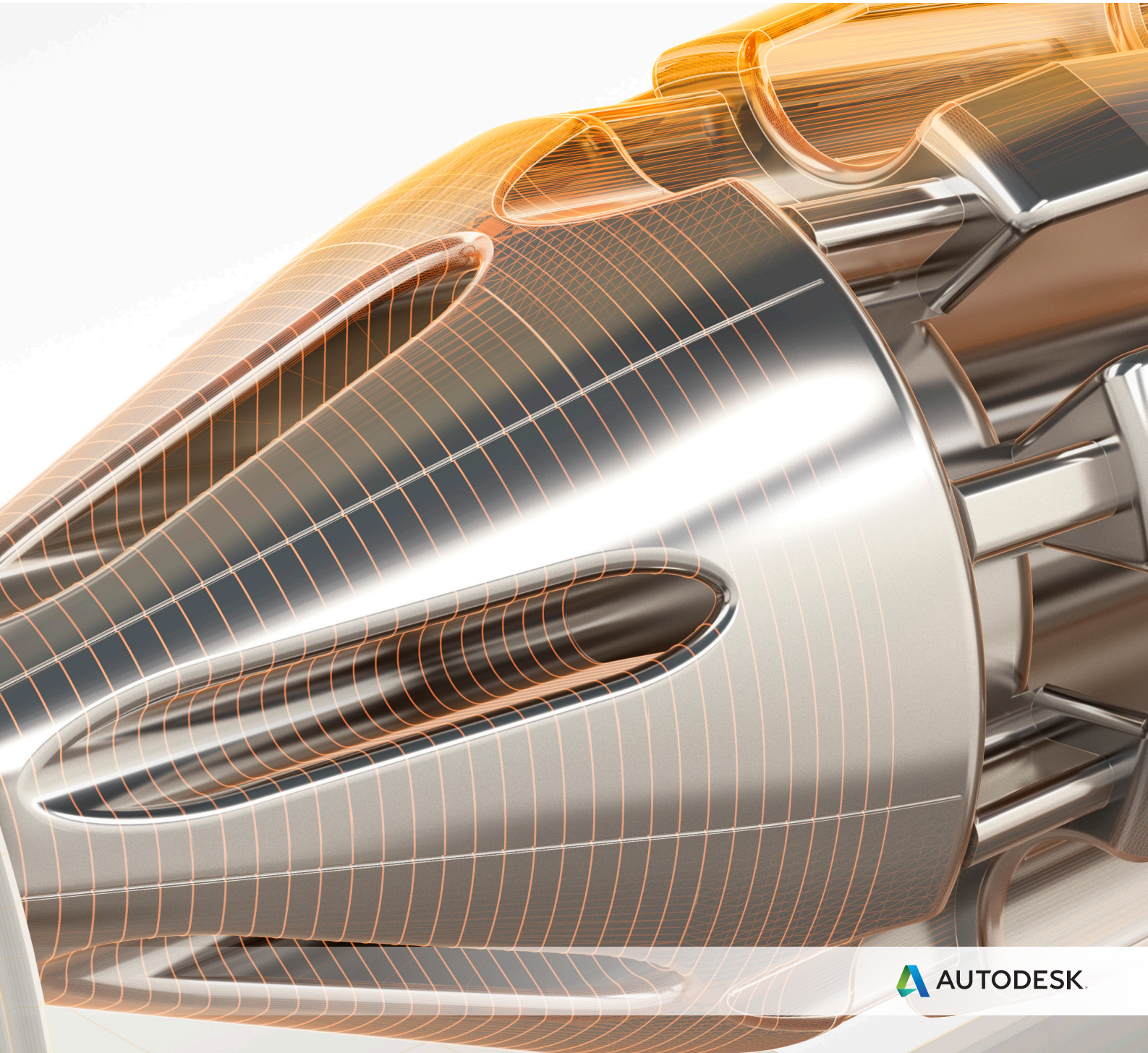
AUTODESK® FEATURECAM®

Automated to make parts faster

Training Course



AUTODESK.

Contents

Introduction to FeatureCAM Swiss Turning 3
CAM Software Strategies 4
Feature-Based Machining 5
Built in intelligence 6
Starting FeatureCAM 6
Evaluation Options 7
Desktop Subscription 10
Access options 10
Flexible term lengths 10
Access to the latest software 10
More about updates and releases 10
Autodesk support 11
Starting a new FeatureCAM part document. 11
User Interface 11
Screen Layout 12
Steps Menu 15
Steps to take to create a part in FeatureCAM 15
Importing a file into FeatureCAM 16
Stock 16
Steps (See menu in PartView) 17
Customize Manufacturing. 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 36
What is a Feature? 37
How is a Feature Created? 38
From Dimensions 39
From Curve 39

© Copyright 2016 Autodesk, Inc. All rights reserved

Saving your work 42
Save Options 43
Send Part Files dialog 44
FeatureCAM file types 45
Part Documentation 45
Documentation tab 45
Preview Picture tab 46
Saving your settings 47
Import/Export 48
Importing Files 50

© Copyright 2016 Autodesk, Inc. All rights reserved

Introduction to FeatureCAM Swiss 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.*

*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.*

© Copyright 2016 Autodesk, Inc. All rights reserved

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.*

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?

© Copyright 2016 Autodesk, Inc. All rights reserved

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. The toolpath is automatically optimized for faster machining.

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.**

© Copyright 2016 Autodesk, Inc. All rights reserved

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.*

Starting FeatureCAM

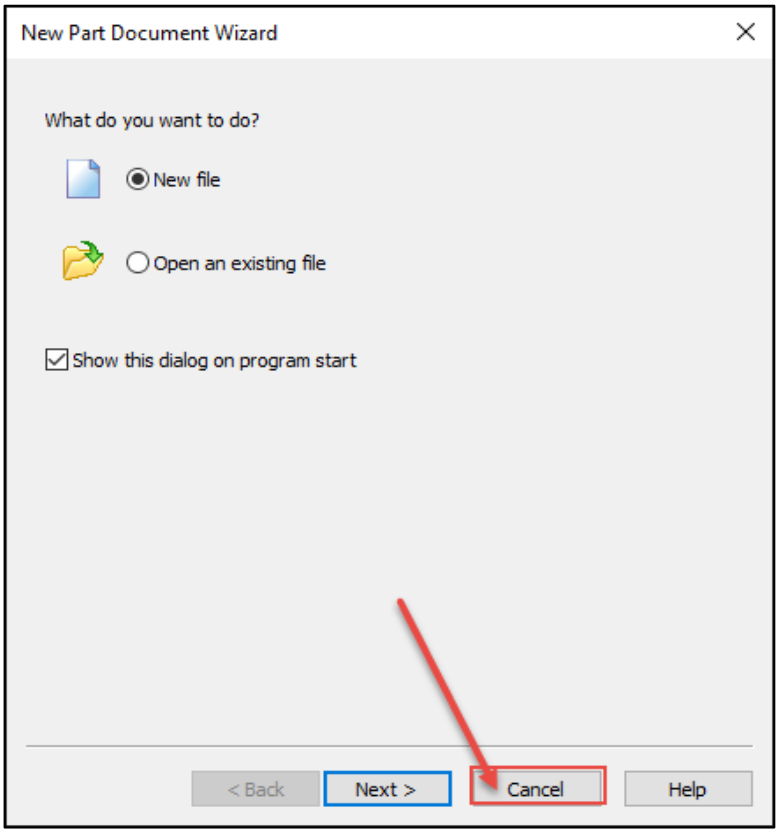
1. Select the FeatureCAM icon

from your desktop and select **New File** and **Swiss Turning 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.

© Copyright 2016 Autodesk, Inc. All rights reserved

Evaluation Options

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



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

© Copyright 2016 Autodesk, Inc. All rights reserved

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			✓

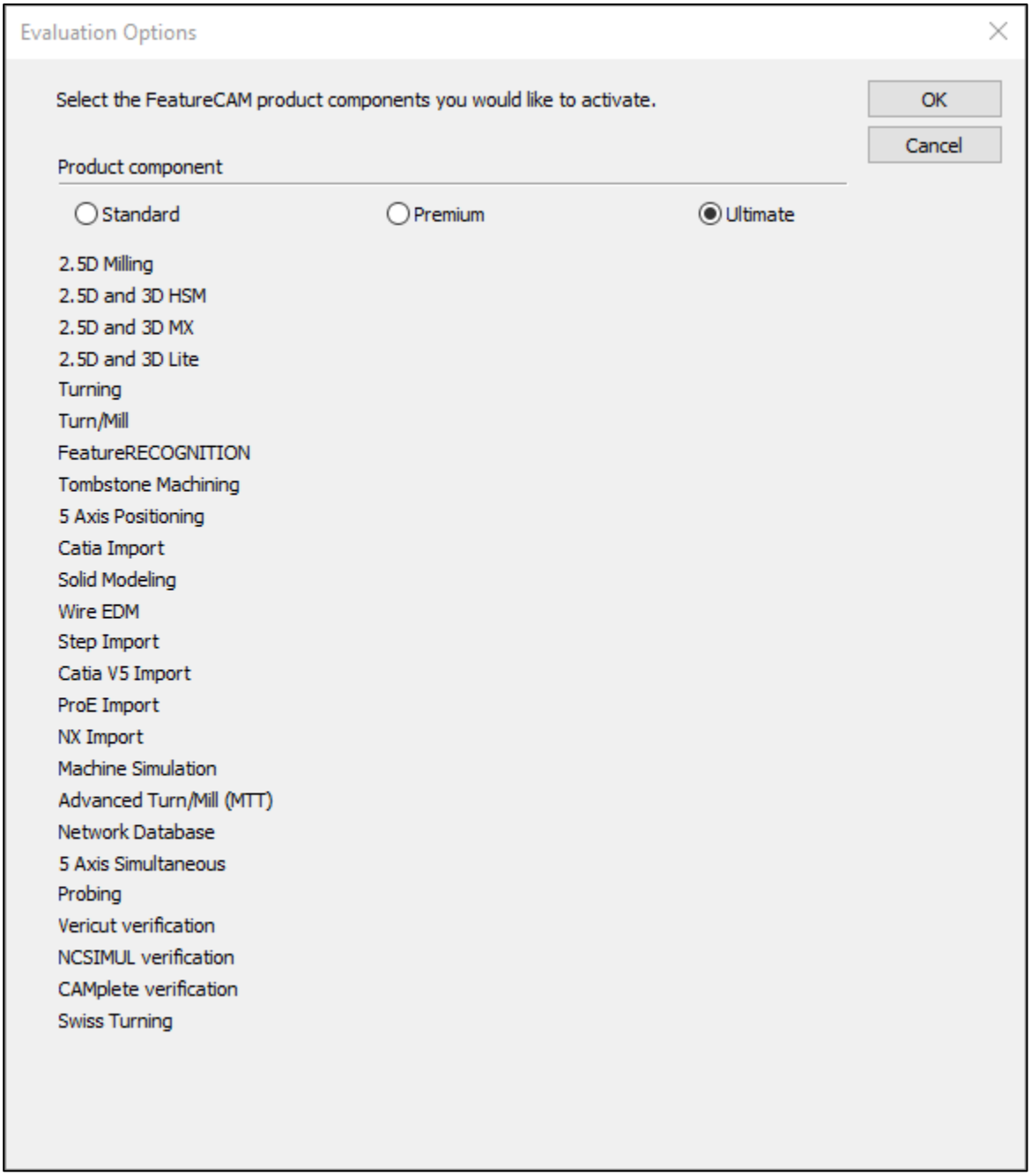
__FeatureCAM__ has now been split into __three levels__. __Standard, Premium__ and __Ultimate__. The next few images show the available options for all modules.

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

Please note that the only options shown are those for __Ultimate as this module is the only one that can be used for Swiss Turning.__

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

© Copyright 2016 Autodesk, Inc. All rights reserved

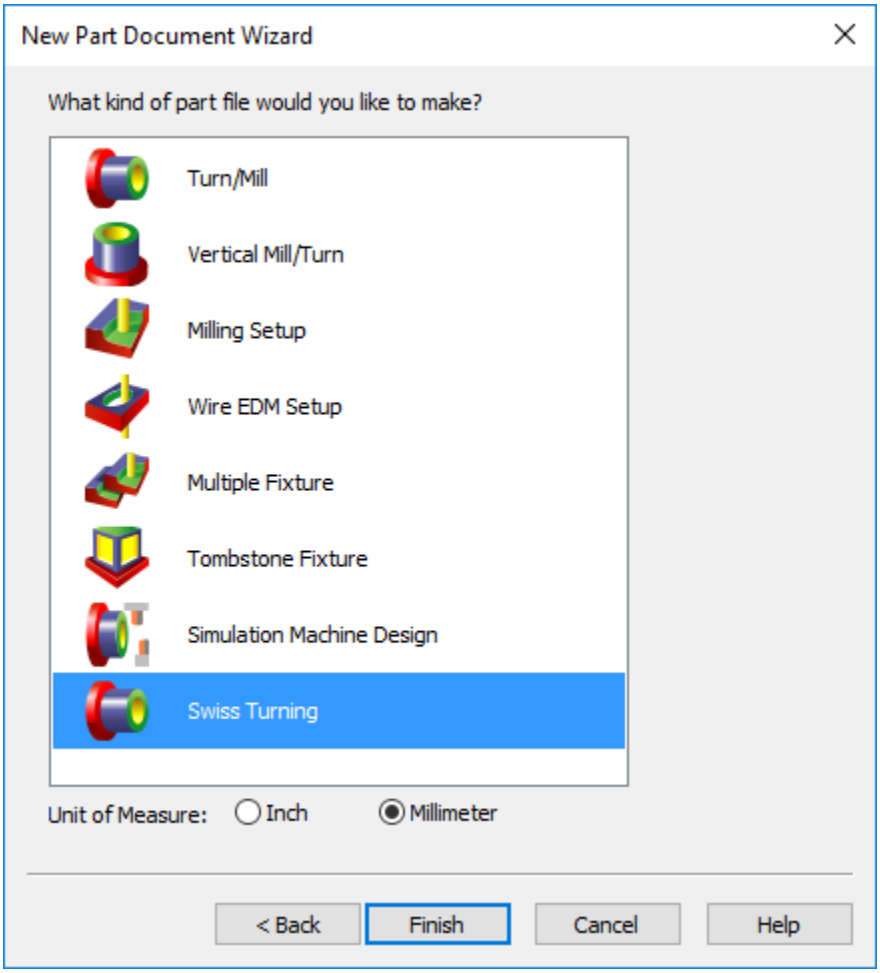
Autodesk support

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.

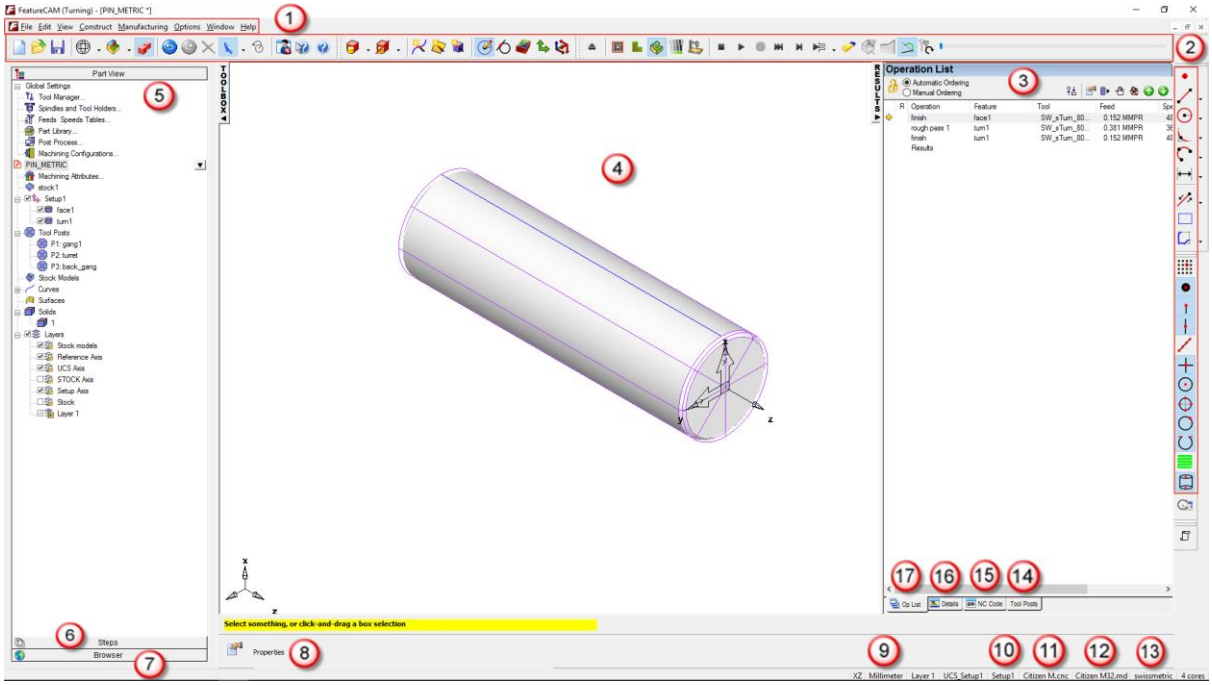


User Interface

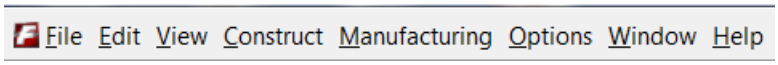
- **FeatureCAM** is a windows-based product and has been a windows based product since its inception unlike other **CAM** products. Many of the functions and keystroke shortcuts that you are familiar with in windows are available inside **FeatureCAM**. Examples are **Ctrl+C** to copy, **Ctrl+V** to paste,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

© Copyright 2016 Autodesk, Inc. All rights reserved

Screen Layout

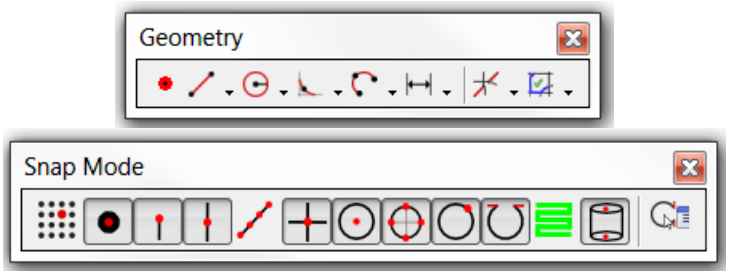


1 **Menu Bar** - Located at the top. Like many other Windows based software, packages **FeatureCAM** includes a series of drop down menus. These menus are unique to **FeatureCAM** and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.



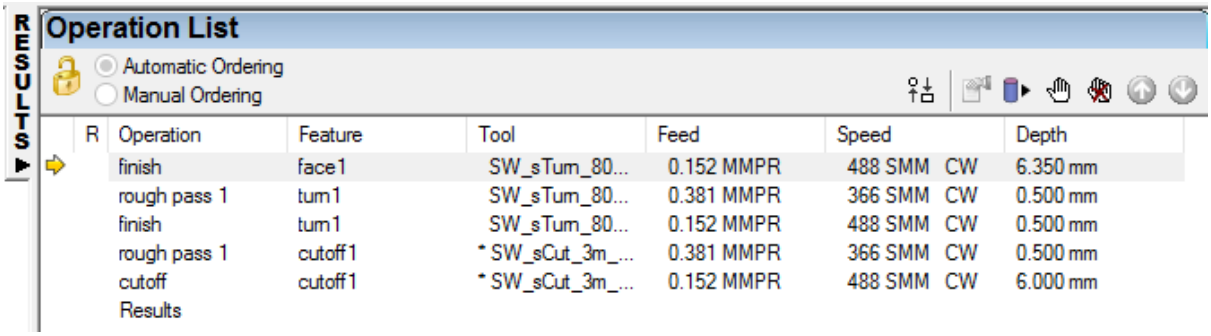
2 **Tool Bars -** usually located at the top but, may be docked in other locations. Can consist of one or more buttons and used to create objects in the Graphics Window such as the Geometry constructors for drawing the shape of your part.

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	face1	SW_sTurn_80...	0.152 MMPR	488 SMM	CW	6.350 mm
	rough pass 1	turn1	SW_sTurn_80...	0.381 MMPR	366 SMM	CW	0.500 mm
	finish	turn1	SW_sTurn_80...	0.152 MMPR	488 SMM	CW	0.500 mm
	rough pass 1	cutoff1	* SW_sCut_3m_...	0.381 MMPR	366 SMM	CW	0.500 mm
	cutoff	cutoff1	* SW_sCut_3m_...	0.152 MMPR	488 SMM	CW	6.000 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**.*

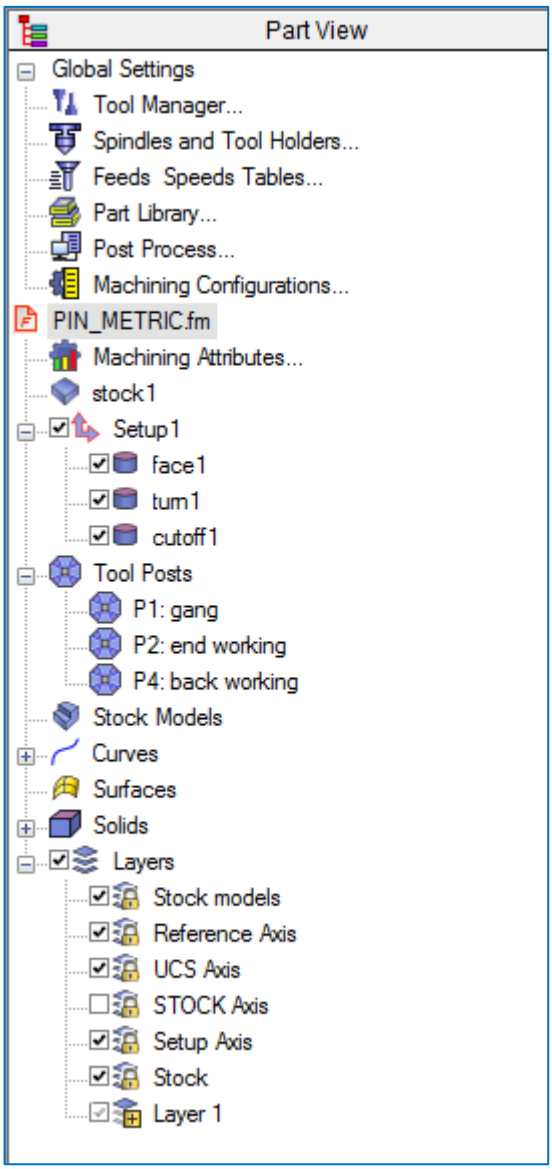
*The screen shots shown above and on the next page show the **Results Window**. This shows you the order in which the machining is carried out. The Feature in which it is associated too, plus the Tools used in that sequence and the corresponding Feeds and Speeds. At the end you have the depth that the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

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.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Steps Menu

Steps to take to create a part in FeatureCAM

Drawing a part in FeatureCAM

1. **Start FeatureCAM Swiss Turning.**
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 Swiss Turning.**
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.*

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

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

© Copyright 2016 Autodesk, Inc. All rights reserved

***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.*

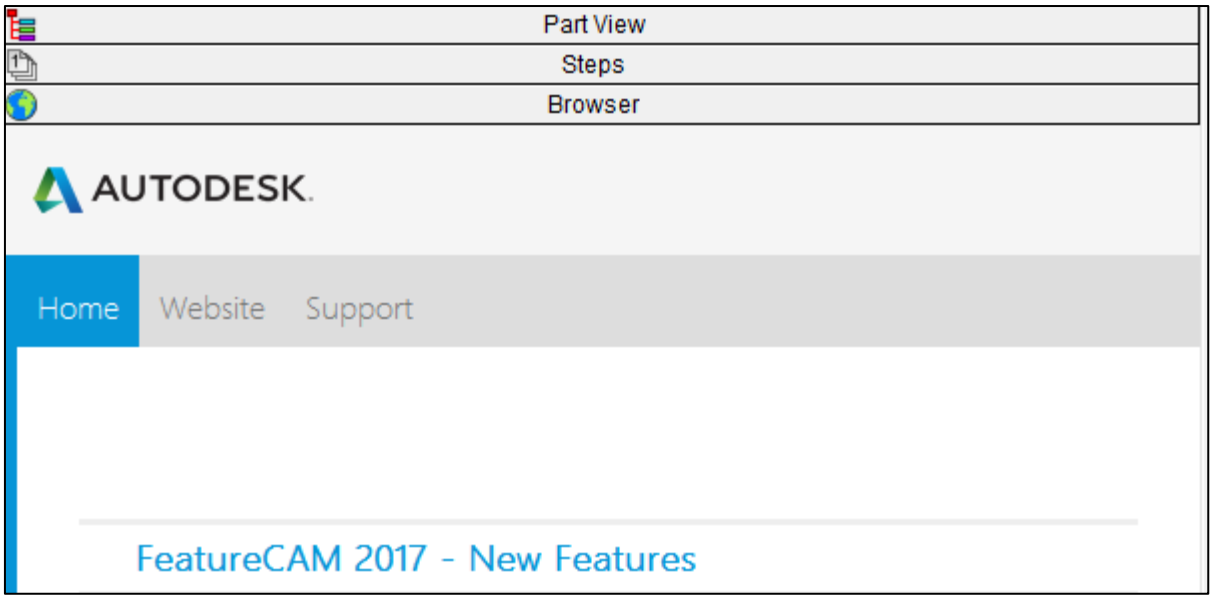
Steps (See menu in PartView)

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

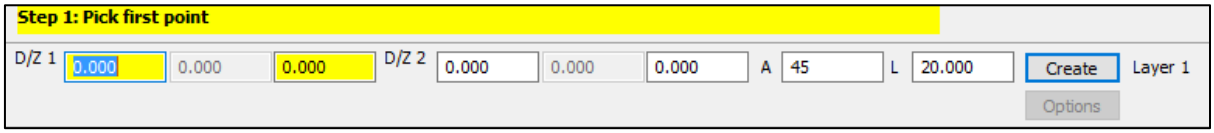
1. Open **FeatureCAM** and select **New File** (Default). Select **Next**. Choose your module of choice and your units. Select **Finish**. Cancel the dimensions wizard. Hide the stock by selecting Stock in **PartView**, right click and select **Hide Stock**. Draw you part using geometry creation techniques. 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

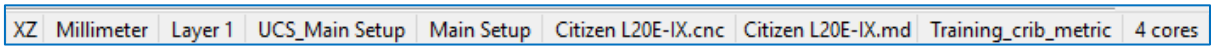


8 **Geometry Bar -** Located near the bottom. This area is constantly used and is where the dimension boxes are used to locate the Geometry on your stock material in the Graphics Window.



