reduce these steps by grouping operations in standard processes. **KBM** is used as the program remembers the used processes and repeats them for future operations. However, this method only works when a shop uses the same processes over and over for the same types of parts.*

*When a process is applied to different types of parts, or parts made of different materials, the **CNC programmer** must still verify that the correct operations were used and that the correct tools, stepover and speeds and feeds were selected because a static machining process does not automatically adapt to every part.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Below is a chart showing a comparison of the three different CAM strategies:

Feature Based CAM ✓	**Operations-Based CAM**	**Process-Based CAM**
1. Draw or Import Your Part **2.** Identify Features **3.** Click Simulation **You Are Done!** **FeatureCAM Automatically:** • Determines Rough and Finish Operations • Selects ALL Tools Sizes • Calculates Feeds & Speeds • Determines Stepover & Z Increments • Generates Toolpath • Creates NC Code	**1.** Import or draw part. **2.** Select a rough operation. **3.** Select rough boundary. **4.** Select type of toolpath. **5.** Select roughing tool. **6.** Select feeds and speeds **7.** Select stepover and Z steps. **8.** Select a finish operation. **9.** Select finish boundary **10.** Select type of toolpath **11.** Select a finishing tool. **12.** Select stepover and Z steps. **13.** Select feeds and speeds. **14.** Click simulation. **15.** Create NC code	**1.** Import or draw part. **2.** Identify features. **3.** Select machining processes. **4.** Click simulation. **5.** Create NC code. **???Are You Done? No.** You must analyse and verify: Does the process have the correct operations? Does the process use the correct size tools? Are the correct feeds and speeds used? Does the process have the correct stepover and Z increments?

Feature-based CAM systems use a set of interrelated machinable features to describe a complete part. The beauty of features is that they not only describe shape, but these also are made up of one or more associative operations that describe the preferred method for cutting that shape at the NC machine. A predefined set of machining rules and user preferences are applied to a part. KBM in these systems streamlines the manufacturing process by building the highest level of machining intelligence right into the CAM system. 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 Centre drill and preferred machining strategies for roughing and finishing. The CAM system evaluates the part geometry and part material, selects the most appropriate tools and operations, recommends machining strategies, calculates feeds and speeds, then automatically generates the NC code.

Feature-Based Machining

Has many advantages such as:

1. Dramatically shorter part programming times.
2. Part revisions do not require reprogramming.
3. NC code is consistent and predictable regardless of the CNC programmer.
4. Tools are optimized and tool changes reduced.
5. Toolpath is automatically optimized for faster machining.

© Copyright 2016 Autodesk, Inc. All rights reserved

__FeatureCAM__ is a Feature Based programming system with advantages listed above. Additionally __FeatureCAM__ is very easy to use. A programmer new to __FeatureCAM__ or new to programming can quickly learn the product and create G-code with the appropriate tools, feeds and speeds.

Below is a list of key points that make **FeatureCAM** very easy to use.

1. **User Friendly Interface (Steps & Wizards).**
2. **Automatic Tool Selection (2D & 3D).**
3. **Automatic Feature RECOGNITION (AFR).**
4. **Interactive Feature RECOGNITION (IFR).**
5. **Standard Databases (Tools, Materials, Feeds & Speeds).**
6. **Customizable Post Processors.**
7. **Outstanding Technical Support.**

Built in intelligence

A major advantage of __FeatureCAM__ is the users' ability to define their own set of machining preferences. __FeatureCAM__ comes with predefined machining rules programmed into the CAM system which are based on a standard set of material properties and standard tool cribs. You can then customize these settings for specific materials that you use and the tooling that you actually have available in your shop. Once you have these defined, you simply select a material for your work piece and the tool crib you want to work with and __FeatureCAM__ will then make selections of tooling for a feature automatically. If you change the tool crib or the material then __FeatureCAM__ automatically updates all of the toolpaths in the part.

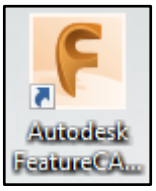
This means that a part may be programmed for one machine with the tools available there and then shifted to another machine simply by changing the tool crib. Similarly if you wish to cut a test part in Aluminium and then cut the final part in Stainless Steel you program the part with Aluminium set as the material. Once you have cut the test piece you change the material to be Stainless Steel and __FeatureCAM__ will update the toolpaths by changing feeds, speeds, depth of cut, stepover etc. without any further intervention from the operator.

By providing a set of standard machining rules already in place, it's easier for you to customize the software to reflect the machining preferences of your Machine shop.

Setting up your own machining preferences and rules in __FeatureCAM__ not only reduces the time it takes to generate programs, but it also streamlines production because the approach to machining every part is predictable and consistent. Machine operators can rely on the fact that each job will be machined in a predictable manner regardless of who did the programming

© Copyright 2016 Autodesk, Inc. All rights reserved

Starting FeatureCAM

1 Select the FeatureCAM icon 

from your desktop and select **New File** and **Milling Setup** and **Millimetre**. Select **Finish** twice and then **Ok** to close the Stock menus. The stock menu will be covered later on in the training notes.

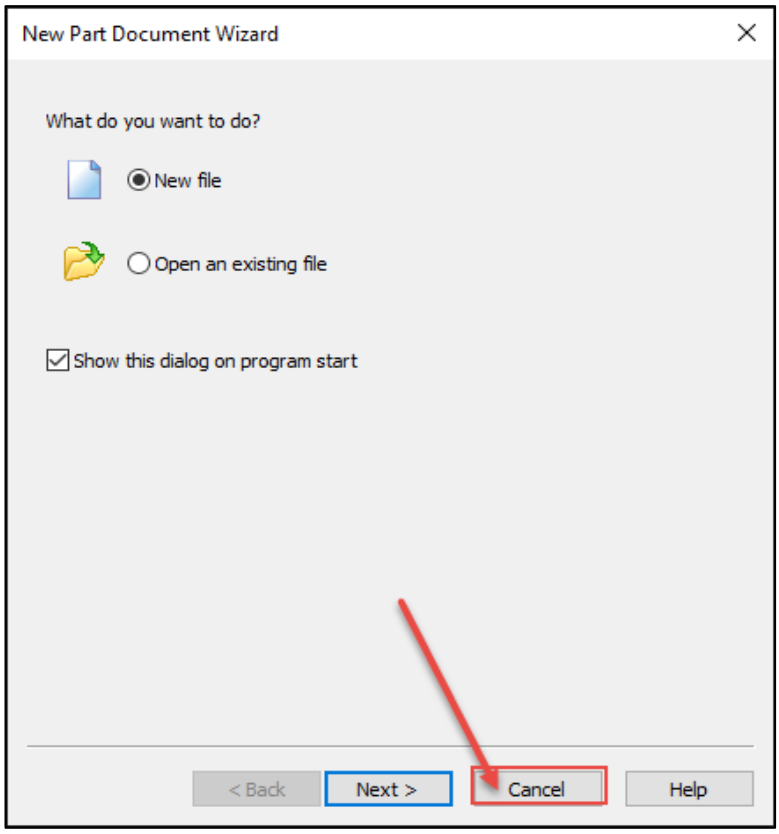
Evaluation Options

2 Click on the shortcut

that is on the Desktop. This will start FeatureCAM.

3 When it has finished **cancel** the new **Part document wizard** that is on the screen.

© Copyright 2016 Autodesk, Inc. All rights reserved



4 Go up to the top left of the screen and go to the **File** menu**.** Then Select **Evaluation Options.**

AUTODESK® **FEATURECAM®**

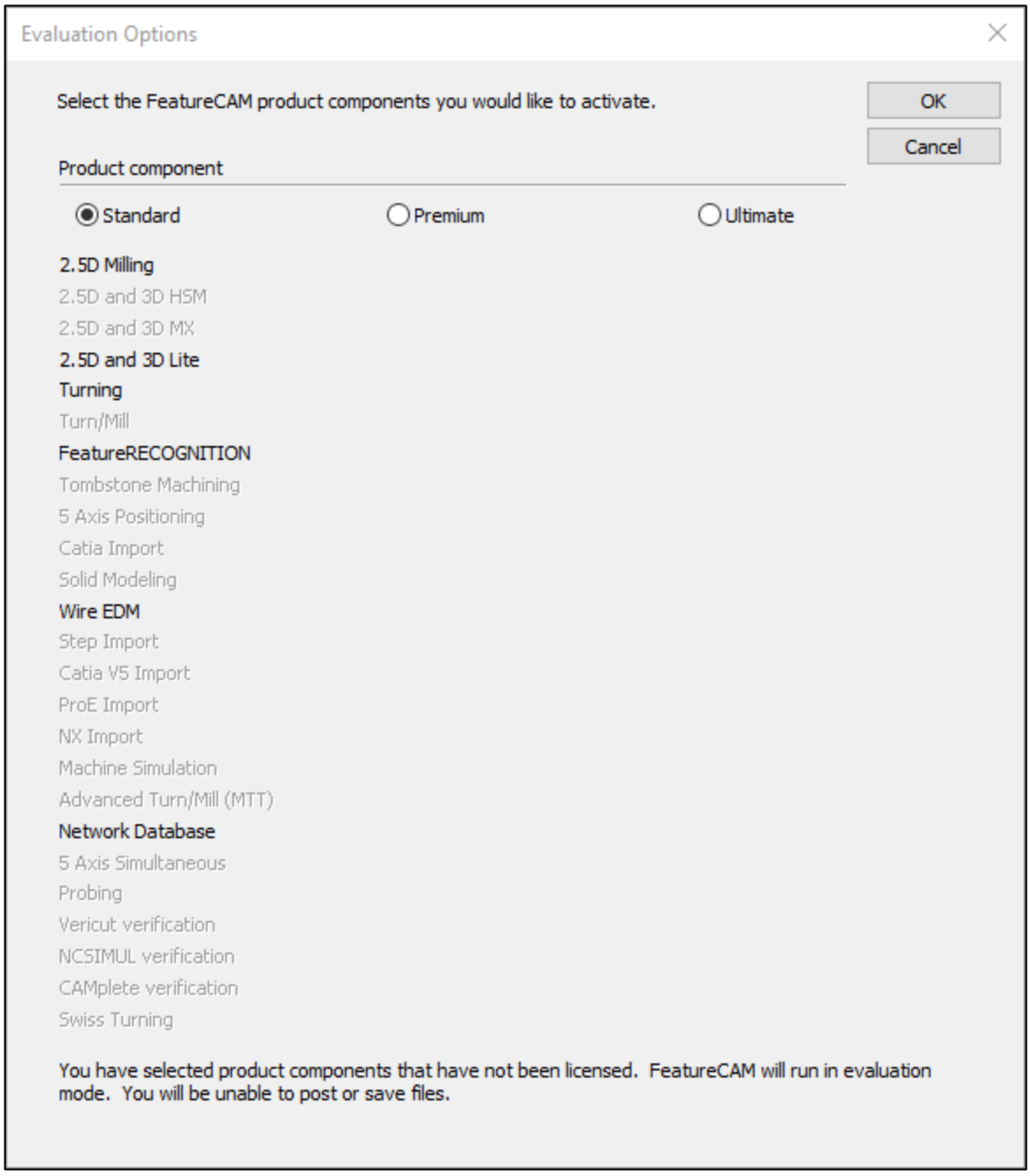
Feature Comparison

Feature Description	FeatureCAM Standard	FeatureCAM Premium	FeatureCAM Ultimate
Automated Programming			
Feature recognition	✓	✓	✓
Built-in intelligence	✓	✓	✓
Programming control	✓	✓	✓
Solid modeling		✓	✓
Simulation and Safety			
Collision avoidance	✓	✓	✓
Clamps & fixtures	✓	✓	✓
Stock models	✓	✓	✓
Machine simulation		✓	✓
Machine limits		✓	✓
Vericut, NCSIMUL & CAMplete			✓
CNC Milling Applications			
2.5-axis milling & 3-axis lite	✓	✓	✓
2-axis turning	✓	✓	✓
2-axis & 4-axis wire	✓	✓	✓
3-axis milling & 3+2 positioning		✓	✓
Tombstone machining		✓	✓
Probing			✓
Turn-mill		✓	✓
Advanced turn-mill			✓
Swiss-type lathes			✓
5-axis milling			✓
PartMaker CAM for Swiss Lathes			
Swiss machining			✓
Cycle time efficiency			✓
CNC compatibility			✓
NC code			✓

© Copyright 2016 Autodesk, Inc. All rights reserved

***FeatureCAM** has now been split into **three levels**. **Standard, Premium** and **Ultimate**. The next few images show the available options for all modules.*

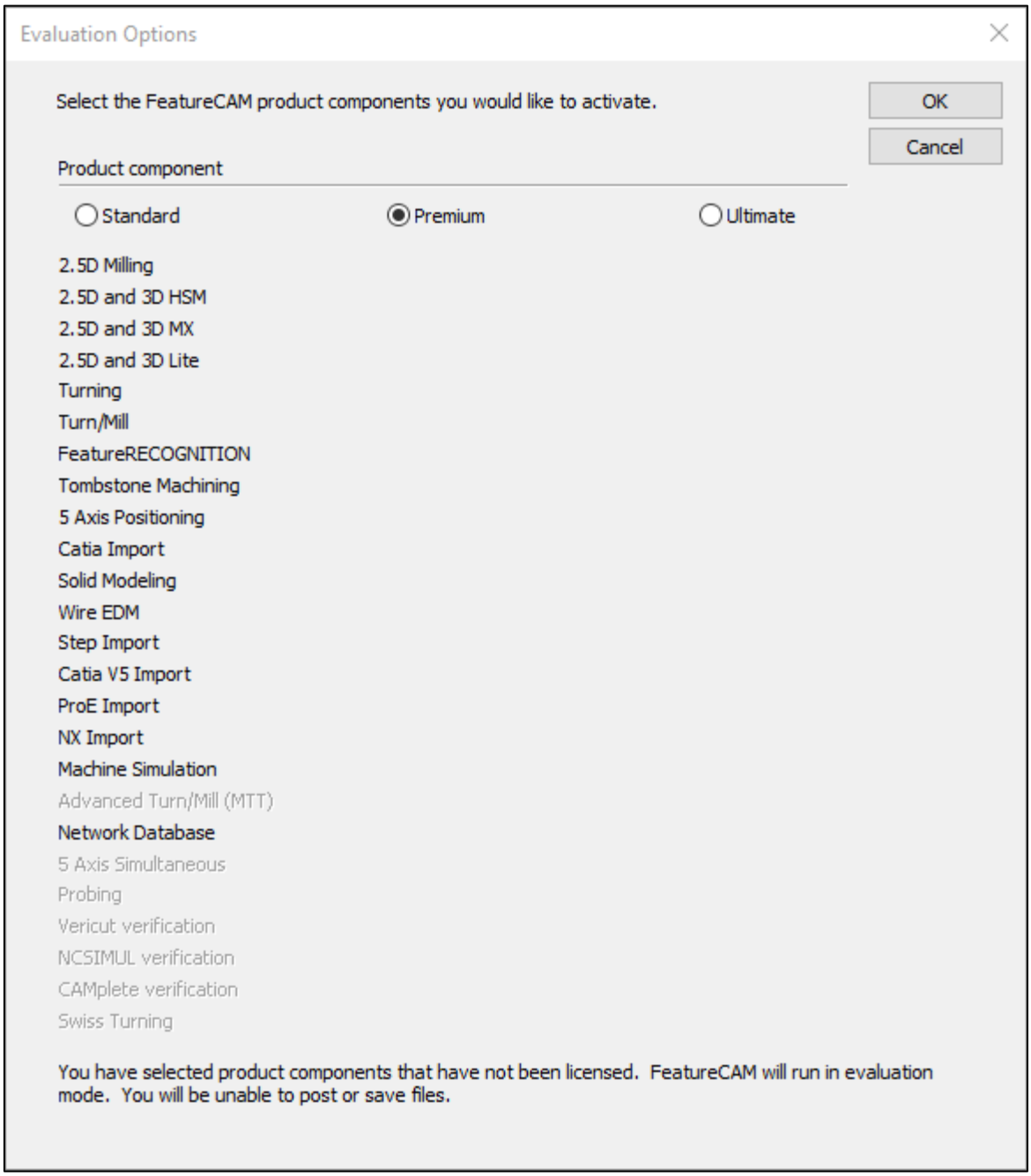
*The image below show the available options in **Standard FeatureCAM***



It is here that you will see the activated products that you have purchased.

© Copyright 2016 Autodesk, Inc. All rights reserved

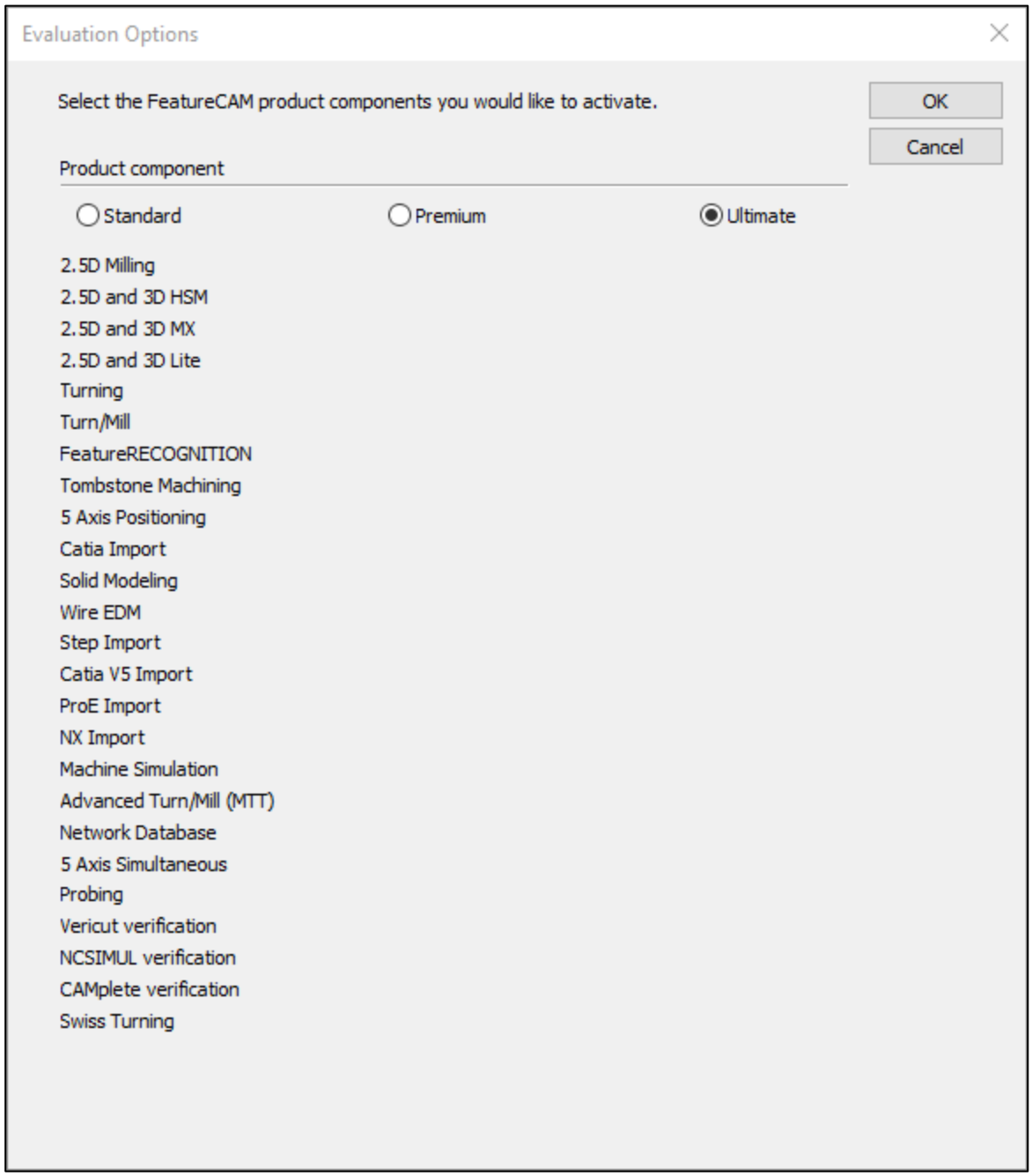
*The image below show the available options in **Premium FeatureCAM**.*



It is here that you will see the activated products that you have purchased.

© Copyright 2016 Autodesk, Inc. All rights reserved

*The image below show the available options in **Ultimate FeatureCAM**.*



It is here that you will see the activated products that you have purchased.

© Copyright 2016 Autodesk, Inc. All rights reserved

Desktop Subscription

This is what you get when you subscribe:-

- Individuals, teams and enterprises have more choices than ever before Regardless of the size of your business - small, mid-size or enterprise.
- Autodesk offers the flexibility to choose the subscription options that best fit your business needs.
- Try out new tools without making a big up-front investment.
- Better manage changing software and budget needs.

Access options

When subscribing, select the option for access and use of your software:

- **Single-user access** - best for when software is needed for one person
- **Multi-user access** - ideal for situations in which you need more than one person to share software licences

Flexible term lengths

With monthly, quarterly, annual and multi-year options, you can get exactly the software you need for as long as you need it.

Access to the latest software

When you subscribe, you have instant access to the latest product releases and enhancements.

- Stay current on the latest software rather than waiting for major product releases
- Update software straightaway or choose a time that is right for your entire team

More about updates and releases

- Software when you need it, where you need it

Use your software in more places with access to both current and preceding versions.

- Access preceding versions :-

  You will be able to download and use preceding releases of most Autodesk software.
  More about preceding version rights :-
- Use your software at home - Access both current and preceding versions of your software on your home computer.
  More about home use rights
- Use your software when travelling :-

© Copyright 2016 Autodesk, Inc. All rights reserved

- Use your software and services globally when travelling outside of your home country. More about global travel rights

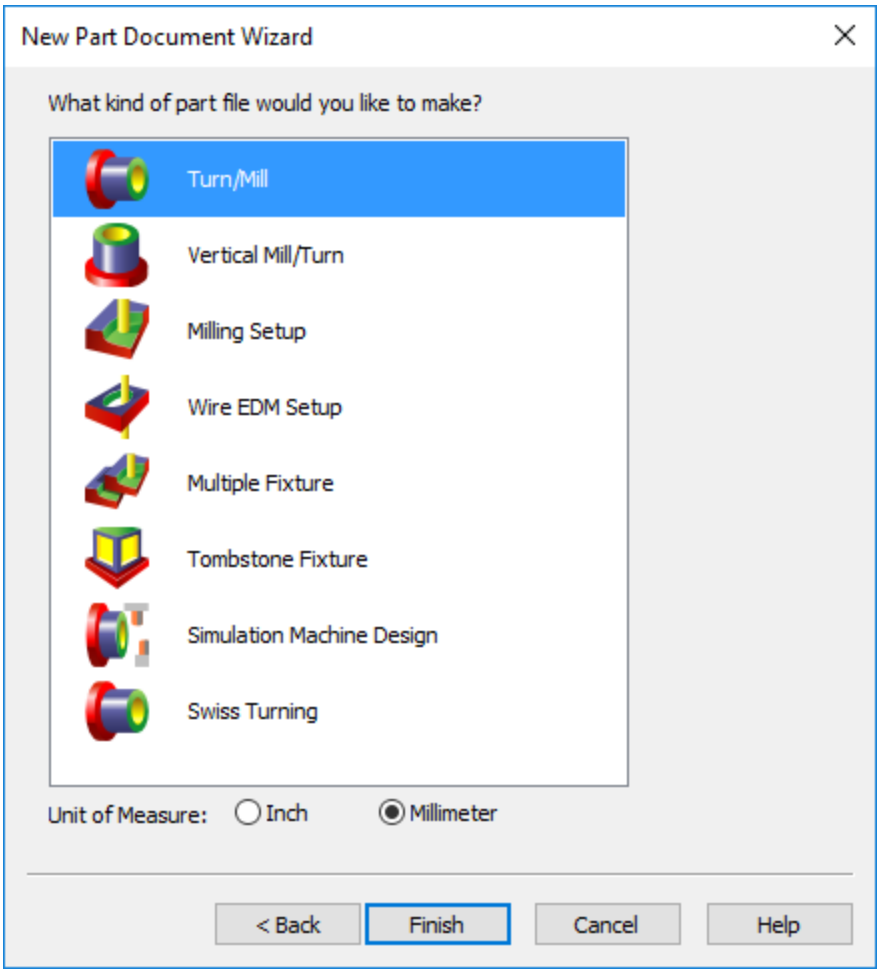
Autodesk support

Let us help you to resolve issues quickly. You will have access to:

- Senior Autodesk support specialists.
- Moderated community support forums.
- E-learning opportunities and exclusive training web casts New Part Document.

Starting a new FeatureCAM part document.

- Select **File**>**New** from the menu or select **Ctrl + N** keys.

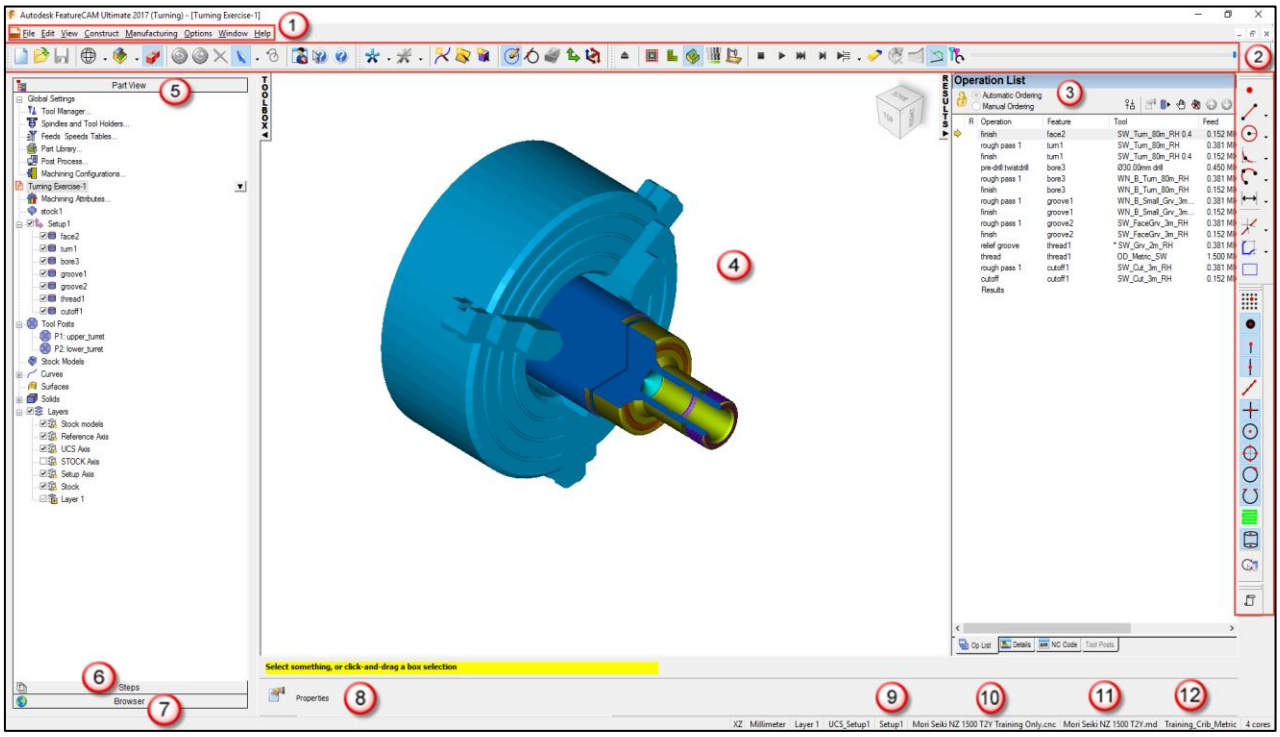


User Interface

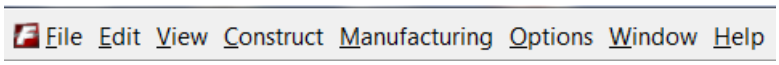
- **FeatureCAM** is a windows-based product and has been a windows based product since its inception unlike other **CAM** products. Many of the functions and keystroke shortcuts that you are familiar with in windows are available inside **FeatureCAM**. Examples are **Ctrl+C** to copy, **Ctrl+V** to paste, **Ctrl+Z** Undo last operation. etc.

© Copyright 2016 Autodesk, Inc. All rights reserved

Screen Layout

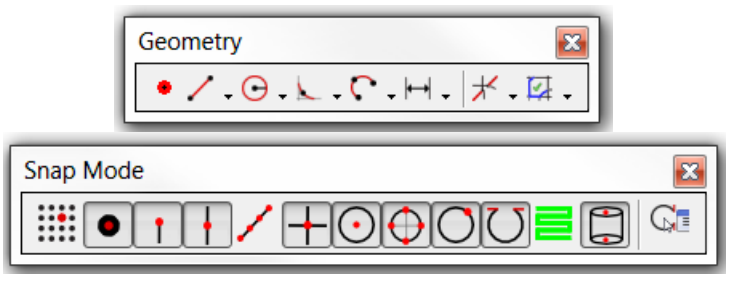


1 **Menu Bar** - Located at the top. Like many other Windows based software, packages **FeatureCAM** includes a series of drop down menus. These menus are unique to **FeatureCAM** and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.



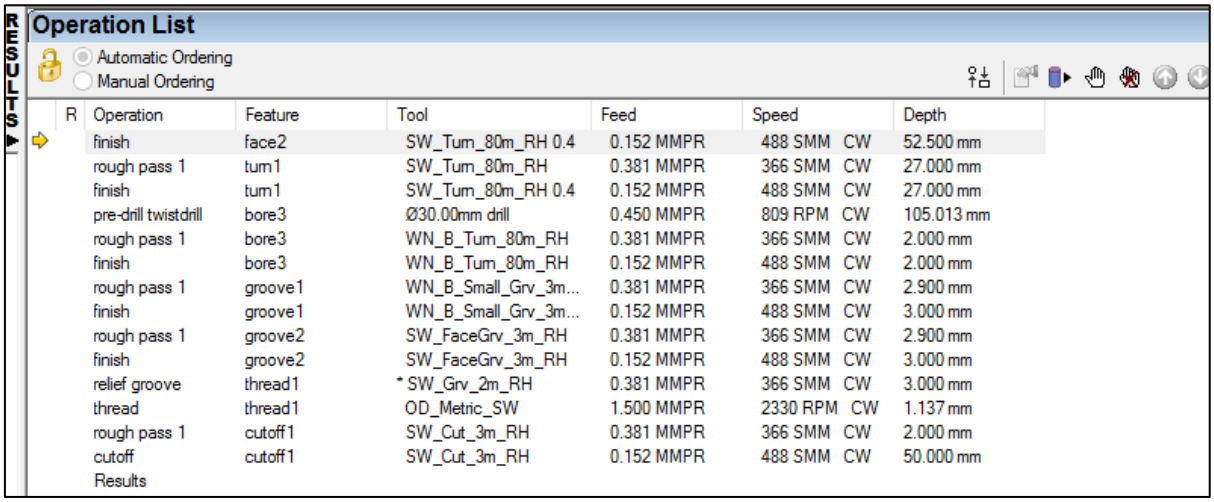
2 **Tool Bars -** usually located at the top but, may be docked in other locations. Can consist of one or more buttons and used to create objects in the Graphics Window such as the Geometry constructors for drawing the shape of your part.

2 **Geometry and Snap Icons**: - Use geometry constructors to create 2D geometry. Snapping is a CAD concept that helps you position lines, points or shapes as you construct geometry for the part. Entering points by coordinates is exact but not always convenient or feasible. Picking points with the mouse is not always precise enough. The bridge between the methods of point selection is snaps. From left to right, the buttons set the snap to:



© Copyright 2016 Autodesk, Inc. All rights reserved

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



R	Operation	Feature	Tool	Feed	Speed	Depth
	finish	face2	SW_Turn_80m_RH 0.4	0.152 MMPR	488 SMM CW	52.500 mm
	rough pass 1	turn1	SW_Turn_80m_RH	0.381 MMPR	366 SMM CW	27.000 mm
	finish	turn1	SW_Turn_80m_RH 0.4	0.152 MMPR	488 SMM CW	27.000 mm
	pre-drill twistdrill	bore3	Ø30.00mm drill	0.450 MMPR	809 RPM CW	105.013 mm
	rough pass 1	bore3	WN_B_Turn_80m_RH	0.381 MMPR	366 SMM CW	2.000 mm
	finish	bore3	WN_B_Turn_80m_RH	0.152 MMPR	488 SMM CW	2.000 mm
	rough pass 1	groove1	WN_B_Small_Grv_3m...	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove1	WN_B_Small_Grv_3m...	0.152 MMPR	488 SMM CW	3.000 mm
	rough pass 1	groove2	SW_FaceGrv_3m_RH	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove2	SW_FaceGrv_3m_RH	0.152 MMPR	488 SMM CW	3.000 mm
	relief groove	thread1	* SW_Grv_2m_RH	0.381 MMPR	366 SMM CW	3.000 mm
	thread	thread1	OD_Metric_SW	1.500 MMPR	2330 RPM CW	1.137 mm
	rough pass 1	cutoff1	SW_Cut_3m_RH	0.381 MMPR	366 SMM CW	2.000 mm
	cutoff	cutoff1	SW_Cut_3m_RH	0.152 MMPR	488 SMM CW	50.000 mm
	Results					

4 **Graphics Window** - Located in the Centre. This is where most of the activity is taking place, creating **Stock**, **Importing Drawings**, Creating **Solid** and **Surface** models, constructing **Geometry**, **Chaining** with **Curves**, creating **Features** and **Simulating** the tool path in **3D** or using the **Centreline Simulation**.

In the Centre of the screen is the Graphics Window. All drawings and toolpath simulations happen in this window. On the right is the Manufacturing Results Window. The automatically generated documentation and part programs are shown in this window.

*At the bottom of the screen is the **Assistance Bars** for **Geometry** and **Status**. The **Assistance Bar** is where you are prompted when creating **Geometry**.*

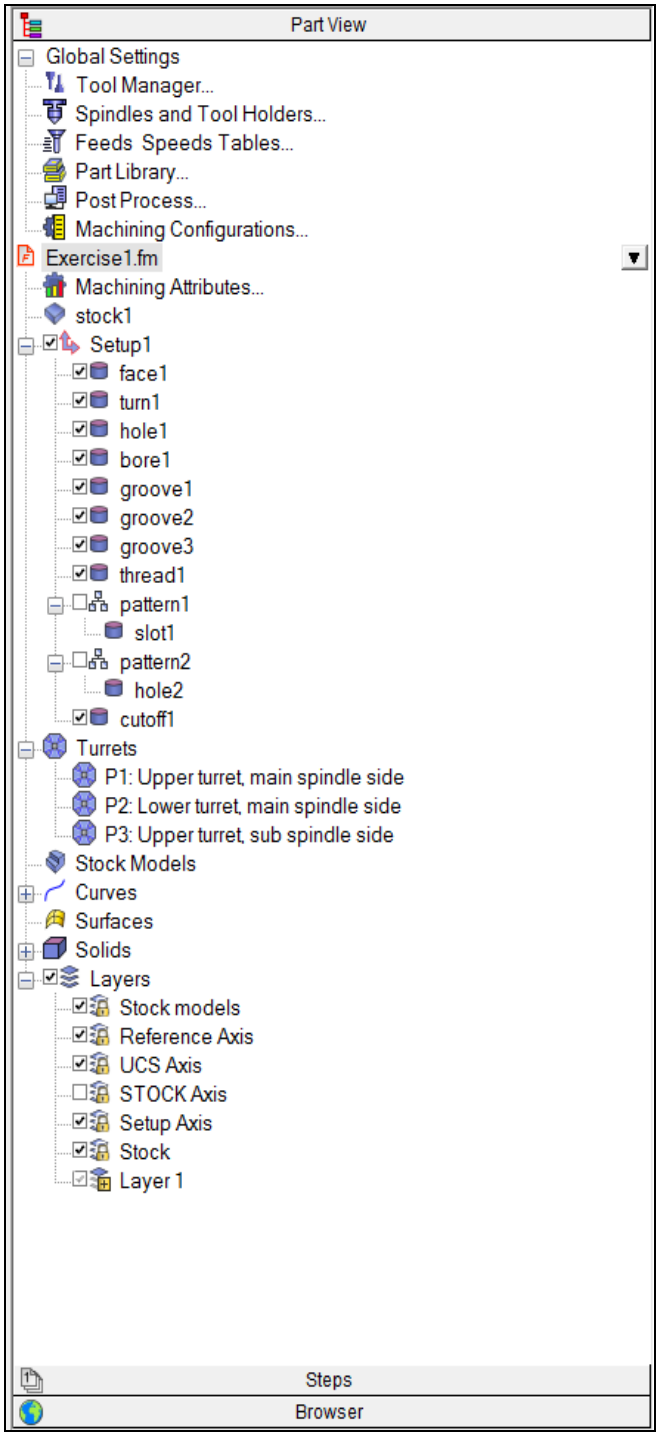
*The screen shots shown above and on the next page show the **Results Window**. This shows you the order in which the machining is carried out. The Feature in which it is associated too, plus the Tools used in that sequence and the corresponding Feeds and Speeds. At the end you have the depth that the Tool will go to for example **25mm***

There are three tabs available on the results window at the bottom.

These are **Op List, Details** and **NC Code**.

© Copyright 2016 Autodesk, Inc. All rights reserved

5 In the **Part View Tool Box** this contains the names of the objects that are created on the screen. Objects that can be contained in this area are **Stock, Setups, Curves, Surfaces, and Solids,** which can include **Stl** triangulated models.



*Clicking on the objects in the **Part View** enables different functions. Left-click selects the object in the **Graphics Window**. Right-click reveals menu of options available for the object. Clicking on a **"+"** will expand the view to reveal objects that are **subordinate**.*

© Copyright 2016 Autodesk, Inc. All rights reserved

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

*The **Steps** toolbar is an outline for the flow of the program. The Steps are numbered sequentially in the order they should be executed and can be a guide for creating your part.*

Steps Menu

Steps to take to create a part in FeatureCAM

Drawing a part in FeatureCAM

1. **Start FeatureCAM.**
2. **Draw geometry in FeatureCAM.**
3. **Resize Stock to fit part.**
4. **Create and move Setup to desired location.**
5. **Create curves.**
6. **Create Features.**
7. **Select Crib, Select Post, and Generate NC code.**

© Copyright 2016 Autodesk, Inc. All rights reserved

Importing a file into FeatureCAM

1. **Start FeatureCAM.**
2. **Import 2D or 3D files into FeatureCAM.**
3. **Use Wizard to align part and create Setup.**
4. **For Solid models use Automatic Feature Recognition (AFR) or use Interactive Feature Recognition (IFR) to create features.**
5. **Select Crib, Select Post, and Generate NC code.**

*Another example of the steps to take. Suppose you have a 2D drawing on your desk and you would like to create a solid model and a **CAM** program. To do this, we would **follow the steps sequentially**:*

1. Specify type of stock and stock dimensions,
2. Draw Geometry
3. Create Curves from Geometry
4. Create Surfaces
5. Create Solid from Curves. (Extra Cost Option)
6. Use AFR (Automatic Feature Recognition) or IFR (Interactive Feature Recognition)
7. Create Features
8. View toolpaths and
9. Select Crib, Select Post, and Generate NC code.

*You can access the majority of **FeatureCAM's** functions through these buttons. A summary of each of the **Steps** function is listed below.*

***Tip:** We recommend that if you are just starting out with the program or demonstrating it to others that you use the **steps toolbox** instead of toolbar buttons in other areas of the interface to keep the interface simple.*

Stock

*The **Stock Wizard** steps you through entering the shape and dimensions of the stock, the stock material, part program zero and the coordinate system for modeling.*

***Geometry.** This gives you the ability to create part shapes using a series of Geometry commands for example Points, Lines, Arcs, Clip, Fillet etc. These are used to define the overall shape of parts. Many different geometry tools are available. Geometry can also be imported from other CAD systems.*

***Curves.** Shapes that involve more than a single line or arc are described as curves.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Surfaces. *This opens the* ***Surface Wizard.***

Solids. *This opens the* ***Solid Wizard.***

AFR. (Automatic Feature Recognition*) This creates 2.5D geometry from Solid Models ready for machining.* ***IFR*** *is also available.*

Features. *This relates to common shop terms like* ***Hole****,* ***Pocket****,* ***Slot*** *or* ***Thread Milling*** *etc. They are created from curves or dimensions. These objects are created in 2.5D and are used to generate toolpaths.*

Toolpaths. *Toolpaths are generated from collections of features. You can simulate them in* ***FeatureCAM*** *using toolpath Centerlines, 2D shaded or 3D solid shaded simulations. Plus full* ***Machine Tool Simulation****.*

NC Code. *Machine-specific G-codes are generated from the toolpaths. Translators are provided for many different NC controls and include a program for creating new translators.*

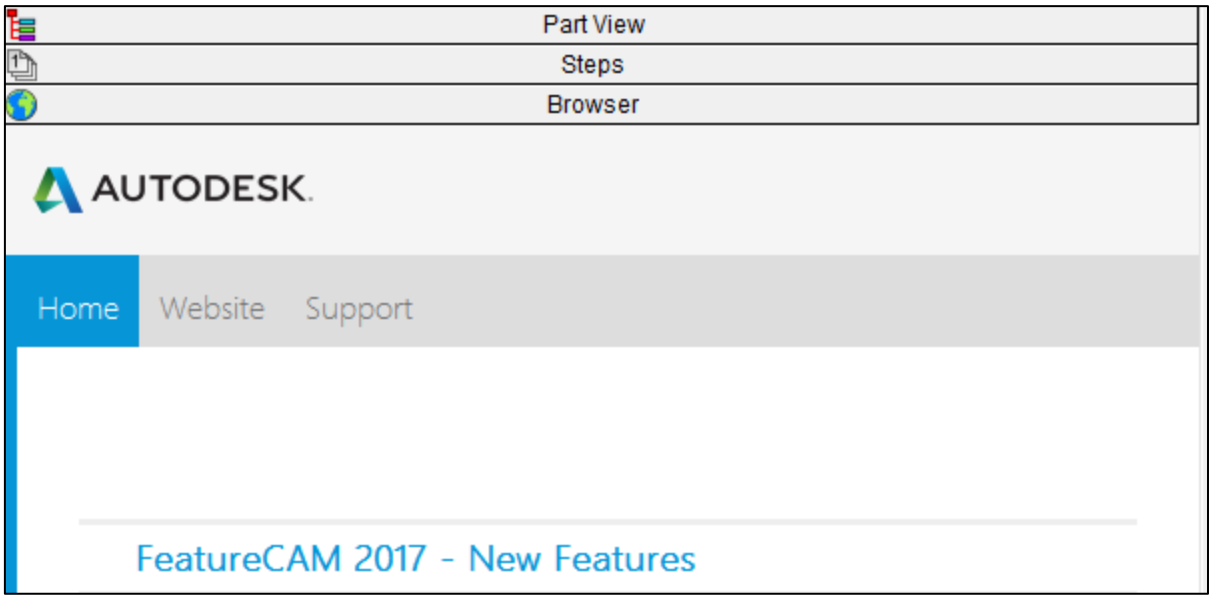
Steps (See menu in PartView)

The following is a guide to help you choose the correct way in which to machine a component in ***FeatureCAM****.*

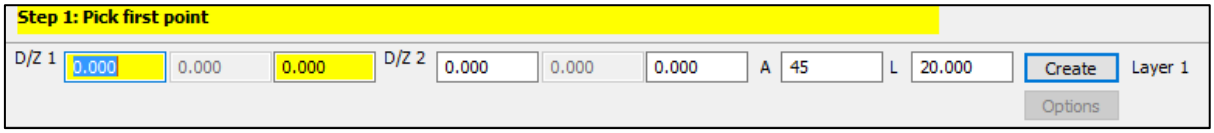
1. Open **FeatureCAM** and select **New File** (Default). Select **Next**. Choose your module of choice and your units. Select **Finish**. Cancel the dimensions wizard. Hide the stock by selecting Stock in **PartView**, right click and select **Hide Stock**. Draw you part using geometry creation techniques. Select **Show Stock** in **PartView**. Create your stock around your geometry. Double click on the stock. Use the **Resize** button to create your stock boundary. Use **Compute Stock from the size of the part**. Re-establish your **Setup1** position. Select your **ToolCrib**. Create **Curves**. Create **Features,** which in turn creates **Toolpaths**. Select your desired post. Set your tool change position. Run **2D centreline Simulation** or **3D Simulation**. Check for errors. **Output code** to your desired location.
2. Import **2D (.dxf) (.dwg) drawing** and follow the steps above from **resizing Stock**.
3. Import Solid Model. Follow the wizard for **Stock size** and **Setup**. Use either **Automatic Feature Recognition** for simple 2.5D plate work, i.e. 2D pockets and holes. Use **Interactive Feature Recognition** for more control over extracting features. Run **3D Simulation**. Check for errors, Collisions. **Output code**.

7 **Browser -** Located on the Left side**"Toolbox**" below **Steps**. Your computer must be connected to the Internet. The Browser provides a link to the Internet. Learn about the Browser by participating in the various sections with downloadable examples on the opening page, then click on the links and explore the contents.

© Copyright 2016 Autodesk, Inc. All rights reserved

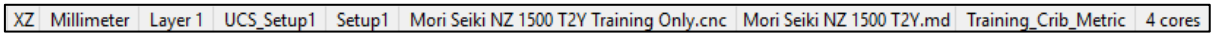


8 **Geometry Bar -** Located near the bottom. This area is constantly used and is where the dimension boxes are used to locate the Geometry on your stock material in the Graphics Window.



Metric

9 **Status Bar -** Metric or Inch. This is Located at the very bottom of **FeatureCAM**. It contains the plane you are working in **(XY–XZ-YZ),** the unit of measure (**Inch** or **Metric**), the layer you are working in, the Setup, Post Processor and the Tool Crib. Normally: - **BasicMetric** for **Metric Tools**. We also show the number of Cores Used.



*Analyzing the above screenshot from the **Status Bar**.*

- **XZ is the current drawing plane** being used. **Inch** or **Millimetres** are the type of units being used e.g. length **100mm. Layer1** is the current active Layer. **UCS_Setup1** is the current **UCS (User Coordinated System)** being used. **Setup1** is the current active setup in which features are created.

10 **Mori Seiki NZ 1500 TZY.CNC** is the current **Post Processor** being used.

11 **Mori Seiki NZ 1500 TZY.MD this is the Machine Simulation file.**

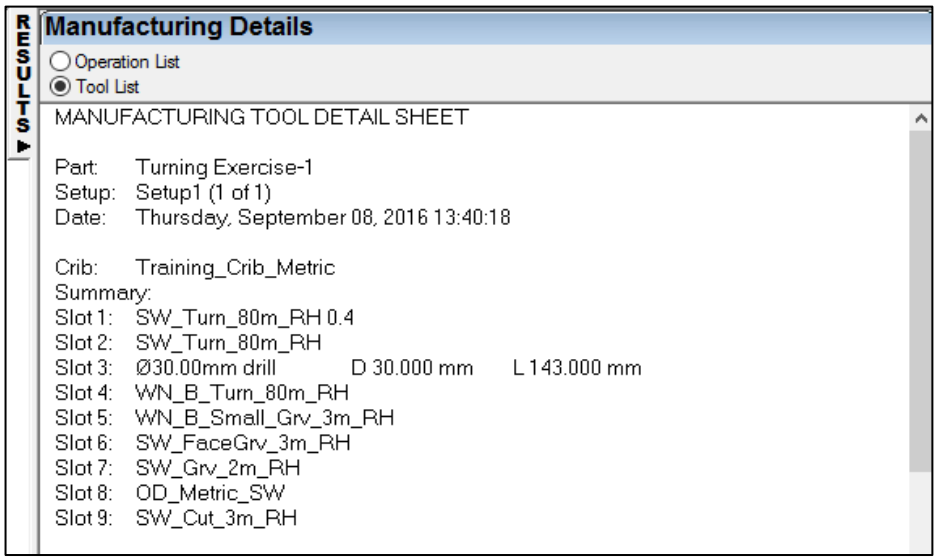
12 This is the current **Tool Crib** and we are using **4 cores** on this computer.

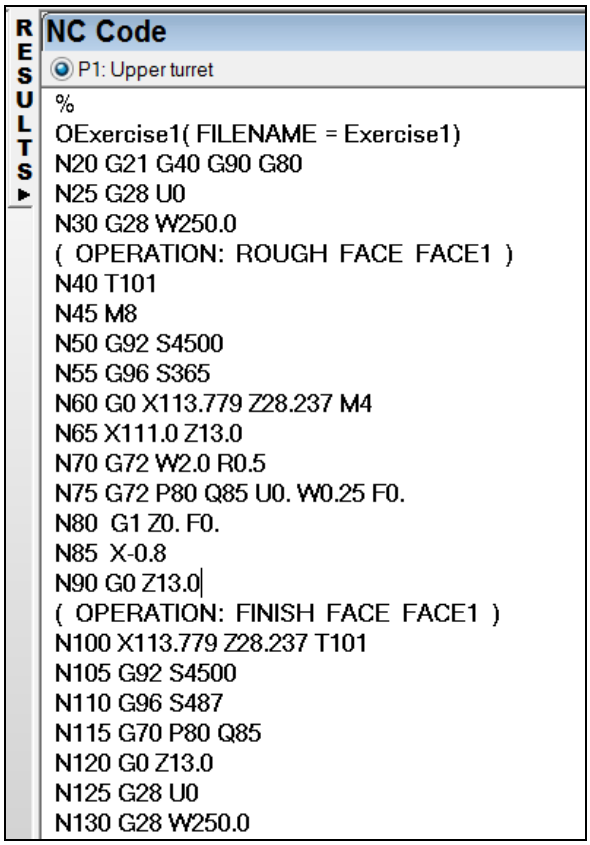
Customize Manufacturing.

- **FeatureCAM** automates the entire part programming process; you can customize all of the System Settings including **Feed/Speed Tables, Tooling Databases or Feature settings.**

© Copyright 2016 Autodesk, Inc. All rights reserved

*Note: Some **Steps** are displayed or not displayed depending on which options you have purchased or have turned on.*





- **Operation List Tab: -** This contains a table of different sequences in order of operation.
- **Details,** gives you a breakdown of the sequence of operations.
- **Tool List.** This gives a breakdown of the tools used.
- **NC Code.** CNC code that will be output from the currently **selected Post Processor**.
- Please note you have to run the **Simulation** before the **NC Code** is made available for output.

© Copyright 2016 Autodesk, Inc. All rights reserved

ViewCube

The ViewCube is a new feature in FeatureCAM, which enables you to change and identify the viewpoint of the Graphics window. By clicking the ViewCube's corners, faces, edges, and icons, you can use it to directly manipulate the view in the Graphics window. In addition, when you re-orient the view using a cube option, toolbar button, or keyboard shortcut, the ViewCube automatically reflects the new viewpoint.

The ViewCube is displayed in the upper-right corner of the Graphics window. Click and drag the cube to re-orient the view in any direction. Alternatively, click:

- A face to show it as an orthogonal view.
- An edge to show the adjacent faces.
- A corner to show the three adjacent faces.

In addition, when you move the cursor near the cube, the Home icon is displayed. Click the icon to show the Home view.

Manipulating orthogonal views

When you select a single-face view and move the cursor near the cube, control icons are displayed:

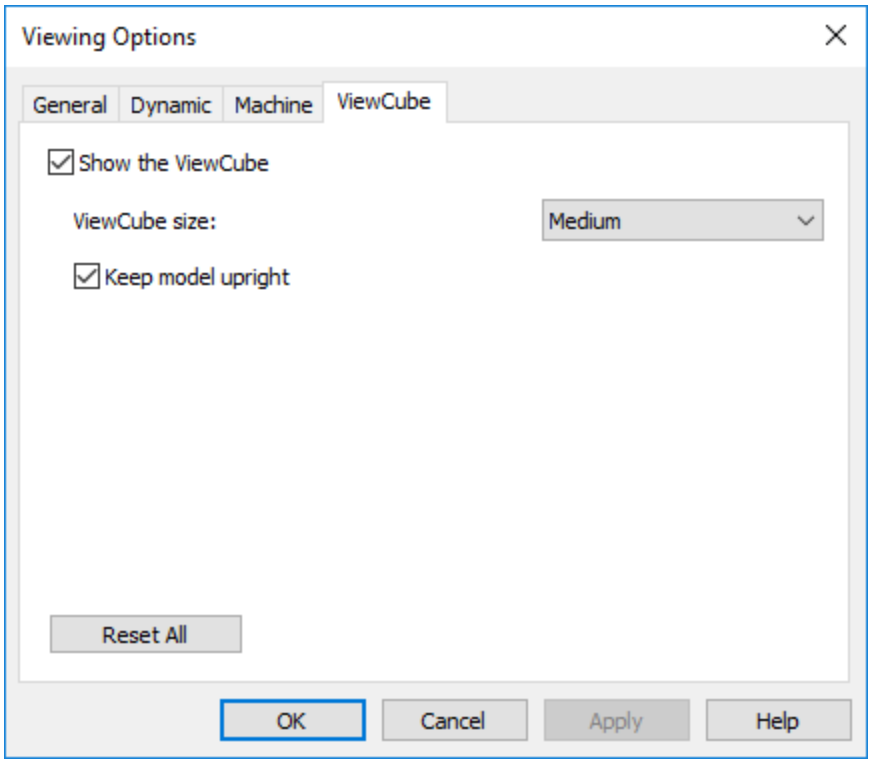
Click:

To show the view of an adjacent face.

 To rotate the view clockwise through 90 degrees.

 To rotate the view counter-clockwise through 90 degrees.

© Copyright 2016 Autodesk, Inc. All rights reserved



Mouse buttons

- Each of the three mouse buttons performs a different dynamic operation in **FeatureCAM**.

Left mouse button Picking and selecting.

- This button is used for **Selecting Items** of the pull down menus, options within **forms**, and **entities** in the **graphics area**.

© Copyright 2016 Autodesk, Inc. All rights reserved

Middle mouse button Dynamics.

- In these options, you can set the behaviours to be specific viewing modes such as **Trackball, Rotate, Pan, Zoom, Pan** and **Zoom**, and so on, or choose the **Current Viewing Mode** option that allows the middle-mouse button (along with key combinations) to perform the viewing mode that is currently set in the toolbar.
- The most commonly used mouse feature is to hold the **Centre wheel down** and moving the mouse, this will rotate your part. Hold the shift key down at the same time and you can **Pan** anywhere on the screen. Hold the **shift key** down and where ever the mouse is it will **Zoom** to that point by using the **Wheel**. This is excellent for looking closely at your component.
- **Zooming in and out: -** Hold down the **Shift** key and **mouse button 2**. Move the mouse up and down to zoom in and out.
- **Pan around the model: -**Click once with mouse button 2 to position in the Centre of the screen. Click and hold to zoom and then move the mouse in the required direction.
- **Zoom Box –** Hold down **mouse buttons 1** and then **button 2**, drag a box around the area to zoom. Release both buttons at the same time. **Rotate mode:** Hold down the **Ctrl** key and **mouse button 2**. Move the mouse up and down to zoom in and out.

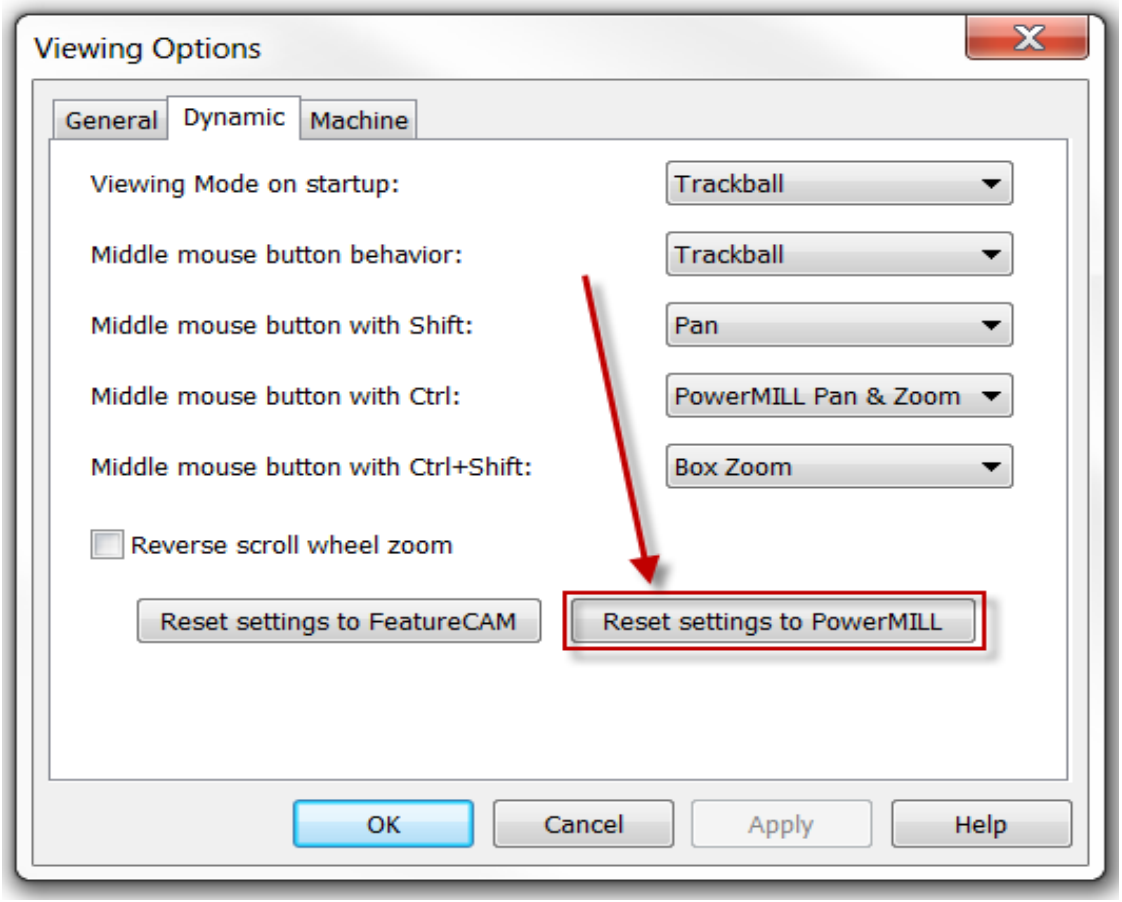
Right mouse button Special Menus & FeatureCAM Options.

© Copyright 2016 Autodesk, Inc. All rights reserved

- When this button is pressed it brings up a local menu relating to whatever the Mouse Pointer is over, such as a named item in the **FeatureCAM Part View** or a physical entity in the graphics area. If nothing specific is selected the **View** menu appears.

Viewing Options

- For those users who are already familiar with other **Delcam ®** products **DON'T PANIC!** Just do the following.
- **View** changes the way you interact with the view of the part. Selecting any of the options from the **View** menu puts you in *view mode.*
- Dynamic Viewing Options.



Dynamic Viewing Options tab, you have two choices to reset the settings to:

Reset to FeatureCAM Settings

- This button resets all the options in the dialog to **FeatureCAM** defaults, which are:

Dynamic Viewing Choice upon **FeatureCAM** Start-Up - **Pan** and **Zoom**

- Middle-mouse button behaviour - **Current Viewing Mode**
- Middle-mouse button with Shift - **Pan and Zoom**
- Middle-mouse button with Ctrl - **Rotate**
- Middle-mouse button with Ctrl+Shift – **Zoom**

© Copyright 2016 Autodesk, Inc. All rights reserved

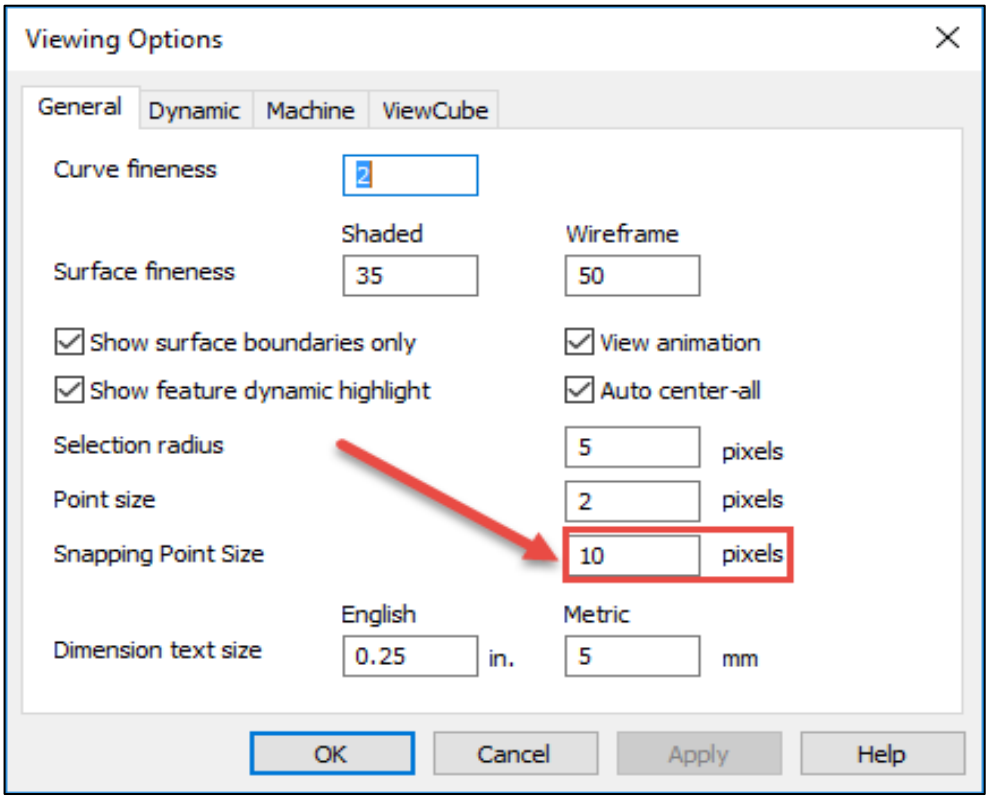
Reset to PowerMILL Settings

- This button resets all the options in the dialog to **PowerMILL** defaults, which are:

  Dynamic Viewing Choice upon **FeatureCAM** Start-Up - **Trackball**
- Middle-mouse button behaviour - **Trackball**
- Middle-mouse button with Shift - **Pan**
- Middle-mouse button with Ctrl - **PowerMILL Pan & Zoom**
- Middle-mouse button with **Ctrl+Shift** - **Box Zoom**
- This menu controls the default viewing mode when **FeatureCAM** first opens. For example, if you select **Trackball**, the next time **FeatureCAM** starts up, the viewing mode is **Trackball.**

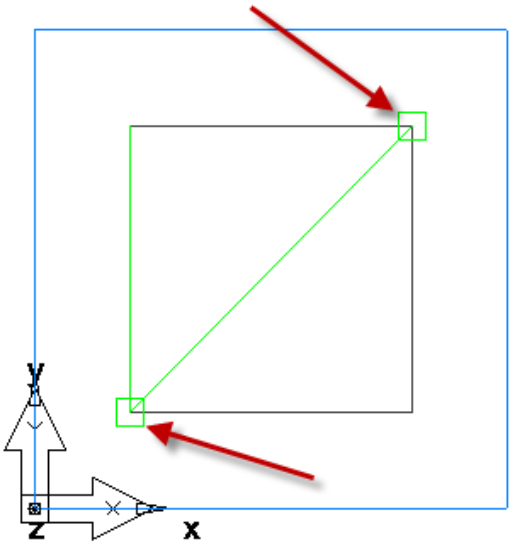
Viewing Options General

In the General Viewing Options we now have the ability to change the size of the Snapping point. By default this size is normally 4 pixels. In order to create larger points, change this to 8 pixels.



© Copyright 2016 Autodesk, Inc. All rights reserved

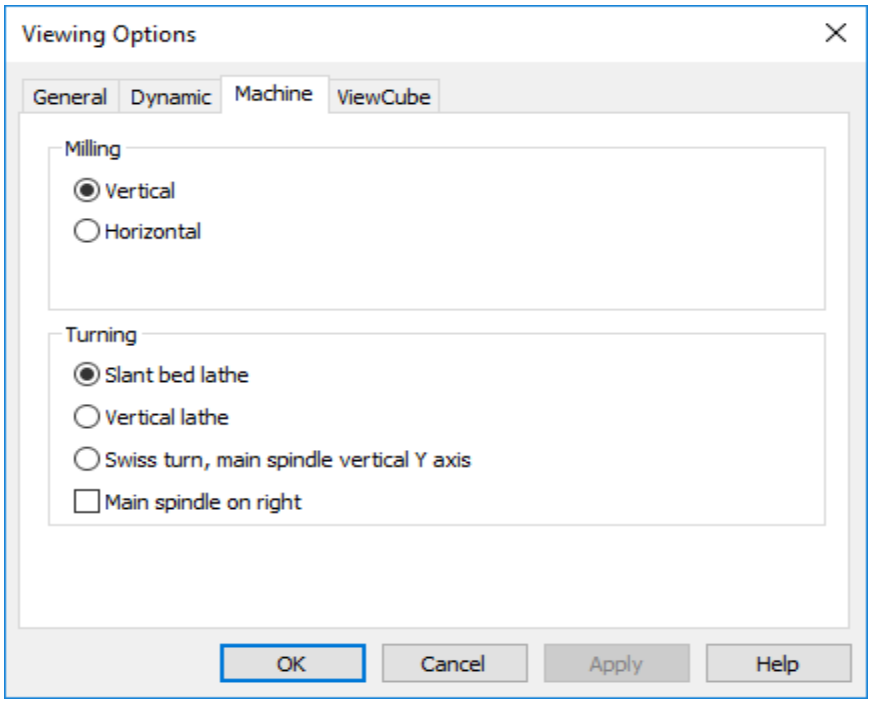
An example of this is shown below



Viewing Options Machine

When viewing machine axis, by default the view is shown vertical: If you have a Horizontal machine or a Mill/Turn machine (Vertical turret lathe) use this option to change the view.

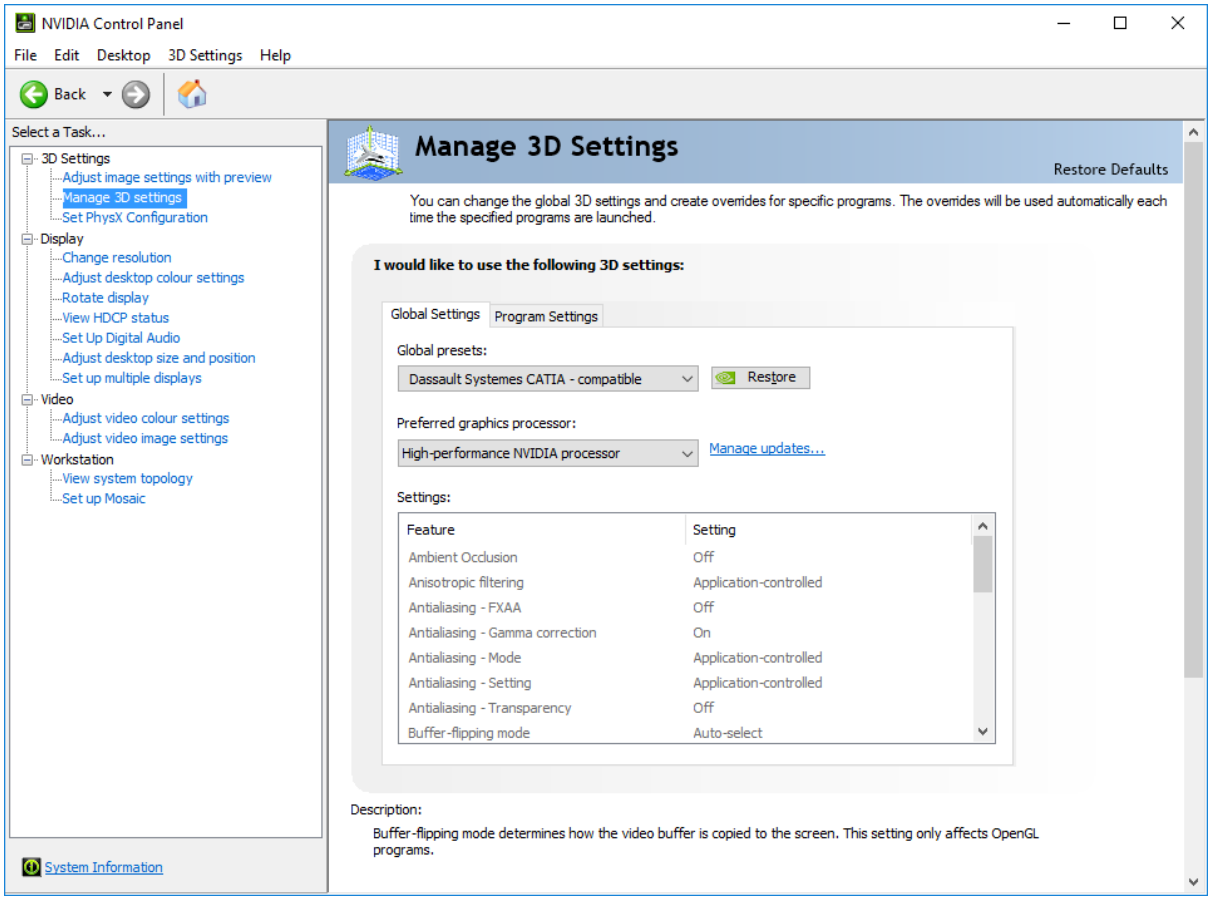
- **Milling** — Select the type of milling machine from **Vertical** and **Horizontal**, to improve viewing during machine simulation and tool previews.
- **Turning** — Select the type of turning machine from **Slant bed lathe** and **Vertical turret lathe**, to improve viewing during machine simulation and tool previews.



© Copyright 2016 Autodesk, Inc. All rights reserved

Display options

- For the best **Display Options** for your computer set the **N-Vidia ® Control Panel** options to the following.

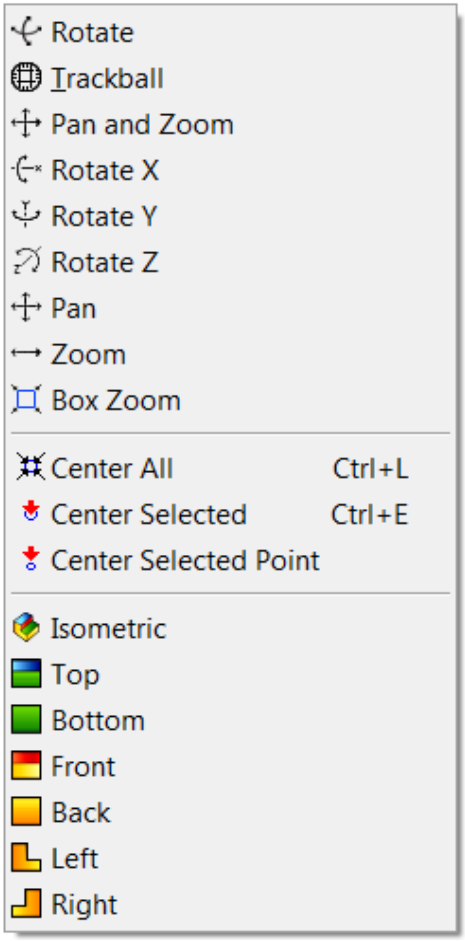


*All of the viewing options are available via **Right mouse button**. Click in the Main Graphics Area*

Principle views

- Front Changes to a view of the part from the front with no other surfaces visible.
- Back Changes to a view of the part from the back with no other surfaces visible.
- Left Changes to a view of the part from the left side with no other surfaces visible.
- Right Changes to a view of the part from the Right side with no other surfaces visible.
- Top Changes to a view of the part from the top only. Useful for drawing geometry, but harder to see the wireframe model of the part.
- Bottom Changes the view to the bottom of the part. No sides of the part are visible from this perspective.

© Copyright 2016 Autodesk, Inc. All rights reserved



Right Click in the graphics area to show this menu.

- All the viewing modes are available, plus an option called **Mode from Last Session.** This option keeps track of the viewing mode last set before **FeatureCAM** was closed, and uses that viewing mode the next time **FeatureCAM** starts up. For example, if the option is set to **Mode from Last Session**, and you had set the viewing mode to **rotate** before you closed **FeatureCAM, Rotate** is the viewing mode that is set in the toolbar the next time you open **FeatureCAM**.
- **Isometric** - changes the view to a three quarter view of the part showing the top and two sides with the current UCS near the bottom of the viewing area.
- **Perspective** - If selected, the view is a perspective view. If deselected, the view is an orthographic view.
- **As on setup** - If selected, the view is relative to the current setup. If deselected, the view is relative to the world coordinate system.
- **As on UCS** - changes your view to that of the current **UCS**.
- **As on world** - changes your view to that of the current **UCS**.

© Copyright 2016 Autodesk, Inc. All rights reserved

Keyboard shortcuts

- You can use the keyboard to navigate around the menus. To do this, press and hold the **Alt** key, and then press the letters on your keyboard that correspond to the letters underlined on the **Menu** bar and in the menu options. For example, to rotate your part, press and hold the **Alt** key, press **V** for **View** menu, press **V** for Viewing Modes and press **R** for **Rotate.**

Key sequence	Action
Alt+click-and-drag on any toolbar button	Move button around in toolbar.
Alt+**1** (**2**,**3**,**4**)	User View 1 (2,3,4)
Alt+**Enter**	Properties
Alt+**F1**	Centreline simulation.
Alt+**F2**	Play/pause simulation.
Alt+**F3**	Single-step simulation.
Alt+**L**	Last view.
Alt+**R**	Refresh
Alt+**Shift**+**V**	Save view.
Ctrl+click a column of any list box that does sorting	Activates the second level of sorting.
Ctrl+click a viewing mode button	Stay in that viewing mode.
Ctrl+click the **3D Simulation** button. Release key and click the **Play** button.	Run 3D simulation in hidden line mode and do continuous looping.
Ctrl+click **Fast Forward to End**	Stops the simulation when it encounters the next rapid (in addition to the next operation).
Ctrl+click the **Machine Simulation** button. Release key and click the **Play** button.	Run machine simulation in hidden line mode and do continuous looping.
Ctrl+**A**	Select all.
Ctrl+**C**	Copy
Ctrl+**E**	Centre selected.
Ctrl+**F**	Find (text in a text window). The cursor must be in a text window.
Ctrl+**H**	Replace (text in a text window). The cursor must be in a text window.
Ctrl+**L**	Centre all.
Ctrl+**N**	New file
Ctrl+**O**	Open file
Ctrl+**P**	Print
Ctrl+**R**	Opens the **New Feature** wizard.
Ctrl+**S**	Save file
Ctrl+**V**	Paste
Ctrl+**X**	Cut
Ctrl+**Y**	Redo
Ctrl+**Z**	Undo

© Copyright 2016 Autodesk, Inc. All rights reserved

Ctrl+click the **Pick Curve** button	Causes the warp status for that dialog to toggle.
Ctrl+click the **Pick Curve** button while a **Key sequence**	Toggles between simulation of the selection feature and the entire setup.
Ctrl+click the **Play** button	Run simulation in hidden line mode.
Hold **Ctrl** while toolpath is being computed	Temporarily turns off Toolpath Computation Minimization. Toolpaths will be recomputed for all features even if that feature did not change.
Ctrl+**Alt**+click-and-drag on any toolbar button	Duplicate and move button around in toolbar.
Ctrl+**Shift**+**click** the **3D Simulation** button. Release keys and click the **Play** button.	Continuous loop simulation for 3D simulation.
Ctrl+**Shift**+click the **Machine Simulation** button. Release keys and click the **Play** button.	Continuous loop simulation for machine simulation.
Ctrl+**Shift**+**A**	Shade selected.
Ctrl+**Shift**+**C**	Unshade all.
Ctrl+**Shift**+**N**	Unshade selected.
Ctrl+**Shift**+**P**	Toggles perspective.
Delete	Delete the selected object
Double-click a simulation button	Activate that simulation without clicking the play button.
Esc	Stop current simulation.
F1	Context-sensitive help.
Middle-click-and-drag in the graphics window	Performs viewing based on the current viewing mode. The mouse wheel-click-and-drag also performs this function.
Mouse scroll wheel	Zoom
Shift+**click** an edge when filleting	Selects all edges of the face and adds them to the list.
Shift+**click** the **NC Code** tab	Show ACL instead of NC code.
Shift+**click** while clipping when **Multiple Region** is **On**	Removes the entire picked region instead of to the nearest intersection.
Shift+**right-click** in graphics window	Dynamic viewing using current view mode.
Shift+**click** the **Show** button in the **SCL** dialog	Brings up a dialog that shows the attributes of the model selected in SCL dialog.

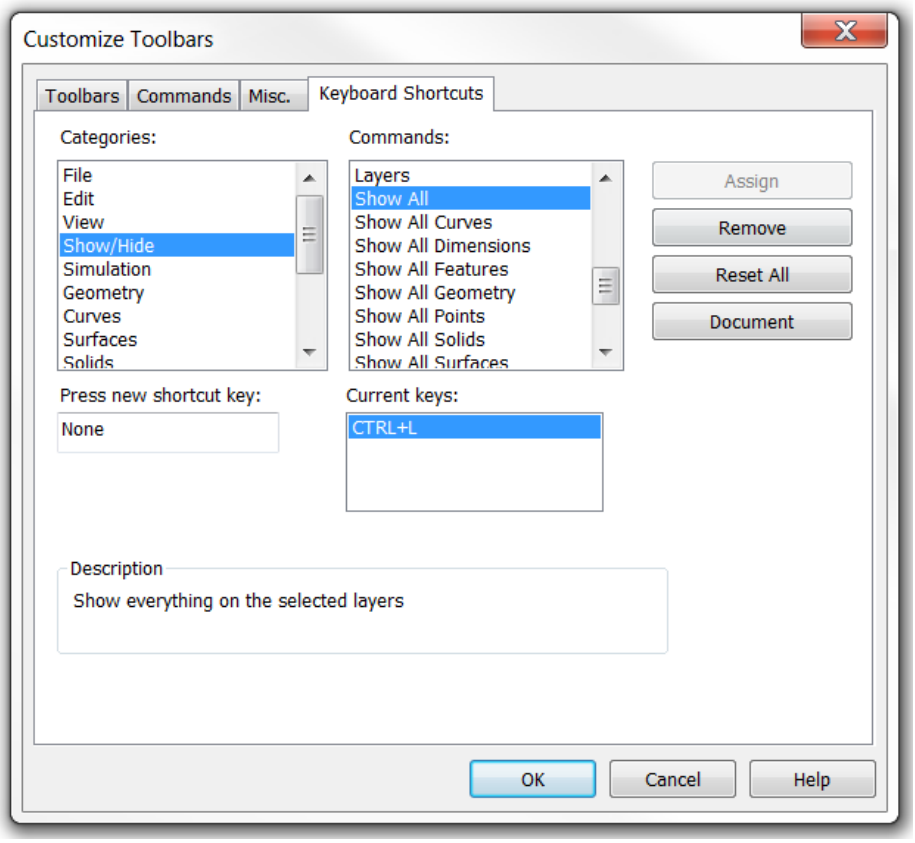
© Copyright 2016 Autodesk, Inc. All rights reserved

Commonly used Ctrl Keys

- **New Feature Menu** or **Ctrl + R**
- **Isometric View** or **Ctrl + 1**
- **Top View** or **Ctrl + 5**
- **Refresh Alt + R**

User interface: Keyboard shortcuts

*There is a new tab in the **Customize Toolbars** dialog, **Keyboard Shortcuts**:*



*This tab lets you assign your own shortcut keys to the **FeatureCAM** menu commands. To set a new shortcut key:*

Select a category from the **Categories** list.

Select a command from the **Commands** list.

*If the command already has a shortcut, it is listed under **Current keys**.*

To assign a new key, select the **Press new shortcut key** field and press the keys that you want to use as the shortcut.

Press the keys on the keyboard, for example, press and hold down the **Alt** key and press the **N** key; do not type **ALT + N**.

© Copyright 2016 Autodesk, Inc. All rights reserved

A warning displays if the shortcut you entered is already assigned to a command.

Click the **Assign** button to save the new shortcut.

If you click **Assign** after receiving a warning that the shortcut is already assigned to a command, the shortcut is removed from the existing command and assigned to the new command.

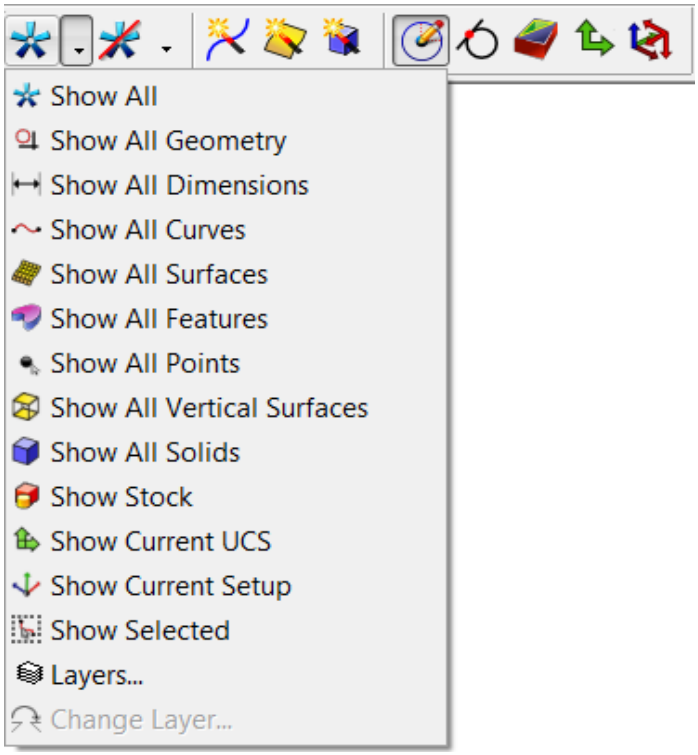
- **Remove** — Click this button to remove the shortcut assigned to the selected command.
- **Reset All** — Click this button to reset all shortcuts to the **FeatureCAM** defaults.

Blanking model entities

- Use the **Blank Selected** (**Ctrl + J**) option to temporarily hide all visible components.
- **(Ctrl K)** Hide Selected components
- To re-display all of the components. Right-click in the graphics area and select **Unblank (Ctrl L)**. The blanked model components are displayed. Please note this option will only work if you have set this up as explained in the previous chapter.

Show menu

- Show functions help control what is displayed. This is useful as you place and model intricate features in a complex part.
- You access the Show functions by selecting **View >** Show from the menu or using the Show **Menu** button on the **Advanced** toolbar:



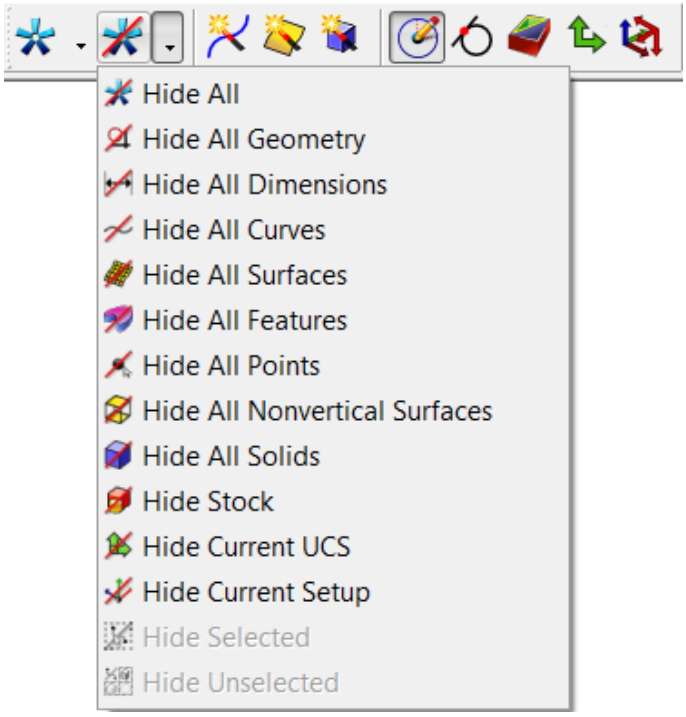
- Show **All** — Use this menu option to Show everything in the part model.

© Copyright 2016 Autodesk, Inc. All rights reserved

- Show **All Geometry** — Use this menu option to Show all geometry (points, lines, arcs, and circles).
- Show **All Dimensions** — Use this menu option to Show all dimension information added with the Dimensions Tool.
- Show **All Curves** — Use this menu option to Show all Curves.
- Show **All Surfaces** — Use this menu option to Show all surfaces (available only in **FeatureCAM 3D**).
- Show **All Features** — Use this menu option to Show all features.
- Show **All Solids** — Use this menu option to Show all solids
- Show **All Vertical Surfaces** — Use this menu option to Show the vertical surfaces in the model. This is useful for identifying surfaces that are part of 2.5D features like holes or pockets contained in a surface or solid model.
- Show **Stock** — Use this menu option to Show the stock outline.
- Show **Current UCS** — Use this menu option to Show the current UCS icon.
- Show **Current Setup** — Use this menu option to Show only the features and drawing elements that are in the current Setup.
- Show **Selected** — Use this menu option to Show only the selected elements

Hide Menu

- Hide controls what is displayed at any given time. This is useful as you place and model intricate features in a complex part. Besides the display factors, you can't snap, select or build curves from hidden entities. The Hide functions are not exclusive. You can click different buttons sequentially, hiding different entities until only the ones you want are still in view.
- You access the Hide functions by selecting **View >** Hide from the menu or using the Hide **Menu** button on the **Advanced** toolbar:



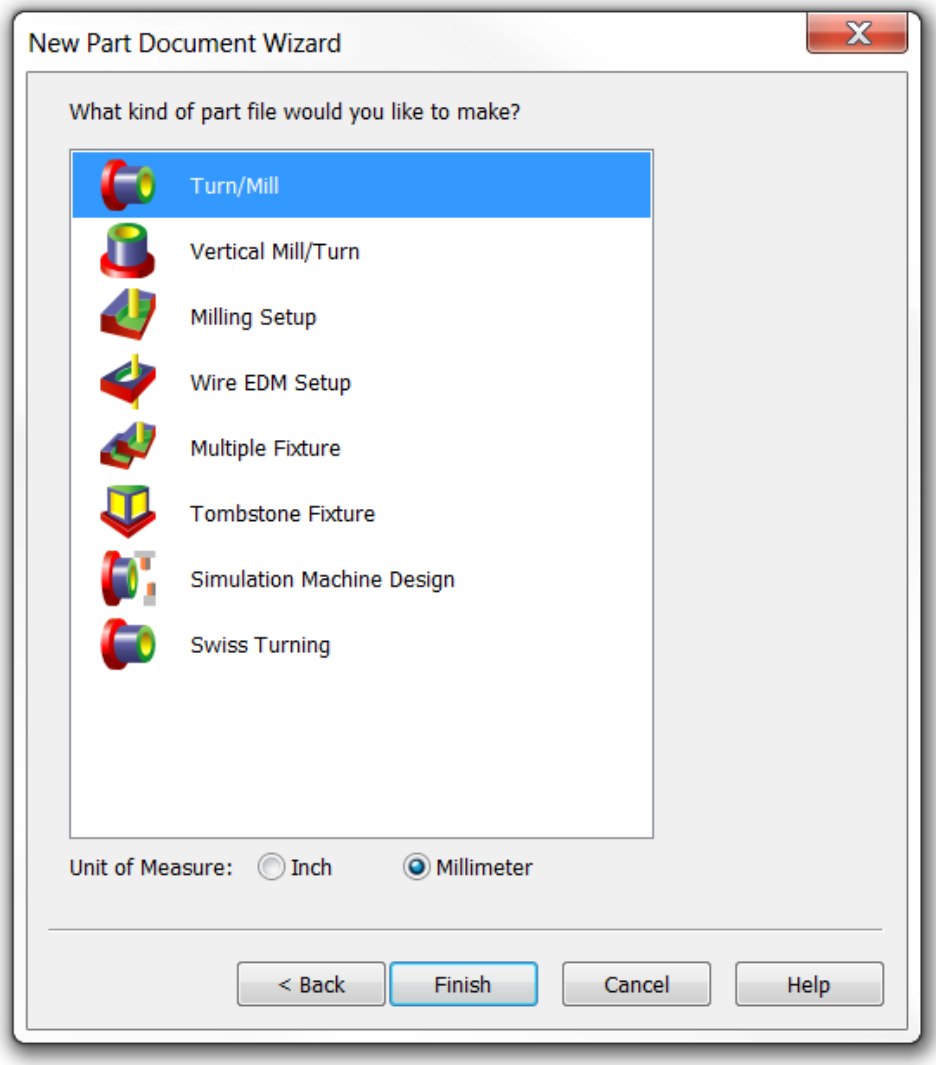
© Copyright 2016 Autodesk, Inc. All rights reserved

- Hide **All** — Use this menu option to hide all geometry, curves, Features. The stock and axis icon remain visible. A common procedure is to Hide **All**, then **Show** only one type of entity, for example features.
- Hide **All Geometry** — Use this menu option to hide all geometry. Other entities remain visible.
- Hide **All Dimensions** — Use this menu option to hide all dimension information added with the FeatureCAM Dimension tools.
- Hide **All Curves** — Use this menu option to hide all curves. Other entities remain visible.
- Hide **All Surfaces** — Use this menu option to hide all surfaces in the part model. Only available in the 3D version.
- Hide **All Features** — Use this menu option to hide all features.
- Hide **All Points** — Use this menu option to hide all points.
- Hide **All Non-Vertical Surfaces** — Use this menu option to hide all surfaces that are not vertical relative to the current setup. This is helpful if you want to isolate the surfaces that are part of 2.5D features in a surface or solid model.
- Hide **All Solids** — Use this menu option to hide all of the solid models.
- Hide **Stock** — Use this menu option to hide the stock outline. All other entities remain visible.
- Hide **Current UCS** — Use this menu option to hide the current user coordinate system.
- Hide **Current Setup** — Use this menu option to hide the axis of the current Setup. All other entities remain visible.
- Hide **Selected** — Use this menu option to hide all selected entities. Non selected entities are still visible.
- Hide **Unselected** — Use this menu option to hide all entities other than the selected ones.

Setting file location options

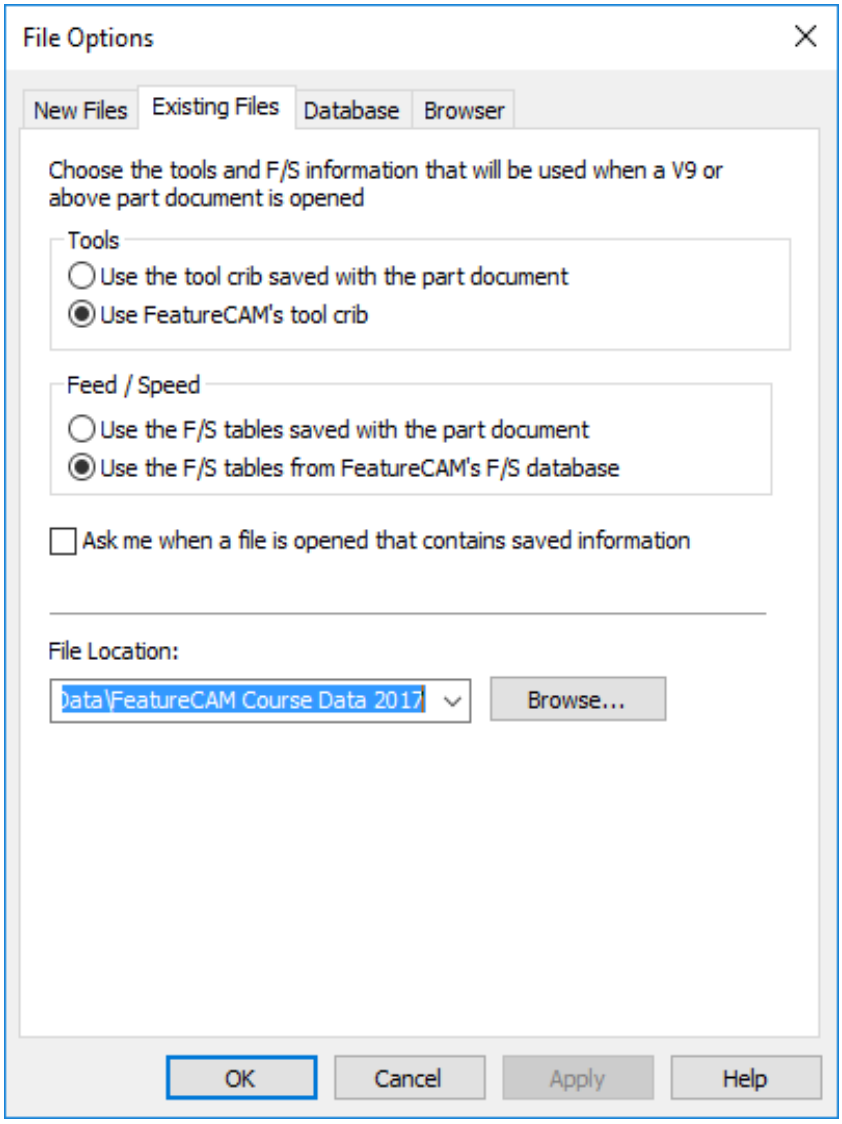
1. Open **FeatureCAM** by double clicking the **FeatureCAM** shortcut
2. Select **New File**. **Turn/Mill** Setup. Select **Finish**.

© Copyright 2016 Autodesk, Inc. All rights reserved



© Copyright 2016 Autodesk, Inc. All rights reserved

3 Before we start please select **Options/File Options/Existing Files.**



4 Select Browse. Then select the following location. **C:\Training_Data\FeatureCAM Course Data 2017**.

5 Select **Apply** and **Ok** to close the form. When we save or import files **FeatureCAM** will take us to this location.

6 Please save all files in **Coursework**.

7 Tool Crib = **BasicMetric**.

8 Post processor used = **Fanuc 21i – T.cnc (2 Axis)**

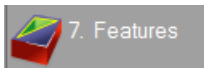
What is a Feature?

- If we look at a Part such as that shown on the next few pages and consider how we would manufacture it we can see that it can be broken down into a series of **Machining Operations**. For example we would probably want to start by **Facing** off the top of the part, then **Turn** the outside profile**,** and then drill and bore the centre **Hole** feature etc.

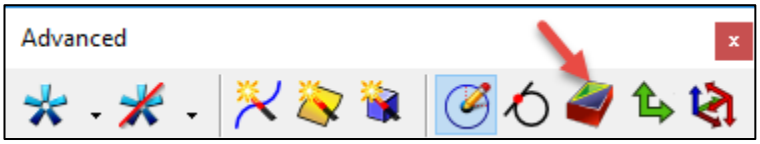
© Copyright 2016 Autodesk, Inc. All rights reserved

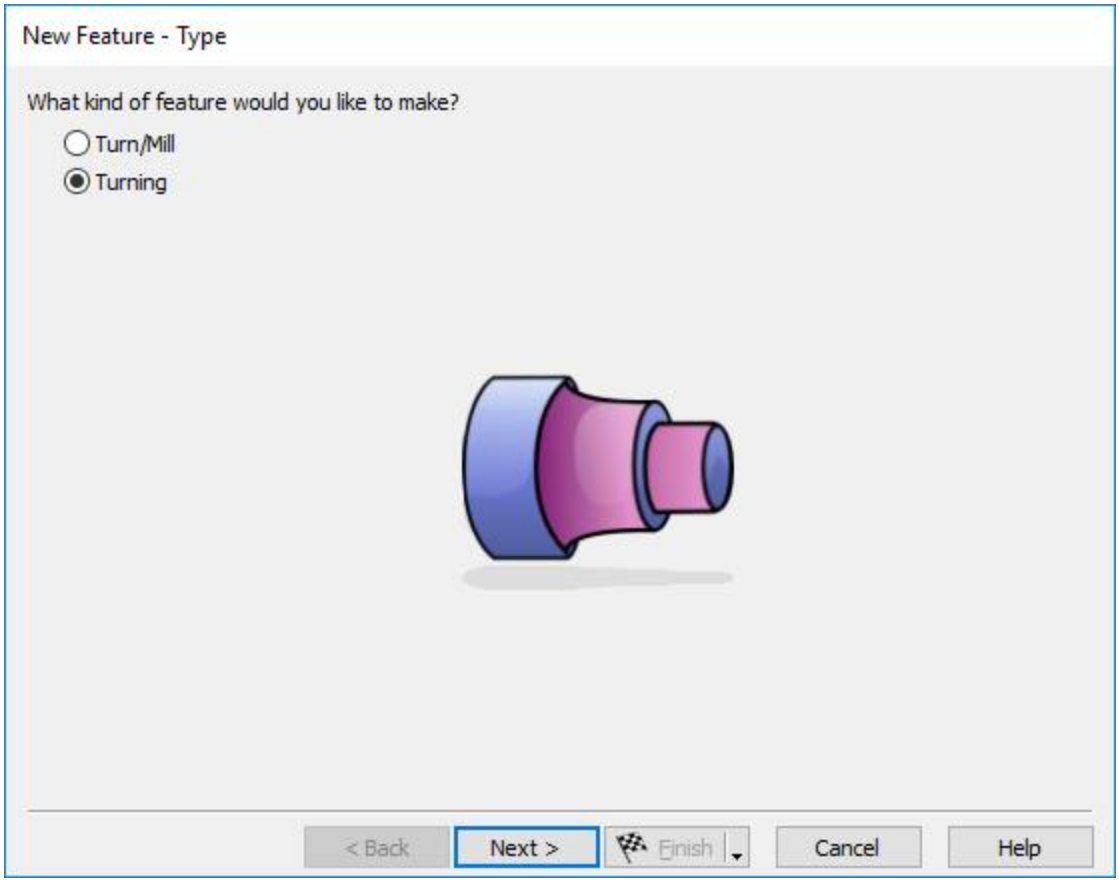
- **FeatureCAM** defines a part in terms of **Features** using common shop floor terminology for instance an area that looks like a **Groove** can be machined using a **Groove Feature**. When you define a profile as a **Turn** Feature, **FeatureCAM** will then automatically decide what operations to use to machine that profile. All you have to do is say "This is a **Turn**" profile and **FeatureCAM** will select suitable **Roughing** and **Finishing Operations**, **Tooling**, **Speeds** and **Feeds**. You are then free to modify these default values to suit your particular situation. By automating the generation of operations the overall time taken to program a part can be drastically reduced.

*There are four different ways to activate the **New Features** menu.*

- Select **Features** 

from the **Steps** menu.
- Select **Ctrl + R New Feature.**
- Select **Construct>New Feature.**
- Select **New Feature** from the **Advanced Toolbar** then select **Turning.**

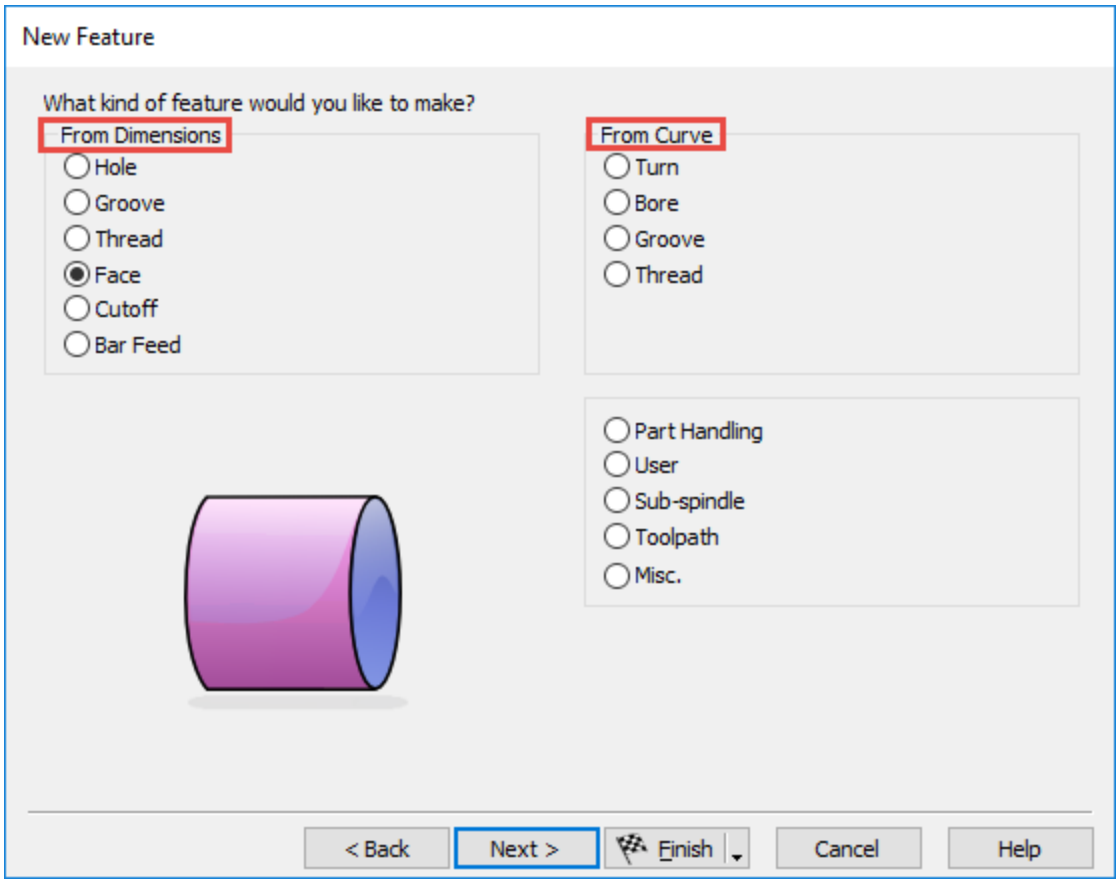




© Copyright 2016 Autodesk, Inc. All rights reserved

How is a Feature Created?

- There are a number of different ways in which **Features** can be created in **FeatureCAM**, which method you choose will depend upon the data available to you. In broad terms the methods are: - **From Dimensions** and **From Curve**.

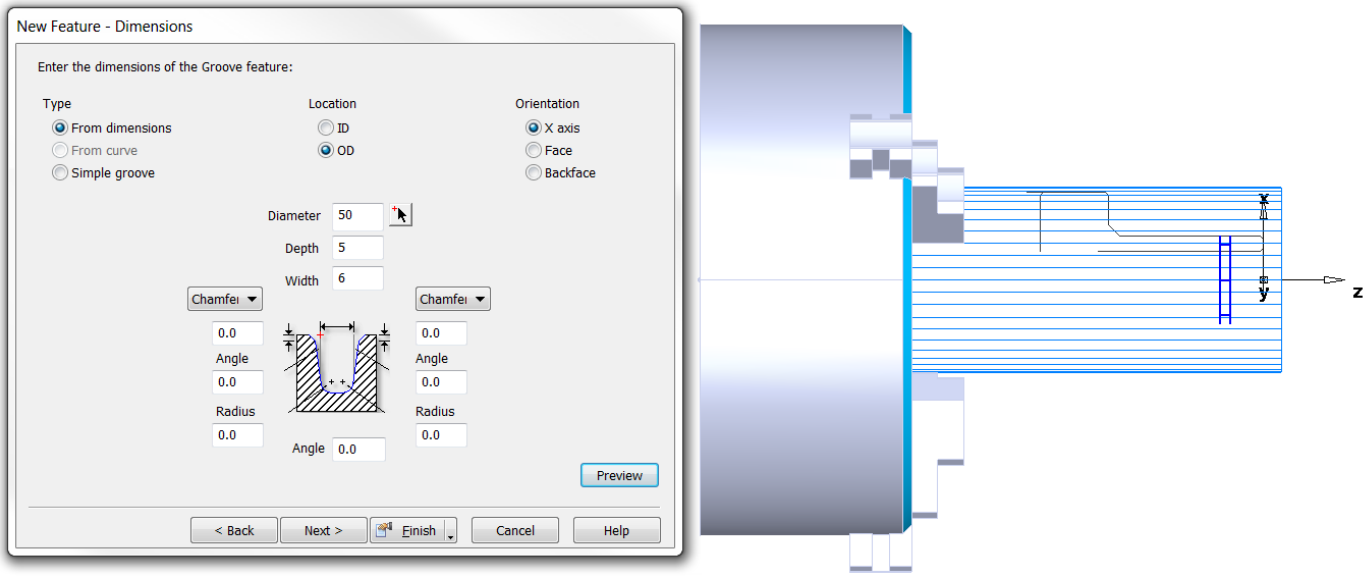


From Dimensions

*If you are working from a print (Paper Drawing) then often the quickest way of creating features is to pull the **Dimensions** from the drawing and enter them directly into **FeatureCAM**. For example you can create a **Groove** by entering its Location and Orientation and **Position (Location)**. Additional attributes for the **Feature** can also be added such as Chamfer or Radius on the groove.*

- **Hole** Feature
- **Groove** Feature
- **Thread** Feature
- **Face** feature
- **Cutoff** feature
- **Bar Feed** feature

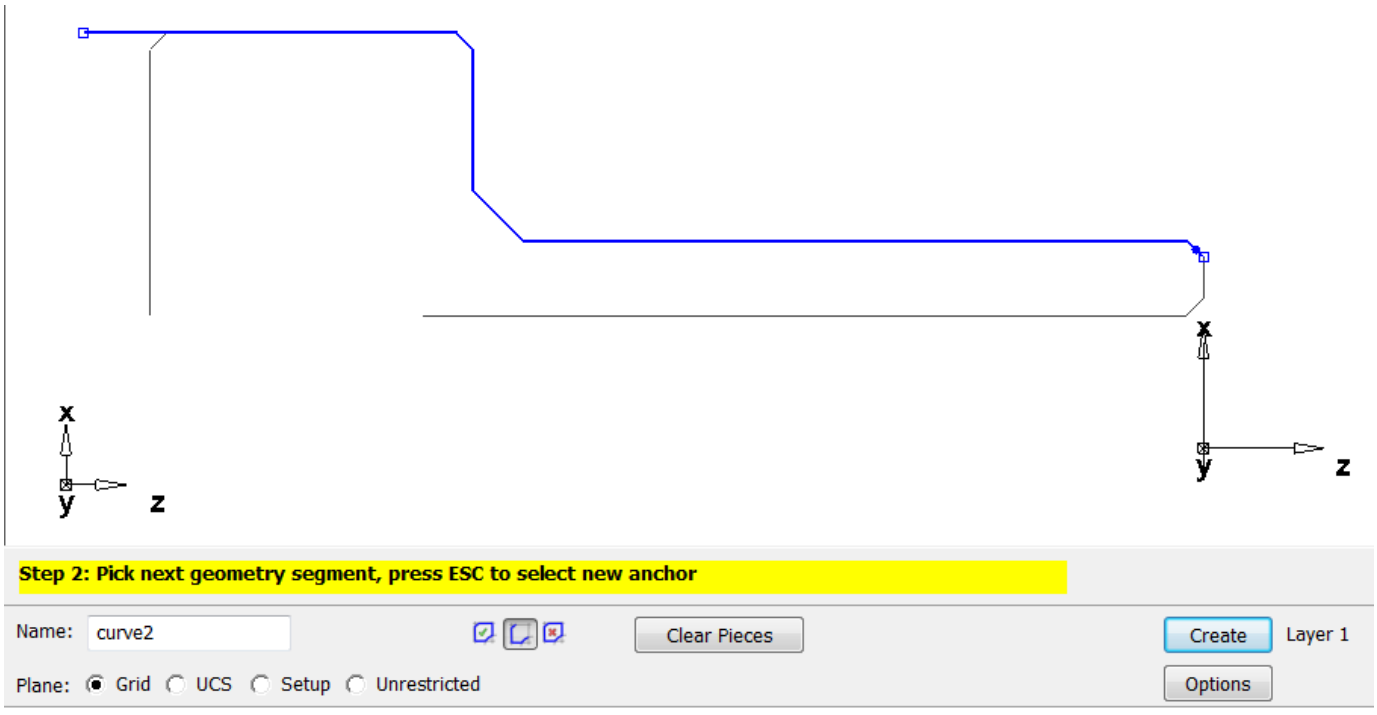
© Copyright 2016 Autodesk, Inc. All rights reserved



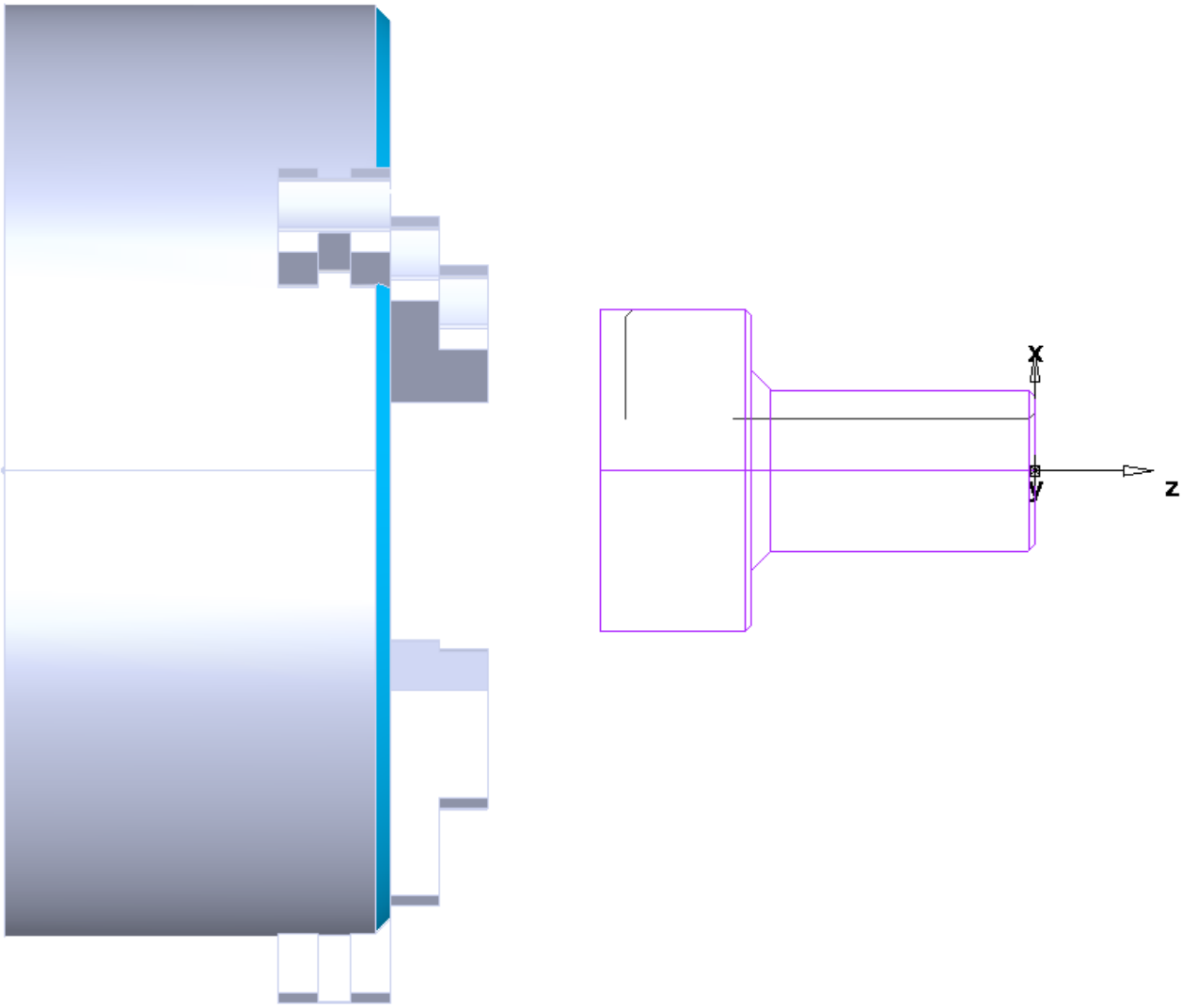
From Curve

*If you have a **print** or a **2D drawing** of a part in electronic format you can create **Curves** for more irregularly shaped parts and then create the **Feature** from the **Curve**. Just select **Pick Curve pieces** select the 2D Geometry and select the **Create** button.*

- **Turn** Feature
- **Bore** Feature
- **Groove** Feature
- **Thread** Feature

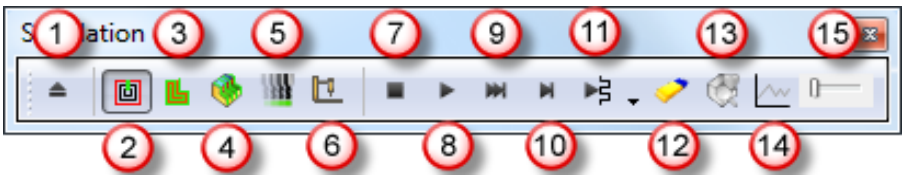


© Copyright 2016 Autodesk, Inc. All rights reserved



*Select **3D Simulation** and then press **Play** (8) on the **Simulation Toolbar**.*

Examine the diagram below of the Simulation ToolBar; Read the purpose of each button



(1) **Eject**: Erases the Simulation from the Graphics window.

(2) Show Centreline: A line drawn at the Centre of the tip of the tool is displayed.

(3) **2D Simulation**: A two-dimensional Simulation showing the regions cut by each operation is displayed. The view is changed to the top view automatically.

(4) **3D Simulation**: A 3D solid Simulation is displayed where the tool is animated through all of its moves.

© Copyright 2016 Autodesk, Inc. All rights reserved

5 **3D Rapid Cut**: In this mode a 3D Simulation is performed but the tool is not animated. Only the final result is displayed. For most parts, the simulation takes only a few seconds to complete. Note this type of Simulation is only available in FeatureMILL3D.

6 **Machine Simulation**: A 3D solid Simulation is displayed where the tool is animated through all of its moves along with the machine tool

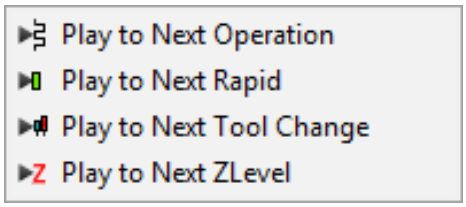
7 **Stop**: Cancels a Simulation.

8 **Play:** Starts the selected Simulation (Centreline, 2D, 3D or Rapid Cut), or restarts a paused Simulation.

9 **Fast Forward to end:** Skips to the end of the animation

10 **Single step:** Moves the Simulation ahead one tool move. The keyboard accelerator for this button is **ALT+F3** or **ALT+Right Arrow**

11 **Play to Next Operation:** continues to simulate until the next operation. This button is actually a fly-out menu. By clicking on the triangle to the right of the button the following additional options are revealed.



Next Rapid simulates until the next rapid tool move.

Next Tool Change simulates until the next tool change.

Next ZLevel will simulate the next Z of a Z level toolpath. For other toolpaths it will play the entire next operation

12 **Clear Toolpath:** Erases any Centreline toolpaths on the screen.

13 **Region of interest** limits the portion of the part that is simulated.

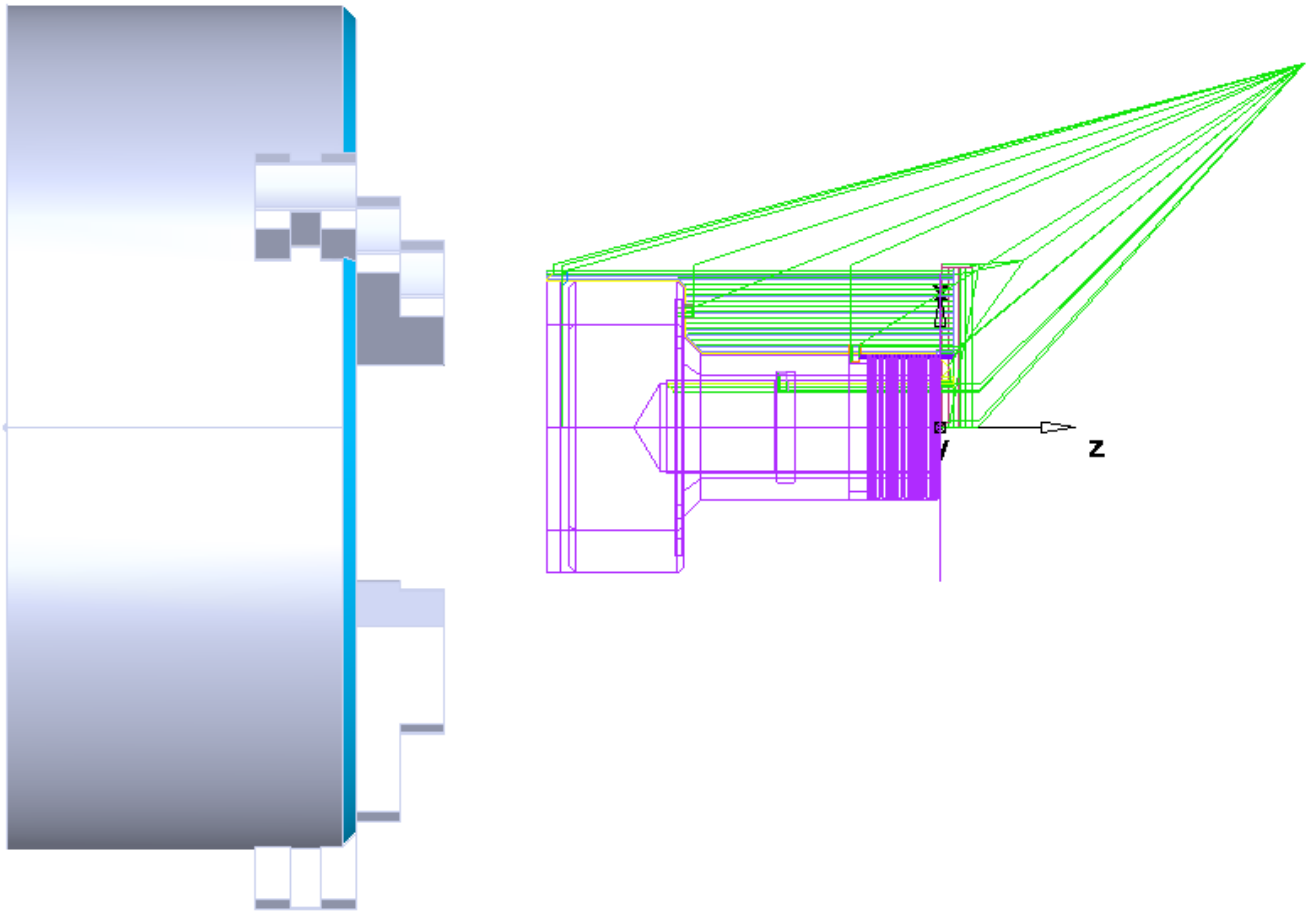
14 **Show tool load** indicates whether or not to display a graph of the tool load when the next 3D Simulation is performed.

15 **Sim Speed:** To adjust the speed of a **Centreline**, **2D** or **3D Simulation**, use the **Slider** on the right-hand side of the controls.

*Slide to the right to speed up, and move to the left to slow down the **Simulation**. The Slider on the **Simulation Toolbar** also affects the display for **Rapid Cut Simulation**. If the Slider is all the way to the right, only the final **Simulation** result is displayed. Position the slider bar further to the left to see slower Rapid results.*

© Copyright 2016 Autodesk, Inc. All rights reserved

You should have a result similar to that shown below.



*A **Centreline** display of the toolpath is shown above where the lines displayed represent the centre of the bottom of the tools movement. Green lines represent rapid movement where blue lines show feed moves. Press the **Eject** button to close the simulation.*

*The image shown is an **isometric View** of the **Toolpath Simulation** with different **colours** representing each operation.*

Select the **3D Simulation** button and press **Play** to show the image shown on the previous page.

Code can only be output after the Simulation has finished. If the simulation fails. NC code will not be output.

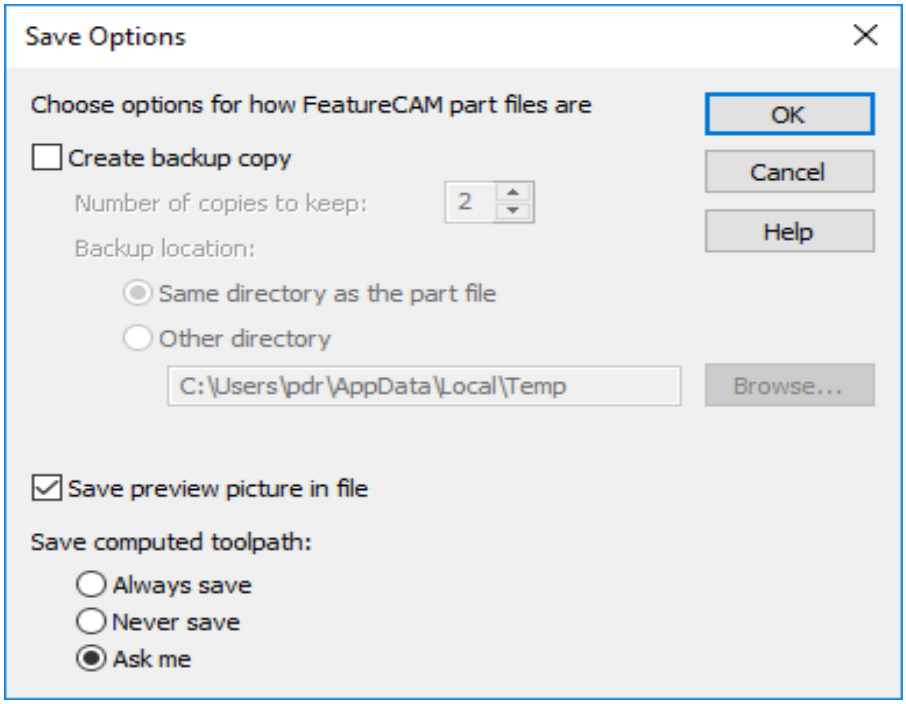
© Copyright 2016 Autodesk, Inc. All rights reserved

Saving your work

- **Question?** What is the difference between **Save** and **Save as**?
- **Answer:** When you are working on a document, it is a good idea to save your work every couple of minutes. In the **File** menu of the program, you will typically find the options "**Save**" and "**Save As**"
- Choosing "**Save**" simply saves your work by updating the last saved version of the file to match the current version you see on your screen.
- Choosing "**Save As**" brings up a prompt to save your work as a file with a different name. For example, you might choose to save a document called "**FeatureCAM_opp1**". This way, you can save you file at different stages and keep multiple versions on your hard drive.
- Choosing "**Save**" and then "**Save As**" is also a good way to make a backup of your file. Just be sure to rename the file something new when you choose "**Save As**" or you will overwrite the current saved version, just like choosing the "**Save**" command would do.

Save Options

- In the **File** menu you have **Save options.**
- See **Save Options** on the next page. Adjust your computer to the same as the options shown.



Create backup copy: - Select this option to save a number of previous versions of your part as you work. Enter the **Number of copies to keep** and set the **Backup location**. When you save a file, the previous version(s) are saved to disk using the name, but prefixed with **Backup of**. The latest version of the file is always saved using the name of your FeatureCAM part.

Compress file:- Select this option to reduce the size of the FeatureCAM files on disk.

© Copyright 2016 Autodesk, Inc. All rights reserved

Compressed files cannot be read into earlier versions of FeatureCAM.

Save preview picture in file: - Select this option to store an image of the part in the file. This image is displayed in the **Open** dialog.

Save computed toolpath: - For certain parts, generating toolpaths can be time-consuming and you may want to save them for the next time you open a part. The setting of Save computed toolpath controls the default behaviour for saving toolpaths. Select from **Always save**, **Never save**, or **Ask me** to be prompted each time you save a part. The toolpath is saved as an **.fmp** file.

This is different than saving the NC text file that the NC machine reads. Instead you are saving the FeatureCAM internal toolpath representation.

If you upgrade to a newer version of FeatureCAM, the toolpath is disregarded and recomputed.

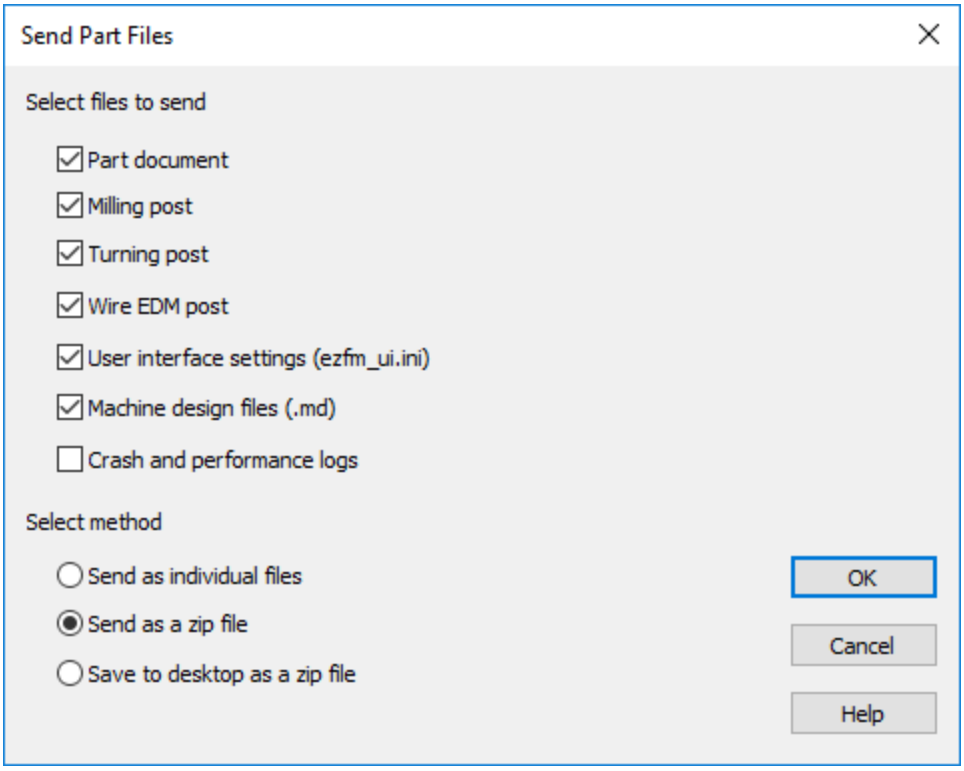
***OK: -** Click the **OK** button to save your settings and close the dialog.*

***Cancel: -** Click the **Cancel** button to close the dialog without saving any changes.*

***Help: -** Click the **Help** button to open this Help topic.*

Send Part Files dialog

To display the **Send Part Files** dialog, select **File > Send** from the menu.



© Copyright 2016 Autodesk, Inc. All rights reserved

Select the files you want to send from:

Part document: - The current **.fm** file.

Milling post: - The current milling **.cnc** file.

Turning post: - The current turning **.cnc** file.

Wire EDM post: - The current Wire EDM **.cnc** file.

User interface settings: - The **.ini** file containing your user interface preferences.

Machine design files (.md): - the current **.md** file.

Crash and performance logs: - Select this option to create a separate .**zip file** of your system's crash and performance logs.

Select how you want to send the files:

As individual files: - The individual files are attached to a new email.

As a zip file: - The files are zipped and the **.zip** file is attached to a new email.

Save to desktop: - The files are zipped and the **.zip** file is saved to your desktop.

Size confirmation

If you are sending **.zip** files, a **Size Confirmation** warning dialog is displayed:

The average email server limits attachments to 10 MB.

The total size of the attachment(s) is: N (where **N** is the size of your attachment(s))

Would you like to continue sending?

Click **Send** to attach the files to an email.

Click **Don't Send** to keep the files on your desktop.

Click **Cancel** to discard the files.

FeatureCAM file types

You can save several different types of files for your part.

- .op is the Manufacturing Operation Sheet and is the same information shown on the **Details** tab when you select **Operation List**.
- .tl is the Manufacturing Tool Detail Sheet and is the same information shown on the **Details** tab when you select **Tool List**.
- .txt is a text file containing the NC code for the particular part file.
- .tdb is a FeatureCAM tooling database that contains just the tools you used to create the part.
- .fdb is a FeatureCAM material database that contains the feed and speed tables used for the part.
- .cdb is a FeatureCAM machine configuration database that contains the settings for default attributes.

© Copyright 2016 Autodesk, Inc. All rights reserved

Part Documentation

The Part Documentation dialog enables you to add comments to the printed documentation and set a permanent preview picture for the part.

- To display the Part Documentation dialog, select **File > Part Documentation** from the menu.

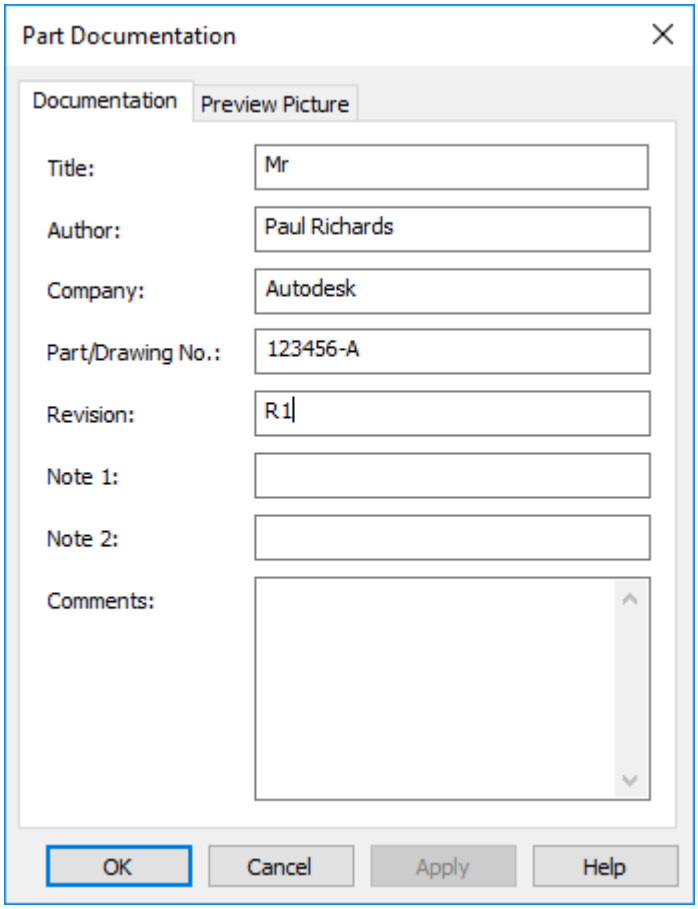
The **Part Documentation** dialog contains two tabs:

Documentation

Preview Picture

Documentation tab

- On the Documentation tab of the Part Documentation dialog, optionally enter a **Title, Author, Company, Part/Drawing No., Revision, Note 1, Note 2, and Comments**.
- To print these values along with the documentation, select Comments in the **Printing Options** dialog.



*If you use the custom setup sheet add-in, these values are copied to the **Setup Sheet Options** dialog.*

Preview Picture tab

On the **Preview Picture** tab of the **Part Documentation** (see page 95) dialog, you can set a permanent preview image for the part. This image is displayed in the preview pane in the **File > Open** dialog when you select a file.

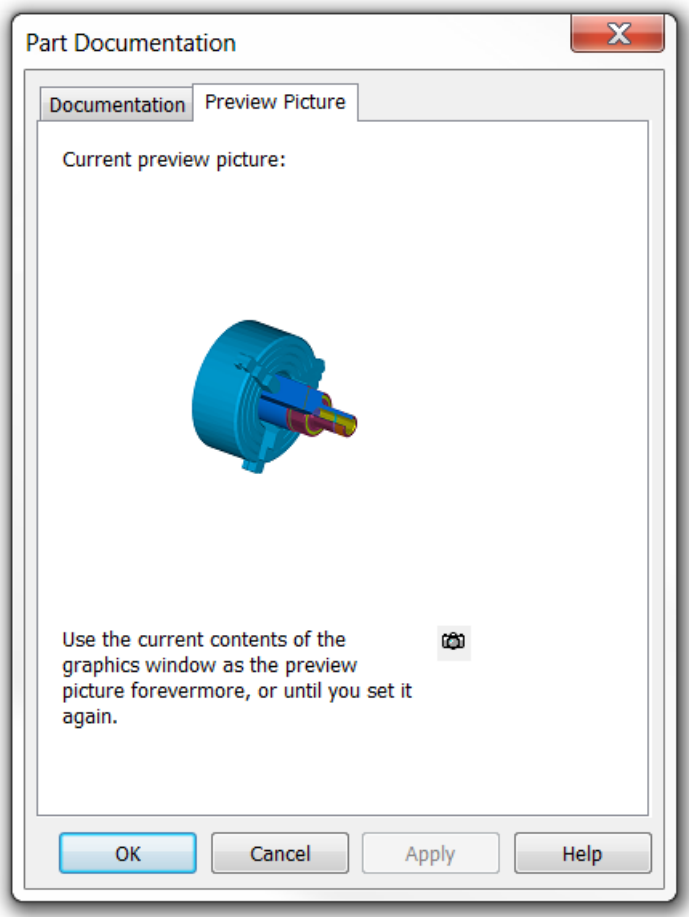
© Copyright 2016 Autodesk, Inc. All rights reserved

This image is normally updated each time you save the part. If you want to store a permanent image with the part:

9 Create the view of the part you want to store.

10 Click Update preview picture

*The current view in the graphics window is displayed as the **Current preview picture**, for example:*



11 Click **OK** to save the preview picture or **Cancel** to close the dialog without saving the preview picture.

Saving your settings

- FeatureCAM uses two .ini files to store your settings:
- **ezfm_ui.ini** contains toolbars, dialog locations, graphics settings, colours, and other user settings.
- **ezfm_mfg.ini** contains manufacturing defaults and .cfg settings.

The ezfm_mfg.ini file contains default values for both inch and metric attributes. For example, there is an entry for **zrapid** in inches and also an entry for **zrapid_mm** in mm. In general, the name for the metric attributes is the same as the inch name, with a suffix of **_mm**.

© Copyright 2016 Autodesk, Inc. All rights reserved

*There are three items from the **Options** menu affect the communication with the ezfm_ui.ini and ezfm_mfg.ini files:*

- **Options > Save Settings Now: -** Writes the current settings to the files.
- **Options > Reload Settings: -** Reads the settings contained in the files into the program.
- **Options > Save on Exit: -** Saves the current settings when you exit the program. If this option is not selected, the settings for your current session are not saved to the files when you exit.

Import/Export

- You can import and export various file types into and from FeatureCAM.
- The settings for import and export are in the **Import/Export Options**
- To open this dialog select **File > Import/Export Options** from the menu.

Import/Export Options

The **Import/Export Options** dialog contains settings for importing and exporting.

To display the **Import/Export Options** dialog select **File > Import/Export Options** from the menu.

The dialog has three tabs:

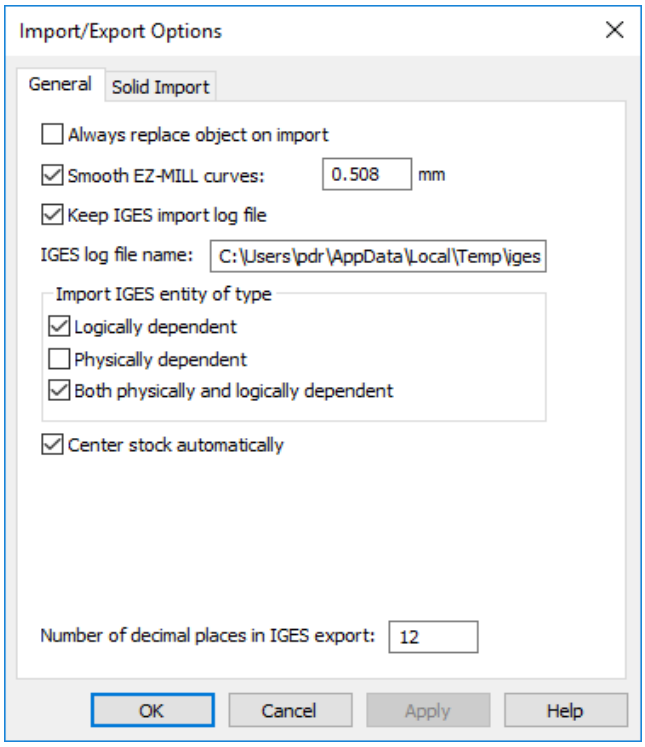
General

Solid Import

Digitized Data

General tab

The General tab controls how FeatureCAM **Imports** or **Exports** files. It contains the following options:



© Copyright 2016 Autodesk, Inc. All rights reserved

Always replace object on import: - Select this option to overwrite objects of the same name when you import. Deselected prompts for every replacement. You can't have two objects with the same name.

Smooth EZ-MILL curves: - Select this option to improve EZ-MILL curves on import because FeatureCAM has a higher resolution curve format.

Keep IGES import log file: - Select this option to keep a log of the import process for later review or troubleshooting.

IGES log file name: - Enter the path and filename for saving a record of the import process.

Import IGES entity of type:

Logically dependent: - can exist by itself but is also referenced by another entity (case of groups or grouping situation).

Physically dependent: - generally, you should deselect this option to indicate you do not want physically dependent entities to be imported. But if you are having trouble with your IGES file and it is not importing properly, try selecting this option and re-importing. Entities in an IGES file are marked to be either physically dependent or not. Those that are marked to be physically dependent are entities that are used in the construction of other entities. For example, a trimming loop is physically dependent upon the trimmed surface that uses it. The trimming loop is not important all by itself, but is instead a building block of something else. The trimmed surface is in turn marked physically dependent to indicate that it is a building block of a parent entity, a solid. By selecting **physically dependent**, you cause FeatureCAM to show everything in the IGES file - not just the top-level entities that are usually shown, but also all of the lower-level building blocks. This is generally not what you want. Generally you want only the top-level entity, for example the solid.

Both physically and logically dependent: - meets both logically and physically dependent criteria (is referenced by at least two other entities**: -** the parent for the logical link cannot be the parent for the physical as well).

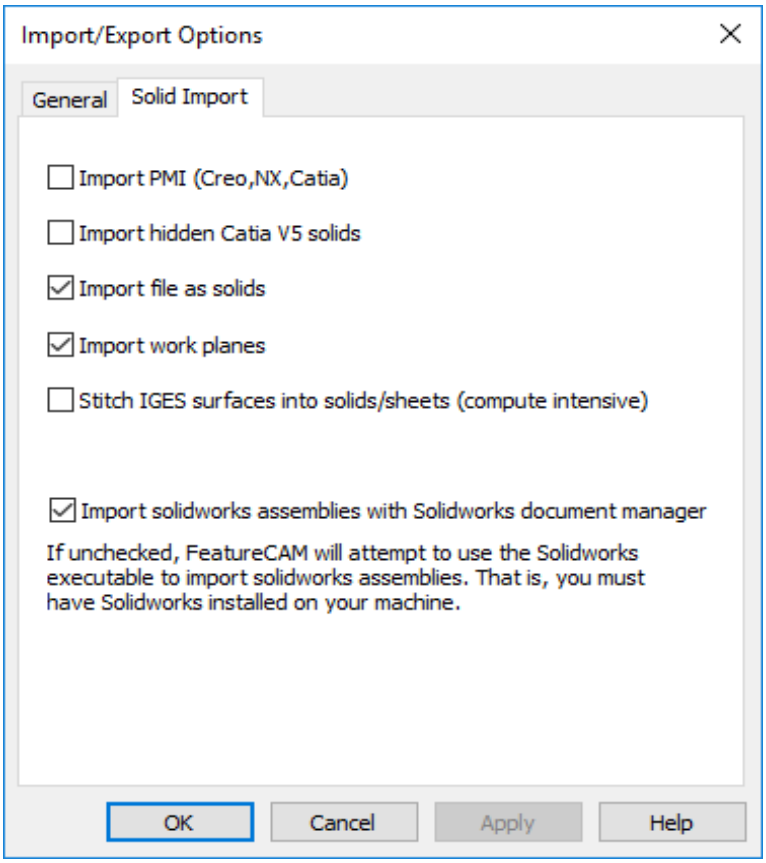
Centre stock automatically: - Select this option to automatically size and position the stock so that it covers the imported data.

Number of decimal places in IGES export: - determines how finely data is exported to the IGES format.

Solid import tab

The **Solid Import** tab contains the following options:

© Copyright 2016 Autodesk, Inc. All rights reserved



Heal Catia solids: - Select this option to try to repair the faces or surfaces contained in the CATIA .mod file. This option can be time-consuming because it attempts to:

- retrim the surfaces/faces against each other.
- force edges of surfaces to actually lie on the surface.
- Simplify surfaces like converting a general surface into a cylinder.

Import hidden Catia V5 solids: - Select this option to import all parts contained in the CATIA file even if those parts are hidden.

Import file as solids: - Select this option to import files as solid models. If this option is deselected, the models are imported as surface models. We recommend that you import models as solids. If this option is selected and the solid fails to import properly, you are asked if you would like to attempt to heal the solid to try and fix the import problem.

Import work planes: - When selected, planar surfaces in solids are imported. When deselected, planar surfaces in solids are ignored.

Stitch IGES surfaces into solids/sheets: - Select this option to try to create one or more solids or sheets from an IGES file. If an IGES file contains more than one solid or sheet, this is the most efficient way to create the multiple objects from the file.

Importing Files

1. Use the **File > Import** menu option to load a CAD model from file.
2. To import a CAD model from a file:

© Copyright 2016 Autodesk, Inc. All rights reserved

3 Open a new or existing part file. You must have a part open to import geometry.

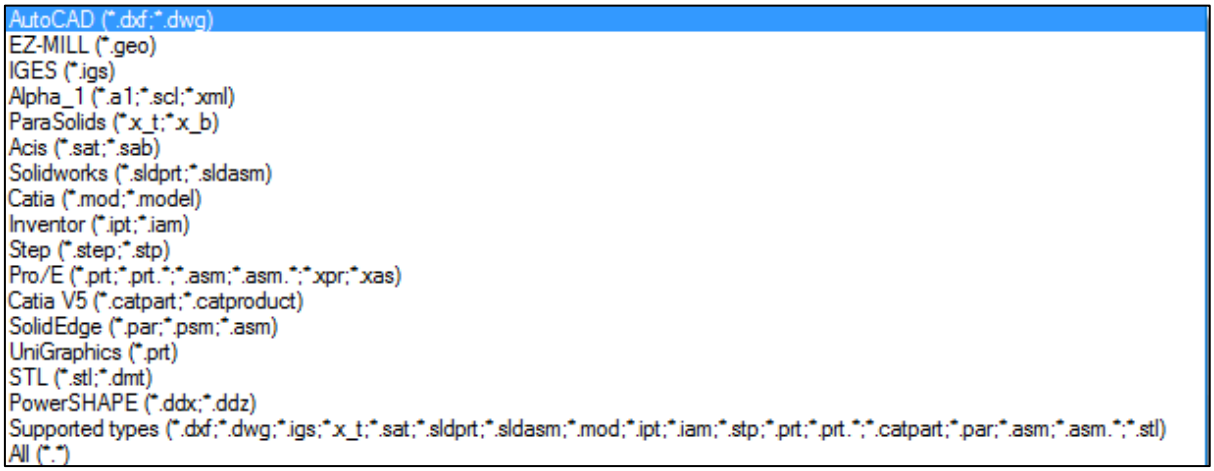
4 If you want to set or change the import options, select **File > Import/Export Options**.

5 Select **File > Import** from the menu. The **Import** dialog is displayed.

6 Select the file you want to import, and click **Open**.

7 *A message asks if you want to review the log file: click **Yes** to display the results, or click **No** to continue. The **Import Results** wizard is displayed.*

8 Follow the instruction in the wizard to complete the process.



*Unless you have specifically ordered the FeatureCAM **.step** or **Stp** file import, you might get an error to say that this option needs to be licenced.*

*If this is the case then use the **Import using exhange** where the .step file import option is **free**.*

Wherever possible avoid importing .iges files as these are surfaces.

*Please import solid models, where possible. The best option is Parasolid which has a file extension of **x_t** or **x_b** which is a binary file which is compressed. Another file format is **.SLDPRT** which is a **solidworks** file.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

FeatureCAM Basic Turning ... 3
Introduction ... 3
Creating a New Turn/Mill Part Document ... 4
Basic Turning Exercise ... 6
Turning Input Modes ... 8
Geometry Creation Techniques ... 9
Curve Creation ... 12
Creating a Face Feature ... 14
Creating a Turn Feature ... 23
Multi –Turret options. ... 25
Creating a Bore Feature ... 26
Creating a Groove Feature ... 30
Creating a Thread Feature ... 35
Results ... 40
2D Simulation ... 40
3D Simulation ... 41
Rapid moves to home position ... 41
Part View All Operations ... 42
NC Code output example ... 44
Part Documentation ... 45
Changing Post Processors and Saving NC Code ... 45
Tool Mapping ... 46
Tool Numbering ... 49
Custom Setup Sheet Add-In ... 49
Exercise 2 ... 54
Turn Feature ... 57
Cut-Grip ... 65
Second groove Feature using Cut-Grip ... 66
Groove from Curve ... 70
Face Boring ... 70
Offset Boring ... 70
Boring below Centreline ... 70
Boring with Canned Cycles ... 71
Tool Nose Radius Compensation ... 71
Back Face Boring ... 71
Back Boring ... 71
Boring with a groove tool ... 72
Turning Exercise 3 ... 72
Exercise 3 in detail ... 74
Turning Properties – Rough - Turning ... 74
Face Turning ... 75
Stock Curve ... 78
Turning Setups Explained ... 80
How do setups relate to UCSs? ... 80
Setups dialog ... 80
Setup – Definition ... 82
A few quick notes about Part Name ... 83
Creating a new Setup2 ... 83

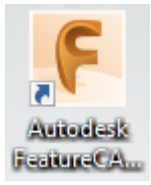
© Copyright 2016 Autodesk, Inc. All rights reserved

Specifying the machine simulation file 87
Setting up a part for multi-spindle turning 87
Setups..... 87
Associating features with setups..... 88
Moving the part 88
Machine Simulation 88
Sub spindle overview..... 88
Creating a Sub spindle feature 89
Setting up a part for multi-turret turning 91
Turning Exercise 4 Lightbulb (Class exercise)..... 93
Lightbulb drawing exercise in detail 93

© Copyright 2016 Autodesk, Inc. All rights reserved

FeatureCAM Basic Turning

Introduction

*Double click on the **FeatureCAM** Icon* 

this will start FeatureCAM and allows you to create a new Part document.

*In this **FeatureTurn** module the user will learn **basic 2-axis***

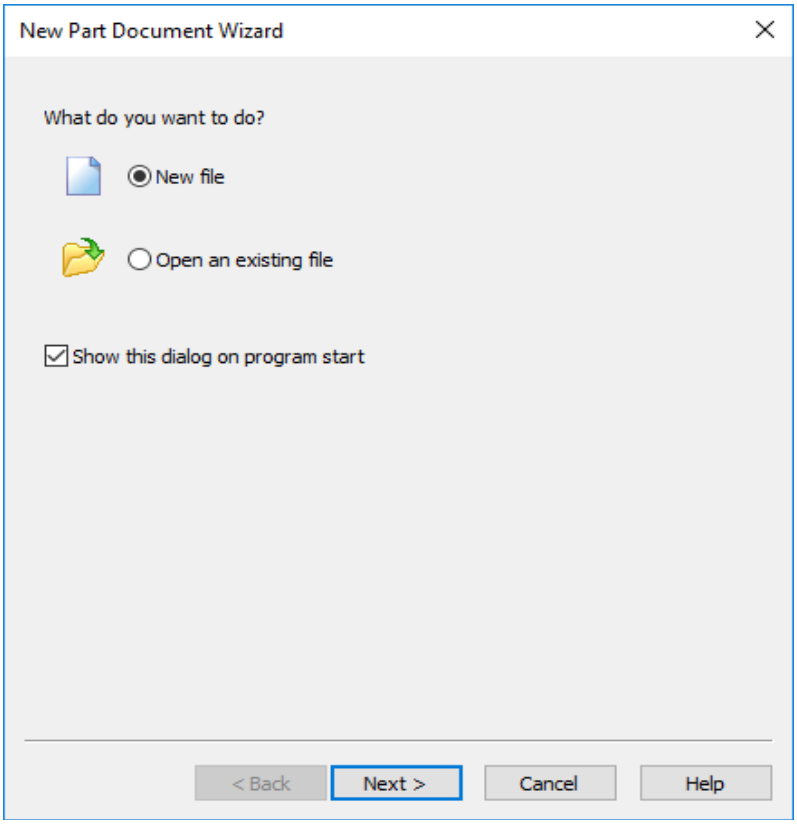
X and Z turning, boring, and grooving along with the basic user interface and different stock types. At the end of the module the user will be able to generate a program for a simple 2 axis part.

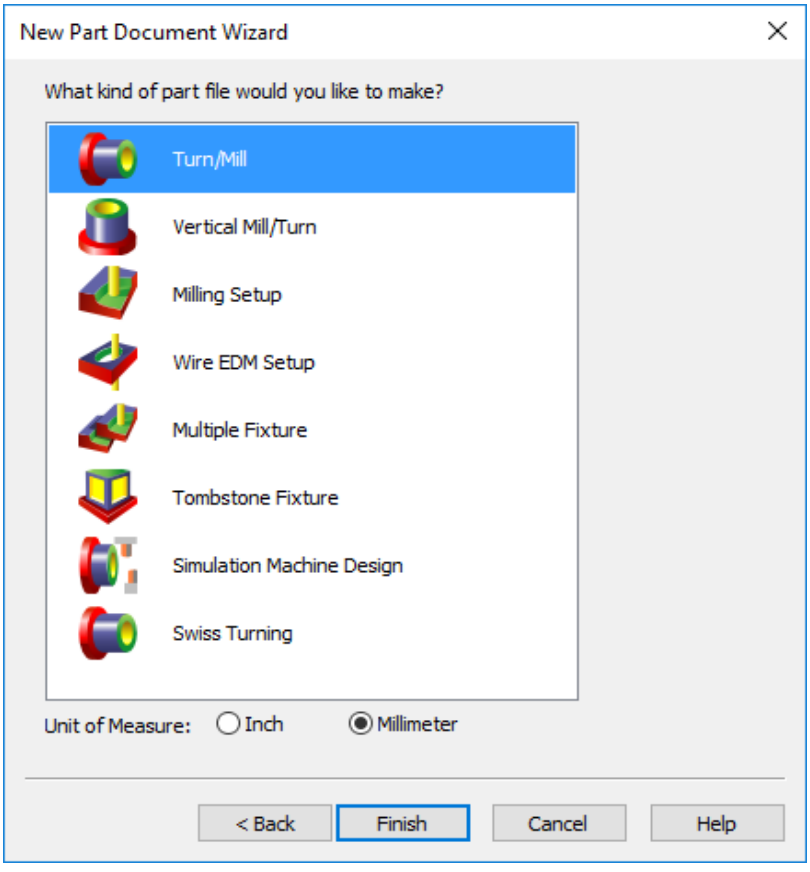
*This chapter will show the user the basic steps required to create a finished **FeatureTURN** Part. The following Tutorial is designed for **Metric** use.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Creating a New Turn/Mill Part Document

- Open a **New file> Turn/Mill** Part Document, the following menu will appear. Select **Next**.

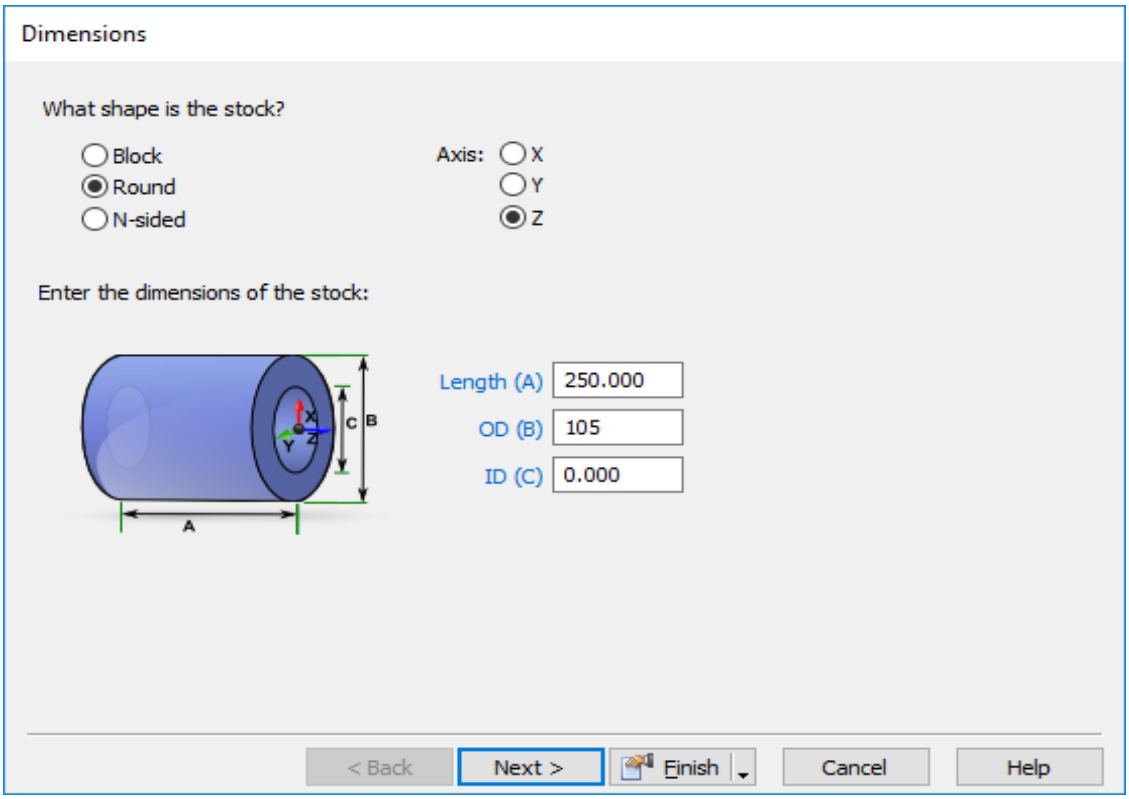




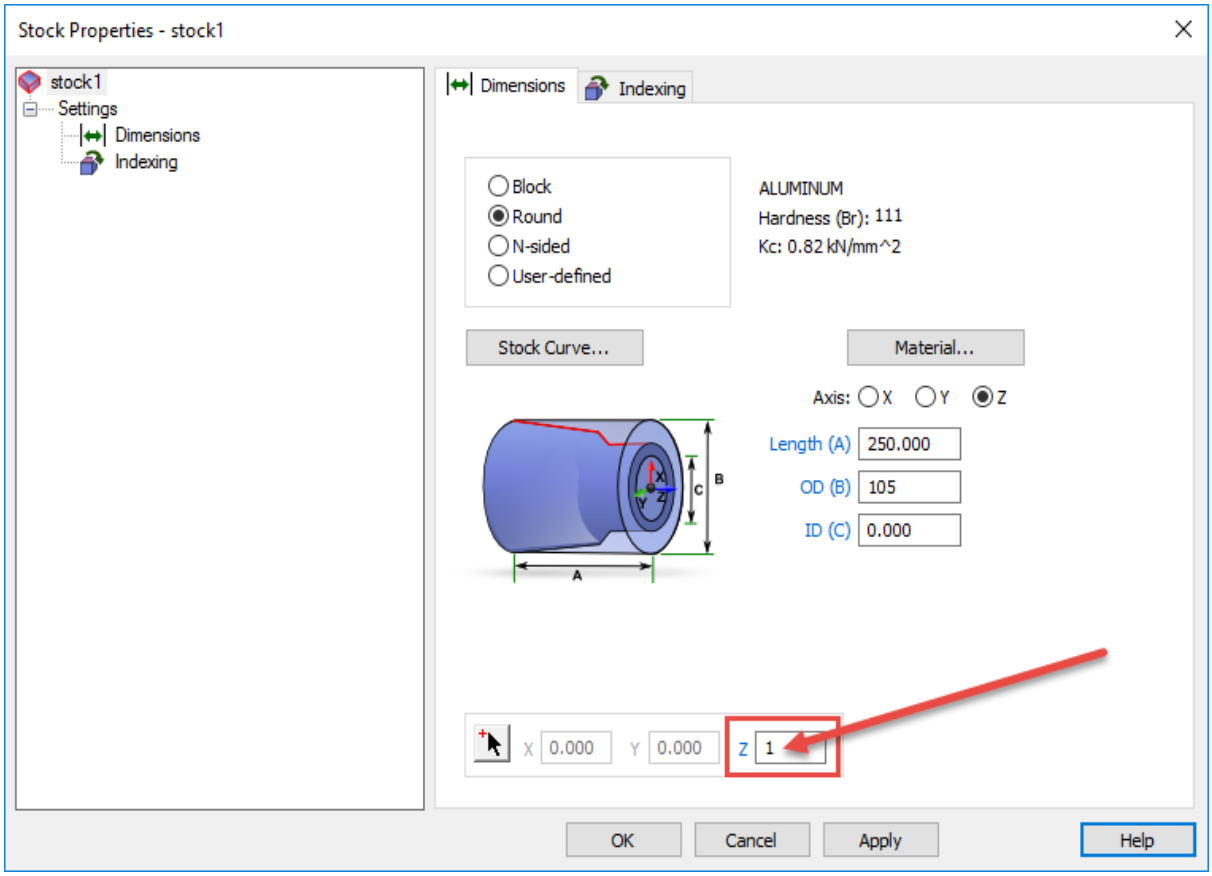
© Copyright 2016 Autodesk, Inc. All rights reserved

1 Select **Finish.**

2 For your Stock enter **Length 250mm** and **OD 105mm** then Select **Finish**.

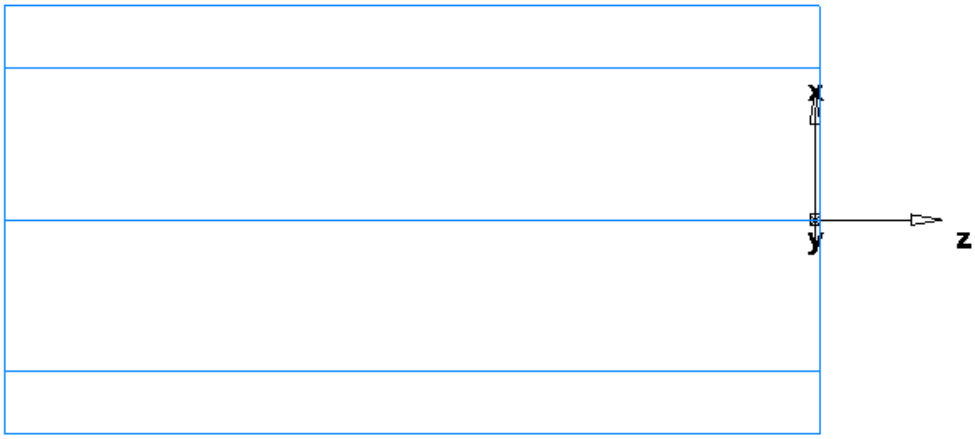


3 We need to set excess stock at the front of the part after selecting **Finish** you will be presented with another form. See the image below.



© Copyright 2016 Autodesk, Inc. All rights reserved

4 Enter **1mm** in **Z** as shown in the menu on the previous page. This will now set the **Z Setup1** as shown below.



5 Hide stock by selecting the **Stock1** in **PartView** and Right click – Select **Hide Stock**.

6 In the **Z Offset** enter **-1mm** this will move the Offset **1mm** to the left of the material face leaving stock to be turned off later.

Basic Turning Exercise

1 Before we start please select **Options/File Options/Existing Files**.

2 **Select Browse. Then select the following location. C:\Training_Data\FeatureCAM Course Data 2017.**

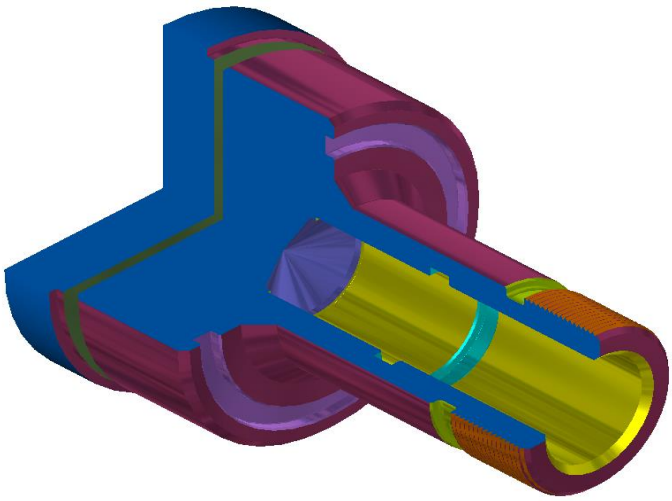
3 Select **Apply**. When we save or import files FeatureCAM will take us to this location.

4 Please save all files in **Coursework**.

5 Tool Crib = **Training_Crib_Metric**.

6 Post processor used = **Fanuc21i-T.cnc**

The part we are going to machine is shown below.

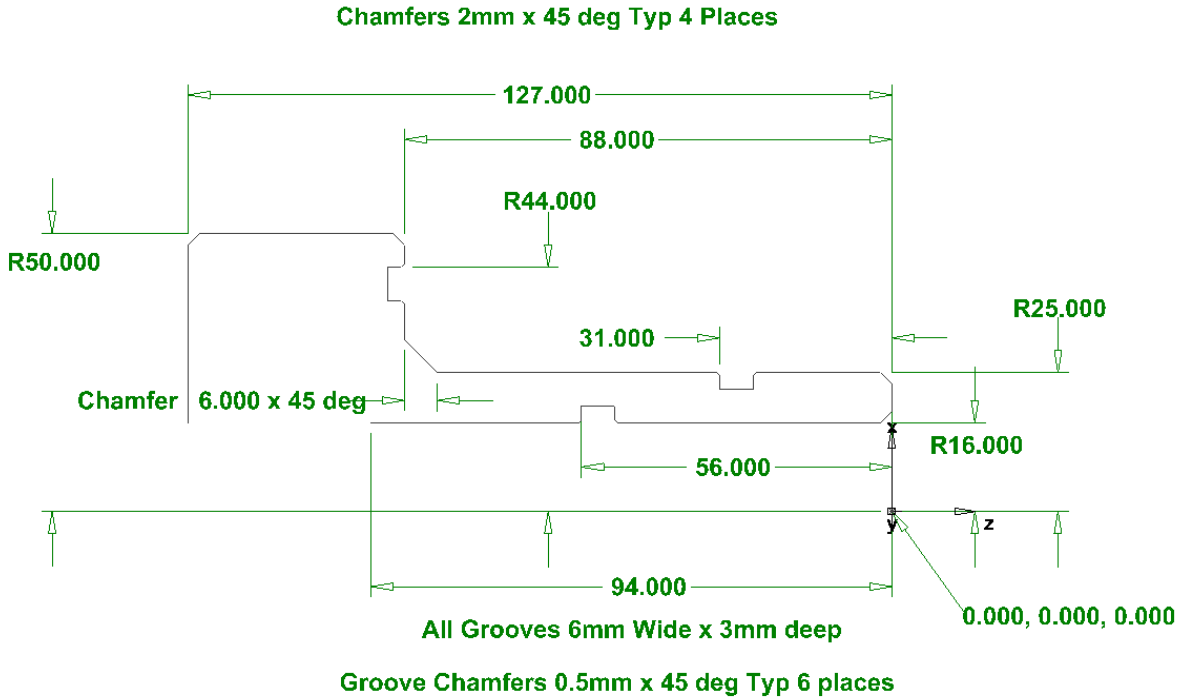


© Copyright 2016 Autodesk, Inc. All rights reserved

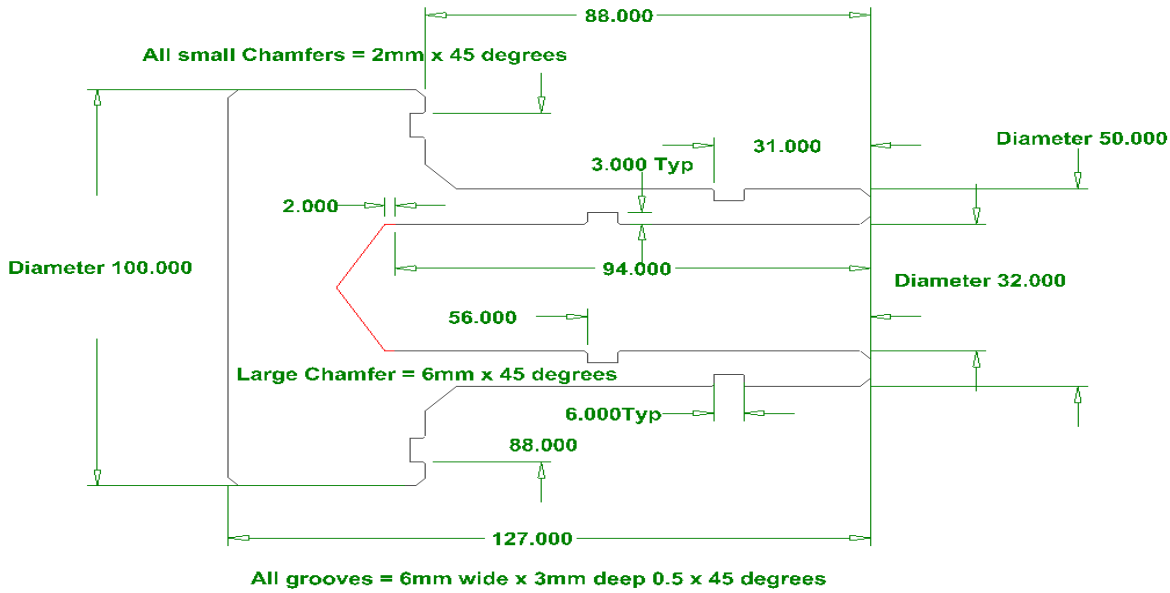
*The example shown is a simple turned part designed to introduce you to several of the drawing and machining tools available in **FeatureTURN.** The tutorial walks you through creating the part and introduces you to **FeatureTURN's** automation and ease of use.*

*For this exercise the **ID**, **OD** and **Face** grooves do not have to be defined by creating geometry, these features will be created automatically later by creating **Groove** features **from dimensions**.*

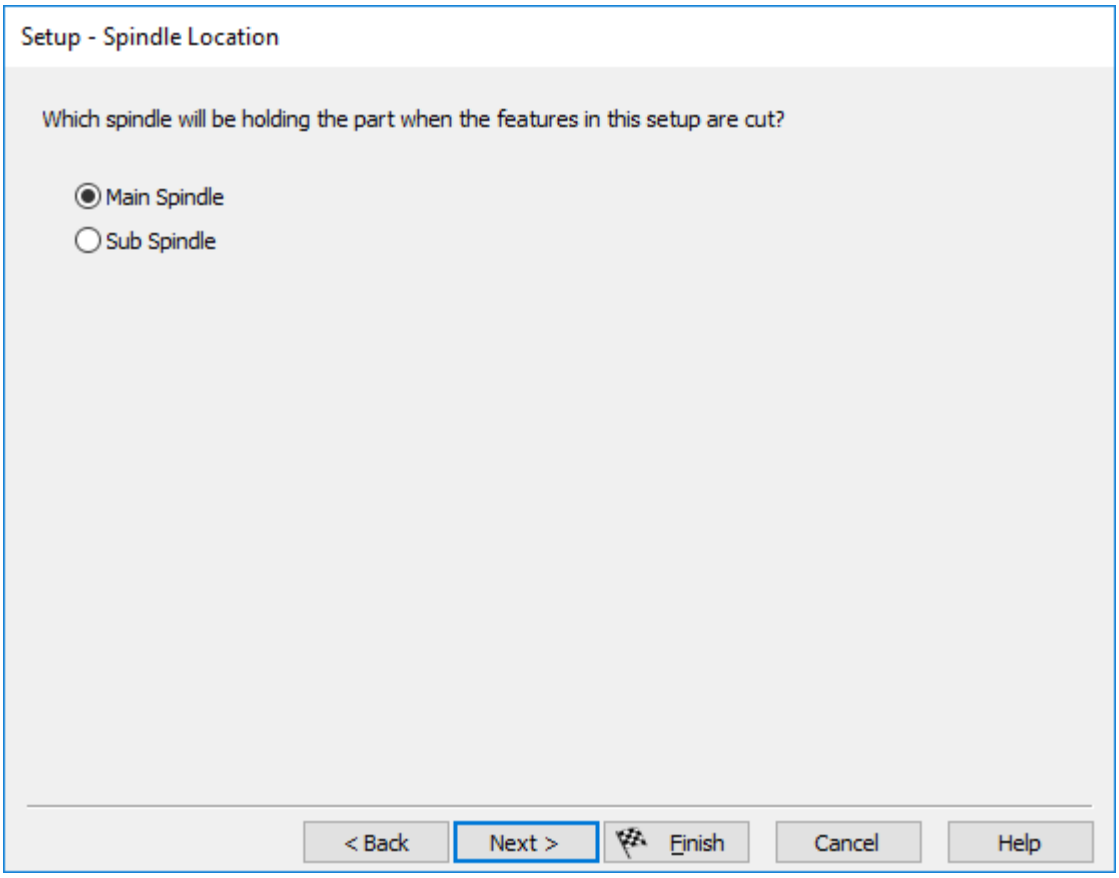
Radius (RZ)



Diameter (DZ)



© Copyright 2016 Autodesk, Inc. All rights reserved



7 **Setup1** – Spindle Location, the **Main Spindle** will always be selected when there is no **Sub Spindle** being used.

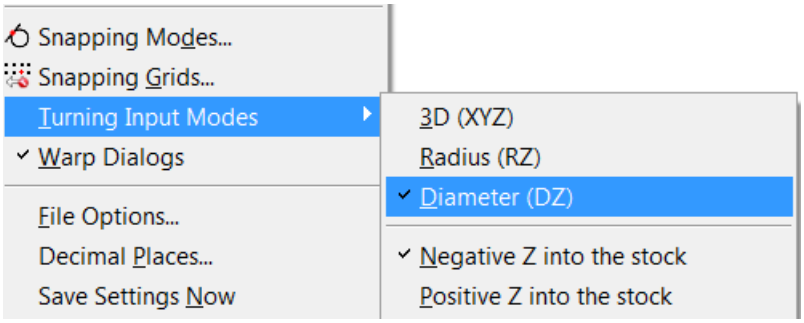
8 Select **Main Spindle**

A sub spindle is a second spindle at the tailstock end of the machine opposing the main spindle and used to run the second side of the part. Machine dependant.

9 Select **Finish**.

Turning Input Modes

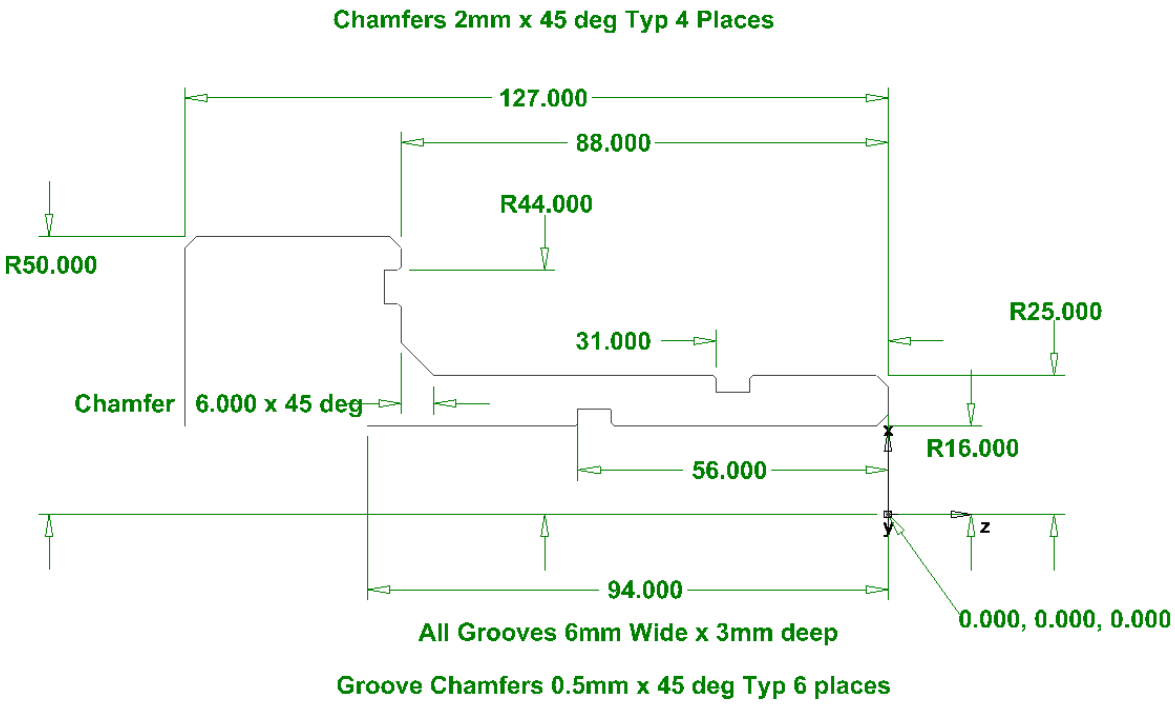
*Before we start to create out geometry we need to select **Options>Turning input modes** and select the following **Radius (RZ)** or **Diameter (DZ)** input modes. Our drawing is dimensioned in **Radius (RZ)** and **Diameter (DZ)** values. You choose your preferred values.*



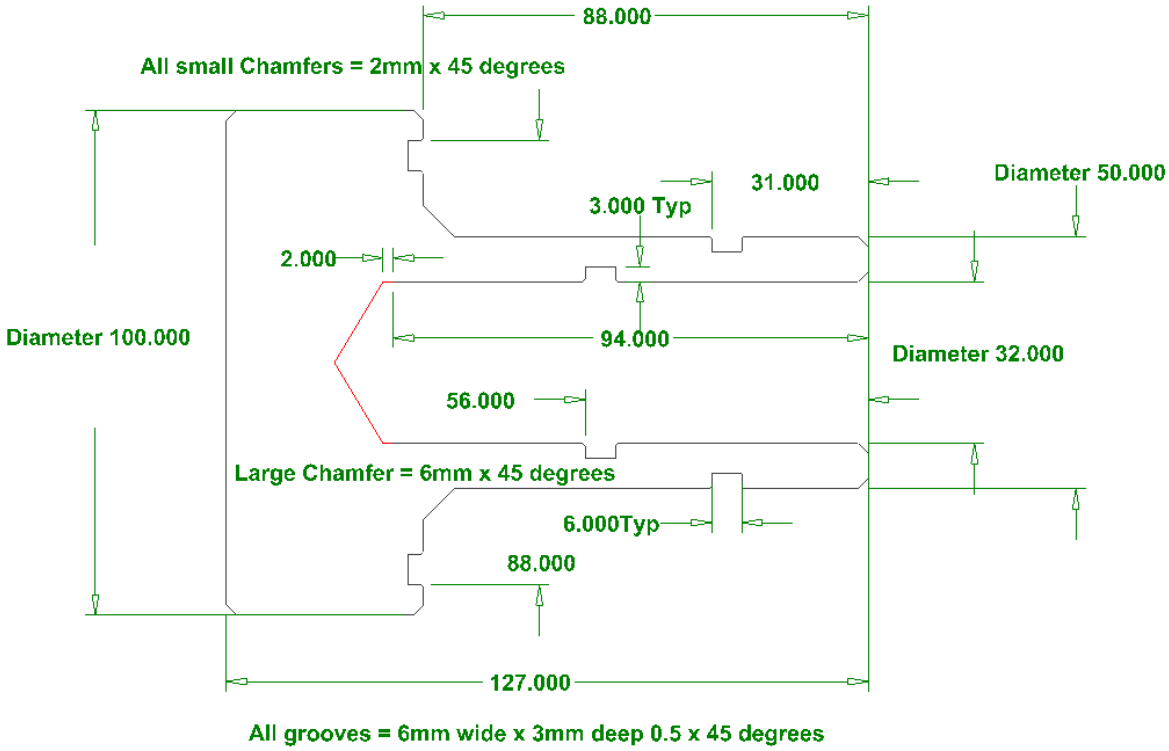
© Copyright 2016 Autodesk, Inc. All rights reserved

Geometry Creation Techniques

Radius (RZ)



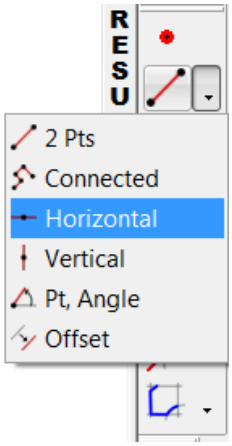
Diameter (DZ)



© Copyright 2016 Autodesk, Inc. All rights reserved

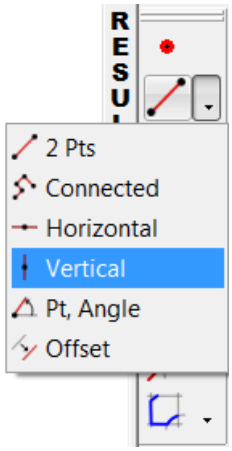
There are many ways to draw this job but the following method is by far the easiest. We are going to use Diameter values.

10 From Geometry Tool Bar. Select Line **Horizontal** as shown.



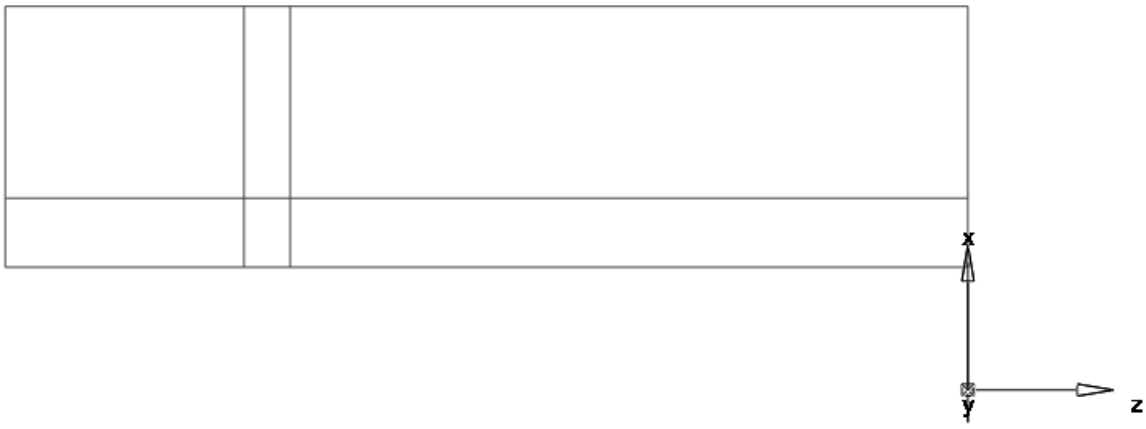
11 Input the following diameter Values **32mm, 50mm, 100mm.**

12 Then Select the Line **vertica**l and input the following values



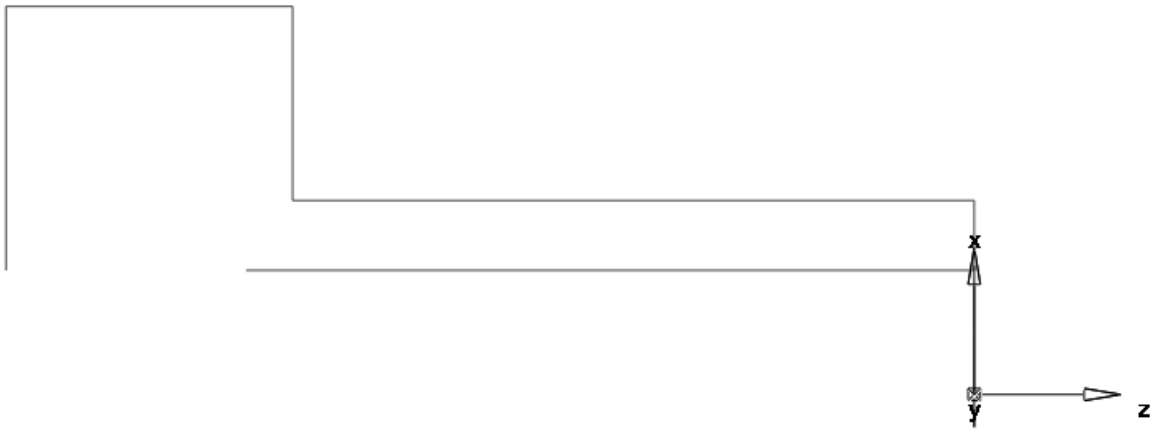
13 **0, -88mm, -94mm, -127mm.**

14 Clip the surrounding lines as shown.

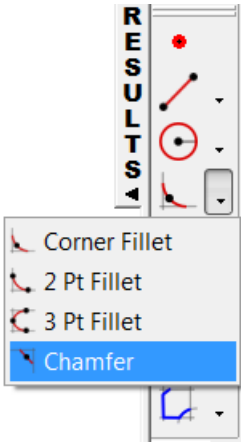


15 Now Clip the unwanted geometry as shown on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved



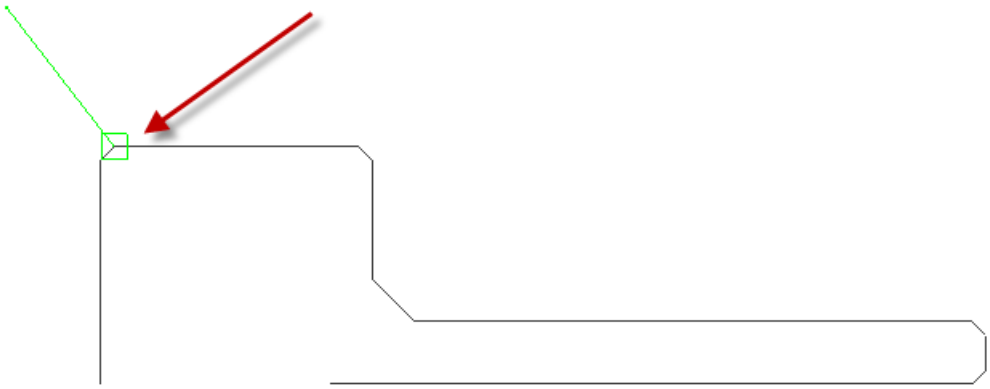
16 Now add the Chamfers by using the Chamfer command.



17 The large chamfer is **6mm** x **6mm** all other chamfers are **2mm** x **2mm**. We have one more item to change.

We need to extend the OD (Outside Diameter) to allow the Cut-Off tool to start at the correct diameter.

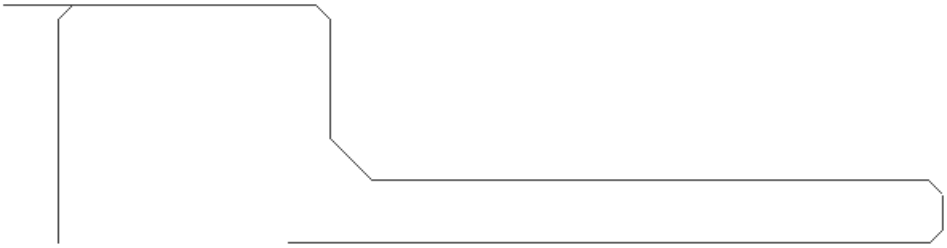
18 To do this, select a **Line from two points** from the **Geometry** menu. Snap to the chamfer as shown.



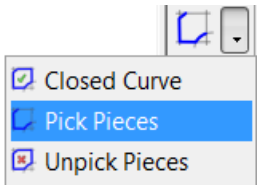
19 We now need to type in a few values as shown. Use the Tab key to move to the next field and type in an Angle of 180 and a length of 10mm

© Copyright 2016 Autodesk, Inc. All rights reserved

20 That is it the Geometry is now complete as shown.

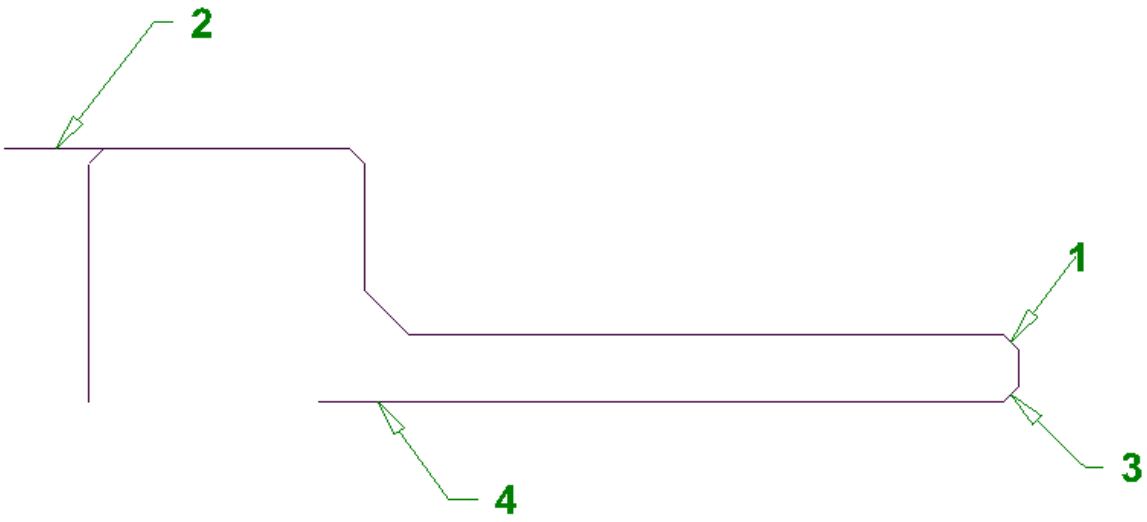


Curve Creation



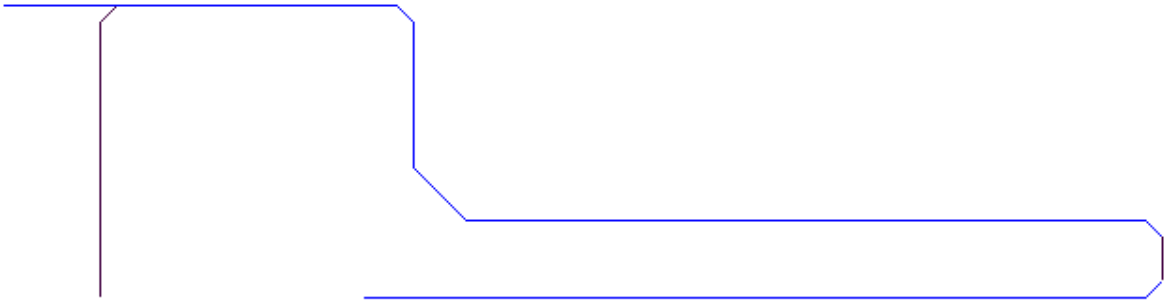
21 Select **Pick Pieces** from the Geometry Toolbar.

22 Select the **Pick Pieces** Icon Click on locations **1, 2**, as shown in the image below. Each line segment will change colour when it is selected. Name this curve "**od profile**" and press the **ENTER** key. **Chain** the Bore **ID** curve by clicking on locations 3 and 4. Label it "**bore profile**" and press the **ENTER** key. Your drawing should look like the one on the next page.



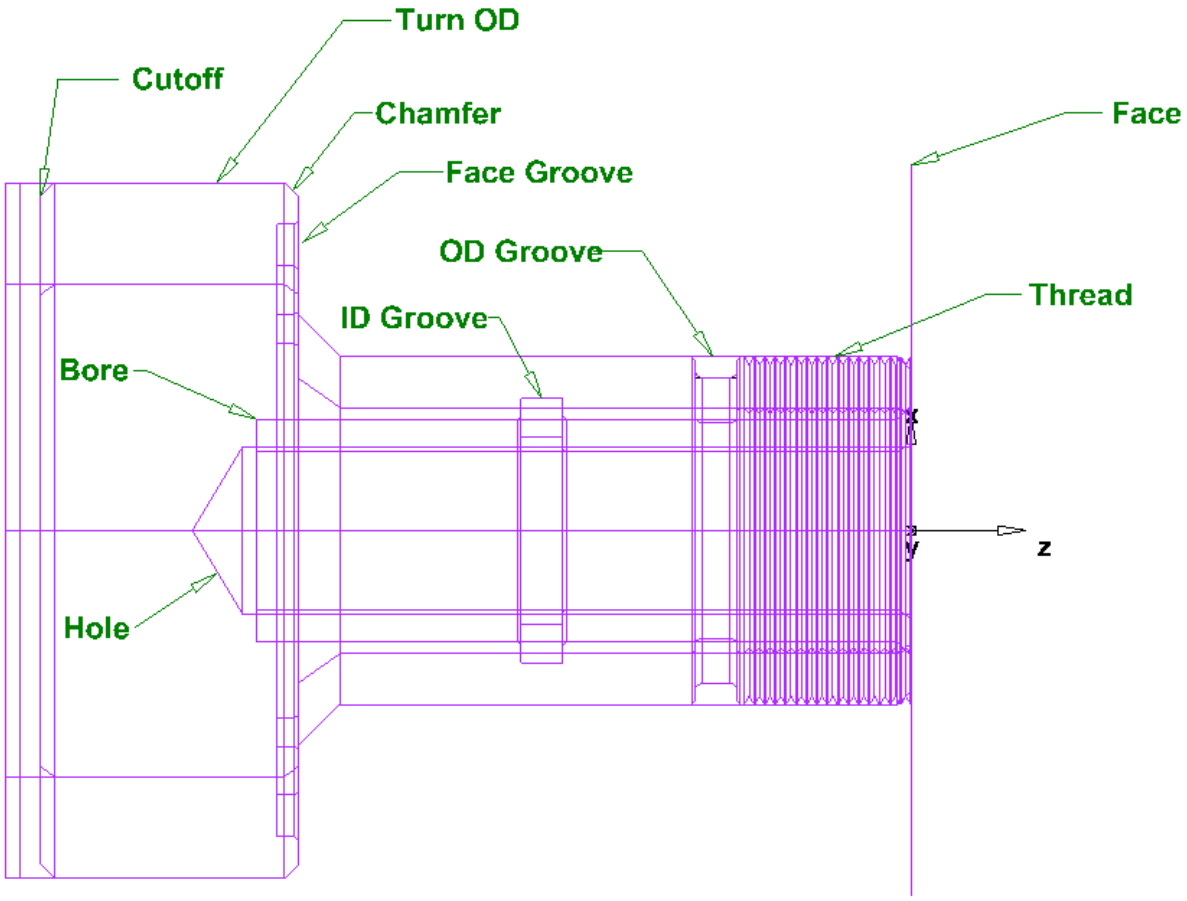
*Remember **Curves** are Blue and **Geometry** is **Black.***

© Copyright 2016 Autodesk, Inc. All rights reserved



23 You should now have **two Curves** that will be shown in the **Part View**.

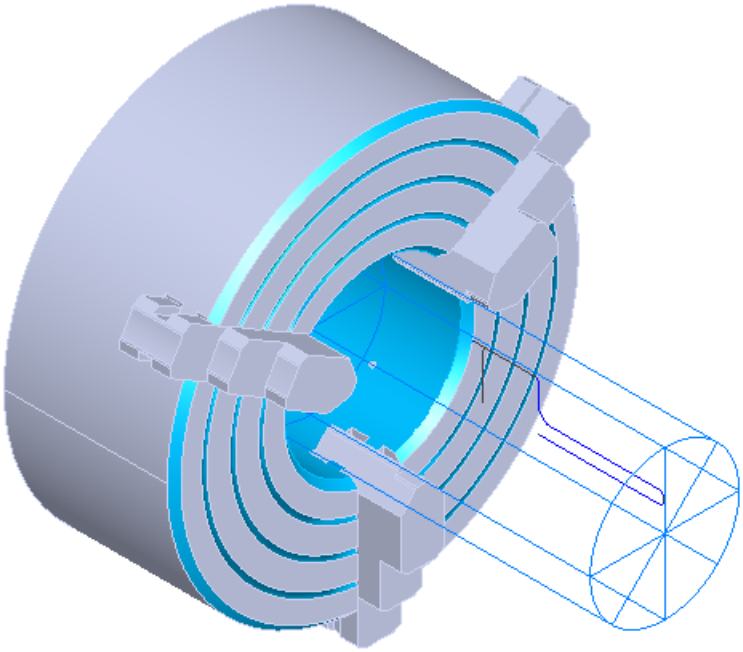
24 There is no need to create the **ID**, **OD** and **Face** grooves in the geometry as we will use **Feature from dimensions** from the **New Feature** menu.



To have a more inclusive visual experience we are now going to import a solid chuck that gives a more realistic view when turning our part.

1 Please Select **Import** from the File menu. The file required is called **Chuck.x_t** this file is located in **C:\ Training_Data\FeatureCAM Course data 2017\Turning files to import.**

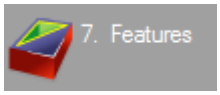
© Copyright 2016 Autodesk, Inc. All rights reserved



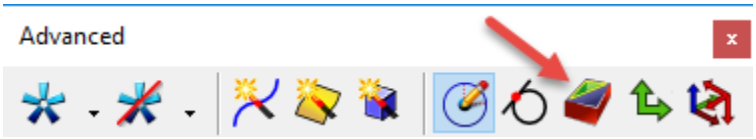
2 Please **cancel** the **import wizard**.

3 In **PartView** please select **all solids** by using **group select** and right click on the solids, from the menu select **Use Solids as Clamps**. This will then allow you to visualise the chuck when you run the **3D Simulation**.

Creating a Face Feature

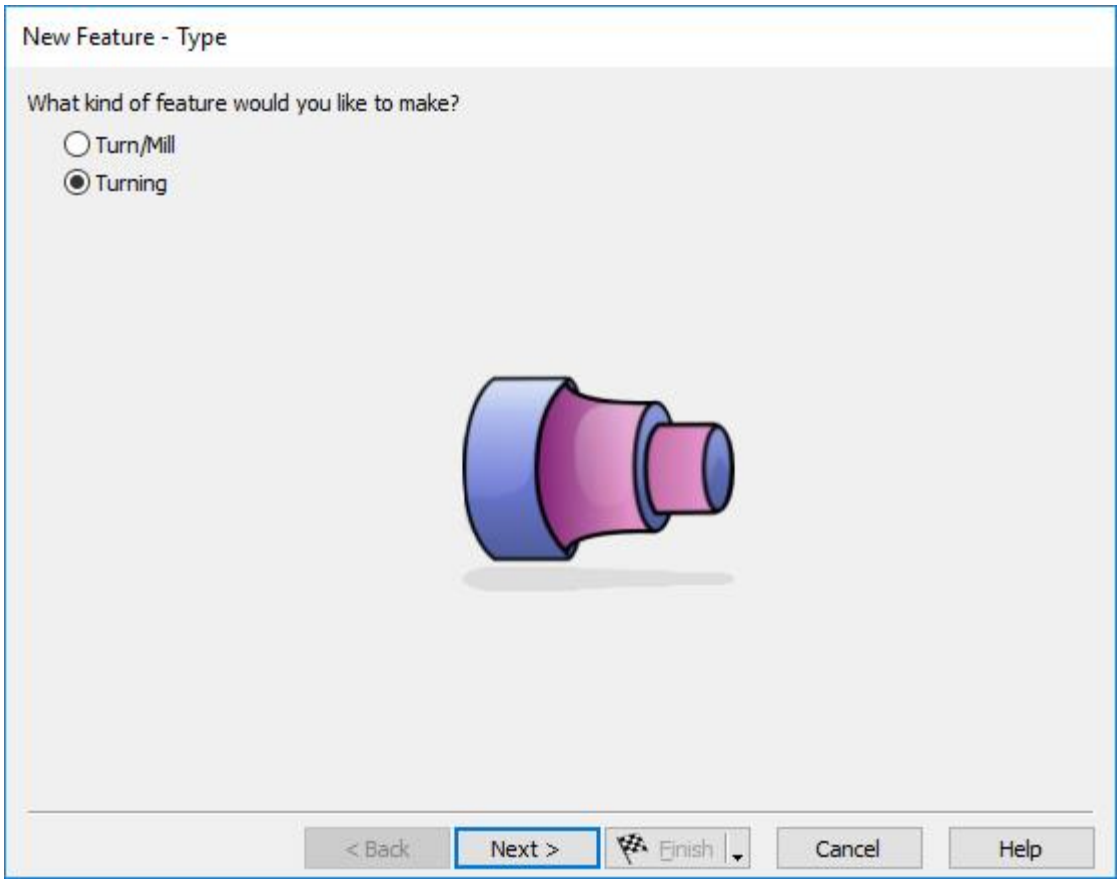
- Select **Features** 

from the **Steps** menu.
- Select **Ctrl + R** New Feature**.**
- Select **Construct>New Feature.**
- Select **New Feature** from the **Advanced Toolbar.**



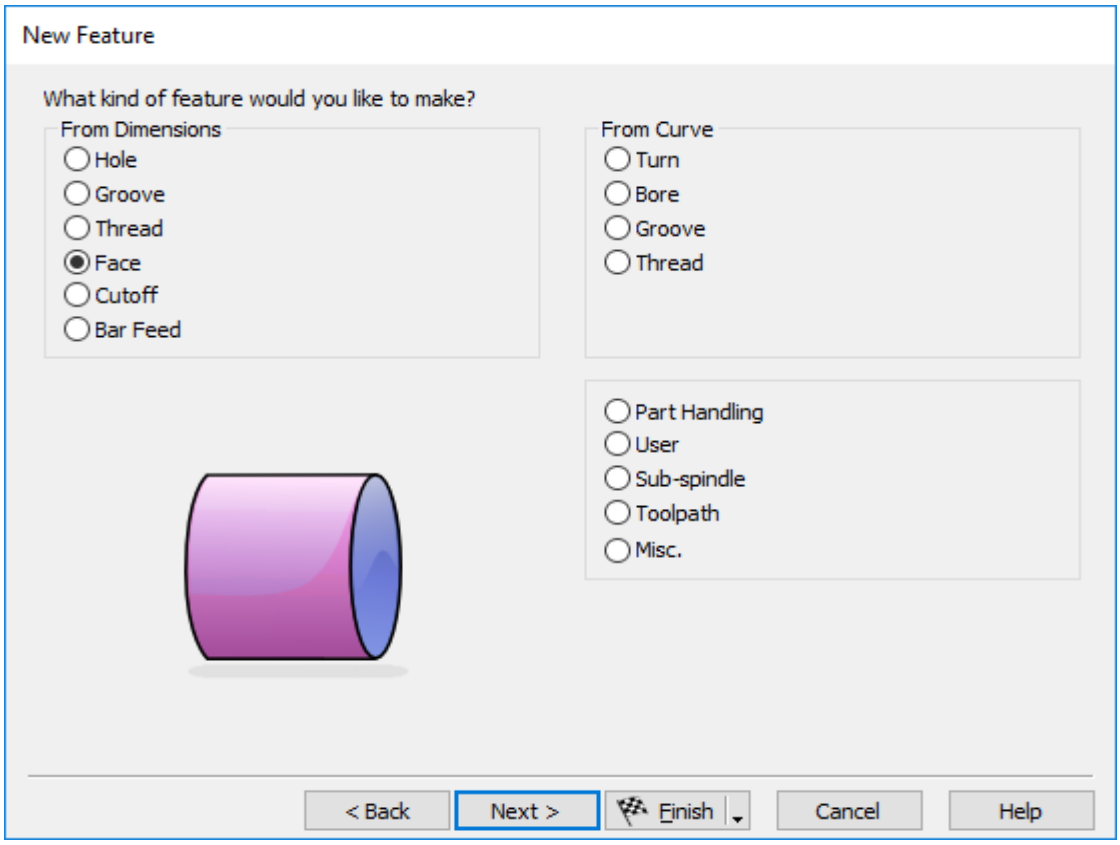
4 Select **Turning.**

© Copyright 2016 Autodesk, Inc. All rights reserved



5 Select **Next.**

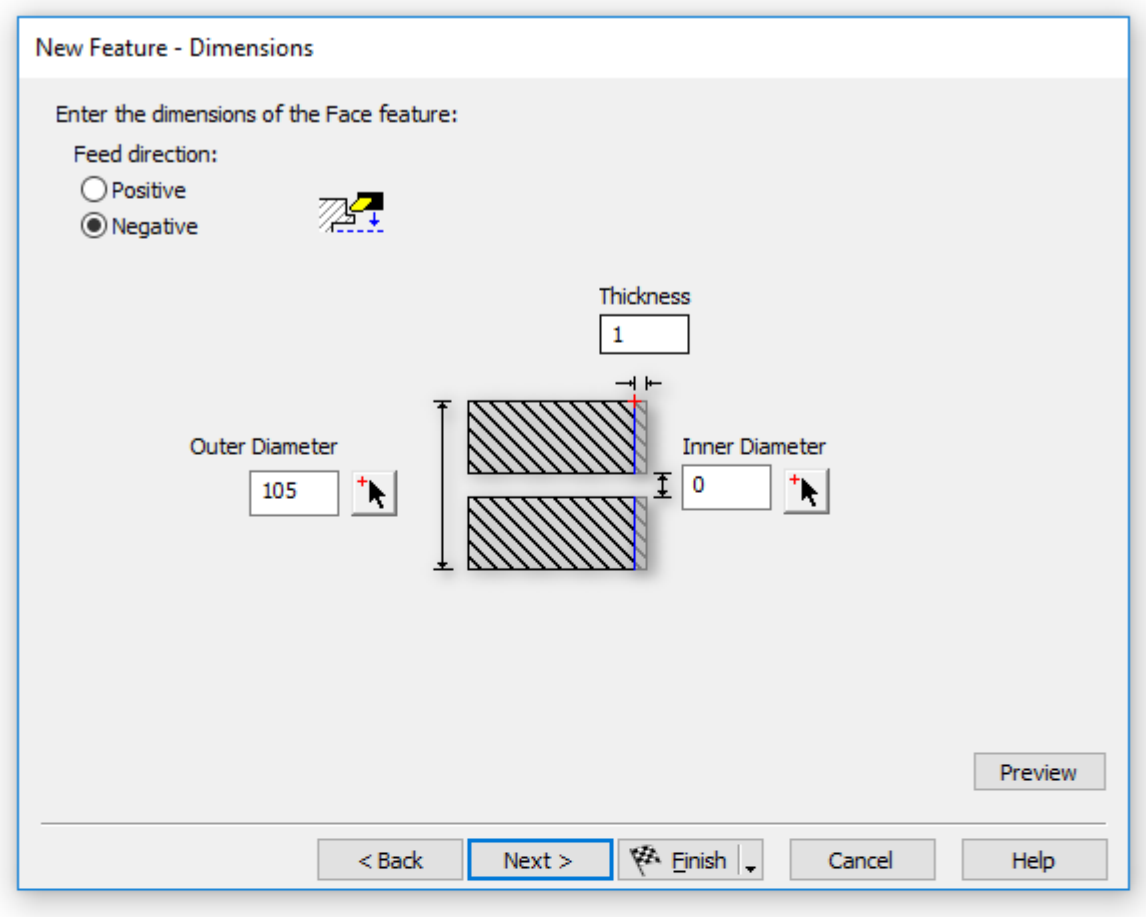
6 **FeatureCAM** will show the following menu. Select **Face** from **Dimensions**.