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 **Setup1** Current datum Setup.

11 **Citizen L20E-IX.cnc** is the current **Post Processor** being used.

12 **Citizen L20E-IX.md** is the current **Machine simulation** file.

13 **SwissMetric** is the current **Tool Crib**.

© Copyright 2016 Autodesk, Inc. All rights reserved

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.**

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

Manufacturing Details

○ Operation List
◉ Tool List

```
MANUFACTURING TOOL DETAIL SHEET

Part:     PIN_METRIC
Setup:    Setup1 (1 of 1)
Date:     Wednesday, March 30, 2016 14:01:21

Crib:     swissmetric
Summary:
Slot 1:   SW_sCut_3m_RH
Slot 2:   SW_sTurn_80m_RH


Tool Name:          SW_sCut_3m_RH
Tool Slot No.:      1
Tool Comp. No.:     1
Tool Offset No.:    1
Orientation:        SW
Cut Type:           Cutoff
Insert Type:        Groove/Cut
Tool Tip Radius:    0.200 mm
Length:             101.000 mm
Width:              3.000 mm
Depth:              9.000 mm
Type:               CutGrip
Handedness:         RIGHT
Tool Grade:         CARBIDE

Tool Name:          SW_sTurn_80m_RH
Tool Slot No.:      2
Tool Comp. No.:     2
Tool Offset No.:    2
Orientation:        SW
Presentation Angle: 5.0 degrees
Cut Type:           Turning
Insert Type:        80 Diamond
Tool Tip Radius:    0.200 mm
Tool Inscribe Circle: 6.000 mm
Length:             101.000 mm
Handedness:         RIGHT
Tool Grade:         CARBIDE
```


© Copyright 2016 Autodesk, Inc. All rights reserved

- **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**.

NC Code

NC Code: PIN_METRIC_NC.txt

```
( BEGIN CHANNEL 1 PROGRAM )

( Citizen L20E-IX )
( DATE: 3-30-2016 )
( TIME: 14:27:55 )

G99 M97 M87
G113 M9
G99
M6
/M52
#500=3.0 (DIST FROM TOOL ZERO TO FARTHEST SIDE OF INSERT)
#501=3.0 (CUTOFF TOOL WIDTH)
#502=0.5 (FACE ALLOWANCE)
G50 Z[#500-#501-#502]
G0 Z-.05
M3 S1=2500
G600 (WAIT1)
( MAIN SPINDLE, TURNING )
( FINISH FACE FACE1 )
T0200 (SW_sTurn_80m_RH)
G50 S6000
G96 S487
G99 G18 G0 Z0. T02
X50.0
X18.7
G1 X-0.4 F0.152
X4.125 Z-2.263

( MAIN SPINDLE, TURNING )
( ROUGH TURN TURN1 )
G50 S6000
G96 S365
G0 X18.3 Z-2.263 T02
G0 X18.7 Z-3.212
X12.2
G1 Z40.2 F0.381
X12.7
X13.407 Z39.846
G0 Z-3.212
G1 X11.024
Z-0.012
X12.024 Z0.488
G2 X12.2 Z0.7 R0.3
G1 X12.907 Z0.346
G0 Z-3.0
```

- Please note you have to run the **Simulation** before the **NC Code** is made available for output.

© Copyright 2016 Autodesk, Inc. All rights reserved

NC Code

NC Code: PIN_METRIC_NC.txt

```
( BEGIN CHANNEL 1 PROGRAM )

( Citizen L20E-IX )
( DATE: 3-30-2016 )
( TIME: 14:27:55 )

G99 M97 M87
G113 M9
G99
M6
/M52
#500=3.0 (DIST FROM TOOL ZERO TO FARTHEST SIDE OF INSERT)
#501=3.0 (CUTOFF TOOL WIDTH)
#502=0.5 (FACE ALLOWANCE)
G50 Z[#500-#501-#502]
G0 Z-.05
M3 S1=2500
G600 (WAIT1)
( MAIN SPINDLE, TURNING )
( FINISH FACE FACE1 )
T0200 (SW_sTurn_80m_RH)
G50 S6000
G96 S487
G99 G18 G0 Z0. T02
X50.0
X18.7
G1 X-0.4 F0.152
X4.125 Z-2.263

( MAIN SPINDLE, TURNING )
( ROUGH TURN TURN1 )
G50 S6000
G96 S365
G0 X18.3 Z-2.263 T02
G0 X18.7 Z-3.212
X12.2
G1 Z40.2 F0.381
X12.7
X13.407 Z39.846
G0 Z-3.212
G1 X11.024
Z-0.012
X12.024 Z0.488
G2 X12.2 Z0.7 R0.3
G1 X12.907 Z0.346
G0 Z-3.0
```


© 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:

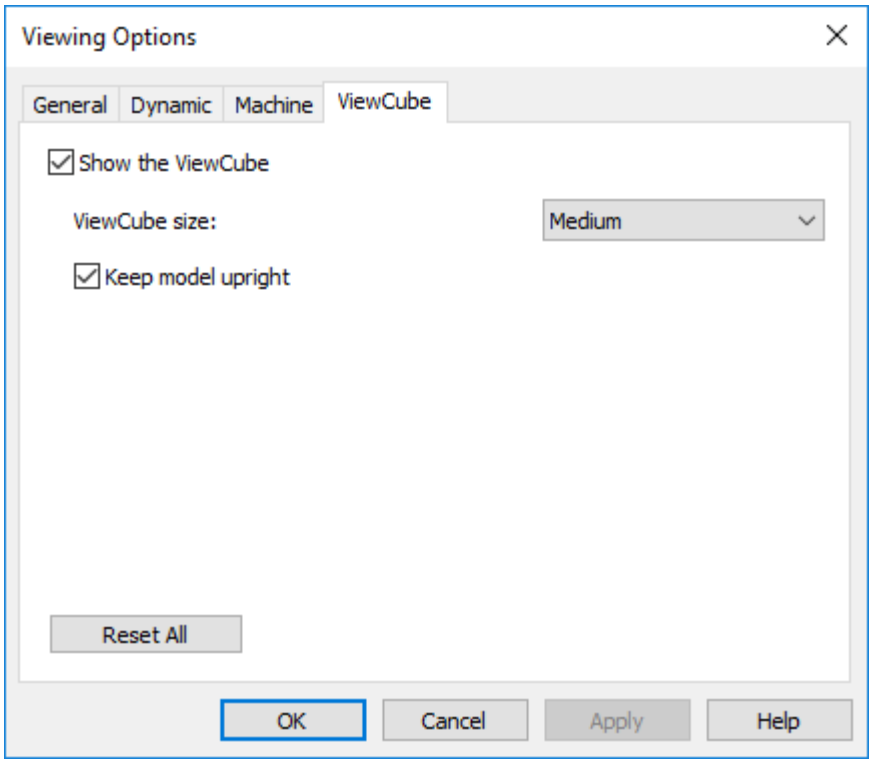
Click:

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.

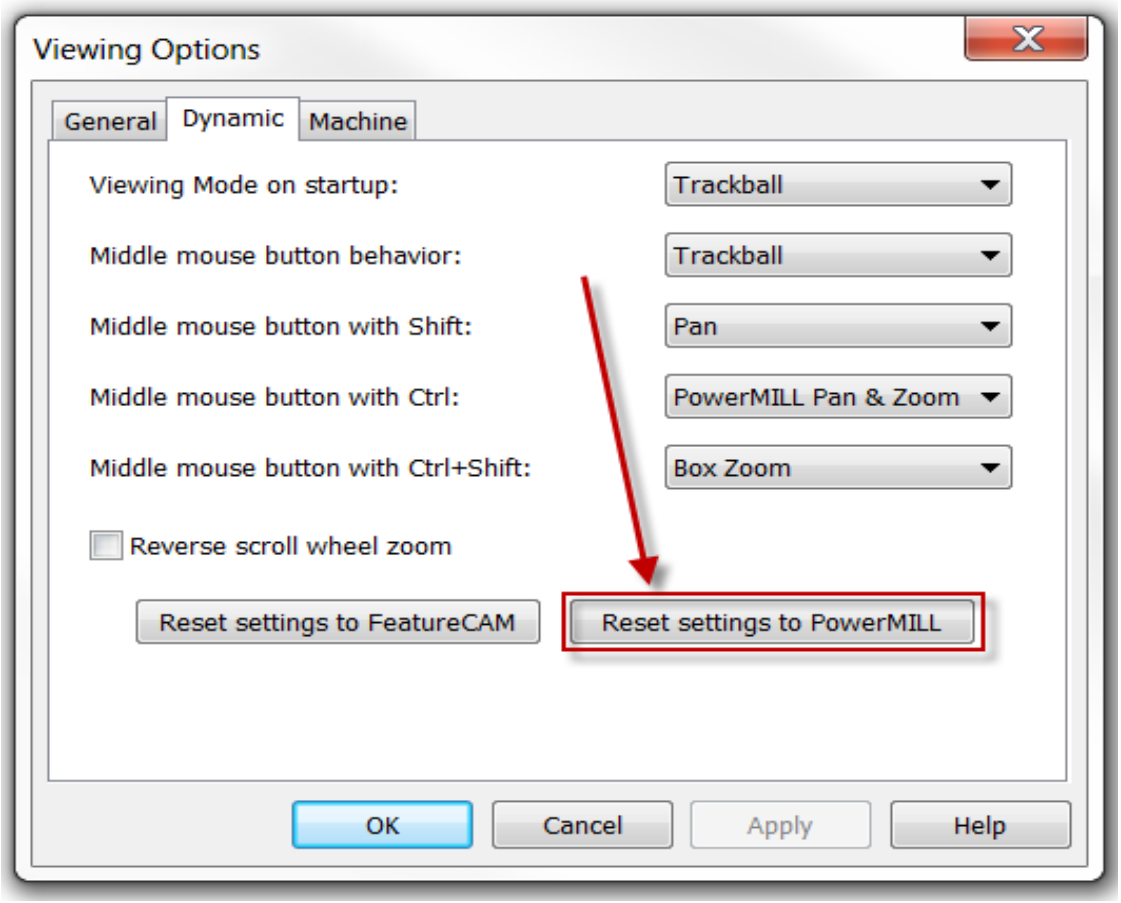
Right mouse button Special Menus & FeatureCAM Options.

© 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

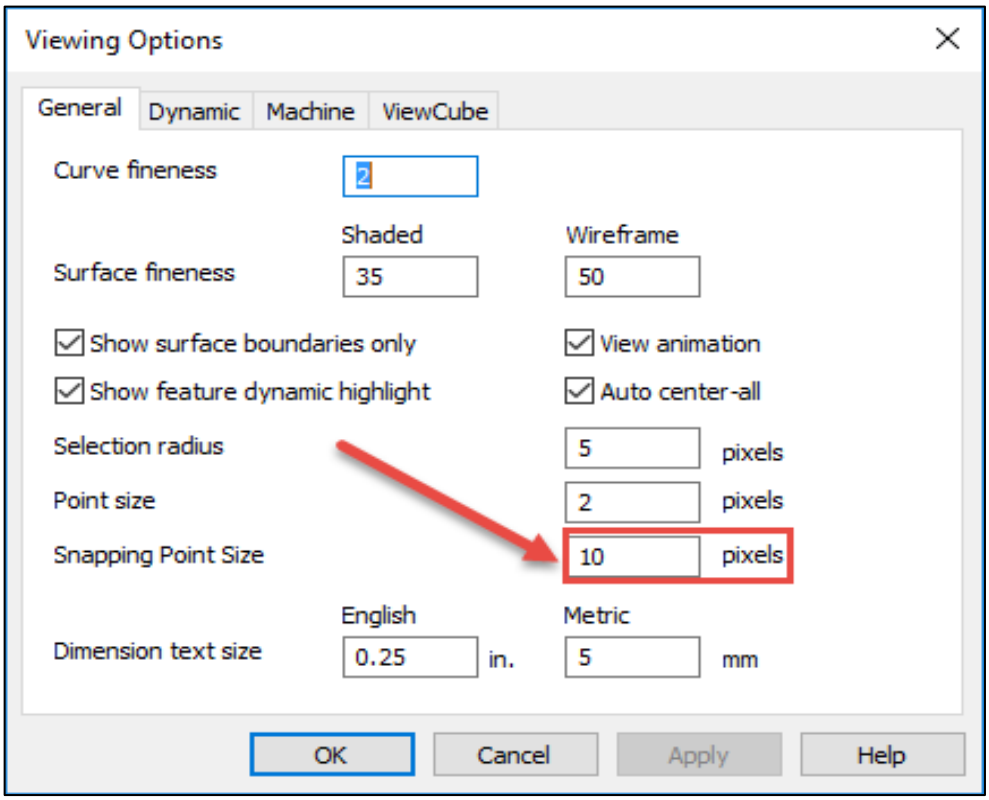
Reset to PowerMILL Settings

- 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.**

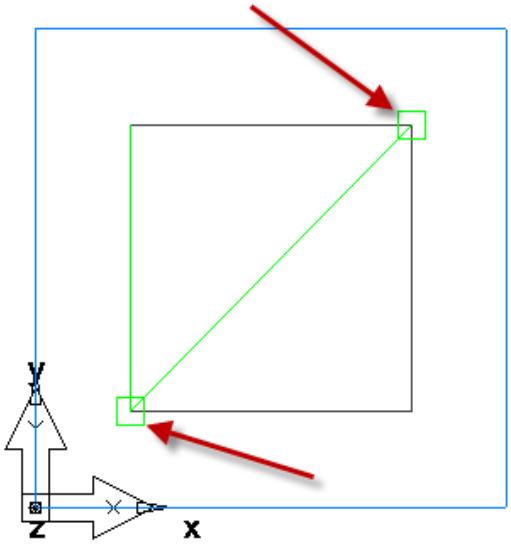
Viewing Options General

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

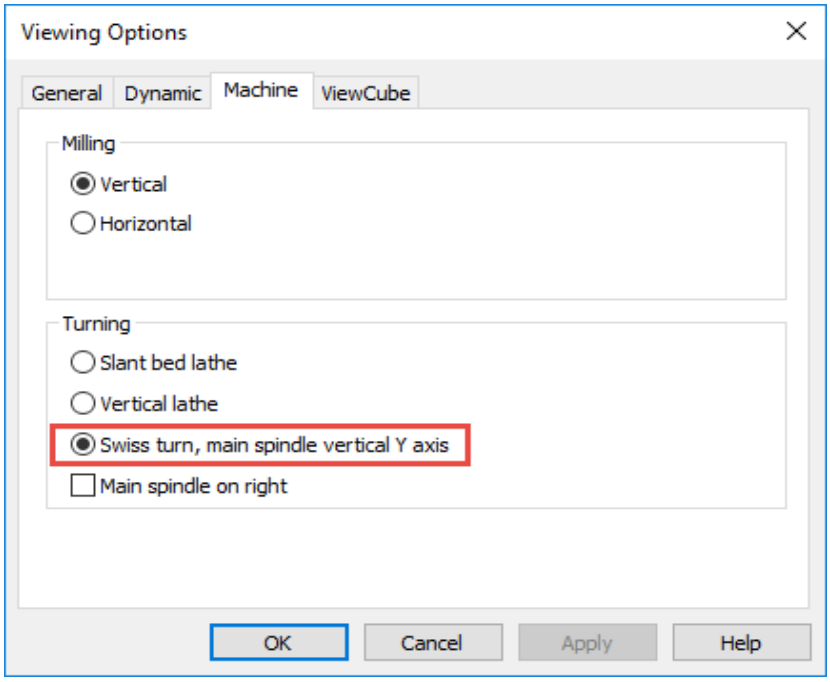
An example of this is shown below



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.

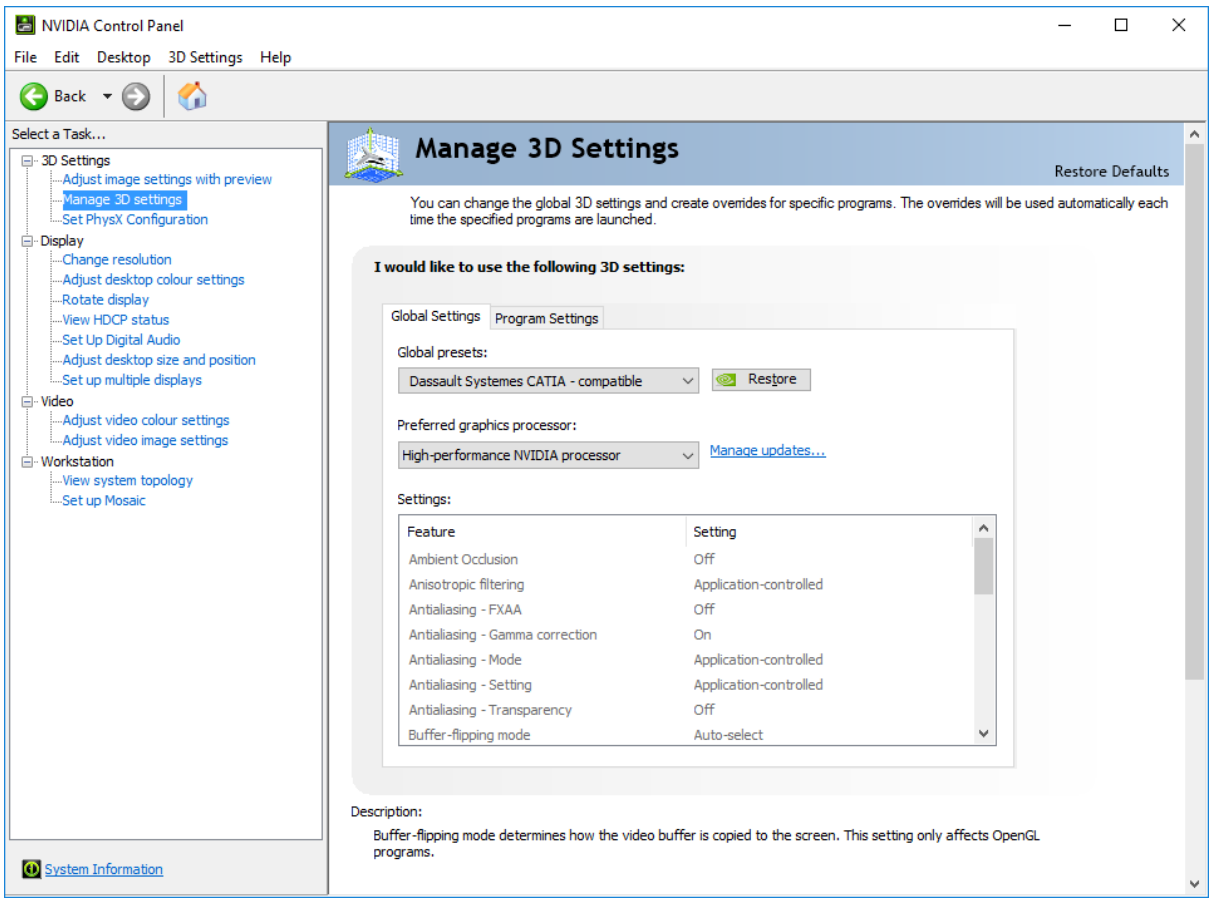
- **Swiss turn, main spindle vertical Y axis** — Use the principal views for Swiss turn machines. For example, the top view shows the YZ plane with the Z axis horizontal on the screen.
- **Main spindle on right** — Select this option to flip the direction of the Z axis in the principal views when using machines with the main spindle on the right.



© 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.

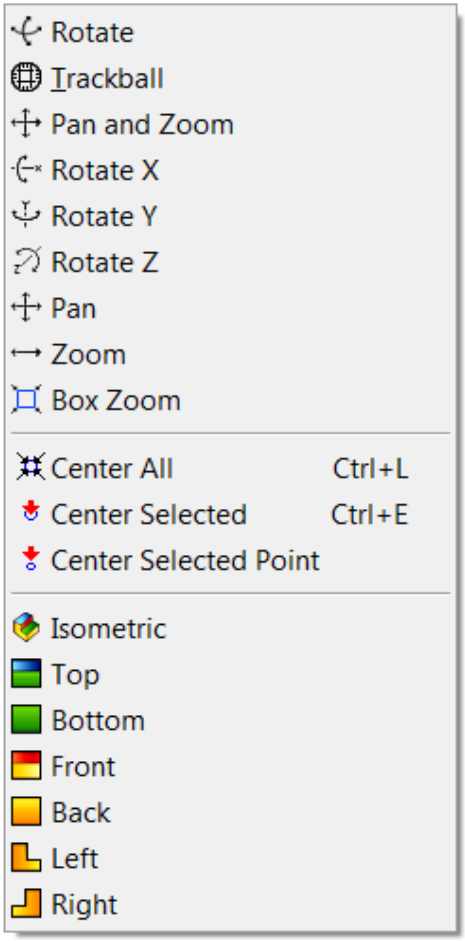


*All of the viewing options are available via **Right mouse button**. Click in the Main Graphics Area*

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+**clic**k 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.