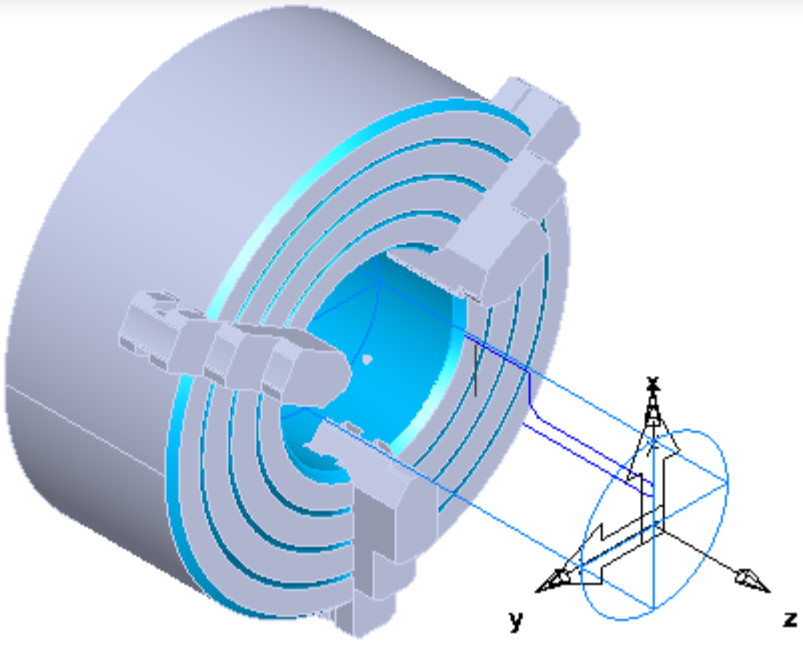


© Copyright 2016 Autodesk, Inc. All rights reserved

7 Select **Next**.

8 The following menu will appear.



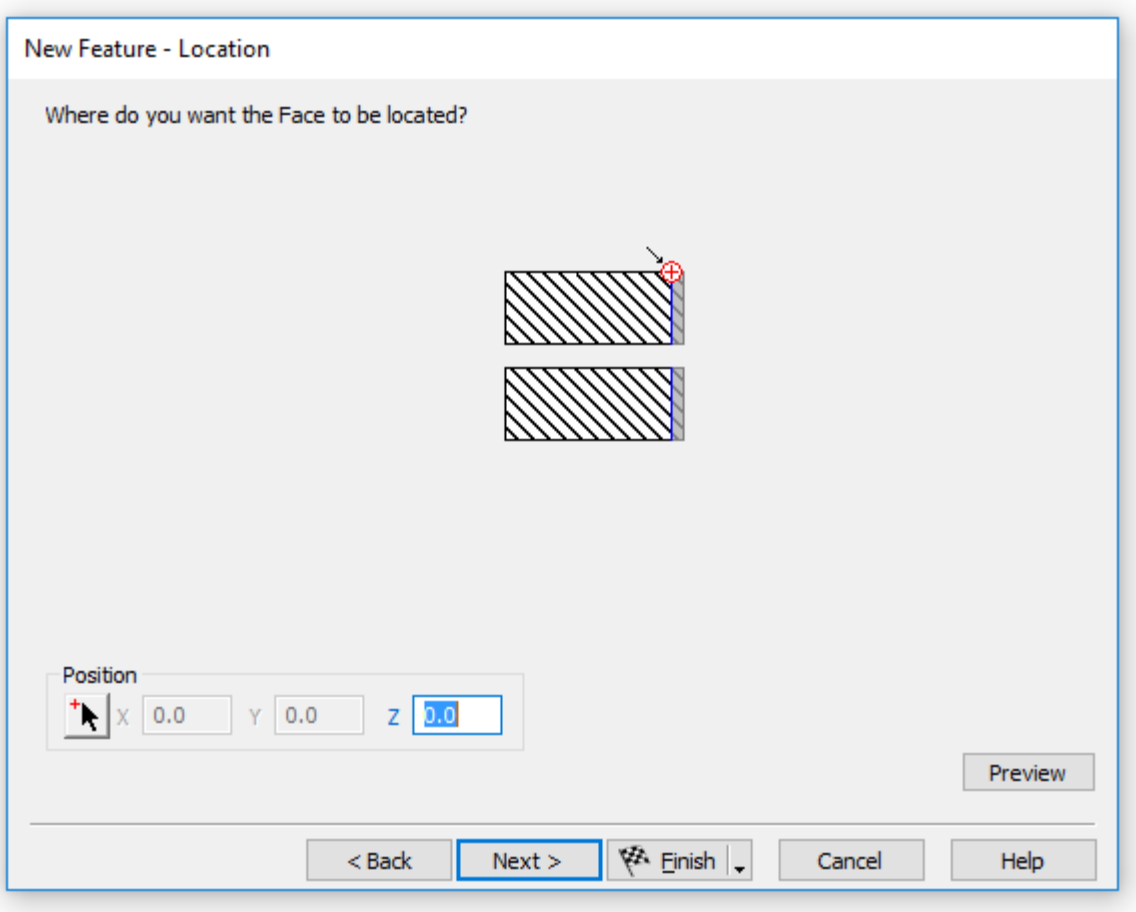


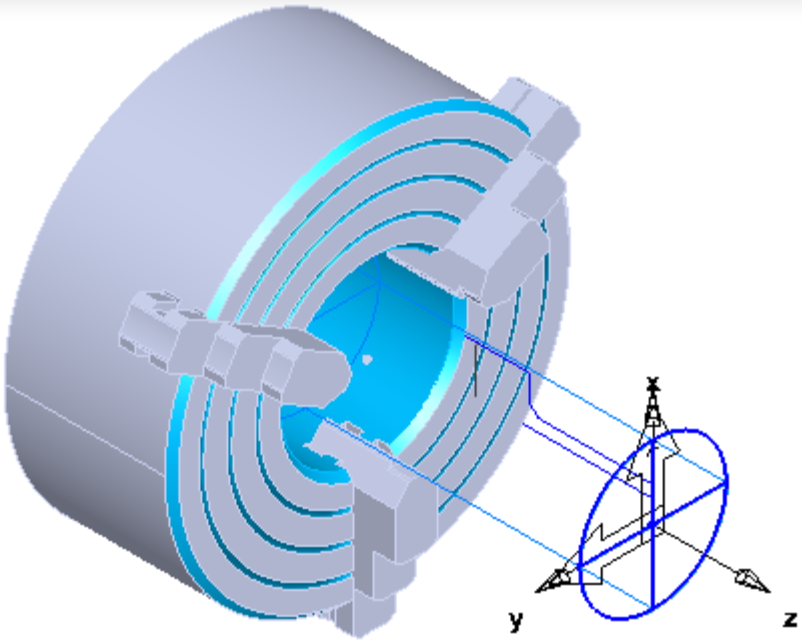
9 **Negative** is **towards** the **chuck** and **Positive** is away from the **chuck**.

10 **Thickness** is the amount of material that **FeatureCAM** will machine away.

© Copyright 2016 Autodesk, Inc. All rights reserved

11 Select **Next.** The following menu will appear.



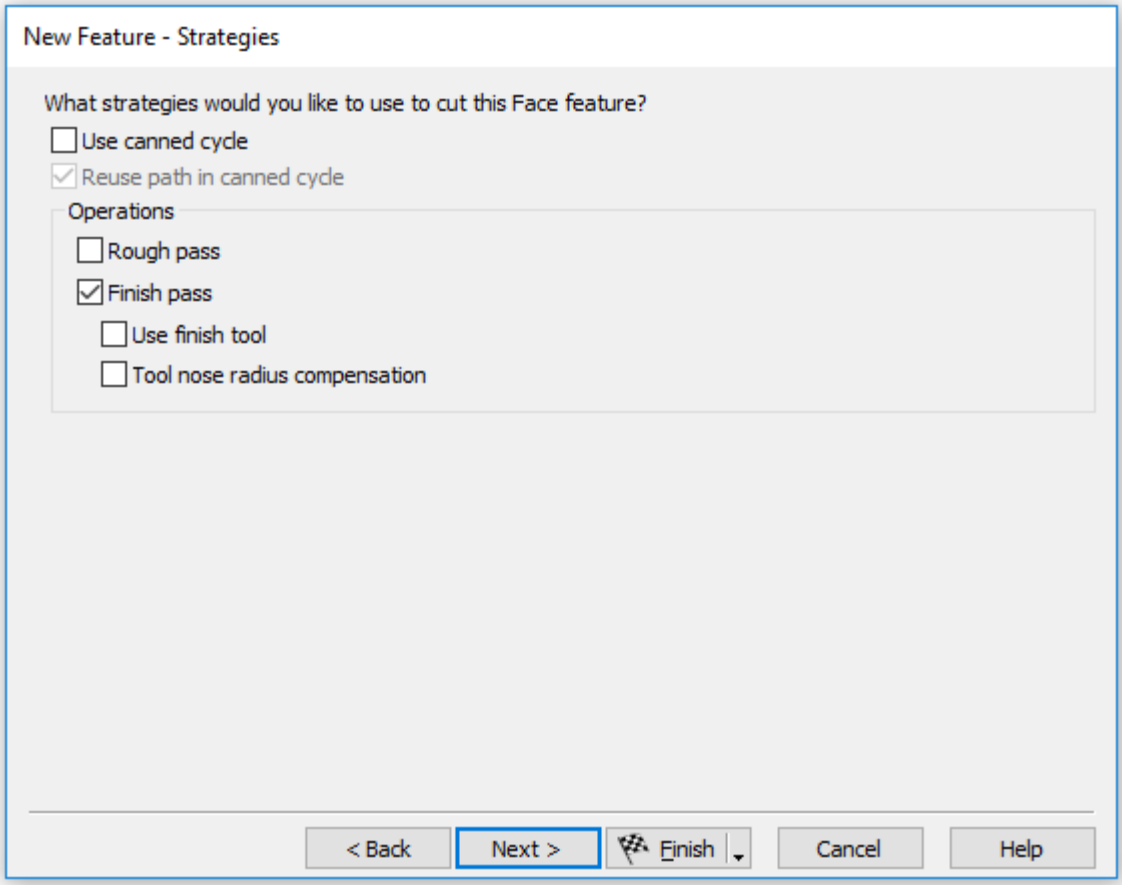


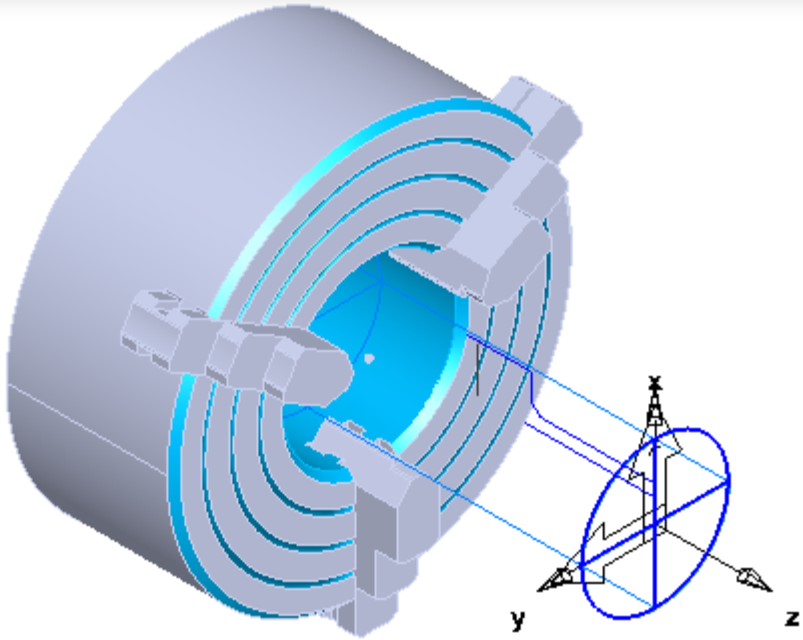
12 Where do you want the **Face** to be located? Leave this at **zero**.

13 Select **Next**. The menu is shown on the next page.

14 Rough Pass is an option if you have a lot of material to turn away.

© Copyright 2016 Autodesk, Inc. All rights reserved





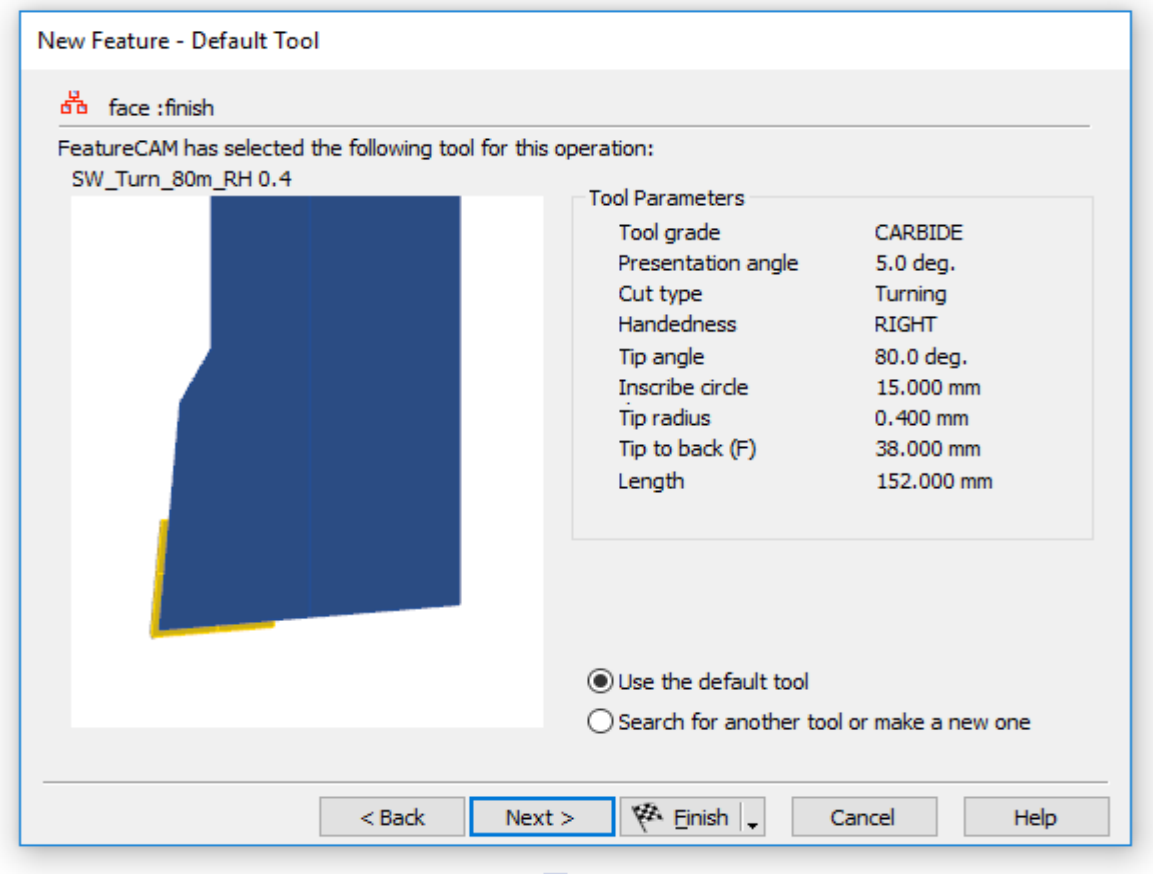
15 Notice that we have the ability to use **Canned Cycles**. The canned cycle will be output through the post. (If supported) This gives greater flexibility to the operator. He can then change this on the machine instead of having to go to the **FeatureCAM** programmer (Production Engineer) and re-output the code. A start point is required. This is explained later on in the chapter.

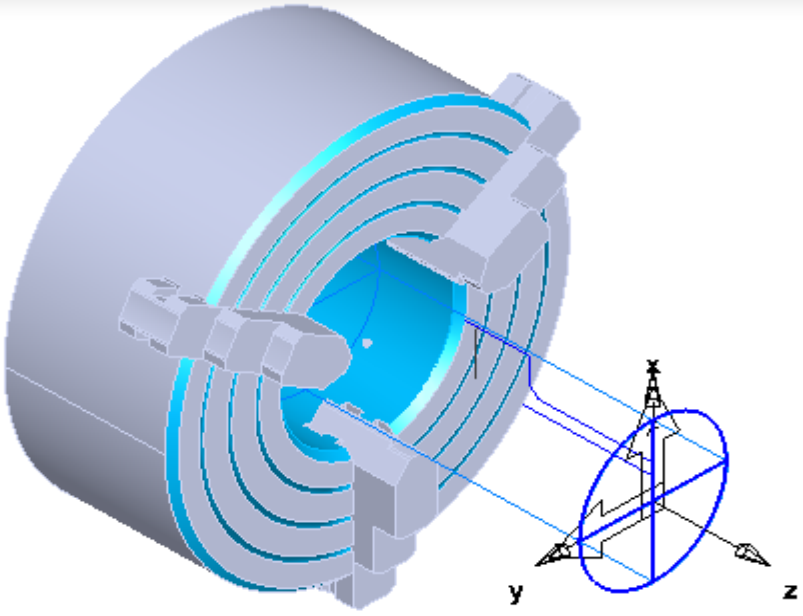
© Copyright 2016 Autodesk, Inc. All rights reserved

*If you have excess stock on the front face, select the **Rough pass** option in the menu. If you do so, you will have the ability to have separate roughing and finishing tools that can be separated in the operations list.*

16 Select **Next** twice.

17 **New Feature – Default Tool**. You can select the default tool or search the tool database for a more suitable tool. See menu below.

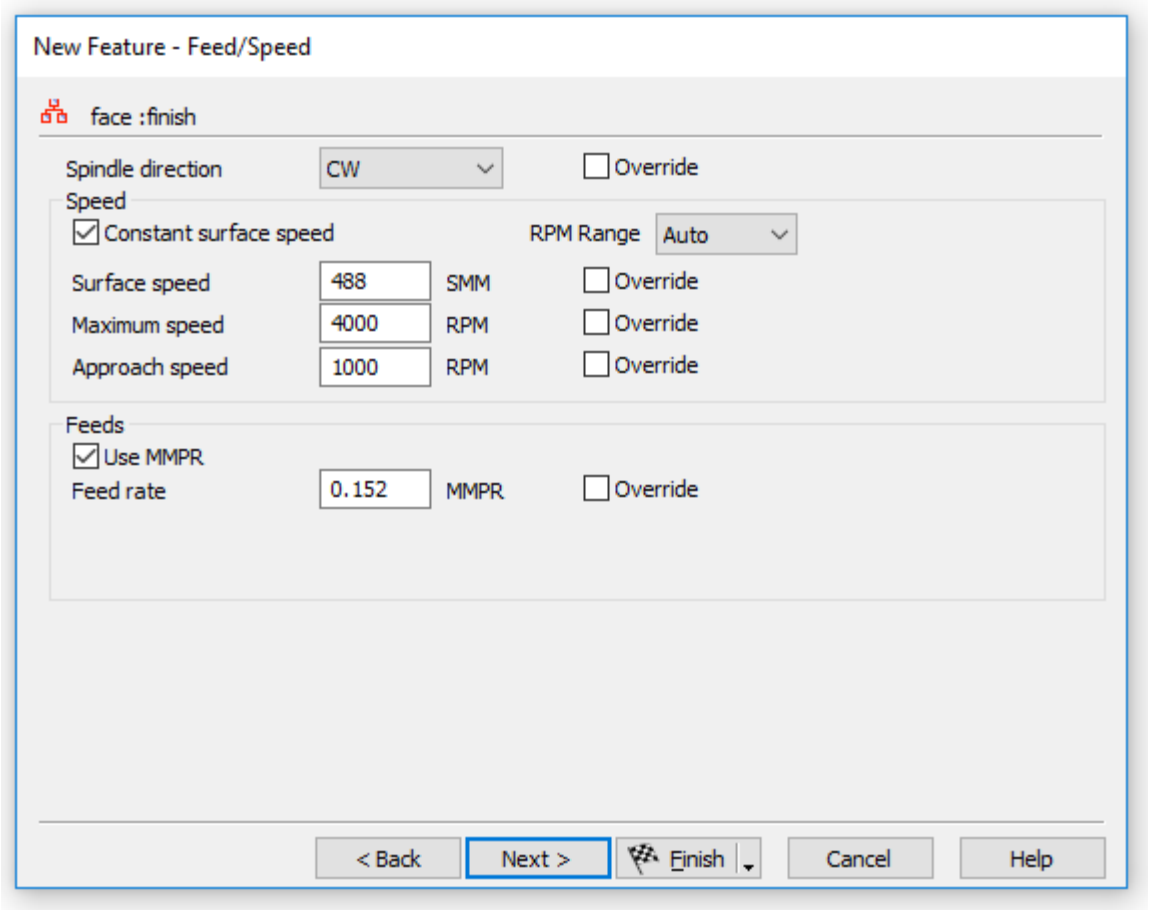


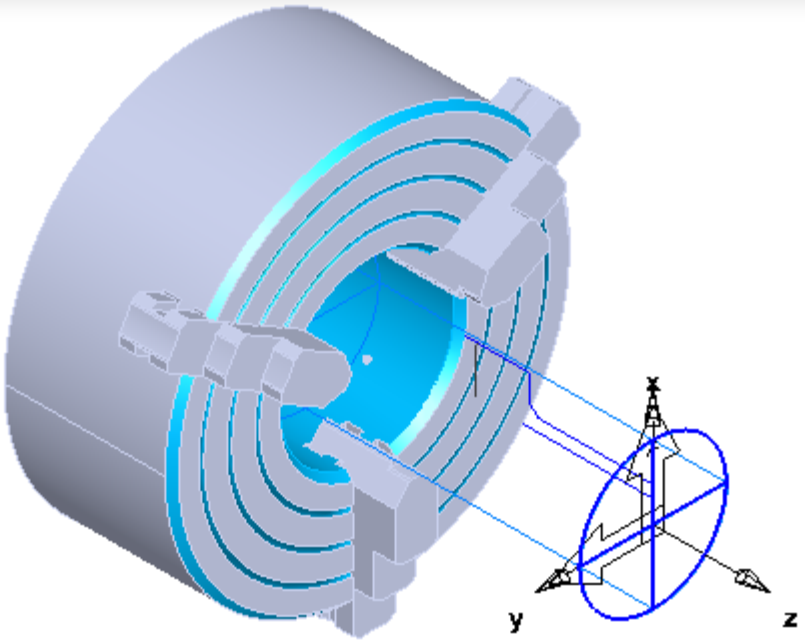


© Copyright 2016 Autodesk, Inc. All rights reserved

18 Select **Next** twice.

19 **New Feature – Feeds/Speeds.** This menu gives the ability to change the feeds and speeds etc.

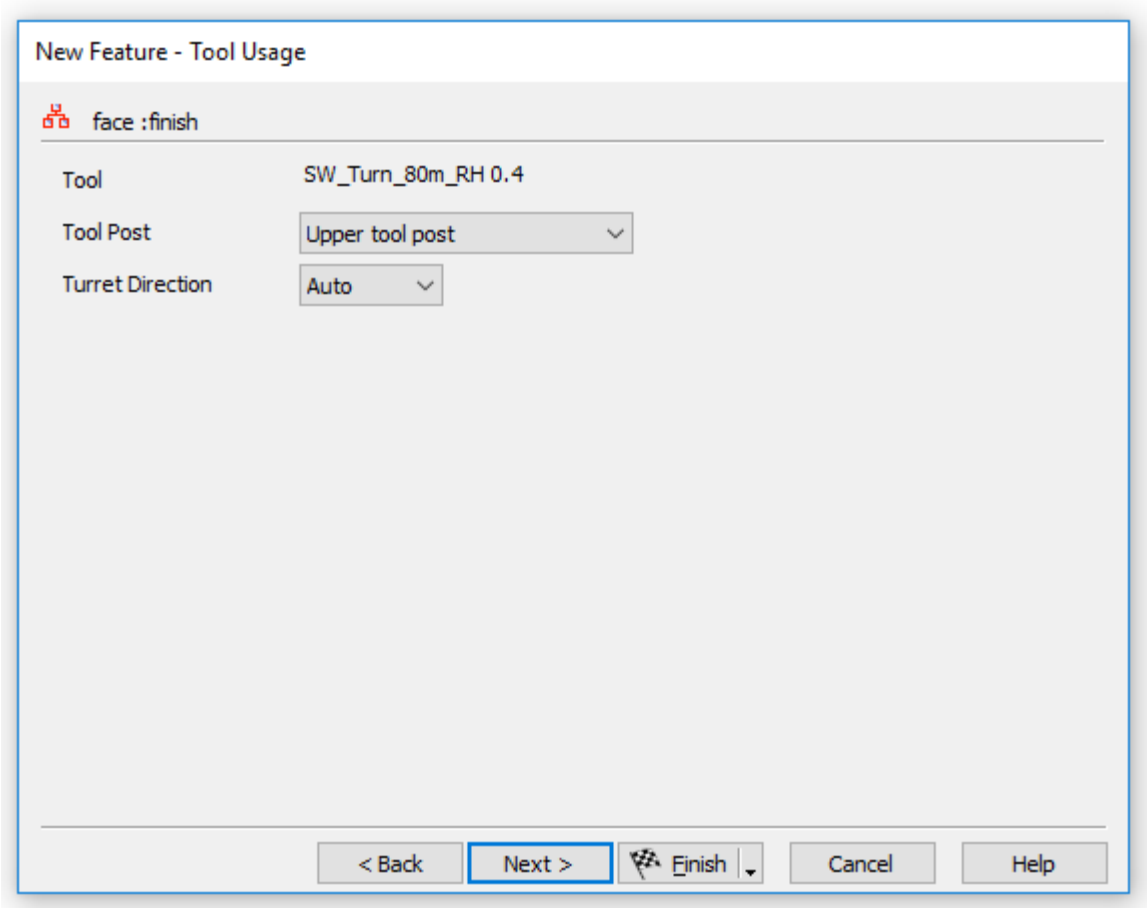


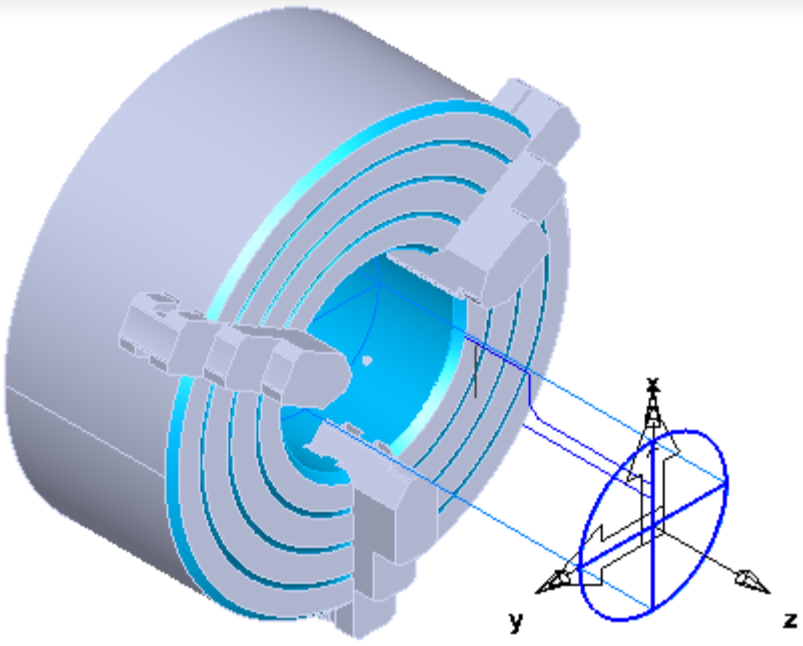


20 Select **Next**.

© Copyright 2016 Autodesk, Inc. All rights reserved

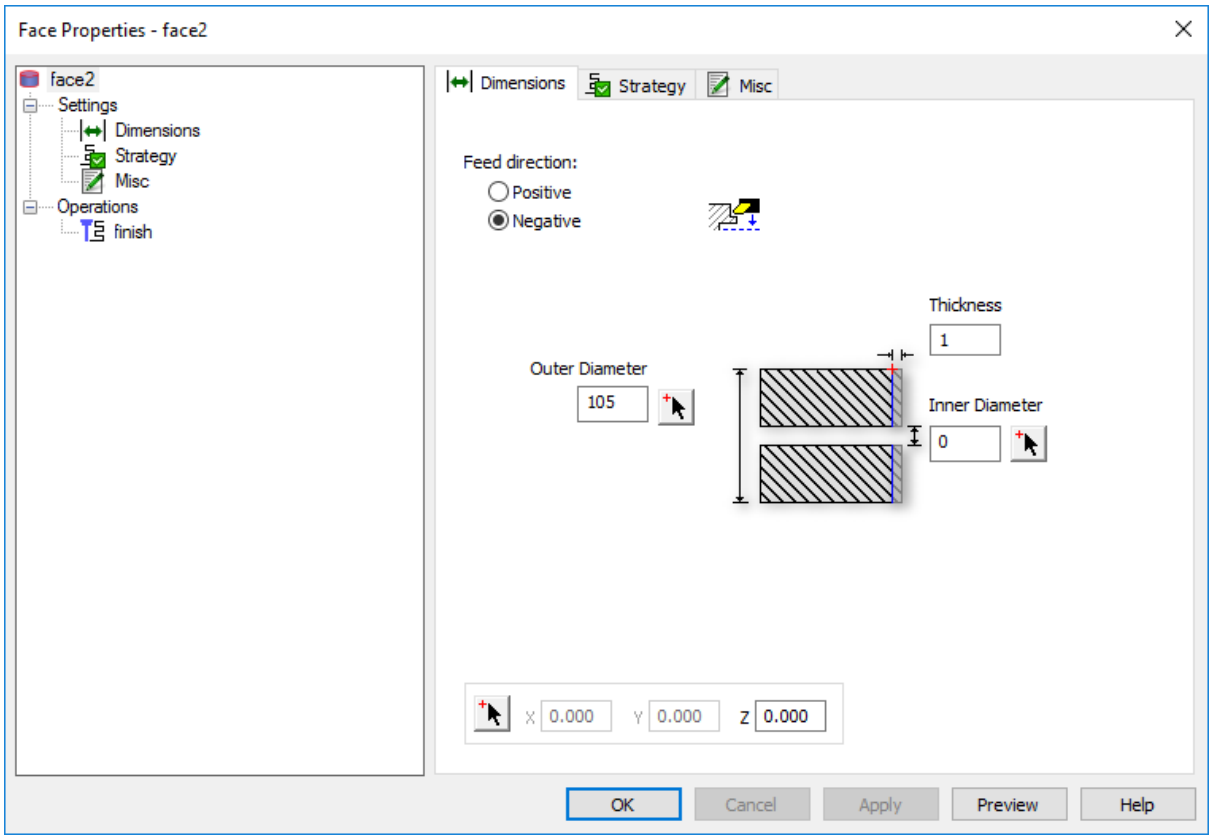
21 **New Feature – Tool Usage.** This menu allows you to select what coolant type also Turret type etc. Select **Next** for a summary of all the options.



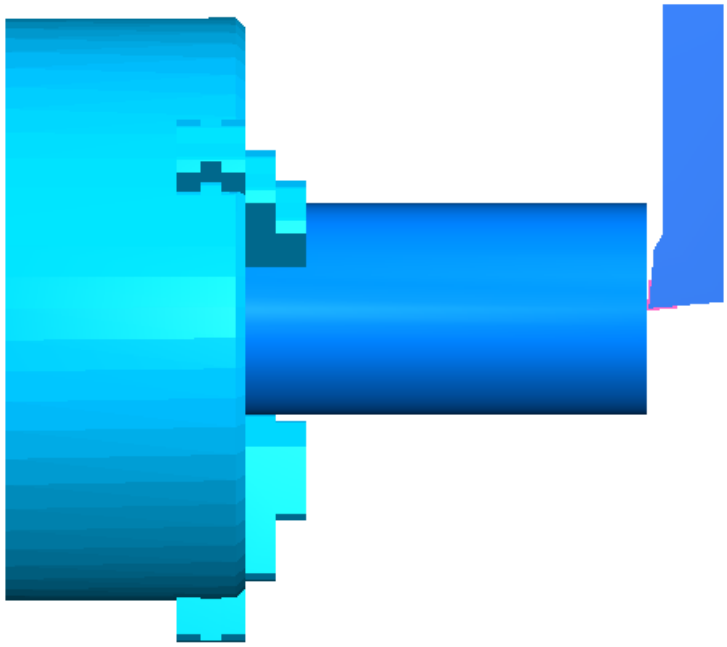


22 Select **Finish** to accept the default values for the rest of the dialog boxes. **Face** will be the first operation.

© Copyright 2016 Autodesk, Inc. All rights reserved



23 Select the **3D Simulation** Icon and then the **Play** button.



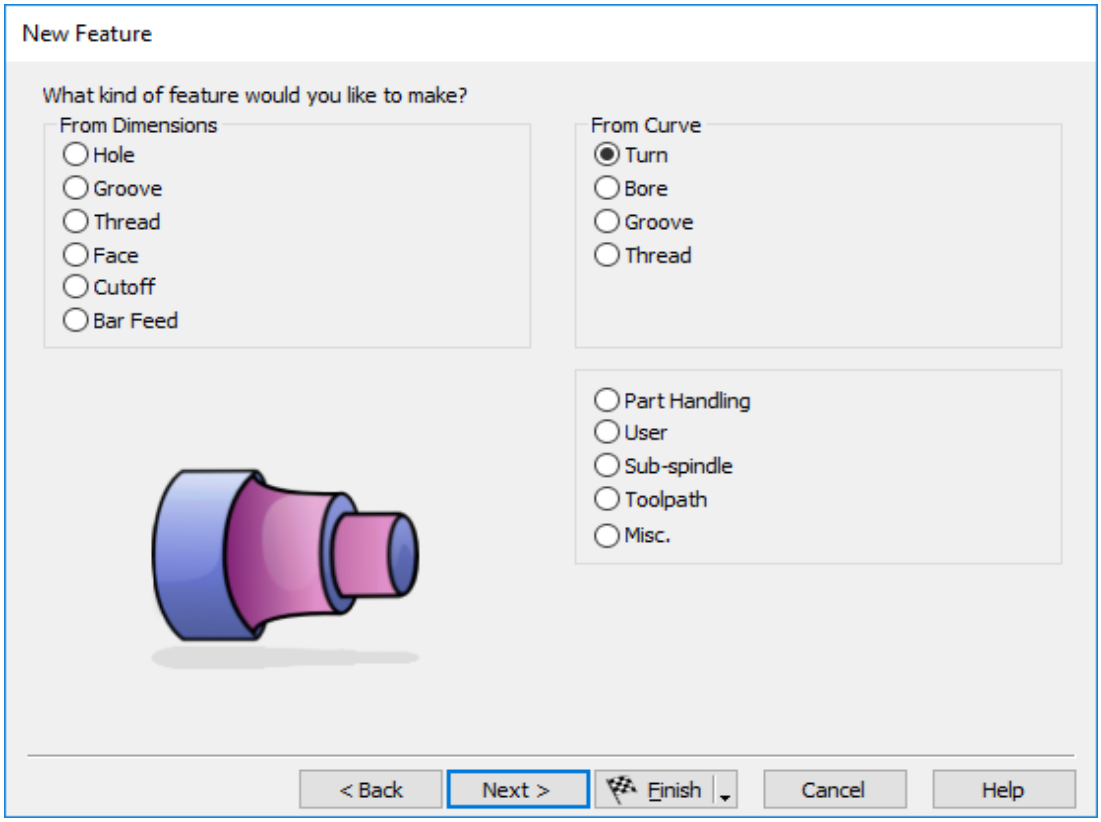
24 Select the **Eject** button to return to the drawing view.

© Copyright 2016 Autodesk, Inc. All rights reserved

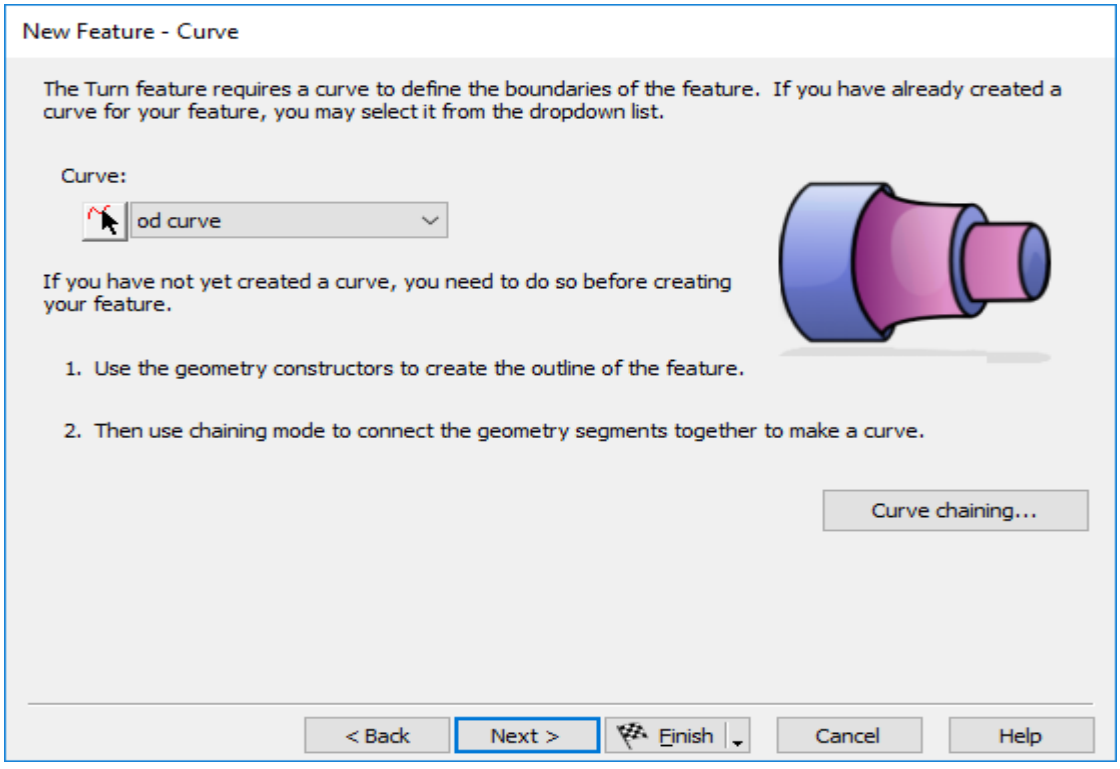
Creating a Turn Feature

25 Select **Ctrl + R** or select **Steps** and **Features** to access the **New Feature** menu.

26 Select **Turning** Select **Next,** Select **Turn.**

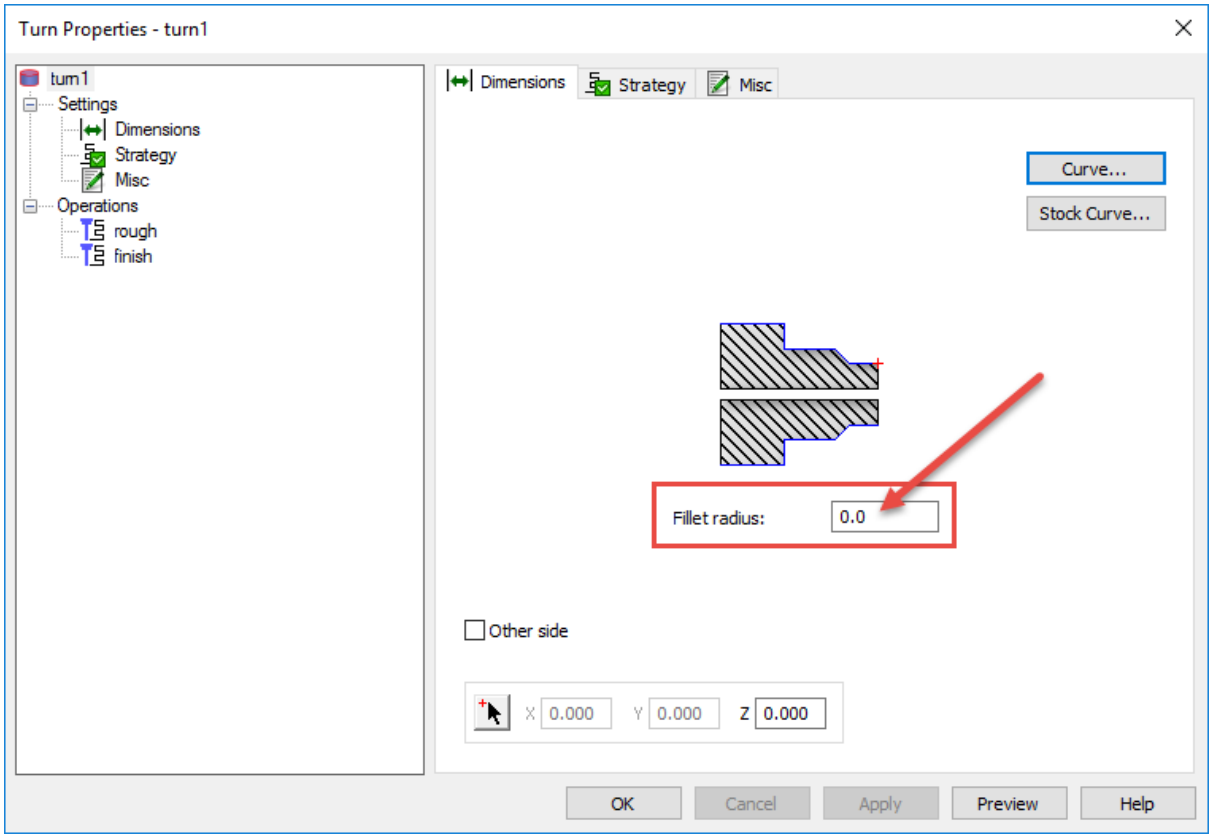


27 Select **Next** and the following form will appear.



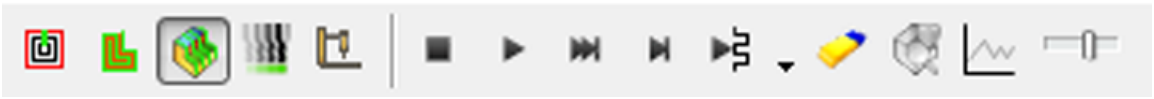
© Copyright 2016 Autodesk, Inc. All rights reserved

28 You can now pick the curve graphically. Click on the curve you have named **"od profile"** earlier. Since this pick could be either the curve or the line you may see the Select dialog box. This box helps to clarify a pick. In this case, select **"od profile"** from the list and click **OK** and then select **Next** in the **New Feature – Curve** dialog box. Accept the default strategy settings by selecting **Finish**.

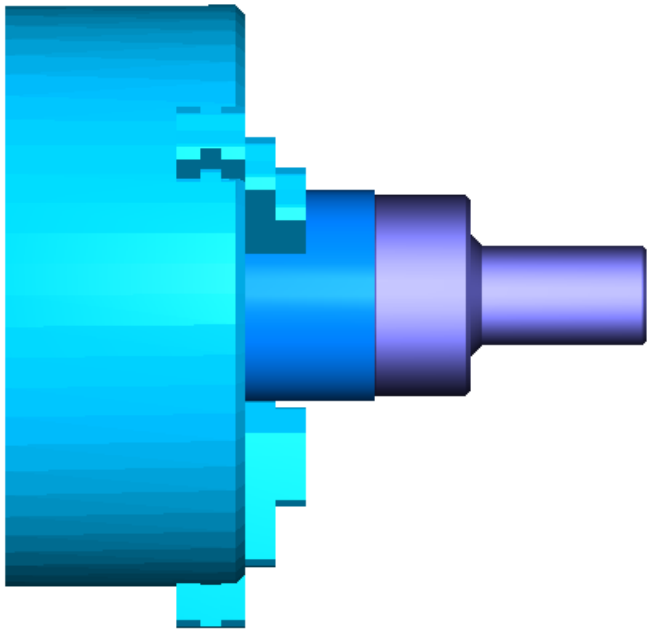


29 We can now automatically Fillet external or internal sharp corners. For internal bore fillet corners use **(Other side)**

30 Run the **3D simulation** as shown.



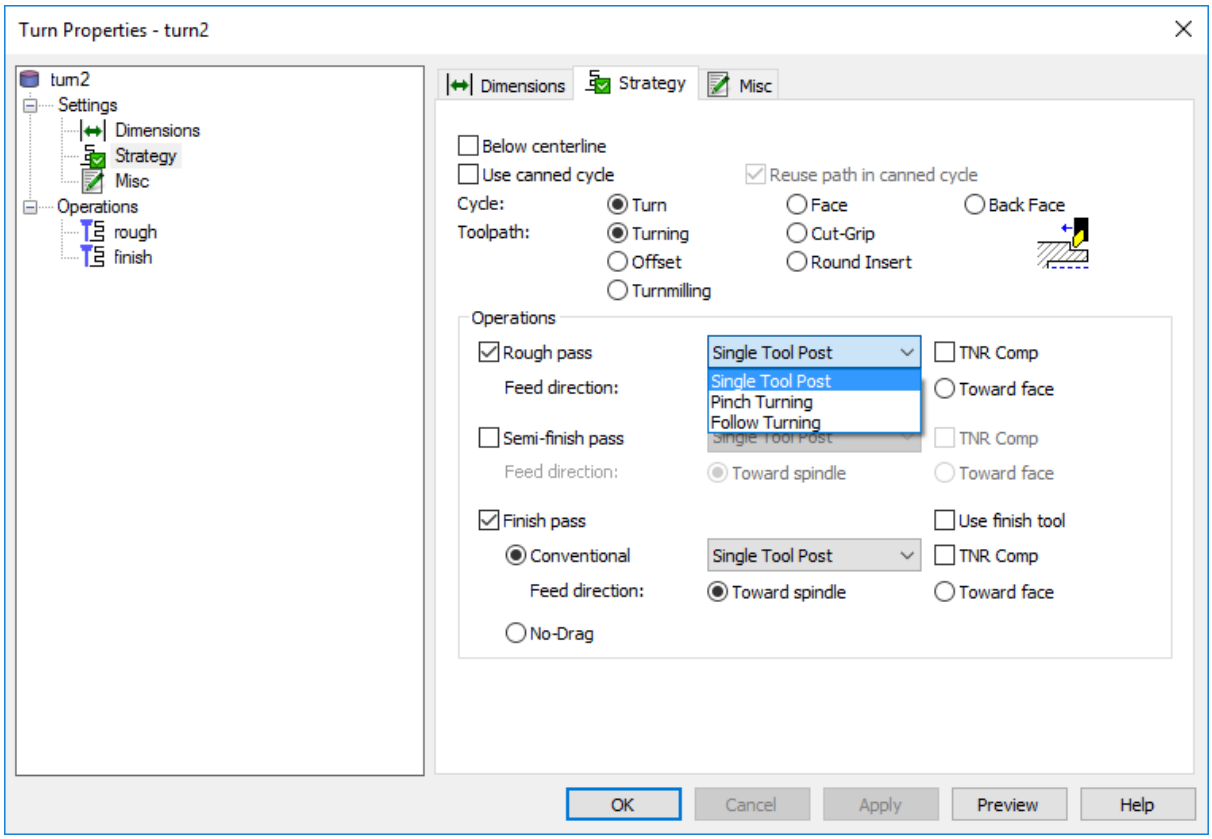
© Copyright 2016 Autodesk, Inc. All rights reserved



31 Select the **Eject** button to return to the drawing view.

Multi –Turret options.

Customers that have multi - turret machines can use the options of **Pinch turning** and **Follow turning**. These option are quick and easy to select.



© Copyright 2016 Autodesk, Inc. All rights reserved

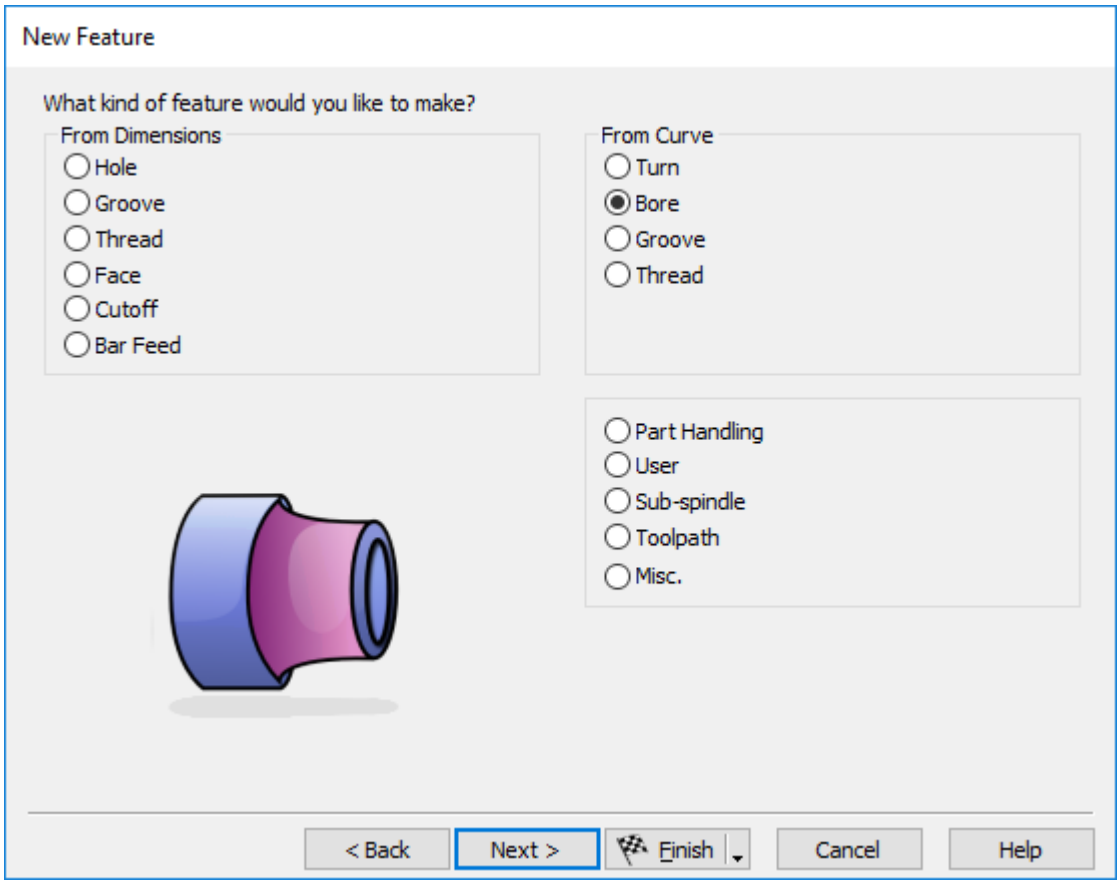
Creating a Bore Feature

32 Select **New Feature** or **Ctrl+R.**

33 Select a **Bore** feature **from curve**, use the curve you named "**idcurve**".

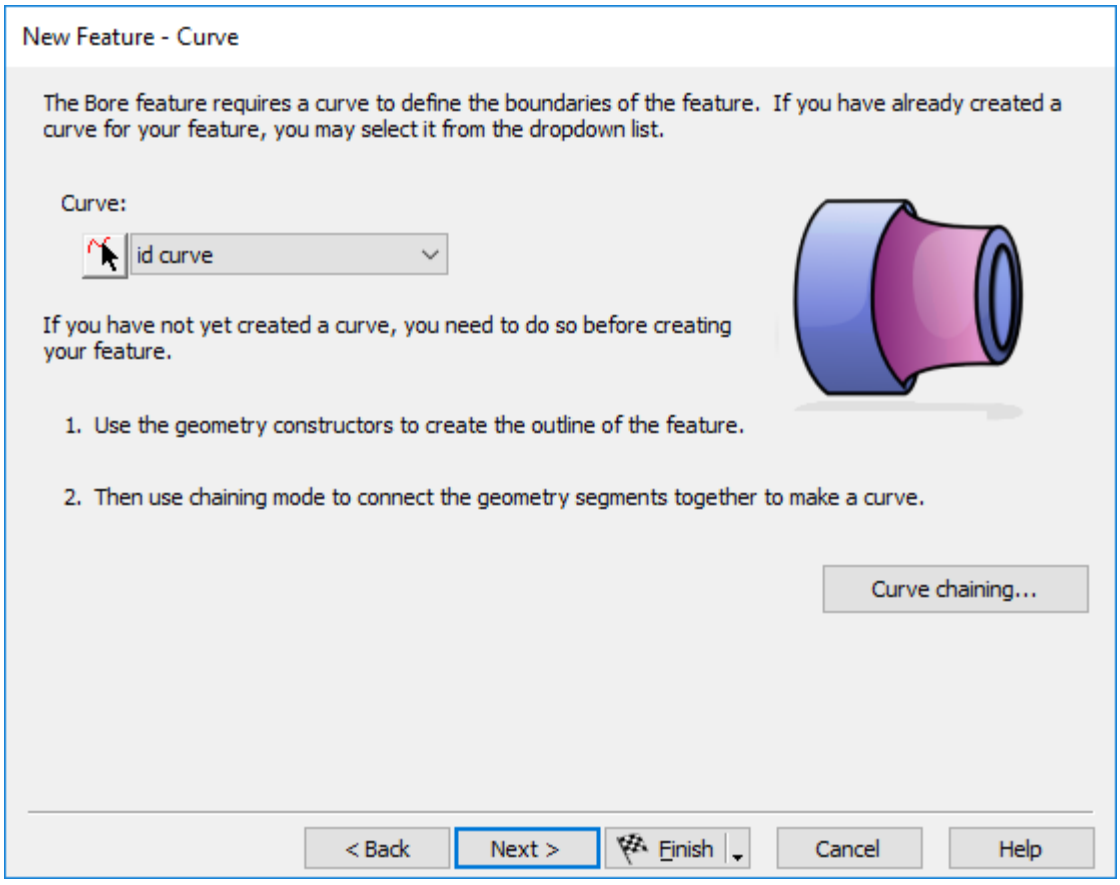
With this option we have the ability to specify a pre-drilled hole.

This then created a drill cycle and inserts this before the boring bar.

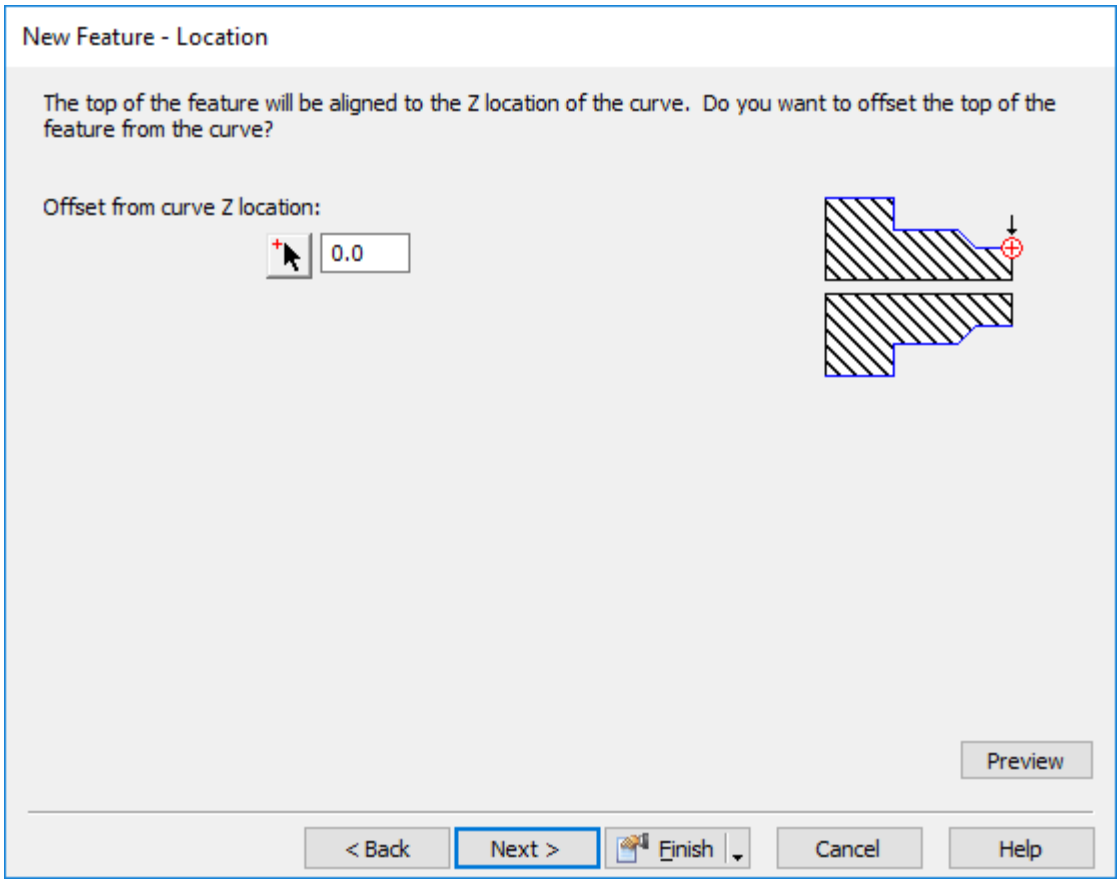


34 Select **Next.**

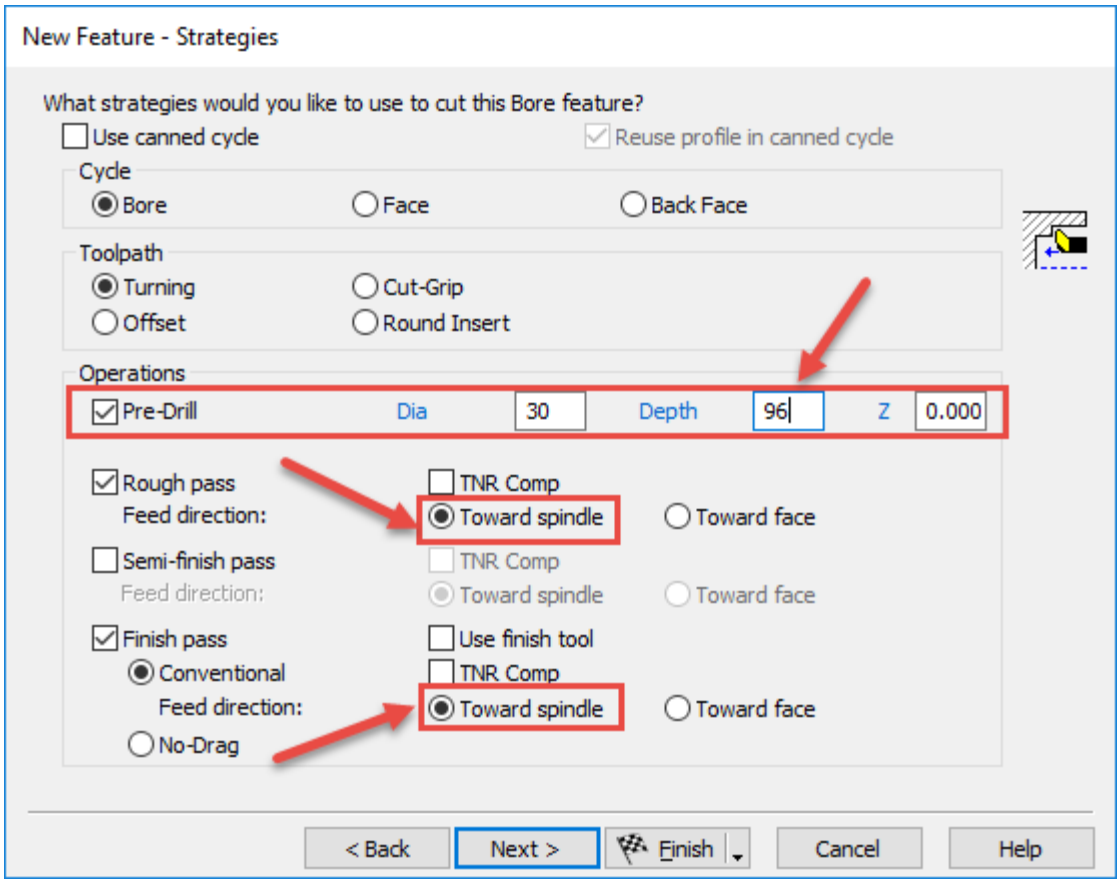
© Copyright 2016 Autodesk, Inc. All rights reserved



35 Select **Next** twice.

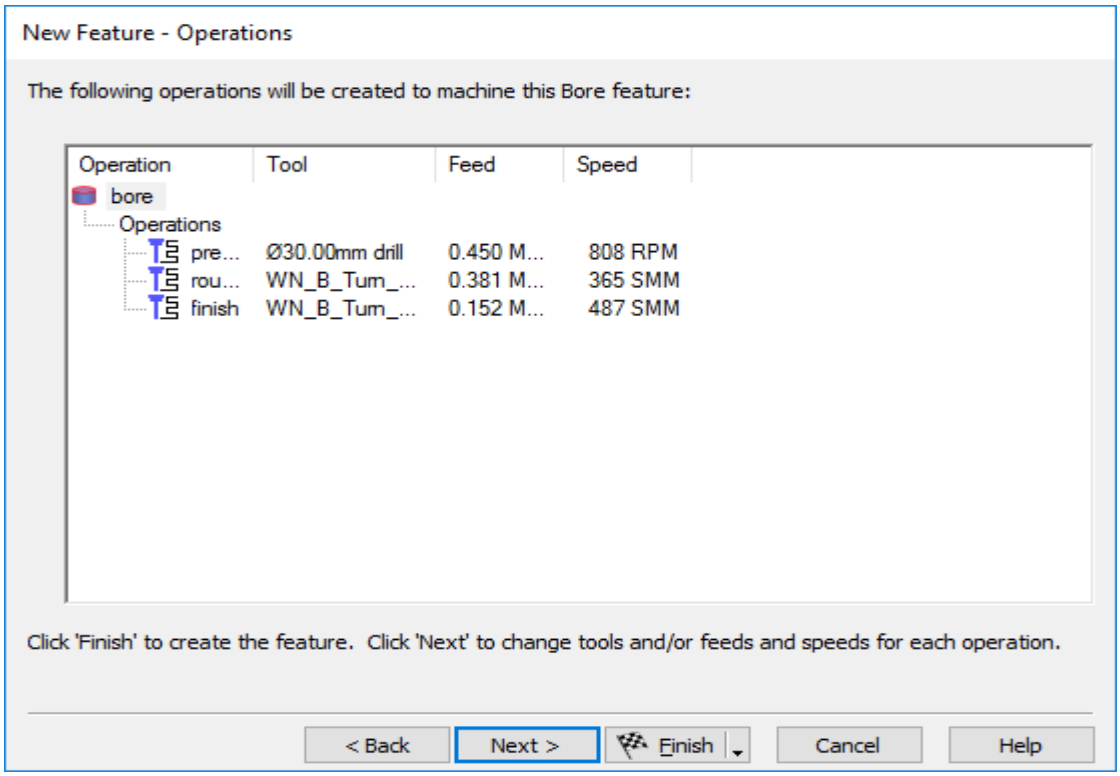


© Copyright 2016 Autodesk, Inc. All rights reserved



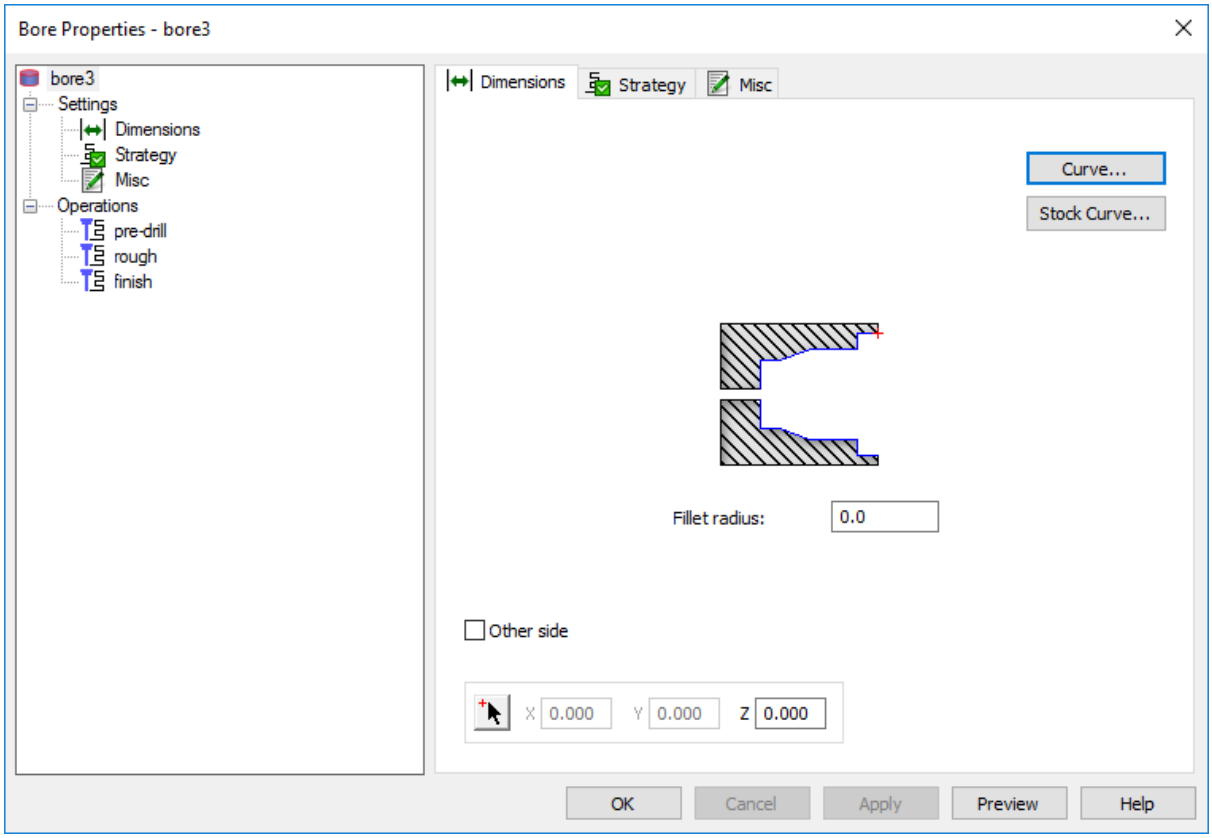
Towards spindle machining is normal convention when turning towards the Chuck. The pre-drill diameter is 30mm which leaves 1mm each side to finish of the bore using a boring bar.

36 Select **Next.**



© Copyright 2016 Autodesk, Inc. All rights reserved

37 Select **Finish**.



38 Before we close this menu we can look at the bore operation in **Single step mode**. Select **Preview** as shown in the above menu.

39 Have a look at the Simulation Menu. We have the following options.

Preview menu

Play selected.

Fast forward to End Selected.

Single Step selected.

Play to Next operation selected.

40 Select single step mode and see the boring bar start from the middle.

41 This will potentially crash the boring bar into the part. (Depending on the size of the boring bar).

42 Double click on **Bore1** in **PartView**.

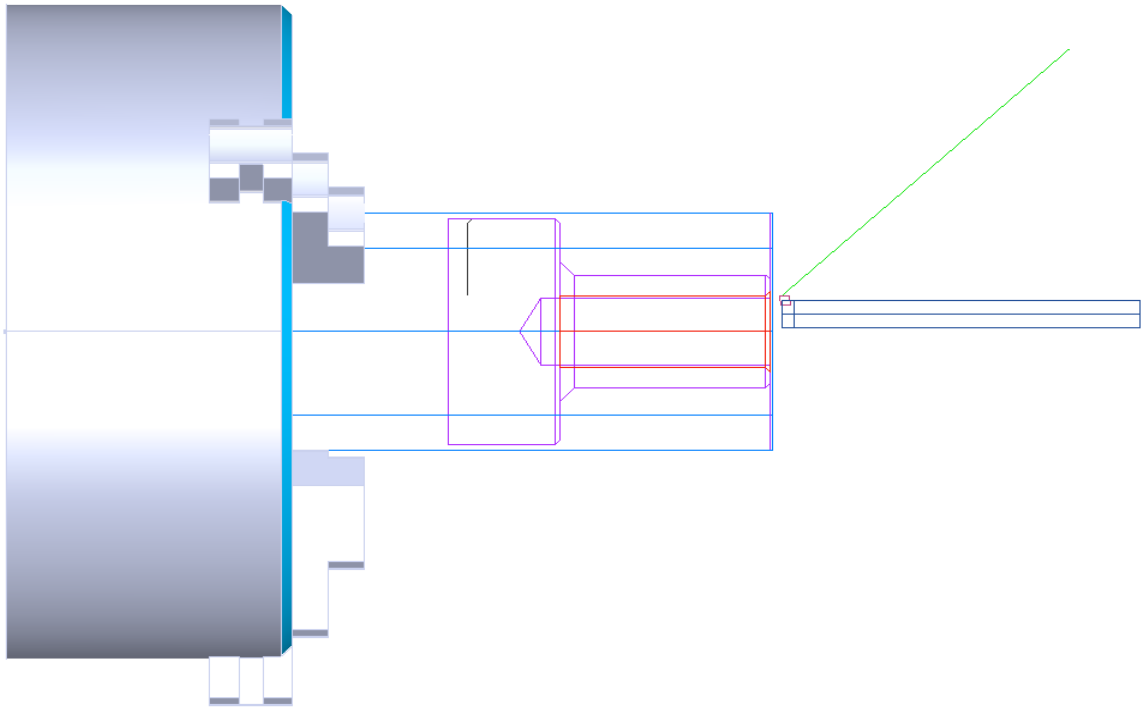
43 Select **Rough**. Select **Boring**.

44 Select **Min radius boundary**. Enter **15**

45 **Preview** the changes you have just made.

46 Select **Esc** to close the **Preview** Simulation.

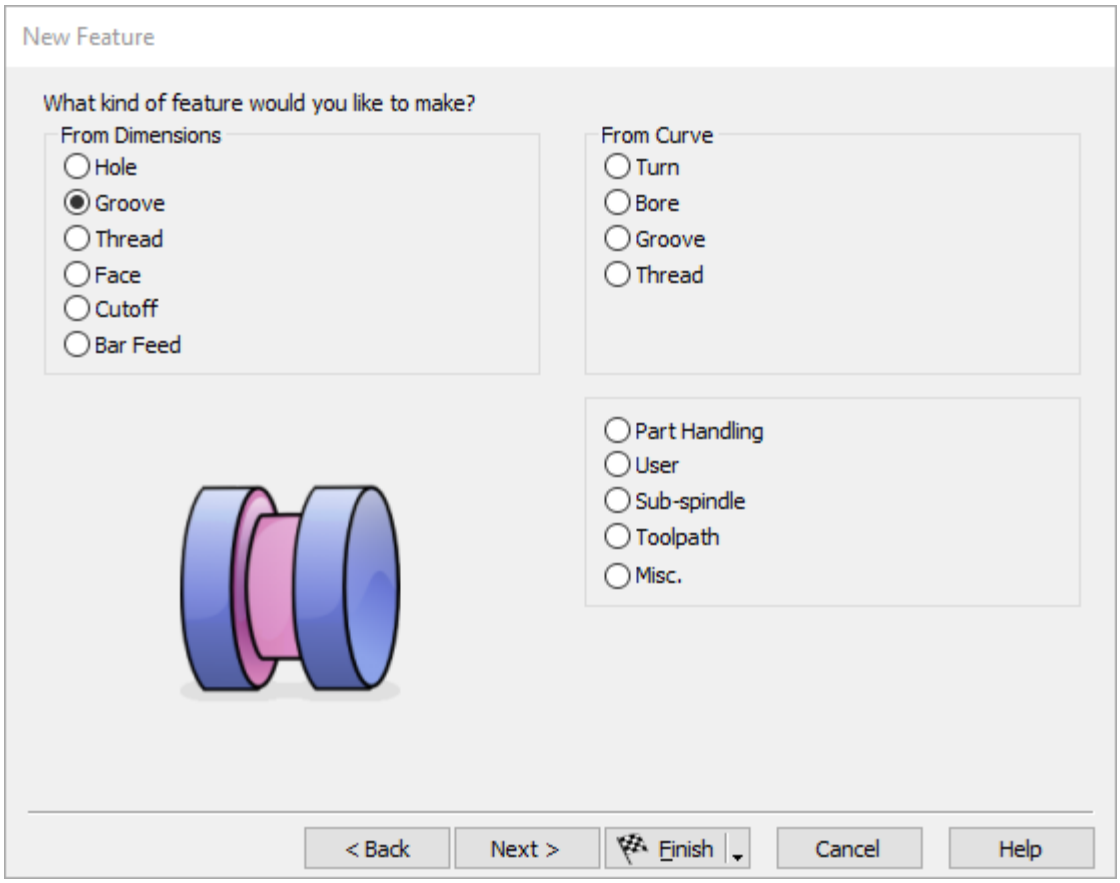
© Copyright 2016 Autodesk, Inc. All rights reserved



Creating a Groove Feature

47 Select New Feature or **Ctrl+R.**

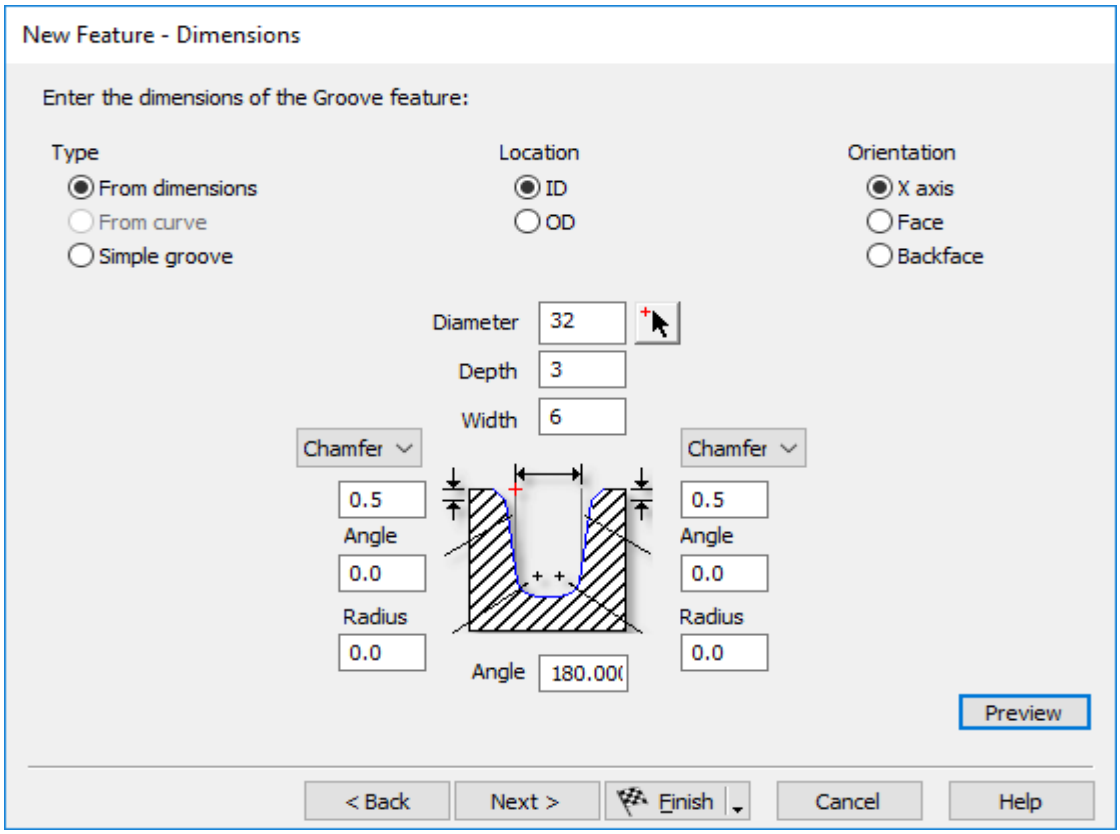
48 Select the **Groove** radio button in the **From Dimensions** group.



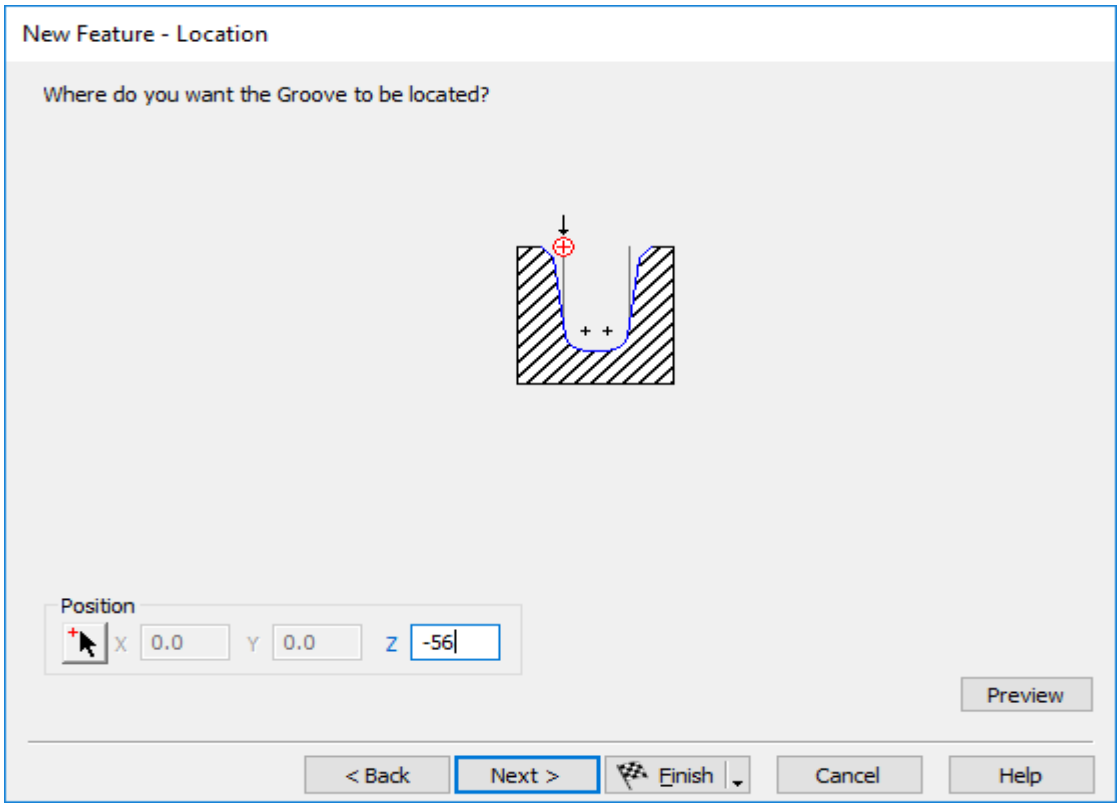
© Copyright 2016 Autodesk, Inc. All rights reserved

49 Select **Next.**

50 On the Dimensions page set Location to **ID**, Orientation to **X-axis**, and Diameter to **32mm**, and Depth to **3mm**, and Width to **6mm**. Chamfer settings at **0.5mm**.

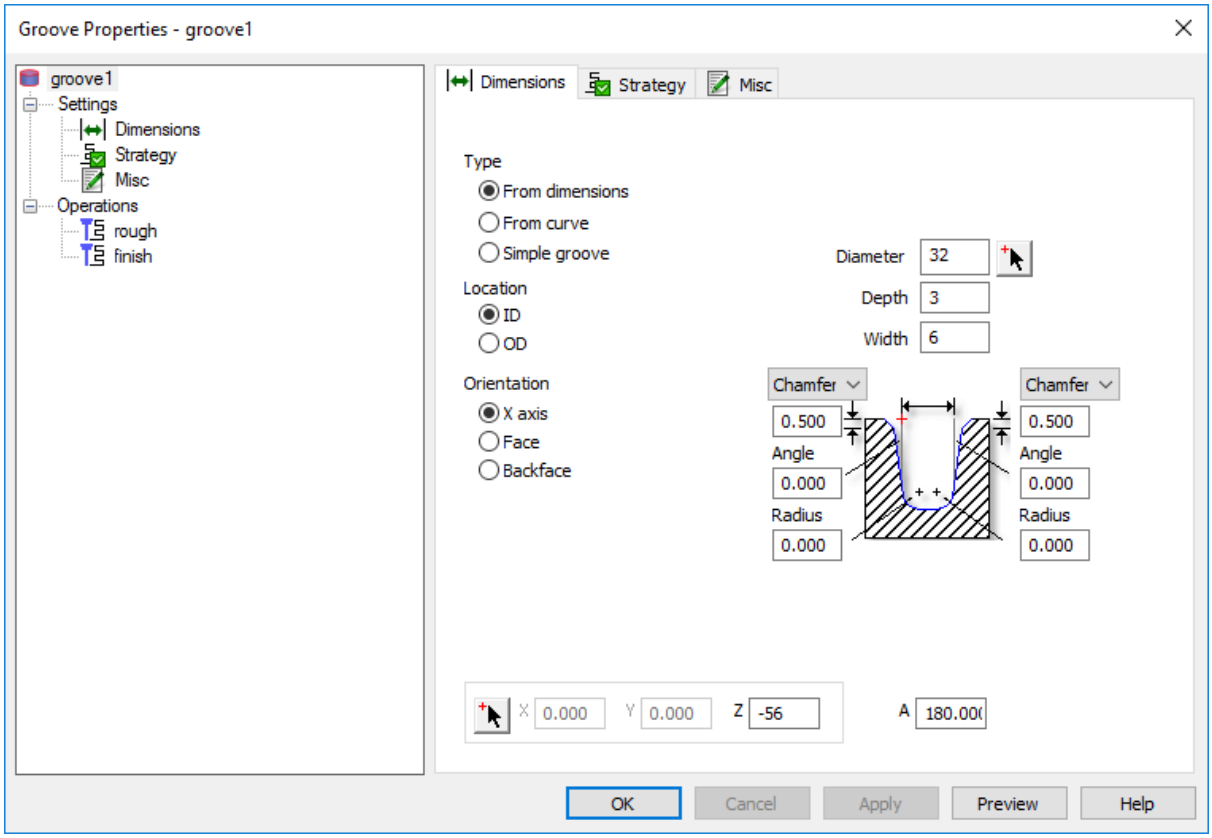


51 Click **Next** and set **Z-56mm**. Click **Finish.**

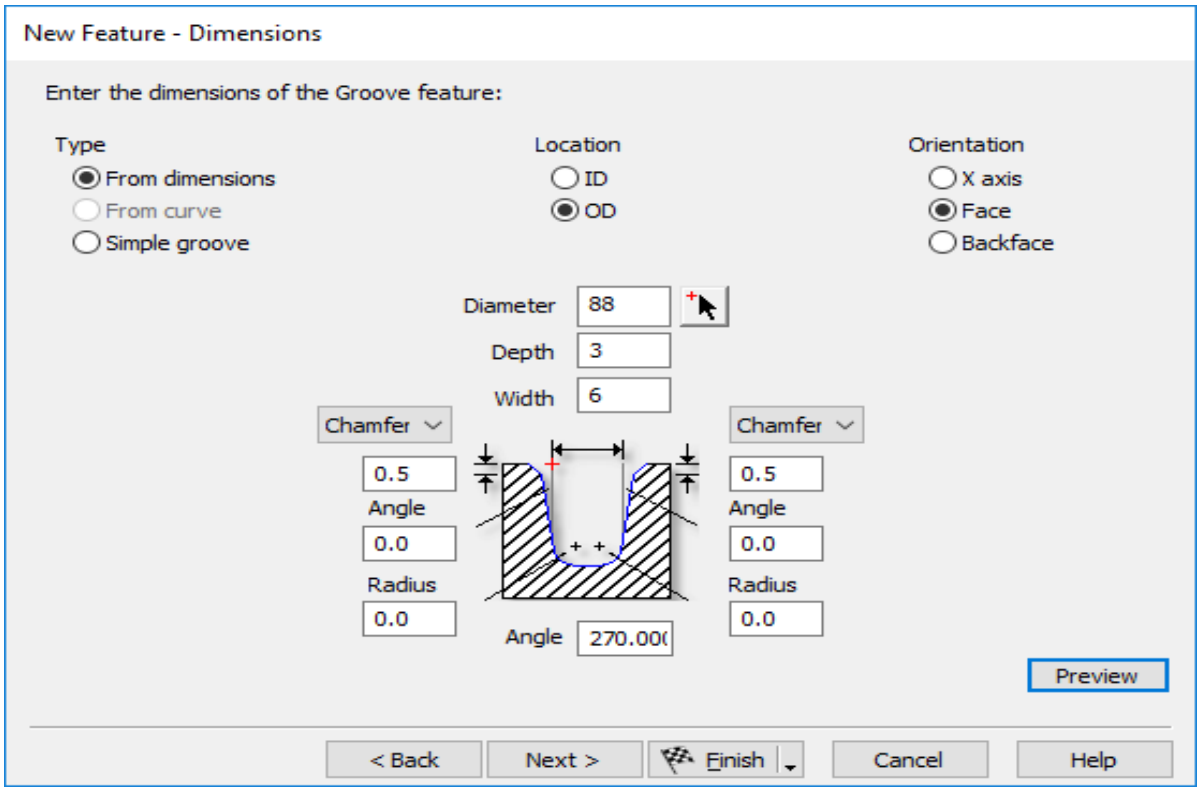


© Copyright 2016 Autodesk, Inc. All rights reserved

52 Select **Finish**. We will now create a **Face Groove**.

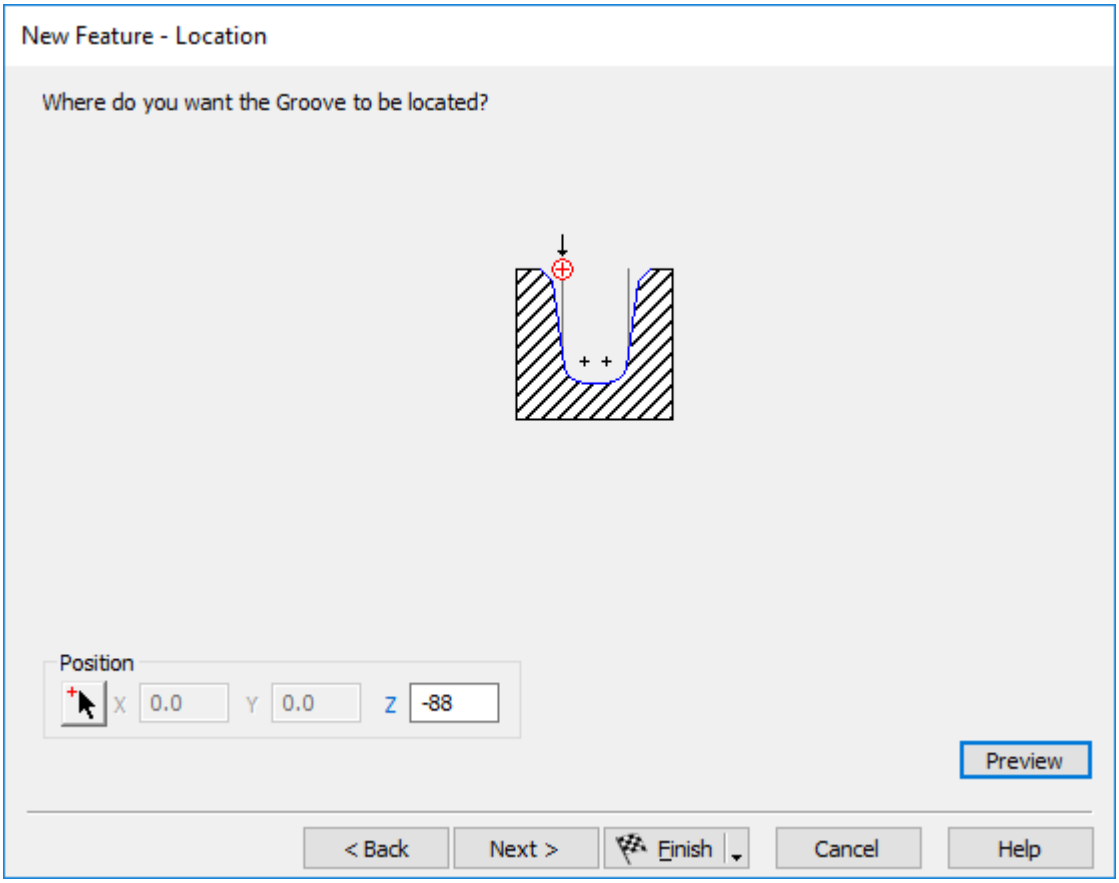


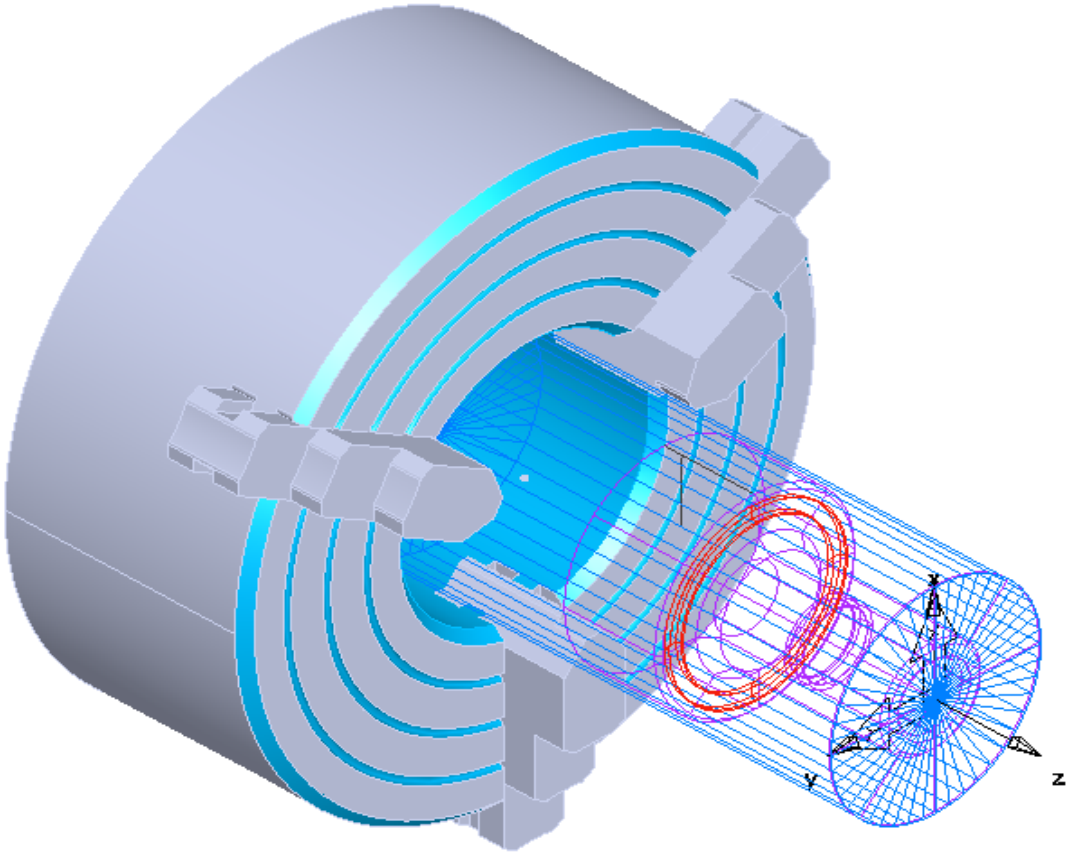
53 Select **Ctrl+R** New Feature, then select **Turning** and then **Next.** We need to select the **Groove** radio button in the **From Dimensions** group. On the Dimensions page set Location to **OD**, Orientation to **Face**, Diameter to **88mm**, Depth to **3mm**, and Width to **6 mm**. Chamfer settings at **0.5mm.** Click **Next**. The **Groove** is located at **Z-88mm**.



© Copyright 2016 Autodesk, Inc. All rights reserved

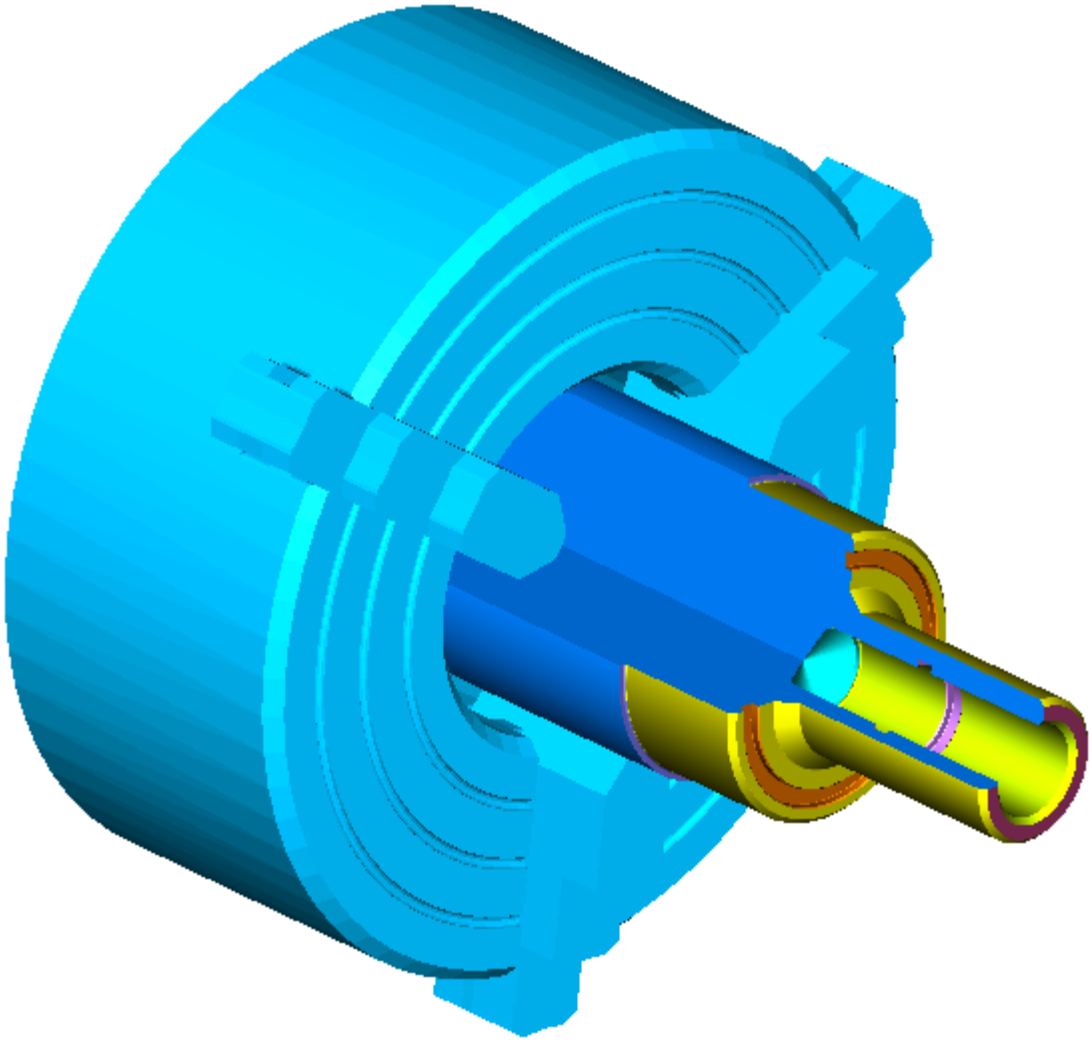
54 Select **Next.** Then enter a position of **-88mm. Then select Finish.**





© Copyright 2016 Autodesk, Inc. All rights reserved

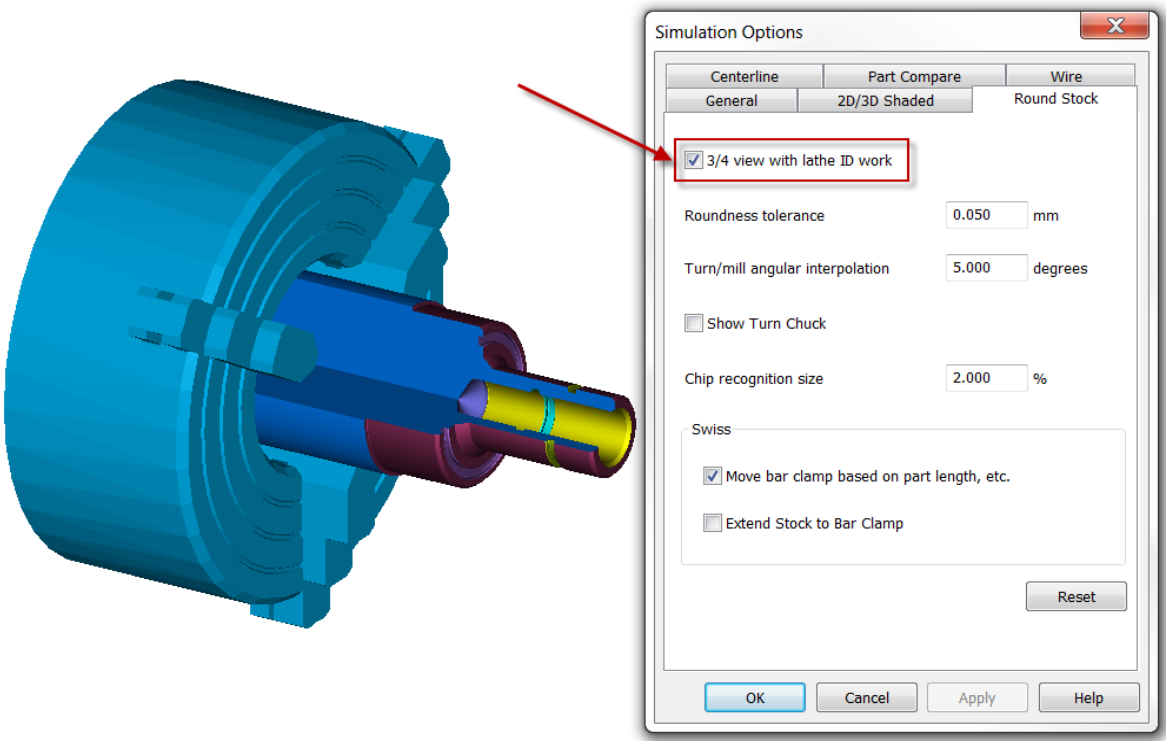
Run a **3D Simulation** to see the results below.



To View the simulation in a ***¾ view*** *select* ***Options>Simulation>Round Stock – ¾ View with lathe ID work.***

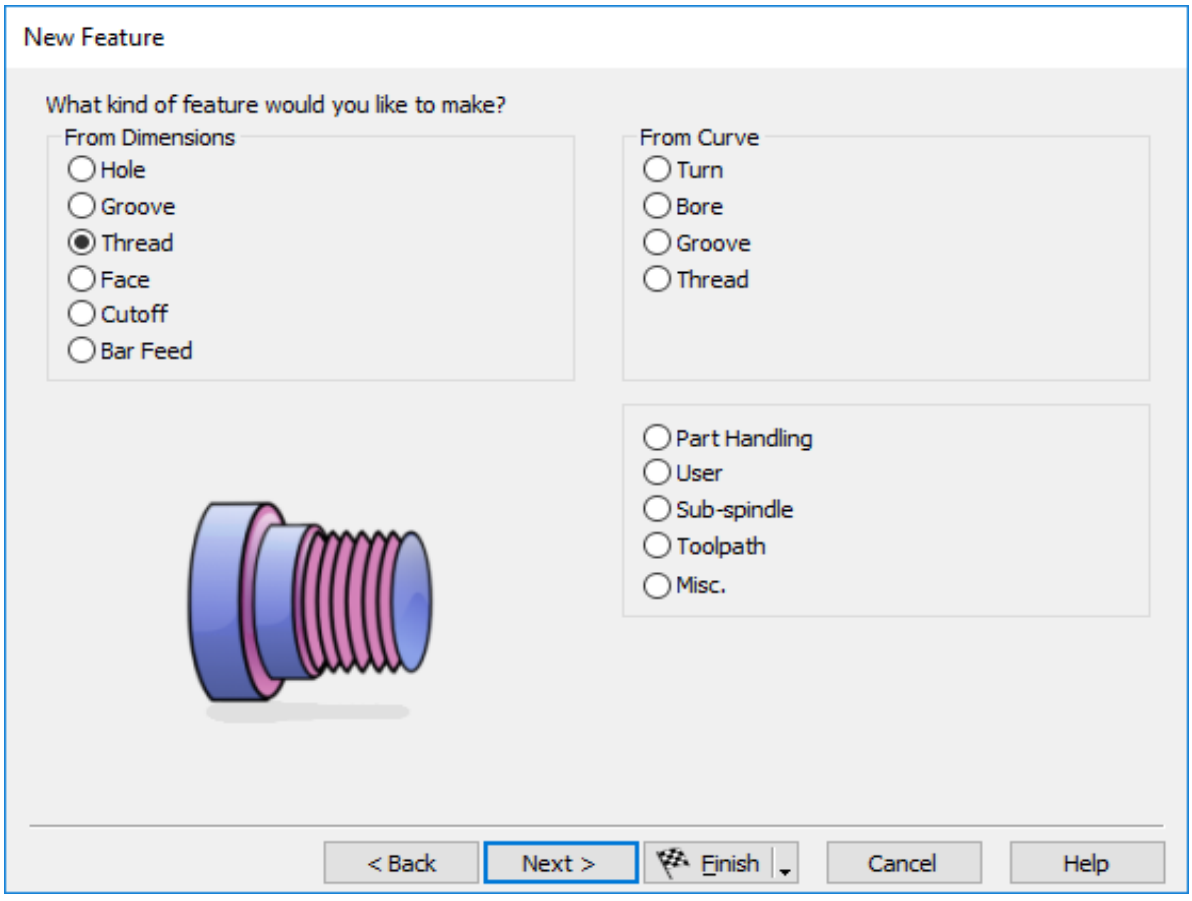
See image on next page.

© Copyright 2016 Autodesk, Inc. All rights reserved



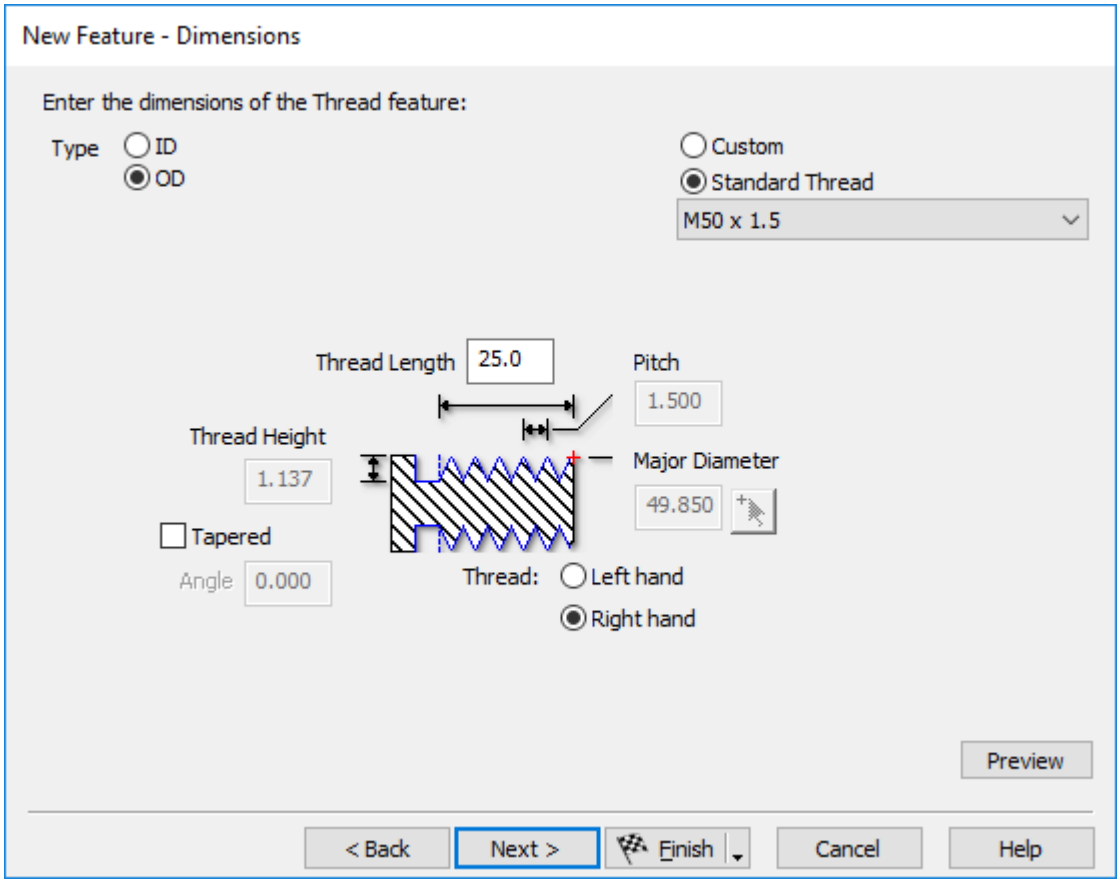
Creating a Thread Feature

55 Select **Ctrl+R (New Feature)** select **Turning** then the **Thread** radio button in the **From Dimensions** groups.



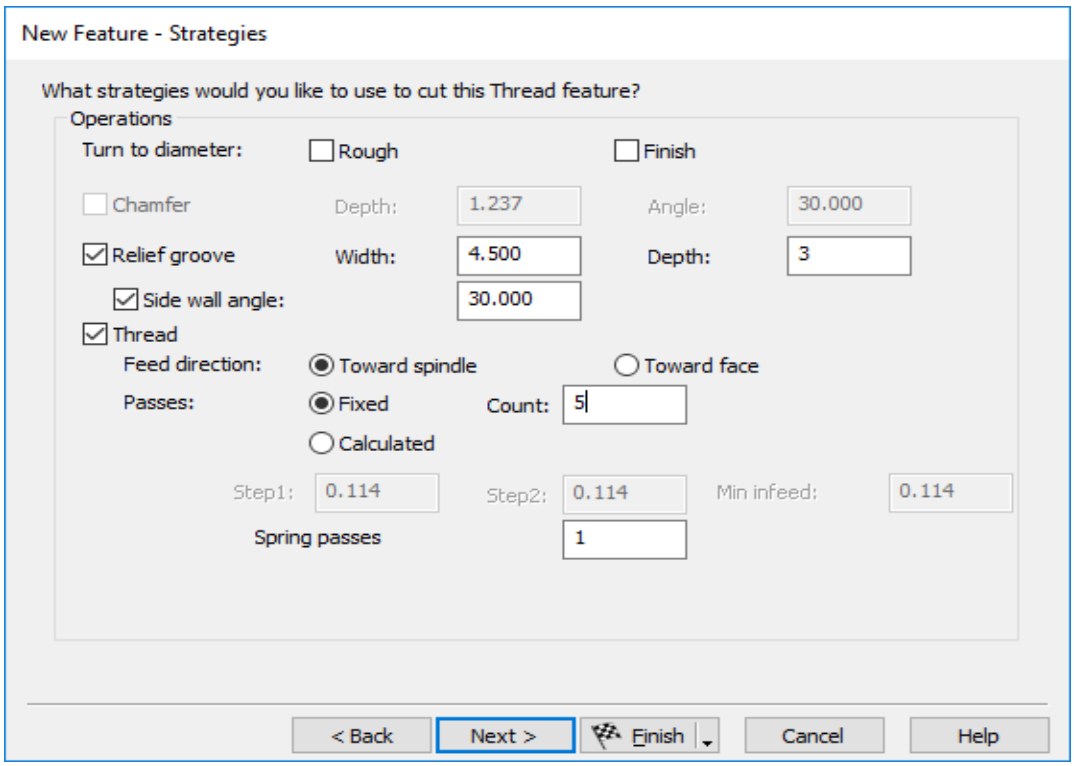
© Copyright 2016 Autodesk, Inc. All rights reserved

56 On the Dimension page, select **Standard thread**, click **OD** and select the **M50-1.5** thread. Select **Next**.



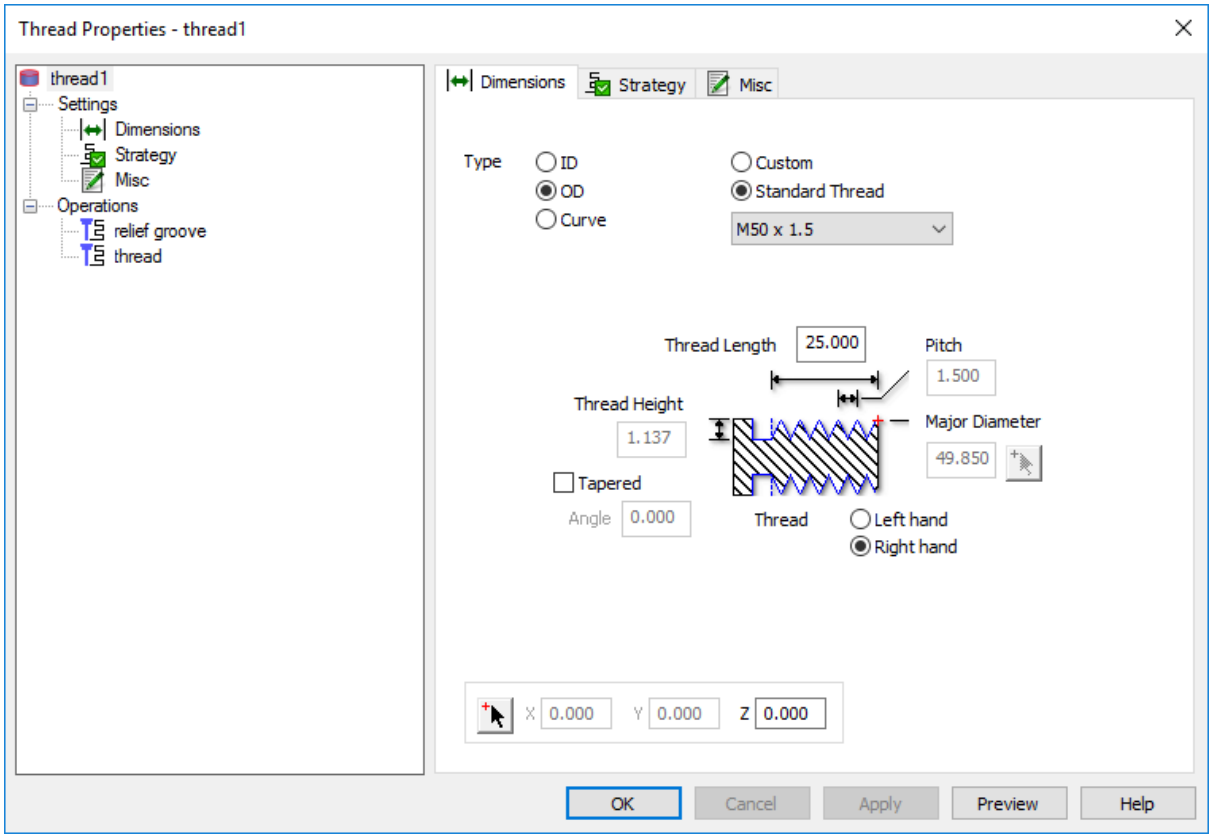
57 Location is at **zero**. Select **Next** twice.

58 By default **FeatureCAM** will create a **relief groove**. Change the depth to 3mm and then change the number of passes to **5**.

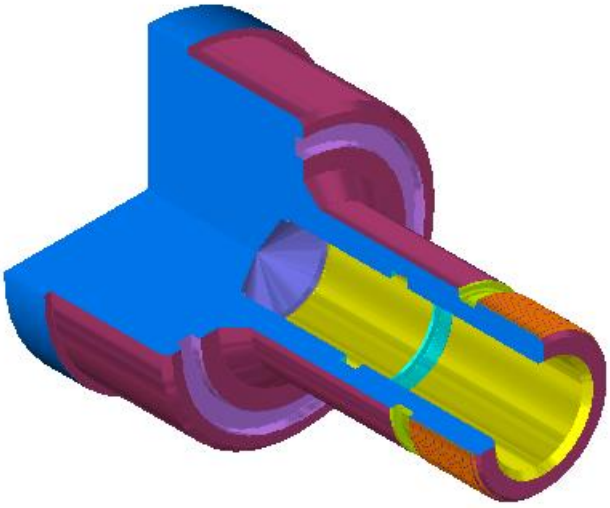


© Copyright 2016 Autodesk, Inc. All rights reserved

59 Select **Finish**.



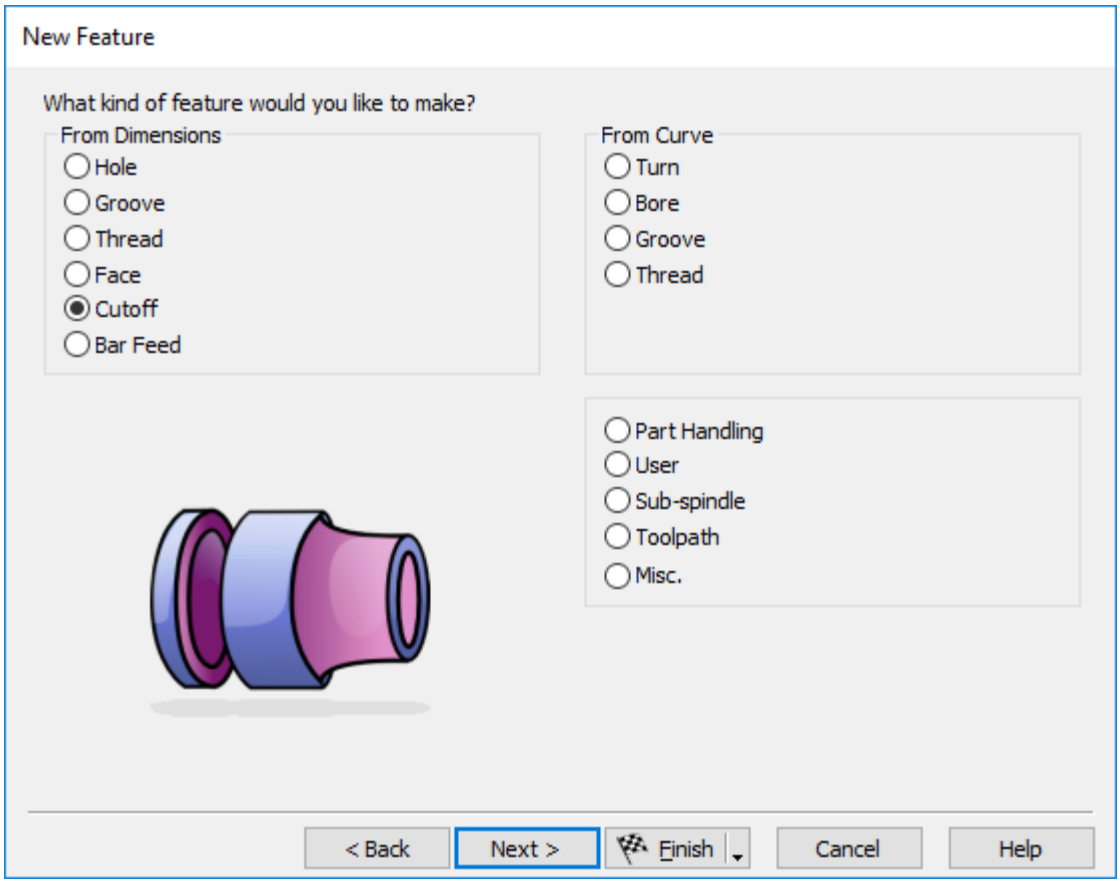
60 Run the **3D simulation** as shown.



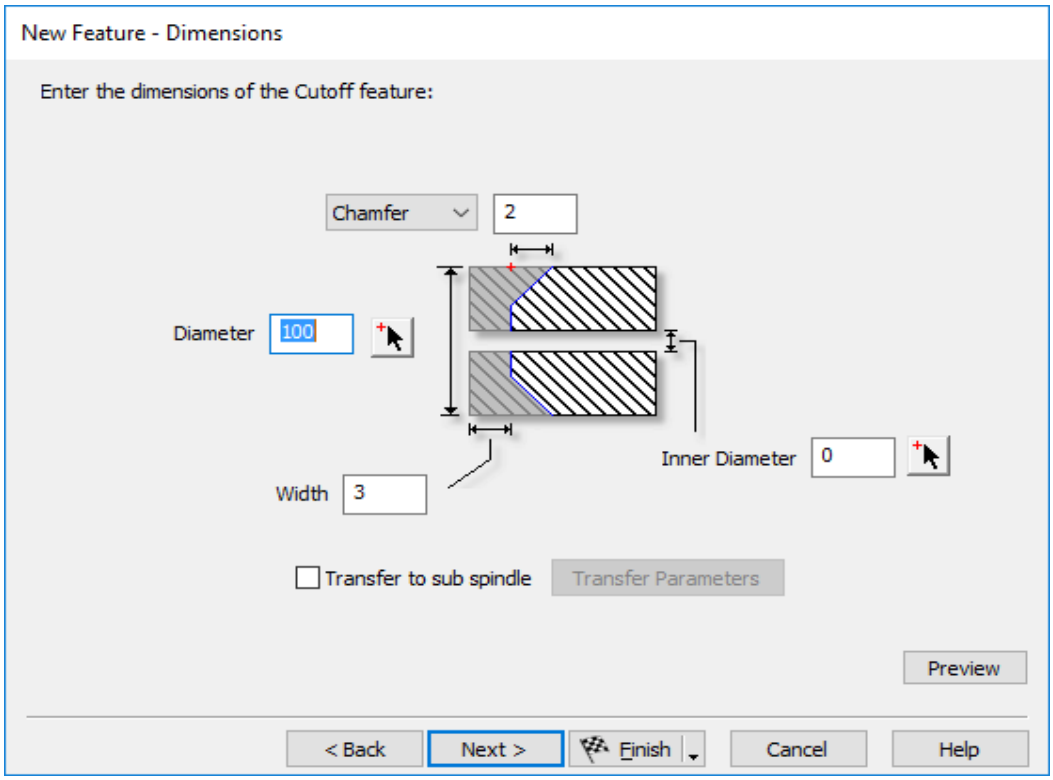
61 Remember to select the **Eject** button to clear the screen.

62 Select **Ctrl+R** New Feature and select **Turning.** Select **Cutoff**.

© Copyright 2016 Autodesk, Inc. All rights reserved

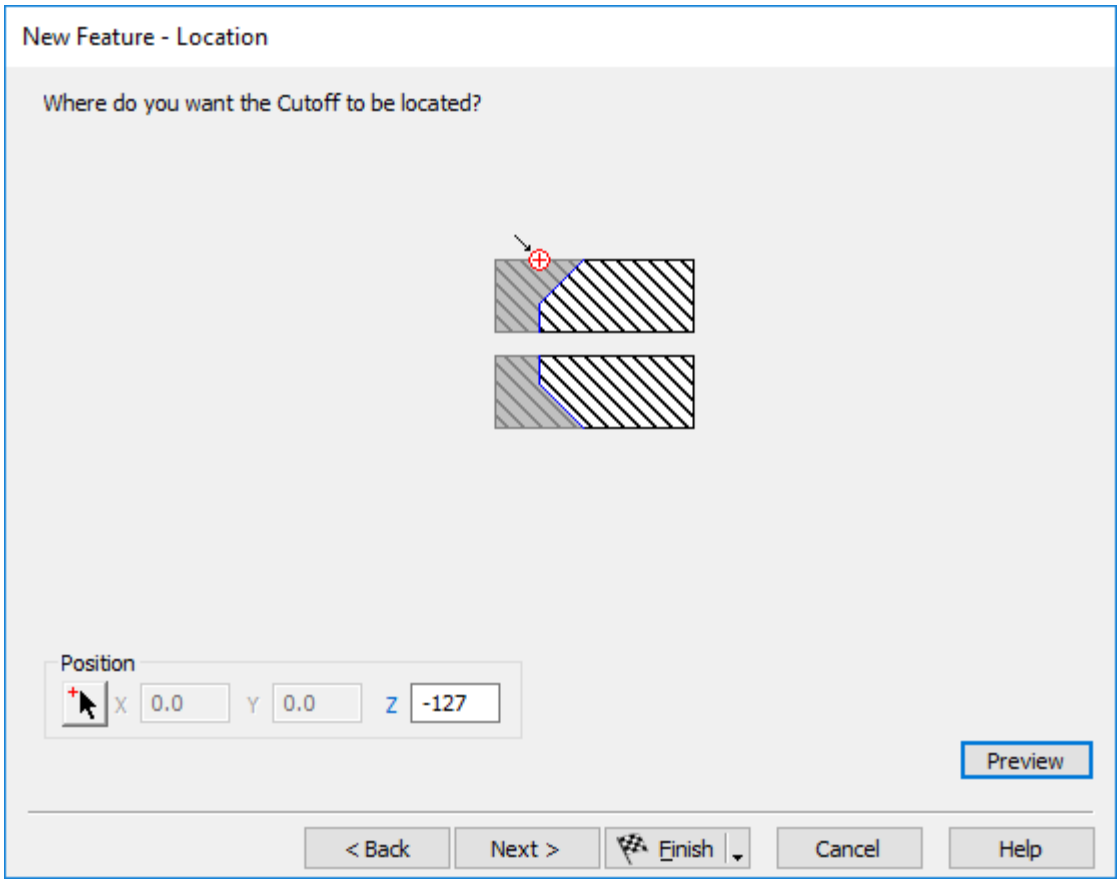


63 On the Dimensions page set Chamfer to **2** then set **Diameter** to **100mm** and **Inner Diameter** to **0** and Width **to 3.0**

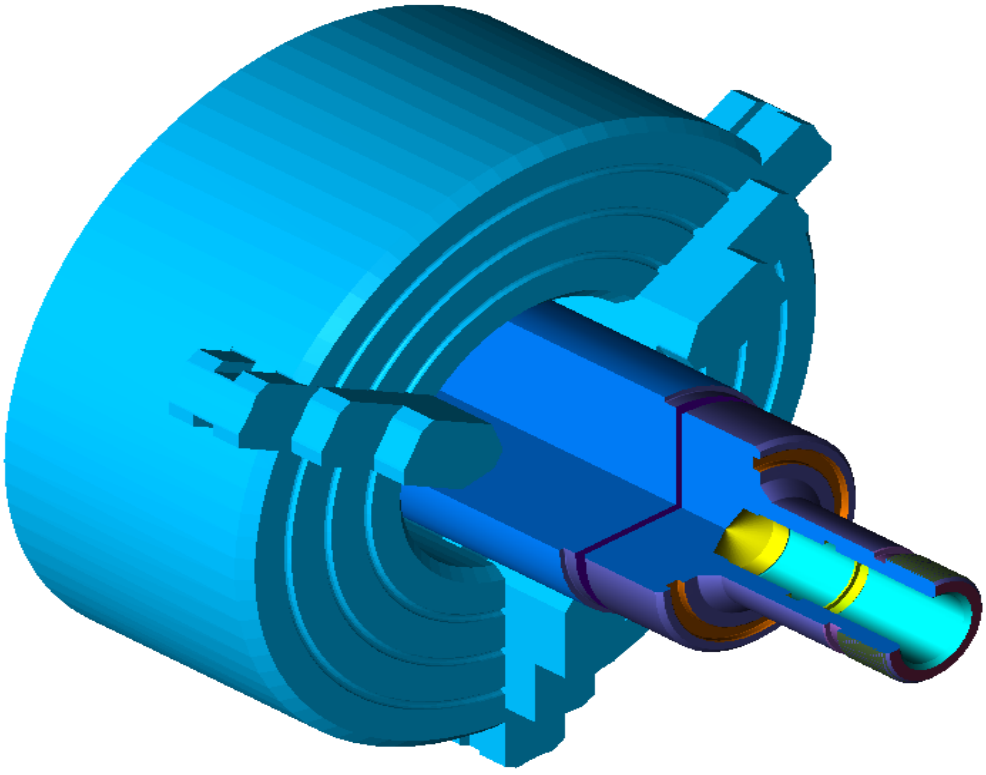


64 Select **Next** and set **Z-127mm.** Select **Finish.**

© Copyright 2016 Autodesk, Inc. All rights reserved



65 Select **Finish**. Run a **3D** Simulation



© Copyright 2016 Autodesk, Inc. All rights reserved

Results

RESULTS

Operation List

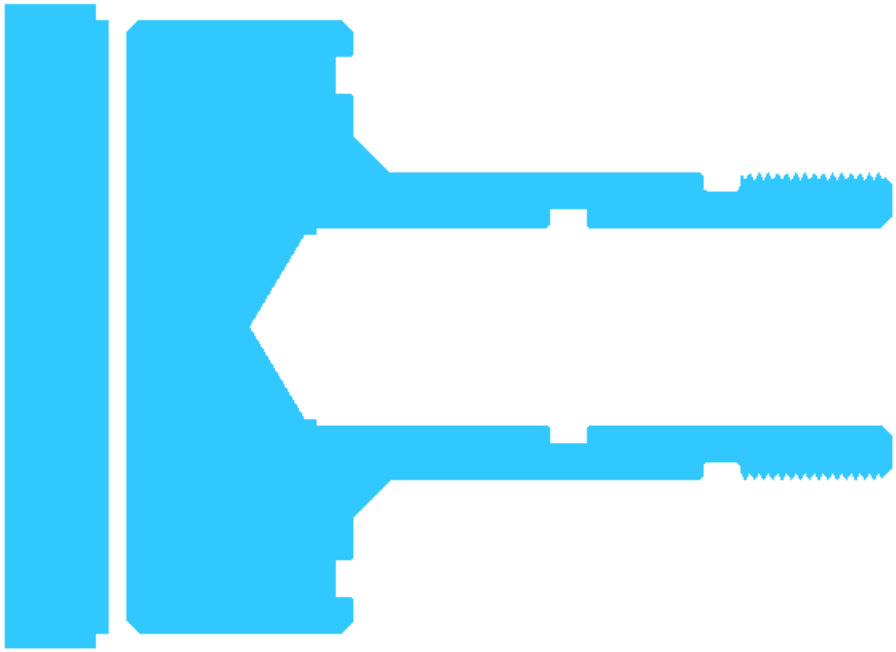
Automatic Ordering
Manual Ordering

F	Operation	Feature	Tool	Feed	Speed	Depth
➪	finish	face1	CNMG-0.4	0.152 MMPR	488 SMM CW	52.500 mm
	rough pass 1	turn1	CNMG-0.4	0.381 MMPR	366 SMM CW	27.000 mm
	finish	turn1	CNMG-0.4	0.152 MMPR	488 SMM CW	27.000 mm
	drill	hole1	30mm drill	0.450 MMPR	809 RPM CW	105.013 mm
	rough pass 1	bore1	WN_B_Sma...	0.381 MMPR	366 SMM CW	2.000 mm
	finish	bore1	WN_B_Sma...	0.152 MMPR	488 SMM CW	2.000 mm
	rough pass 1	groove1	WN_B_Sma...	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove1	WN_B_Sma...	0.152 MMPR	488 SMM CW	3.000 mm
	rough pass 1	groove2	SW_Grv_3...	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove2	SW_Grv_3...	0.152 MMPR	488 SMM CW	3.000 mm
	rough pass 1	groove3	SW_FaceGr...	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove3	SW_FaceGr...	0.152 MMPR	488 SMM CW	3.000 mm
	thread	thread1	OD_Metric_...	1.500 MMPR	2330 RPM CW	1.137 mm
	rough pass 1	cutoff1	SW_Cut_3m...	0.381 MMPR	366 SMM CW	2.000 mm
	cutoff	cutoff1	SW_Cut_3m...	0.152 MMPR	488 SMM CW	50.000 mm
	Results					

NC Code can only be output after a complete simulation has been performed. If any errors are found. The Simulation will not continue.

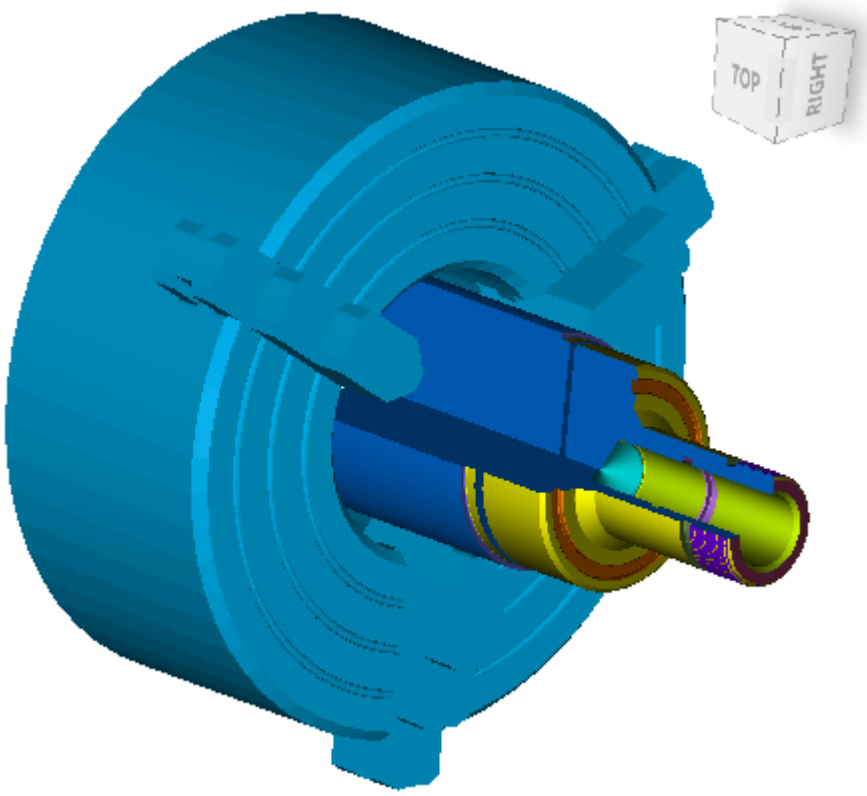
*The sample part is complete. View the **2D Centreline Simulation** followed by a **3D simulation** to view the **finished part.***

2D Simulation

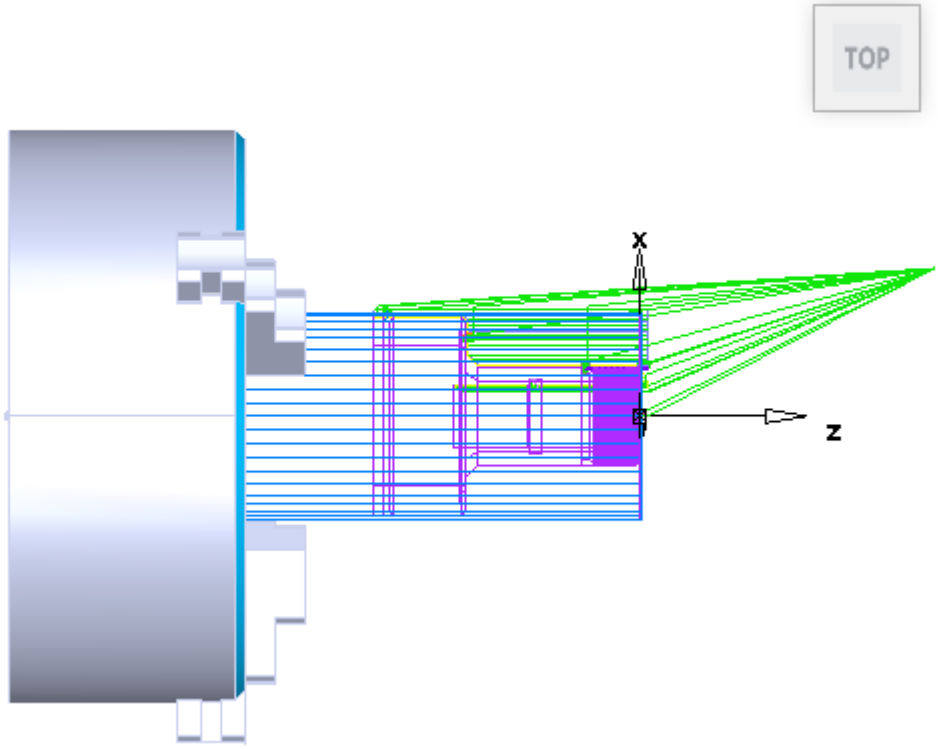


© Copyright 2016 Autodesk, Inc. All rights reserved

3D Simulation

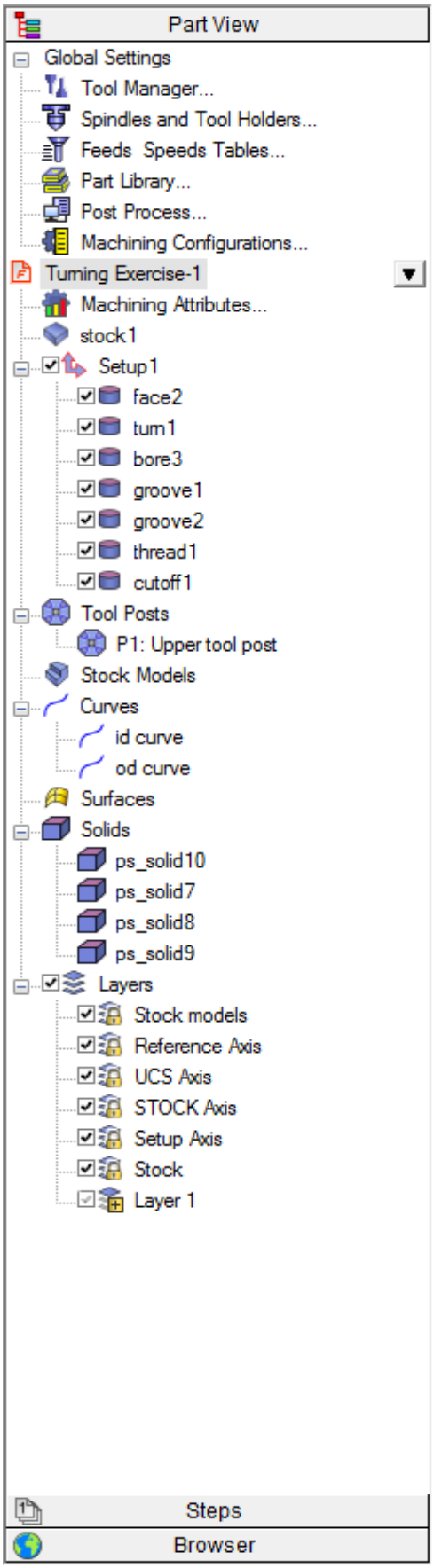


Rapid moves to home position



© Copyright 2016 Autodesk, Inc. All rights reserved

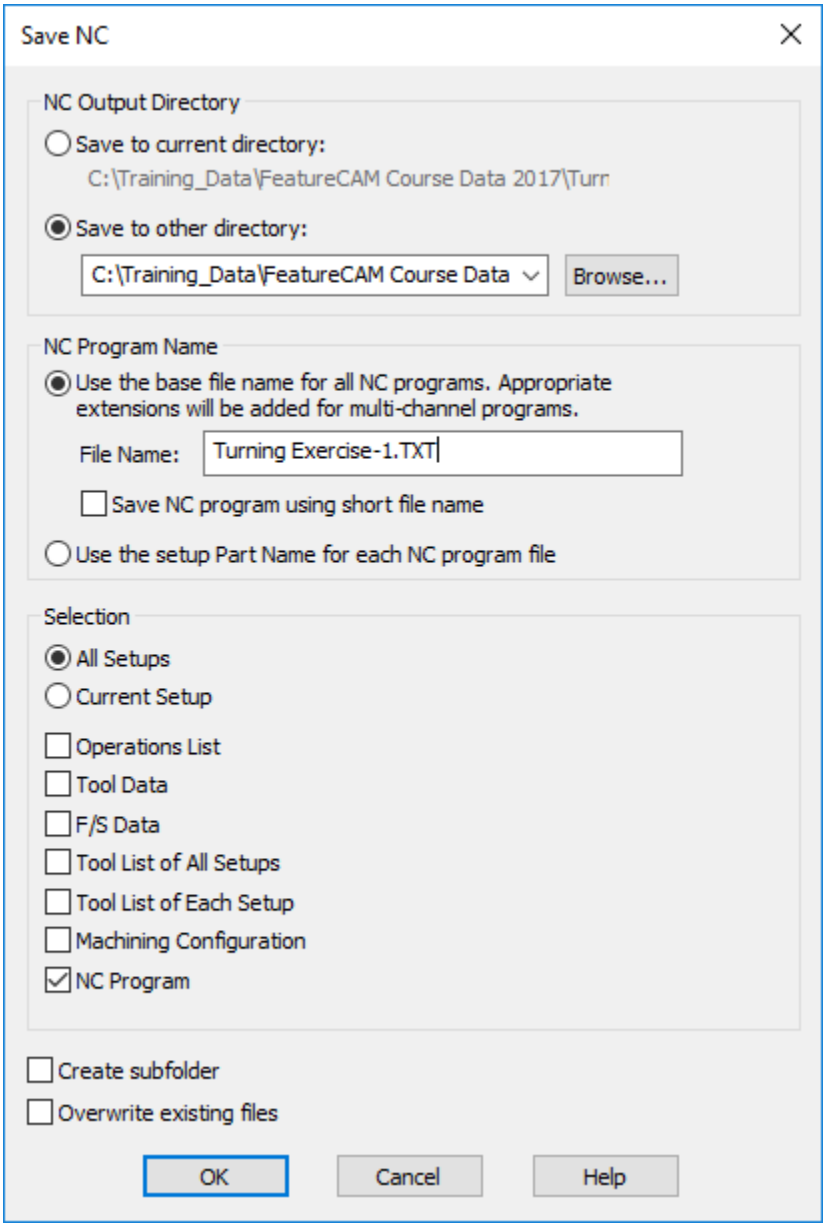
Part View All Operations

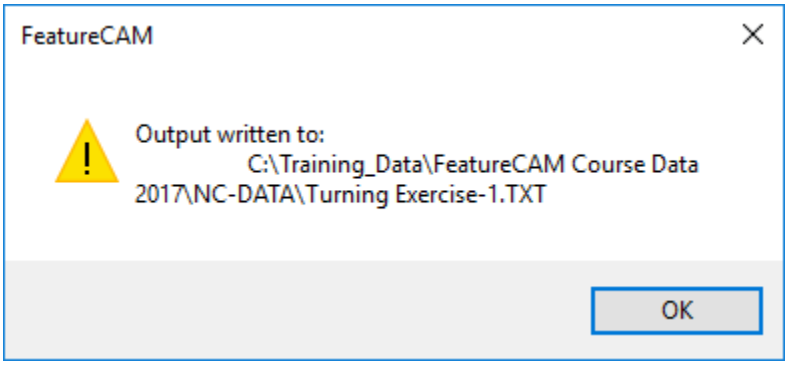


© Copyright 2016 Autodesk, Inc. All rights reserved

To output the code to your CNC machine follow these steps.

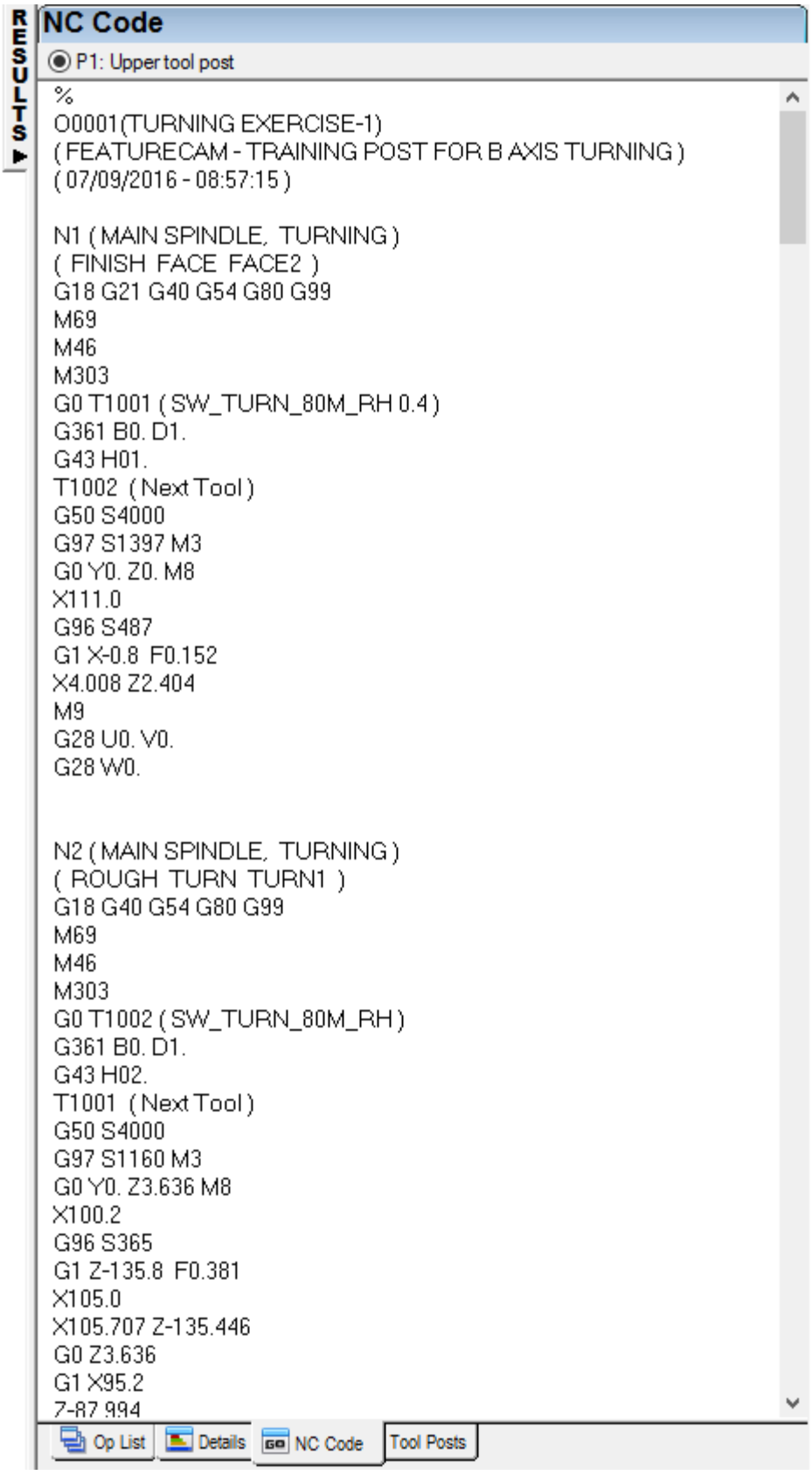
1. **Run the 3D Simulation** to check for errors etc**.**
2. From the **File** menu **select SaveNC.**





© Copyright 2016 Autodesk, Inc. All rights reserved

NC Code output example



© Copyright 2016 Autodesk, Inc. All rights reserved

Part Documentation

FeatureTURN will automatically generate a Manufacturing Operation sheet and a Tool list. The Manufacturing Operation sheet appears by default in the Results window. It contains information such as machining time, tools selected, feeds and speeds, and horsepower requirements for the various operations. A Tool list is also created that gives all the specification and slot assignments for the tools used to manufacture your part.

1. Click on the **Details** tab at the bottom of the Results window. Each operation of the process plan is listed in order.
2. Click on the **Tool List** radio button at the top of the Details tab. Now the tool list is presented in the **Results window**. This document provides a summary of all the tools required for the job as well as details of each tool.
3. You can print this documentation by selecting Print from the File menu. NC Code
4. After you have simulated the part, you can generate NC code. FeatureTURN comes with many post processors and the ability to create custom post processors as well.
5. Before you can generate NC code for a part, you have to run a Simulation to calculate toolpaths. Simulate your part now.
6. Click the NC code step in the Steps toolbar.
7. Click the Display the NC code button in the dialog box to generate.

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

NC code.

Changing Post Processors and Saving NC Code

1. If you are running FeatureTURN, the default post processor is for a Bridgeport machine. Follow the remaining steps to change your post processor and save the NC code.
2. From the Manufacturing menu, select Post Processor.
3. Click Browse to view the post processors.
4. The default directory is C:\Program Files\FeatureCAM\T-library\Metric or Inch. Select your processor and click Open. Click OK in the Post Options dialog box. Your processor is now selected. Run a simulation of the part to regenerate the NC code.

© Copyright 2016 Autodesk, Inc. All rights reserved

5 Clicking Close will not change your post processor.

6 Click the NC code step in the Steps toolbar.

7 Click the Save NC button from the dialog box.

8 Accept the default filename and directory and click OK.

9 Load your file onto your cnc machine via RS232 or Memory stick.

10 You are ready to machine the part.

11 Save the FeatureTURN Part

12 Select the Save button from the Standard toolbar.

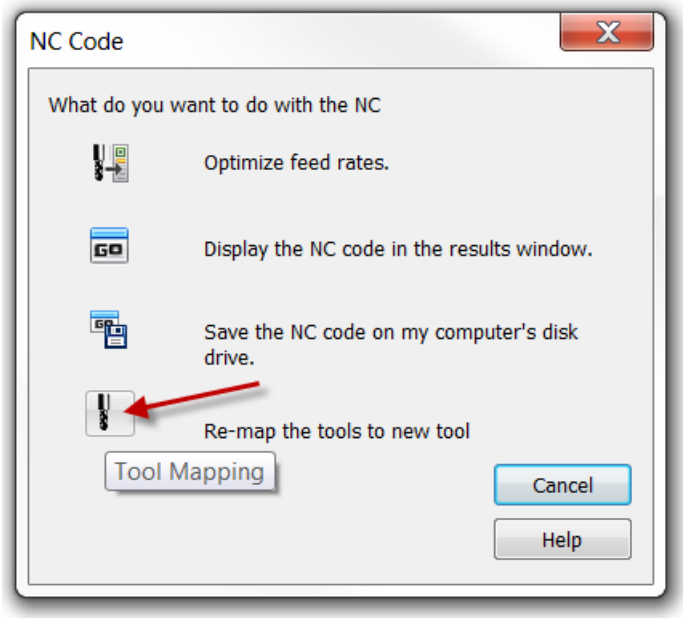
13 Select the directory and then type your part name and click OK.

Tool Mapping

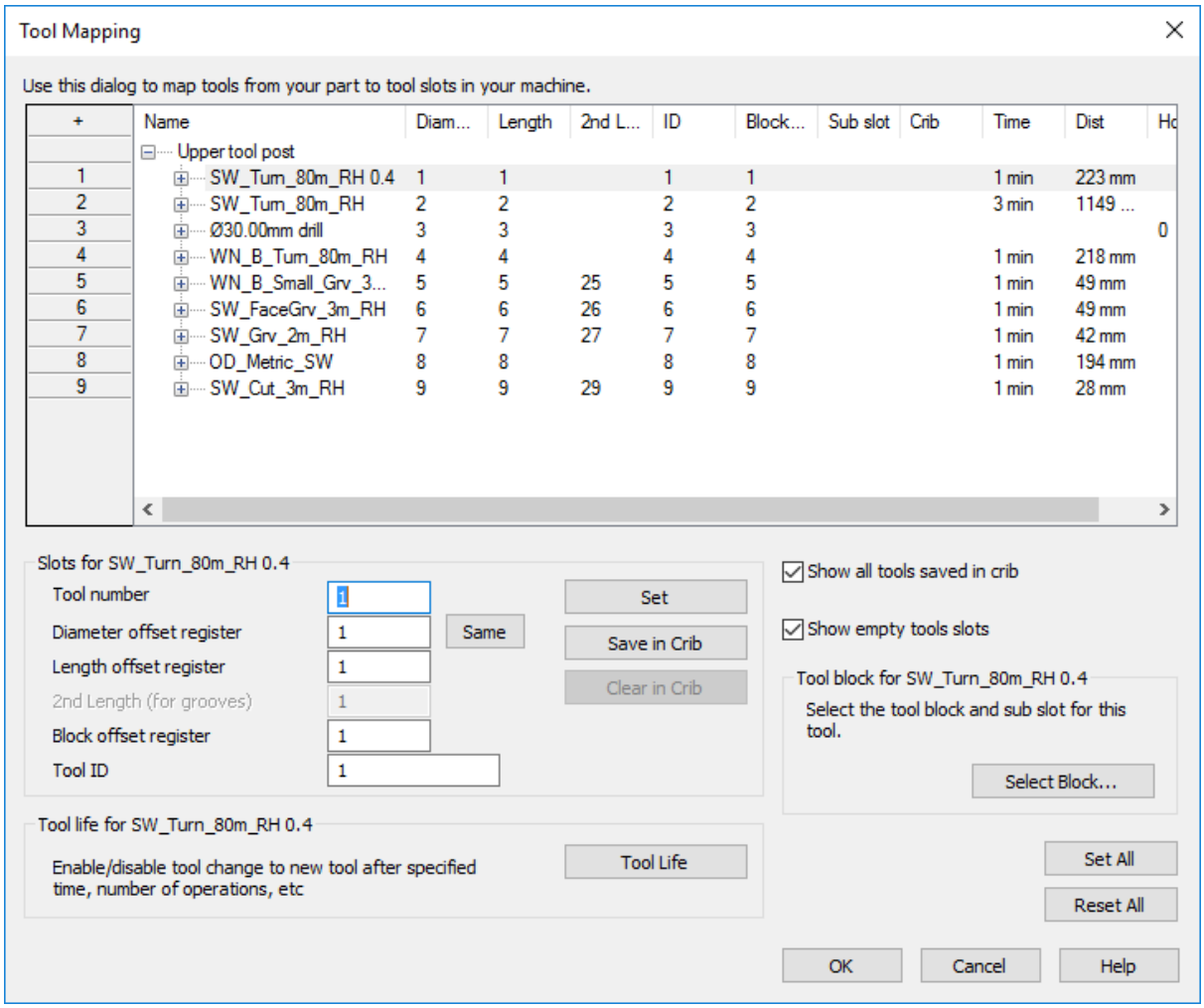
You can open the Tool Mapping dialog in one of these ways:

14 Select **Manufacturing > Tool Mapping** from the menu.

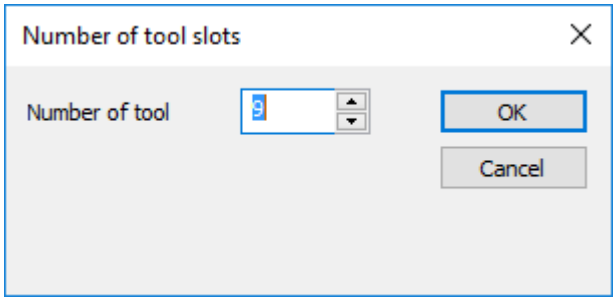
15 Click NC Code in the Steps panel, then click Re-map the tools to new tool slots in the NC Code dialog.



© Copyright 2016 Autodesk, Inc. All rights reserved



- The **Tool Mapping** dialog is where you change the tool slot assigned to the selected tool. You can change the Cutter comp. offset register for any tool here too.
- The dialog has a table at the top. Each row of the table represents a tool. Select a tool to edit its values in the fields below the table.
- Double-click on a tool name, or click the + to the left of the tool name to see the list of operations that use that tool.
- Click the Add tool slots + button at the top left of the table to open the Number of tool slots dialog. It enables you to increase the number of tool slots listed; you cannot reduce this number.



© Copyright 2016 Autodesk, Inc. All rights reserved

- **Tool number** -This corresponds to the first (grey) column in the table and is the current tool slot number for that tool. To move a tool to a different position, enter a new Tool number and click the Set button, or drag-and-drop the name of the tool in the table onto the tool slot number in the left column. More than one turning tool can occupy the same tool slot.
- **Diameter offset register** - Specifies the diameter cutter compensation offset register number for the tool. This value is passed to XBUILD as <COMP-NUM>. It corresponds to the **Diameter** column in the **Tool Mapping** table. Enter up to 8 digits.
- **Length offset register** - Specifies the tool length offset register number. Most lathe controllers have a single register that contains the length and diameter offset values. In this case, the **Length offset register** is the important field to set in FeatureCAM. This value is passed to XBUILD as <OFFSET#>. It corresponds to the **Length** column in the **Tool Mapping** table. Enter up to 8 digits.
- To set the **Diameter offset register** and the **Length offset register** by operation, select the operation in the table. You may need to expand the tool name to view the operation.
- Most lathe controllers have a single register that contains the length and diameter offset values. In this case, the **Length offset register number** is the important field to set in FeatureCAM.
- **Tool ID** - This corresponds to the **ID** column in the table and is the tool ID register for the tool. This is a seldom-used field that is used by Bridgeport lathes and occasionally for Cincinnati machines.
- The **Tool Mapping** dialog has these buttons:
- **Same** - This sets the cutter comp. offset registers for all tools to the value of their tool slot number.
- **Set** - Select a tool in the table, enter a **Tool number** and click the **Set** button to assign this tool a number specific for this part.
- *This assignment is for the current part only. If you want to assign a tool to a default tool slot for all parts, use the **Overrides** tab of the **Tool Properties** dialog.*
- **Save in Crib** - This permanently assigns the tool number with the tool in the database. The tool is then locked in this position for all parts that use the tool.
- **Clear in Crib** - The tool number slot for the selected crib is erased. This means you want FeatureCAM to assign a tool number automatically.
- **Set All** - All tools numbers are set to their current values and are not changed.
- **Reset All** - This returns all tool slot numbers and cutter comp offset registers to their initial values.
- **Select Block** - Click this button to display the **Tool Block Selection**, which you can use to specify which tool block is used to hold the selected tool.

© Copyright 2016 Autodesk, Inc. All rights reserved

- **Tool Life** - Tool life management enables you to limit the use of a tool and automatically switch to another tool when that limit is reached. It is useful when cutting hard material that may wear out a tool during a single program run. The table in the **Tool Mapping** dialog displays the number of **Holes** that are cut by each drilling tool and the **Time** (number of minutes) that each milling tool is used during a single run of the NC program. Select a tool in the table and click the **Tool Life** button to open the **Tool Life** dialog.
- *This button is not available until after you have run a simulation.*

Tool Numbering

FeatureCAM automatically selects tools from the active tool crib. These tools are assigned a tool number (also referred to as a tool slot or tool pocket) for an automatic tool changer. The numbering is assigned according to these rules:

16. Use the number assigned in the **Tool Mapping** dialog.
17. *This numbering is in effect for the current part only.*
18. If no number has been assigned via tool mapping, then the number assigned to the tool in the crib (the **Tool number** field for milling tools or the **Tool slot** for turning tools), is used as the tool number. If two tools have the same permanent number in the crib, the first tool used is assigned the pre-set number and the other tool is given a new number.
19. If no number has been assigned via tool mapping or in the crib, FeatureCAM assigns a tool number.

Custom Setup Sheet Add-In

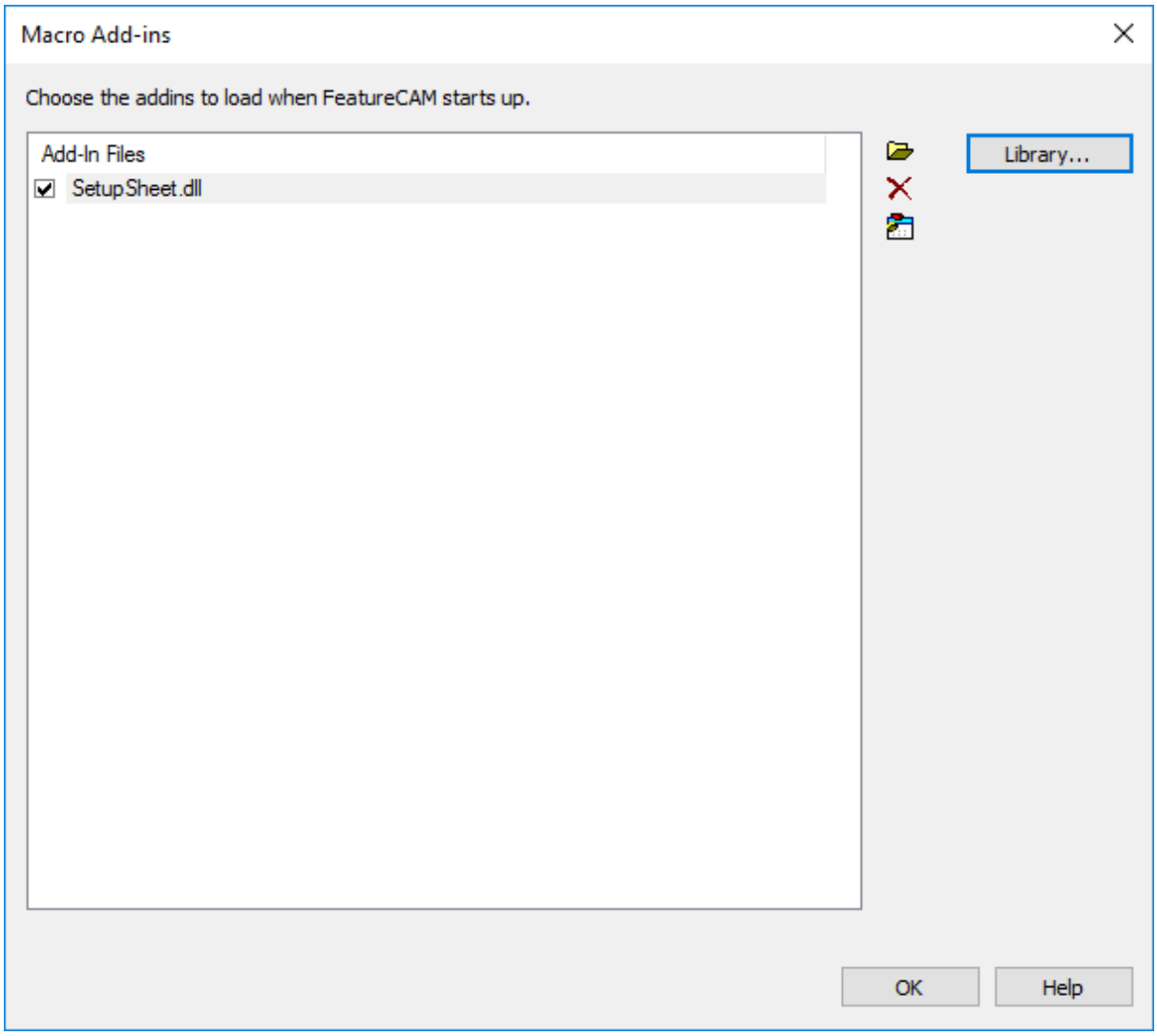
You can create custom setup sheets for milling, turning, and turn/mill parts using the **SetupSheet.dll** add-in.

The add-in extracts information from the **Part Documentation** dialog and enables you to take images for each Setup.

You can use setup sheets to give information to the machine operator about the manufacturing, tooling, and toolpaths of a part.

20. To load and run the add-in:
21. Select Options > Add-Ins from the menu. The Macro Add-ins dialog is displayed.

© Copyright 2016 Autodesk, Inc. All rights reserved



22 Click the Browse button and browse to the SetupSheet.dll file. If you installed FeatureCAM in the default location, the file is at **C:\ProgramFiles\Autodesk\FeatureCAM\Addins\SetupSheet\SetupSheet.dll.**

23 In the Macro Add-ins dialog, in the Add-In Files list, ensure that the check box to the left of the SetupSheet.dll file address is selected.

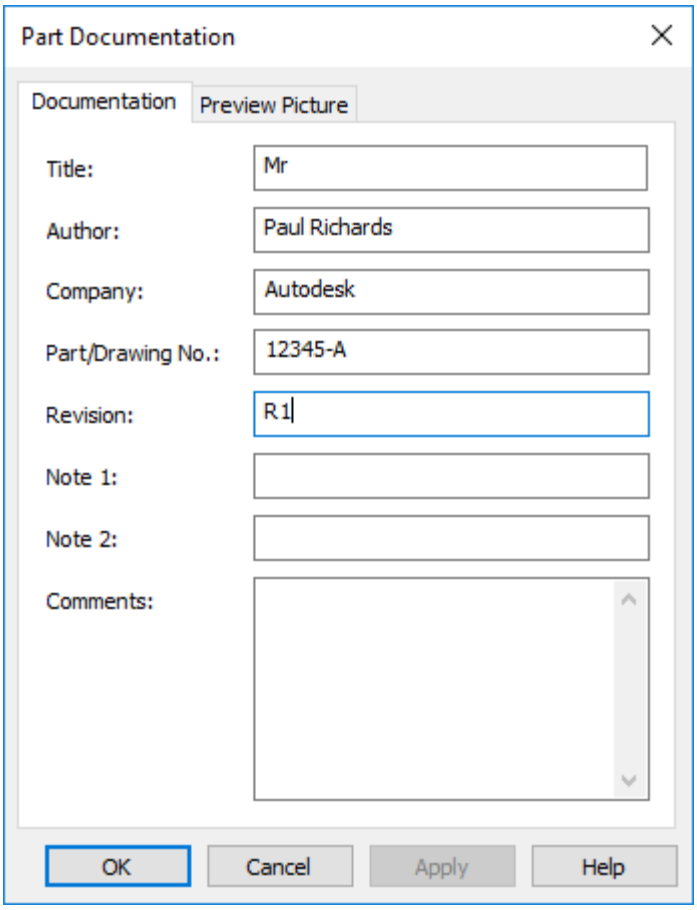
24 Click **OK**. The Utilities toolbar is displayed, containing the SetupSheet button

25 Click the SetupSheet button to run the add-in. The Setup Sheet Options dialog is displayed:

26 Remember to set the Part Document information in **File>Part Documentation.**

27 The Title, Author, Note 1, Note 2, Company, Part/Drawing No., Revision, and Comments values are copied from the Documentation tab of the Part Documentation dialog and you can edit them there.

© Copyright 2016 Autodesk, Inc. All rights reserved



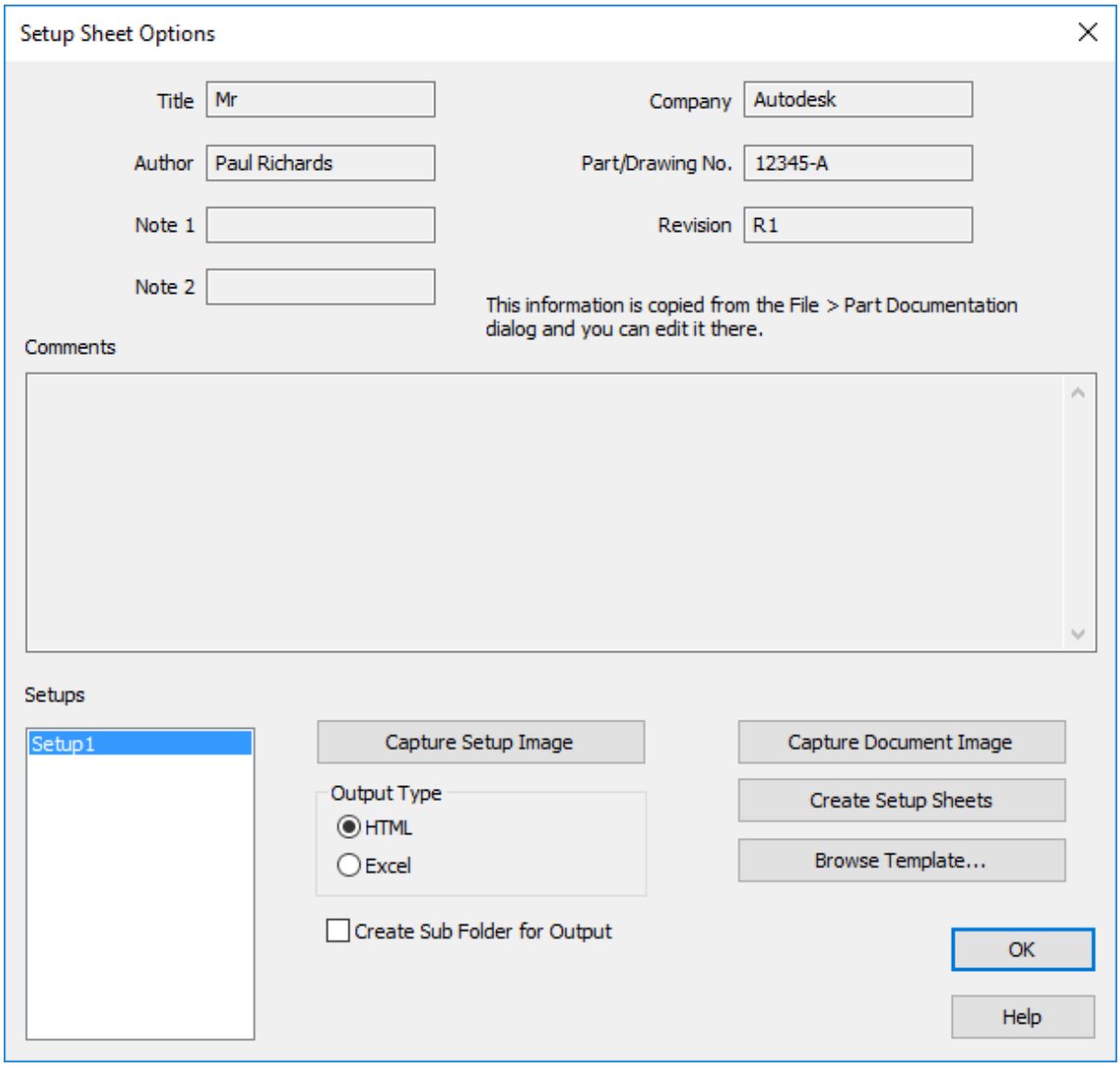
28 Select Ok to close the dialog.

29 Click **OK**. The Utilities toolbar is displayed, containing the SetupSheet button

30 You can use this dialog to capture an image for each Setup in the current document, as well as an image to represent the whole document.

31 To capture an image for a Setup, select the Setup name in the Setups list and click the Capture Setup Image button to capture the current contents of the graphics window.

© Copyright 2016 Autodesk, Inc. All rights reserved



*First run a simulation only for the Setup you want to capture by deselecting other Setups in the **Part View**; then adjust the view to show a good orientation of the Setup, and open the **Setup Sheet Options** dialog to capture the image in the graphics window.*

You must use a template to create the setup sheets. Click the Browse Template button to find and set the template you want to use. There is a template, SetupSheetTemplate.html, in the Addins\SetupSheet folder.

32 To create the setup sheets, click the **Create Setup Sheets** button. The part is simulated to generate toolpaths and the setup sheet is displayed in your web browser. You can save the HTML file from your browser.

33 To open the setup sheet in the FeatureCAM Browser, select **File > Open** from the menu and browse to the setup sheet **HTML** file.

© Copyright 2016 Autodesk, Inc. All rights reserved

Turning Exercise-1

FeatureCAM SetupSheet

Title	Mr	Company	Autodesk
Author	Paul Richards	Part/Drawing No.	12345-A
Note 1		Revision	R1
Note 2			

Comments Info

Stock Info

Type	Material	OD	ID	Z
Round	ALUMINUM	105.000000	0.000000	250.000000

Tool List

Tool Number	Tool Name	Tool Diameter	Tool Exposed Length	Tool Length*	Tool Details	Tool Image
1	SW_Turn_80m_RH 0.4	- - -	0.0000	152.0000	Tool Name: SW_Turn_80m_RH 0.4 Tool Material: CARBIDE Exposed Length: 0.0000 Tool Finish: - - - Comment: - - - Holder: - - - Maximum Depth: 52.5000 Maximum Setup Depth: 162.0000 Maximum Height: 0.0000 Coolant Override: No Override Length: 152.0000 Tip Radius: 0.4000 Holder Orientation: SW	
2	SW_Turn_80m_RH	- - -	0.0000	152.0000	Tool Name: SW_Turn_80m_RH Tool Material: CARBIDE Exposed Length: 0.0000 Tool Finish: - - - Comment: - - - Holder: - - - Maximum Depth: 27.0000 Maximum Setup Depth: 162.0000 Maximum Height: - - - Coolant Override: No Override Length: 152.0000 Tip Radius: 0.8000 Holder Orientation: SW	
3	Ø30.00mm drill	30.0000	165.8000	143.0000	Tool Name: Ø30.00mm drill Tool Material: HSS Exposed Length: 165.8000 Tool Finish: BRIGHT Comment: - - - Holder: BT40 ER40X80 Maximum Depth: 105.0129 Maximum Setup Depth: 105.0129 Maximum Height: 0.0000 Coolant Override: No Override	

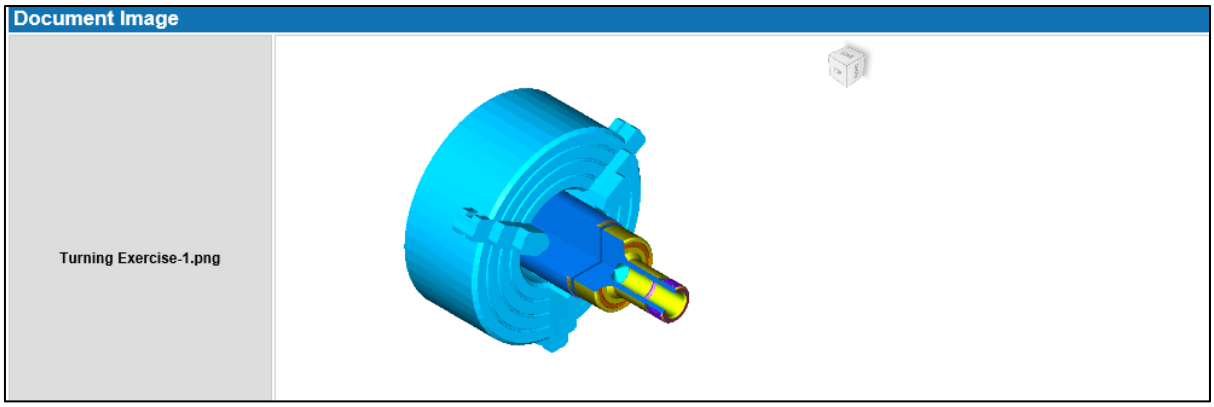
Setup : Setup1

Setup Origin	0.0000, 0.0000, 0.0000

Feature Name	Feature Type	Operation	Tool Number	Tool Name	Operation Feed	Operation Speed	Operation Time	Operation Comment
face2	Face Turn	Finish	1	SW_Turn_80m_RH 0.4	0.1524	487.6800	0:00:19.4	
turn1	Turn Bore	Rough	2	SW_Turn_80m_RH	0.3810	365.7600		
turn1	Turn Bore	Finish	1	SW_Turn_80m_RH 0.4	0.1524	487.6800		
bore3	Turn Bore	Spiral	3	Ø30.00mm drill	0.4500	808.5071		
bore3	Turn Bore	Rough	4	WN_B_Turn_80m_RH	0.3810	365.7600		
bore3	Turn Bore	Finish	4	WN_B_Turn_80m_RH	0.1524	487.6800		
groove1	Groove Turn	Rough	5	WN_B_Small_Grv_3m_RH	0.3810	365.7600		
groove1	Groove Turn	Finish	5	WN_B_Small_Grv_3m_RH	0.1524	487.6800		
groove2	Groove Turn	Rough	6	SW_FaceGrv_3m_RH	0.3810	365.7600		
groove2	Groove Turn	Finish	6	SW_FaceGrv_3m_RH	0.1524	487.6800		
thread1	Thread	Rough	7	SW_Grv_2m_RH	0.3810	365.7600		
thread1	Thread		8	OD_Metric_SW	1.5000	2330.0000		
cutoff1	Cutoff	Rough	9	SW_Cut_3m_RH	0.3810	365.7600		
cutoff1	Cutoff	Rough	9	SW_Cut_3m_RH	0.1524	487.6800		

Total Setup Time

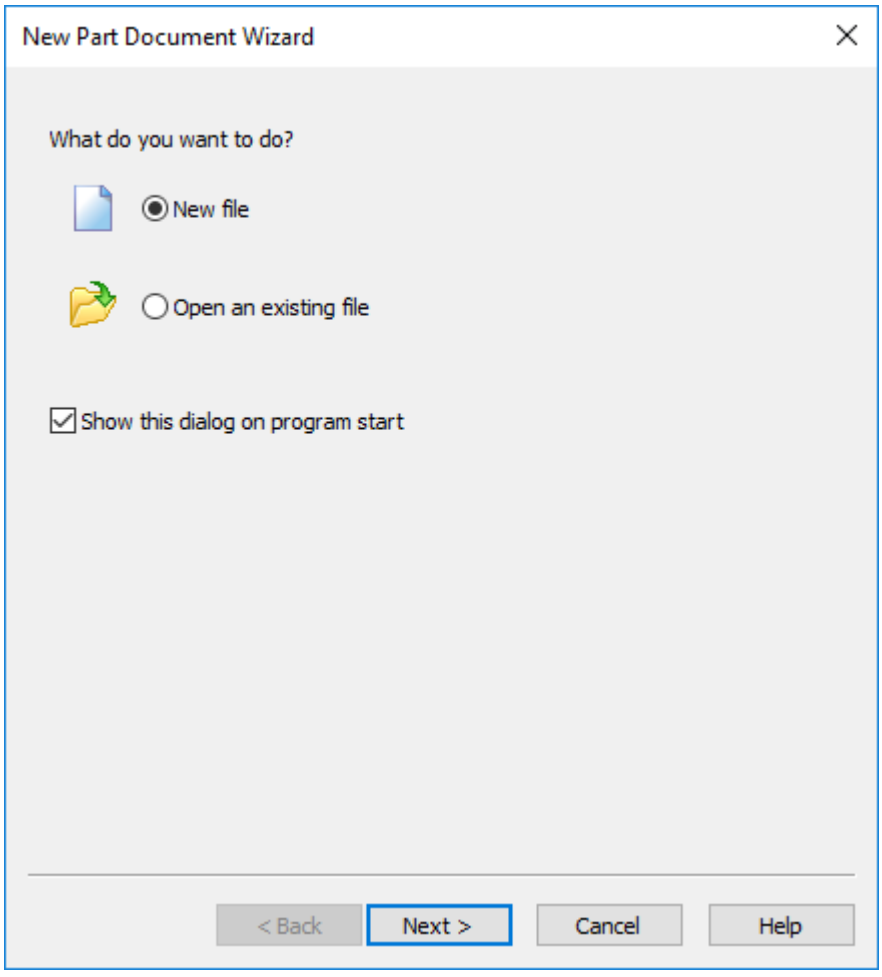
© Copyright 2016 Autodesk, Inc. All rights reserved



Save your File and then close this document.

Exercise 2

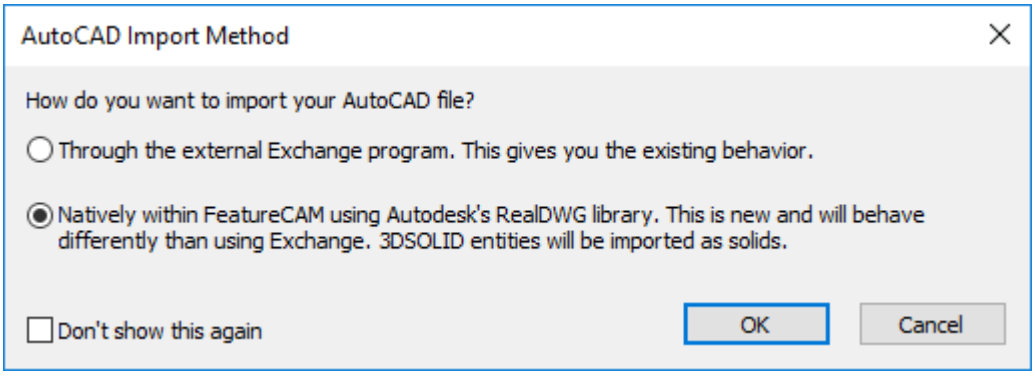
The first exercise is now complete. *Please select* ***New file*** *to create a new document****.*** *Select* ***Next*** *and then* ***Turn/Mill.*** *Select* ***Finish.***



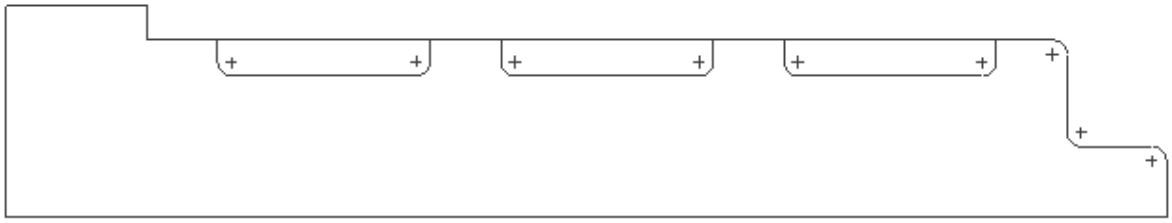
1. Cancel the **Stock Wizard**.
2. Import – **Turning Example 2.dxf** select **Open** to import the file into FeatureTURN**.** (Class Exercise)

© Copyright 2016 Autodesk, Inc. All rights reserved

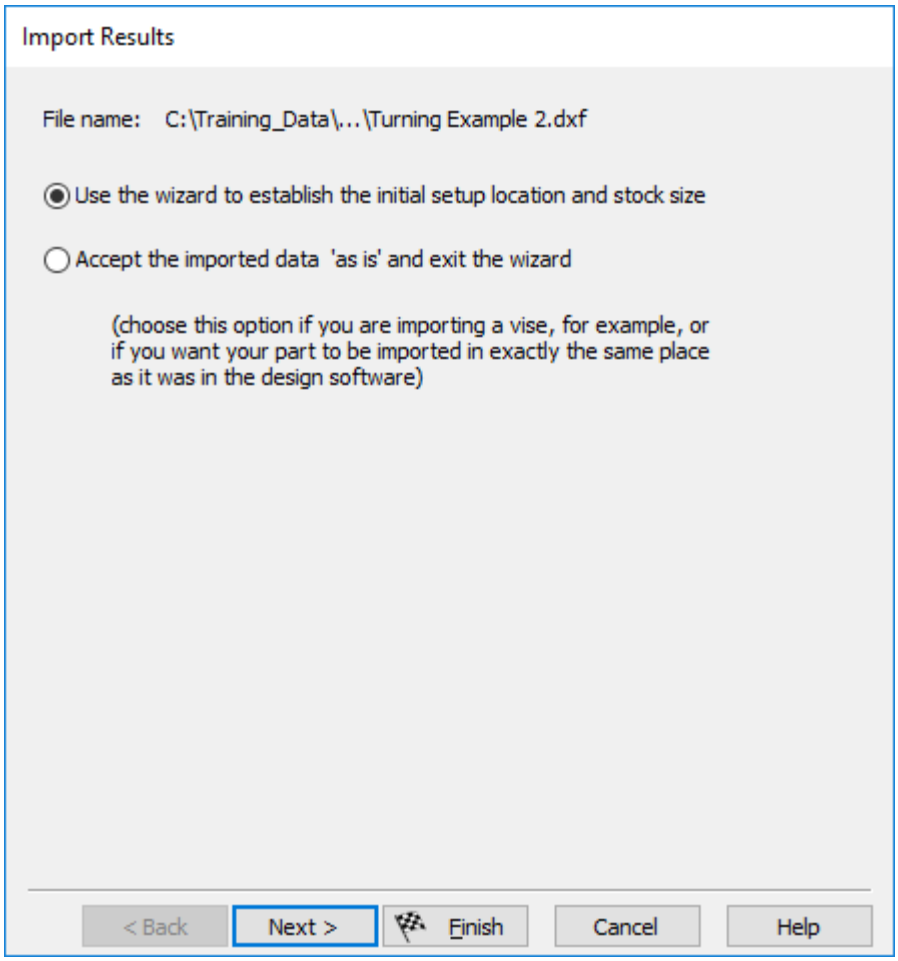
3 We are going to use the RealDWG Library file (see below)



4 Select **OK** to action this command.



5 Use the **Wizard** to establish the **Stock Size** and **Setup1** position.

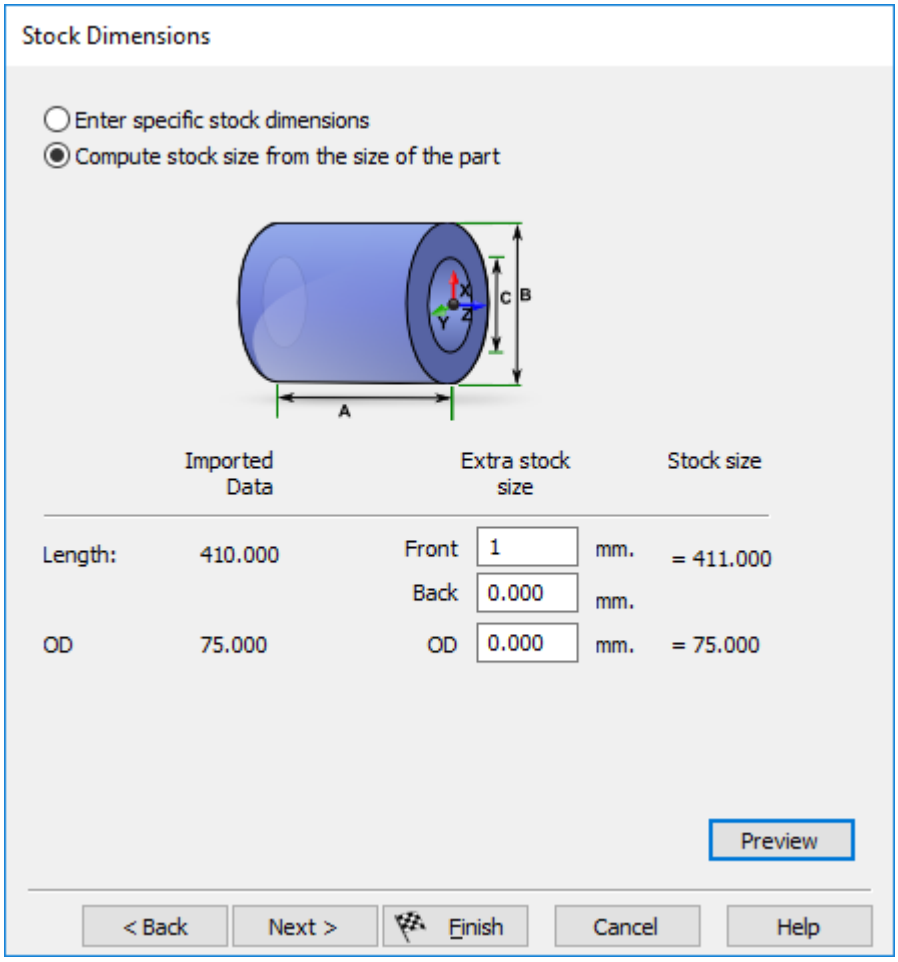


6 Keep selecting **Next** until you get to the **Stock Dimensions.**

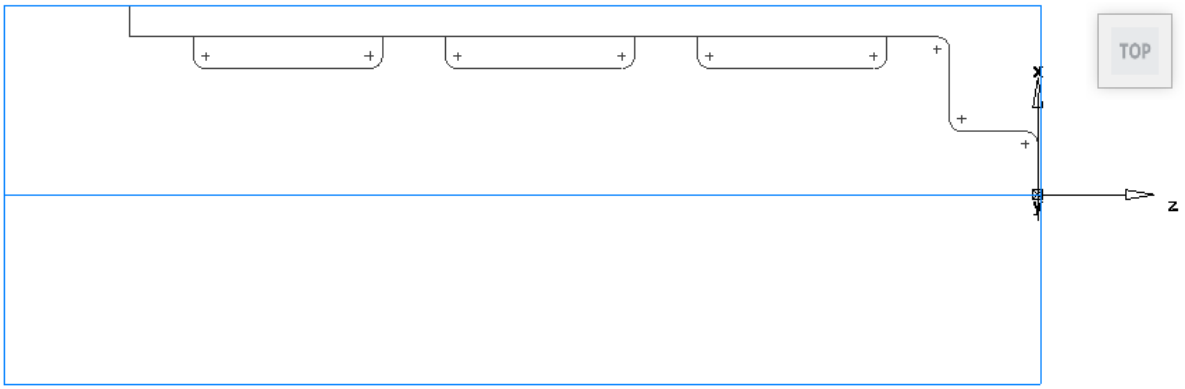
7 Remember to use the option **Compute stock size from the size of the part**.

© Copyright 2016 Autodesk, Inc. All rights reserved

8 All we are going to use is **Front 1 Back and OD are set to Zero.**



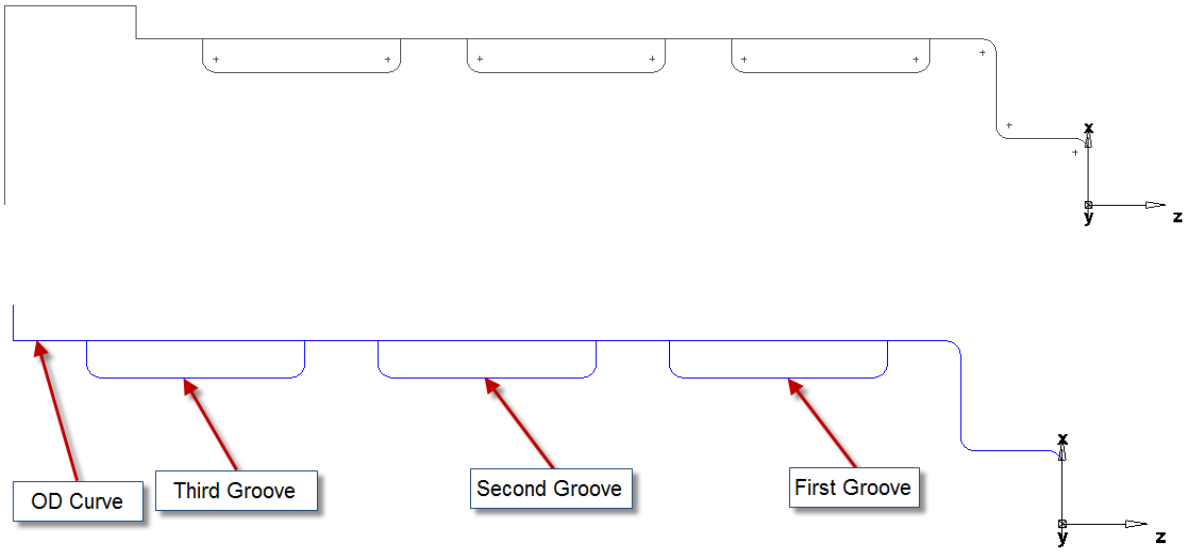
9 Select **Finish**.



*The main reason for machining this part is to create 3 different machining methods for machining the same groove. We will be starting with **Normal turning** and **back turning**. Followed by **Cut-Grip** and then for the last groove we will be using a more traditional machining strategy with **conventional grooving**. All 3 methods will produce the same result. When you are back at work you will use the machining strategy that suits your company.*

© Copyright 2016 Autodesk, Inc. All rights reserved

*To chain your geometry pick each element in turn, after you have made your selection press the Create key so all of the elements will be joined together. Each selection will be given a unique name. If you select the wrong geometry, select **Clear pieces** and then reselect the elements again.*



10 On the illustration above, the Curves shown are the **OD Curve**, **First Groove, Second Groove, and Third Groove.**

Turn Feature

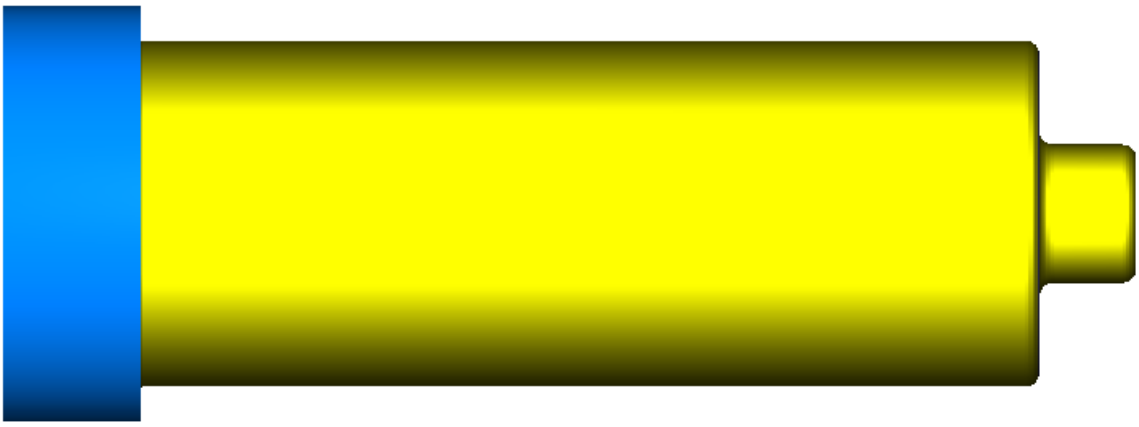
11 Cancel this command by pressing **ESC**.