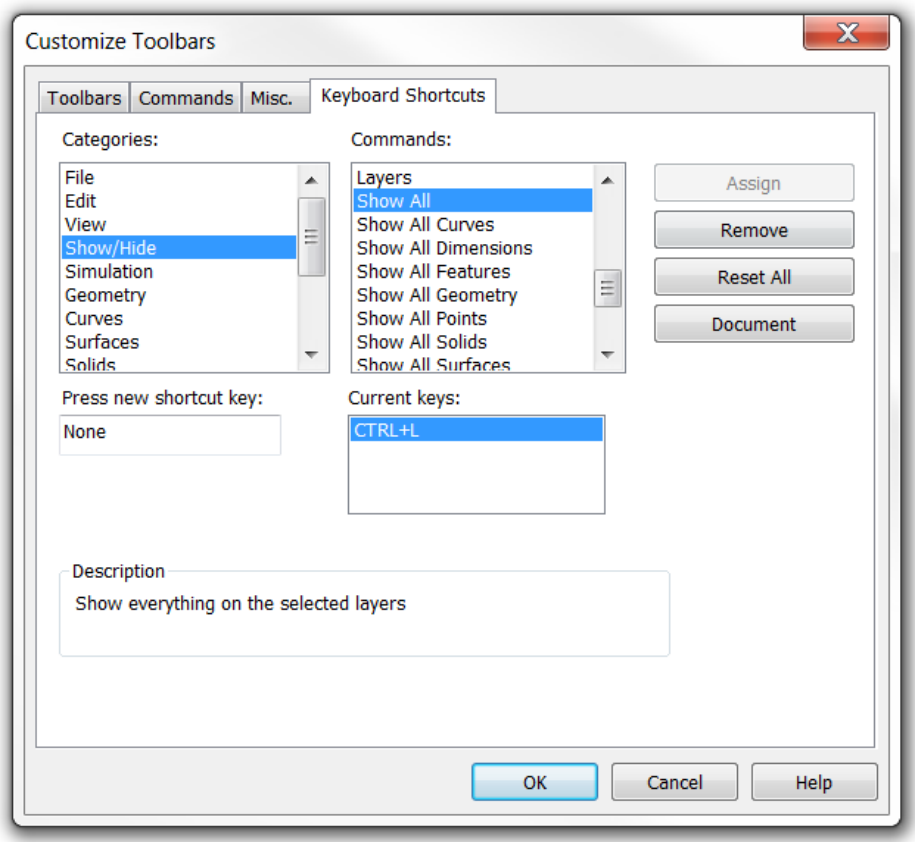
© Copyright 2016 Autodesk, Inc. All rights reserved

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.

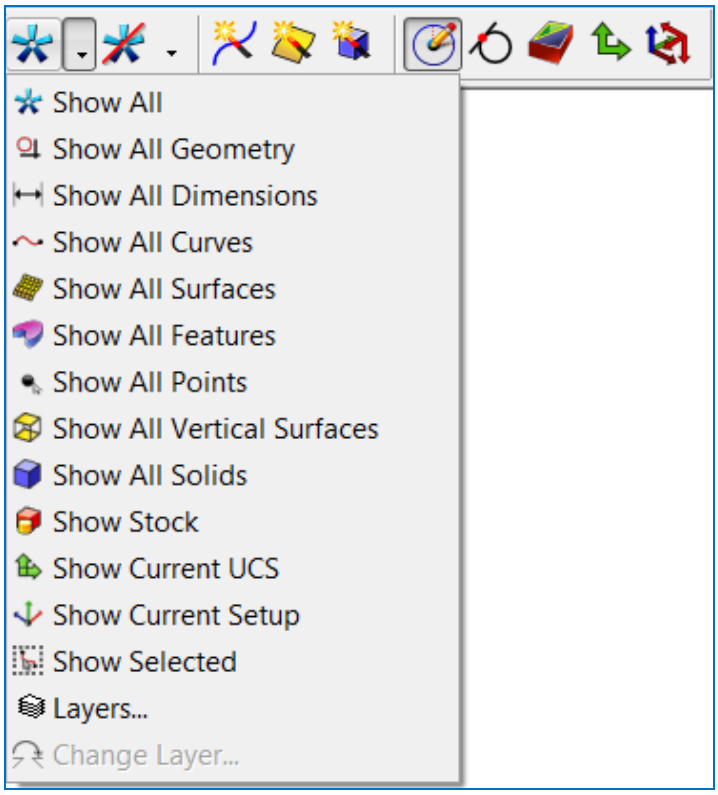
- **Remove** — Click this button to remove the shortcut assigned to the selected command.
- **Reset All** — Click this button to reset all shortcuts to the **FeatureCAM** defaults.

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:

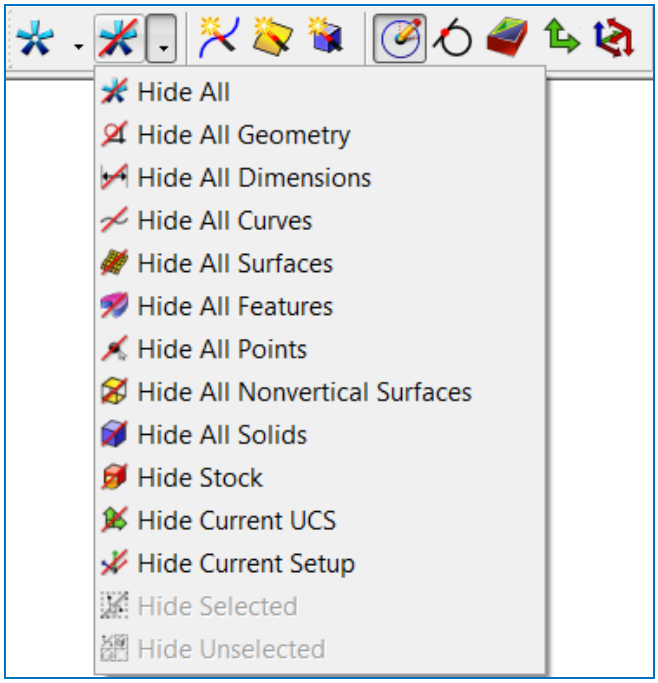


© Copyright 2016 Autodesk, Inc. All rights reserved

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

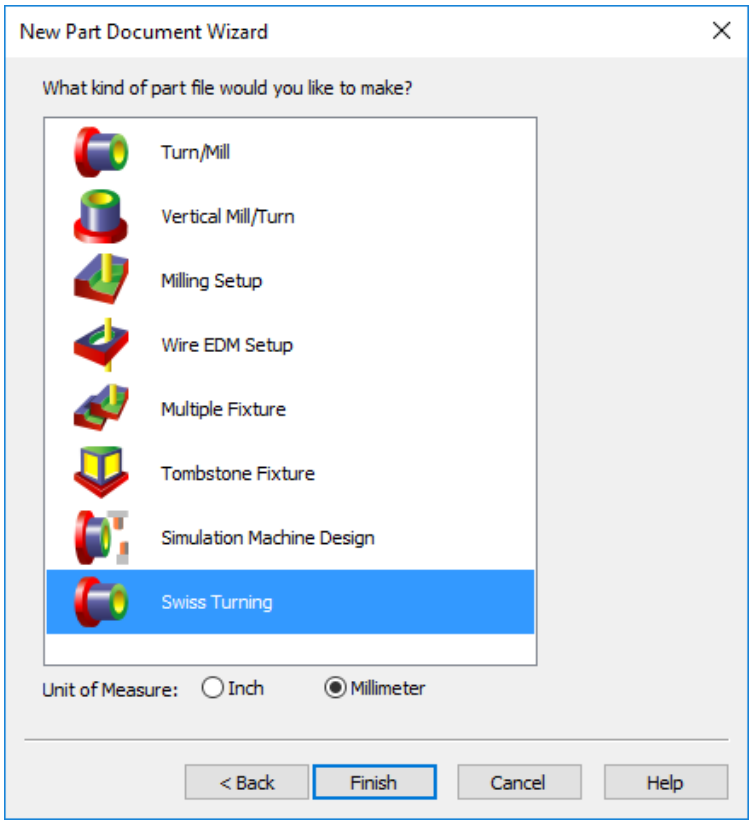
- Hide **All** — Use this menu option to hide all geometry, curves, Features. The stock and axis icon remain visible. A common procedure is to Hide **All**, then **Show** only one type of entity, for example features.
- Hide **All Geometry** — Use this menu option to hide all geometry. Other entities 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.

© Copyright 2016 Autodesk, Inc. All rights reserved

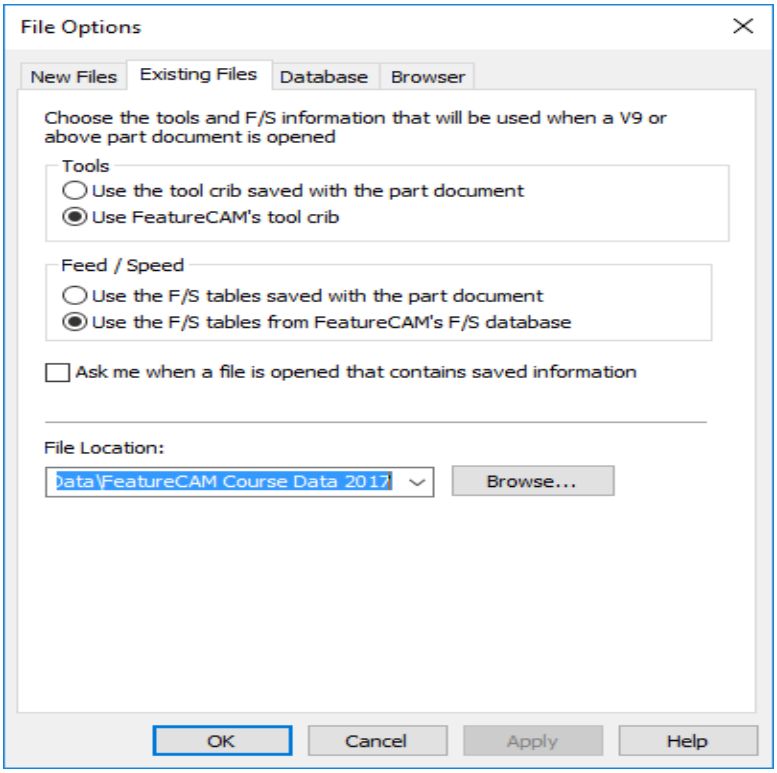
Setting file location options

1 Open **FeatureCAM** by double clicking the **FeatureCAM** shortcut

2 Select **New File**. **Swiss Turning** Setup. Select **Finish**.



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



© Copyright 2016 Autodesk, Inc. All rights reserved

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 = SwissMetric**.
8. **Citizen L20E-IX.cnc** is the current **Post Processor** being used.
9. **Citizen L20E-IX.md** is the current **Machine simulation** file.

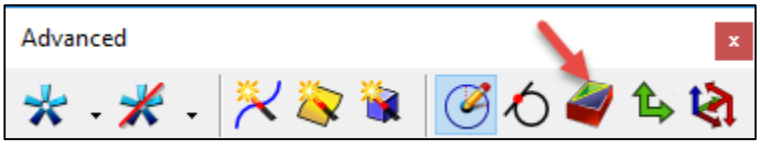
What is a Feature?

- 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.
- **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.

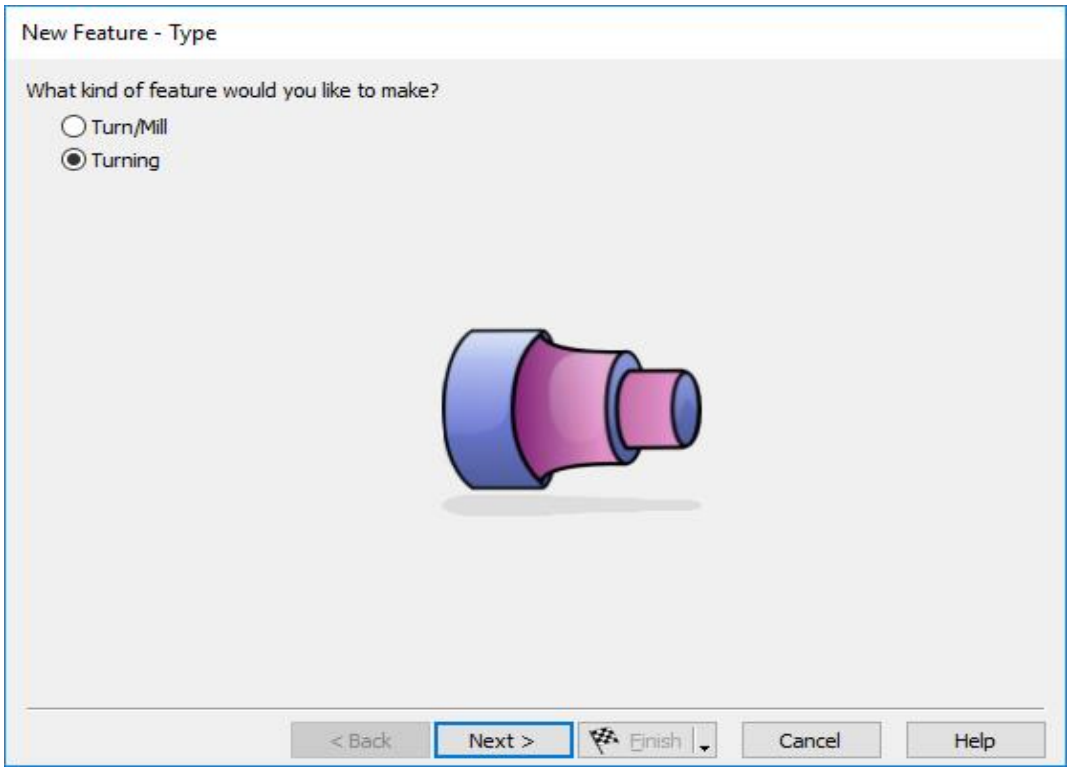
*There are four different ways to activate the **New Features** menu.*

- 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.**

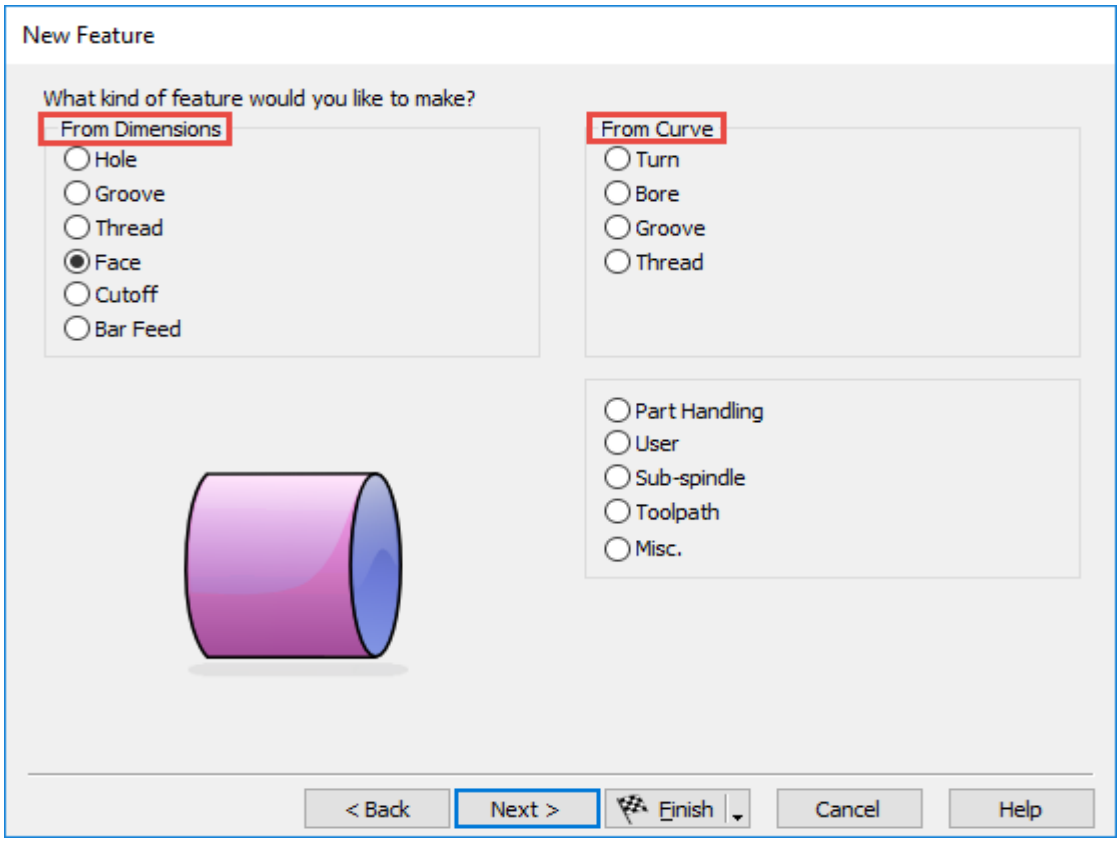


© Copyright 2016 Autodesk, Inc. All rights reserved



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**.

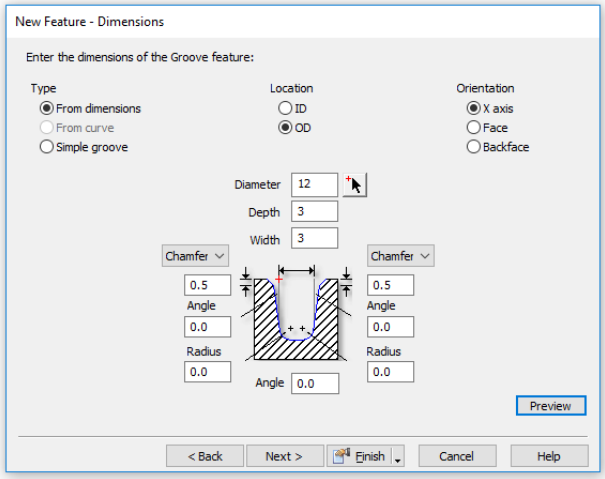


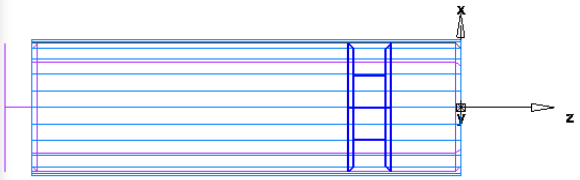
© Copyright 2016 Autodesk, Inc. All rights reserved

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



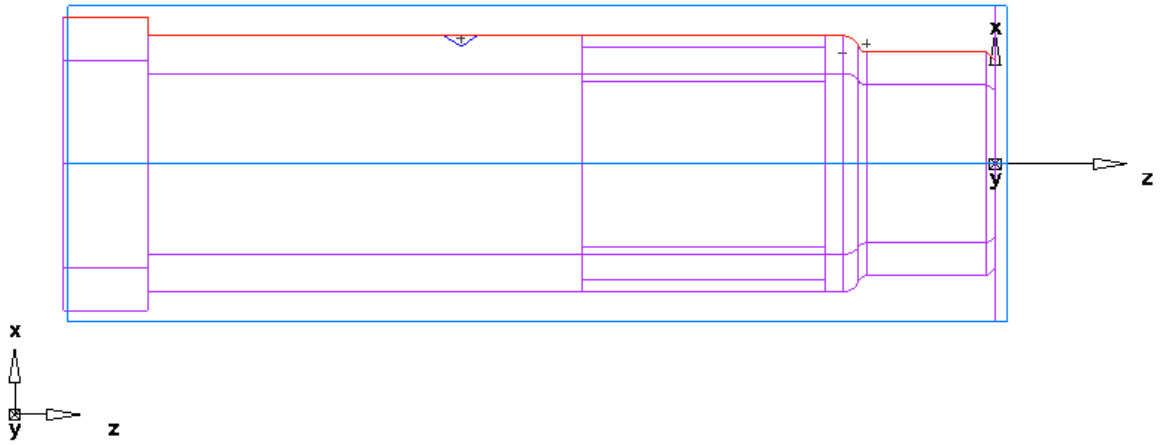


From Curve

*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

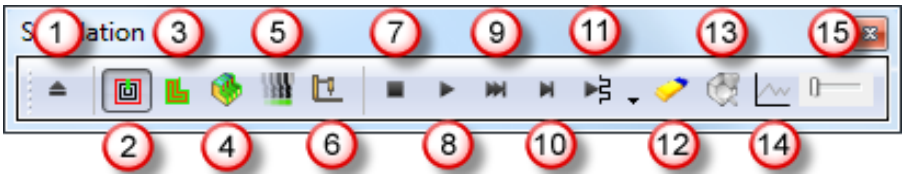
© Copyright 2016 Autodesk, Inc. All rights reserved



*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.

(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.

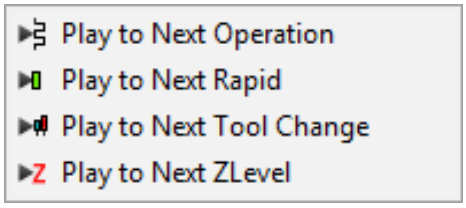
(8) **Play:** Starts the selected Simulation (Centreline, 2D, 3D or Rapid Cut), or restarts a paused Simulation.

© Copyright 2016 Autodesk, Inc. All rights reserved

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.

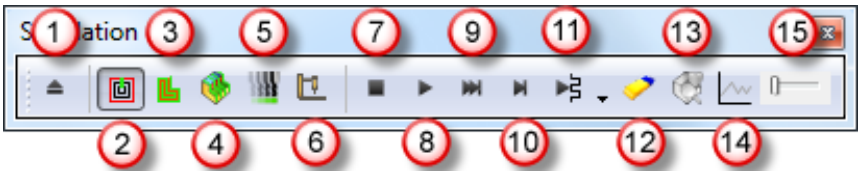
Next Tool Change simulates until the next tool change.

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.

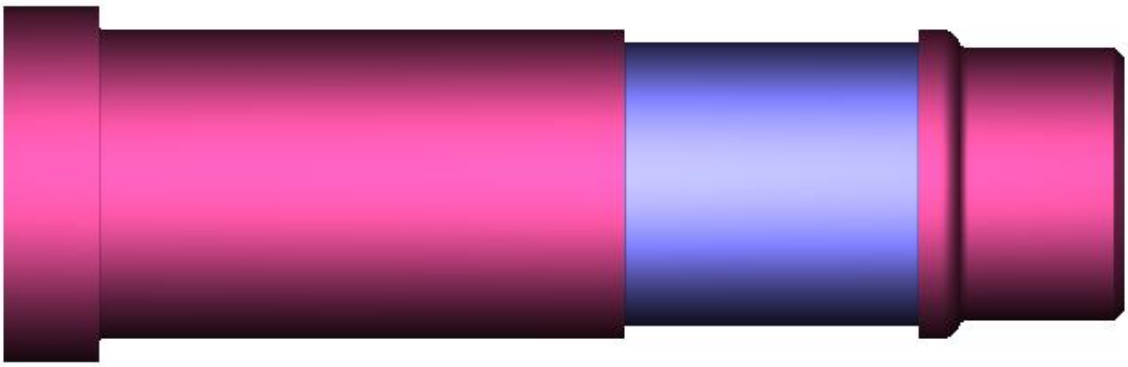
13 **Region of interest** limits the portion of the part that is simulated.