12 Hide the Geometry by selecting **View>Hide>Hide all Geometry**.

13 Select **Ctrl+R New Feature** Select **Turning**, select **next**, select **Face** Select **Finish.**

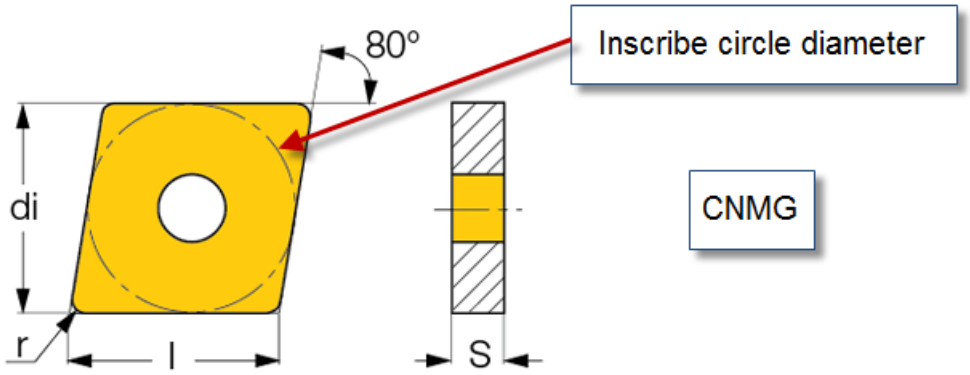
14 Select **Ctrl+R New Feature** and select Turning> **Turn** select next then select **odcurve** from the **New Feature Curve** menu. Select **Finis**h**.**

*New to **FeatureCAM** we can automatically Fillet external or internal sharp corners. For internal bore fillet corners use **Other side.***

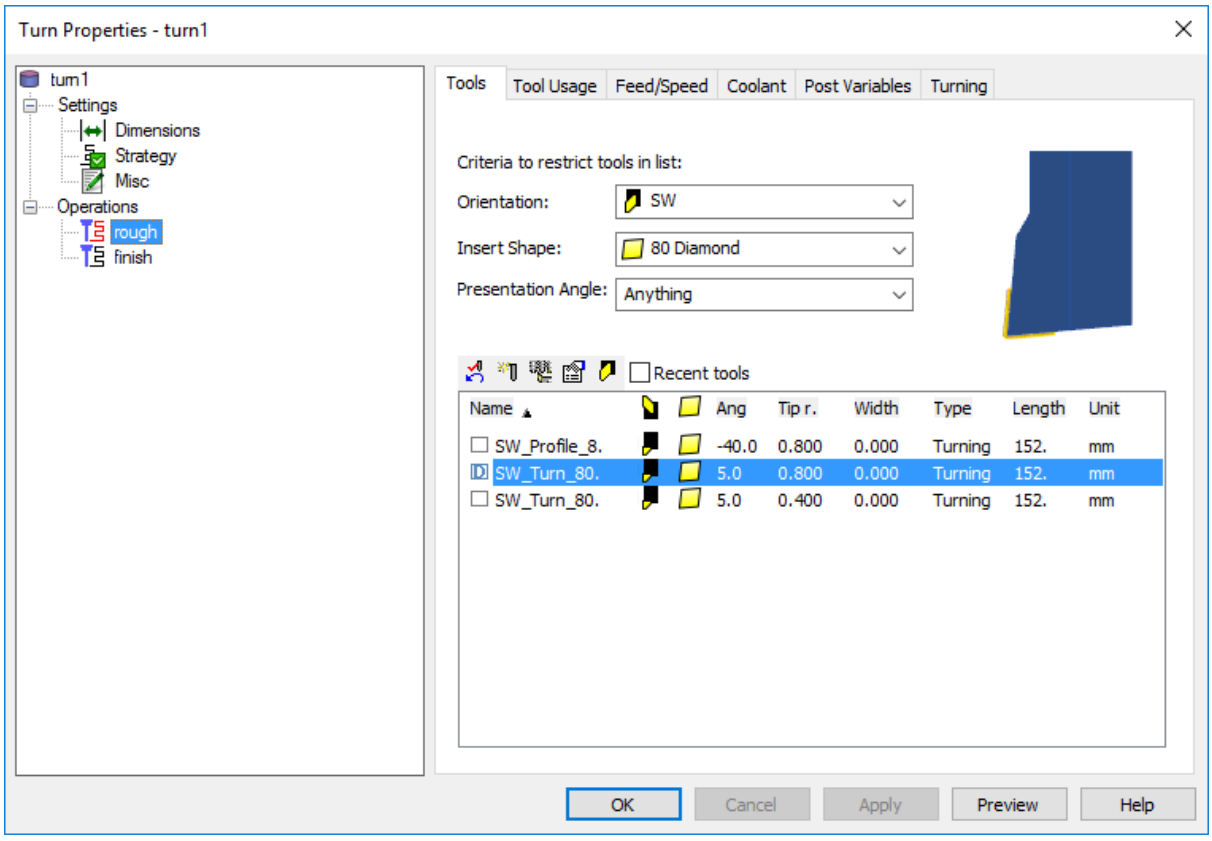


© Copyright 2016 Autodesk, Inc. All rights reserved

*In the Properties, please select **Rough** and then the **Tools** tab. This is a listing of the tool that **FeatureCAM** picked as a default tool with a **(D) Default** in the check box and other tools that may be chosen to replace the **80 degree tool.***

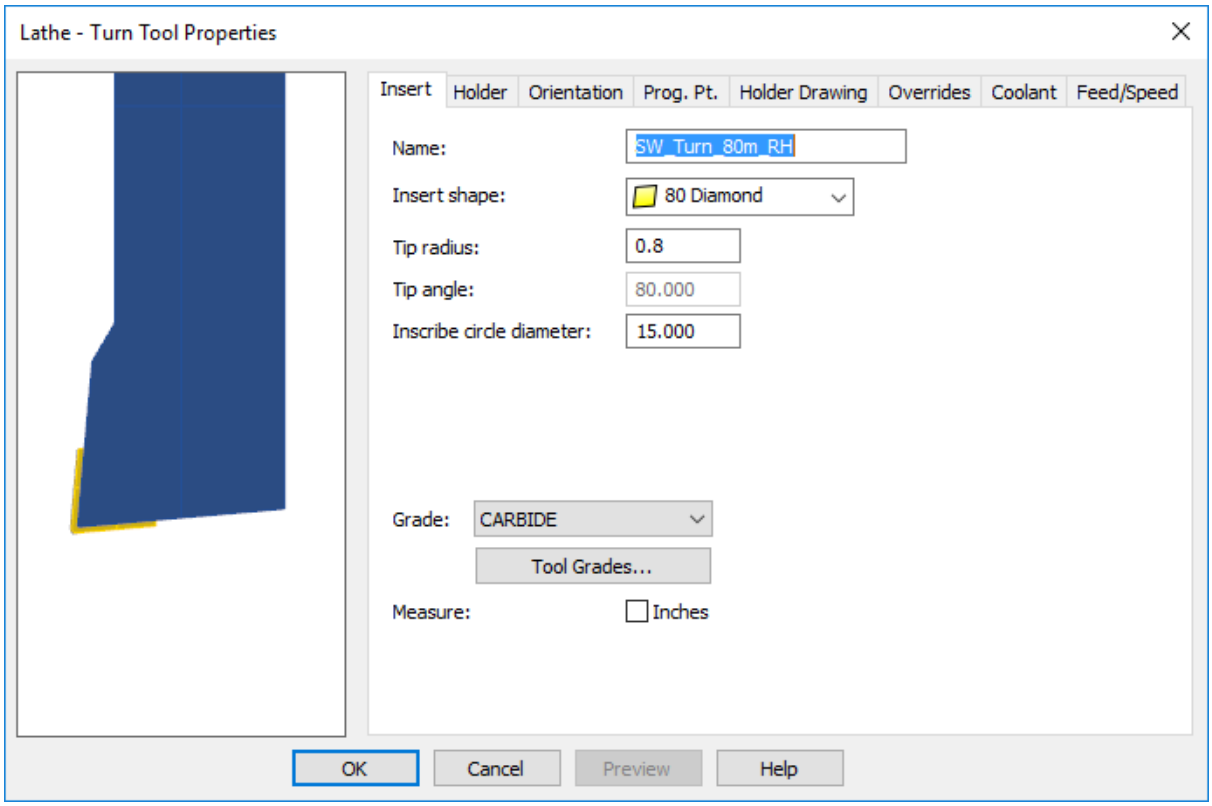


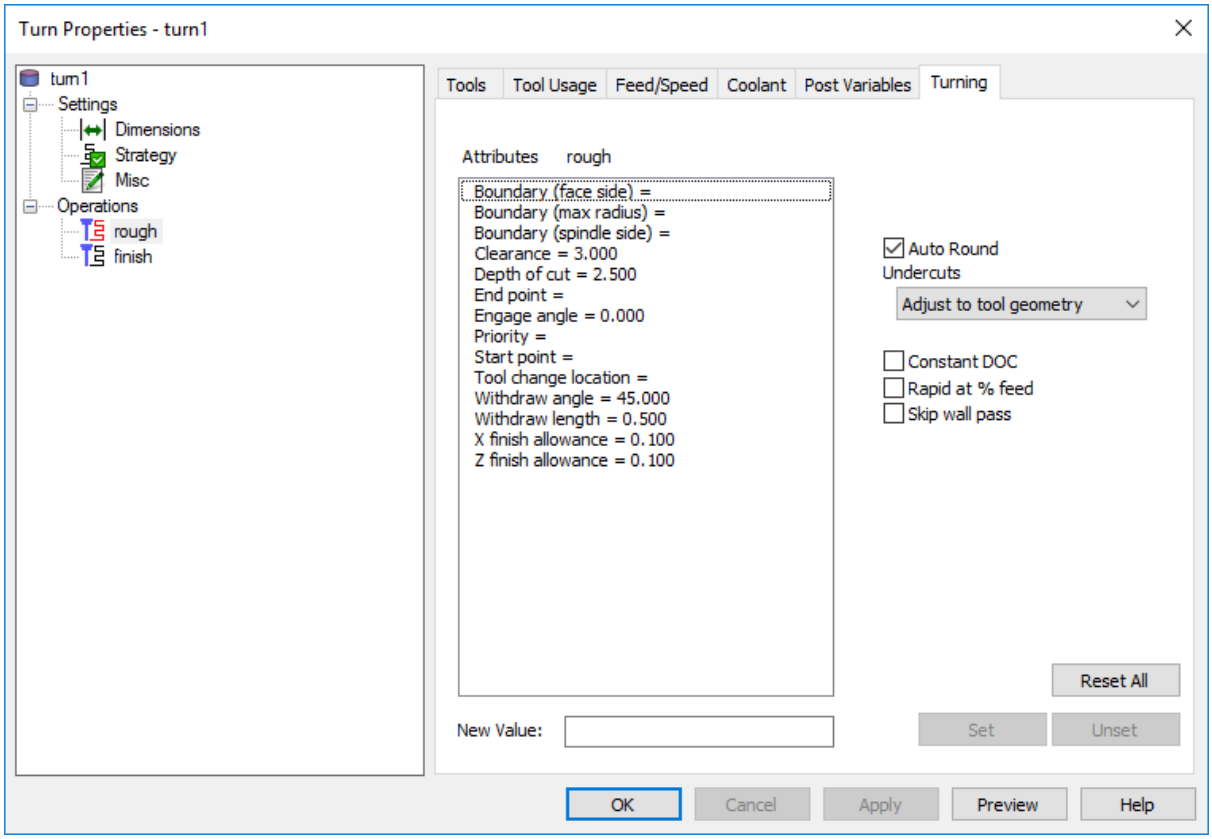
Typical inscribed circle diameters 9.52, 12.7, 15.88,19.05



*The **CNMG** 80 degree diamond tool may be the most used style in the industry today. It is strong and allows for turning and facing with the same tool because of the 5 degree relief on two sides of the insert.*

© Copyright 2016 Autodesk, Inc. All rights reserved



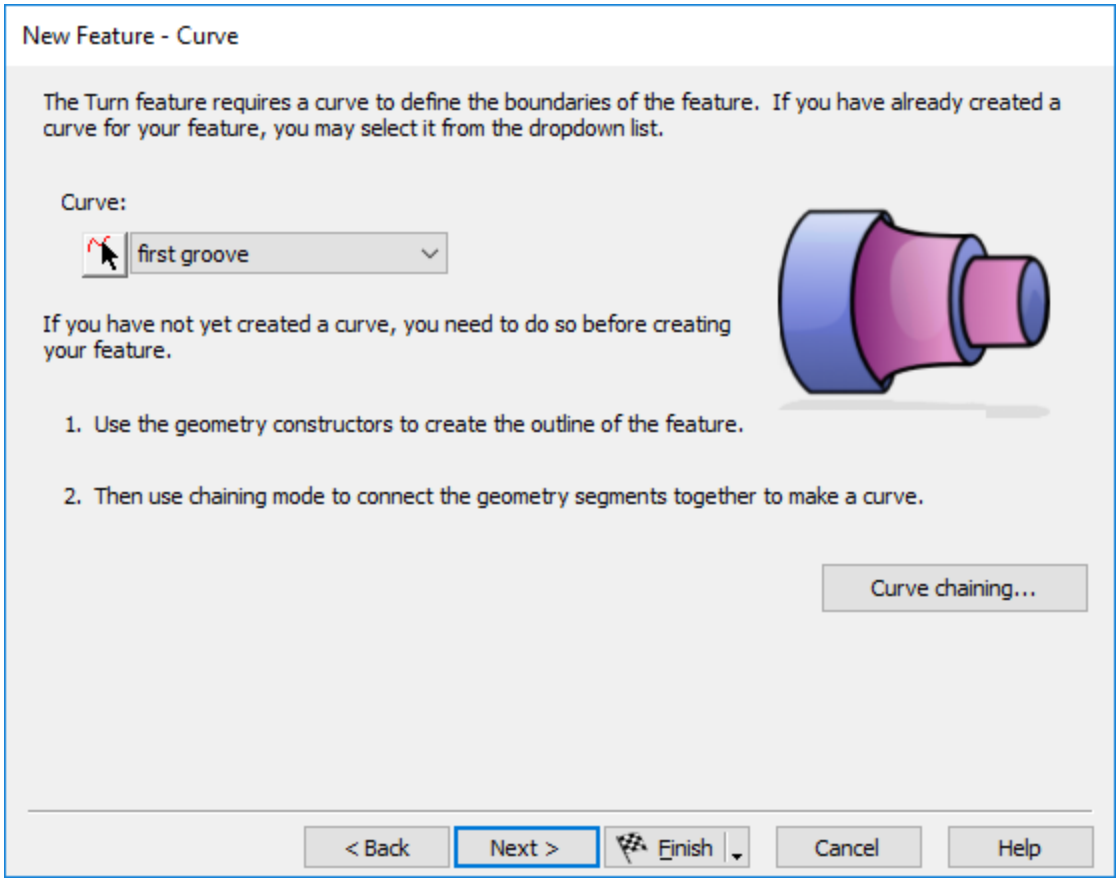


For the next operation we are going to machine the first groove using a turning operation and then create a back turning operation to finish the groove.

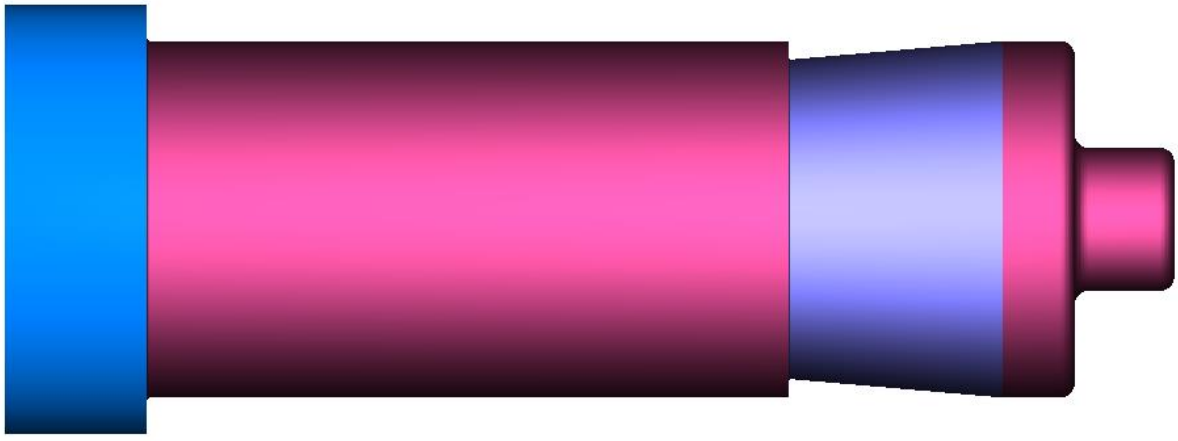
© Copyright 2016 Autodesk, Inc. All rights reserved

15 Create a new feature. Select **Ctrl+R** and select **Turning and** then **Turn.** Select **Next.**

16 Select the **first groove** curve. Then select **Finish**.



In this case by introducing the undercut in the turning operation this tool is unable to even finish one end of the undercut.



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

© Copyright 2016 Autodesk, Inc. All rights reserved

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

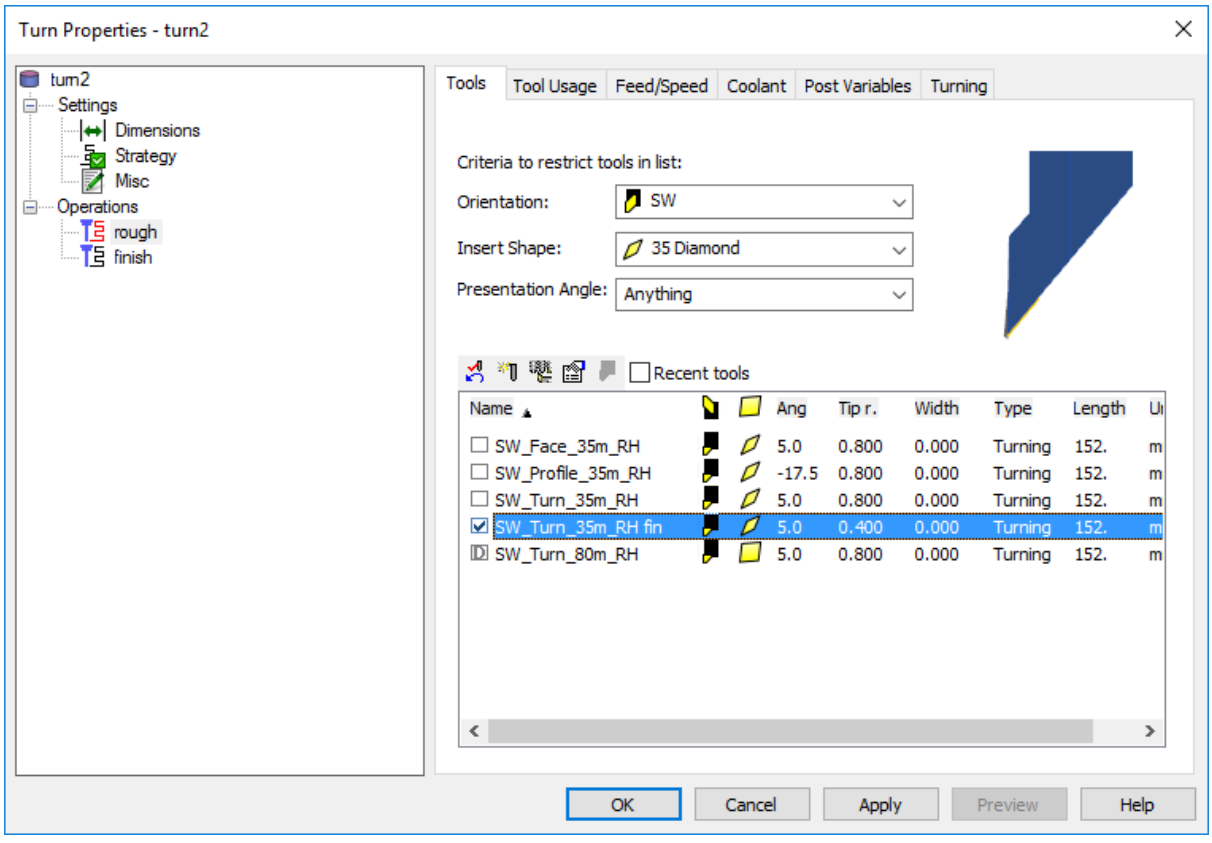
17 First, in the Orientation box the tool is described as **SW** or **South West**. This is the orientation of the current tool and will be the same for the replacement.

18 Drop the menu down in the **Insert Shape** box and select **35 Diamond**.

19 The presentation angle will remain the same.

20 When you Clicked on the **35 Diamond**, **FeatureCAM** shows you the **35 Diamond tools** but this does not select one, you must do that manually.

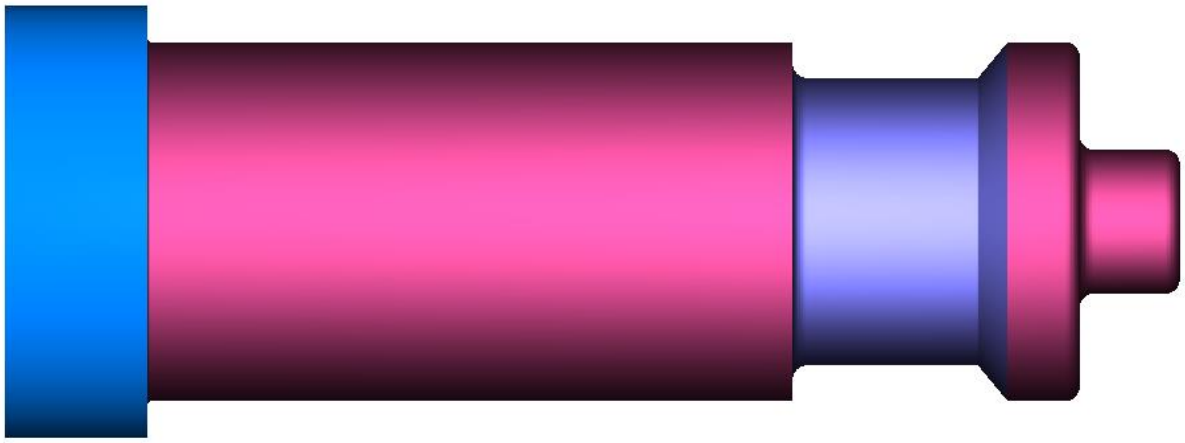
21 Click on each tool description until you find the right tool displayed for the application. You must check the box to the left ☑ of the tool to select the new tool.



*The check in the box means the Default tool has been over ridden. **FeatureCAM** will change the finishing tool as well.*

22 Run the **3D Simulation**. It is readily visible that you can cut much more of the undercut now.

© Copyright 2016 Autodesk, Inc. All rights reserved

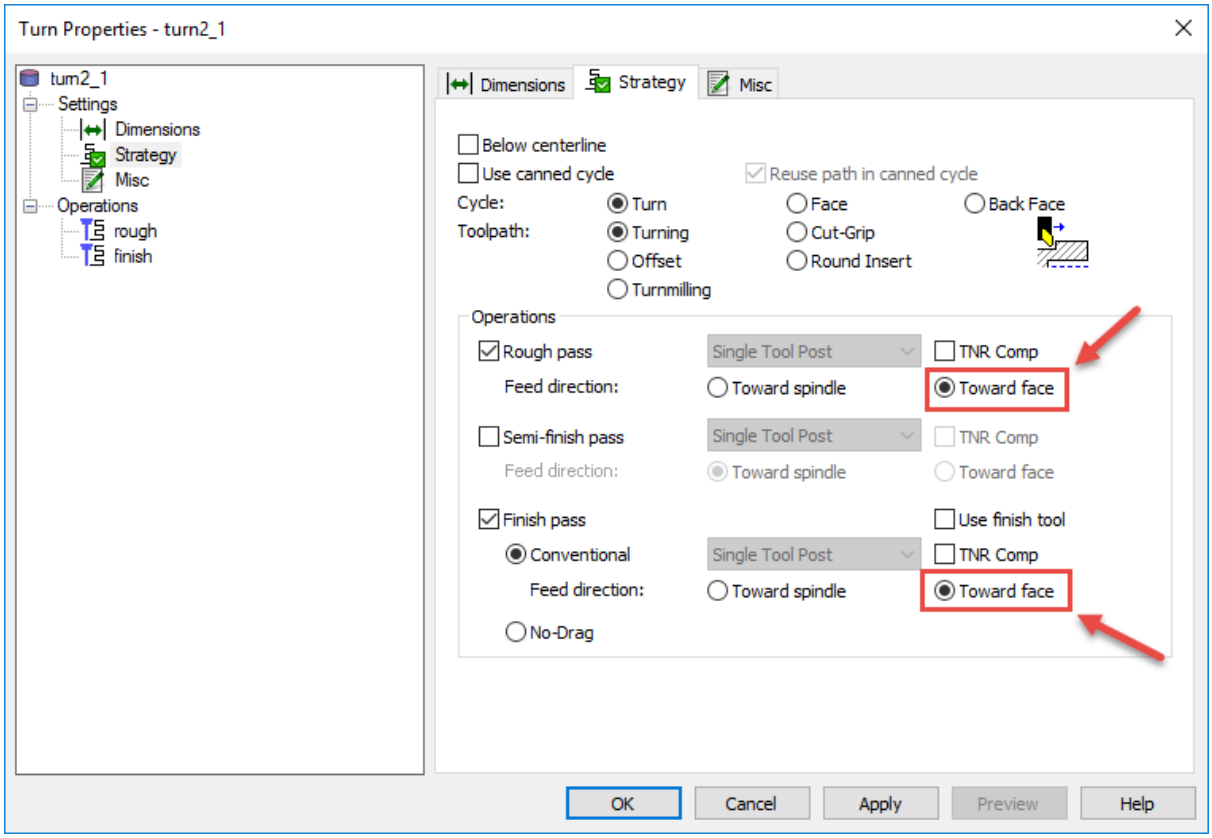


You will notice that when FeatureCAM starts it cuts fresh air at the start.

23 To change this Eject the simulation. Then double click on **turn2** in **PartView**. Select **Rough>Turning**. Change the **Max radius boundary** to **62.5mm**. Then select **Set** and **Apply** to action the new values.

*The material in the corner on the right side of the undercut **needs** to be removed to finish the **OD** of the part.*

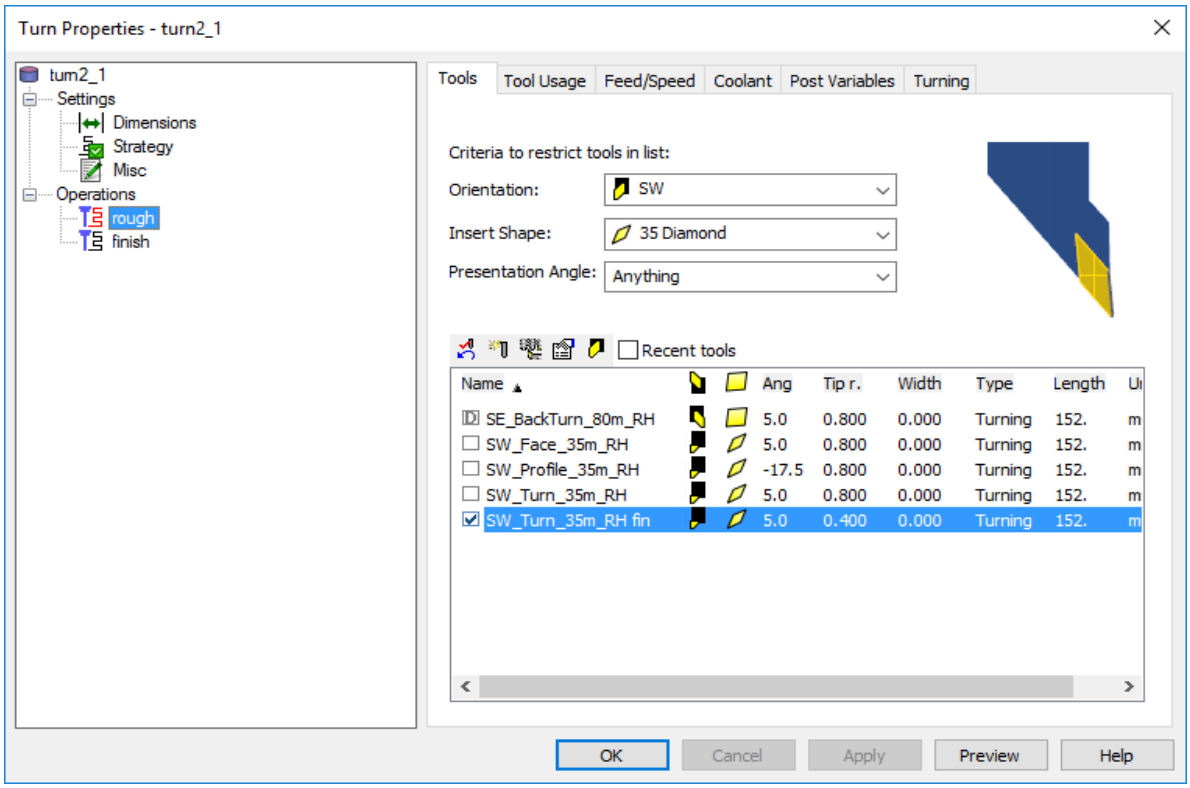
24 Create a copy of the **first groove Feature** in **PartView**. We need to change the **Rough** and **Finish** to cut in a **Toward Face** direction, away from the main spindle.



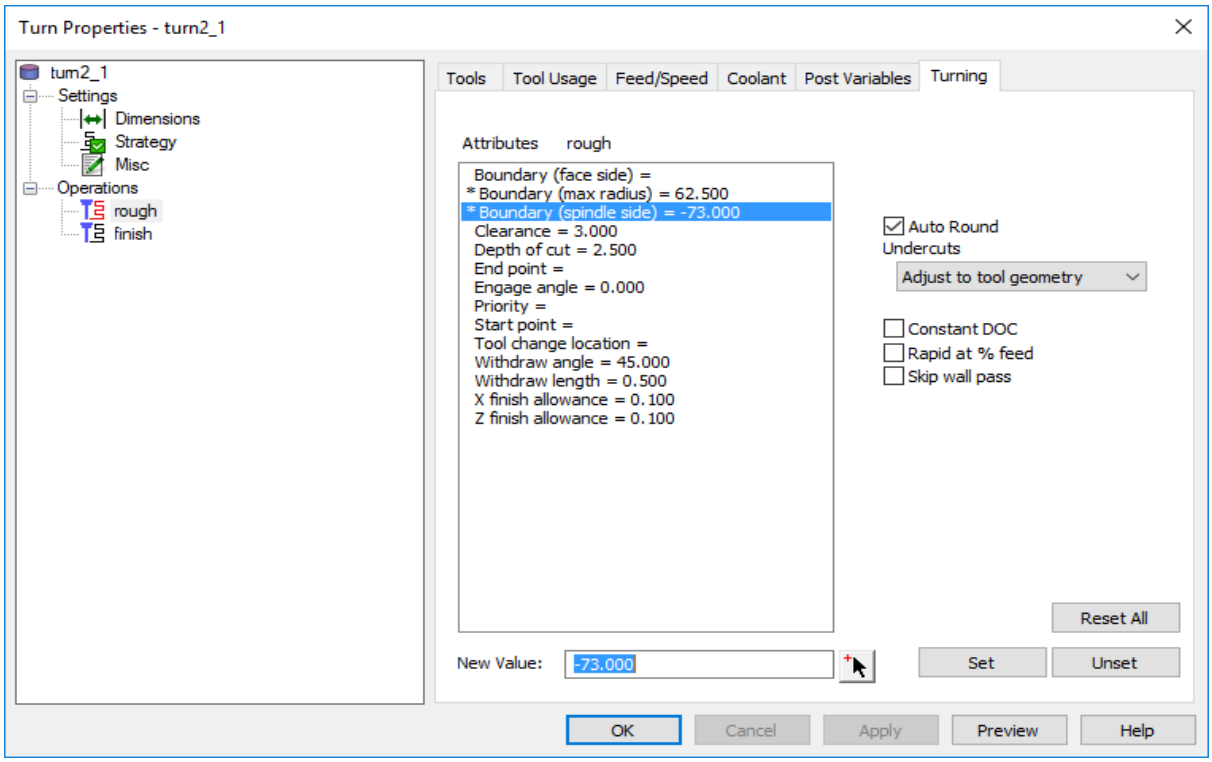
© Copyright 2016 Autodesk, Inc. All rights reserved

By creating a copy of the original Feature the tooling and maximum radius boundary will be maintained.

Double click on the copy of the groove feature in PartView

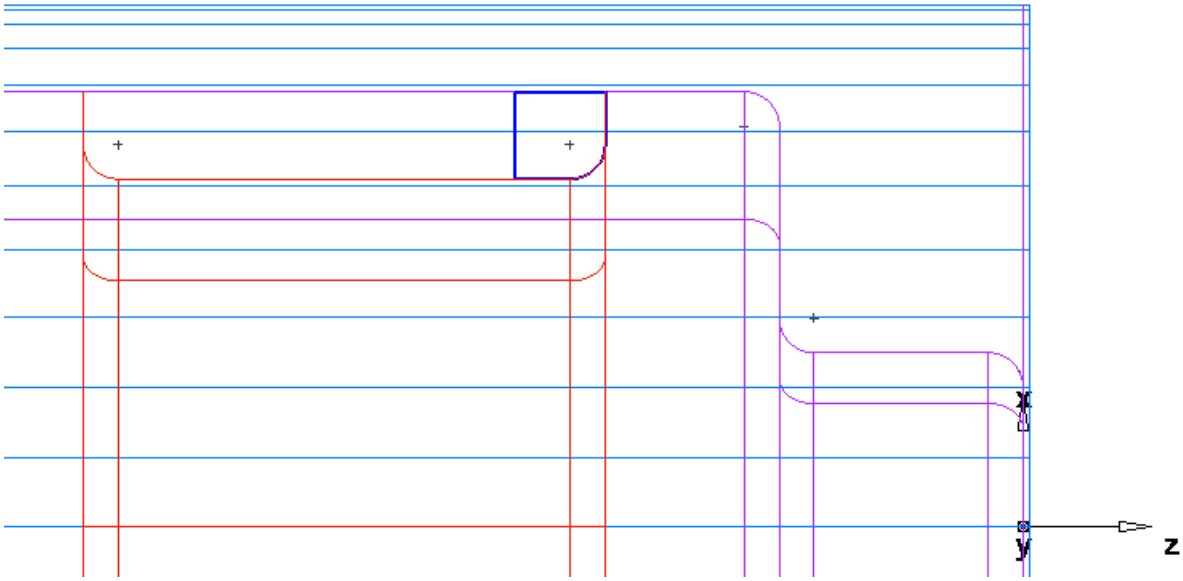


25 Double click on the **Rough** Operation. Then select Turning. The following menu will appear. Select **boundary (Spindle side)** and change this to **-73mm.**



© Copyright 2016 Autodesk, Inc. All rights reserved

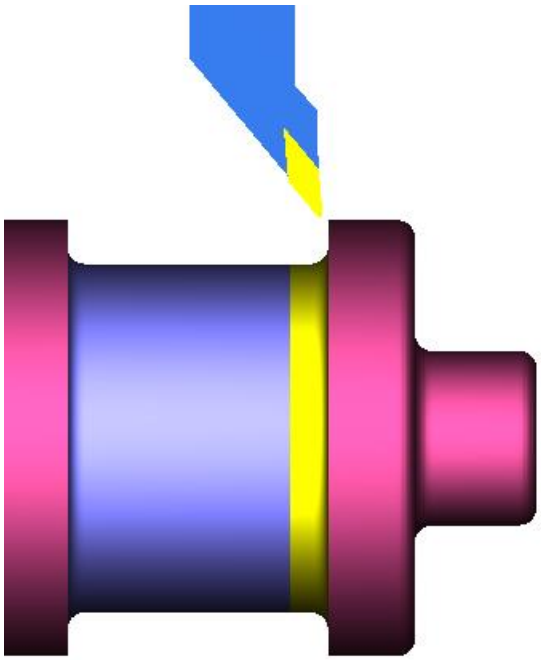
26 Click on **boundary (Spindle side) =** and either Click on the **Arrow** next to the New Value box at the bottom and then click the location on the part in the Graphics Window where you want the finish cut to start or enter a dimension using a negative number measuring from the Left end of the Curve for example **-73mm**. The finishing tool will then start in that location and blend to the previous finish cut.



Boundary (Spindle side)** as shown above. Select **Apply** and **Finish.

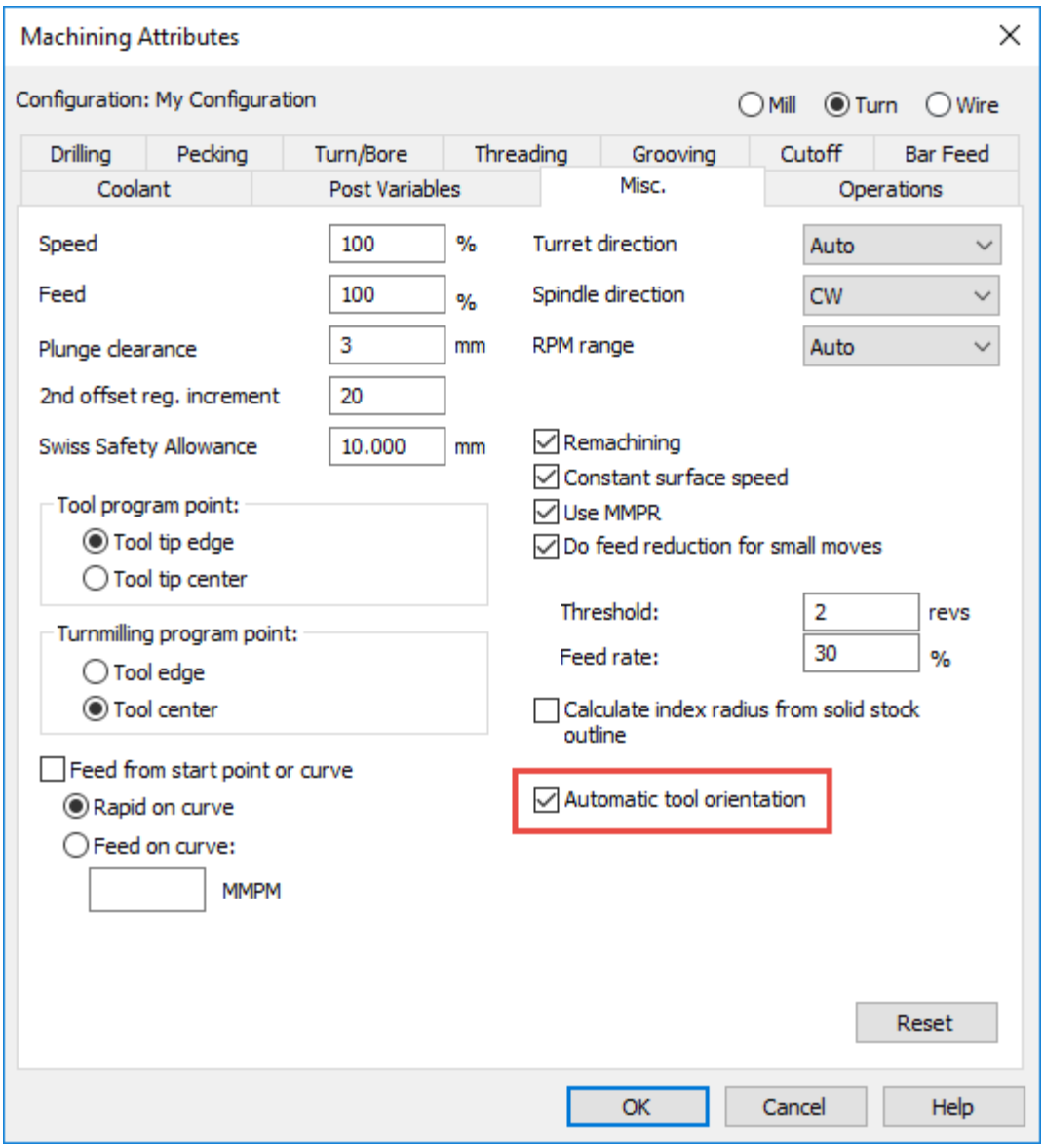
27 You may wish to set the **Right boundary** position. Activate the same menu again and set a value into the **Boundary (face side)** position. This will limit the amount the tool has to travel.

28 Run **3D Simulation** to see results.



© Copyright 2016 Autodesk, Inc. All rights reserved

Special note: - Make sure you have selected the following option in your Machine Configuration.



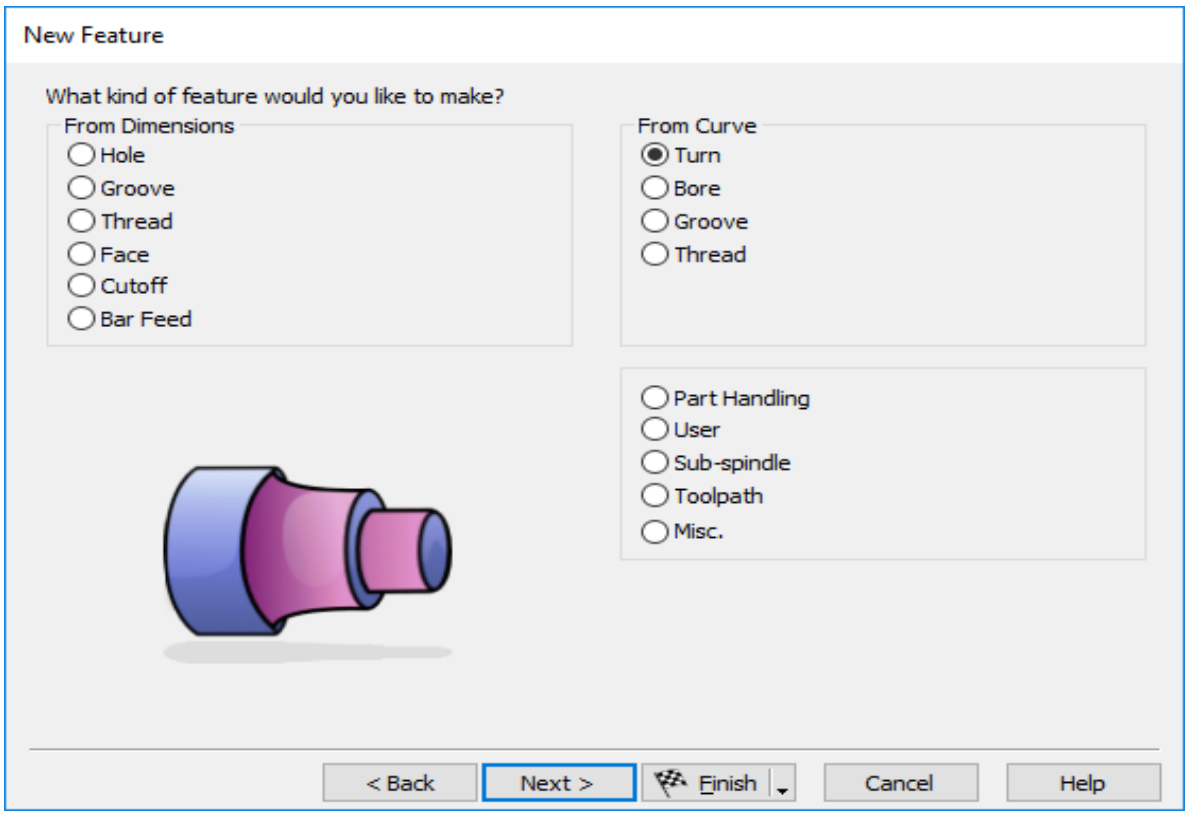
Cut-Grip

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

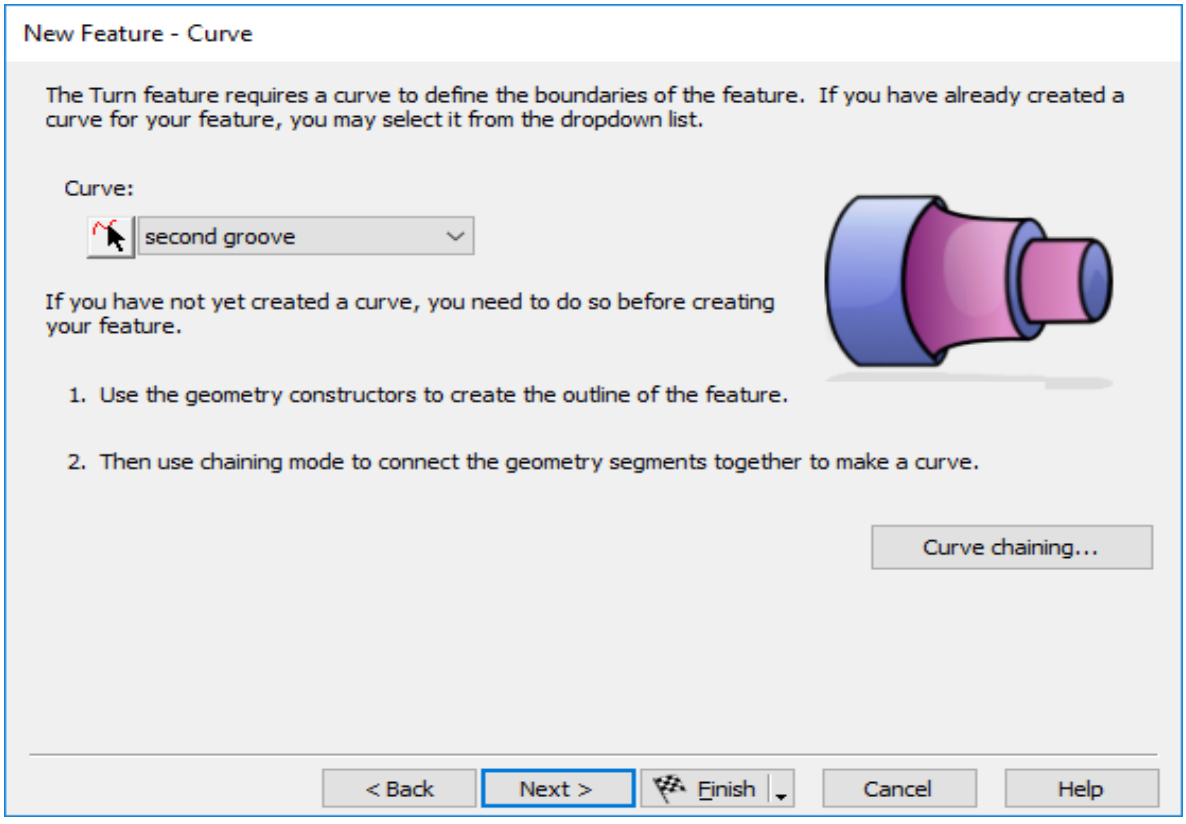
© Copyright 2016 Autodesk, Inc. All rights reserved

Second groove Feature using Cut-Grip

29 The Next feature to consider is **Cut-Grip**. Click on **New Feature** and select **Turn** from Curve.



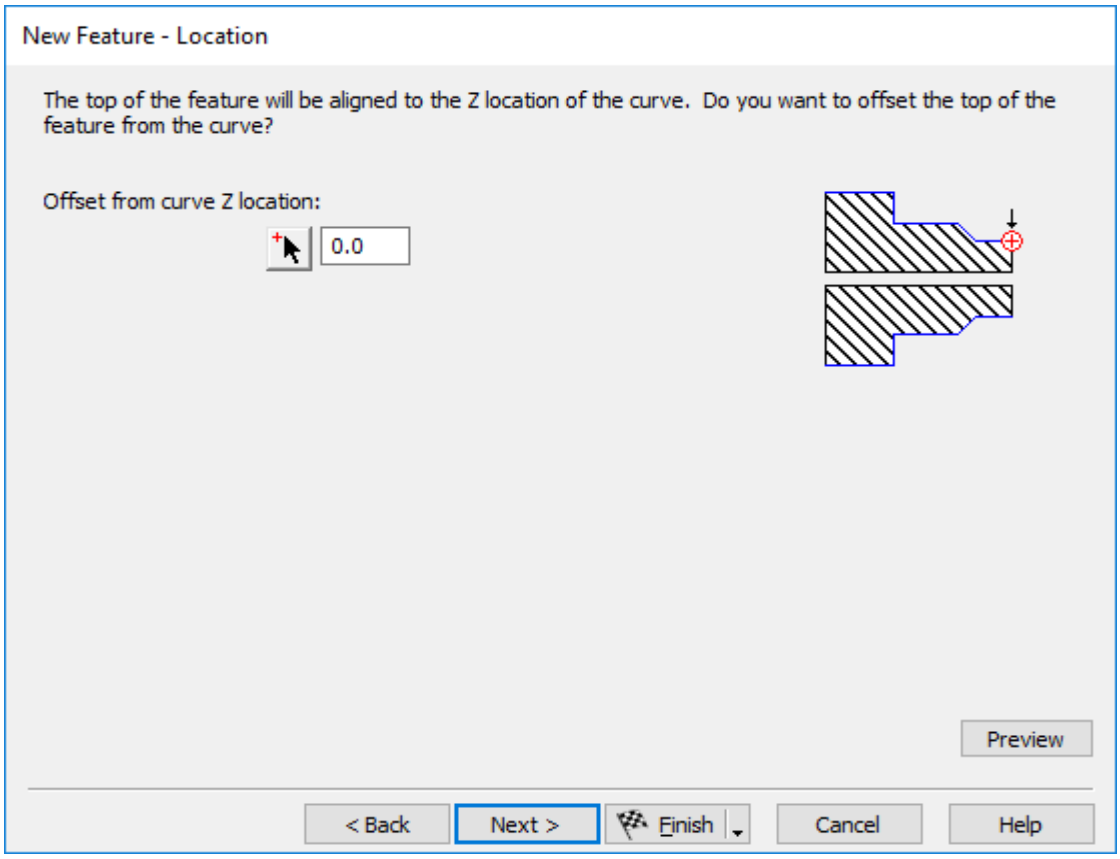
30 Select **Next**.



© Copyright 2016 Autodesk, Inc. All rights reserved

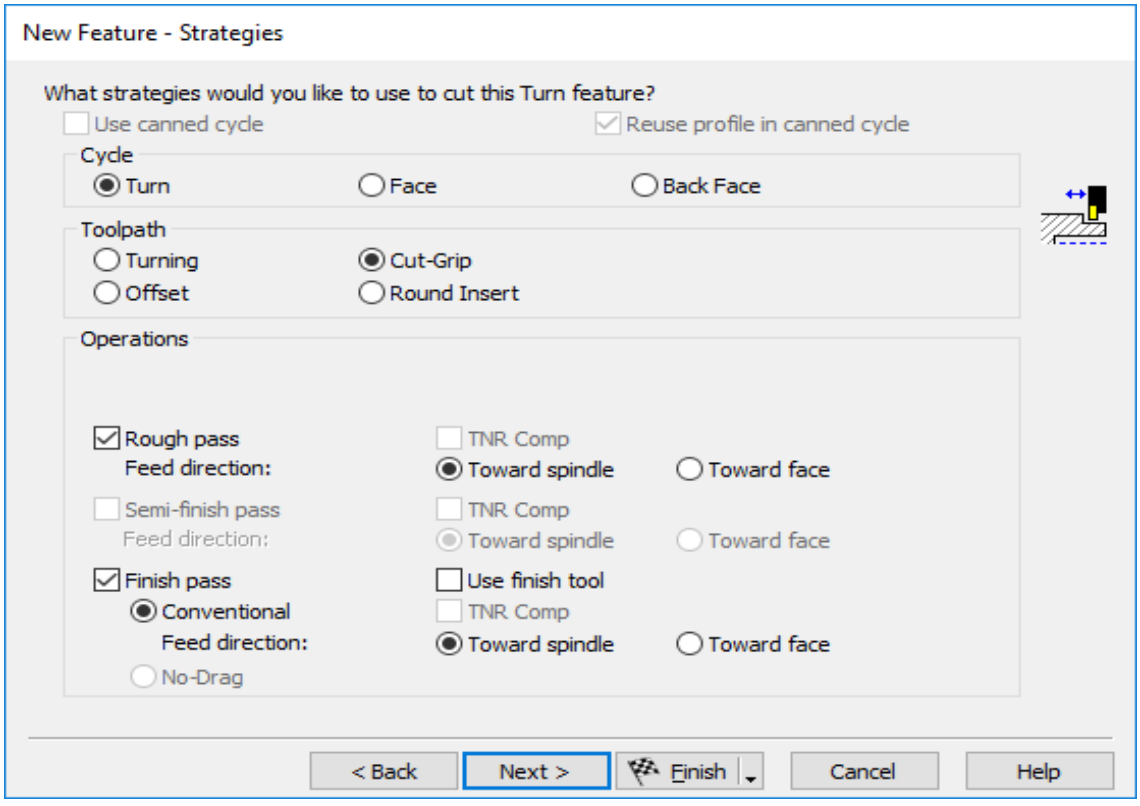
31 Select second groove. Your Curve may have a different name.

32 Select **Next**. Accept the location.



33 Select **Next**.

34 **New Feature Strategies**

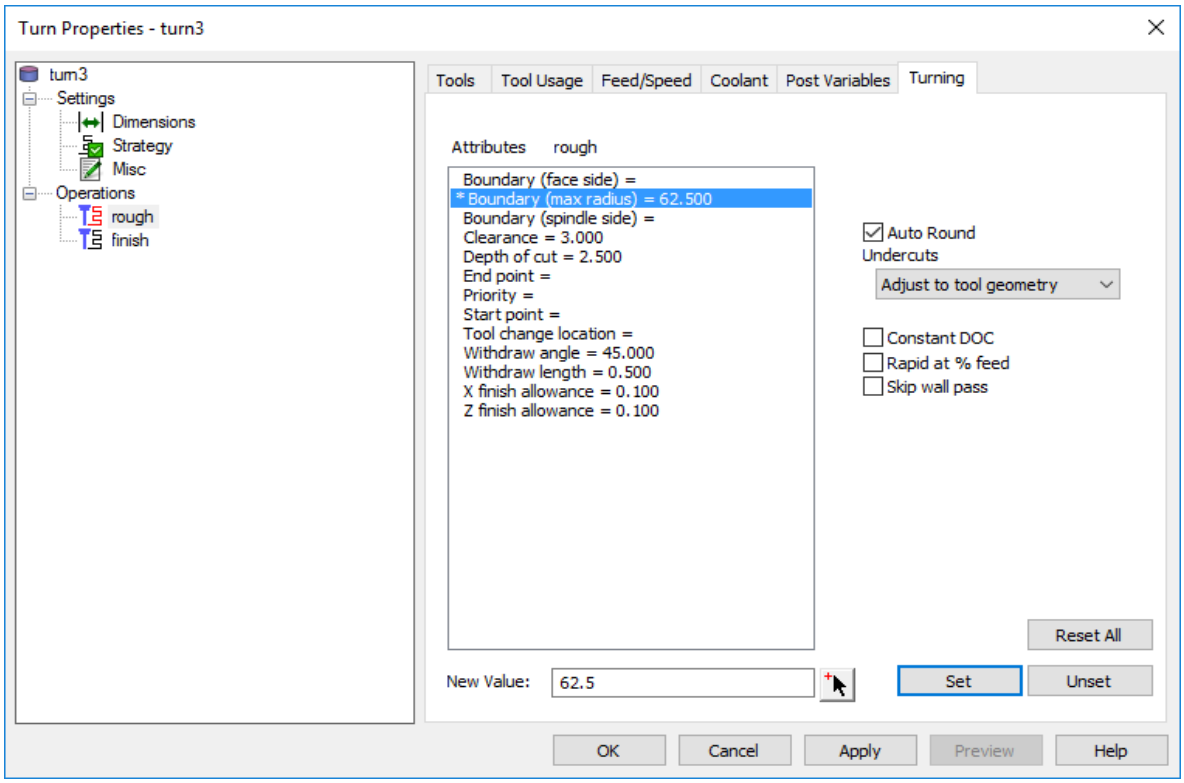


© Copyright 2016 Autodesk, Inc. All rights reserved

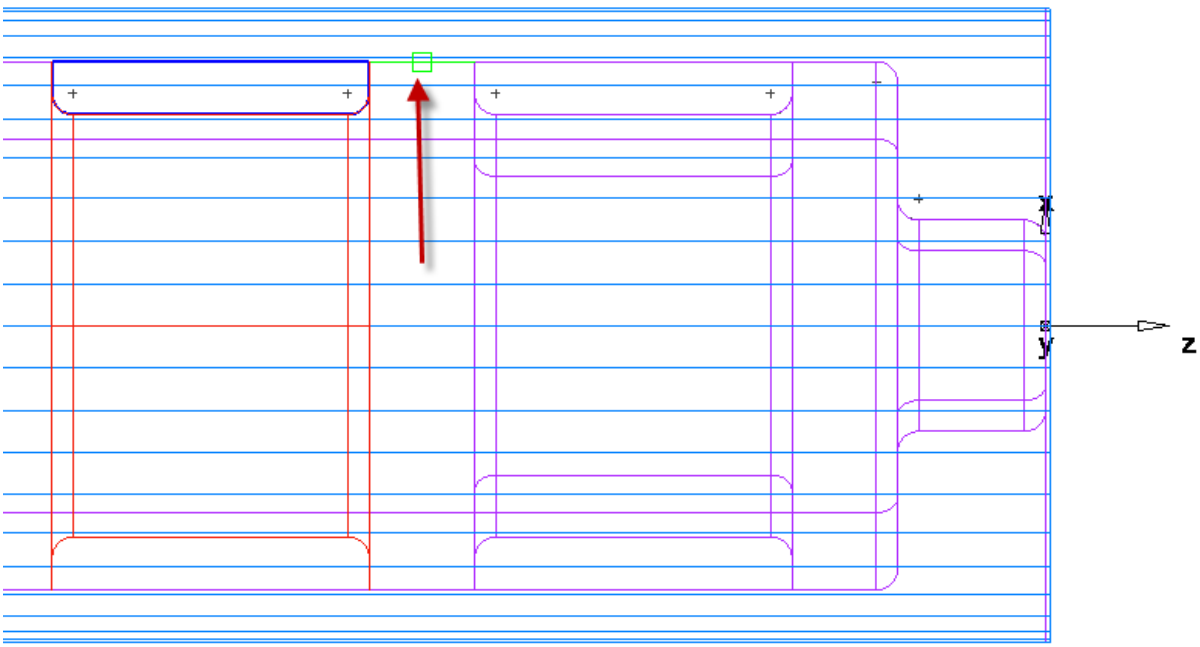
35 Select **Cut-Grip** then select **Finish**.

36 Select **Rough and Turning**

37 Select **Maximum Radius Boundary** and **pick X location** or enter **62.5mm** into the New value field.

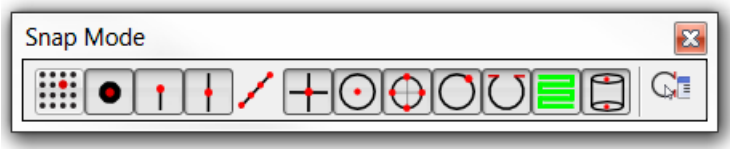


38 Pick Boundary edge.



Make sure you're snapping Toolbar is active and snap to object is highlighted.

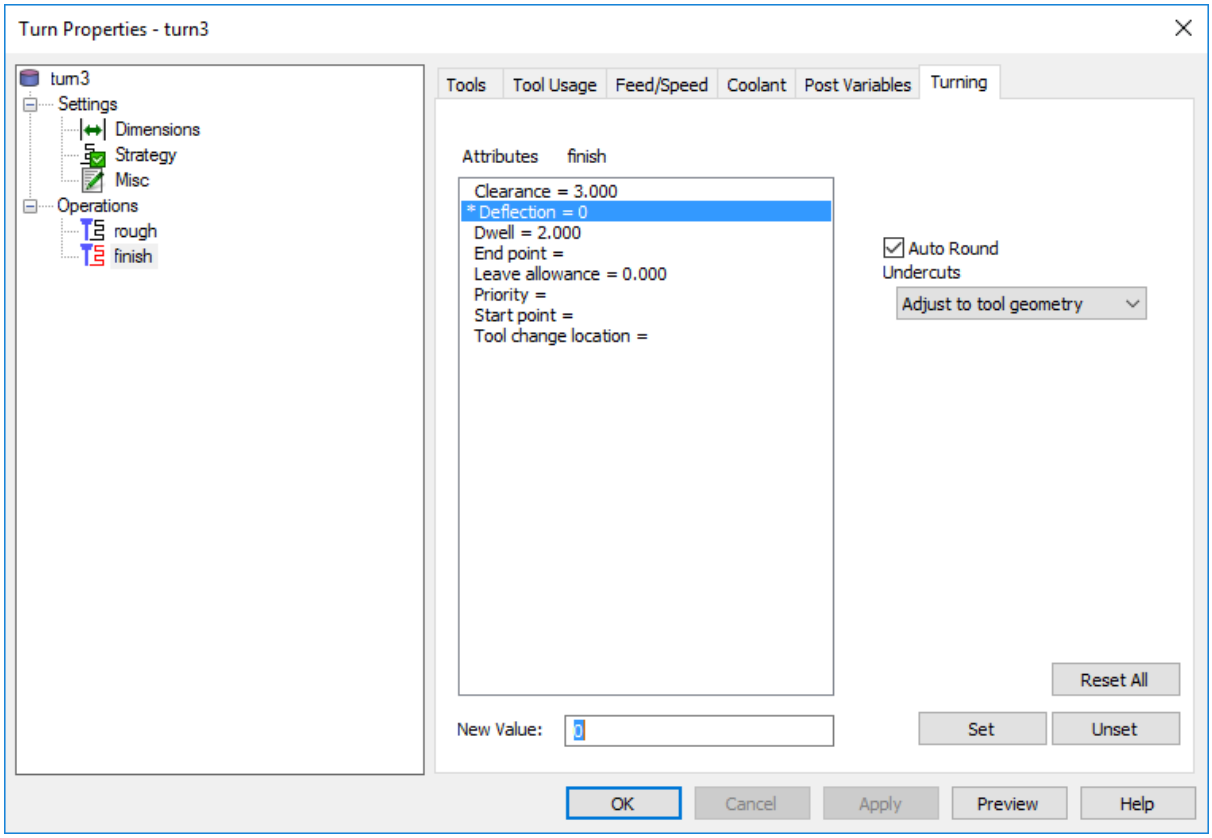
© Copyright 2016 Autodesk, Inc. All rights reserved



39 The Boundary is now adjusted to **62.5mm** Radius **125mm** Diameter.

40 Shown below is the Boundary Adjusted.

*The **Cut-Grip** groove is finished. You will need to adjust the deflection, click on the finish toolpath and adjust the value to **0** (See image on the next page) remember to select **set** to update the value.*



41 **Cut-Grip** is now completed

© Copyright 2016 Autodesk, Inc. All rights reserved

Groove from Curve

The third and last groove will be roughed and finished using a grooving strategy. This is probably one of the most commonly used strategies for removing material from a groove feature.

42 **Eject** the **Simulation**.

43 Click on **New Feature** and select **Groove** from Curve. Select **Next**.

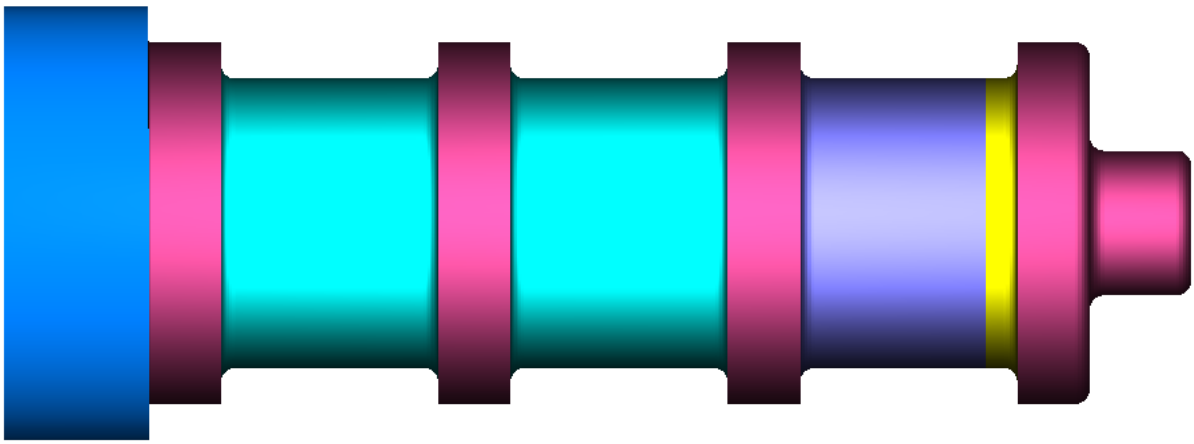
44 Select the **Third Groove** Curve.

45 On the New Feature – Dimension menu make sure you select **X-axis.**

46 Select **Finish**. Please select the **strategy** page.

*The **Negative** option allows the tool to start from **Right** to **left**. **Depth first** allows the tool to plunge to full depth allowing for any corner rads and finishing allowances. **Plunge centre first** allows the tool to start from the middle and work out. **Positive** allows the tool to start from **Left** to **right**. **Width first** splits the cuts into two, still allowing for a finish allowance.*

The part is now complete see image on the next page.



Face Boring

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

Offset Boring

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

Boring below Centreline

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

© Copyright 2016 Autodesk, Inc. All rights reserved

Boring with Canned Cycles

Canned cycles can be generated in the NC code for nearly every turned feature. To generate these macros, your post processor must support them, and you must turn this function on for the post and for some features you must also activate the canned cycles on the feature level.

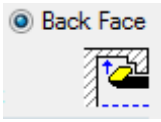
Reuse path in canned cycle outputs the path geometry only once for both roughing and finishing.

Tool Nose Radius Compensation

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

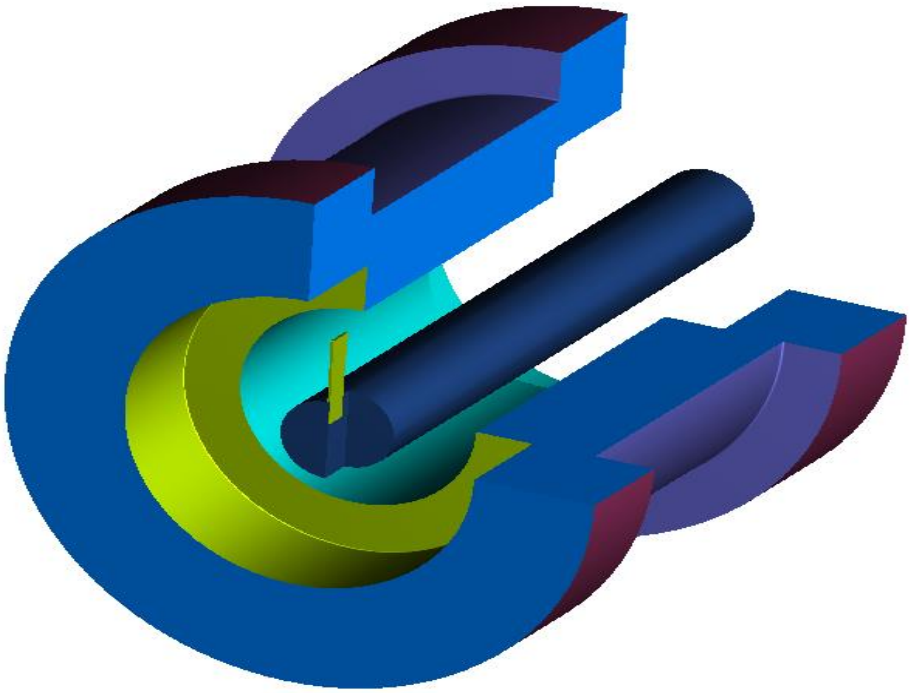
Back Face Boring

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



Back Boring

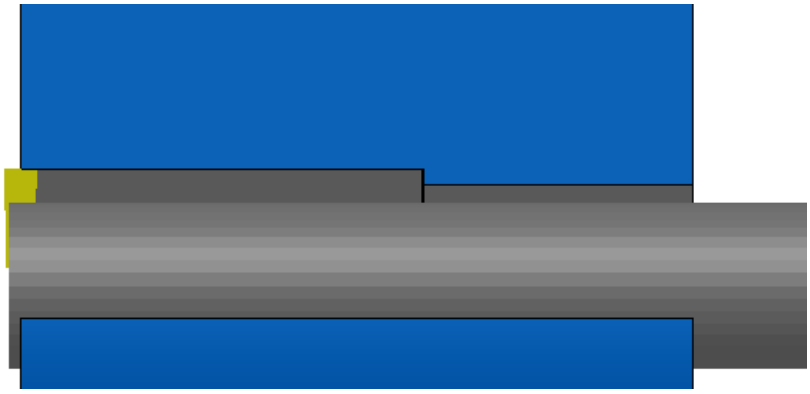
Back Boring cuts parallel to the Z and is performed using a positive direction in Strategies.



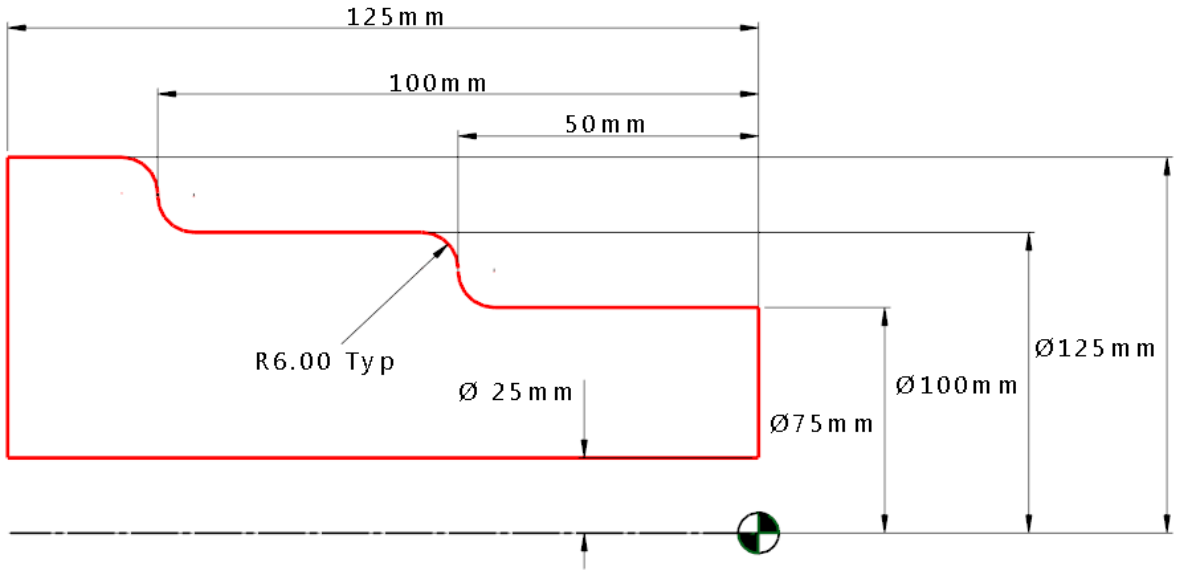
© Copyright 2016 Autodesk, Inc. All rights reserved

Boring with a groove tool

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

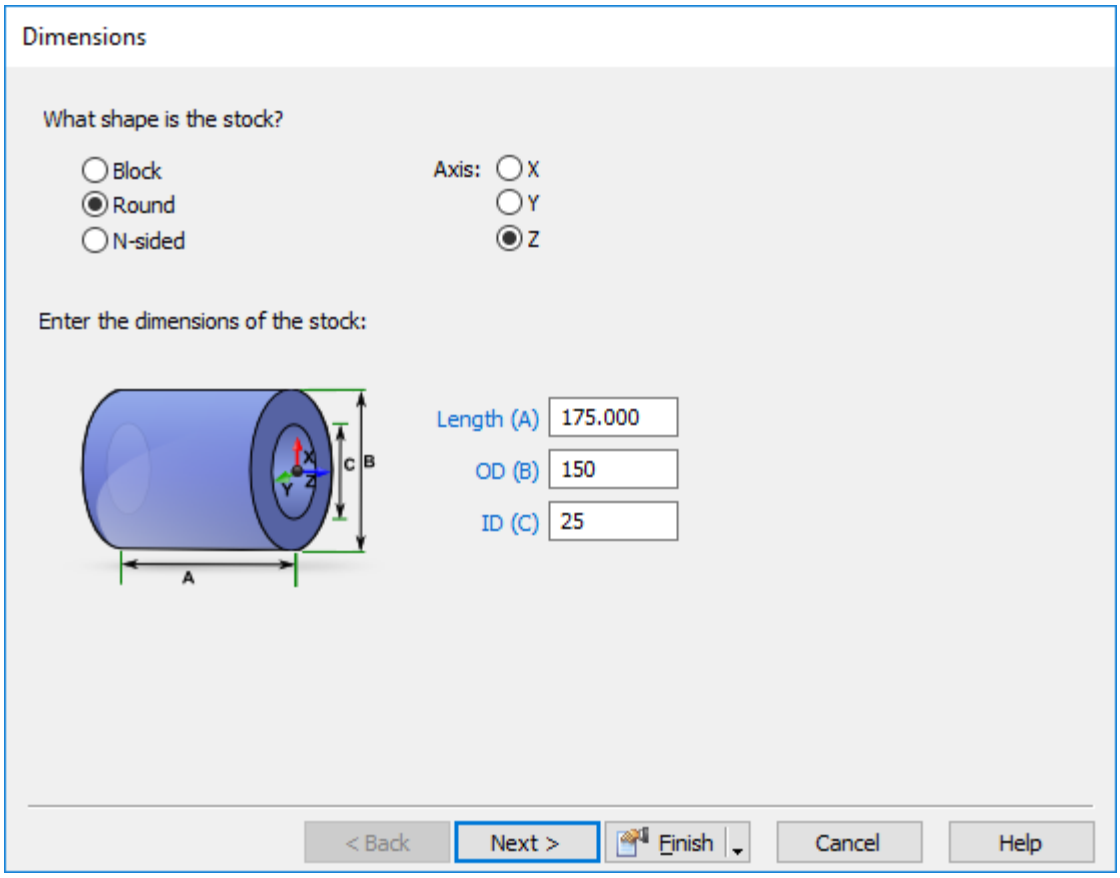


Turning Exercise 3



1. Open a New **Turn Part Document**.
2. Round Stock. OD 150mm, Length 175mm, ID 25mm Z Stock 1mm

© Copyright 2016 Autodesk, Inc. All rights reserved



3 Select **Next.**

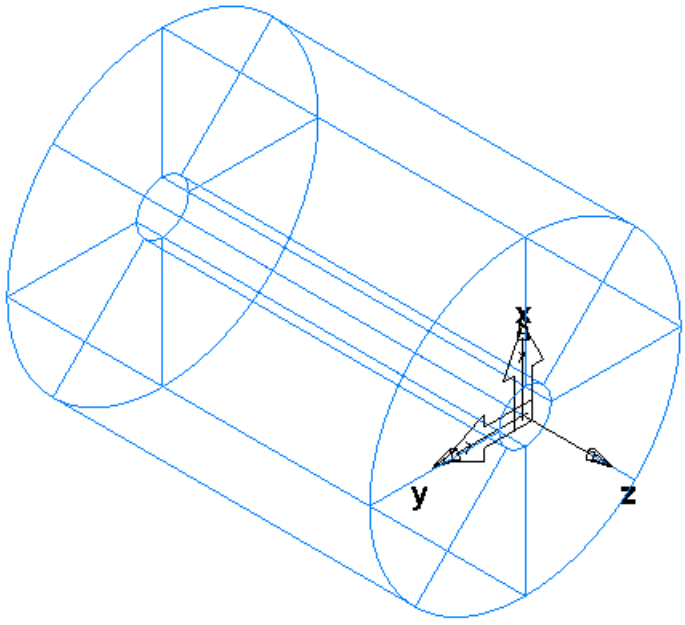
4 Select **Stainless-304L Material**. Select **Next**.

5 No Multi-Axis. Select **Next twice**.

6 Setup, Use current location. Select Next.

7 Main Spindle. Select **Next**.

8 Select **Finish**.



© Copyright 2016 Autodesk, Inc. All rights reserved

Exercise 3 in detail

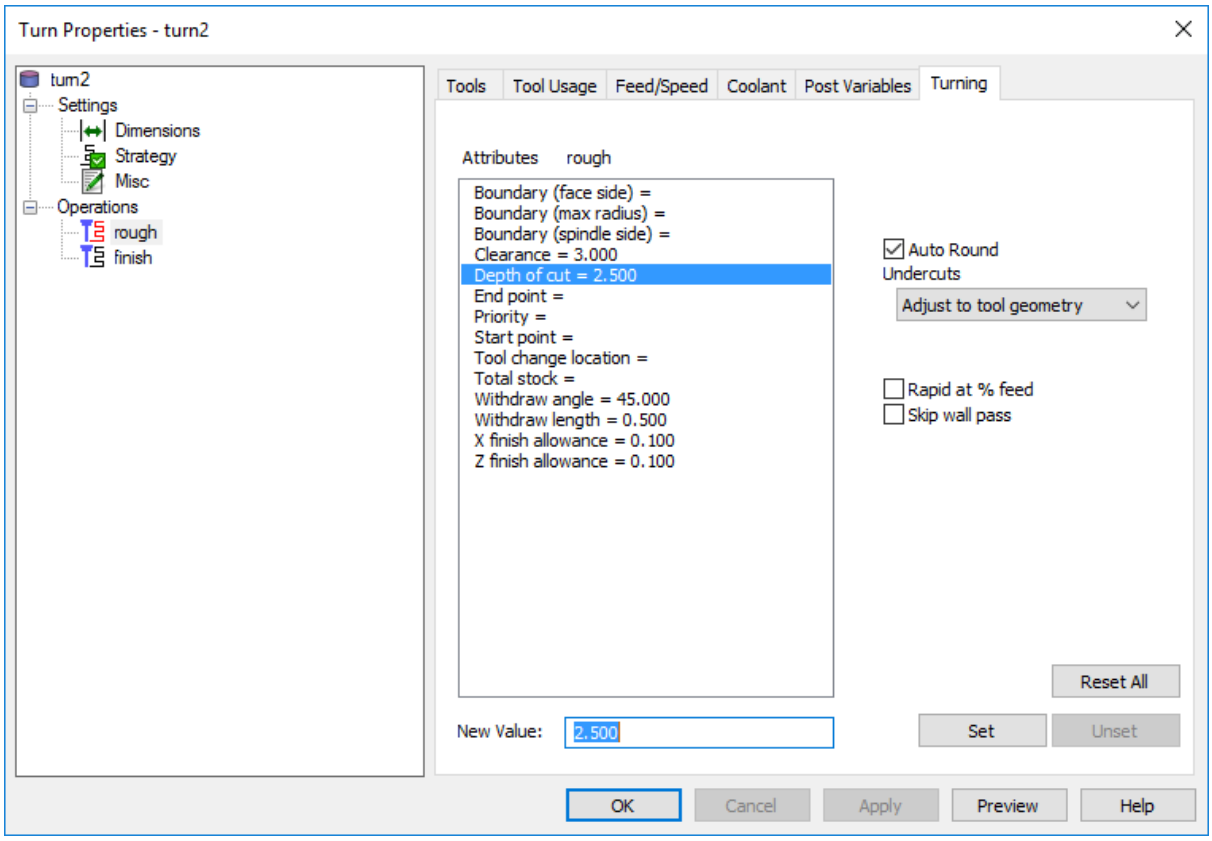
9 Draw the part shown on the previous page using geometry constructors and then **chain curves**.

Remember that the geometry only needs to be drawn above the centreline

10 Select **New Feature** or **CTRL + R** and select **Face.** Select **Finish.**

11 Select **New Feature** or **CTRL + R** and select **Turn from curve.**

12 Select **Next**, Select the **curve** then select **Finish.**



Turning Properties – Rough - Turning

- The cuts are straight but move up the face to remove the scallops left by the angle of the tool.
- **Depth of cut** can be specified as a default in the attributes or click on the feature to open the properties. Then click on rough>turning and select Depth of Cut. It defaults to **5mm** but may be changed by clicking on **Depth of Cut>New Value** at the bottom and entering the new depth and then click **Set**.
- **Constant Depth of Cut (DOC)** The roughing operation leaves the specified finish allowance on all diameters but sometimes that makes some passes come between the 5mm depths. FeatureCAM must cut each diameter to satisfy the leave allowance but the depth of cut might be less than 5mm. It makes this pass but starts at the end of the part when it only needs to cut a short distance on the affected diameter. It may also take extra full length cuts.

© Copyright 2016 Autodesk, Inc. All rights reserved

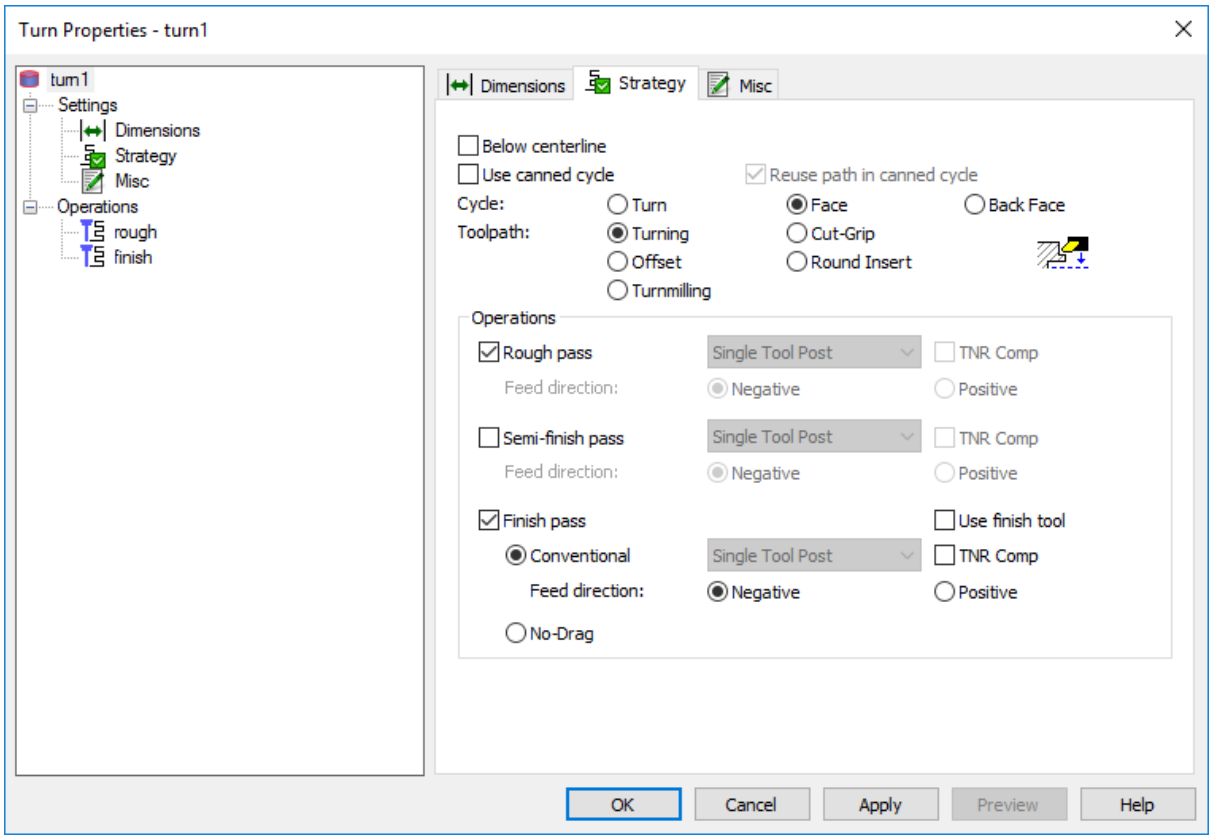
- **Constant depth of cut** always cuts at 5mm for the full length of the part and only cuts the affected diameters surface to the finish allowance when necessary and is a shorter cut. This will reduce the number of full length passes reducing cycle time.
- **Skip Wall Pass** moves the tool up each face that it cuts to smooth the face and can be turned off by checking the Skip Wall Pass box in the Turn Tab located on the right side.
- **Clearance** is the distance from the part that the tool rapids too off the material and then goes into feed rate.
- **End Point** is a specified location (vector) that the tool can be directed to retract to when operation is completed.
- **Max Radius Boundary is used to restrict the tool from starting the cuts from the stock** boundary in the event the material has already been removed.
- **Priority** can be used to set the order in which feature operations are executed. You should use this sparingly because it can affect the ordering that FeatureCAM does automatically.
- **Start Point** allows you to select a location for the tool to rapid too other than the normal clearance location it then rapids to the clearance location from the Start Point. Another selection associated with start point is feed from Start; the tool then feeds to the clearance location. An example is when supporting with a tail stock, you might choose to rapid away from the tailstock and feed to the clearance to control its approach.
- **Withdraw Angle** controls the **angle** at which the tool retracts at the end of the cut.
- **Withdraw Length** controls the **length** that the tool retracts at the end of the cut.
- **X Finish Allowance** is the amount of material left on the **diameter** by the rough tool to be removed by the finish tool.
- **Z Finish Allowance** is the amount of material left on the **face** by the rough tool to be removed by the finish tool.
- **Auto Round** simply rolls the tool around any sharp corner to remove the sharp edge. If there is any radius at the corner then Auto Round is not active.
- **Post Variables** can pass information directly to the NC code from the feature but the post processor must be setup for variables.

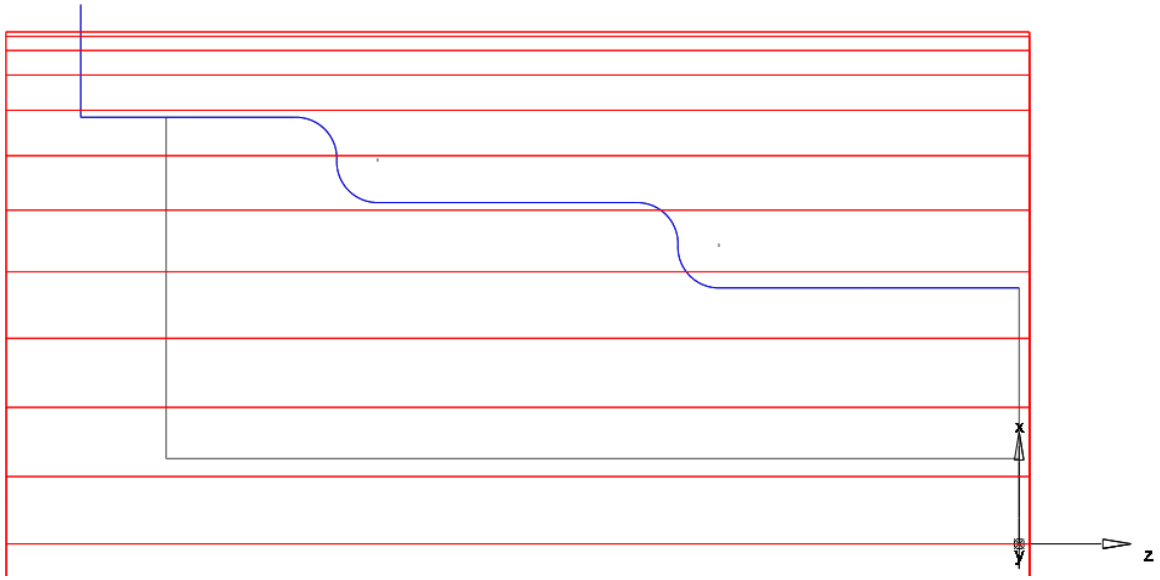
Face Turning

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

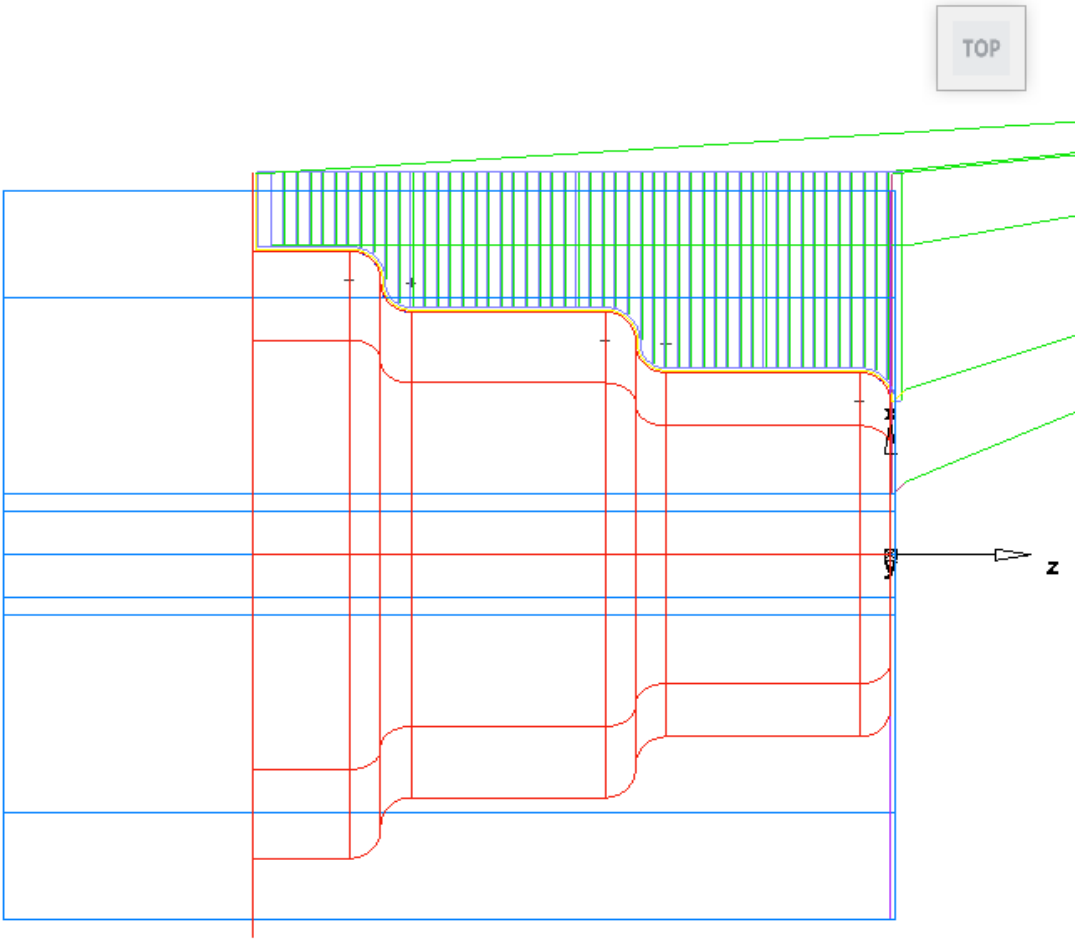
You may need to extend the geometry past the largest diameter above the stock and create your Curve from this. See example below.

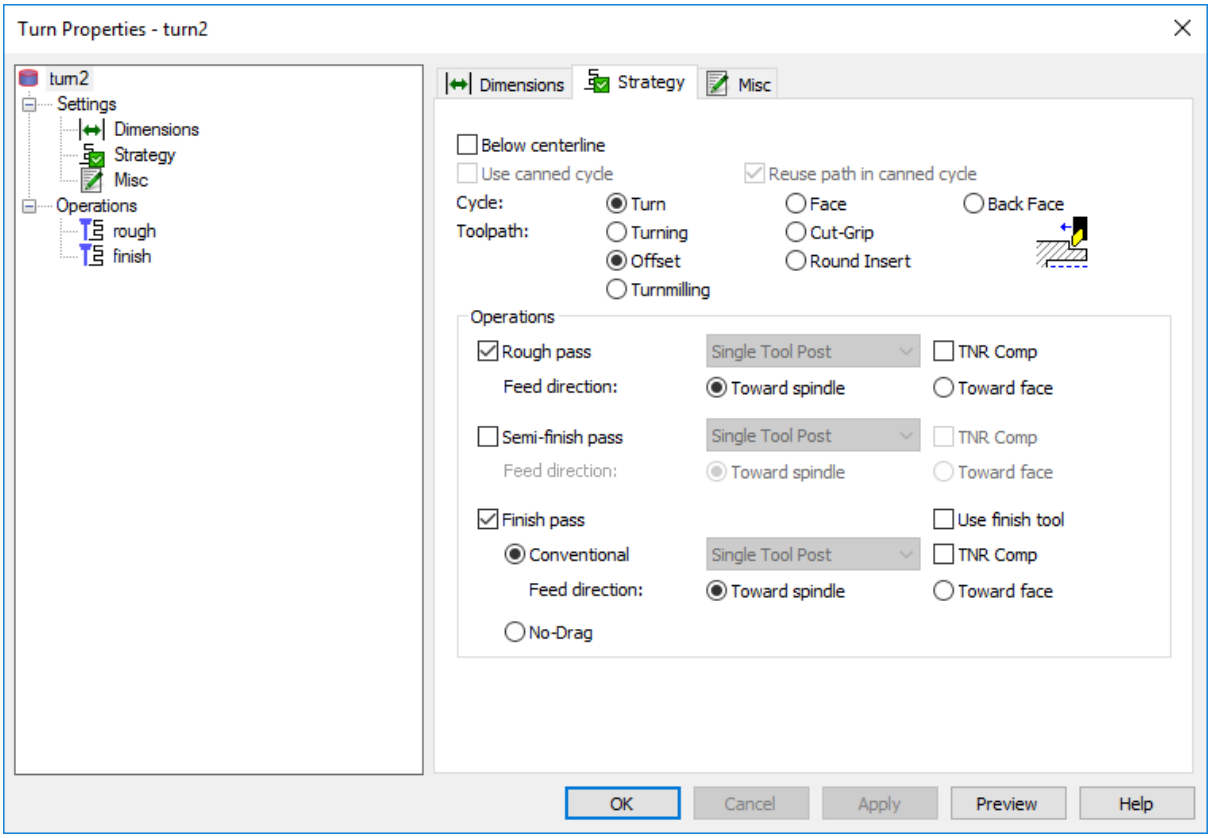
© Copyright 2016 Autodesk, Inc. All rights reserved





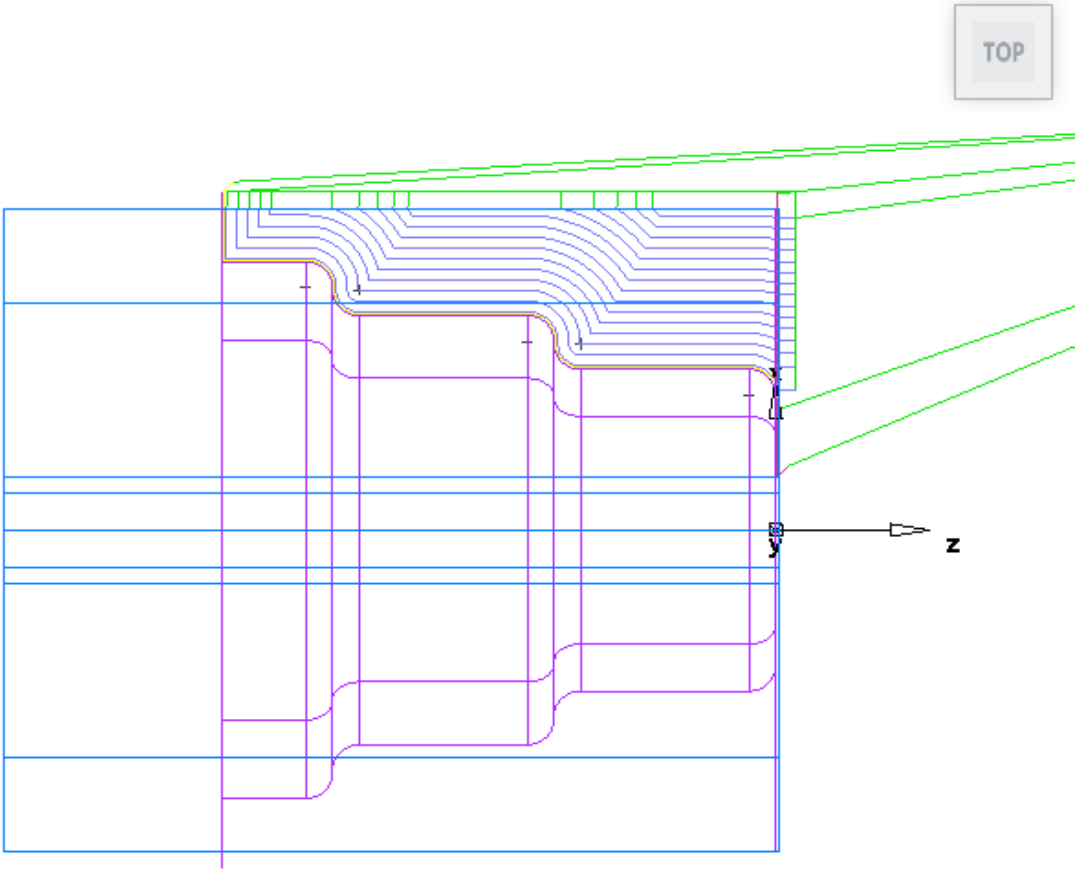
© Copyright 2016 Autodesk, Inc. All rights reserved





© Copyright 2016 Autodesk, Inc. All rights reserved

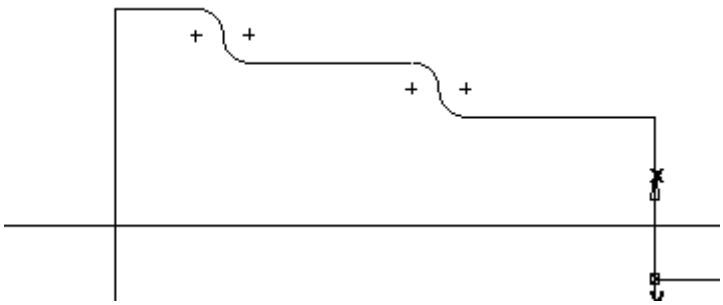
Offset Turning *uses the shape of the curve and offsets it to the end of the part for the tool to follow. It begins to cut the shape of the curve on the first pass and cuts parallel until finished. Good for castings.*



Stock Curve

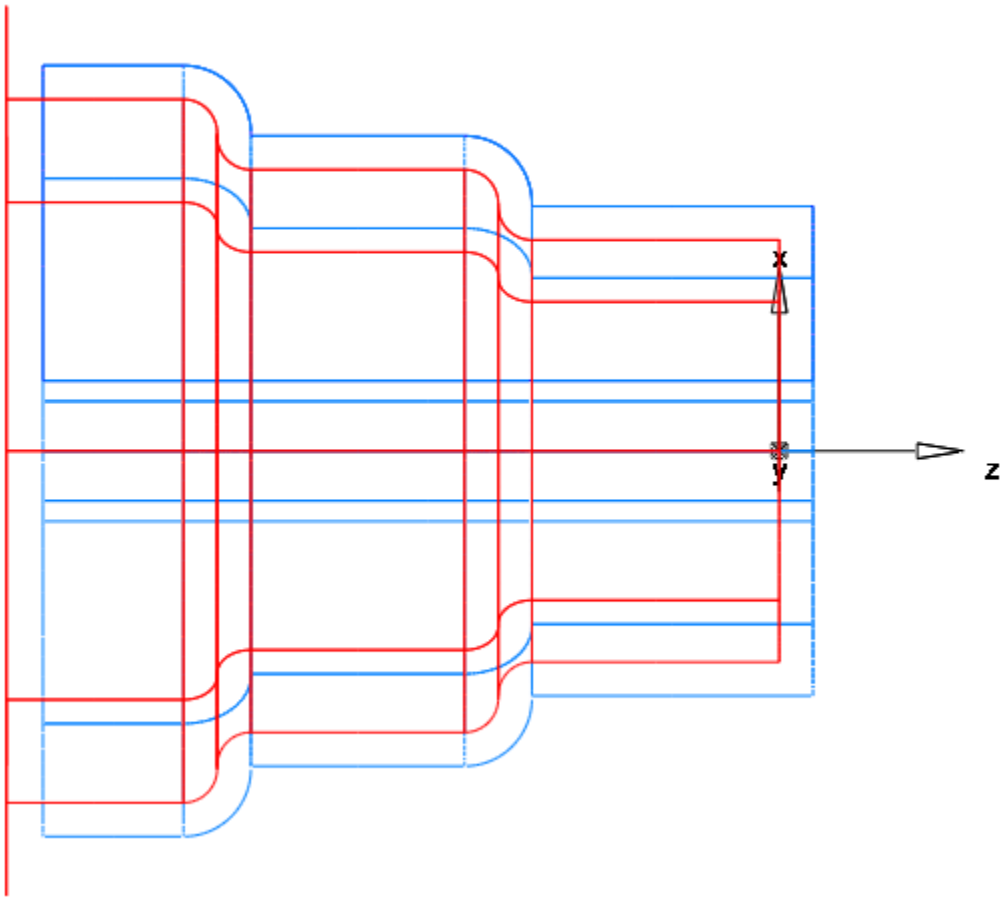
A stock curve can be used to represent a specific shape of the stock rather than bar stock. An example is when a forging or casting is used for the stock. The shape of the material is near the finished part shape with enough extra material to be turned down to the finish part size and shape.

To draw a stock shape and then chain a curve, hide all features, curves and stock. Draw a horizontal line at 25mm diameter

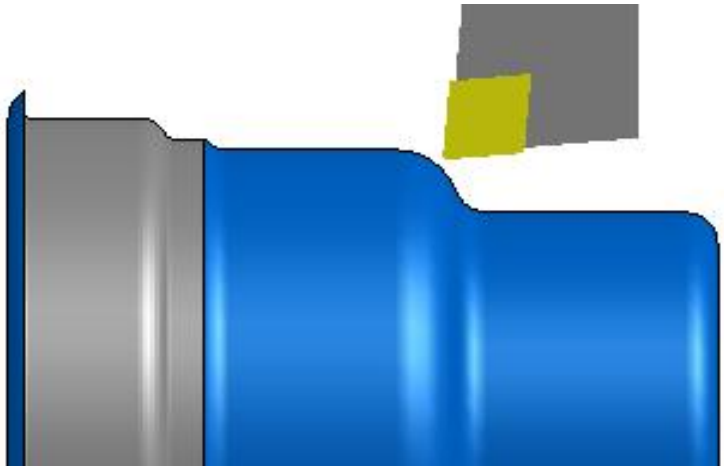


© Copyright 2016 Autodesk, Inc. All rights reserved

- Offset all geometry by **6mm** as illustrated on the next page.



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



© Copyright 2016 Autodesk, Inc. All rights reserved

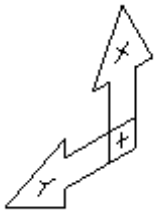
Turning Setups Explained

Turning Setup: - for 2-axis turned parts. Your part can contain multiple setups and these setups can mix different manufacturing techniques. Adding setups to parts lets you work in different orientations on the same part. Or, if you have two parts that share most of the same design, you could create the common features in one setup, and the other different features in two other setups. Then, depending which setups you include in the plan and manufacture, you can have two parts with only slightly more work than one part. The description that follows is for information only.

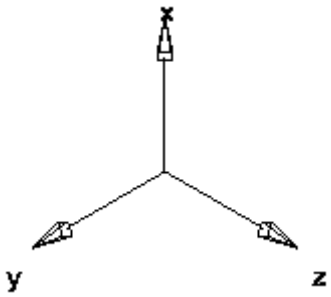
How do setups relate to UCSs?

A User Coordinate System (UCS) is an origin and three vectors (X, Y, and Z) that determine a position and orientation in three-dimensional space. You can use multiple USC's on a turned part to model the features to manufacture your part.

UCS



Setup



One UCS is associated with a setup. A setup is an orientation and part program zero for a physical setup on the machine tool. The orientation and program zero are determined by the associated UCS and the setup contains additional information like the fixture ID and the name of the NC program that will be generated.

*If setups are created directly by aligning with the stock, special UCSs are created with the string **UCS** appended to the setup name. For example a UCS called **UCS_setup2** is automatically created for **Setup2**. These UCSs are used to store the location/orientation information for the setup. They cannot be deleted as long as their setup exists.*

Setups dialog

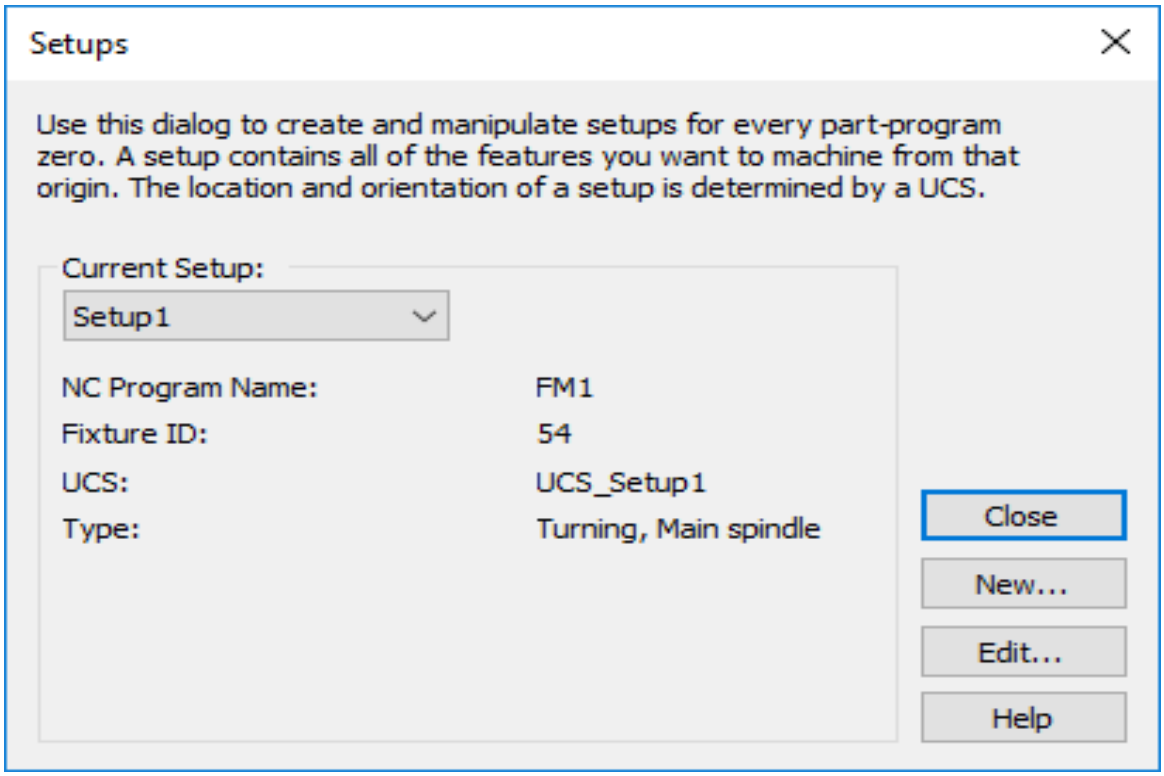
*You use the **Setups** dialog to set the active setup, create new setups and edit existing setups.*

*You access the **Setups** dialog in one of these ways:*

- Select **Manufacturing > Setups** from the menu

© Copyright 2016 Autodesk, Inc. All rights reserved

- Click the **Setups** button on the far right of the **Advanced** toolbar

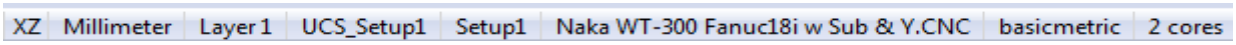


*To create a **New** setup click the **New** button and follow the steps in the wizard.*

*To **edit** an existing setup select it as the **Current Setup** and click the **Edit** button to open the Setups - Definition dialog.*

*To **set** active setup select the setup name from the **Current Setup** list and click the **Close** button.*

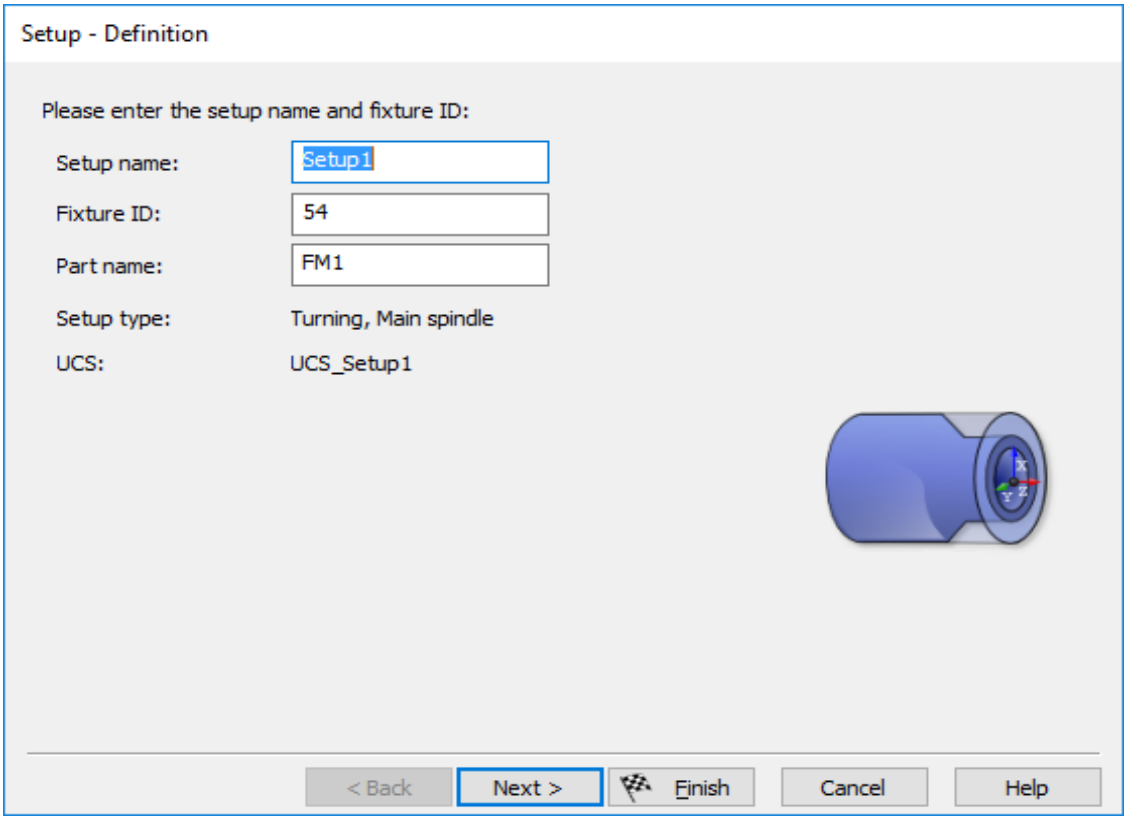
*You can also set or change the active setup in the **Status** bar. Click the existing setup name and select a different setup from the context menu.*



© Copyright 2016 Autodesk, Inc. All rights reserved

Setup – Definition

The Setup-Definition page lets you name the setup and specify the fixture ID.



Optionally enter the **Setup Name**. This name is used only as a label for the setup

- Verify the **Fixture ID** for the setup. The default value is usually correct because it is taken from the current *.cnc post processor template file. If the default value is not correct, enter the correct value

  *Fixture IDs allow the machine to offset the fixture to correspond to the program zero, instead of having to move the fixture to the spot where the program is. For machines that use NC codes **G54 - G59** or registers **54 - 59**, the default should be **54**. For machines that use NC codes **H1 - H3**, the default should be **1**.*

- Optionally enter a different Part Name for the part. This defaults to the file name, but you may need to override this for Fanuc controls to give the part a numeric part name

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

© Copyright 2016 Autodesk, Inc. All rights reserved

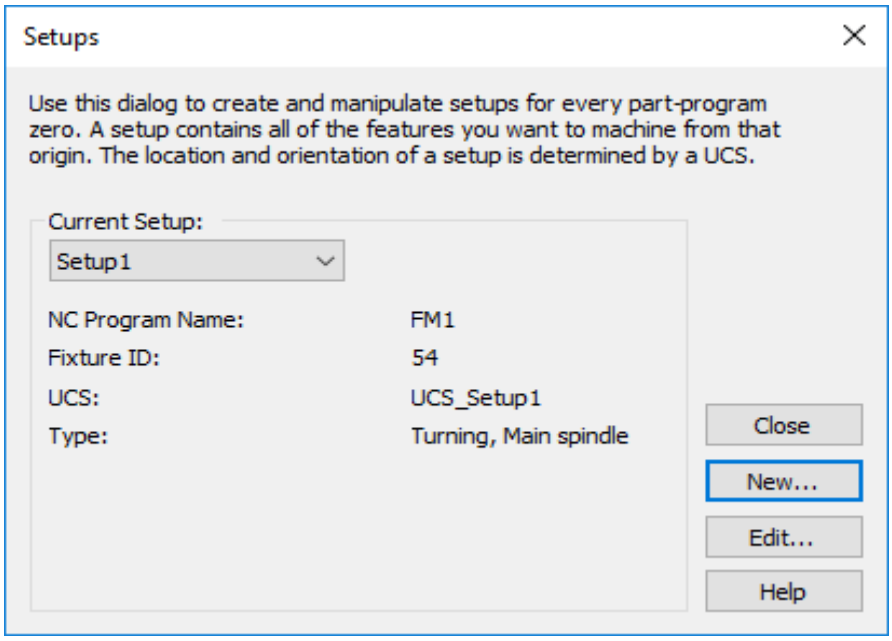
A few quick notes about Part Name

The Part Name are inserted in and must be what is recognized in the NC code ***(00001).*** ***The fixture ID*** *comes from the post processor and the call is applied to the NC code, such as* ***G-54****.* Also, the ***Part Name*** is used *to identify the program text file when you* ***"Save NC"*** *(Use Setup Part Name) under the File menu located upper left.*

Creating a new Setup2

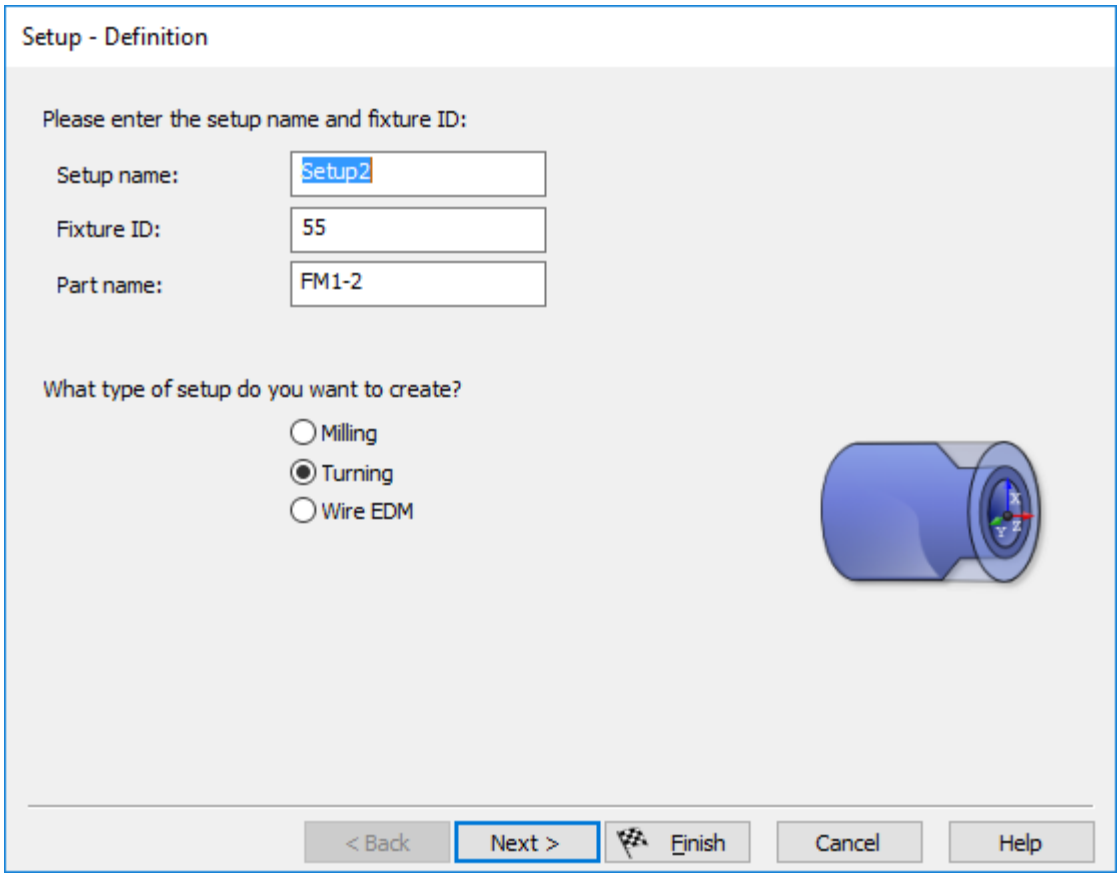
To create a new Setup2 for Main or Sub Spindle follow the guidelines shown below.

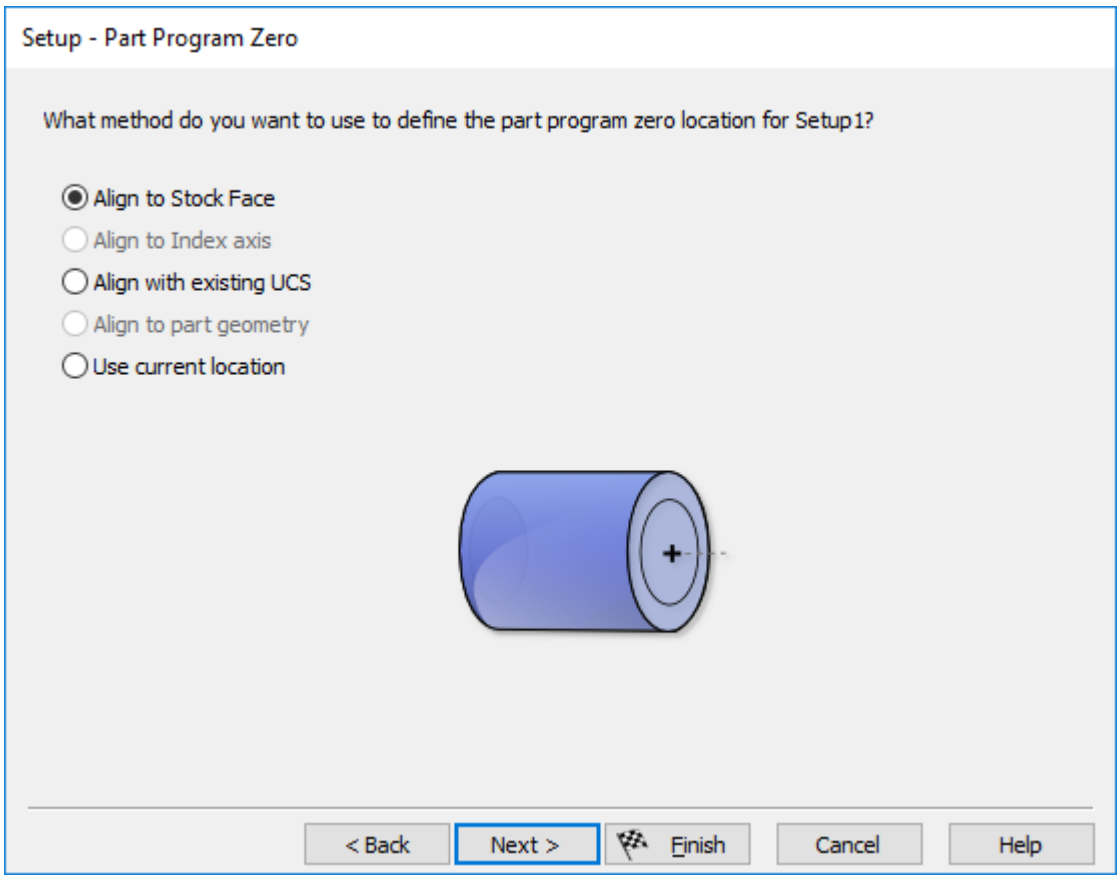
1 Double click on **Setup1** in **PartView**. The following menu will appear.



2 Select **New**. Then select **Next**.

© Copyright 2016 Autodesk, Inc. All rights reserved



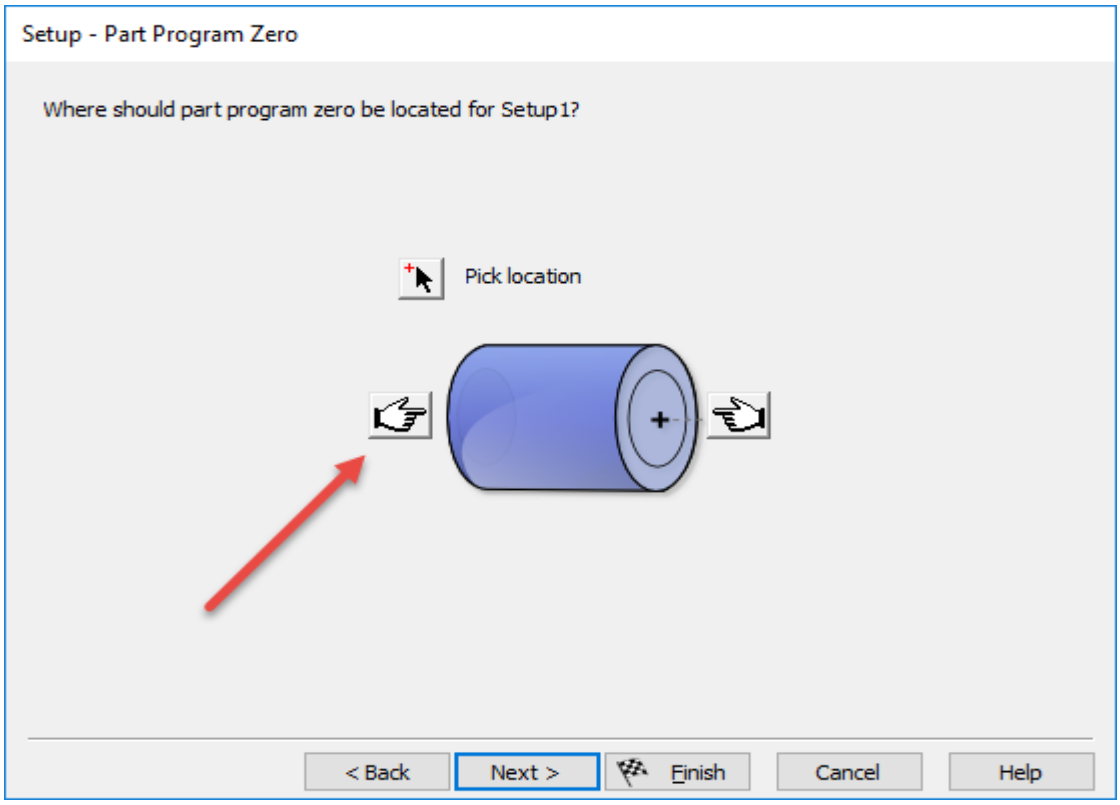


3 *Choose* **Align to Stock Face**

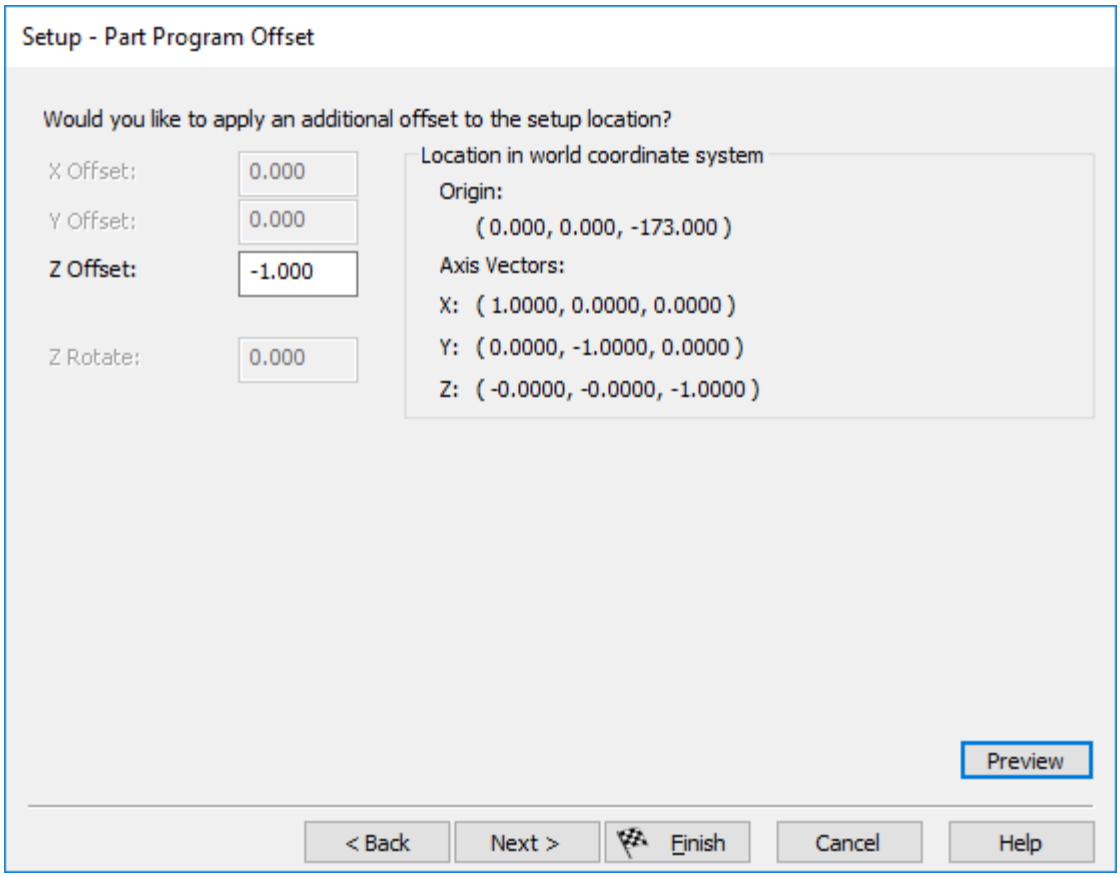
4 Select **Next**

© Copyright 2016 Autodesk, Inc. All rights reserved

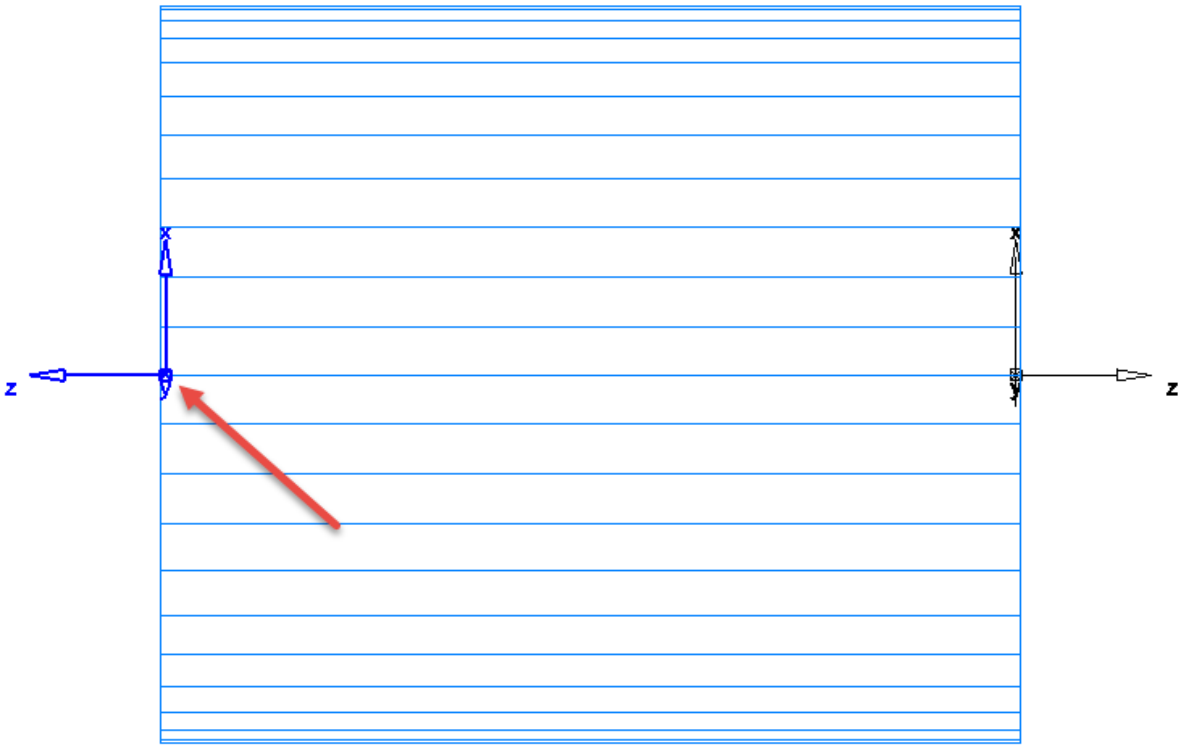
Choose the Left hand Icon as shown on the next image.



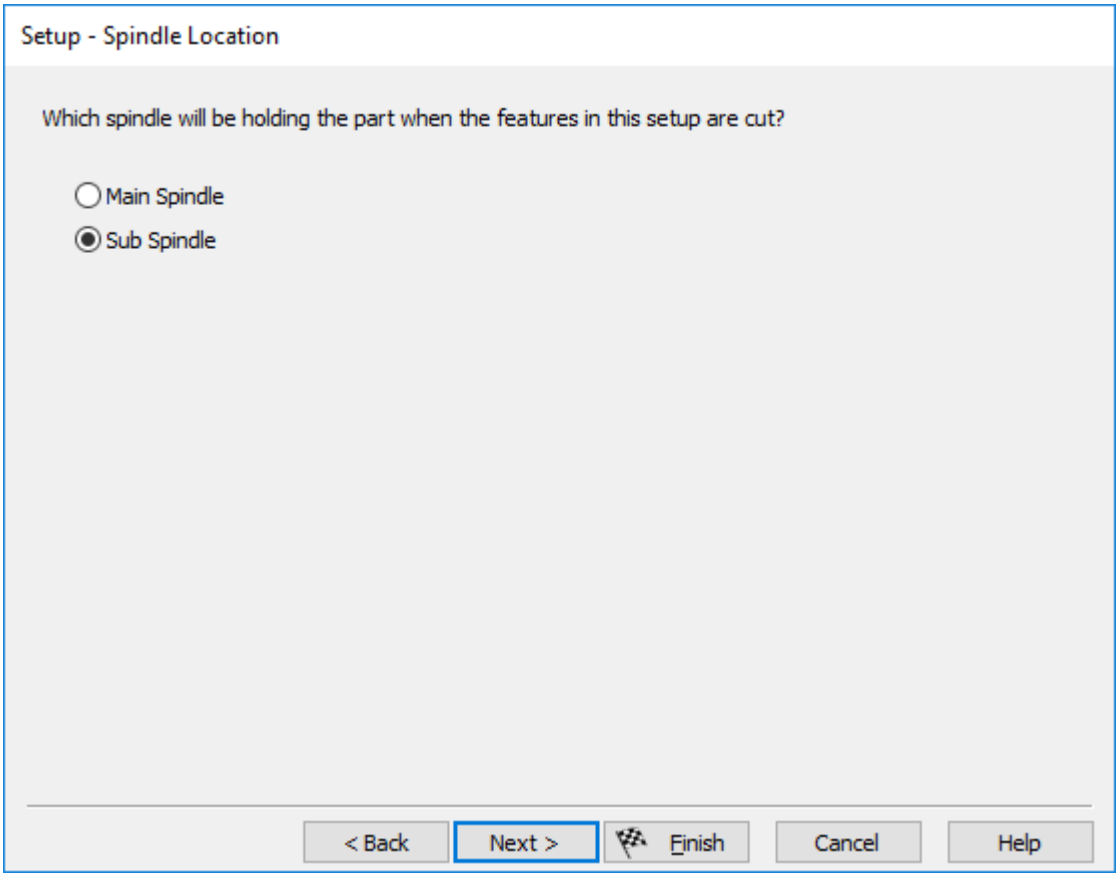
5 Select **Next**. I have entered -1 to move the **Setup2** into the work piece.



© Copyright 2016 Autodesk, Inc. All rights reserved



6 Select **Next**.



Setup – Spindle Location, the Main Spindle will always be selected when there is no Sub Spindle being used.

© Copyright 2016 Autodesk, Inc. All rights reserved

7 Select **Main Spindle or Sub Spindle.**

- A sub spindle is a second spindle at the tailstock end of the machine opposing the main spindle and used to run the second side of the part), covered later in this module.

Specifying the machine simulation file

The Machine Simulation file is associated with the Setup. For files where you simulate multiple setups such as indexed or turn/mill parts, you should associate the machine file with the first setup of a part.

- At any time, when in a **FeatureCAM** document and the need may arise to specify a machine simulation file
- **Open** the Setups dialog in one of the following four ways:
- Select **Manufacturing** > **Setups** from the menu.
- Double-click the **Setup** name in the Part View.
- Select **Properties** from the Setup's context menu in the Part View.
- Double-click the **Setup** in the graphics window.
- Click the **Edit** button.
- Click the **Next** button until the **Setups** > **Simulation Information** dialog is displayed.
- Enter the Z Offset parameters. These represent offsets for loading the part onto the machine. For simulating single turning setups these offsets are applied to the setup after the part is aligned with the top-most location.
- In the Simulation machine design file section, select one of these options:
- **Always use this one** - if you want to identify a specific file. Browse to the location of the Machine Design file.

Remember that the number of turrets and locations of the turrets must match the selected post processor.

- **Use the one specified in the .cnc file** - to use the machine specified in the post processor *.cnc file.

Setting up a part for multi-spindle turning

Setups

- You need at least two setups for multi-spindle turning, one setup for the main spindle and a second setup for the sub spindle. The Z axes of the two setups must point in opposite directions.

If your NC controller requires that both coordinate systems point in the same direction, this adjustment is made in the post processor

- The origin of the two setups can be positioned at any convenient point on the part.
- **FeatureCAM** automatically creates setups with non-descriptive names like Setup1 or Setup2.

© Copyright 2016 Autodesk, Inc. All rights reserved

*For organizational purposes, you may want to rename your setups with more descriptive names like **Main Spindle**.*

- The first setup will automatically be assigned to the main spindle. If you want to perform the first operations on the sub spindle, you can do either of the following:
- Drag the sub spindle setup above the main spindle setup in the **Part View** panel.
- Go to **Manufacturing > Process Plan** in the menu and use the arrow buttons to change the order of the setups in the third list.

Associating features with setups

- Features are associated with the current setup. For multiple turret cutting, the setup associated with the feature will dictate which spindle will hold the part while the feature is being cut. You can see what setup the feature is associated with by looking in the part view.

Moving the part

- The part must be moved between the two spindles using a collection of sub spindle features and perhaps bar feed features.

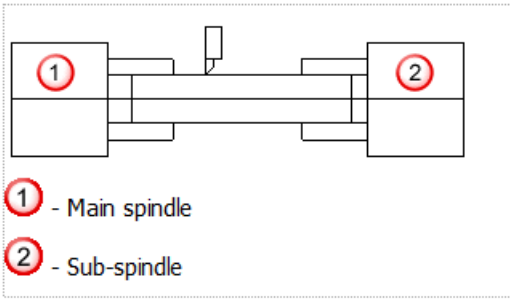
Machine Simulation

- Machine simulation shows each setup cut twice if there is a sub spindle move. This is to verify that we do not crash into the part once it is on the sub-spindle.

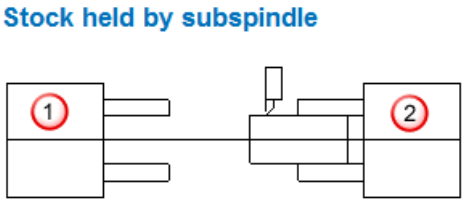
Sub spindle overview

- Sub spindles can be used to support the part from both ends of the stock, or to change which spindle is used to hold the stock. To cut features on two different ends of your stock, you must first create setups at each end of your part with the Z directions pointing out from the stock, as shown below. (If your machine requires that the Z of each setup point in the same direction, this can be changed in the post.)

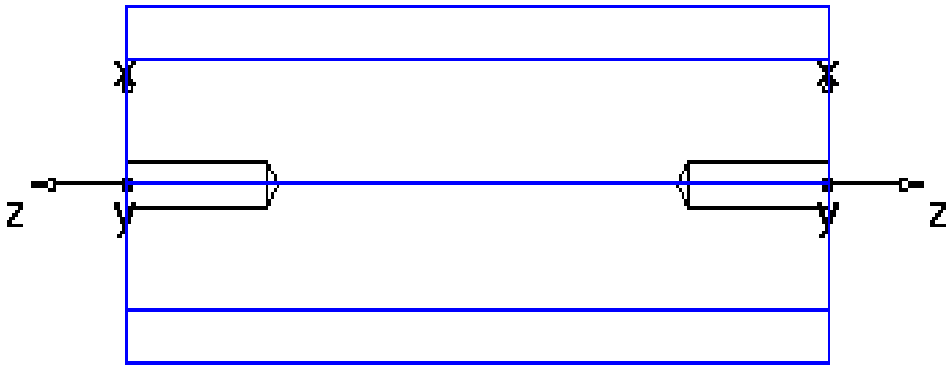
Stock held by main spindle



Stock held by sub spindle

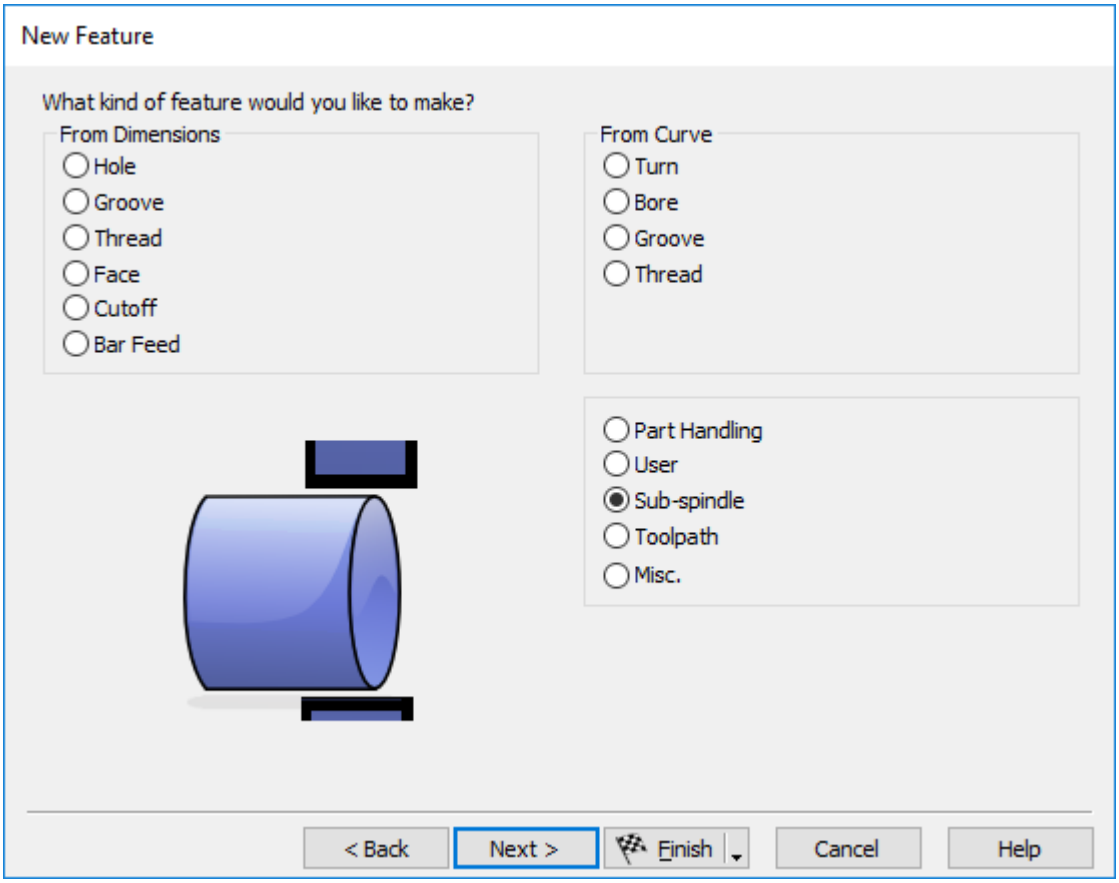


© Copyright 2016 Autodesk, Inc. All rights reserved



Sub-Spindle **Main spindle**

Creating a Sub spindle feature



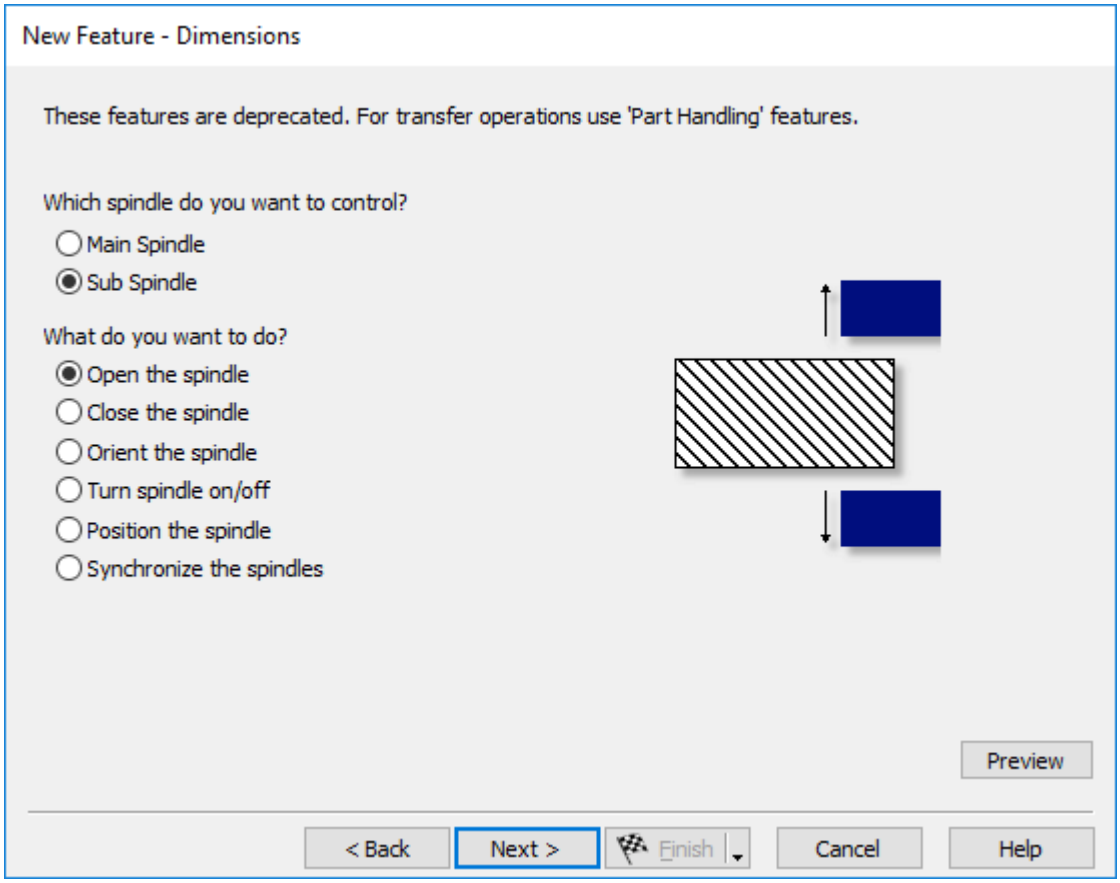
- To create a sub spindle feature: Select **Features** in the **Steps** panel.

  *If you are using the **Turn/mill** module, first select **Turning** as the **Feature type** and click **Next**.*

- Select **Sub spindle** as the **Feature type** and click **Next** to open the **New Feature - Dimensions** page

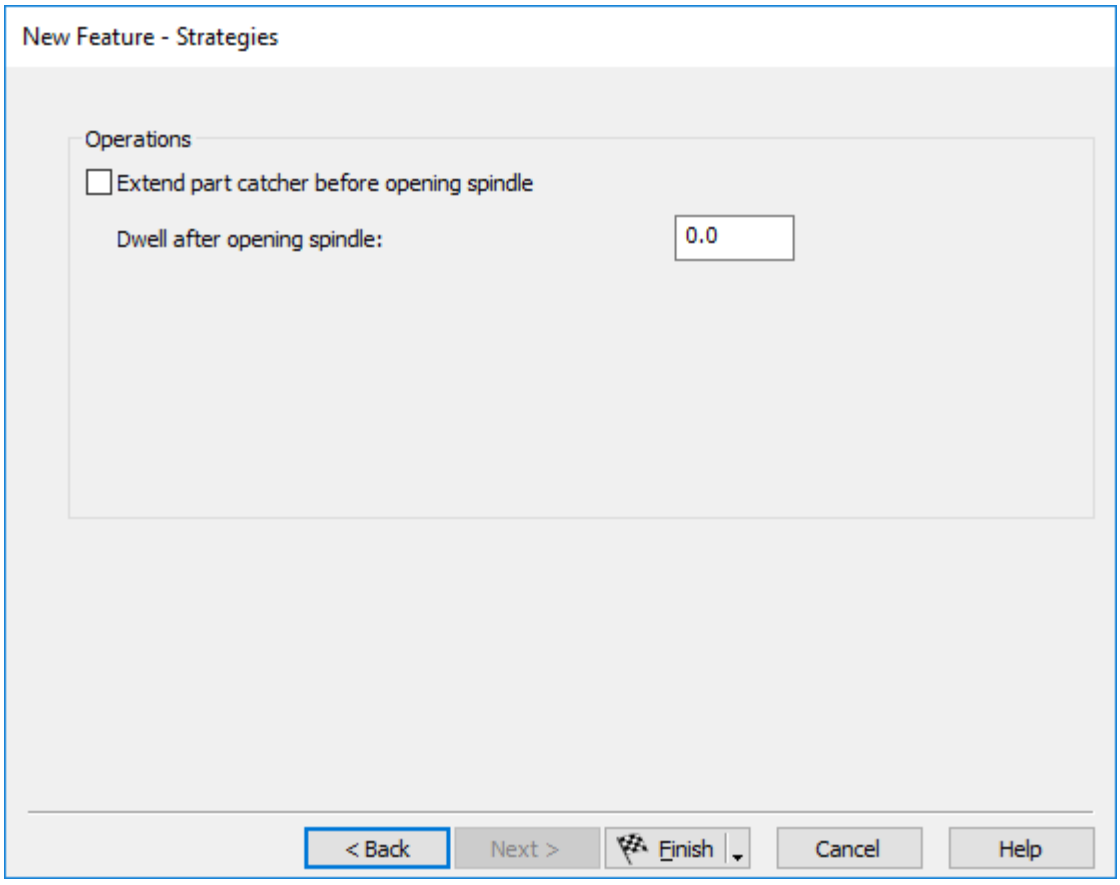
© Copyright 2016 Autodesk, Inc. All rights reserved

- Select which spindle you want to control, either the **Main spindle** or the **Sub spindle**.



- Select the action you want to perform.
- If you selected the action as:
- **Open the spindle**, optionally select **Extend the part catcher before opening spindle** and optionally enter a **Dwell after opening spindle**.
- **Close the spindle**; optionally enter a **Dwell after closing spindle**.
- **Orient the spindle**; enter an angle for **Set spindle angle to**.
- **Turn the spindle on/off**, select **Off**, **CW** (Clockwise), or **CCW** (Counter-Clockwise). If you are turning on the spindle, you must enter the **Spindle speed**.
- **Position the spindle**, the **New Feature - Location** page for a sub spindle opens.
- **Synchronize the spindles**; specify the type of synchronization to use.
- Click **Next**.

© Copyright 2016 Autodesk, Inc. All rights reserved



- Click **Finish**.

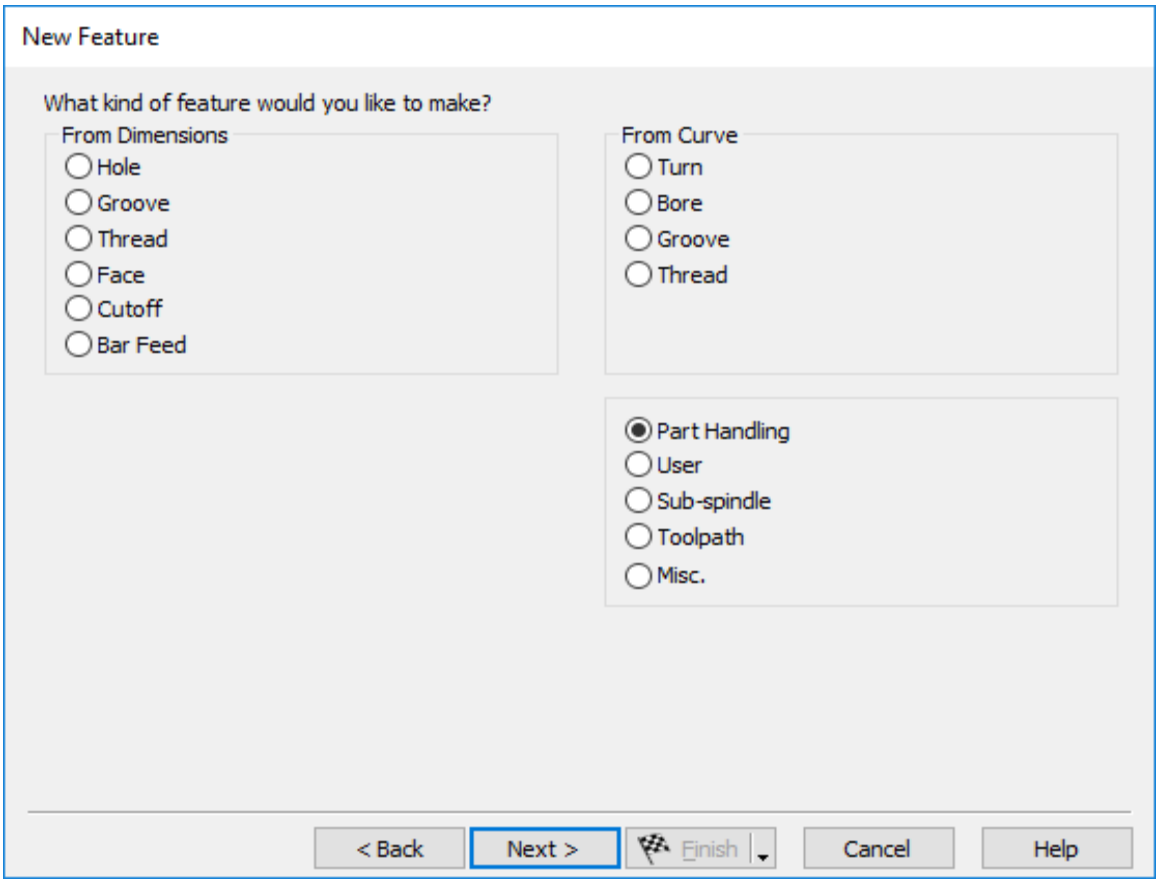
Setting up a part for multi-turret turning

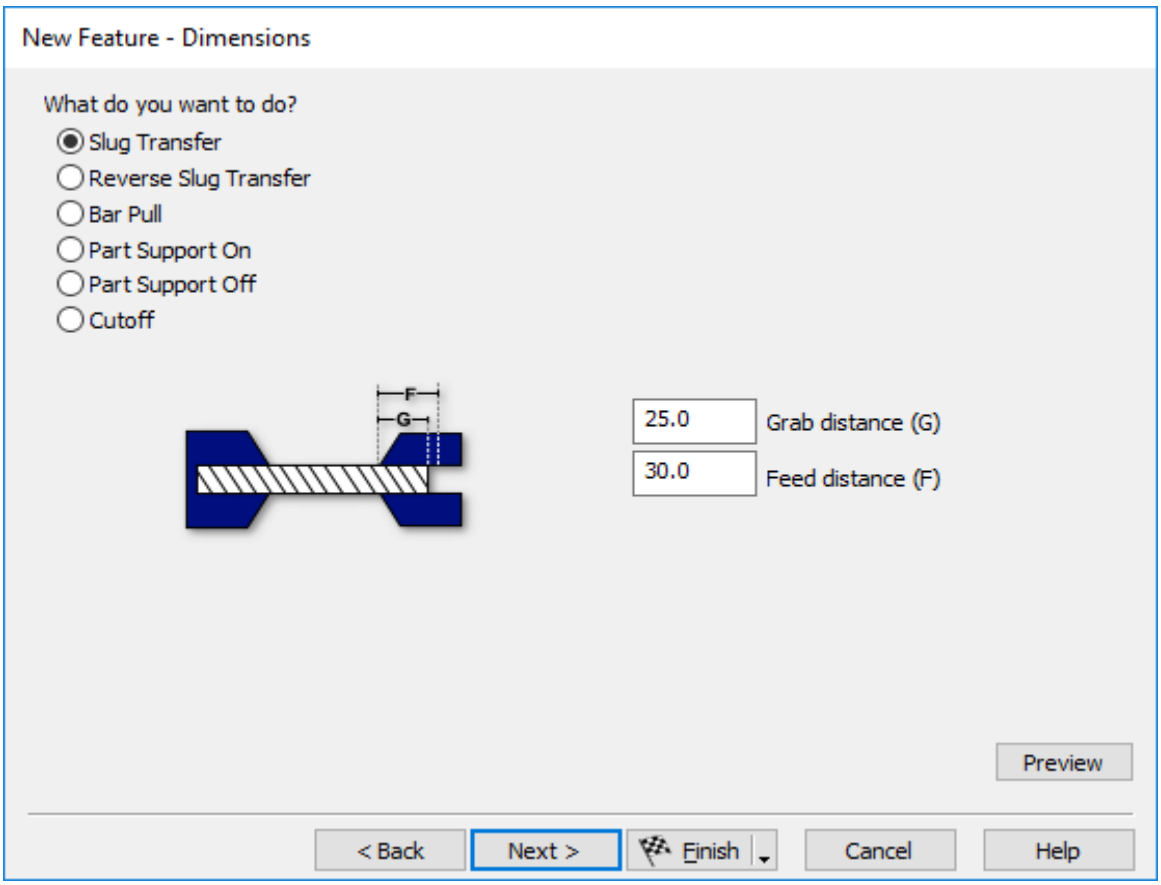
The following issues must be considered for multi-turret turning:

Multi-turret machining can be performed with as few as one setup The post processor that is loaded while you are creating your part must support multiple turrets For pinch or follow turning, your post processor must have opposing turrets

For appropriate machine simulation the post processor must have the same turret and spindle description as the machine design file With the exception of automatically synchronized features, features are automatically associated with the first turret. To move an operation to another turret, you must use the Turrets tab.

© Copyright 2016 Autodesk, Inc. All rights reserved

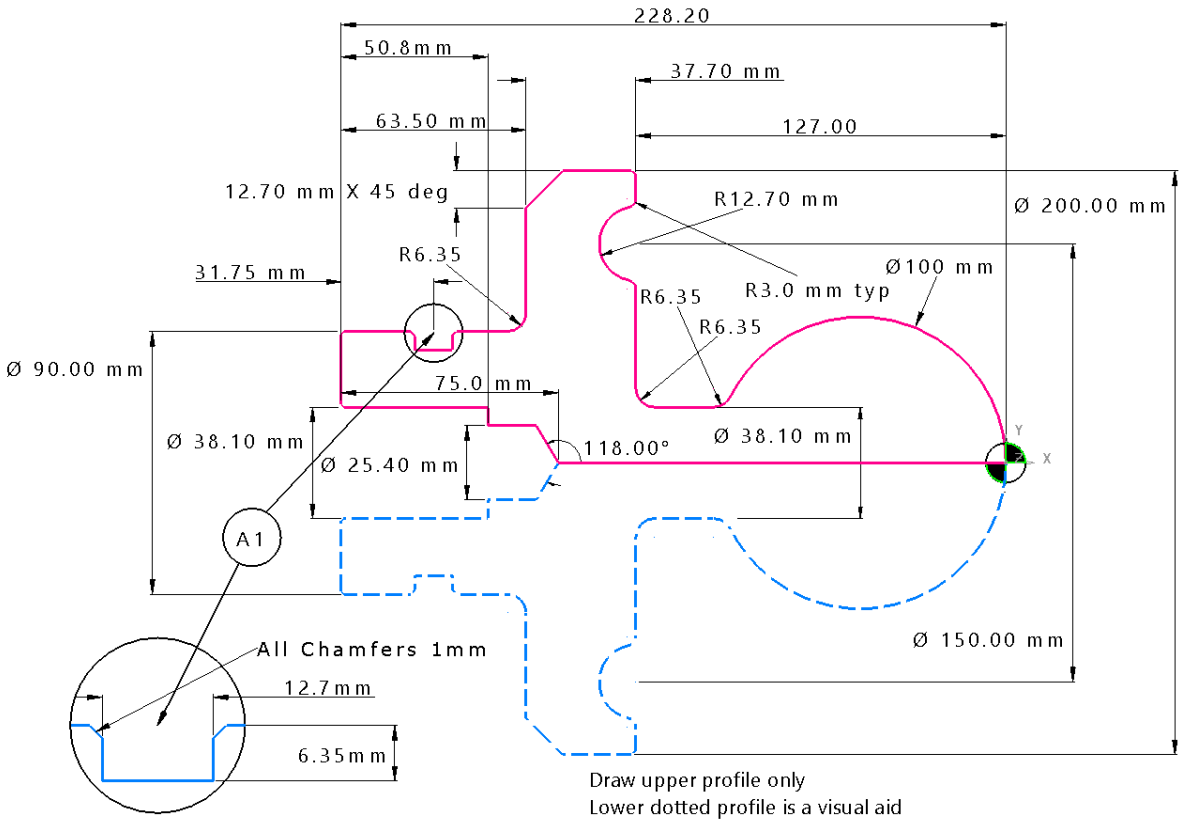




© Copyright 2016 Autodesk, Inc. All rights reserved

Turning Exercise 4 Lightbulb (Class exercise)

Work through this Exercise on your own.



The images above show an outline of the geometry in Blue. This is not a Curve.

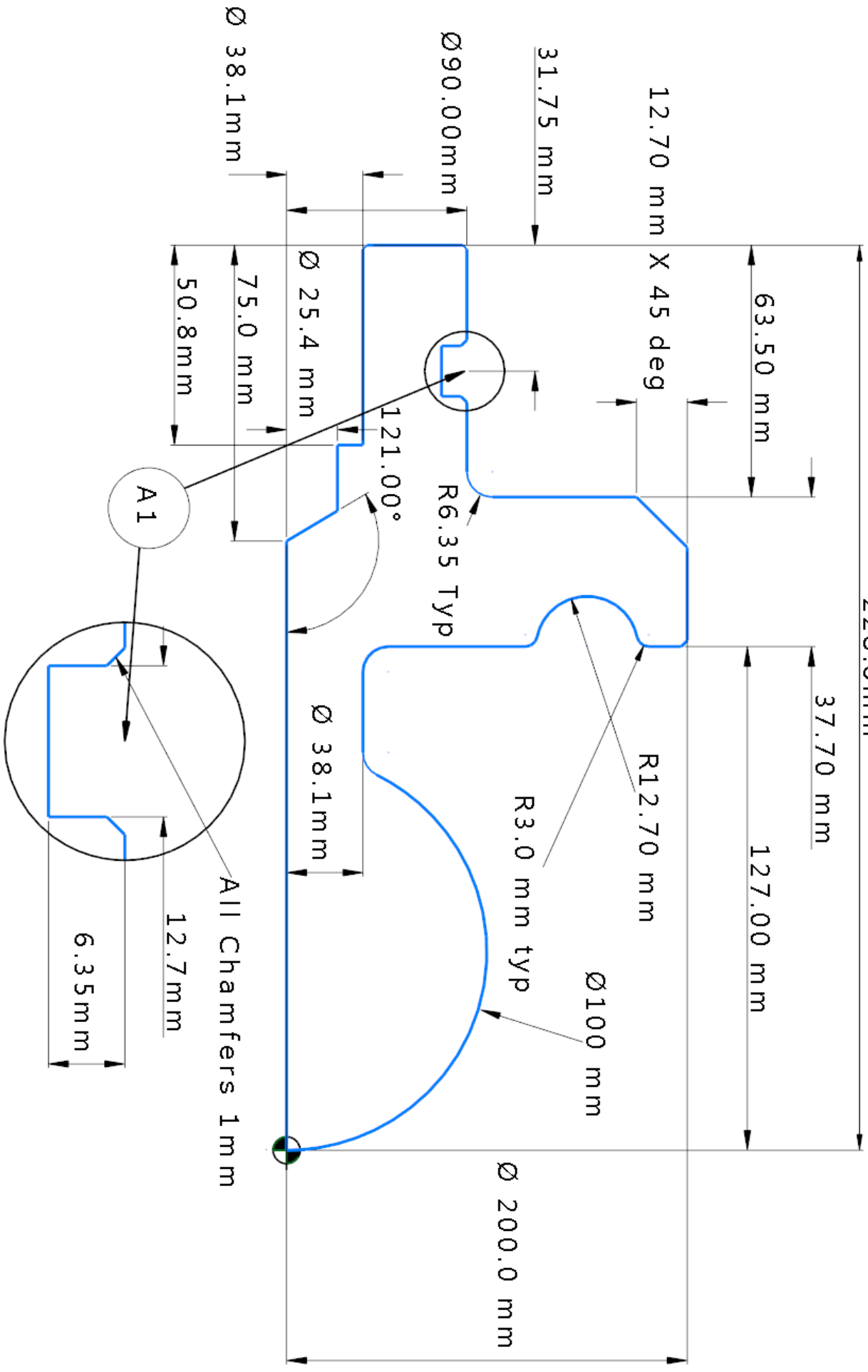
Remember to create a Curve we use pick pieces. All Curves for Turning are open Curves.

Lightbulb drawing exercise in detail

Please note the following drawing exercise can be created in many different ways

1. Open a new Turn/Mill FeatureCAM document Millimeters.
2. Start by creating the outer edges and then create the inner features later on.
3. Work through this on your own. Ask your Instructor for help if needed.
4. A larger drawing image on the next page can be taken out of the folder.
5. Machine the opposite end by creating Setup2 and choosing Main or Sub-Spindle options.

© Copyright 2016 Autodesk, Inc. All rights reserved


Ø 38.1mm
Ø90.00mm
31.75 mm
12.70 mm X 45 deg
63.50 mm
228.6mm
Ø 25.4 mm
75.0 mm
50.8mm
121.00°
R6.35 Typ
A1
37.70 mm
127.00 mm
R12.70 mm
R3.0 mm typ
Ø100 mm
Ø 38.1mm
Ø 200.0 mm
All Chamfers 1mm
12.7mm
6.35mm

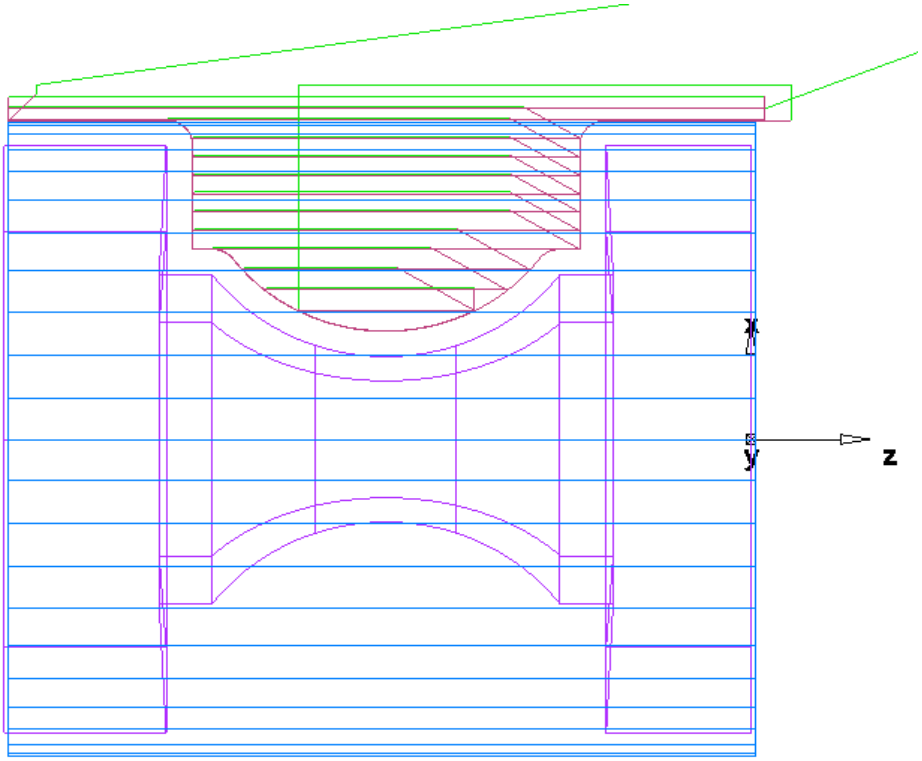
Round Insert Strategy

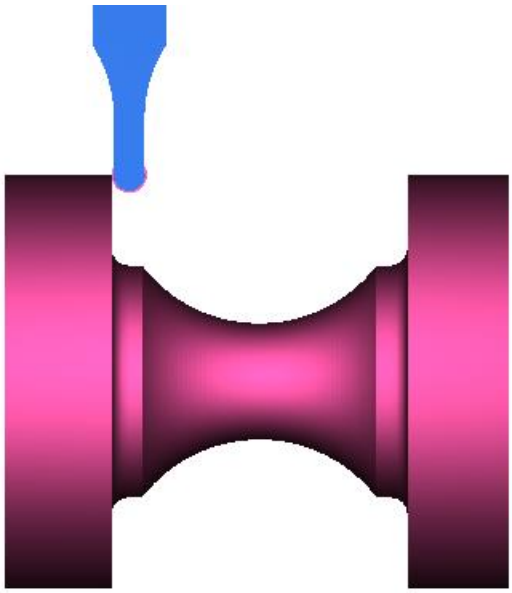
Round Insert— Round tool roughing toolpaths are designed to ease the tool more gently into a groove shape using a Button Style insert.

1 Please load file - Round Insert Strategy.fm from the folder path shown below.

C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm Files

This exercise shows a simple turned profile with a slight undercut using a button insert tool. Notice the angled lines to the right showing the tool entry into the groove.





© Copyright 2016 Autodesk, Inc. All rights reserved

This Page Intentionally Left Blank

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Basic Turning Attributes / Configurations 2
Introduction 2
Machining Configurations 2
Machining Attributes 4
Drilling Tab 4
Pecking 6
Turn/Bore tab (Turn) 7
Rough depth of cut 7
Engage angle 8
Tool nose radius compensation 9
Use canned cycle 9
Hole features 9
Turn/Bore features 9
Groove features 9
Thread features 9
Constant DOC 10
Threading tab (Turn) 11
OD Thread Feature 12
ID Thread feature 12
Grooving tab (Turn) 15
Cut type 15
Cutoff tab (Turn) 18
Bar Feed tab (Turn) 19
Coolant 20
Misc. tab (Turn) 21
Turret location 24
Operations 25
Base priority 26
Use Rules 27
Use Template 28
What are Machining Attributes? 29
Creating Machining Configurations 31
The Attributes 32
Drilling Tab 32
Pecking Tab 32
Turn/Bore 32
Threading 32
Grooving 32
Cut-off and Bar Feed 32
Miscellaneous 33
Operations 33
Help Contents 33

© Copyright 2016 Autodesk, Inc. All rights reserved

Basic Turning Attributes / Configurations

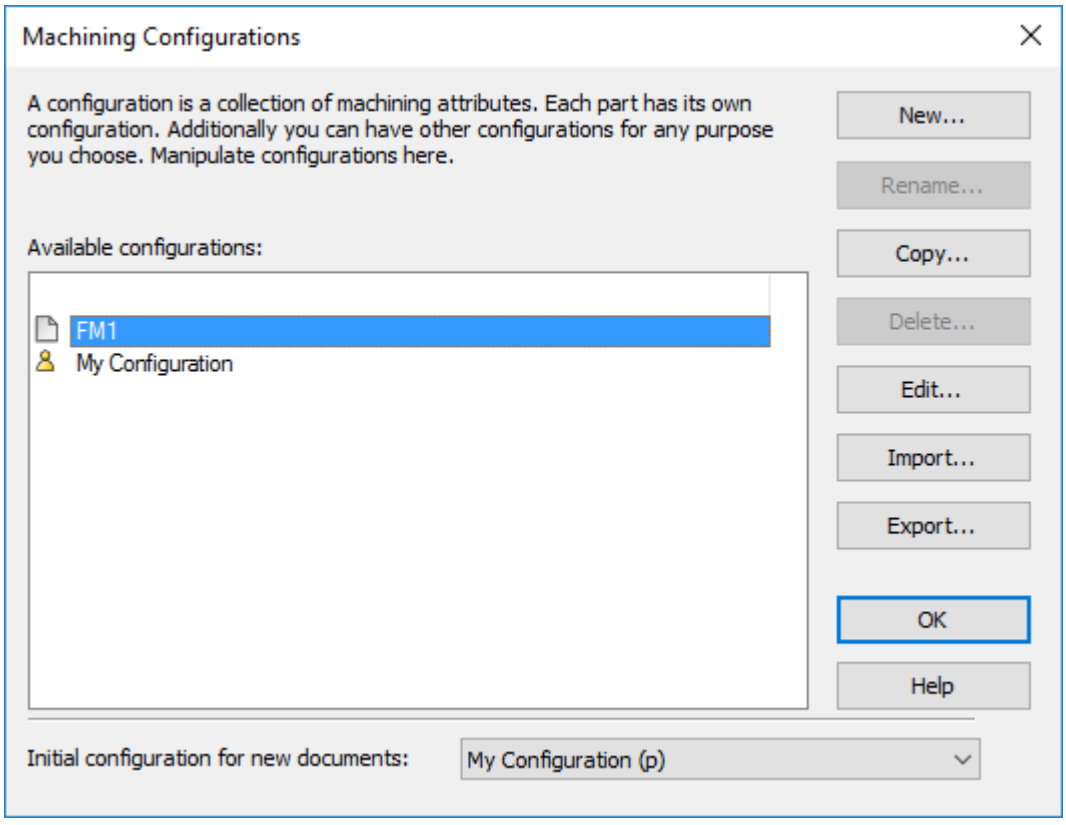
Introduction

This module will provide the user with an understanding of what attributes are, (characteristics and features) and where they reside Also covering when an attribute is chosen, where it shows up in **FeatureCAM**, and how it affects the feature.

By the end of the module the user will be able to select, prior to creating the program, the machining configuration of attributes, as the defaults for the material and/or machine to be used. This saves time in programming by not having to make the same repetitive entries for the choices you wish to be defaults when that particular material, or machine, is chosen. It also eliminates having to remember the different settings.

Machining Configurations

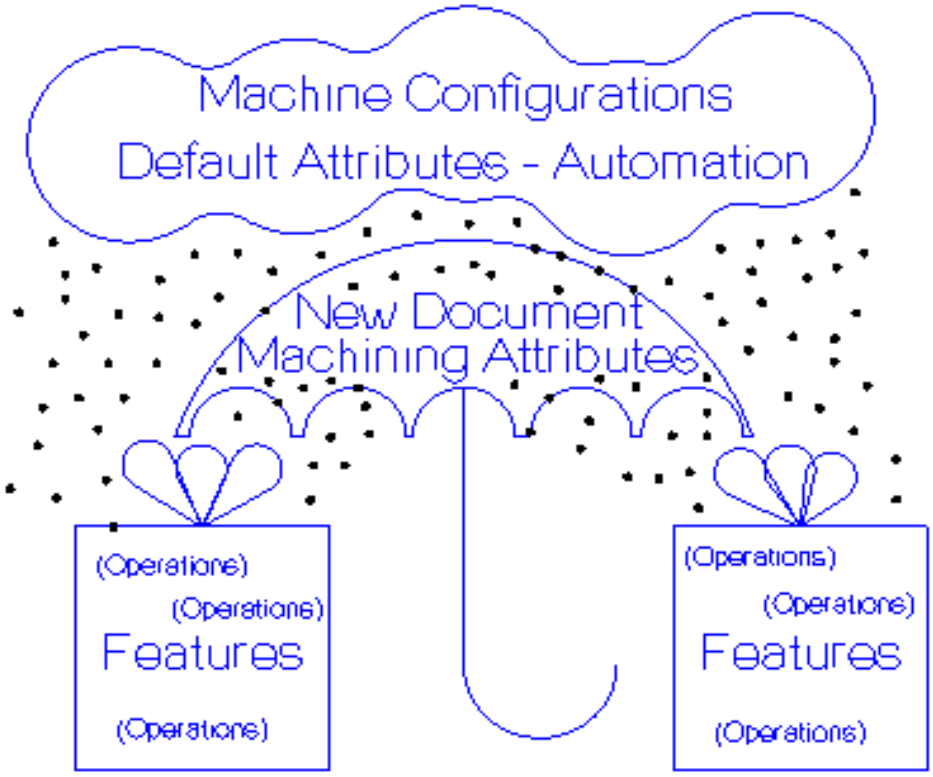
- The **Machining Configurations** can be accessed from the **Manufacturing** drop down menu.
- Machining Configurations are predetermined default functions that are selected, and used in all features that are created in the document.
- A Machining Configuration contains approximately 100 settings.
- An unlimited number of Machining Configurations may be created.
- The user can name a Configuration to reference the material and/or machine to be used.



- "Initial Configuration for new documents", is in the drop down menu at the bottom.

© Copyright 2016 Autodesk, Inc. All rights reserved

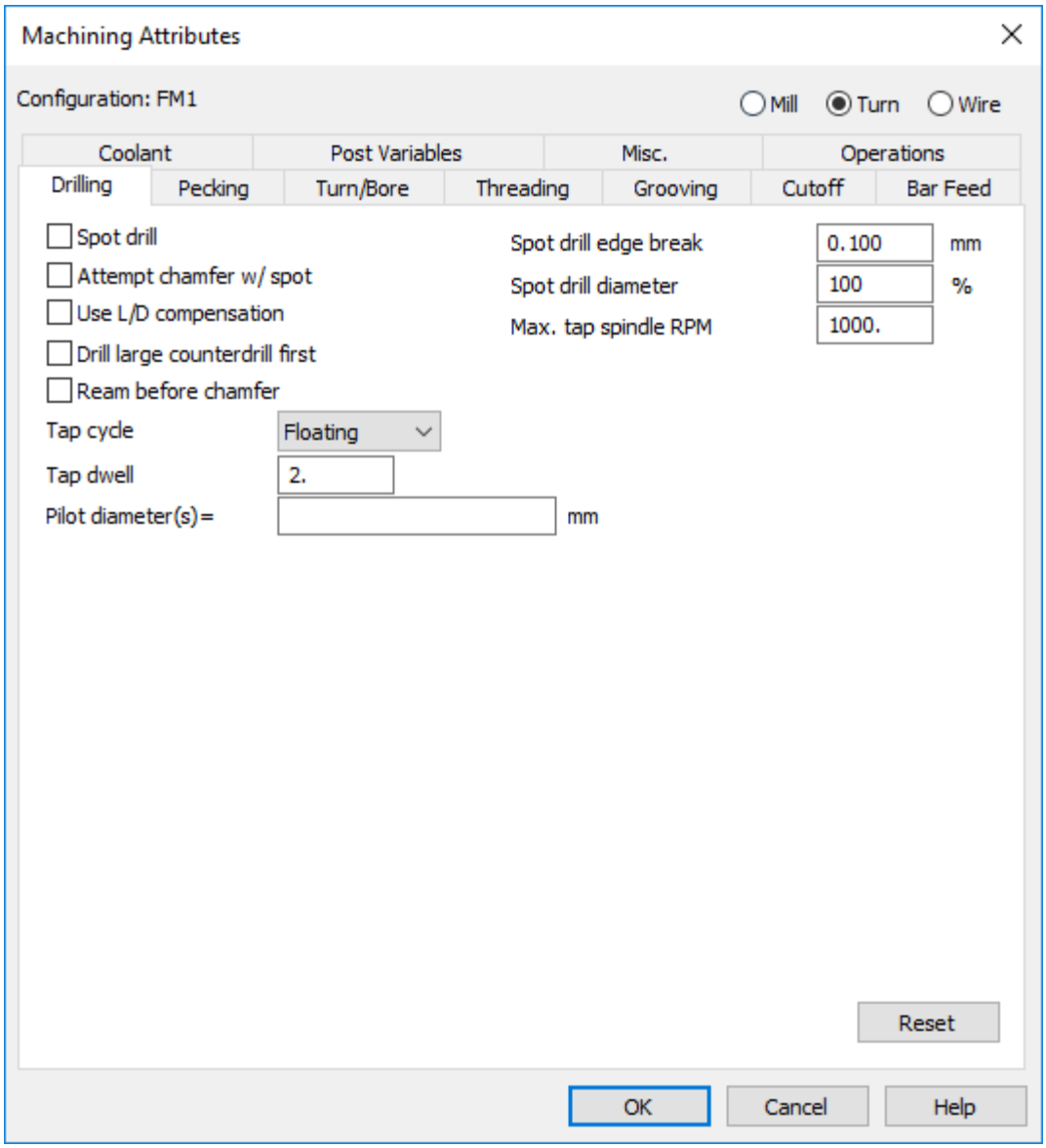
- When a Machining Configuration is selected prior to programming, the attributes, so to speak, rain down into the new document just opened. These are used as the default settings. They are divorced from the Machining Configurations and become Machining Attributes, assigned to, and a permanent part of the new document.



- These settings may be changed in the document. You can click Manufacturing and Machining Attributes at any time. Changing any attributes will assign them to this document only, but does not affect the original, or default, Machining Configuration.
- In other words, these changes do not become the defaults and when a new document is opened the old defaults are active. These settings stay with the document and are active anytime this same .fm document is opened.

© Copyright 2016 Autodesk, Inc. All rights reserved

Machining Attributes



Drilling Tab

Spot drill

This operation has some wide-ranging effects, however, especially when used with the Attempt chamfer w/ spot and tool optimization. Of those three settings, tool optimization has the highest priority and its decisions override settings with a lower priority.

For example, a spot drill operation could be performed with either a spot drill or a centre drill. Spot drills with a tip angle of 90 can also perform a chamfering operation. You specify a specific tool to cut the hole's chamfer and also turn on Attempt Chamfer /w Spot and tool optimization. If there is an appropriate spot drill in the tool crib, FeatureCAM optimizes things and use this tool in spite of your lower priority override. Even though you selected a specific tool, your other settings conflicted with and superseded your choice.

© Copyright 2016 Autodesk, Inc. All rights reserved

This is the advantage of the optimization and simulation functions in FeatureCAM. As you work through the optimization settings, and see where you can optimize automatically and where you cannot, you can find ways to group your parts for faster production, but still use specific tools for specific effects when needed.

You can set this attribute for an individual feature in the **Feature Properties** dialog. Feature-level attributes override Machining.

Spot drill edge break — to create an edge break or chamfer using the spot drill tool, enter the radial distance of the edge break/chamfer. The spot drill creates an edge break/chamfer by cutting deeper than it normally would to create the spot drill operation alone. The default value **0.0050**" or **0.1** mm results in a chamfer 0.0100" or 0.2 mm greater than the hole size. The angle of the chamfer depends on the spot drill tool used.

Enable this option to try to cut the chamfer during spot drilling. If no available tool can spot and chamfer without gouging the hole, a separate chamfer operation is created.

Spot drill diameter %: - This percentage is used to select a spot drilling tool. A value of **100** specifies that the spot drill should be the same diameter as the hole. A smaller value creates only a starter hole.

Use L/D compensation: - This reduces speed and feed for holes that have a ratio of hole depth (L) to hole diameter (D) of greater than 2.5.

The greater this ratio, the greater the speed/feed reduction.

Max tap spindle RPM: - This is the maximum speed (in RPM) for tapping.

Tap cycle affects how a tap operation is performed. Select from:

Floating: - Floating and tension-compression holders

Rigid: - This is most commonly available on current machines

Deep Hole: - The tool pecks and retracts to the **Plunge clearance** and returns to the previous depth.

Chip Break: - The tool stops feeding only to break the chip.

All cycles use the same **Tap** program format, but logical reserved words exist in **XBUILD** to distinguish the tap type.

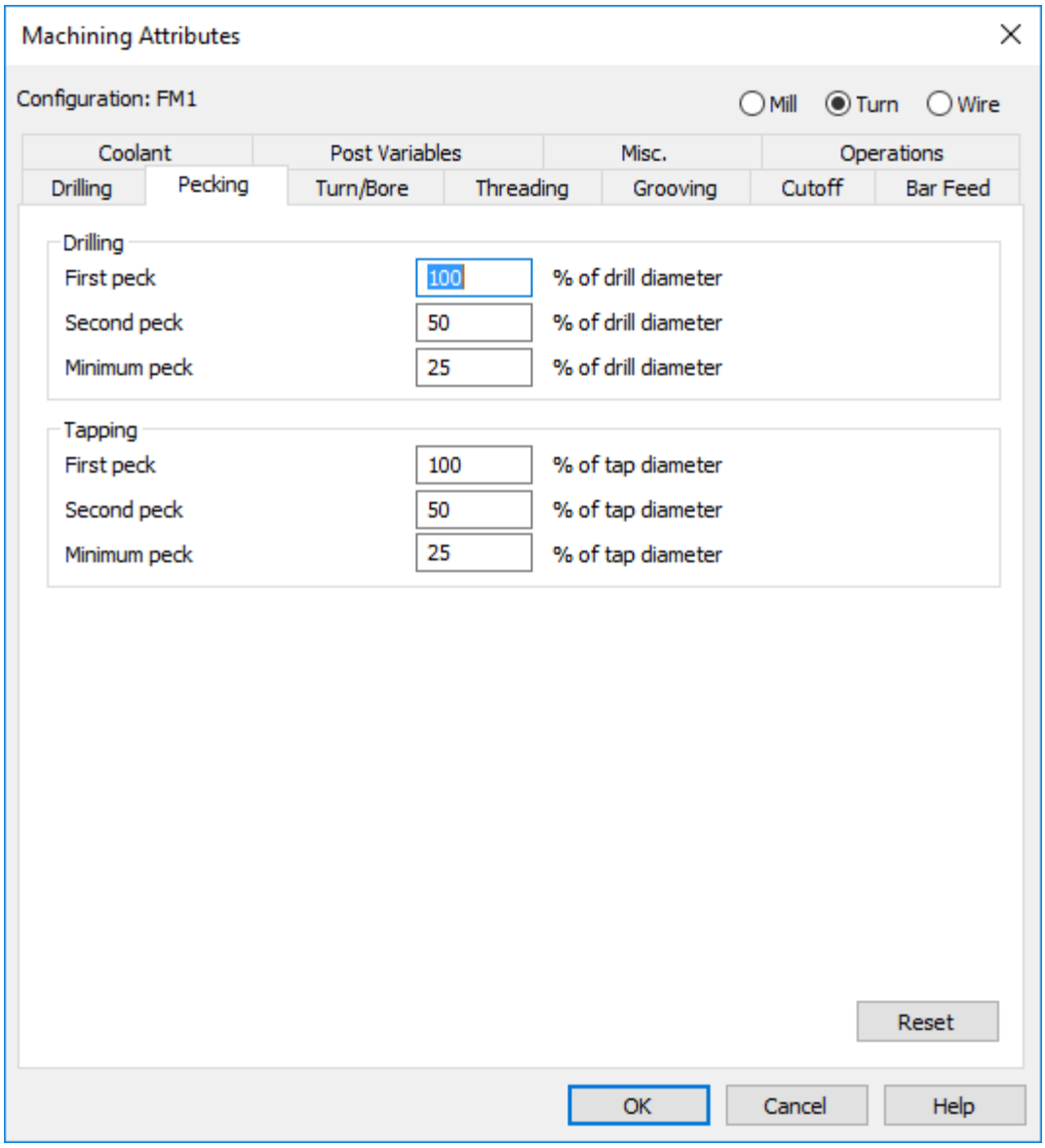
Pilot diameter(s): - This enables and sets a list of drill sizes used to drill pilot holes. Enter a comma-separated list of drill diameters. For example, entering **12, 25, 40** in metric, causes holes to be pilot drilled with the 12mm drill for final hole sizes up to 25mm. A hole in excess of 40mm is pilot-drilled with all three of the specified drills before being drilled to size. No list of drill sizes turns off pilot drilling for the feature, although this attribute can also be set up as a default for all parts.

© Copyright 2016 Autodesk, Inc. All rights reserved

Pecking

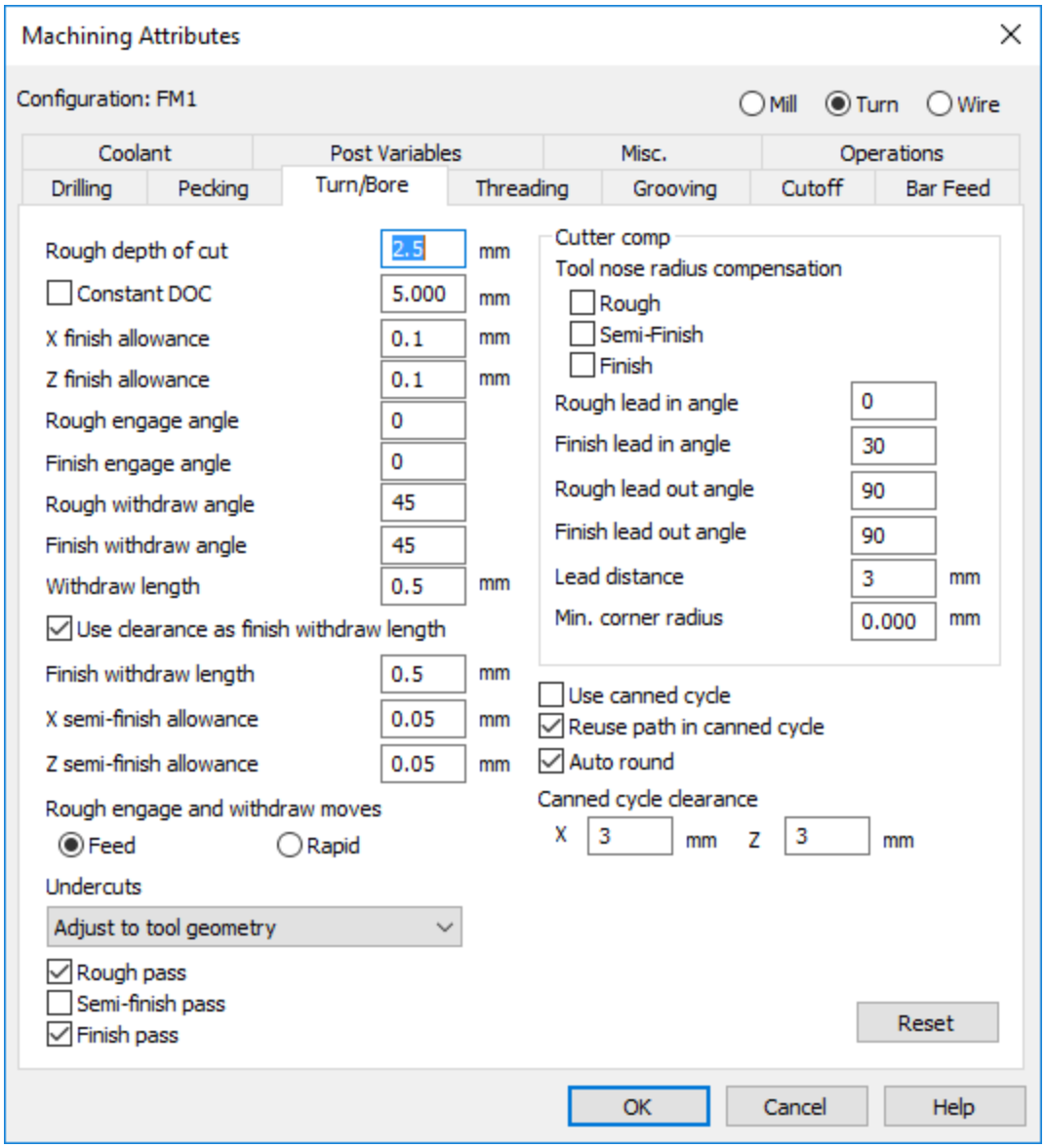
Pecking applies to Deep Hole, Chip Break, and Tap operations. FeatureCAM supports four styles of pecking. These styles are listed in the post processor. Three different attributes control the pecking and they are used differently depending on the style of pecking. FeatureCAM checks the pecking type in the currently loaded post processor to duplicate canned cycles when simulating toolpaths. Set these attributes separately for **Drilling** and **Tapping** operations:

- **First peck: -** This is the depth of the first peck of a drilling/tapping operation specified as a percentage of tool diameters. If the depth of the hole is less than First peck, the hole is drilled in a single peck.
- **Second peck: -** This is the depth of the second peck of a drilling/tapping operation specified as a percentage of tool diameter. The post handles the conversion.
- **Minimum peck: -** This is the minimum step size for a peck used for value reduction pecking methods or factor reduction pecking methods.



© Copyright 2016 Autodesk, Inc. All rights reserved

Turn/Bore tab (Turn)



Rough depth of cut

Enter a step increment for each pass that the roughing routine performs on the part. The interpretation of Depth of cut depends on the Constant **DOC** setting in the Turn/Bore document-level options.

If Constant **DOC** is deselected, the Depth of cut value you set is the maximum depth of cut for the feature. If the Depth of cut evenly divides the depth of your feature, your increment is used. If it results in a final pass that is quite shallow, the Depth of cut is adjusted to result in even roughing passes. For example if you have a feature that is 12mm deep and specify a Depth of cut of 4mm, the feature is roughed in two even passes 6mm deep instead of one pass of depth 10mm and another pass with depth of 2mm

© Copyright 2016 Autodesk, Inc. All rights reserved

If Constant **DOC** is selected, the feature is cut using this depth for each pass. With Constant DOC selected, you can also list a series of depths, separated by commas, to control the depth of each cut. For example a Depth of cut specified as 6mm, 4mm, 2.5mm results in the first pass being cut at 6mm, the second at 4mm and the remaining pass at 2.5mm. If there are more cuts than depths specified, the last depth is repeated.

Engage angle

Enter the approach angle for the tool, measured away from the part. An angle of 0 degrees approaches along the path. An angle of 90 approaches perpendicular to the path. If the beginning of a scan line begins with a shoulder, a value of 90 is used automatically for that scan line. The only valid values are 0 or 90 degrees.

The Engage angle and Withdraw angle are specified from the path (or extension of the path), relative to the side of the path that the tool is on, and the direction in which the tool is traveling.

Withdraw length: - This is the distance along the withdraw angle line in which the tool withdraws before returning for the next step.

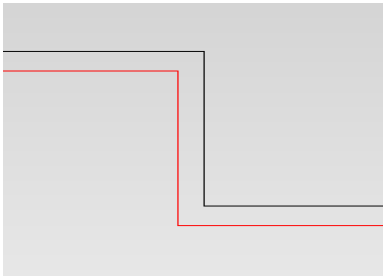
Use Clearance as finish with draw length: - When selected, the **Clearance** value is used as the approach and withdraw length for finishing moves. When deselected, the **Withdraw length** value is used.

Auto round

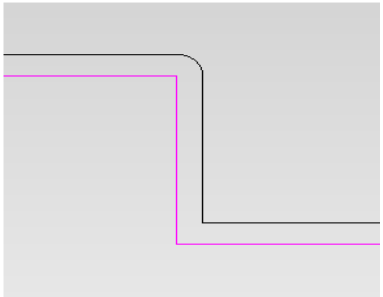
This turning attribute applies to both rough and finish passes. When **Auto round** is enabled, **FeatureCAM** automatically inserts arc moves to connect two non-tangent elements. The effects are:

- Minimum of wasted motion by the machine; however, the posted part program may be slightly longer in the number of blocks used.
- Burrs are removed, but otherwise the part has the same shape and dimensions given by the feature curve because the radius of the inserted arc is the same as the tool nose radius.
- Machine motion is smoother.

This is an example with **Auto round** turned off:



This is the same example with **Auto round** turned on:



© Copyright 2016 Autodesk, Inc. All rights reserved

Tool nose radius compensation

- Enable this option to ignore the tool radius when generating passes for Turn, Bore, and Face features. The actual part geometry is output as the toolpath. It is assumed that the tool radius compensation will be performed by the operator at the machine tool when this option is enabled.
- Select whether you want **TNR comp** for **Rough**, **Semi-Finish**, and **Finish** operations. Enter the **Lead-in angle**, **Lead-out angle,** and **Lead distance** parameters for **TNR comp**.

Use canned cycle

Enable this option to perform the feature's operations using canned cycles. You must use a post that has support for roughing and finishing canned cycles.

Canned cycles can be generated in the NC code for nearly every turned feature. To generate these macros, your post processor must support them, and you must turn this function on for the post and for some features you must also activate the canned cycles at feature level.

Hole features

If Enable drilled canned cycles is deselected in the Post options dialog, then all hole drilling operations are computed in the post. This includes spot drilling, drilling, bore, ream, and tapping operations. If Enable drilled canned cycles is selected, then canned cycles will be output if the post you are using has g-codes defined for the hole canned cycles. If the post does not have these G-codes defined, the hole operations will still be computed.

There is no way to control the output of canned cycles on an individual feature basis.

Turn/Bore features

Canned cycles for Turn and Bore features must be enabled by selecting Enable **turn canned cycles** in the Post options dialog. You must then go to the Properties dialog for each **Turn/Bore** feature, click the **Strategy tab** and select **Use canned cycle**.

Also select **Reuse path in canned cycle** if you want to output the path geometry only once for both roughing and finishing. You can also set these values in the default attributes, but remember these values will only apply to features you create after making this change.

Groove features

Enable **grooving canned cycles** in the Post options dialog by selecting Enable **groove path canned cycle**. Then turn on canned cycles for each groove by bringing up the feature's Property dialog, clicking the **Strategy tab**, and then clicking Use path canned cycle. You can also set this attribute on the Groove tab of the default attributes, but this will only apply to features you create after changing this setting.

Thread features

Thread features always use canned cycles.

© Copyright 2016 Autodesk, Inc. All rights reserved

You can set this attribute for an individual feature in the Feature Properties dialog. Feature-level attributes override Machining Attributes

Reuse path in canned cycle: - Relates to Use canned cycle. Enable this option to output the curve to the NC file once and then reference it in both the Rough and Finish canned cycles. This option is enabled by default.

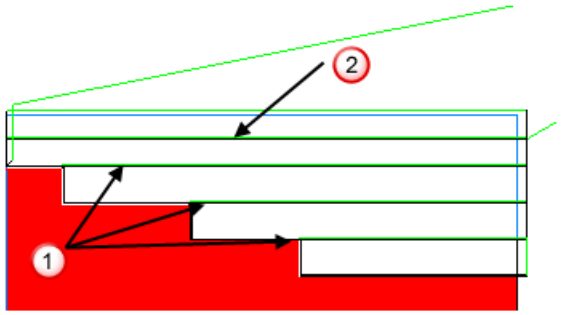
Canned cycle clearance X and Z: - These attributes control the tool location before the start of a turning canned cycle. The tool location is obtained by applying the X and Z clearances to the start point of the curve.

Constant DOC

The Constant DOC option controls how the Depth of cut attribute is used to calculate the X depth of roughing passes.

If **Constant DOC** is deselected, cuts are guaranteed not to exceed the **Depth of cut**, but some cuts may be less than that amount. A pass is performed at each step. The area above each step is divided into equal regions with a depth of less than the **Depth of Cut**.

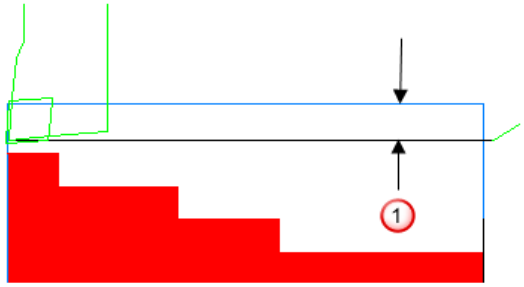
In the example below, the bottom three cutter paths are aligned with the steps. The top region is divided into two equal regions with shallow depths of cut.



1 Aligned with Steps

2 Shallow Cut

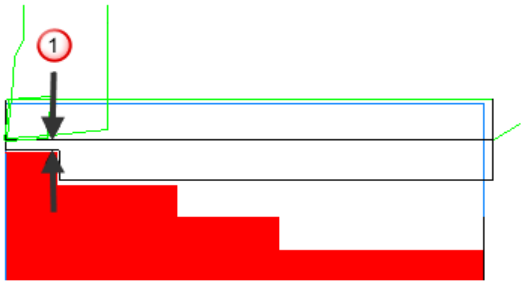
If Constant DOC is selected, each pass is performed at a consistent depth of cut. If required, the pass also climbs and cleans up previous steps. In the figure below, the first cut is performed straight across at the specified Depth of cut.



1 Depth of cut

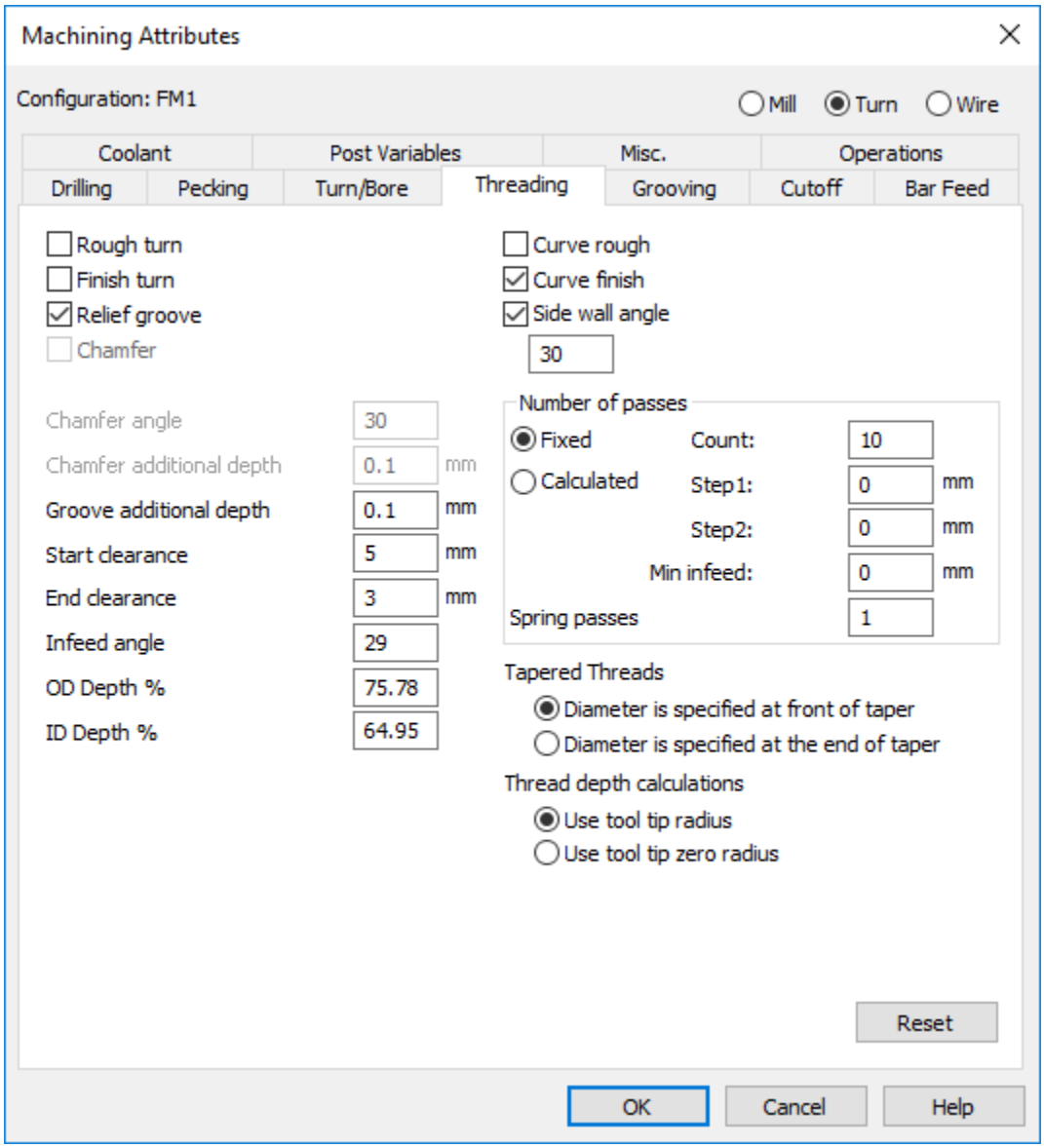
In the next pass, the tool cuts across at the specified depth of cut and then the tool climbs up the wall and the previous step is cut to with the finish allowance.

© Copyright 2016 Autodesk, Inc. All rights reserved



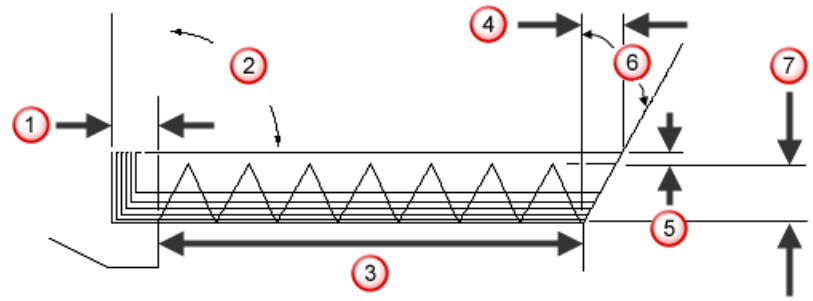
1 Clean up of previous steps

Threading tab (Turn)



© Copyright 2016 Autodesk, Inc. All rights reserved

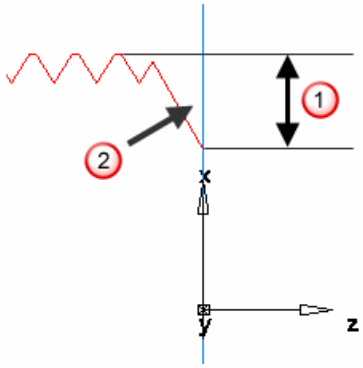
Some of the Threading attributes are shown in the diagram below:-



1. End Clearance
2. Withdraw Angle
3. Depth
4. Start Clearance
5. Clearance
6. Infeed angle
7. Height

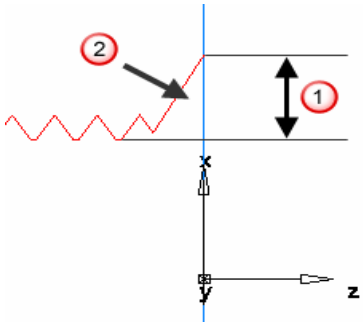
Rough turn and **Finish turn** — Select these options to automatically turn the piece down to the thread diameter. See How a thread feature is manufactured.

OD Thread Feature

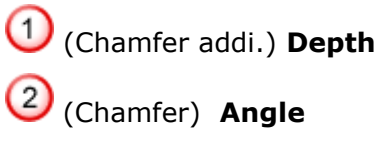


1. (Chamfer addi.) **Depth**
2. (Chamfer) **Angle**

ID Thread feature



© Copyright 2016 Autodesk, Inc. All rights reserved



(Chamfer) Angle: - Enter the angle of the chamfer in degrees (between **0** and **85**) measured from vertical. The default angle is **30**.

(Chamfer addl.) Depth: - Optionally enter additional depth for the chamfer. The default depth is the same as the **Thread Height**. The minimum value is **0**.

Relief groove: - A Thread feature has an option of cutting a relief groove at the end of the thread. Enter the groove parameters in the **Feature Properties** dialog on the **Strategy**.

Side wall angle: - This is the default angle of the relief groove of the thread.

Groove addl. depth: - The depth of the relief groove is the depth of the thread plus the **Groove addl. depth**. This prevents the threading tool from dragging on the bottom of the groove.

Start clearance: - This value is the position to which the tool traverses before engaging into the workpiece.

Chamfer Angle: - Enter the angle of the chamfer in degrees (between **0** and **85**) measured from vertical. The default angle is **30**.

Chamfer add depth: - Optionally enter additional depth for the chamfer. The default depth is the same as the **Thread Height**. The minimum value is **0**.

End clearance: - Enter the distance that the tool feeds past the end of the thread (into the relief groove) before retracting from the part's surface.

Infeed angle: - Enter an unsigned, incremental value from the positive Z axis.

Withdraw angle

This parameter specifies the angle between cross feed movement and the withdraw move. For both roughing and finishing passes this angle is measured against the roughing scanlines.

An angle of **90** retracts perpendicular to the roughing pass. An angle of **45** pulls back away from the part and the chuck. An angle of **135** pulls toward the chuck. The withdraw angle for finishing is not dependent on the shape of the feature curve. Even if no roughing pass is created, the withdraw angle for finishing pass is measured against the roughing scanlines.

The **Engage angle** and **Withdraw angle** are specified from the path (or extension of the path), relative to the side of the path that the tool is on, and the direction in which the tool is traveling.

OD/ID Depth %: - The depth of a thread is based on the thread pitch. The thread depth is calculated as **Depth% * pitch**.

Passes: - This is the number of steps to the bottom of the thread. Select either **Fixed** or **Calculated**.

Fixed: - This refers to a fixed rate of material removal. As the tool cuts further into the part, the area of contact of the tool increases. FeatureCAM reduces the infeed on each pass so that the tool loading remains constant. Enter the number of passes in **Count**.

Calculated: - the number of steps is calculated automatically by FeatureCAM.

© Copyright 2016 Autodesk, Inc. All rights reserved

Step1 is used to specify the incremental step for the first pass across the thread.

Step2 specifies the second pass and is used by the system to determine subsequent passes on the thread, reducing in depth until the **Min Infeed** value is reached.

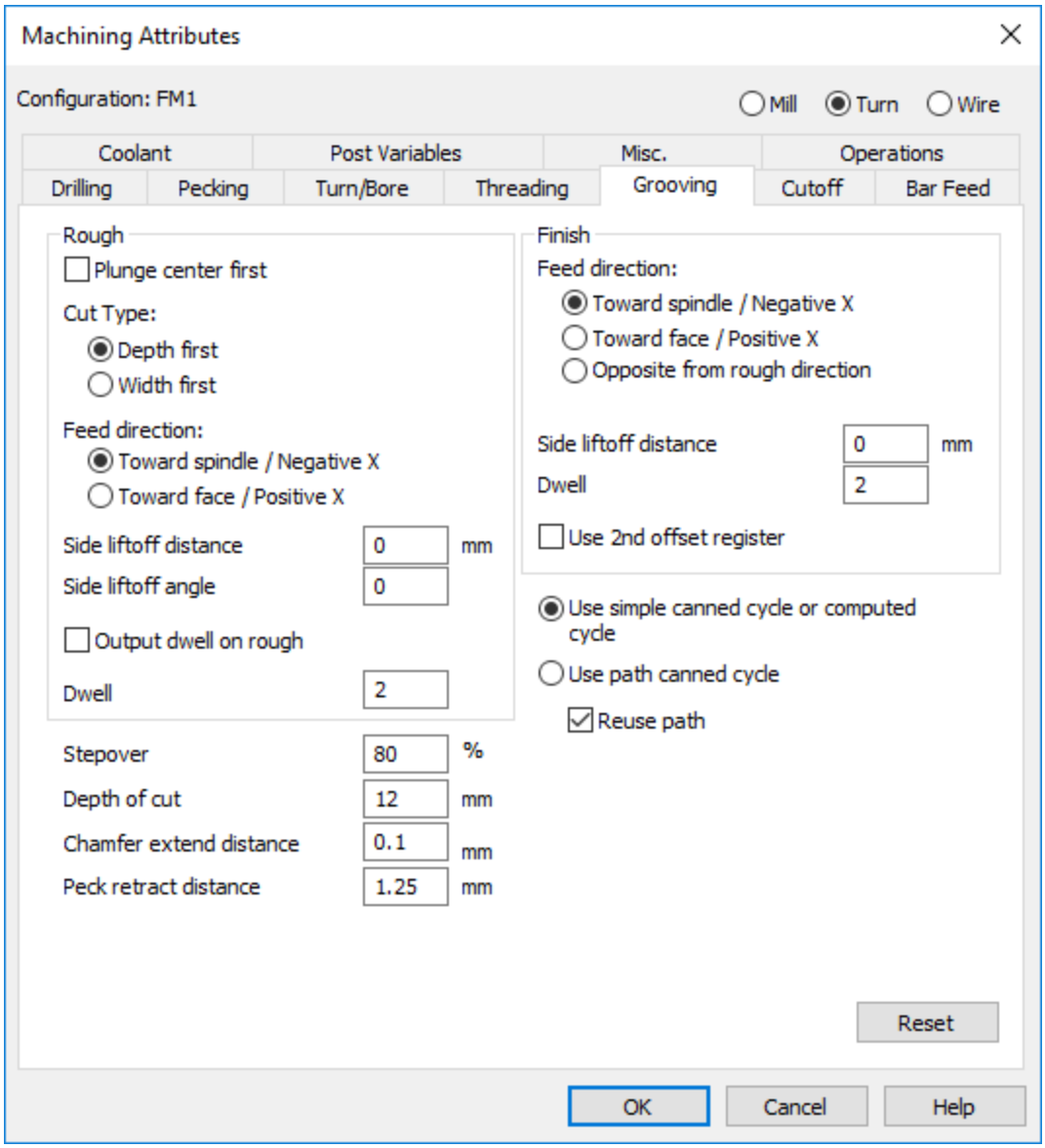
Spring passes: - A *spring pass* is a duplicate of the final threading pass. Enter the number of **spring passes** that you want.

Thread depth calculations - Select how you want FeatureCAM to calculate the **Standard Threads** for pre-populating the dimensions of a Thread (see page 1434) feature. Select from:

- **Use tool tip radius: -** This option takes into account the tool tip radius. For example the standard thread designation **M20 x 2.5** outputs:
- OD thread: **Thread Height** 1.894, **Major Diameter** 19.791
- ID thread: **Thread Height** 1.625, **Major Diameter** 17.519
- **Use tool tip zero radius** — this option outputs the nominal diameter and actual thread depth. This method is more commonly used in Europe. With this option selected, the standard thread designation **M20 x 2.5** outputs:
- OD thread: **Thread Height** 1.534, **Major Diameter** 20.000
- ID thread: **Thread Height** 1.534, **Major Diameter** 16.933

© Copyright 2016 Autodesk, Inc. All rights reserved

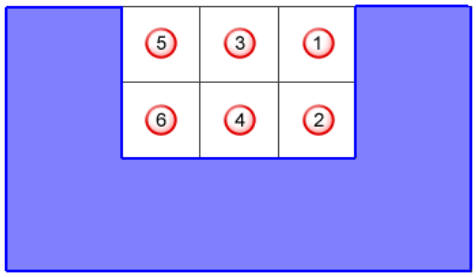
Grooving tab (Turn)



Plunge centre first: - For groove features, if this option is selected, the straight portion of the groove is roughed first and then the angled portions are roughed separately. If **Plunge centre first** is set, the red region of this image is roughed first and then the yellow regions are roughed.

Cut type

A turned groove can be cut in two different styles. The regions in the following figure are used to show the differences.



© Copyright 2016 Autodesk, Inc. All rights reserved

Width first - regions are cut in the order: - 1, 3, 5, 2, 4, 6

Depth first - regions are cut in the order: - 1, 2, 3, 4, 5, 6

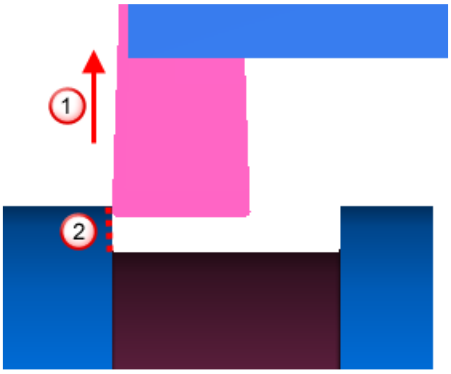
If **Depth First** and **Plunge Center First** are both enabled, then the groove is cut in the following order: 3, 4, 1, 2, 5, 6. If the groove were wider, the subsequent cuts would alternate from one side of the groove until the other until the entire groove is cut.

Feed dir: - This is the direction the tool feeds. The choices are either **Neg** (negative, -Z direction) or **Pos** (positive, +Z direction). For finish, you can also set this to **Opposite from rough dir**.

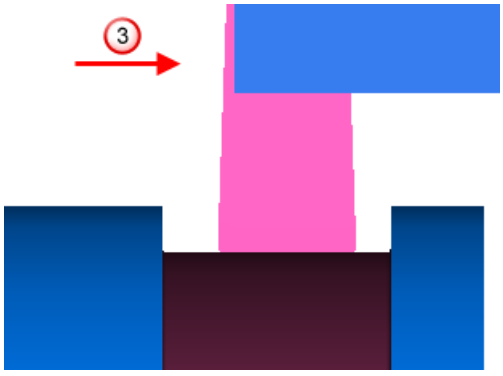
Side lift off dist: - Enter the distance to move the tool after a plunge cut, in the direction opposite to the cutting direction. This increases the tool's life and leaves a better finish on the part. This applies to a Groove feature. See also **Side liftoff angle**.

This part has a Groove feature, shown in pink:

The default behaviour is for the tool to lift off the part at 90, shown in the following image by 1, after each plunge cut. This results in tool contact with the uncut material, at 2, when the tool is retracting at a rapid feed rate along the X axis:



You can avoid this by using the Side lift off dist. attribute, to move the tool back along the Z axis 3, before lifting off.



© Copyright 2016 Autodesk, Inc. All rights reserved

Side lift-off dist. and **Side lift-off angle** are ignored for the retract move at the end of the first plunge. The lift-off move is performed at the plunge feed rate. If the groove is a round-bottomed groove, then lift-off is not used, even when specified.

Side lift-off angle: - Enter the angle to lift the tool off the part after each plunge cut. This increases the tool's life and leaves a better finish on the part. This attribute applies to a Groove feature.

Dwell: - The number of seconds the tool dwells after plunging during a groove roughing pass. It also applies to the roughing of the Cutoff chamfer.

Stepover %: - Enter the distance, as a percentage of the tool's diameter that the tool shifts to position itself for the next plunge cut. This value specifies the maximum stepover distance. If this value evenly divides the width of the feature, it is used. If it results in a final pass that is quite shallow, the cut widths are adjusted to result in even roughing passes.

Chamfer extend dist.: - This provides extra space for the tool so that the tool does not start on the material for the Groove finish pass.

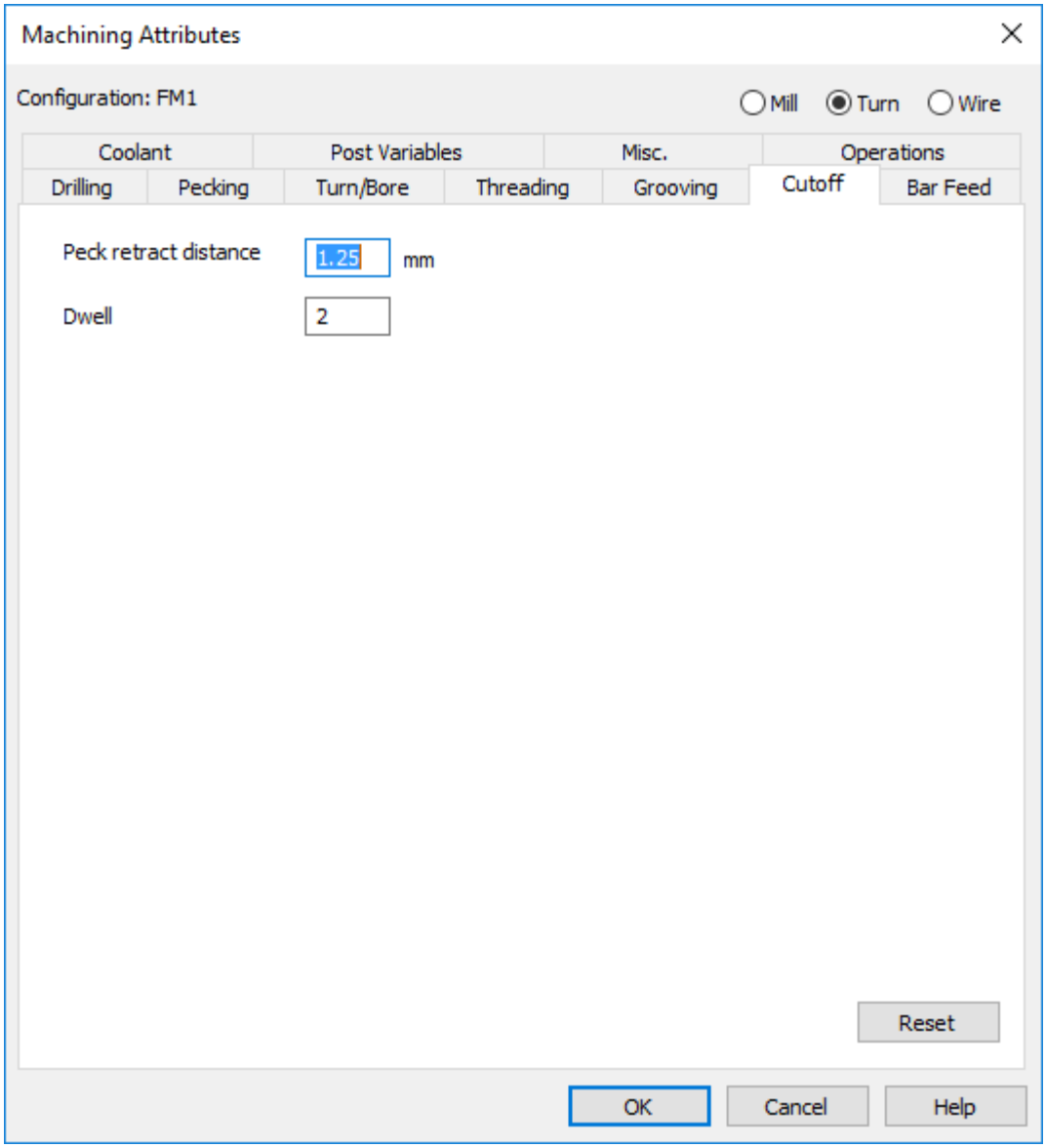
Peck retract dist.: - For Cutoff and Groove features, **Peck retract dist.** is the distance the tool retracts between plunges.

© Copyright 2016 Autodesk, Inc. All rights reserved

Cutoff tab (Turn)

Peck retract dist.: - For Cutoff and Groove features, **Peck retract dist.** is the distance the tool retracts between plunges.

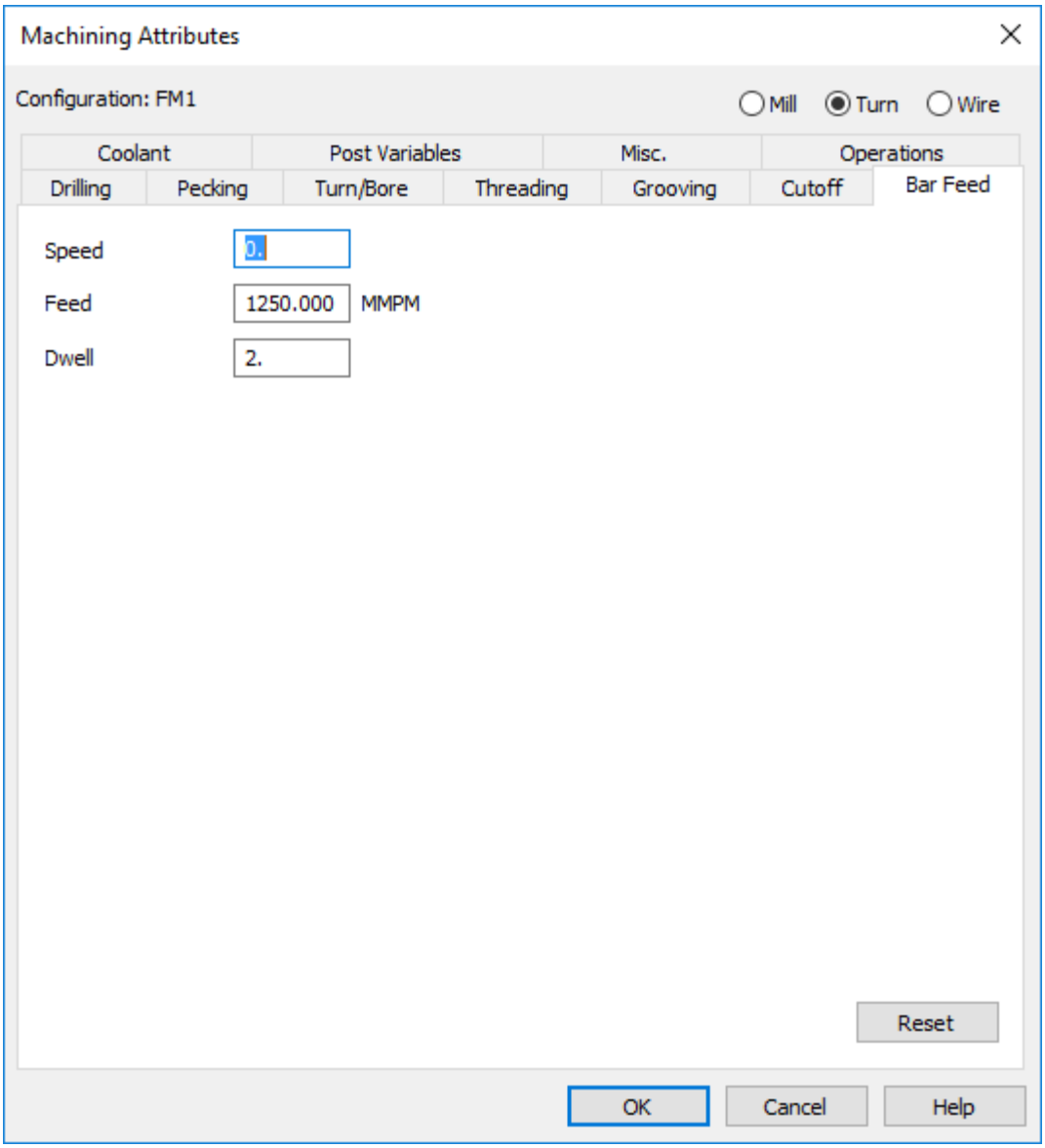
Dwell: - The number of seconds the tool dwells after plunging during a groove roughing pass. It also applies to the roughing of the Cutoff chamfer.



*You can set these attributes for an individual Cutoff feature in the **Feature Properties** dialog on the **Cutoff** tab.*

© Copyright 2016 Autodesk, Inc. All rights reserved

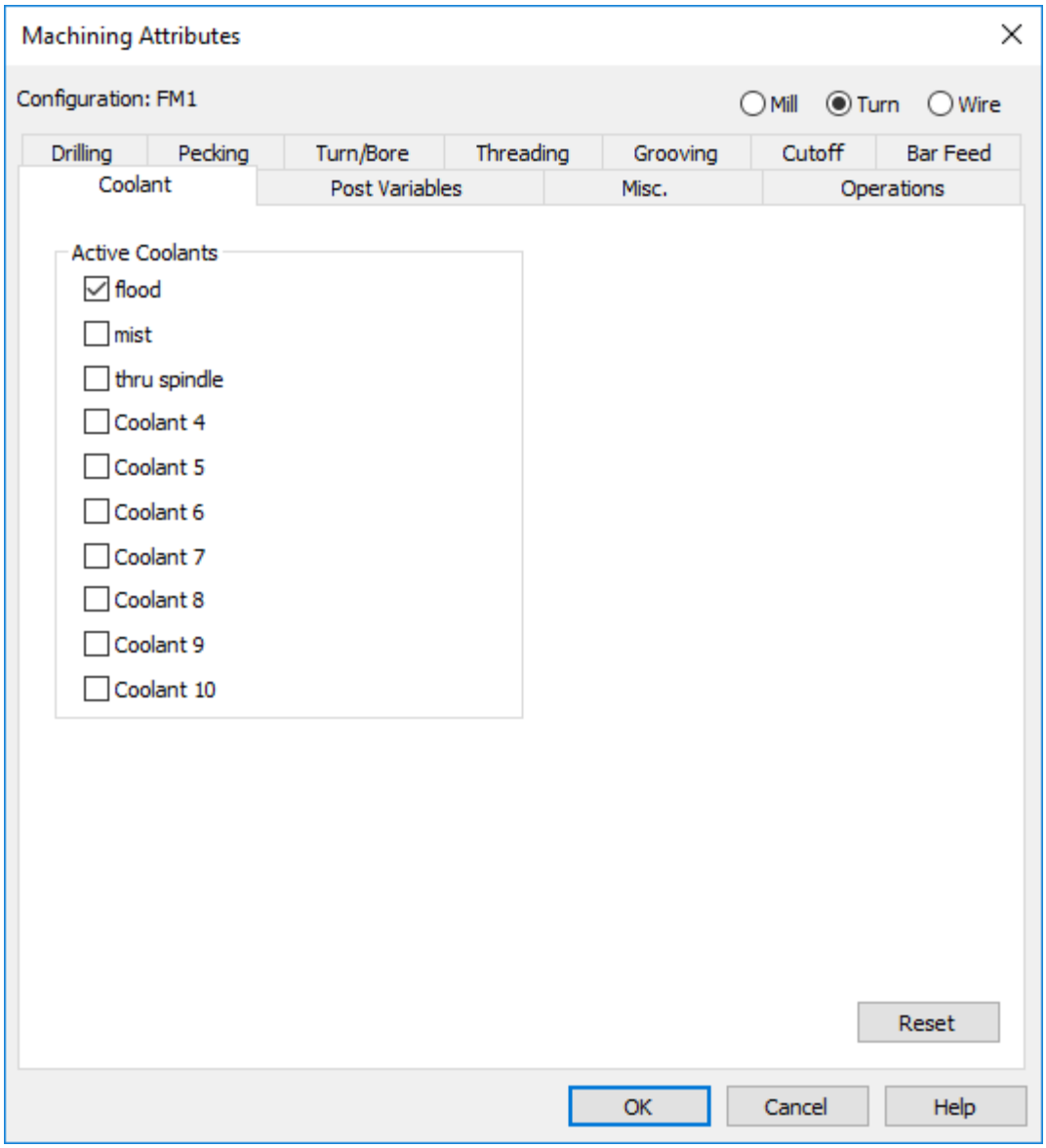
Bar Feed tab (Turn)



***Dwell**: - The number of seconds you want the tool to dwell after plunging during a groove roughing pass. It also applies to the roughing of the Cutoff chamfer.*

© Copyright 2016 Autodesk, Inc. All rights reserved

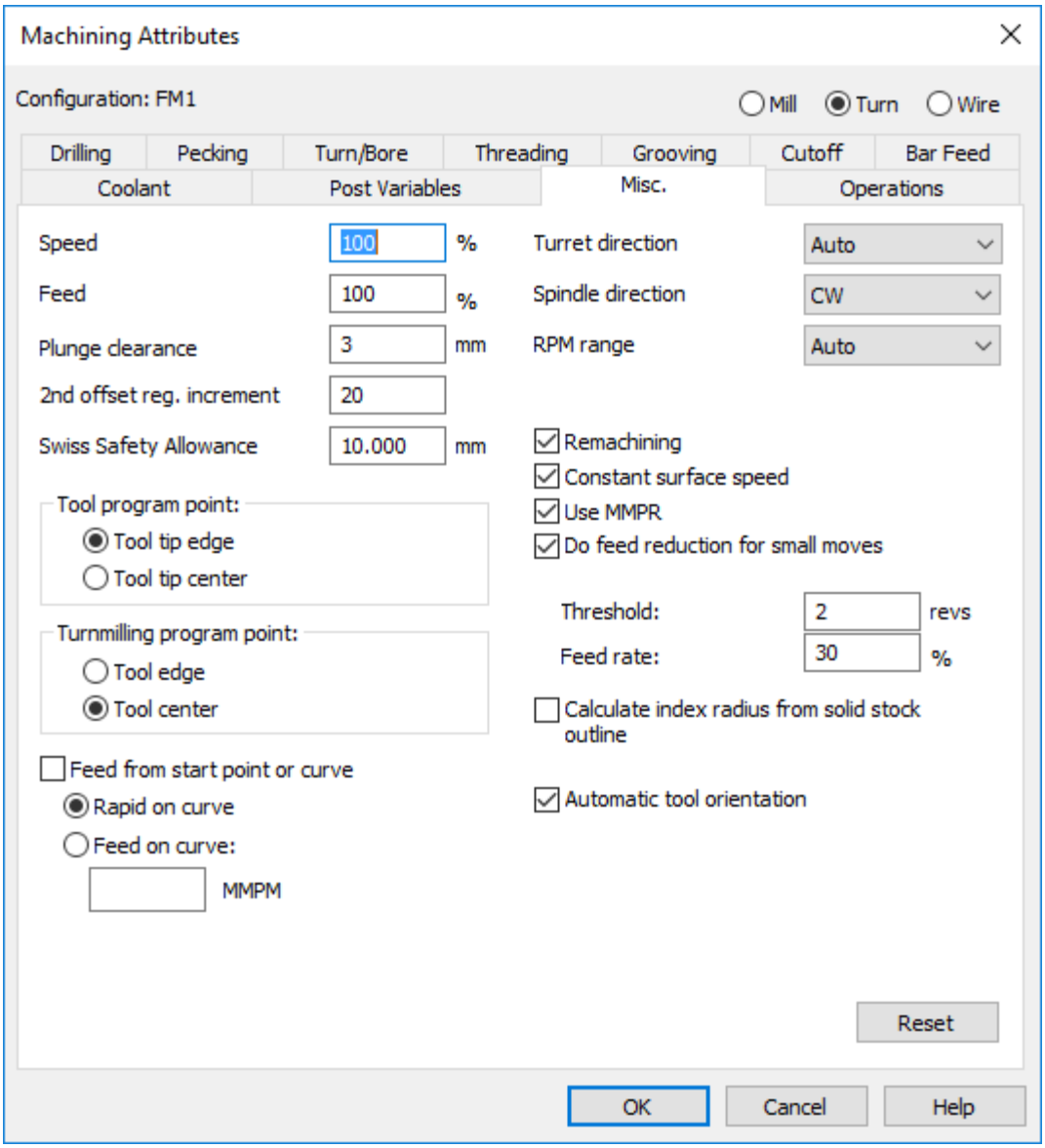
Coolant



*Use the **Coolant** tab of the **Machining Attributes** dialog to specify the default coolants to use for an operation.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Misc. tab (Turn)



Plunge clearance: - This is the distance above an operation at which the tool starts to feed. In the case of deep hole drilling, the drill retracts to this distance between pecks. For milling features, the default is to use the same value for roughing and finishing. As a result, the tool feeds from the top of a pocket to the floor before cutting. To make the tool feed down into the feature, set the **Plunge clearance** for an operation to a negative value, but make sure the value is above the floor of the feature.

The **Starting offset number for shared tool slots** is the first length offset register to use for tools that share the same tool slot.

Tool program point: - Specify the program point for turning tools, select from:

- **Tool tip edge: -** Select this option to adjust the tool program point by the insert radius in the NC code. In this case, adjust the tool's program point by the radius compensation on the **Program. Pt** tab of the **Tool Properties** dialog.

© Copyright 2016 Autodesk, Inc. All rights reserved

- **Tool tip centre: -** Select this option to adjust the tool program point by the insert radius at the machine. In this case, set the tool's program point **X Coordinate** and **Z Coordinate** to **0, 0** on the **Program. Pt tab** of the **Tool Properties** dialog.

- **Turn-milling program point: -** Specify the program point for turn milling tools, select from:
- **Tool tip edge: -** Select this option to adjust the tool program point by the tool radius in the NC code.
- **Tool tip centre: -** Select this option to adjust the tool program point by the tool radius at the machine.

Post vars. dialog

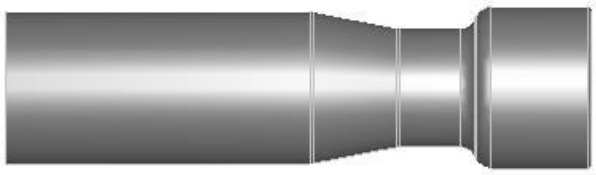
Feed from start point or curve — Select this option to use a feed move from the **Start point** or the end of the **Start curve**, to the beginning of the toolpath. If you are using a start curve, you have two further options:

- **Rapid on curve: -** Select this option if you want to use a rapid move along the start curve.

*Selecting **Feed from start point or curve** and **Rapid on curve** is the equivalent of selecting the **Feed from start** option on the **Turning** tab in previous versions of FeatureCAM.*

- **Feed on curve: -** Select this option if you want to use a feed move along the start curve. Enter the feed rate value(s). You can set the feed move as a single value, or use a comma separator to enter multiple values, for example **200, 100, 50**. If you enter multiple values, the feeds are applied to the curve segments in reverse order. So with these values, the last segment has a feed rate of **50**, the second last has a feed rate of **100** and any remaining segments have a feed rate of **200**. If you do not enter any feed rate values, FeatureCAM uses the default feed rate.

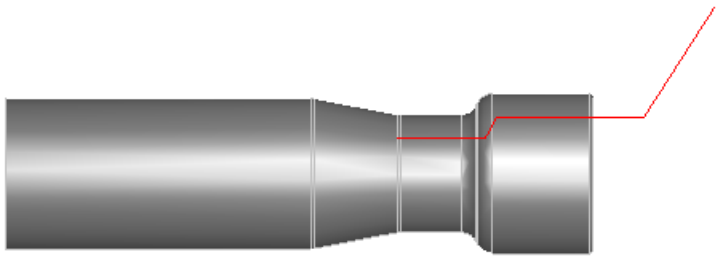
This example turned part has a Turn, Hole, and Groove feature:



A **3D simulation with 3/4 view** shows that the tool has to pass through a narrow channel to access the Groove feature:

To control the movement of the tool through that narrow channel, you can create curves and set them as Start point and End point on the Turning tab.

© Copyright 2016 Autodesk, Inc. All rights reserved



Turret direction: - We recommend that you leave this as Auto so that FeatureCAM can calculate the best direction for a particular operation. You can also explicitly set this option to CW (clockwise) or CCW (counter-clockwise).

RPM Range: - If your machine has explicit spindle speed ranges, you can set this option.

Some turning centres have gear boxes that set the maximum spindle speed of the machine. The **RPM Range** list sets the gear box to a specific maximum range. If you set **RPM Range** to a value of **1**-**4**, then the range is set explicitly. If **RPM Range** is set to **Auto** then FeatureCAM sets the range for you based on the following rules:

If the feature is a turned Hole or another turned feature without **Constant Surface Speed** set, then the range is determined based on the **Spindle Speed**.

If the feature is a turned feature with **Constant Surface Speed** set, then the range is determined based on the **Max RPM**.

Remachining: - This automatically sets the boundaries for subsequent operations that use the same curve. This minimizes air cutting and works between Turn features, Bore features and between Holes and Bore features. The same curve must be used in both features. ID features use the results of a previous turn drill operation if such a feature exists. The stock curve that results from the first operation is the result of undercut clipping with the tool geometry and nothing more than that.

Constant Surface Speed: - Select this option to specify the speed as a constant surface speed.

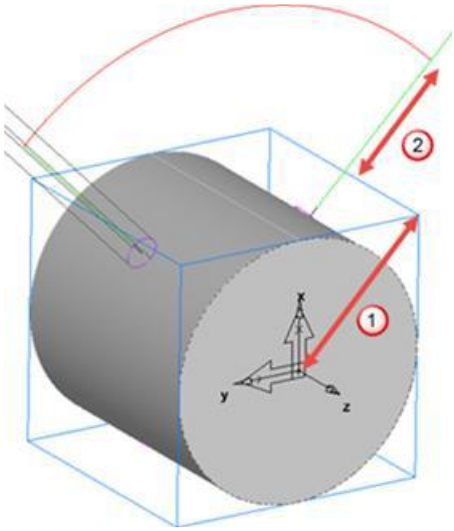
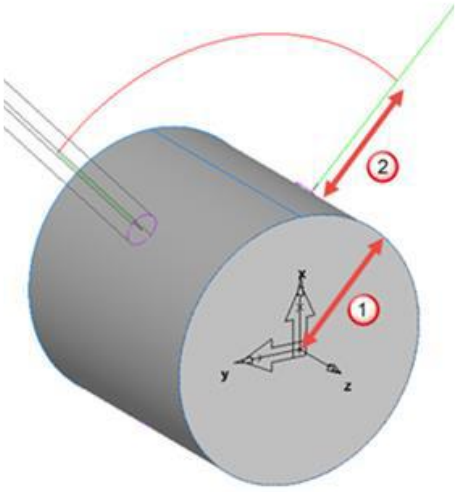
Use IPR/MMPR: - The default feed units are **IPM** (inches per minute) or **MMPM** (mm per minute). Select this option to use **IPR** (inches per revolution) or **MMPR** (mm per revolution).

Do feed reduction for small moves: - This attribute helps FeatureCAM cut small features properly. It is typically applied to small chamfers or small radii but affects any small move. If **Do feed reduction for small moves** is selected, then any move with fewer revolutions than the **Threshold**, is reduced by the **Feed rate %**.

Automatic Tool Selection:- This option set the same tool in the appropriate direction for OD Turning and ID Turning, with respect to Main Spindle/Sub-Spindle.

© Copyright 2016 Autodesk, Inc. All rights reserved

Calculate index radius from solid stock outline: - *Select this option to determine the index height directly from the stock solid, instead of calculating it above a square bounding box.*

Calculate index radius from solid stock outline off	**Calculate index radius from solid stock outline on**
	
1 Distance to bounding box. 2 **Z rapid level** attribute value value	1 Distance to stock boundary 2 **Z rapid level** attribute

Turret location

The Turret location is kept in the *.cnc file, because there may be multiple turret locations.

To change the locations:

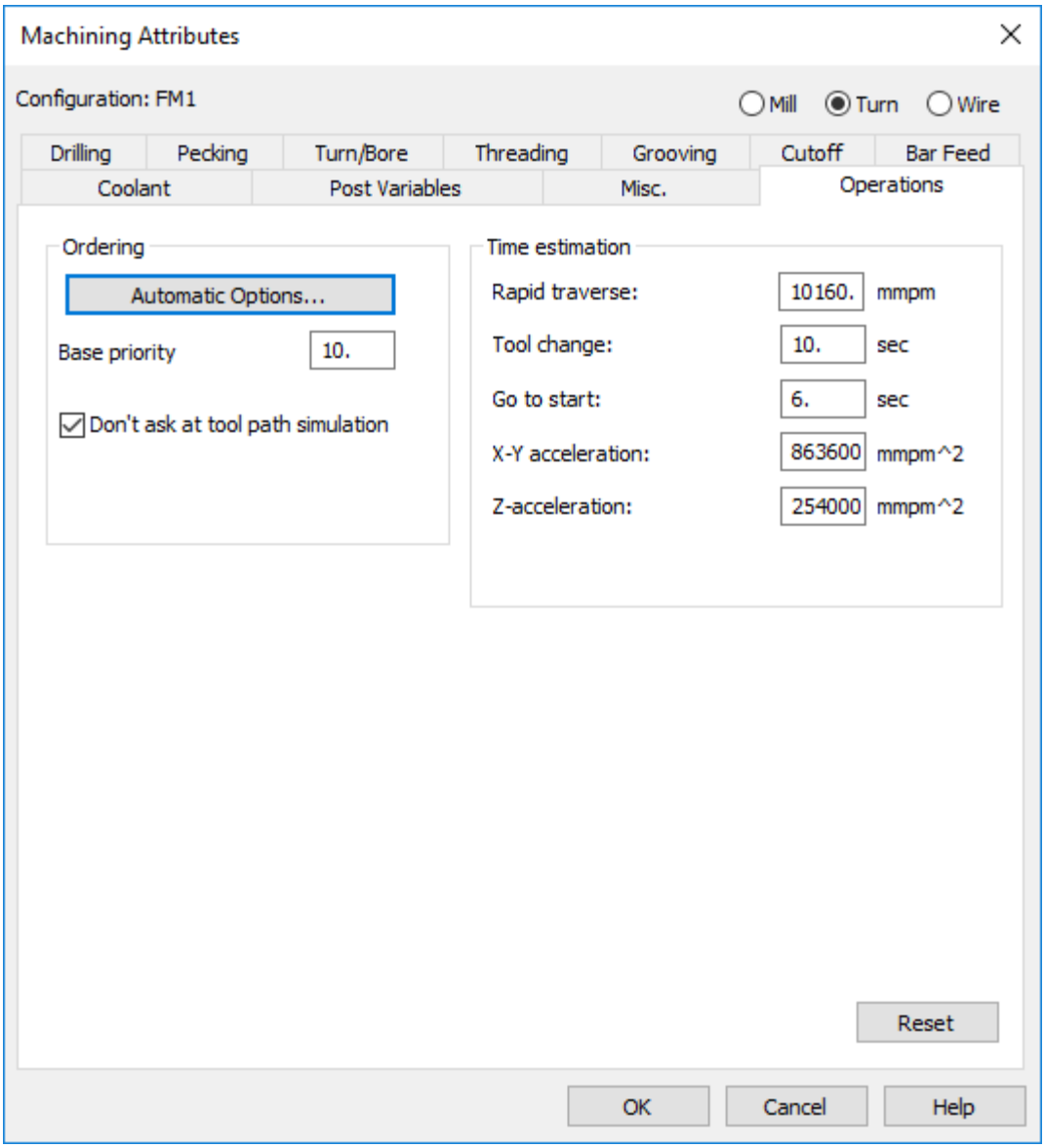
Select Manufacturing > Post Process from the menu.

In the Post Options dialog, click the Turn/Mill tab.

3 Click the Edit button and **XBUILD** opens. Select CNC-Info > Turrets from the menu.

© Copyright 2016 Autodesk, Inc. All rights reserved

Operations



© Copyright 2016 Autodesk, Inc. All rights reserved

Base priority

Features are sorted by their Base priority to determine the order in which they are manufactured. For features that have the same Base priority value, the system uses the Automatic Ordering settings.

To ensure that an individual feature is cut before anything else, you can set its Base priority attribute. All features have a default Base priority of **10**. To ensure that a feature is manufactured first, set its priority to a lower value. To make a feature last, set its priority to a higher value. For example, if you set the Base priority of a pocket to **8**, its roughing pass is the first operation performed, its finish pass is second, and the rest of the operations are ordered according to the Automatic Ordering or Manual Ordering settings.

Although you can specify the order of every feature by priority, you should not do so casually because you lose the automatic optimization sequences built into the system and it is harder to maintain or change the part.

The order of operations in the document is controlled by the operation-level Priority attribute. If you use the Op List to drag-and-drop operations to the order you want, the Priority is updated automatically.

Don't ask at tool path generation**: -** This is a toggle for whether you are prompted with the dialog when you run a simulation. The Ordering dialog settings override the operation defaults set on this tab.

Time estimation attributes**: -** Set these attributes to fit the behaviour of your particular machine. These attributes affect the machining time estimates printed in the operation sheets.

Rapid traverse: - This is the feed per minute of rapid moves.

Tool change: - This is the time in seconds it takes to change a tool (not including the rapid to get to tool change location).

Go to start: - This is the time it takes for the tool head to move to the start location and the spindle or tool head to come to a stop.

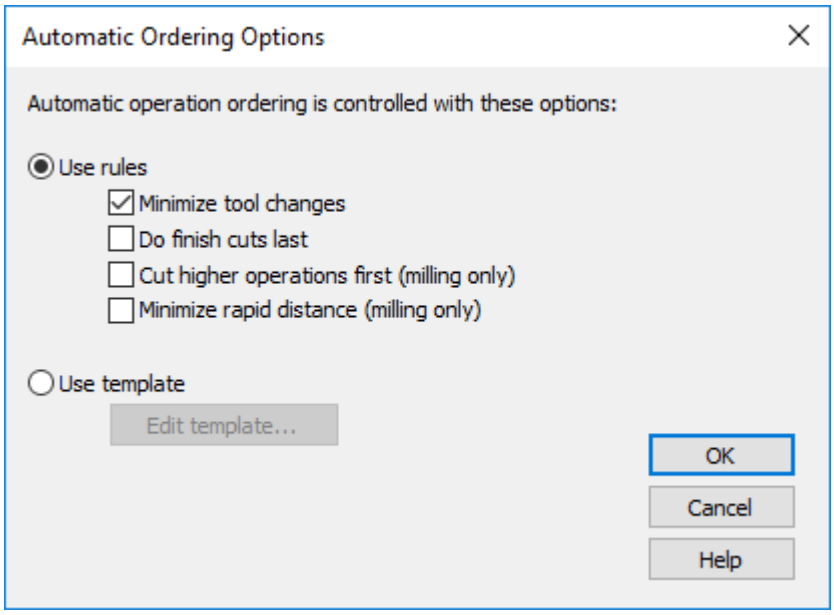
X-Y acceleration: - This is used in the formula to calculate the time for a particular tool move.

Z acceleration: - This is used in the formula to calculate the time for a particular tool move.

These attributes are the same as on the Operations tab for Mill with an additional option in the **Automatic Ordering Options** dialog, **Use template**:

© Copyright 2016 Autodesk, Inc. All rights reserved

Use Rules



Automatic Options (TURN)

These attributes are the same as on the Operations tab for Mill, with one additional option in the Automatic Ordering Options dialog.

Use rules:

Minimize tool changes: - This option groups operations together that use the same tool. This saves time for you by eliminating or reducing needless tool changes. You must select this option if you want to generate hole macros in the NC code.

Do finish cuts last**: -** This option moves the finish milling operations to the end of the Setup without altering the order of the finishing operations? If you want to perform all rough milling operations before finish milling operations, select this option.

Cut higher operations first: - This option affects only milling Setups. Select this option to mill the features from the top of the stock first and work toward the bottom. If you deselect this attribute, you should carefully graphically verify the toolpath before cutting your part.

Sorting by Z coordinate is controlled by the Cut higher operations first attribute. If you deselect this attribute, graphically verify the toolpath before cutting your part.

Minimize rapid distance: - This affects only milling Setups and is the only ordering option that changes the order of features specified in the part view. Minimize Rapid Distance moves to the next closest feature that uses the same tool as the last operation. You must deselect this option if you want to generate hole macros in the NC code.

If you have selected all of the optimization options, the manufacturing order for operations that were derived from different features is determined like this:

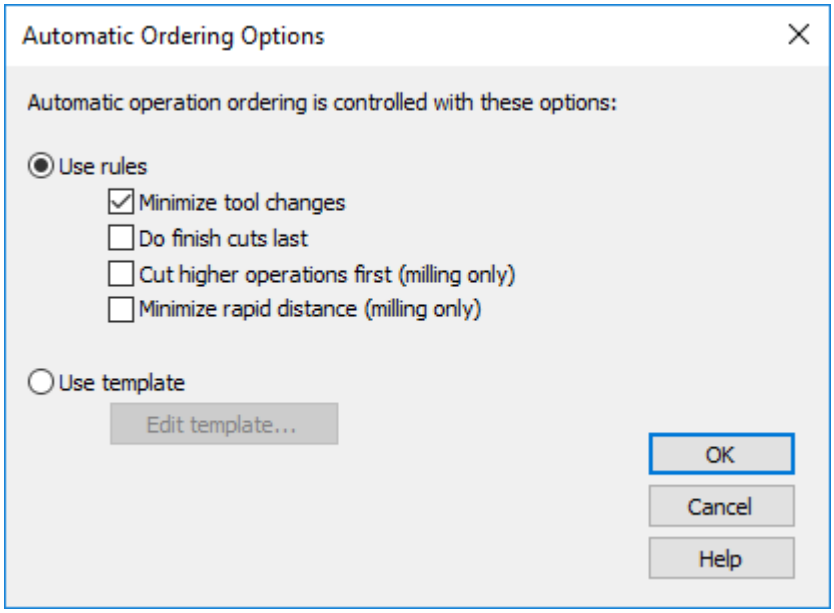
The operations are sorted by their top Z coordinate.

Among operations with the same top Z coordinate, operations are grouped by the tool with which they are cut.

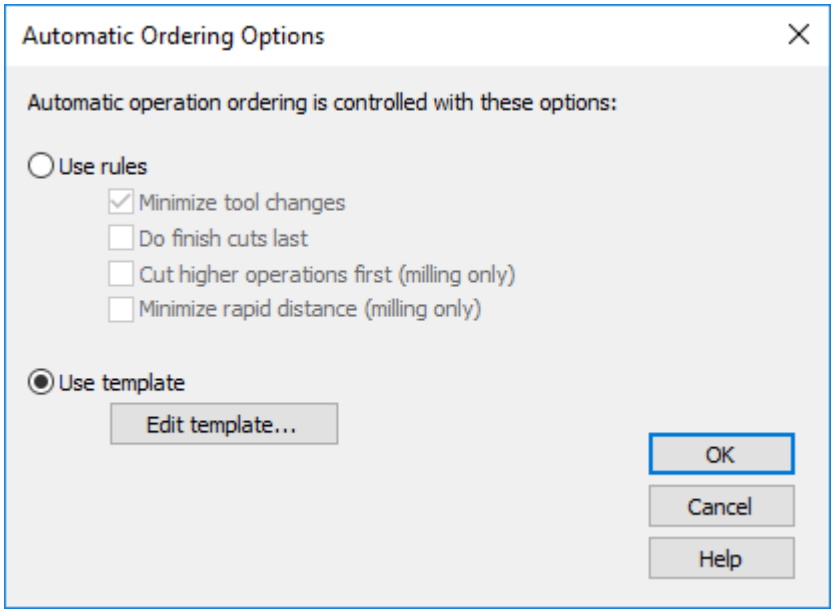
© Copyright 2016 Autodesk, Inc. All rights reserved

After an operation is cut, FeatureCAM moves to the next operation performed with the same tool that is the closest to the current operation.

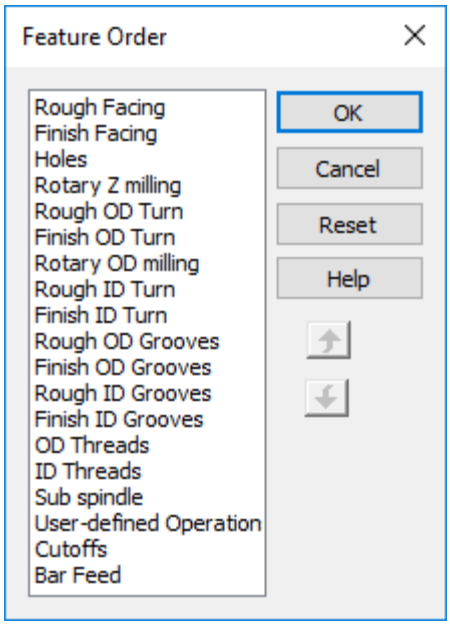
Use Template



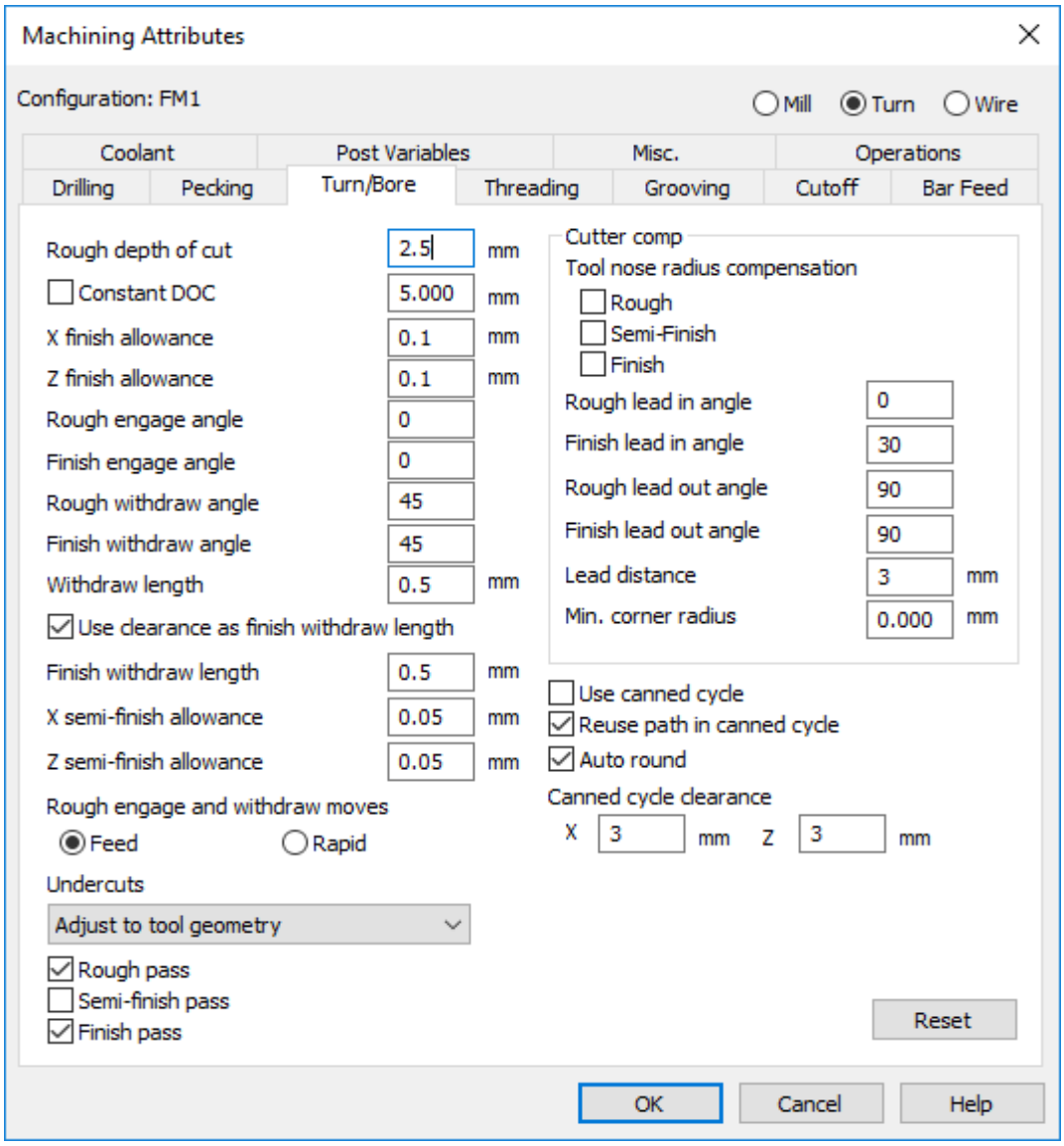
*This attribute is applicable only to turning setups. If you select **Use template** then the order of operations are determined by the outline of operations listed in the Feature Order dialog. Click **Edit template** to open the Feature Order dialog. See turn operation order.*



© Copyright 2016 Autodesk, Inc. All rights reserved

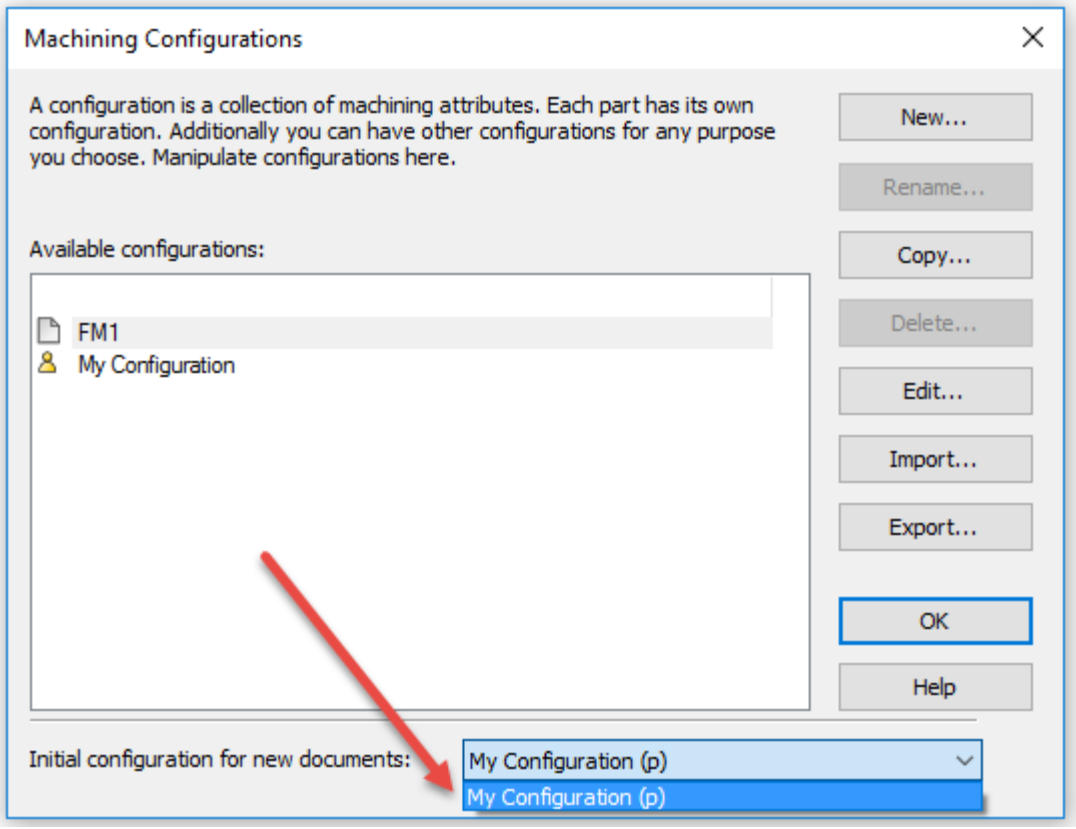


What are Machining Attributes?



© Copyright 2016 Autodesk, Inc. All rights reserved

- When an option in the Machining Attribute is selected, a box is checked or a value is entered, it becomes a default. This means that **FeatureCAM** uses this setting every time a feature requiring that particular setting is selected in a program. There is no input required from the user during programming, as it is now automatically set. However, it may be overridden and changed in the feature as needed.
- Tabs are used to categorize the different operation types and necessary attribute settings.
- Some selections are check boxes, some are numerical values, and some are specific functions. The majority of these show up in the feature that they affect, but some settings do not. These settings may only be changed in the attributes. This is the reason it is so important to know where these settings reside, what their function is, and where they are located in the features.
- To apply the changes you have made, in the Attributes for the document, to become the defaults for any new documents they must also be changed in the Machining Configuration that opened with the file. Once set, and the next time a new document is opened, those changes will then take effect as the default settings.
- Open a New Turn Document and click **Manufacturing** on the top menu and select **Machining Configurations**
- Select the **document you just opened**. Usually there is only one document open but multiple documents may be opened. In this case, you would see multiple documents in the list
- Click **Copy** and **select the My Configuration** from drop down menu. These defaults load into the new document



- To copy the changes from the document's configuration into another configuration to be set as the defaults:

© Copyright 2016 Autodesk, Inc. All rights reserved

- Select the **My Configuration** in the window. This is the configuration you want to **copy the attributes to**
- Click **Copy** and select the document from drop down menu in the Copy Configuration dialog
- Select **OK.**
- This reverses the copy and places the changes, if we had made any, from the document Attributes into the configuration selected in the window of the Machining Configurations dialog.

Creating Machining Configurations

- Open a New Turning Document. Click **Manufacturing** on Menu bar then Machining Configurations
- Click on New and type the word Aluminium in the field. **OK**
- Repeat these steps to create a Stainless configuration. **OK**
- Double click on **Stainless** in the window. Select the **Turn/Bore Tab** in the Machining Attributes dialog. Change the **Rough depth of cut** to **2.5mm**

  The Rough Depth of cut for rough turning was 5mm per pass. It is now 2.5mm to account for the harder to machine Stainless steel.
- Repeat the steps to change the **Aluminium** Configuration's Rough **depth of cut** to **6mm** this may be more than you would want to cut, even in Aluminium, so put whatever value makes you comfortable.

  Note: You now have three separate Machining Configurations. Each has approximately 100 settings.
- Select the Document you have open in the Machining Configurations dialog at the top of the window
- Copy and select **Aluminium** from the **Copy Configuration** dialog
- **OK** twice. These settings will be used in the new document
- Open an existing file named **Turning Attributes.fm**, from the Beginning Data folder, and run a 3D simulation

  The Turning tool cuts in depth increments of 5mm to the shape of the curve, to rough the material to shape. The default attribute specifies the Rough depth of cut to be 5mm per pass.
- Click **Manufacturing** then **Machining Configurations**. Select the document in the window on the left named **Turning Attributes.fm**
- Press the **Copy** button
- Select **Stainless** from the drop down list in the **Copy Configuration** dialog
- Press **OK** twice
- Using the same feature and by only changing the Machining Configuration the Attributes automatically change to the Stainless settings.
- Run a 3D simulation

© Copyright 2016 Autodesk, Inc. All rights reserved

The setting for Stainless is 2.5mm Rough depth of cut, so now the tool makes ten passes at 2.5mm depth of cut instead of five at 5mm with no further input from the user.

The Attributes

- We should take a look a bit more closely at what attributes are where and what settings are available in each grouping, or tab. Special attention will be drawn to the more common/important settings.

Drilling Tab

- The drilling tab has settings such as whether or not a spot drill is used, and other controls for that spot drill. There are some settings for how operations are ordered in a hole feature and active, such as Reaming before chamfering.
- Most importantly there is a tap "cycle" setting that will be loaded as the default when a thread is cut: Floating, Rigid, or Deep tap.

Pecking Tab

The basic peck settings for both drilling and tapping operations.

Turn/Bore

The settings in the Turn/Bore tab are the most commonly used settings in FeatureCAM's turning attributes.

Settings such as Rough depth of cut, the semi-finish, and finish allowance settings and how undercuts are handled highlight the settings used in these feature types.

Additionally, cutter comp, leads, and canned cycle settings are contained herein.

Threading

Material removal settings such as rough and finish turn in preparation to cut the threads are key settings.

Relief settings are important in threading.

Thread depth percentages and calculations also reside in this area.

Grooving

The settings here, obviously address the way grooving features are machined.

Key settings are the default way you would prefer to handle the rough cuts with or without plunging and how those plunges are cut.

Stepover and depth of cuts as well as canned cycles usage are located here.

Cut-off and Bar Feed

The Cut-off and bar feed settings are self-explanatory.

© Copyright 2016 Autodesk, Inc. All rights reserved

Miscellaneous

The basics of turrets and spindles, as well as RPM and coolant types and offset registers are a part of the miscellaneous settings.

CSS (constant surface speed) and the use of MMPR (Millimetres per revolution) by default are set here.

Remachining is addressed a little later on in this module.

Operations

The way operations are handled in this tab, such as the way the automatic ordering of features is done.

An integral part of **FeatureCAM's** tools is the time estimation output in the results window. Many users use this as a base for quoting their customers for manufacturing of the parts. Key settings are your actual rapid traverse speeds and time taken at the machine tool for each tool change, among others.

Help Contents

Using the Contents, Index, or Search tabs is a great way to learn more about FeatureCAM and how to use it efficiently. The settings are found and explained in the Help files.

- Open the file named **Help File.fm**

  *In this file you will notice there is a feature already created, **turn1**. There are a couple settings in the attributes the user can take advantage of to fine-tune the Toolpath.*
  Use the help file to search a few of the attributes to decide which, if any, of the attribute settings available might be some changes you want to make and apply to the final Toolpath.

- Double-click on **turn1** in the Part View of the Toolbox to see the feature properties
- Select **Rough** operation and the **Turn** tab

  You see several attributes in the window of the dialog that may not make any sense to you at this point. Using the help file is a great way to learn what these settings do and how to apply them to this operation of the feature.

- Click **Help** from the File Menu and **Contents** to access the help file

  *A New Window opens up that allows the user to use three methods in the FeatureCAM Reference Help section to learn more about **FeatureCAM**, including attributes.*

- Select the **Search** tab and type in the words "**start point**."
- Press the **List Topics** button

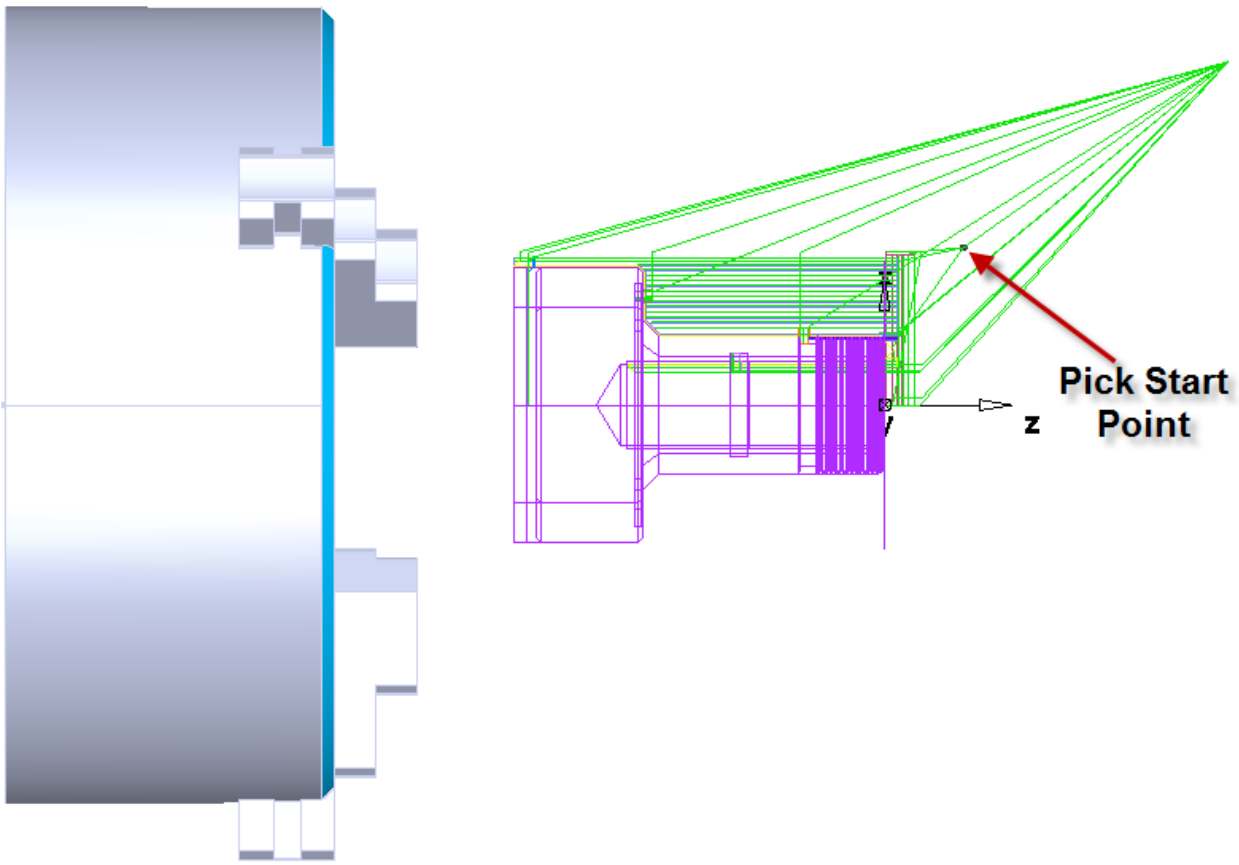
© Copyright 2016 Autodesk, Inc. All rights reserved

The Select topic list shows all the topics where the key words Start and/or Point appear. Since we know that setting exists in the Turning tab of our feature we will want to choose that listing.

- Double click the second item in the list, **Turning tab (TURN)** to update the window at the right of the Reference Help window to see how "**Start Point**" applies in relation to the Turn tab of an operation
- Run a **Centreline** Simulation to see where the tool starts without any changes made

HINT: Use Single Step instead of play to see the initial move to the start point location. As shown on the next page.

- In the rough operation Turning tab highlight **Start point =** and press the **Pick** Arrow to pick a new position for the tool to start, see below



- **Apply** and **OK**
- Run a **Centreline** Simulation, single step to see the change to the Toolpath

Notice the tool starts where you picked it to then rapids over to start the cut.

Try a few other settings after referencing the Help file.

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Turning Tool Reference Guide 2
Tooling 2
Overview of tooling 2
Tooling database 3
Turning Tool Orientation 5
Tool Groups. 7
Insert Shapes 7
Inserts for General Turning 8
CNMG Insert 9
DNMG Insert 10
Insert Shape 13
Holder 14
Orientation 14
Holder Drawing 16
Overrides 16
Coolant 17
Feeds/Speeds 17
Turning tool operation usage 18
Lathe - Bore Tool Properties example. 20
Holder Information 20
Boring Orientation 22
Using an insert drill to drill and bore in the same program 23
Lathe - Bore Programming Point 24
Lathe – Groove tool Properties example 24
Lathe - Cutoff Tool Properties example 25
Lathe - Groove/Cutoff Holder tool Properties example 26
Lathe – Thread Tool Properties example. 27
Adding a Solid Model Turning Tool 28
Mini-turrets 32

© Copyright 2016 Autodesk, Inc. All rights reserved

Turning Tool Reference Guide

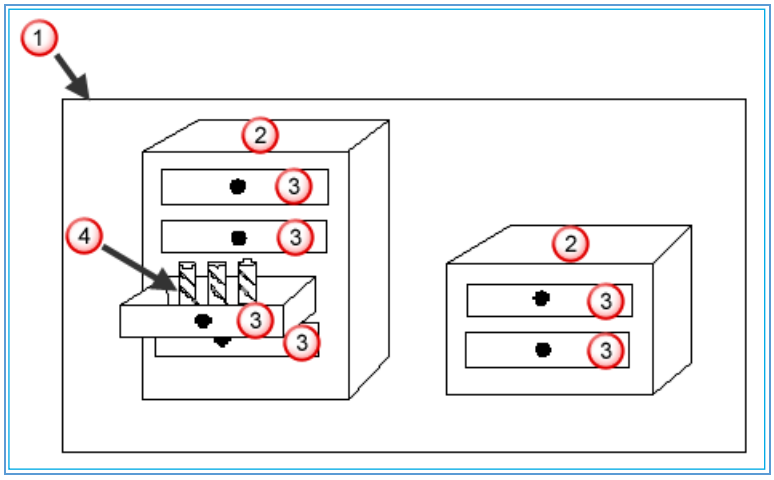
It is possible to create Tool libraries in **FeatureCAM**. These are called Tool **Cribs.** The main reason for creating Tool Cribs is because in any machine shop there is normally a finite amount of tooling available and by creating a custom Tool Crib, **FeatureCAM** will know what tooling is available to choose from.

Tooling

FeatureCAM has extensive tooling databases and automatically selects tools for each manufacturing operation.

Overview of tooling

For each manufacturing operation created for the features of a part, a tool must be selected from the internal tooling database. This database is broken into separate tool cribs to represent individual collections of tools that your organisation might have. The image below shows the structure of the tooling database.



1. Tooling Database
2. Tool Cribs
3. Tool groups for example drills, taps, Endmill.
4. Tools

FeatureCAM comes with three different built-in tool cribs when selecting both. This includes both Metric and Inch tool databases. The **Basic** crib is the default crib that contains standard available tools that most shops own. The **Tools** crib is a large crib containing hundreds of tools. This crib is most often used as a source to copy from into custom tool cribs or into the basic tool crib. **BasicMetric** is purely metric tools for the European market.

Only one tool crib is available at a time and all tools are selected for a part from only the current tool crib. Tool cribs contain individual tools and they are classified into tool groups such as **drills**, **end mills**, and **boring bars**. You cannot create new tool groups, but you can create new tools to reflect the specific tools that your shop owns.

© Copyright 2016 Autodesk, Inc. All rights reserved

Each feature type has **rules** for tool selection.

Double-click a feature in the graphics window or the **Part View** panel to display the **Feature Properties** dialog, then select an operation from the Tree View on the left of the dialog. On the **Tools** tab of the **Feature Properties** dialog, you can: See which tool is currently selected to perform the operation; or Change the tool used to perform the operation.

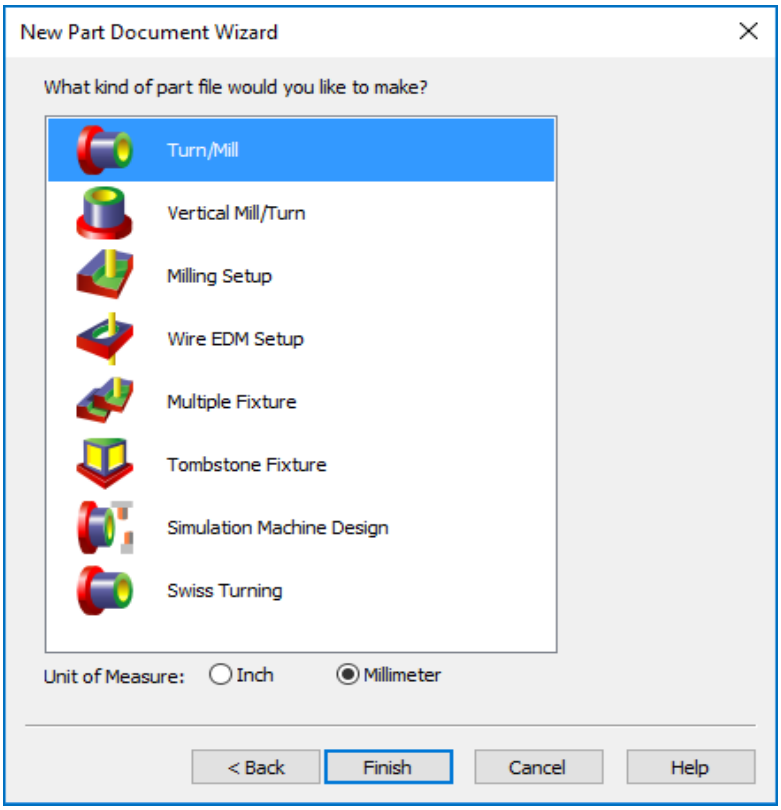
Tooling database

The tooling database defines the set of tools from which FeatureCAM selects tools to perform manufacturing operations. These tool sets are called cribs. FeatureCAM comes with two standard tool cribs, the **Tools** tool crib and the **Basic** tool crib. The **Tools** tool crib is a comprehensive tool crib that contains more tools than your shop probably owns. The **Basic** tool crib contains a smaller set of tools such as HSS endmill and standardly available drills. By default FeatureCAM is set to use the **Basic** tool crib. You should modify the cribs to reflect the tools your shop has. You can create a crib containing tools you commonly use, which can simplify setting up a part for machining.

The name of the active tool crib is displayed on the status bar. To change the current tool crib you are currently using, click the crib name on the status bar. A list of tool cribs is displayed that you can select from.

Please note this document is part reference, with information taken from the help file.

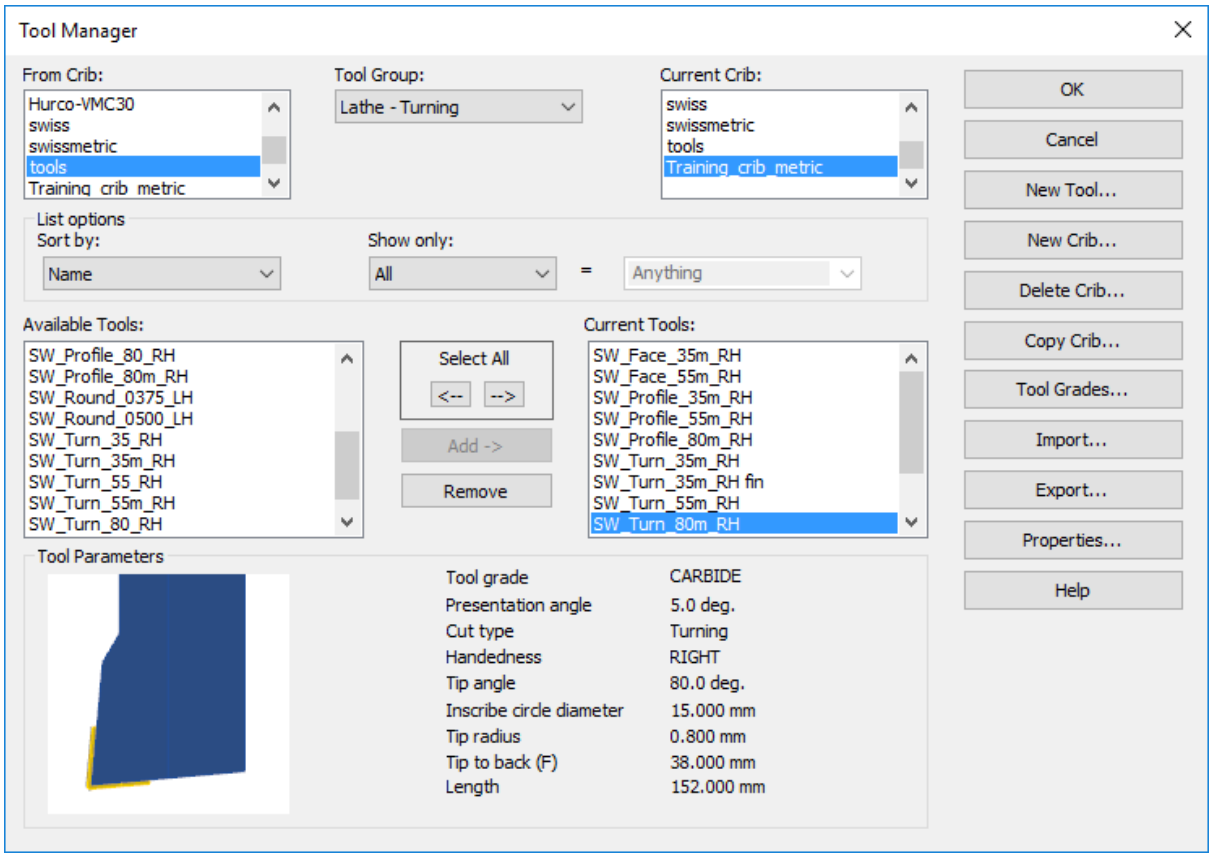
1. **Open** a new **FeatureCAM** part by selecting **File, New.**
2. The following Menu shown on the next page will appear. Select **Turn/Mill** Set-up and click **OK.**



© Copyright 2016 Autodesk, Inc. All rights reserved

*To create or modify a tool crib, you must have a part file open, then select **Manufacturing > Tool Manager** from the menu to display the **Tool Manager***

3 Go up to the **Manufacturing Menu** and down to **Tool Manager**. This will open the following form.



*There are three tool Cribs loaded as standard. **(Basic, BasicMetric and Tools)** and these can be used as a starting point when the user first starts to look at setting up their own Cribs. Another Crib is available **BT40-Training-Crib-Metric** on request or copied from the training material supplied with this course. This can be loaded as an **.XML** file into **FeatureCAM** via the Tool Manager.*

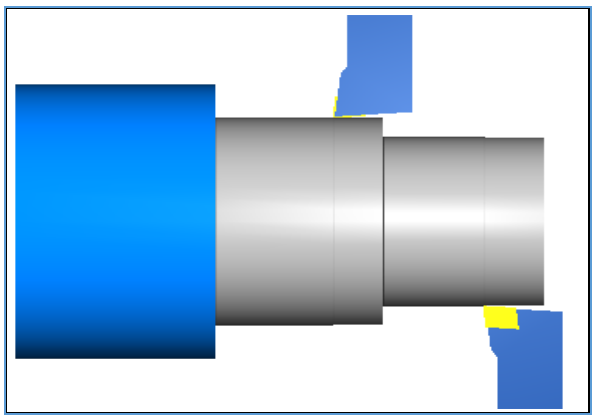
The window is divided into separate areas and at first glance this can look a little intimidating, but by applying some simple rules it becomes much easier to work with.

If the user divides the form in two down the middle of the form. The left side is used to select which Crib to copy from whereas the right hand side of the Form relates to the Crib that is current or the working Crib.

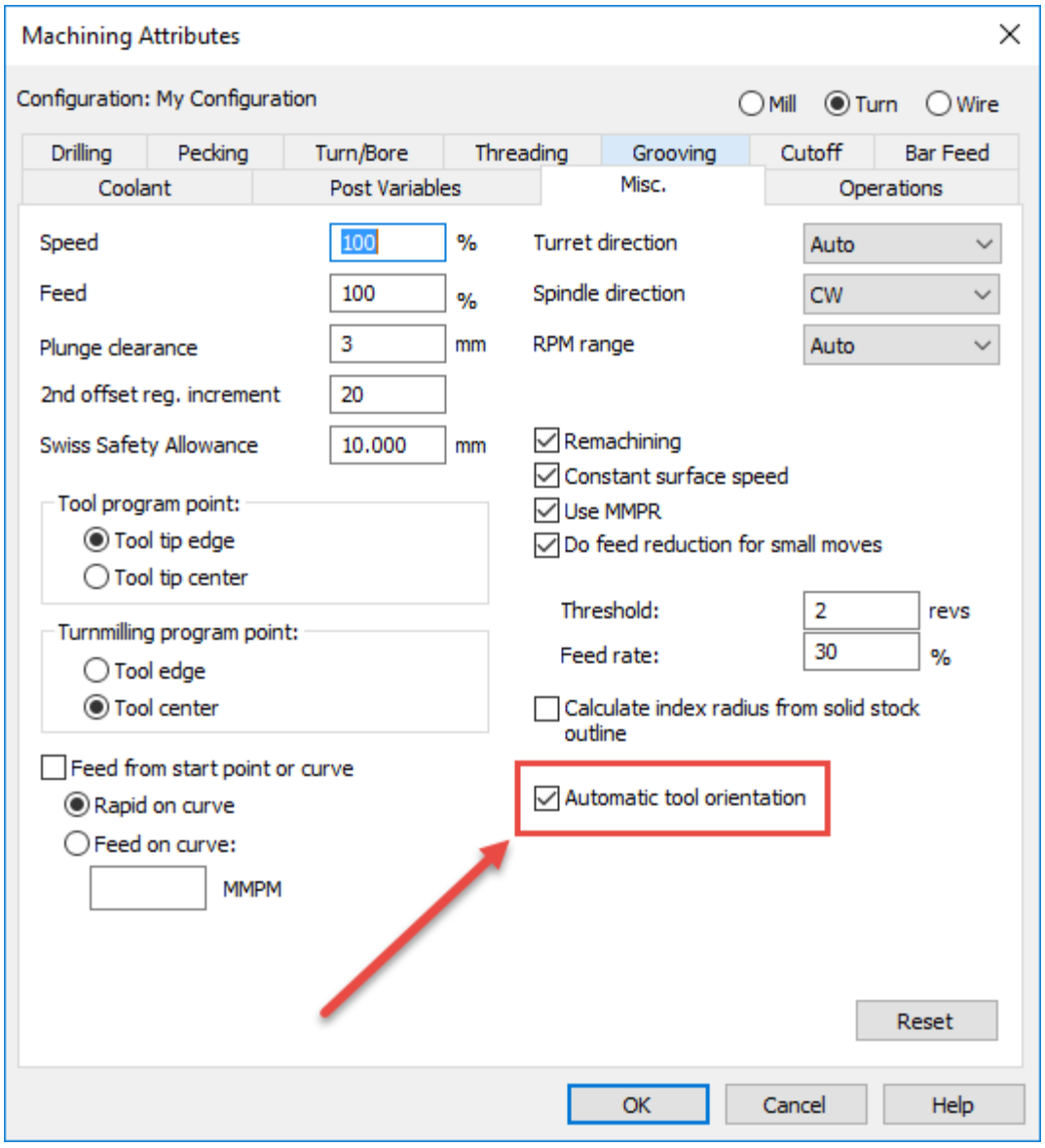
© Copyright 2016 Autodesk, Inc. All rights reserved

Turning Tool Orientation

You can now use a turning tool in multiple orientations without having to create a different tool in the database for each orientation.



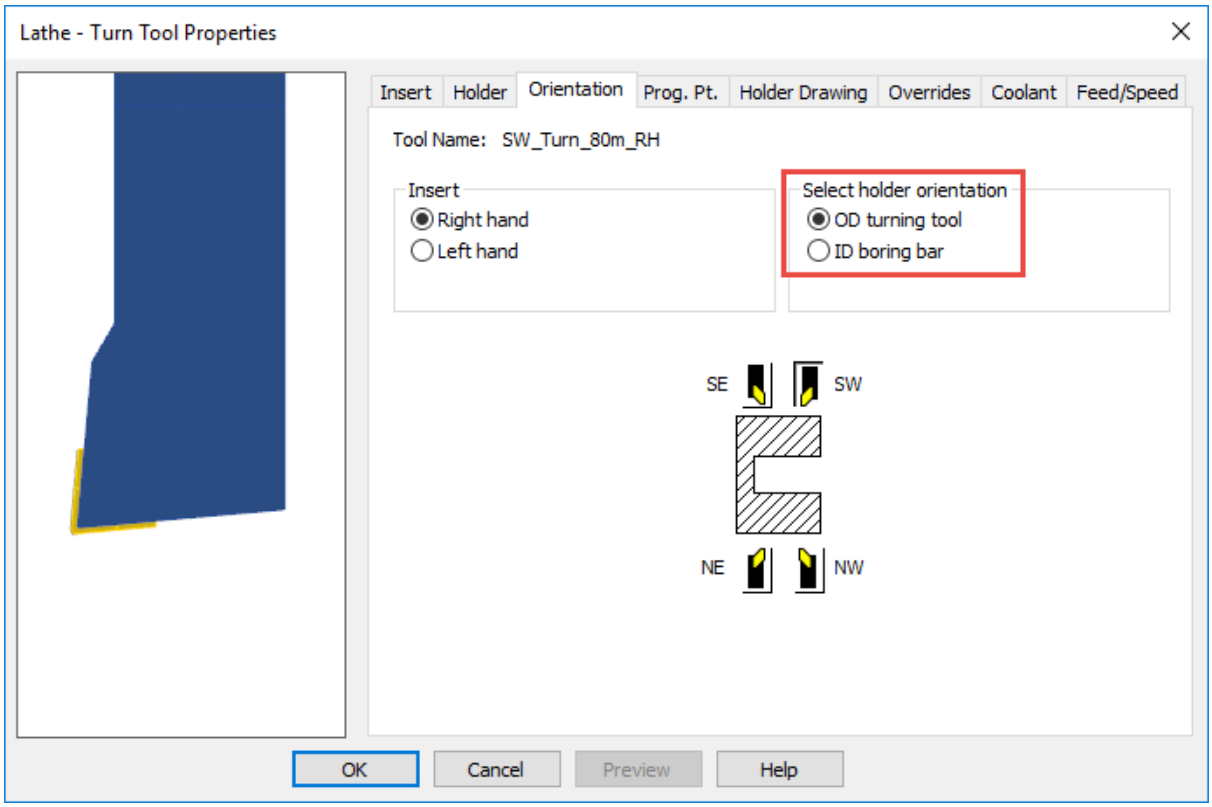
To activate this option please set the following in your configuration file.

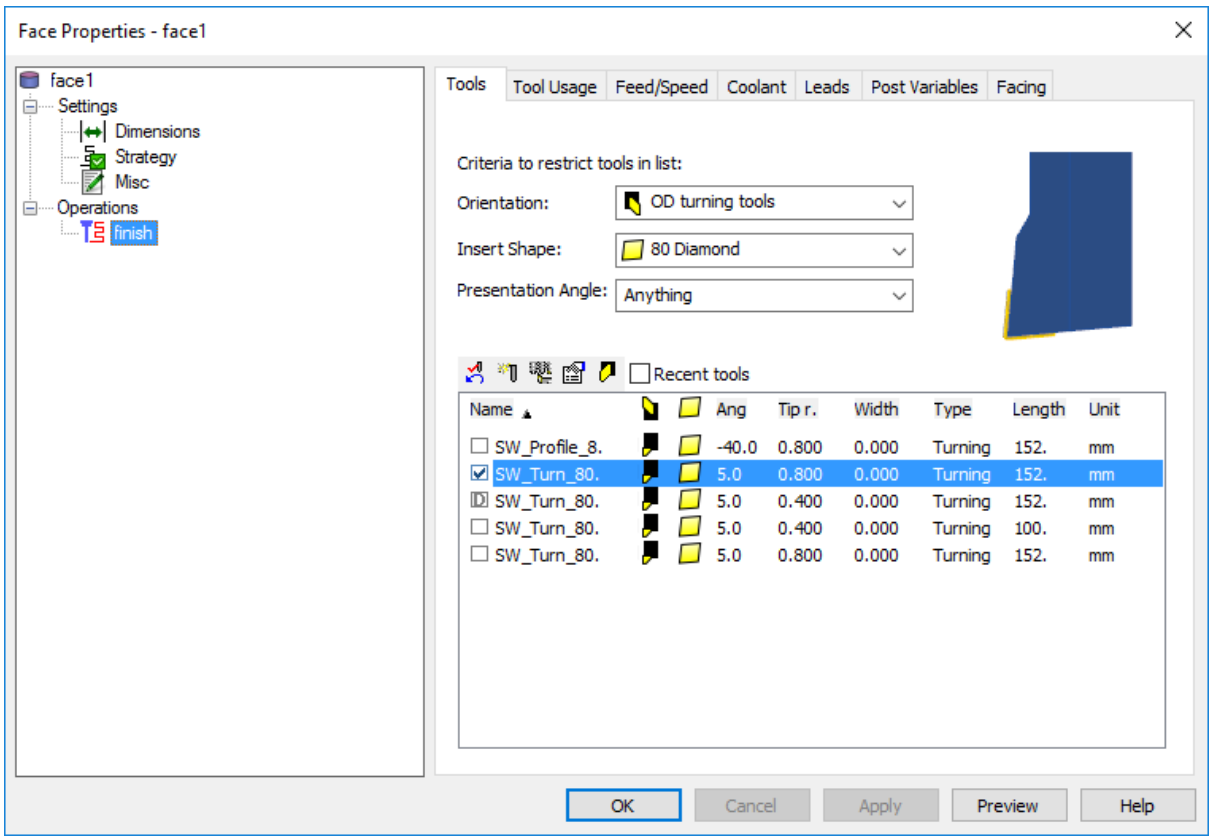


© Copyright 2016 Autodesk, Inc. All rights reserved

The tool orientation is split into two types.

OD Turning or **ID Turning** for **Main- spindle** or **Sub-Spindle.**

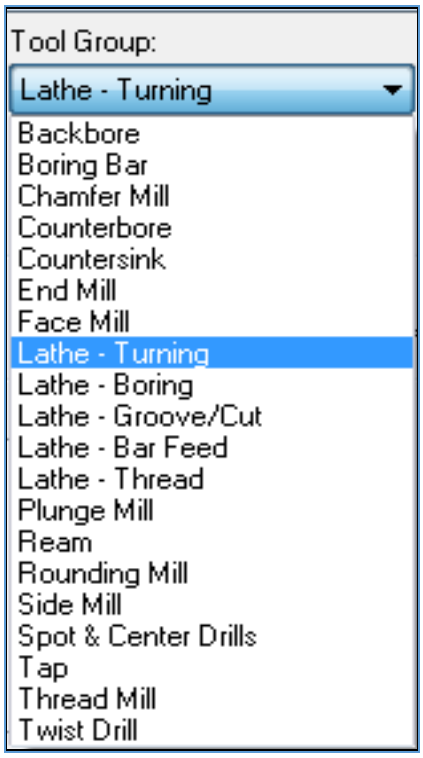




© Copyright 2016 Autodesk, Inc. All rights reserved

Tool Groups.

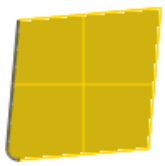
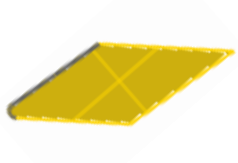
The tools can be grouped together depending upon their type or use.



Insert Shapes

Select the **insert shape**.

For the **Custom diamond insert** shape, you must enter the **Tip Angle**.

			
80 deg Diamond	**Square**	**Triangle**	**35 Deg Diamond**

55 deg Diamond	**Round Insert**	**80 Deg Trigon**	**Custom Diamond**

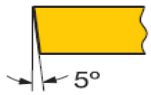
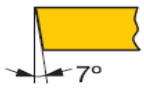
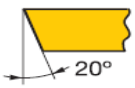
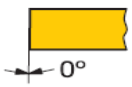
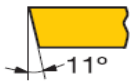
© Copyright 2016 Autodesk, Inc. All rights reserved

Inserts for General Turning

1 Insert shape

C		D	
K		R	
S		T	
V		W	

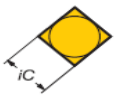
2 Insert clearance angle

B		C	
E		N	
P		O	Specific description

3 Tolerances, metric

Class	*s*	*iC* / *iW*
G	±0.13	±0.025
M	±0.13	±0.05 – ±0.15[1)]
U	±0.13	±0.08 – ±0.25[1)]
E	±0.025	±0.025

[1)]Varies depending on the size of *iC*. See below.



Inscribed circle *iC* mm	Tolerance class M	U
3.97 5.0 5.56 6.0 6.35 8.0 9.525 10.0	±0.05	±0.08
12.0 12.7	±0.08	±0.13
15.875 16.0 19.05 20.0	±0.10	±0.18
25.0 25.4	±0.13	±0.25
31.75 32.0	±0.15	±0.25

For positive inserts iC is valid for a sharp corner. See cutting edge condition F. (Picture 8).

© Copyright 2016 Autodesk, Inc. All rights reserved

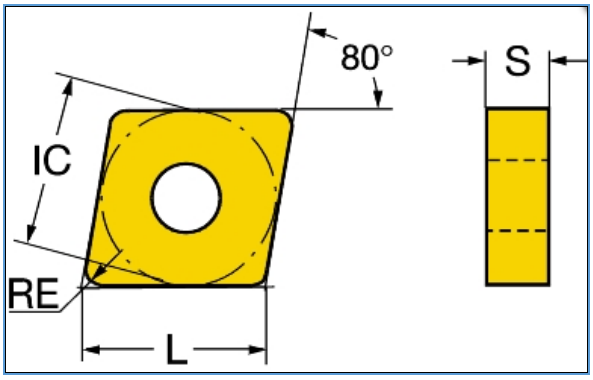
CNMG Insert

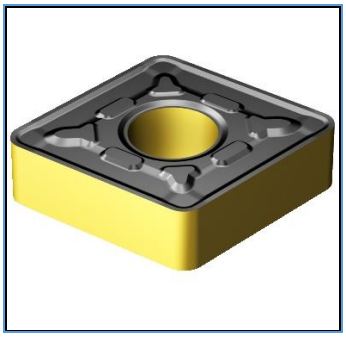
Tip Radius is the radius of the cutting tip of the insert.

Tip Angle is the included angle of the insert.

Inscribe Circle Diameter is the diameter of a circle that fits inside the insert shape.

CNMG 12 04 08-PR 4225

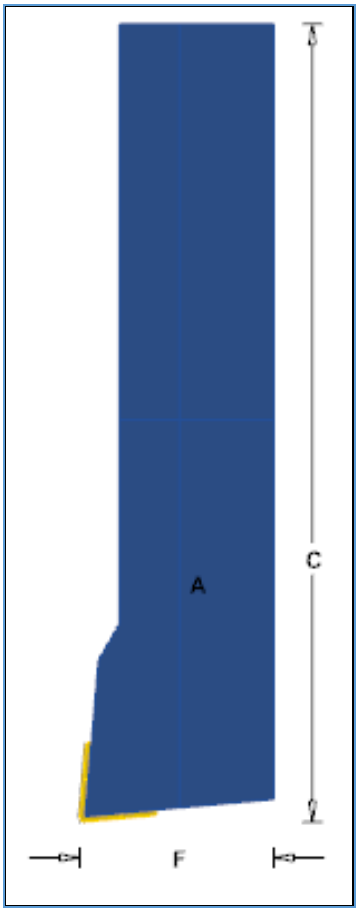




S = Insert Thickness

RE = Corner Radius

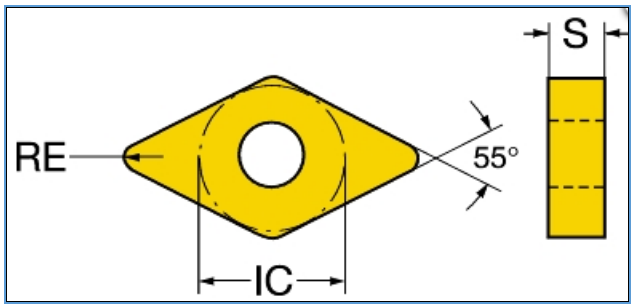
IC = Inscribed Circle Diameter

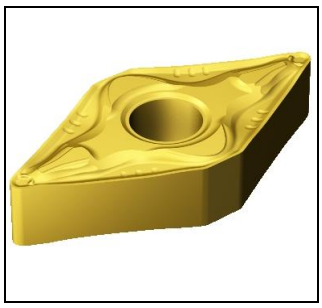


FeatureCAM tool designation = **SW_Turn_80m_RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

DNMG Insert





S = Insert Thickness

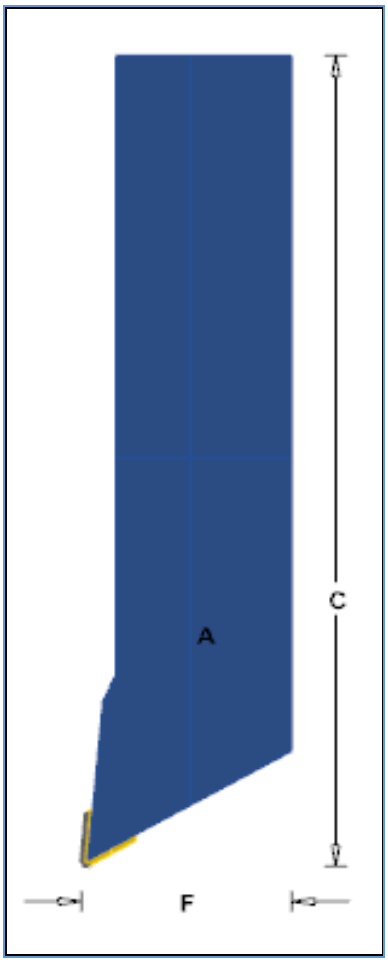
RE = Corner Radius

IC = Inscribed Circle Diameter

Tip Radius is the radius of the cutting tip of the insert.

Tip Angle is the included angle of the insert.

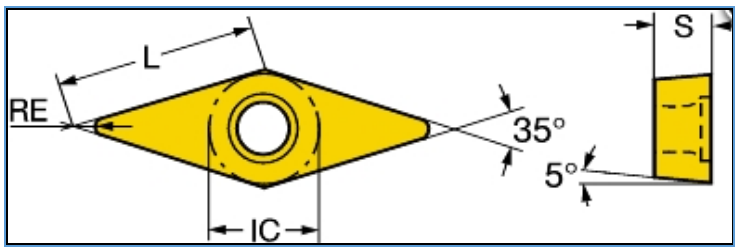
Inscribe Circle Diameter is the diameter of a circle that fits inside the insert shape.

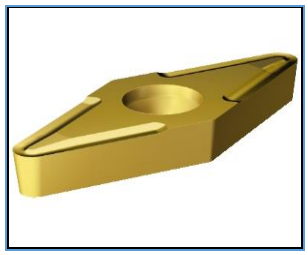


FeatureCAM tool designation = **SW_Turn_55m_RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

VBMT Insert





S = Insert Thickness

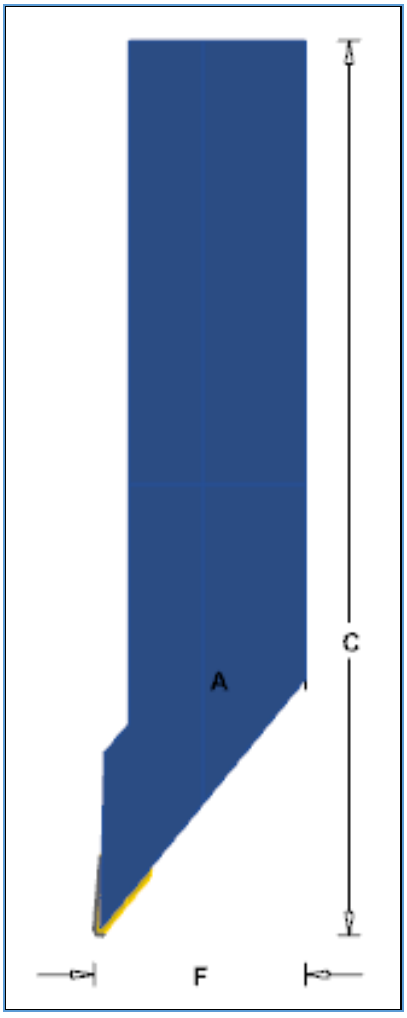
RE = Corner Radius

IC = Inscribed Circle Diameter

Tip Radius is the radius of the cutting tip of the insert.

Tip Angle is the included angle of the insert.

Inscribe Circle Diameter is the diameter of a circle that fits inside the insert shape.

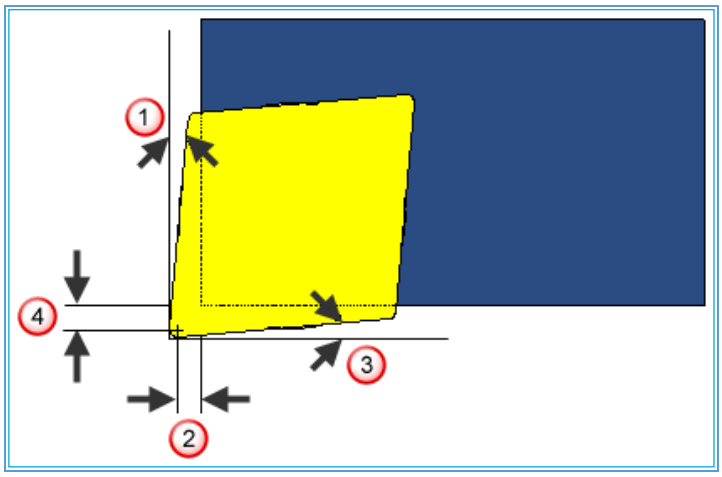


FeatureCAM tool designation = **SW_Turn_35m_RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

End Cut: - The tool cuts in a direction parallel with the length of the holder.

Side Cut: - The tool cuts in a direction perpendicular with the length of the holder.

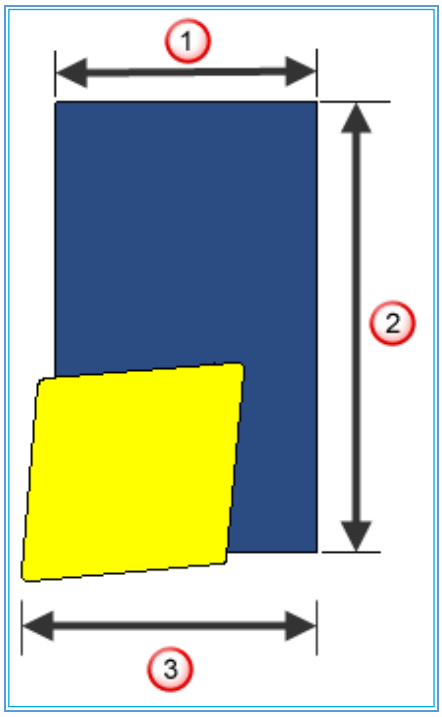


1 End Angle

2 End Clearance

3 Side Angle

4 Side Clearance



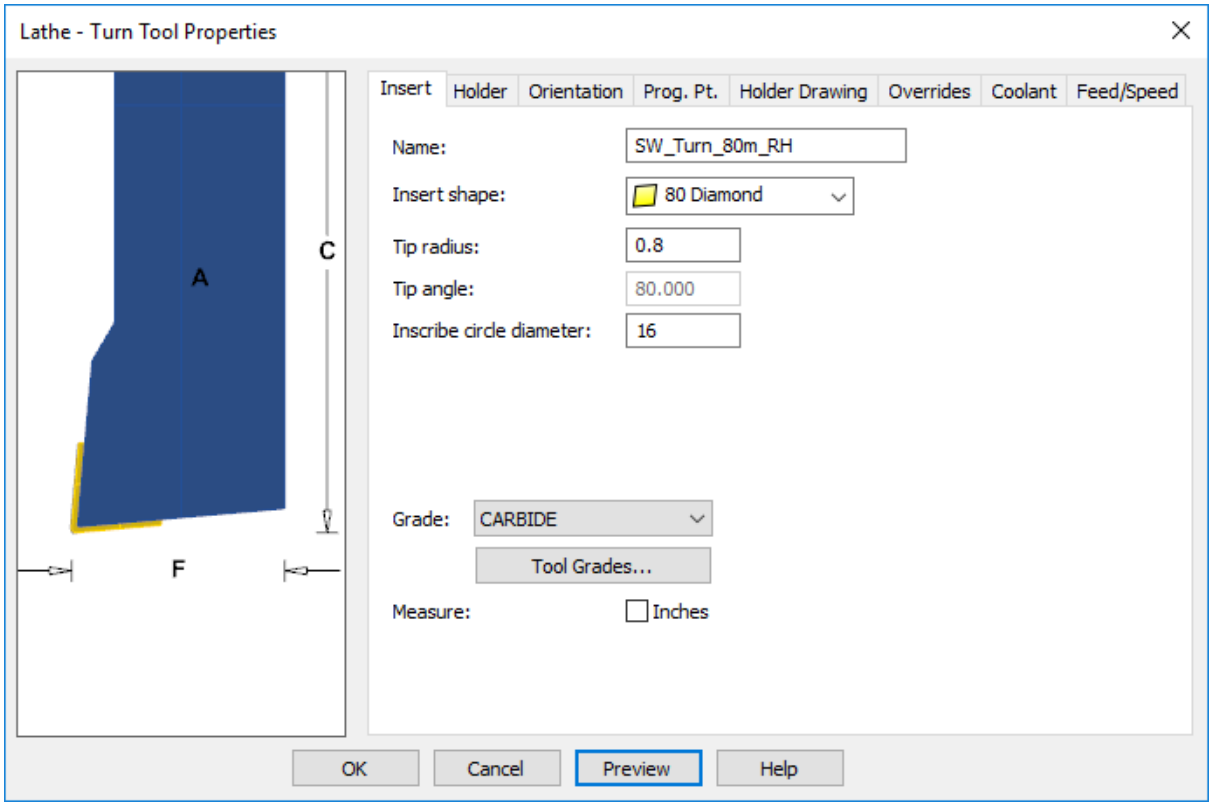
1 Shaft Width (A)

2 Length (C)

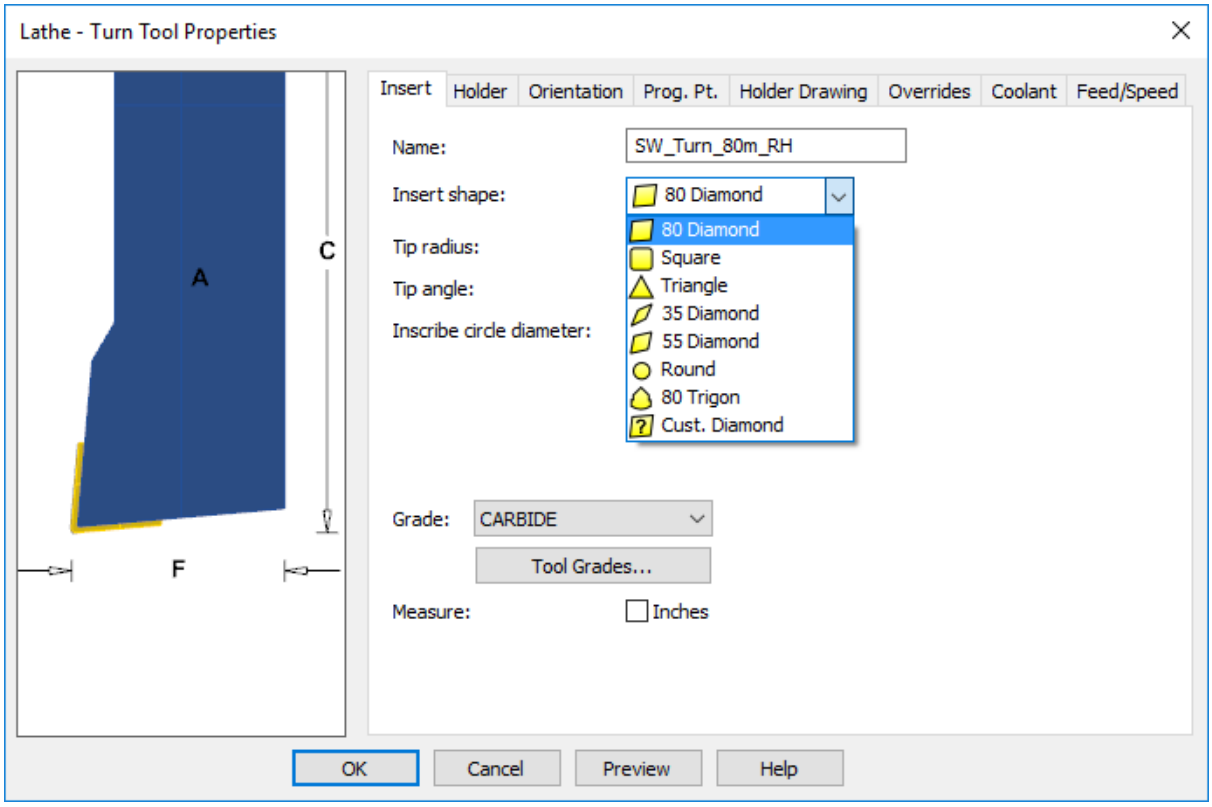
3 Tip to Back (F)

© Copyright 2016 Autodesk, Inc. All rights reserved

Exposed length: - This is the amount of the tool that sticks out of the Turret if the holder is simulated. Lathe - Turning Tool Properties example.

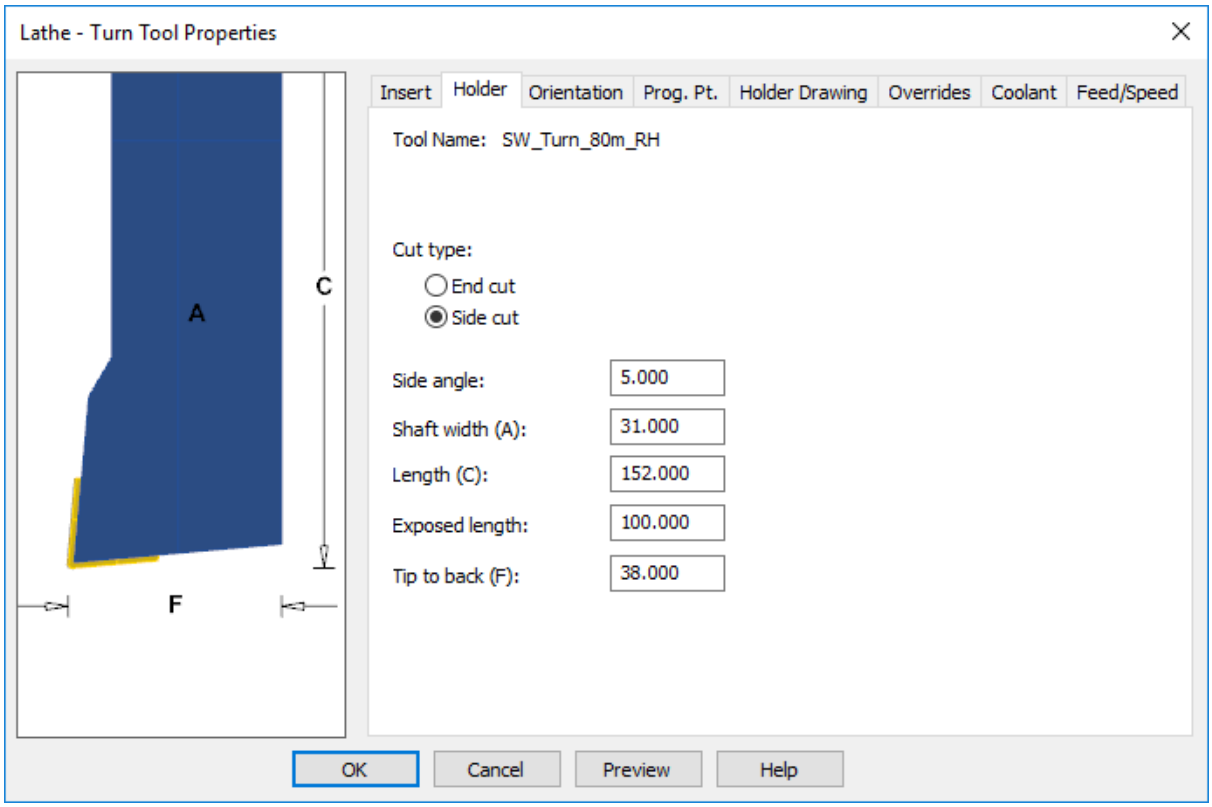


Insert Shape

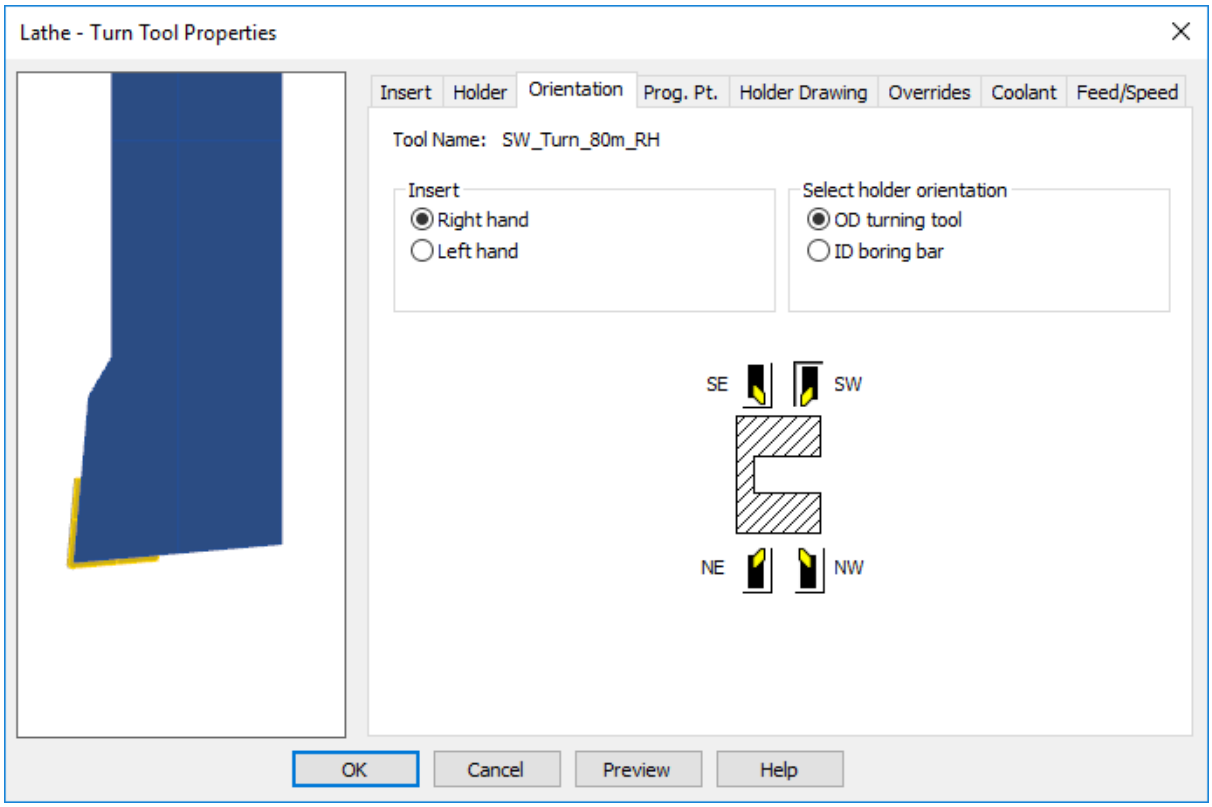


© Copyright 2016 Autodesk, Inc. All rights reserved

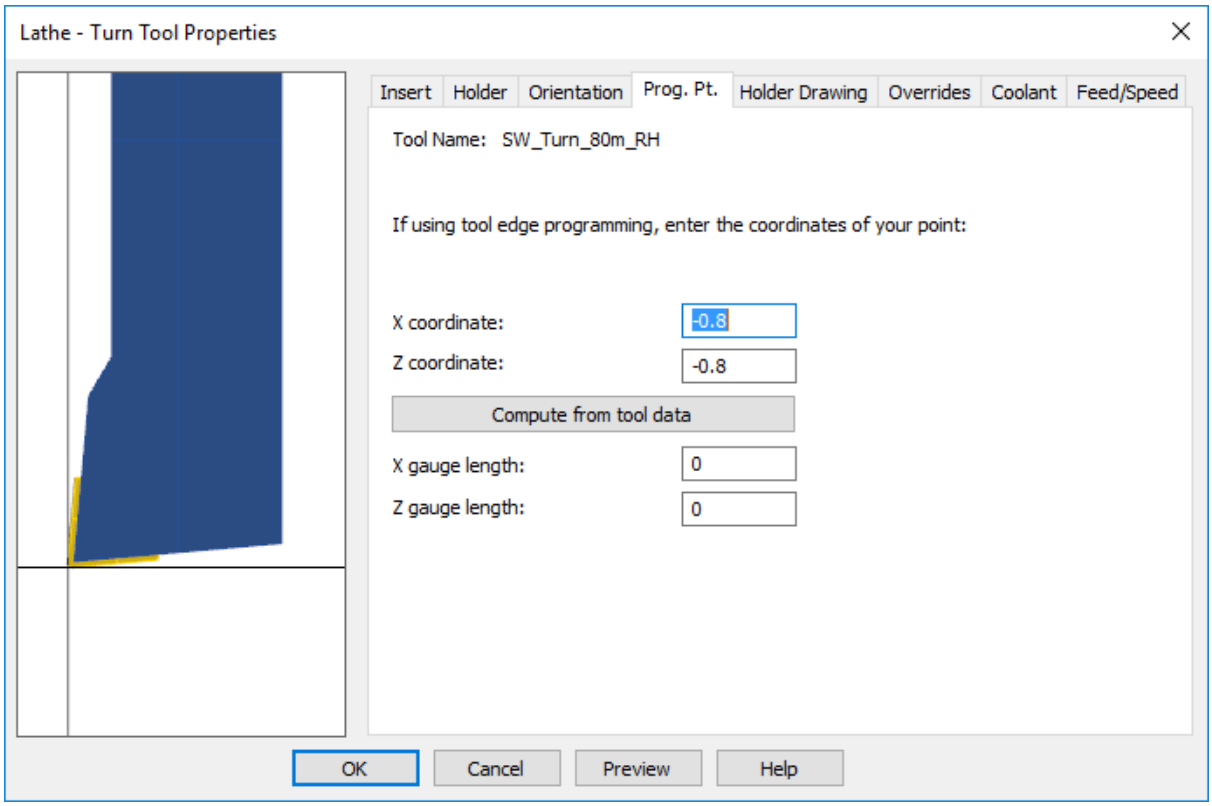
Holder



Orientation



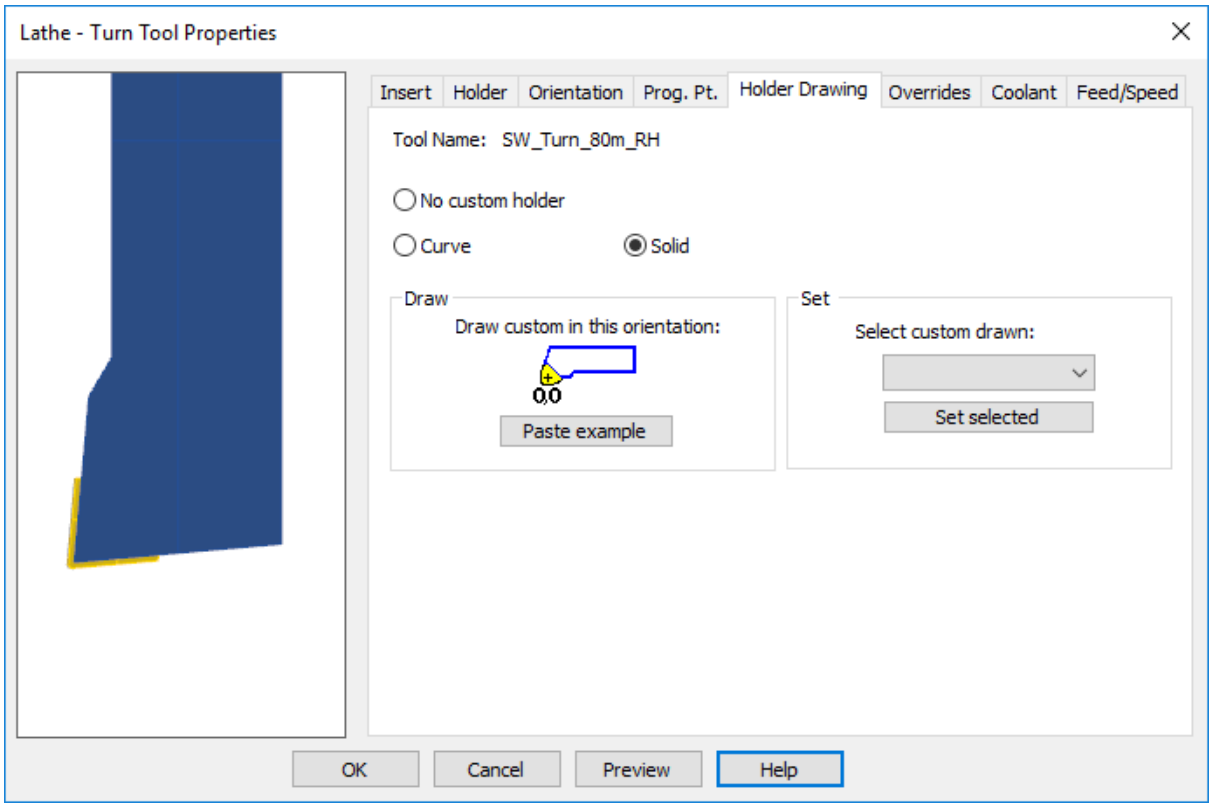
© Copyright 2016 Autodesk, Inc. All rights reserved



*On the **Prog. Pt.** (program point) tab you set the point of the insert that is actually programmed. You can set both the **X Coordinate** and the **Z Coordinate**. **Tool tip centre** - If both **X Coordinate** and the **Z Coordinate** are set to **0.0** then the centre point of the tip arc is programmed. In this case it is expected that you perform insert radius compensation at the machine tool. **Tool tip edge** - If you wish to perform insert radius compensation in **FeatureCAM**, set the **X Coordinate** and **Z Coordinate** to the radius compensation values provided with your tool. You should also set the default **Tool program point** attribute to **Tool tip centre** or **Tool tip edge**.*

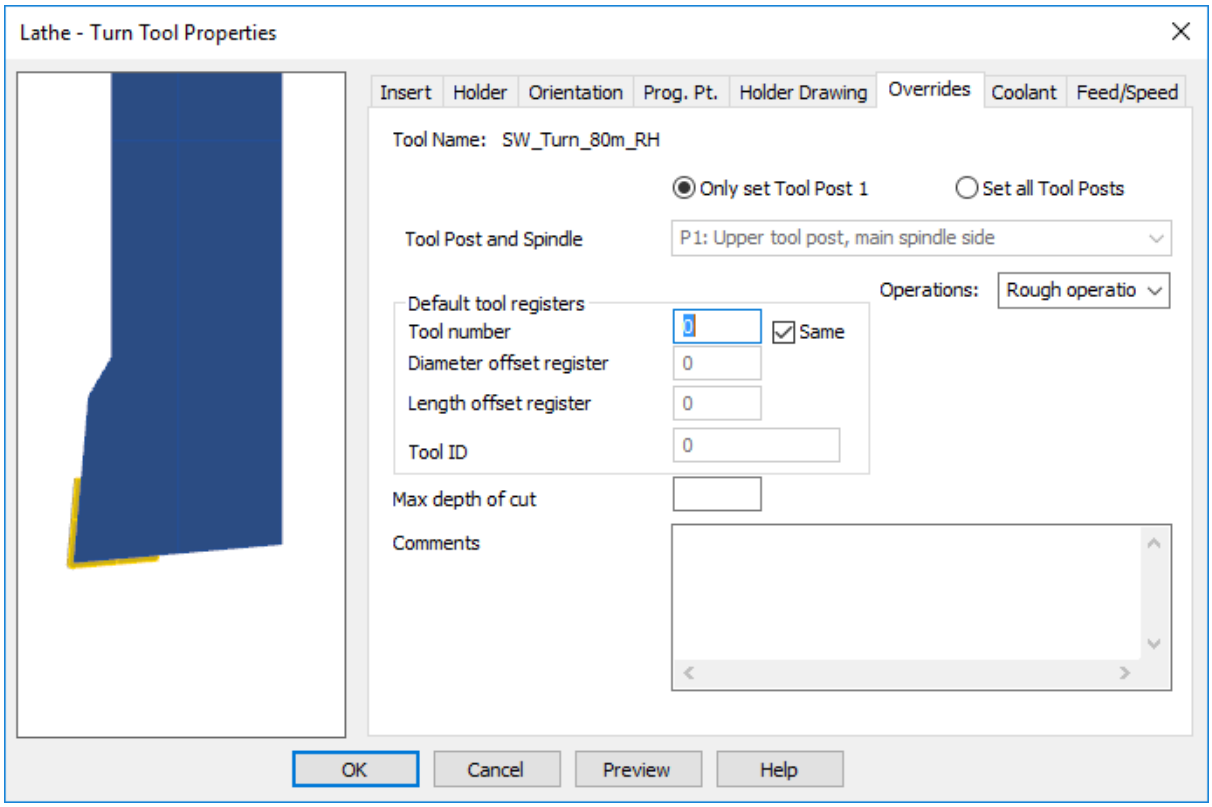
© Copyright 2016 Autodesk, Inc. All rights reserved

Holder Drawing



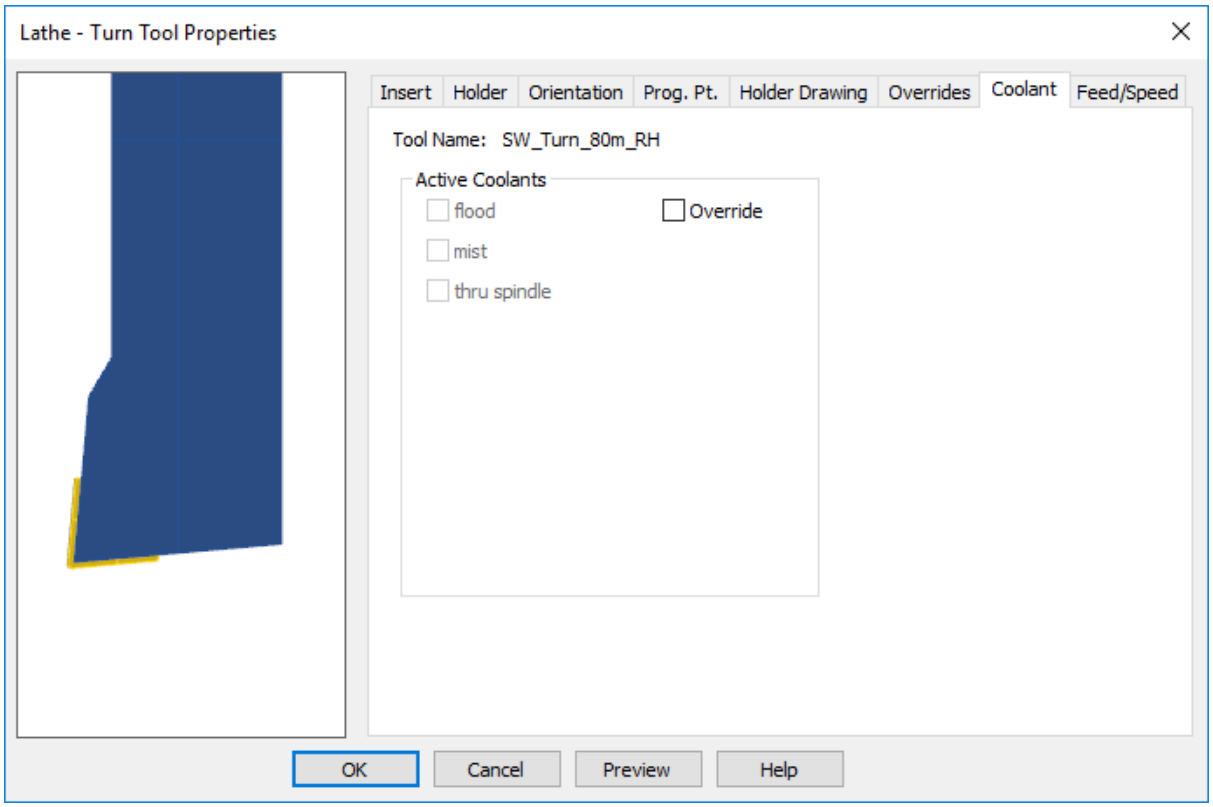
Solid — you can select a solid to define the shape of the Tool Holder. This is the most accurate way to represent the tool holder.

Overrides

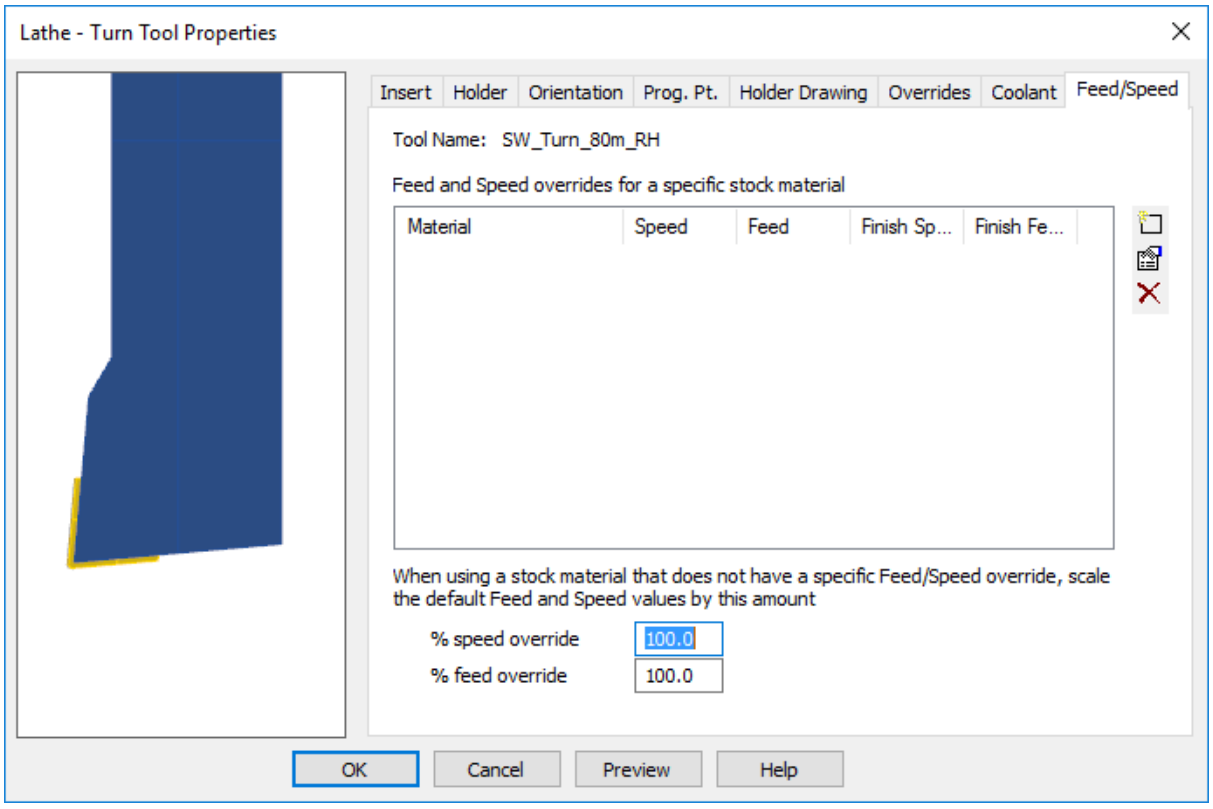


© Copyright 2016 Autodesk, Inc. All rights reserved

Coolant



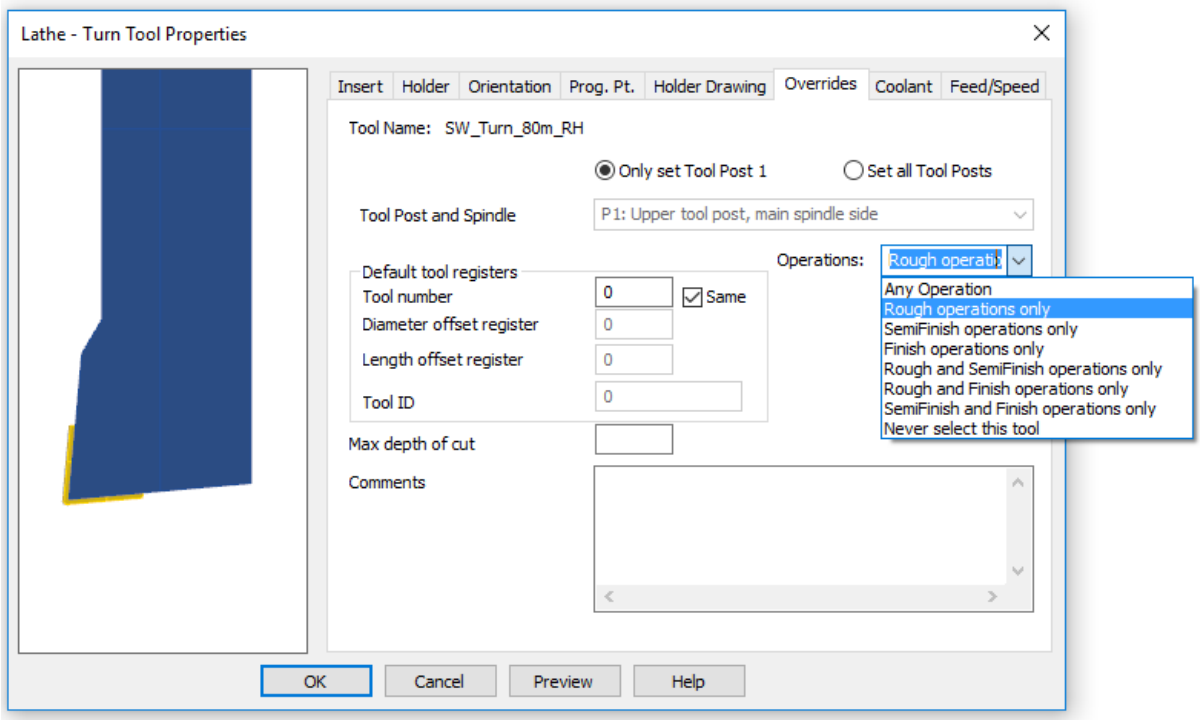
Feeds/Speeds



© Copyright 2016 Autodesk, Inc. All rights reserved

Turning tool operation usage

FeatureCAM gives you the ability to set turning tools operation type. Select **Operations** then select an entry in the list to specify the type of operation for which FeatureCAM can select this tool. You can override this setting by selecting the tool



Plunge clearance *is the distance above an operation at which the tool starts to feed. In the case of deep hole drilling, the drill retracts to this distance between pecks. For milling features, the default is to use the same value for roughing and finishing. As a result, the tool feeds from the top of a pocket to the floor before cutting. To make the tool feed down into the feature, set the Plunge clearance for an operation to a negative value, but make sure the value is above the floor of the feature.*

The ***Starting offset*** *number for shared tool slots is the first length offset register to use for tools that share the same tool slot.*

Tool program point - select from*:*

Tool tip edge *- select this option if you want to adjust the tool programming points by the insert radius compensation. In this case adjust the tool's programming point by the radius compensation.*

Tool tip centre *- select this option if you want to adjust the tool touch-off points by the insert radius compensation at the machine. In this case set the tool's programming point X Coordinate and Z Coordinate to 0.0 on the tool's Prog. Pt. tab.*

Turret direction *- we recommend that you leave this as Auto so that FeatureCAM can calculate the best direction for a particular operation. You can also explicitly set this option to CW (clockwise) or CCW (counter-clockwise).*

© Copyright 2016 Autodesk, Inc. All rights reserved

RPM Range

Some turning centres have gear boxes that set the maximum spindle speed of the machine. The RPM Range list sets the gear box to a specific maximum range. If RPM Range is set to a value of 1-4, then the range is set explicitly. If RPM Range is set to Auto then FeatureCAM sets the range for you based on the following rules:

If the feature is a turned hole or another turned feature without Constant Surface Speed set, then the range is determined based on the Spindle Speed.

If the feature is a turned feature with Constant Surface Speed set, then the range is determined based on the MAX RPM.

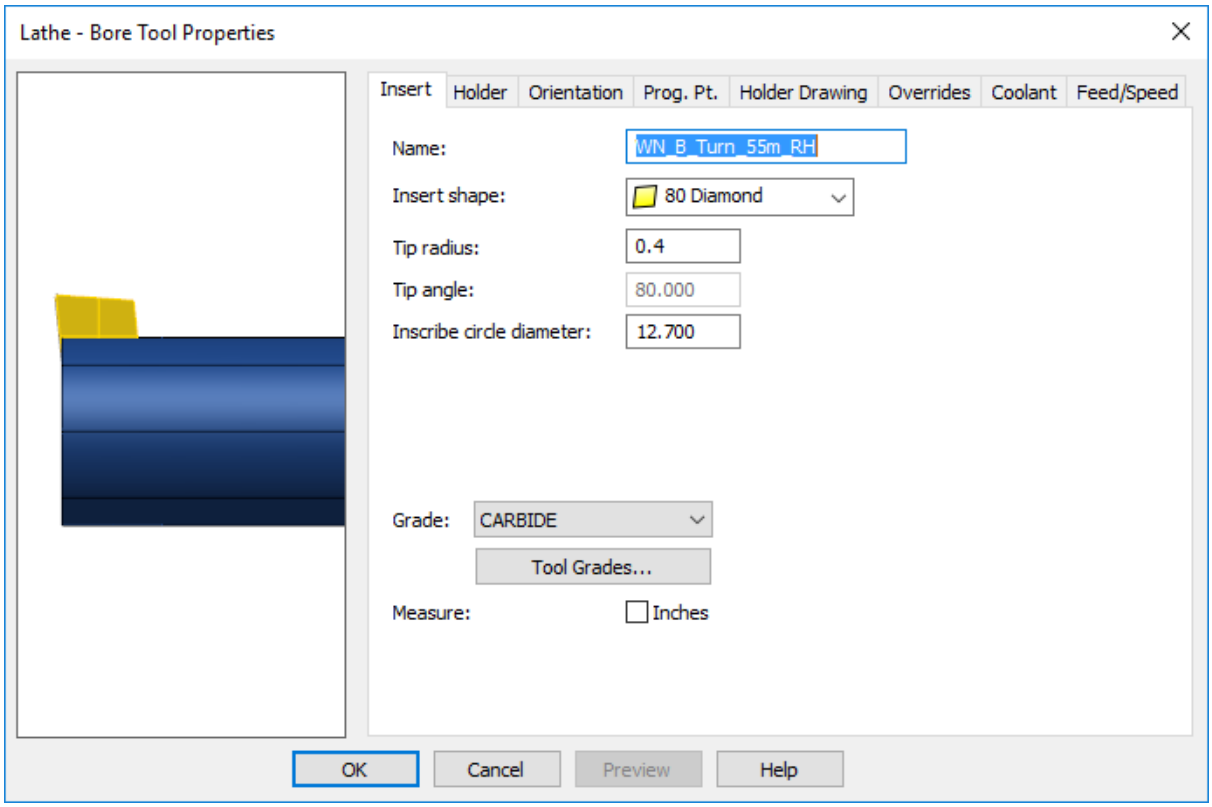
Remachining *- automatically sets the boundaries for subsequent operations that use the same curve. This minimizes air cutting and works between turn features, bore features and between holes and bore features. The same curve must be used in both features. ID features use the results of a previous turn drill operation if such a feature exists. The stock curve that results from the first operation is the result of undercut clipping with the tool geometry and nothing more than that.*

Use IPR/MMPR *- The default feed units are IPM (inches per minute) or MMPM (mm per minute). Select this option to use IPR (inches per revolution) or MMPR (mm per revolution).*

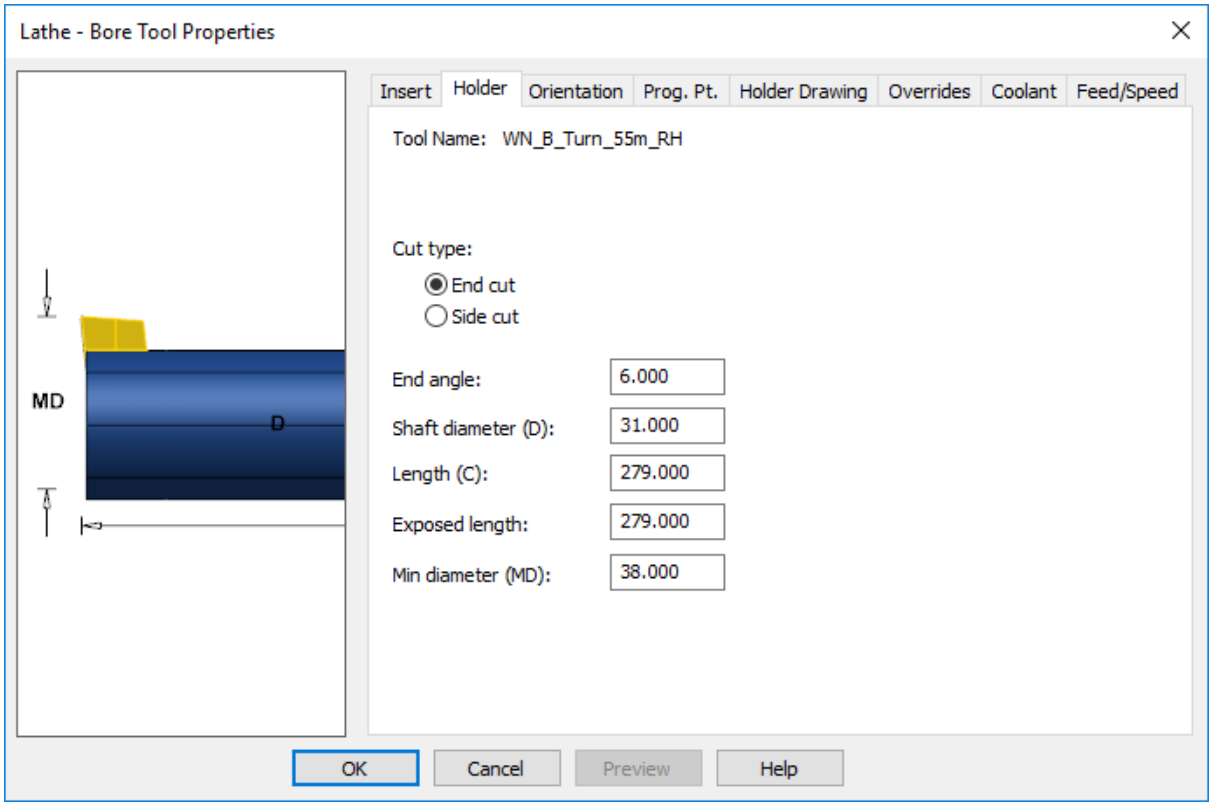
Do feed reduction for small moves *- This attribute helps FeatureCAM cut small features properly. It is typically applied to small chamfers or small radii but affects any small move. If Do feed reduction for small moves is selected, then any move with fewer revolutions than the Threshold, is reduced by the Feed rate %*

© Copyright 2016 Autodesk, Inc. All rights reserved

Lathe - Bore Tool Properties example.



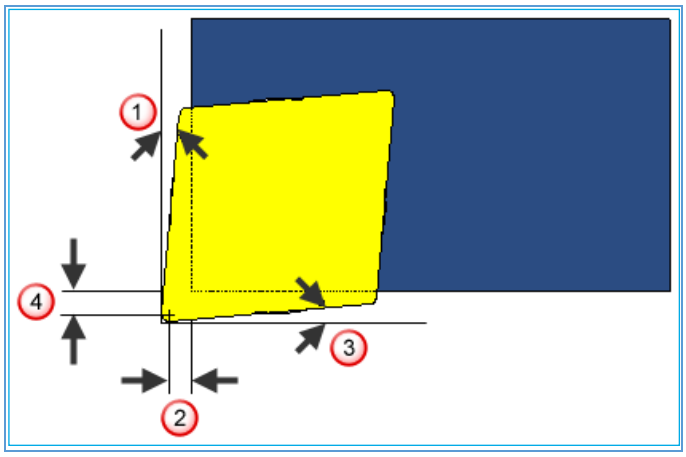
Holder Information



© Copyright 2016 Autodesk, Inc. All rights reserved

***End Cut*: -** *The tool cuts in a direction parallel with the length of the holder.*

***Side Cut*: -** *The tool cuts in a direction perpendicular with the length of the holder.*

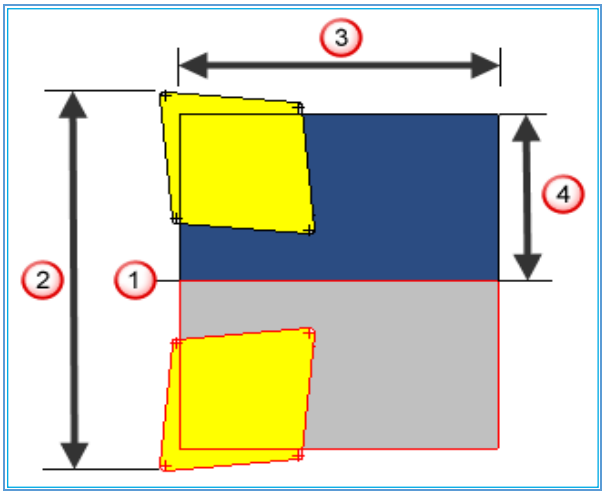


1 End Angle

2 End Clearance

3 Side Angle

4 Side Clearance

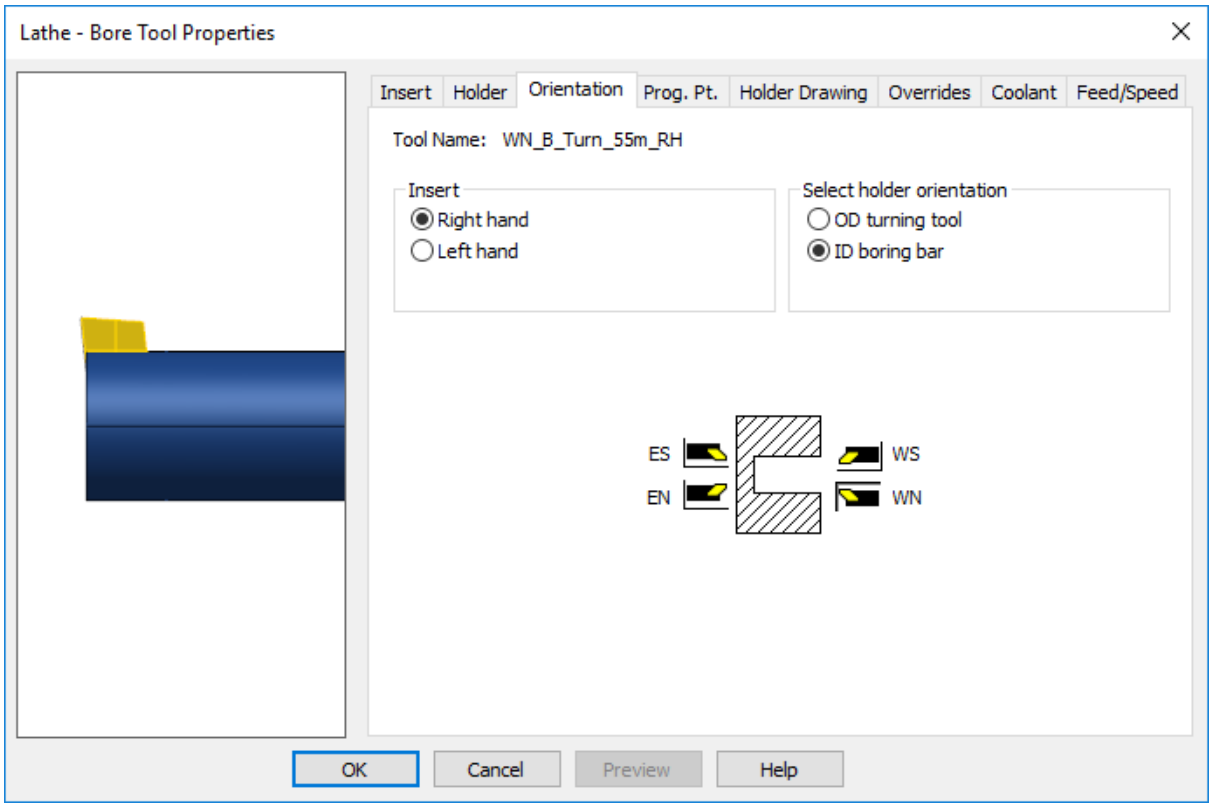


© Copyright 2016 Autodesk, Inc. All rights reserved

1. Z Axis
2. Min. Diameter (MD)
3. Length (C)
4. Shaft Diameter (D)

Exposed length: - This is the amount of the tool that sticks out of the Turret if the holder is simulated.

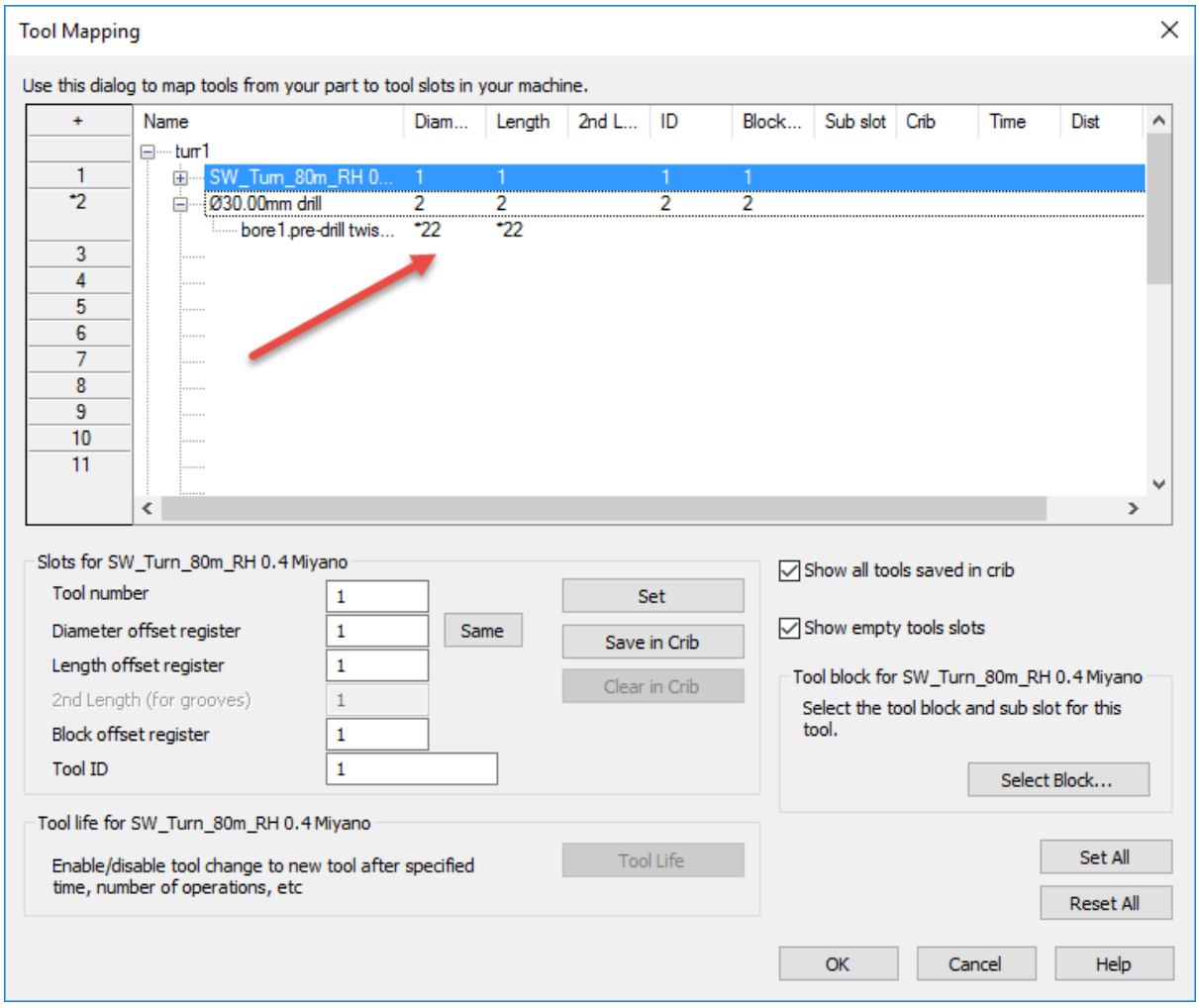
Boring Orientation



© Copyright 2016 Autodesk, Inc. All rights reserved

Using an insert drill to drill and bore in the same program

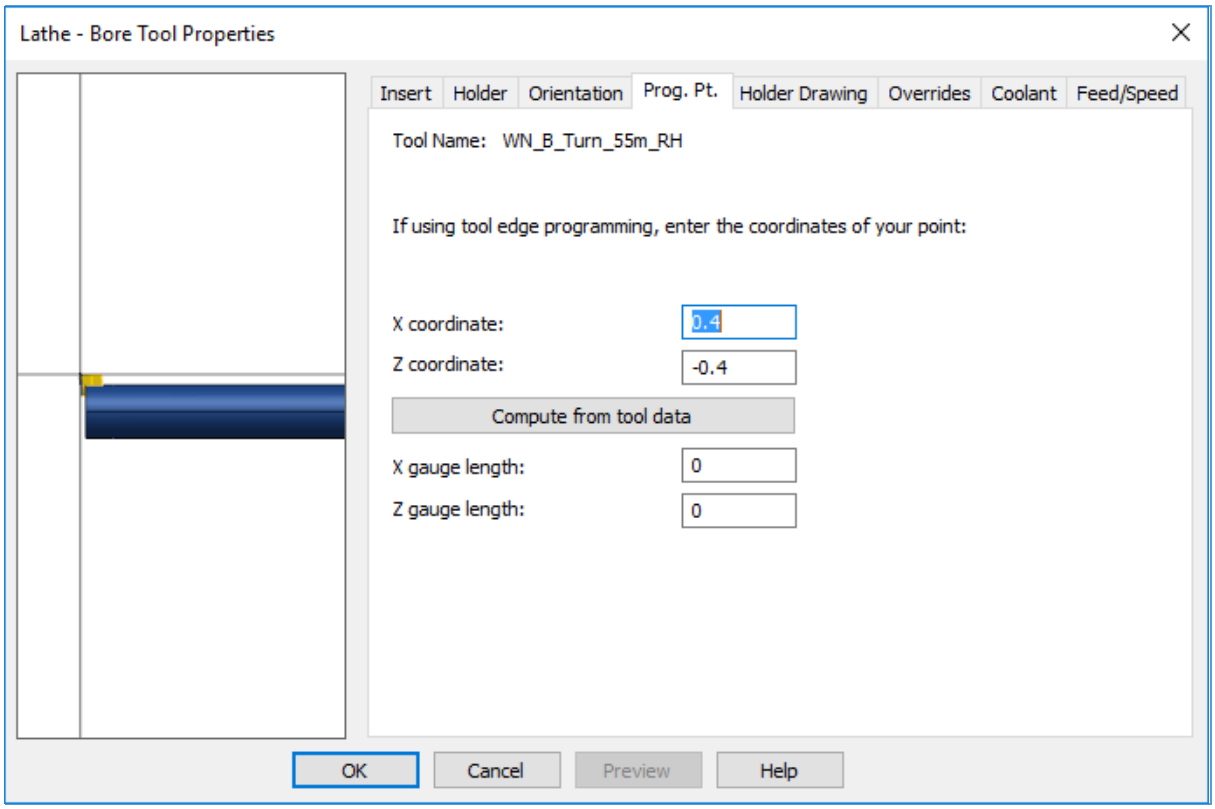
1. Create the Hole feature and override the tool for the drilling operation to be an insert drill.
2. Create the Bore feature and override the tool to be the same insert drill.
3. If you view the tools in the Tool mapping dialog you see that there are two drills listed in the same tool slot, but they each have a different Length offset register.



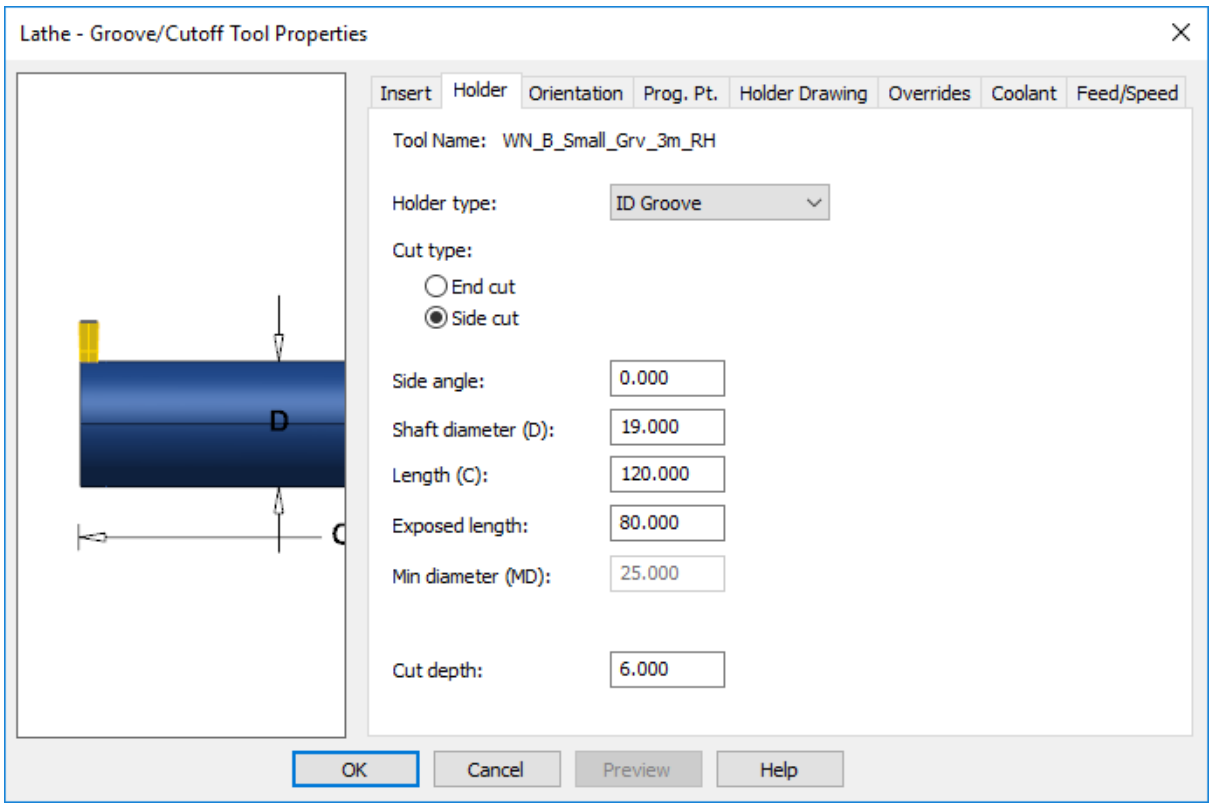
Please note that you need to reduce the retract distance to avoid a collision inside the bore. This needs to be applied to roughing and finishing operations.

© Copyright 2016 Autodesk, Inc. All rights reserved

Lathe - Bore Programming Point

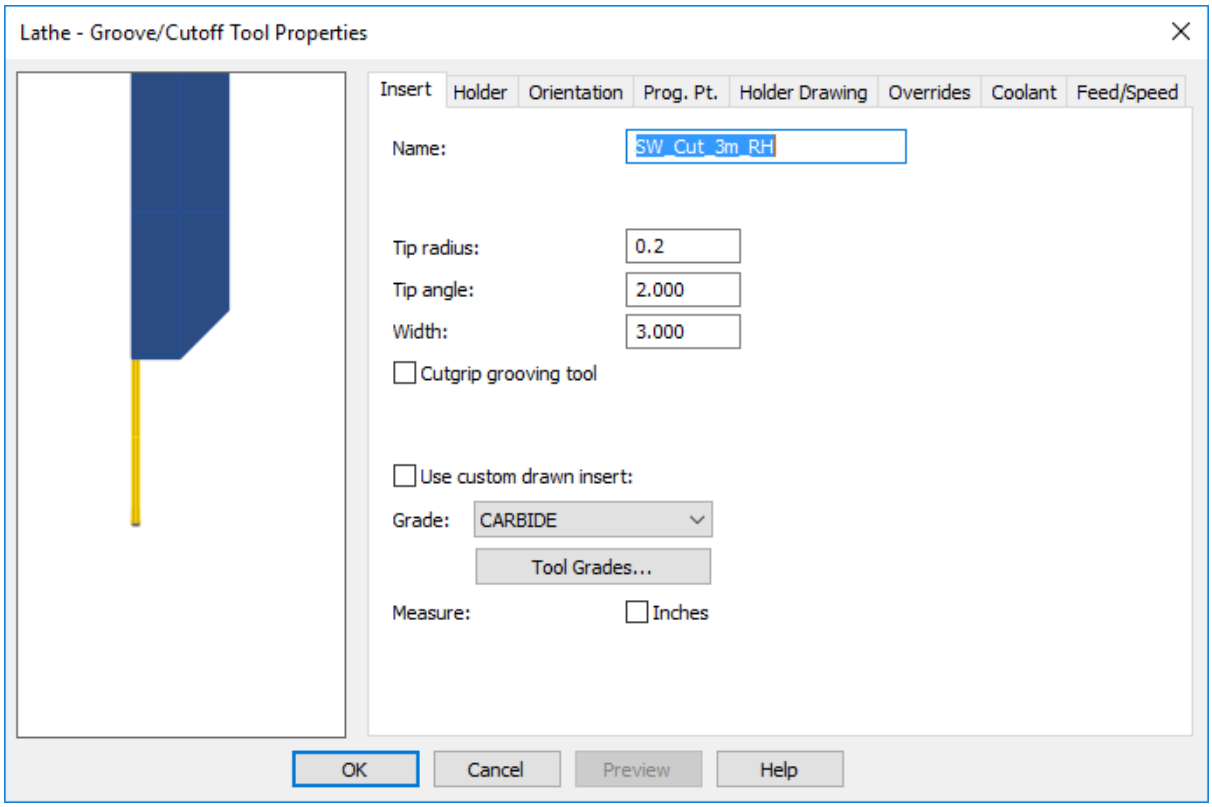


Lathe – Groove tool Properties example



© Copyright 2016 Autodesk, Inc. All rights reserved

Lathe - Cutoff Tool Properties example

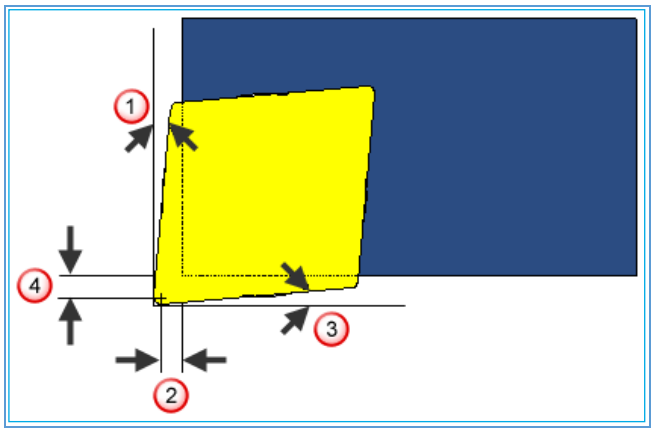


Holder Type: - For grooves possible holder types are:

- **OD Groove: -** Select this option for outer diameter grooving tools.
- **ID Groove: -** Select this option for inner diameter grooving tools.
- **Face Groove: -** Select this option for face grooving tools.
- **Cutting: -** Select this option for Cutoff tools.

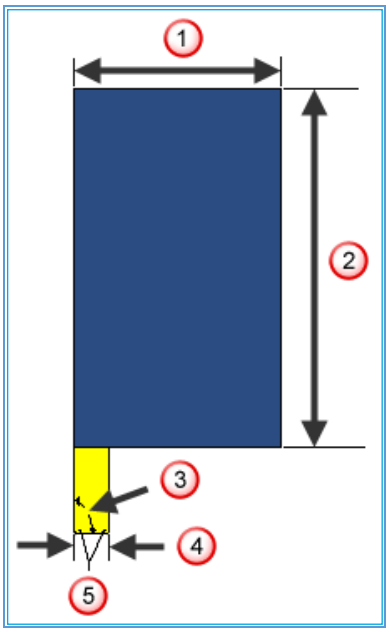
End Cut: - The tool cuts in a direction parallel with the length of the holder.

Side Cut: - The tool cuts in a direction perpendicular with the length of the holder.



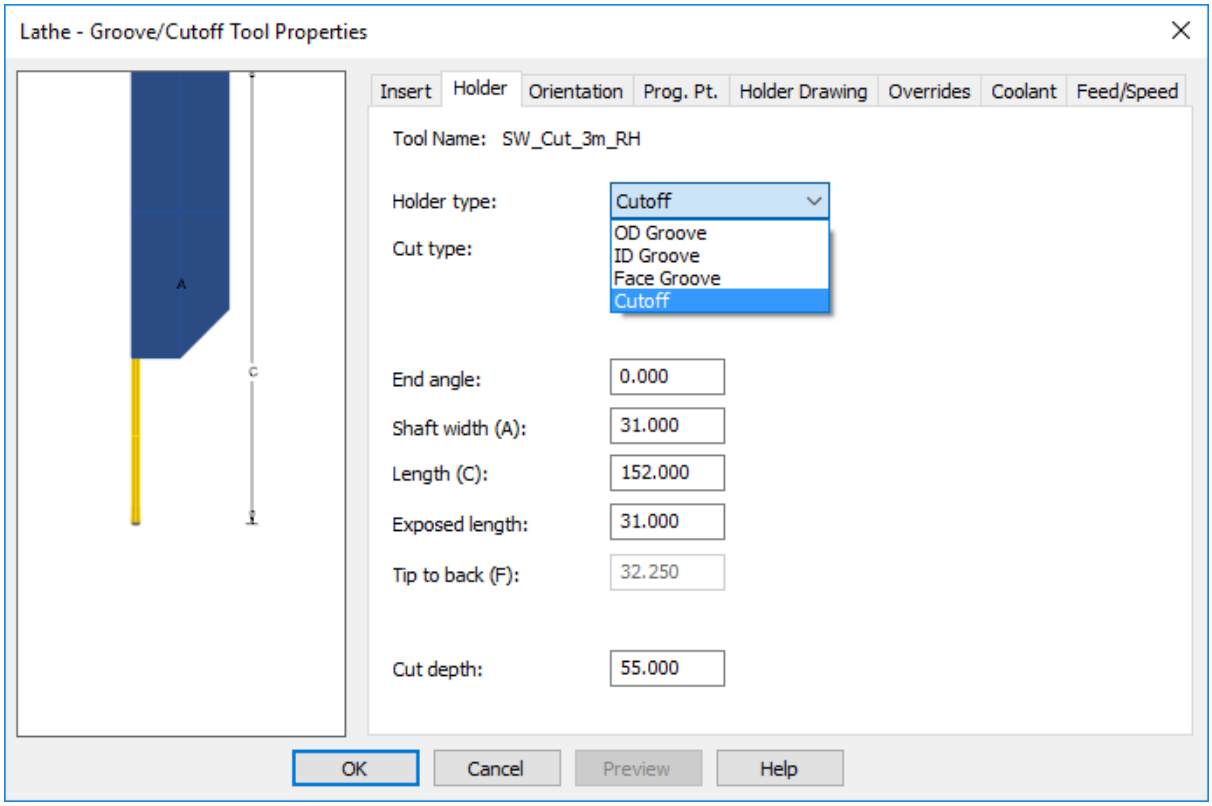
1 End Angle 2 End Clearance 3 Side Angle 4 Side Clearance

© Copyright 2016 Autodesk, Inc. All rights reserved



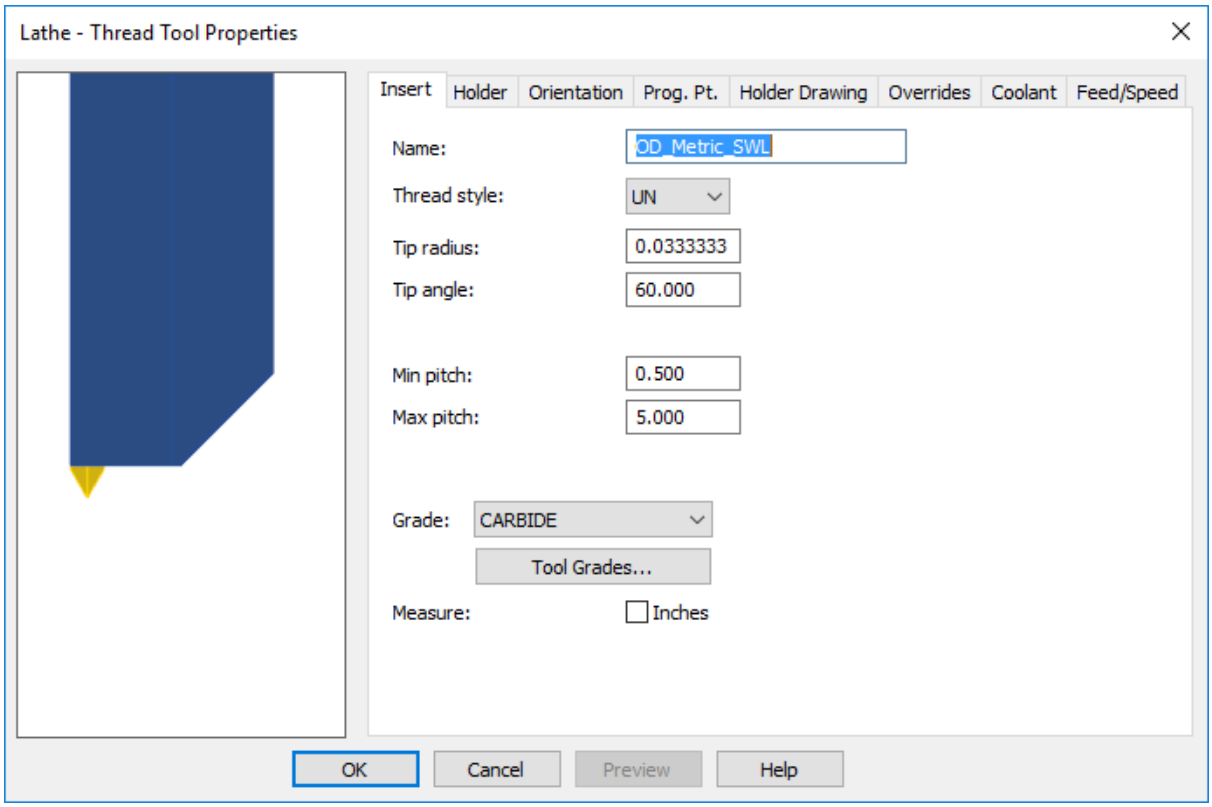
1 Shaft Diameter (D) 2 Length (C) 3 Tip Angle 4 Width 5 Tip Radius.

Lathe - Groove/Cutoff Holder tool Properties example



© Copyright 2016 Autodesk, Inc. All rights reserved

Lathe – Thread Tool Properties example.

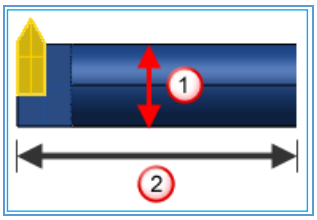


Name *- is a string that identifies the tool. The name must be unique among all the tools in the crib.* ***Tip Radius*** *is the radius of the cutting tip of the insert. See diagram.*

For threading tools, 3D simulation simulates the tool with a tip radius of 0.0. This is just for visualization purposes only. The NC code or tool selection is not affected in any way. Tip Angle is the included angle of the insert. See diagram.

Measure *- indicates the units that are used for reporting the tool's dimensions. Select inches for inches or deselect the box for millimetres.*

This is for reference only. If you want to edit the Name of the tool, do it on the Insert tab.



1 Shaft Diameter

2 Length

Holder Type *- For threads possible holder types are:*

OD Threads *- Select this value for outer diameter threading tools*

ID Threads *- Select this value for inner diameter threading tools*

© Copyright 2016 Autodesk, Inc. All rights reserved

End Cut *- The tool cuts in a direction parallel with the length of the holder.*

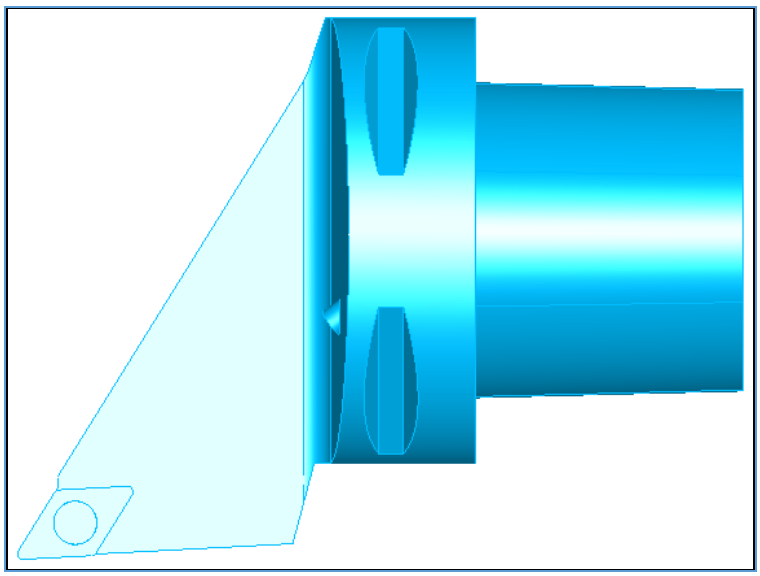
Side Cut *- The tool cuts in a direction perpendicular with the length of the holder.*

Adding a Solid Model Turning Tool

FeatureCAM has the ability to use Solid model Turning Tools and add them to the tooling database. The following steps, take you through the importing of the Solid Model tool and transforming it to the correct position. Then adding the tool to the tool database.

1 Open a new Turning document.

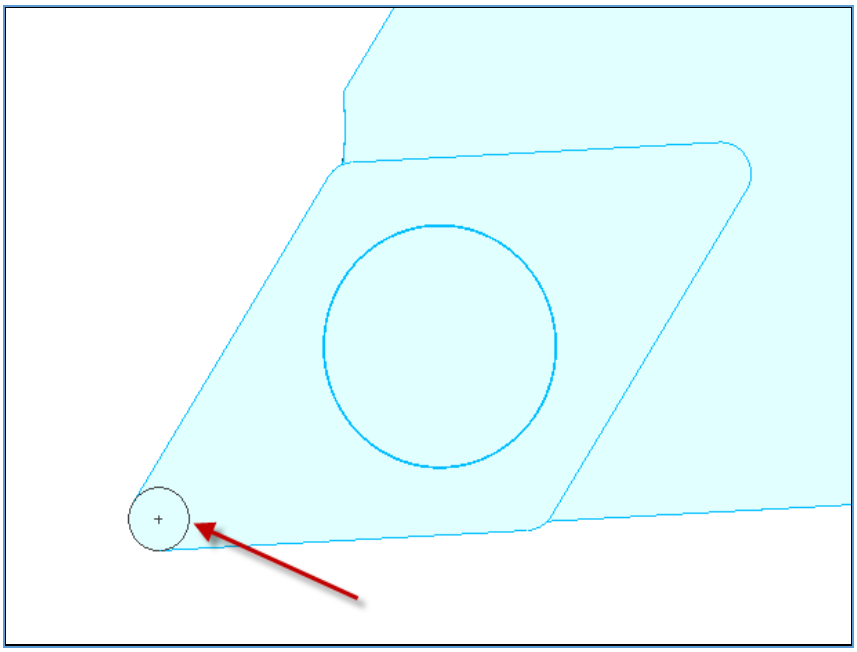
Import the solid model **from C:\Training_Data\FeatureCAM Course Data 2017\Tooling\Turning Tools C6-SDJCR-45065-11-635.x_t**



2 Switch on the **Stock axis** in **Part View**. You will notice that the tool tip radius centre point 0.8 is **not** located in the correct position. The centre point of the tip radius must be at X0, Y0, Z0. (Stock Axis).

3 Draw a circle on the tip radius. To do this use **Circle from 3 points** from the geometry menu. Make sure you have snap to object switched on.

© Copyright 2016 Autodesk, Inc. All rights reserved

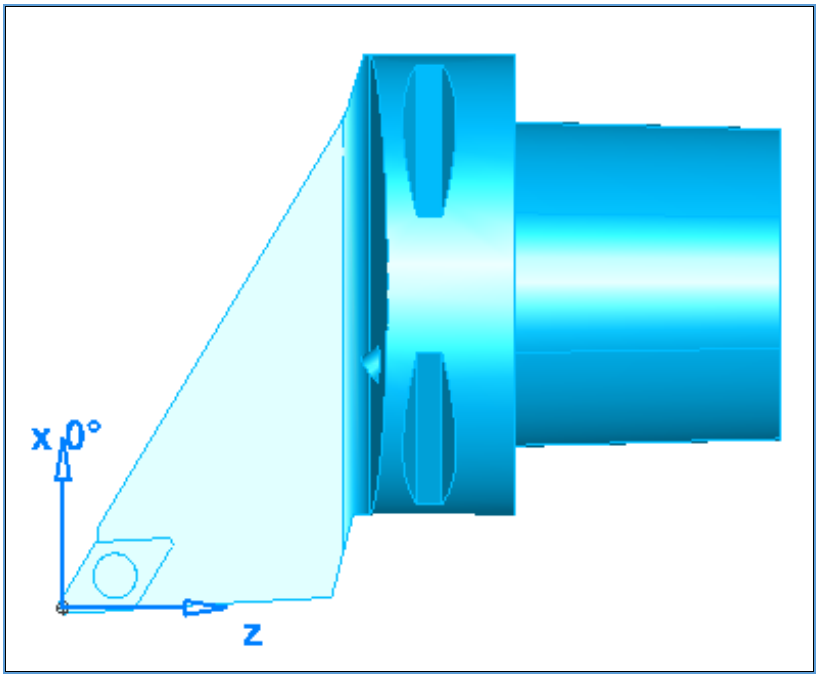


4 In **PartView** Select the solid model tool shape in the Solids directory. This will make it active. Then from the **Edit** menu select **Transform/Translate/**

5 **Move-From.** Snap to the circle centre **To X0, Y0, Z0**.

6 Select **Ok** to action the process.

7 Switch on the **Stock Axis**. You will see the following.

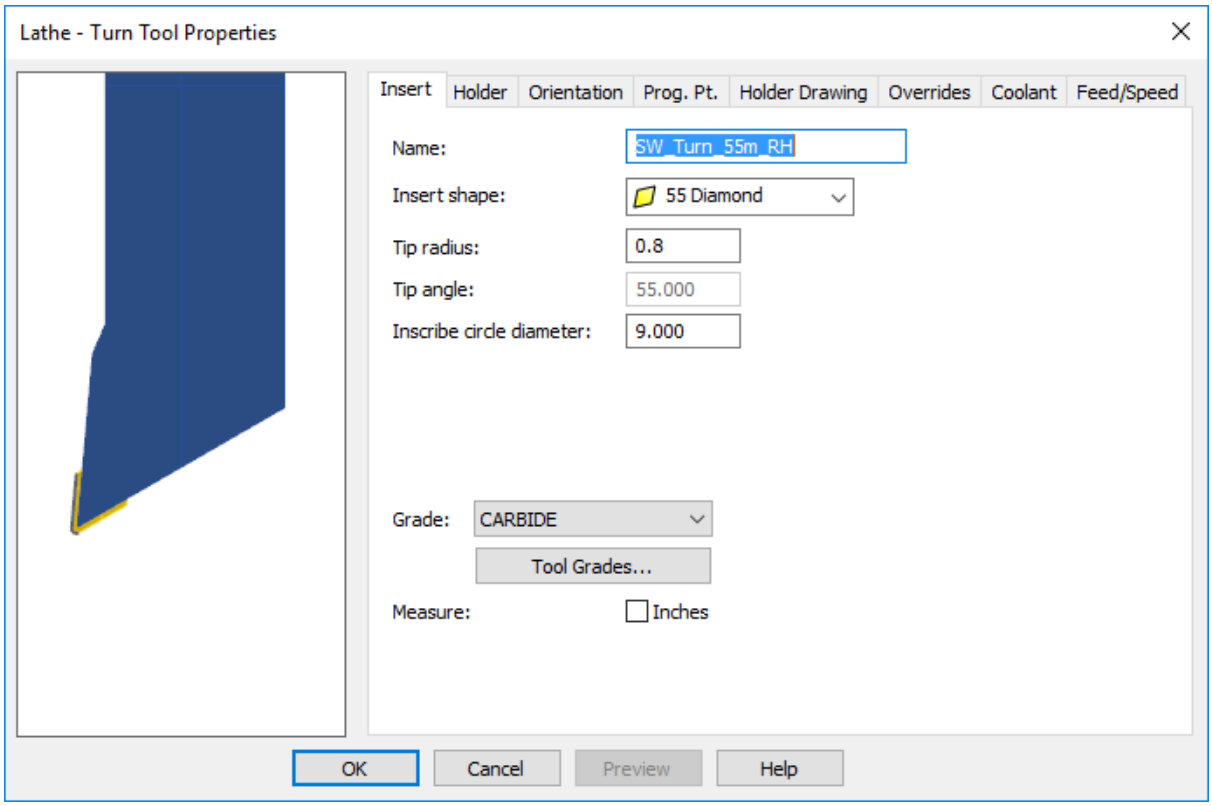


The bottom tool will be used for Back Turning.

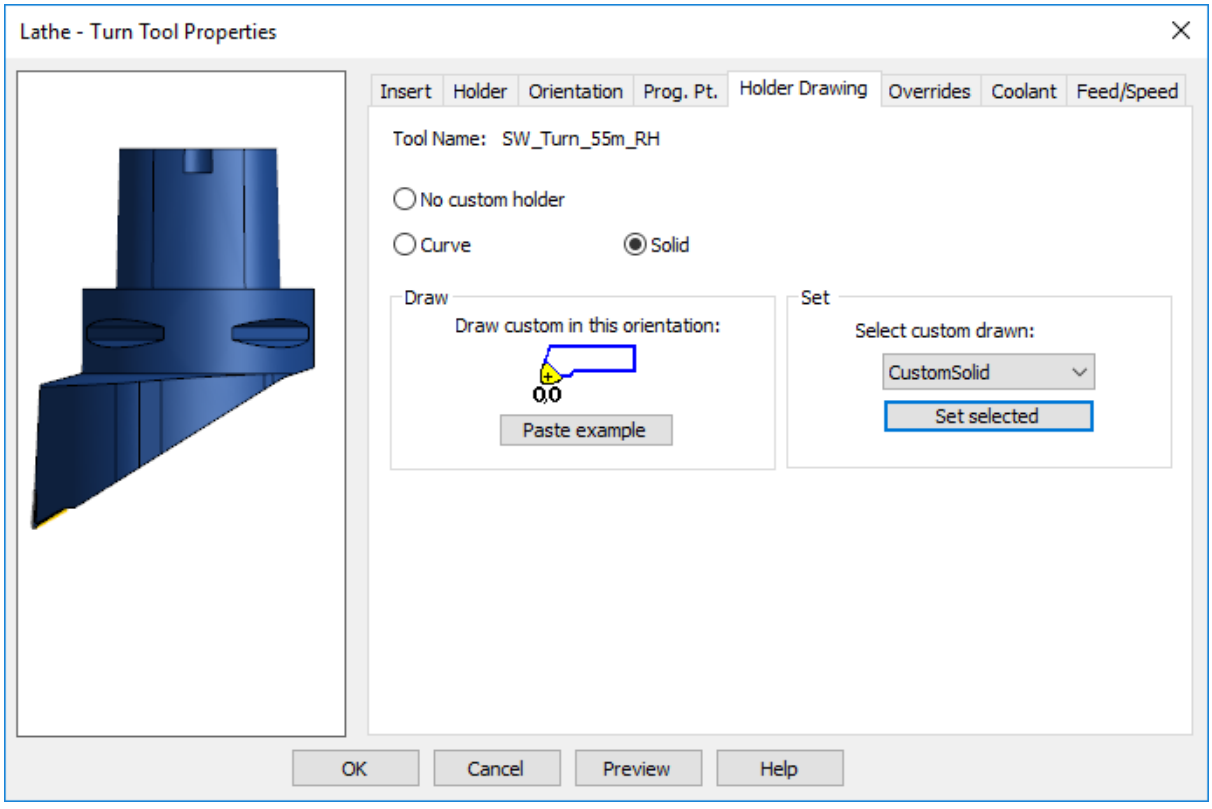
8 Open the Tool manager from the Manufacturing menu.

9 Select the **SW_Turn_55m_RH** turning tool by double clicking on that tool.

© Copyright 2016 Autodesk, Inc. All rights reserved

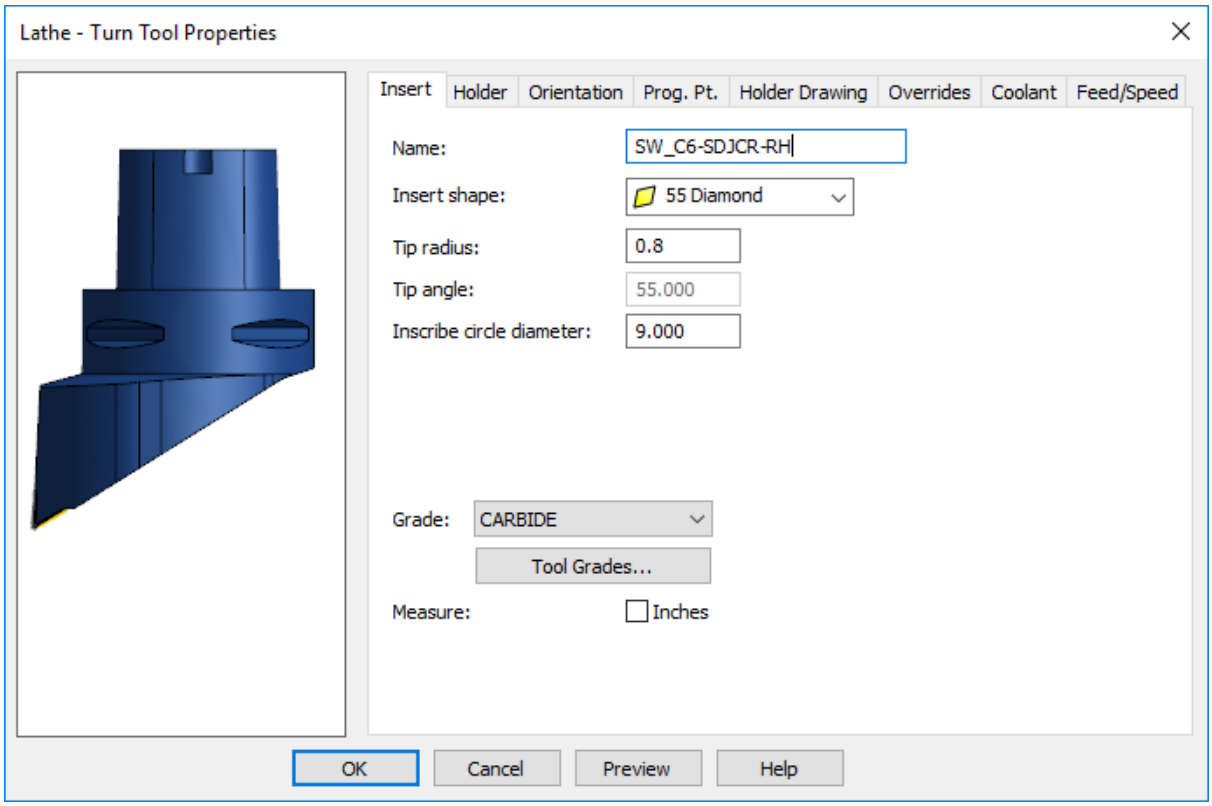


10 Select the **Holder drawing** menu and select the first solid model as shown.



11 Change the name of the Tool to **SW_C6-SDJCR-RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

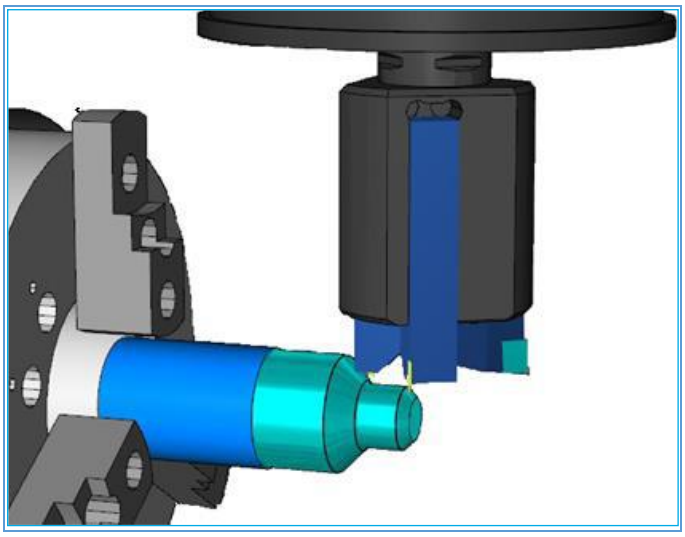


*The insert Inscribe circle is **9.0**, insert Shape is **55deg Diamond**.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Mini-turrets

You can simulate mini-turrets, which are tools with multiple inserts where the tool rotates around the b-axis to access each insert. The tools are simulated simultaneously, which enables you to check for gouges with the tools that are not currently in use.



This is only available in the machine design file (Machine Tool Simulation) which is an extra cost option.

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

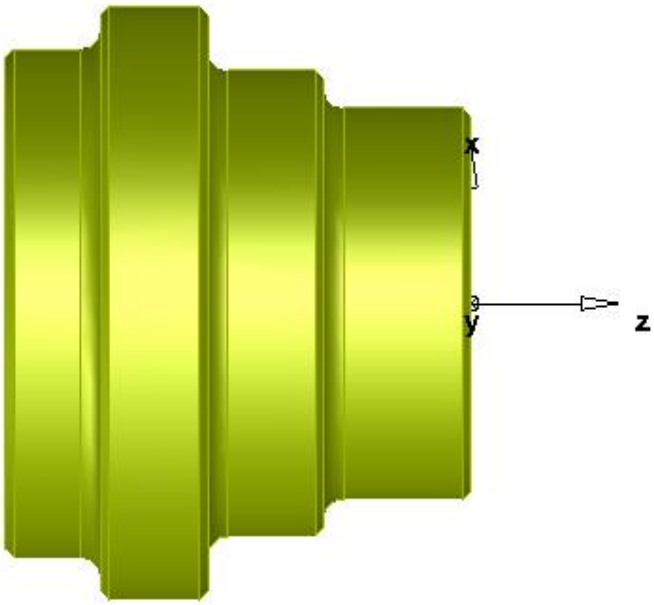
AFR Turning Example 1 - **Part4 Turning** 2
AFR Turning Example 2 - **Turnsolid** 18
Turnsolid - Manipulating the model to the correct orientation ready for AFR. .. 19
Using a Solid model for stock 29
Feature Recognise additional models..... 36

© Copyright 2016 Autodesk, Inc. All rights reserved

AFR Turning Example 1 - **Part4 Turning**

Automatic Feature Recognition recognises different features on solid models and automatically creates toolpaths for different setups. FeatureCAM will select the relevant tooling, but will warn you if it cannot find the correct tooling. Using the **AFR Wizard**, this will guide you through the process from aligning the part to creating the stock and then creating the features for your part. We will go through this process on an actual job from the **Turning Files to Import** directory**. C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**

1 We will start off with a relatively easy part assuming that we have a simple **2 axis cnc turning centre**. The part will be created with **two setups**.

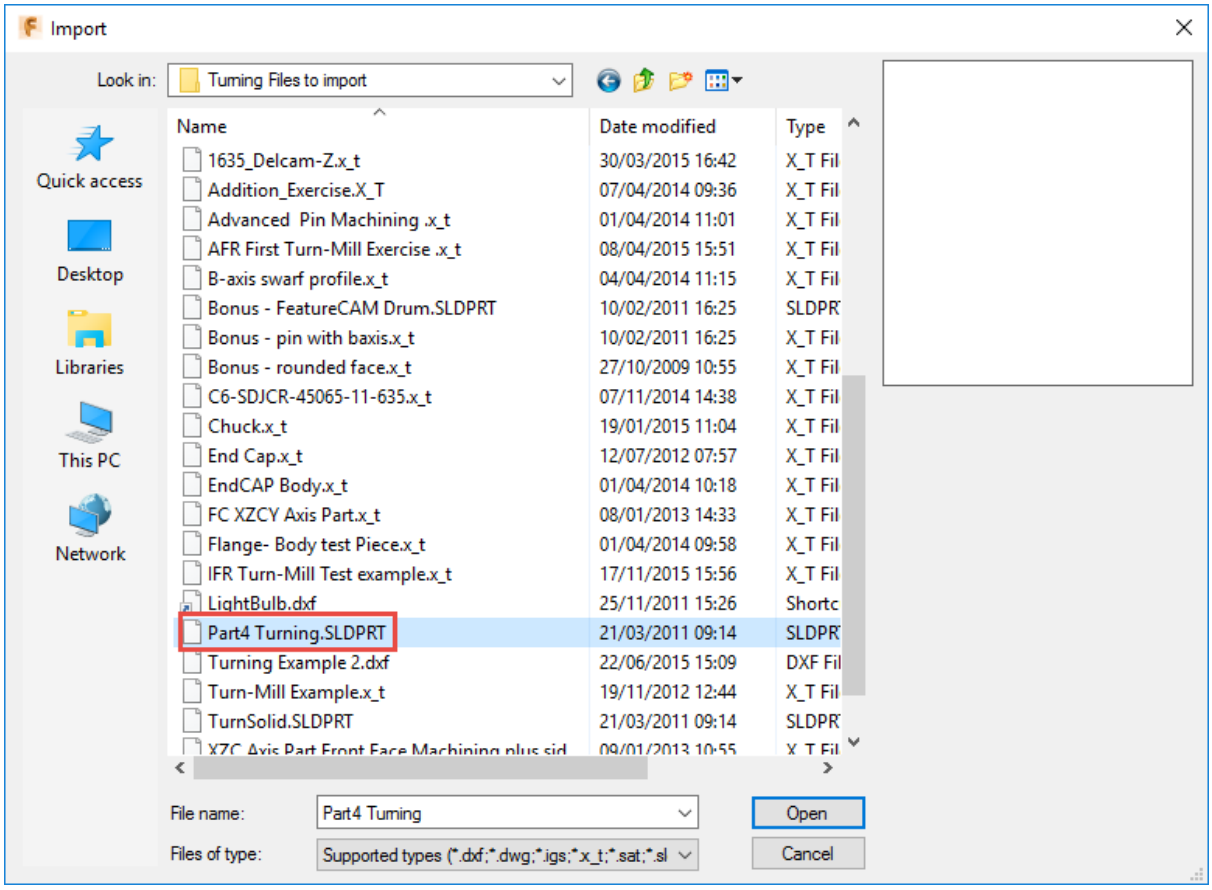


2 We will start by importing the model.

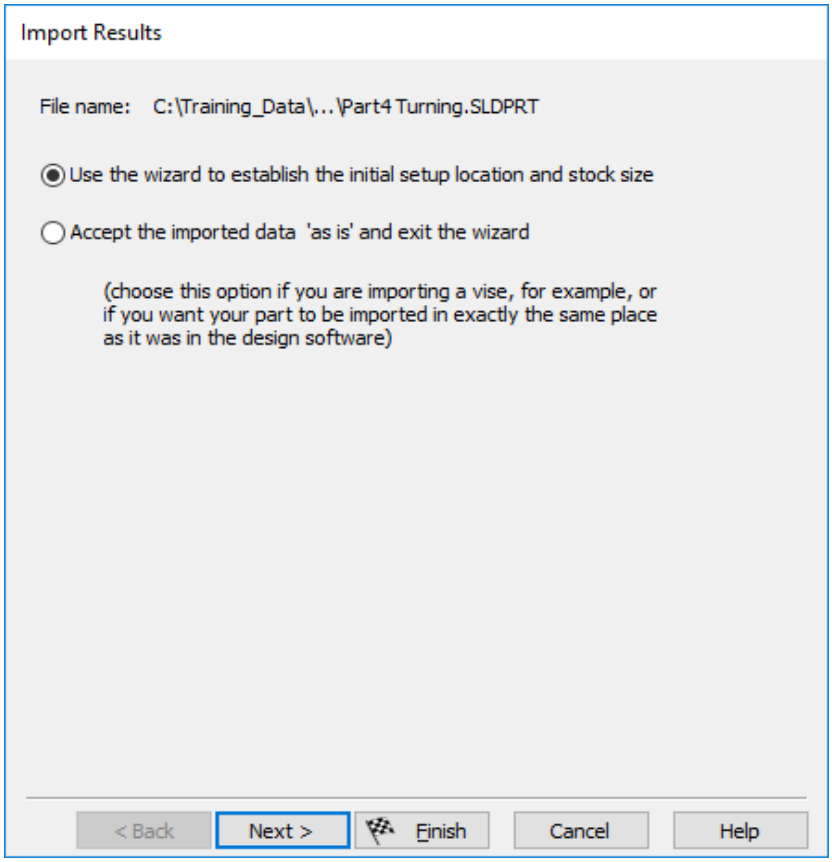
3 Go to **File>Import C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import.**

© Copyright 2016 Autodesk, Inc. All rights reserved

4 Import the following part. **Part4 Turning.SLDPRT**

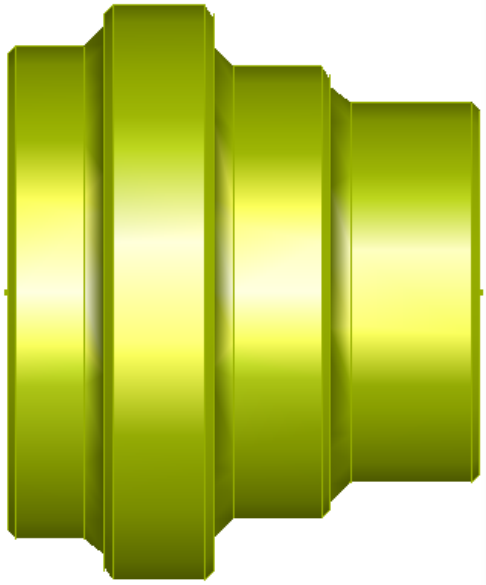


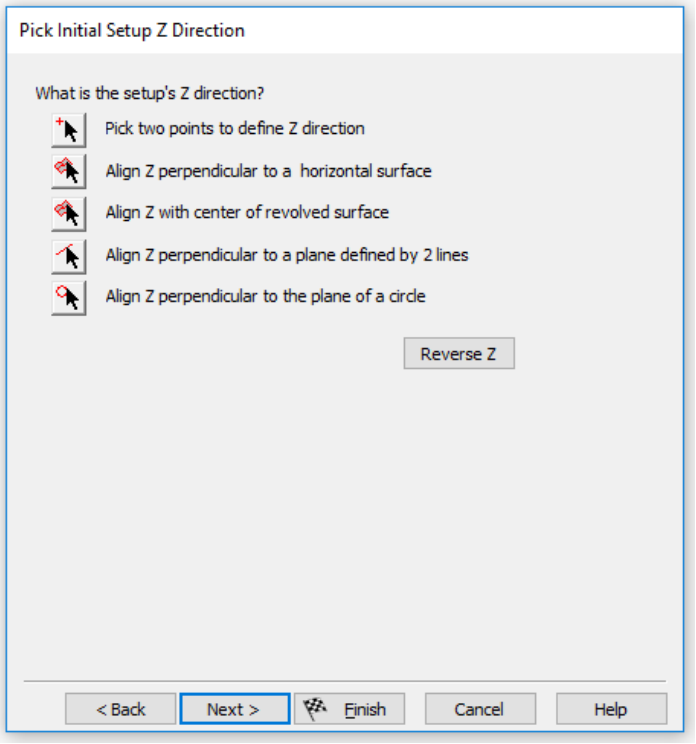
5 The **Wizard** will guide you through the process.