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



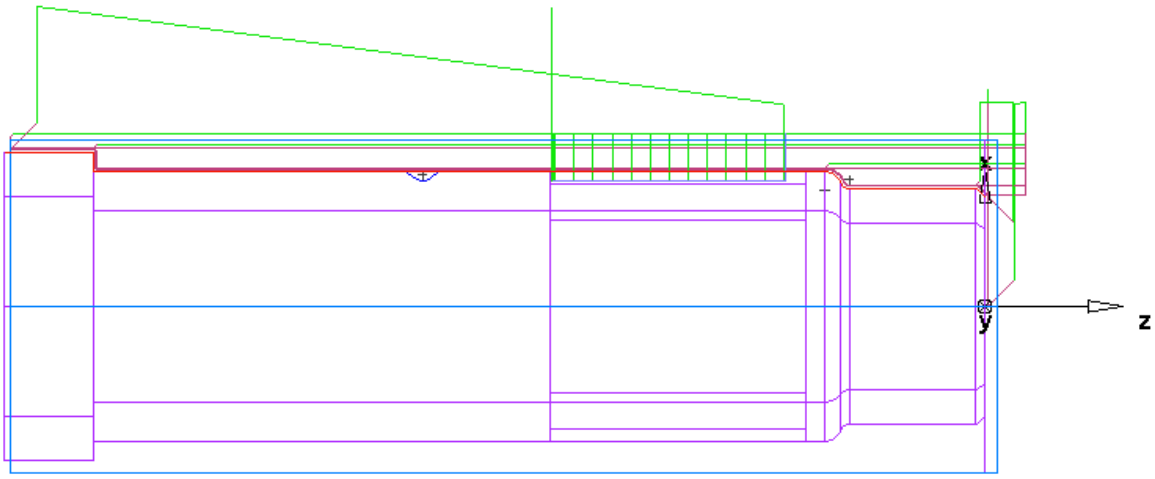
15 **Sim Speed:** To adjust the speed of a **Centreline**, **2D** or **3D Simulation**, use the **Slider** on the right-hand side of the controls.

*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.

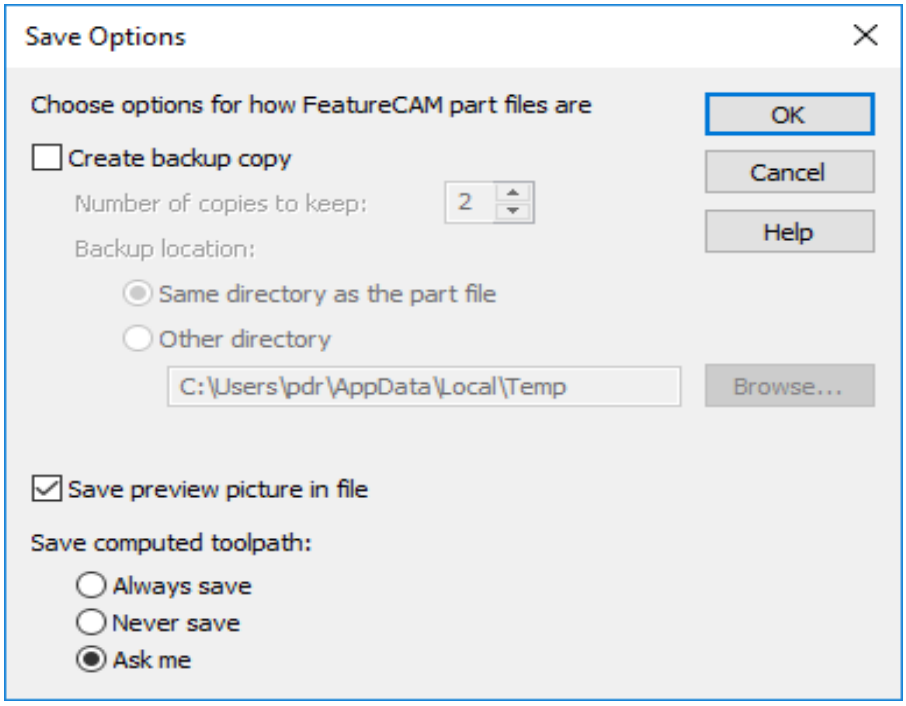
Saving your work

- **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.

© Copyright 2016 Autodesk, Inc. All rights reserved

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.

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.*

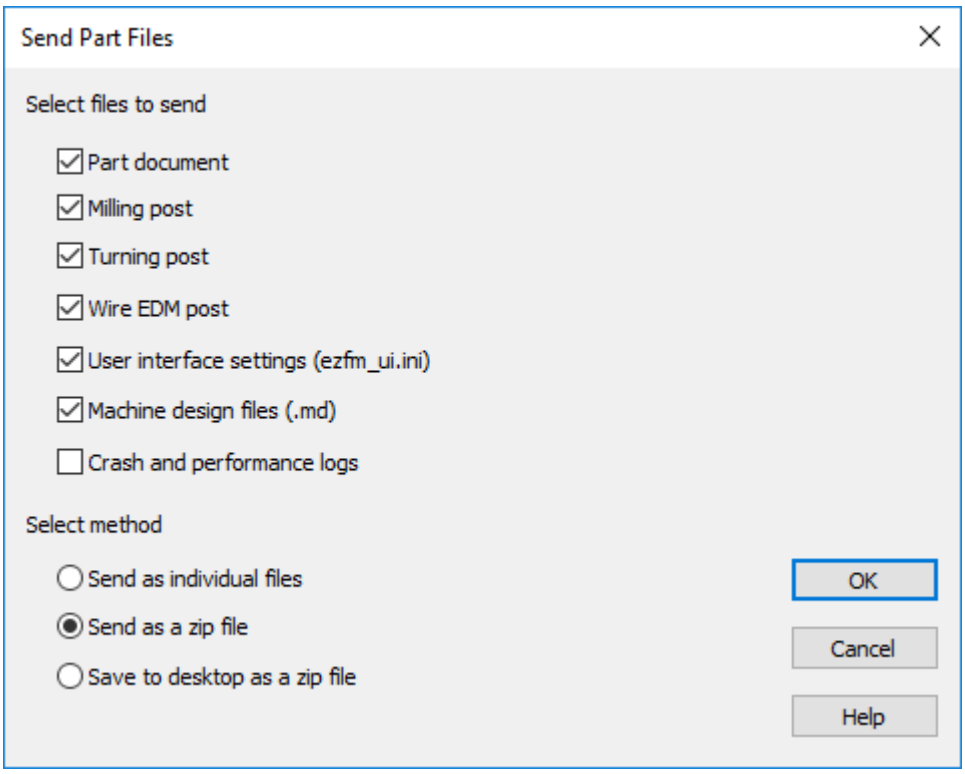
© Copyright 2016 Autodesk, Inc. All rights reserved

***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.



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

© Copyright 2016 Autodesk, Inc. All rights reserved

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.

Part Documentation

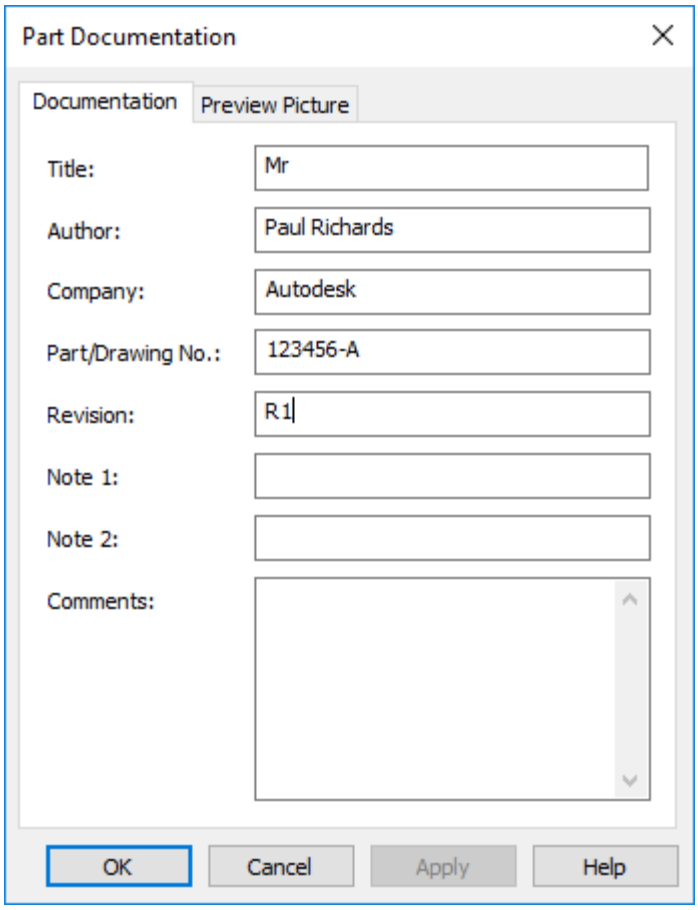
The Part Documentation dialog enables you to add comments to the printed documentation and set a permanent preview picture for the part.

- 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.

© Copyright 2016 Autodesk, Inc. All rights reserved



*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** 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.*

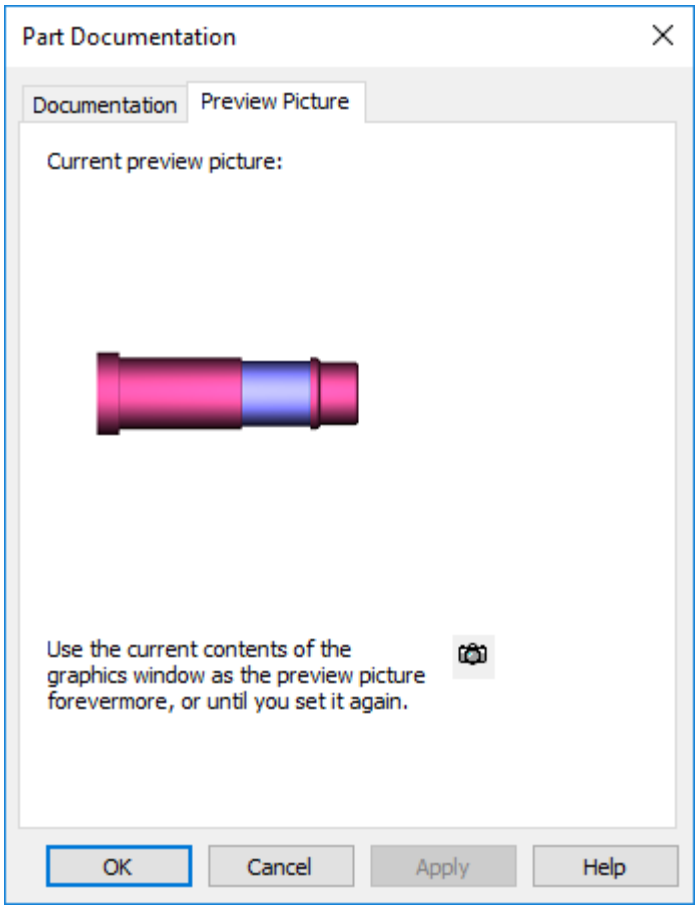
This image is normally updated each time you save the part. If you want to store a permanent image with the part:

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

11 Click Update preview picture

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

© Copyright 2016 Autodesk, Inc. All rights reserved



12 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**.

*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.

© Copyright 2016 Autodesk, Inc. All rights reserved

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:

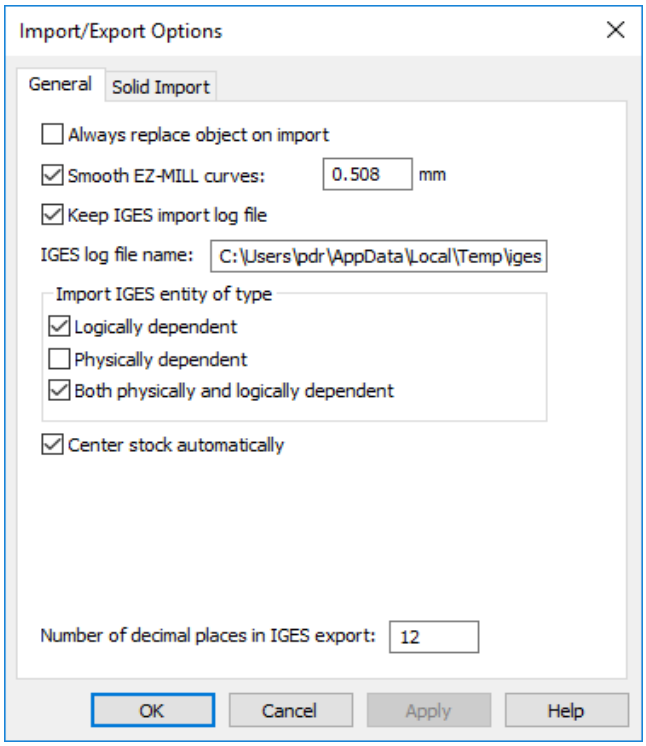
General

Solid Import

Digitized Data

General tab

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



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.

© Copyright 2016 Autodesk, Inc. All rights reserved

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.

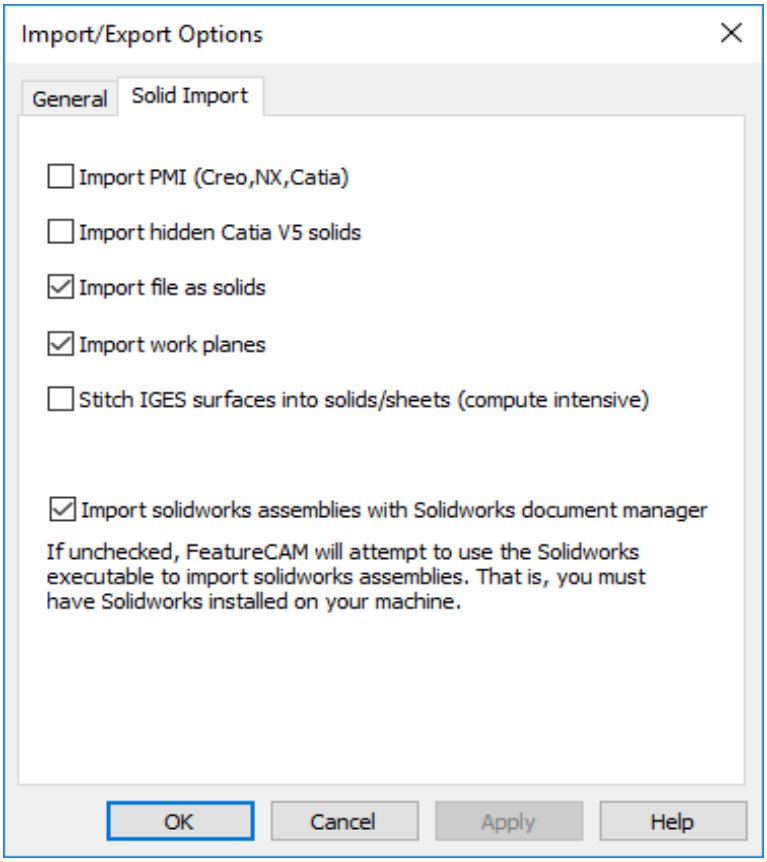
Both physically and logically dependent: - meets both logically and physically dependent criteria (is referenced by at least two other entities**: -** the parent for the logical link cannot be the parent for the physical as well).

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:

- Re-trim 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.

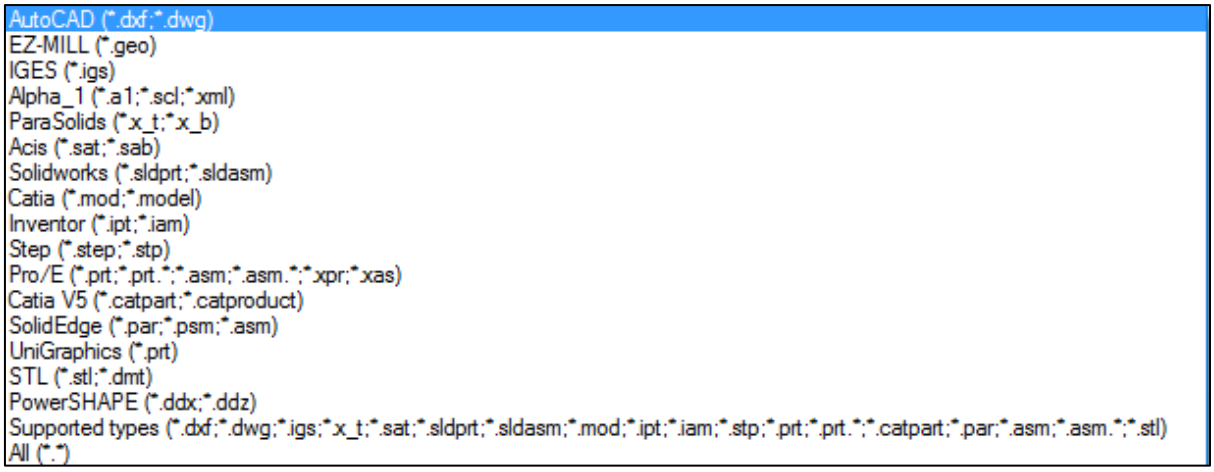
Import file as solids: - Select this option to import files as solid models. If this option is deselected, the models are imported as surface models. We recommend that you import models as solids. If this option is selected and the solid fails to import properly, you are asked if you would like to attempt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



© Copyright 2016 Autodesk, Inc. All rights reserved

*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

Swiss Turning Basics 2
Introduction 2
Basic Swiss Turning Exercise 2
Creating a New Swiss-Turn Part Document 4
Turning Input Modes 10
Geometry Creation Techniques 10
Curve Creation 13
Creating a Face Feature 14
Creating a Turn Feature 19
Results 23
2D Simulation with Rapid moves 23
Machine Simulation 23
Part View All Operations 24
NC Code output example 25
Part Documentation 25
Tool Mapping 30
Tool Numbering 33
Show Machine in Tool Mapping 33

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss Turning Basics

Introduction

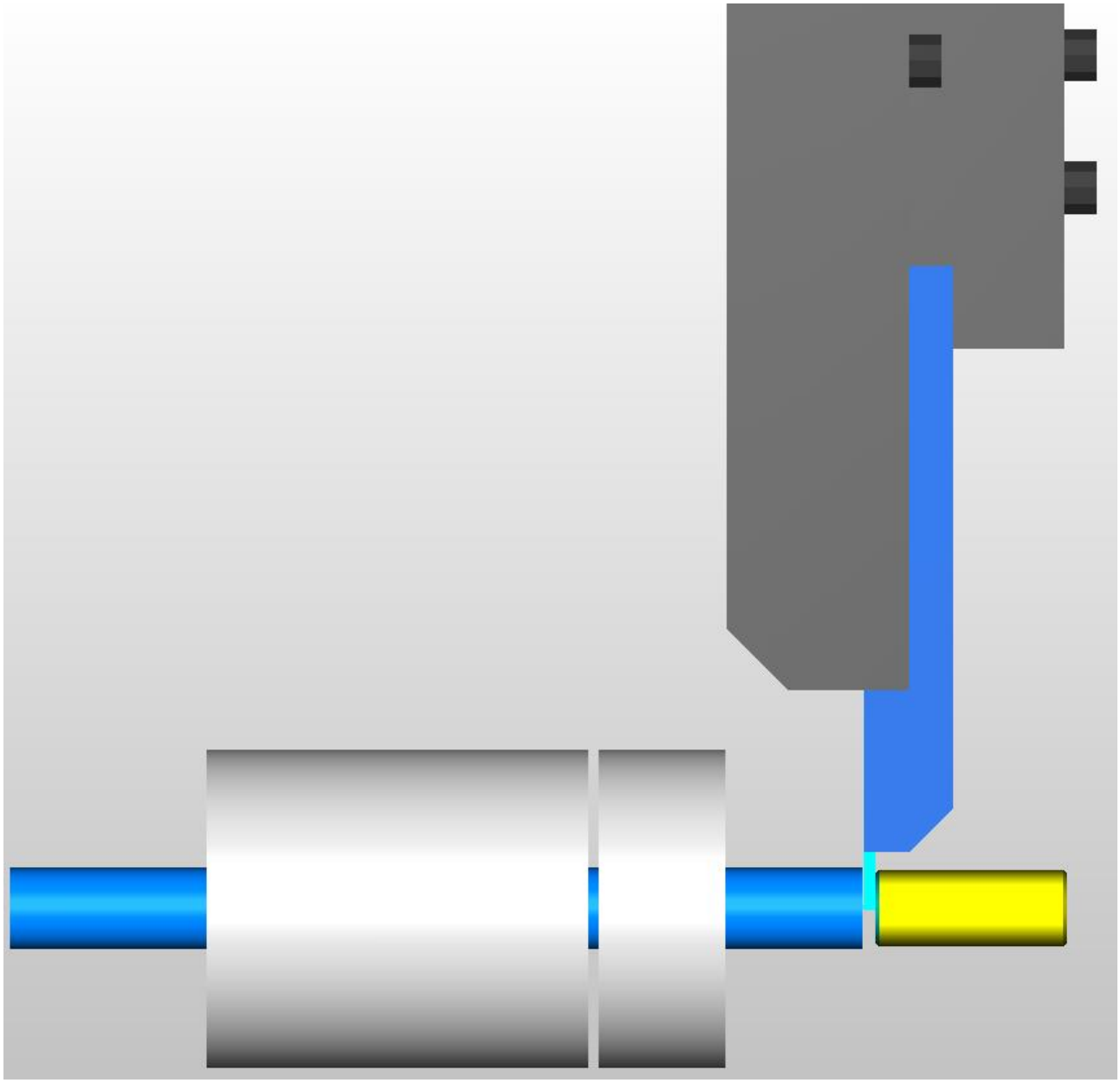
In the **Swiss Turning** module the user will learn basic 2-axis (X and Z) turning, along with the basic user interface. At the end of the module the user will be able to generate a program for a simple 2 axis part.

Basic Swiss Turning Exercise

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

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 **Coursewor**k.
5. **Tool Crib** = **SwissMetric**.
6. Post processor used = **Citizen L20E-IX.cnc**

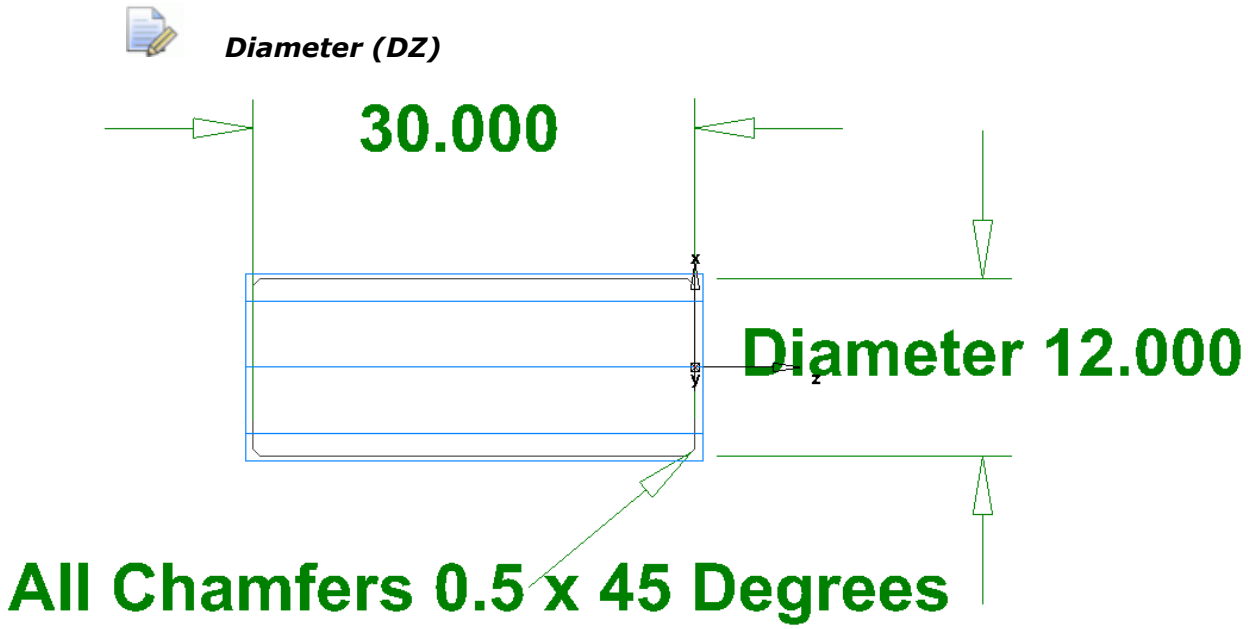
© Copyright 2016 Autodesk, Inc. All rights reserved



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

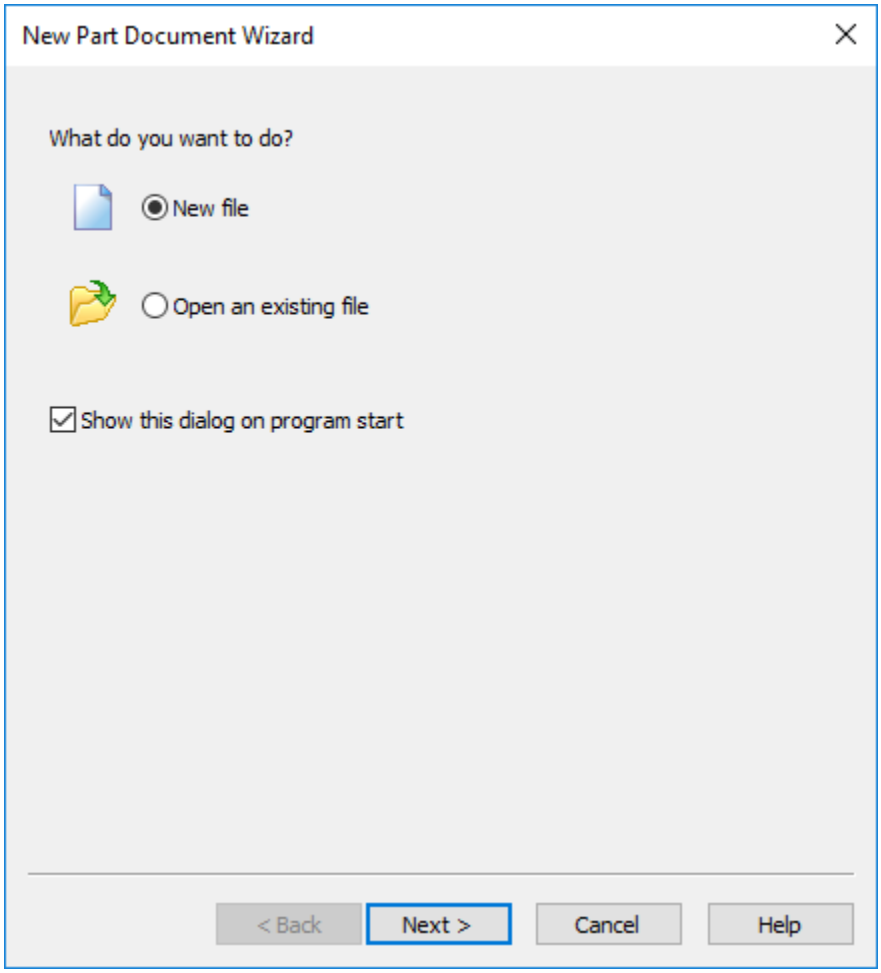
*For this exercise the **Face** and **Cut-off** do not have to be defined by creating geometry, these features will be created automatically later by creating features from dimensions.*

© Copyright 2016 Autodesk, Inc. All rights reserved



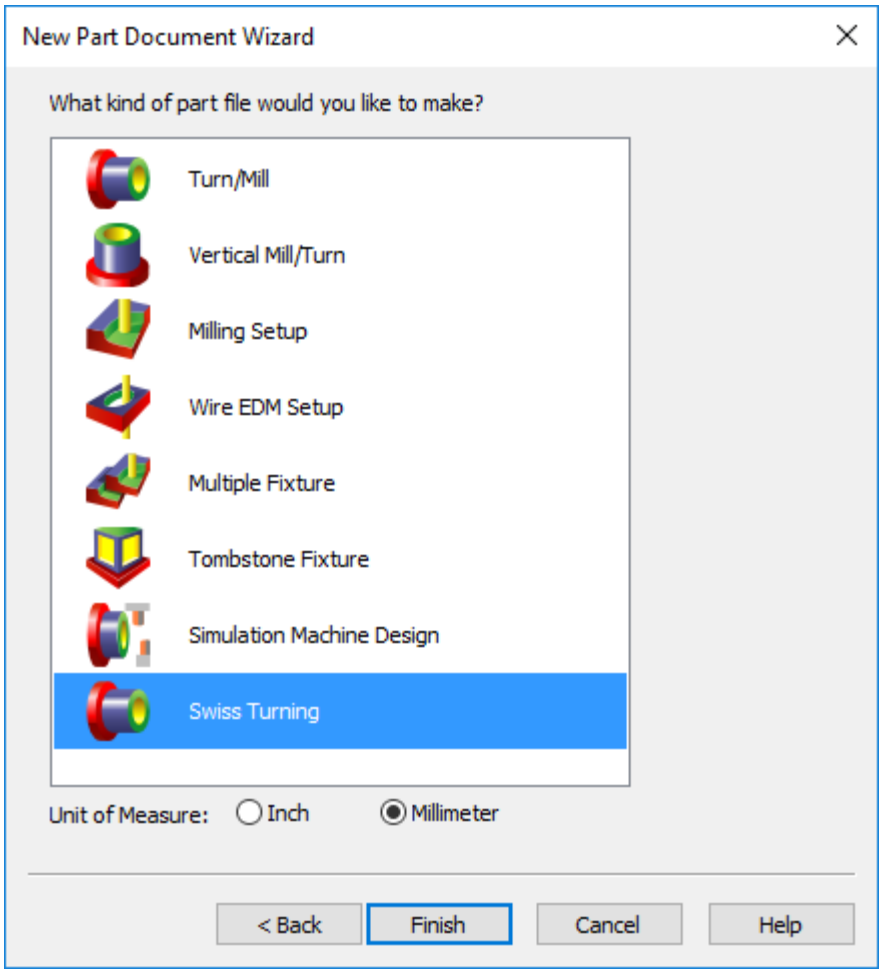
Creating a New Swiss-Turn Part Document

7 Create a **New file**.



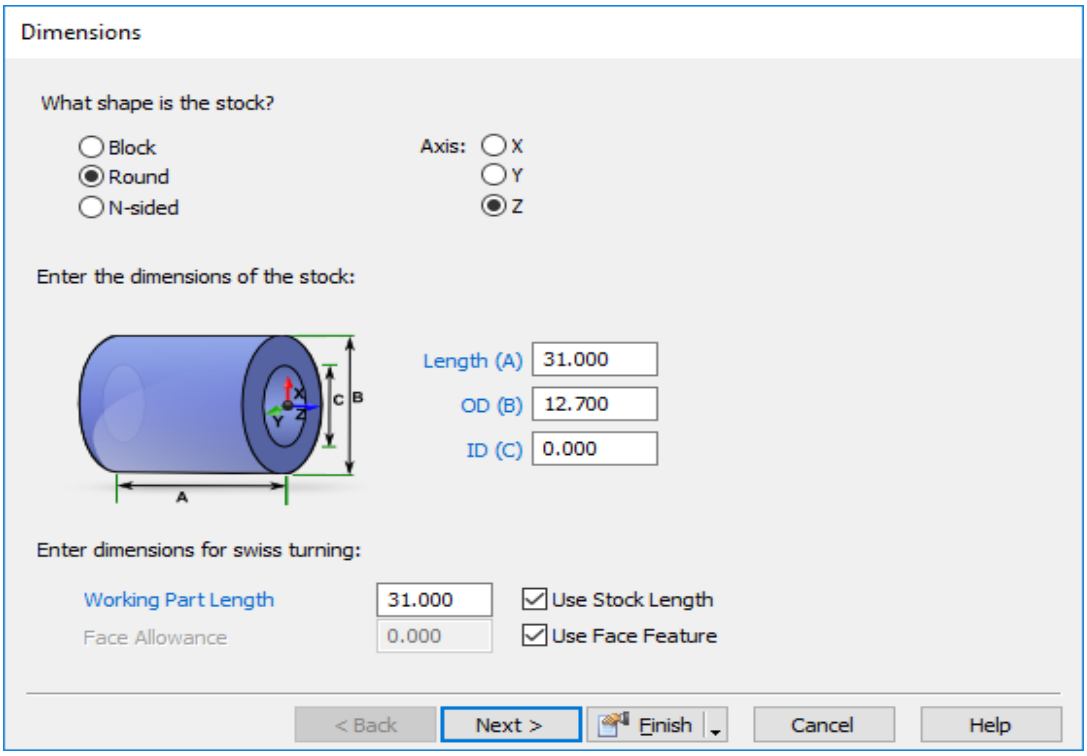
8 Then select the **Swiss Turning** module.

© Copyright 2016 Autodesk, Inc. All rights reserved



*This will create a **New Metric** document. Select **Finish.***

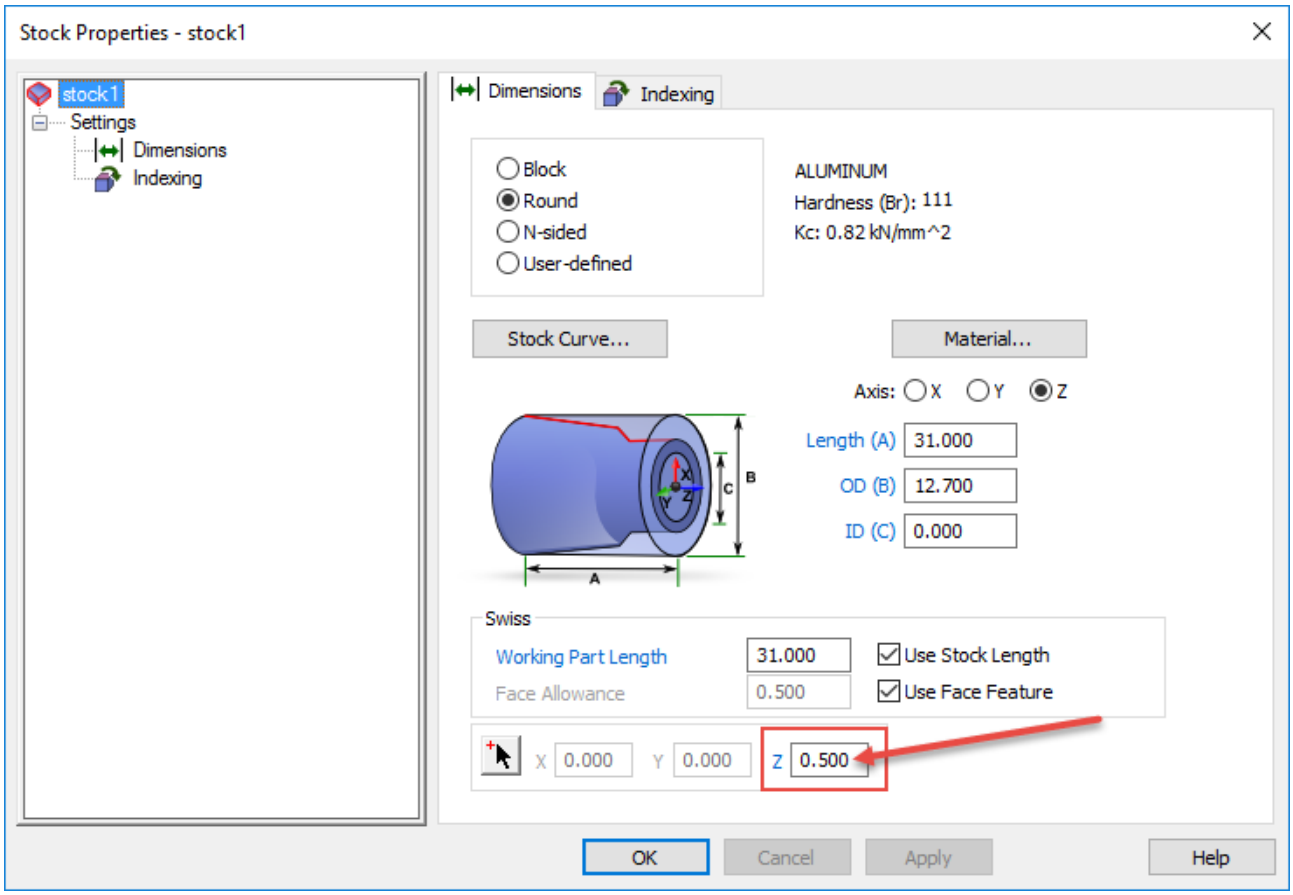
9 **Enter Length (A) 31** OD (B) **12.7 and ID (C) 0**. Select **Finish**.



© Copyright 2016 Autodesk, Inc. All rights reserved

10 Select **Finish.**

11 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.

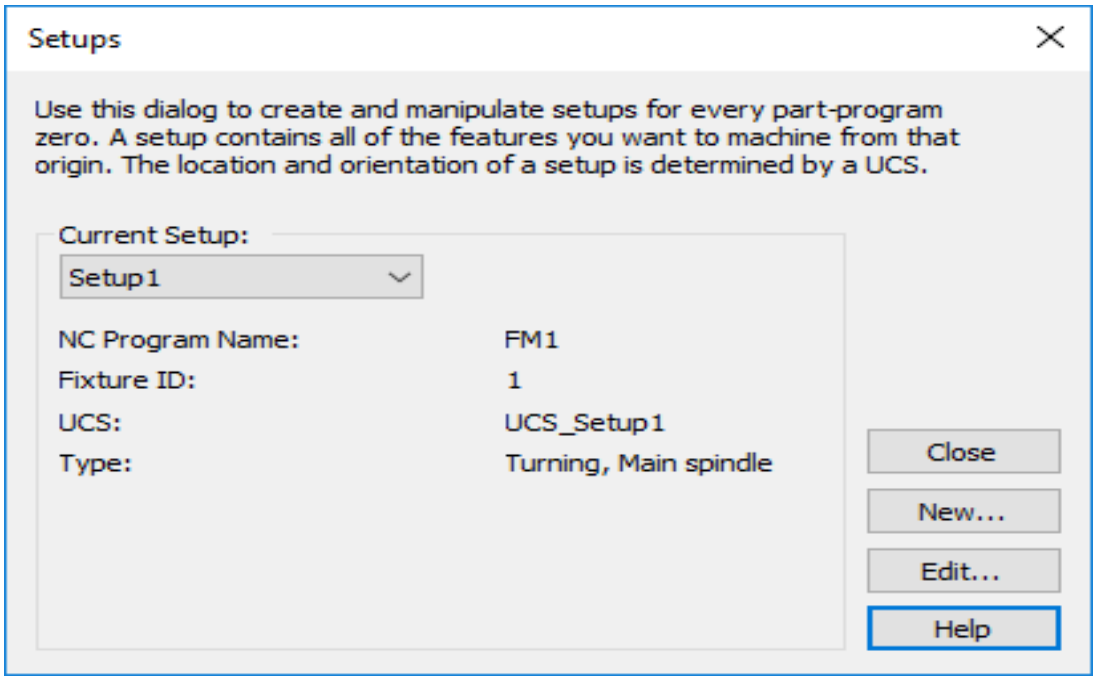


12 Enter **0.5mm** in **Z** as shown in the menu shown above. This will now set the **Z Setup1** as shown below.

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

14 Alternatively you can select **Setup1** in **PartView** and follow the instructions.

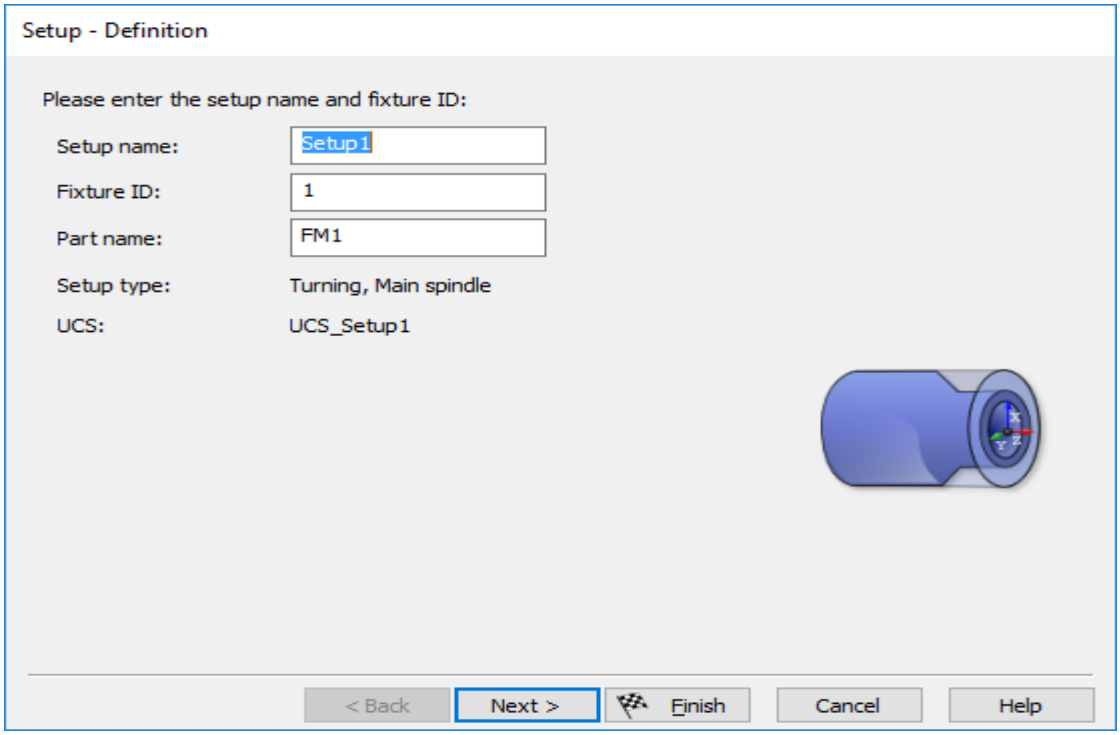
© Copyright 2016 Autodesk, Inc. All rights reserved



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

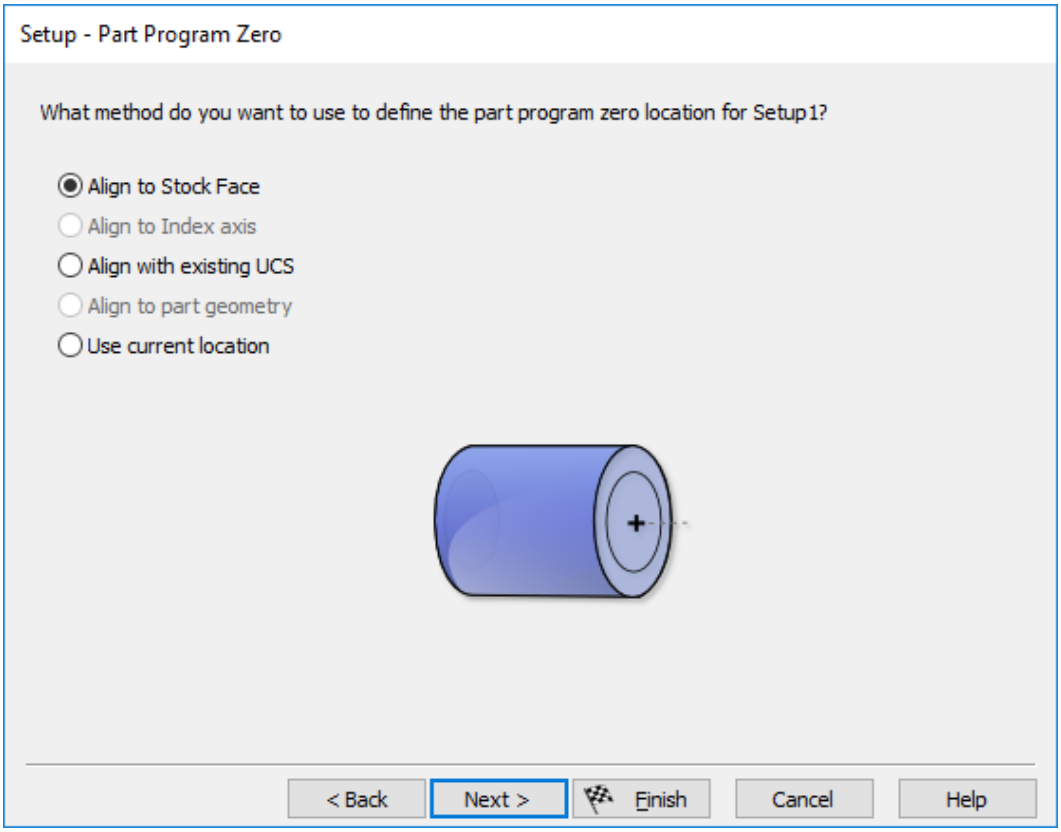
A few quick notes about ***Part Name***
The ***Part Name*** are inserted in and must be what is recognized in the NC code (*O0001*)**.** *The **Fixture ID** comes from the post processor and the call is applied to the NC code, such as **G54**.*
Also, the ***Part Name*** is used *to identify the program text file when you "Save NC" (Use Setup Part Name) under the File menu located upper left.*

15 Select **Edit**.



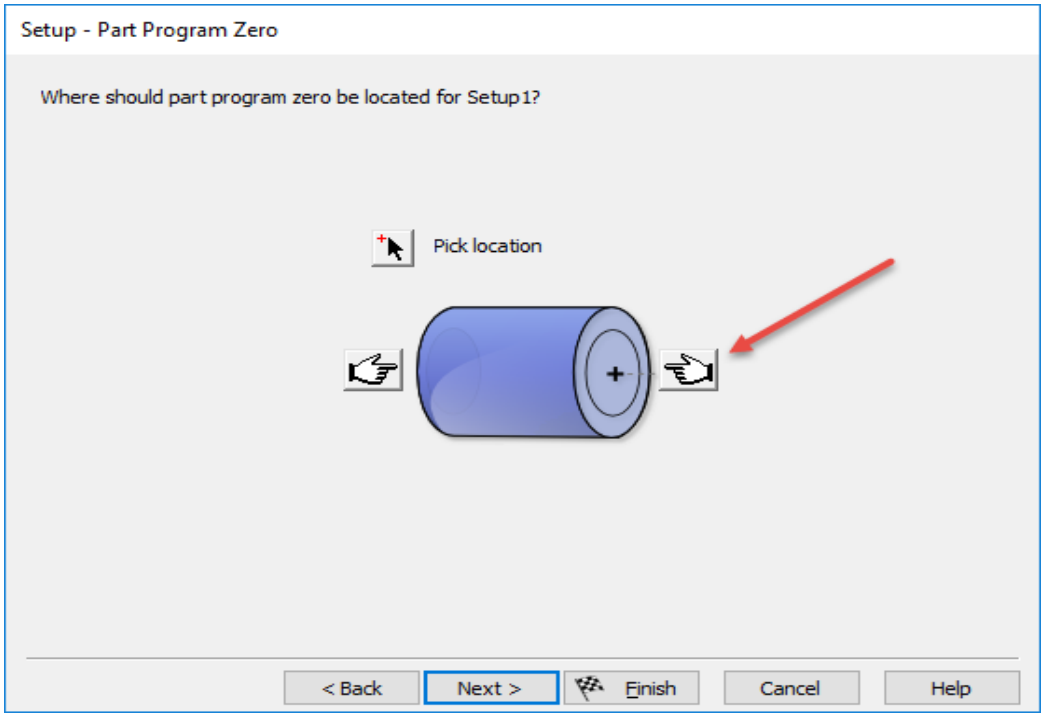
© Copyright 2016 Autodesk, Inc. All rights reserved

16 Select **Next**.



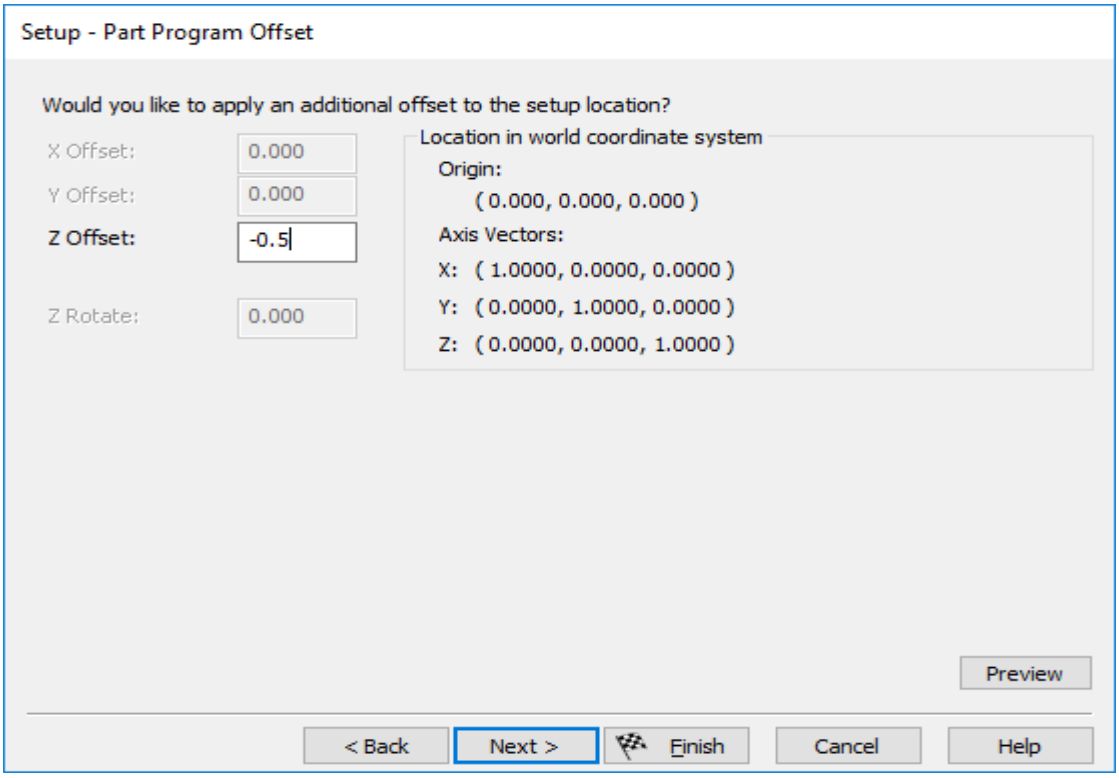
Setup – Part Program Zero provides several methods to define the part program zero location. Align to the face of the material, align with another existing UCS (User Coordinate System) or accept the current location.