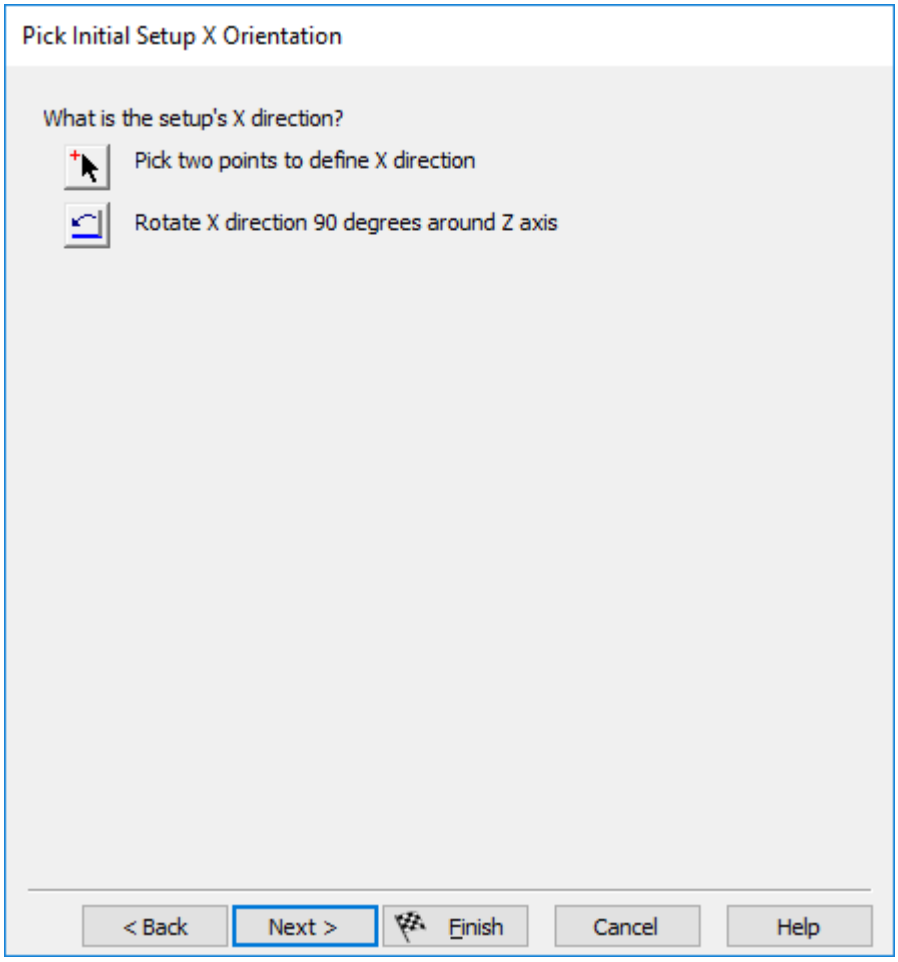


© Copyright 2016 Autodesk, Inc. All rights reserved

6 Is the part properly aligned with the Z axis? In this case the answer is **yes**.





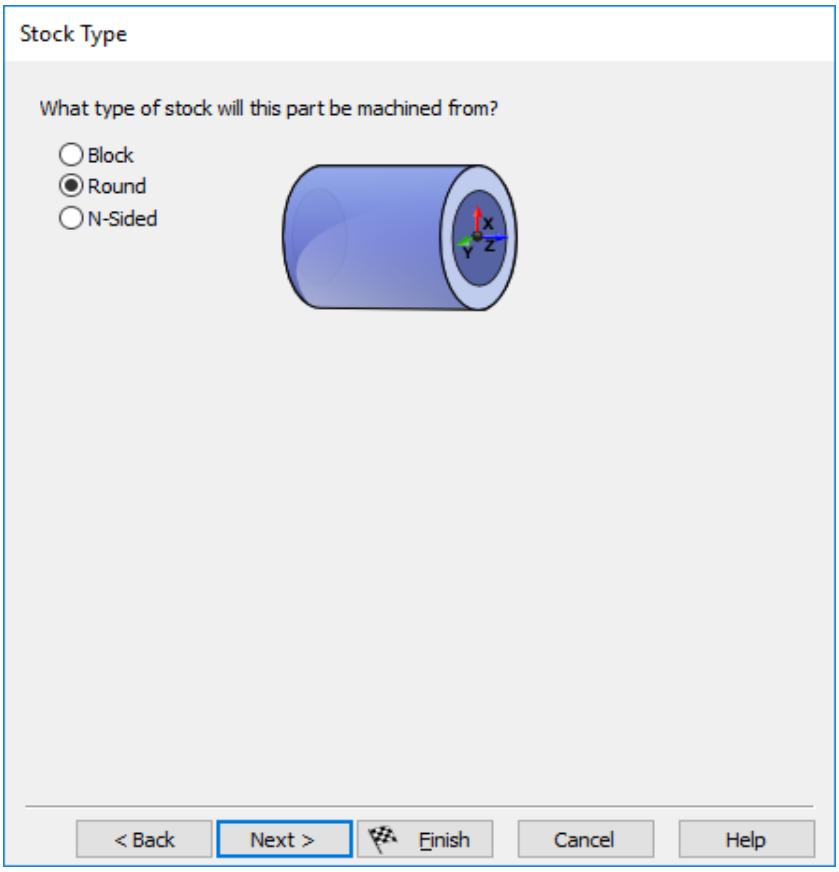


7 Select **Next**.

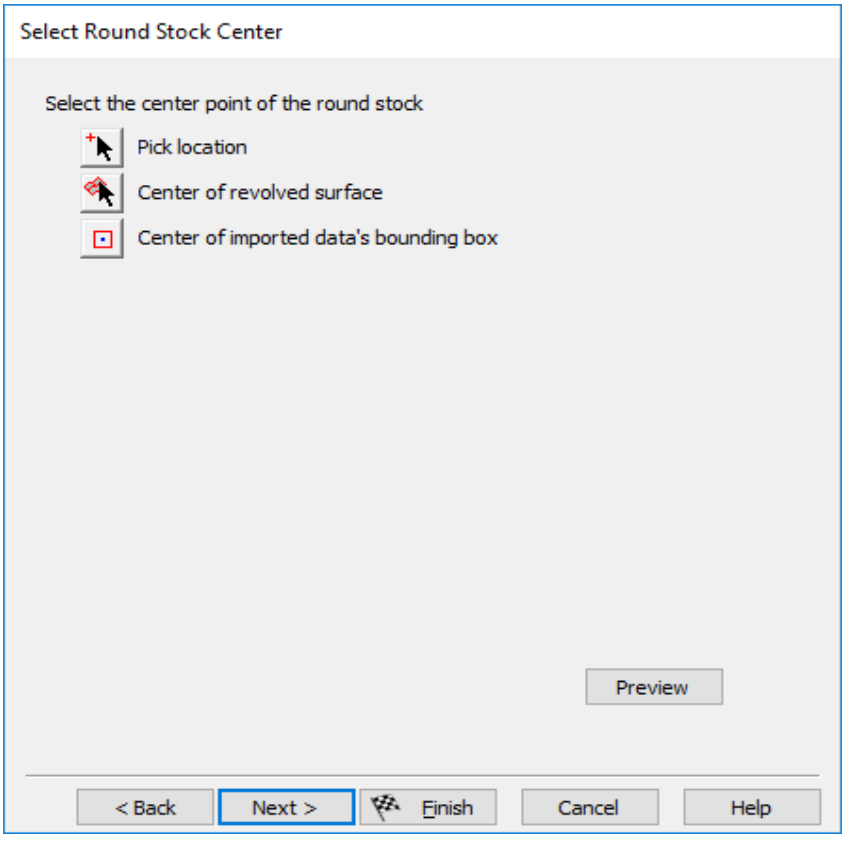
© Copyright 2016 Autodesk, Inc. All rights reserved

8 The next Menu gives you the option to select the Stock Type.

9 For our turned part select **Round** as shown.



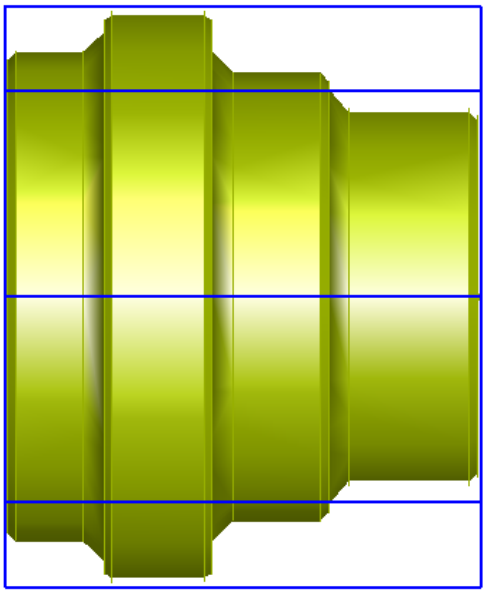
10 The next available menu will ask you to select the **Centre** of the round stock.

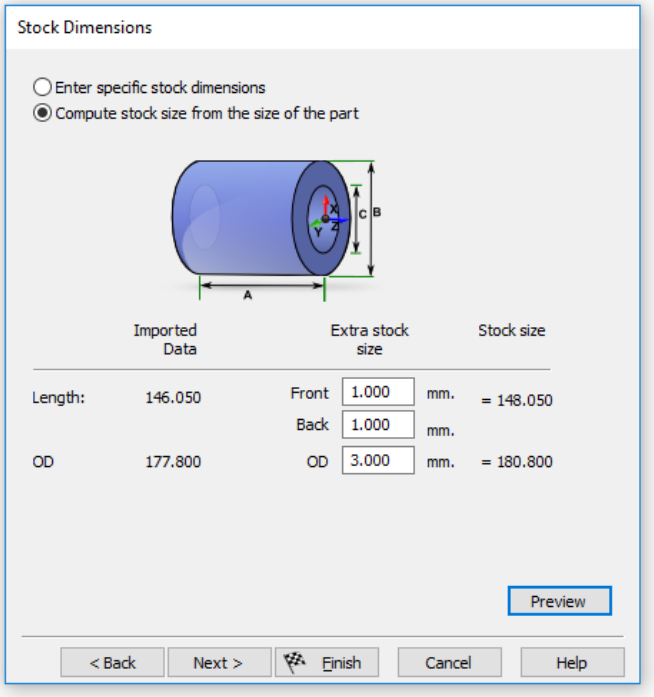


© Copyright 2016 Autodesk, Inc. All rights reserved

11 Select **Next**.

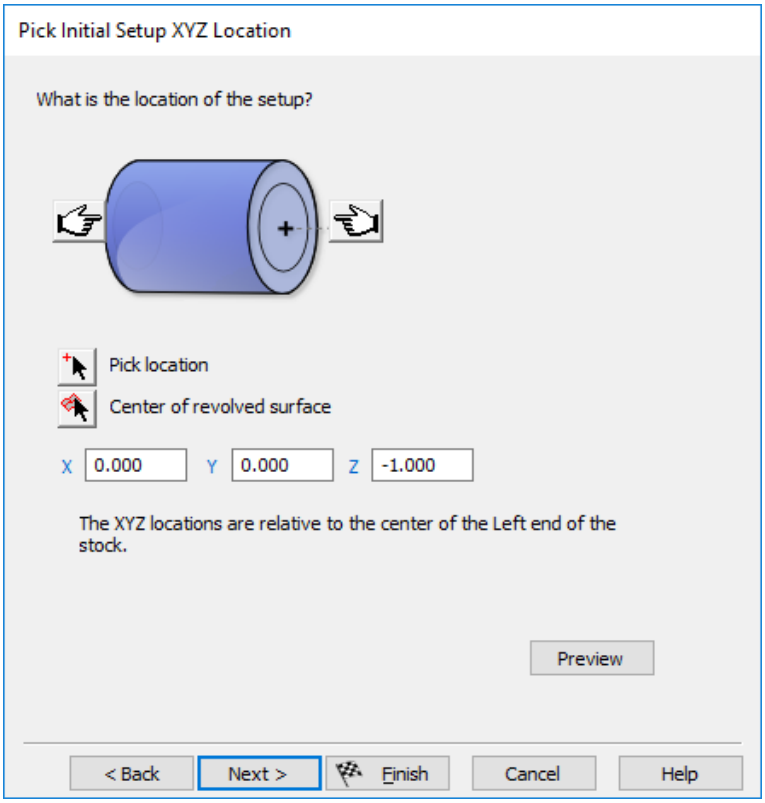
12 The next available menu will give you two options. **Option1**- **Enter specific stock dimensions** or **Option 2** – **Compute stock size from the size of the part**. Please select **option 2**.





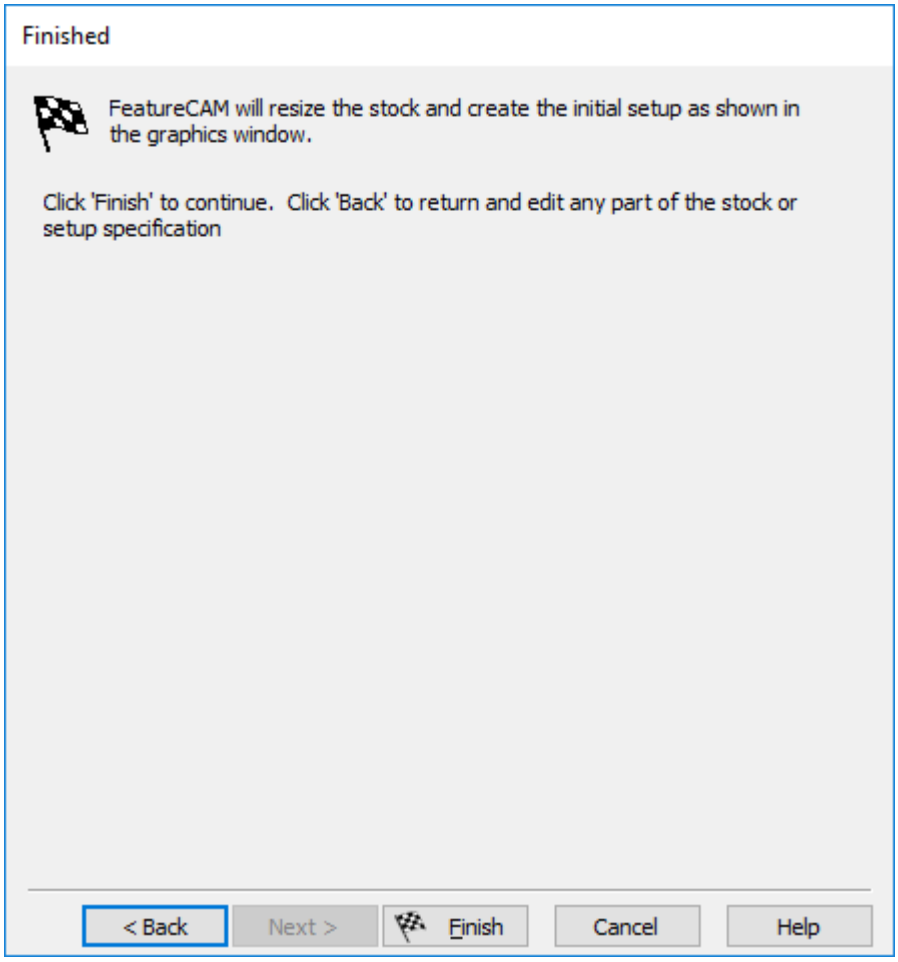
13 Enter the above coordinates. This allows **1mm** on the front and back faces plus **3mm** on diameter.

14 Select Next. You will notice that the Z datum shows **-1.000**. FeatureCAM knows how much material you have left on from the previous menu.

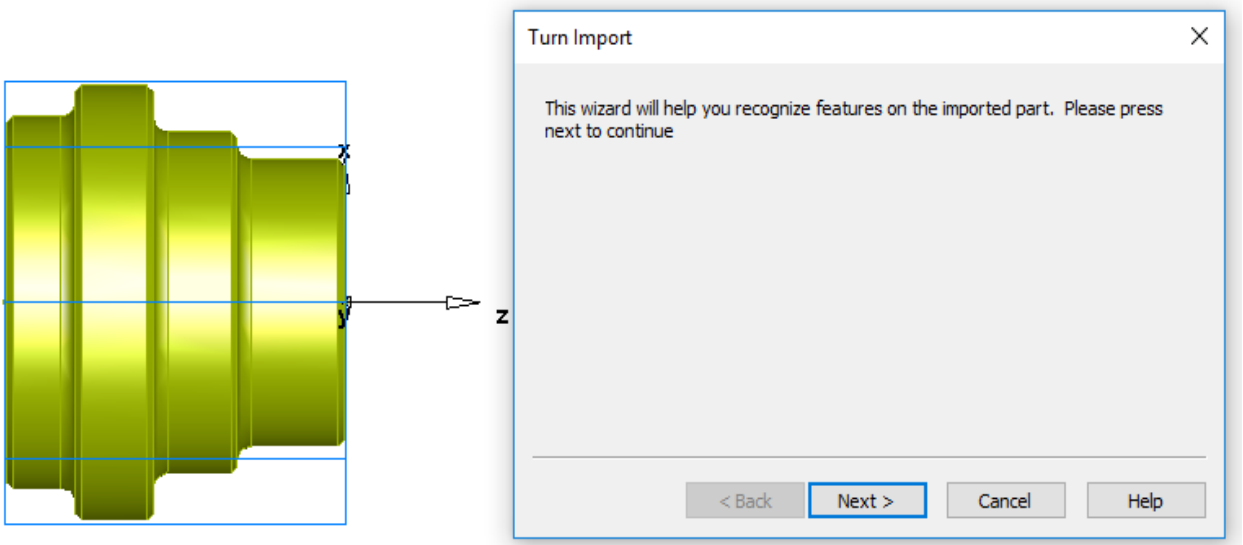


© Copyright 2016 Autodesk, Inc. All rights reserved

15 Select Next.



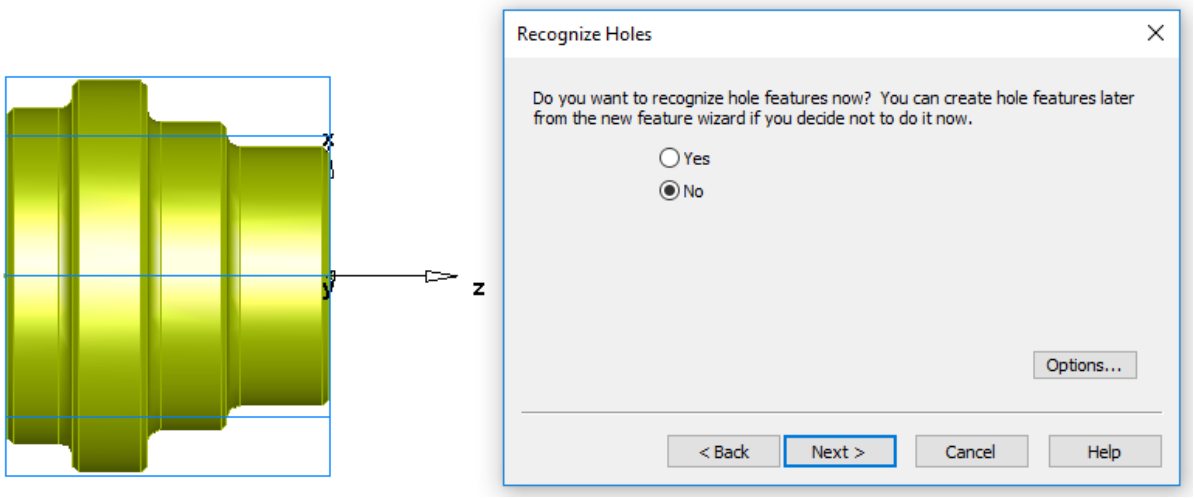
16 Select **Finish.** The following menu will be shown.



17 Select **Next.**

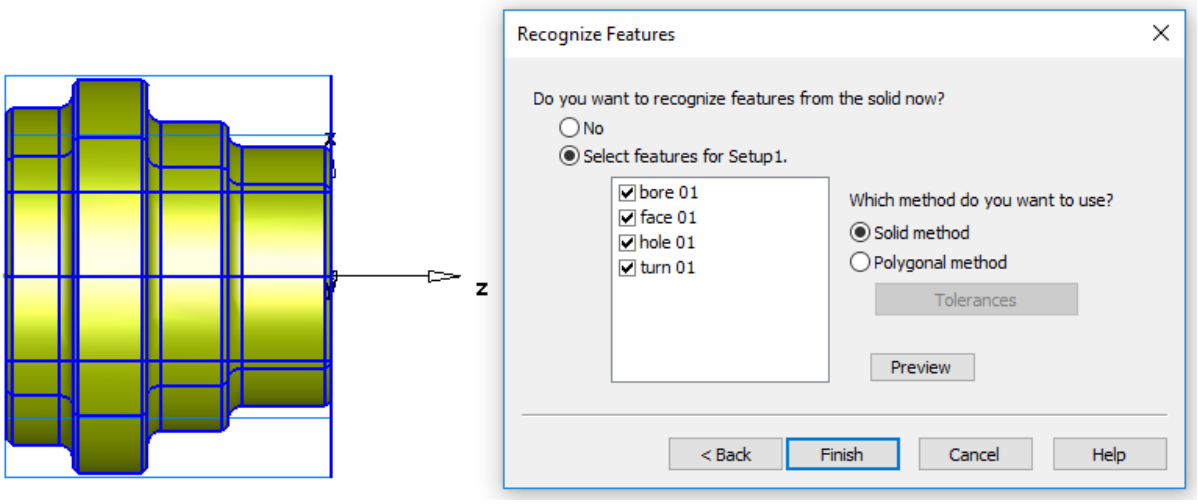
18 You will be asked if you want to **Recognise Hole Features now**. Select **No**.

© Copyright 2016 Autodesk, Inc. All rights reserved



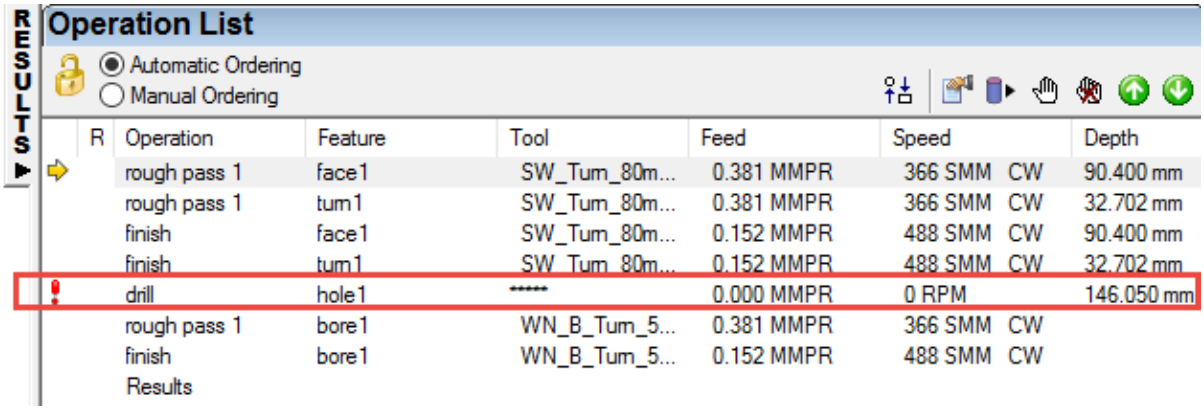
19 Select **Next**.

20 Please select **Select features for Setup1**.



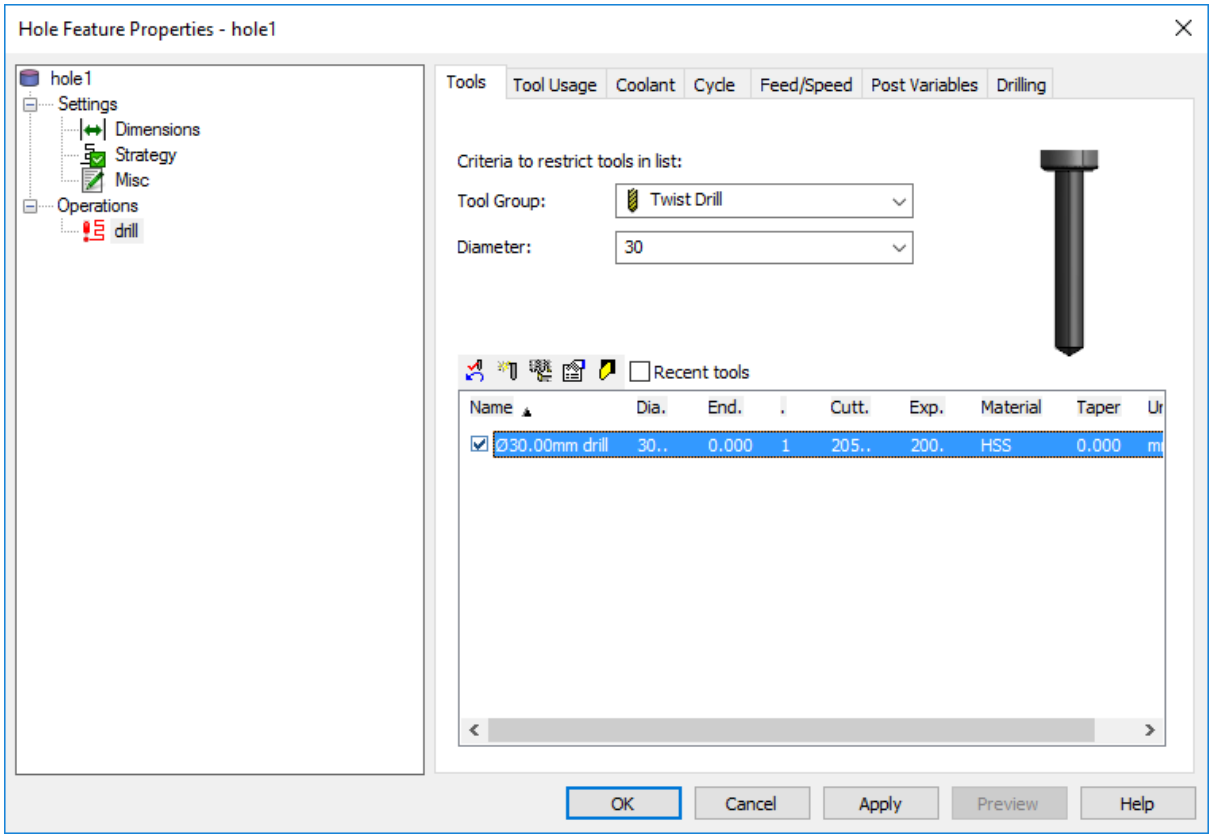
21 Select **Finish**.

22 In your operation list you will get a warning?



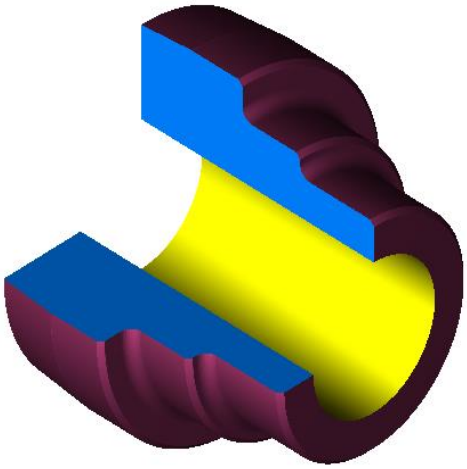
© Copyright 2016 Autodesk, Inc. All rights reserved

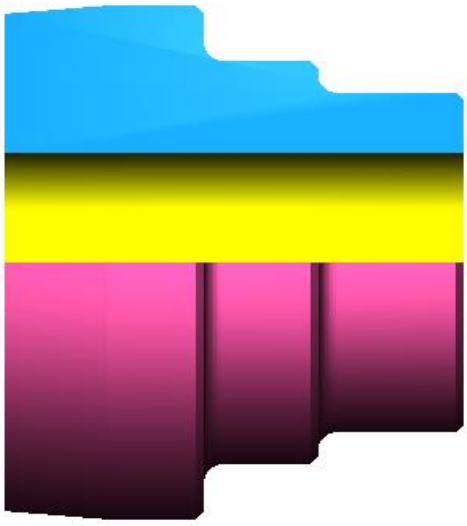
23 For the **Hole1** Feature. Adjust the hole roughing tool to a **30mm Drill**. From the **Training_Crib_Metric** crib this will not exist, so create a new tool. Choose an existing drill of a smaller size and change the values. Remember to change the name so you can create a new tool. To do this double click the on the following ***** this will open the tooling form.



24 After changing the drill you may also have to change the **minimum Radius boundary** to **15mm** in the **Bore1** Feature. Change depth of cut to **2mm**. Run a **3D Simulation** to view results from the **Automatic Feature Recognition**.

25 Remember that a **Bore Feature** has a **Pre-Drill** option that combines the two strategies into one.



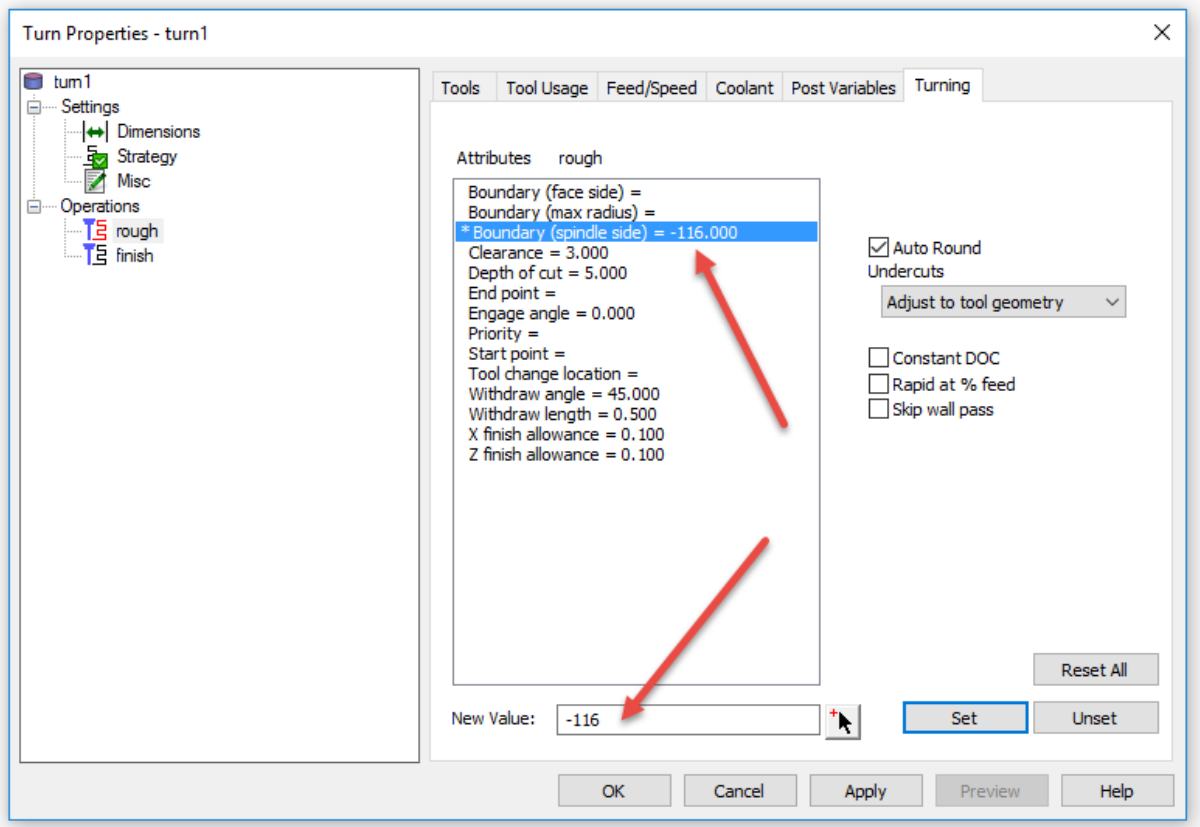


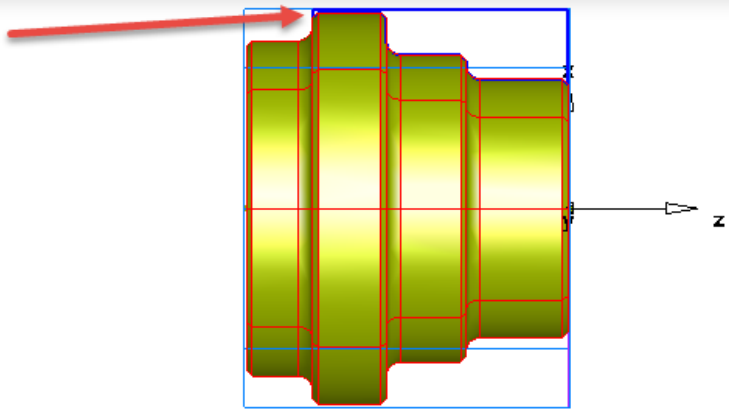
© Copyright 2016 Autodesk, Inc. All rights reserved

Notice how the tool will machine as much as it can, which depends on which tool is selected. (see image)

26 Try using **Select left boundary** and limit the length to the back shoulder. The chuck will be damaged if you have not left enough material at the back of the part. Select **Turn1** by double clicking in **Part View**. Select **Roughing.**

27 Select **Pick Z Location.**

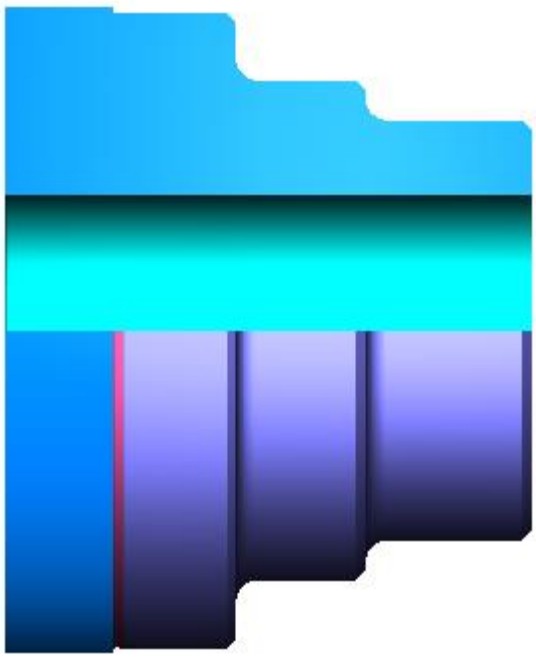




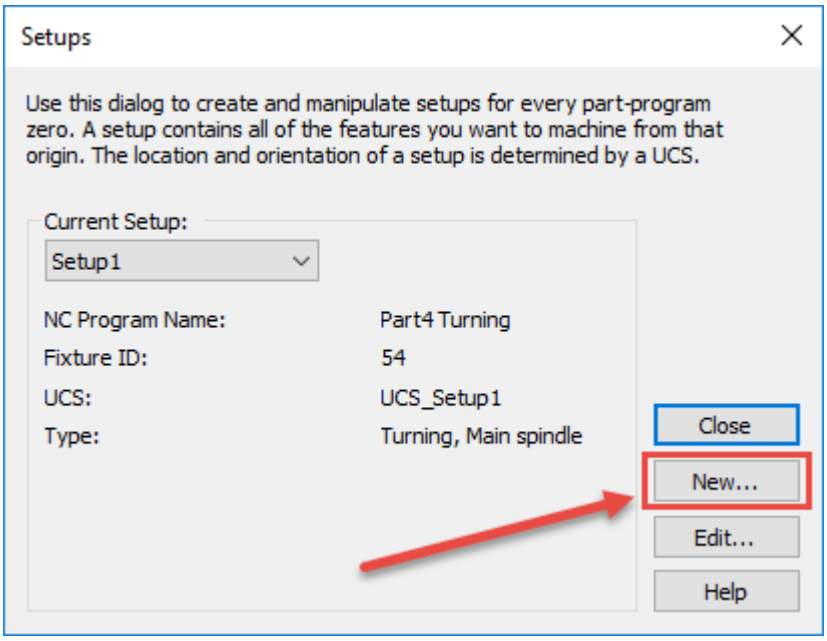
*The Z Value is **-116.0** the * denotes that the value has changed.*

28 Re-Run the **3D Simulation**.

© Copyright 2016 Autodesk, Inc. All rights reserved

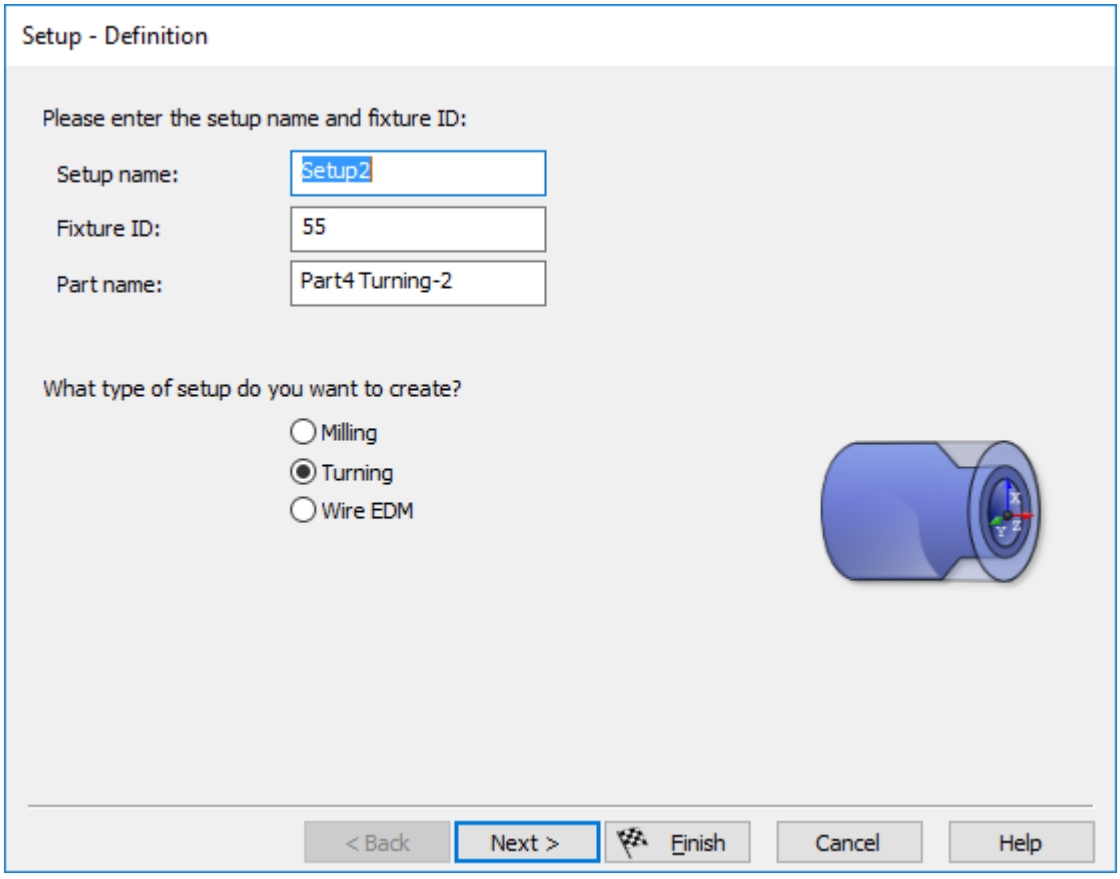


We can now create **Setup2**. Double click on **Setup1** following menu will appear.

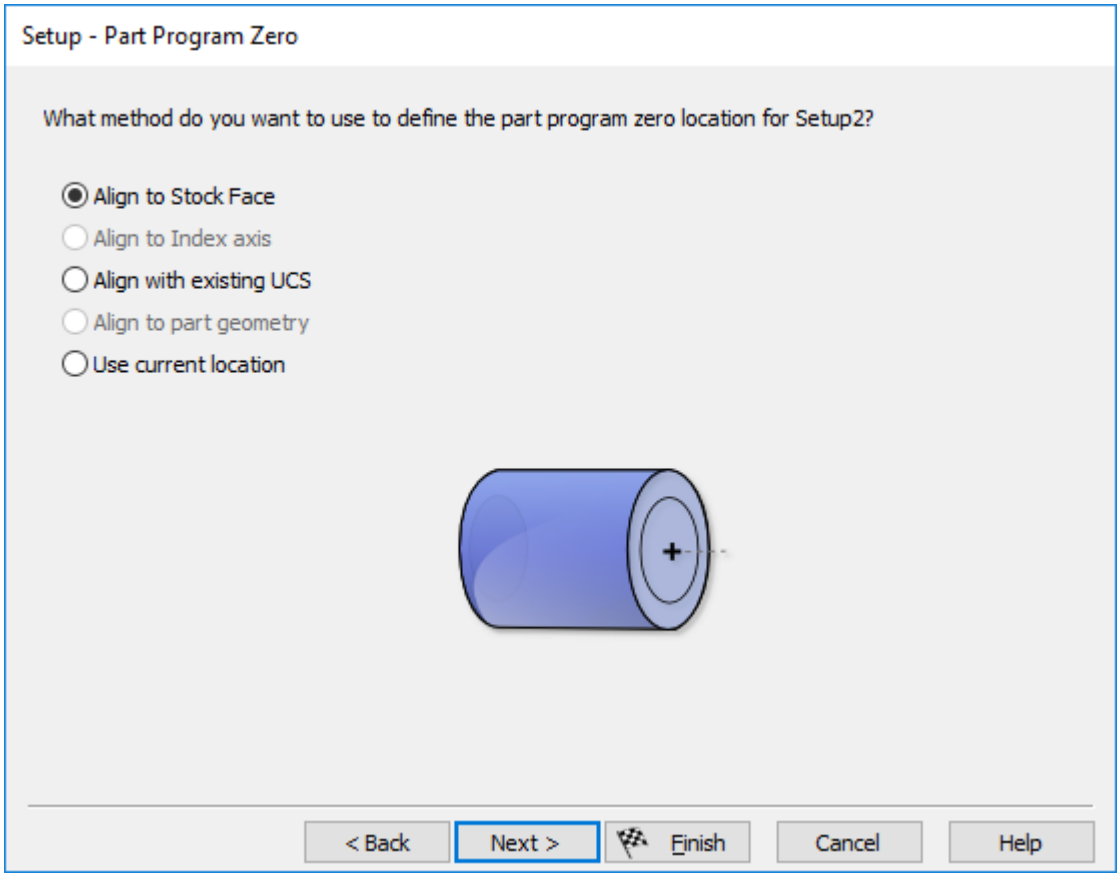


29 Select **New** and the following menu on the next page will be displayed.

© Copyright 2016 Autodesk, Inc. All rights reserved

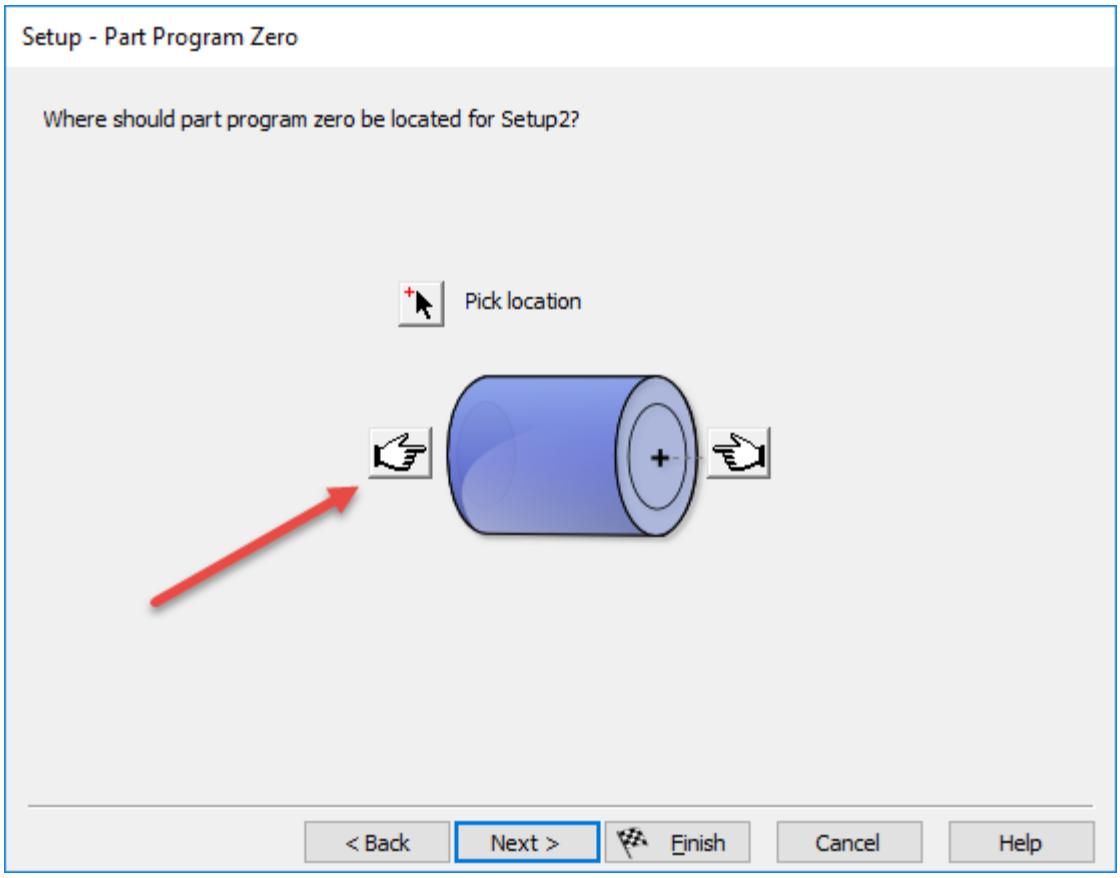


30 Select **Next.**



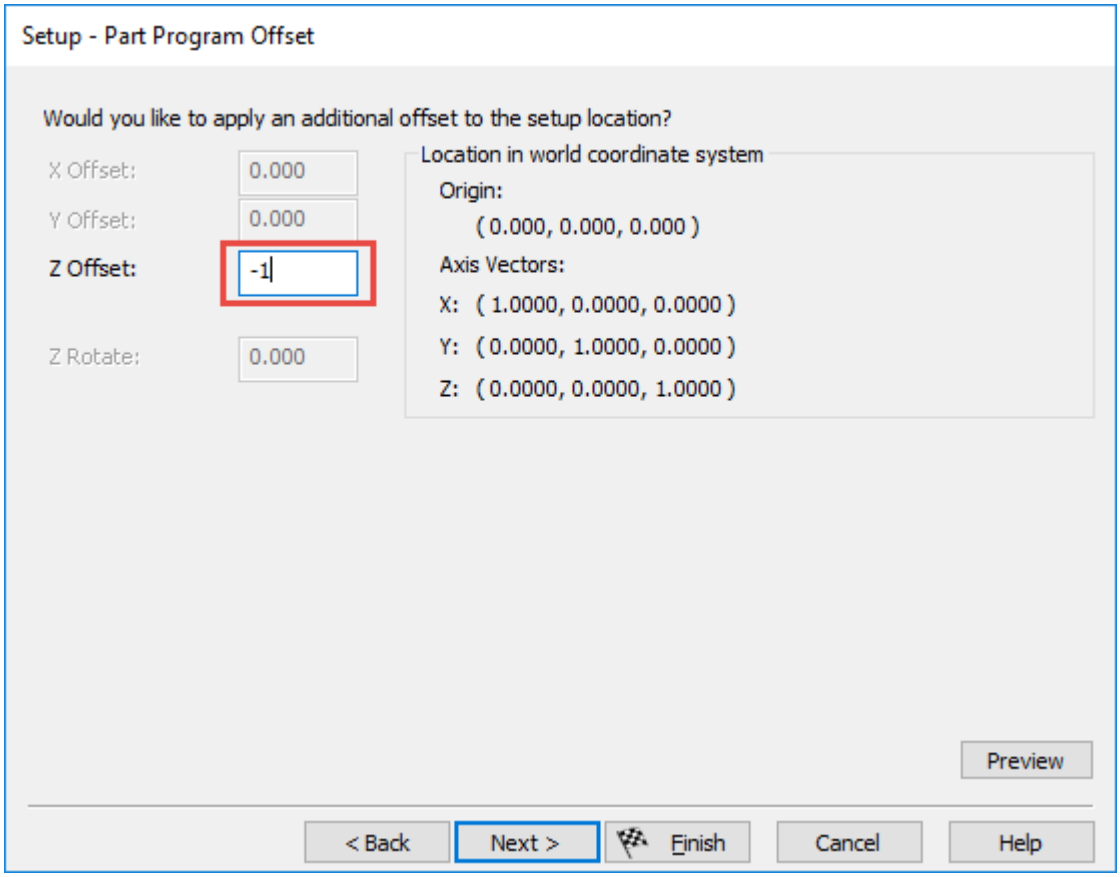
31 Select **Next.**

© Copyright 2016 Autodesk, Inc. All rights reserved



32 Select the **left hand** as shown.

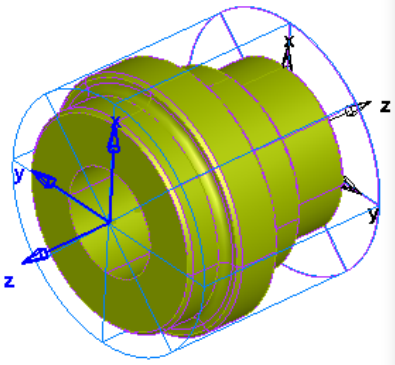
33 Select **Next**. Then set the **Z Offset** to **-1** as shown.

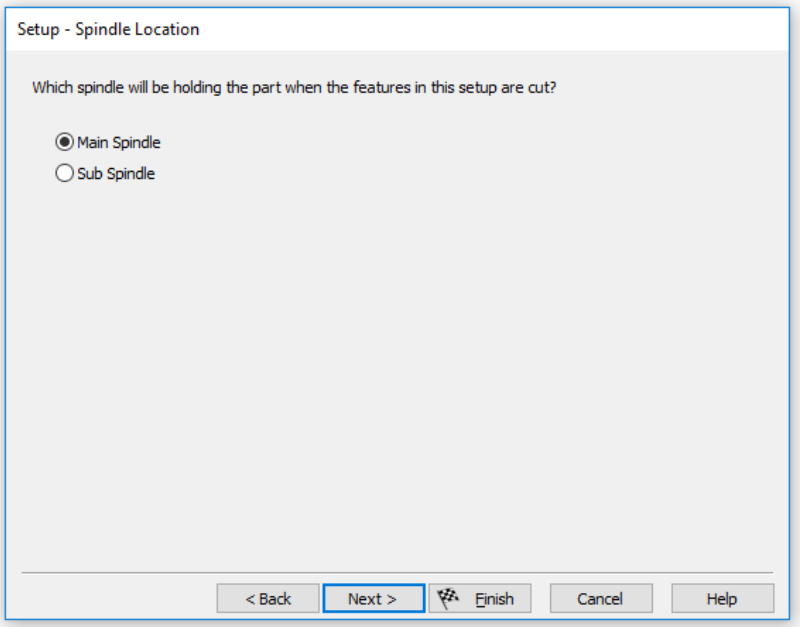


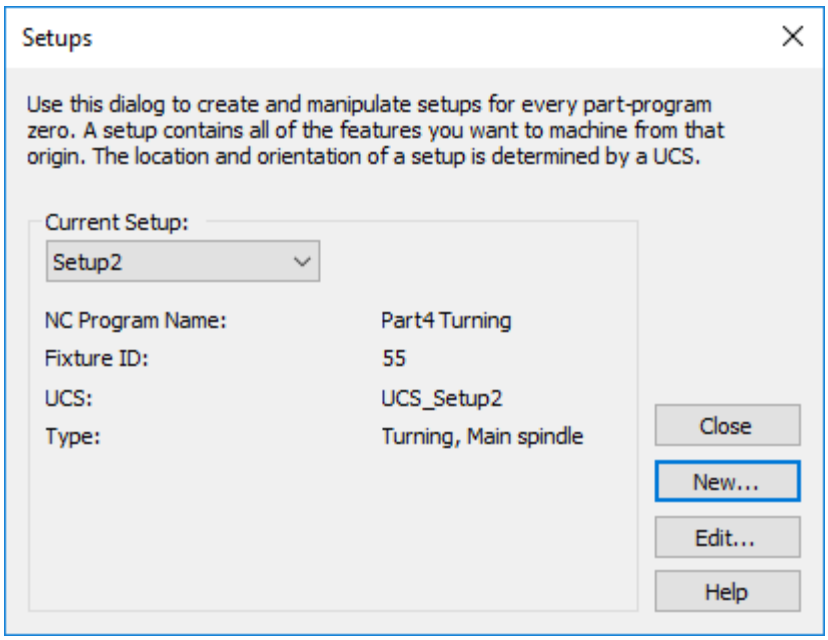
© Copyright 2016 Autodesk, Inc. All rights reserved

This will set the new datum to the opposite face. This is when you take the job out of the chuck and turn it around if you only have one spindle. If you have a Sub-Spindle please select this option. You will have to use Part Handling to transfer the part into the Sub- Spindle.

*FeatureCAM will ask if you want the **main Spindle** or **Sub-Spindle**. See menu on the next page.*



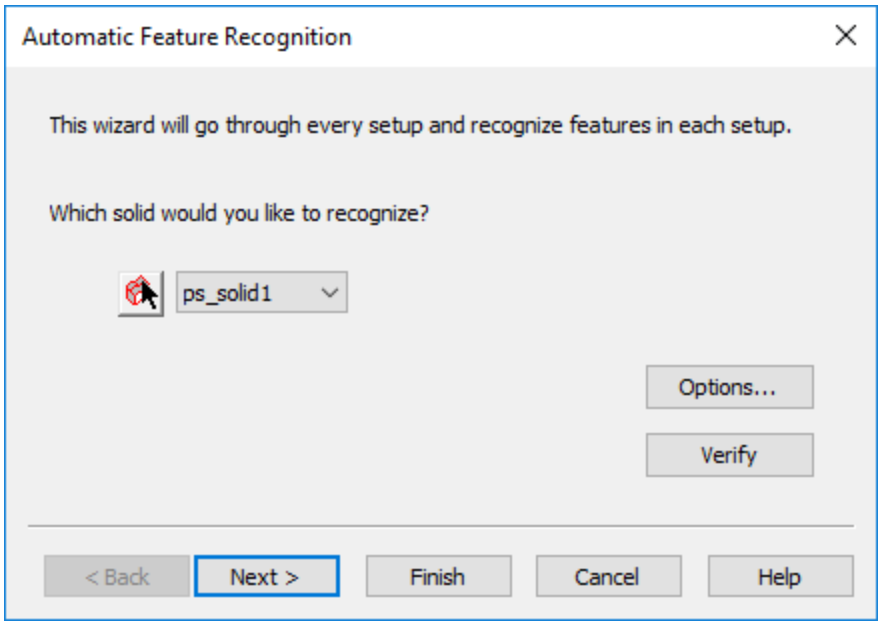




34 We are now ready to apply **AFR** for **setup2**. Select **STEPS** and select **AFR**.

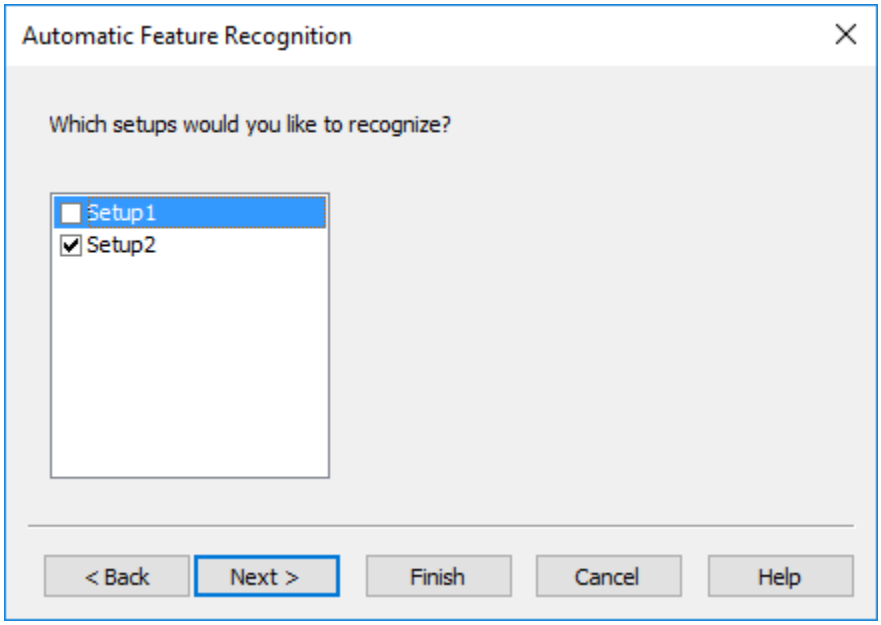
35 From the **Construct** menu please select **Automatic Feature Recognition**.

© Copyright 2016 Autodesk, Inc. All rights reserved



36 Select **Next**.

37 Unselect **setup1** so it will only recognise **setup2**

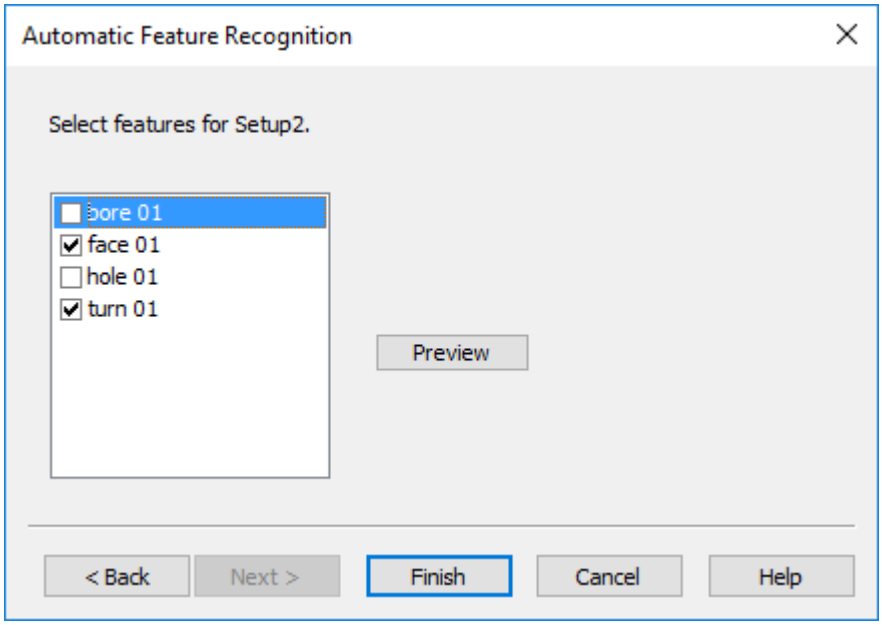


*If you select **Setup1** and **Setup2** this will duplicate operations.*

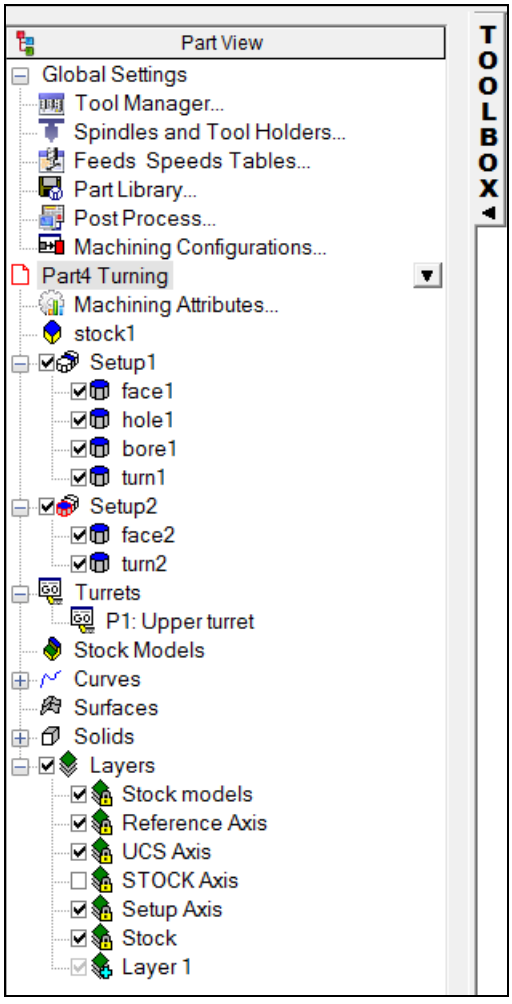
38 You will be given the option to select which Features are relevant to **Setup2**.

39 Please unselect **Bore 01** and **Hole 01** as shown on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved

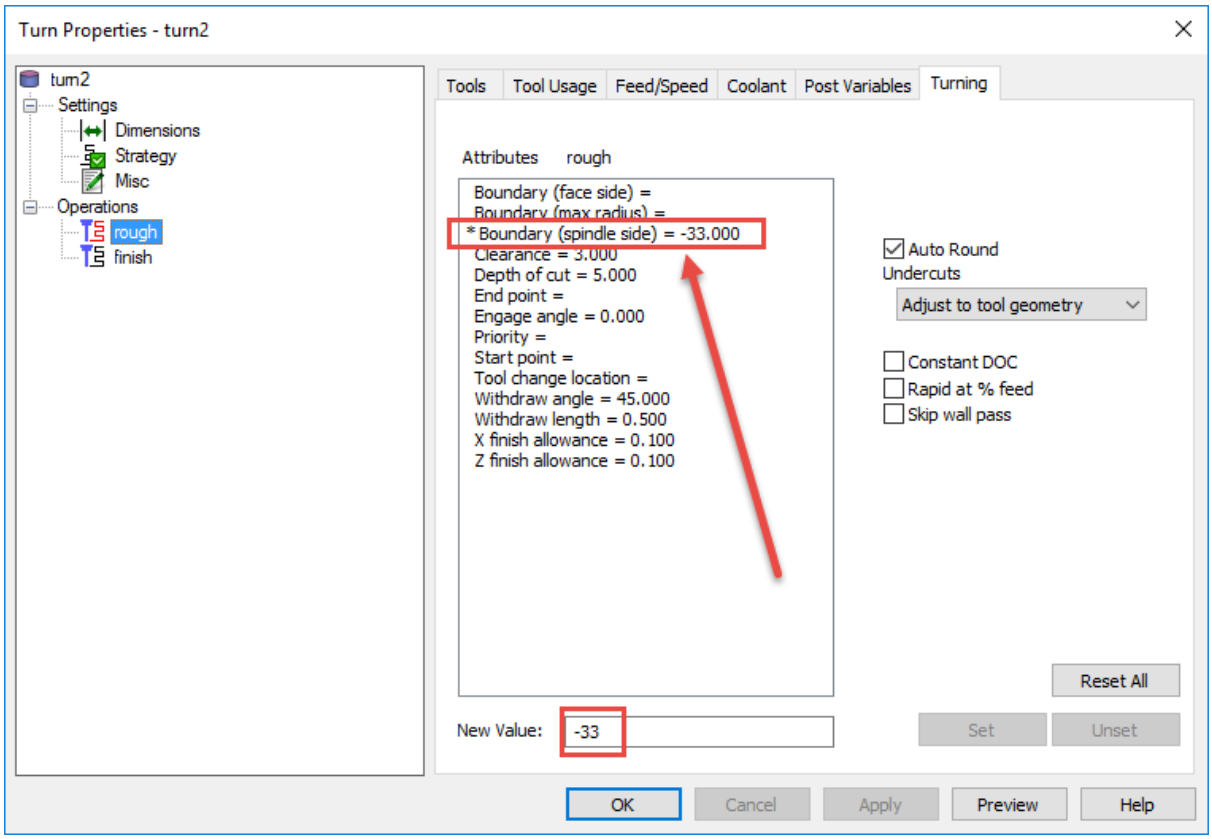


40 Select **Finish**.

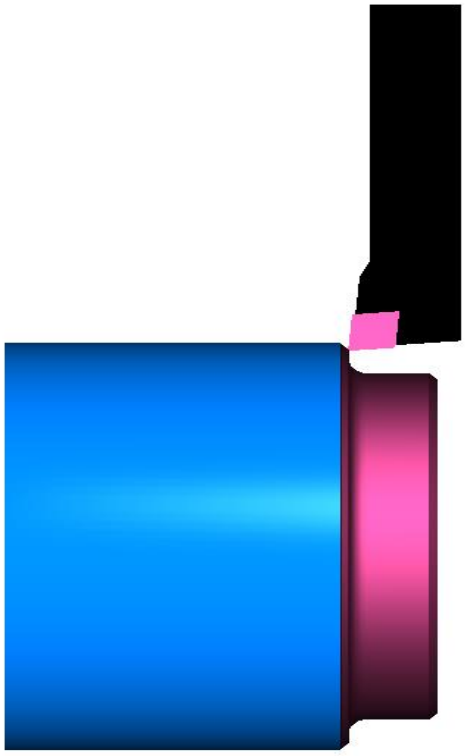


41 We need to adjust **turn2** in **Setup2**. Double click on **turn2** in **Setup2.** Select the Rough pass in the Operation list this will show the following menu.

© Copyright 2016 Autodesk, Inc. All rights reserved

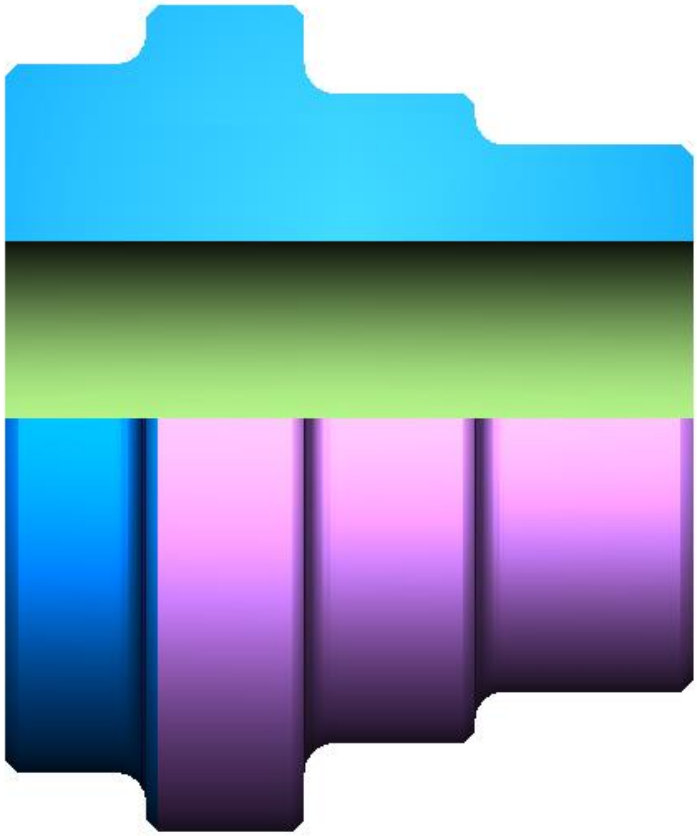


42 This will now only allow the machining up to the point shown. See **3D Simulation.**

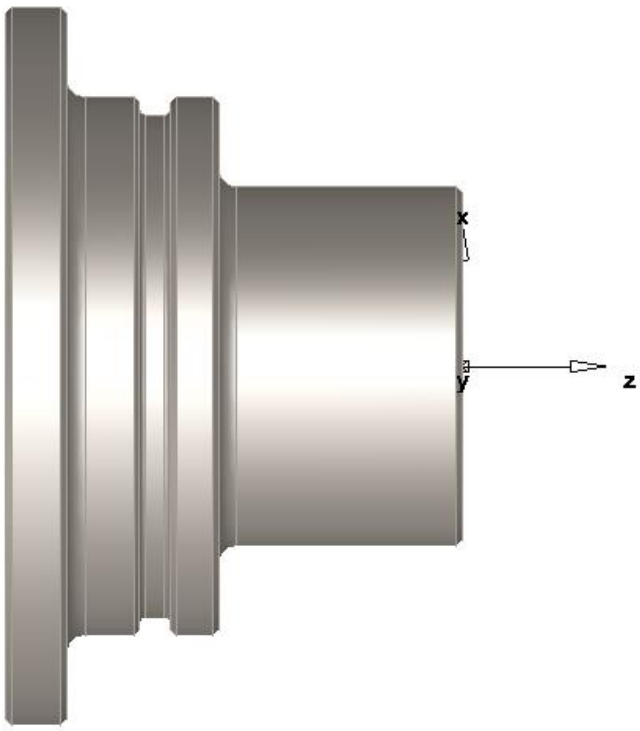


43 The component is now finished see image of both operations on the next page. Change the Post Processor and look at the code output.

© Copyright 2016 Autodesk, Inc. All rights reserved

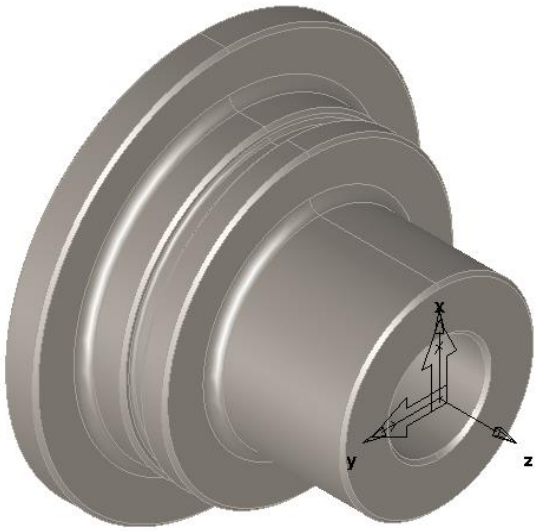


AFR Turning Example 2 - **Turnsolid**



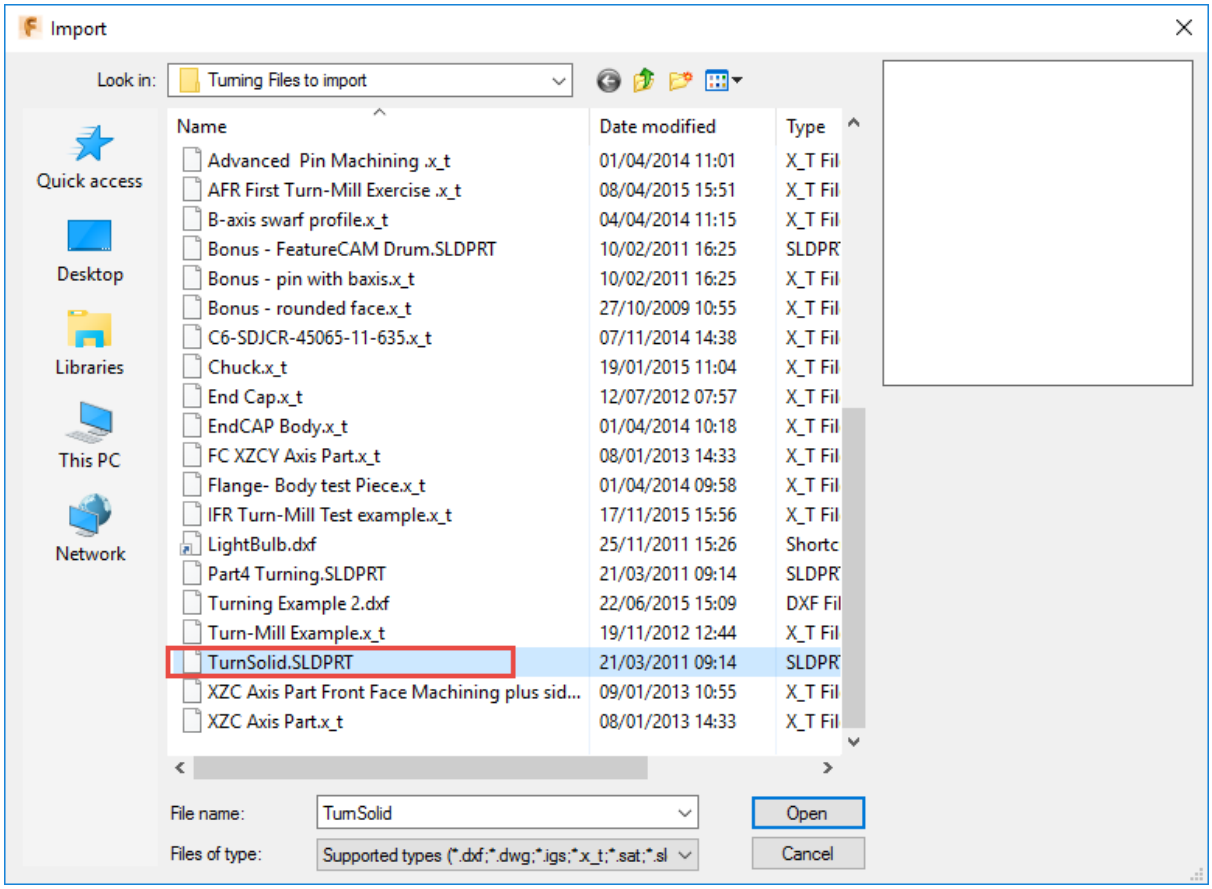
© Copyright 2016 Autodesk, Inc. All rights reserved

Turnsolid - Manipulating the model to the correct orientation ready for AFR.



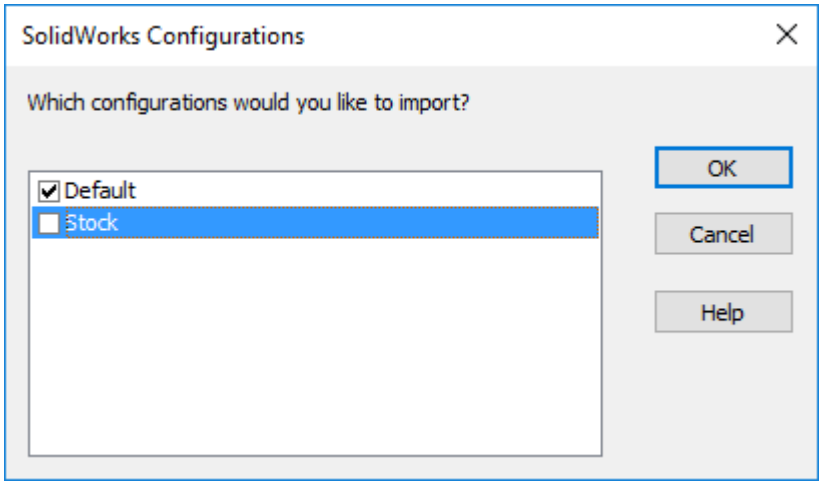
1 We will start by importing the model.

2 Go to **File>Import. C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**

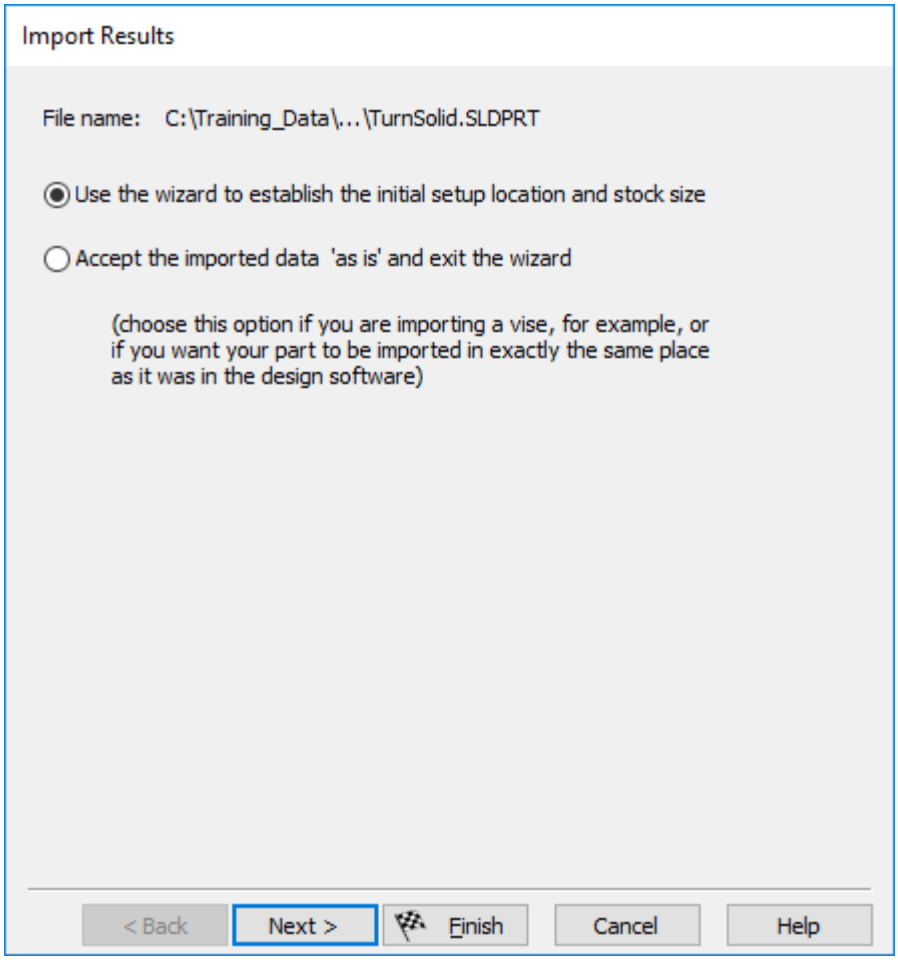


3 Select the file **TurnSolid.SLDPRT** then select **Open**. Please select the Default Configuration only.

© Copyright 2016 Autodesk, Inc. All rights reserved

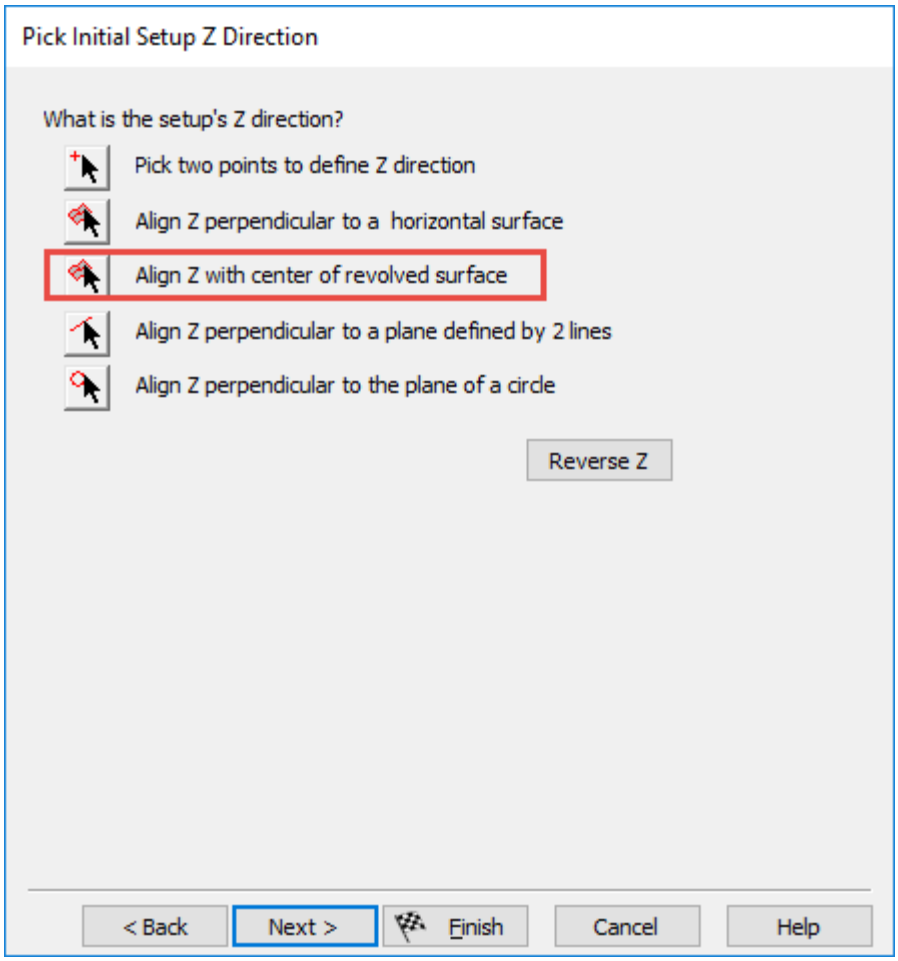


*The part needs to be orientated, we can Use the **Wizard** to establish the initial **Setup location** and **stock size**. This example needs to be rotated.*



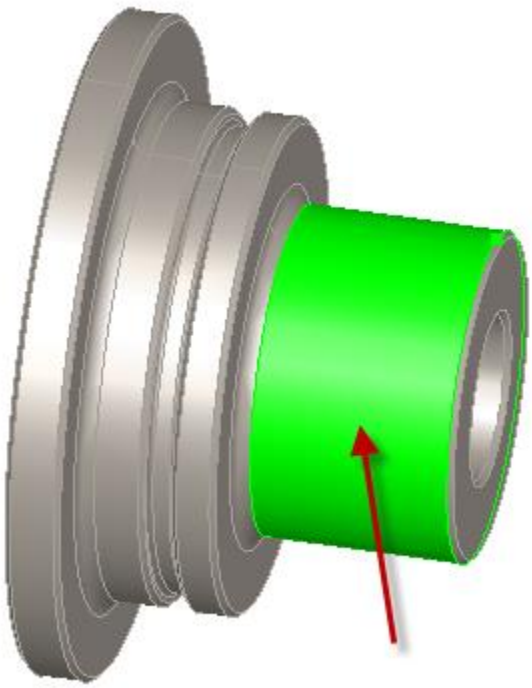
4 Select **Next**.

© Copyright 2016 Autodesk, Inc. All rights reserved



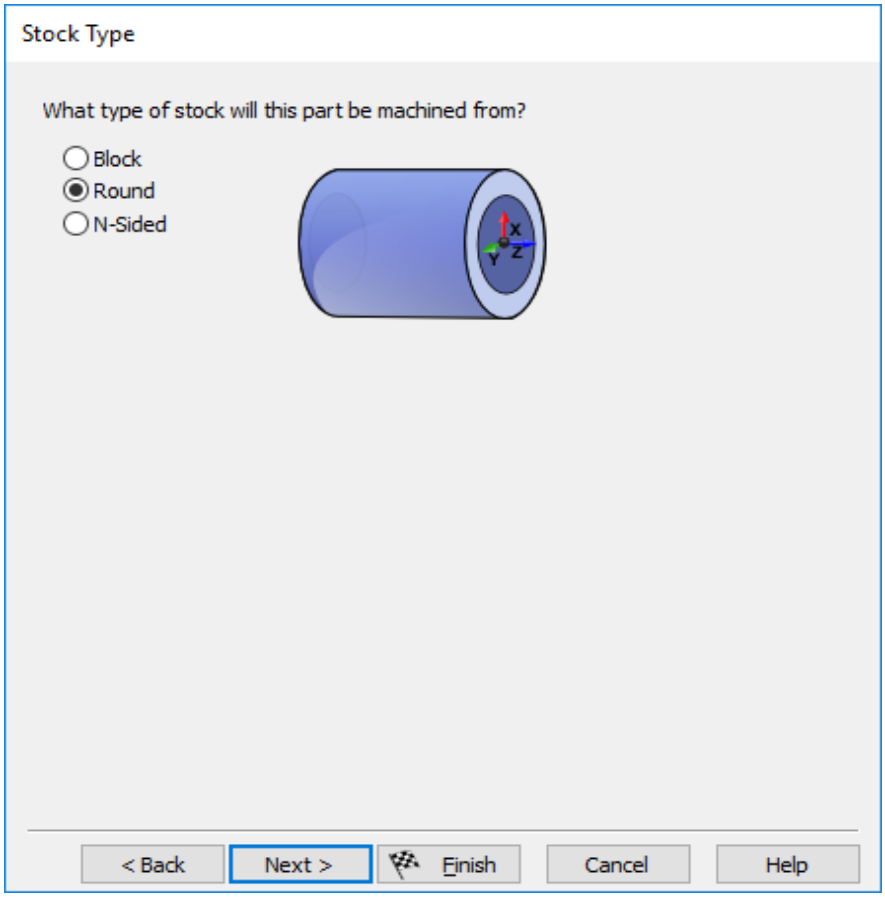
5 Use **Align Z with centre of Revolved Surface.**

6 Select **a revolved Surface**.

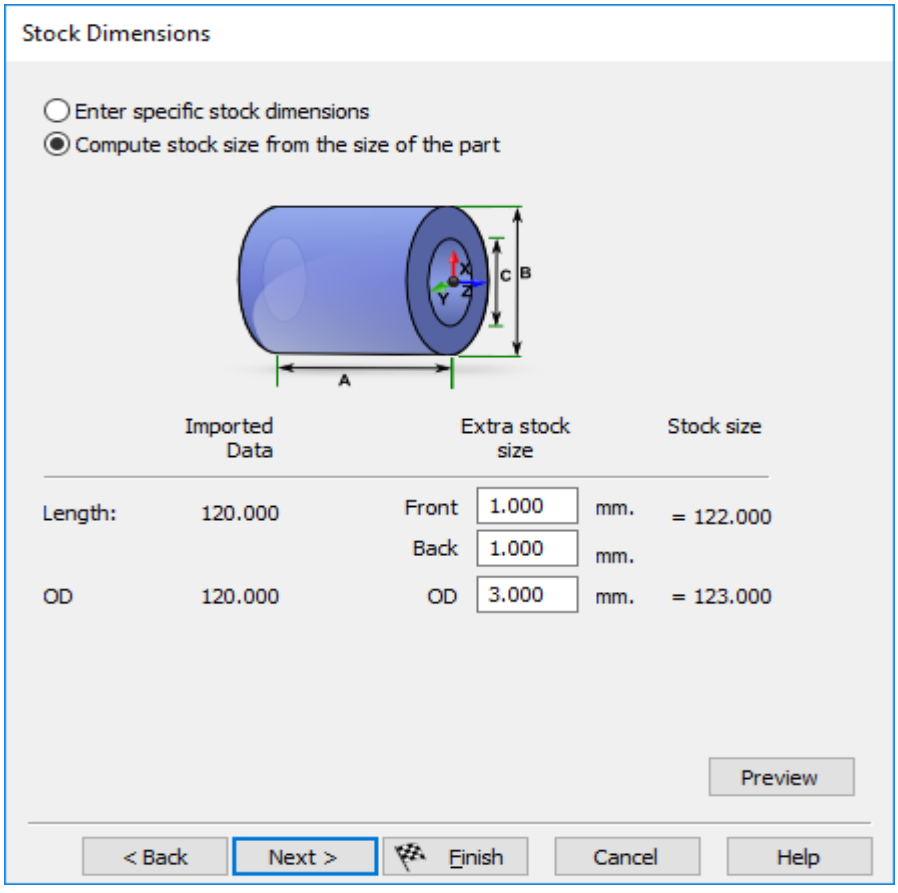


7 Select **Next twice** and then set the **Stock Type** to **Round** (Default)

© Copyright 2016 Autodesk, Inc. All rights reserved



8 Select **Next** twice.



© Copyright 2016 Autodesk, Inc. All rights reserved

9 Select **Next** to set the Stock dimensions.

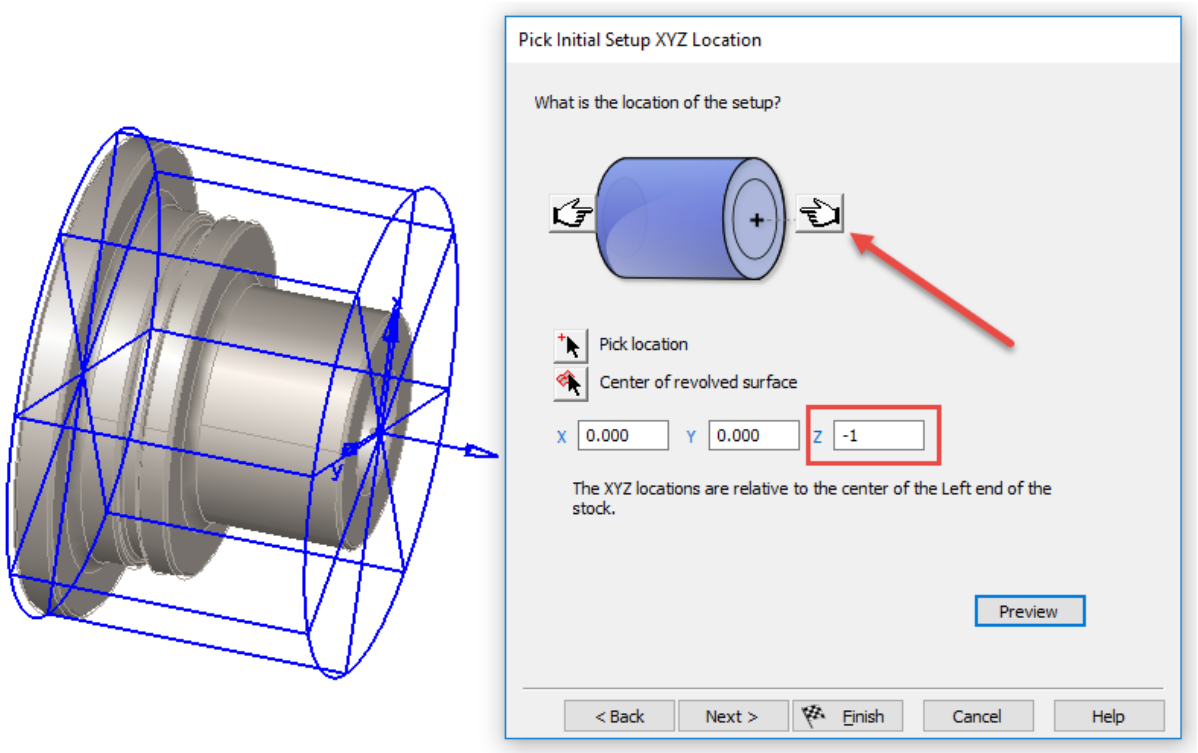
10 Select Stock Size from the size of the part.

11 Enter **1mm** Extra Stock **Front**

12 Enter **1mm** Extra Stock **Back**

13 Enter **3mm** Extra Stock **OD**

14 Select **Next** to align **Setup1** to the front of the part.

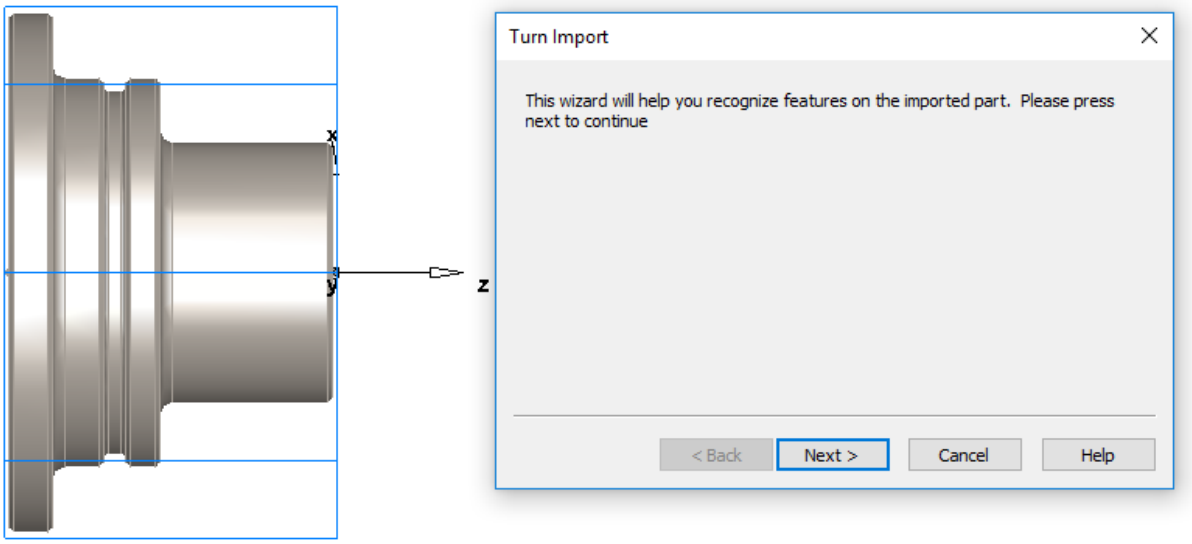


15 Select the **Right Hand** Icon; this will set the value to Zero. **Type** in **-1mm**

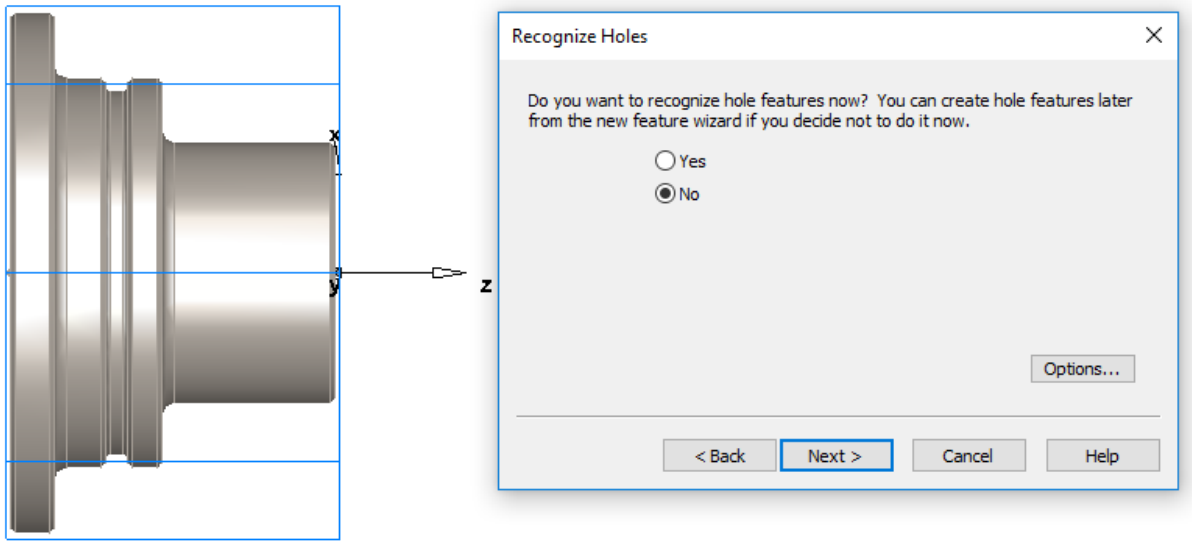
16 Select **Finish**.

17 The **Wizard** will guide you through the Feature extraction process.

© Copyright 2016 Autodesk, Inc. All rights reserved

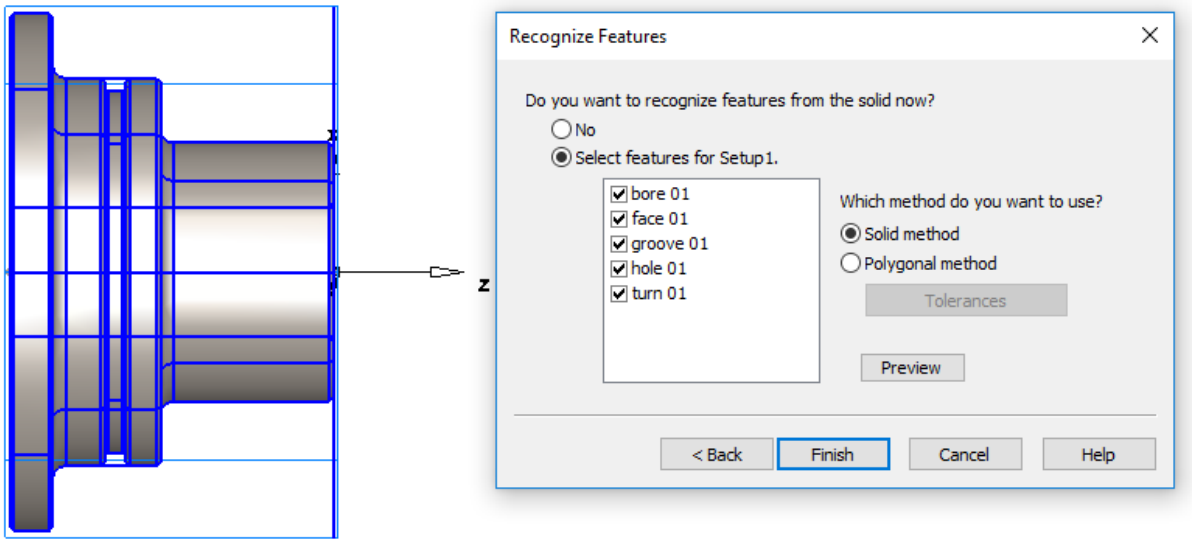


18 Select **Next** to continue.



© Copyright 2016 Autodesk, Inc. All rights reserved

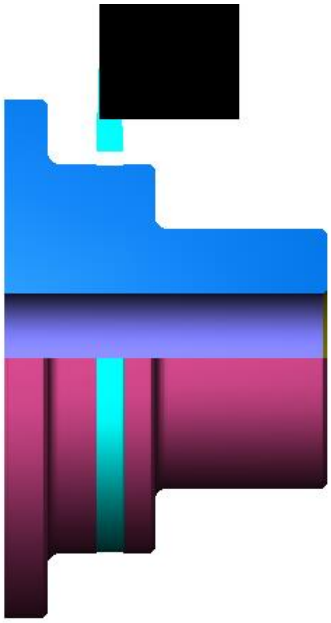
19 Select **Yes** and then **Next**. Accept the features created.



20 Select Finish and Run 3D Simulation.

*If you run the **3D Simulation** we have two problems one the Tool that **FeatureCAM** has picked is too big for the actual groove plus in the process table we have two error messages see below. Which are related to the grooving tool*

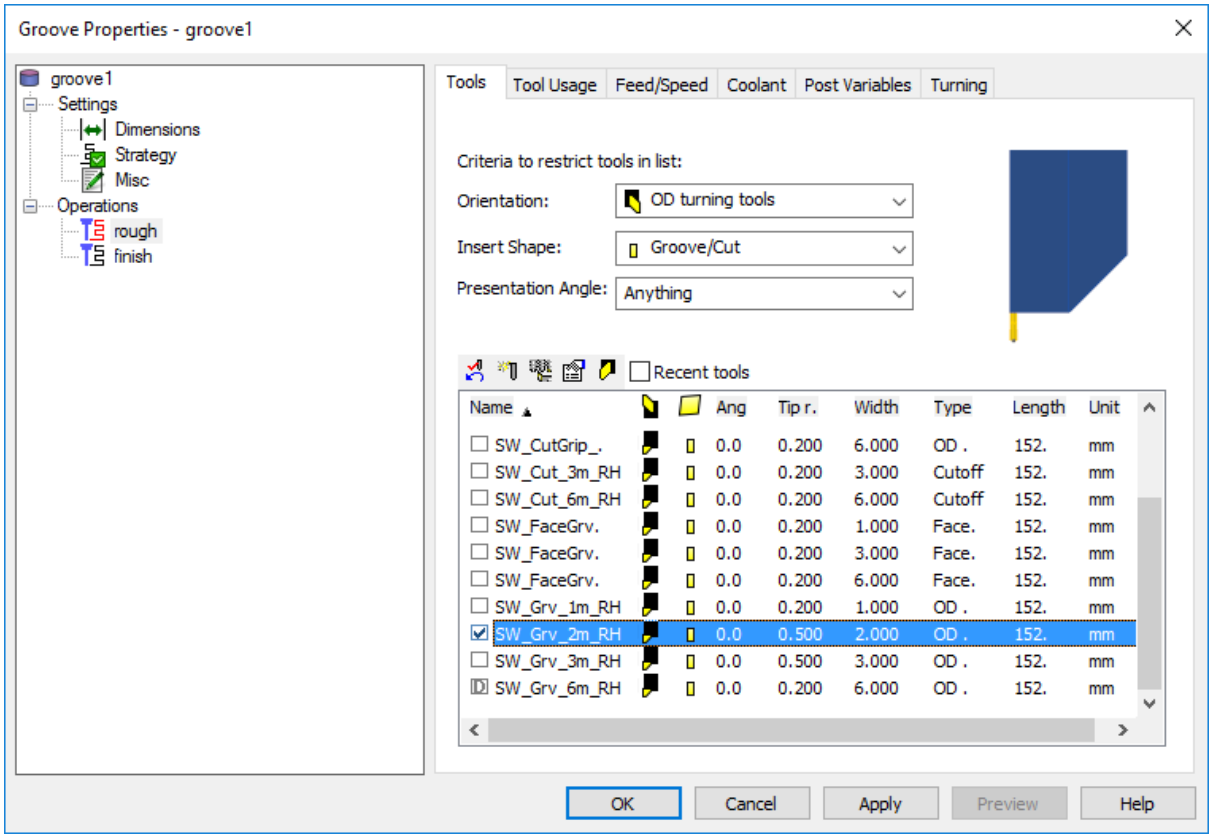
Warning: TPL08W: Undercut detected. Unable to completely finish feature with this tool.



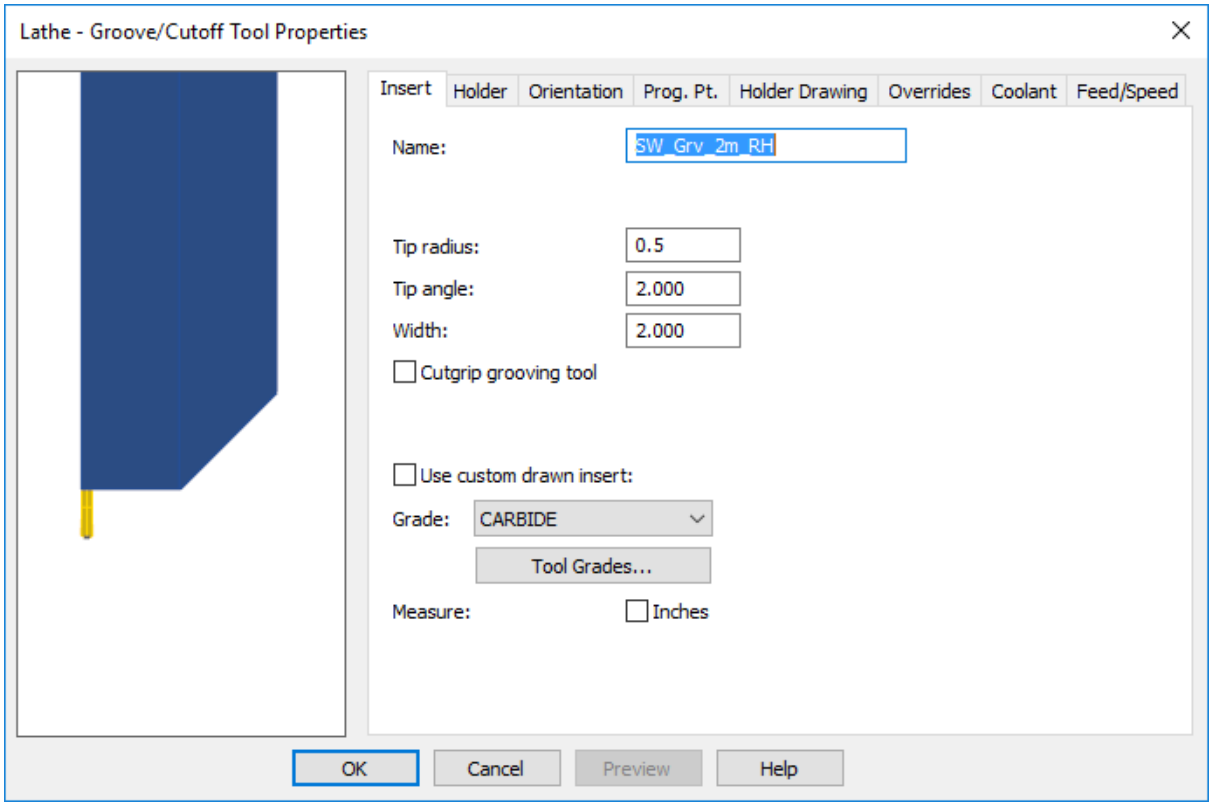
21 To resolve the problems first of all we need to create a grooving tool **2mm R0.5** this will allow the tool to finish the job.

22 **Eject** the **3D Simulation** and double click on the **groove 01** feature. The following menu will appear. Select **Rough**. Select Tools.

© Copyright 2016 Autodesk, Inc. All rights reserved



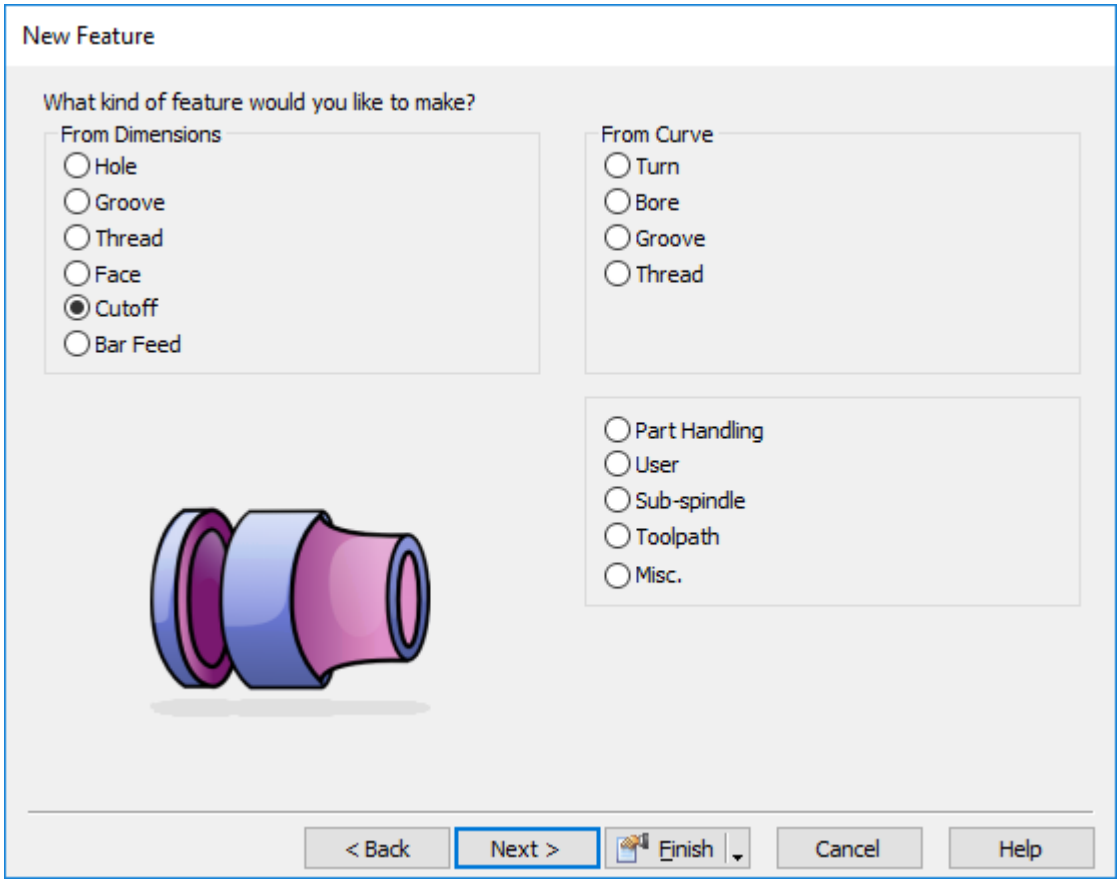
23 Double click on **SW_cut_2** and from this tool create the tool as shown on the next page. Remember to change the name.



24 Change the tool to the one shown and hit apply. This will update the machining process automatically. Double click on the tool that shows the error. See menu. Select the tool listed below. Re-run the **3D Simulation**.

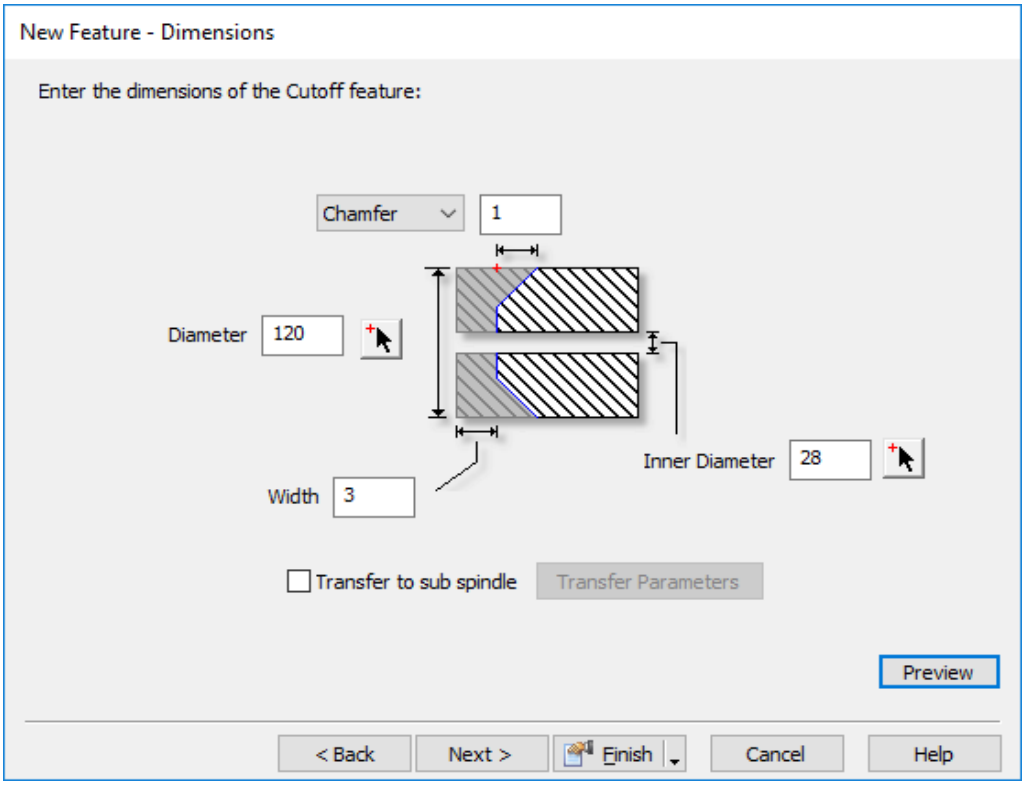
© Copyright 2016 Autodesk, Inc. All rights reserved

25 We now have to create a **Cut-Off** Feature manually to finish the part. Select **New Features** or **(Ctrl+R)** and then select **Cut-off**.



26 Select **Next**.

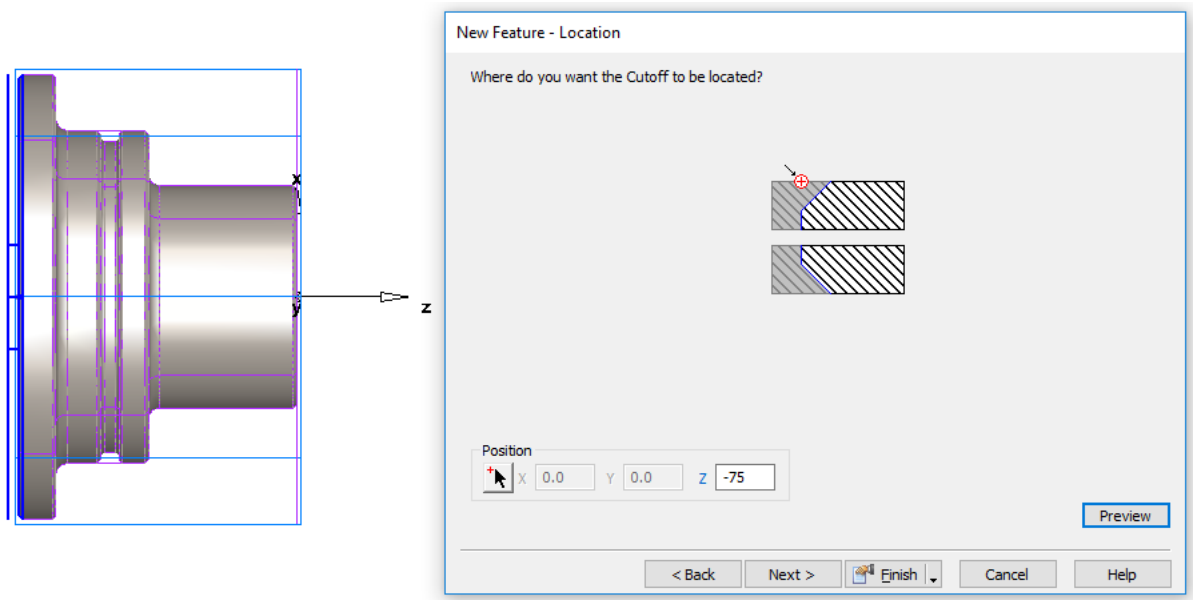
27 Enter the following information into the menu.

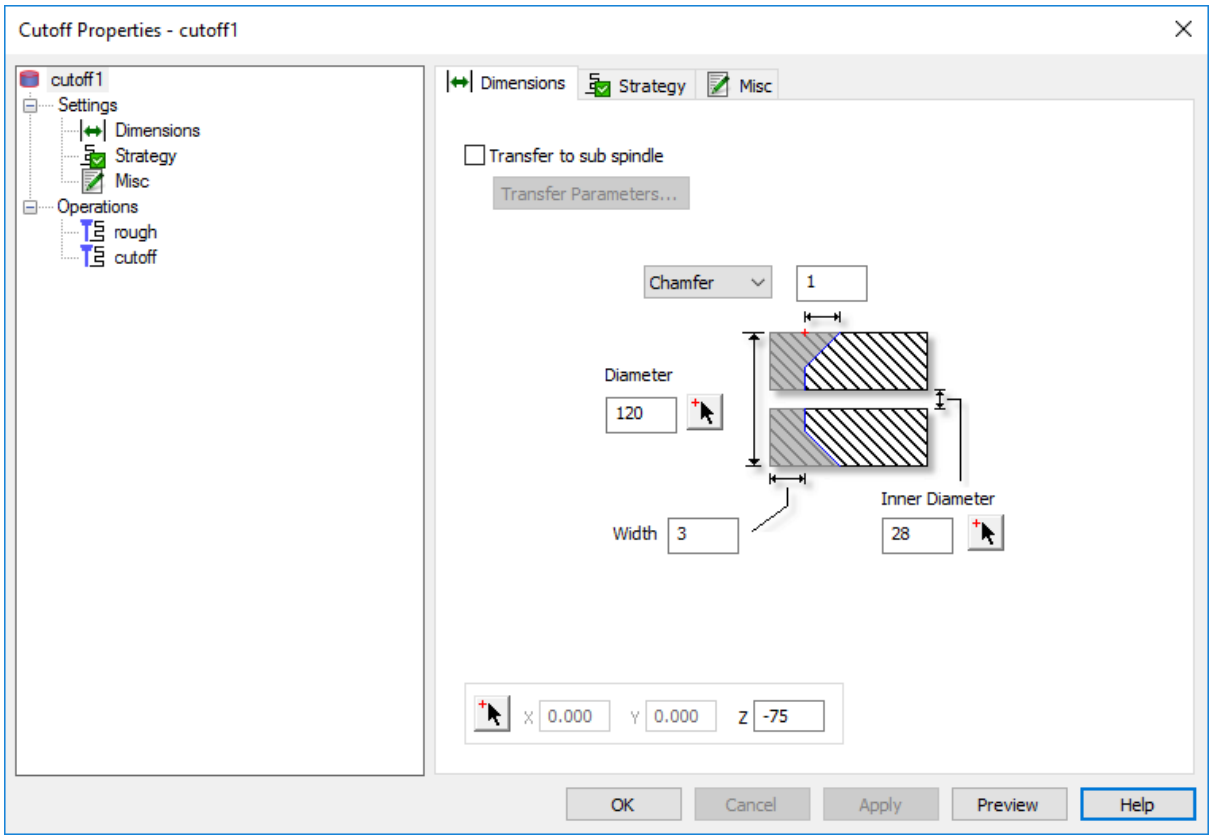


© Copyright 2016 Autodesk, Inc. All rights reserved

28 Select **Next**.

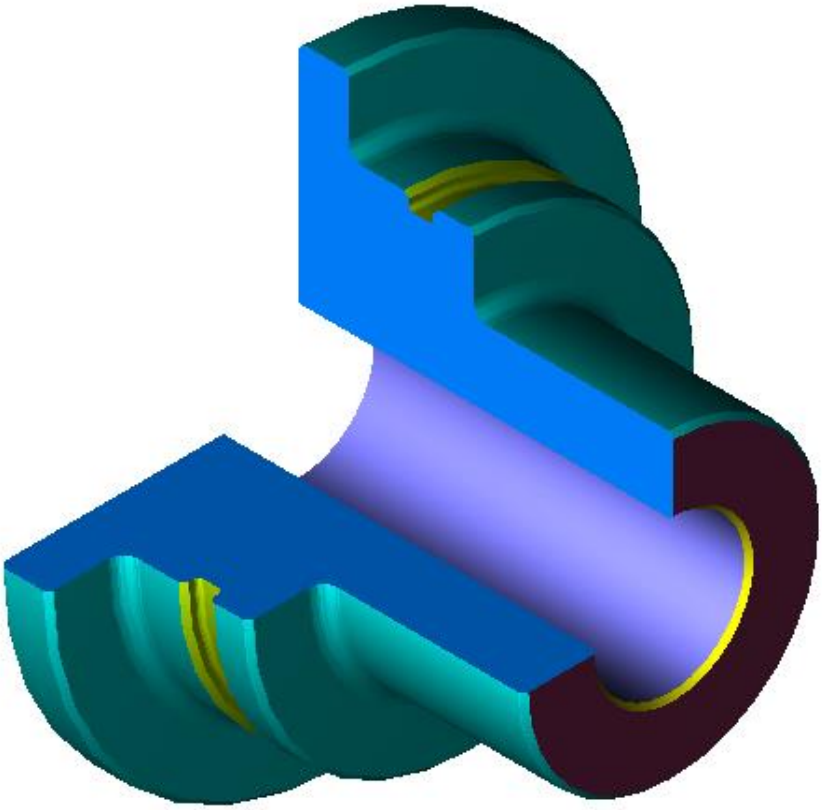
29 The overall length should finish at -**75mm**





© Copyright 2016 Autodesk, Inc. All rights reserved

30 This is the **finished part** shown below.



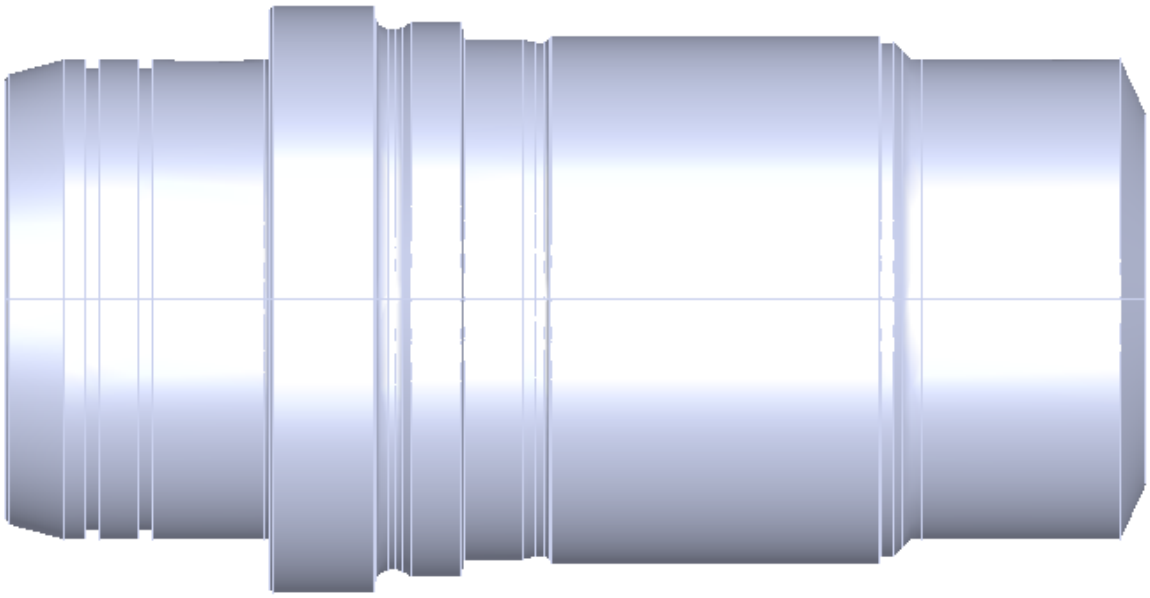
Using a Solid model for stock

The next exercise uses a Solid model to define the stock material. This method minimises the amount of toolpaths required as it only machines where the stock is located.

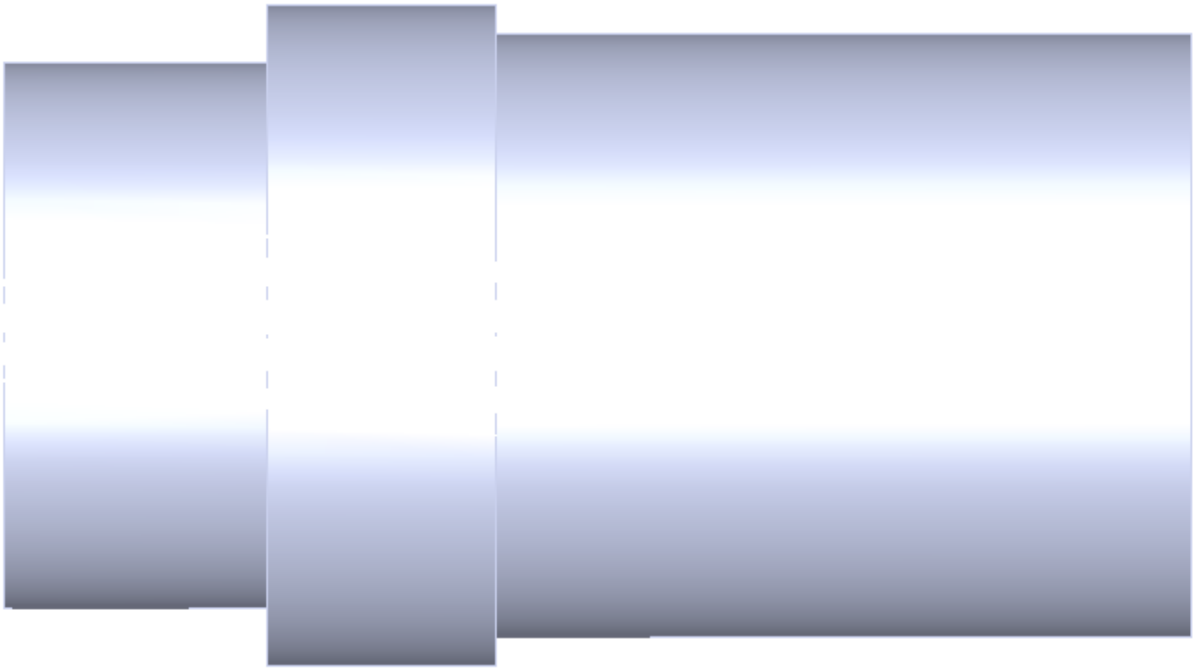
1 Import: - **Solid Stock-2 - All Models .x_t** from.
C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import S**olid Stock-2 - All Models .x_t**

© Copyright 2016 Autodesk, Inc. All rights reserved

*The following model is the **Part***

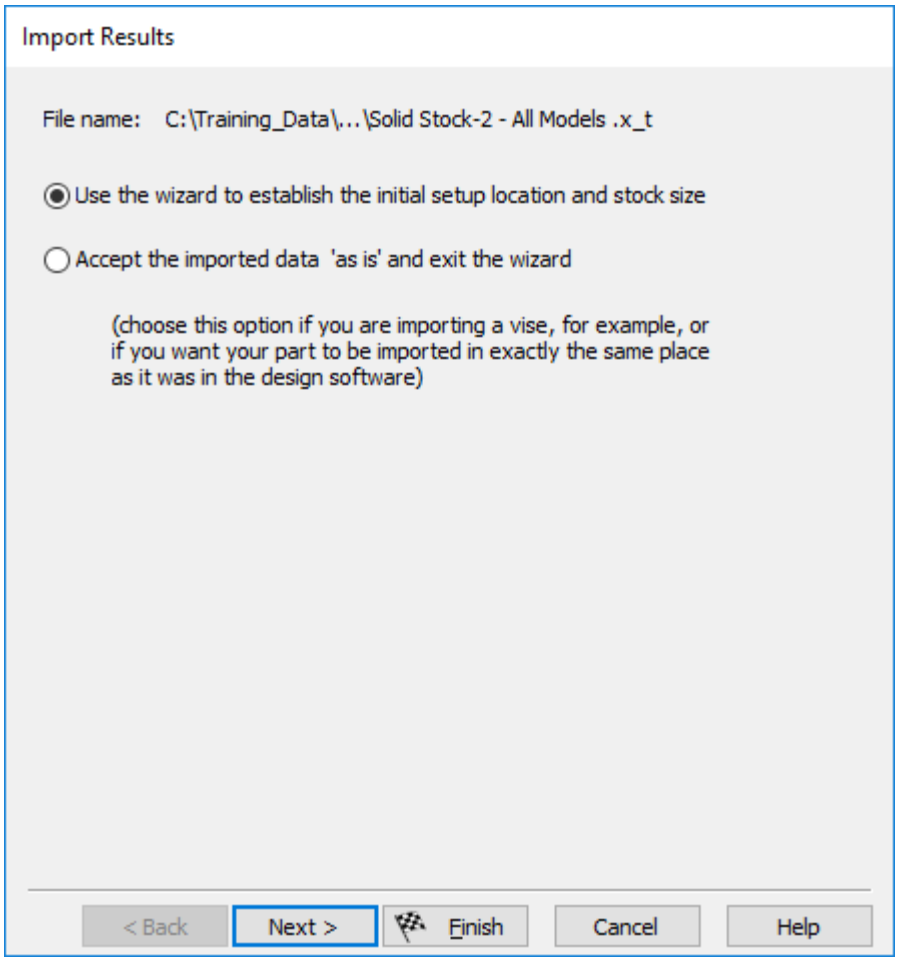


*The following model is the **Stock**.*



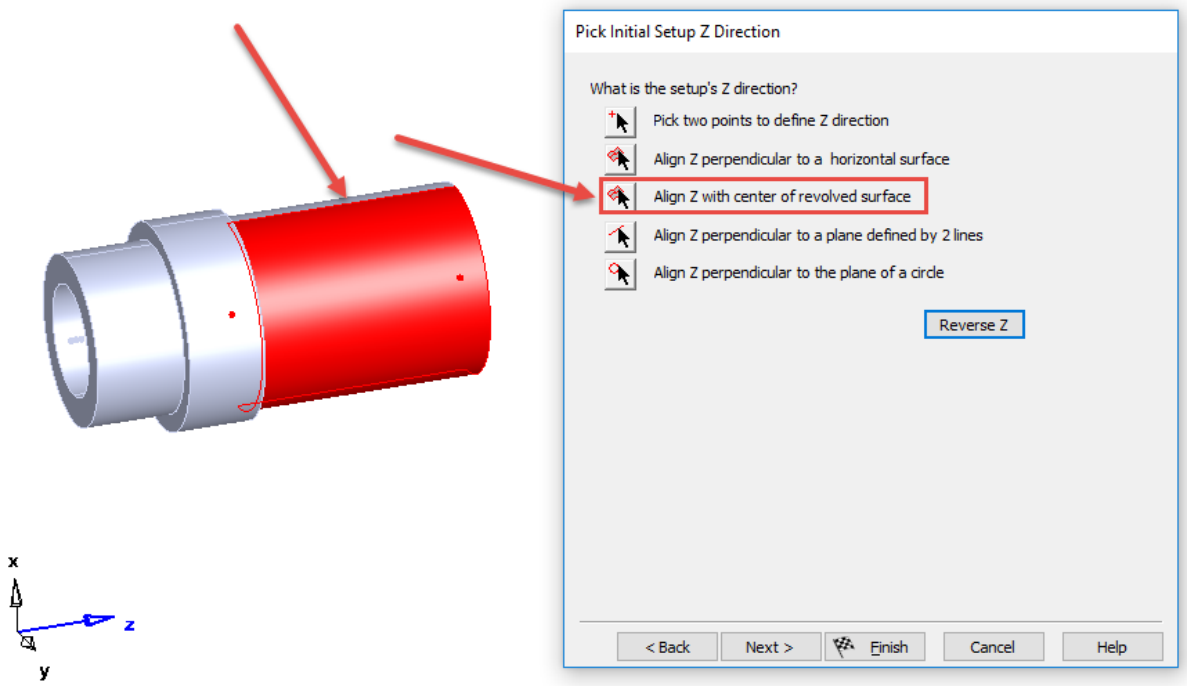
2 Work through the Wizard.

© Copyright 2016 Autodesk, Inc. All rights reserved



3 Select **Next**.

4 Use Align Z with centre of revolved surface.



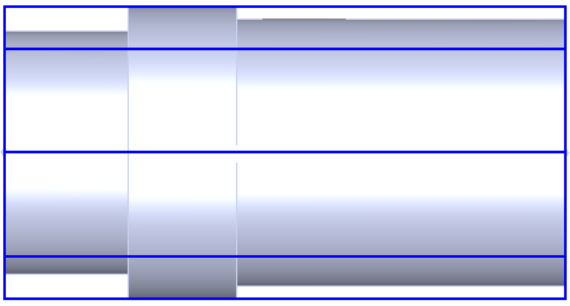
5 Select **Next** Twice.

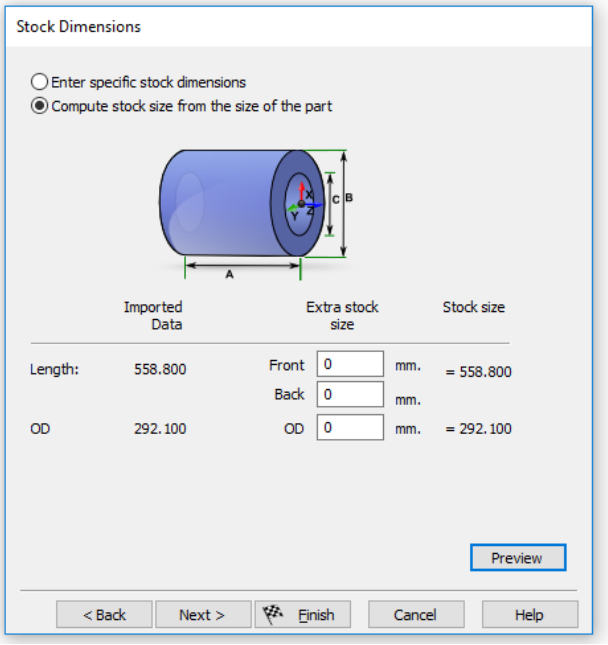
© Copyright 2016 Autodesk, Inc. All rights reserved

6 Stock Type = **Round**. Select **Next**.

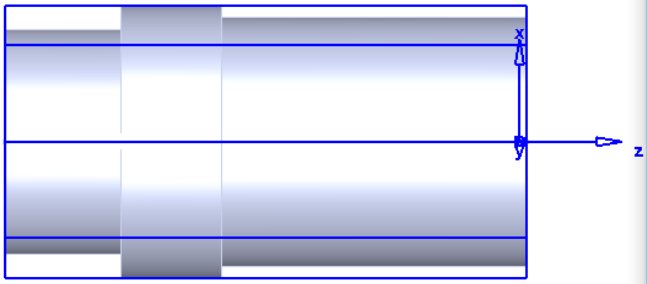
7 Select **Next** Twice.

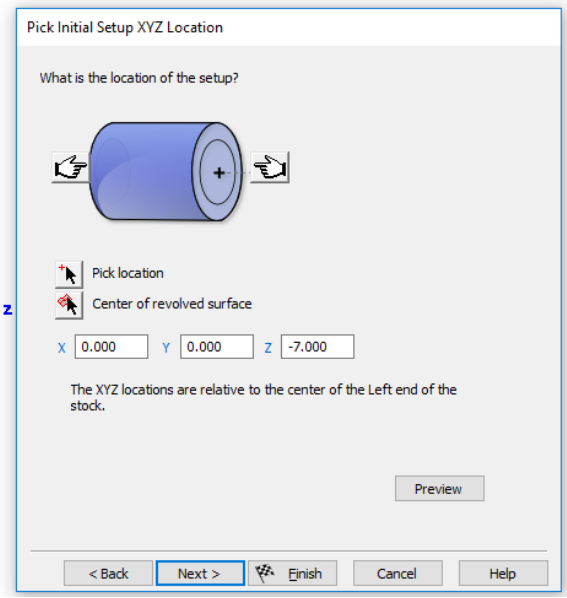
8 Select **Compute Stock size from the size of the part.** Set all data fields to zero.





9 Select **Next**.

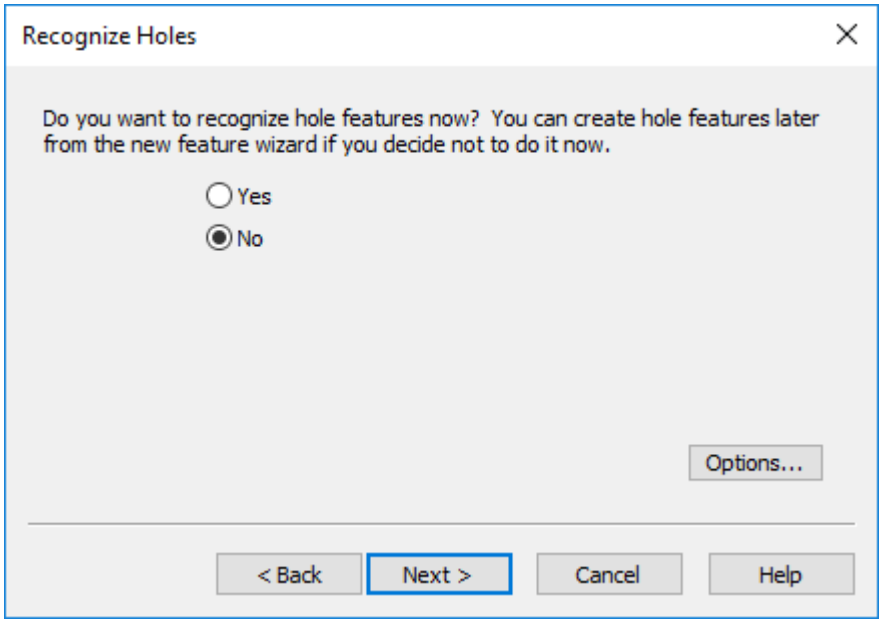




10 The datum will be -7mm. Leave this as it is as it will set to the Part which is inside the Stock solid.

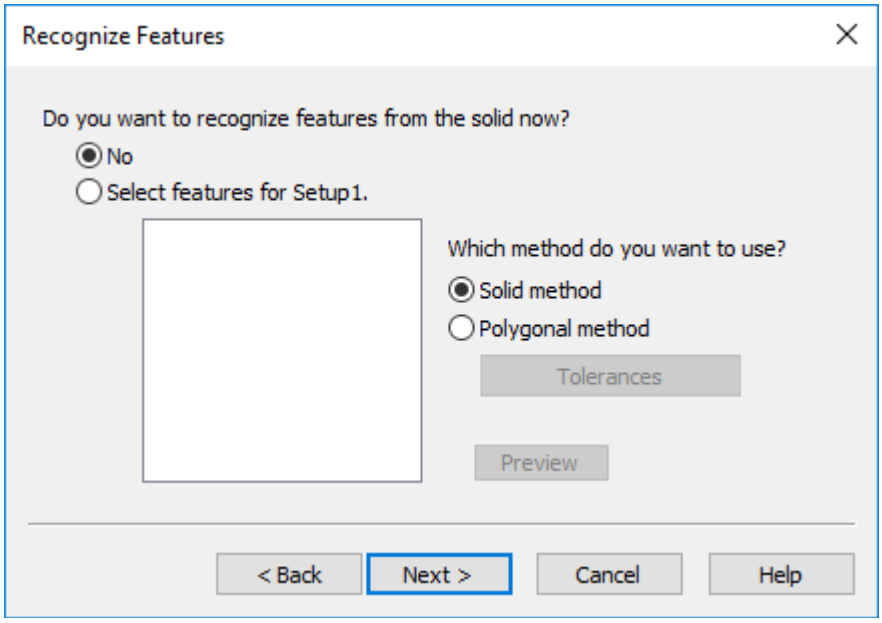
© Copyright 2016 Autodesk, Inc. All rights reserved

11 Select **Finish**. Select **No** to **Hole Features**.



12 Select **Next**.

13 Say **No** to **Automatic Feature recognition**.

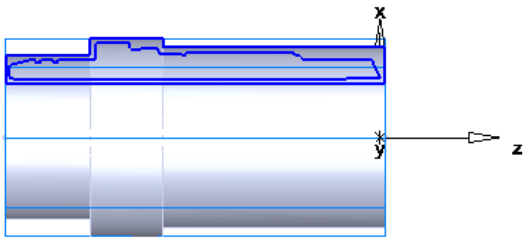


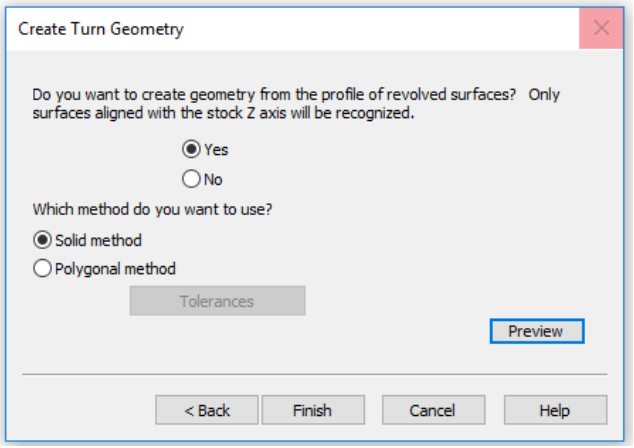
14 Select **Next**.

15 Select **Yes** to extract the turned profile.

Because we have two solid models. Both profiles are extracted.

© Copyright 2016 Autodesk, Inc. All rights reserved



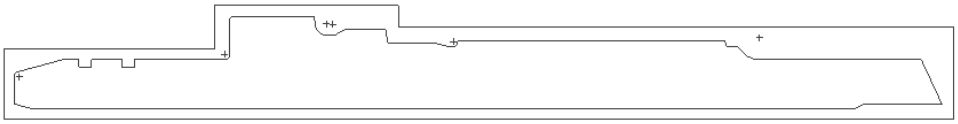


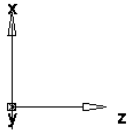
This will extract the 2D Geometry from the two solids we have imported.

16 We can use this Geometry to create Curves to create Stock and machine the Part complete.

17 Select **Finish**.

The geometry shown has been created from the Solid Models.





We have a number of ways to create Stock from the imported information.

18 We can use the larger Solid model and define this solid as stock or

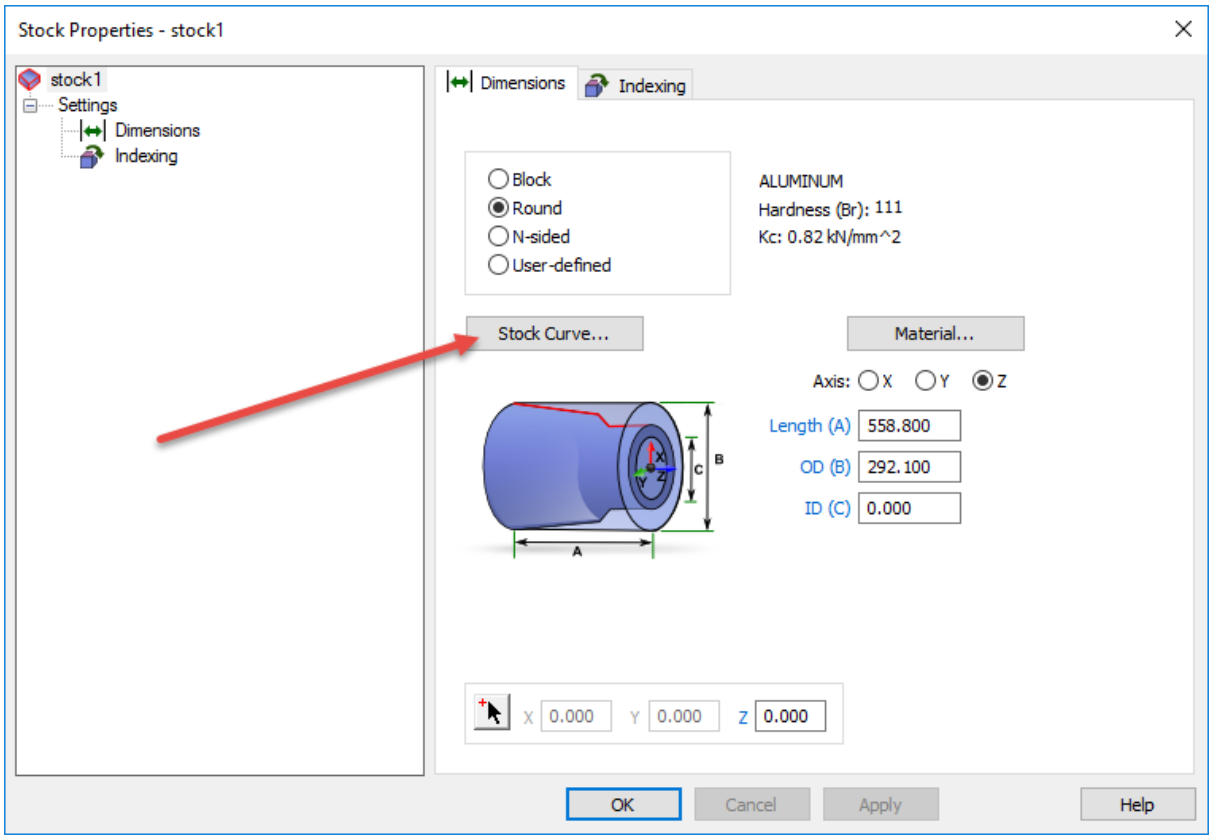
Create a Curve from the outer shape Geometry and define this as a Stock Curve.

19 Remember to **hide** the solid models so you can see the Geometry.

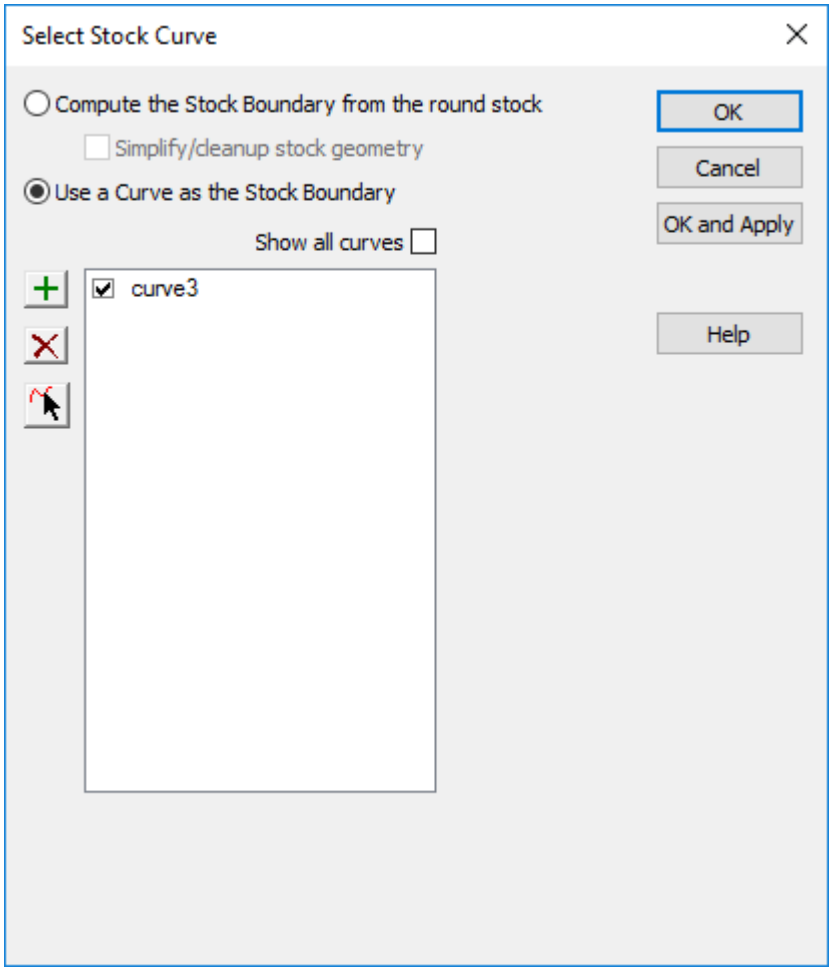
20 In this case my stock Curve is called **Curve5** your curve will be called a different Name.

21 Double click on **Stock1** in **Part View.**

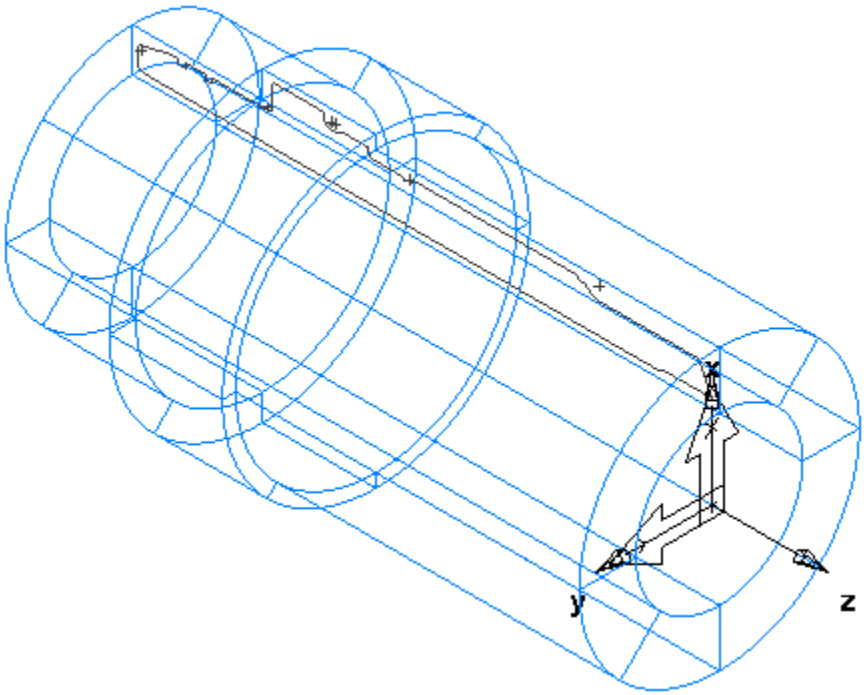
© Copyright 2016 Autodesk, Inc. All rights reserved



22 Select **Stock Curve.**



© Copyright 2016 Autodesk, Inc. All rights reserved



23 Or alternatively use the User Defined button and select the **Stock Solid**.

24 From the Geometry you have extracted have a go at machining the Part.

25 Once completed have a go at **Solid Stock-3 - All Models .x_t**

Feature Recognise additional models

Location of files ***C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import***

1036_Autodesk-A_01.x_t

1036_Autodesk-B_01.x_t

1036_Autodesk-C_01.x_t

1294_Autodesk-D_01.x_t

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

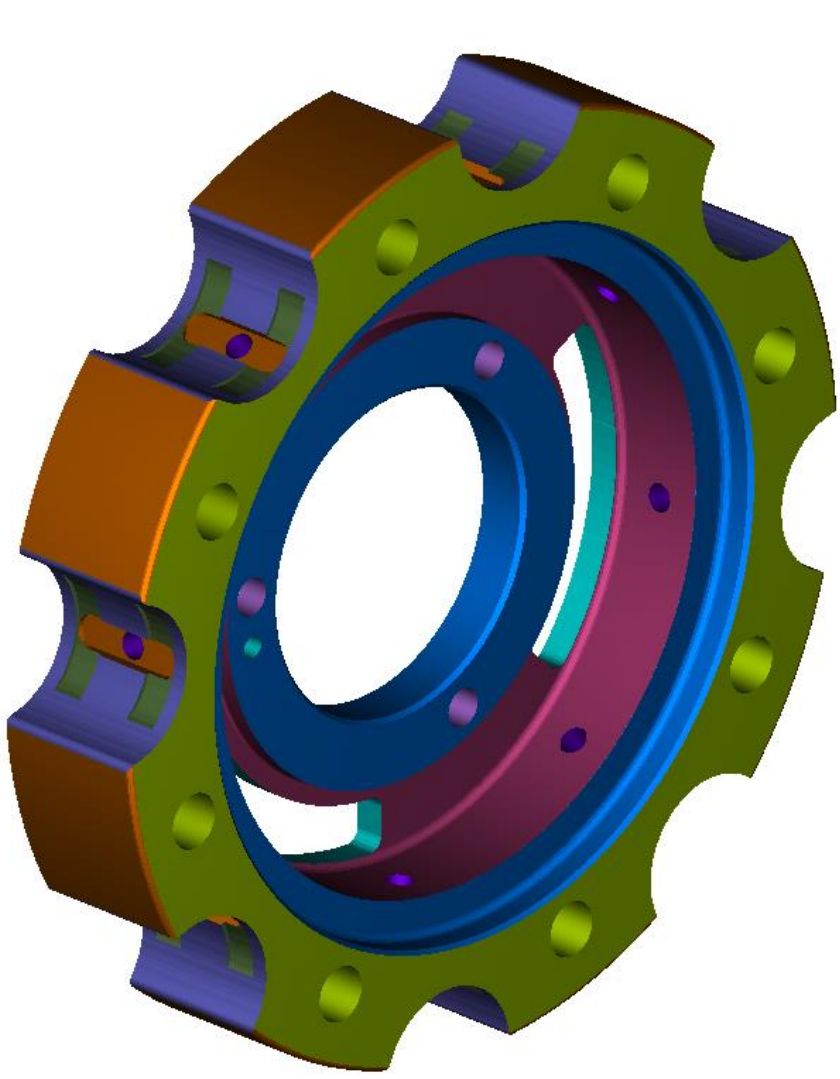
Turn-Mill Features X Z C &Y Axis 2
Introduction 2
Requirements for Turn milling 2
Advantages & Disadvantages of Y axis verses C axis 3
Simulation in Turn/Mill 4
Tool selection for Turn-Mill Features 4
Features appropriate for Turn-Mill 4
C & Y Axis Machining 4
Creating Turn-Mill features in FeatureCAM 4
Viewing Options 5
Stock Axis in Turning 6
Creating Turn/Mill Features from a 2D Drawing 6
AFR First Turn/Mill Exercise 23
FC XZCY Axis Part 33
C Axis IFR (Interactive Feature Recognition) 54
Vortex Machining 69
Vortex Introduction 69
Pocketing properties 70
Flange Body Test Piece. 71
Endcap Body 71
Advanced Pin Machining 72
Additional Exercise 72
1635_Autodesk-Z.x_t 73
Turn milling Example (Information Only) 73

© Copyright 2016 Autodesk, Inc. All rights reserved

Turn-Mill Features X Z C &Y Axis

Introduction

Turn milling is a machining strategy in which you machine a turned feature using a milling tool. Both the Spindle of the lathe and the milling tool are spinning at the same time. This allows for a much faster rate of material removal. Another big advantage of turn milling is that it breaks the chips preventing chips from building up around the head of the cutter which is common with traditional turning methods.



Requirements for Turn milling

In order to use Turn milling your machine has to be capable of spinning both the live tool and the turning spindle at the same time. You should also have a Y axis to use this functionality as it is recommended that you offset the tool from centre so that you are not cutting on centreline when cutting on the diameter of the part. You can also use this strategy for face features. In this case it is recommended to cut on the centre of the part.

© Copyright 2016 Autodesk, Inc. All rights reserved

Advantages & Disadvantages of Y axis verses C axis.

Although we support both methods of machining, there are cases where the person using the software would favour 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 centreline 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 centre 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 **40mm** to **50mm** in each Y direction, measured at X0. The furthest the tool travels in X, the less the Y-axis travels.
- In the screen shots in this chapter you will see how after selecting Turn-Mill the software will allow the user to create milling features exactly like you would be using our basic 2.5D module.

FeatureCAM Turn-Mill *allows the combination of turning and milling features on lathes with powered rotary tools.* ***FeatureCAM Turn-Mill*** *supports the normal Z and X axes of turning combined with the C and optional Y-axis. All Turn-Mill machining centres will support C axis (which has 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.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Simulation in Turn/Mill

- All forms of simulation **(Centreline, 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 on the machine.

Tool selection for Turn-Mill Features

- Turn/mill features use the same tools as the normal milling features, but they are renamed with – Rotary X or – Rotary Z 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 turn mill centre drill operation, the tool is copied and the copy is called center4 – rotary X.
- Rotary tools cannot be explicitly created, but if you manually select a tool for a turn mill operation, it is copied and the copy is designated as a rotary tool. Rotary tools cannot be used for turning operations.

Features appropriate for Turn-Mill

- Milling features can be performed in Turn-Mill with the following considerations:
- Milled features on the Z face: **FeatureCAM Turn-Mill** 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.

C & Y Axis Machining

- 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 features. 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

Creating Turn-Mill features in FeatureCAM

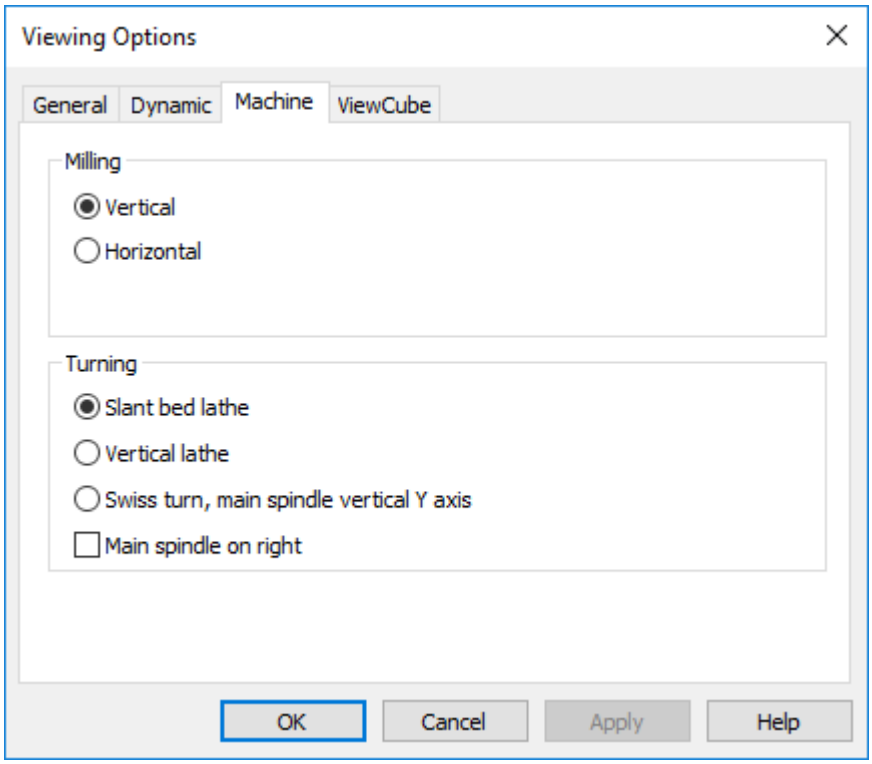
- Turn-Mill features are classified as either **Turn-Mill** or pure **Turning features**. Turn-Mill features assume powered tools and turning features assume that the tool does not spin. For example, you can make a hole along the Z-axis with both feature types, but the milled one uses rotary tools and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

© Copyright 2016 Autodesk, Inc. All rights reserved

- Turning features are identical to those on a **2-axis lathe**. Milling features are created the same as for a **3-axis milling**, except that you are given new choices for positioning and orienting the features either on the **Outside Diameter** or on the **face** of the part in the Z Axis.

Viewing Options

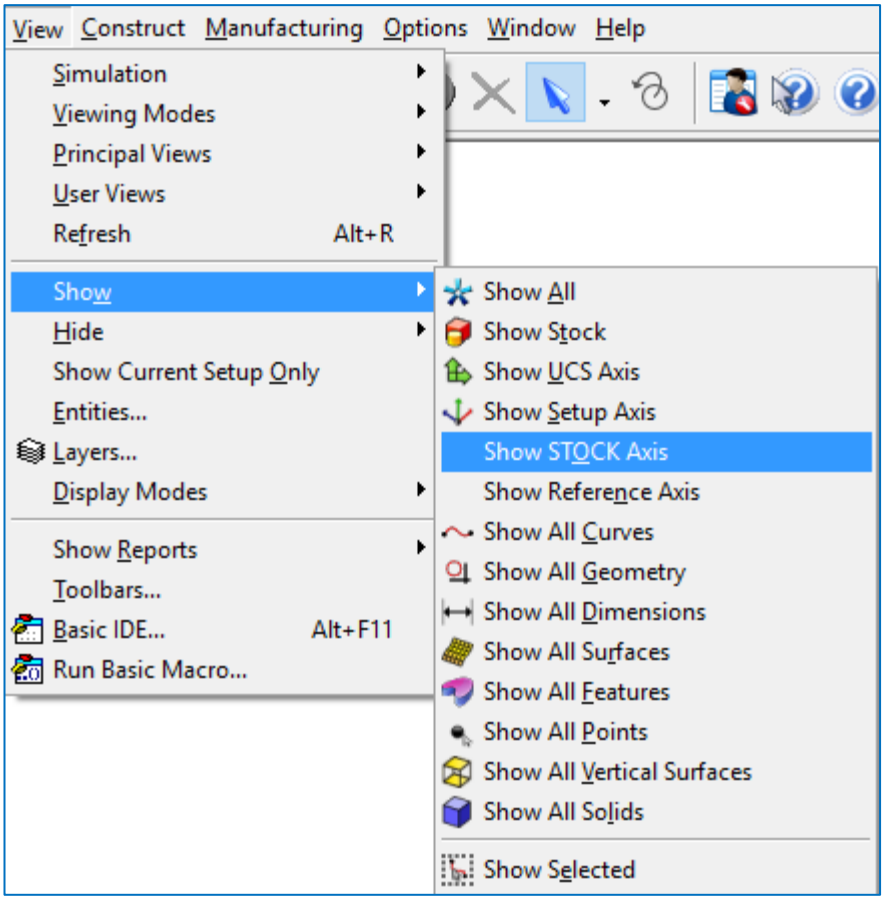
We have the following options for machine orientation. **(Slant bed lathe)**



© Copyright 2016 Autodesk, Inc. All rights reserved

Stock Axis in Turning

- The stock axis shows the orientation of **FeatureCAM** Turn-Mill, this will be the basis for all indexing co-ordinates. See image on the next page.
- *To show the* **STOCK Axis** *select* **View>Show>Show STOCK Axis**.

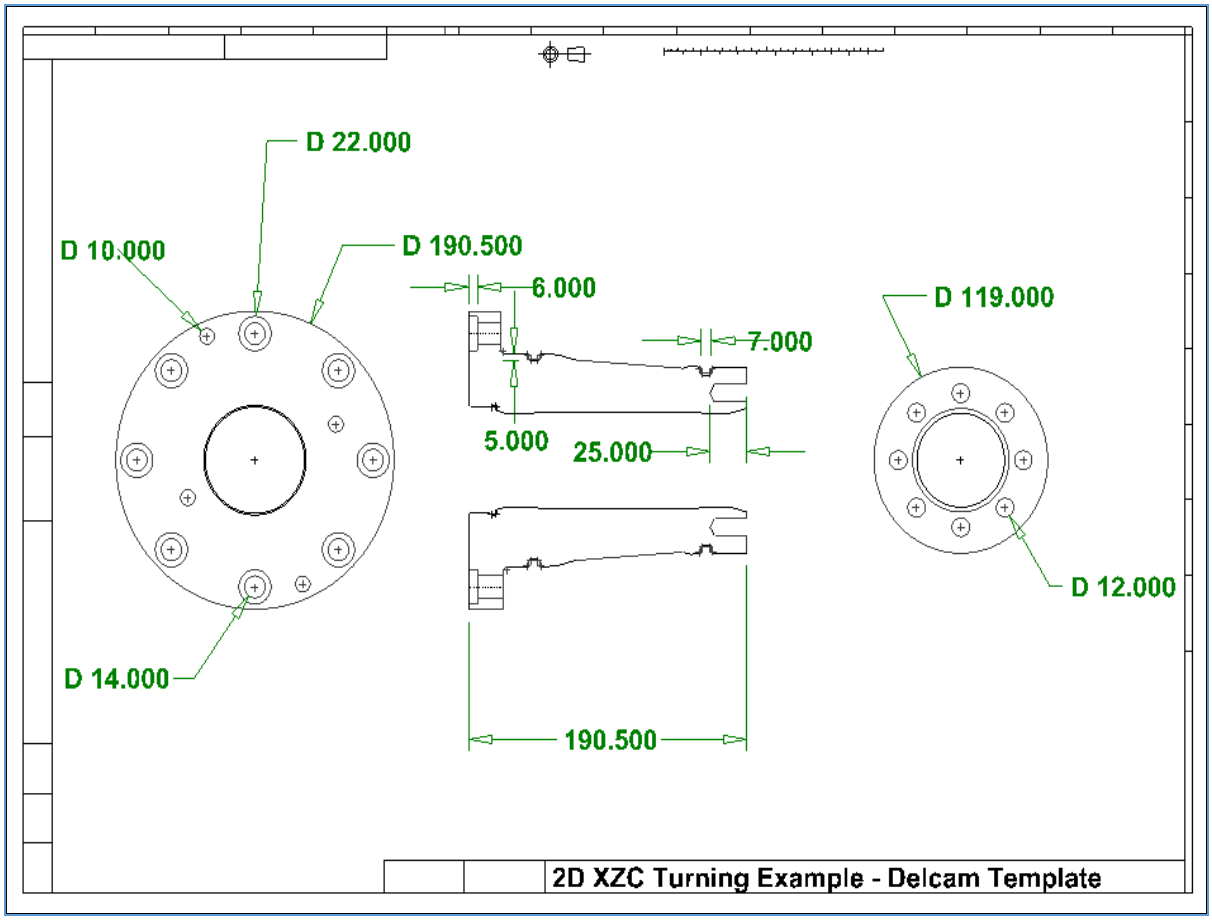


Creating Turn/Mill Features from a 2D Drawing

*Companies still use **2D .dxf** and **.dwg** drawings. The following exercise will demonstrate how to create **C Axis** machining from a **2D drawing**.*

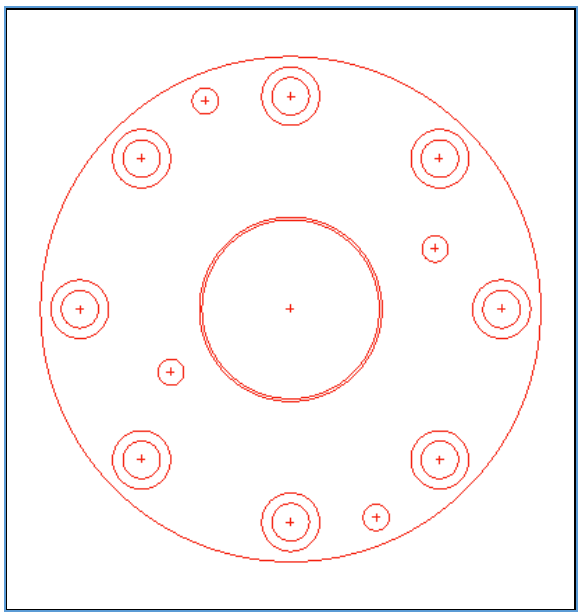
1. Open the file **2D XZC Turning ExampleMetric.fm** from the folder. **C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm files**
2. Load the **Mori Seiki NZ 2000 T2 Training Only.cnc** Post Processor from the same folder.

© Copyright 2016 Autodesk, Inc. All rights reserved



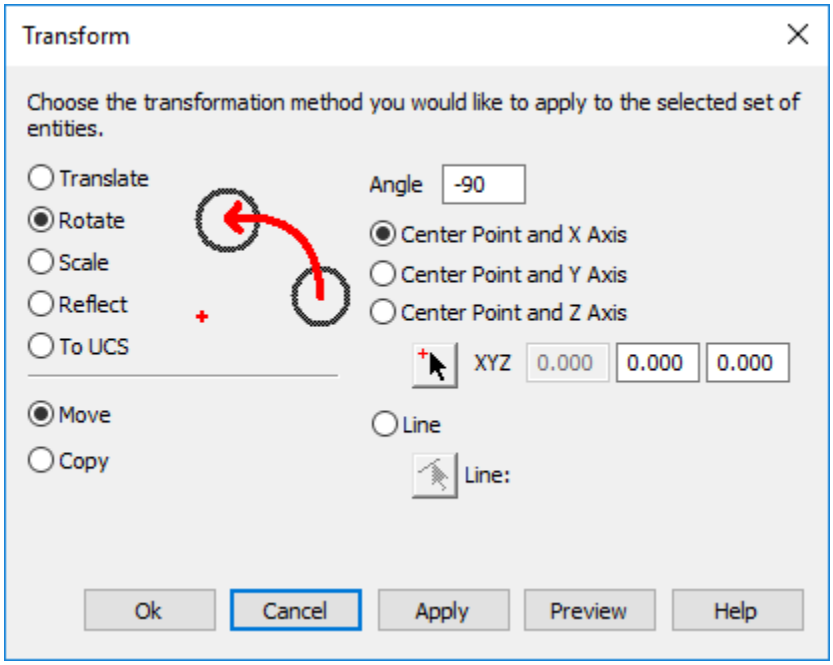
3 **Hide** all dimensions.

4 We are now going to rotate the selected geometry. Select the left hand circular image making sure all holes are selected.



5 Select **Edit**>**Transform** and enter the following values.

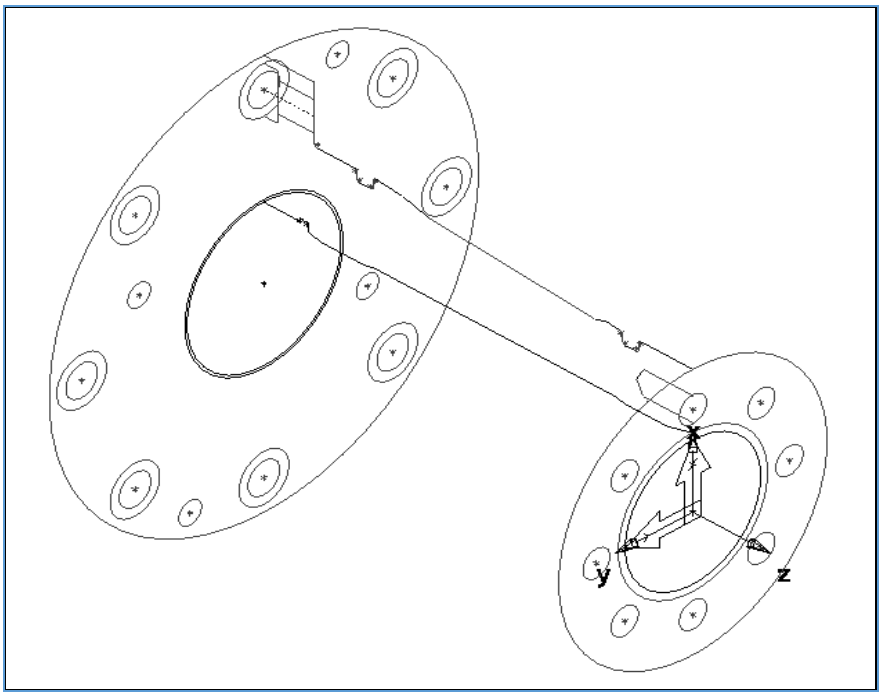
© Copyright 2016 Autodesk, Inc. All rights reserved



This will rotate the part so we can move it to the correct position. Whilst the previous geometry is still selected, reselect Transform; rotate the image on the screen so you will be able to snap to the centre of the circle.

*Select **Translate** and **Move** using **From** and then **To**. **From** is where the current geometry is located. **To** is where you want the geometry to be located. In our case the new position is **X0, Y0, Z-190.5**. The image below shows the selected geometry in position. Follow the same procedure so all geometry is in the correct position.*

Make sure the stock has material all around the part.

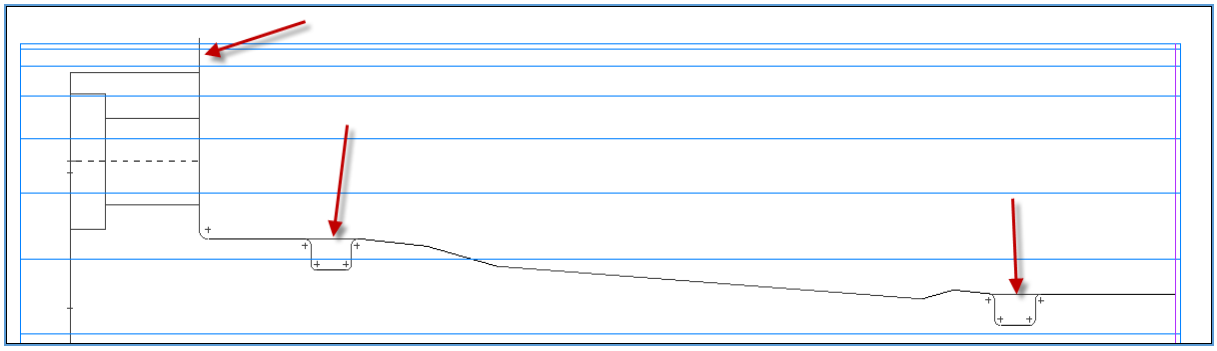


© Copyright 2016 Autodesk, Inc. All rights reserved

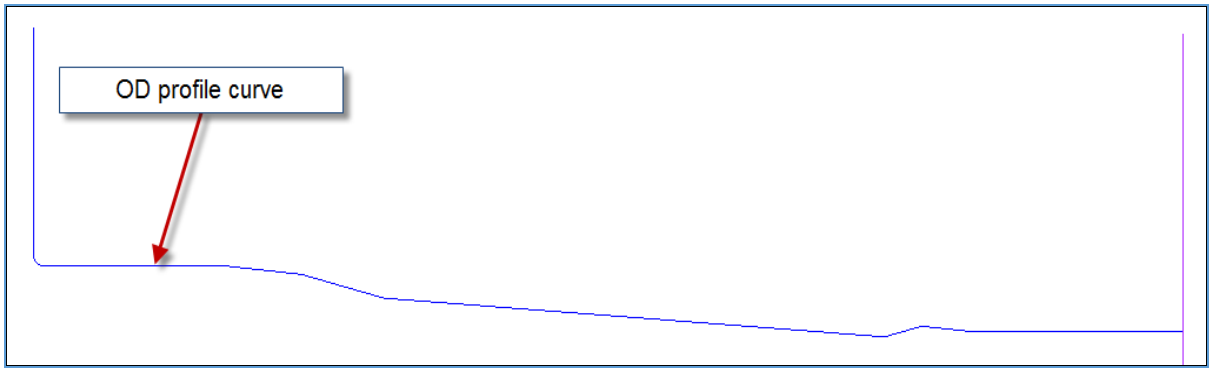
*We are now ready to start machining. Set the **Base Priority** for each Feature as you work through the part.*

6 Select **Ctrl + R New Feature** and create a **Face** Feature.

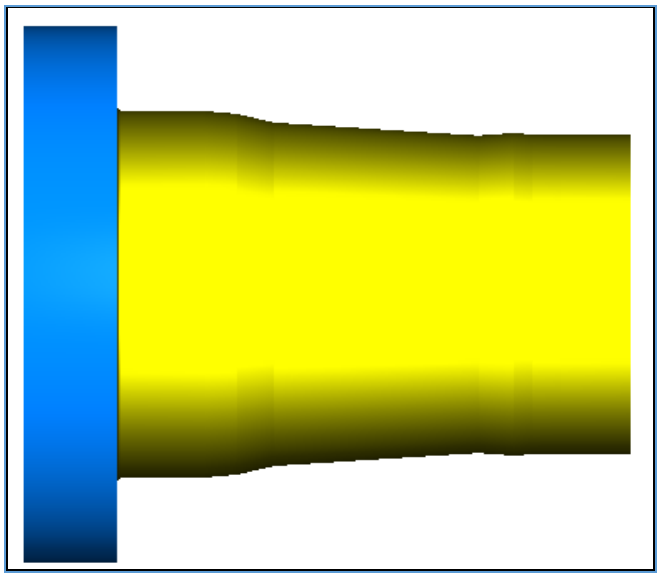
*Before we start turning our Outside diameter we will have to create geometry that will bridge over the two grooves. We can then create a curve for our **OD profile**. You should be able to do this by now. Then extend the geometry for the shoulder profile as shown below.*



7 Create your Curve as shown.



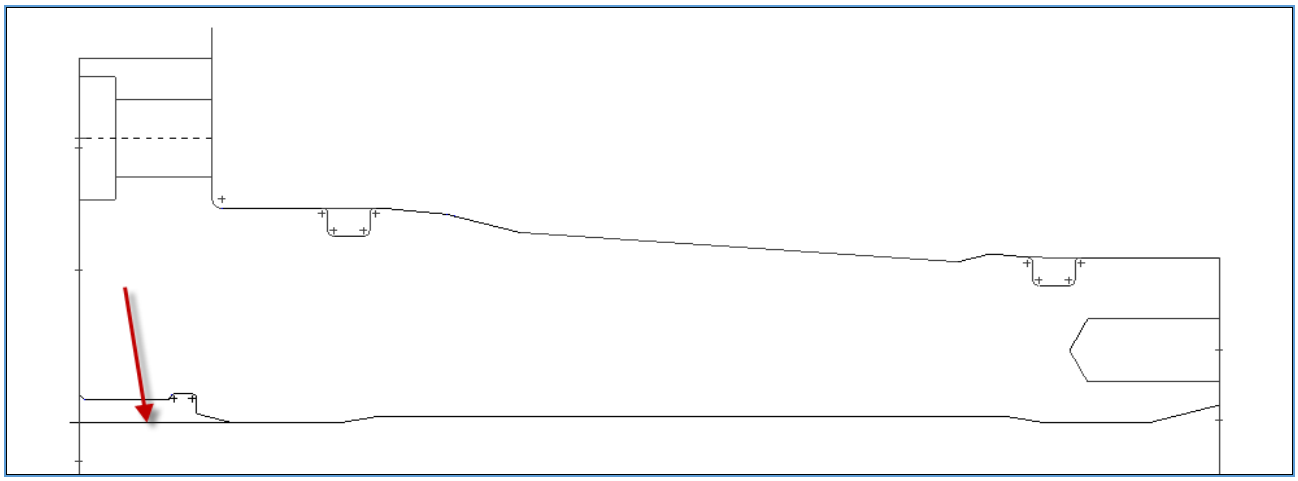
8 Create a **Turn** feature from curve.



© Copyright 2016 Autodesk, Inc. All rights reserved

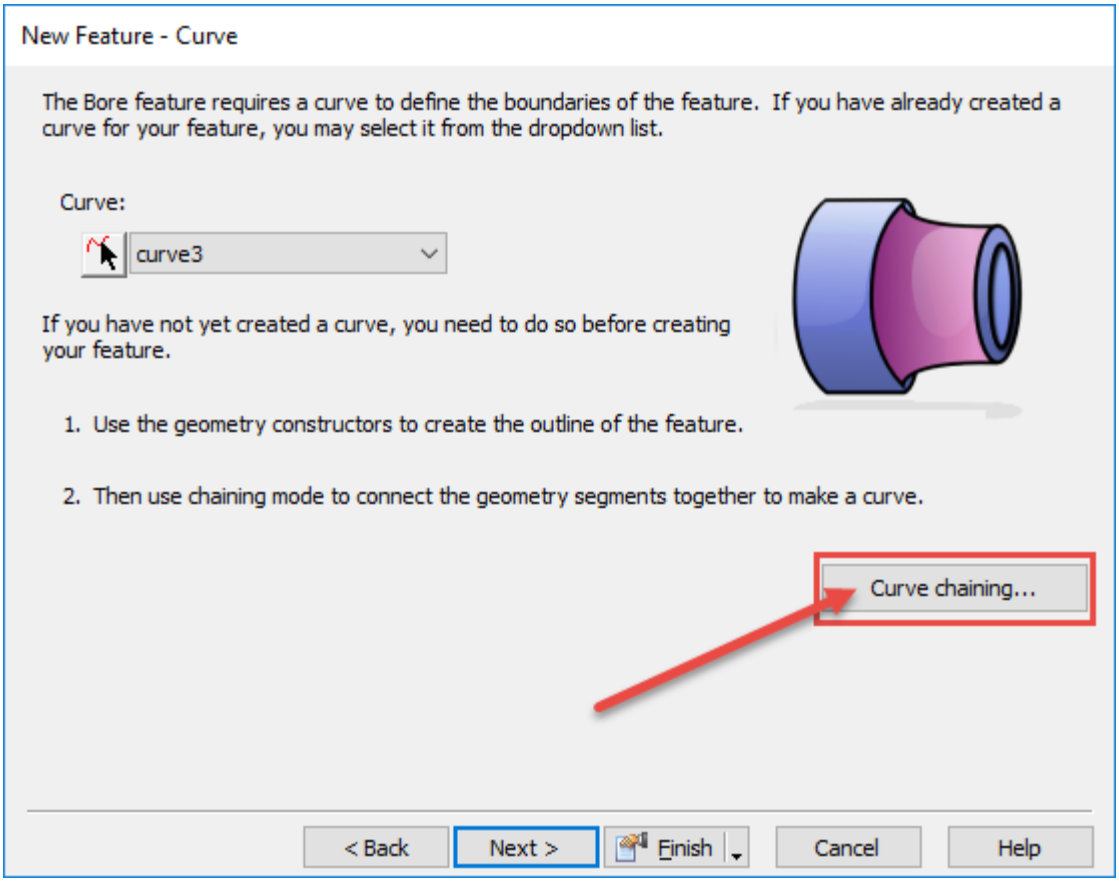
9 **Eject** the **3D Simulation**. We will now create a **Bore** feature using a **52mm U-Drill** (Drilling on centre). This combines two strategies into one.

Before we create our curve please extend the geometry as shown.



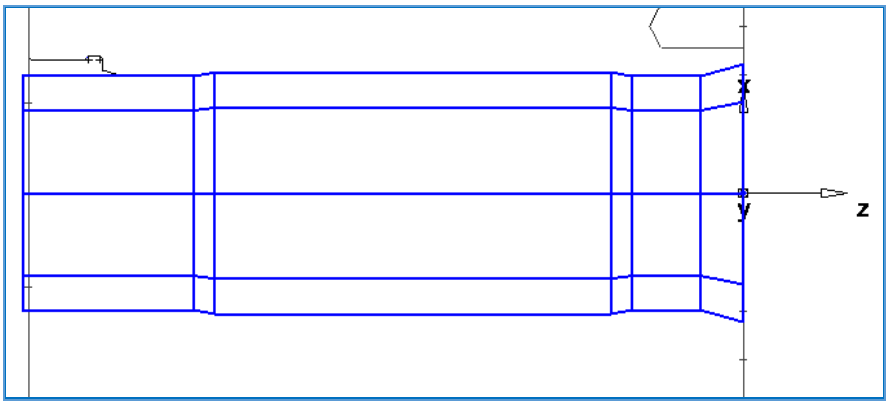
10 Select **Features** then Select **Turning**.

11 Select **Bore** select **Next** and then select **Curve chaining**.



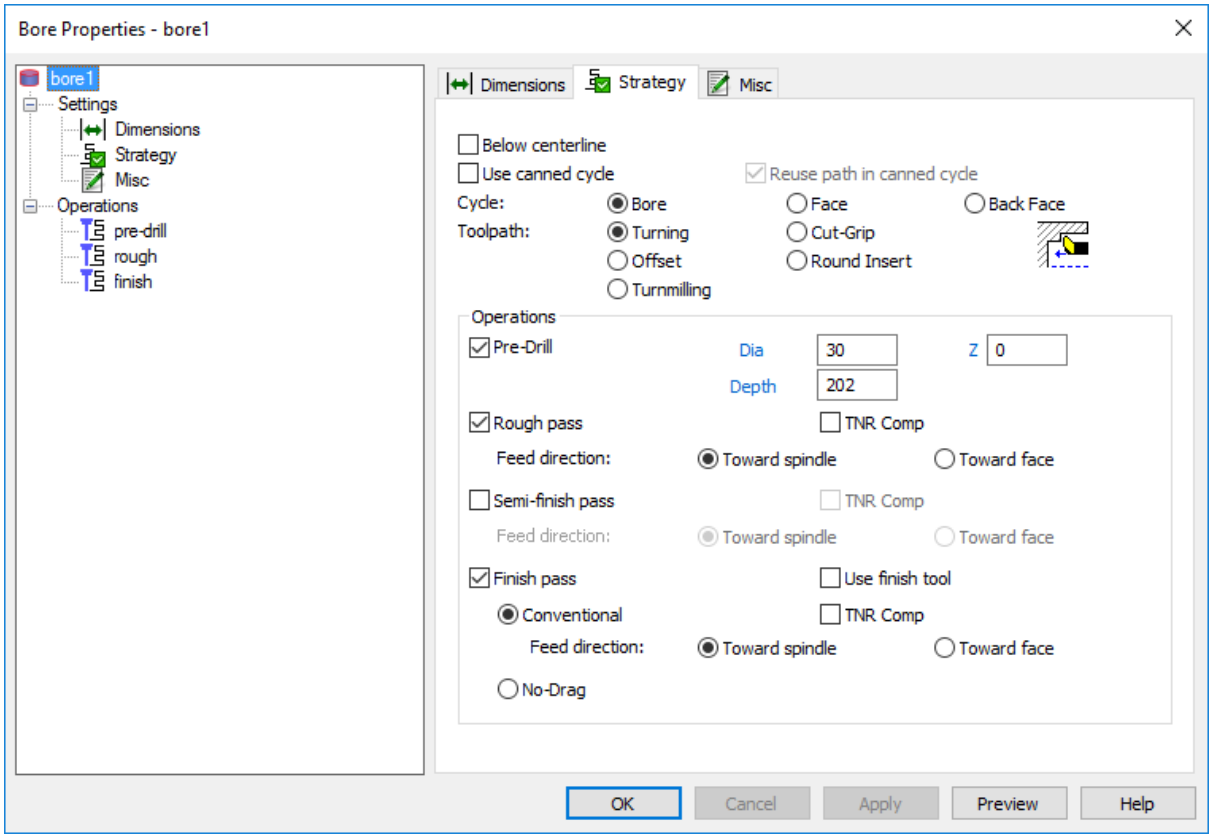
12 Create a **Curve** as shown and then select **Next** twice.

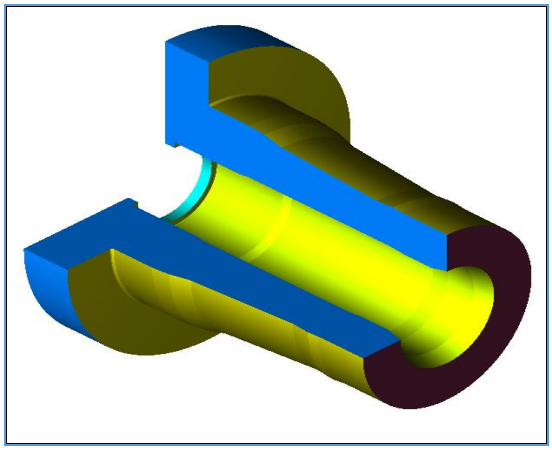
© Copyright 2016 Autodesk, Inc. All rights reserved



13 Select the **Pre-Drill** option and enter **30mm** for the **Diameter** and **Depth** = **202mm**

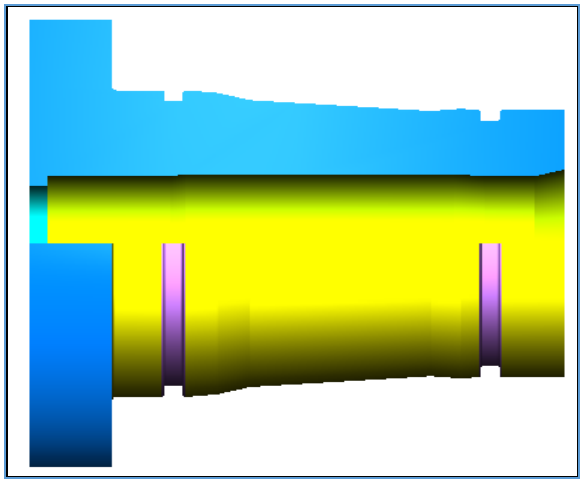
14 The image below shows the **Bore properties Strategy page**.



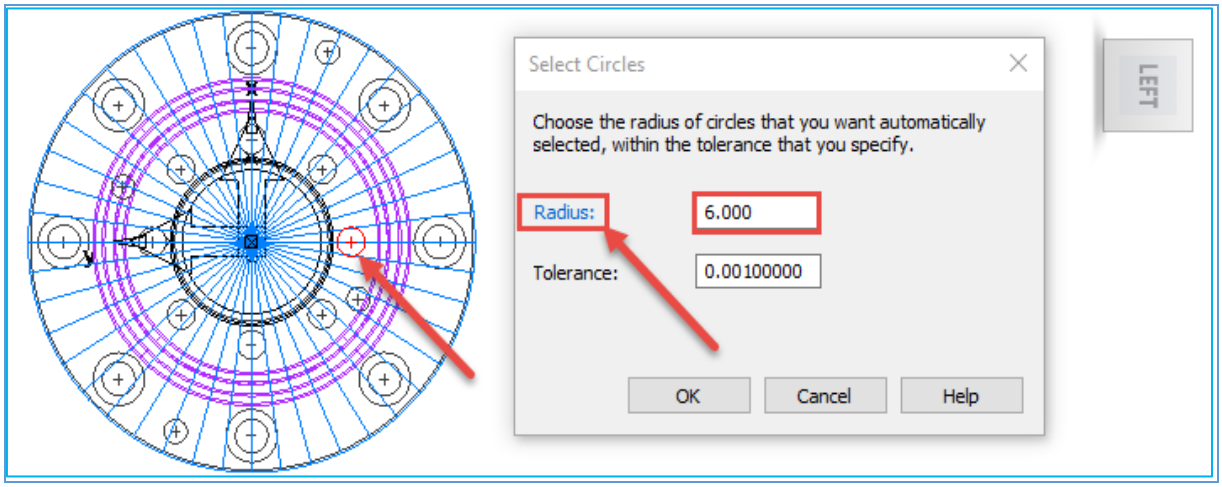


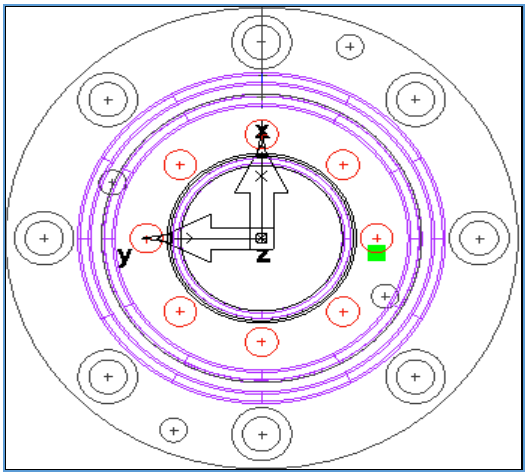
© Copyright 2016 Autodesk, Inc. All rights reserved

*We will now machine the grooves in the **OD**. Create the curves and then create the **Groove** features from curve as shown below.*



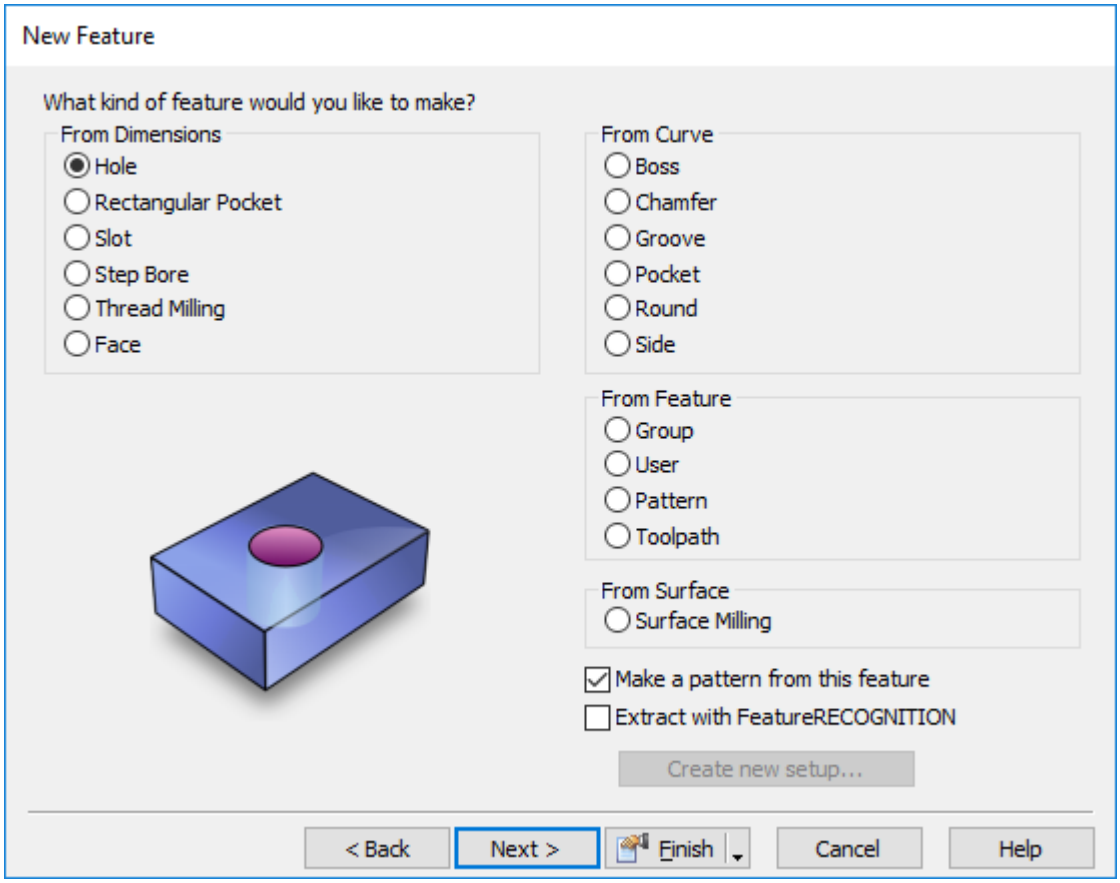
15 Eject the 3D Simulation. We will now machine the holes in the front end of the part. Select **Edit> Select Circles**. Then select the blue Radius Hyperlink as shown.



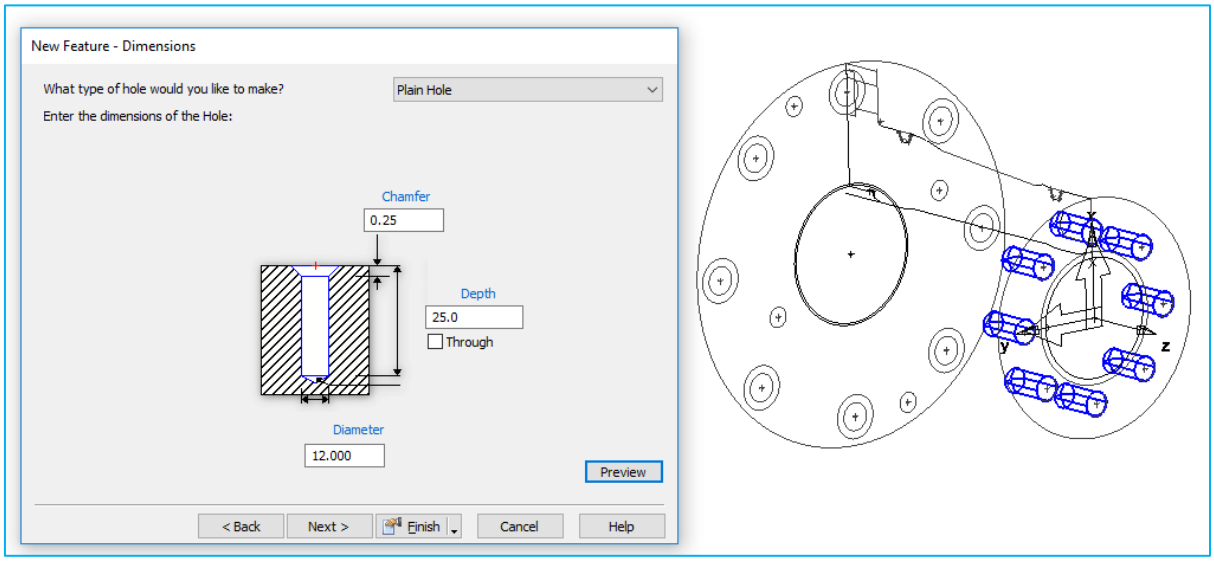


© Copyright 2016 Autodesk, Inc. All rights reserved

16 Create a new **Turn/Mill Hole** Feature.

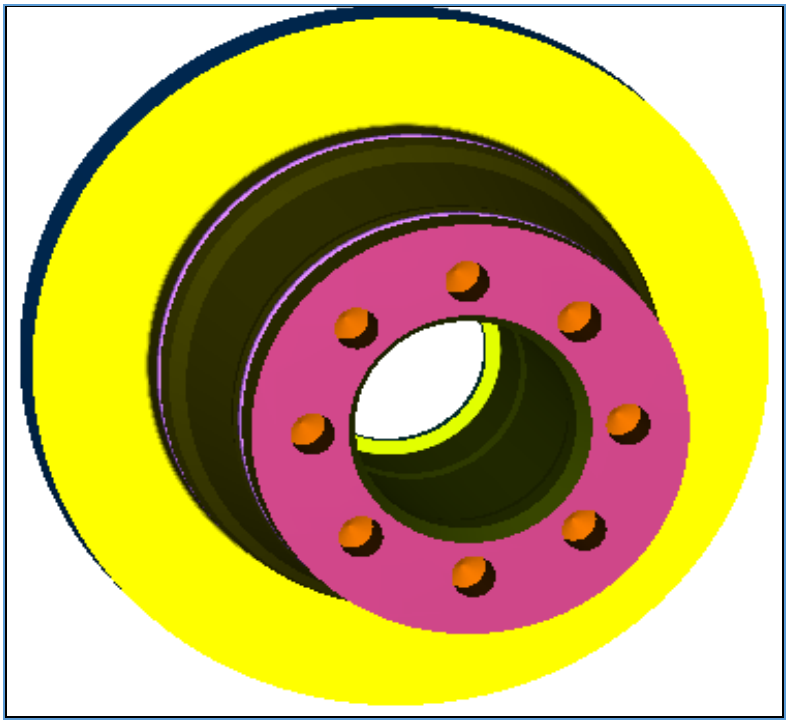


*Because there is more than one hole selected **FeatureCAM** will automatically select **Make a pattern from this feature**.*



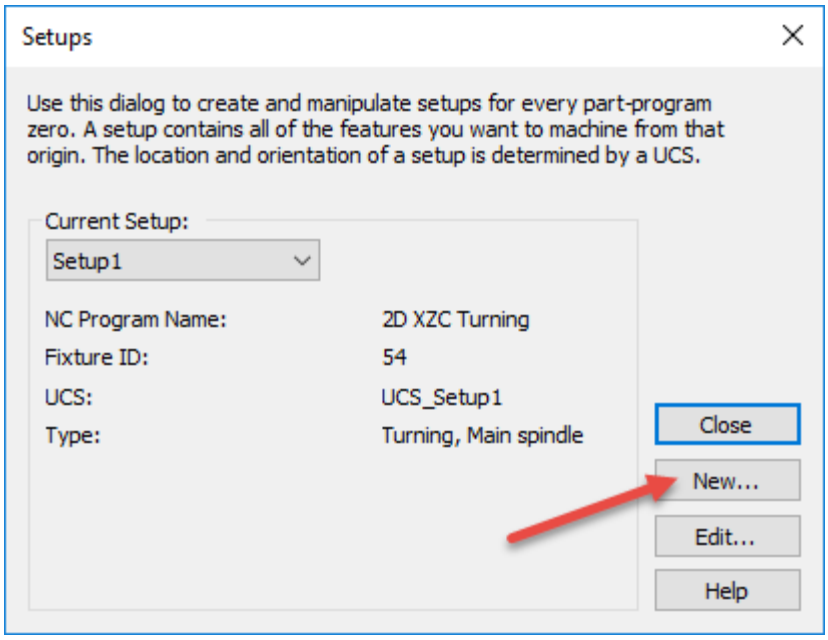
© Copyright 2016 Autodesk, Inc. All rights reserved

17 Select **Finish**. Select **3D Simulation**.

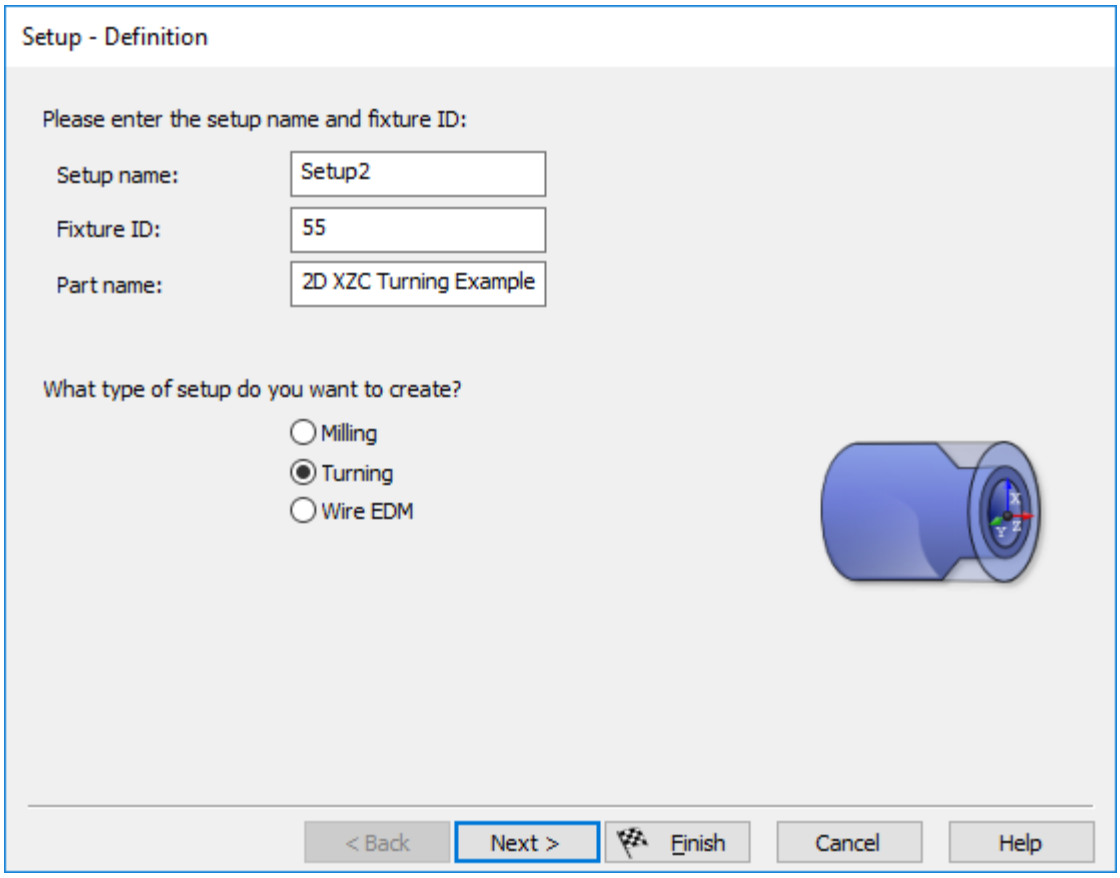


18 **Eject** the **simulation** and then create a new **Setup2** at the other end. For this exercise we will use the Sub-spindle. Double click on **Setup1** in **PartView** use Align to stock face. Select the Left-hand hand and then select pick location. Select the centre point at the large diameter end.