17 *Choose* **Align to Stock Face.** Select **Next.** Choose the **Right hand Icon** as shown on the next page.

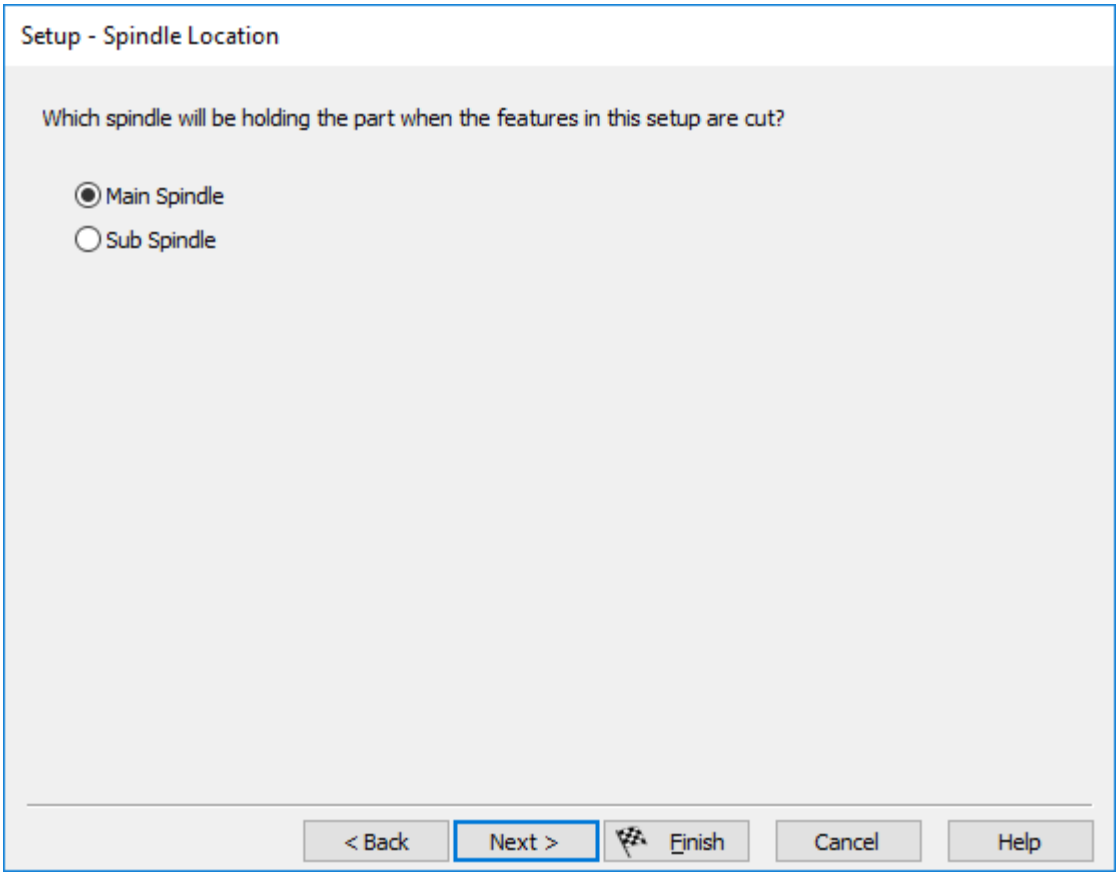


© Copyright 2016 Autodesk, Inc. All rights reserved

18 Select **Next.**



19 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.



© Copyright 2016 Autodesk, Inc. All rights reserved

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

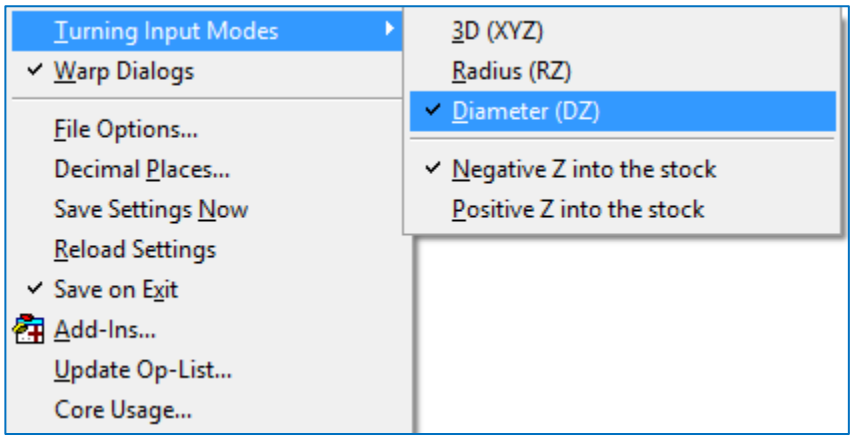
20 Select **Main Spindle**

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.

21 Select **Finish.**

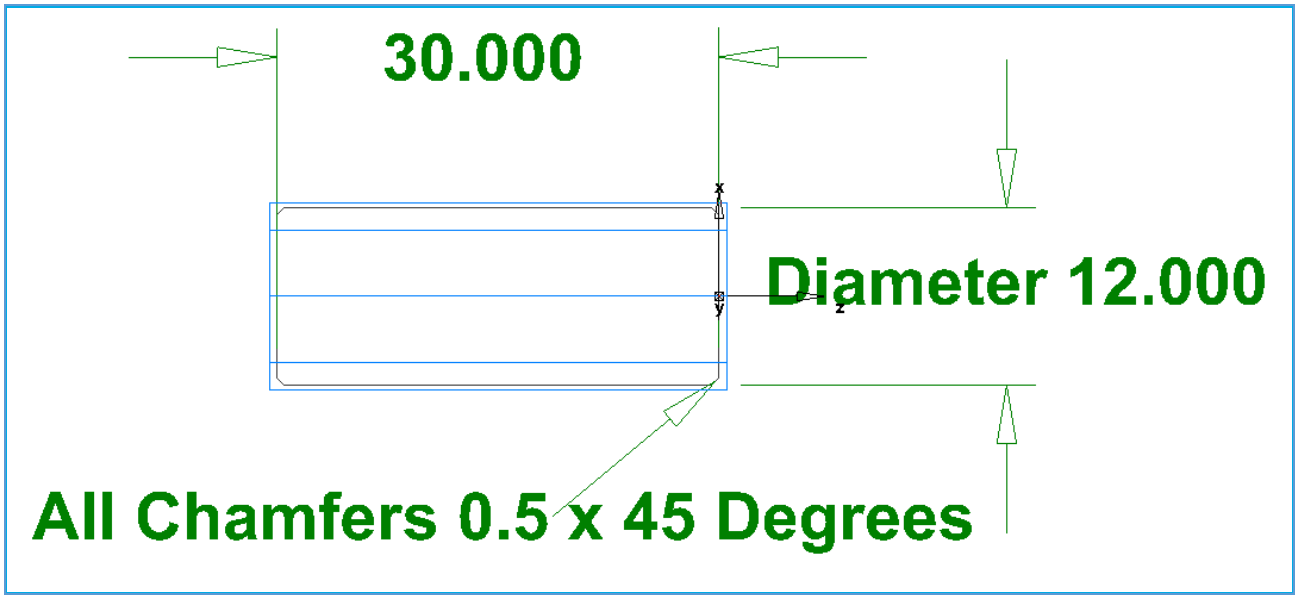
Turning Input Modes

Diameter (DZ) input modes. Our drawing is dimensioned in **Diameter (DZ)** values.



Geometry Creation Techniques

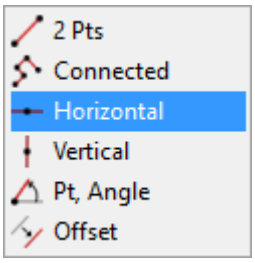
Diameter (DZ)



© Copyright 2016 Autodesk, Inc. All rights reserved

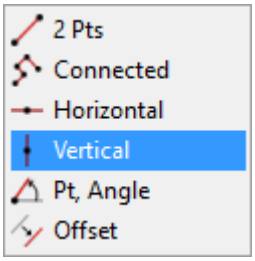
There are many ways to draw this job but the following method is by far the easiest.

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

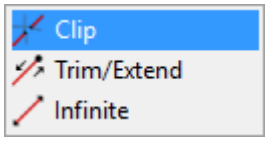


23 Or input the following diameter Values **12mm & -12mm**

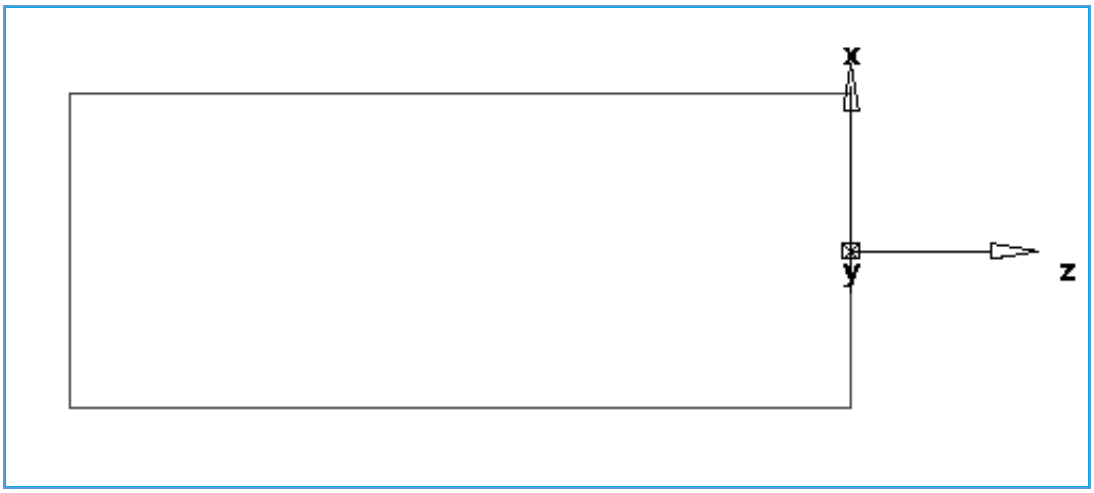
24 Then Select the **Line vertical** and input the following values



25 **0, -30**

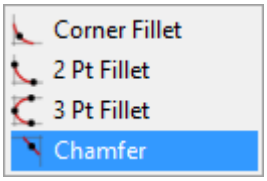
26 **Clip** 

the surrounding lines as shown.



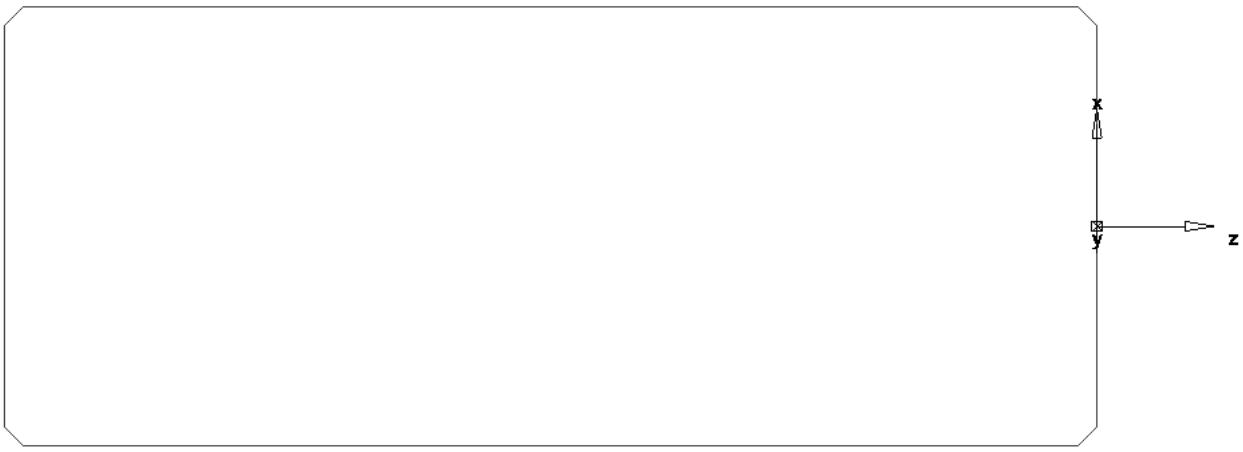
27 Now add the **Chamfers** by using the **Chamfer** command.

28 We do not need to draw the bottom half of the part. I have just done this for effect.

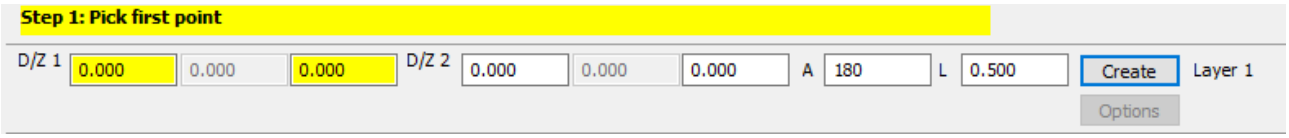


29 The **chamfer is 0.5mm x 45 degrees apply to all corners.**

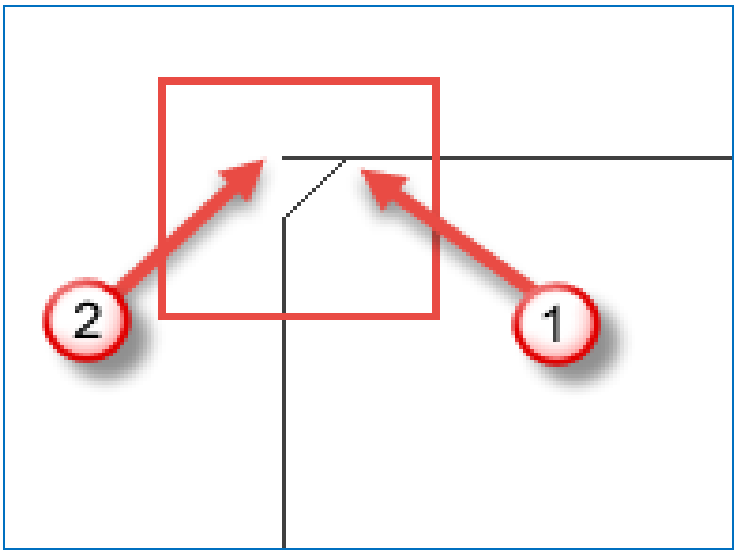
© Copyright 2016 Autodesk, Inc. All rights reserved



30 We now need to draw a line from 2 points. Snap to the top part of the left hand chamfer. Type in the values as shown. Use the Tab key to move to the next field and type in an **Angle** of **180** and a **length** of **0.5mm**



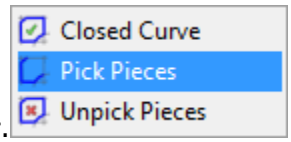
31 That is it, the Geometry is now complete.



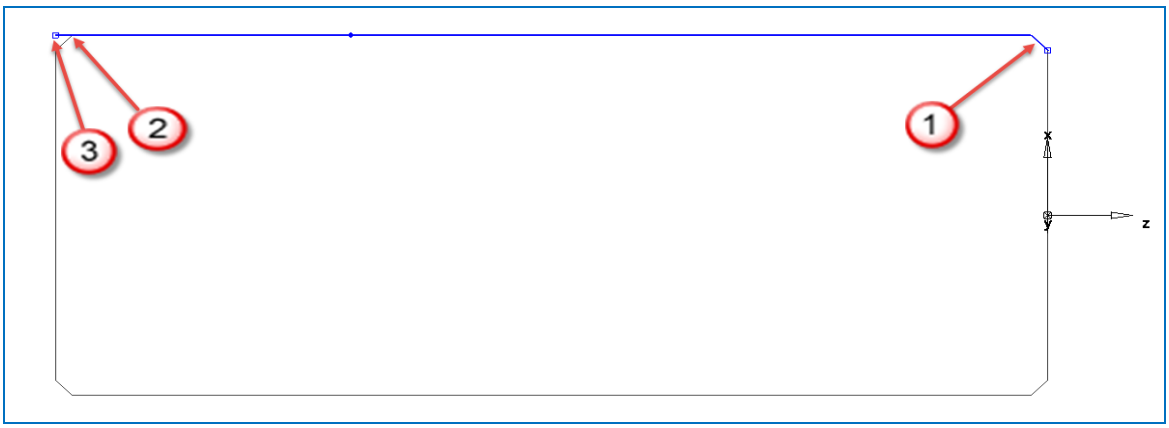
© Copyright 2016 Autodesk, Inc. All rights reserved

Curve Creation

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

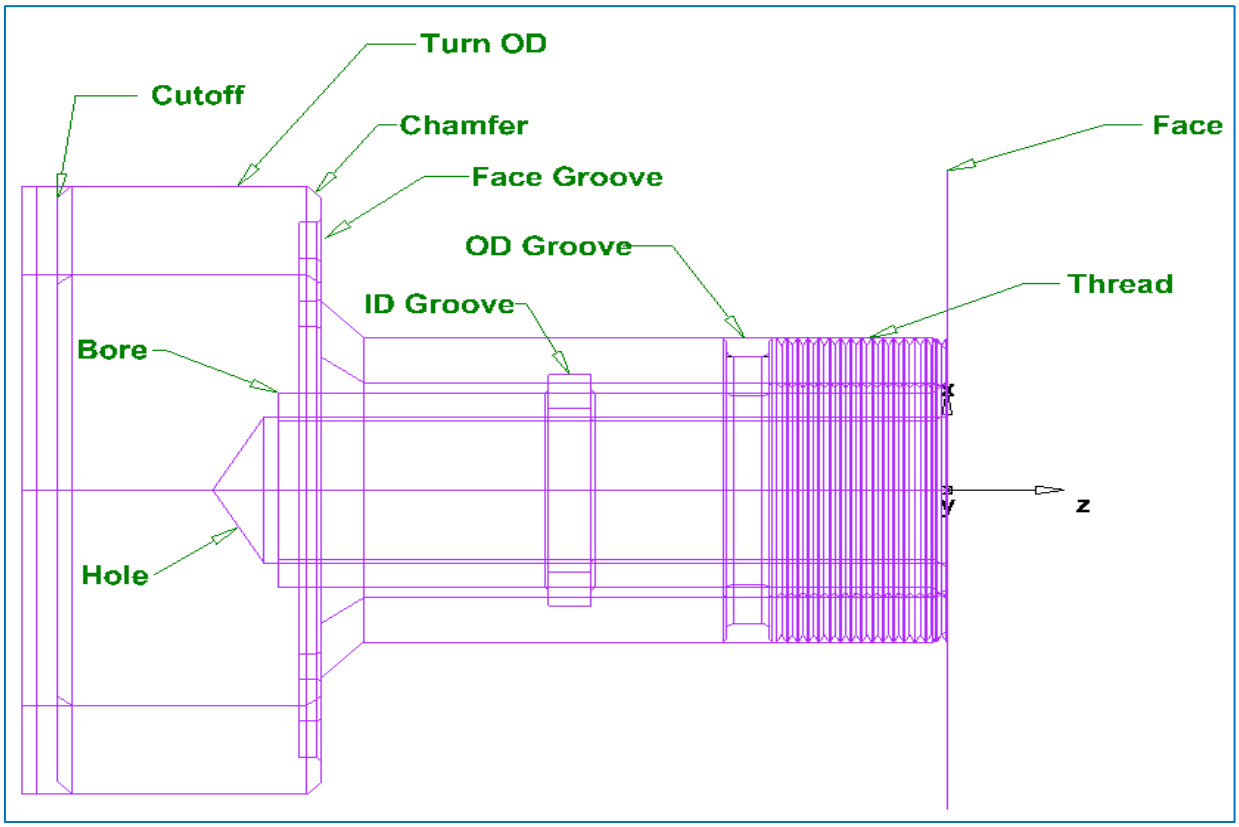


33 Select the Pick Pieces Icon Click on locations 1, 2, and 3 as shown in the image below. Each line segment will change colour when it is selected. Name this curve "od profile" and press the ENTER key. Your drawing should look like the one below.



34 You should now have one Curve that will be shown in the **Part View**.

35 The image below shows the different types of features that can be created using **Swiss Turning.**

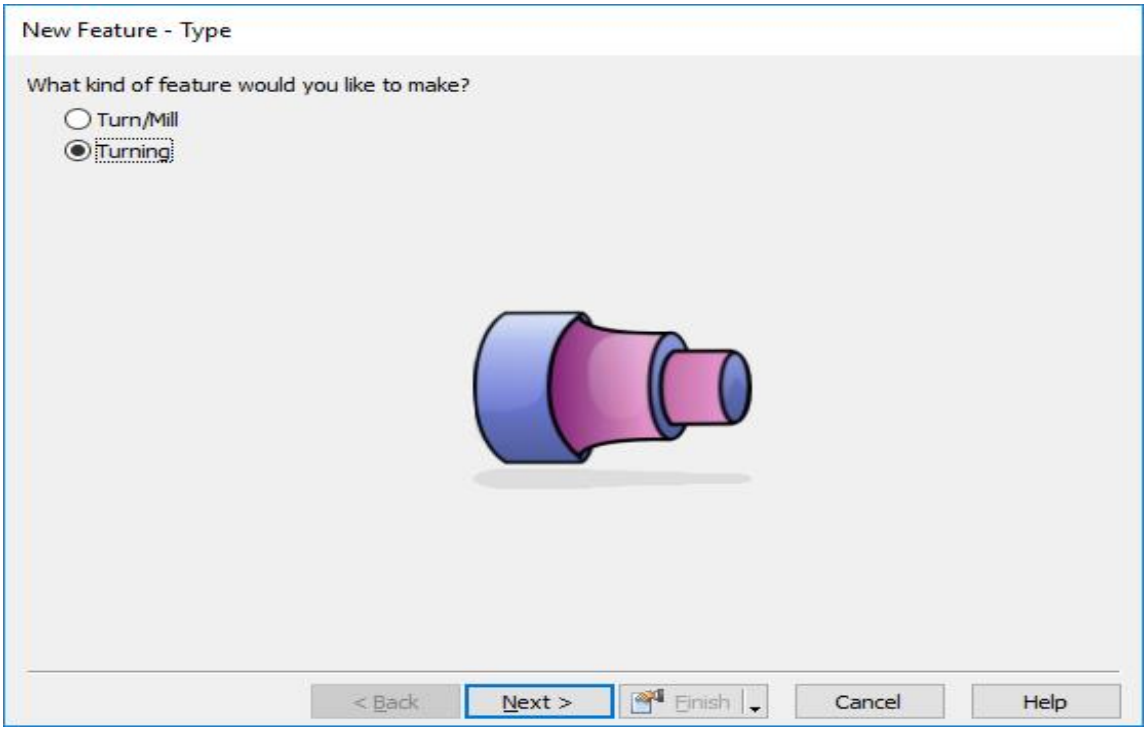


© Copyright 2016 Autodesk, Inc. All rights reserved

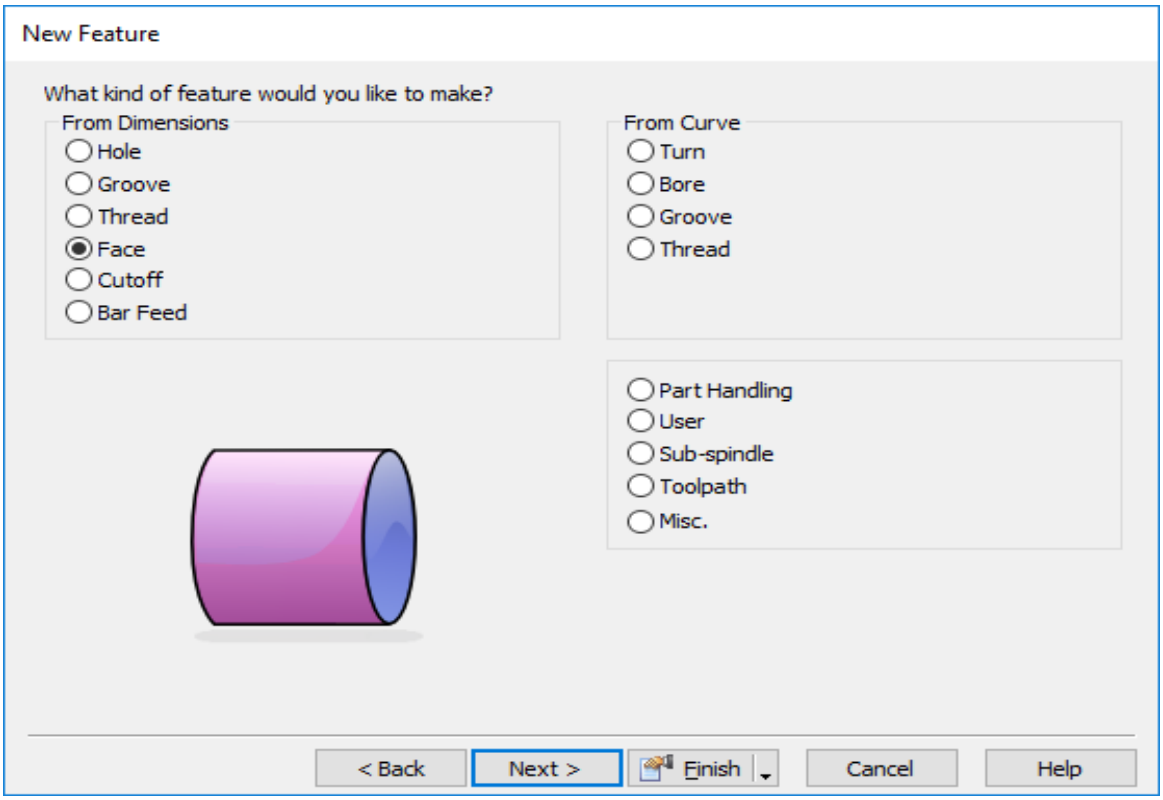
Creating a Face Feature

36 Click **Features** from the **Steps** menu in **PartView** or select **Ctrl + R (New Feature)**

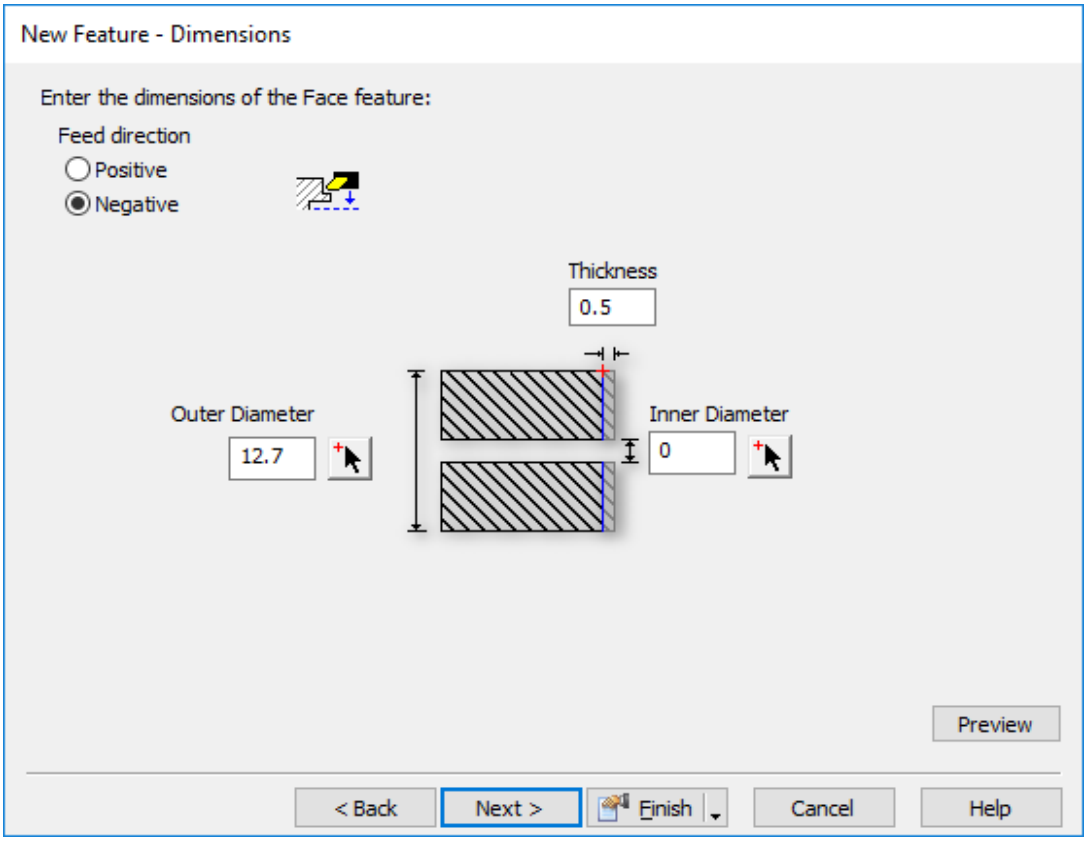
37 Select **Turning.**



38 **FeatureCAM** will show the following menu. Select the **Face from Dimensions** radio button. Select **Next** twice.

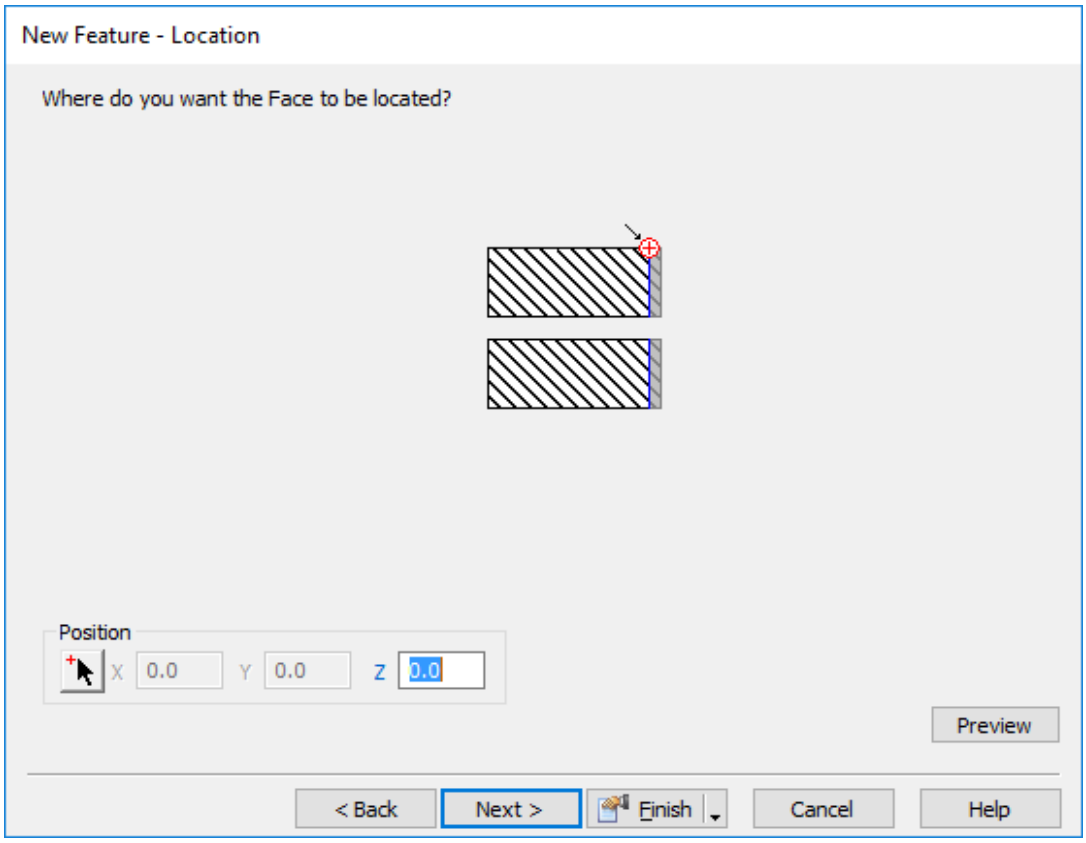


© Copyright 2016 Autodesk, Inc. All rights reserved



39 **Negative** is **towards** the **chuck** and **Positive** is away from the **chuck**. **Thickness** is the amount of material that **FeatureCAM** will machine away.

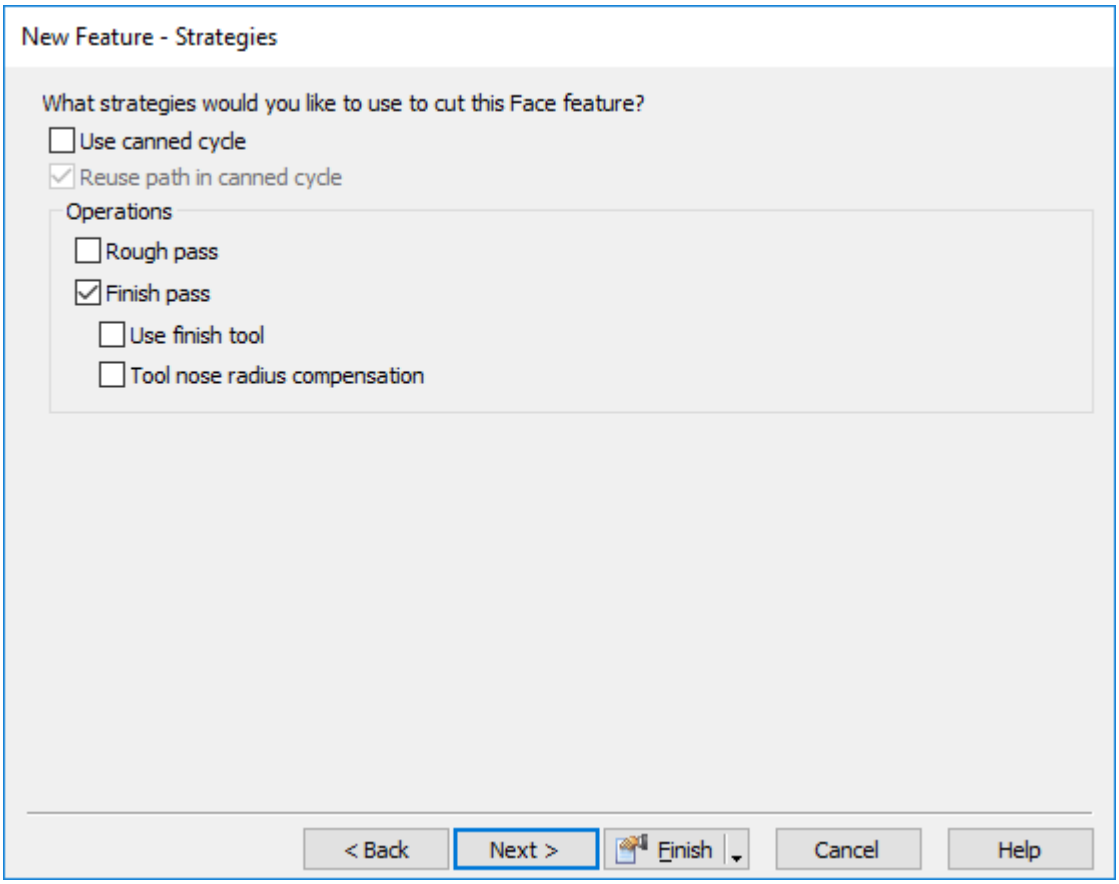
40 Select **Next.** The following menu will appear.



© Copyright 2016 Autodesk, Inc. All rights reserved

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

42 Select **Next**. The menu is shown below.



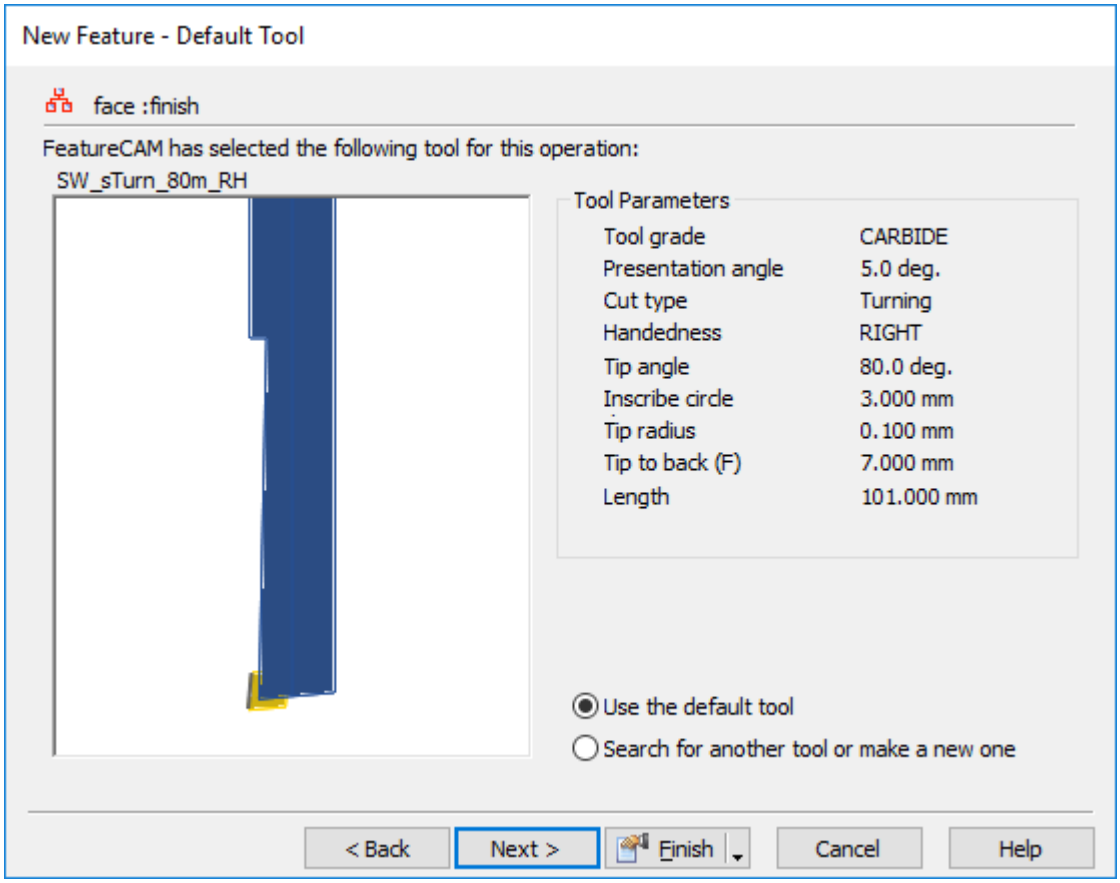
43 Notice that we have the ability to use **Canned Cycles**. The canned cycle will be output through the post. (If supported) This gives greater flexibility to the operator. He can then change this on the machine instead of having to go to the **FeatureCAM** programmer (Production Engineer) and re-output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

*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.*

44 Select **Next** twice.

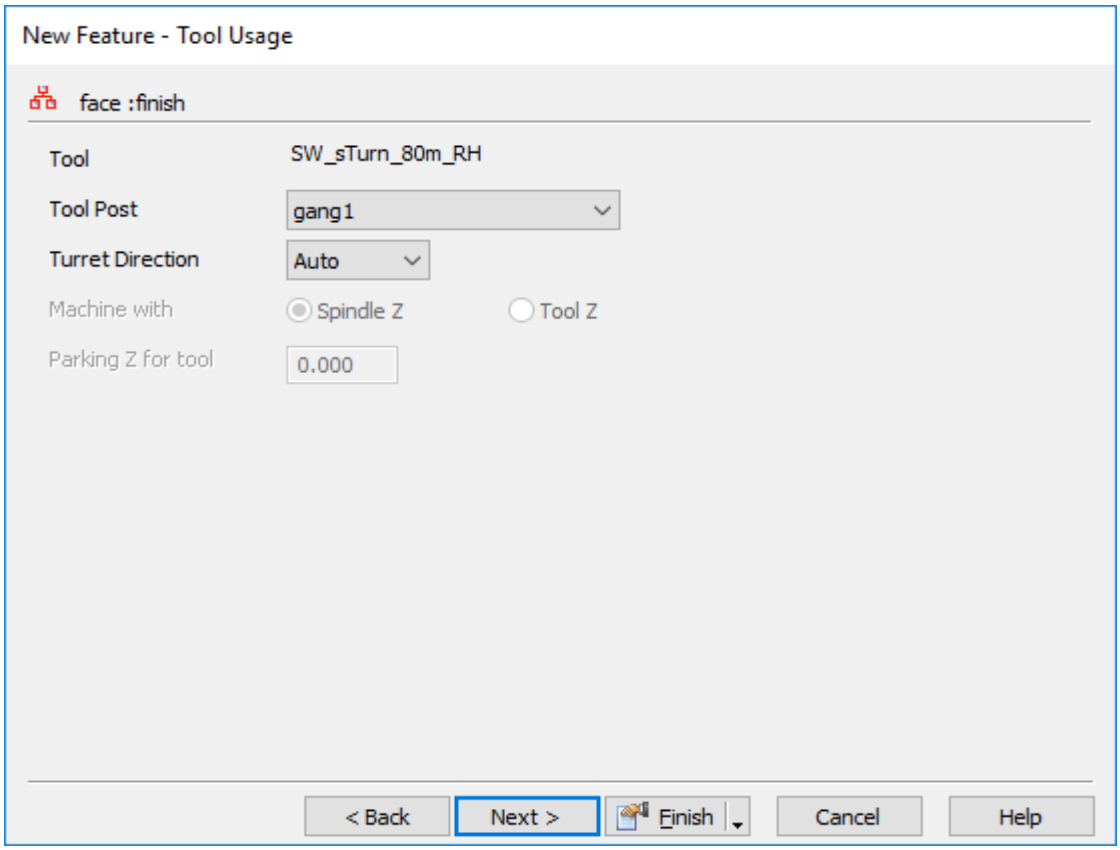
45 **New Feature – Default Tool**. You can select the default tool or search the tool database for a more suitable tool. See menu on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved



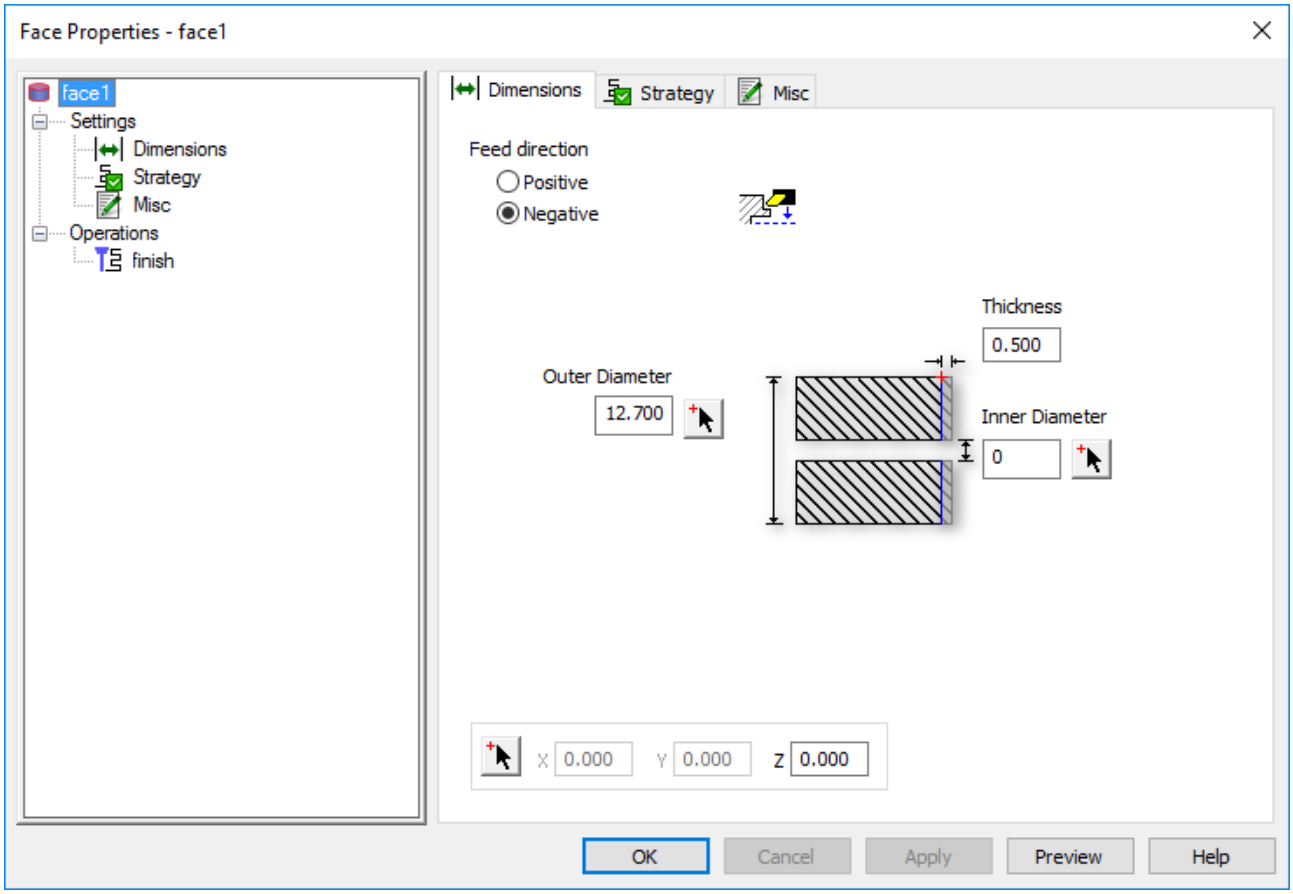
46 Select **Next** twice.