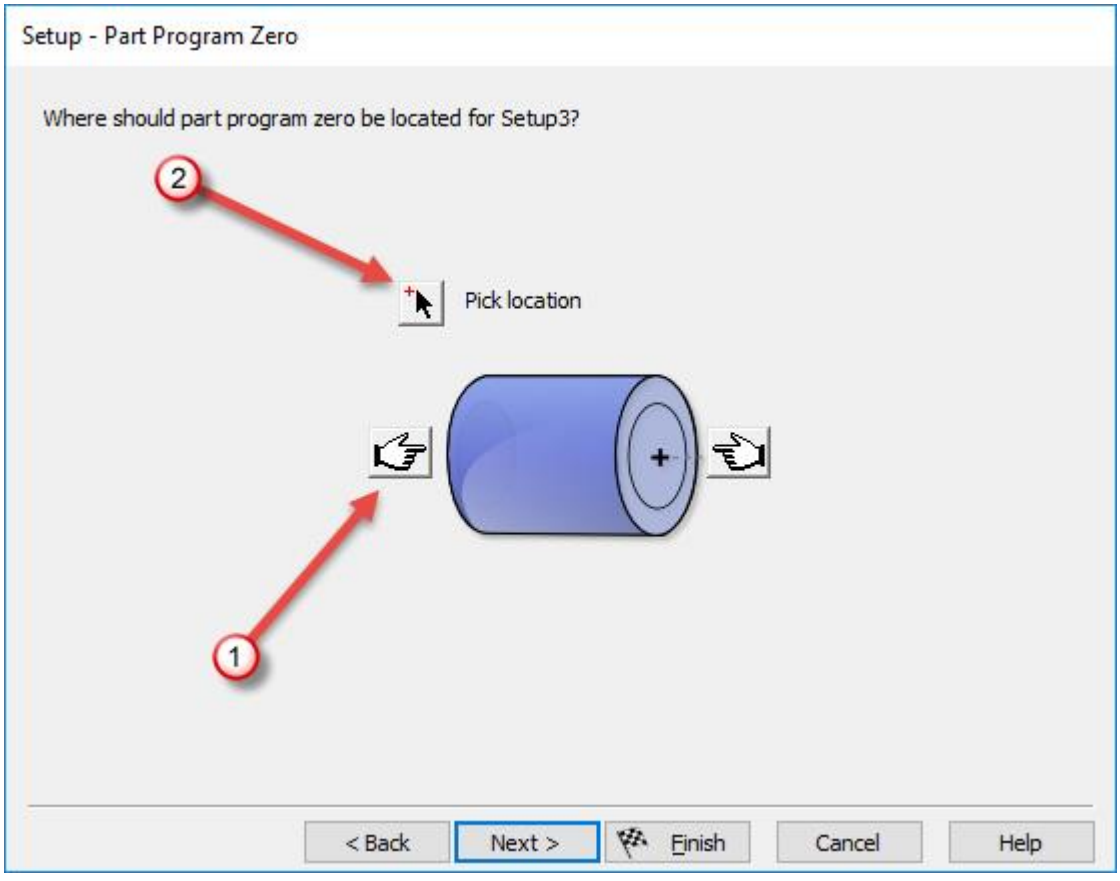
19 Double click on **Setup1** in **PartView** then select **New**.



© Copyright 2016 Autodesk, Inc. All rights reserved

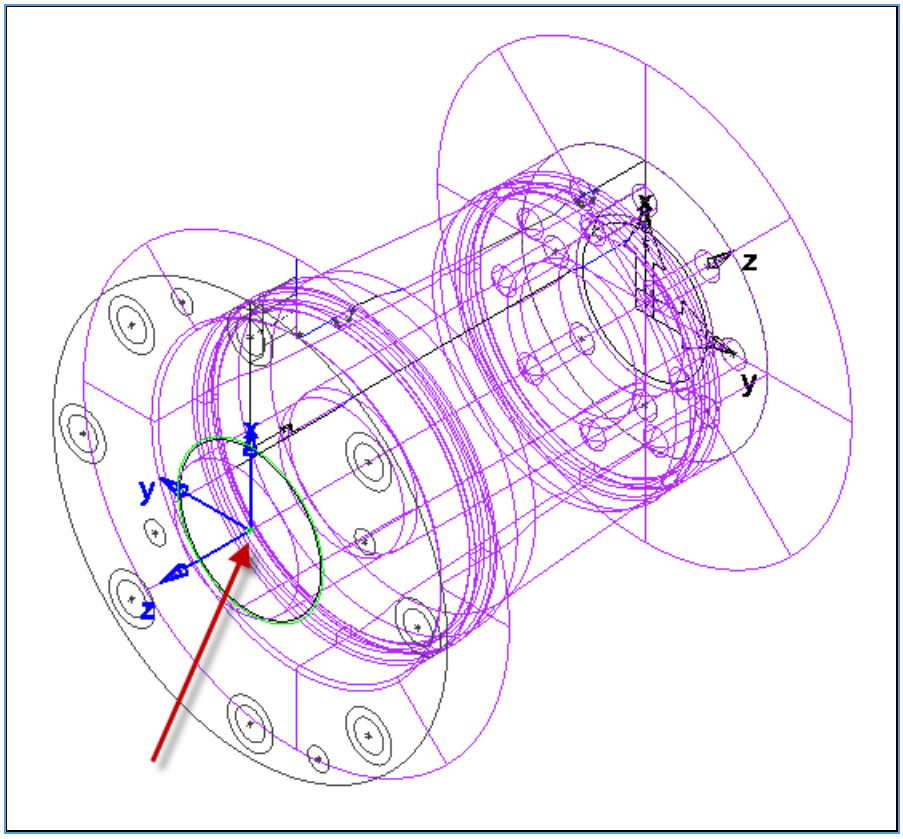


20 Select Next. Select **Align to stock Face**. Select **Next**. Select Left-hand hand. Then select **Pick location**.

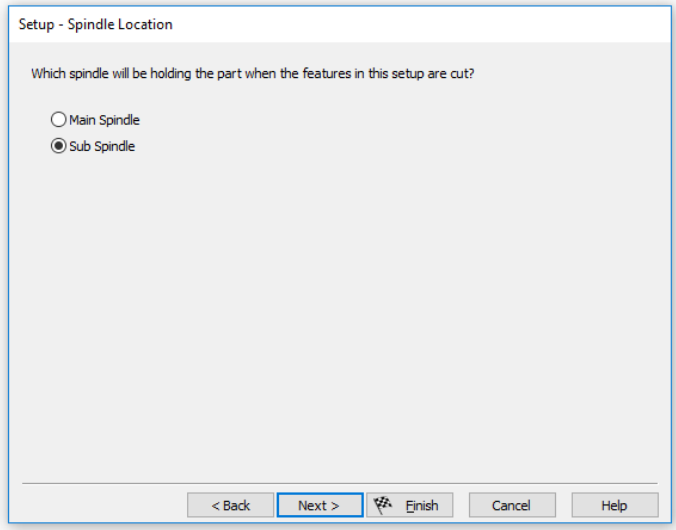


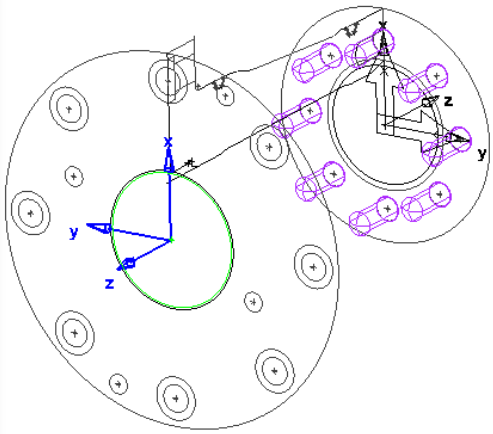
© Copyright 2016 Autodesk, Inc. All rights reserved

21 Pick the point as shown.



22 Then Select **Next** and select **Sub-Spindle**. Select **Finish** and **Close**.

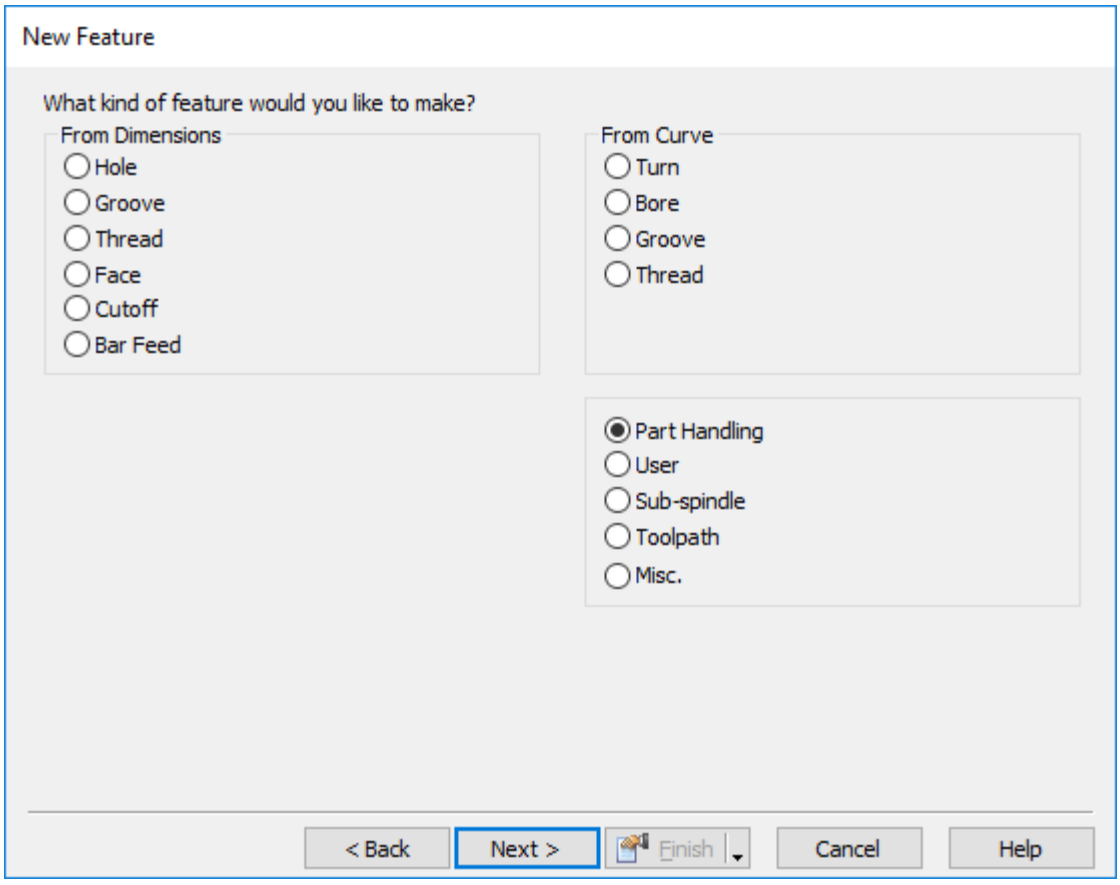




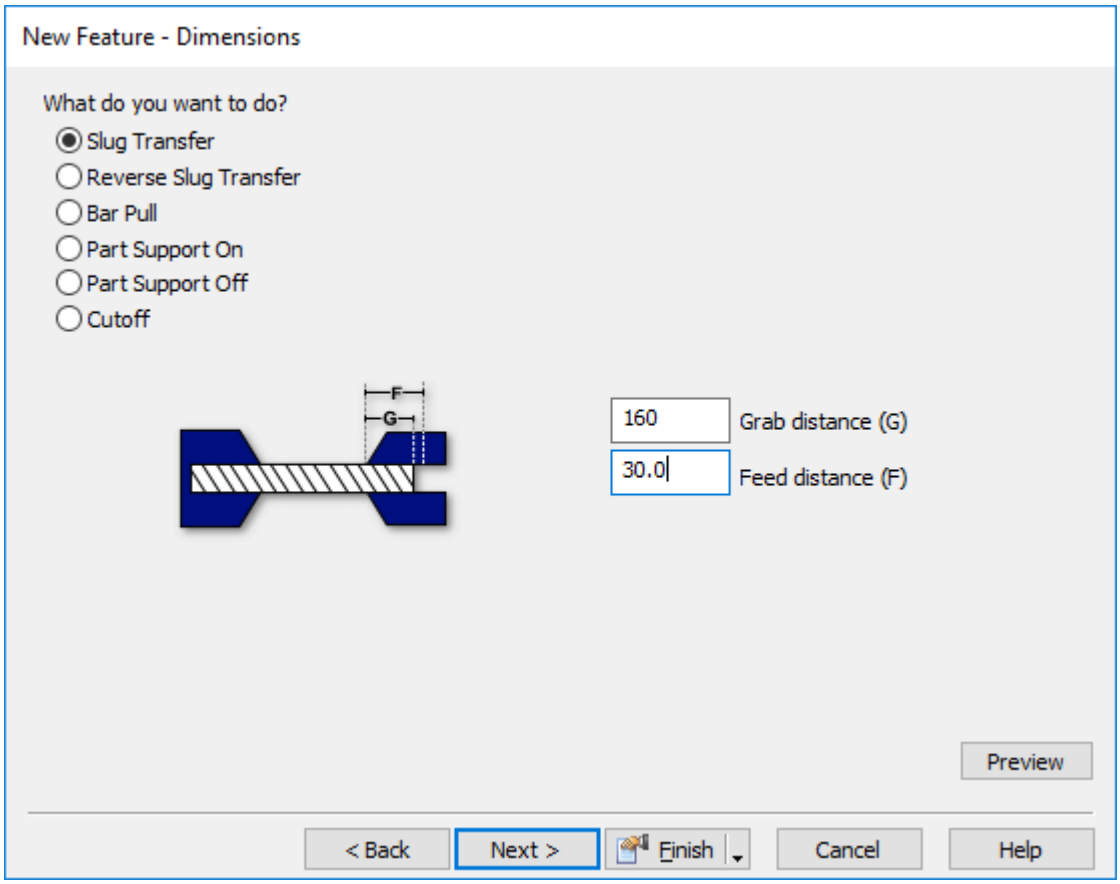
23 Select **Setup1** in **PartView**.

24 Create a **new Feature**. **Ctrl+R>Turning>Part Handling**.

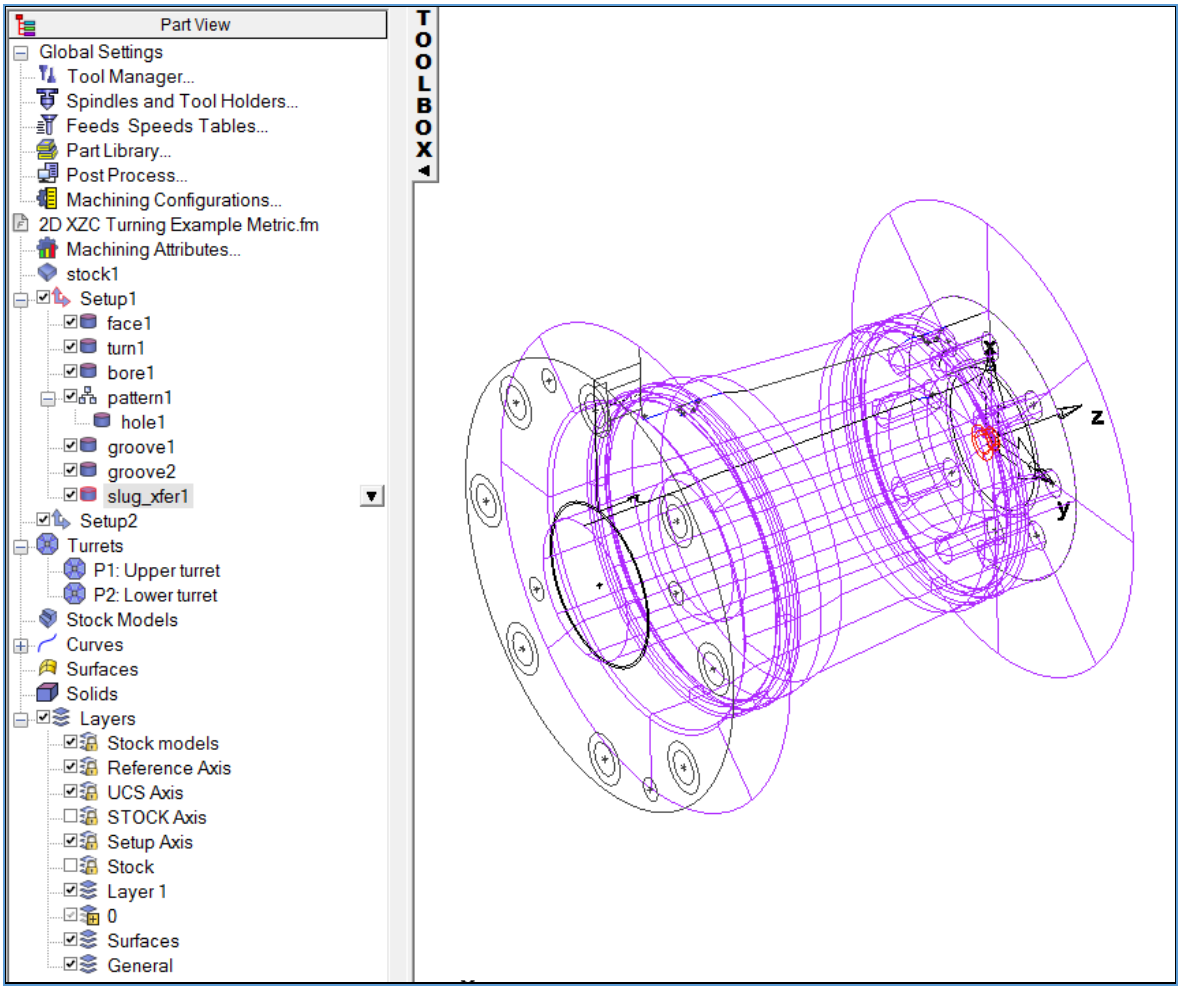
© Copyright 2016 Autodesk, Inc. All rights reserved



25 Select **Next**. The following menu will appear. Enter **160mm** for the **Grab distance.** Then select **Finish.**



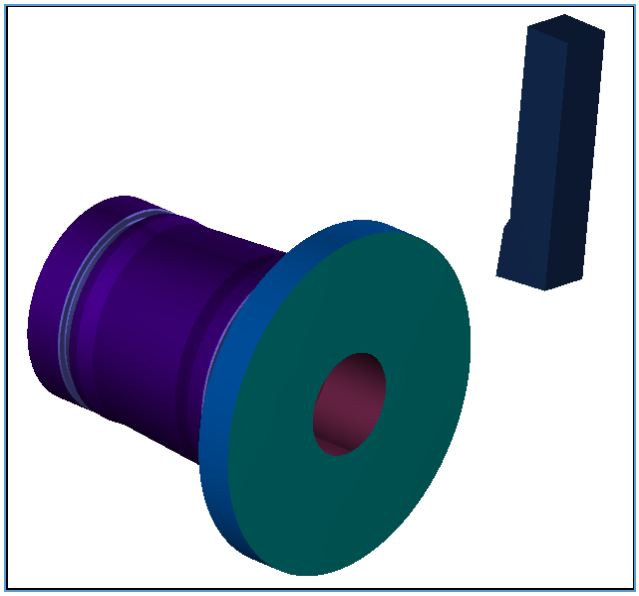
© Copyright 2016 Autodesk, Inc. All rights reserved



26 Select **Setup2** in **PartView**.

27 Create a **Face** operation in Turning for **Setup2**. Make sure you add a Roughing operation. Add 8mm to thickness.

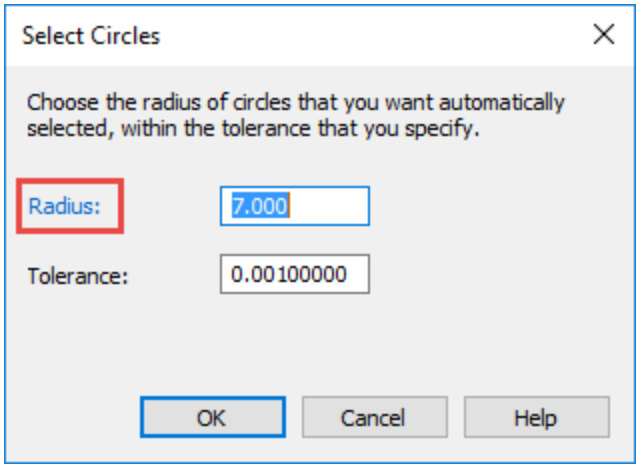
28 Run a **3D Simulation**.



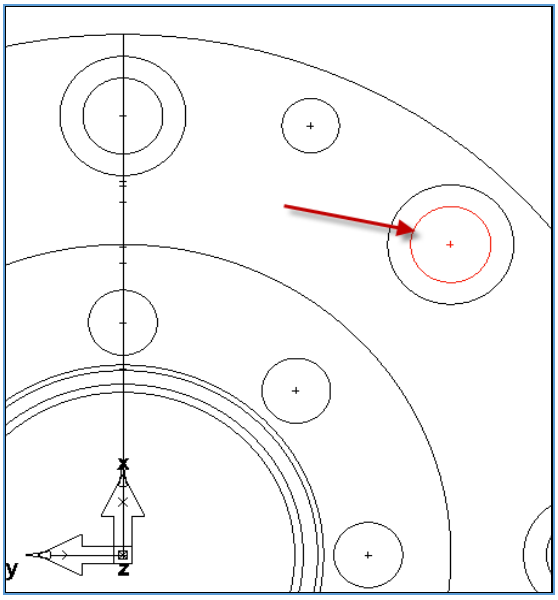
© Copyright 2016 Autodesk, Inc. All rights reserved

29 Please create a drilling operation for all the holes in **Setup2**.

30 From the menu select the Blue Radius Hyper-Link.



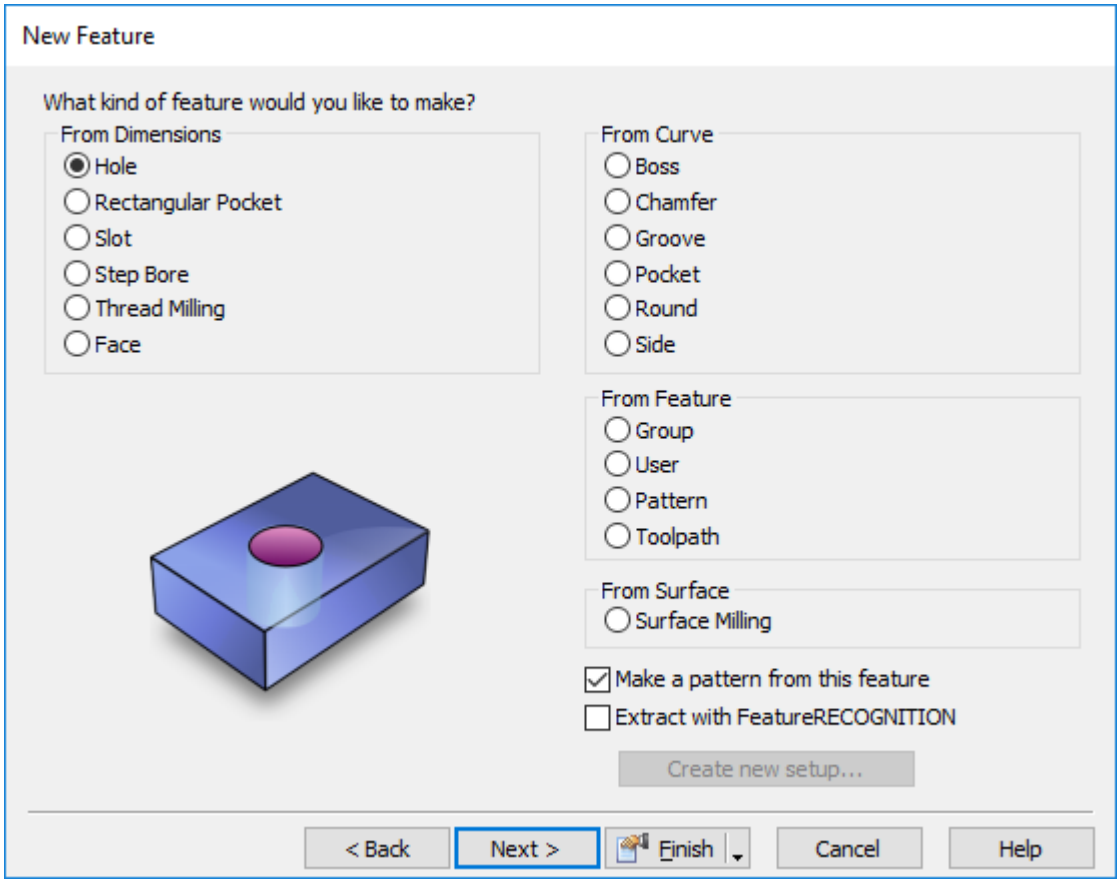
31 Select a **14mm** diameter Hole shown below. All of the same diameter circles will now be selected.



32 Select **New Feature**

33 Select **Turn/Mill**. Select **Next**. Select **Hole**

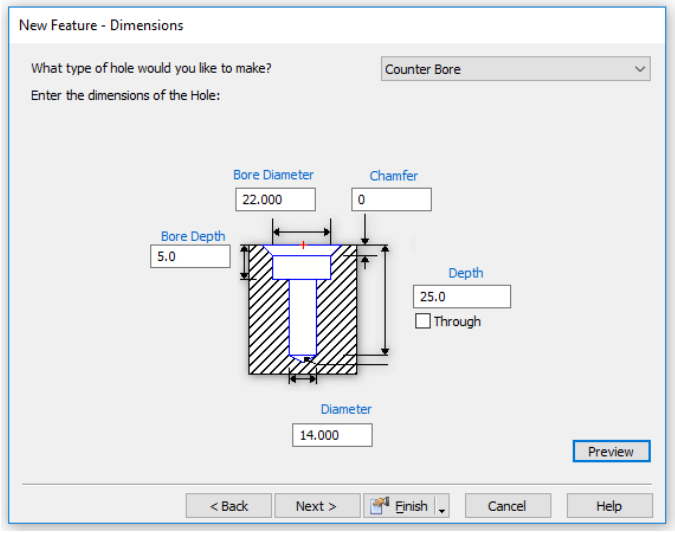
© Copyright 2016 Autodesk, Inc. All rights reserved

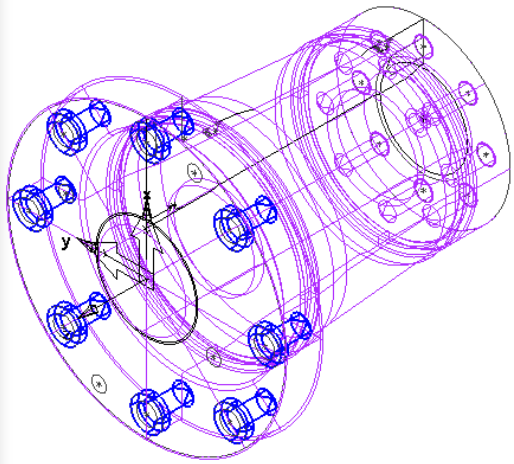


34 Select **Next**.

35 We need to select **Counter Bore** as our **Hole Type**.

36 Enter the **Metric** values shown below. Then select **Next**.



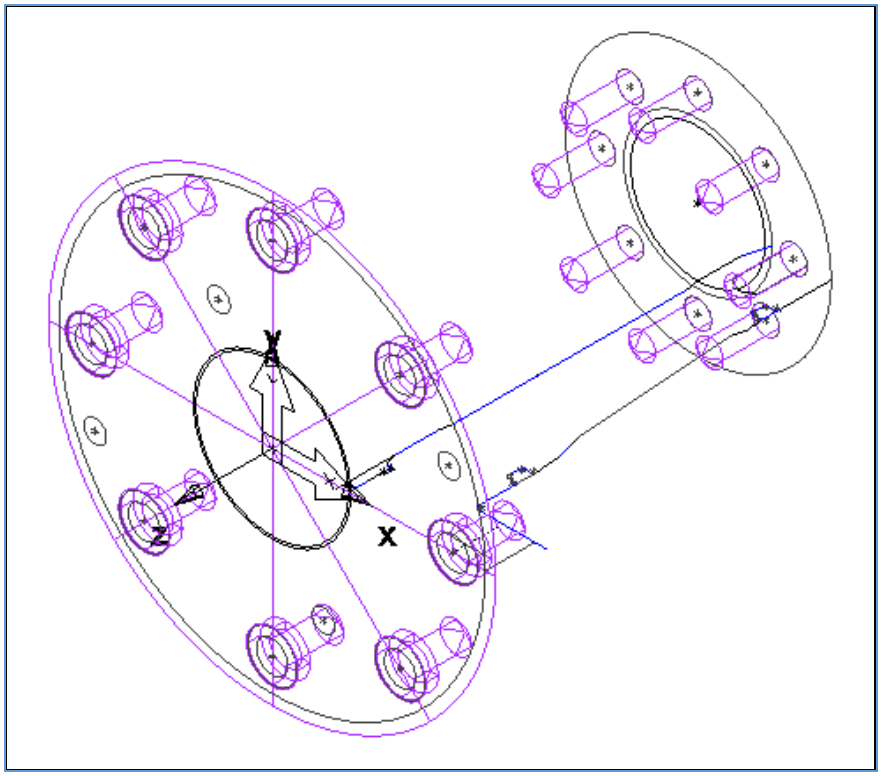


37 Please select **Points List Pattern in the setup XY Plane**. (Default)

38 Select **Finish**. All the selected holes on **Setup2** will now be machined.

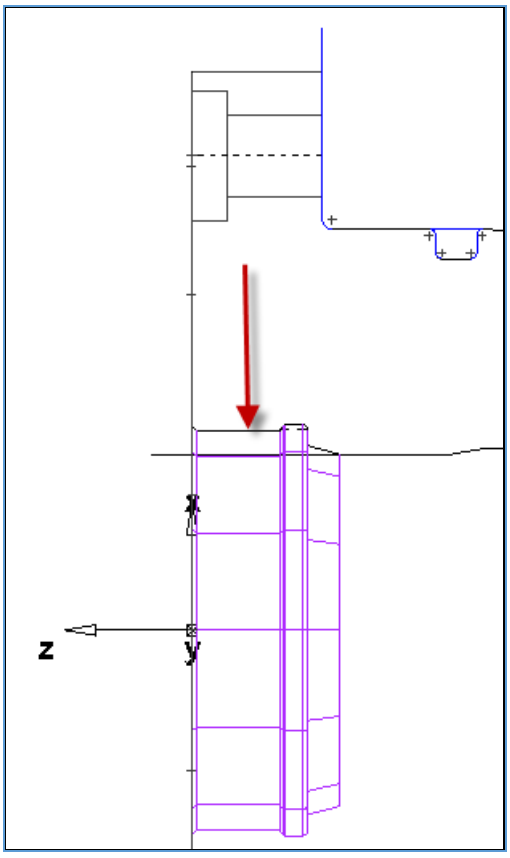
Apply this method to all of the remaining holes.

© Copyright 2016 Autodesk, Inc. All rights reserved



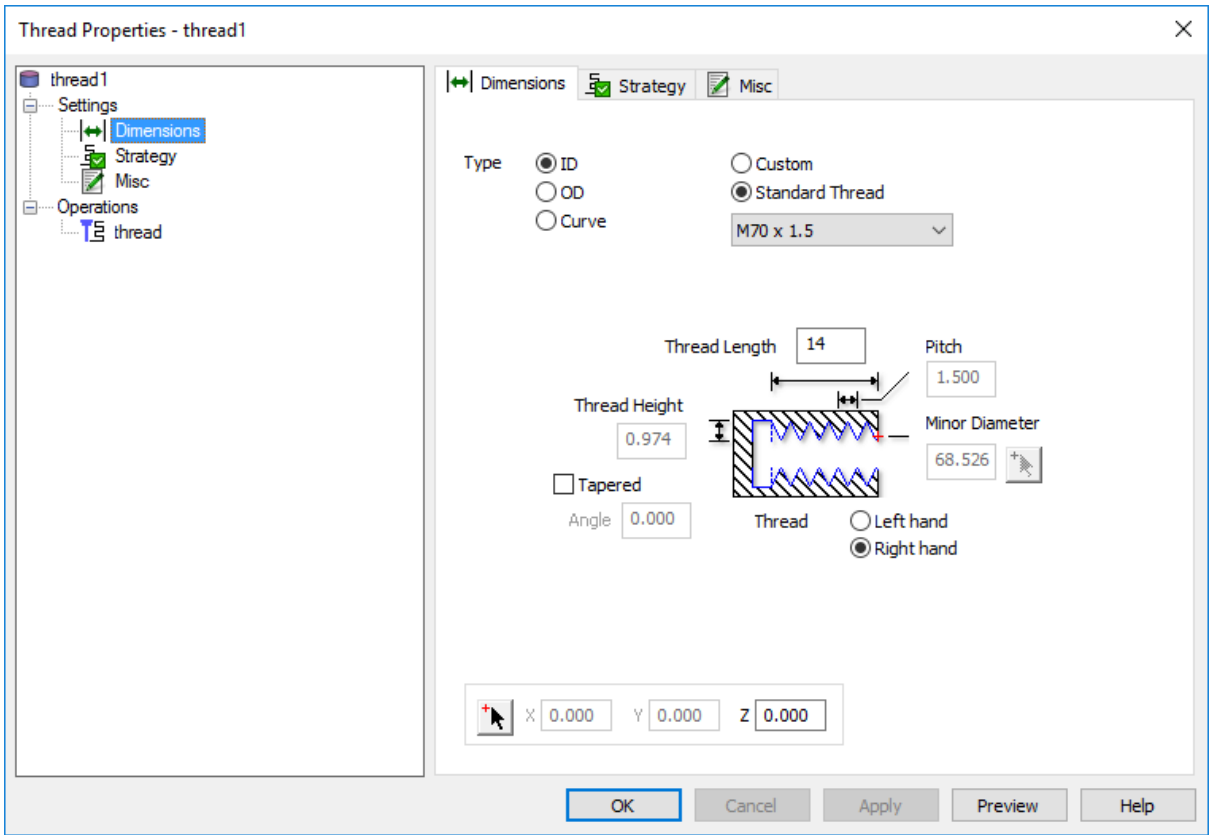
39 We will rough and finish the threaded profile in the bore. Create a **Curve** to machine the **Bore** feature. Make sure the boring bar has the correct orientation.

40 **Bore** Feature.

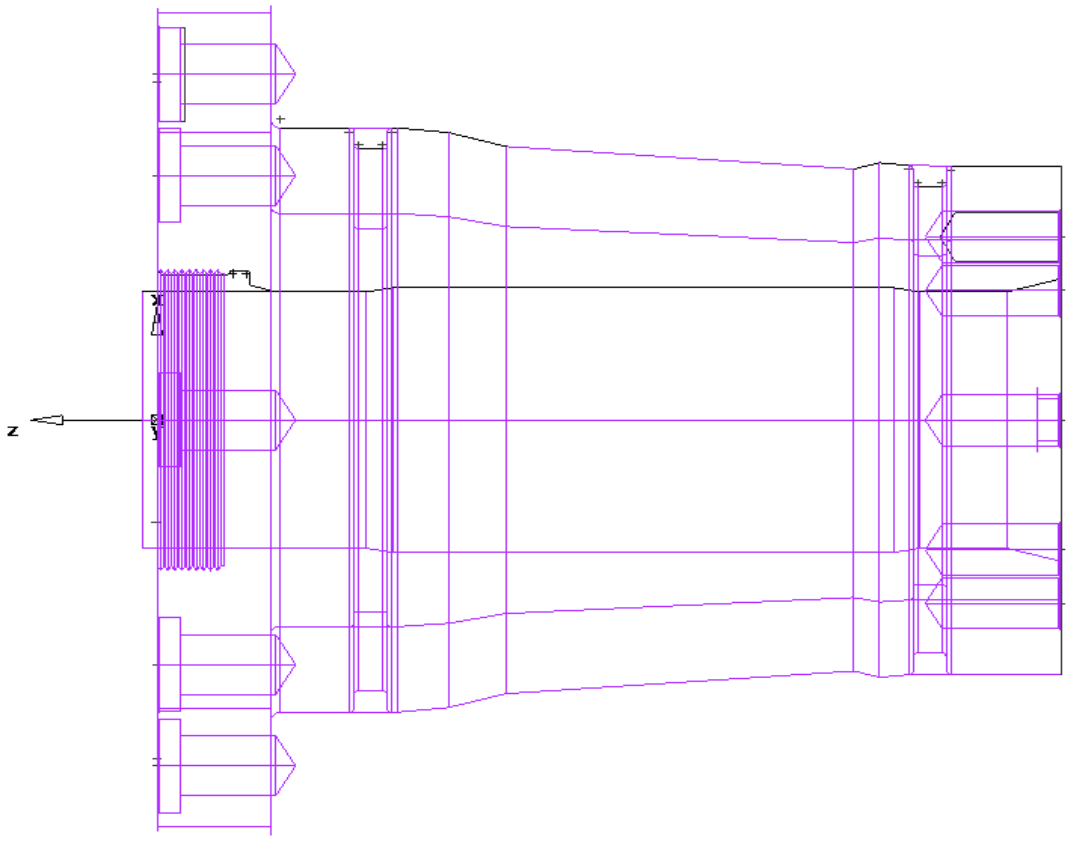


© Copyright 2016 Autodesk, Inc. All rights reserved

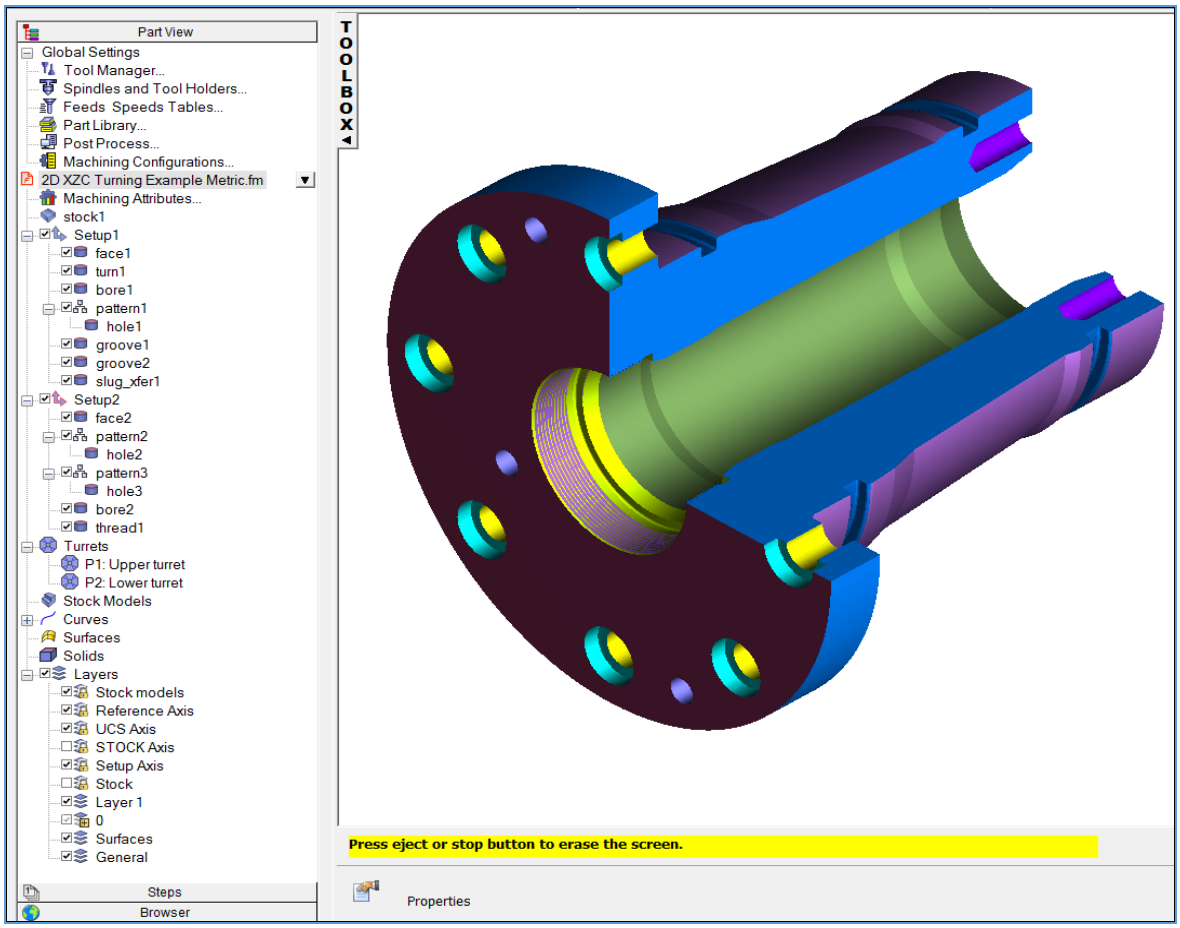
41 Create a New **Thread** Feature from **dimensions**.



42 Enter the following Metric Values shown above. Please **unselect relief groove**.



© Copyright 2016 Autodesk, Inc. All rights reserved



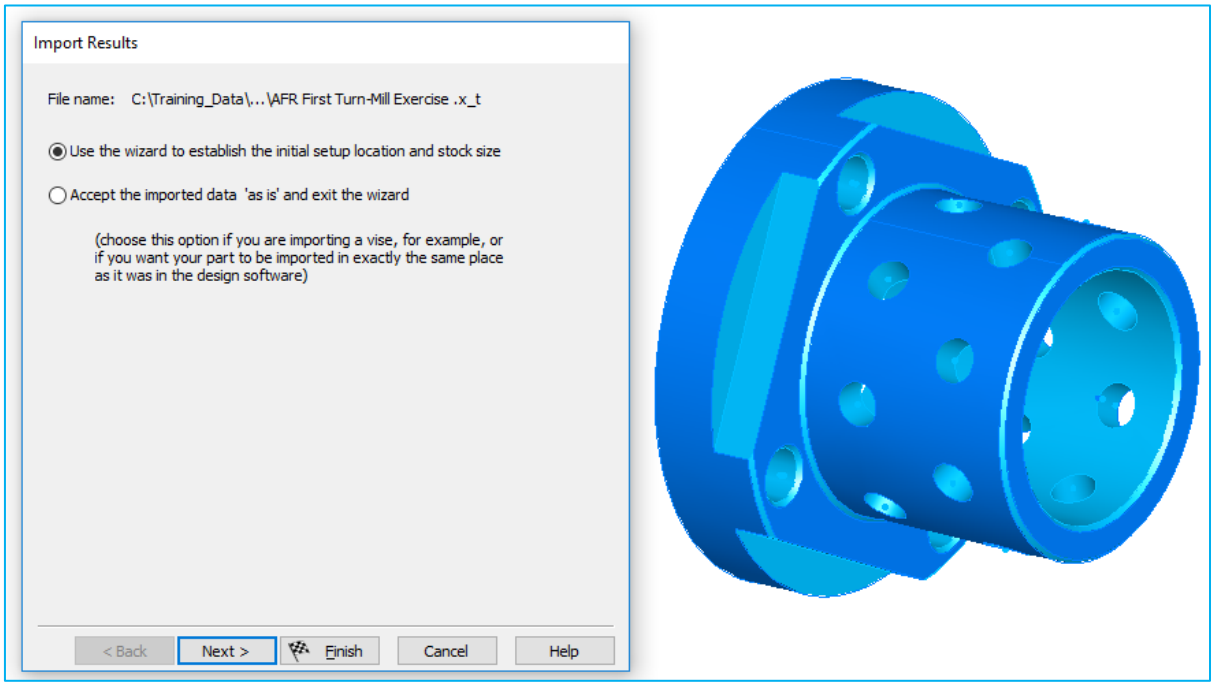
AFR First Turn/Mill Exercise

Previously we have extracted features from 2D drawings. What we are about to do is extract X, Z, C & Y axis features from a solid model.

1. Please start a new **Turn/Mill document**. Accept the stock defaults.
2. ***Import*** *the solid model* ***AFR First Turn-Mill Exercise*****. X_t**
3. Load the Mori Seiki NZ 2000 T2 Training Only.cnc Post Processor from the folder Training Posts.

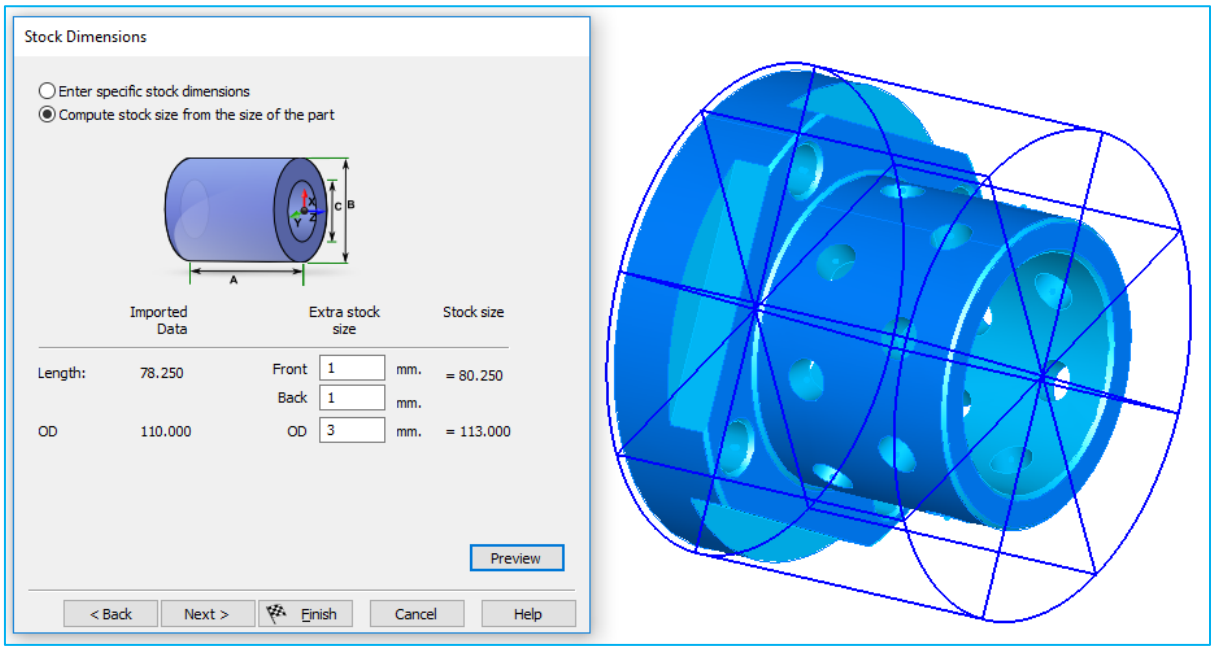
The part is already aligned in the ***Z axis;*** *use the wizard to establish the* ***Setup1*** *and* ***Stock1*** *size.*

© Copyright 2016 Autodesk, Inc. All rights reserved



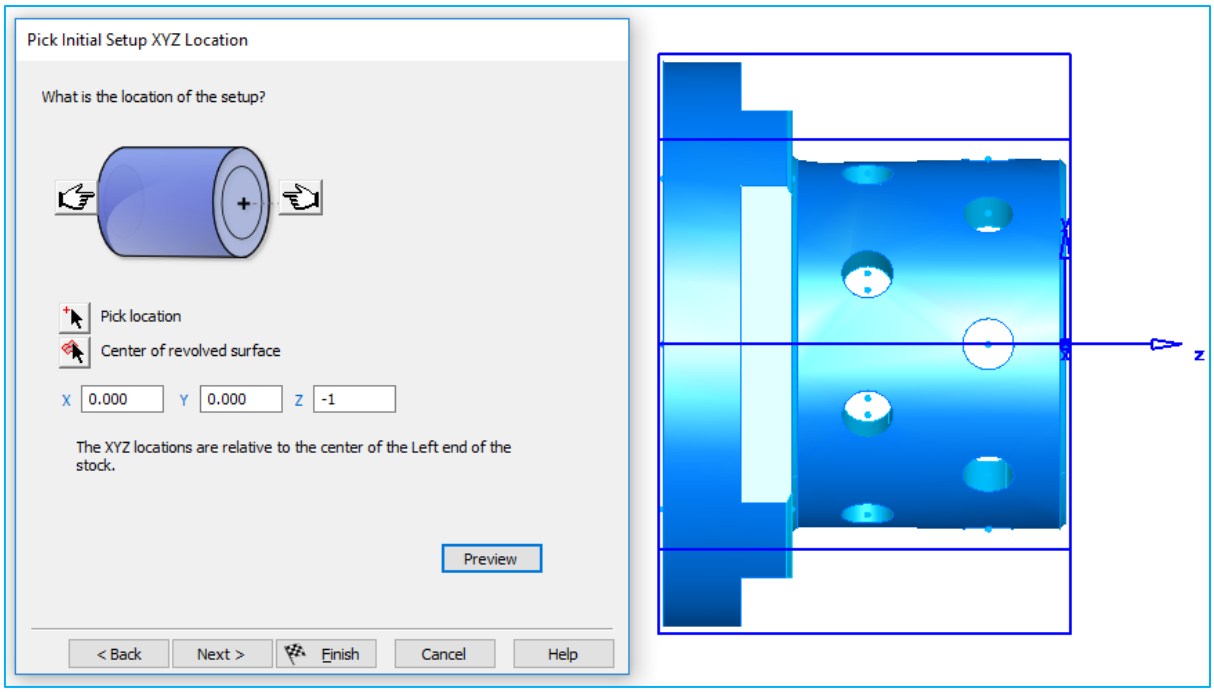
*You will notice that we have **Hole** features around the diameter **(Rotary C axis)** and **Side** features in the **Z axis**. We have the option to change this to **Y axis** if we prefer using interactive Feature Recognition **(IFR)**.*

4 Select **Next** five times until you get to the **Stock Dimensions**. Enter the values as shown on the next page.

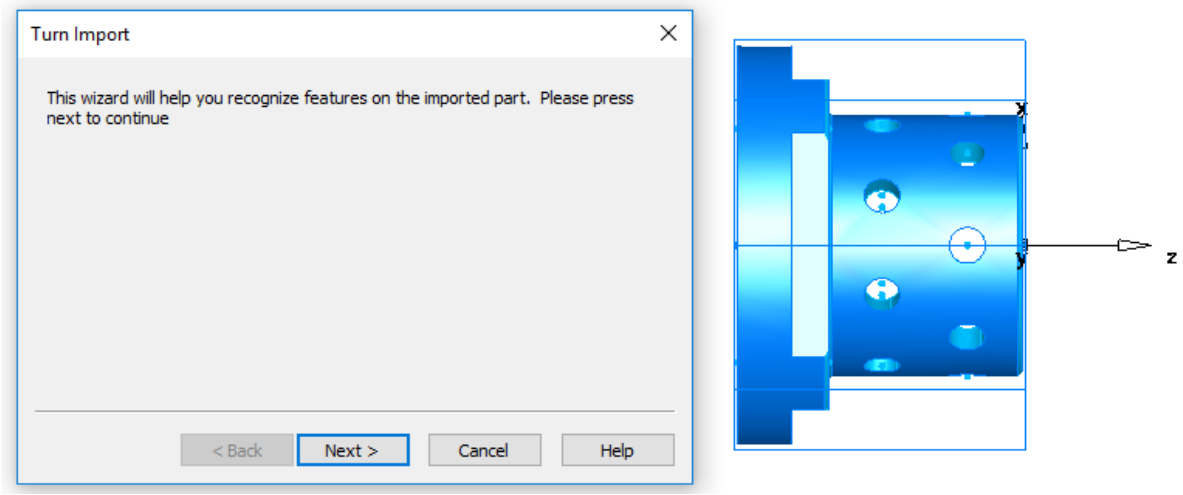


5 Select **Next**. The Setup Position needs to be reset. Select the right hand Icon and the enter **-1mm** then Select **Finish**.

© Copyright 2016 Autodesk, Inc. All rights reserved



The Wizard will help you recognise all the Features from the solid model.

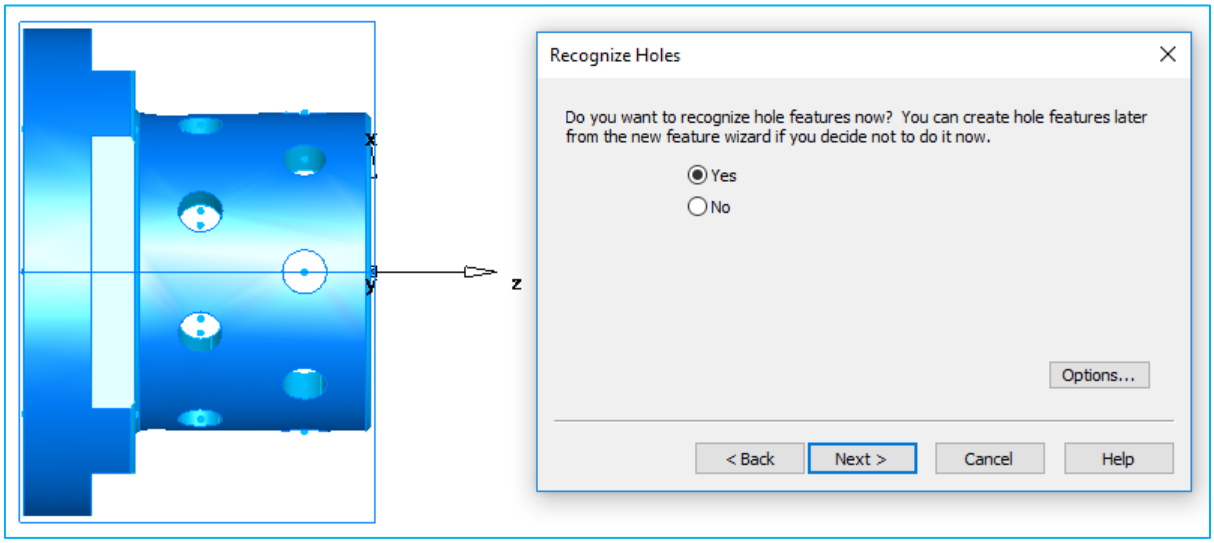


6 Select **Next**. FeatureCAM will ask if you want to recognise hole features now or later. Select **Yes** and then select **Next**. Then select the option **Select Features for Setup1**. All the recognised features will be shown.

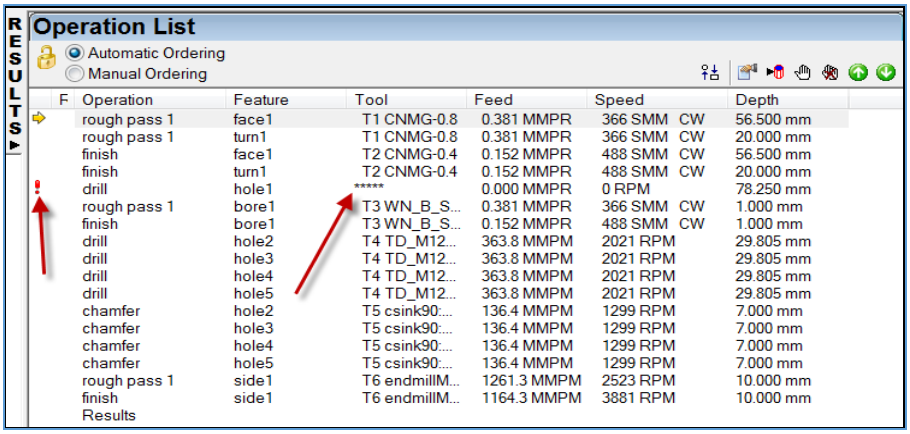
Please note that all of the features will be extracted but errors will appear. For example we do not have a 42mm diameter drill in the training crib. Plus the boring Curve is not long enough.

We will delete the hole feature for the centre hole and add this to the Bore feature as a Pre-drill option. We can then increase the Bore curve length by using the Trim extend option.

© Copyright 2016 Autodesk, Inc. All rights reserved



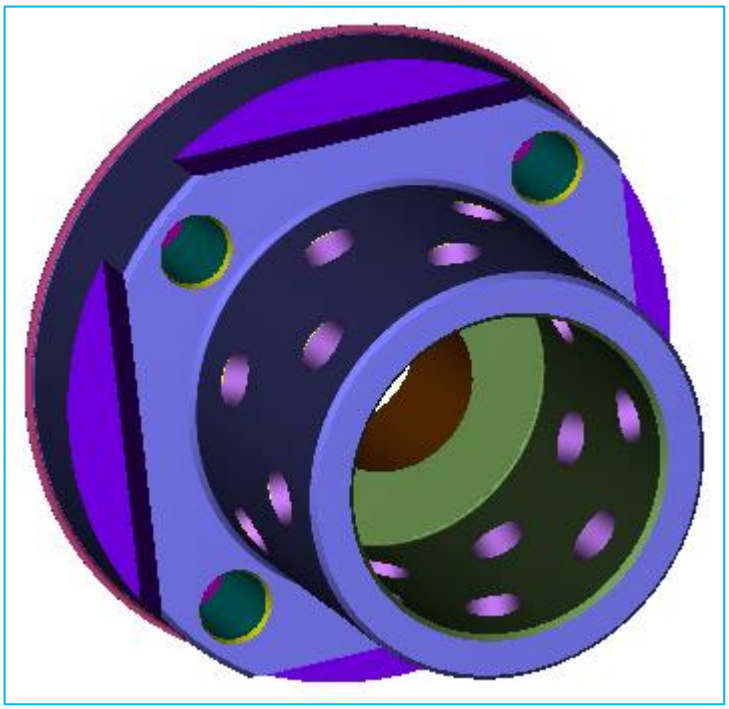
7 Select **Finish**. FeatureCAM will most likely come up with an error to say it cannot find a suitable drill.



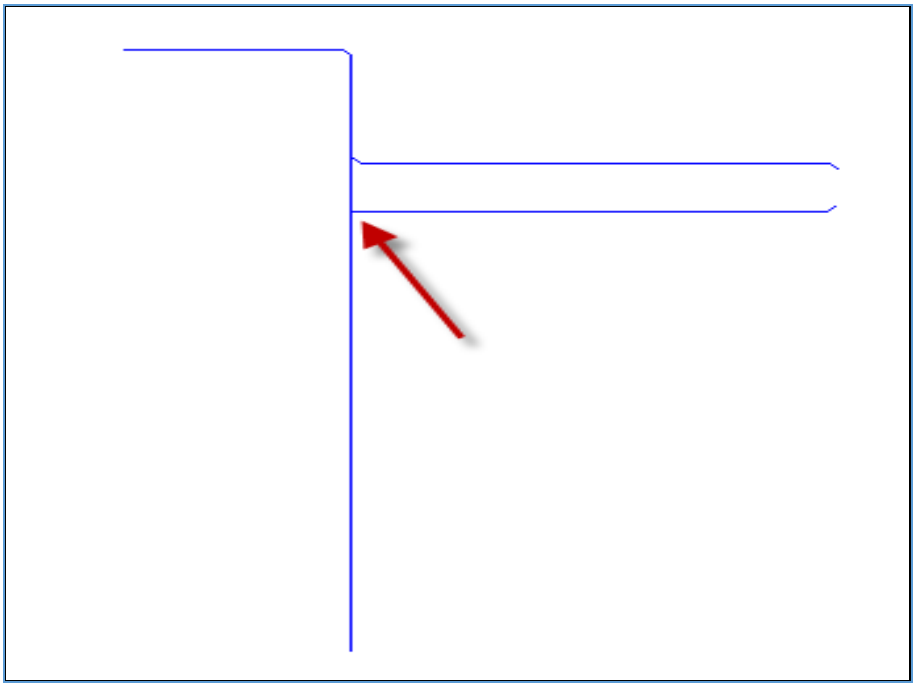
*Double click on the ***** and then select a suitable drill from your tool database or create a new one. The finished size is 56mm. Or you can delete the Hole feature and add a Pre-drill to the Bore Feature. Add a **30mm drill** and change the **depth to 80mm**. Remember to start boring from **15mm** by setting the **Minimum radius boundary** from **Rough>Boring**.*

© Copyright 2016 Autodesk, Inc. All rights reserved

*Run a **3D simulation** to see the features that have been machined. You will notice that the bore feature has only machined up to the shoulder. See the image on the next page. Select **ESC** to **eject** the simulation. Then Select **Hide all** and then select **Show all curves**. Then select **Ctrl+5** or **Top View**.*

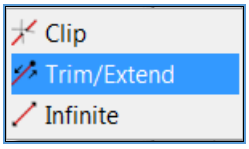


8 You will just see the curves on the screen as shown below.

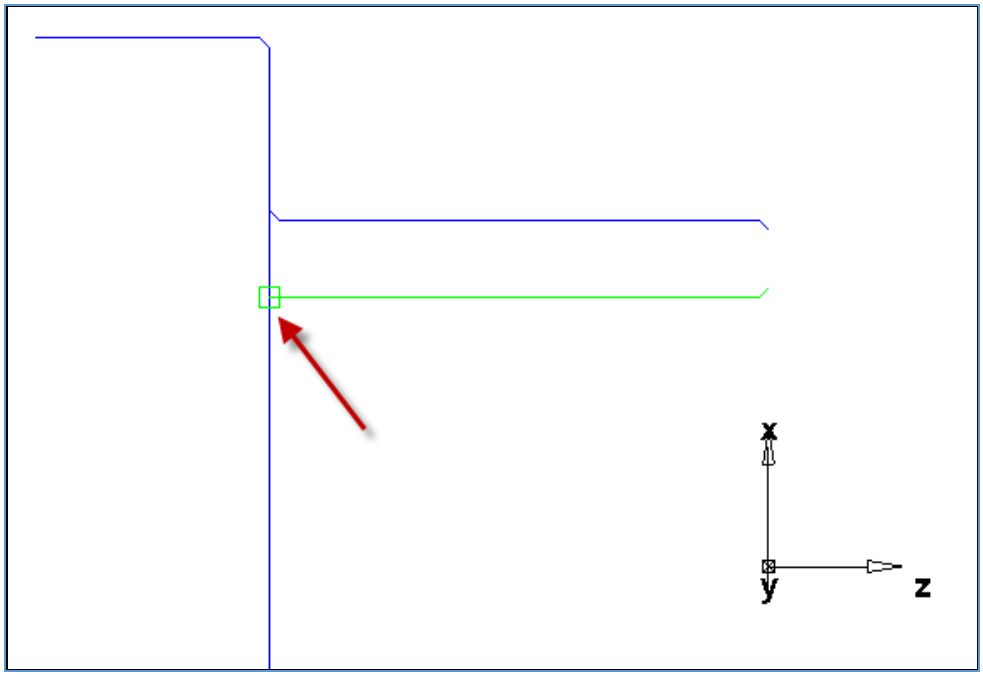


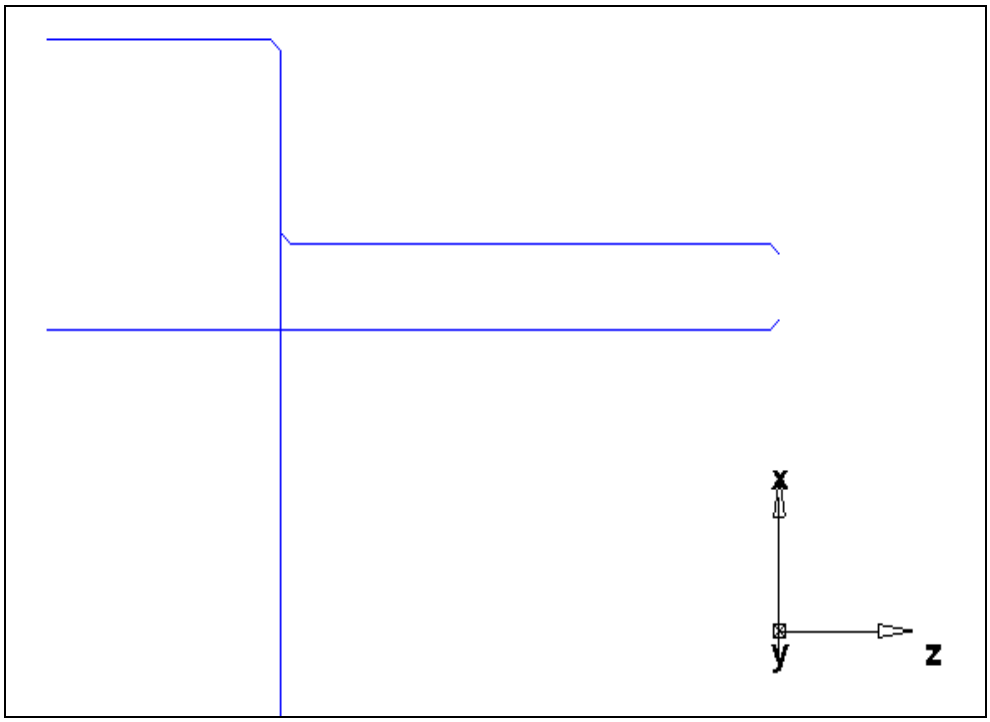
*What we need to do is extend the curve using the **Trim/Extend Icon**?*

© Copyright 2016 Autodesk, Inc. All rights reserved



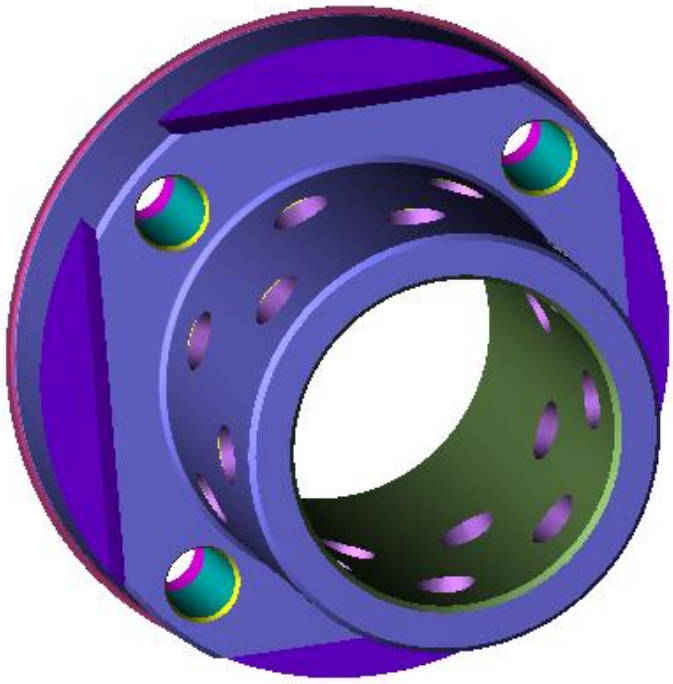
9 Select the **end point** as shown and hold the left mouse button down to extend the Curve. You may have to extend this passed the excess material.





*By extending the curve **FeatureCAM** will automatically update the machining.*

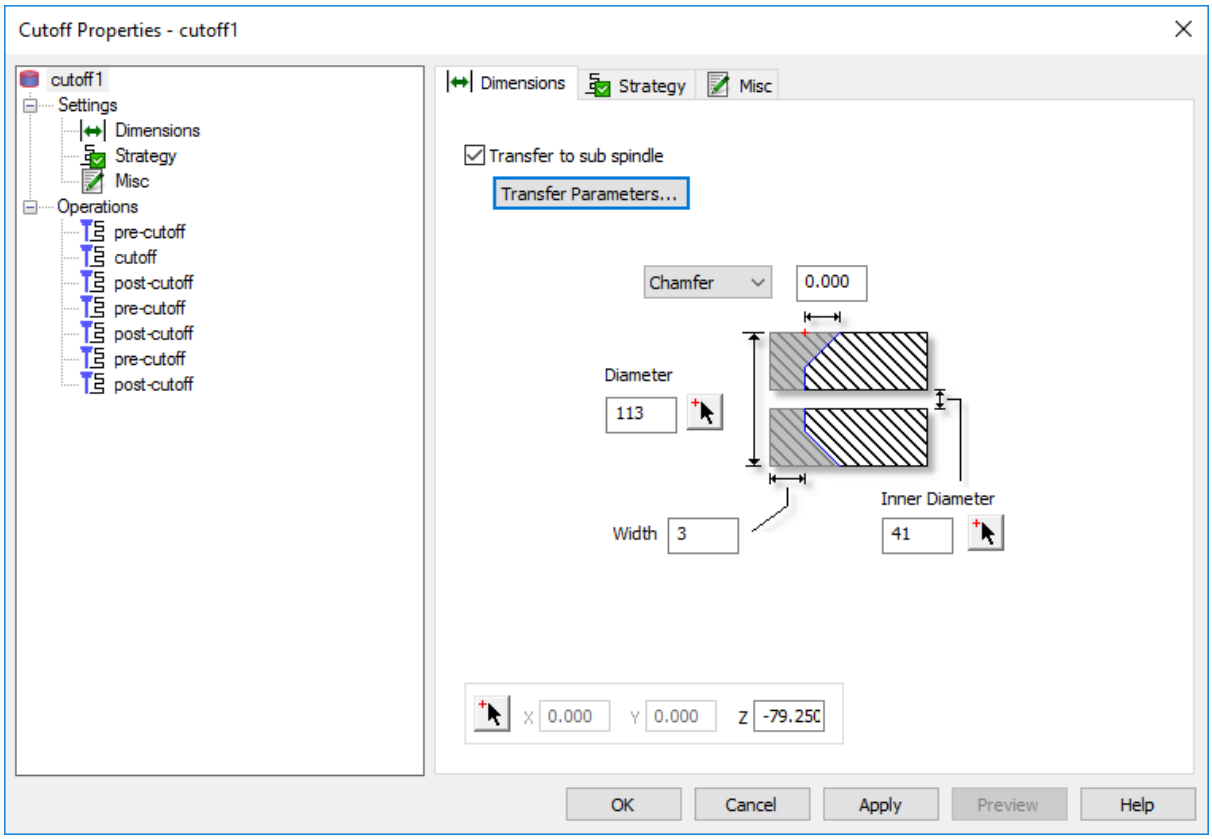
© Copyright 2016 Autodesk, Inc. All rights reserved



10 Select **Setup1** from the **PartView** menu.

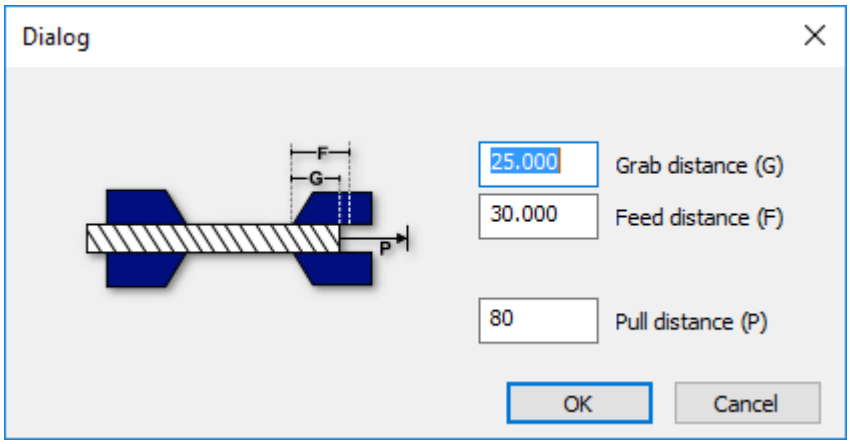
*What we are going to do is use **Cutoff** and allow **1mm** to machine off after transferring to the **Sub-Spindle**.*

11 We are still using **Setup1**. Create a new **Cutoff** Feature with the following Parameters.



© Copyright 2016 Autodesk, Inc. All rights reserved

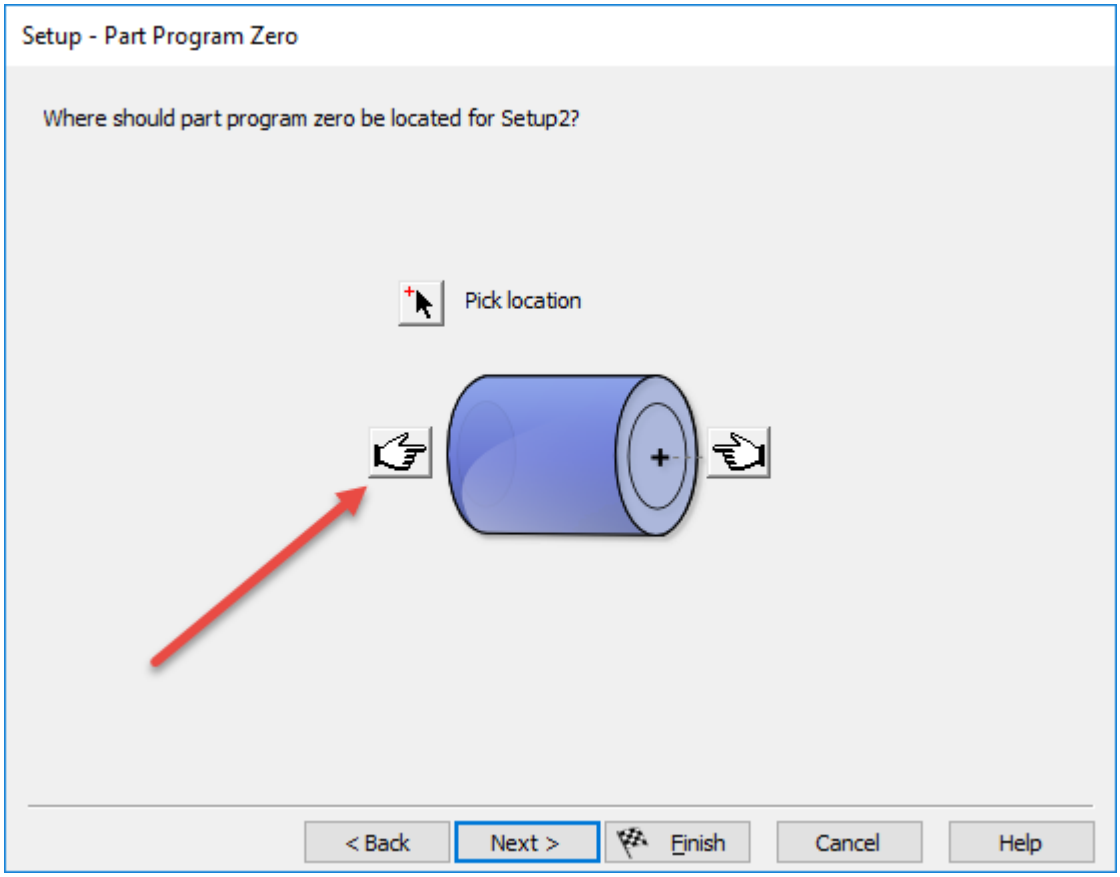
12 You will notice from the previous menu that we now have the option to **Transfer** to the **Sub-Spindle** from within the **Cutoff** menu. Please select this option and accept the default **Grab distance** of **25mm plus 80mm** for the **Pull distance**.



13 Select **OK** to action the command.

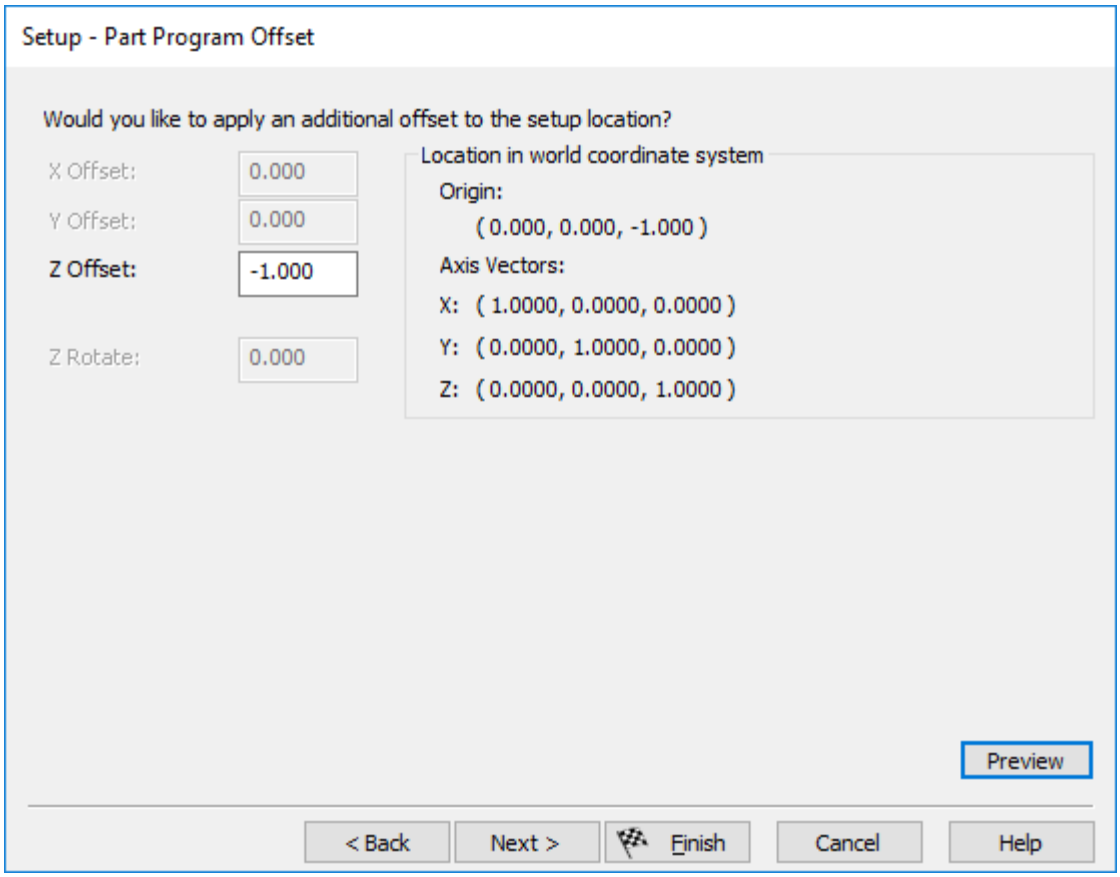
*We will now create **Setup2** so we can machine the excess material of the other end.*

14 Double click on **Setup1** and select **New** and then **Next** and select **Align to Stock Face.** Select **Next.** Then select the **Left-hand Hand.**

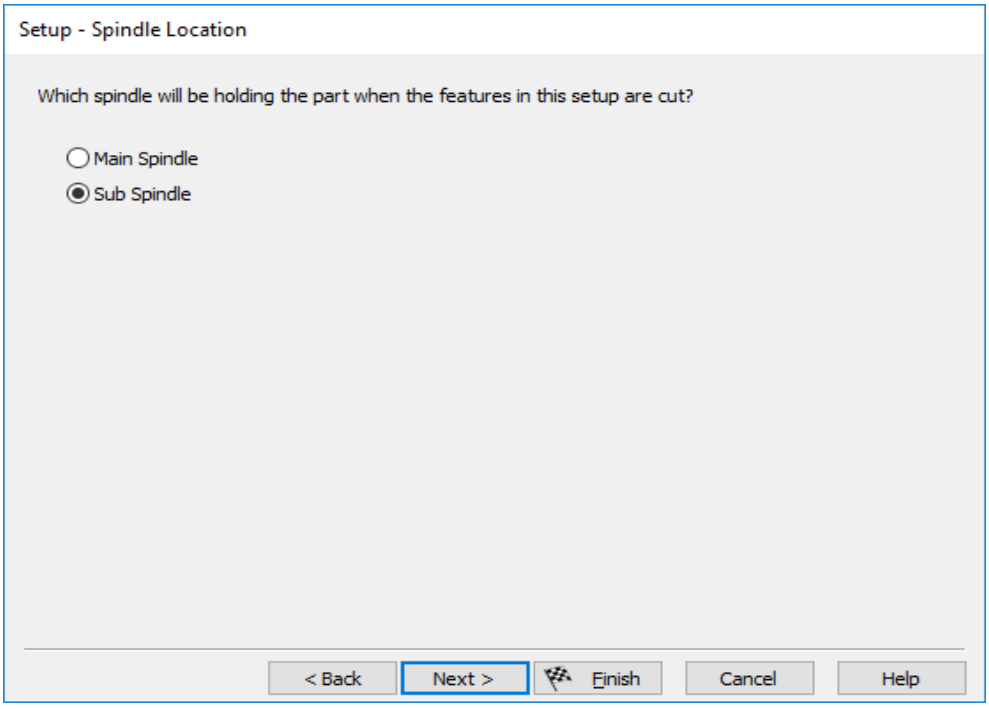


15 Select **Next**. On the next menu enter **-1mm** in the **Z Offset**.

© Copyright 2016 Autodesk, Inc. All rights reserved



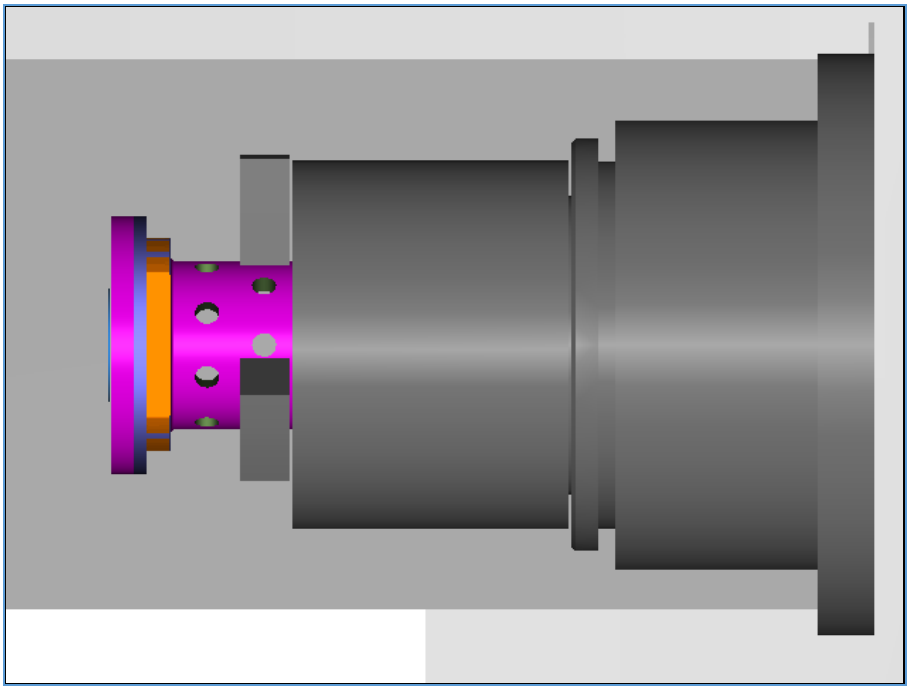
16 Select **Next** and then select **Sub-Spindle**.



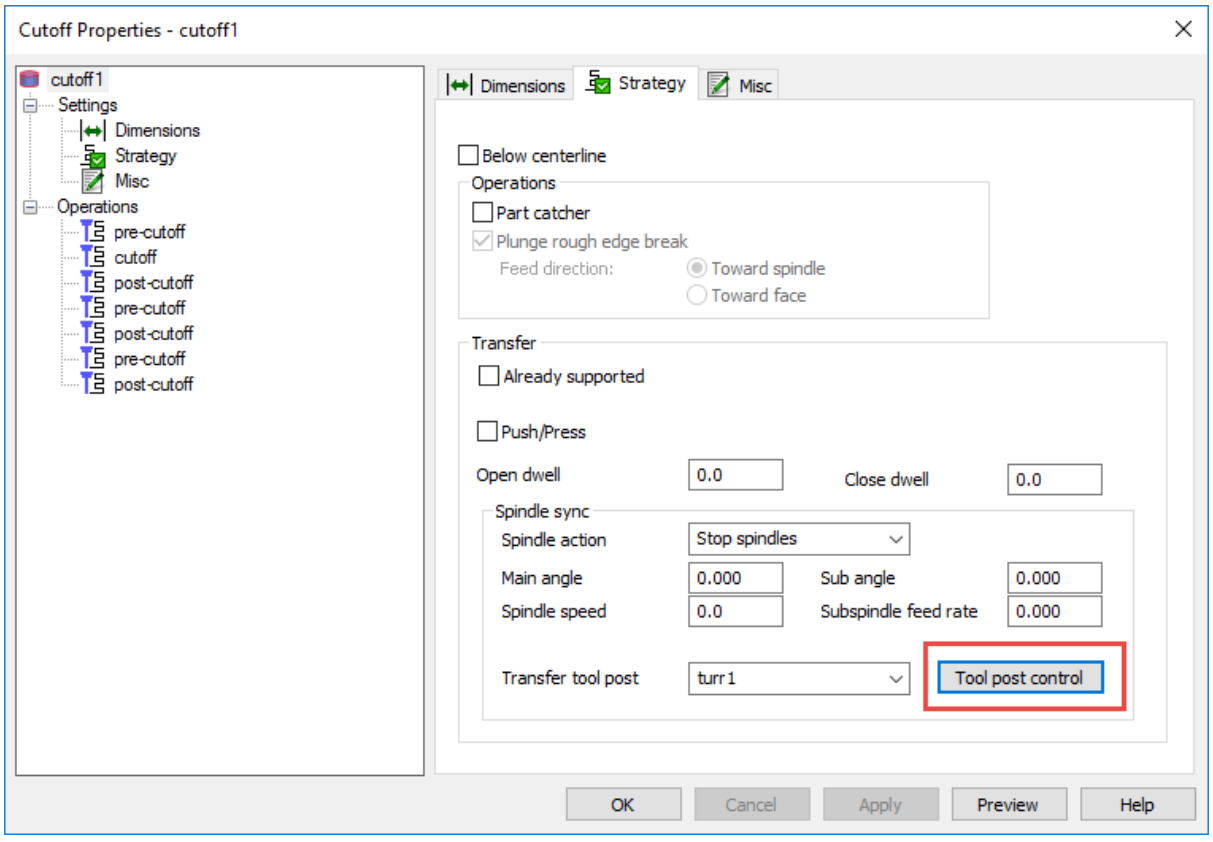
17 Select **Finish**.

18 Please select **Setup2**. Then create and new **Face** feature. See the finished image on the next page. This is shown in the sub spindle.

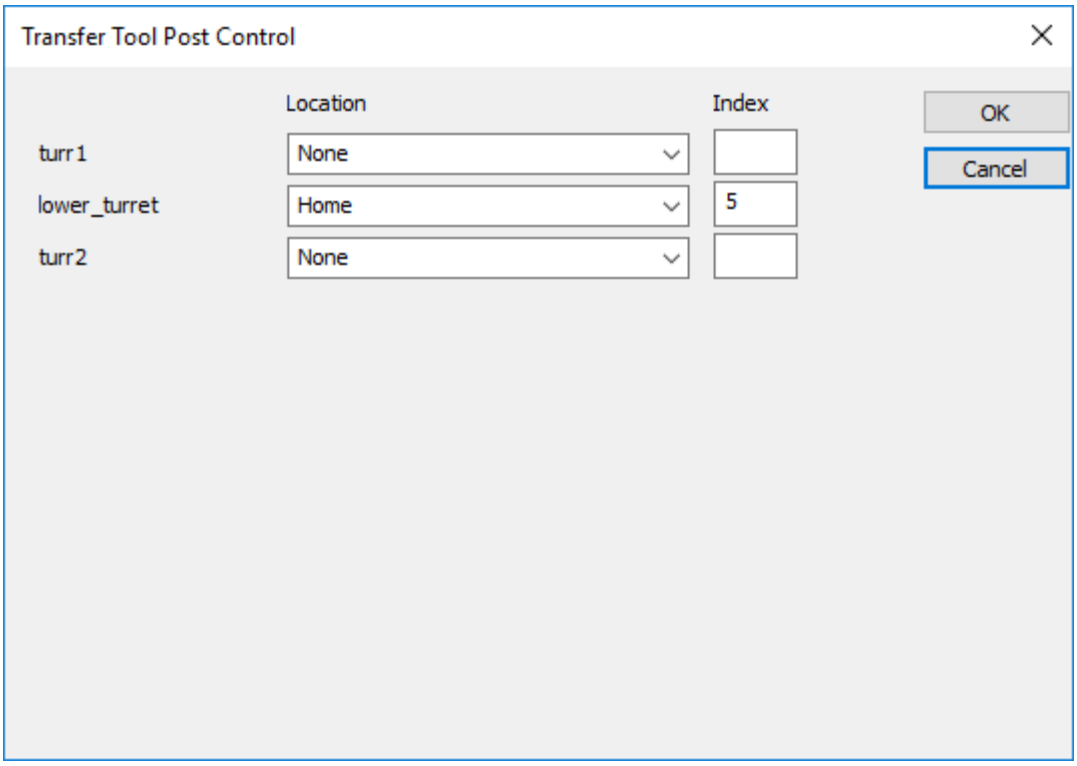
© Copyright 2016 Autodesk, Inc. All rights reserved



Another option that can be useful is to orientate the lower turret when transferring from the Main spindle to the Sub-Spindle. See menu below to activate this option.

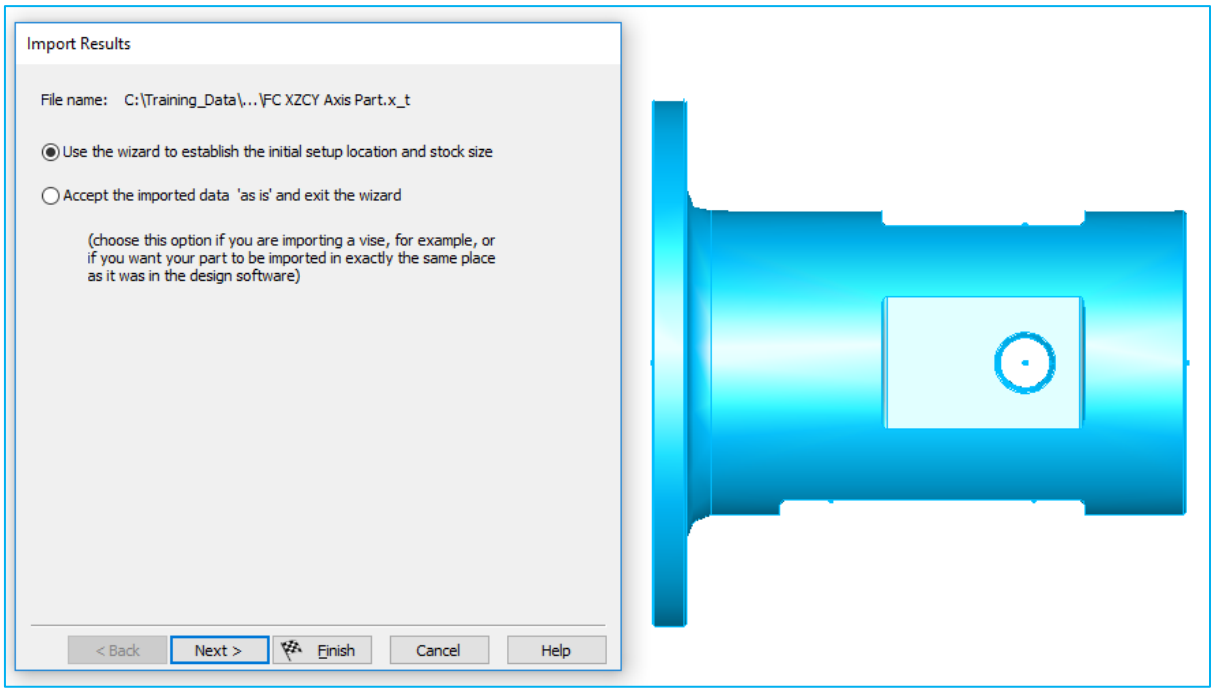


© Copyright 2016 Autodesk, Inc. All rights reserved



FC XZCY Axis Part

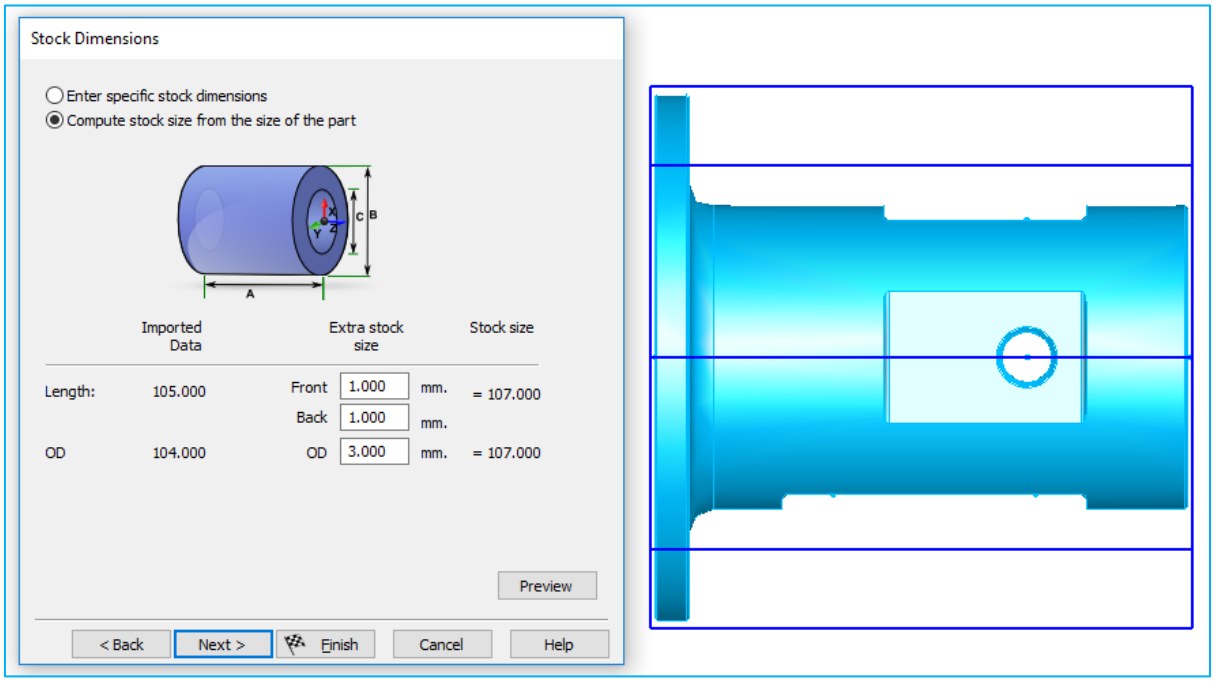
1 Import file FC XZCY axis Part from the Turning Data Folder. **C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**



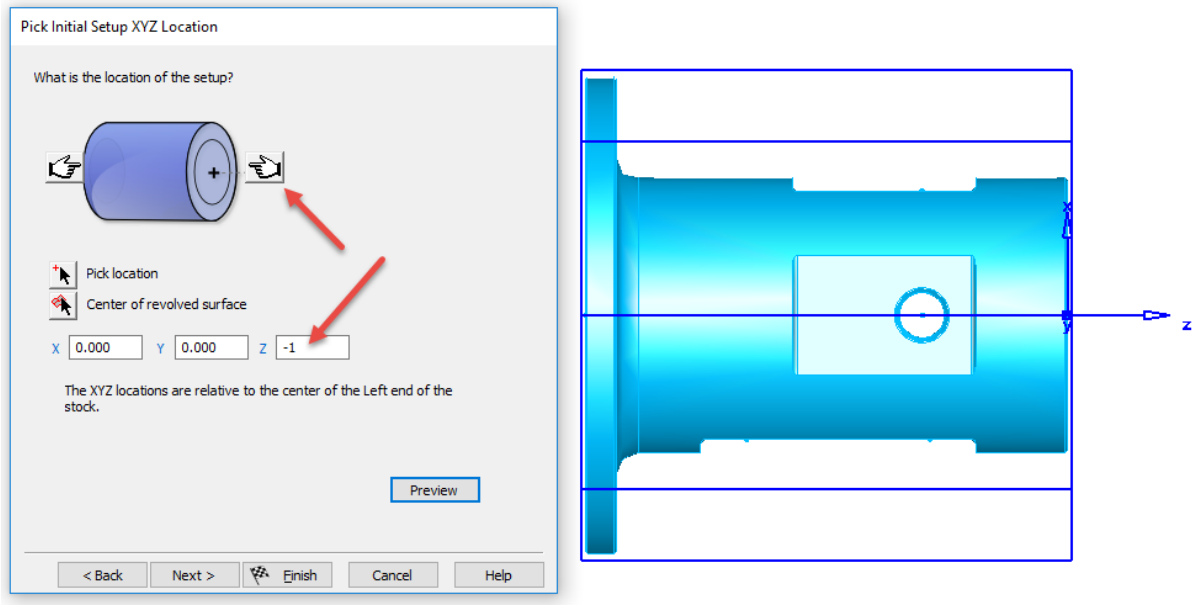
2 Work through the **Wizard**.

3 Select next **5 times** until you get to **Stock dimensions**. Enter the values shown on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved

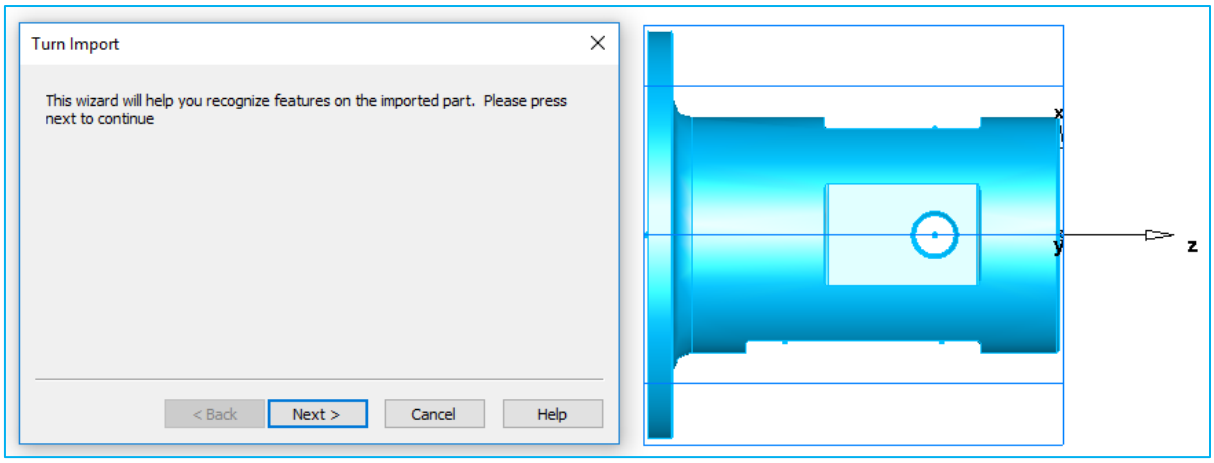


4 Select **Next**. Then select the **Right-hand hand** and enter **-1** into the **Z field**.



5 Select **Finish**. The Wizard will help you recognise features on the imported part.

© Copyright 2016 Autodesk, Inc. All rights reserved



6 Select **Next**.

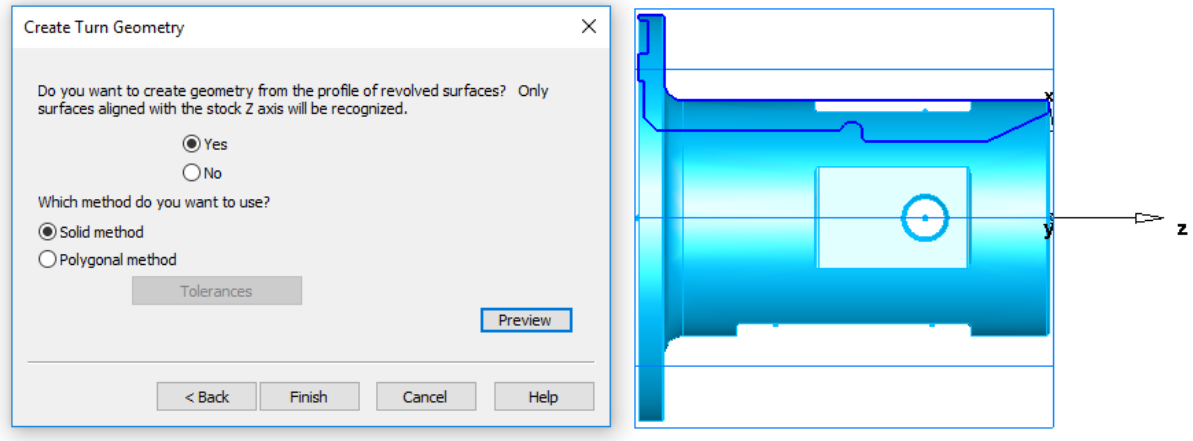
7 Select **No** to **recognise holes**.

8 Select **Next**.

9 Select **No** to **recognise features**.

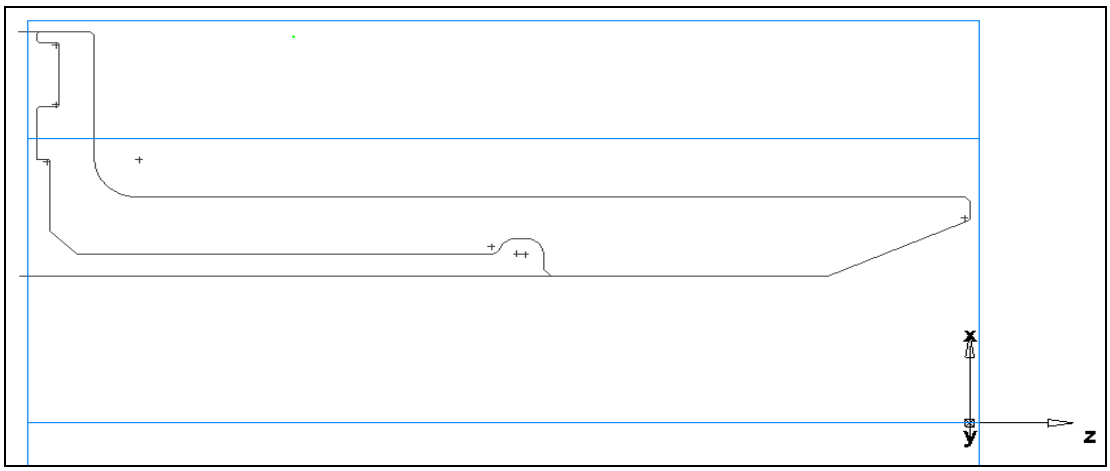
10 Select **Next**.

We do want the profile for turning and boring. We will be using the Solid method for the 2D profile.



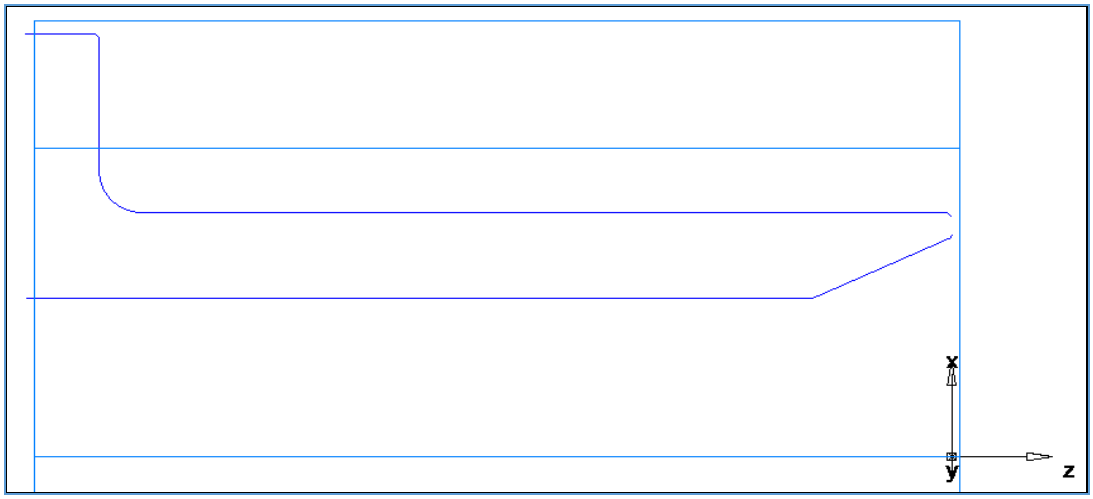
11 Select **Finish.** The **2D profile** has been extracted.

12 Modify the geometry so it looks like this example. Use **Trim/Extend**.



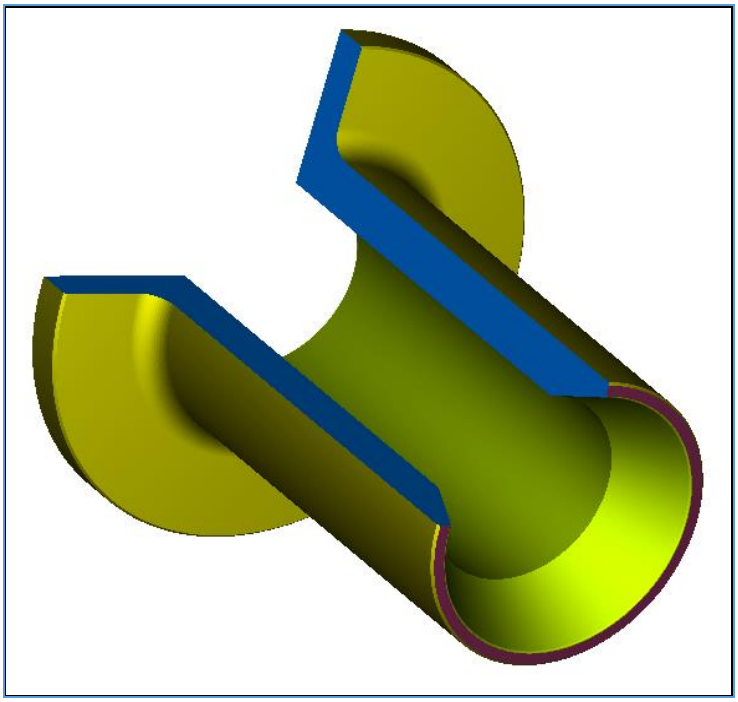
© Copyright 2016 Autodesk, Inc. All rights reserved

*Notice how the geometry goes past the material. Create a **Curve** for the **Bore ID** and **OD** profiles like the ones shown below.*



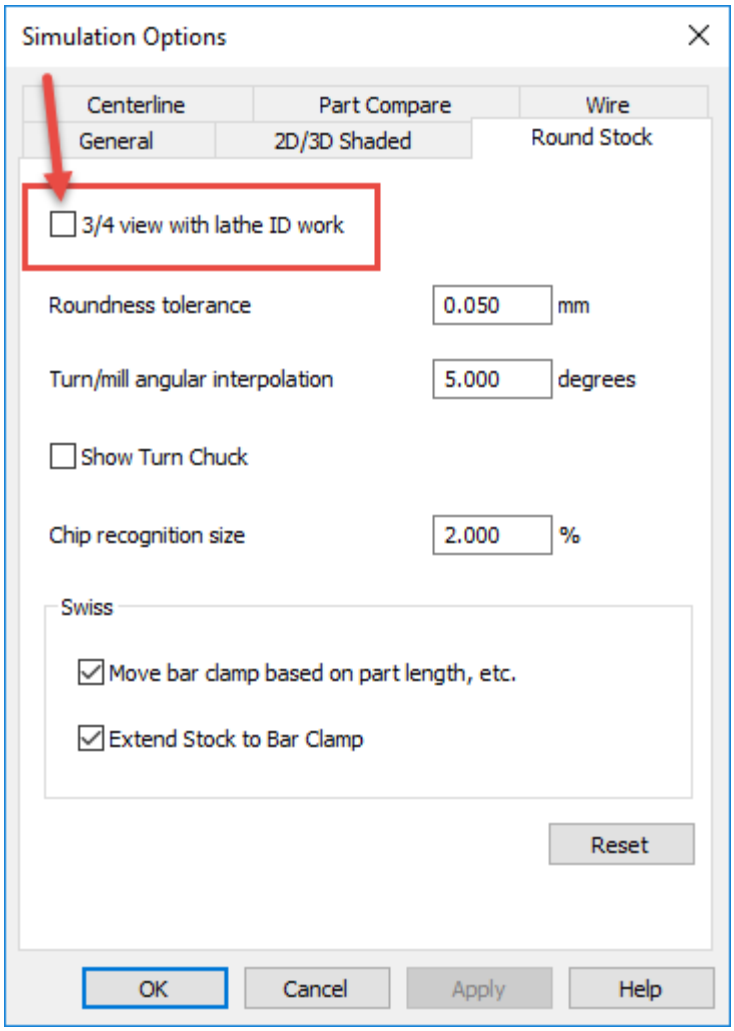
13 Create **Face**, **Turn** and **Bore** features. Use the **Pre-drill** option in **Bore** and use a **30mm u-drill** to rough out the bore. Set the **minimum radius boundary** to **15mm** as a starting point for the Rough boring operation. Also check the shaft diameter of your boring bar. Reduce this value if necessary.

The image below shows all of the above operations completed.

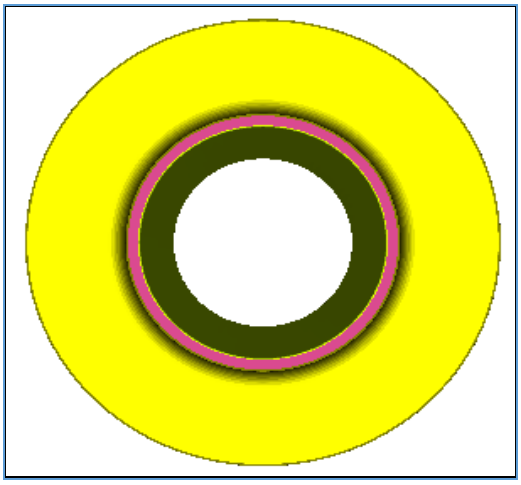


*In order to see the complete simulation instead of a ¾ view go to **Options>Simulation>Round Stock** and **untick ¾ view with lathe ID work**.*

© Copyright 2016 Autodesk, Inc. All rights reserved



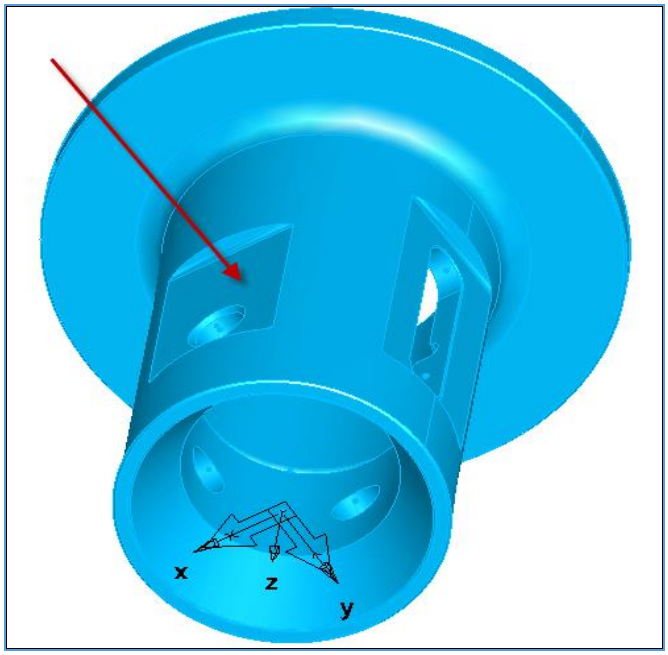
14 Select **Apply** to action the command.



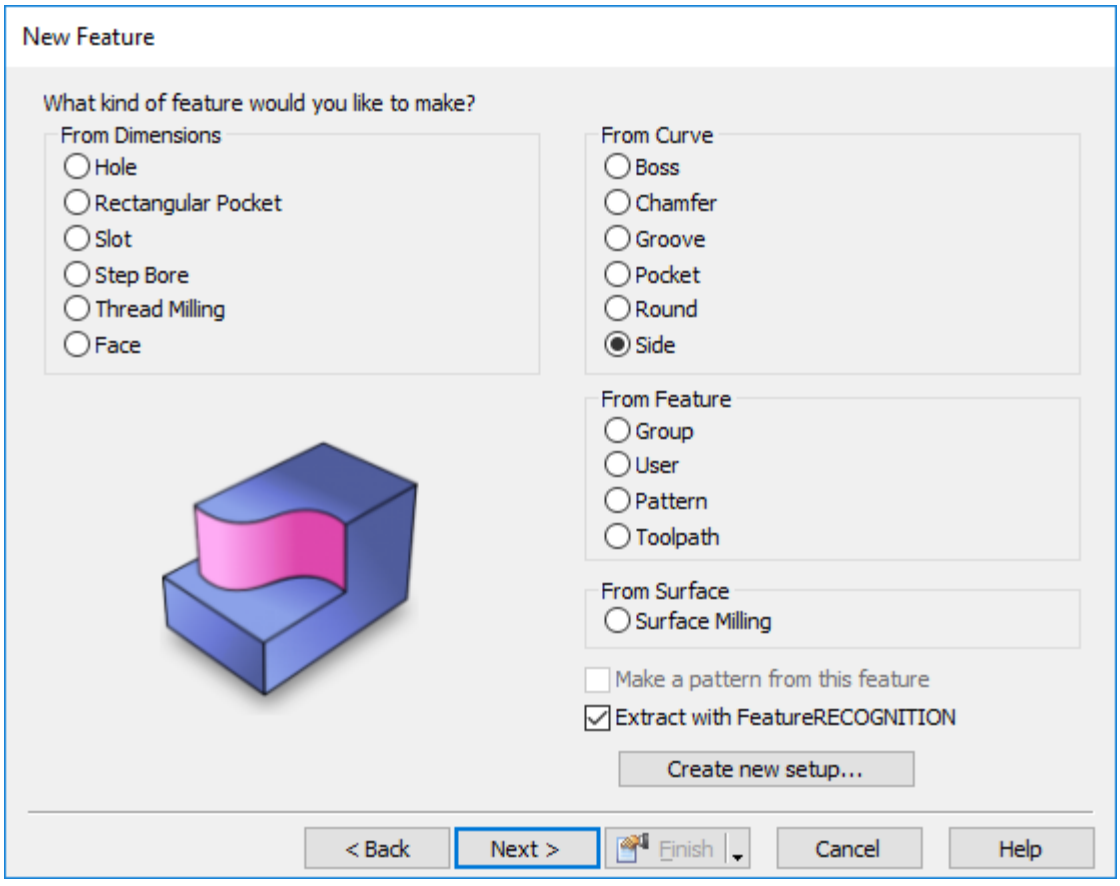
*We will now machine the Flats, slots and holes around the index axis. **Eject** the **3D Simulation**. In **View** select **Show all Solids**.*

*Rotate the solid model in the viewing area so that the flat shown can be easily extracted using **Interactive Feature recognition**. The reason for this is so that we can create a Pattern of the Side Feature.*

© Copyright 2016 Autodesk, Inc. All rights reserved



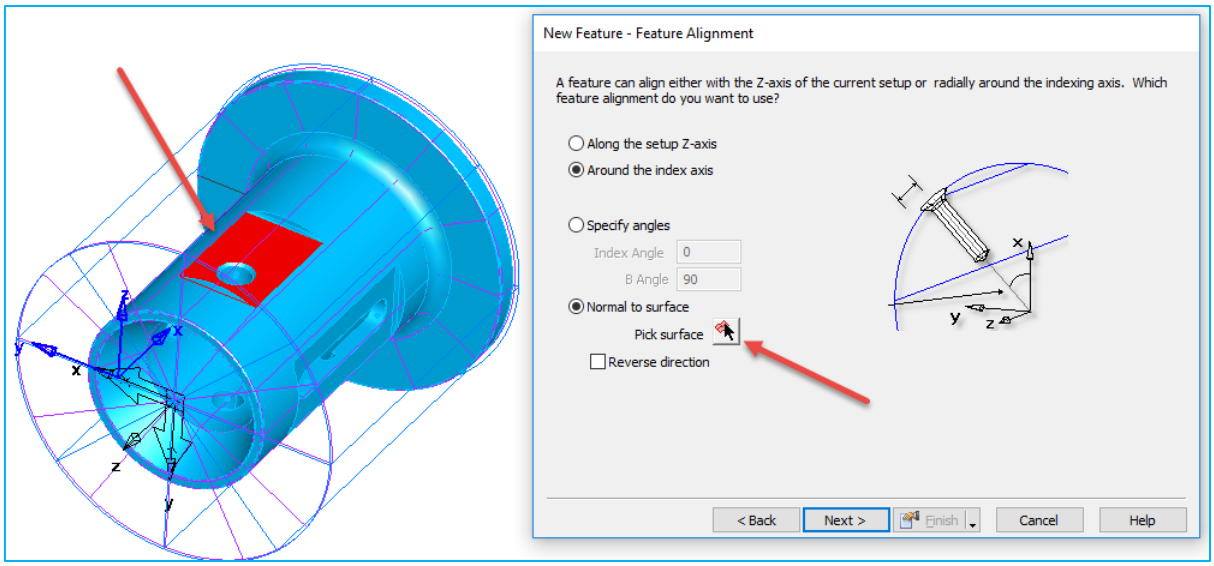
15 Please create a new **Turn/Mill** Side Feature using Feature Recognition. Select **Ctrl+R** to create a new feature. Select **Turn/Mill**. Select **Next**. Please select **Side** and **Extract with Feature Recognition**.



16 Select Next. Please select **around the index axis** and **Normal to surface.**

17 Select the **Pick surface** icon.

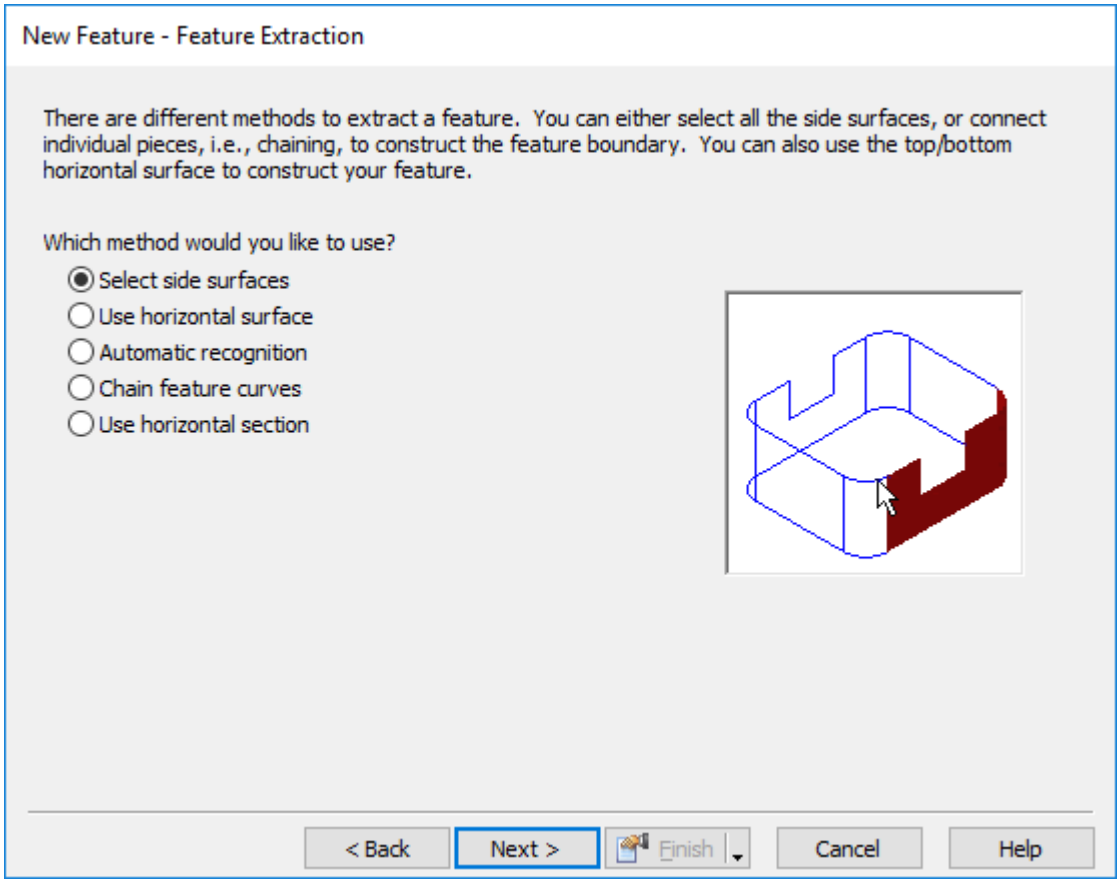
© Copyright 2016 Autodesk, Inc. All rights reserved



Then select the following surface.

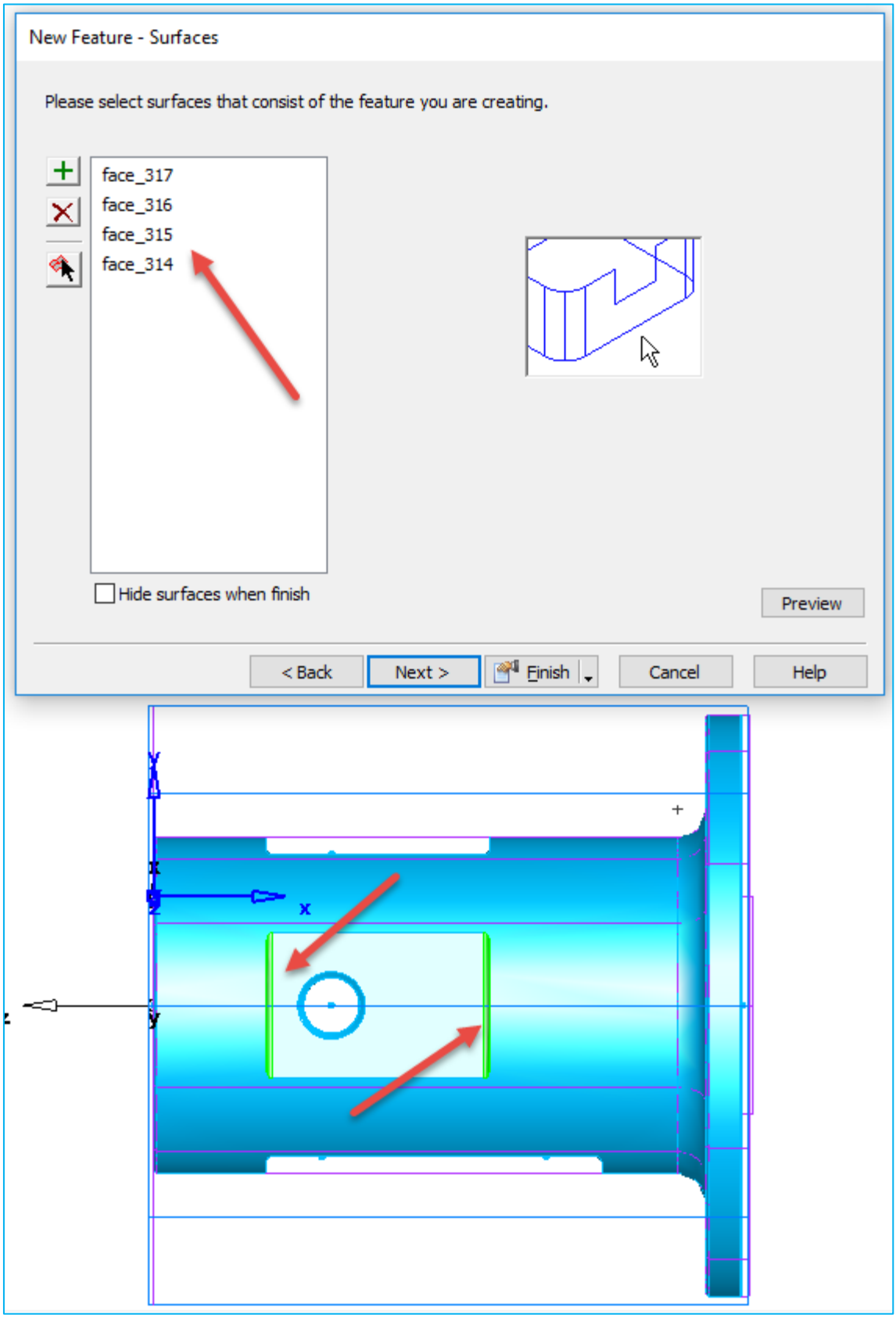
18 Select **Next**. Please select the viewing area to **de-select** the selected surface.

19 We are going to select **side surfaces** so leave it on this **option**.



20 Select **Next** and then select the following four surfaces.

© Copyright 2016 Autodesk, Inc. All rights reserved



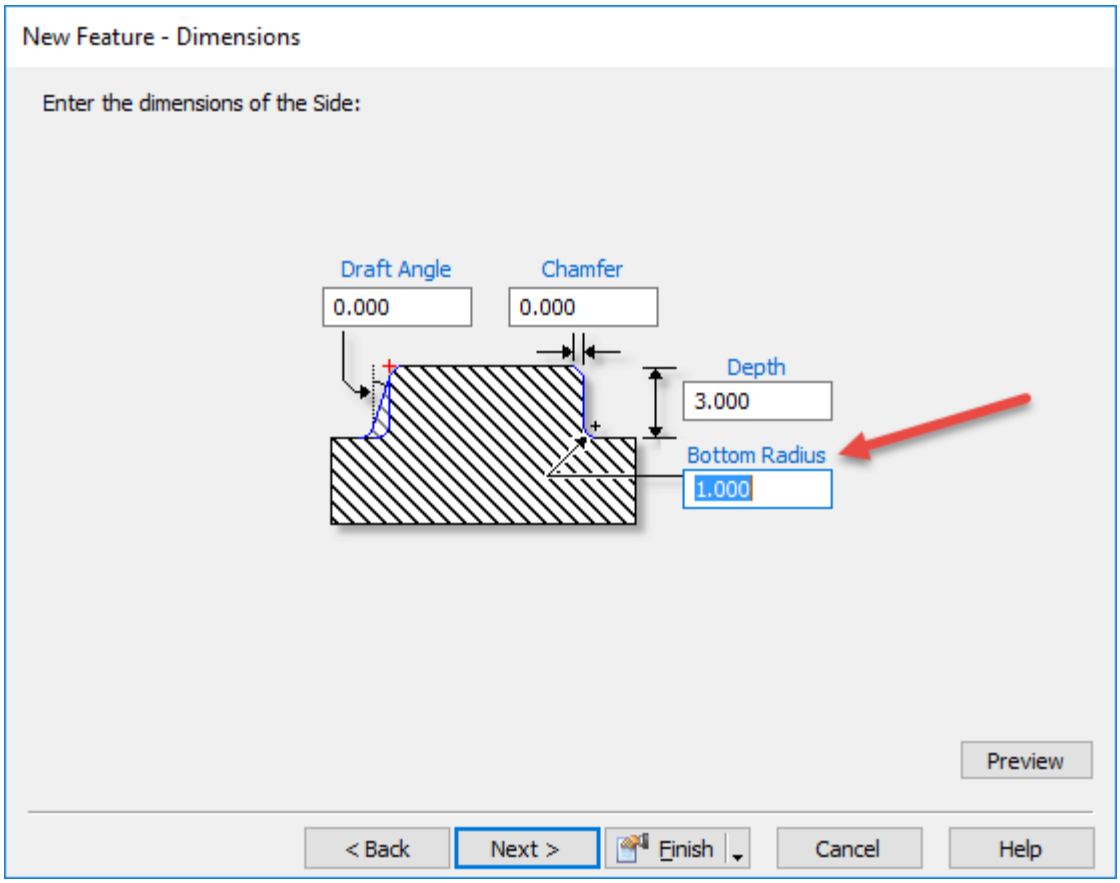
21 Select **Next** and then we will be given the option to set the direction of the two side features. Accept the direction for both curves.

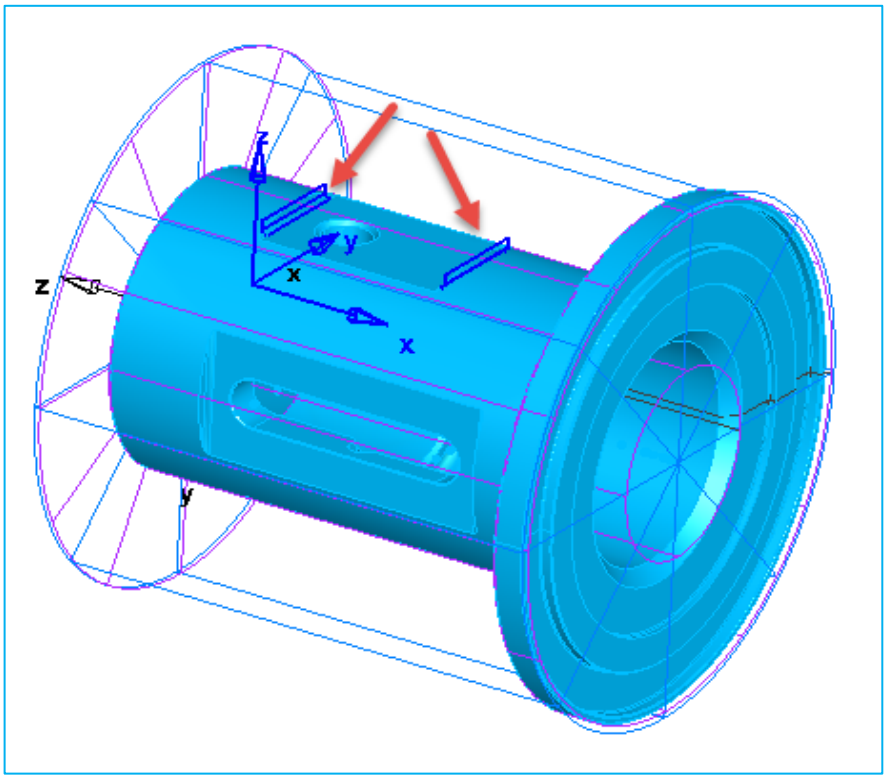
22 Select **Next**. Accept the **Top** and **Bottom** location values.

© Copyright 2016 Autodesk, Inc. All rights reserved

23 Select **Next**.

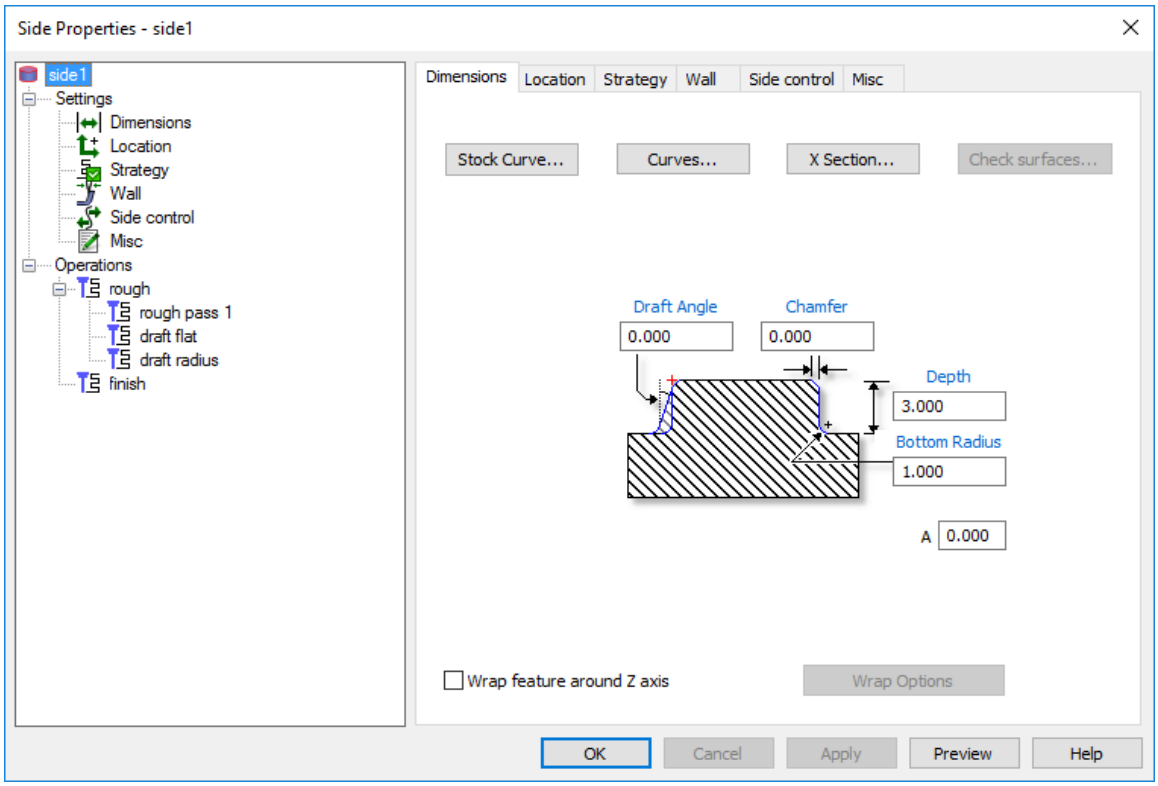
We have a radius in each corner of our feature. In this menu select on the blue Bottom radius Hyperlink and select the radius on the model.



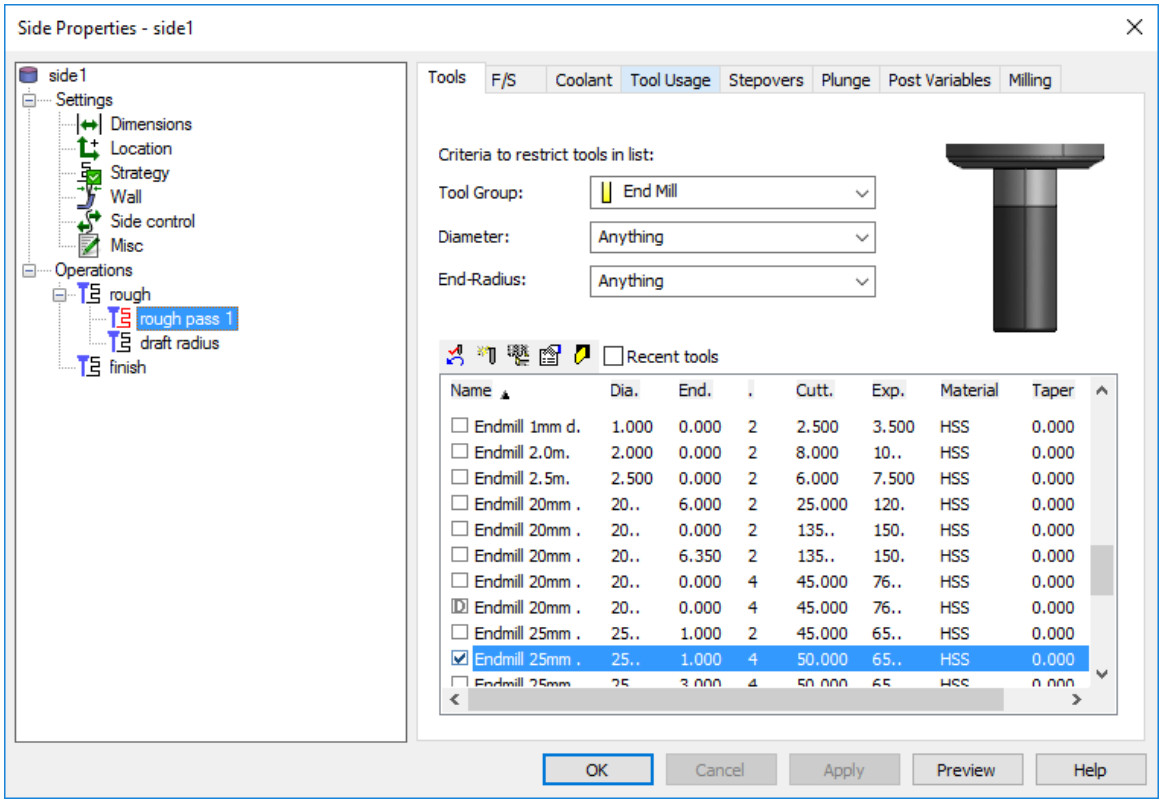


© Copyright 2016 Autodesk, Inc. All rights reserved

24 Select **Finish**.

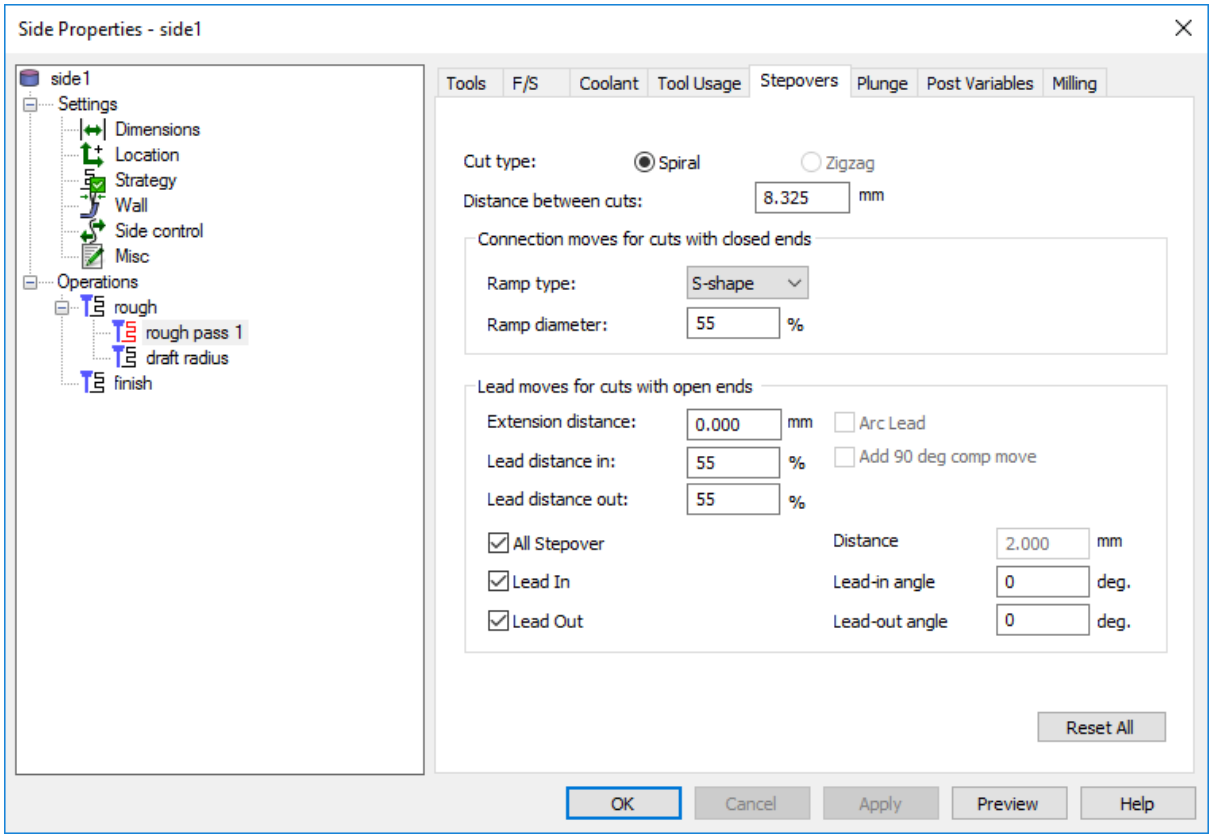


25 **FeatureCAM** has now automatically machined the selected surface. To make the operation more efficient go to the strategy page and change the options as shown. Then change the finishing tool to a tool that is **25mm diameter** and has a **1mm radius**. This will result in having only one tool to complete the machining operation.



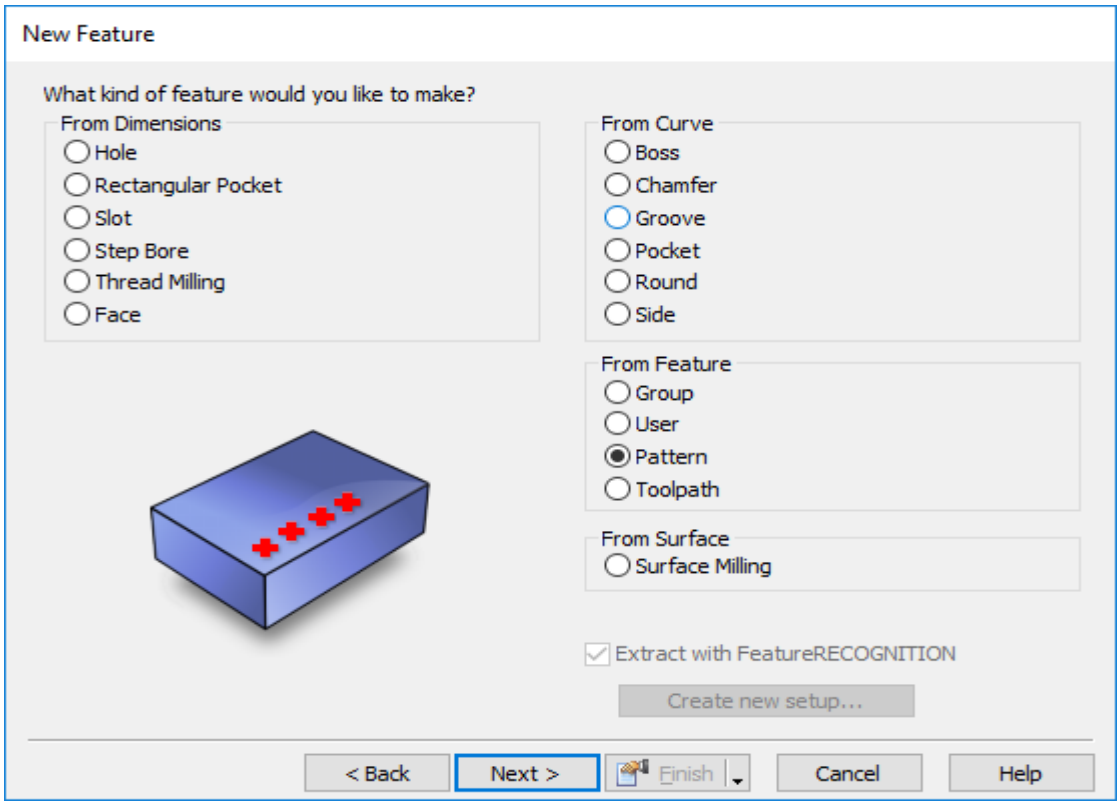
© Copyright 2016 Autodesk, Inc. All rights reserved

26 The only problem is the **Leadin/out** is **not tangential**. Select the **Rough** and **finish** operations and change the **Leadin/out** in the Stepovers tab. Change the **Leadin/out** angle to **zero**. Like the example shown below.



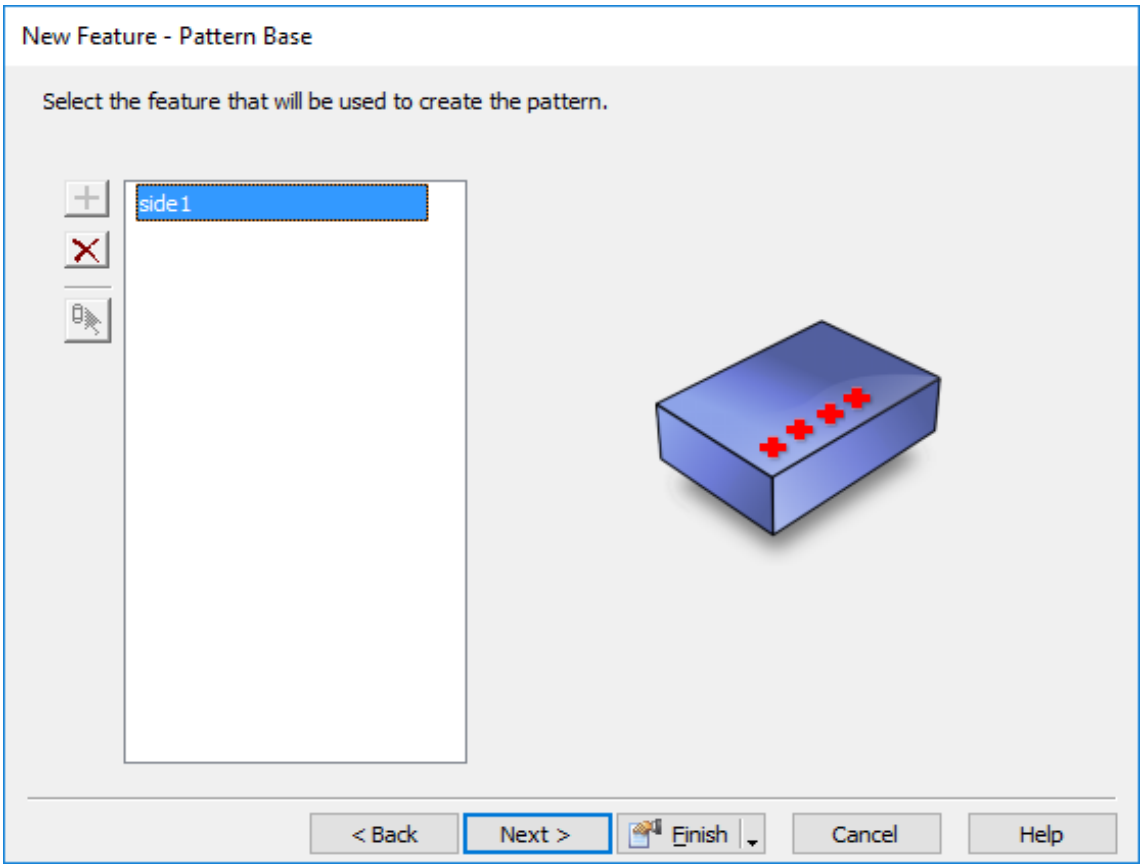
27 We will now create a **Pattern** to machine the other two similar features.

28 Create a new Feature **Ctrl+R>Turn/Mill>Pattern**.

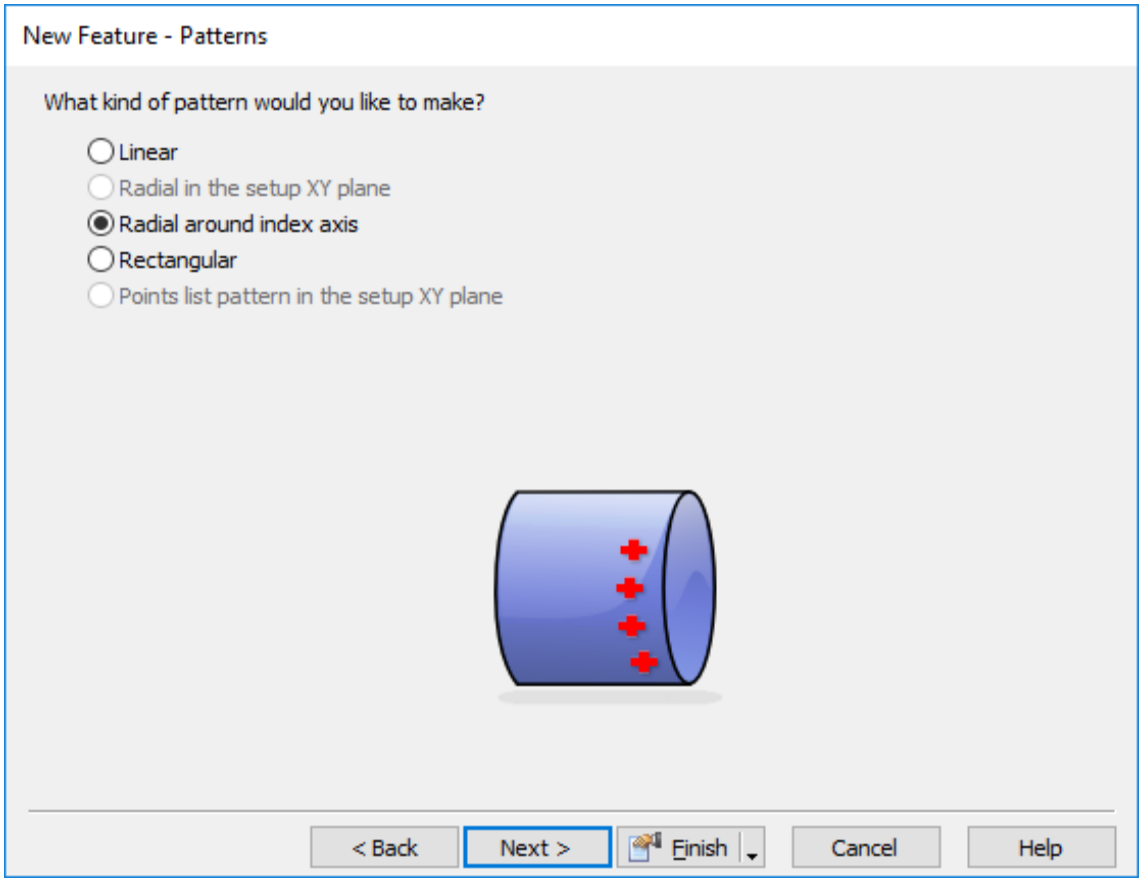


© Copyright 2016 Autodesk, Inc. All rights reserved

29 Select **Next** and then select **Side** from **PartView**.

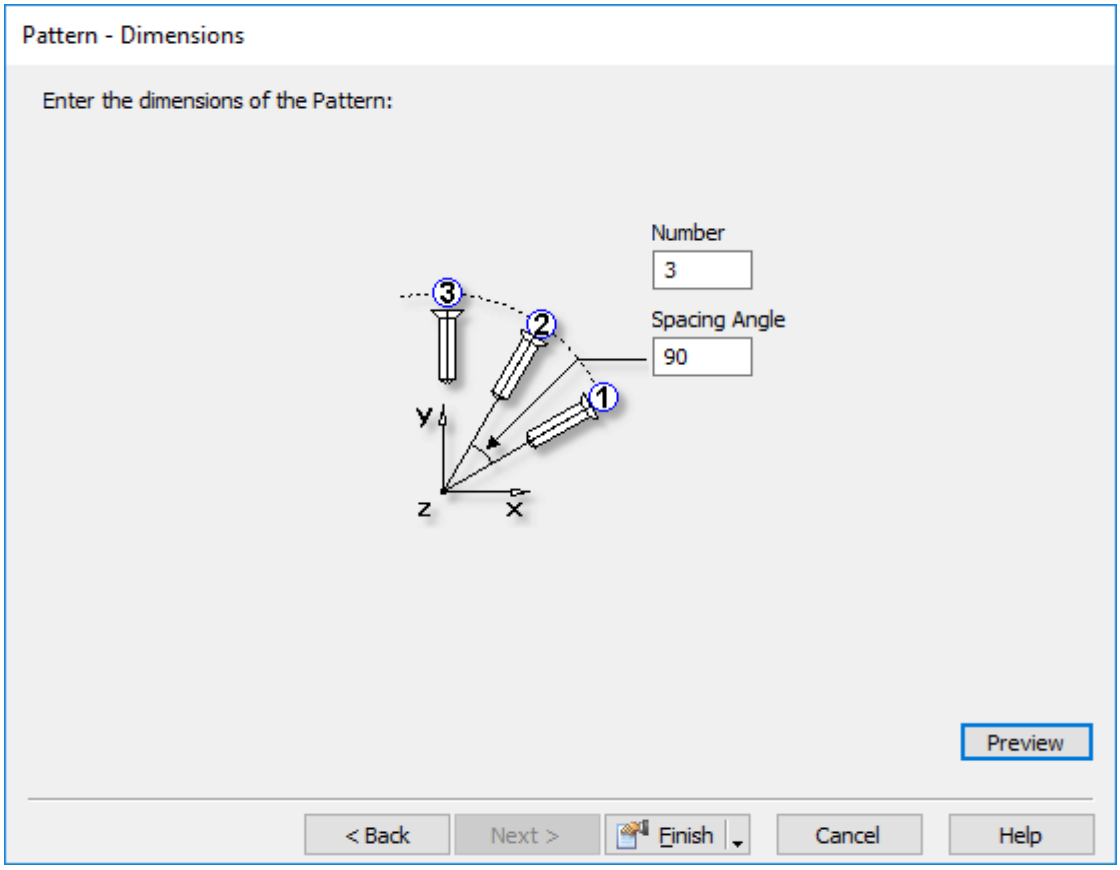


30 Select Next. Then select **Radial around index axis**.

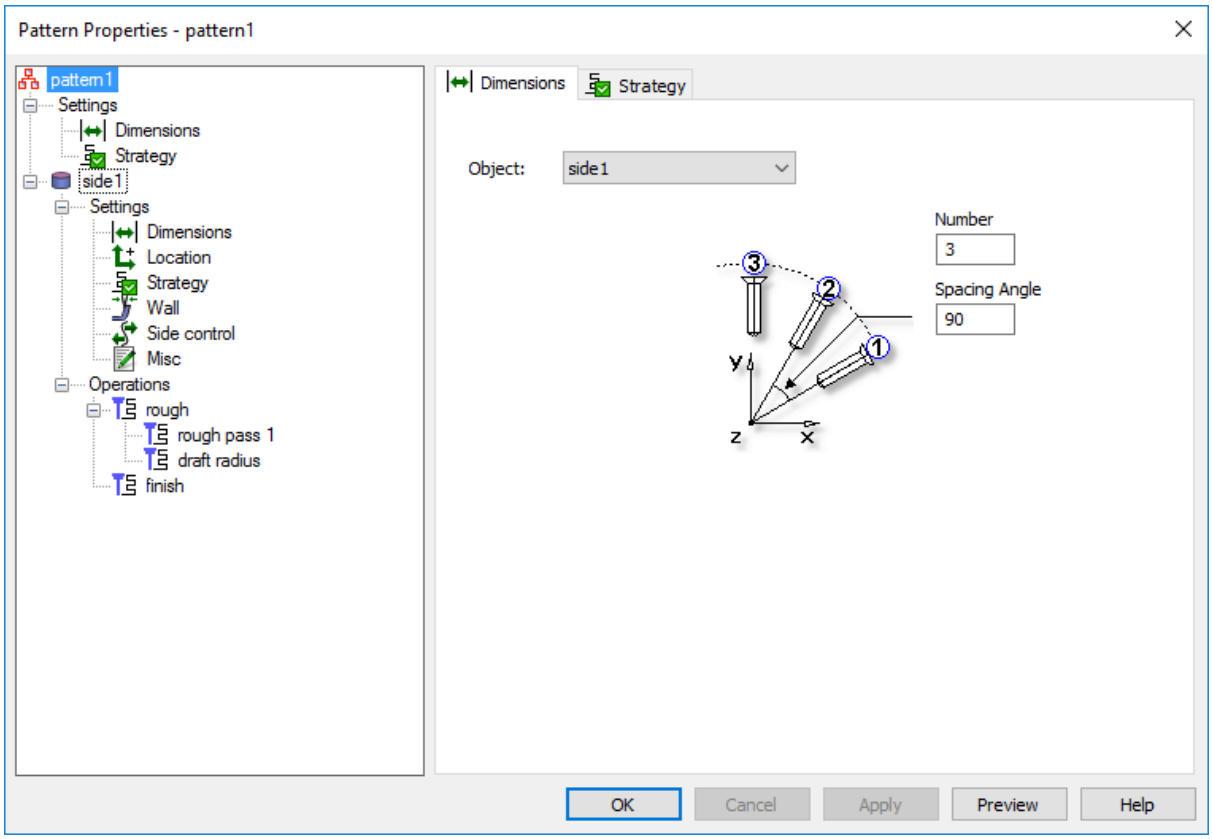


© Copyright 2016 Autodesk, Inc. All rights reserved

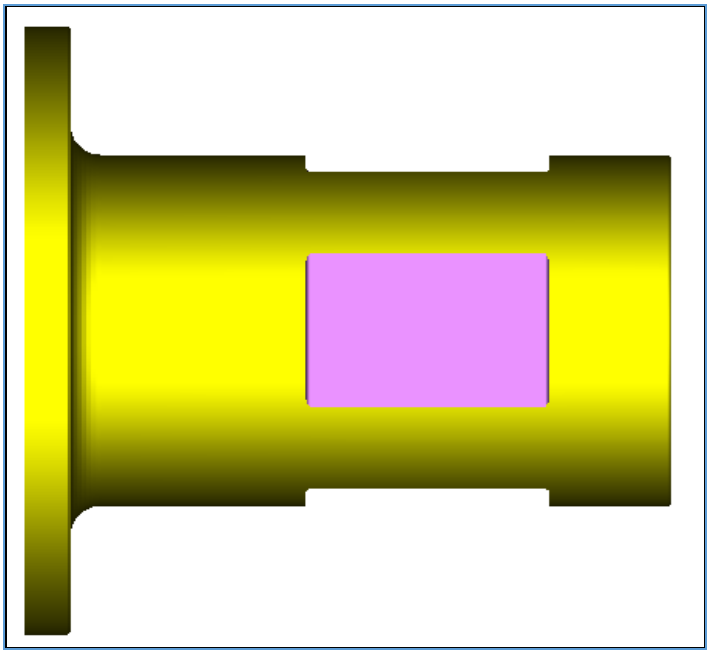
31 Select **Next**. **Number = 3 and Spacing angle = 90** select **Finish**.



Please note the number of copies includes the original.

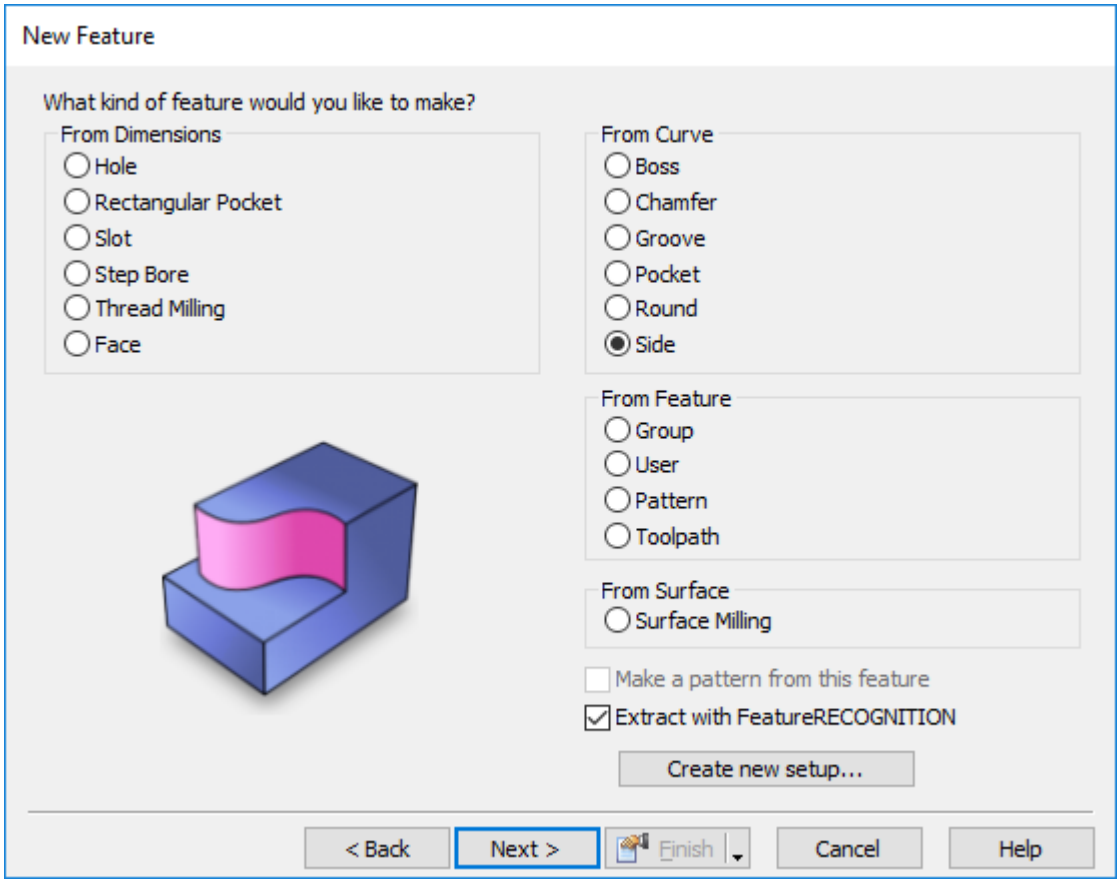


© Copyright 2016 Autodesk, Inc. All rights reserved



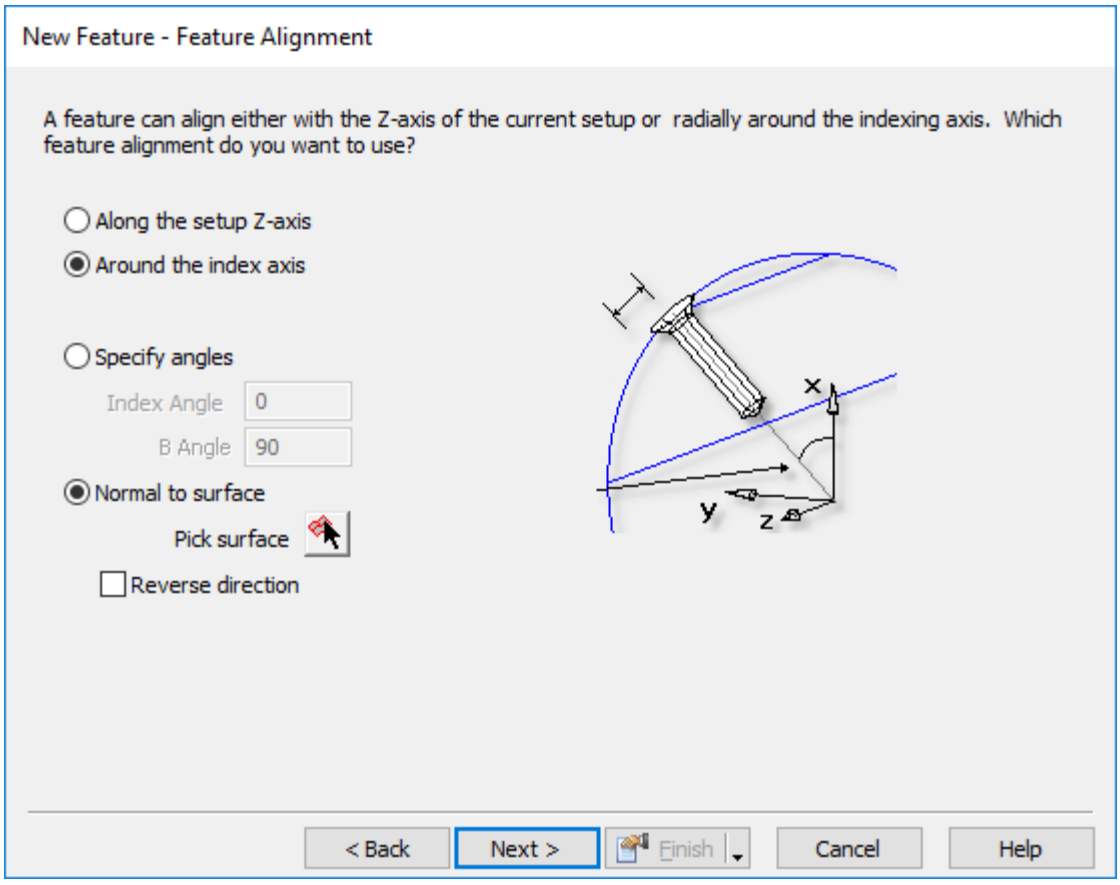
Adopt the same techniques to machine the large single flat face.
***Eject** the simulation.*

32 Create a new feature using your preferred method. **Ctrl+R**. Select **Turn/Mill** and create a **Side** feature using extract with **Feature Recognition**.

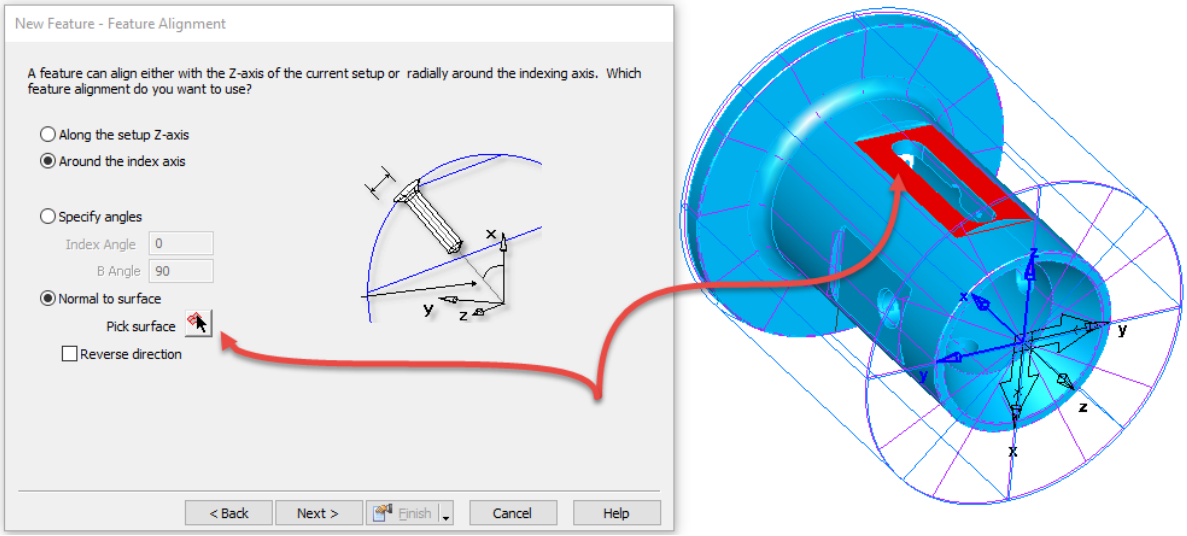


33 Select **Next**. We need to select **Around the Index axis** and **Normal to Surface**.

© Copyright 2016 Autodesk, Inc. All rights reserved

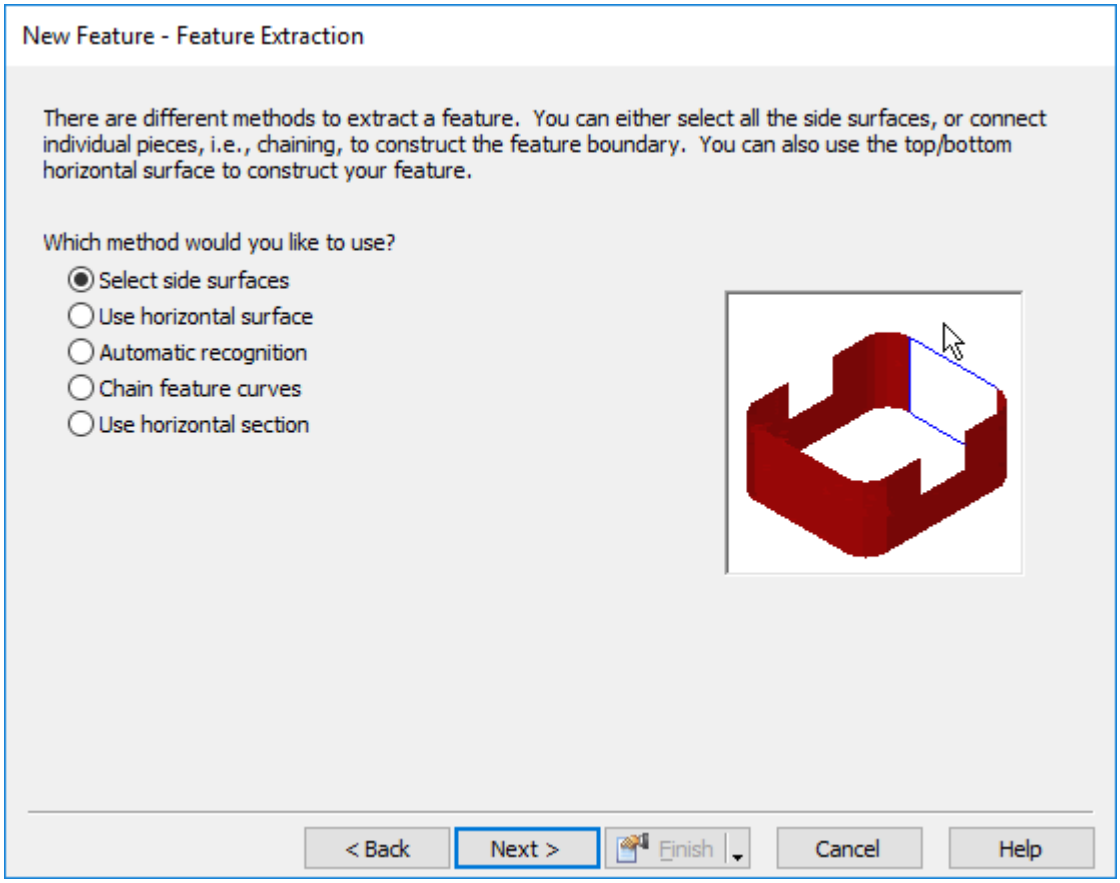


34 Use the **Pick Surface** Icon and select the surface shown.



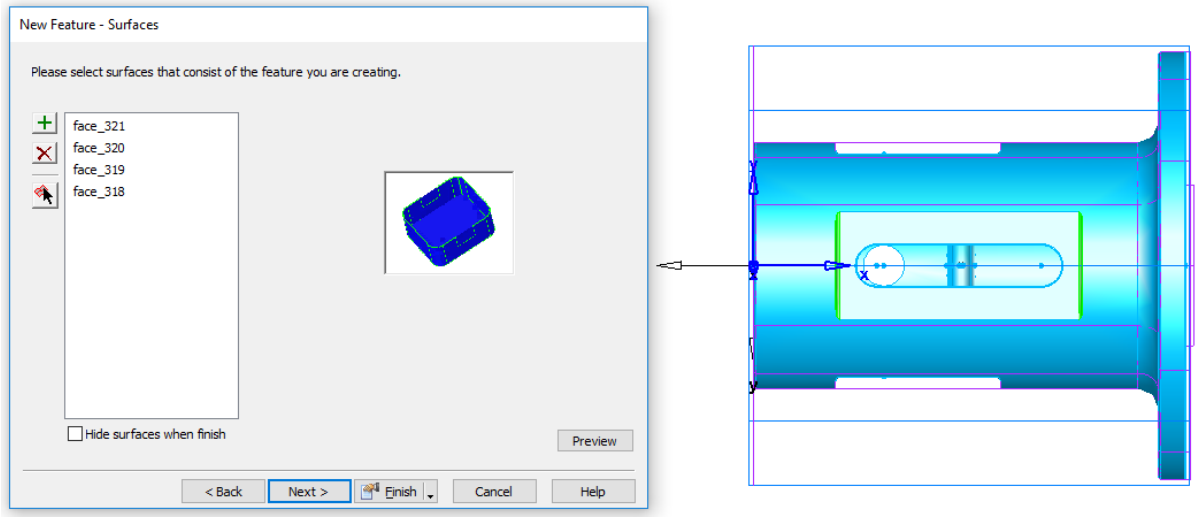
35 Select **Next**. Please select **Select Side Surfaces**. Then select **Next.**

© Copyright 2016 Autodesk, Inc. All rights reserved



36 Please select anywhere on the screen to deselect the surface you have used to align the surface.

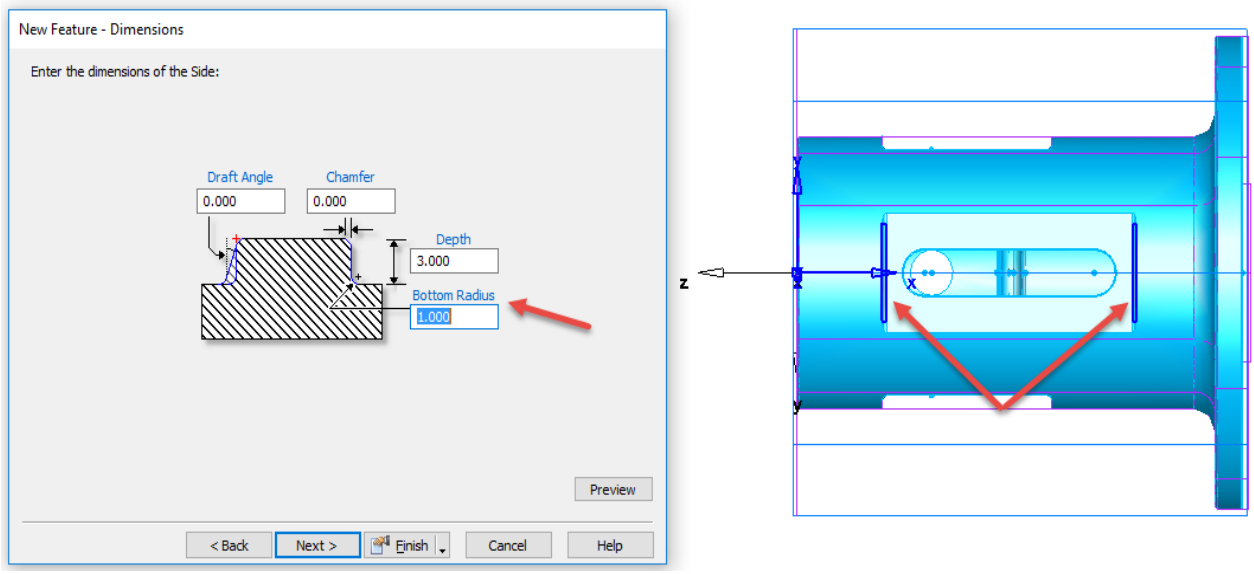
37 Please select the **four faces** shown on the next page.



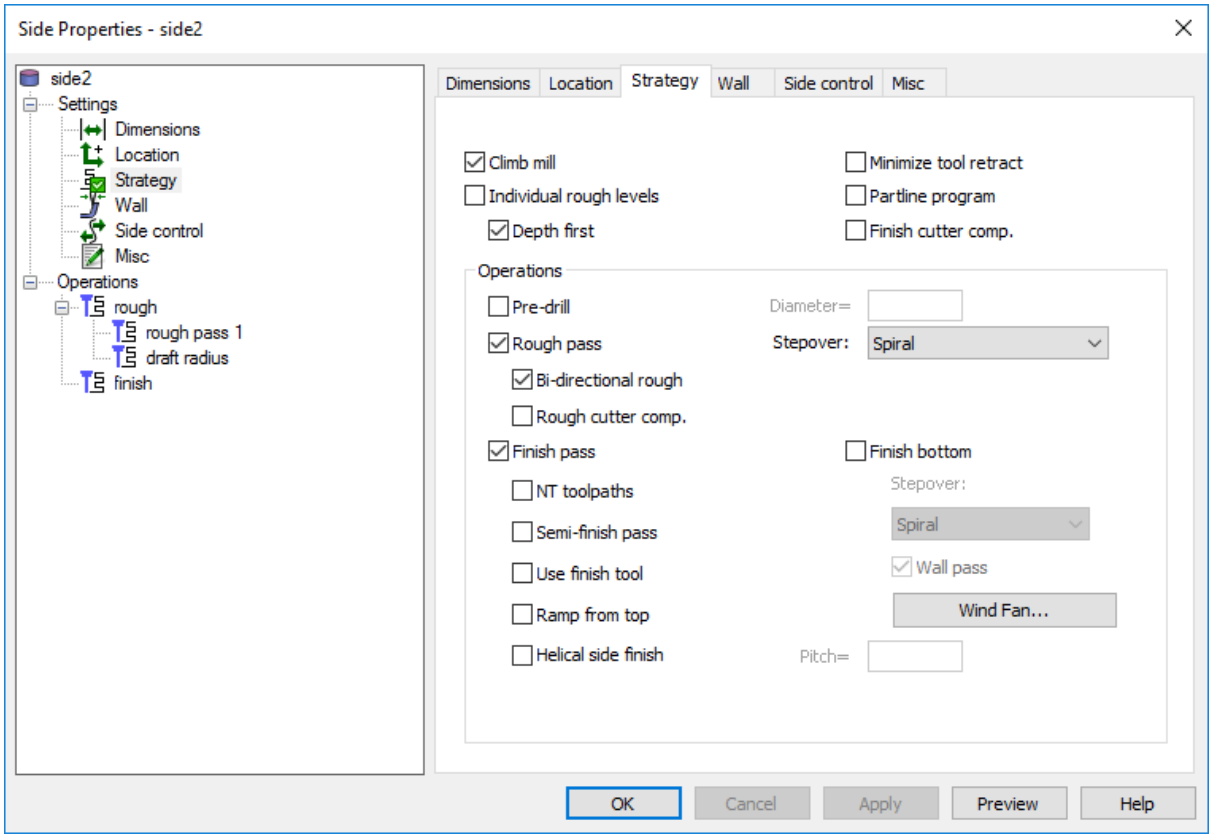
38 Select **Next**. Make sure the machining side is pointing in the correct direction.

39 Select **Next** twice. Then select the Bottom Radius hyperlink and pick the radius on the model. This will input the radius into the bottom radius input field.

© Copyright 2016 Autodesk, Inc. All rights reserved

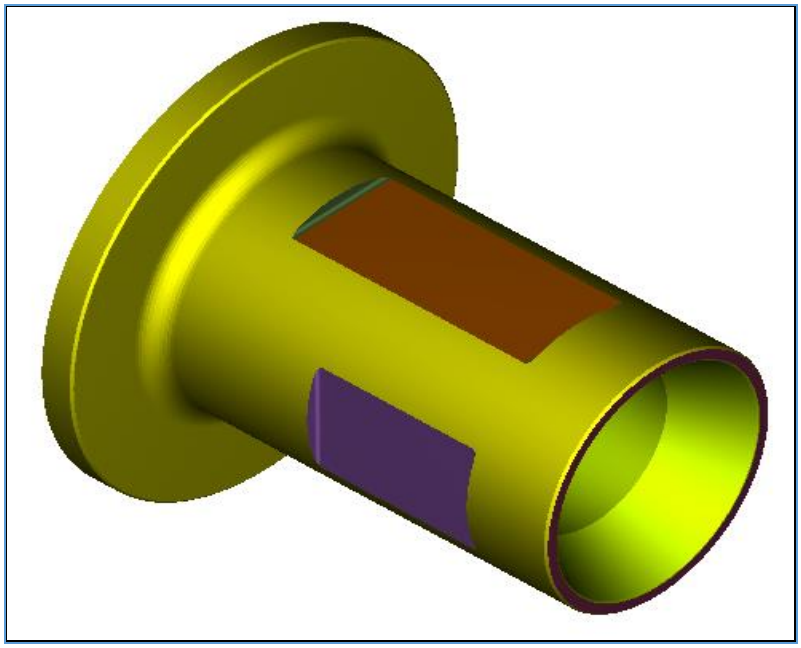


*Select **Finish**. Go into the **strategy** tab and untick **Ramp from top.***



*Remember to change the Leadin/out under the **Finish** tab>**Stepovers**. Change the **extension distance** to **5mm** and **Leadin/out angle** to zero which is tangential. See the finished image below.*

© Copyright 2016 Autodesk, Inc. All rights reserved



40 Create a **Side** Feature for the slot, remember to add the **chamfer**. **Eject the simulation**. The work plane setup will be remembered from the last feature.

41 Create a new Feature using your preferred method. **Ctrl+R**

42 Select **Turn/Mill**.

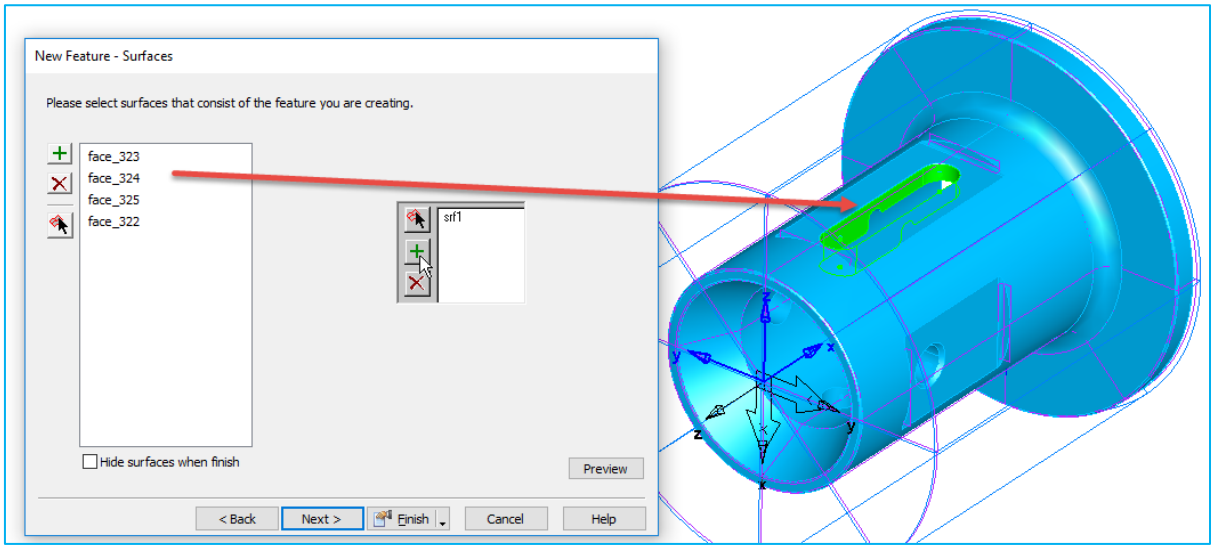
43 Select **Side** with **Feature Recognition**.

44 Select **Next** twice.

45 Select **Side Surfaces.**

46 Select **Next**.

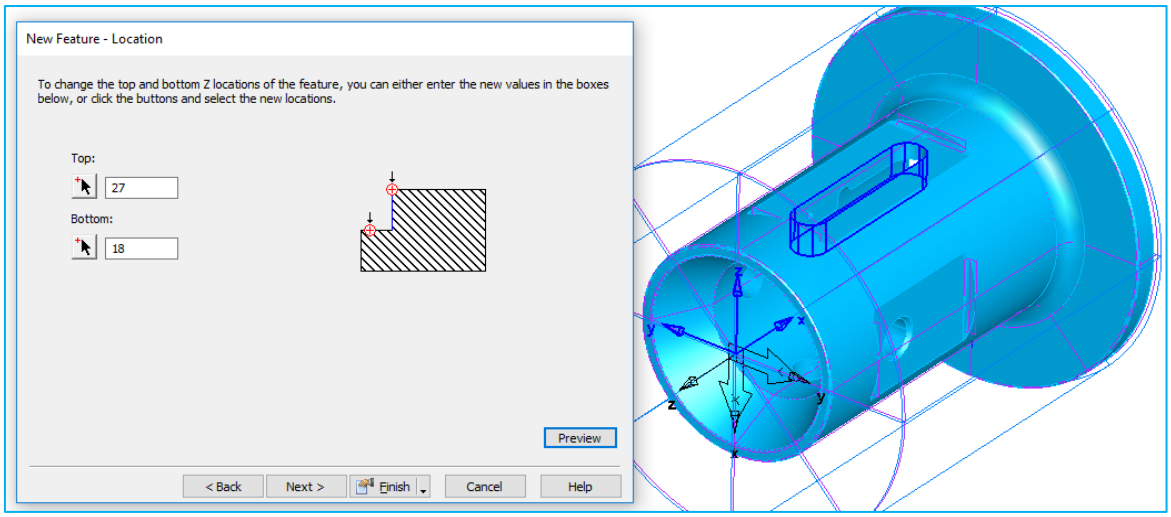
47 Select the **four surfaces** for the **slot** as shown.



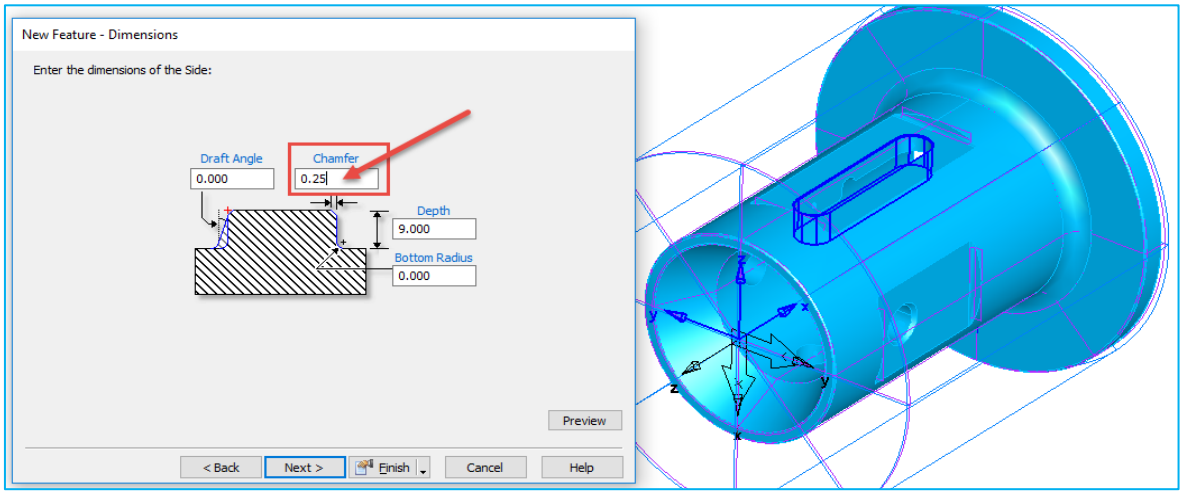
48 Select **Next** and make sure the arrow is pointing inside. Select **Next**.

*The height of the slot is **26.750mm** this is incorrect as we have a **0.25mm chamfer** this should be **27mm** from the **Setup1** position. Change the **Top** value to **27mm** and the **Bottom** to **18mm** as shown in the image on the next page.*

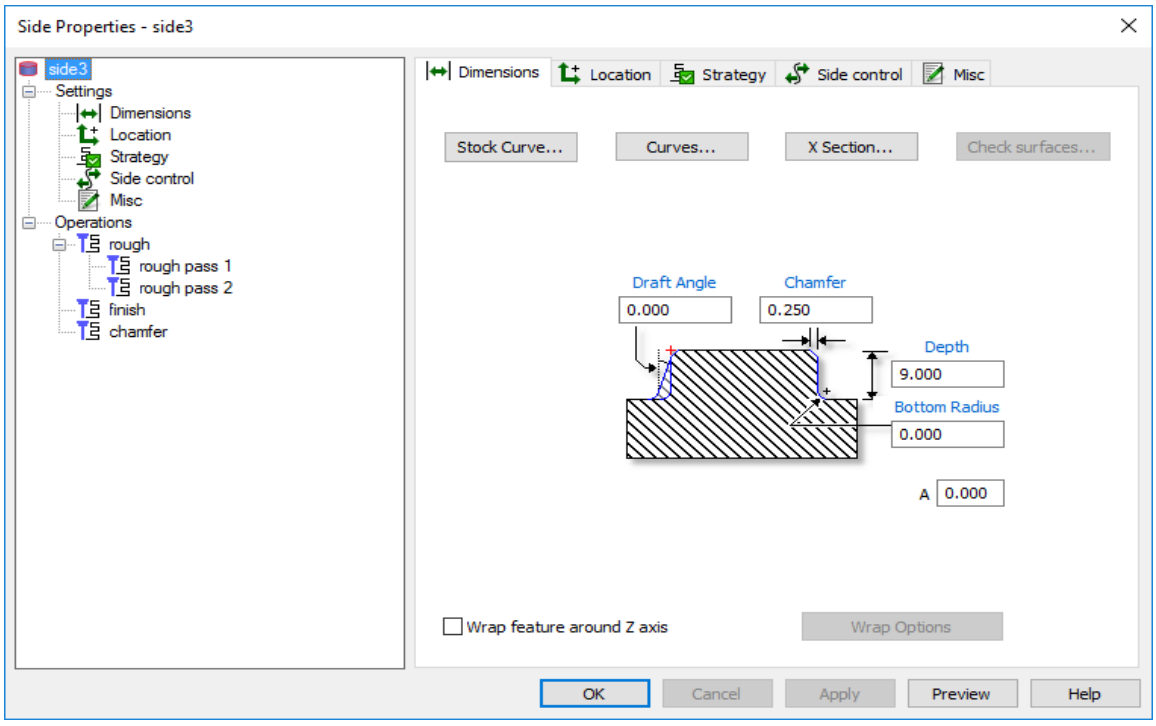
© Copyright 2016 Autodesk, Inc. All rights reserved



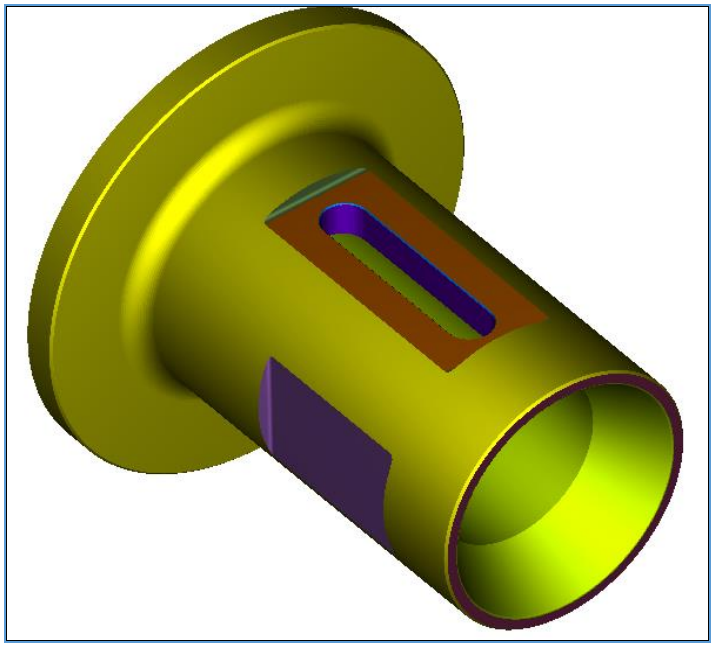
49 Select **Next**. Change the chamfer to **0.25mm**



50 Select **Apply** and **Ok** to close the form. The finished image is on the next page.



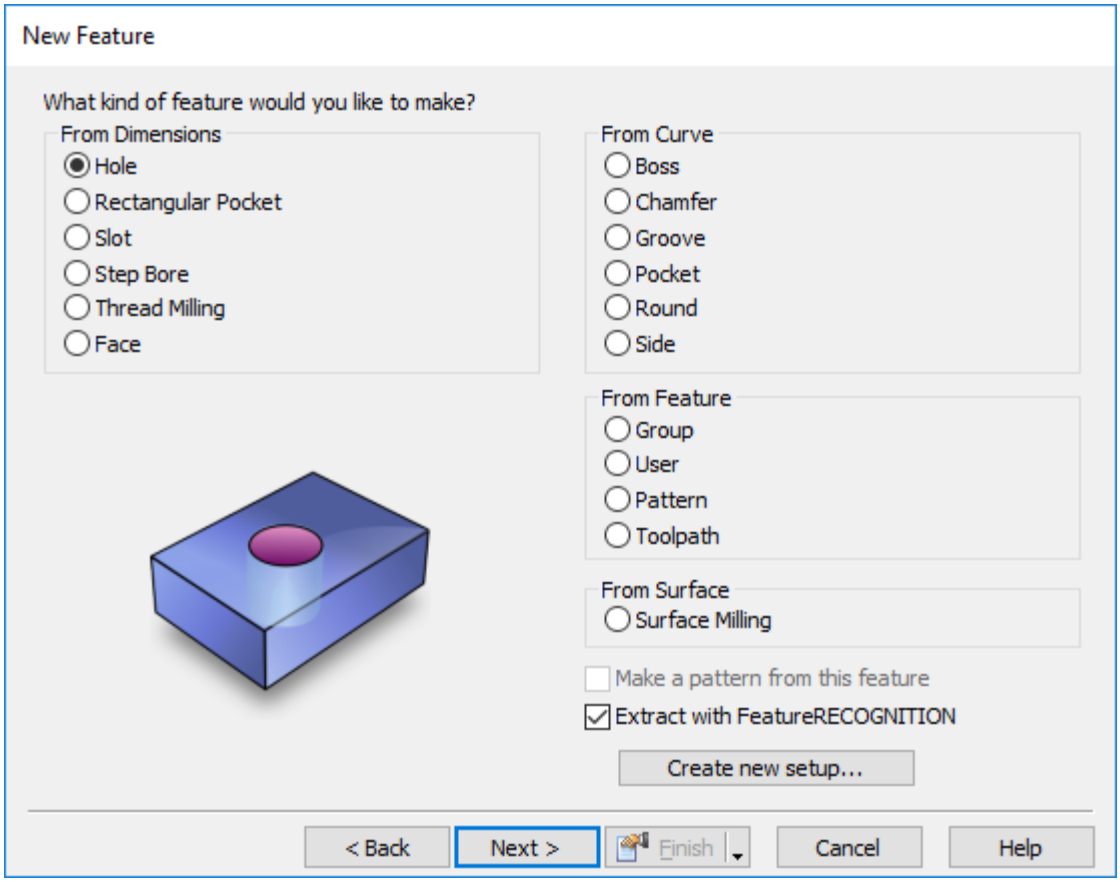
© Copyright 2016 Autodesk, Inc. All rights reserved



51 **Eject the Simulation**.

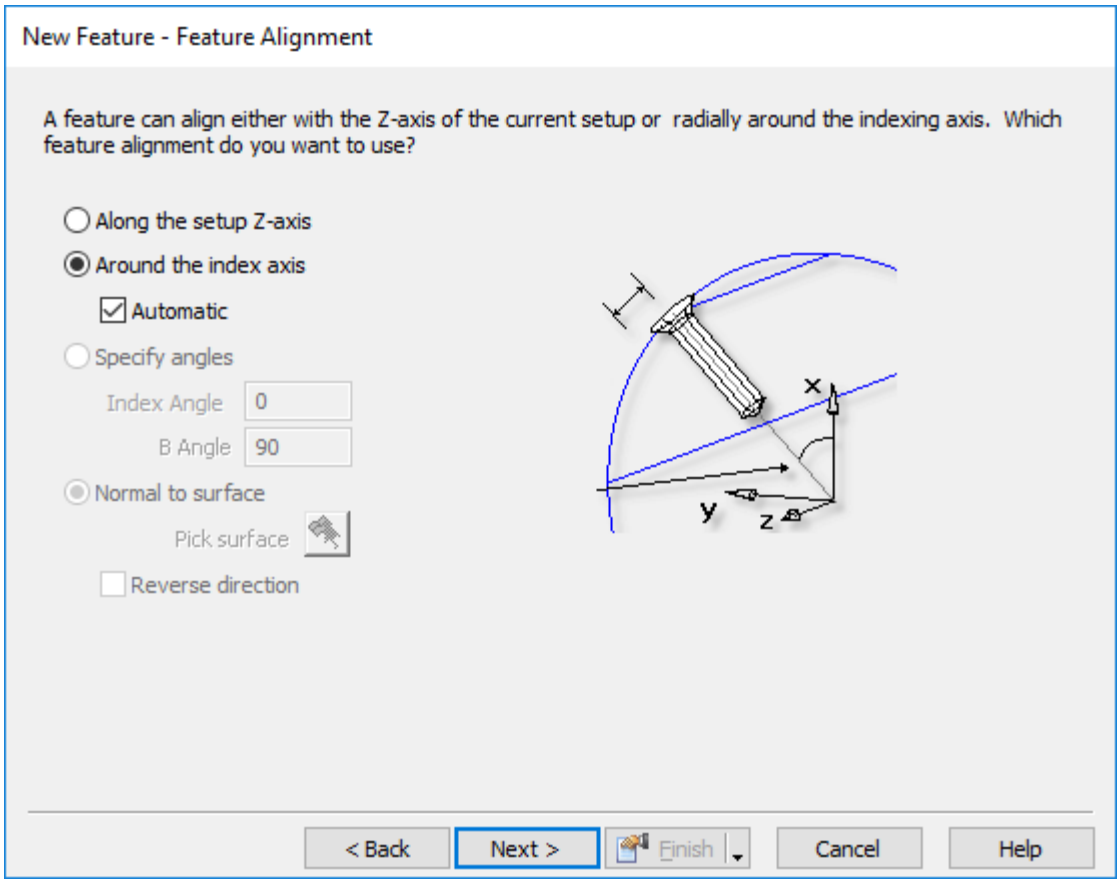
We now need to extract the 3 holes and chamfers around the diameter.

52 Create a new **Hole Feature**. Select **Turn/Mill>Next** then Select **Hole** and **Extract with Feature Recognition**.



53 Select **Next**. Then select **Around the Index Axis and Automatic**.

© Copyright 2016 Autodesk, Inc. All rights reserved

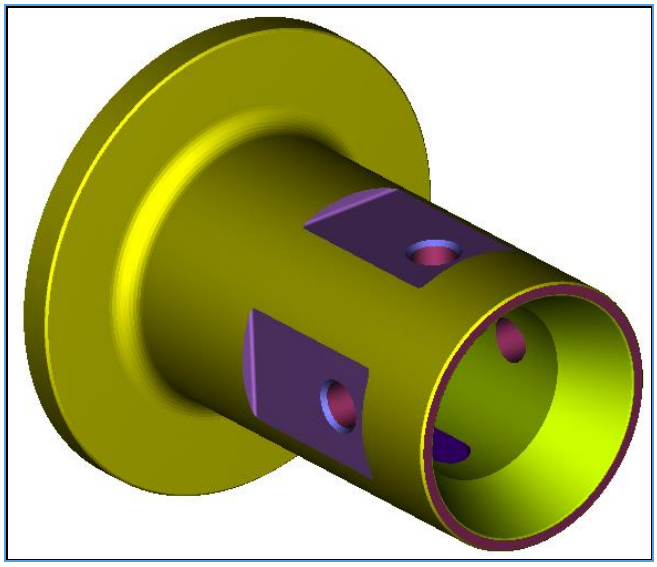


54 Select **Next**.

55 We need to select Recognise and construct multiple holes.

56 Select **Next**. Please Select, **Select All**. Then select **Finish**.

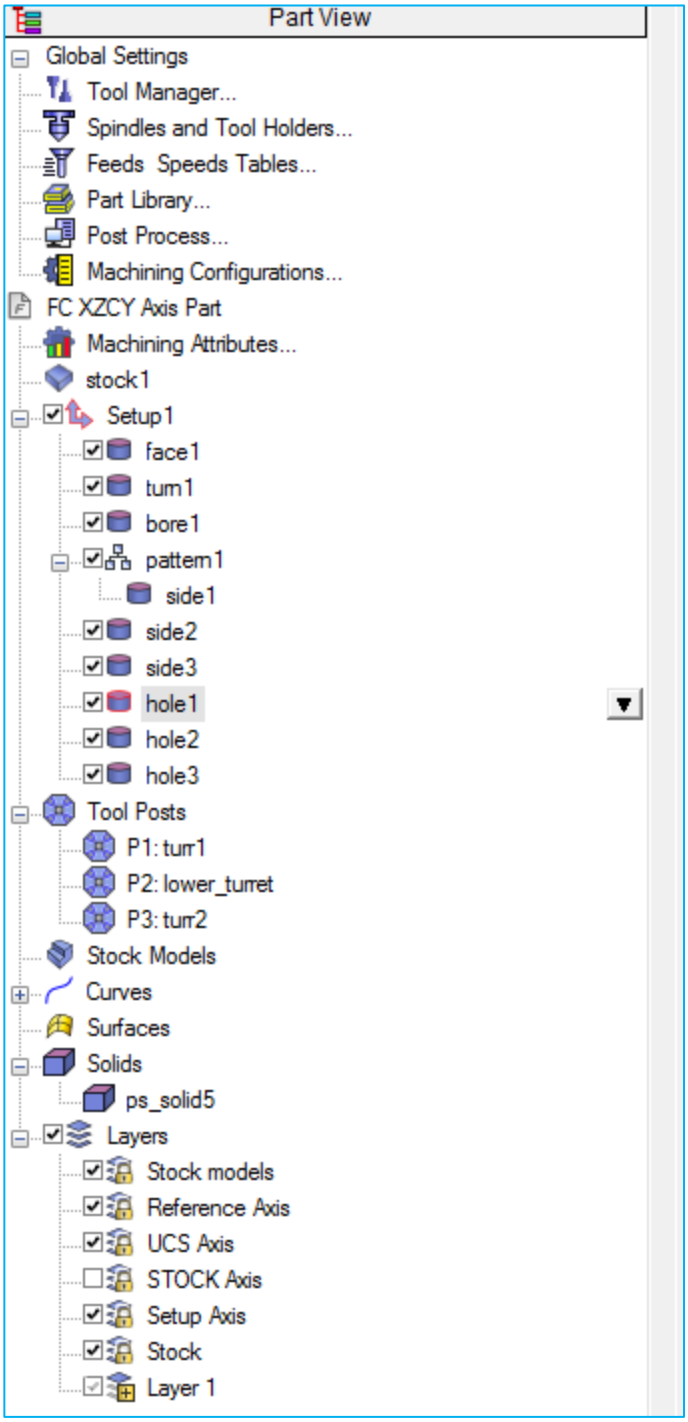
The image below shows the simulation with all holes.



57 Please create **Setup2** and add the **Face**, **Groove**, **Bore**, **thread** and **features**.

*The image above shows the operations completed up to this point. Please add a **Part Transfer** to the **Sub-Spindle** as described in the previous chapter.*

© Copyright 2016 Autodesk, Inc. All rights reserved

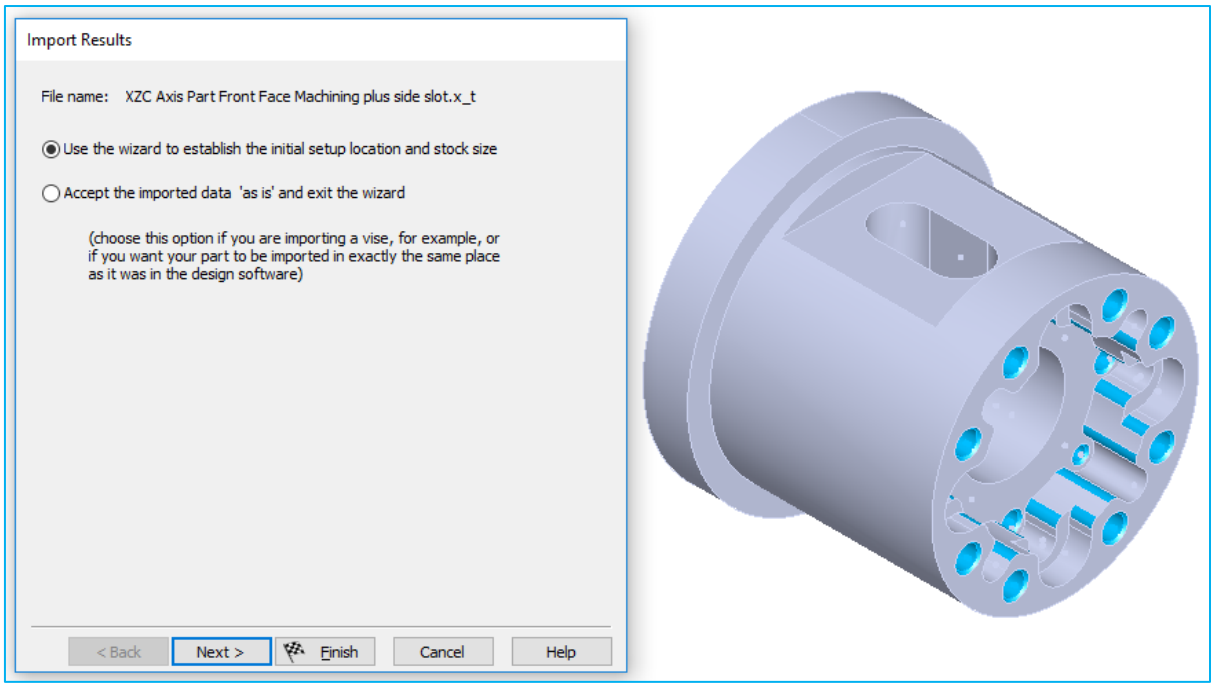


C Axis IFR (Interactive Feature Recognition)

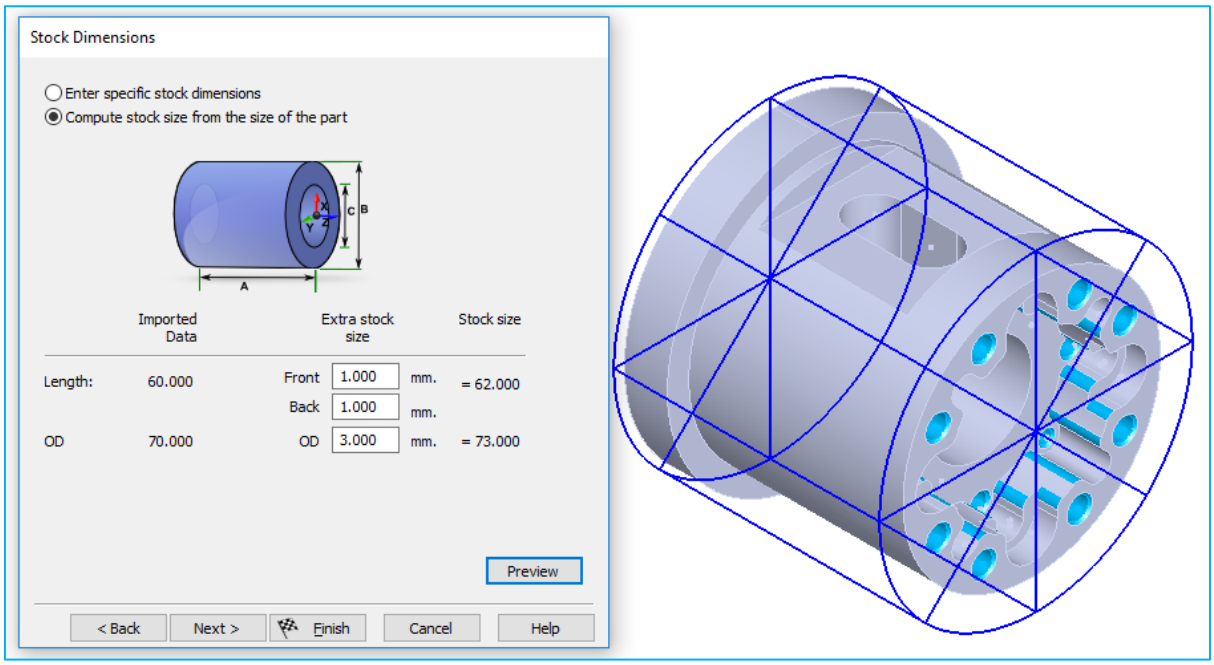
*The next exercise will take you through a fairly basic C Axis Part using interactive Feature Recognition (**IFR**).*

1 Import File **XZC Axis Part Front Face Machining plus side slot.x_t** from **C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**

© Copyright 2016 Autodesk, Inc. All rights reserved

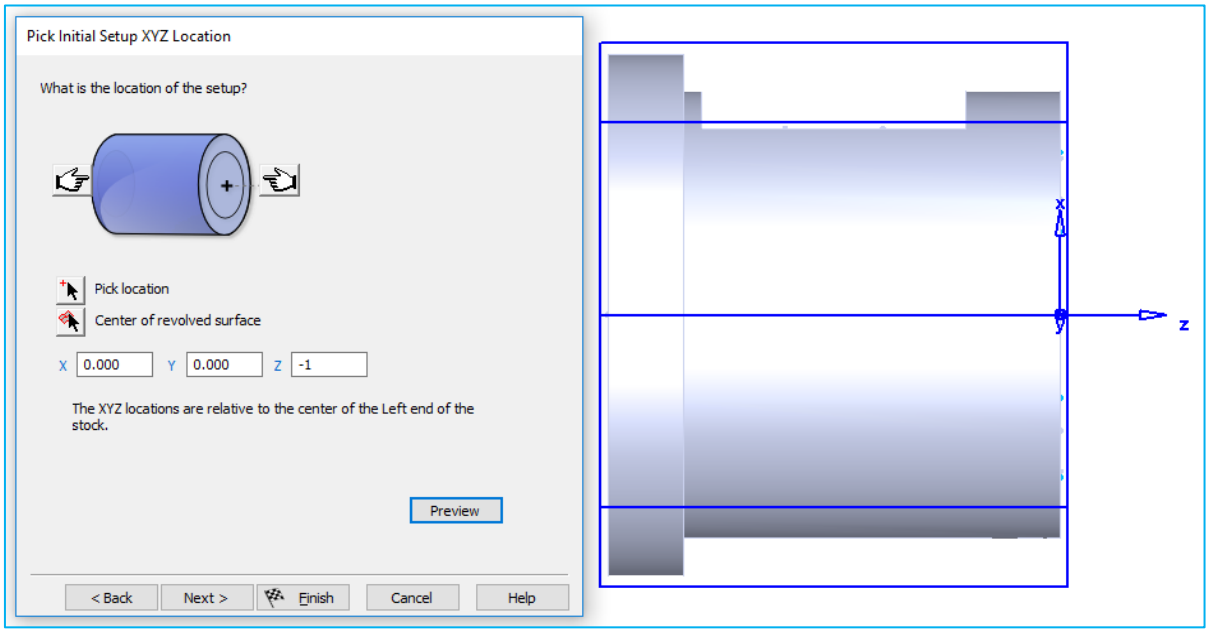


2 Select **Next** 5 times until you get to the **Stock dimensions** page.

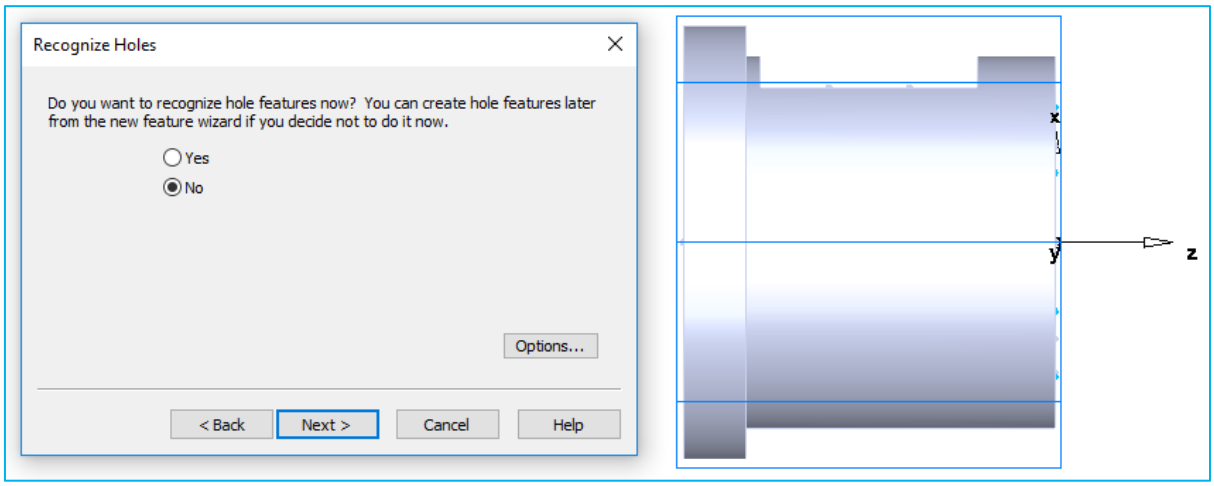


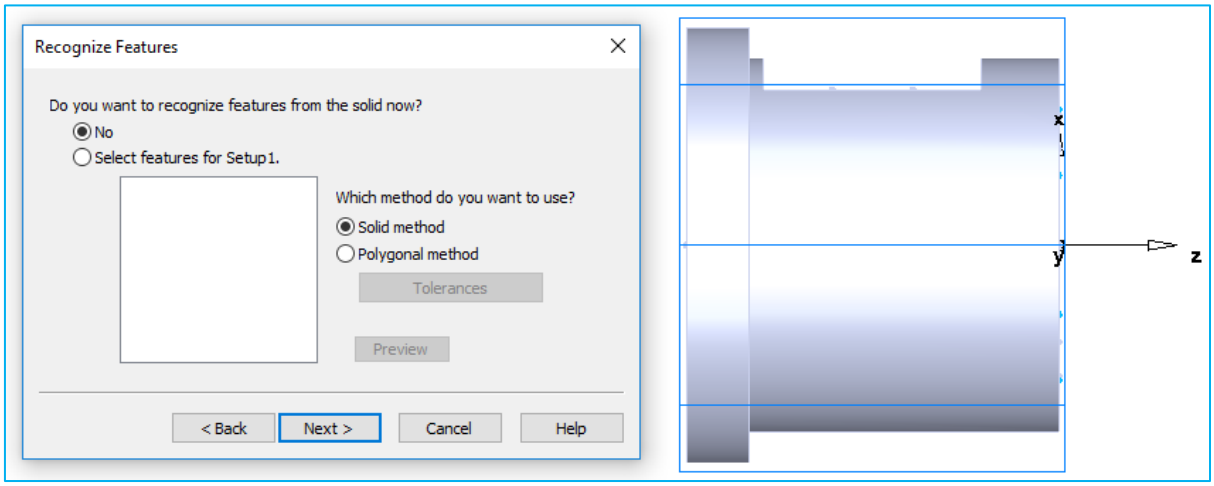
3 Enter the values as shown. Then select **Next**. Select the Right hand. This sets the datum to the stock face. We need to set this to -1mm.

© Copyright 2016 Autodesk, Inc. All rights reserved



4 Select **Finish**. With the wizard still available select **Next**. Select **No** to recognising hole features. Select **Next**.

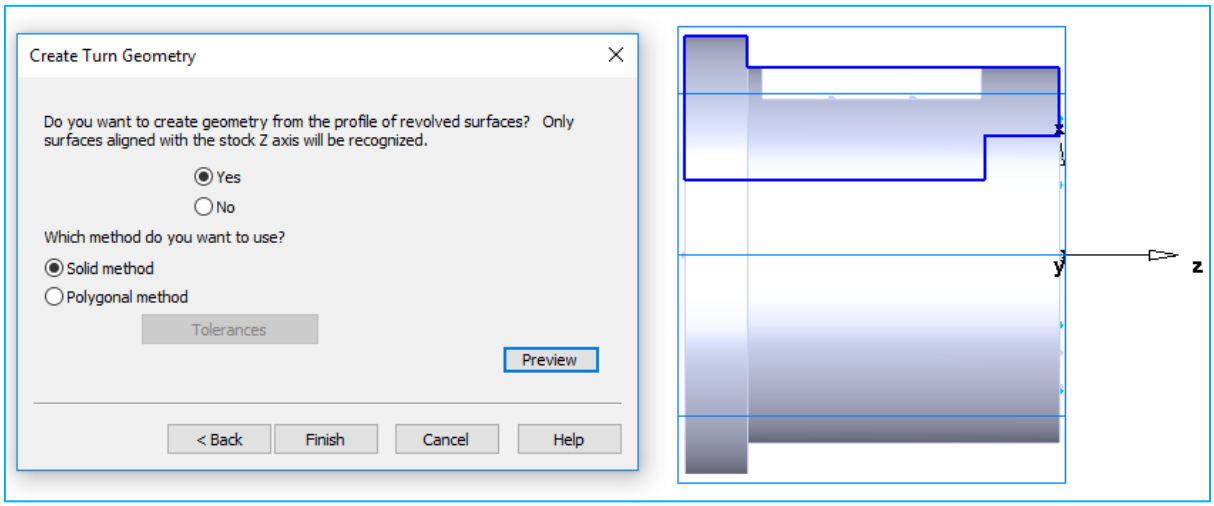




5 Select **No** to Do you want to recognise features from the solid now?

© Copyright 2016 Autodesk, Inc. All rights reserved

6 Select **Next**. Select **Yes** to Do you want to create geometry from the profile of the revolved surface. Select **Preview** to look at the profile.

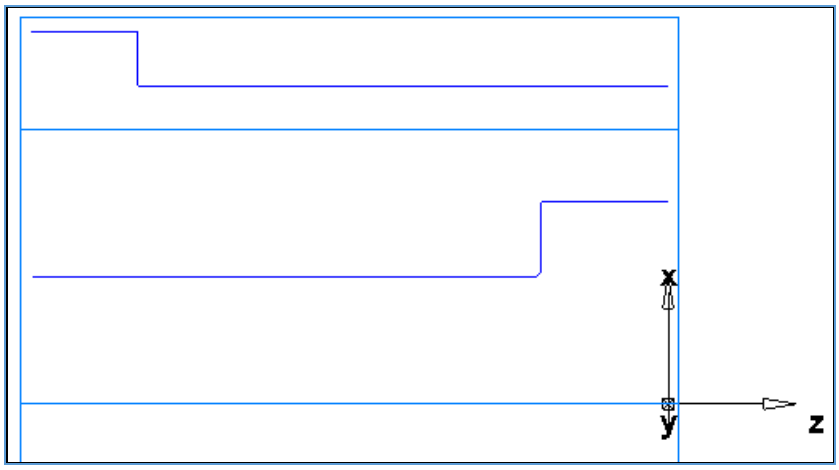


7 Select **Finish**.

*You can manually do this by selecting **Construct>Curve>From Surface>Revolved Surface boundary.***

8 **Hide** the solid model from **PartView**.

9 Create a **Curve** for the **outer profile** and another **Curve** for the **Bore**.



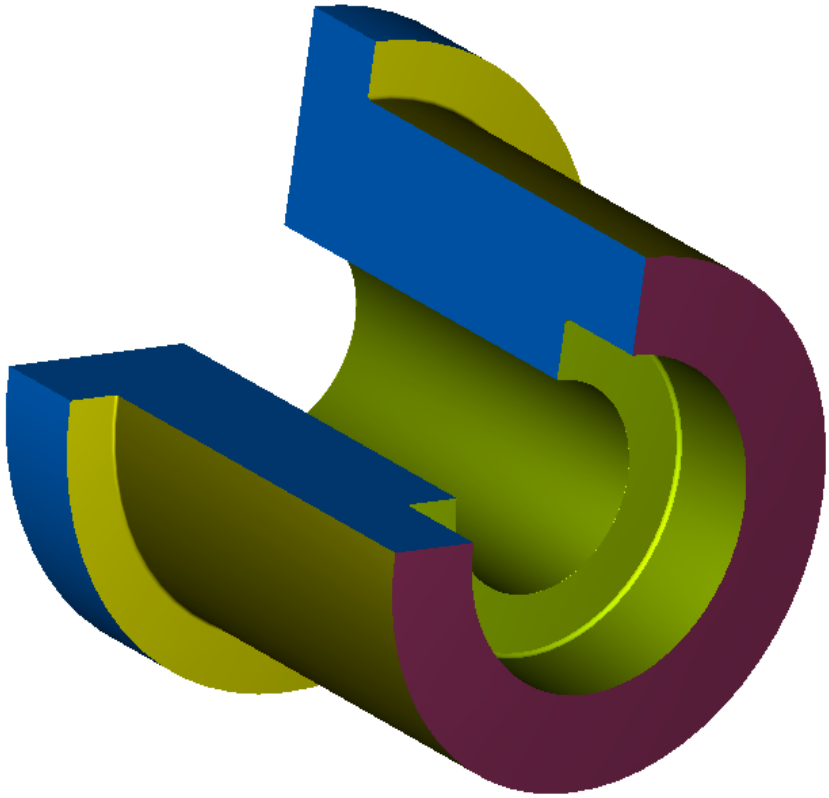
10 Create a **Face** feature.

11 Create **Turn** Feature using the outer profile curve.

12 Create a **Bore** Feature and select the inner profile curve. Select pre-Drill hole using a **22mm U-Drill** to rough out the centre material.

Features created so far as shown on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved



13 **Eject** the **3D Simulation**.

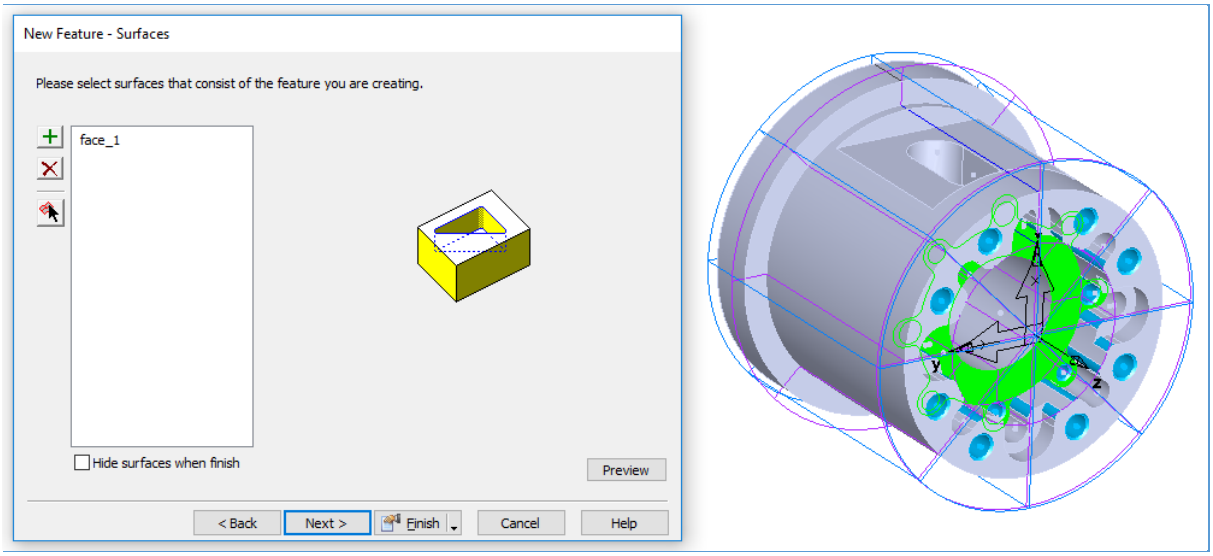
14 Select **Show all Solids** from the **View** menu.

15 Select **Ctrl + R** to create a New Feature. Select **Turn/Mill**.

16 **Select a Pocket from Curve**. Remember to select **Extract with Feature Recognition**.

17 Select **Next** and then select **Along the setup Z-axis**.

18 Select **Next** and select **Use Horizontal Surface**. Select the **face** shown. See image below.



19 Select **Next**.

© Copyright 2016 Autodesk, Inc. All rights reserved

20 The Top is Zero and the Btm is **-12mm**

21 Select **Finish**.

22 Go to the **Pocket Strategy** page **untick Rough** and Tick **Semi Finish**.

23 This will reduce the amount of fresh Air cuts. Select **Semi-Finish** and Helical side finish. Set a pitch of 4mm. For every revolution **Z will move by 4mm**.

*We will now machine the **Counter bores** which are **5mm deep**.*

24 Select **Ctrl + R** and Select **Turn/Mill**.

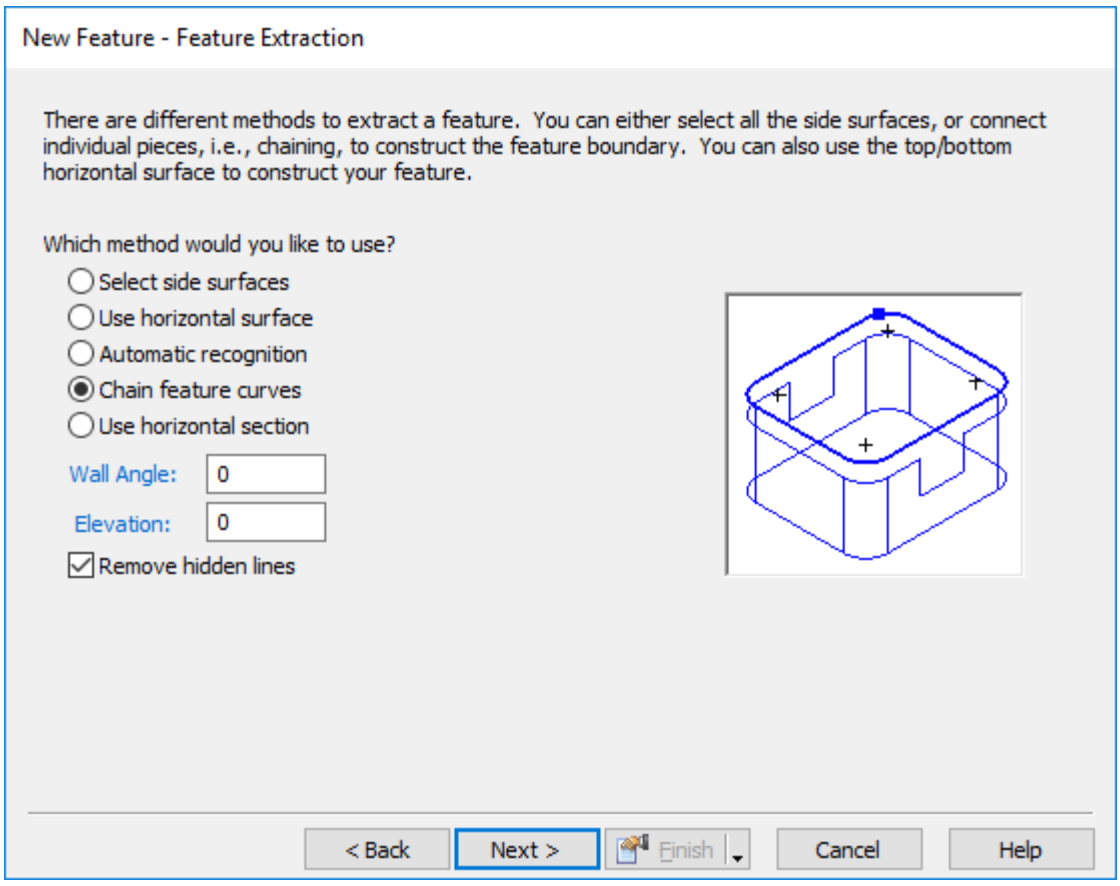
25 Select **Side** with **Extract with Feature Recognition**.

26 Select **Next**.

27 Select **Along the Z Setup Axis**.

28 Select **Next**.

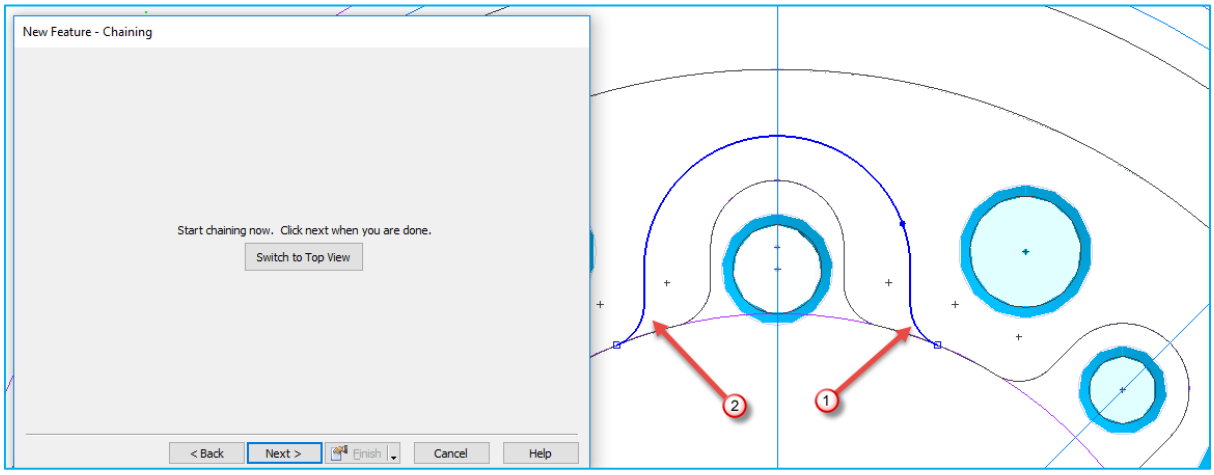
29 Select **Chain Feature Curves**.



30 Select **Next**.

31 FeatureCAM will say **Start chaining now**.

© Copyright 2016 Autodesk, Inc. All rights reserved

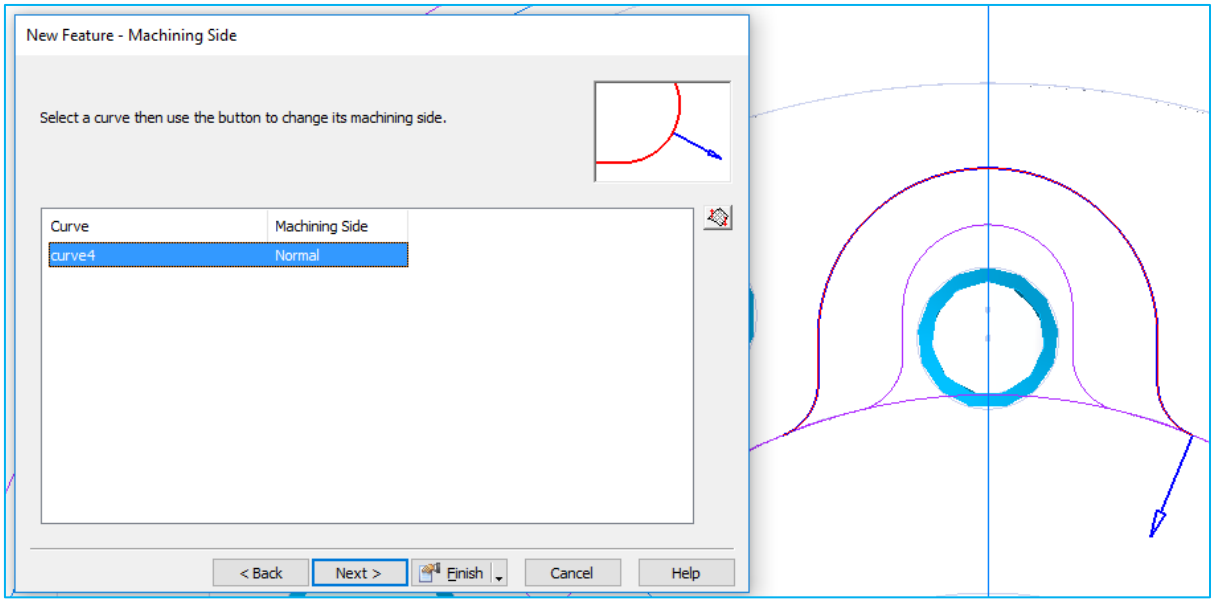


32 Chain the profile as show.

33 Select **Next**.

34 FeatureCAM shows you the arrow to decide which side needs to be machined.

35 Select reverse direction if your curve arrow is pointing the wrong way.



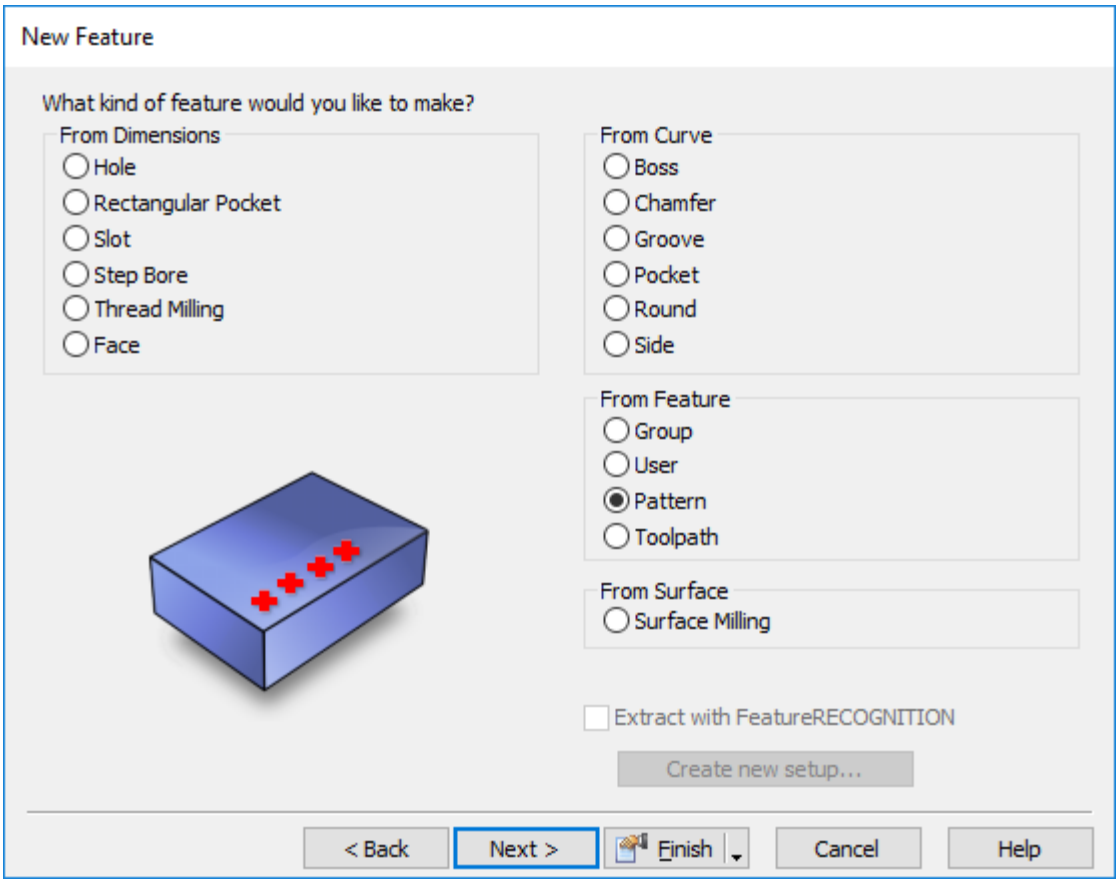
36 FeatureCAM will ask you for Z Heights. The **Top** is **Zero** and the **Bottom** Is -**5mm**

37 Select **Finish**.

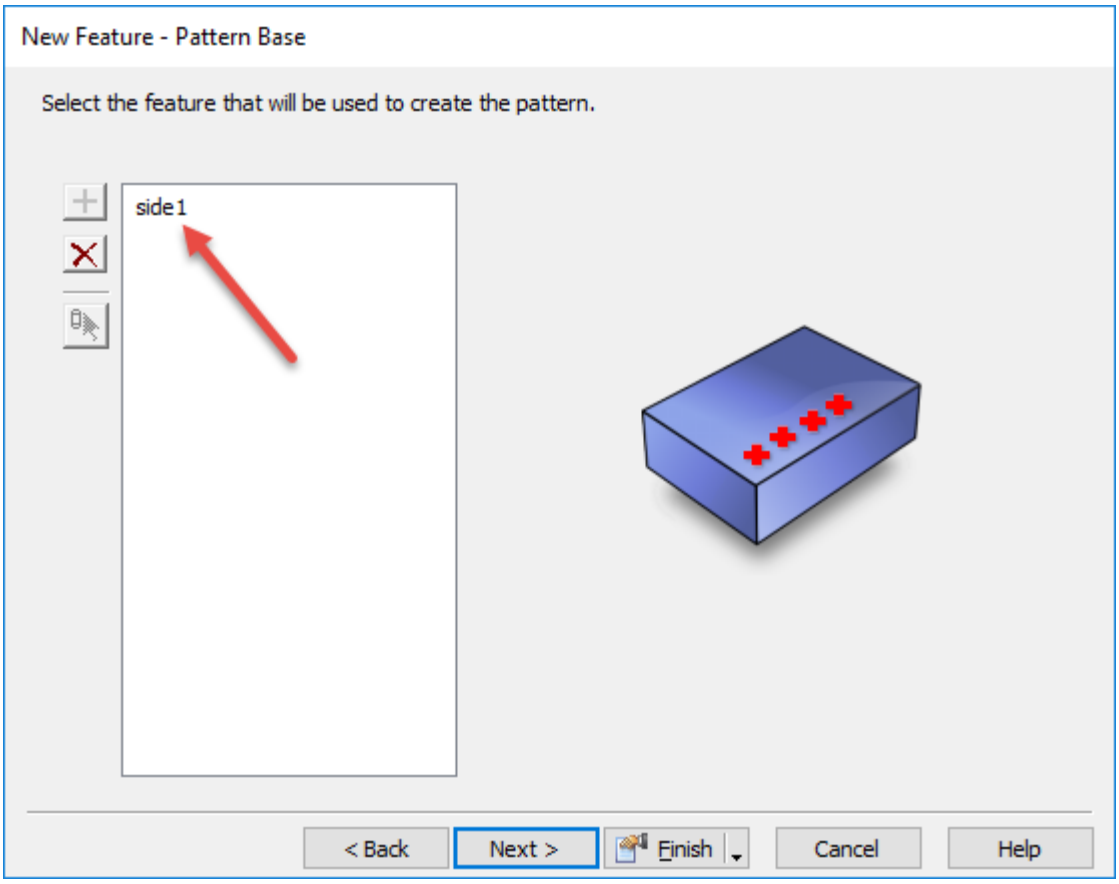
38 What we are going to do now is make a pattern from the side feature.

39 Create a New Feature **Ctrl+R Turn/Mill** then select **Pattern.**

© Copyright 2016 Autodesk, Inc. All rights reserved

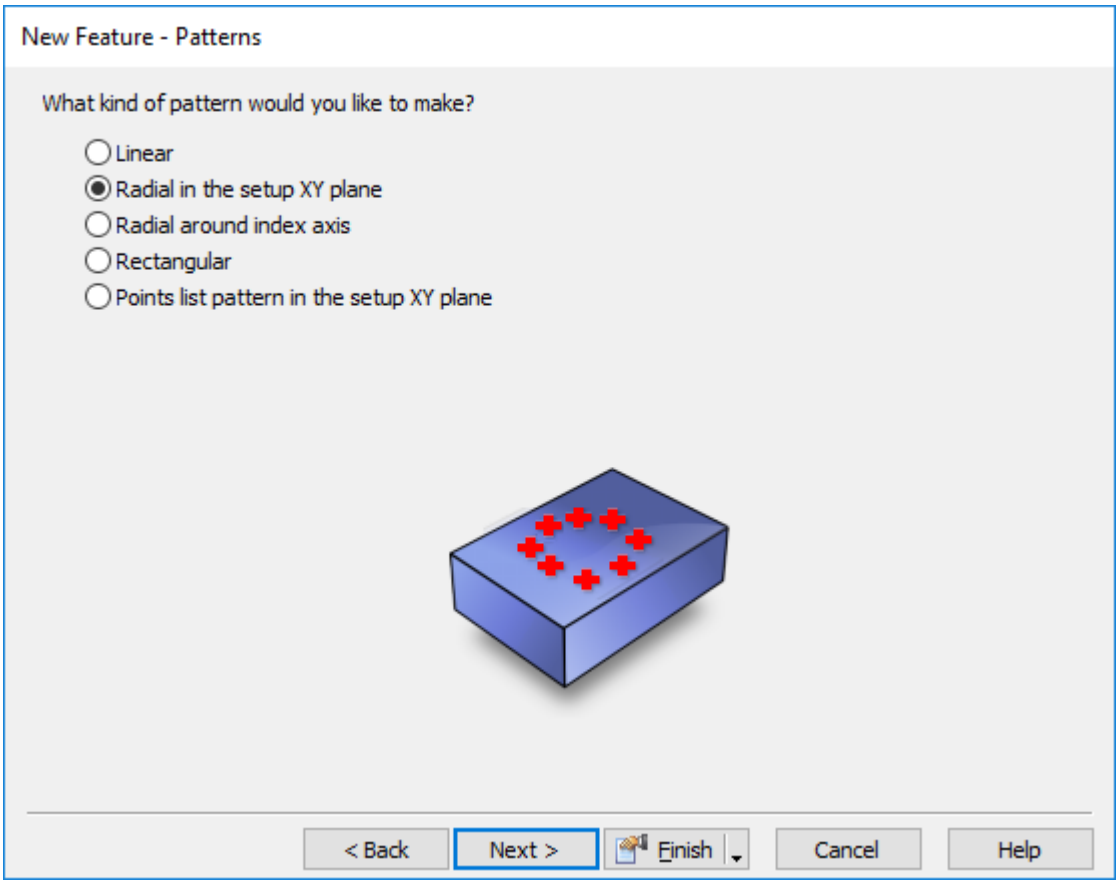


40 Select **Next**. Select the **Side** feature you have just created.

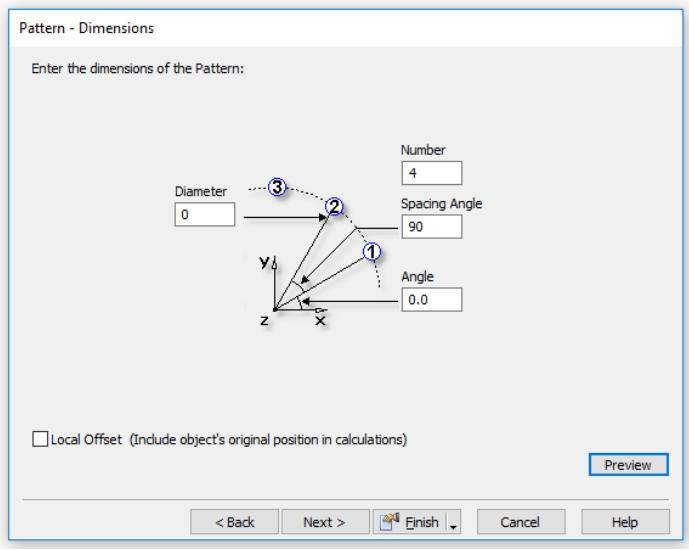


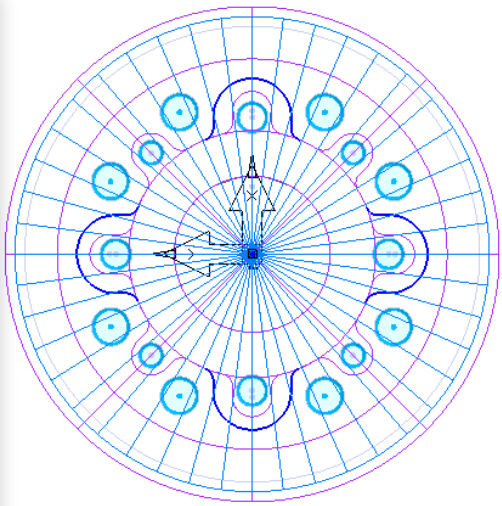
© Copyright 2016 Autodesk, Inc. All rights reserved

41 Select **Next**. Select **Radial in the setup XY plane**.



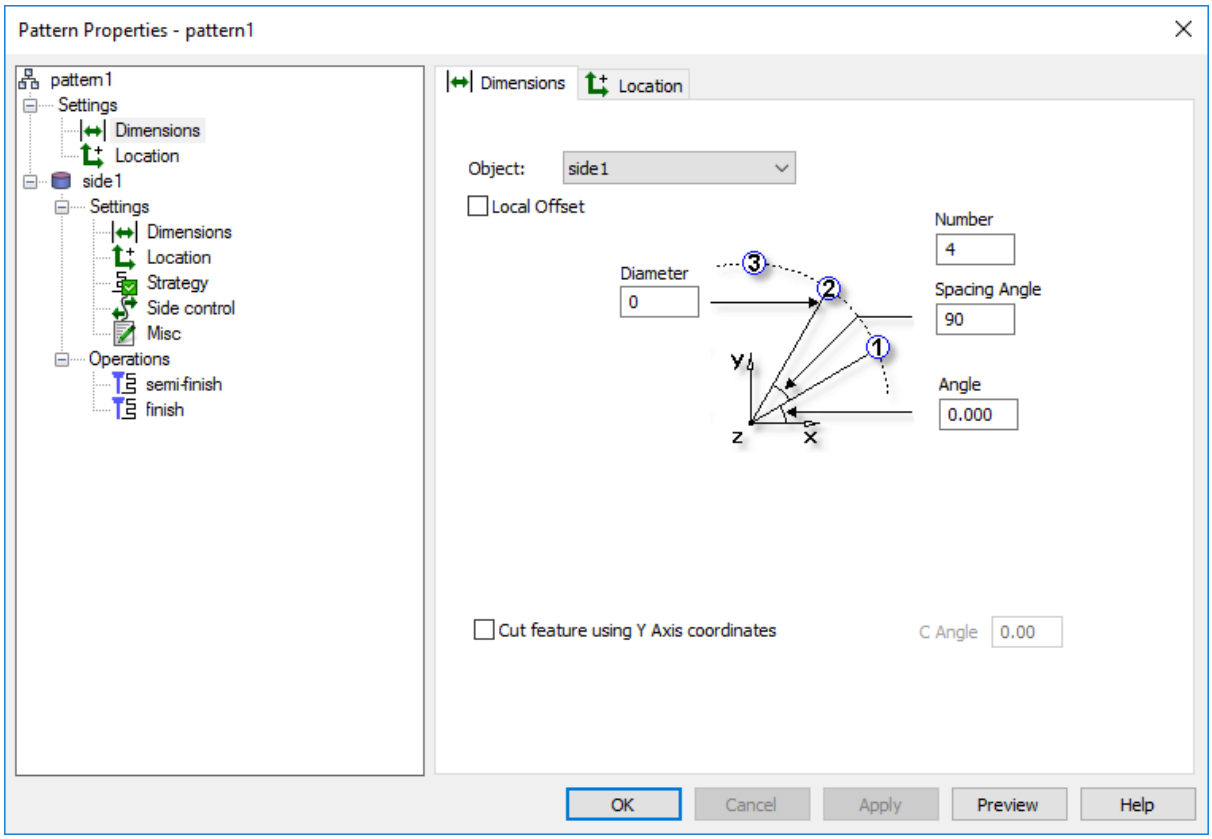
42 Select **Next**.

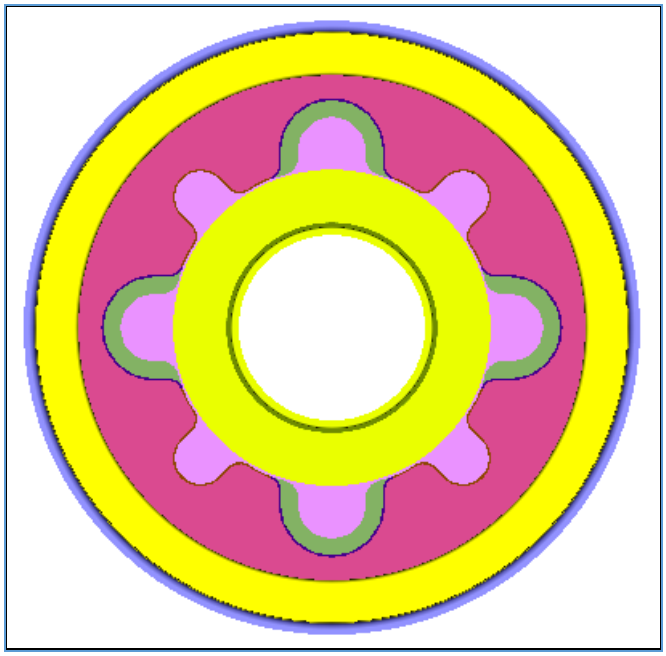




43 Enter the values as shown above. Then select **Finish**. Change the Chamfer to 0.25mm and then in the strategy page unselect the **roughing operation**. Add a **Semi finishing pass**.

© Copyright 2016 Autodesk, Inc. All rights reserved





44 **Eject** the Simulation.

45 We need to Drill and Chamfer all holes.

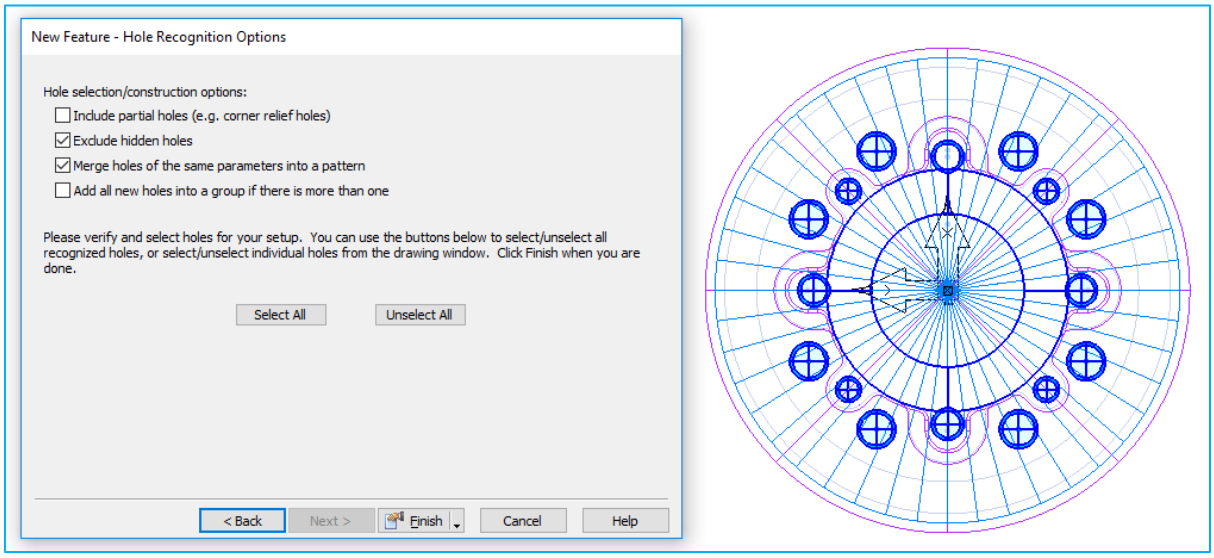
46 Select **Ctrl + R** and Select **Turn/Mill**.

47 Select **Hole** with **Interactive Feature Recognition**. Remember to unselect the centre holes.

48 Along the **Z Axis**. Select **Next.**

49 Change the **Spot-drill tools** otherwise it will damage the part.

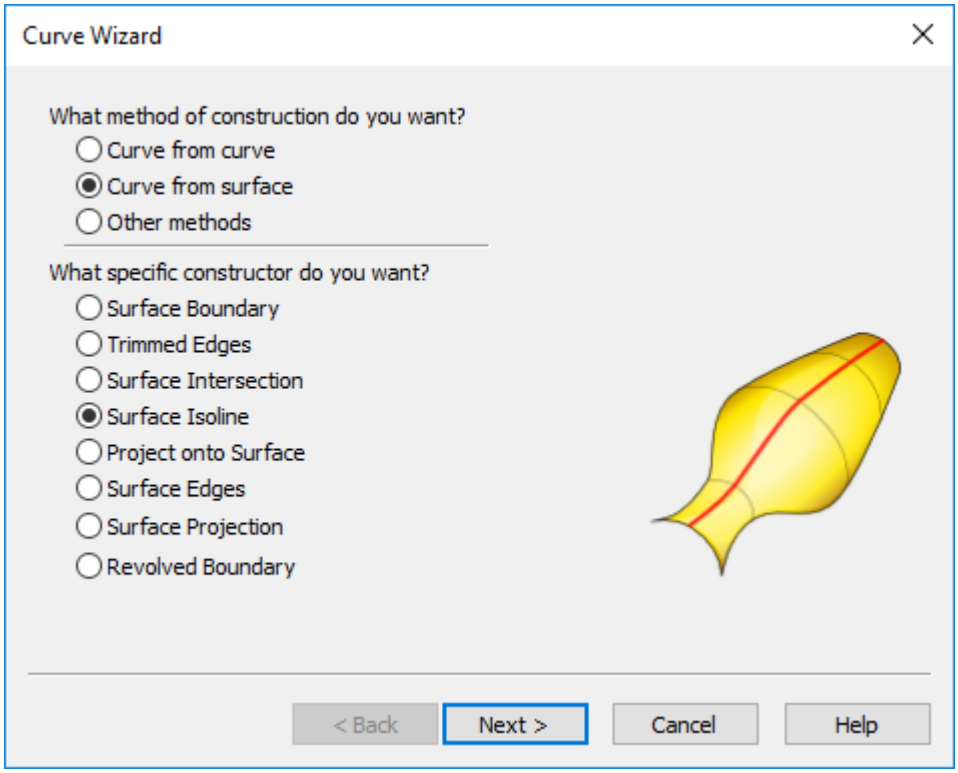
© Copyright 2016 Autodesk, Inc. All rights reserved



50 Run a **3D Simulation** to check for collisions. Once you are happy with your tool choice. **Eject the Simulation**.

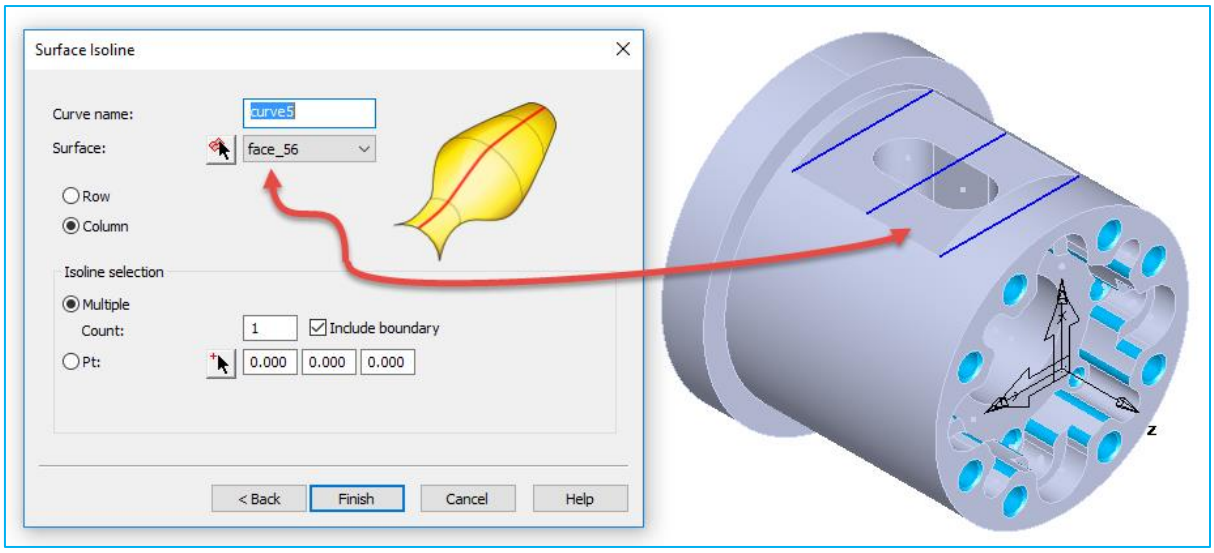
*We will now concentrate on the **Slot** and the **Flat**.*

51 Select the **Curve Wizard**. Select **Curve from Surface** and then **Surface Isoline**.



52 Select **Next**. Select the **Surface** and change to **Column**.

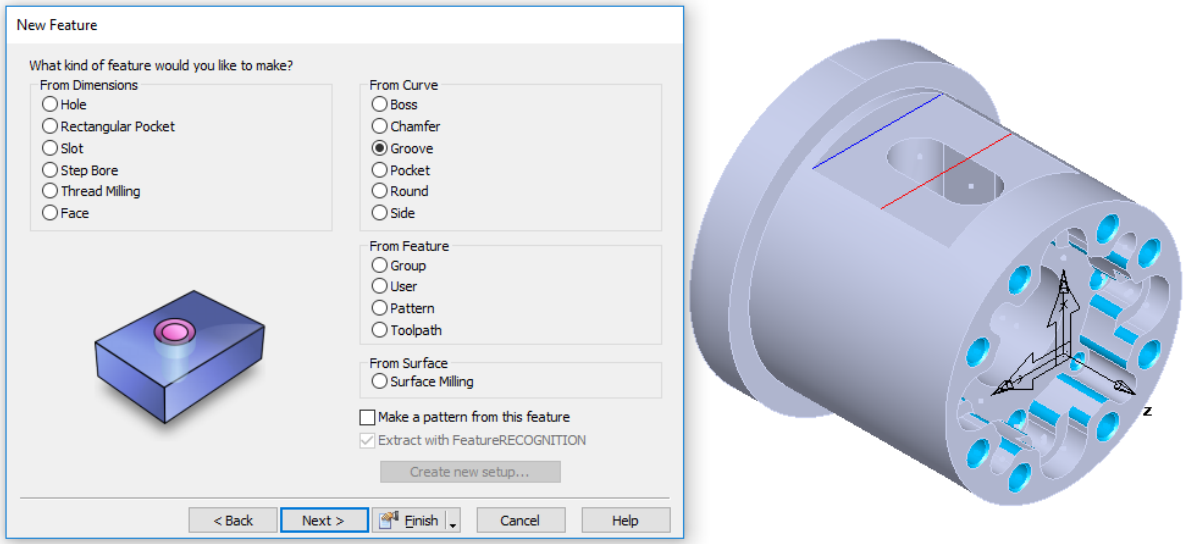
© Copyright 2016 Autodesk, Inc. All rights reserved



53 Select **Finish** and then select the middle curve.

54 Select **Ctrl + R** and Select **Turn/Mill**.

55 Select **Groove from Curve**.



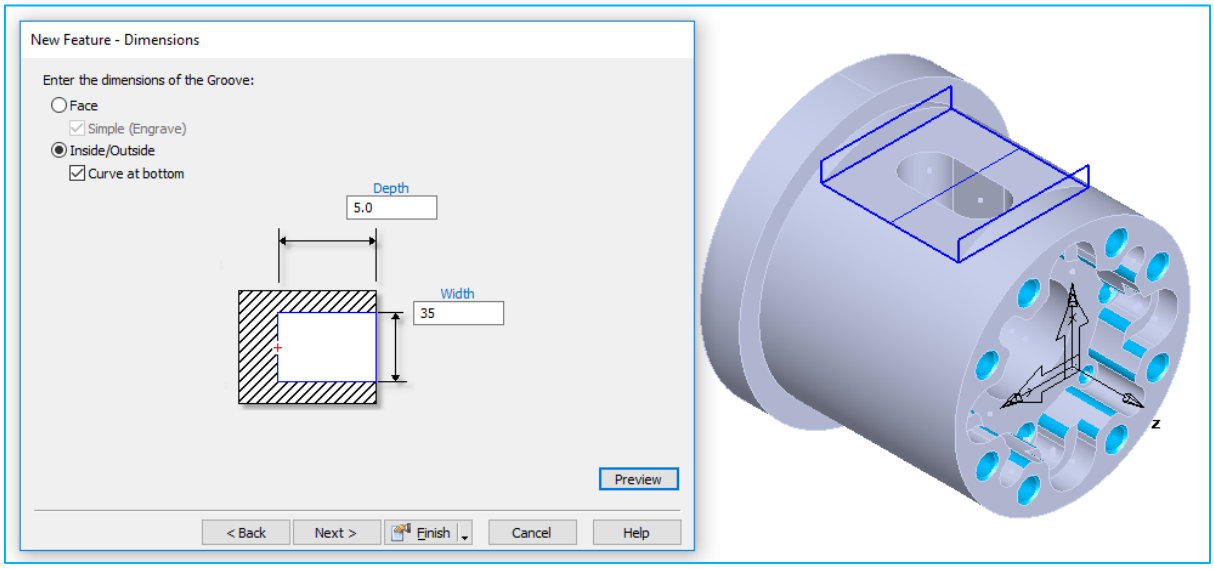
56 Select **Next** three times.

57 The width is **35mm** and the depth is **5mm.**

58 Select **Inside/Outside**. And **Curve at Bottom**.

59 Select **Next**. You may have to **reverse** the **direction** of the curve.

© Copyright 2016 Autodesk, Inc. All rights reserved



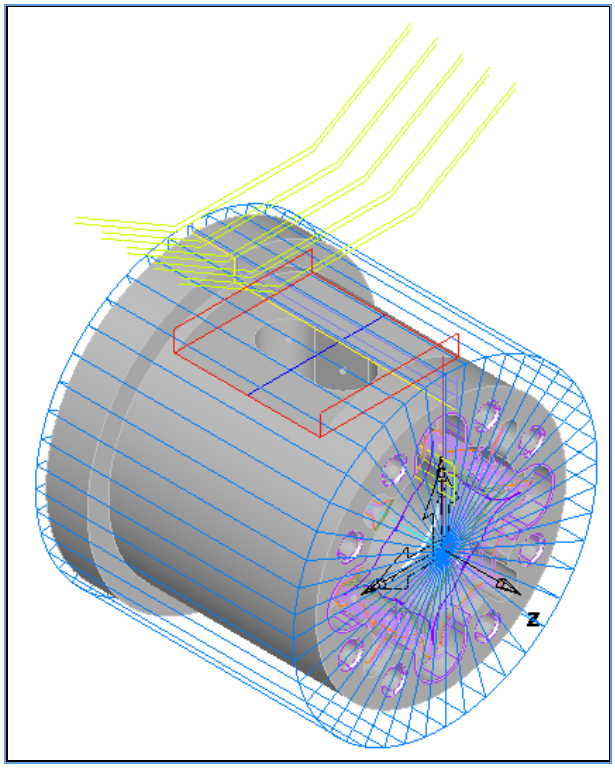
60 To reverse the Side click **Next** to get to the next menu.

61 Select **Next**.

62 Pick the Tool and Machine the Slot.

63 I have used a **50mm** Dia cutter **12mm wide** with a shank of **25mm**

64 See image below.



65 Select **Finish** and **OK**.

66 We will now machine the **12mm** wide slot on the same face.

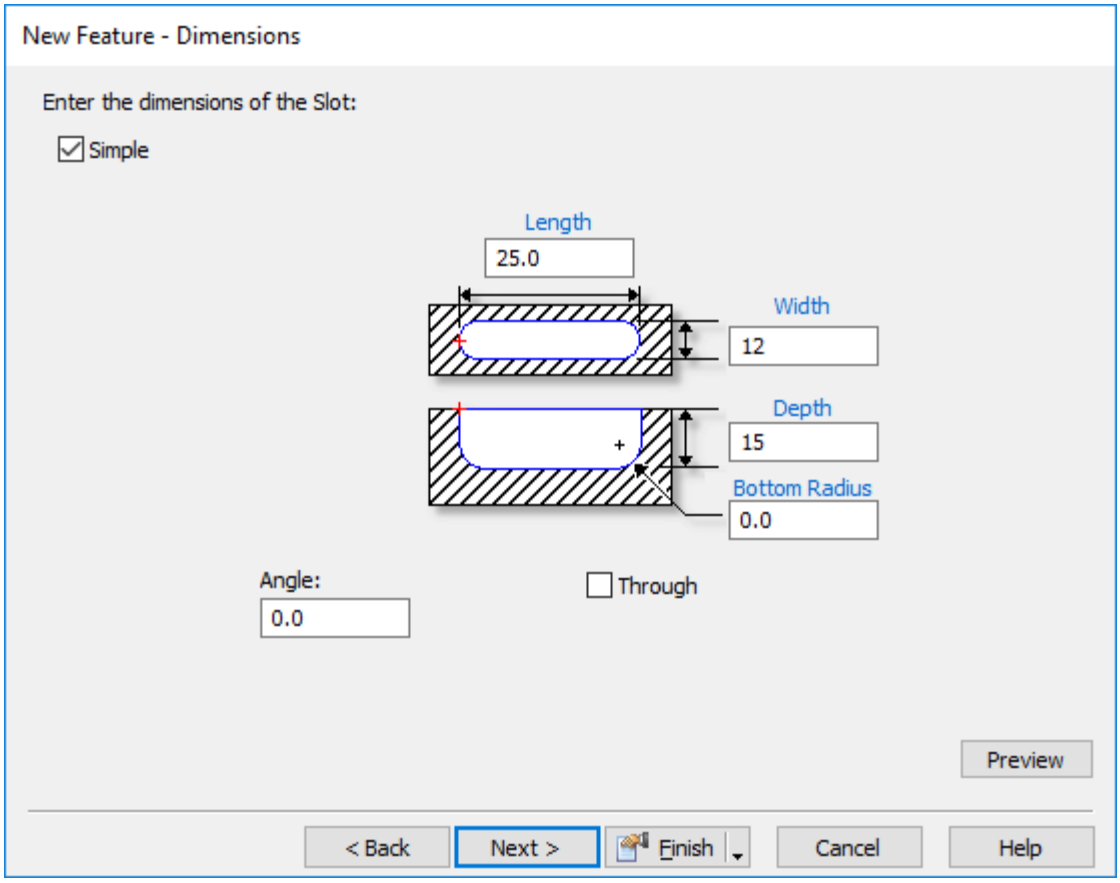
*If your machine has **C Axis** only you will have to use a smaller cutter first to rough it out then use a cutter **12mm** wide to finish. If your machine has **Y axis** then the machining of the slot will not be a problem as you can use a smaller cutter and use cutter Compensation **G41/G42**.*

© Copyright 2016 Autodesk, Inc. All rights reserved

67 Select **Ctrl + R** for a **New Feature**.

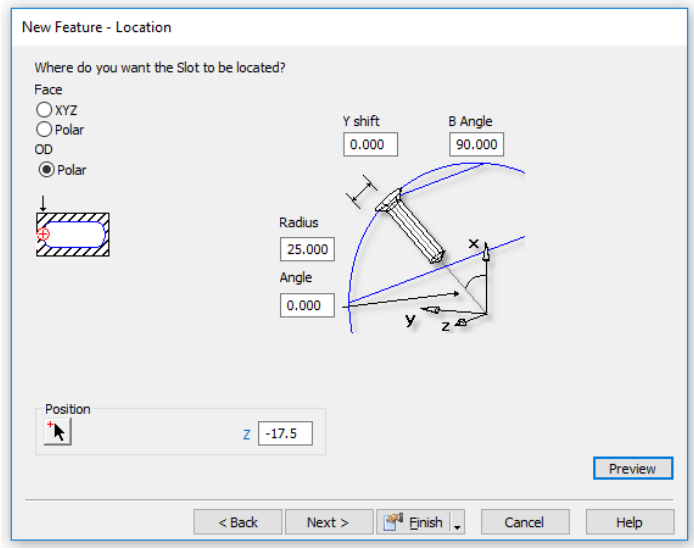
68 Select **Turn/Mill**. Select **Next**.

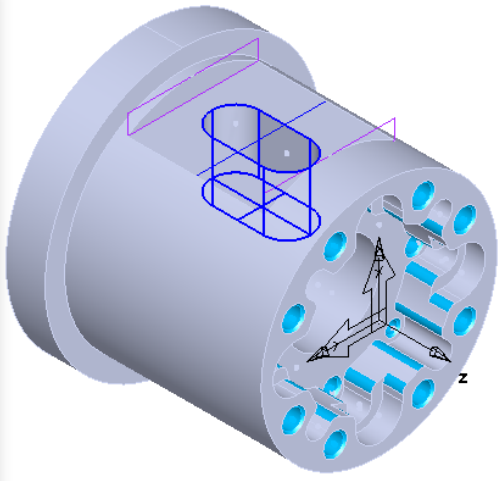
69 Select **Slot from Dimensions.** Enter the following Figures. Select **Simple.**



70 Select **Next.**

71 Select **OD Polar**. Change the **Z** value to **-17.5**





72 Enter the above dimensions.

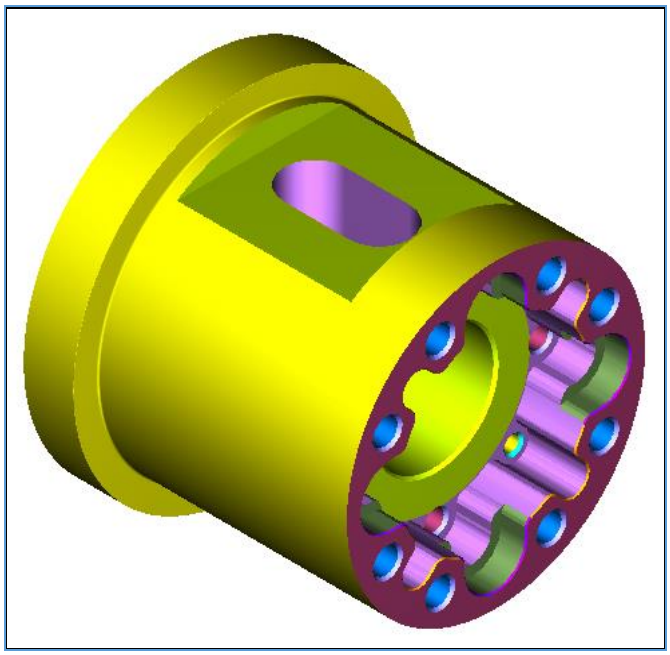
73 Remember you can use the **Hyperlinks** in Blue to retrieve dimensional data from the model.

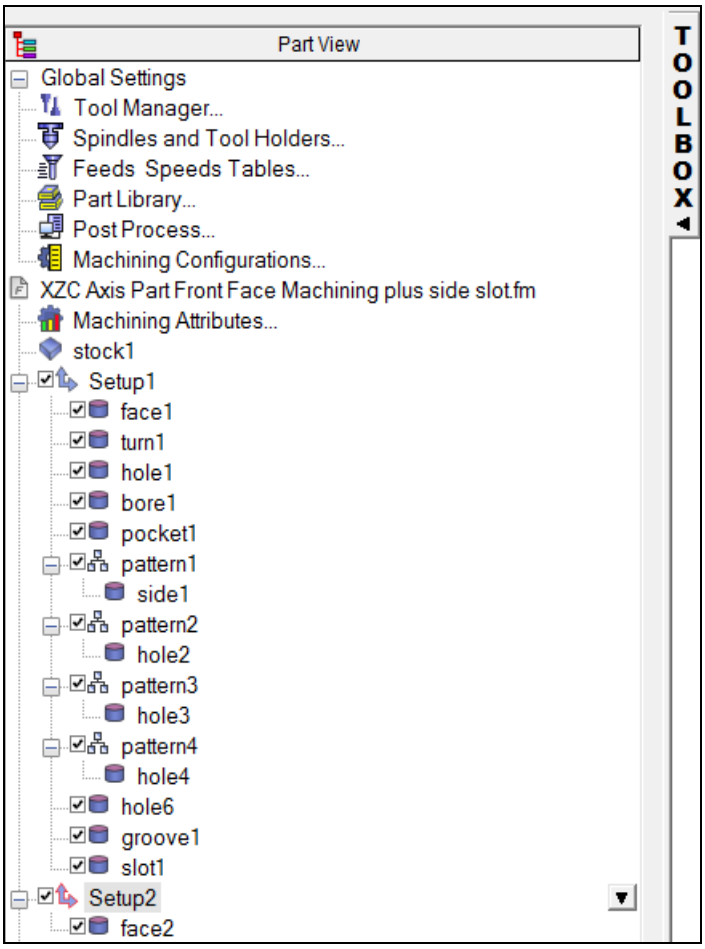
© Copyright 2016 Autodesk, Inc. All rights reserved

74 Select **Finish**.

75 Setup **Setup2** to finish the back face.

76 The image below is the finished part.





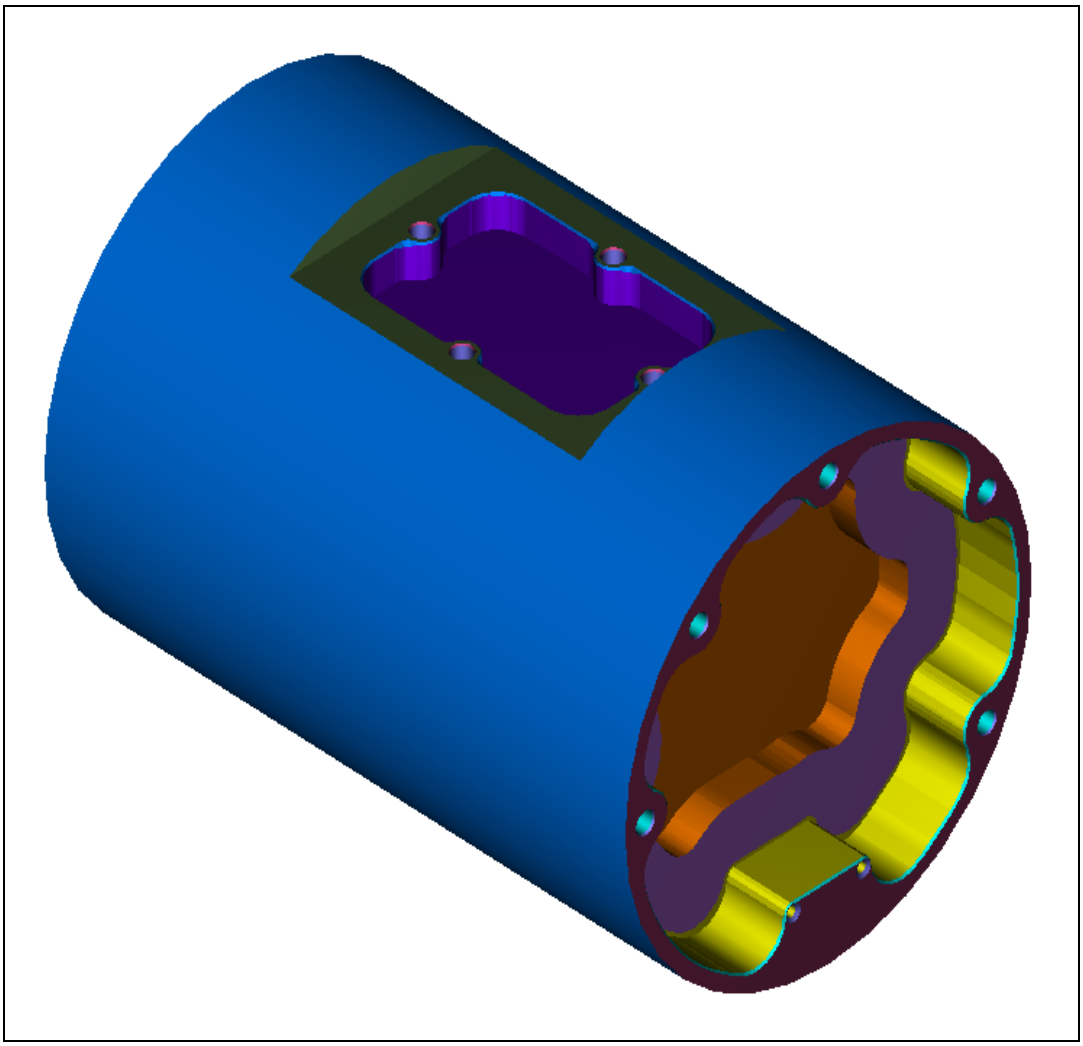
© Copyright 2016 Autodesk, Inc. All rights reserved

Vortex Machining

Vortex Introduction

- Because Vortex machining cuts with the side of the tool, it is designed for solid carbide tools, but there may be other types of tools suitable for Vortex. These tools work best when taking deep cuts with a relatively small stepover.
- To machine effectively when taking large depths of cut, you must ensure the tool engagement angle never exceeds the specified value. This eliminates excessive tool load and all full-width cuts. FeatureCAM achieves this by introducing Trochoidal moves to prevent the tool from exceeding the maximum tool engagement value.

  Open file **Turnmillvortex.fm** from your instructors chosen location.
  C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm Files



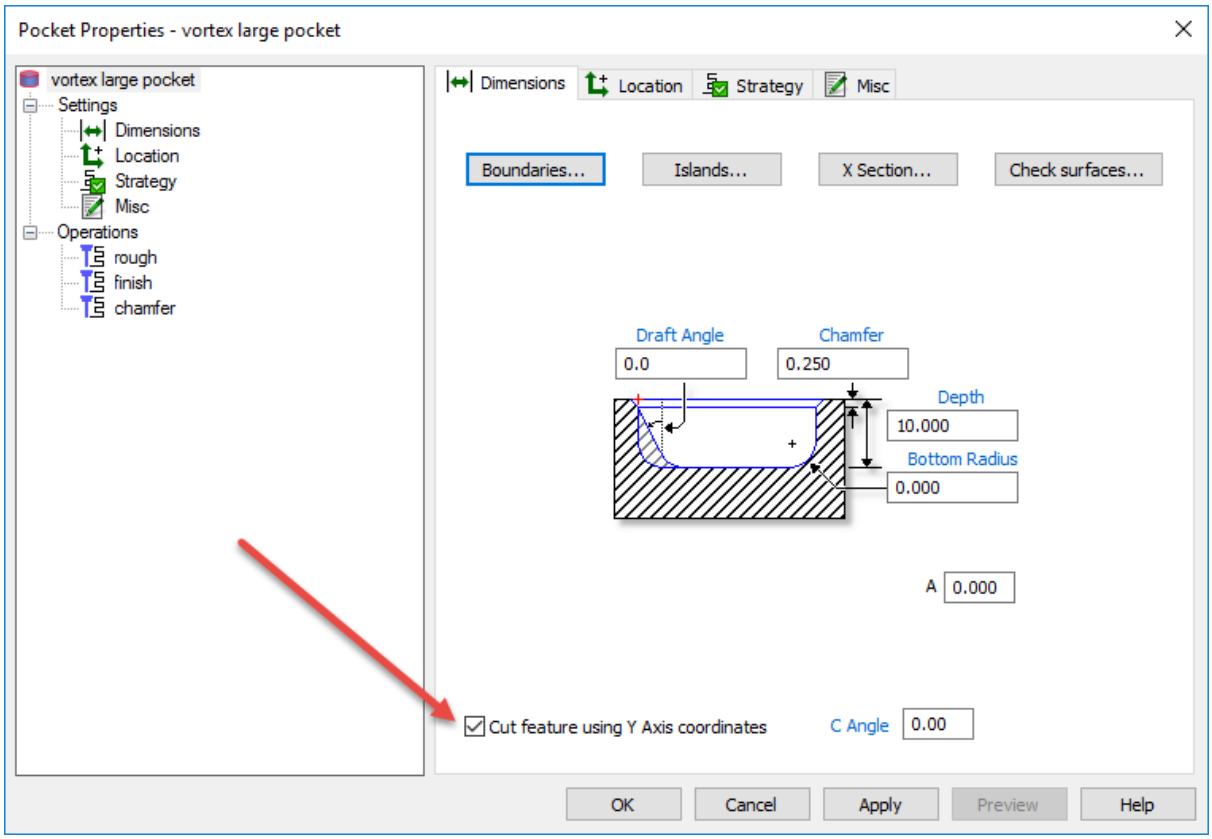
- When creating a 2.5D Vortex toolpath using the C Axis, excessive moves are created. To avoid these sharp movements we can lock the C axis in position and only use the Y axis.

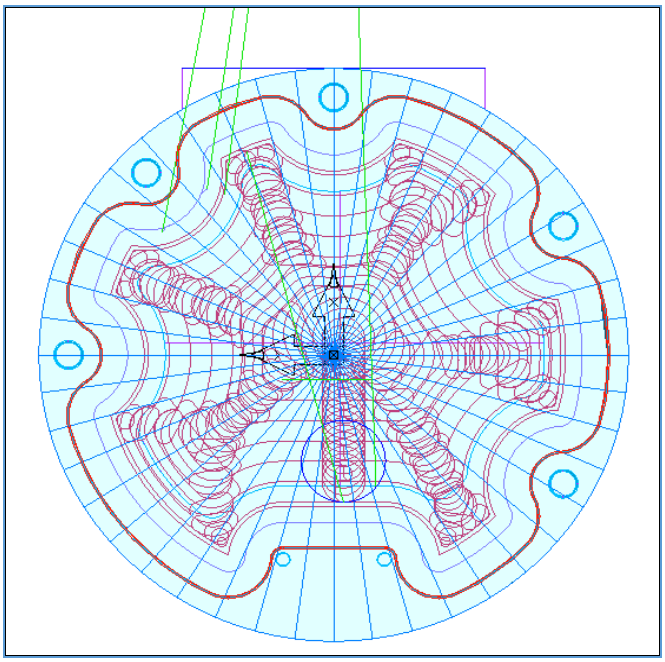
© Copyright 2016 Autodesk, Inc. All rights reserved

Please note that the component must be stable enough to accept large cutting forces.

Pocketing properties

- To change from the C axis to Y axis, select the following option as shown below.

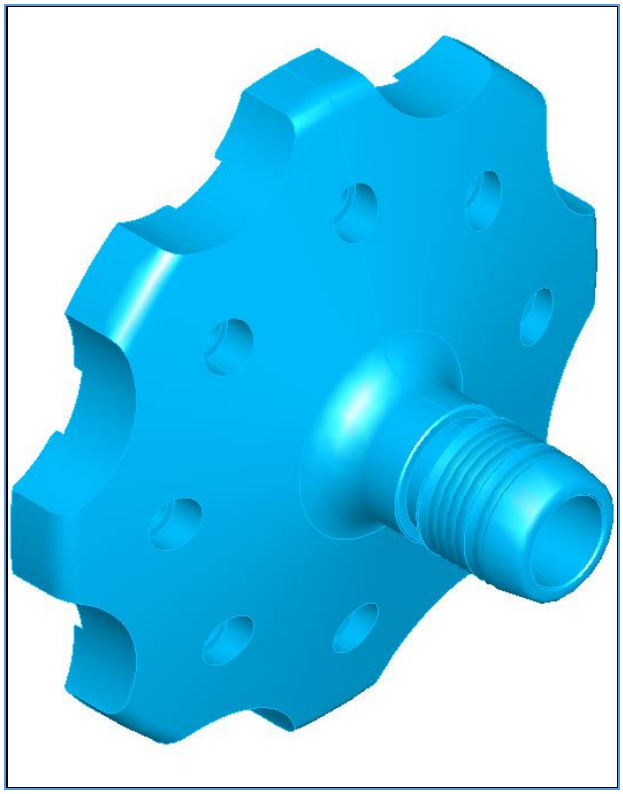




© Copyright 2016 Autodesk, Inc. All rights reserved

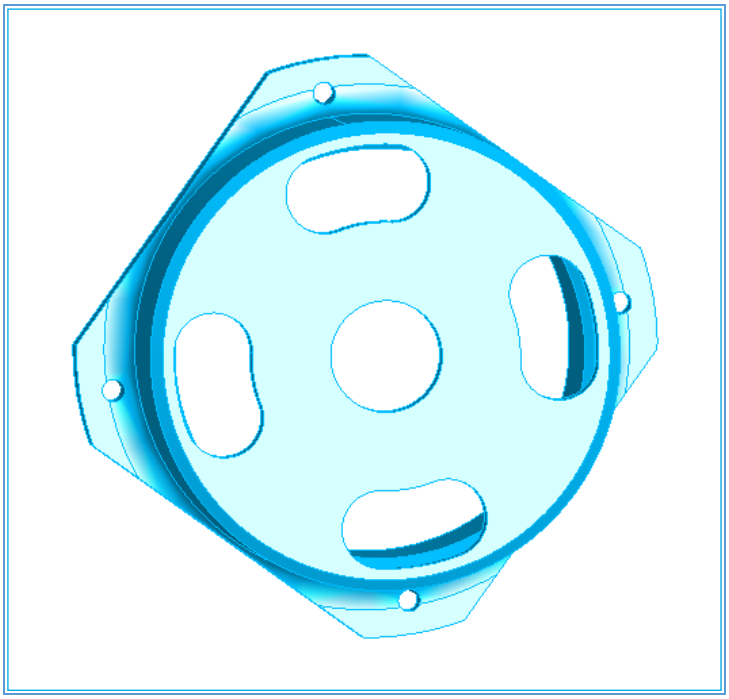
Flange Body Test Piece.

Time permitting work through this example on your own.



Endcap Body

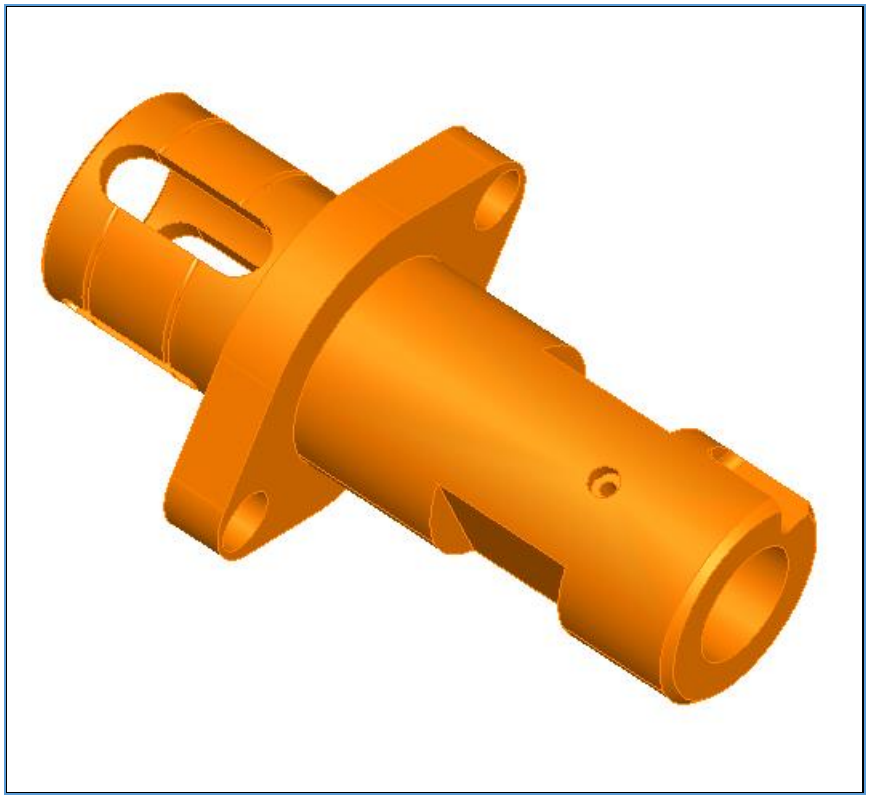
Time permitting work through this example on your own.



© Copyright 2016 Autodesk, Inc. All rights reserved

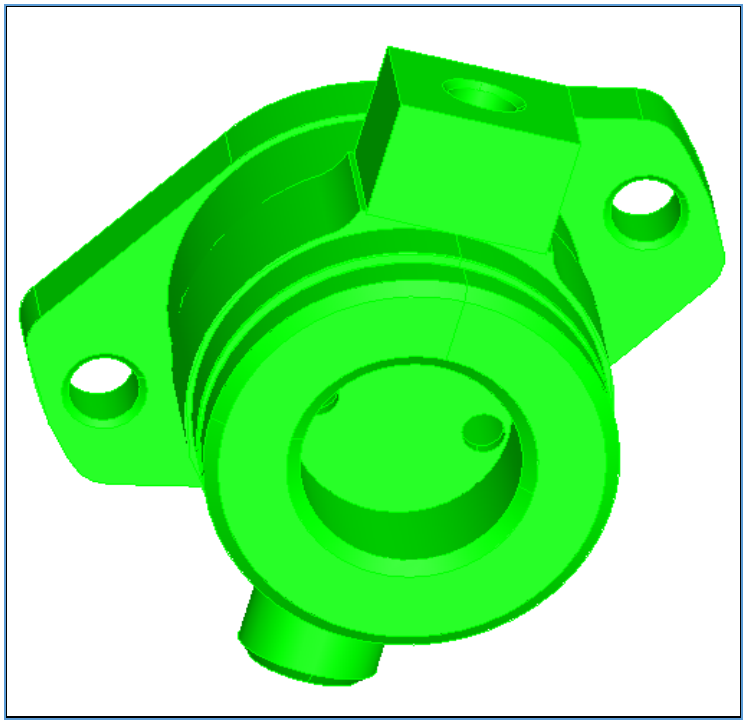
Advanced Pin Machining

Time permitting work through this example on your own.



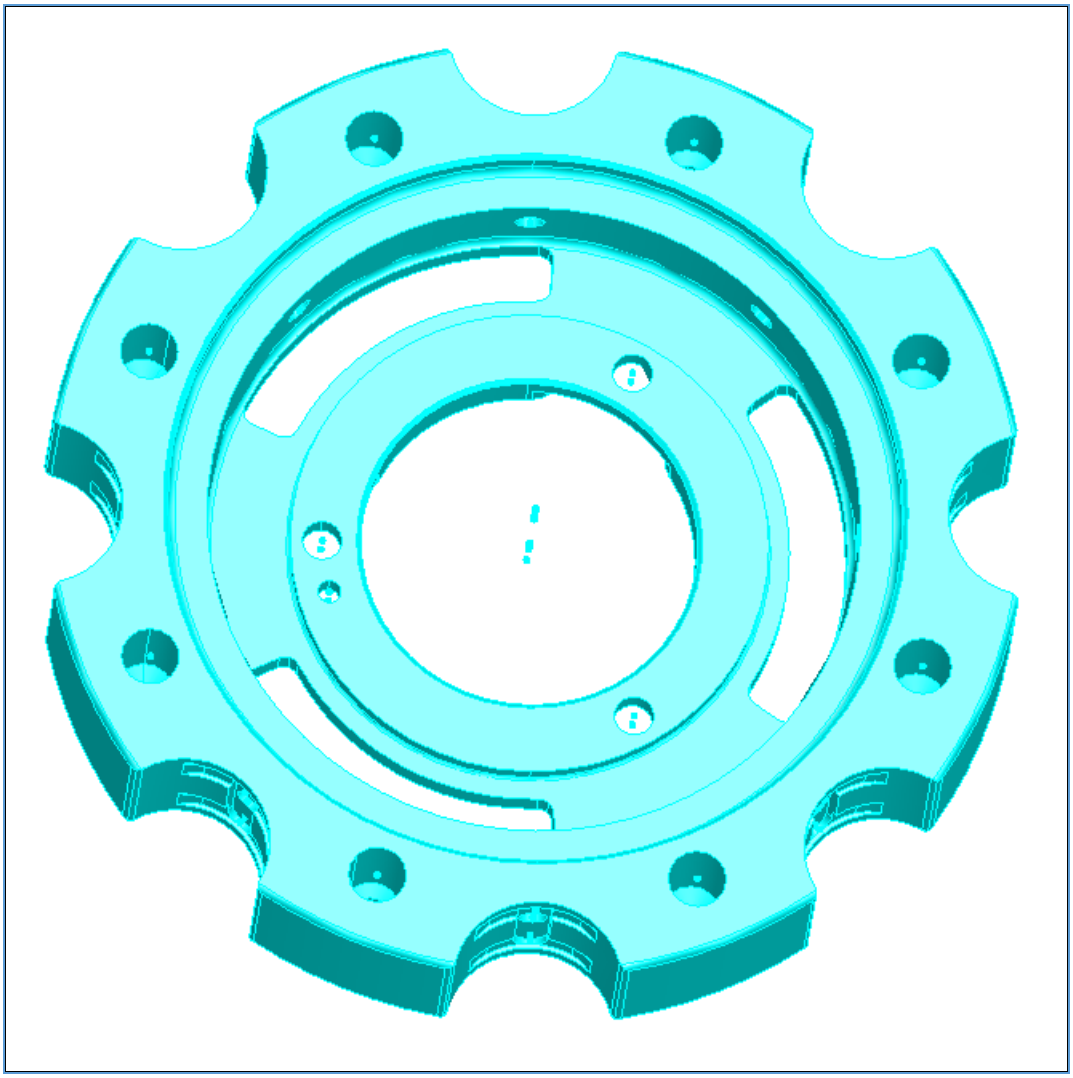
Additional Exercise

Time permitting work through this example on your own.



© Copyright 2016 Autodesk, Inc. All rights reserved

1635_Autodesk-Z.x_t



Turn milling Example (Information Only)

1. Import the Solid Model Import: - **Solid Stock-1 - All Models .x_t** From Location C:/Training_Data. FeatureCAM_Data. FeatureCAM Course Data 2017. Turning Data. **Solid Stock-1 - All Models .x_t**

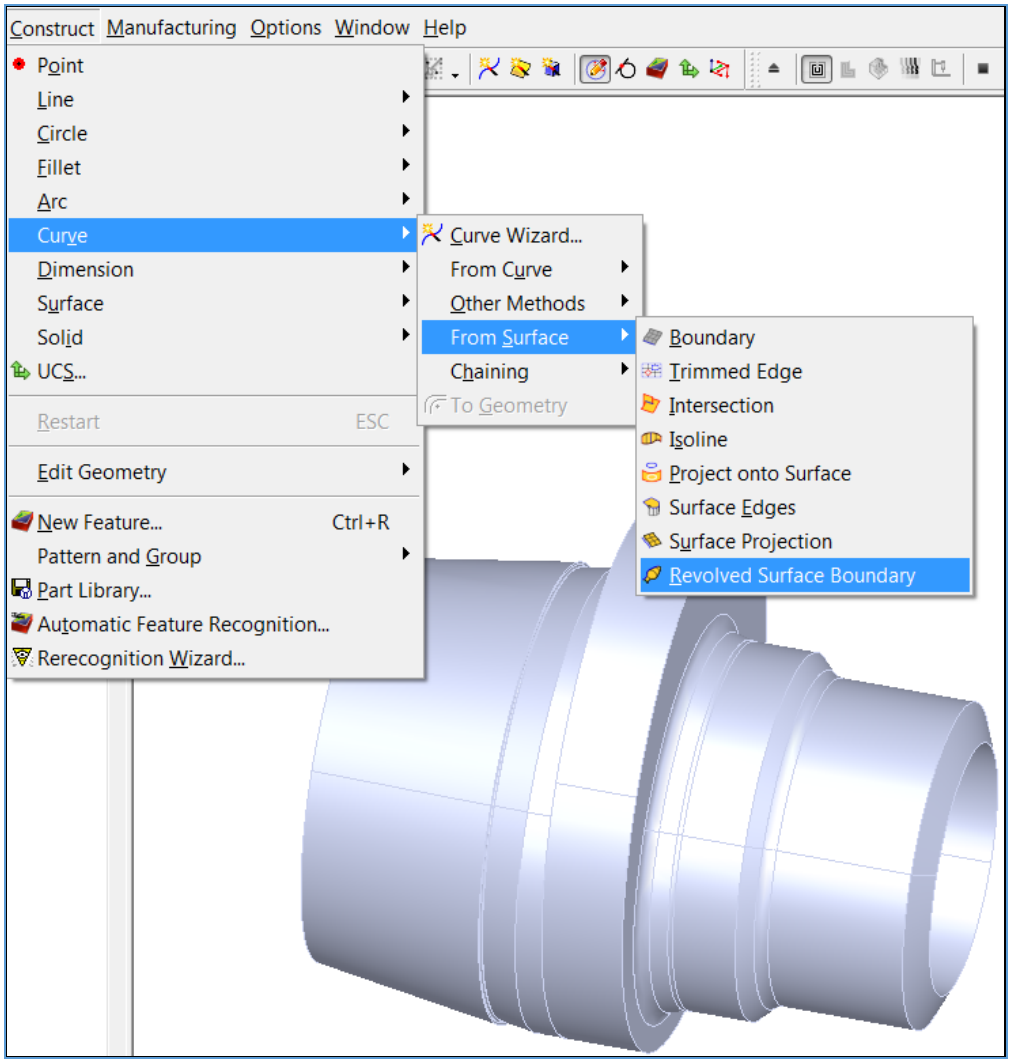
   *Please note - There are two solid models. One is the part and the other is stock. Rename the files in **Part View**. Then hide the stock model.*

2. Change the first Solid model in **PartView>Solids** into the Stock Material.
3. Select **Stock1>User Defined>Solid** and select the Solid in **PartView** that we need to be turned into **Stock**.
4. Click on **Setup1** and Align to the Front Stock face and move the Z to **-6.35mm**
5. From the following folder **C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm Files.** Select the **baxisturningtraining.cnc** post processor from the **Training Posts** folder.

XZ	Millimeter	Layer 1	UCS_Setup1	Setup1	baxisturningtraining.cnc	Basic_Metric_Training	4 cores

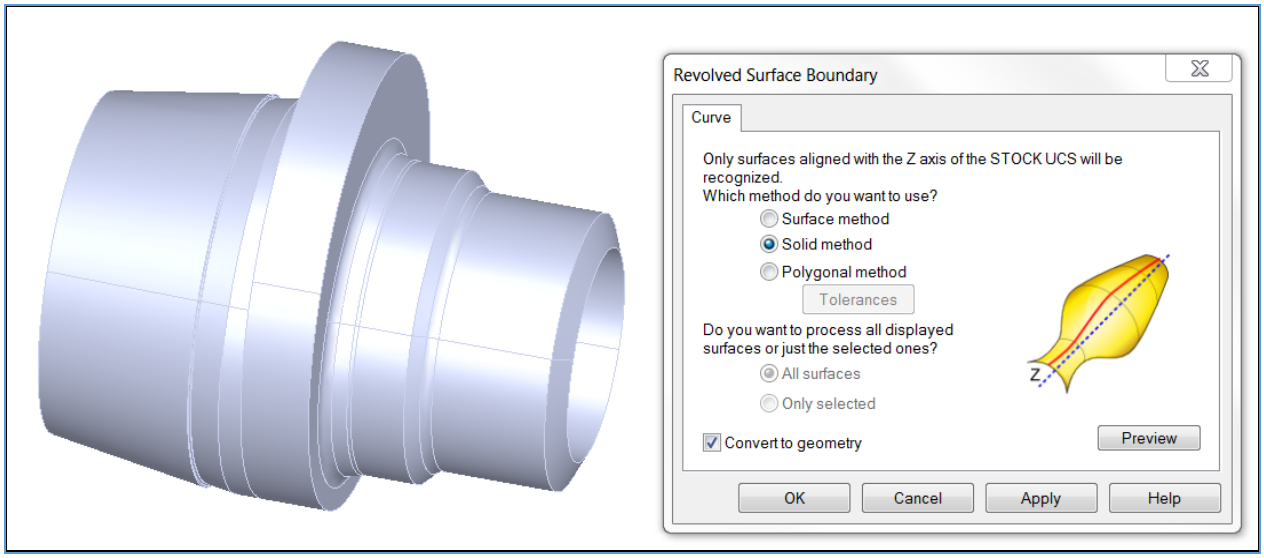
© Copyright 2016 Autodesk, Inc. All rights reserved

6 Select **Curve>From Surface>Revolved Surface Boundary**. (See Below)



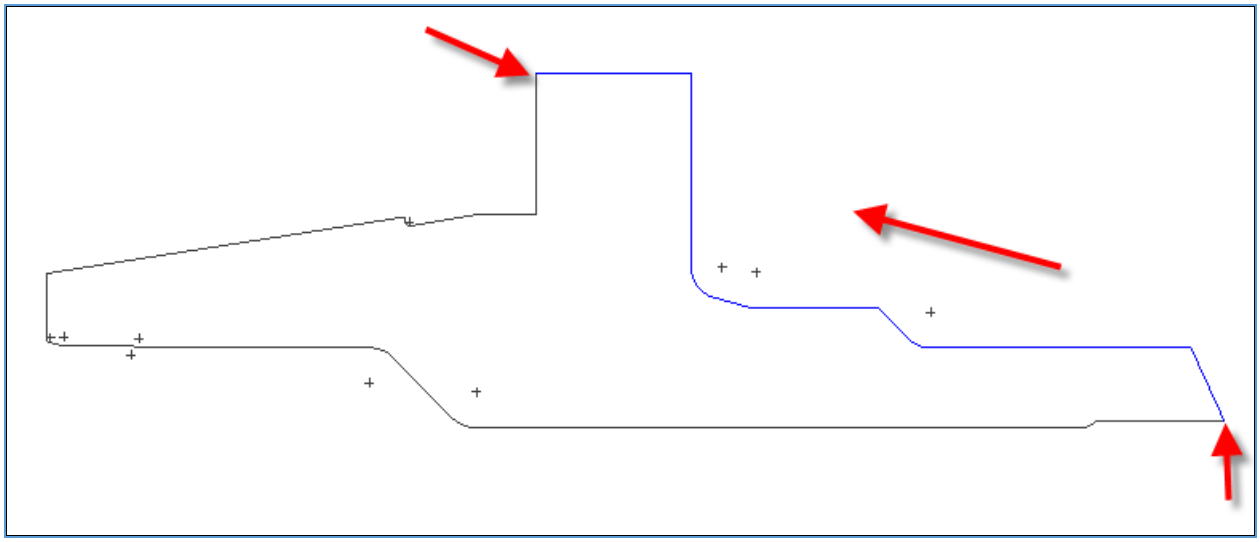
7 Select **Solid Method** Extraction.

8 Select **Convert to Geometry**.



© Copyright 2016 Autodesk, Inc. All rights reserved

9 Create a Curve from the outline Geometry as shown using **Pick Pieces**.



10 Select **Turning** as the **New Feature** - Type

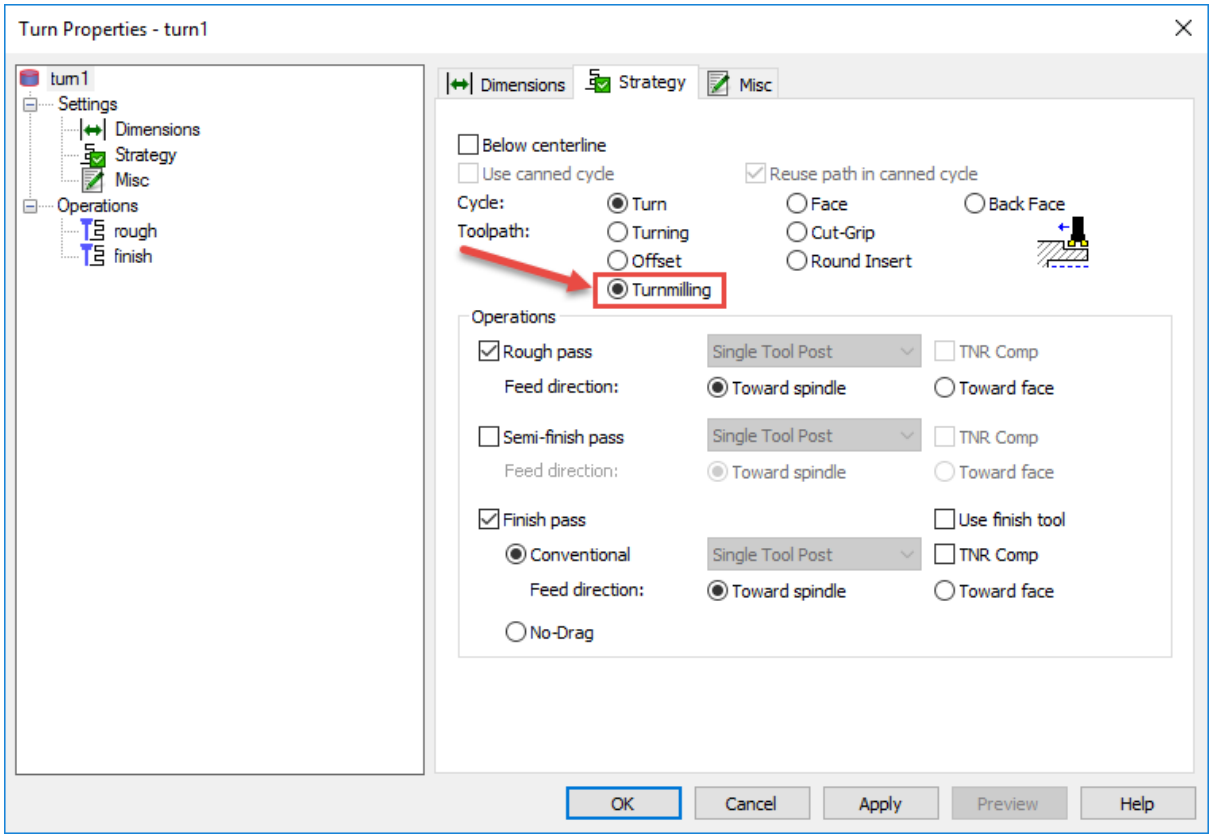
11 Select **From Dimensions** – **Face**. **OD 450mm,** Select **Finish**.

12 Select **From Curve** – **Turn**. Select **Curve1.**

13 Select **Next**.

14 Select **Finish**.

15 Select **Turn1** in **PartView** and you will be presented with the following menu.



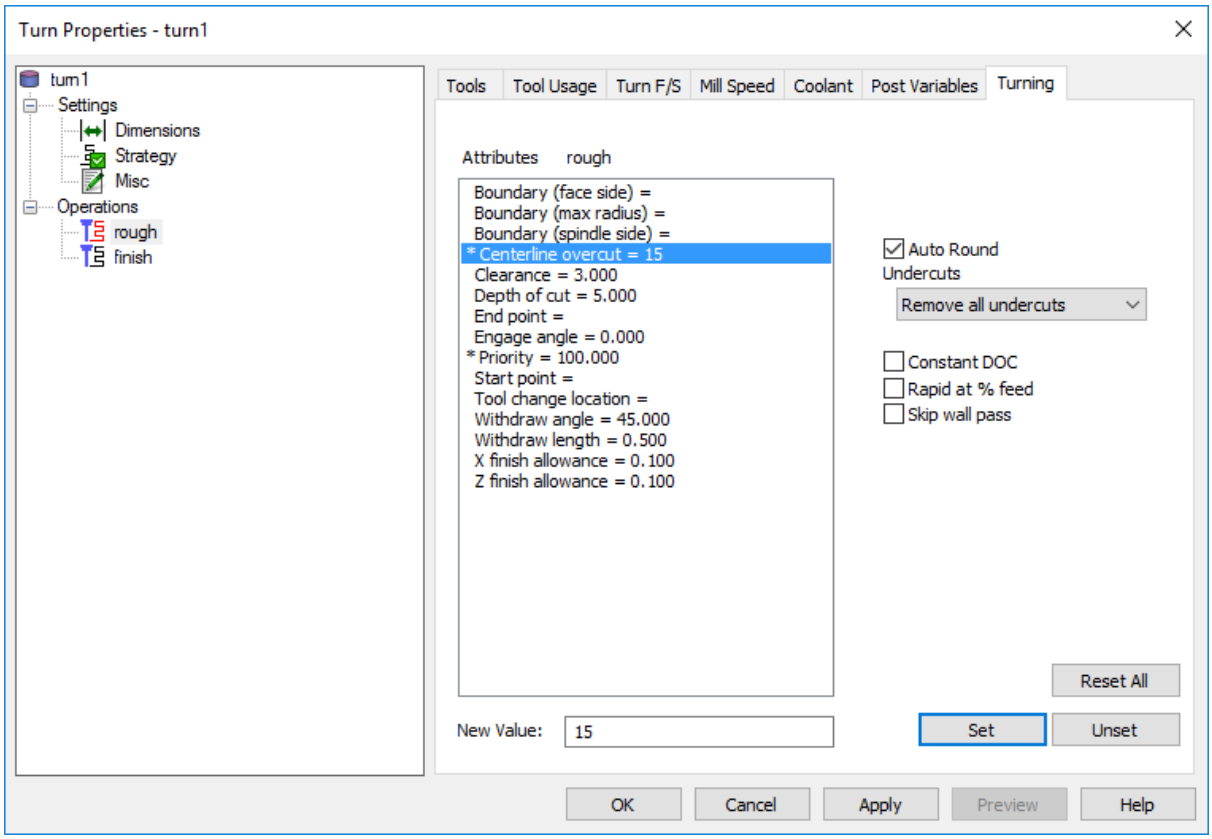
16 You can now see the **Turn milling** option in the **Strategy** Tab.

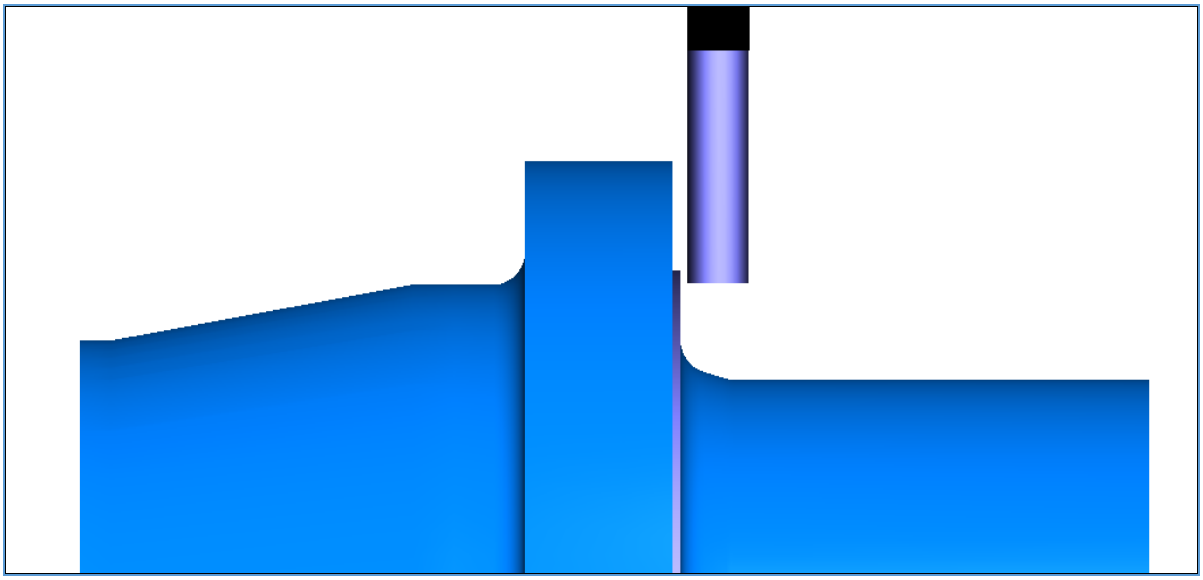
17 **Select** this option.

© Copyright 2016 Autodesk, Inc. All rights reserved

18 Select a larger Endmill for example **25mm** for the roughing operation.

19 We now need to change the **Centreline overcut** value to **15mm** as shown below.





20 The **Roughing operation** will now be roughed out using a **25mm** Endmill.

21 The **Finishing operation** will still be using a Turning tool.

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

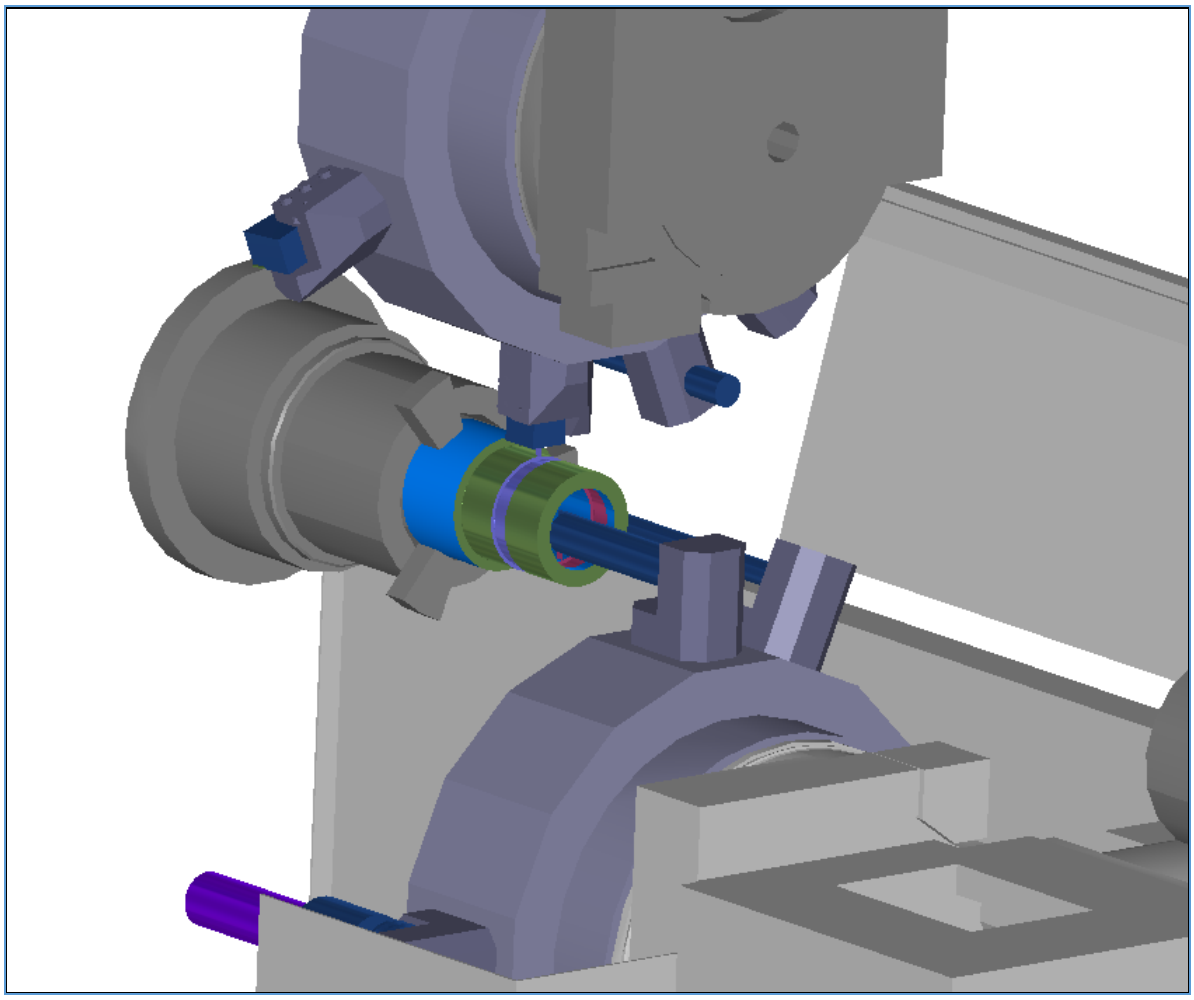
Synchronization Basics 2
Introduction 2

© Copyright 2016 Autodesk, Inc. All rights reserved

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 machines with up 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 shown below.



1. Open a new FeatureCAM document and Select **Turn/Mill**.
2. Open file 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 the ***Mori Seiki NZ 1500 T2Y Training Only.CNC*** *Post.*

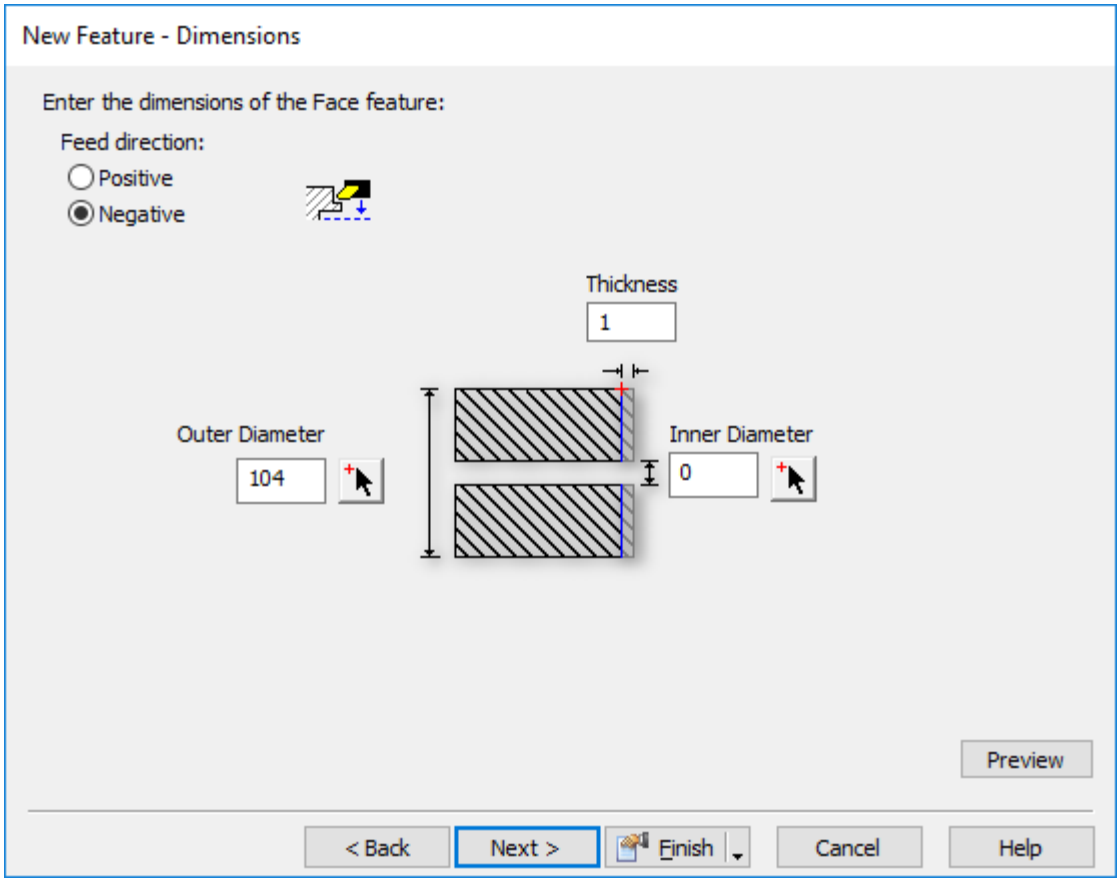
You will find the post in C:\Training_Data\FeatureCAM Course Data 2017\Training Posts.

© Copyright 2016 Autodesk, Inc. All rights reserved

We will make all turning features for this file using the curves that have already been created.

3 Select **Ctrl + R** or from **Steps** select **Features.** Select **Turning** and then **Next.**

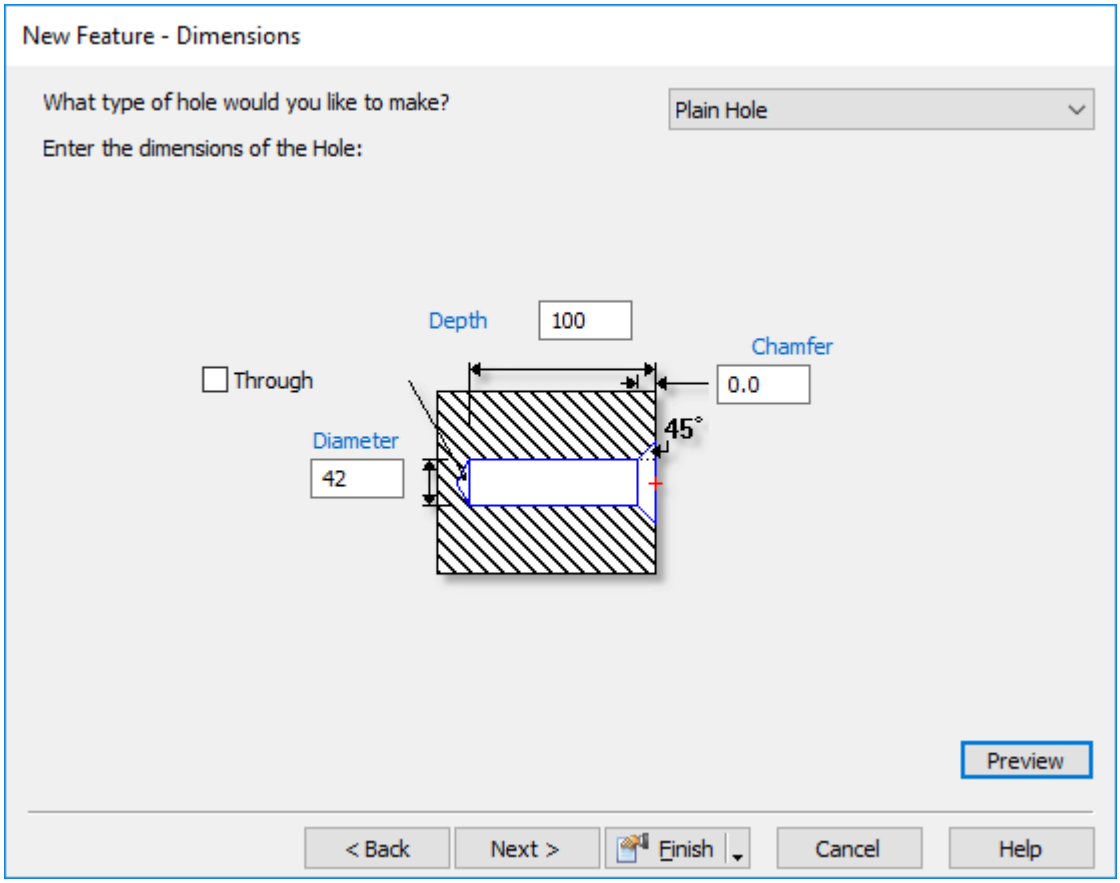
4 Select **Face** then **Next** and the following menu will appear. Select **Finish**.



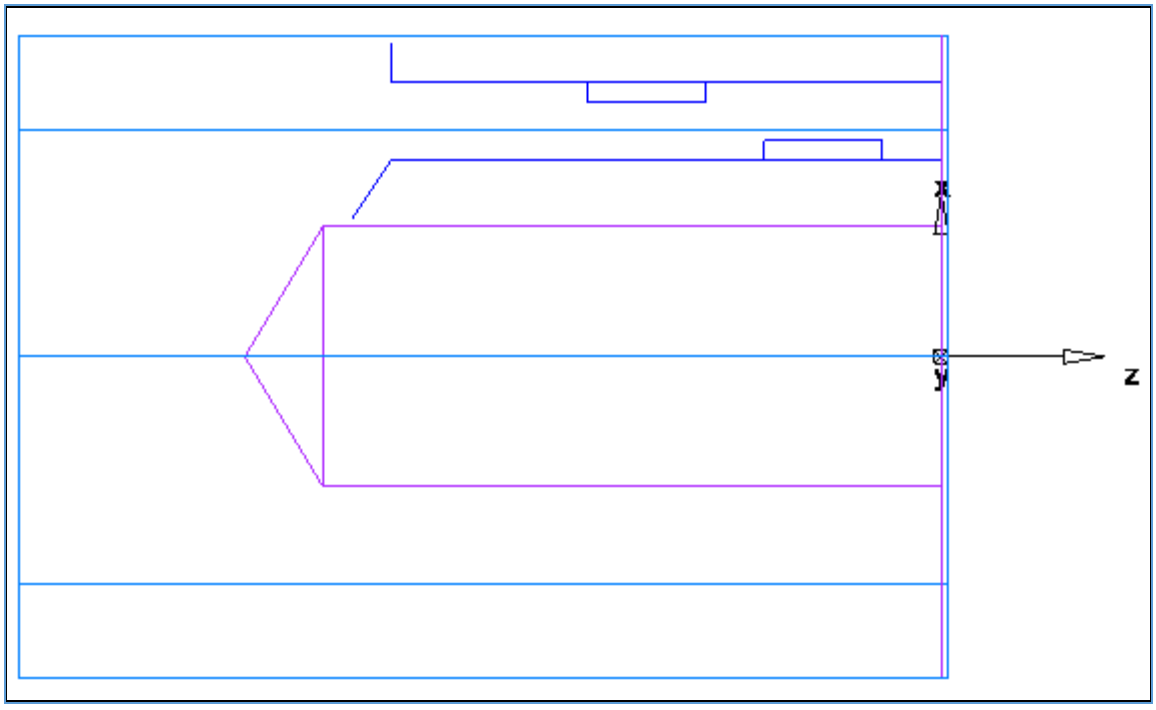
*Now let's make a **Hole** feature on the face of the part to open up the **Internal Diameter**.*

5 From the steps menu click on **Features** and choose **Turning** and press **next**. Select **From Dimensions**>**Hole** and then select **Next**. Make sure the parameters in the page are correct and click on **Finish**. You should enter a **diameter** of **42mm** and a **depth** of **100mm.** This drill will not be available from a standard **Crib** so you will have to make one.

© Copyright 2016 Autodesk, Inc. All rights reserved



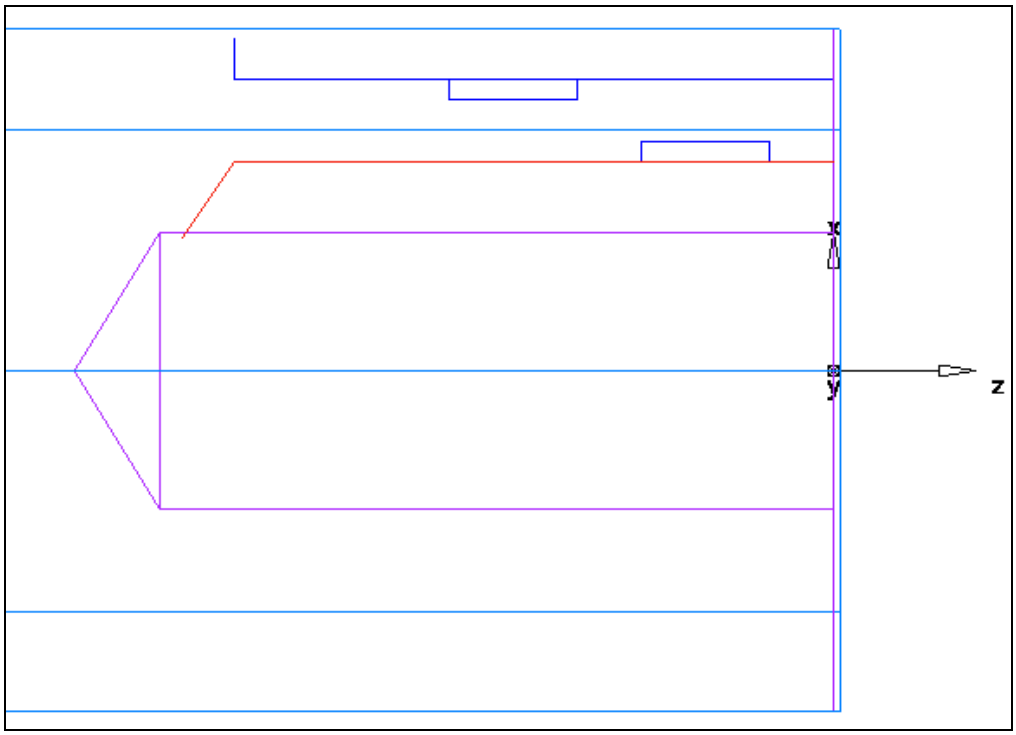
*This image below shows the **Drilled hole**.*



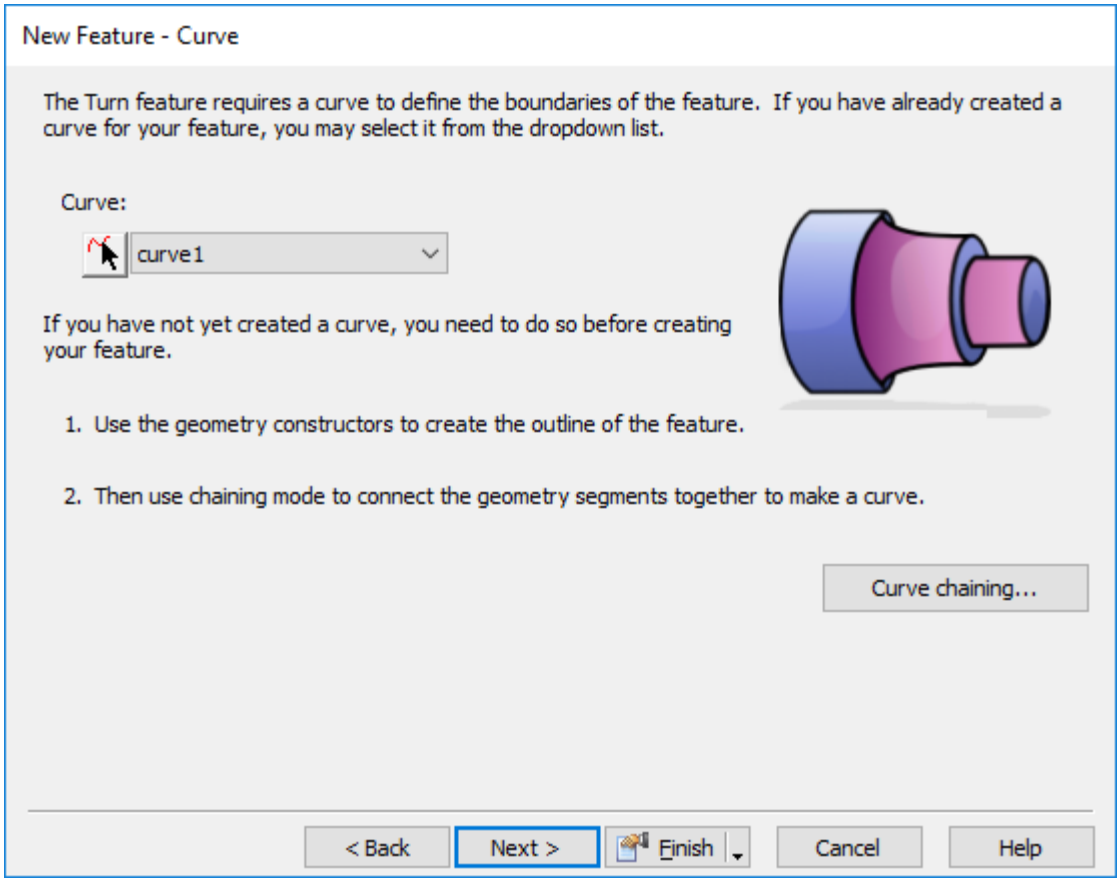
6 We are now ready to **Rough** and **Finish** the **internal Bore**. Press **Ctrl + R** this will activate the **New Feature** Menu - Select **Turning** then **Next**.

7 Select **Bore** and then select **Curve 2** shown below. Select **Finish**.

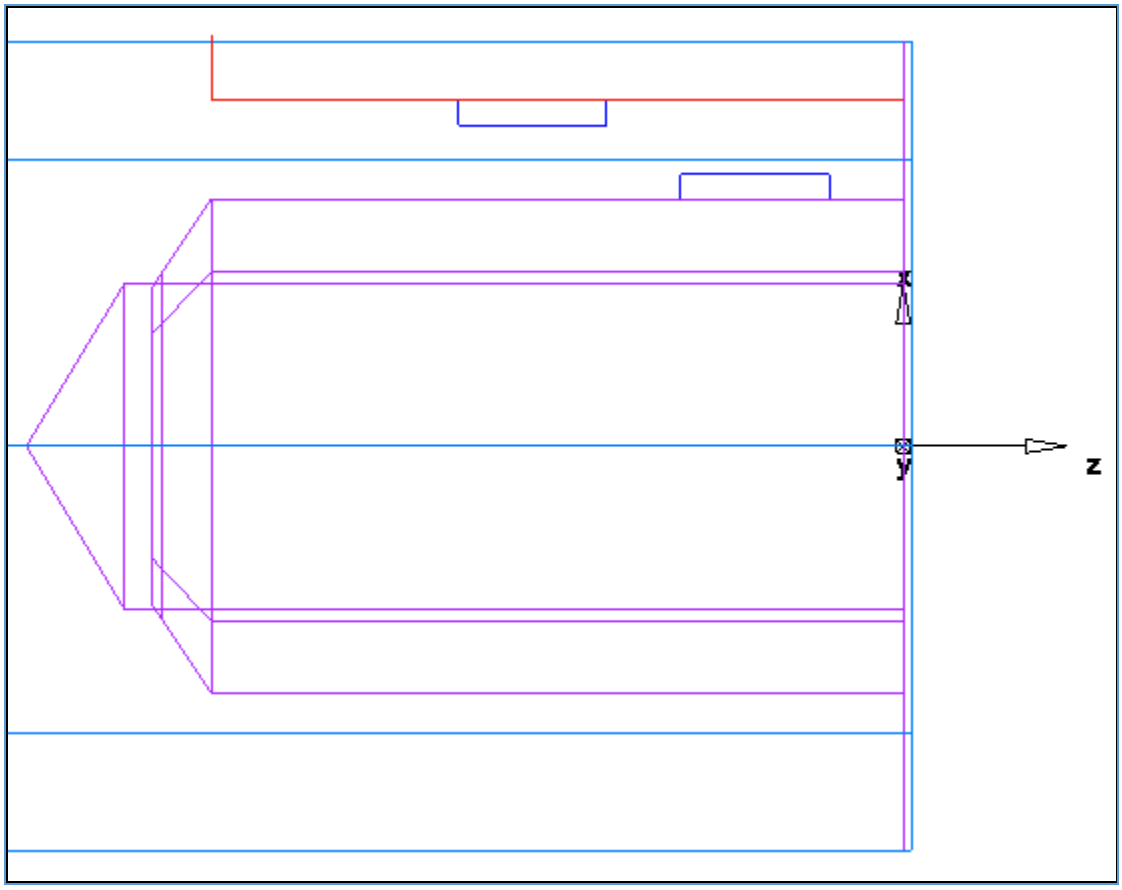
© Copyright 2016 Autodesk, Inc. All rights reserved



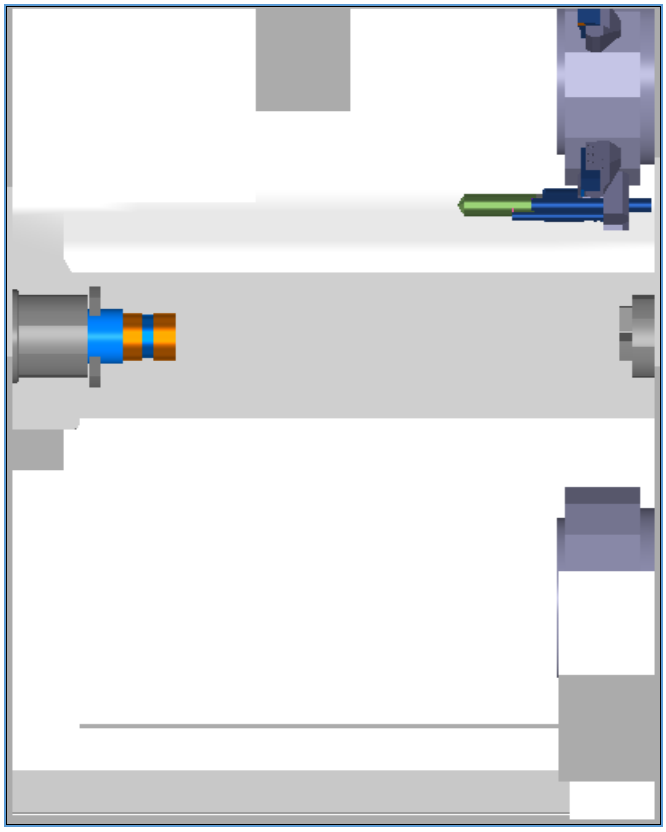
8 We are ready to machine the **Outside Diameter**. From the **steps** click on the **New Feature** wizard, choose **turning** and press **Next**. Select **From curve** > **Turn** and press **Next**. From the following menu on the next page select **Curve1**



© Copyright 2016 Autodesk, Inc. All rights reserved



9 Select **Finish** and run the **3DMachine Simulation.**



10 **Eject** the **Machine Simulation**.

© Copyright 2016 Autodesk, Inc. All rights reserved

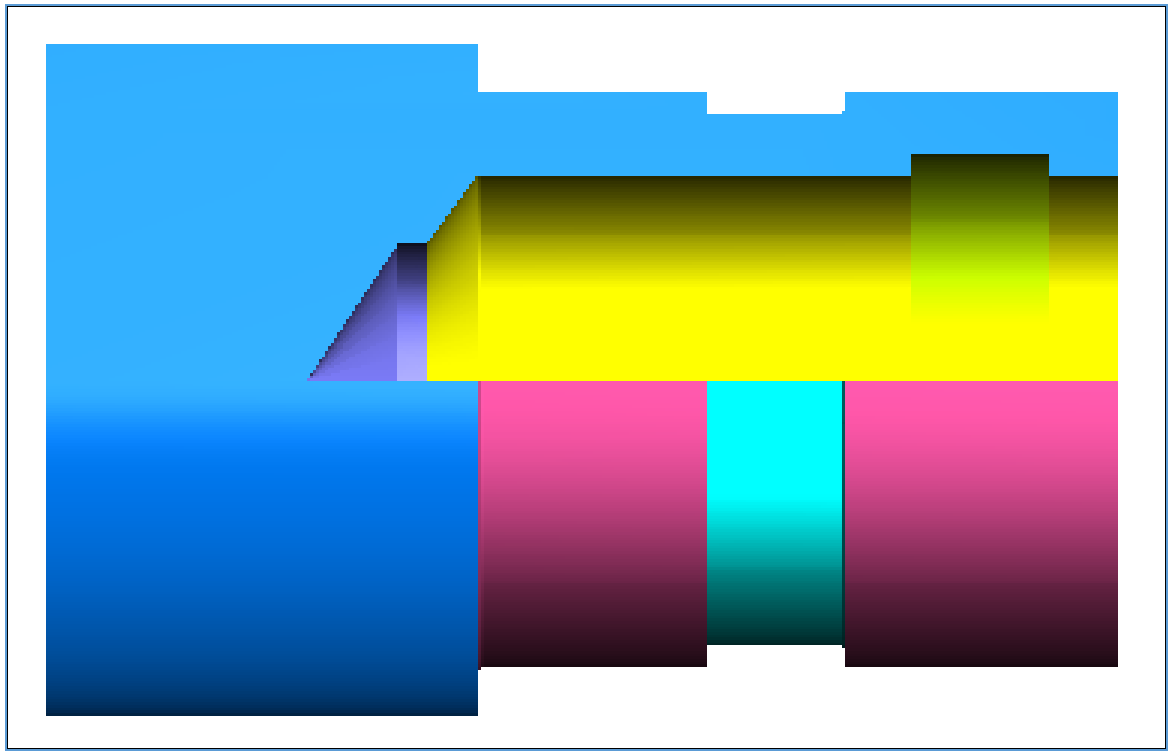
*Now we are ready to machine the **2 grooves**. We will use the curves available in the document.*

11 From the steps click on the **New Feature Wizard**, choose **Turning** and press **Next**. Select from **Curve** > **Groove** and press **Next**. Select **Curve3.**

12 It is important that you select the correct orientation of the feature. In this case this is an **OD** feature, so we do not need to change anything.

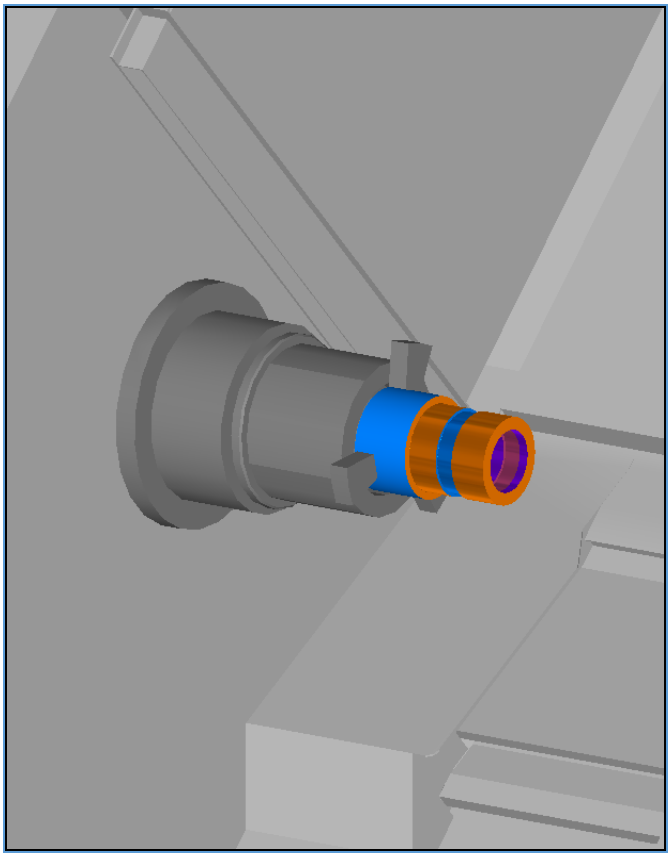
13 Now let's follow the same steps and machine the **inside groove** feature. Select **Curve4**.

14 Now pay attention to the **location**. You need to make sure that the location is **ID** for this groove. Select **Finish**. The image below shows **ID** and **OD** grooves complete.



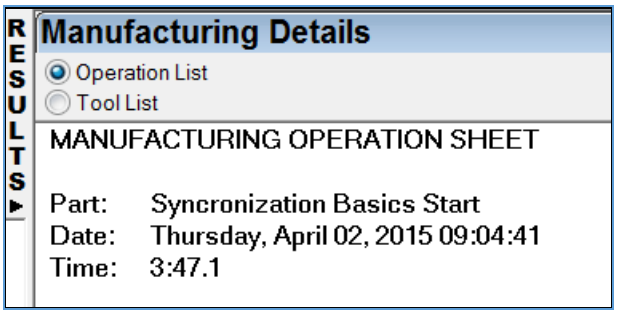
*Let's run a **machine simulation** and take a look at the finished machined part.*

© Copyright 2016 Autodesk, Inc. All rights reserved



Now, why do we need synchronization?? The part is finished; we didn't gouge the part as you can see from the machine simulation. 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.

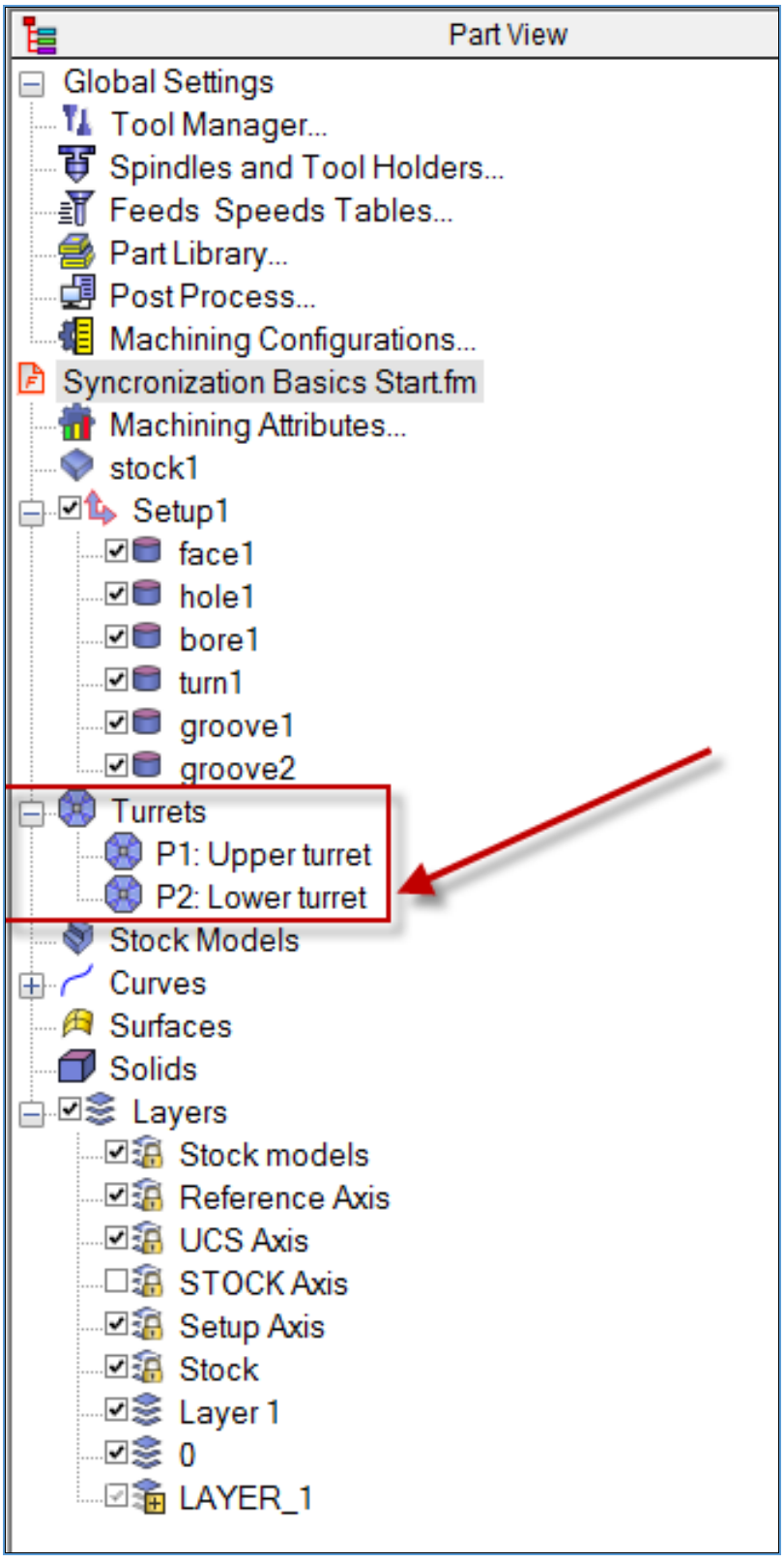
*Run a **centreline simulation**, then go to the **details** tab and look at the current time it takes to machine this file. Your results could be different based on tool grade selected, material, max travel, etc.*



Now let's analyse 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**.*

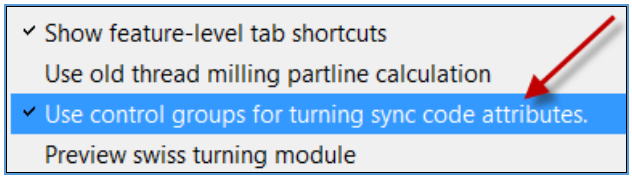
© Copyright 2016 Autodesk, Inc. All rights reserved



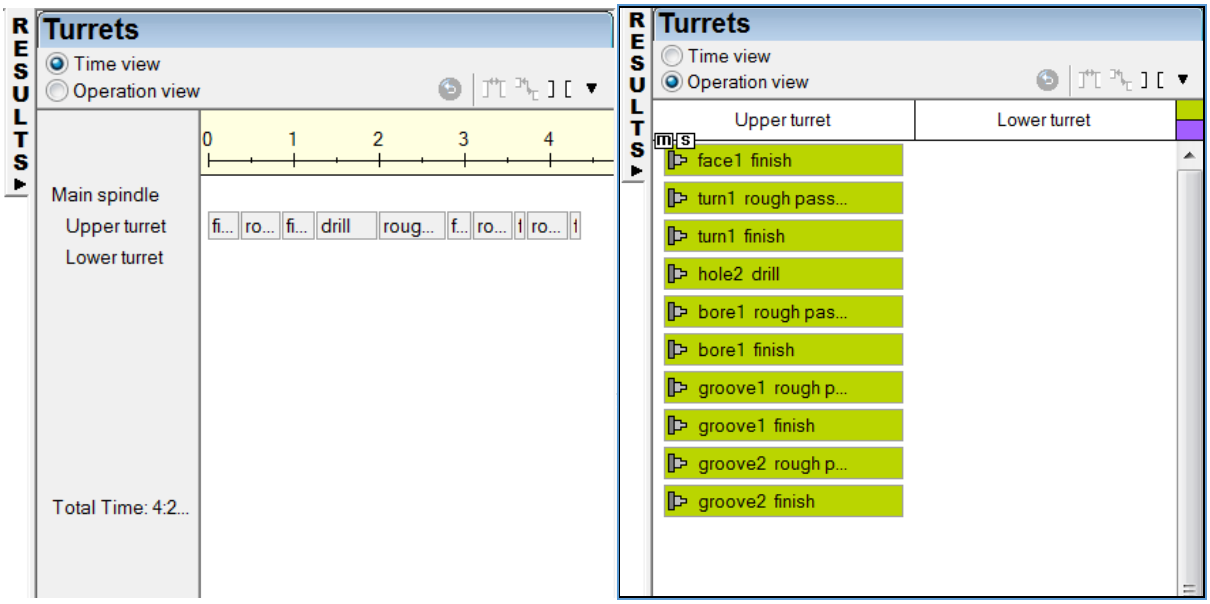
You will get a new tab in the results window called turrets. Turrets is where you can see what operations are being assigned to each turret

© Copyright 2016 Autodesk, Inc. All rights reserved

15 Using the secret menu please select Use control groups for turning sync code attributes as shown.



16 Eject the simulation and click on **turrets**. Turrets have 2 view modes that you can select. **Time based** and **Operation based**.

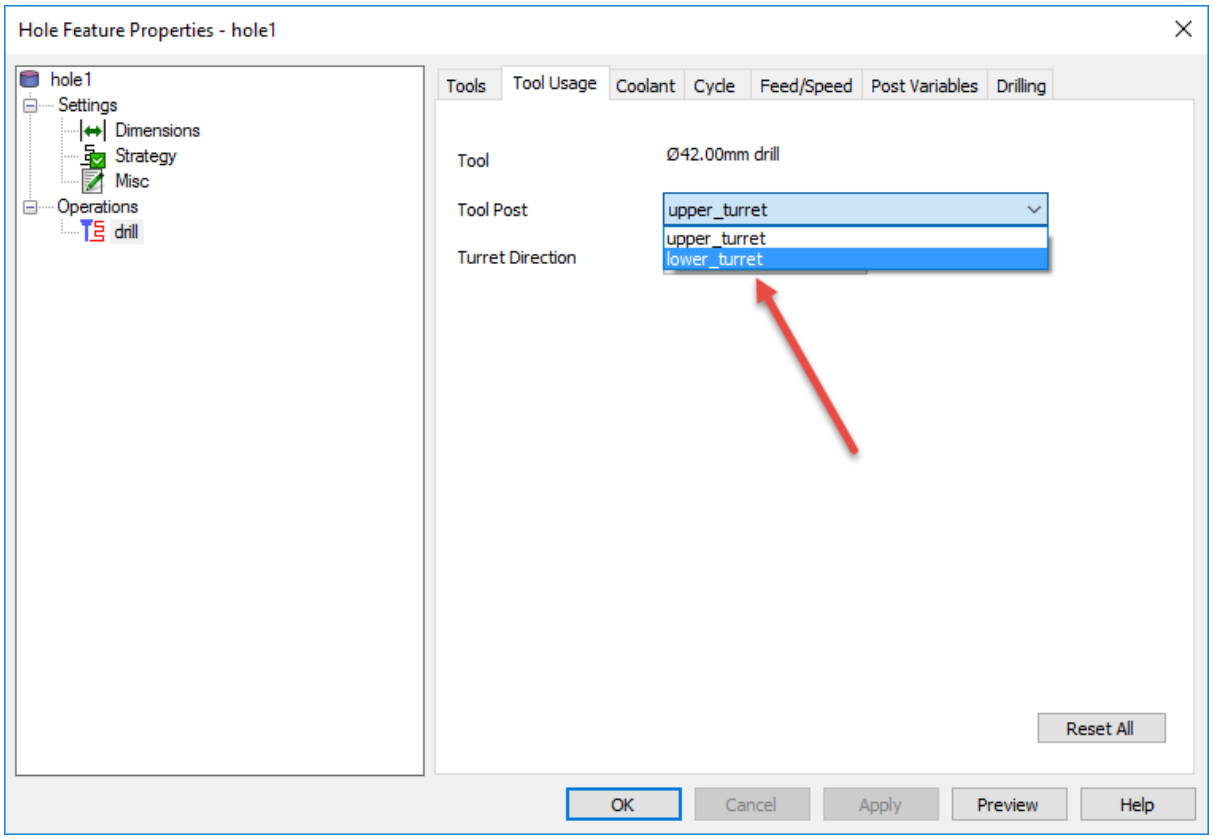


Time based offers a timeline that shows how long each operation takes. It is useful in the sense that you can visually check for operation time and adjust the order to accommodate for the shortest possible machining time. Operation based shows operations being displayed in columns. The operation based 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.

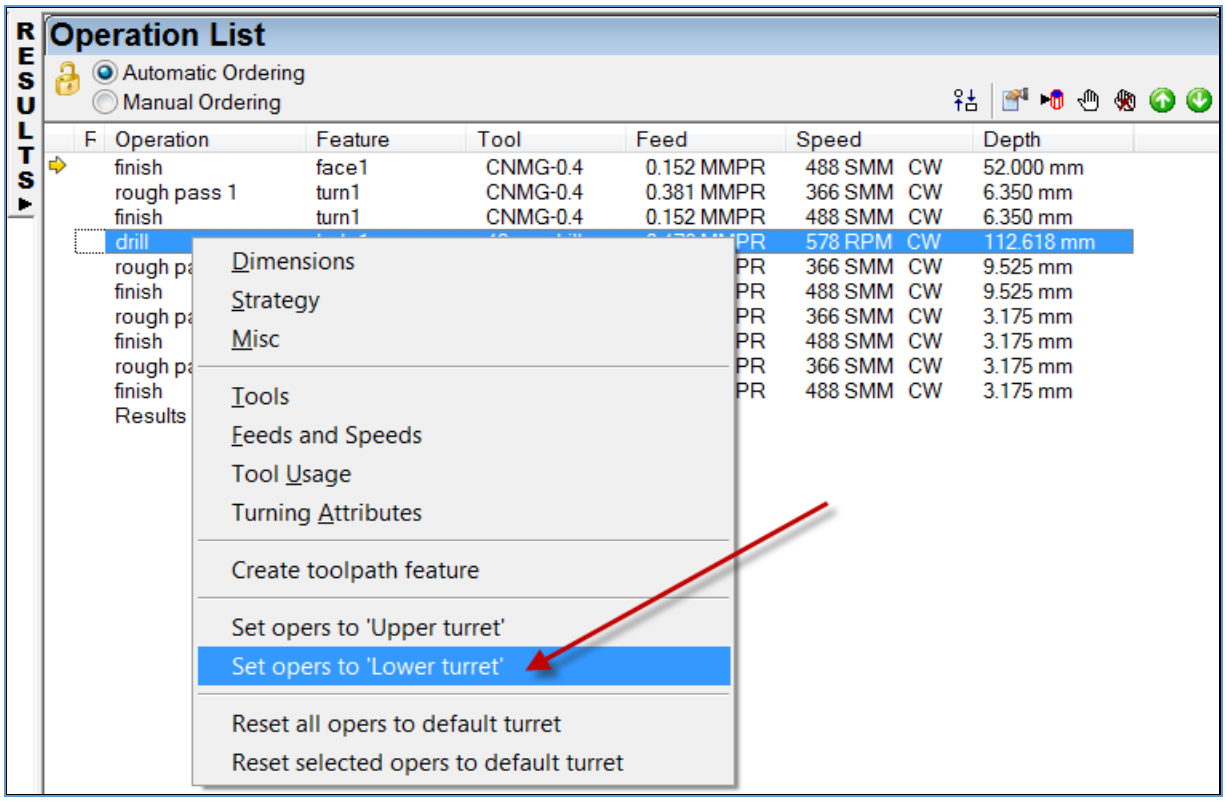
*Let's start by moving the drill operation **(hole1 drill)** to the lower turret. There're are multiple ways to achieve this:*

*By opening the feature at the **operation** level and switching from **upper** to **lower** turret*

© Copyright 2016 Autodesk, Inc. All rights reserved

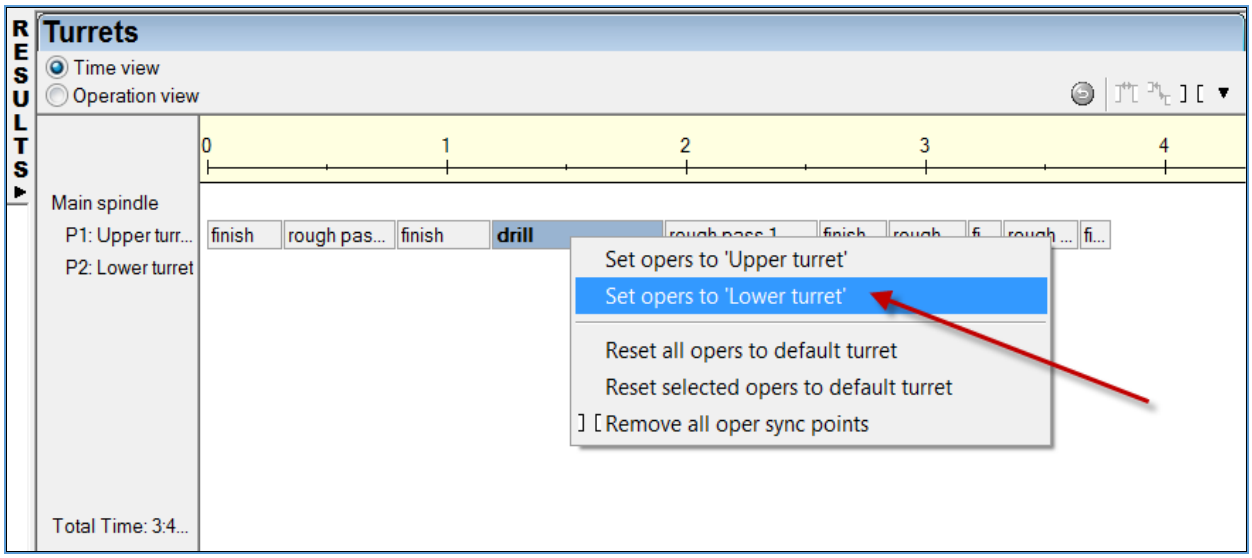


By right-clicking on the specific operation in the op list tab and selecting set operations to 'Lower Turret'. Note that you can move multiple operations with this method if you pre-select them at the same time.

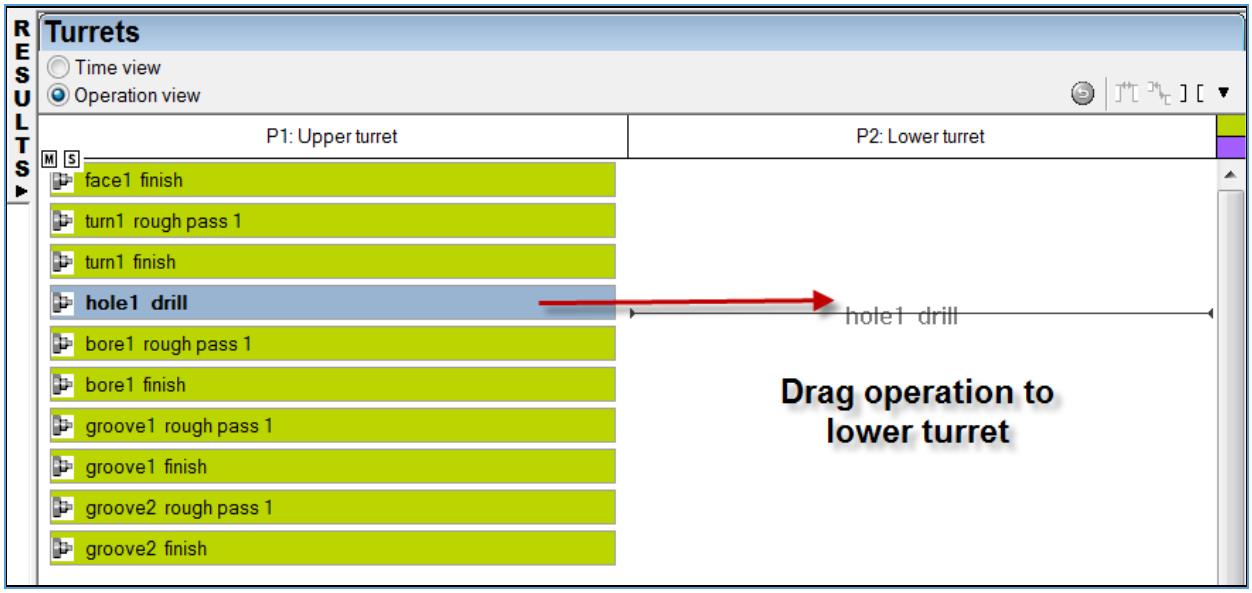


© Copyright 2016 Autodesk, Inc. All rights reserved

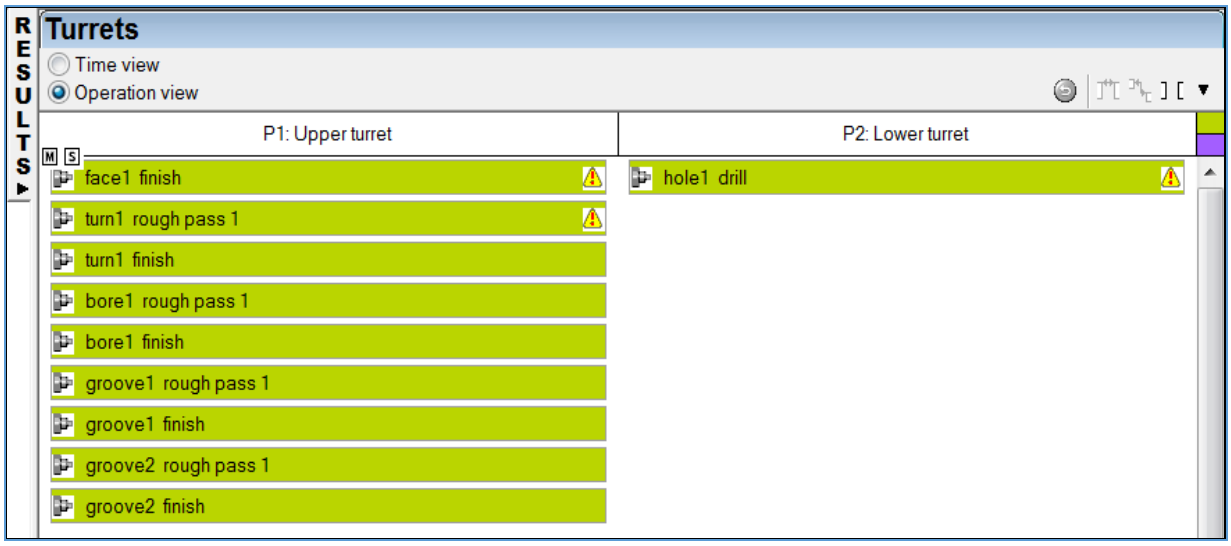
By right-clicking on the operation in the turrets tab > Time view and selecting the appropriate choice



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

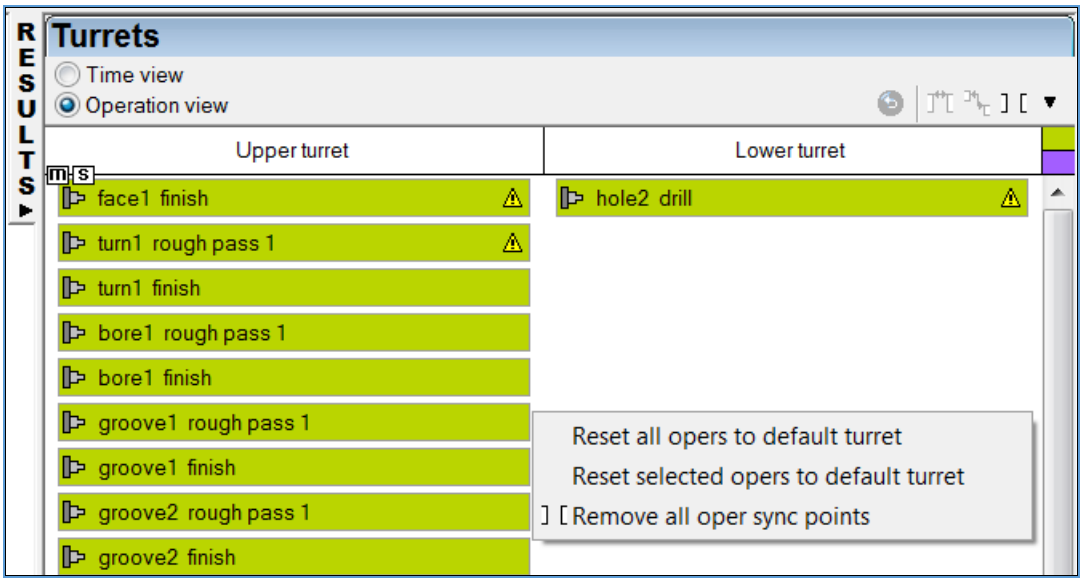


© Copyright 2016 Autodesk, Inc. All rights reserved



*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).

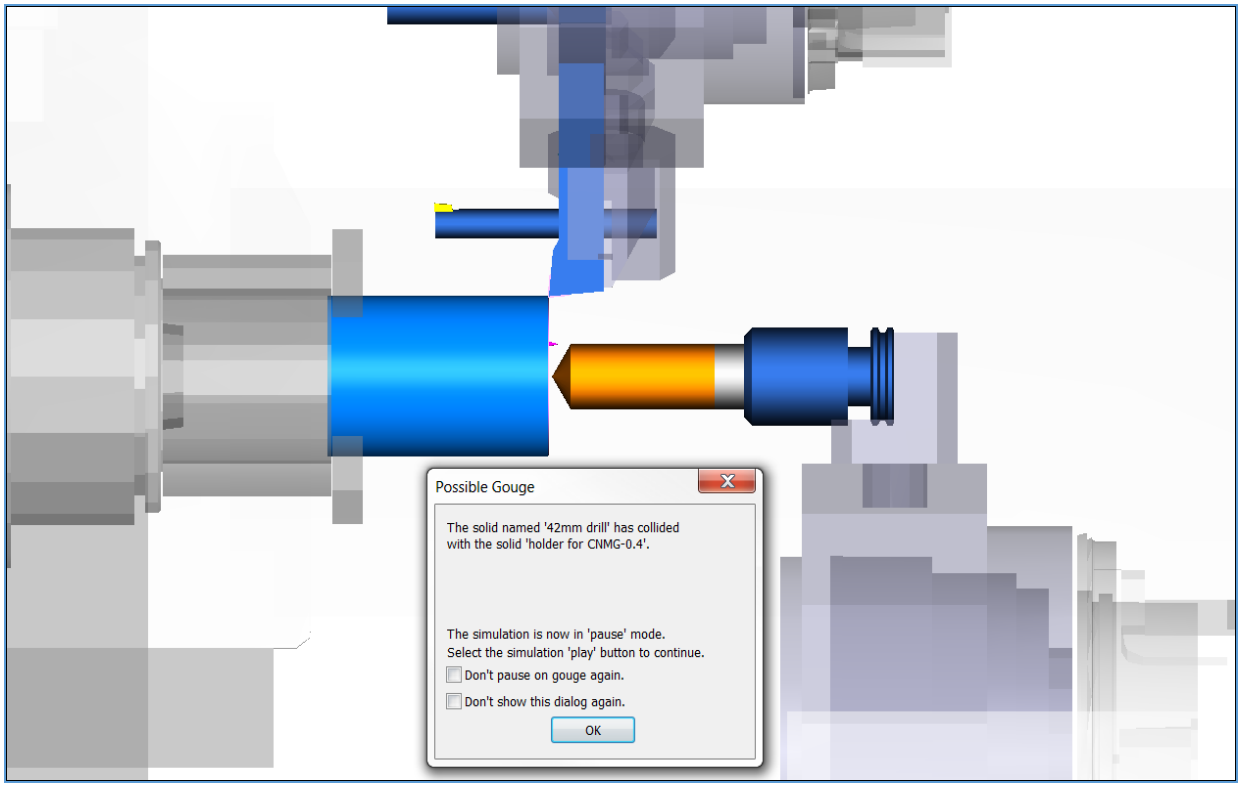


17 Let's try it. Left-click on the operation you want to move (**hole1 drill**) and then drag it to the lower turret.

© Copyright 2016 Autodesk, Inc. All rights reserved

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…

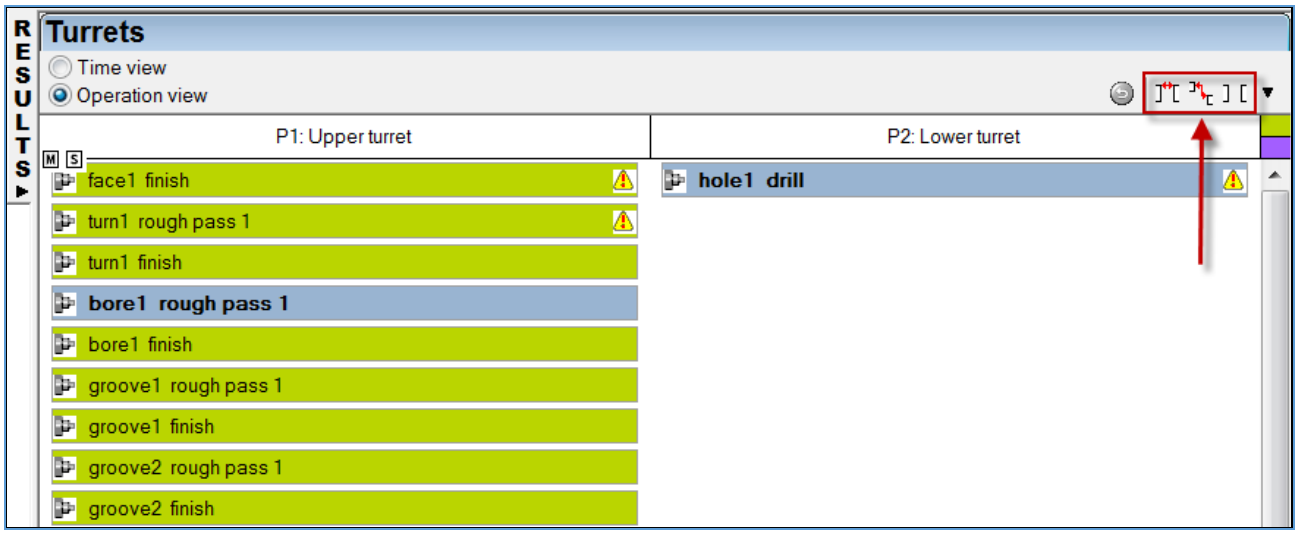
18 Let's run machine Simulation to see how things look so far. You should get a gouge like below



*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.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Let's first introduce the User Interface 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 User Interface (either in the time view or operation view). 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*

19 Let's try our first synchronization exercise:

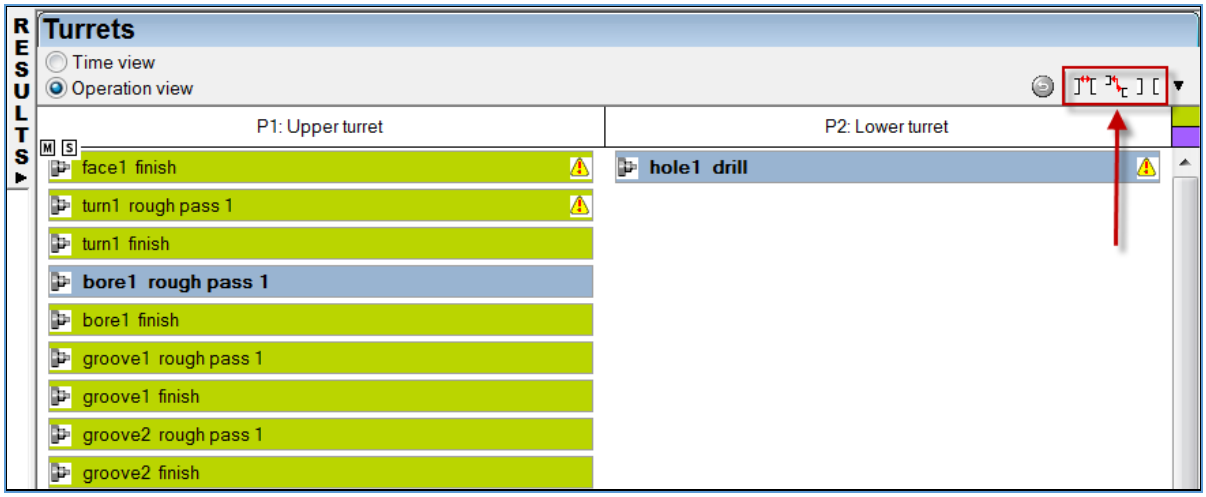
20 Hold the **Ctrl** Key – we need to synchronize 2 operations, therefore we need to select multiple operations with the **Ctrl** key

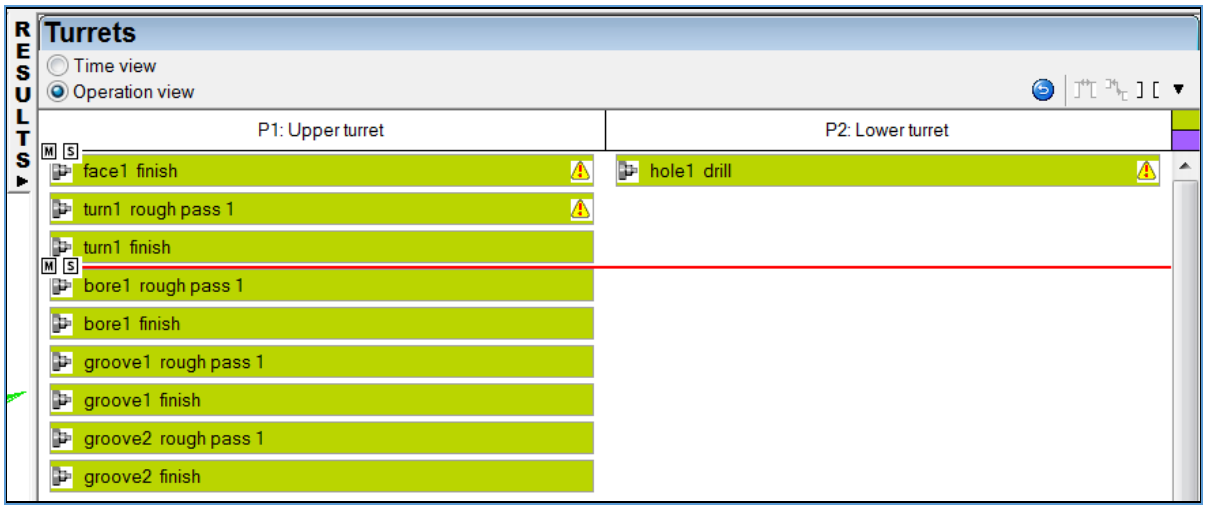
21 Click on the operation **hole1 drill (P2: Lower turret)**

22 Click on the operation **bore1 rough pass 1 (P1: Upper turret)**

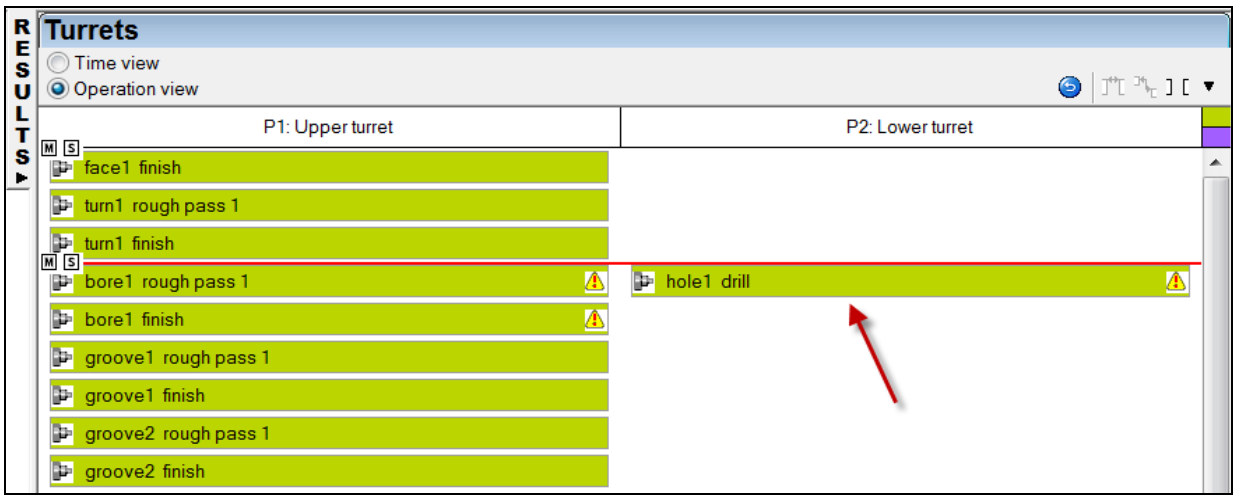
23 Click on the **middle sync** icon (one operation after the other)

© Copyright 2016 Autodesk, Inc. All rights reserved





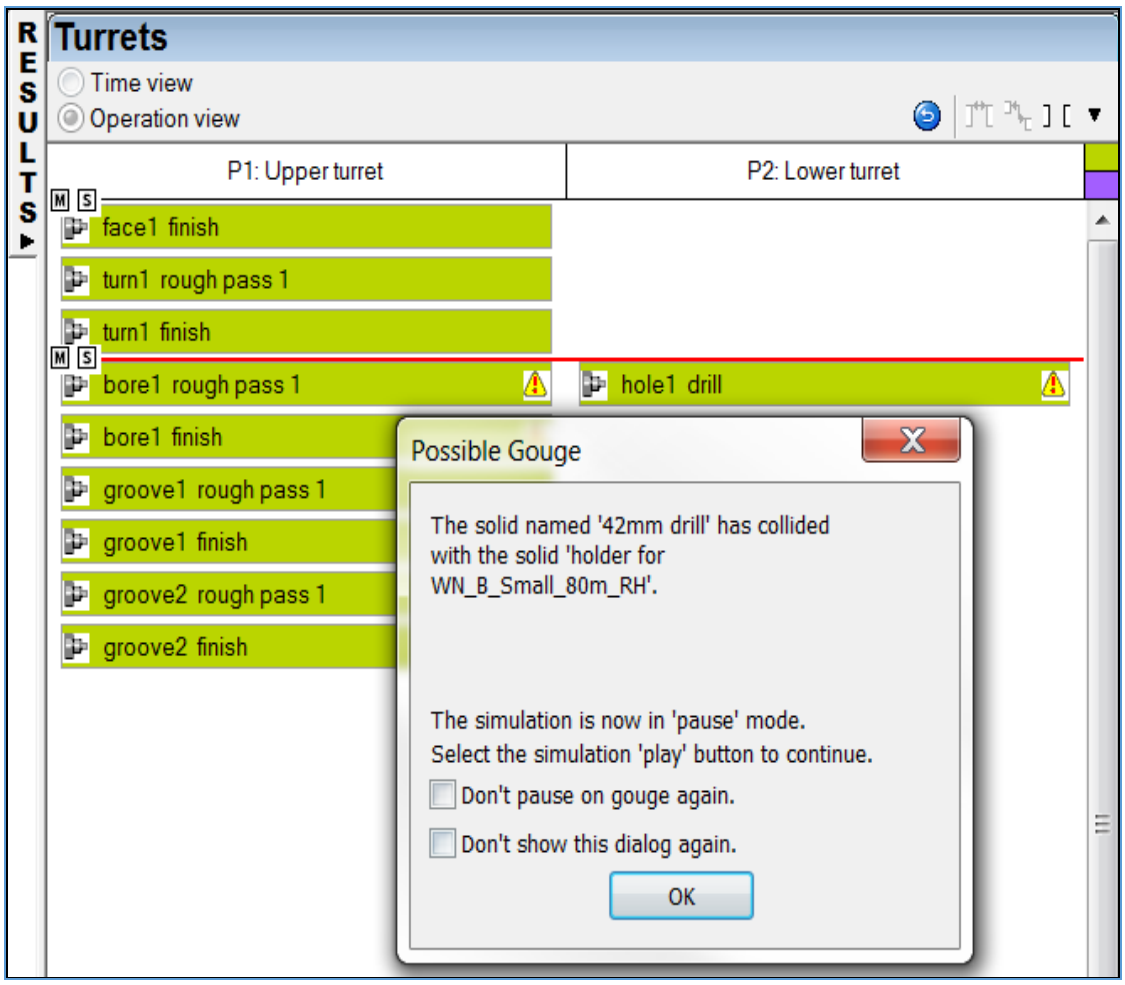
24 Now drag **hole1 drill** to below the sync point as shown below.



© Copyright 2016 Autodesk, Inc. All rights reserved

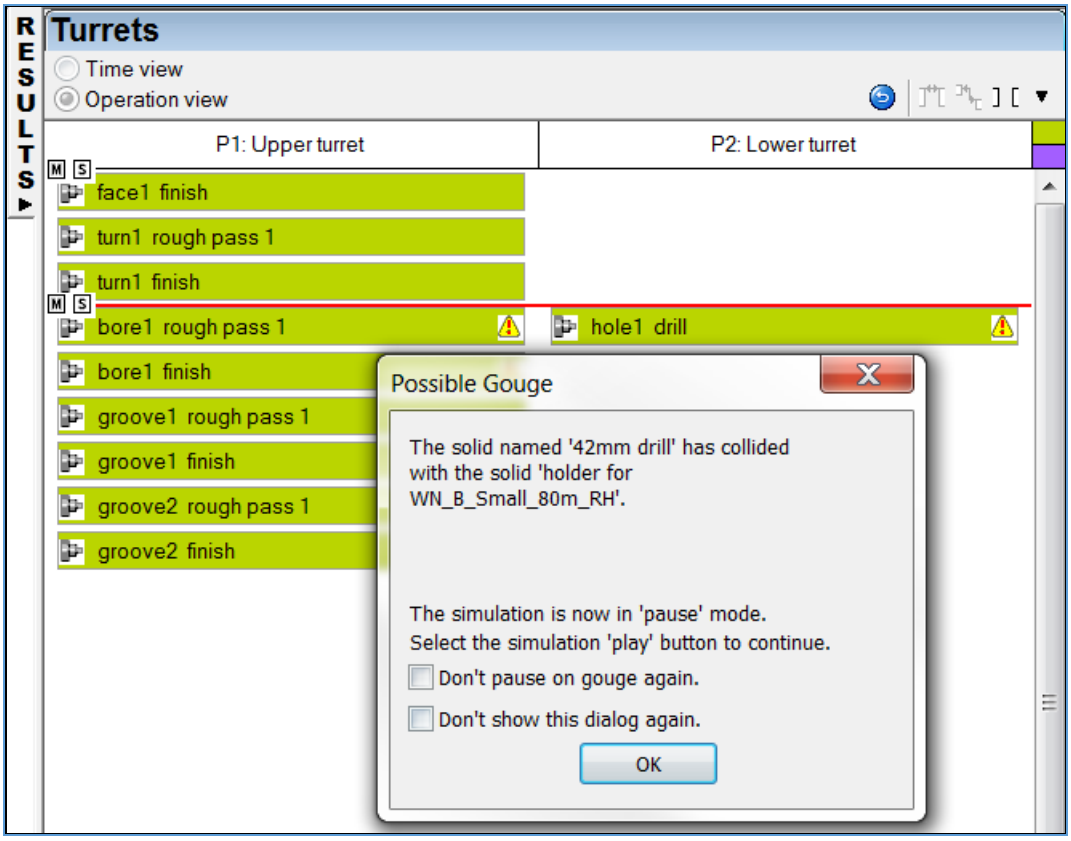
*Another way to synchronize that will not be covered in this tutorial is to select the operations with the **CTRL** key and then (with the operations selected) right-click on them and select the appropriate synchronization option. Both methods are valid and work in the same manner. Just different ways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

Now we have another possible gouge issue.

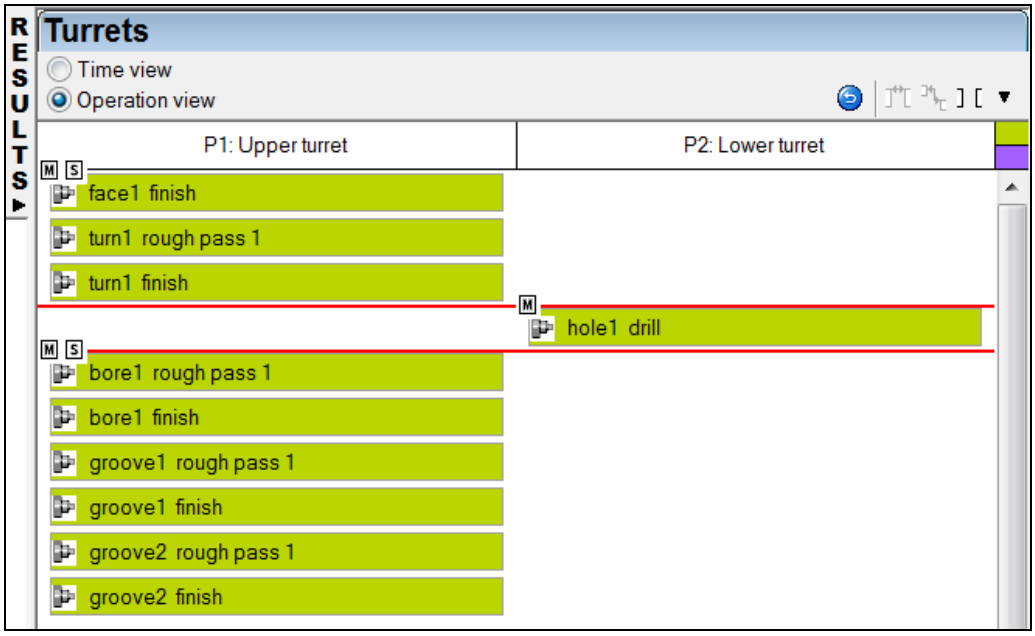


*We need to make sure that the **drill** operation will be done before machining the **bore1 rough pass** operation. Follow the steps above to make the **hole1 drill** happen before **bore1 rough pass**. Your turrets tab should look like the picture on the next page. Run machine simulation and see how the turrets now sit waiting for the next operation to happen.*

© Copyright 2016 Autodesk, Inc. All rights reserved



25 Move the bore operation after the drill operation as shown below.



If you now run a machine simulation you will see that there are no collisions.

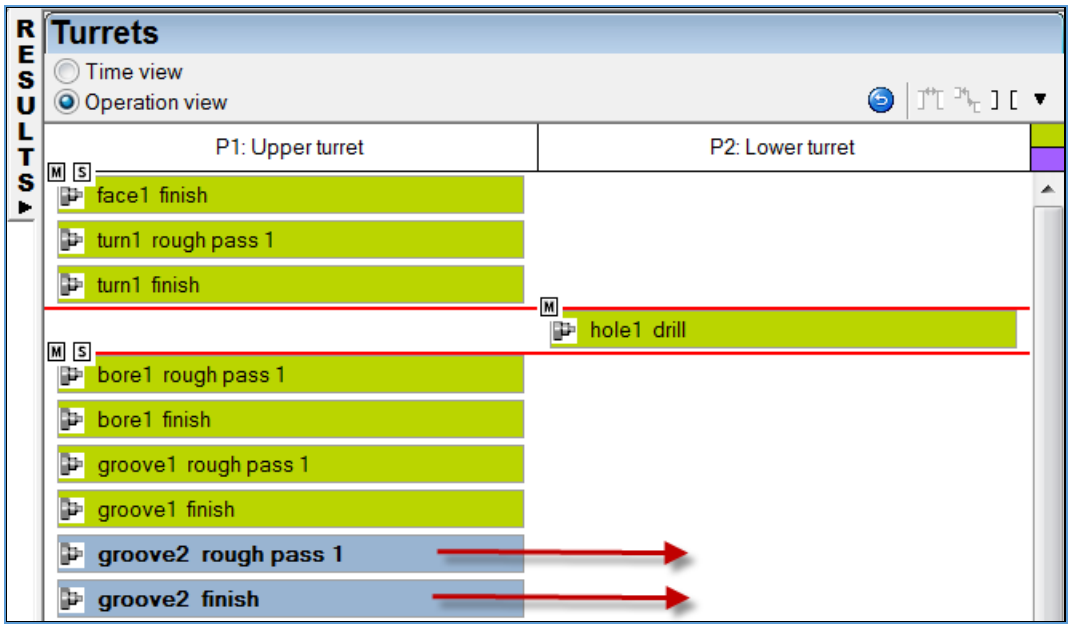
26 Move groove2 rough and finish to the lower turret.

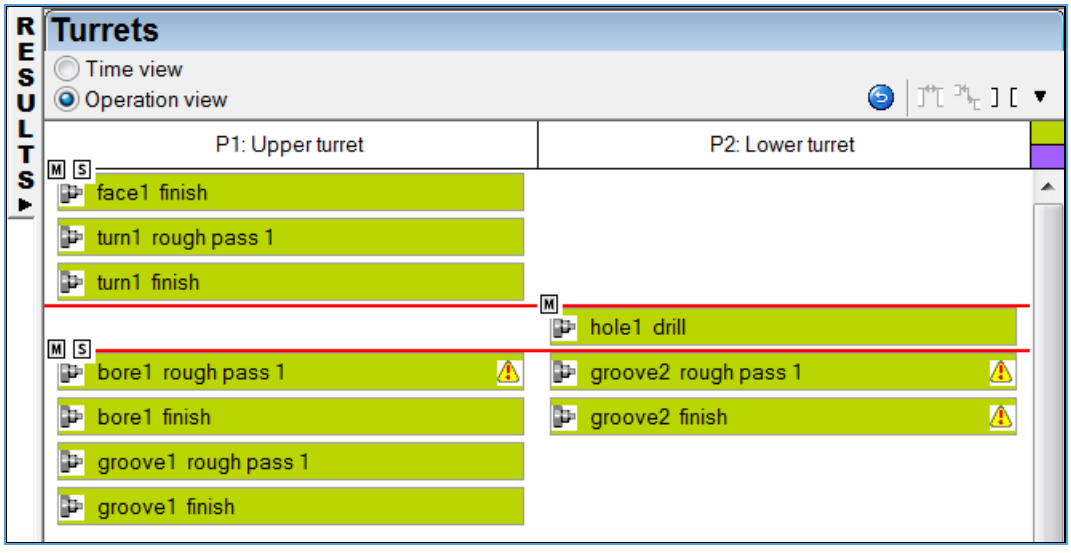
© Copyright 2016 Autodesk, Inc. All rights reserved

*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 sub-spindle 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 sub-spindle, the lower turret will control the speed.

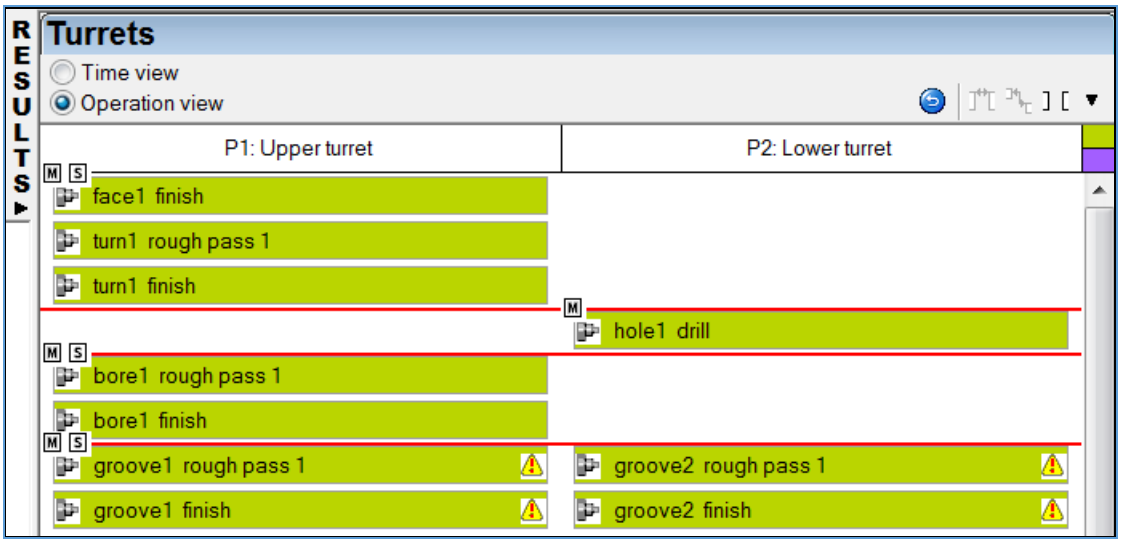
*To do this hold the **Ctrl** Key and select the 2 operations shown below. Then drag them to the lower turret*





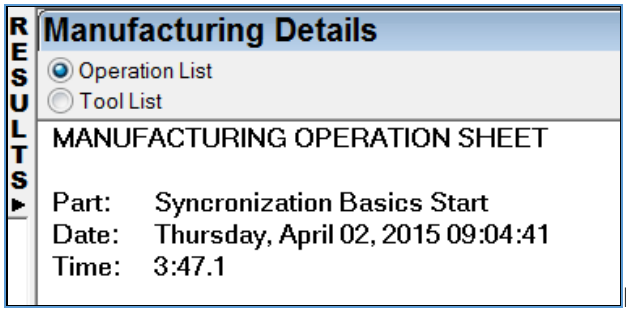
© Copyright 2016 Autodesk, Inc. All rights reserved

27 Now we need to set a sync point for the **groove2 rough and finish pass** with **groove1 rough and finish pass** as shown on the next page. **(set sync point at operation start)** Then drag the finish pass down as well.

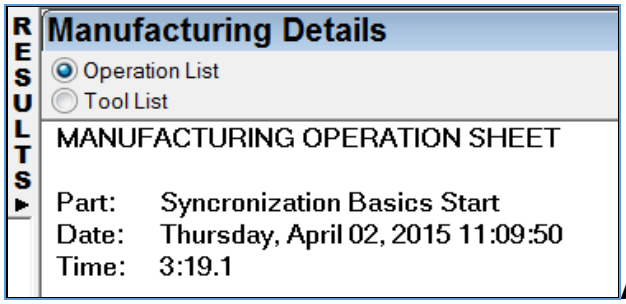


Let's look at the time. Now… see how much time we can save by synchronizing features. Remember, previous time without synchronization was approx. 3:47

Run centreline simulation and go to the details tab to look at the new time is now 3.19. So by synchronizing features, we save close to 30 seconds per part!



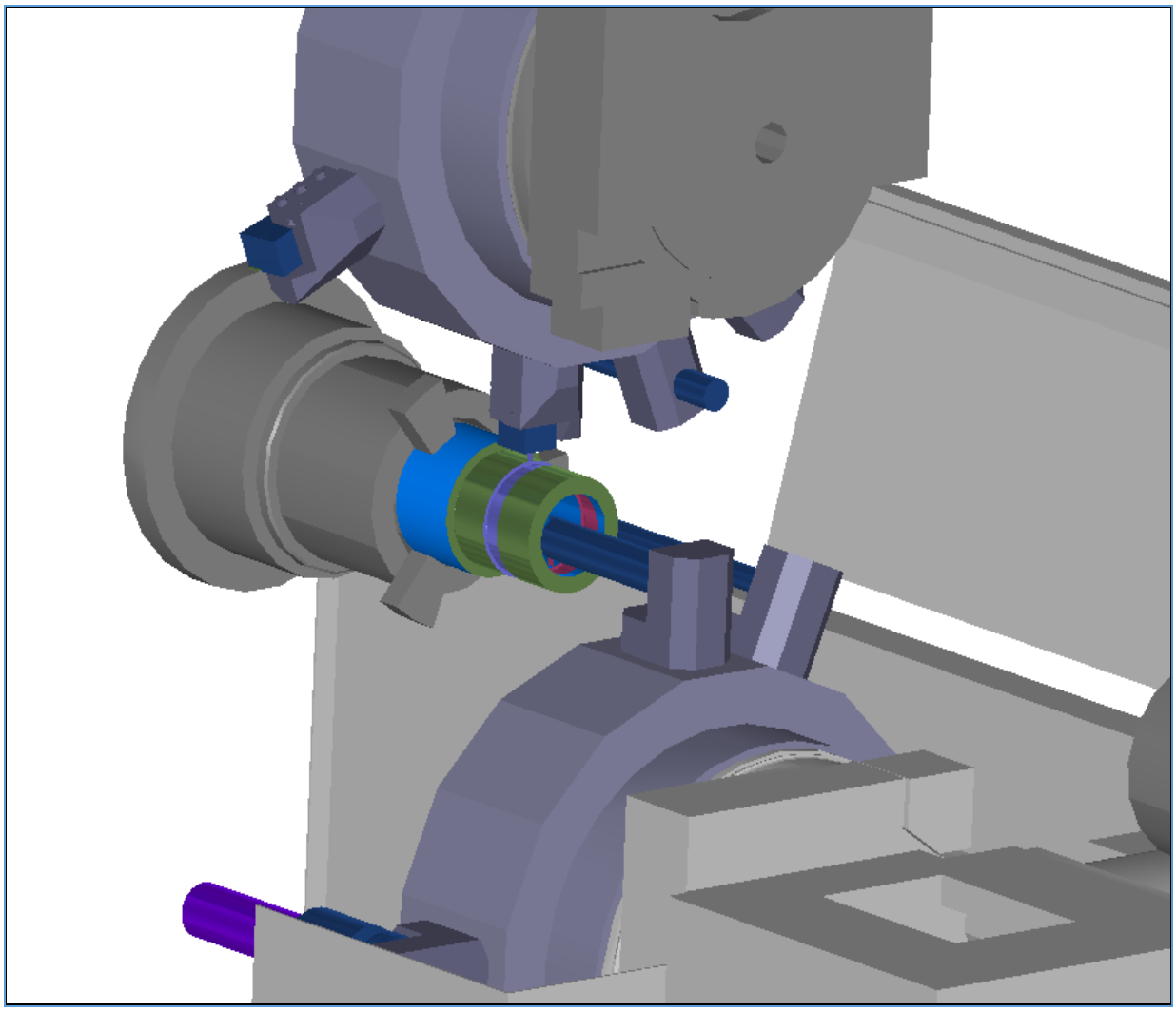
No Sync



After Sync

You're done!!! *Your simulation should look like the one on the next page. There should be no gouges or collisions.*

© Copyright 2016 Autodesk, Inc. All rights reserved



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

© Copyright 2016 Autodesk, Inc. All rights reserved

This page is intentionally left blank

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Integrated part handling (TURN) 2
Transfer Turret Control 4
Transfer Turret Control Dialog 5
Part Handling feature types 6
Slug Transfer 7
Reverse Slug Transfer 7
Bar Pull 8
Part Support Off 9
Cutoff Feature (from Dimensions) 9
An Overview of how Transfers Work (Information Only) 10
Overview 10
Key Positioning Parameters 14
Sub Position in NC Code 14
Cumulative Prior Pulls 14
Part Support Interaction with the Other UDFs 15
Cutoff Parameters 15
Part Handling Parameters 15
Initial Turret Motion 16
Limitations on Machine Simulation 16
Setting up the Post 17
Part Handling Exercise (Optional Exercise) 17

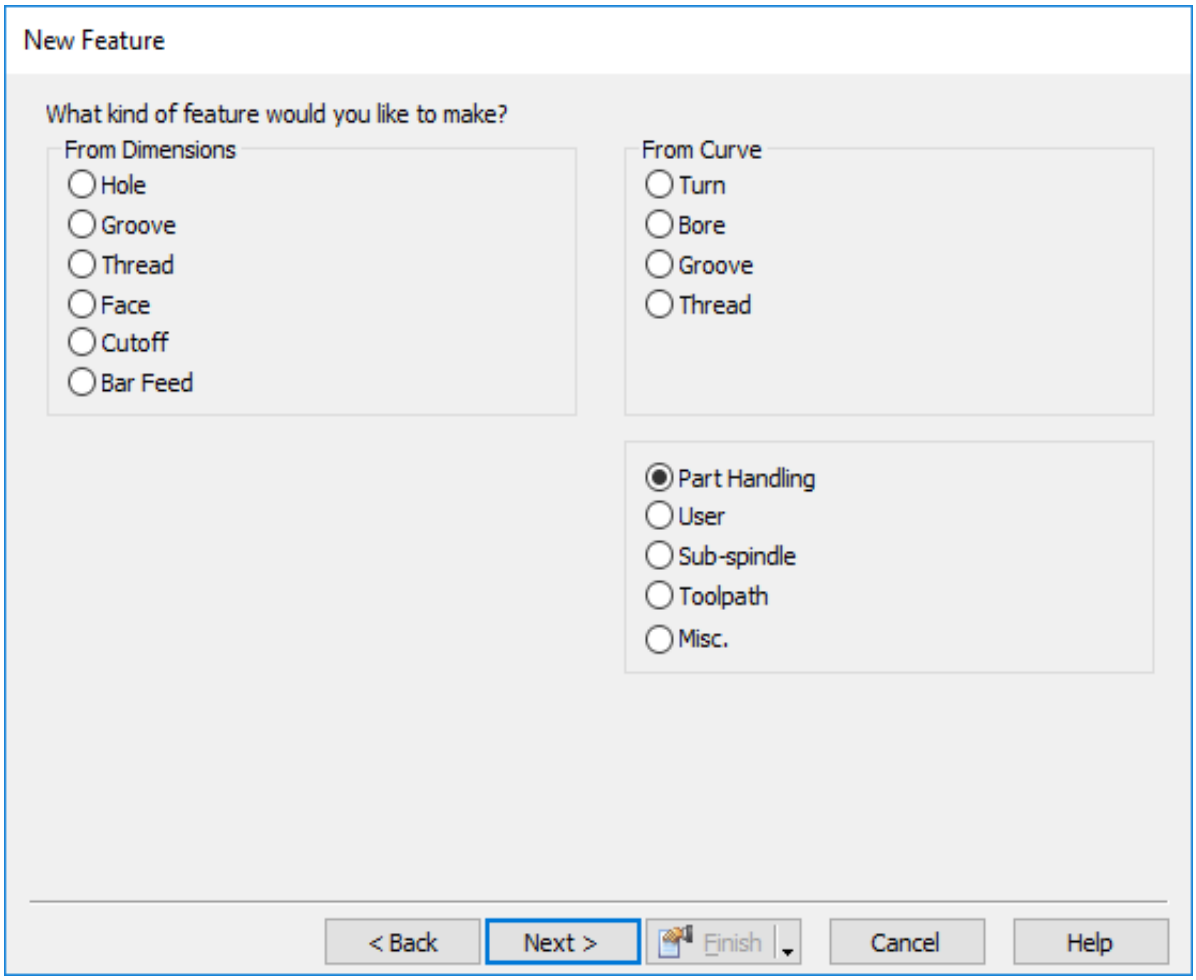
© Copyright 2016 Autodesk, Inc. All rights reserved

Integrated part handling (TURN)

- Part transfers are now fully integrated. The transfer types that are available are shown below:
- **Slug transfer**
- **Reverse slug transfer**
- **Bar pull**
- **Part support on**
- **Part support off**

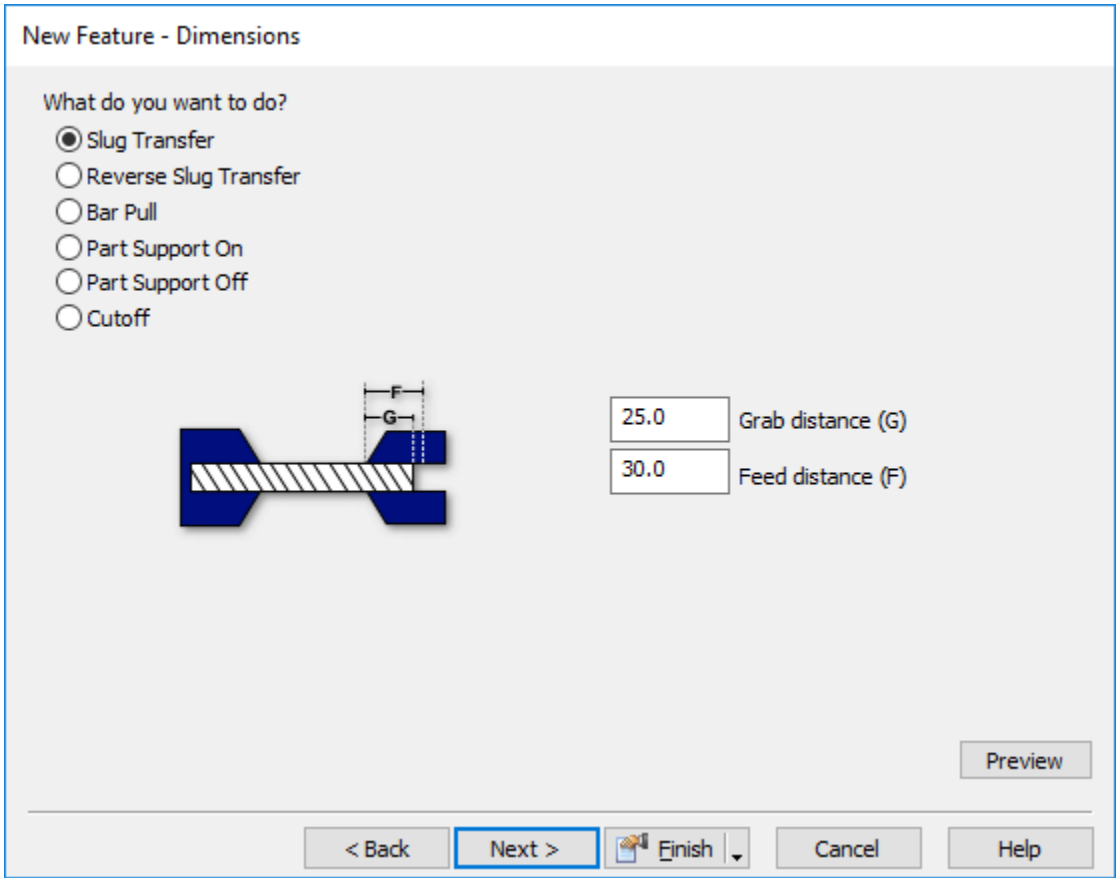
Post and simulation are supported.

- So how do we create a Part Handling feature?
- There is a new **Part Handling** feature found in the Turn **New Feature** wizard:

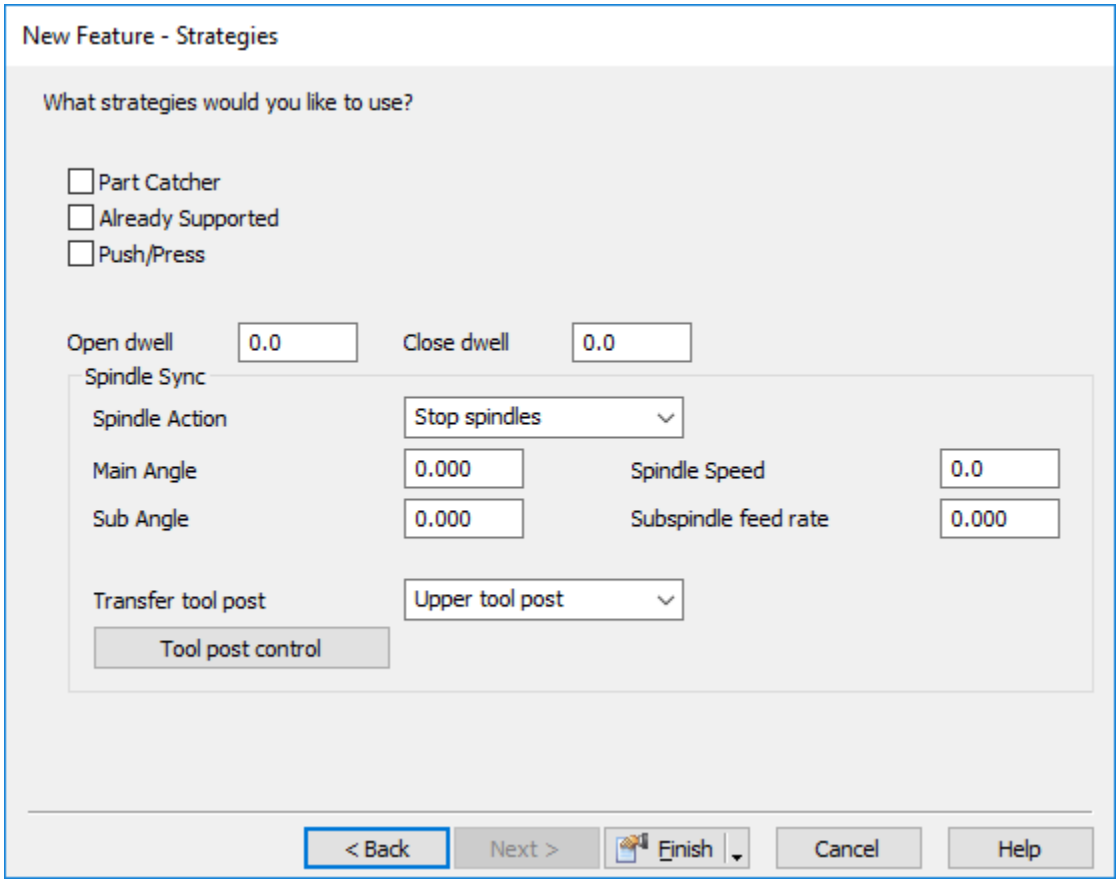


1 Select **Part Handling** and click **Next** to open the **New Feature** - Dimensions page:

© Copyright 2016 Autodesk, Inc. All rights reserved



2 Select the type of **Part Handling** feature you want and enter the dimensions. Click **Next** to open the **New Feature - Strategies** page:



© Copyright 2016 Autodesk, Inc. All rights reserved

3 Set the attributes for the Part Handling feature, for example:

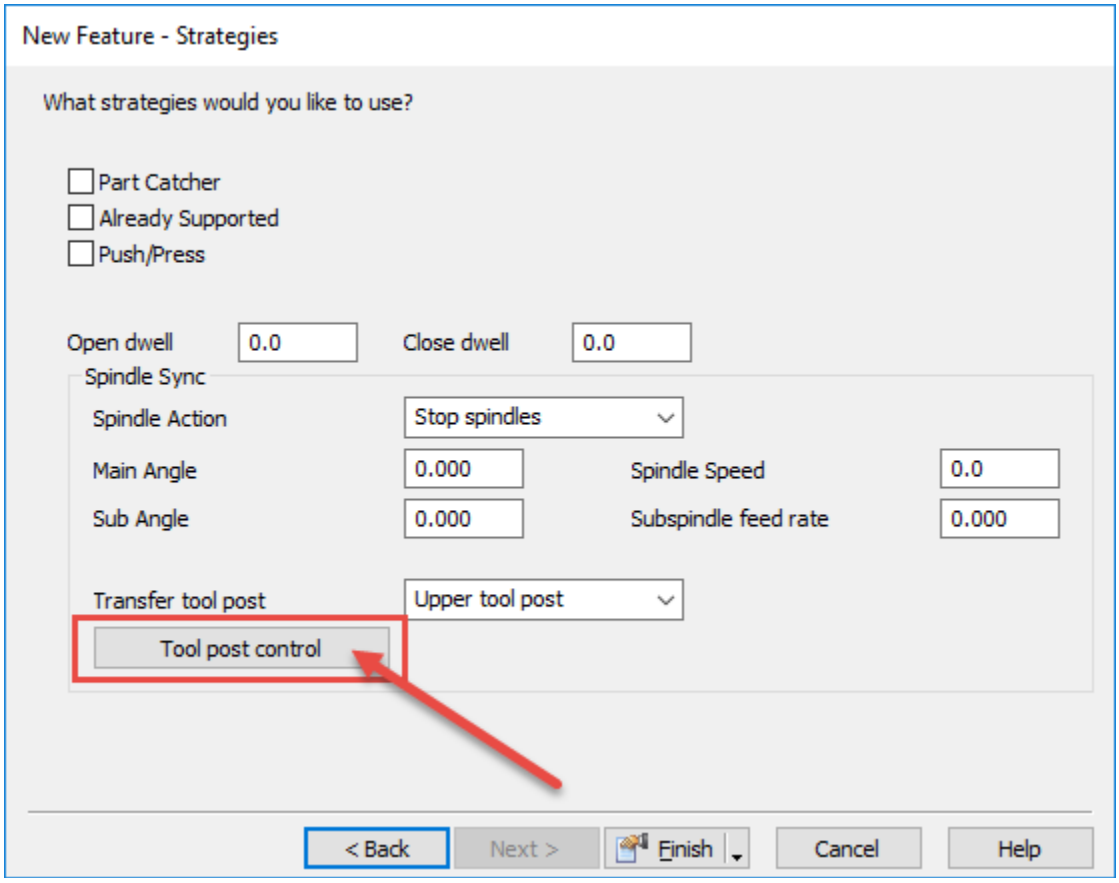
4 **Part catcher** — Select this option if you want to instigate the part catcher.

5 **Already Supported** — Select this option to indicate that the part is already supported.

6 **Sub Angle** — Enter the angle that you want the sub-spindle to rotate to before it moves to collect the part.

Transfer Turret Control

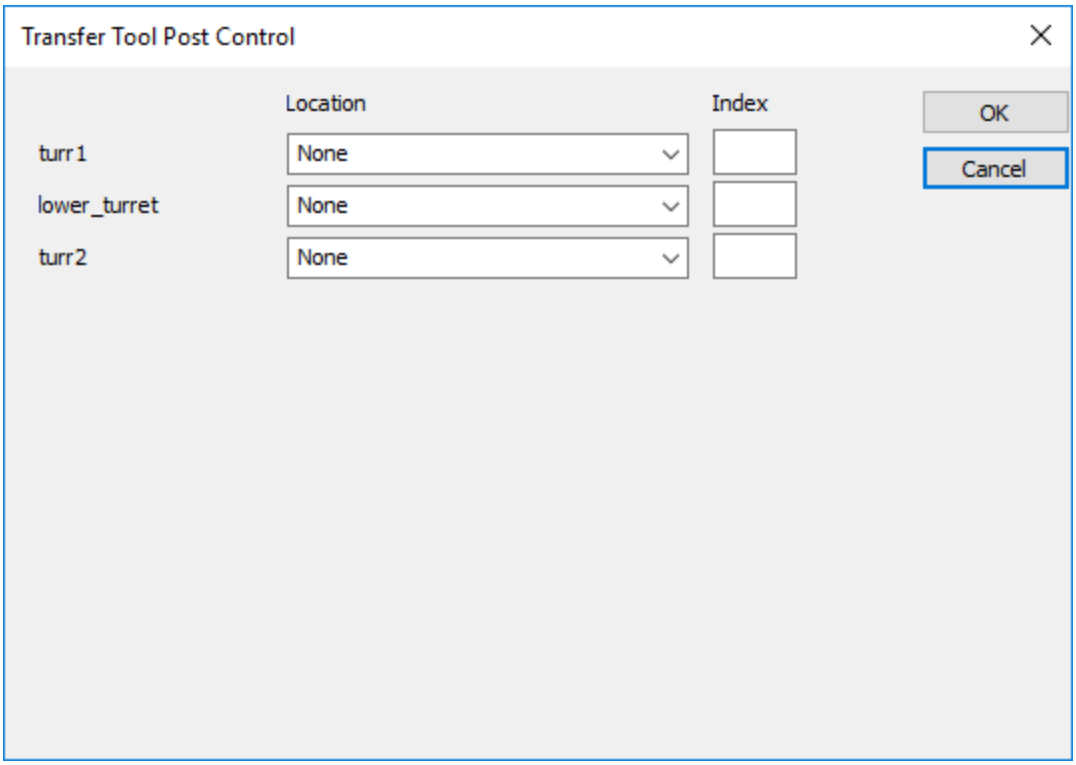
- After you select the type of Part Handling Feature you would like to use, press Next and you will see the option to set the **Strategies** for the feature. One of these settings is **the Transfer Turret Control**.



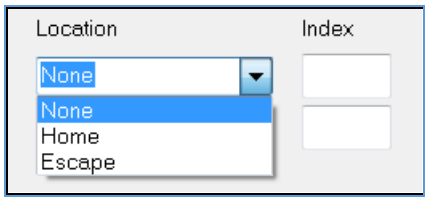
- In this dialog you can set the **Location** and **Index** that you want to use for each turret on your machine during part handling.

© Copyright 2016 Autodesk, Inc. All rights reserved

Transfer Turret Control Dialog



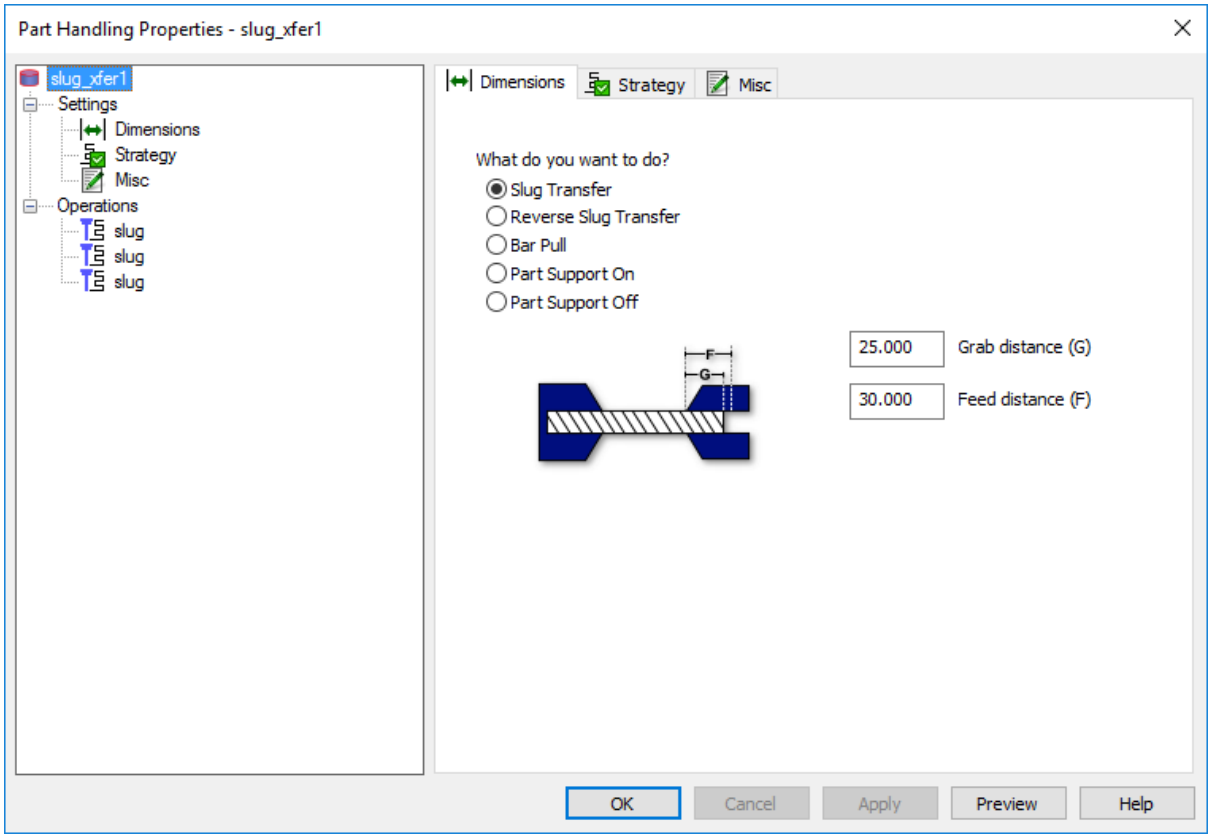
- **Location** — Select where you want the turret to be during the part handling: Select the down arrow to show the sub menu.



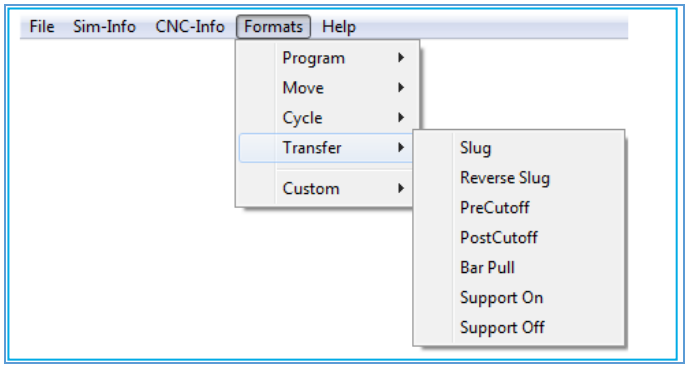
- **None** — Select this option to leave the turret at its current position before starting the part handling.
- **Home** — Select this option to move the turret to its home position before starting the part handling.
- **Escape** — Select this option to move the turret to a safe location away from the part before starting the part handling.
- **Index** — if necessary, you can rotate the turret to ensure that tools are not in the way. Enter the **Index** number

- Once you have set the **Strategies** for the Part Handling Feature, click **Finish** to save the Part Handling feature.
- In the same way as for other features, the Part Handling feature name is displayed in the **Part View** and on the **Op List** tab, and you can edit the feature in the **Part Handling Properties** dialog:

© Copyright 2016 Autodesk, Inc. All rights reserved



*There is also a new **Transfer** format type in the XBUILD menu:*

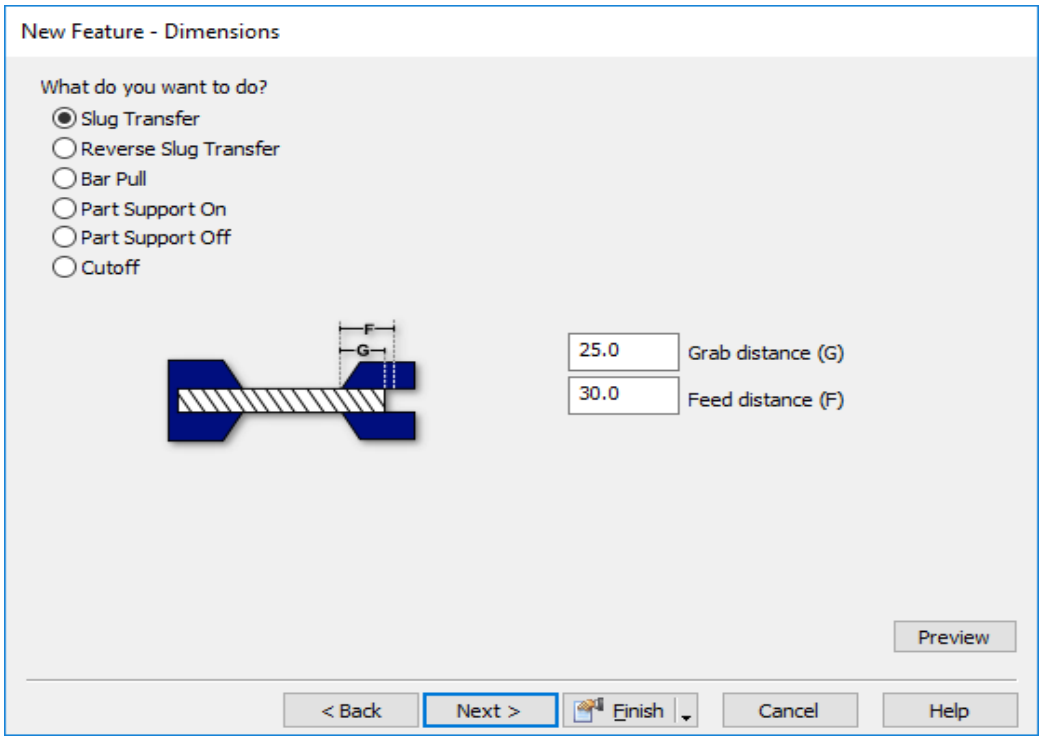


Part Handling feature types

- Select what you want to do from:
- **Slug Transfer** — Select this option to transfer the part from the main spindle to the sub-spindle.
- **Reverse Slug Transfer** — Select this option to transfer the part from the sub-spindle to the main spindle.
- **Bar pull** — Select this option to use the Sub-Spindle to grab and pull the bar of material further out of the main spindle.
- **Part Support On** — Select this option to enable part support such as a steady rest or tailstock.
- **Part Support Off** — Select this option to remove the part support.

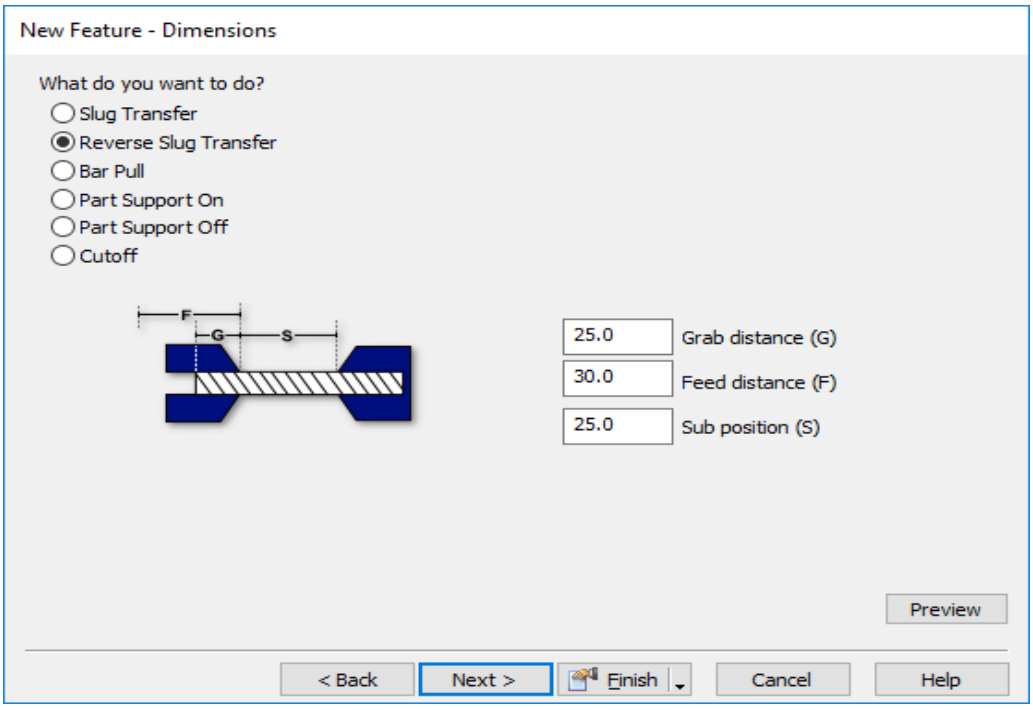
© Copyright 2016 Autodesk, Inc. All rights reserved

Slug Transfer



- **Grab Distance** – Enter the Grab Distance
- **Feed Distance** – Enter the Feed Distance

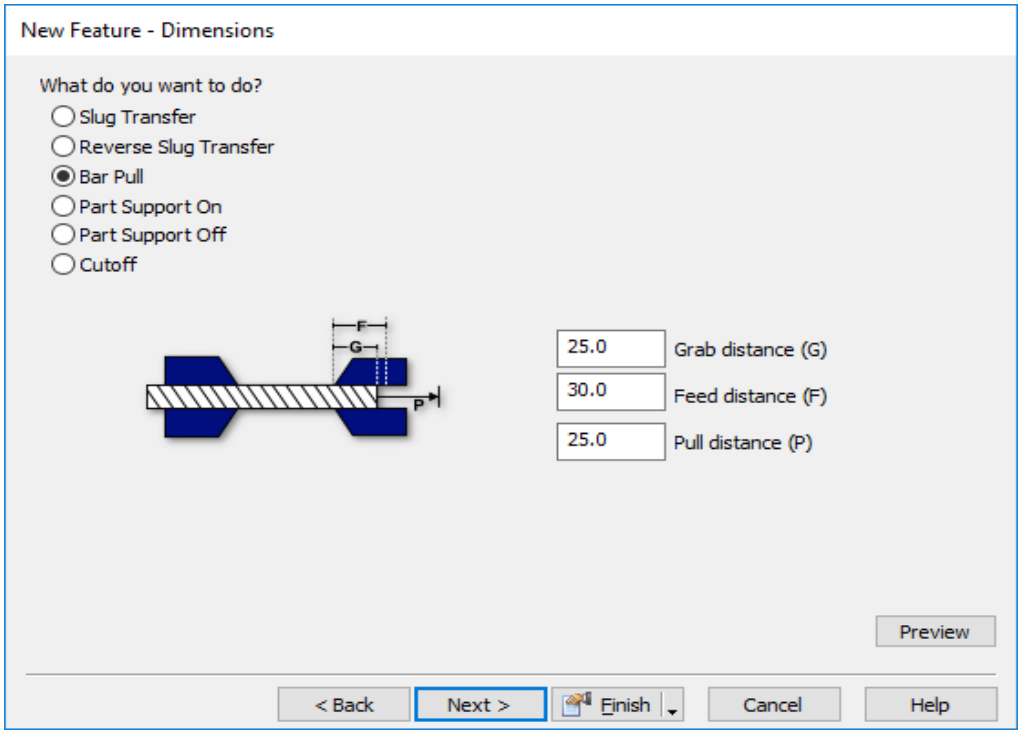
Reverse Slug Transfer



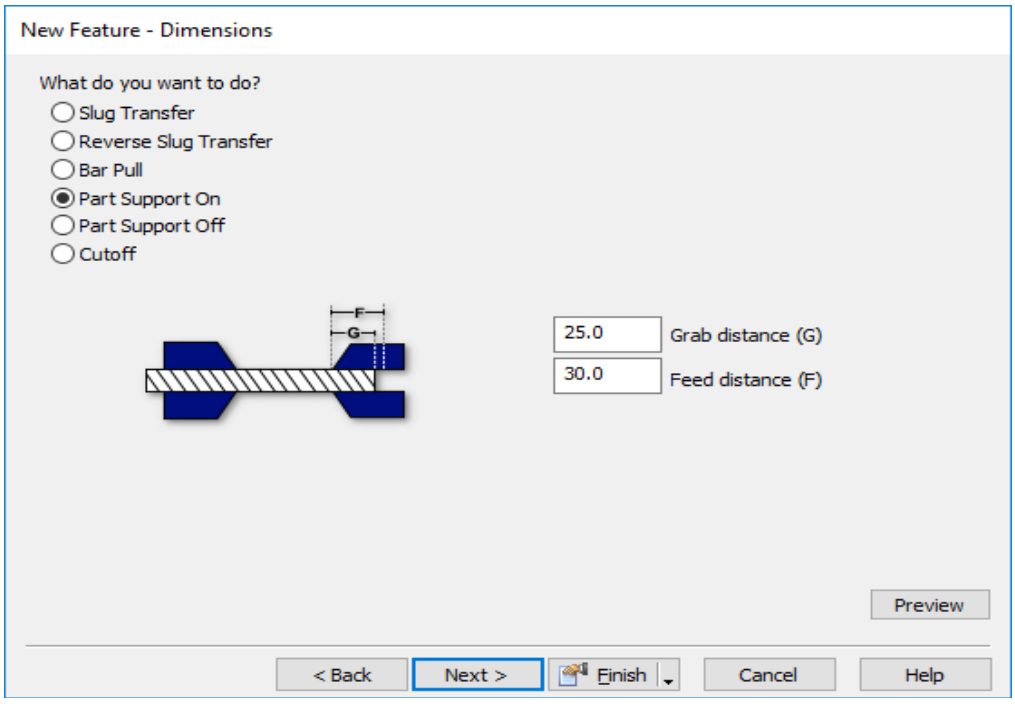
- **Grab Distance** – Enter the **Grab** Distance
- **Feed Distance** – Enter the **Feed** Distance
- **Sub Position** – Enter **Sub-Spindle** position

© Copyright 2016 Autodesk, Inc. All rights reserved

Bar Pull



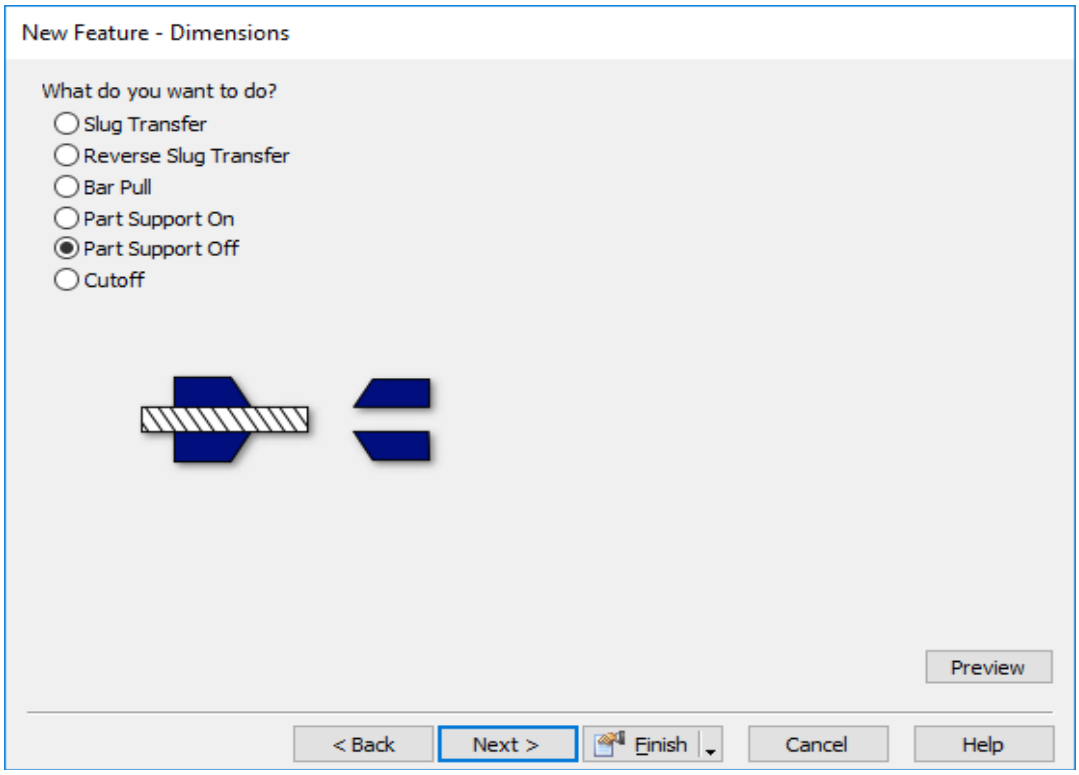
- **Grab Distance** – Enter the Grab Distance
- **Feed Distance** – Enter the Feed Distance
- **Pull Distance** – Enter Pull Distance
- **Part Support On**



- **Grab Distance** – Enter the Grab Distance in Z where you want the support to move to
- **Feed Distance** – Enter the Feed Distance

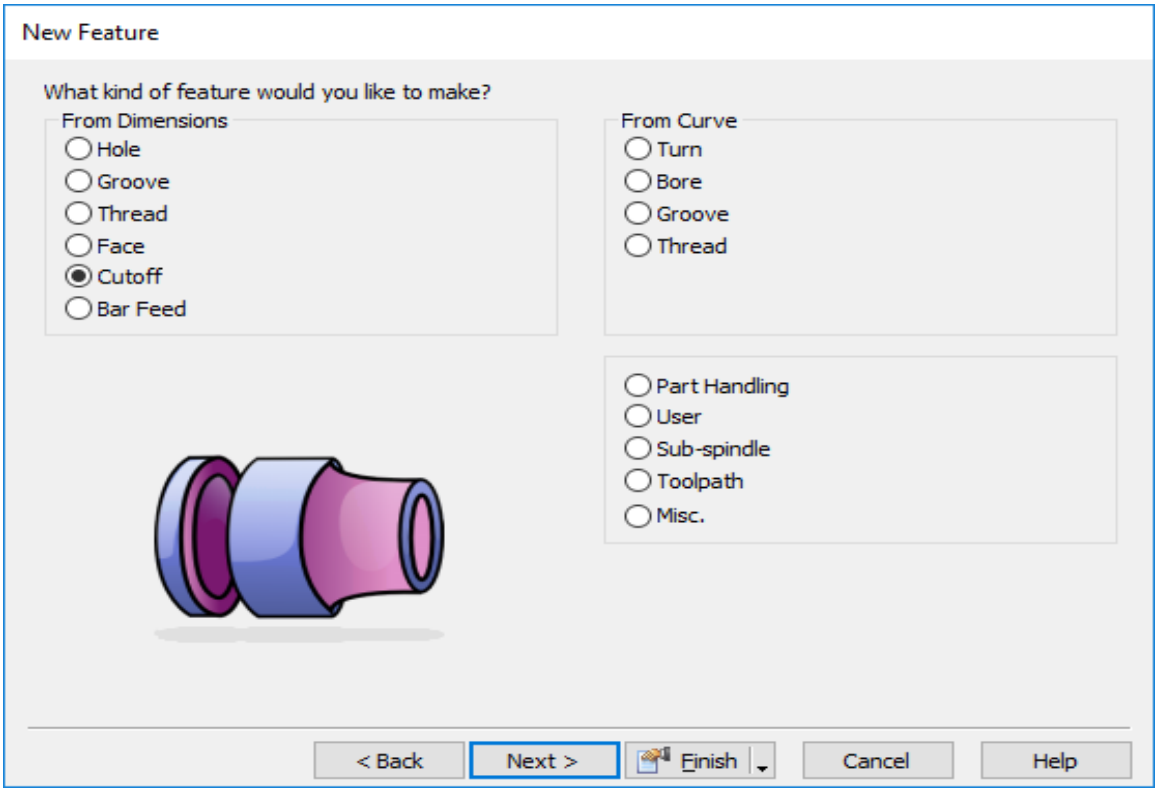
© Copyright 2016 Autodesk, Inc. All rights reserved

Part Support Off

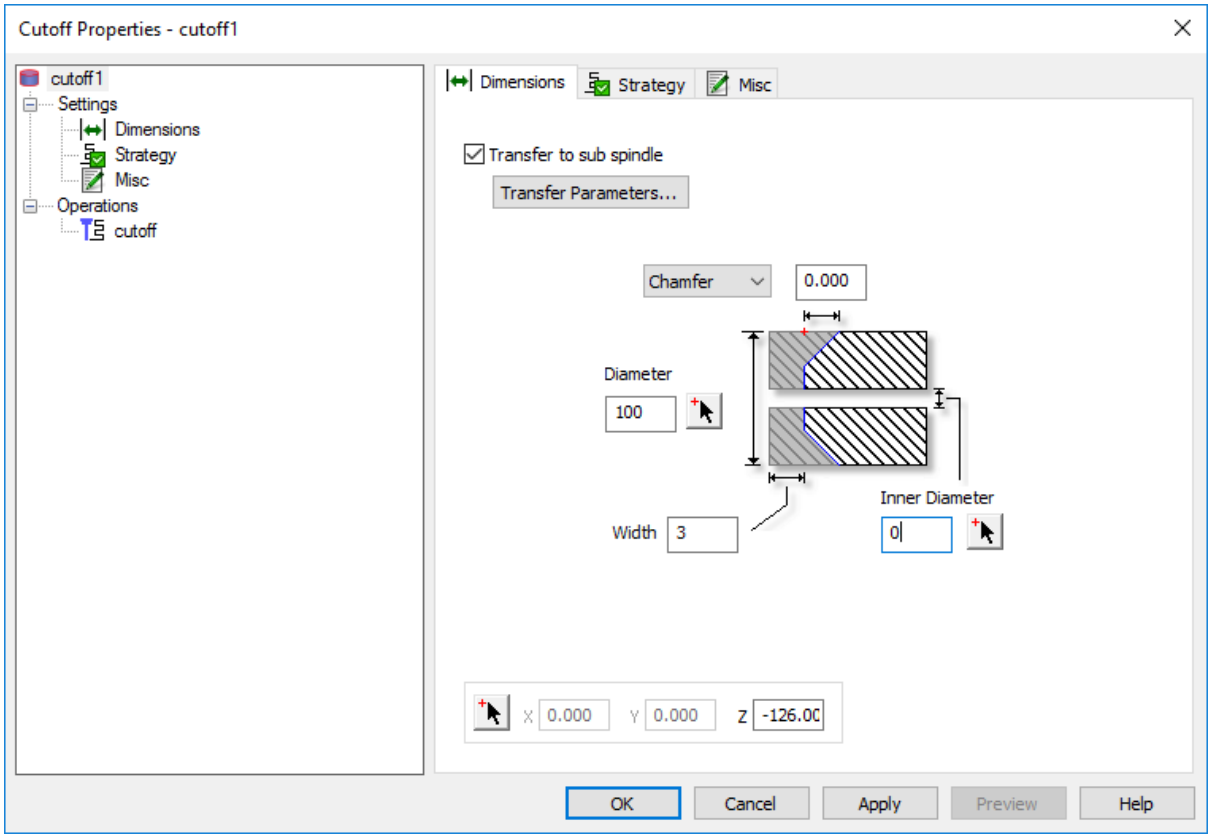


Cutoff Feature (from Dimensions)

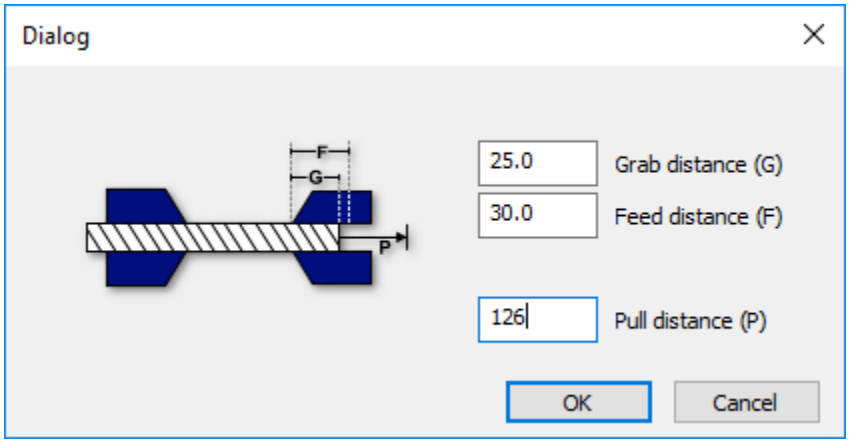
- The only **Part Handling** style feature that is **NOT in Part handling** area of the New Feature Wizard is the **Cutoff feature with a Slug Transfer** after the cutoff has been performed.



© Copyright 2016 Autodesk, Inc. All rights reserved



- When you press the Transfer Parameters button a new dialog pops up for you to set the parameters required for the process.



An Overview of how Transfers Work (Information Only)

Overview

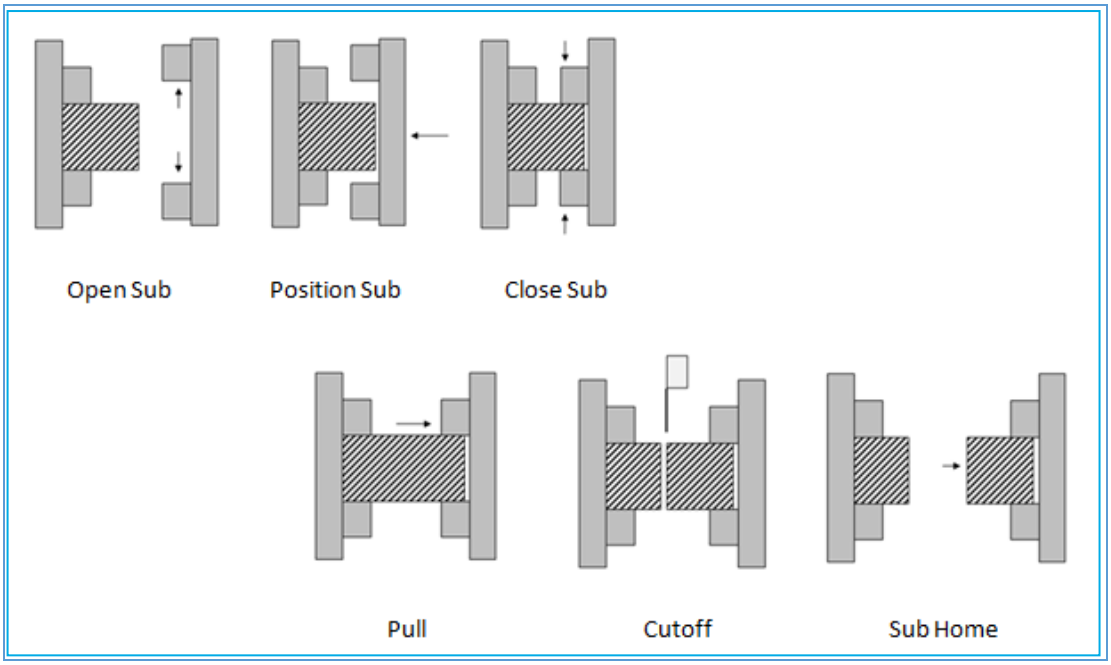
- A lot of parts, that have a complex set of features, have very simple transfer requirements. In most cases, all of the work on the front face will be done on the main spindle, the part transferred to the sub-spindle, and then all the back face features are cut. In these cases, it is only necessary to place a single Cutoff Transfer or Slug Transfer at the end of the first setup.

Cutoff Transfer. The basic cutoff transfer assumes that:

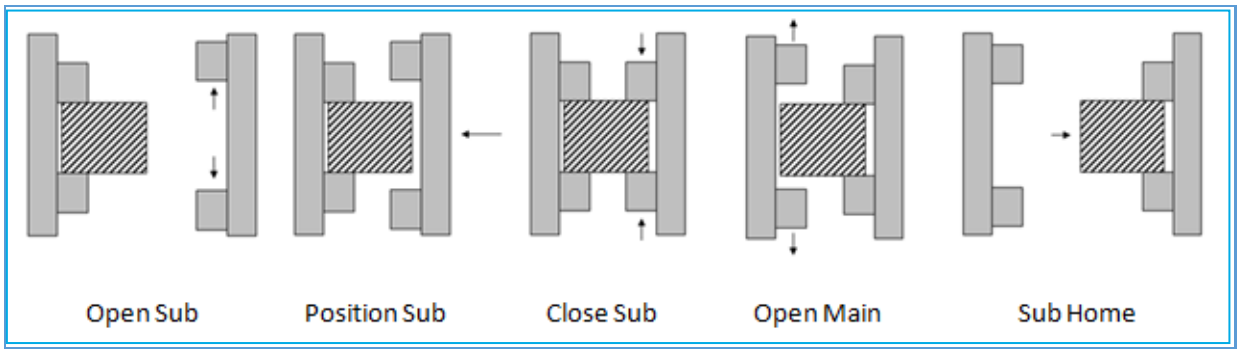
- The **sub-spindle** will travel to the face of the part and grab it

© Copyright 2016 Autodesk, Inc. All rights reserved

- The **sub-spindle** may optionally pull the part out by a certain distance
- The **bar** is cut, using the standard FeatureCAM cutoff operation, which is created by the UDF
- The **sub-spindle** retracts to its home position, carrying the cut stock with it



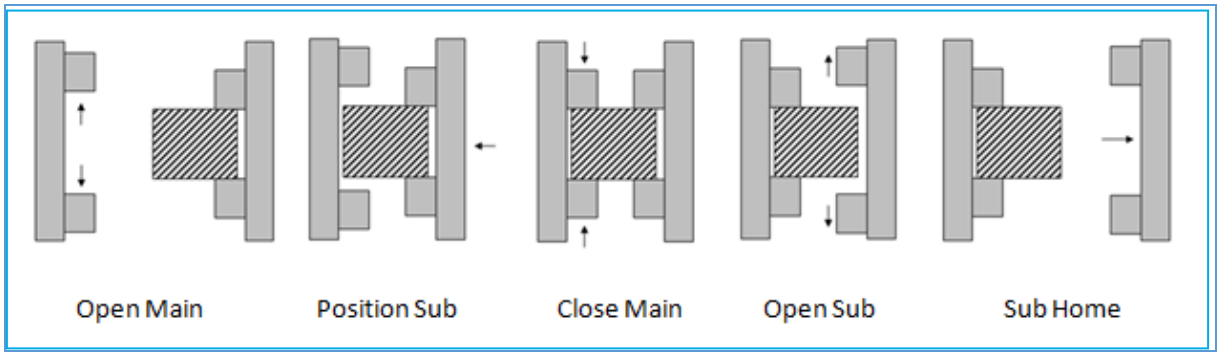
Slug Transfer. A slug transfer is similar to the cutoff transfer, but assumes that the stock was previously cut to the proper size, so the transfer is simply a hand-off of the stock from the main to the sub-spindle.



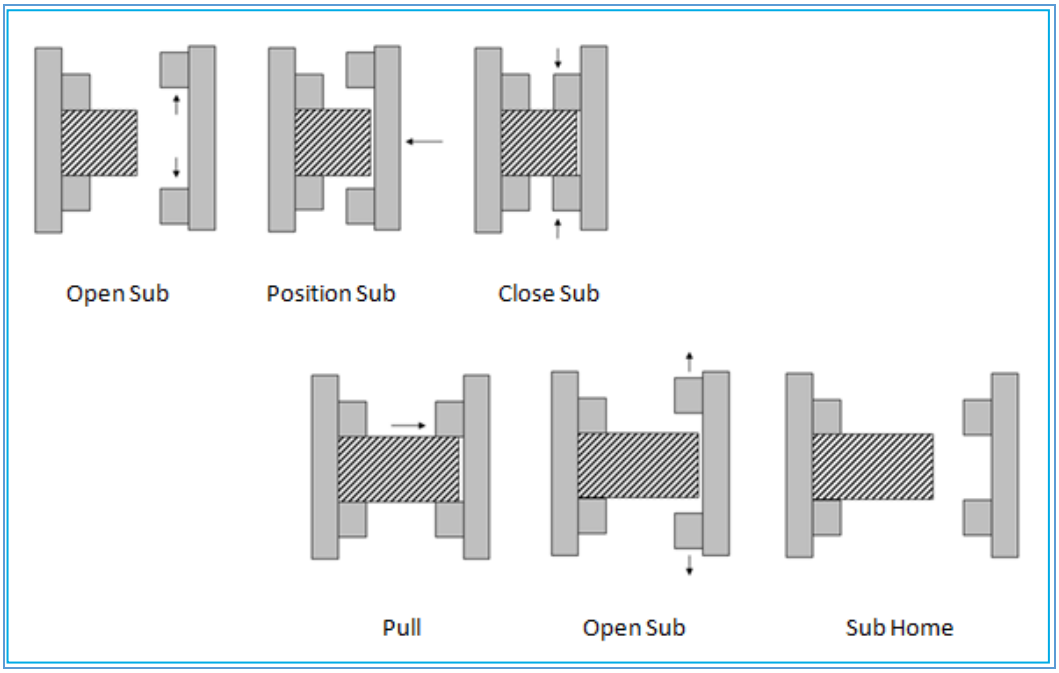
- The additional UDFs that are provided are for more complex transfer and spindle support situations. In these cases, several of the transfer UDFs will likely have to be combined to completely cut the part. Often, these situations involve longer parts in which it is desirable to pull bar stock out from the spindle in several steps in order to improve stability, or in which complex features on the front and back face must be alternated.

Slug Reverse Transfer. A slug reverse transfer is very similar to a slug transfer, but the part begins on the sub-spindle and moves to the main spindle.

© Copyright 2016 Autodesk, Inc. All rights reserved



Bar Pull. A bar pull leaves the stock clamped in the main spindle, but pulls the bar out in order to expose more of the stock for machining.



Support On

This UDF indicates that the stock (which must be in the main spindle) is to be supported in some way while machining the next set of features. There are 3 types of part support which can be specified (Sub-Spindle is illustrated in the diagram below):

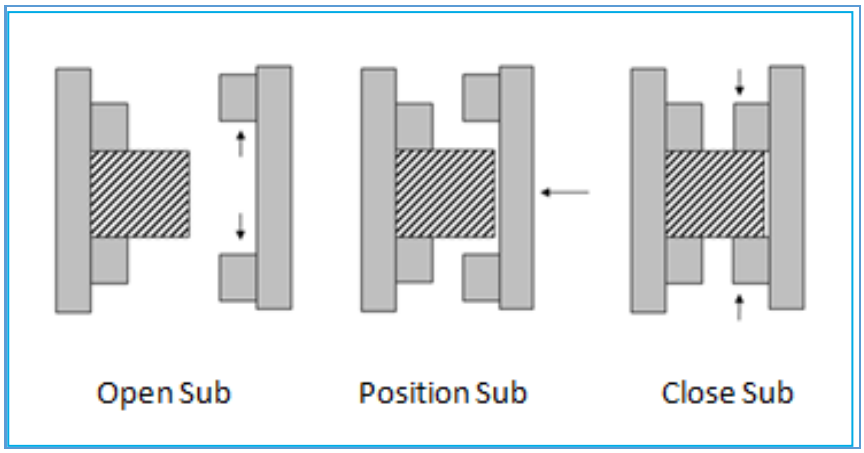
Sub-Spindle

Identical to the initial Sub-Spindle action for a Cutoff Transfer or a Slug Transfer,

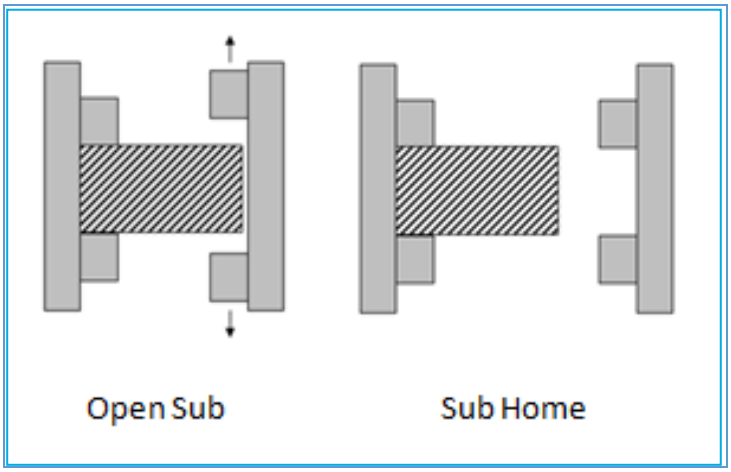
The Sub-Spindle grabs the face of the part and remains until specifically dismissed.

- **Tailstock** The machine has a tailstock rather than a sub-spindle.
- **SteadyRest** The machine has a steadyrest rather than a sub-spindle.

© Copyright 2016 Autodesk, Inc. All rights reserved



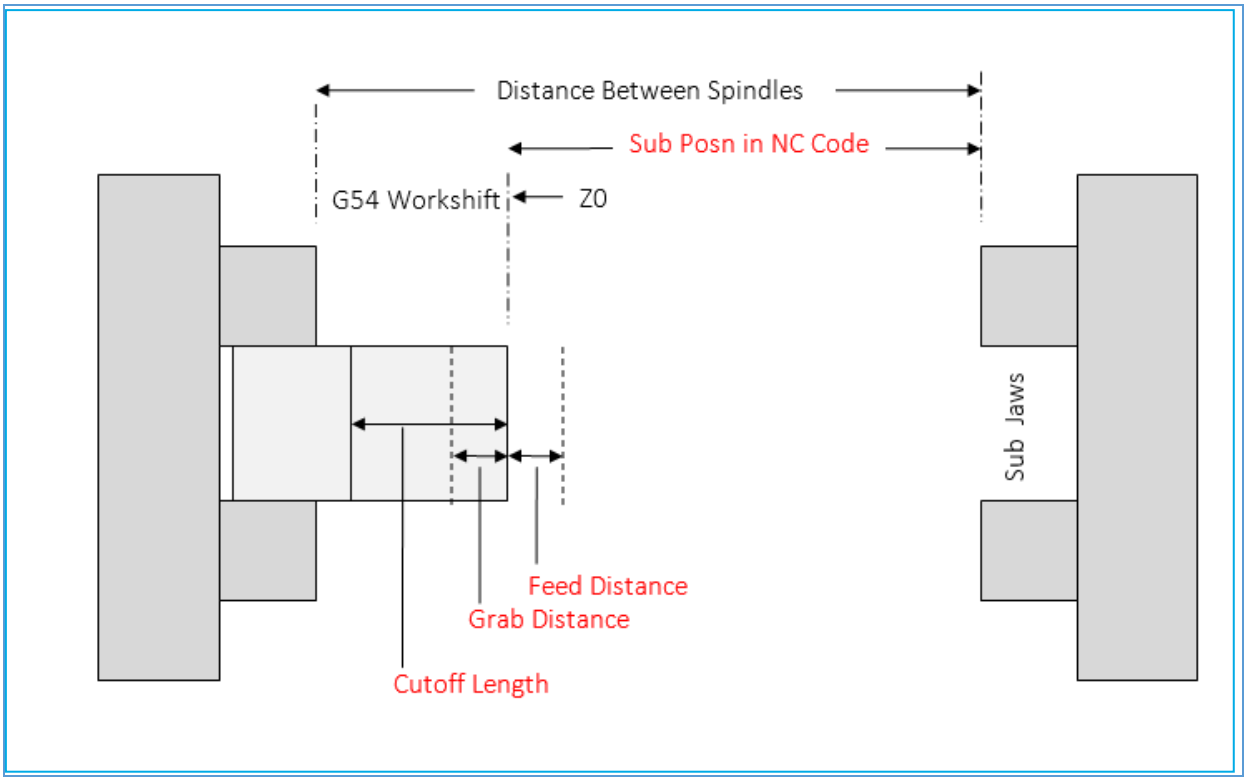
Support Off. This UDF indicates that the part support which was previously initiated by a Support On feature should now be removed.



© Copyright 2016 Autodesk, Inc. All rights reserved

Key Positioning Parameters

The figure below shows the key positioning parameters that are used by the transfer macros. The four values highlighted in red can be specified in the UDF, and are the most important ones to understand.



Sub Position in NC Code

- The base position (z-coordinate), for the sub-spindle during the transfer. In most cases you do not need to set this value. If the G54 workshift is set to the face of the part, then this number is typically zero. This is the simplest way to work with these UDFs. If you are not setting the fixture offset, then this should be the z coordinate at which the sub-spindle would just touch the face of the part.
- **Feed Distance** The distance from the part face at which the sub-spindle should stop rapid motion and begin to feed onto the part. This number could be negative.
- **Grab Distance** The distance for the sub-spindle to grab (or overlap) the part before closing the jaws

Cumulative Prior Pulls

- The cumulative length of any prior BarPull operations, which were done without a cutoff. This is required so that the sub-spindle is positioned correctly.

© Copyright 2016 Autodesk, Inc. All rights reserved

***Note**: all of the parameters are relative (distances) not absolute (Z positions), with the exception of the Sub-spindle Position in Z, which (again) is typically zero.*

Part Support Interaction with the Other UDFs

- Since the "Support On" UDF (when the type is "Sub-Spindle") is functionally a component of several of the other transfer types, two parameters are available to provide part support directly from one of the other UDFs:
- **Already Supported** indicates whether a previous "Support On" operation is already supporting the part when the transfer starts (i.e. sub-spindle will be assumed to already be holding the part). This parameter is available for Slug Transfer, Pull Cutoff, and Bar Pull.
- **Leave Supported** indicates whether the part should remain supported at the end of the transfer (i.e. sub-spindle will not be sent home). This parameter is available for Slug Reverse Transfer and Bar Pull.
- These support parameters in the other transfer functions mean that in many cases you can use part support without explicitly creating Support On and Support Off features. For example, you could do a series of bar-pull operations, leaving the part supported after each one, and finish with a pull-cutoff transfer without ever explicitly creating a Support **On** or Support **Off UDF**.

Cutoff Parameters

- These parameters are available only for the Part Cutoff UDF:
- **Cutoff Length -** The length of stock to be cut off (usually the length of the finished part)
- **Cutoff Tool Width** – The width of the tool that will be used for the cutoff
- **Cutoff OD -** The outer diameter for the cutoff
- **Cutoff ID – The** inner diameter for the cutoff
- **Cutoff Chamfer -** chamfer value for the cutoff
- **Cutoff Turret -** specifies which turret should be used for the cutoff operation *[not available if the post is single turret]*
- **DoPull -** whether or not to pull out the barstock (by the length of the part plus the cutting tool width) during a cutoff operation.

Part Handling Parameters

- All UDF parameters not listed above are shown below. Those which apply only to specific UDFs are indicated; if none are listed, then the parameter applies to all of the Transfer UDFs except Support Off.
- **Feed Rate -** The feed rate to be used for the spindle when feeding
- **Pull Distance -** The distance to pull *[Bar Pull only]*

© Copyright 2016 Autodesk, Inc. All rights reserved

- **Position -** The Z position, relative to the main setup, to which the sub-spindle should return the part *[Slug Reverse only]*
- **Support Type -** which type of part support will be used: Sub-Spindle, Tailstock, or SteadyRest *[Support On, Support Off only]*

Initial Turret Motion

- specifies location to which each turret should be moved at the start of the transfer *[All, unless the post is single turret]*
- **Initial Turret Index** specifies the location to which each turret should be indexed at the start of the transfer *[All, unless the post is single turret]*
- **Transfer Turret** specifies which turret must issue Sub-Spindle control commands *[All, unless the post is single turret]*
- **Part Catcher** whether part catcher command should be issued during the transfer
- **Use Push/Press** whether the Push-Press option should be used
- **Open/Close Dwell Time**
- pause (in seconds) after opening or closing main or sub-spindle jaws
- **Sub Grab Action:** Whether spindles should be stopped, stopped and oriented, or kept spinning. Default is stopped
- **Main/Sub Angle** If Sub Grab Action is orient, this is the indexing angle on main and sub
- **Spindle Direction** If Sub Grab Action is keep-rotating, then start (sync'd) spindles in this direction
- **Spindle Speed** If Sub Grab Action is keep-rotating, then start (sync'd) spindles at this speed

Limitations on Machine Simulation

- All of the functionality that is described above can be taken advantage of in the post-processor, but there are a few of the options that cannot be correctly handled by machine simulation in the current release. These are listed here:
- The "initial turret motion" and "turret index" options are not supported in the simulation. There are a couple of checkboxes in the machine design file that can be used to control turret locations, but they are global and cannot interact with the settings in the UDFs.
- The part support types "tailstock" and "steadyrest" are simulated by declaring the tailstock or steadyrest to be the sub-spindle in the machine design file. This allows simulation of one or the other within a single MD file, but not both.

© Copyright 2016 Autodesk, Inc. All rights reserved

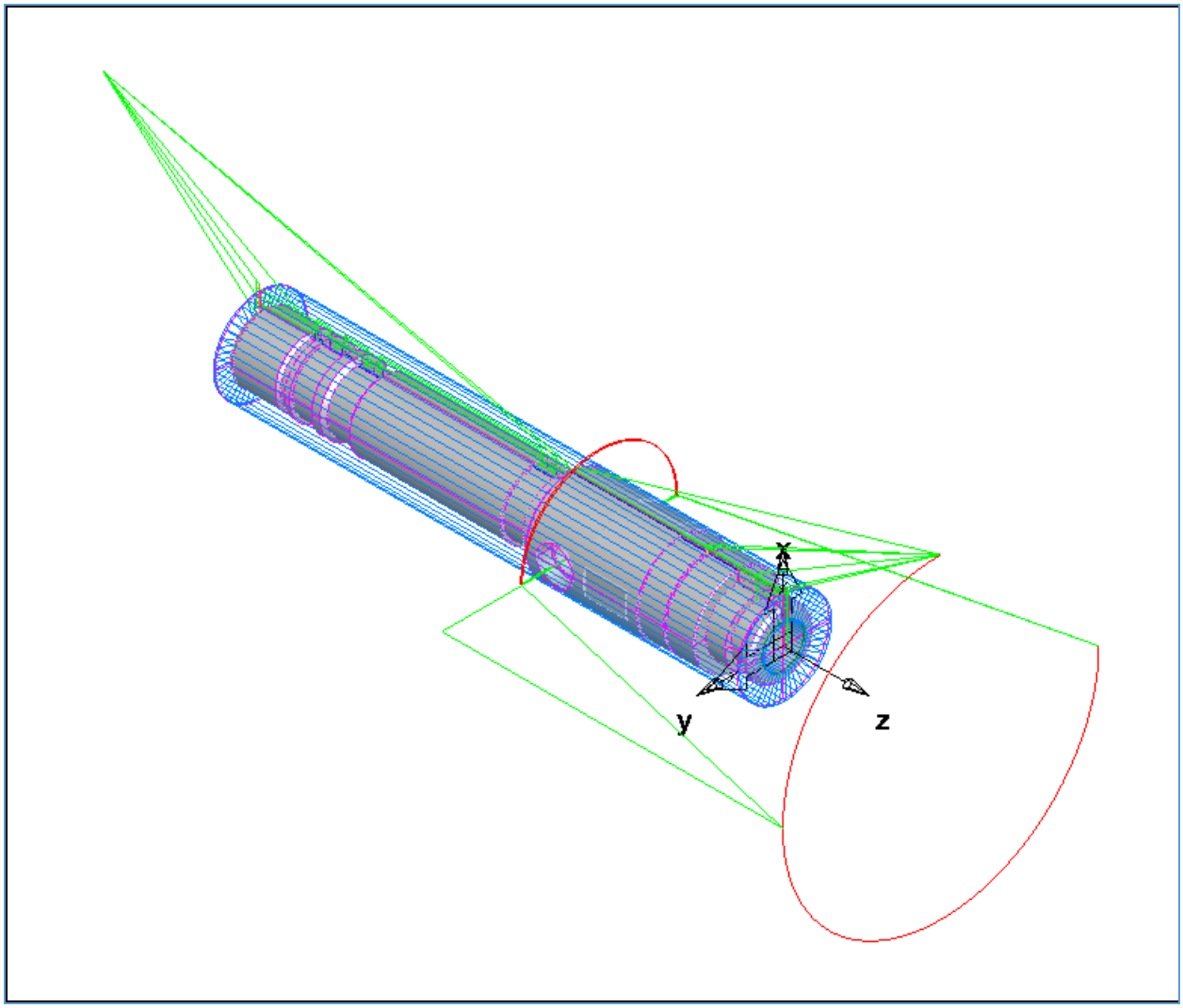
Setting up the Post

Important: *Currently it is likely that your post is not configured for the new Part Handling functionality. Please contact your post support group to make sure your post is configured for these operations.*

Part Handling Exercise (Optional Exercise)

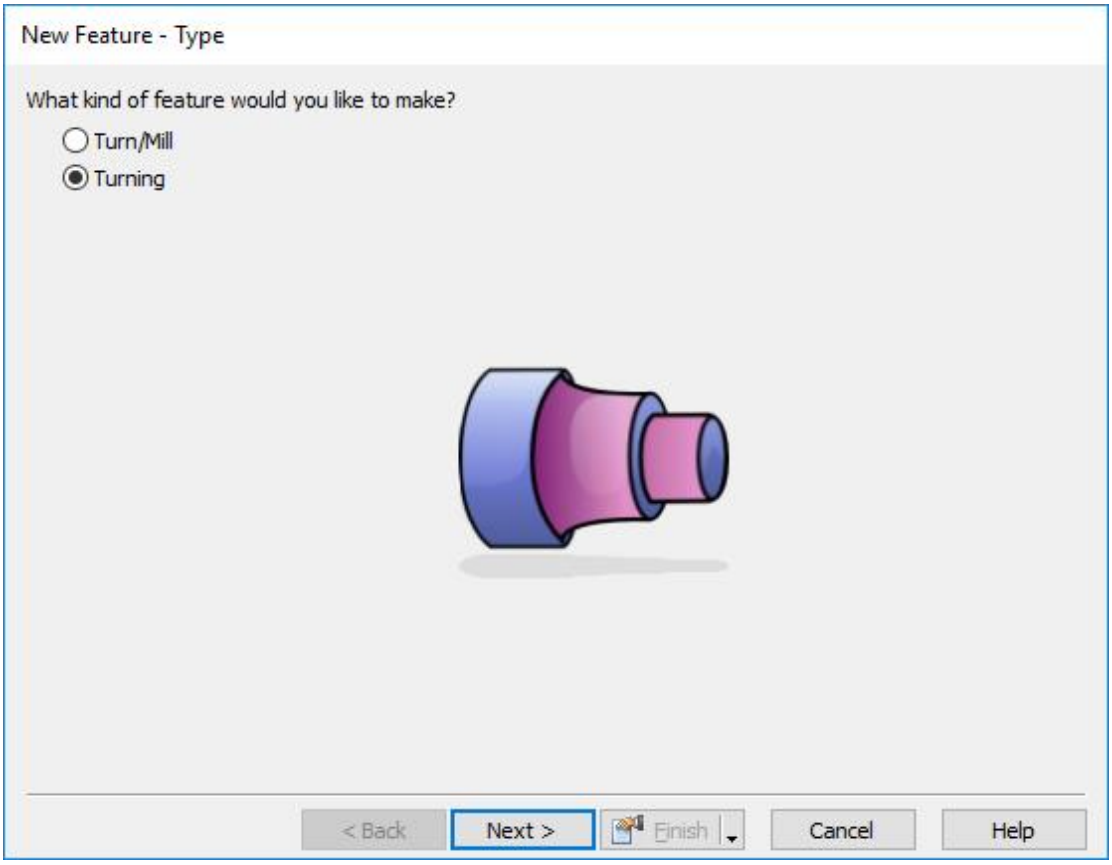
1. Open the part **FCAM2017_PartHandling.fm C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm Files**
2. Rename the file and call it **My_FCAM2017_PartHandling.fm**
3. Load the post from the same folder **- Mori Seiki NZ 1500 T2Y Training Only.cnc.**

We already have a part with Toolpaths.



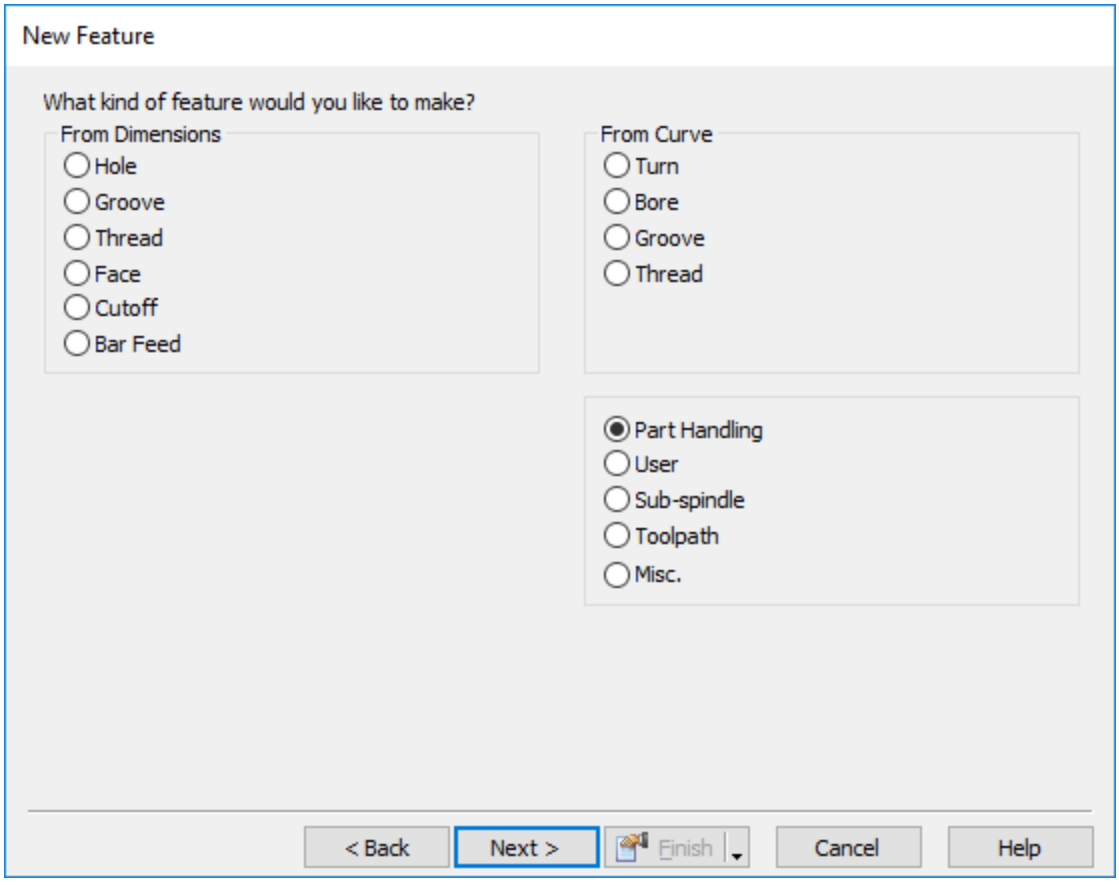
4. We are now going to add the part transfer to **PartView Setup1**
5. Select **Ctrl** + **R** to create a **New Feature**. Select Turning

© Copyright 2016 Autodesk, Inc. All rights reserved



6 Select **Next**

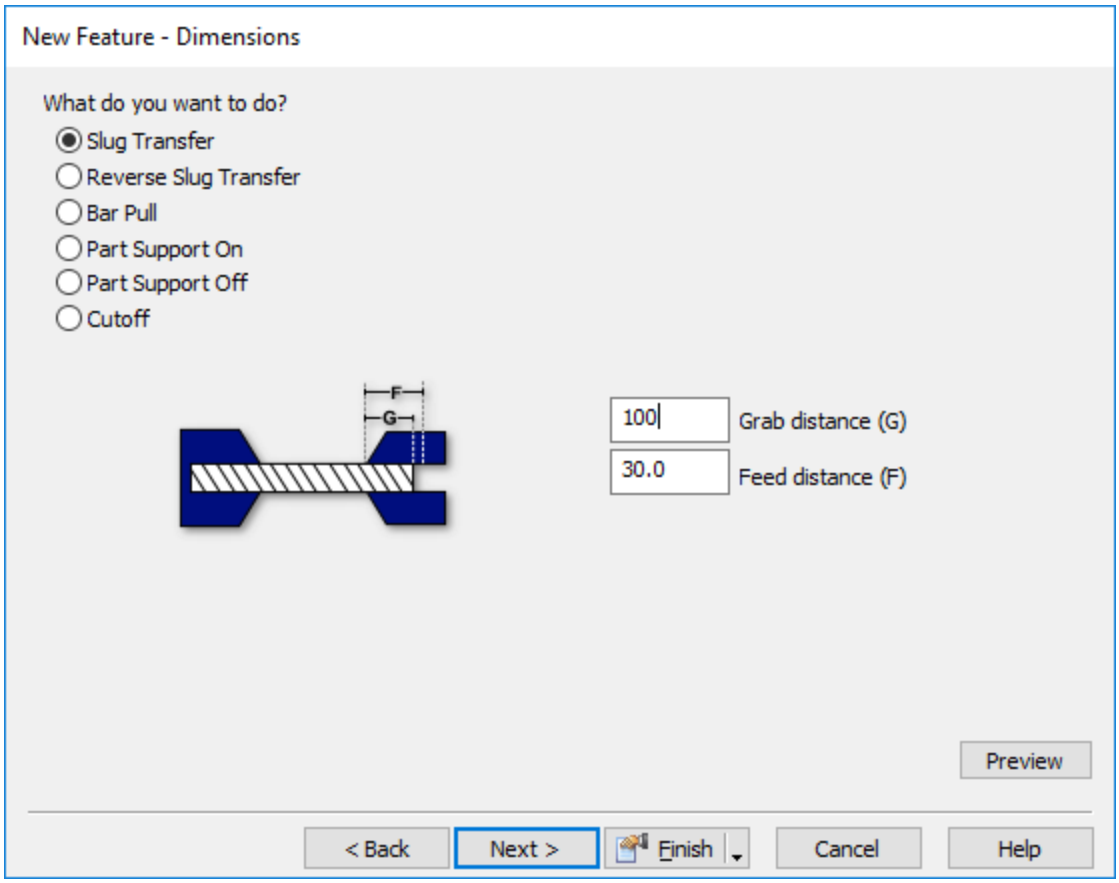
7 Select **Part Handling**



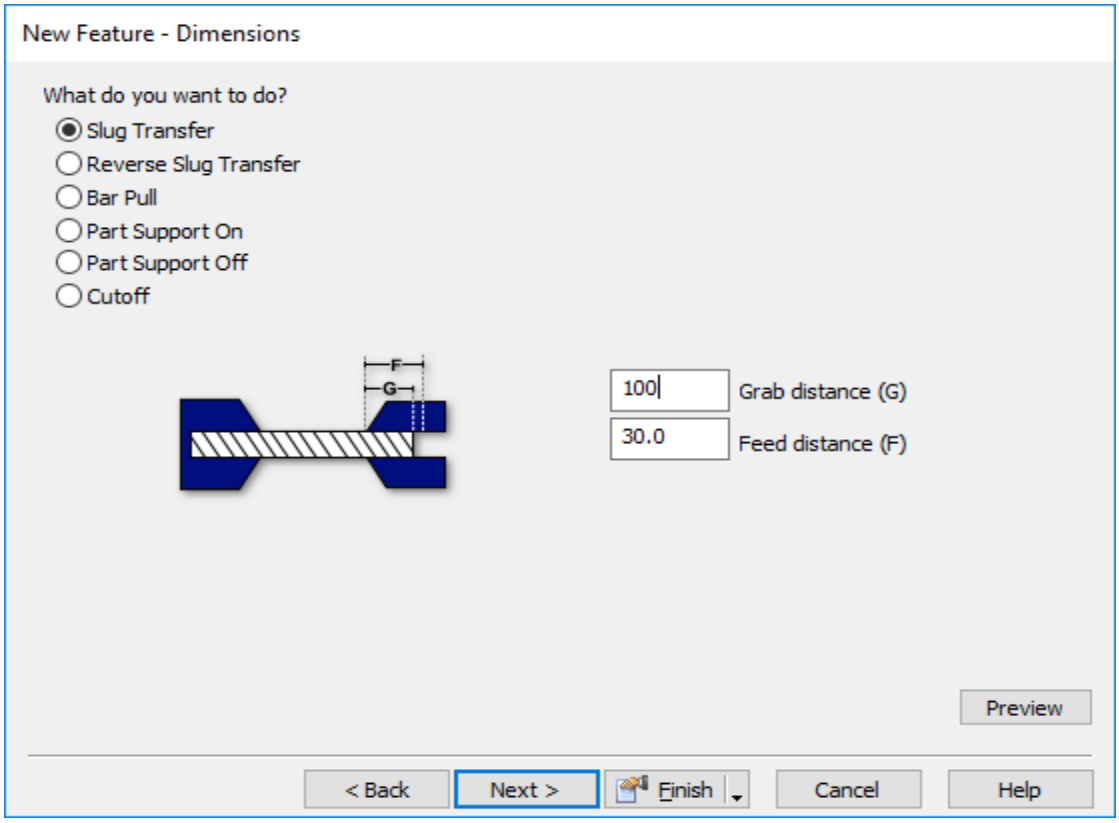
© Copyright 2016 Autodesk, Inc. All rights reserved

8 Select **Next**

9 Select **Slug Transfer** and enter a value of **100mm** to the **Grab distance**.



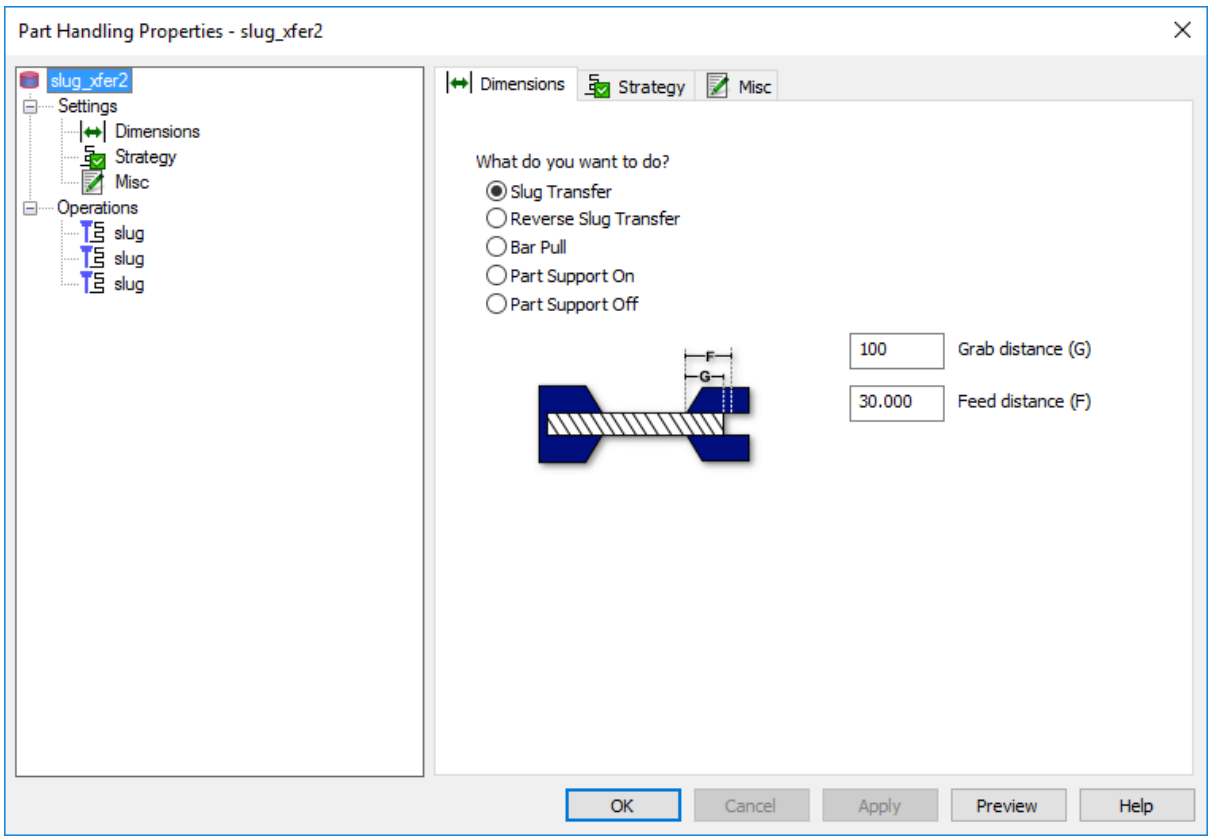
10 Select **Next.**



© Copyright 2016 Autodesk, Inc. All rights reserved

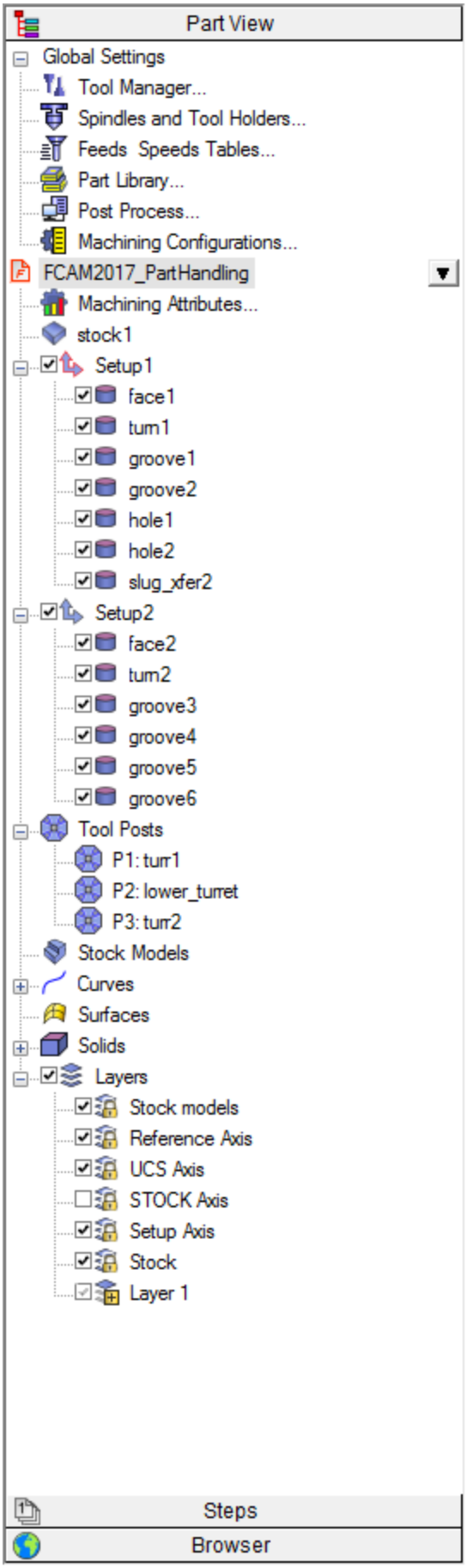
11 Select **Finish**.

12 This has now created an Operation called **Slug Xfer1** as shown.



*Remember that the turret can be indexed out of the way by using the **Tool Post Control** option and setting the index position that does not have ant tools. This will avoid the possibility of the Sub-Spindle hitting any tools when the transfer takes place.*

© Copyright 2016 Autodesk, Inc. All rights reserved

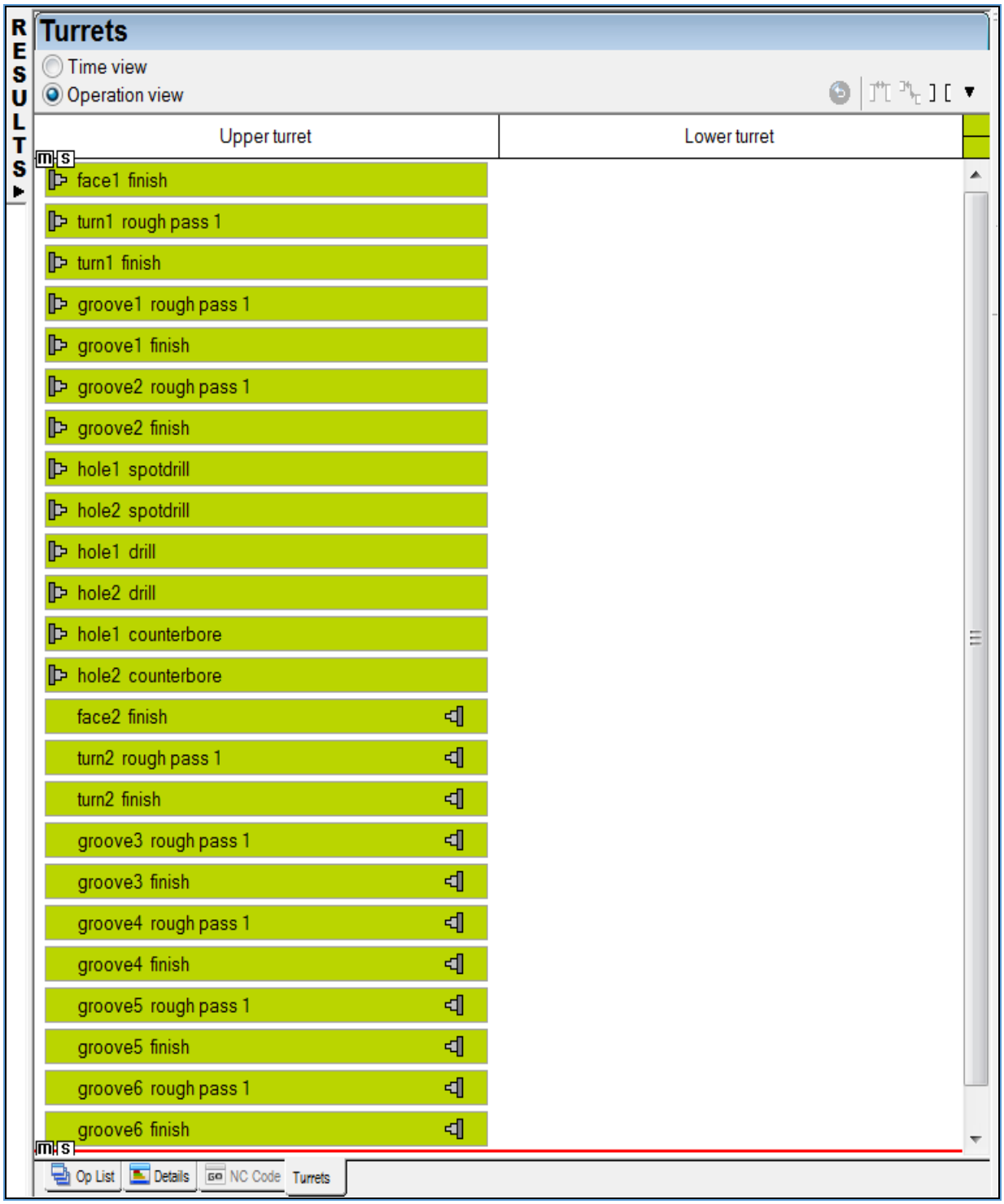


© Copyright 2016 Autodesk, Inc. All rights reserved

We now need to synchronize the two Turrets.

13 Select Turrets in the Results Tab.

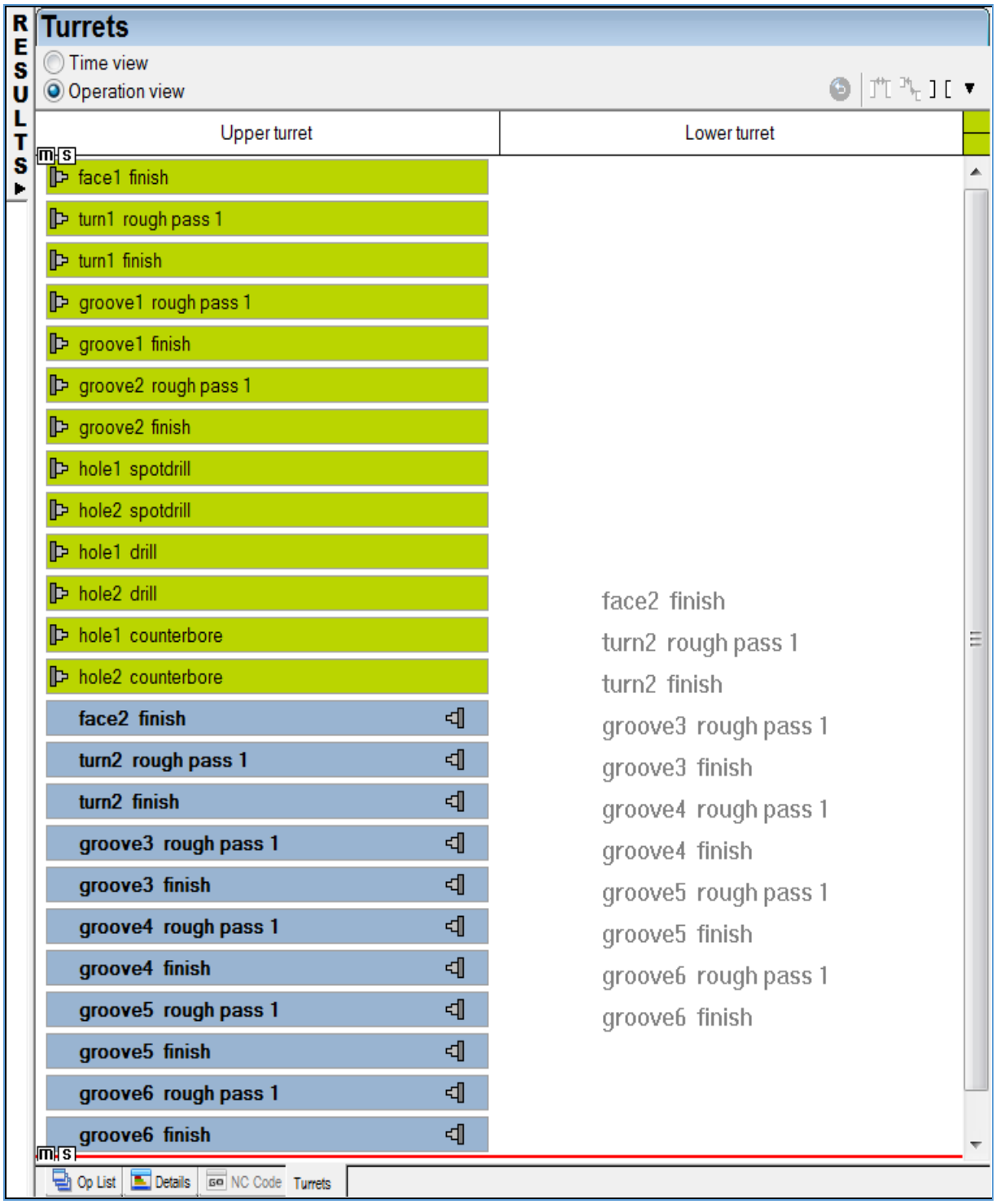
14 This is how it looks at the moment.



15 We have to move the Sub-Spindle operations to the Lower turret

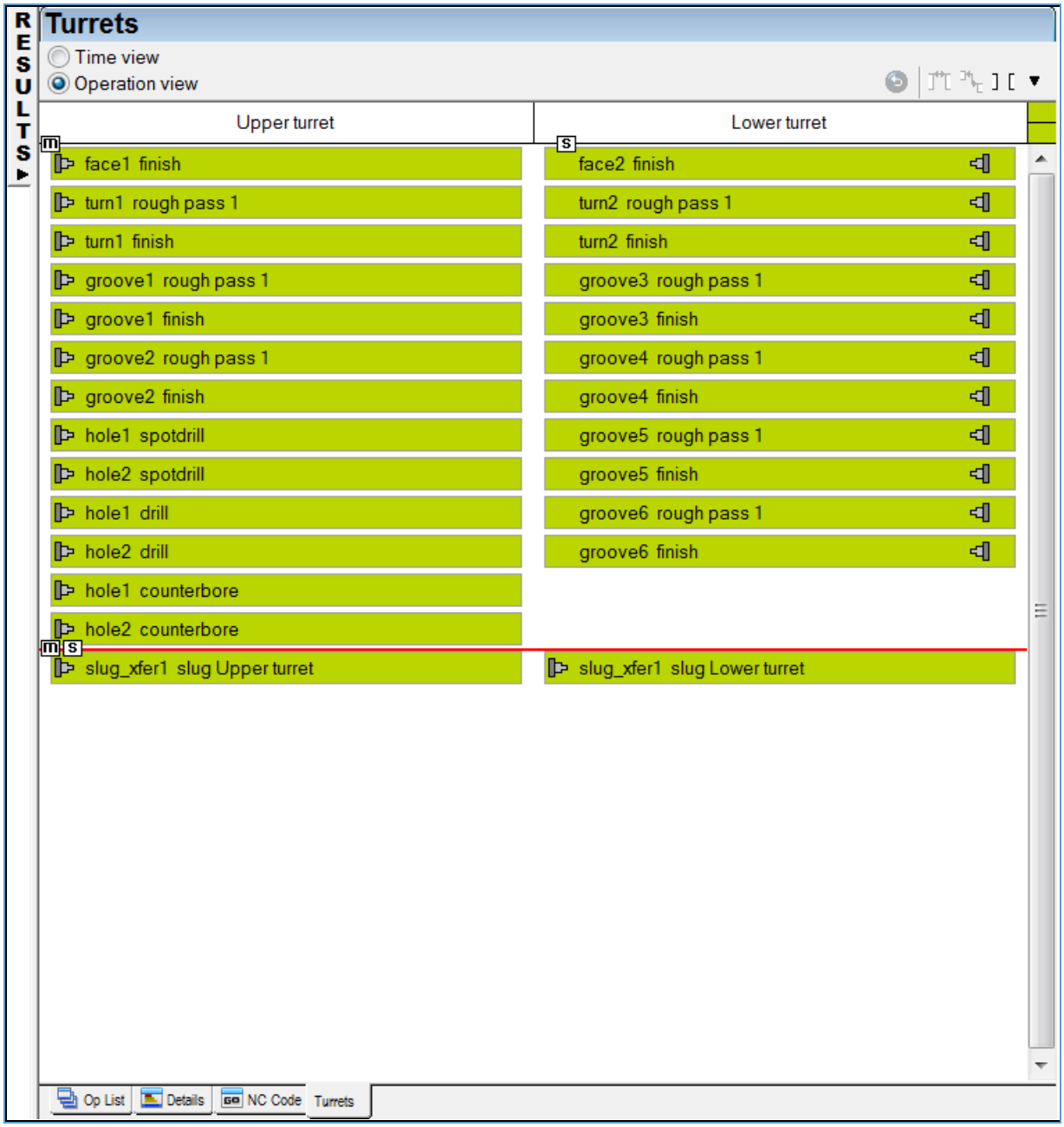
16 Select the 11 operations and drag them to the Lower turret as shown.

© Copyright 2016 Autodesk, Inc. All rights reserved

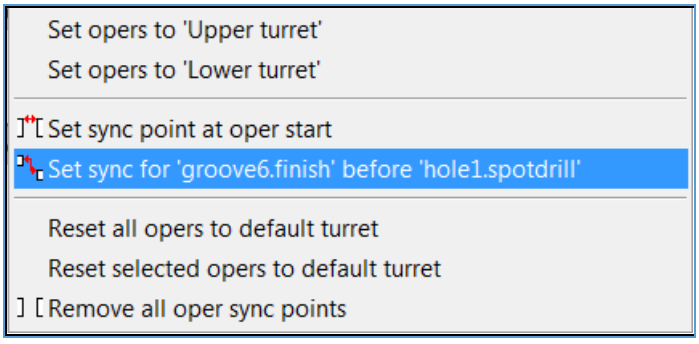


The next page shows an image of the operations moved to the Lower turret.

© Copyright 2016 Autodesk, Inc. All rights reserved

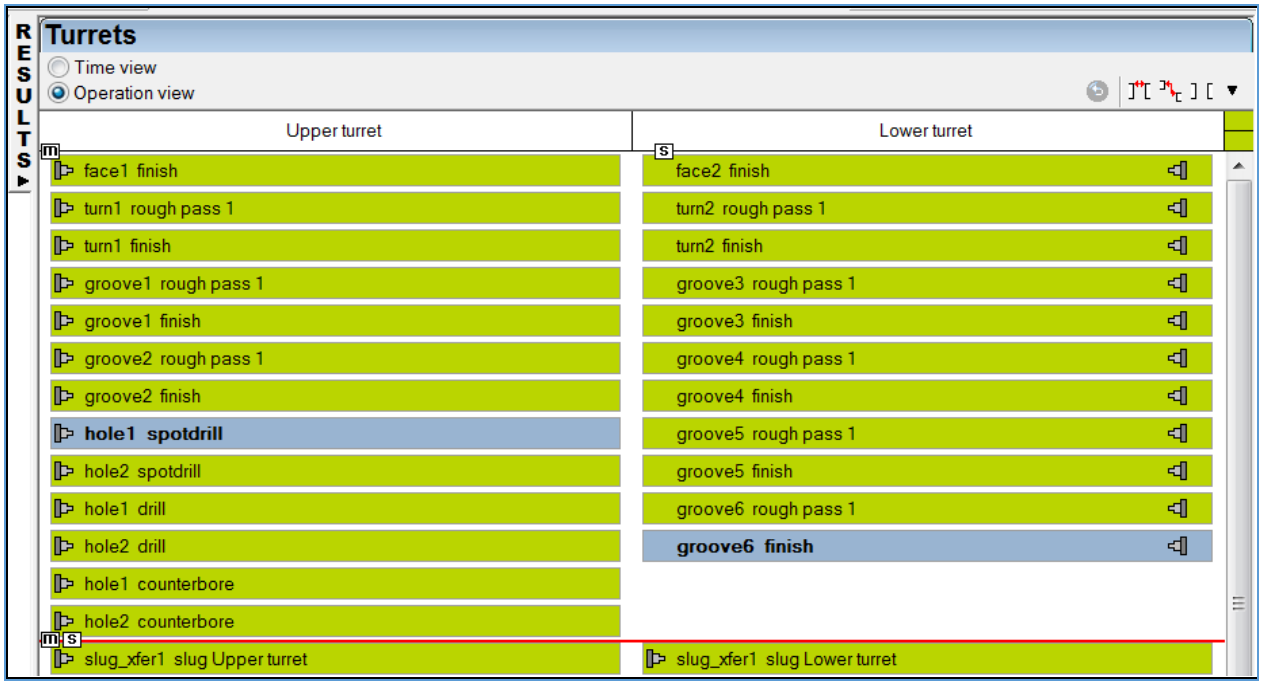


17 We now need to *synchronize* the two turrets. Select Groove6 finish in the Lower turret and hole1 spot drill in the Upper Turret. As shown.

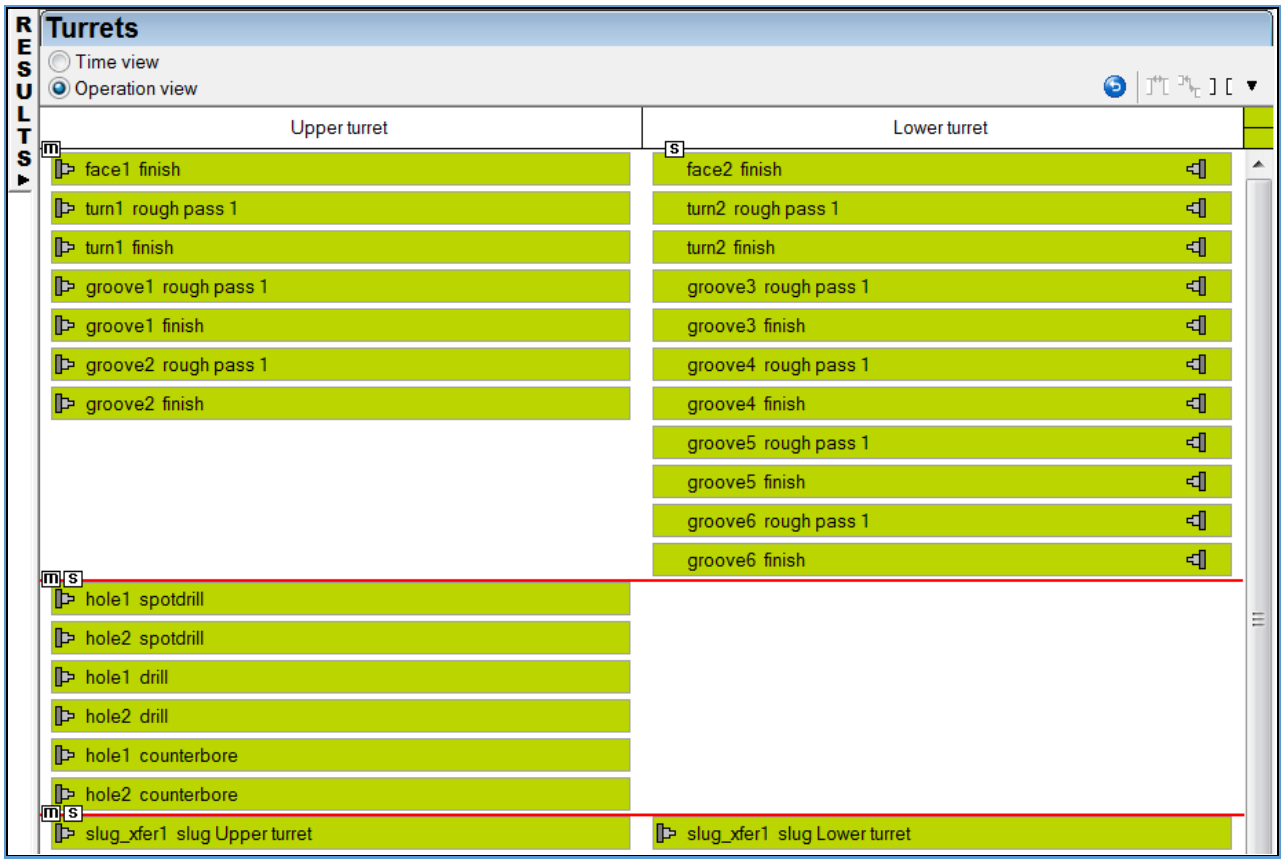


© Copyright 2016 Autodesk, Inc. All rights reserved

18 Remember to hold down the **Ctrl key** to select the two points and then set the Sync points.



19 This is how it should look



© Copyright 2016 Autodesk, Inc. All rights reserved

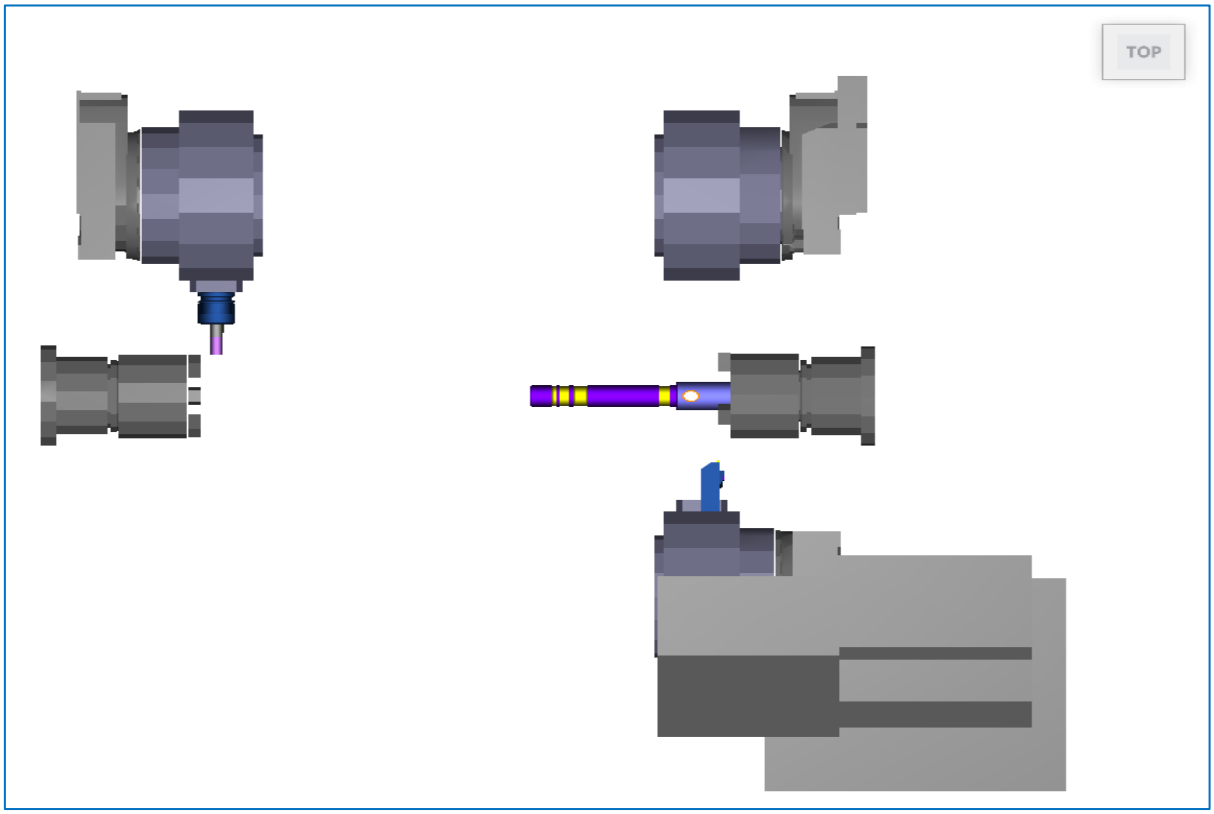
20 Now run the Machine Simulation to see the transfer of the part to the sub-Spindle.

RESULTS

Operation List

Automatic Ordering
Manual Ordering

R	Operation	Feature	Tool	Feed	Speed		Depth
	finish	face1	SW_Turn_80m...	0.152 MMPR	213 SMM	CW	36.500 mm
	rough pass 1	turn1	SW_Turn_80m...	0.381 MMPR	146 SMM	CW	14.986 mm
	finish	turn1	SW_Turn_80m...	0.152 MMPR	213 SMM	CW	14.986 mm
	rough pass 1	groove1	SW_Grv_6m_RH	0.381 MMPR	146 SMM	CW	4.470 mm
	finish	groove1	SW_Grv_6m_RH	0.152 MMPR	213 SMM	CW	4.470 mm
	rough pass 1	groove2	SW_Grv_1m_RH	0.381 MMPR	146 SMM	CW	1.651 mm
	finish	groove2	SW_Grv_1m_RH	0.152 MMPR	213 SMM	CW	1.651 mm
	drill	hole1	Ø23mm drill par...	27.2 MMPM	169 RPM		33.008 mm
	drill	hole2	Ø23mm drill par...	27.2 MMPM	169 RPM		33.008 mm
	counterbore	hole1	Endmill 20mm d...	77.6 MMPM	243 RPM		11.430 mm
	counterbore	hole2	Endmill 20mm d...	77.6 MMPM	243 RPM		11.430 mm
	slug turr1	slug_xfer2					
	finish	face2	NE_BackTurn_...	0.152 MMPR	213 SMM	CW	19.863 mm
	rough pass 1	turn2	NE_BackTurn_...	0.381 MMPR	146 SMM	CW	11.811 mm
	finish	turn2	NE_BackTurn_...	0.152 MMPR	213 SMM	CW	11.811 mm
	rough pass 1	groove3	NE_Grv_6m_RH	0.381 MMPR	146 SMM	CW	2.883 mm
	finish	groove3	NE_Grv_6m_RH	0.152 MMPR	213 SMM	CW	2.883 mm
	rough pass 1	groove4	NE_Grv_6m_RH	0.381 MMPR	146 SMM	CW	2.819 mm
	finish	groove4	NE_Grv_6m_RH	0.152 MMPR	213 SMM	CW	2.819 mm
	rough pass 1	groove5	NE_Grv_6m_RH	0.381 MMPR	146 SMM	CW	2.883 mm
	finish	groove5	NE_Grv_6m_RH	0.152 MMPR	213 SMM	CW	2.883 mm
	rough pass 1	groove6	NE_Grv_6m_RH	0.381 MMPR	146 SMM	CW	2.819 mm
	finish	groove6	NE_Grv_6m_RH	0.152 MMPR	213 SMM	CW	2.819 mm
	slug lower_turret	slug_xfer2					
	slug turr2	slug_xfer2					
	Results						



© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Extract the Turned Geometry 2
Extracting Surface Edges 2
To extract curves from 3D data: 2
Unwrapping Features 4
Unwrapping Curves 4
Machining the unwrapped Curve 5
Wrapping Text around a diameter 11
Wrapping a 2D Logo around the outside diameter. 16

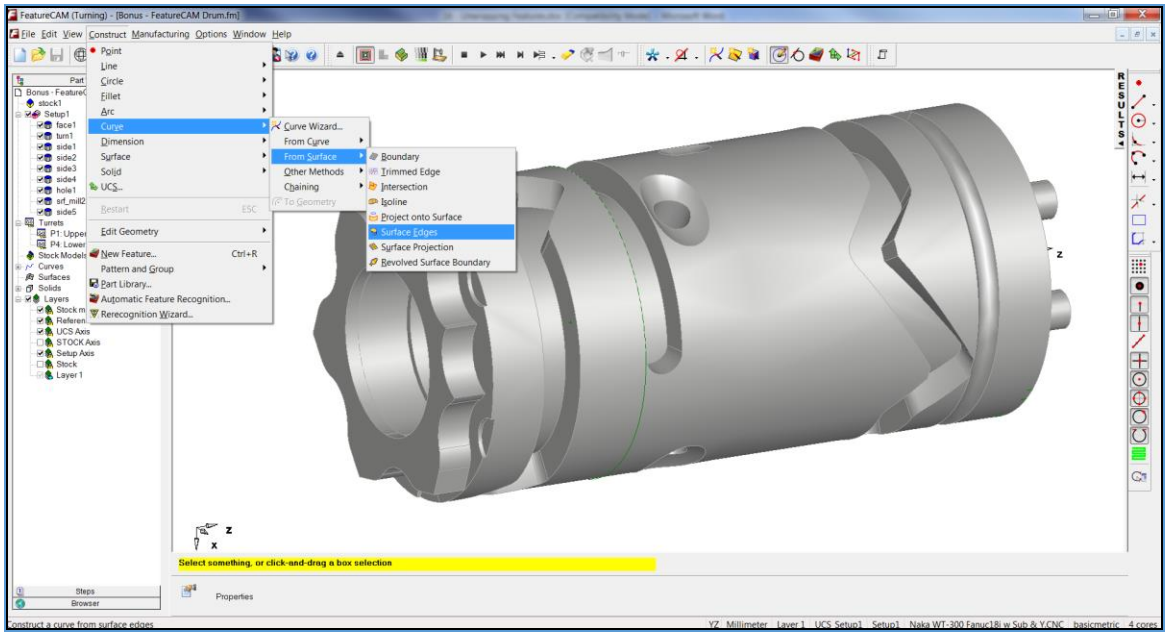
© Copyright 2016 Autodesk, Inc. All rights reserved

Extract the Turned Geometry

1. Import Model **Bonus - FeatureCAM Drum.SLDPRT** from **C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**
2. Work through the wizard to align the part so the longest edge is parallel with the Z axis.
3. Extract the Turned geometry shape, then **Face** and then **turn** the outside profile.

Extracting Surface Edges

- **Surface edges: -** Enables you to extract trimming curves from a collection of surfaces, join them into a curve and then project or Unwrap the curve onto the XY plane of your current UCS. You can then use this curve to create features.



To extract curves from 3D data:

4. Open the Surface Edges dialog in one of these ways:
5. In the **Curve Wizard** select **Curve from surface**, then **Surface Edges**, and Select **Next**.
6. On the **Curves and Surfaces** toolbar, click the **Surface Edges** button in the **Curve from surface** menu.
7. From the menu, select **Construct > Curve > From Surface > Surface Edges**.
8. Optionally enter a curve name in the **Name** field, or leave the default name.
9. Select the edge you want to use from the **Edges** menu, or use the **Pick curve on surface** button to select it in the graphics window. The edge is added to the table.
10. Applies to solids only (not a random collection of surfaces).

© Copyright 2016 Autodesk, Inc. All rights reserved

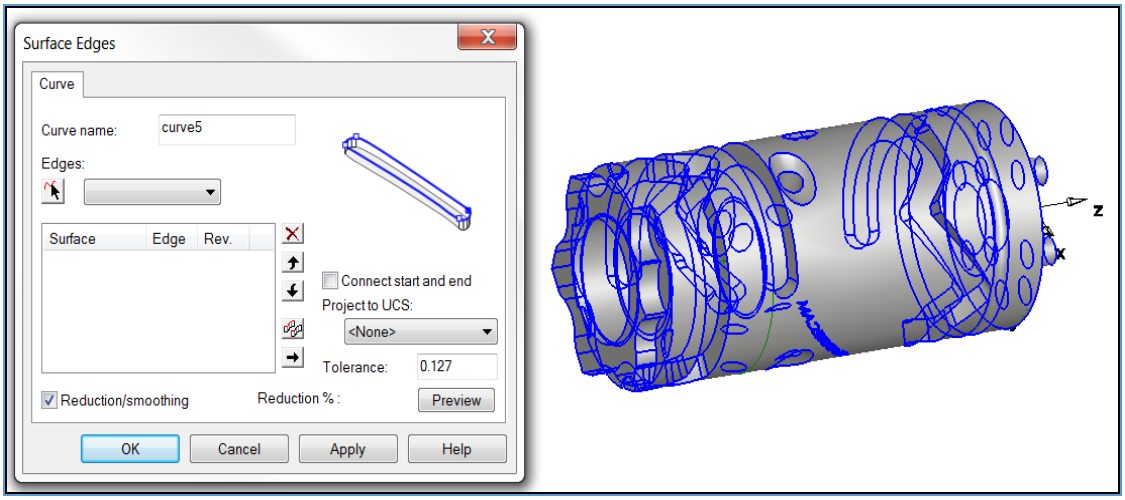
11 Ctrl+ click two edges to select all edges that make up the shortest path between them.

12 Double-click an edge to select all tangential edges.

13 The location of the pick is used to select a surface *and* extract a trimming edge. Click the **Use next edge of surface** button if you picked the correct surface, but the wrong edge.

14 Continue selecting edges in order until you have selected all edges.

15 Use the **Move item up** and **Move item down** buttons to reorder the edges if necessary.



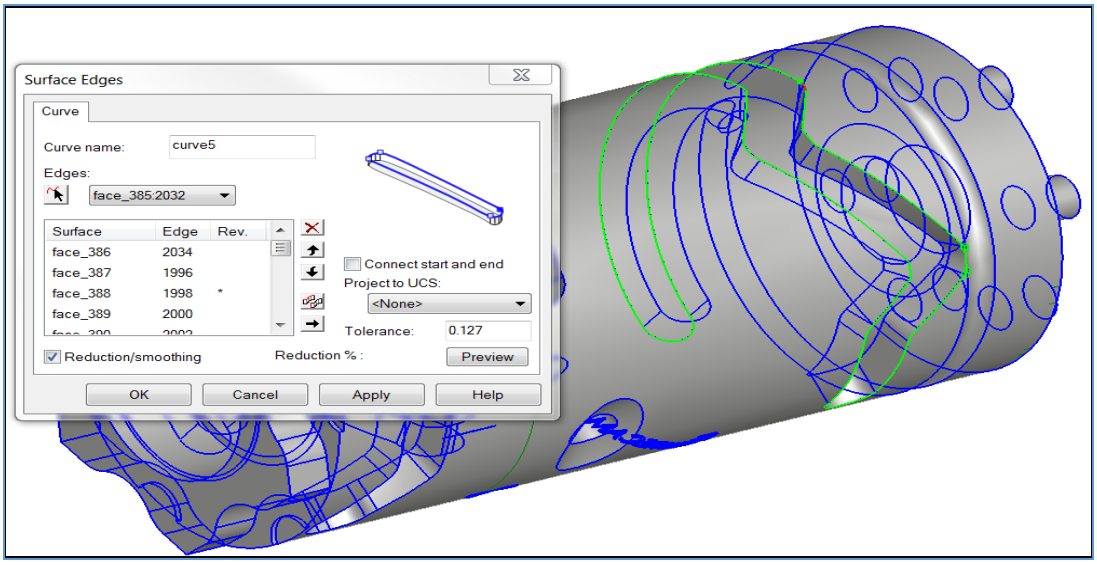
16 If the edges are in the correct order, but the endpoints are matched up incorrectly, select the surface name in the table and click the **Reverse** button.

17 Select **Connect start and end** if you want the curve to be closed automatically.

18 Select from the **Project to UCS** menu to have the curve automatically projected to the 2D XY plane of the UCS.

19 If a hollow blue square is displayed at the intersection of two edges, then the edges have not quite matched up. You can correct this by either selecting the missing edge that will connect the endpoints, or if it is just an issue of floating point error, increase the **Tolerance** parameter.

20 Click **OK** (or **Finish** if you're using the wizard).

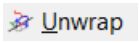


© Copyright 2016 Autodesk, Inc. All rights reserved

Unwrapping Features

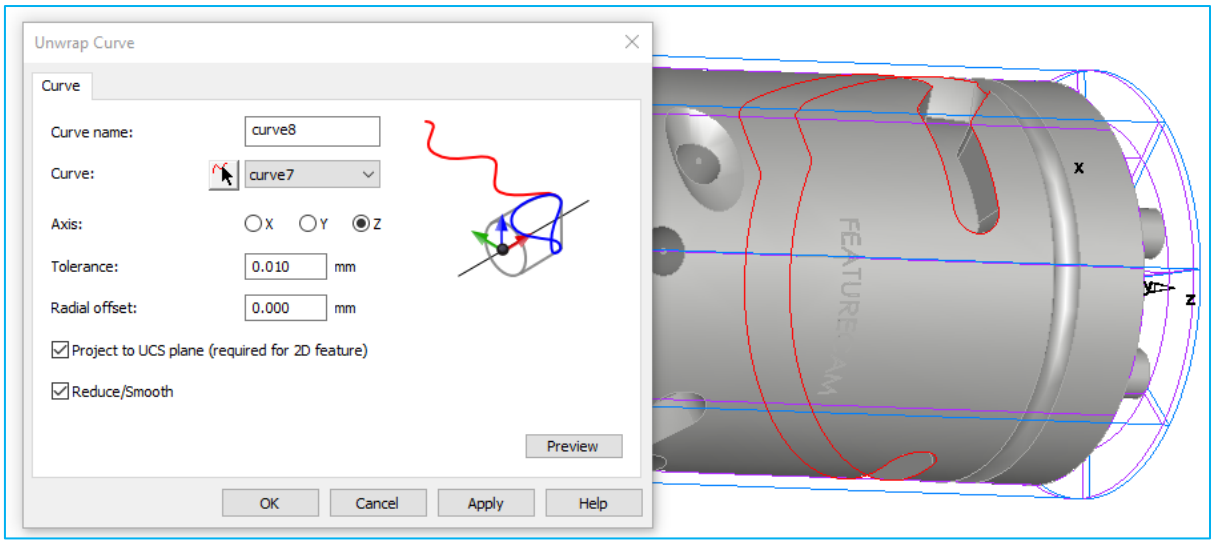
Unwrapping Curves

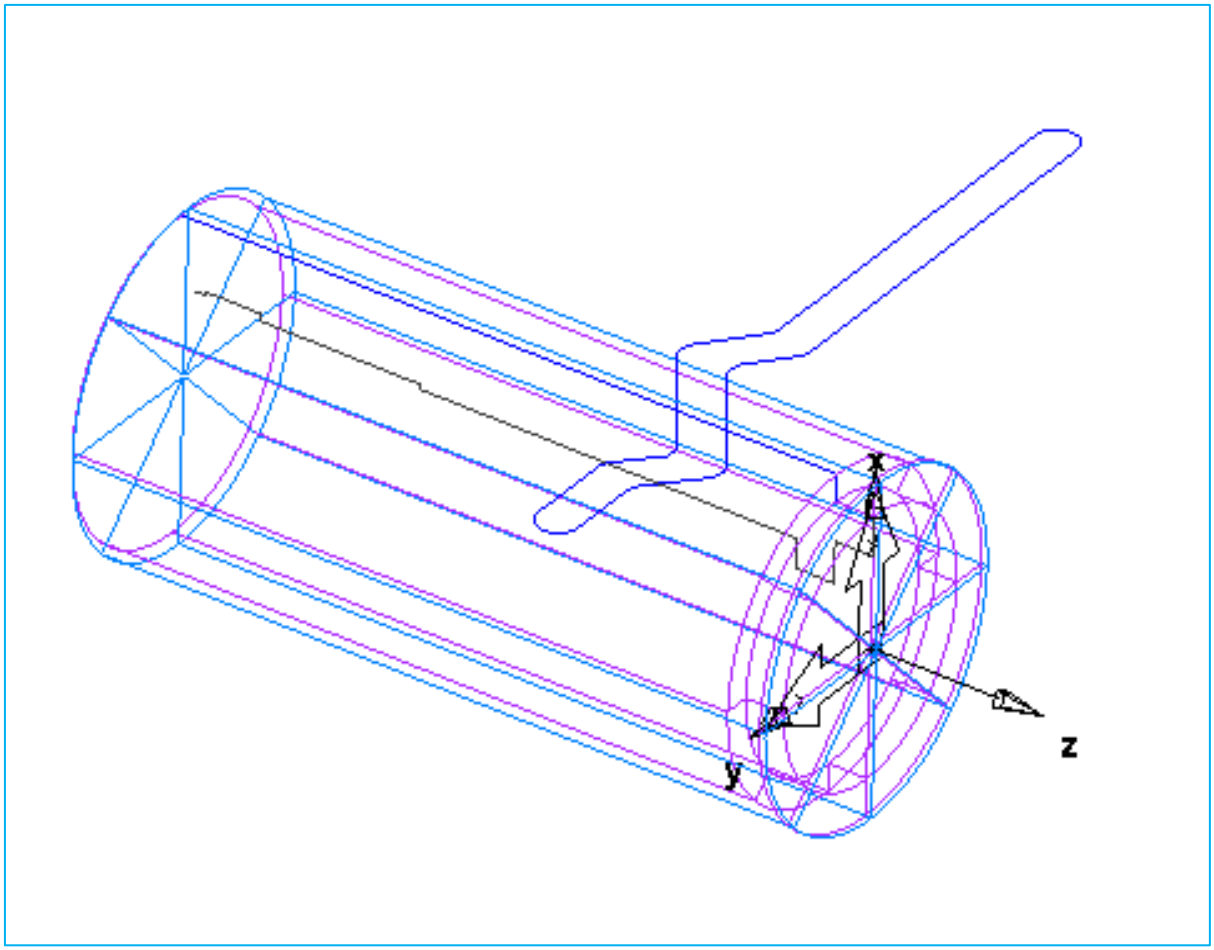
- If you are working with a curve that has been wrapped around an axis and you want to unwrap the curve into a planar curve, use **Curve unwrap**. Wrapped features in FeatureCAM require a planar curve; this function is helpful if you want to generate toolpaths from data that is already wrapped.
- To construct a curve using Unwrap:
- Use one of the following methods to display the

Unwrap Curve dialog:
- In the **Curve Wizard** select **Curve from curve**, then **Unwrap Curve**, and click **Next**.
- On the **Curves and Surfaces** toolbar, click the **Unwrap Curve** 

button in the **Curve from Curve** menu.
- From the menu, select **Construct > Curve > From Curve > Unwrap Curve**.
- Optionally enter a curve name in the **Name** field, or leave the default name.
- Select the curve you want to use from the **Curve** menu, or click the **Pick Curve** button and select it in the graphics window.
- Select the **Axis** that the curve is wrapped around. This setting defaults to the wrapping axis specified in the **Stock** wizard.
- The **Tolerance** controls the accuracy of the unwrapped curve. All unwrapped curves are piece-wise linear, so the smaller the tolerance, the more points your curve has.
- The **Radial offset** should normally be set to **0** because the curve is unwrapped in place. If you have extracted the curve at the bottom of the feature, then you need to set the **Radial offset** to the depth of the feature to offset the curve to the top.
- Select **Project to UCS plane** if you are creating a 2D feature, or deselect it if you want a 3D curve.
- Select **Reduce/Smooth** if you want to reduce the number of points in your curve.
- Click **OK** (or **Finish** if you're using the wizard).

© Copyright 2016 Autodesk, Inc. All rights reserved

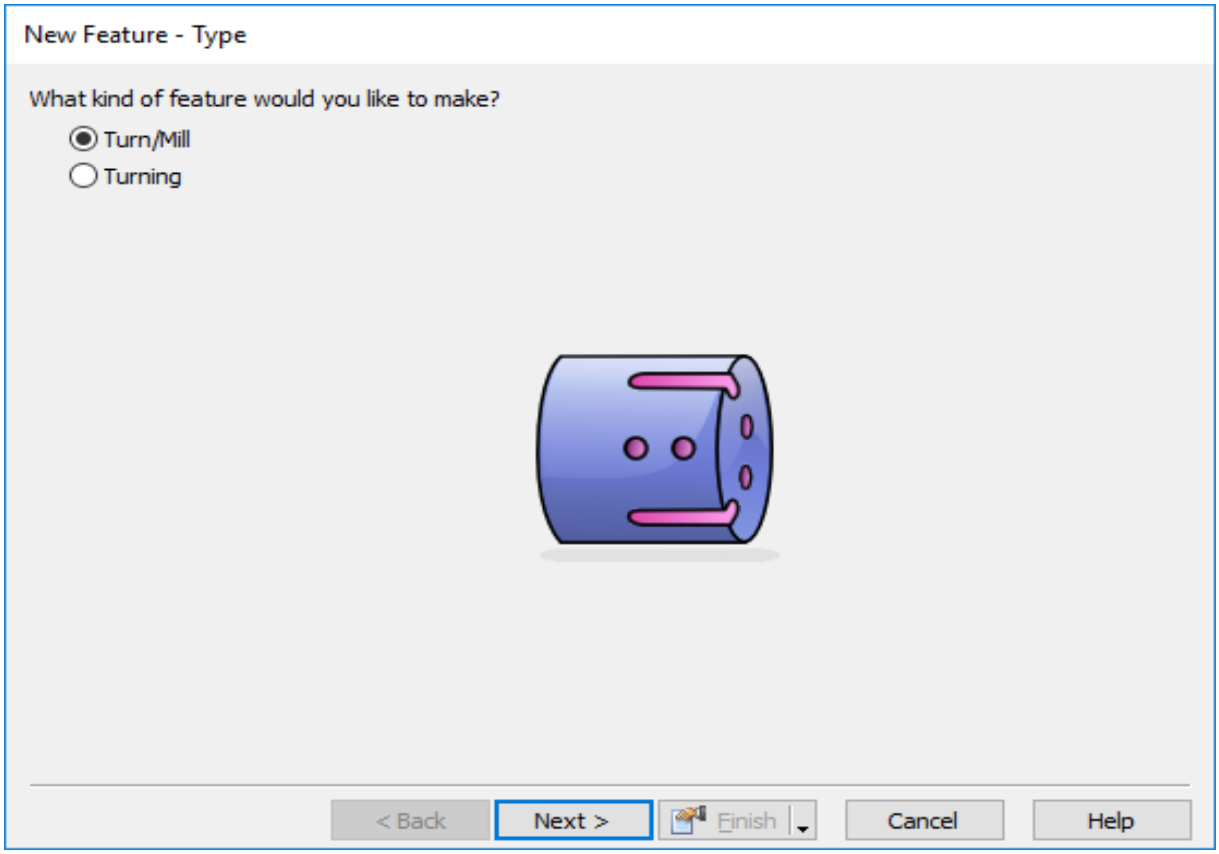




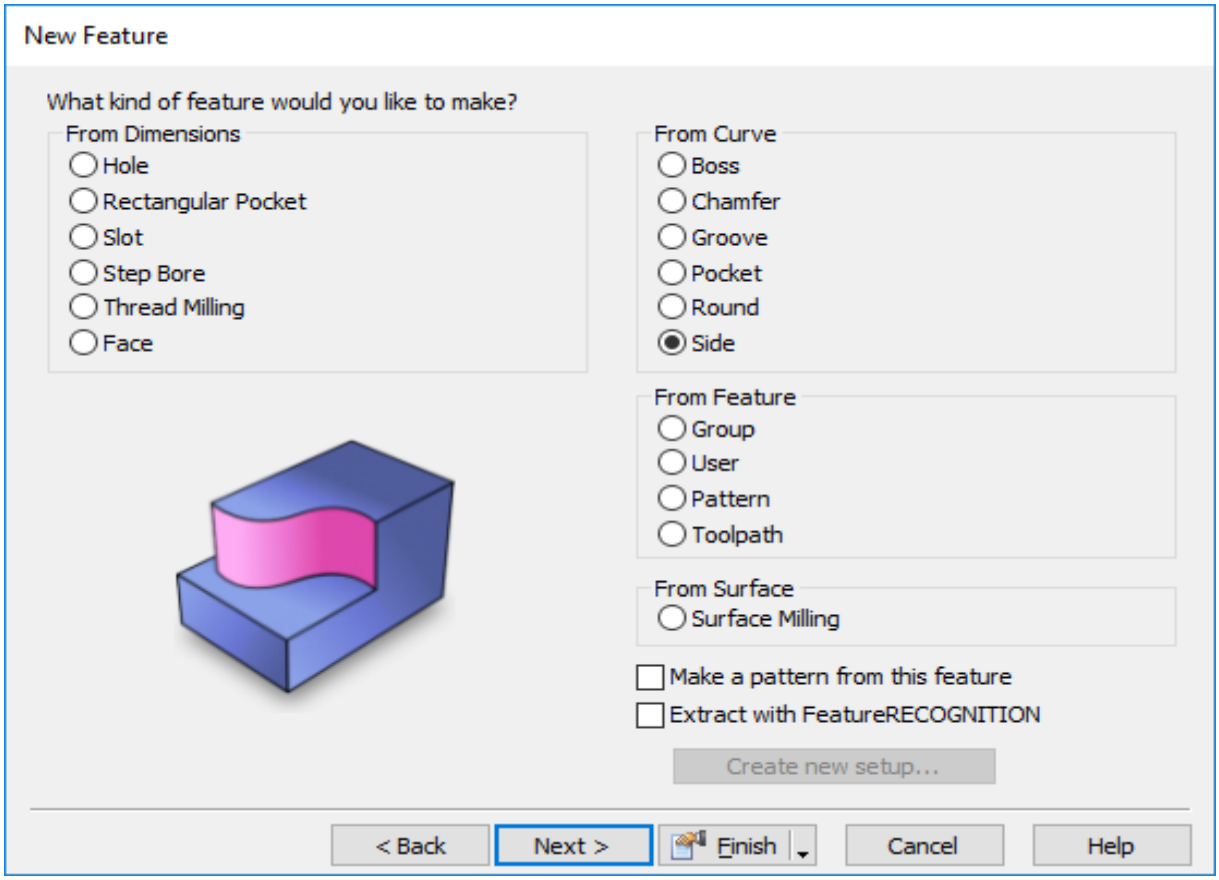
Machining the unwrapped Curve

1 To machine the unwrapped curve select a **New Feature** and select the following menu.

© Copyright 2016 Autodesk, Inc. All rights reserved

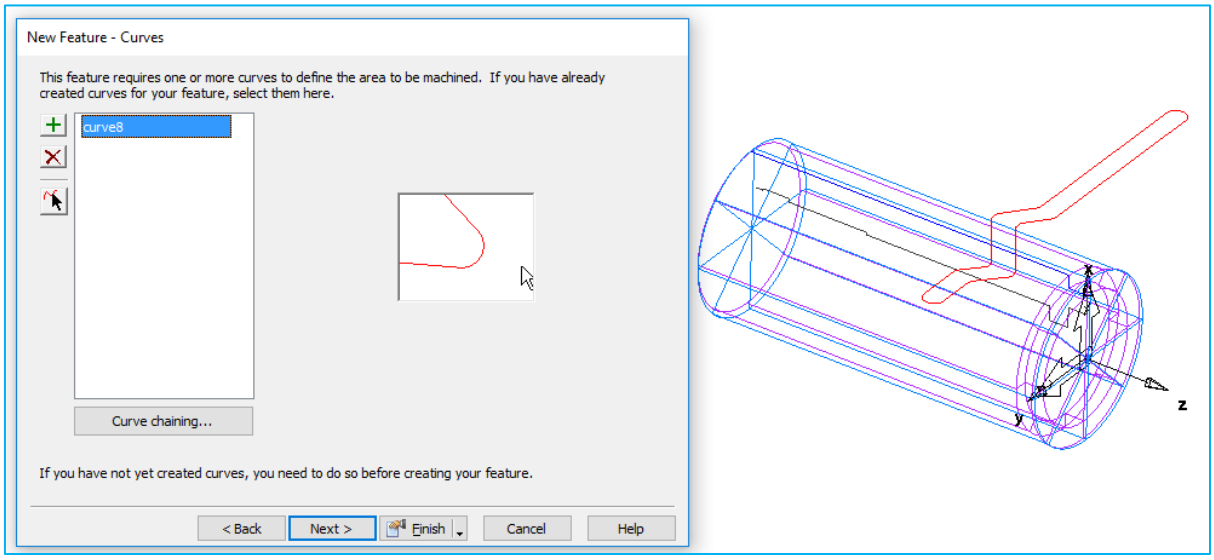


2 Select **Next**. Select **Side**.

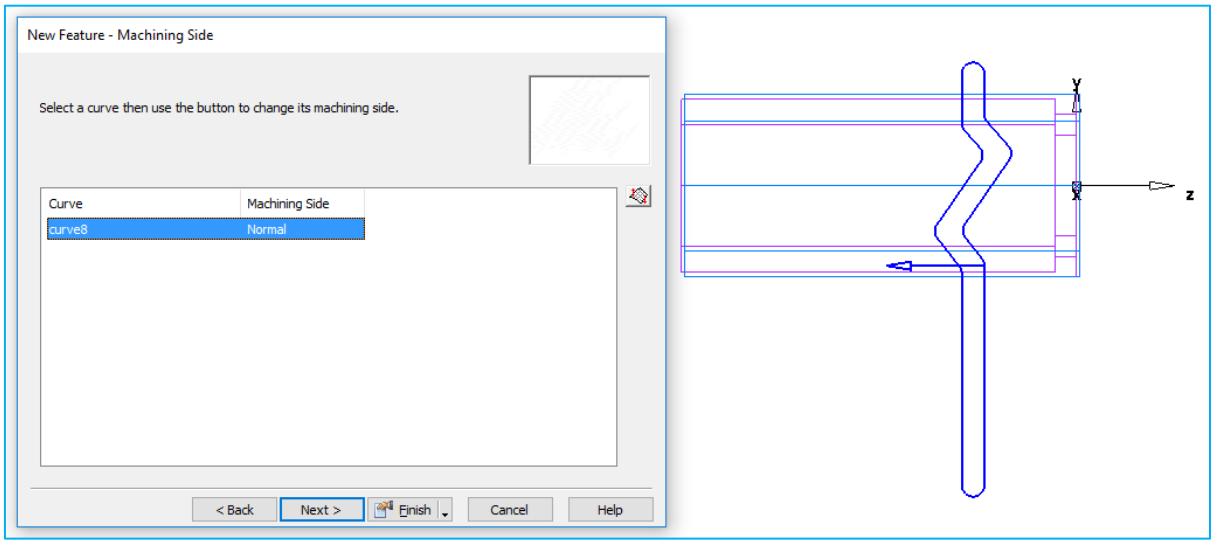


© Copyright 2016 Autodesk, Inc. All rights reserved

3 Select **Next**. Then select the curve that was unwrapped.



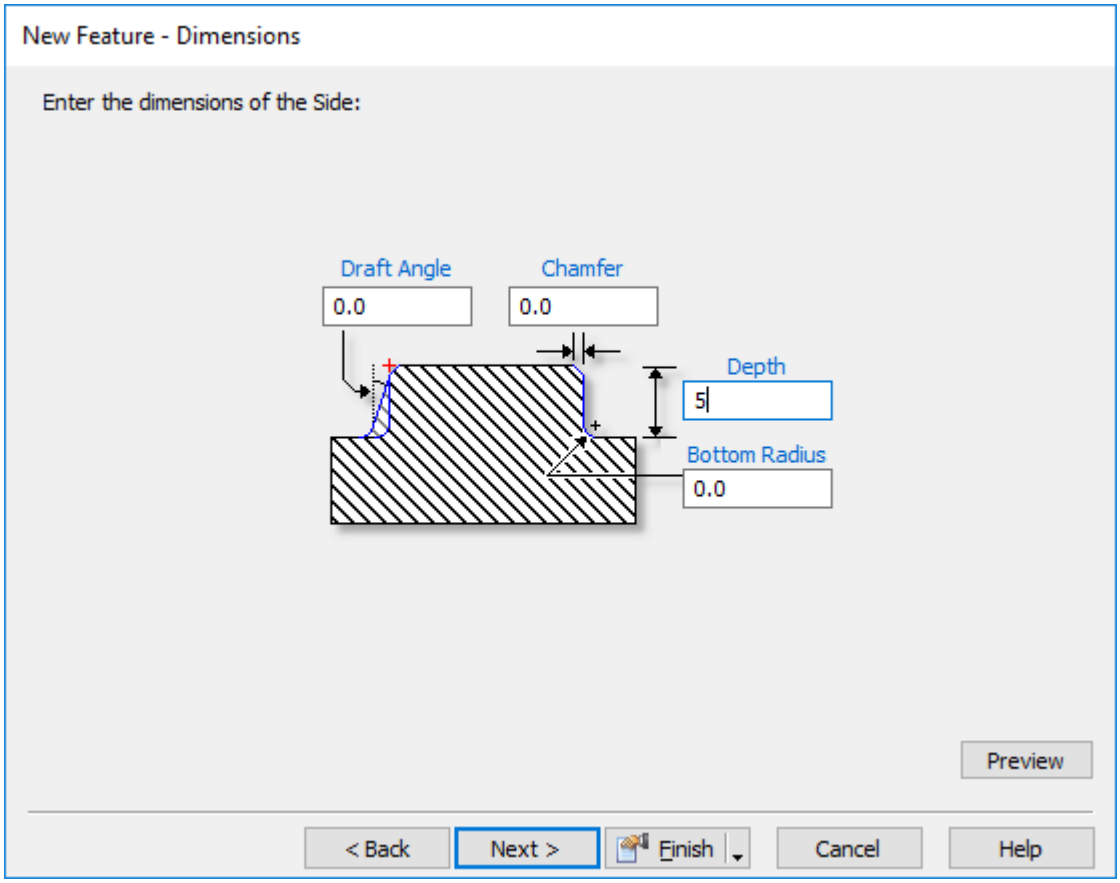
4 Select **Next.** Check Curve direction.



5 Select **Next** twice.

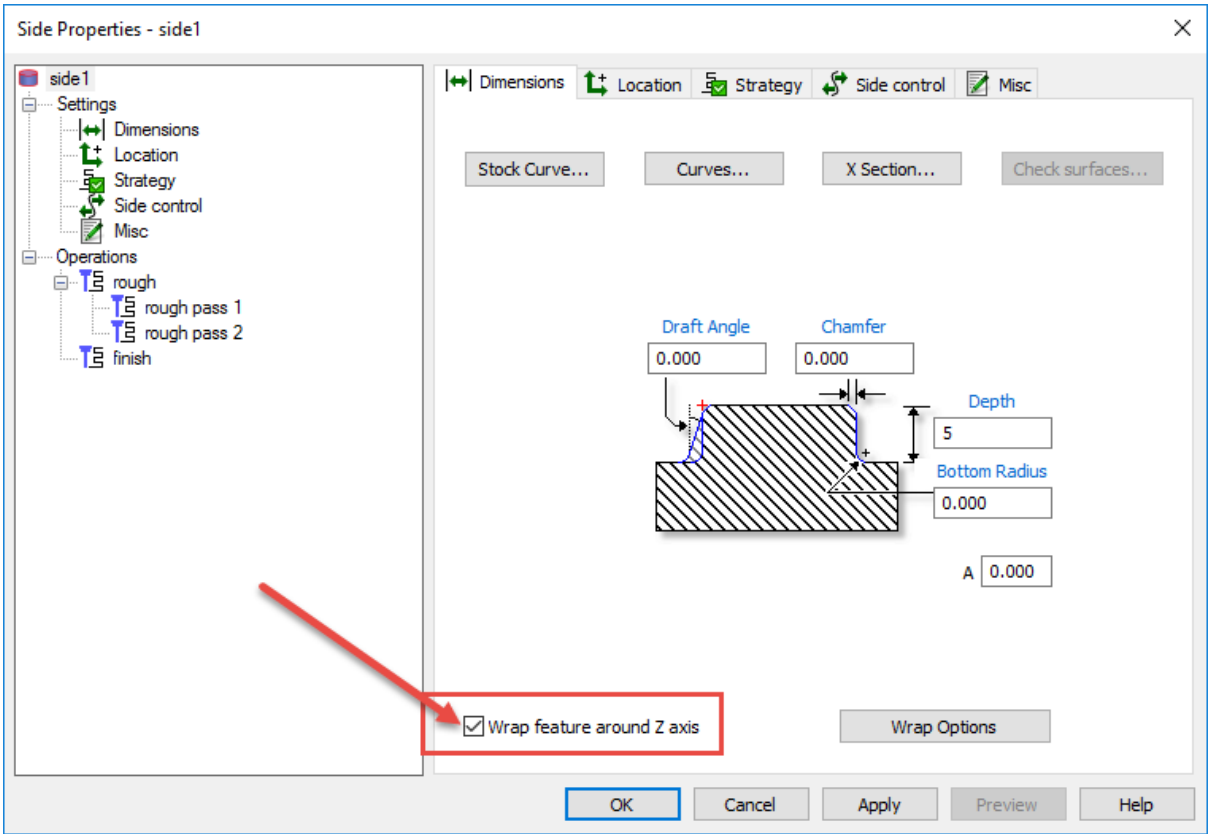
6 Enter the following information.

© Copyright 2016 Autodesk, Inc. All rights reserved



7 Select **Finish**.

8 Then select **Wrap features around the Z axis.**

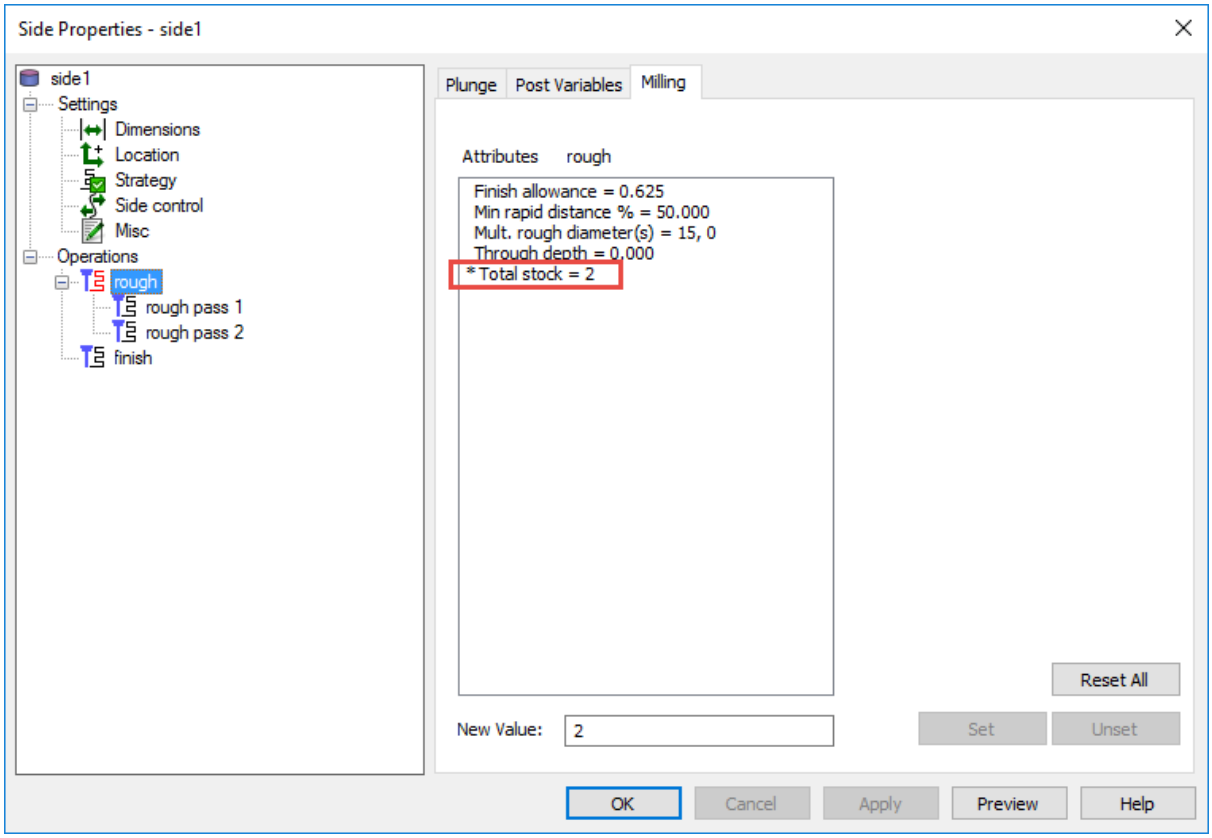


© Copyright 2016 Autodesk, Inc. All rights reserved

9 You will get an error message to set total stock.

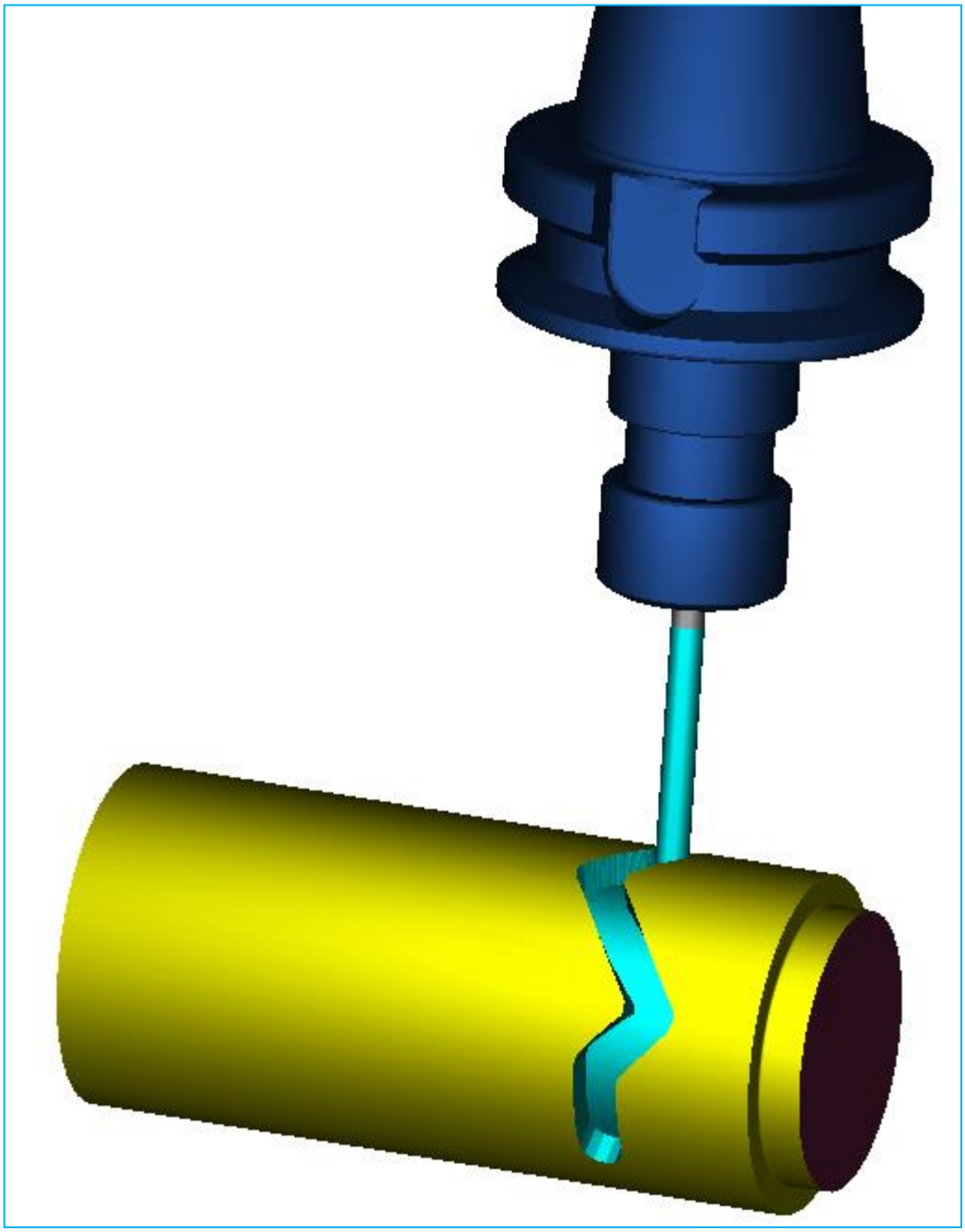
The Total Stock attribute changes the way that the feature is roughed. Instead of roughing within the boundaries of the Stock, the region that is roughed is determined by offsetting the feature's curve by the total stock amount. The toolpaths are then performed parallel to the feature's curve.

10 Set **Total stock** to **2mm**



11 Run **3D Simulation**.

© Copyright 2016 Autodesk, Inc. All rights reserved



Apply this technique to the other slots on the Part.

© Copyright 2016 Autodesk, Inc. All rights reserved

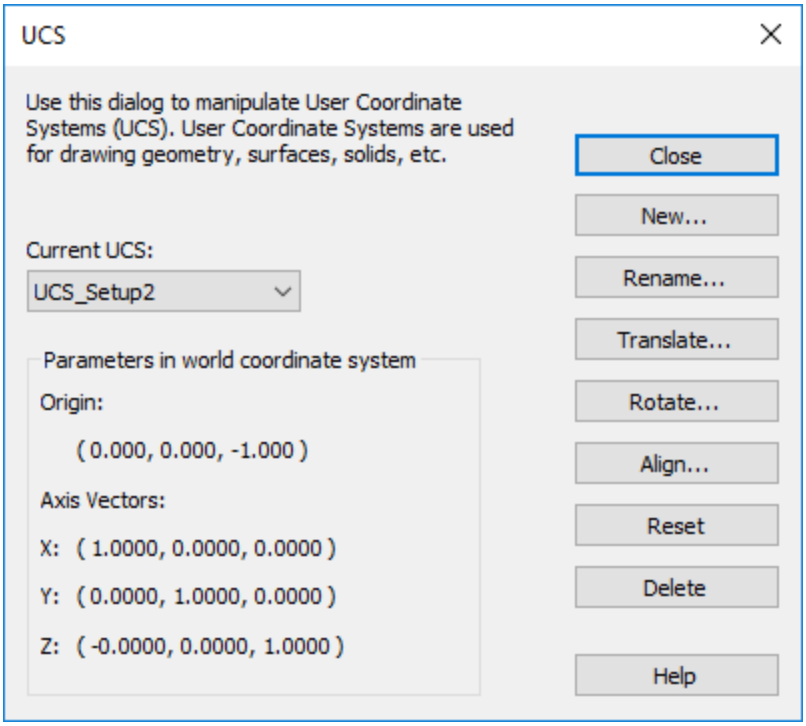
Wrapping Text around a diameter

One of the most commonly asked questions on a training course is how can you machine Text around the diameter of a turned component. The following example shows the stages you have to go through to complete this task.

*We will be using the same example **FeatureCAM Drum** for wrapping our text. We will be adding a new grooving operation to the existing FeatureCAM document.*

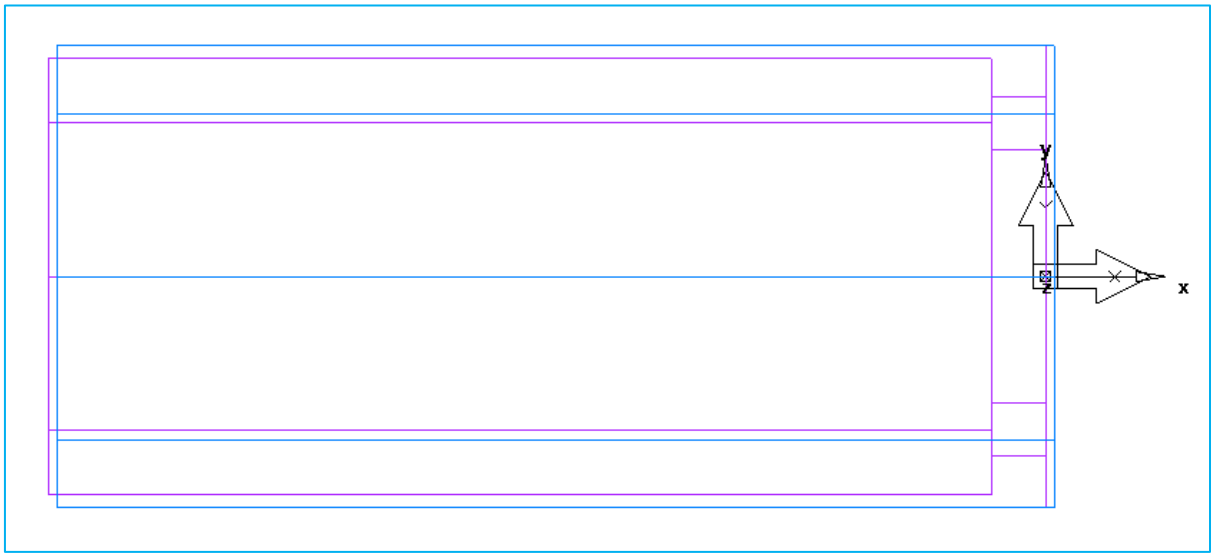
*The quickest way to achieve this is to make a temporary **Setup2***

1. Double click on Setup1 and then select **New** then select **Next**.
2. Select Align with existing UCS. Select **Next.**
3. Select **UCS_Setup1.** Select **Finish.**
4. Double click on the large arrows on **Setup2**.
5. The following form will appear.



6. Select Rotate. Then rotate the axis so that the Z axis is pointing in the direction of the Diameter. Select front to see if the angle is correct.

© Copyright 2016 Autodesk, Inc. All rights reserved



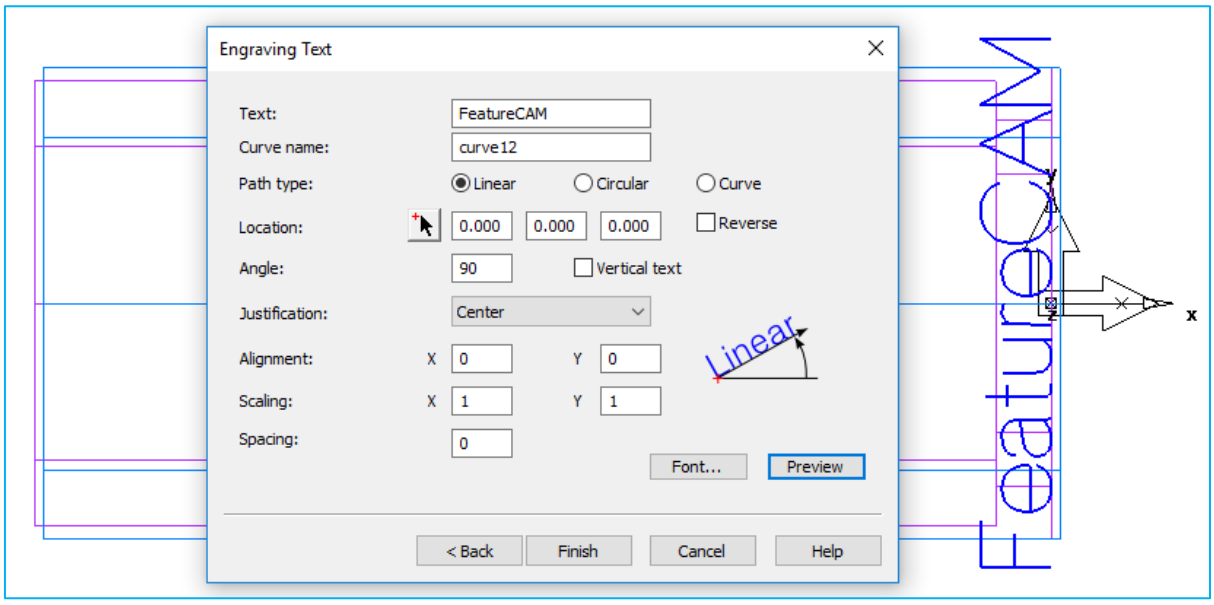
You now have a new setup2 with X, Y, Z coordinates. We can manipulate this to create our Text onto a new plane.

7 We are now ready to create our text. Select the **Curve Wizard** from the **Advanced menu** tool bar, or select **Construct>Curve>Curve Wizard** from the main menu.

8 Select **Text** from **Other Methods**. Type in **FeatureCAM** into the **Text Field**.

9 Click on the preview button to see where the text is.

10 Select **Centre** in **Justification** and **Angle 90**.

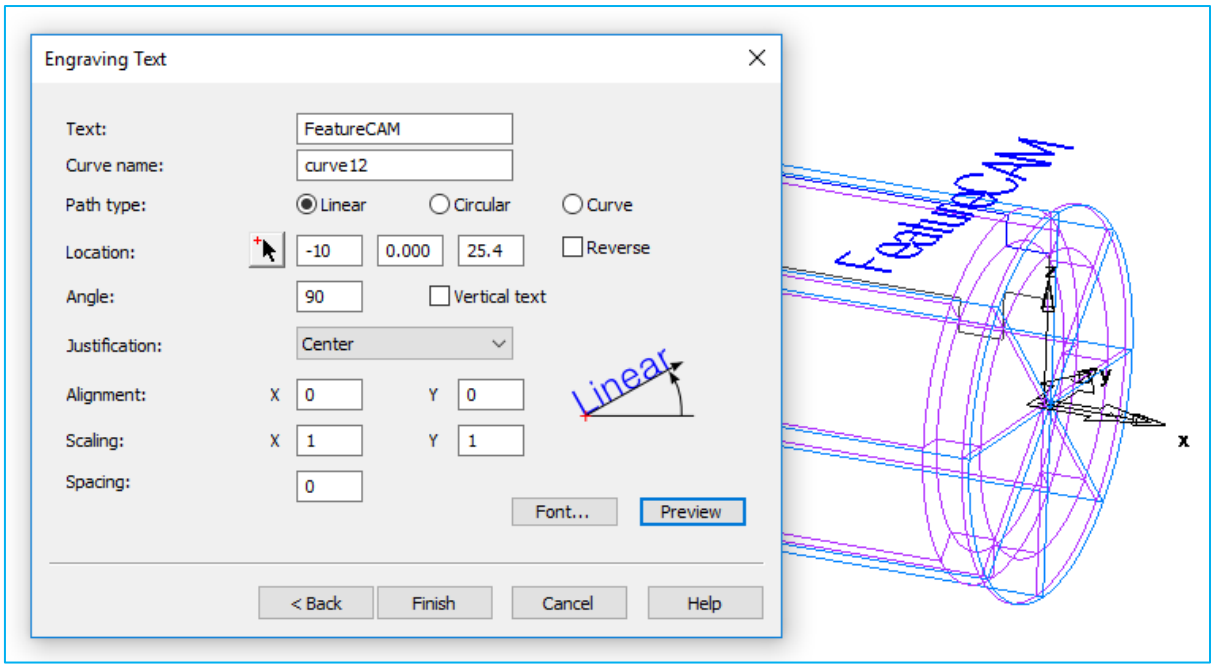


11 We now need to move the Text to the correct position. Type in the following.

12 **X-10** and **Z25.4**

13 Select **Finish** closing the form.

© Copyright 2016 Autodesk, Inc. All rights reserved

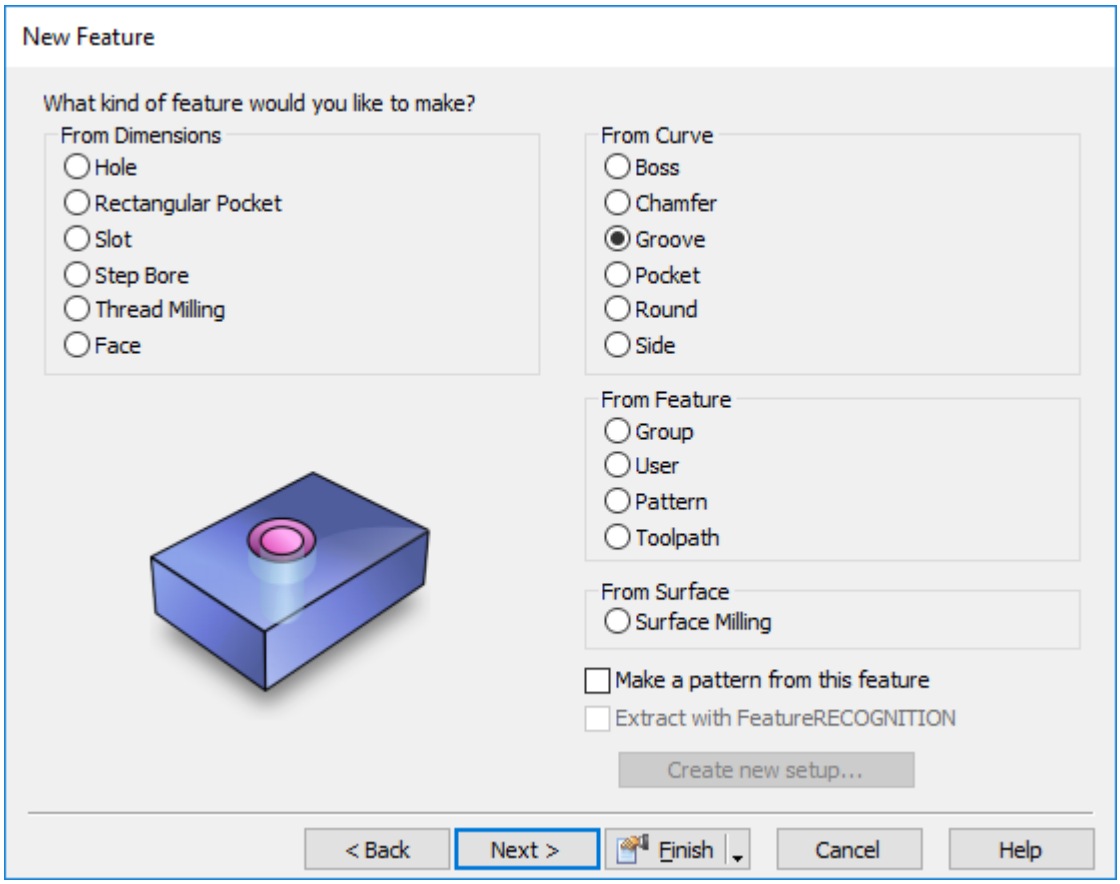


14 Once in position delete **Setup2**

15 We are now ready to machine the curve using a **Simple (Engraving) Groove Feature**. Set width to 0.1 and depth to 0.1.

16 Select **New Feature>Turn/Mill**.

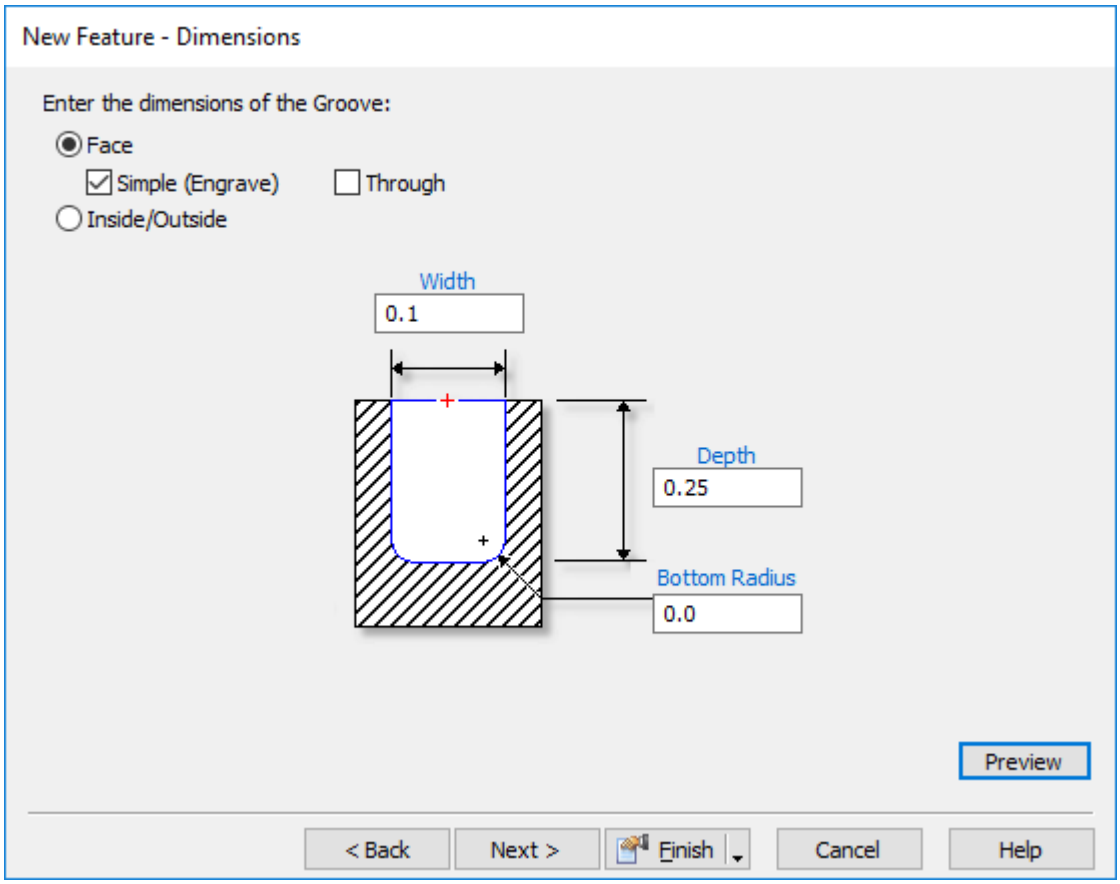
17 Select **Next**. Select **Groove**. Select **Next**.



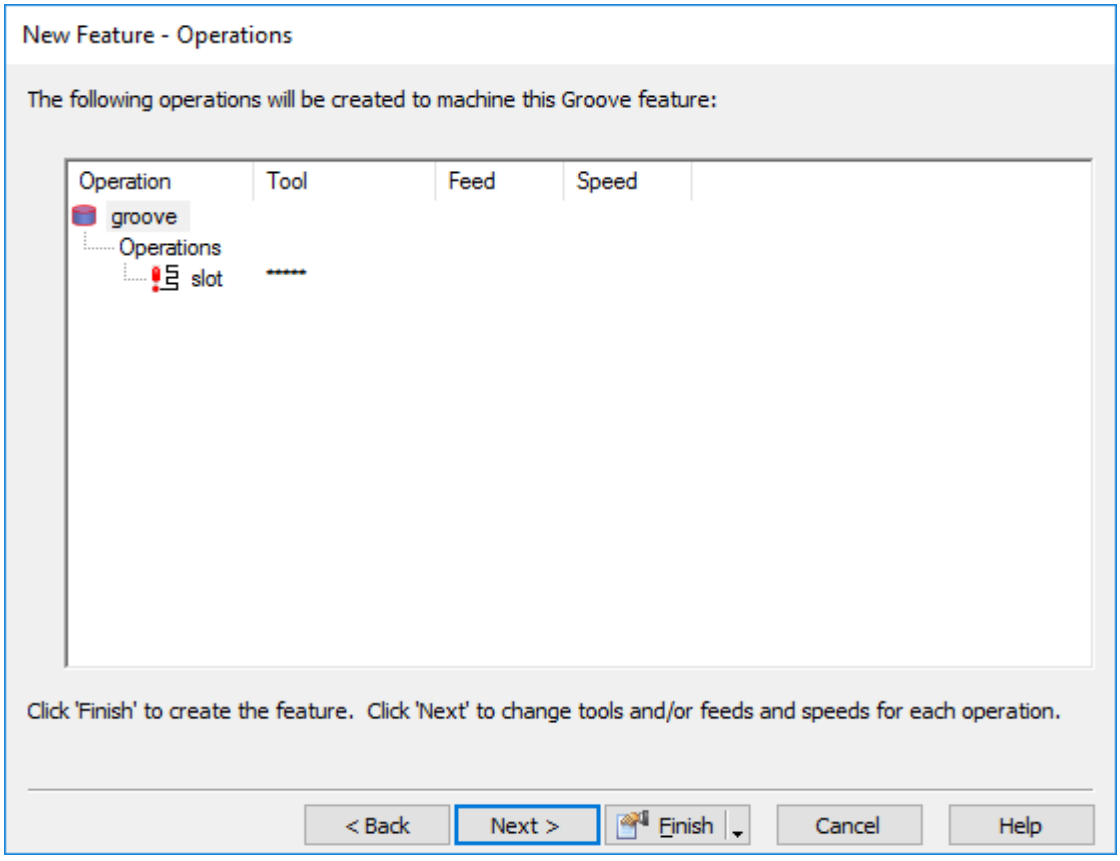
18 Select the **Curve** we have just created. Select **Next** twice.

© Copyright 2016 Autodesk, Inc. All rights reserved

19 Enter the Feature Dimensions as shown.

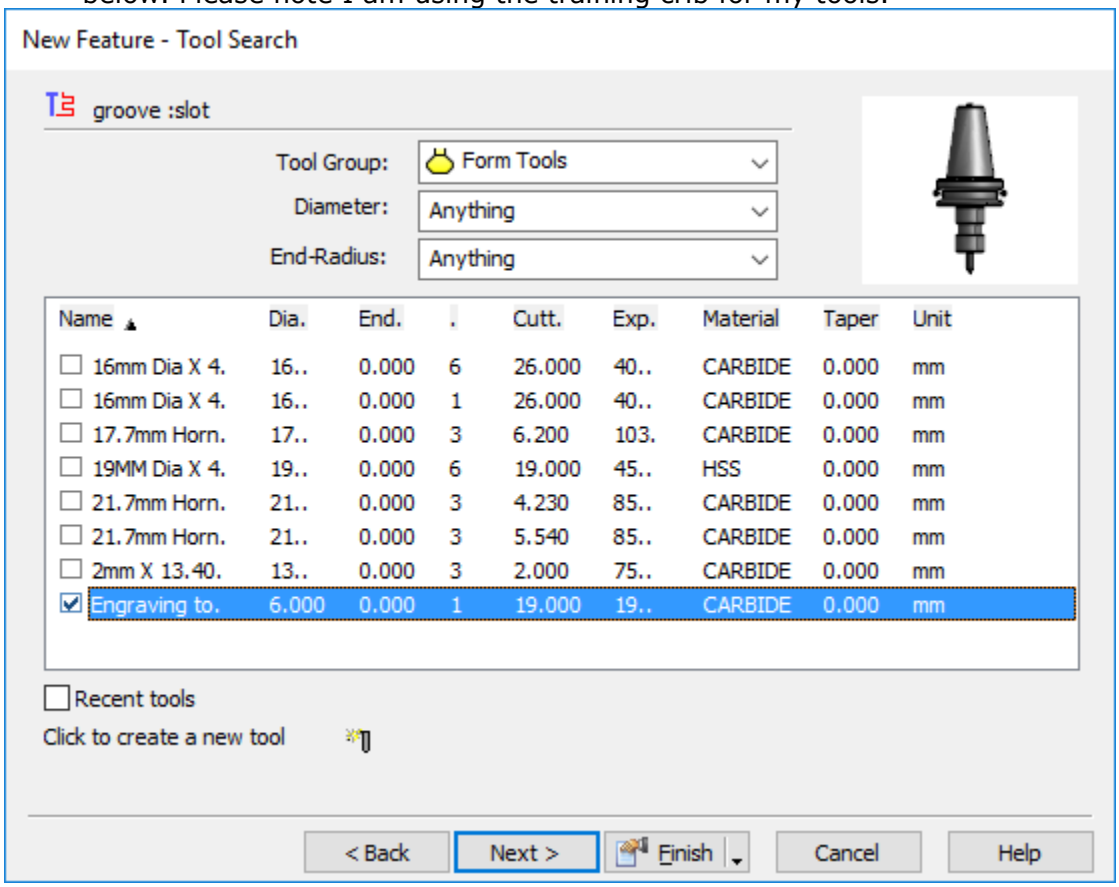


20 Enter **Next** until you get to the form where you can select your tool.

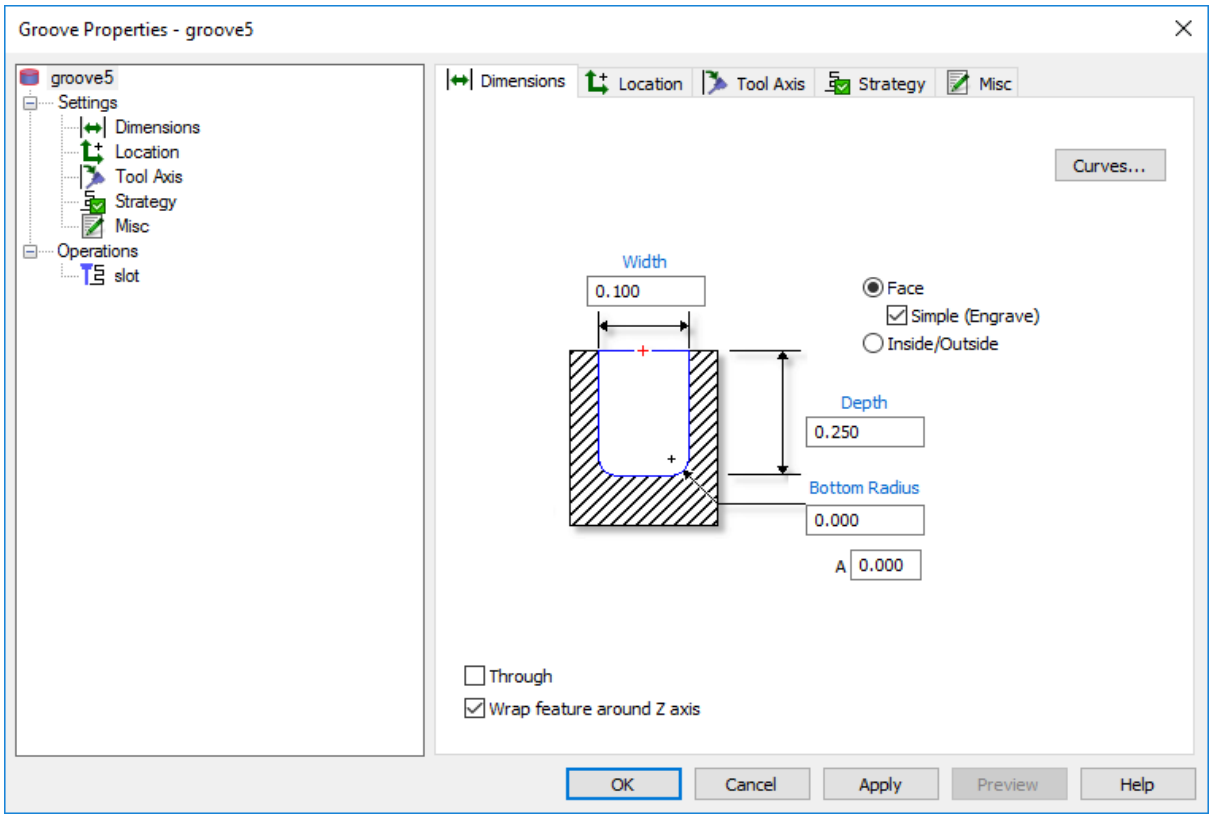


© Copyright 2016 Autodesk, Inc. All rights reserved

21 Select Next where you can search for another tool. Select the tool shown below. Please note I am using the training crib for my tools.



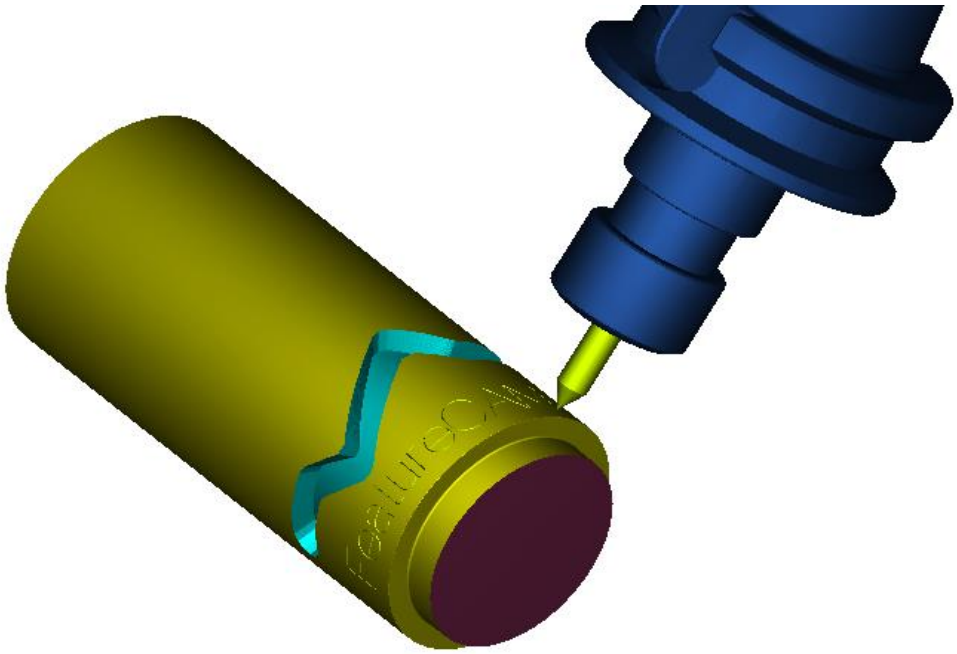
22 Select **Finish**. The following menu will appear. Select **Wrap feature around the Z axis.**



© Copyright 2016 Autodesk, Inc. All rights reserved

Select **Wrap Feature around Z axis**. Select A**pply** and **Ok**.

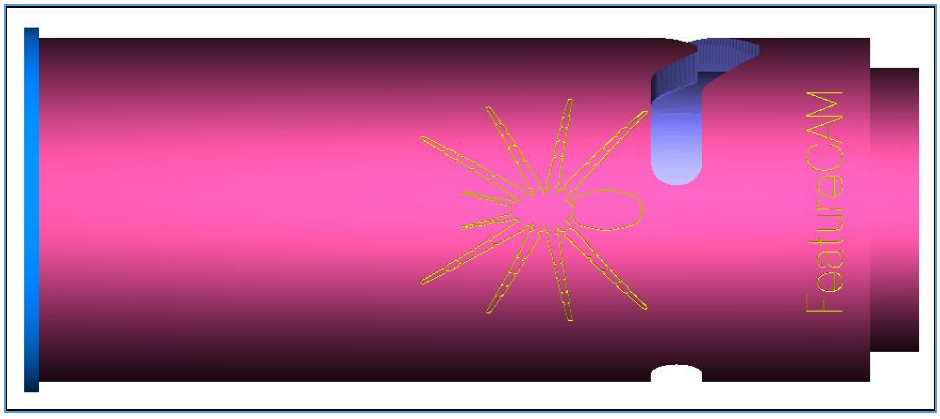
Run **3D Simulation**.



Wrapping a 2D Logo around the outside diameter.

For customers who wish to wrap a 2D logo around the outside diameter please load file:- ***Bonus - FeatureCAM Drum with Text + Spider logo Start.fm*** *from the* ***Turning Data .fm Files*** *Folder.*

1. **Open** File.
2. Create a **Curve** from the **2D Spider Geometry**.
3. **Transform** the Curve to the middle of the turned profile.
4. Remember to translate the curve to the radius of the Part (25.4mm)
5. Create a **Simple Groove** Feature.
6. Use the same tool you have used for the text.
7. Depth 0.1mm Width 0.1mm.



© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

B-Axis Interactive Feature Recognition 2
Exercise - Bonus – rounded face 8

© Copyright 2016 Autodesk, Inc. All rights reserved

B-Axis Interactive Feature Recognition

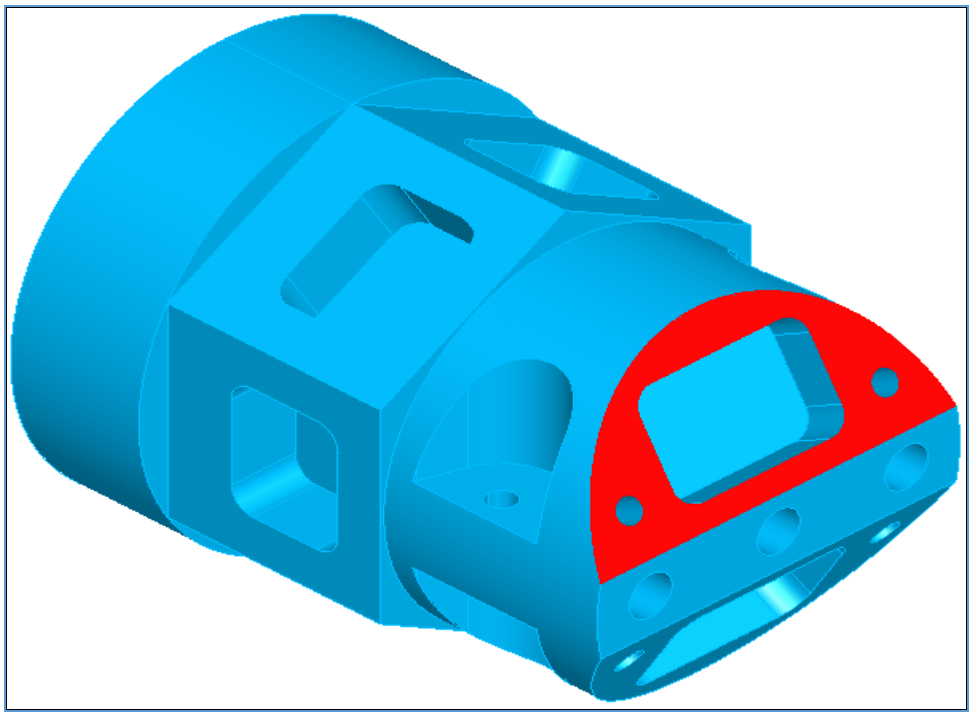
In this example we will be looking at various ways of identifying features from solids when using a **B-Axis Turn/Mill**. In previous versions of **FeatureCAM** it was necessary to create a setup with its Z axis aligned with the direction the tool would approach the feature before it could be recognized. 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.

1. Open the file **B-Axis tutorial start.fm** from **C:\Training_Data\FeatureCAM Course Data 2017\Turning Data .fm Files**
2. Load the **baxusturningtraining.CNC** file from the Training Post folder.
3. Create a **Turn feature** from the curve **curve1** using the **default values**
4. 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.

*Show in **Isometric** view and **Shade** the model*

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



5. Open the **New feature** wizard
6. Under New Feature Type select a new **Turn/Mill** feature
7. Click **Next**
8. Check **Face** and **Extract with Feature RECOGNITION**.
9. Click **Next**

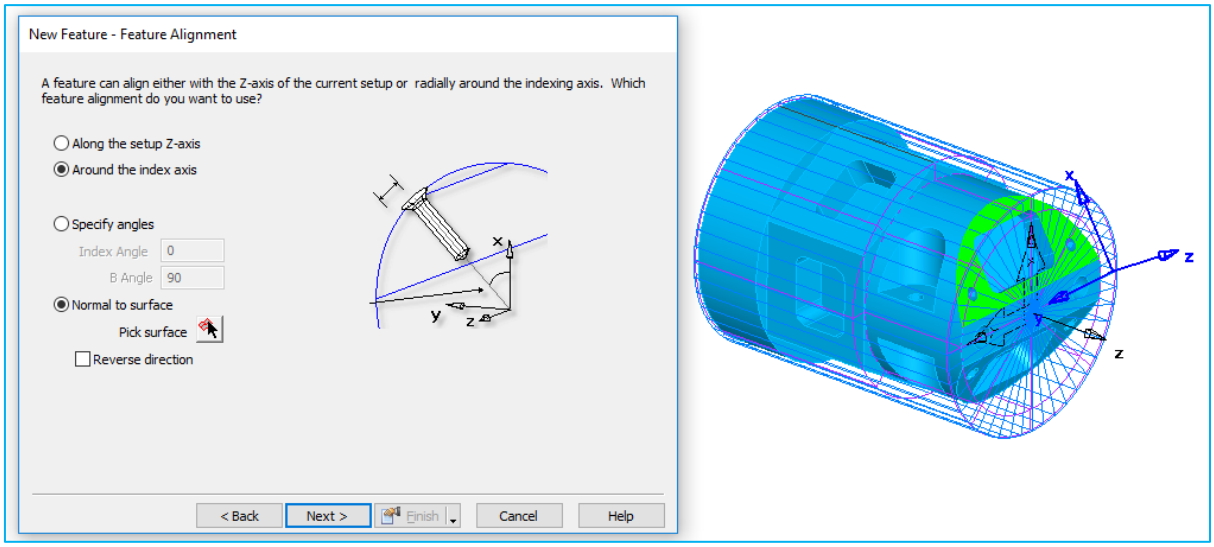
© Copyright 2016 Autodesk, Inc. All rights reserved

*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.*

10 Check **Around the index axis** and then **Normal to surface**

11 Click on **Next**

12 Click on the **Pick surface** button and then select the **face**



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.

13 Click **Next**

14 Click **Add from selected items** (the green plus on the upper left)

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

Select a roughing operation and change the Z increment to 2.5mm

Try changing the direction of cut to 90 degrees.

The face feature has been automatically recognized.

Repeat the process to machine the face on the opposite side.

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.

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

© Copyright 2016 Autodesk, Inc. All rights reserved

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

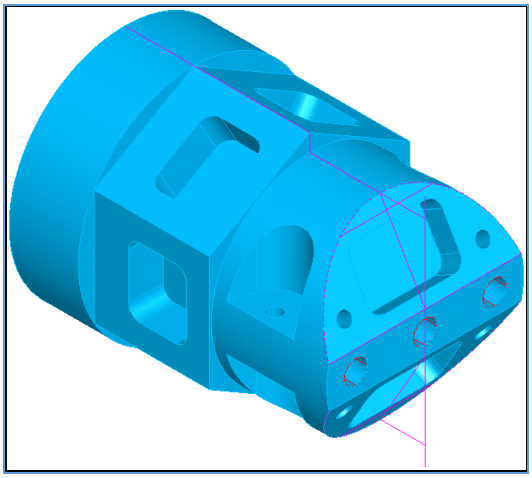
17 Create a new **Turn/Mill** feature

18 Select **Hole** and **Extract using Feature RECOGNITION**

19 Click **Next** and **select Along the setup Z axis**

20 Click **Next** twice until you reach the **Hole recognition options** page

21 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.

22 Open the **New feature** wizard

23 Under New Feature Type select a new **Turn/Mill** feature

24 Click **Next**

25 Check **From curve** - **Pocket** and **Extract with Feature RECOGNITION**

26 Click **Next**

27 Check **Around index axis**

28 Check **Normal to surface**

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

30 Click on **Next**

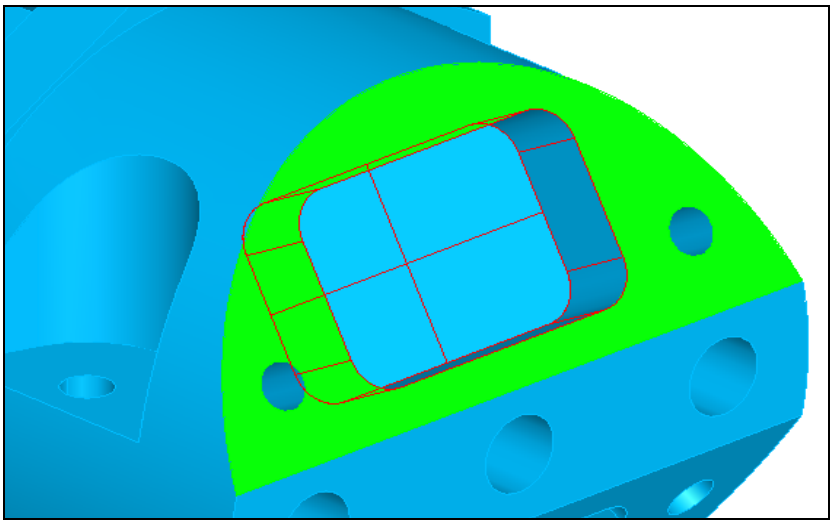
31 Select **Automatic recognition**

32 Click on **Next**

33 Click **Select all**

34 Click **Finish**

© Copyright 2016 Autodesk, Inc. All rights reserved



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.

35 Open the **New feature** wizard

36 Under New Feature Type select a new **Turn/Mill** feature

37 Click **Next**

38 Check **From curve** - **Side and Extract with Feature RECOGNITION**

39 Click **Next**

40 Check **Around index axis**

41 Check **Specify Angles** and set **0** and **90** as shown.

42 Click **Next**

43 Check **Automatic recognition**

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

44 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.

45 Open the **New feature** wizard

46 Under New Feature Type select a new **Turn/Mill** feature

47 Click **Next**

48 Check **Hole** and **Extract with Feature RECOGNITION**

49 Click **Next**

© Copyright 2016 Autodesk, Inc. All rights reserved

50 Check **Around index axis and Automatic**

51 Click **Next**

52 Check **Exclude holes**… and set the **diameter** to 0.3in.

53 Click **Next**

54 Click **Select all**

55 Click **Finish**

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

56 Open the New feature wizard

57 Under New Feature Type select a new Turn/Mill feature

58 Click Next

59 Check Side and Extract with Feature RECOGNITION

60 Click Next

61 Check Along setup Z axis

62 Click **Next**

63 Check **Automatic recognition**

64 Click **Next**

65 Click **Select all** and then click on the round side feature to deselect it

66 Click **Finish**

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

67 Open the **New feature** wizard

68 Under New Feature Type select a new **Turn/Mill** feature

69 Click **Next**

70 Check **From curve** - **Pocket and Extract with Feature RECOGNITION**

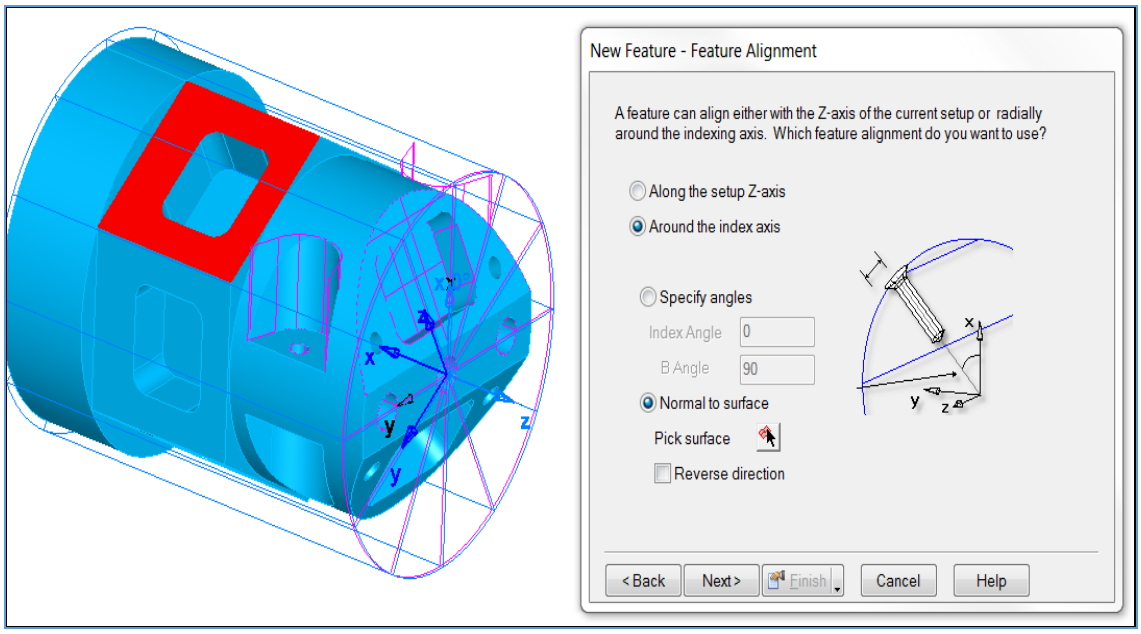
71 Click **Next**

72 Check **Around index axis**

73 Check **Normal to surface**

74 Click on the **Pick surface button** and then select **face_395**

© Copyright 2016 Autodesk, Inc. All rights reserved



75 Click **Next**

76 Select **Automatic recognition**

77 Click **Select all** and then **Finish**

78 Open the **New feature** wizard

79 Under New Feature Type select a new **Turn/Mill** feature

80 Click **Next**

81 Select **Pattern** and click **Next**

82 Make sure that **pocket3** is selected then click **Next**

83 Check **Radial** about the **index** axis and click **Next**

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

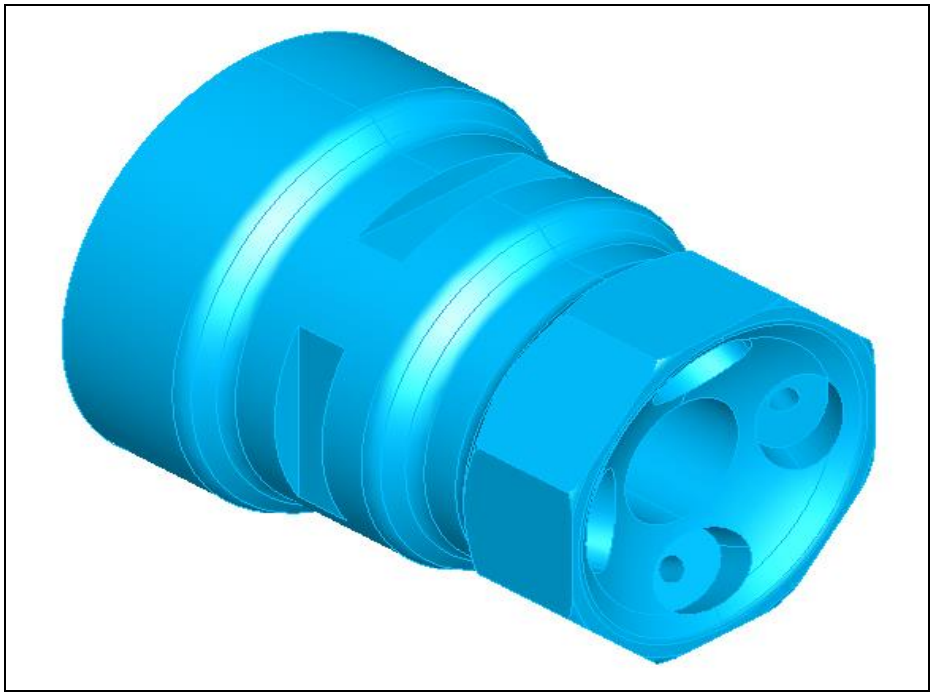
85 Click **Finish**

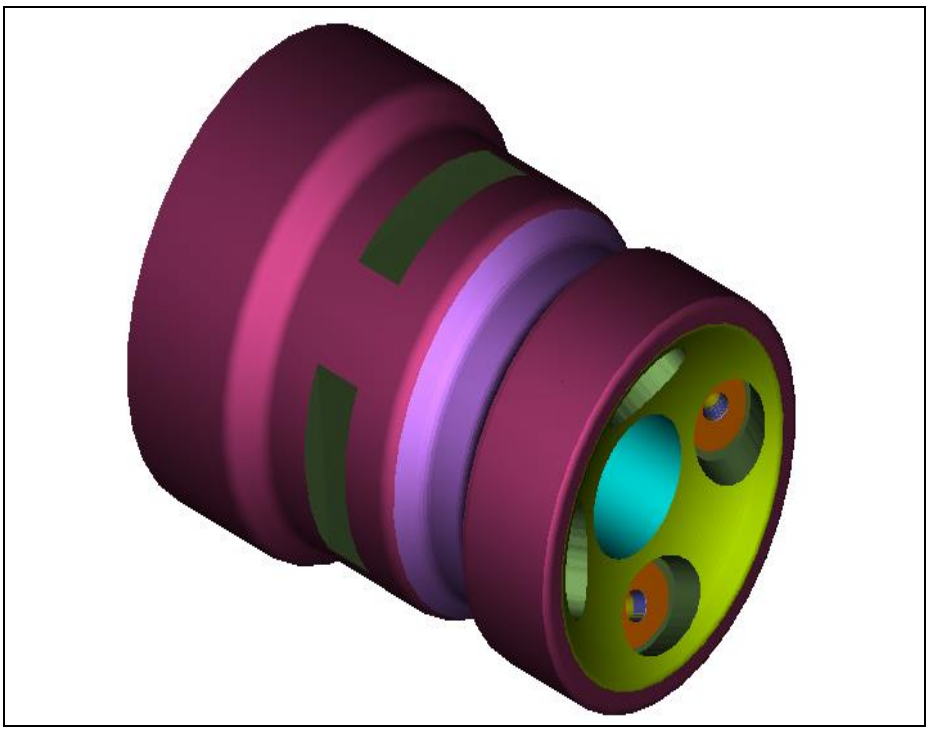
86 Run a Machine simulation to see the results.

© Copyright 2016 Autodesk, Inc. All rights reserved

Exercise - Bonus – rounded face

- Import the model: - **Bonus - rounded face. x_t** from **C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**
- Have a go at the following exercise. Use the same post used in the previous exercise.





© Copyright 2016 Autodesk, Inc. All rights reserved

© Copyright 2016 Autodesk, Inc. All rights reserved