*It is important to know if the tool is located in **Gang1***



© Copyright 2016 Autodesk, Inc. All rights reserved

47 Select **Finish**.



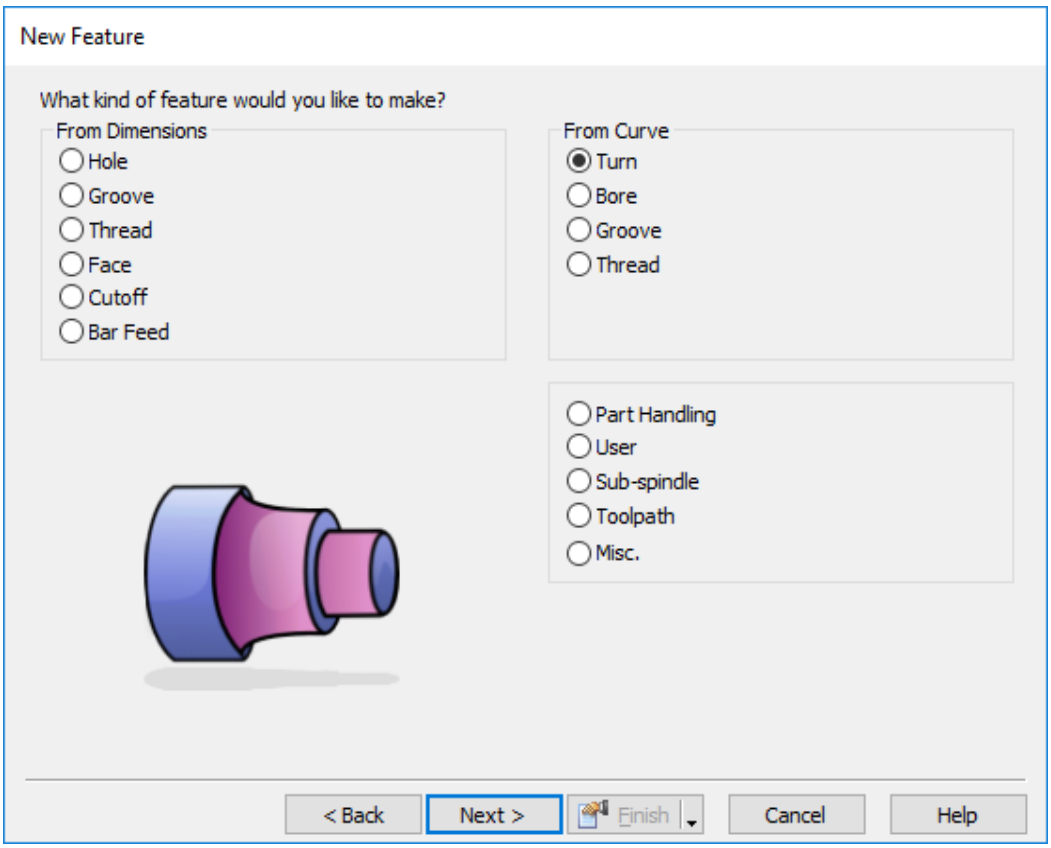
48 Select the **3D Simulation** Icon and then the **Play** button.

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

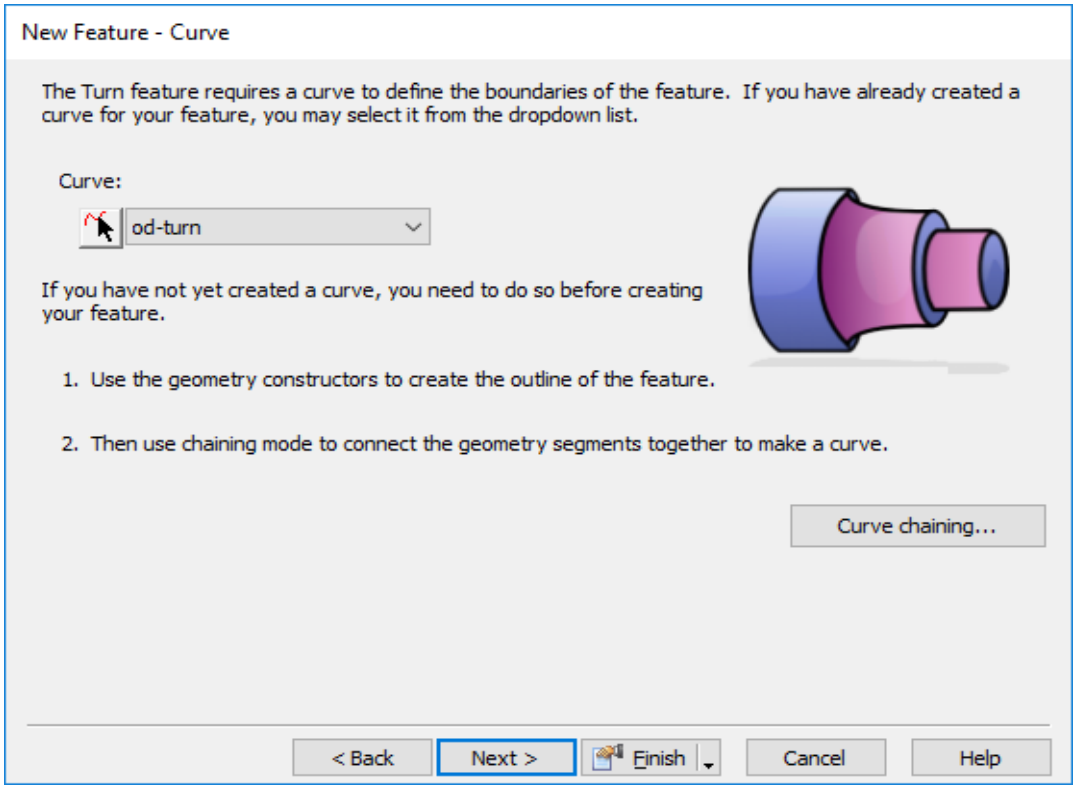
© Copyright 2016 Autodesk, Inc. All rights reserved

Creating a Turn Feature

50 Select **Ctrl + R** or select **Steps** and **Features** to access the **New Feature** menu. Select **Turning** Select **Next,** Select **Turn.**

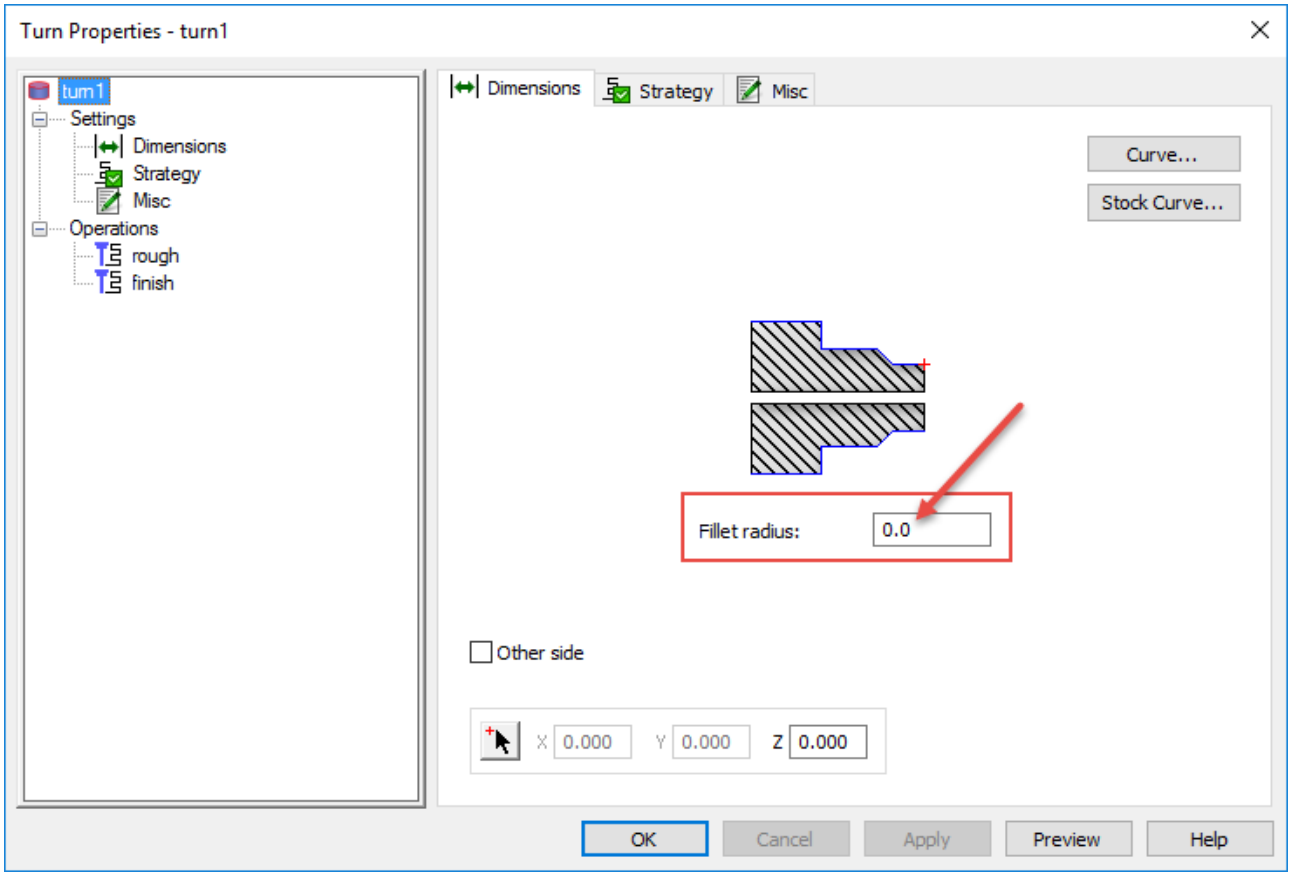


51 Select **Next** and the following form will appear.



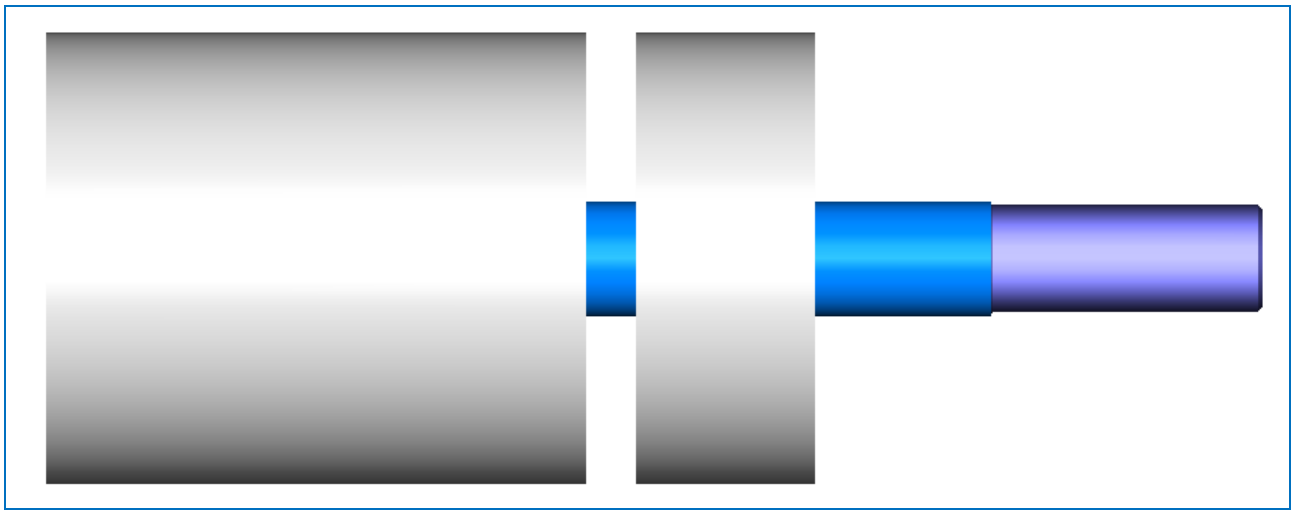
© Copyright 2016 Autodesk, Inc. All rights reserved

52 To make the machining more efficient unselect the Roughing operation. We only have **0.7mm** on for finishing. This will drastically reduce the cycle time.



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

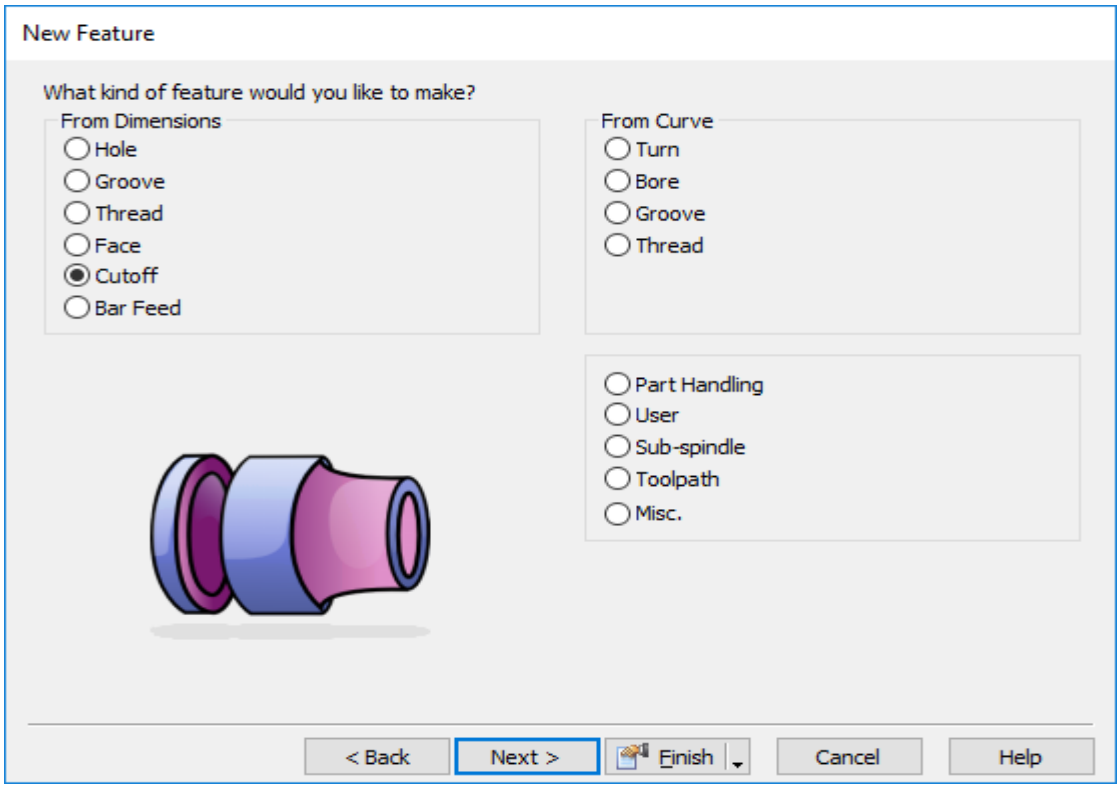
54 Run the **3D simulation** as shown.



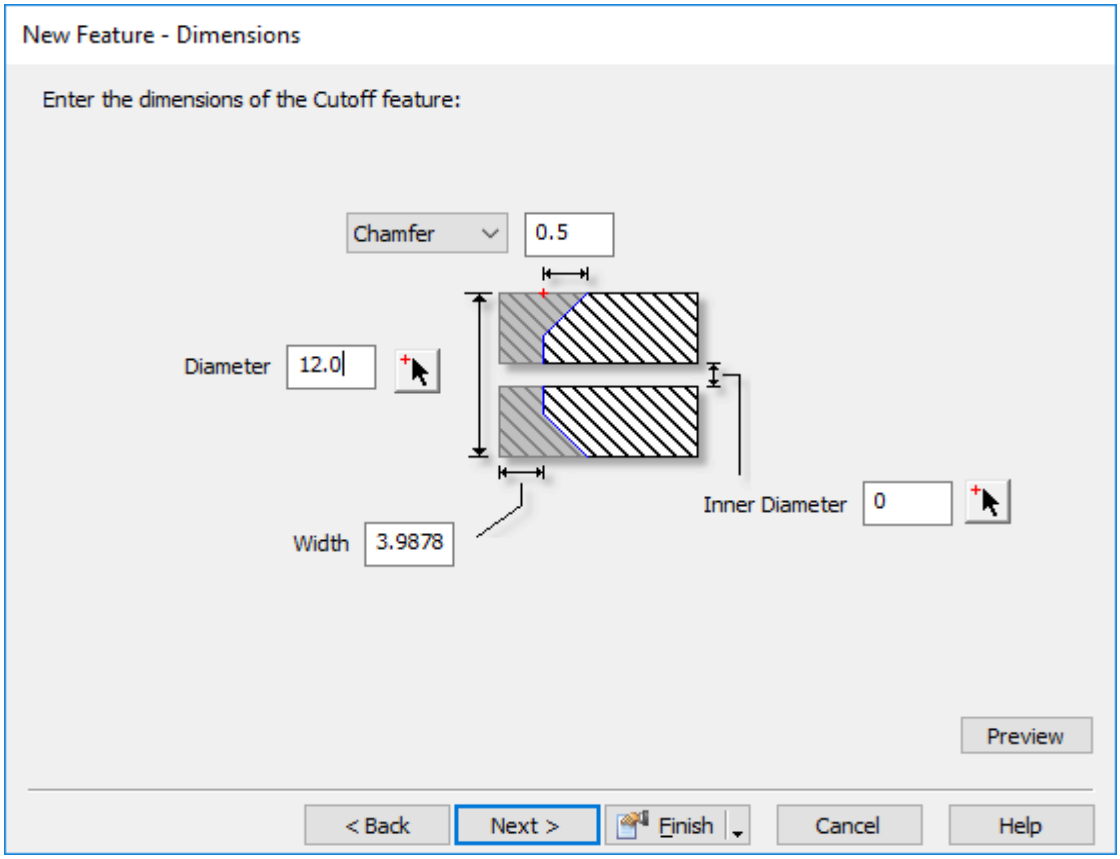
55 Remember to select the **Eject** button to clear the screen.

© Copyright 2016 Autodesk, Inc. All rights reserved

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

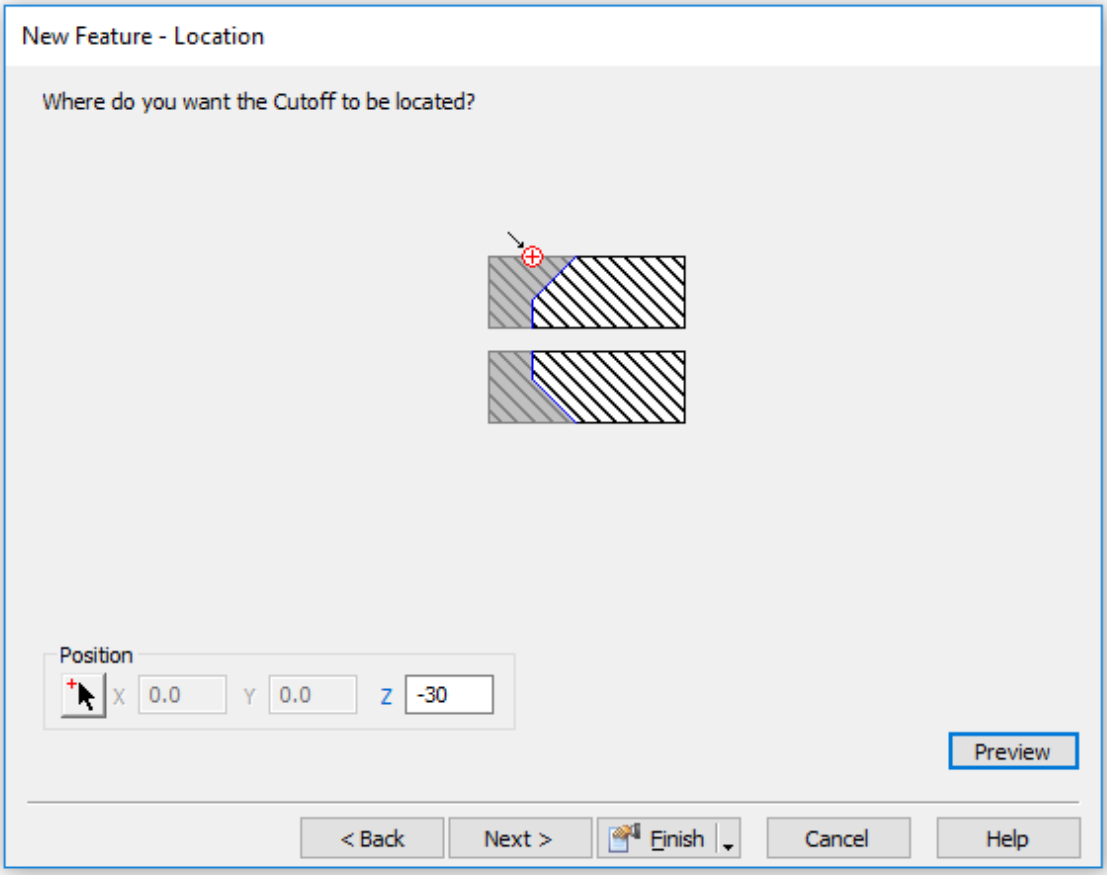


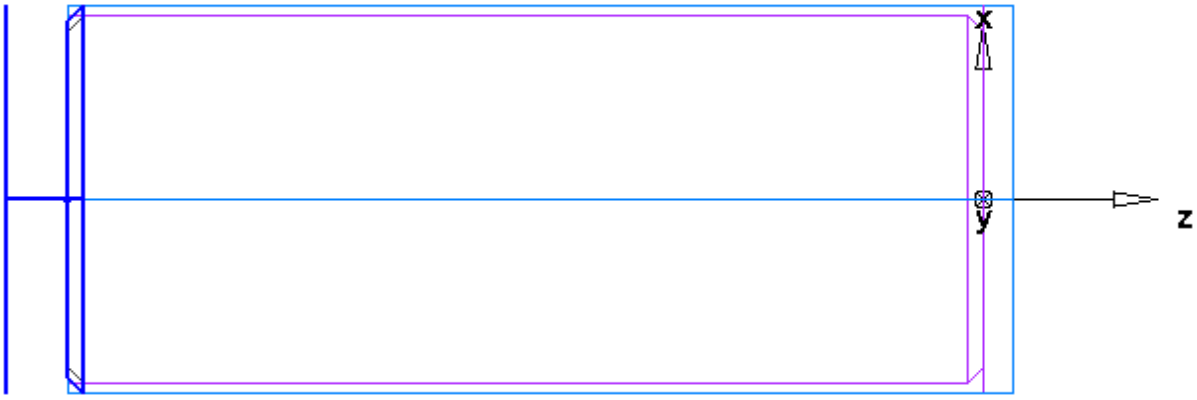
57 On the Dimensions page set Chamfer to **0.5** then set **Diameter** to **12mm** and **Inner Diameter** to **0** and Width **to 2.0**



© Copyright 2016 Autodesk, Inc. All rights reserved

58 Enter a Z position of -30mm





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

60 If the tool used in the cutoff process collides with the part, complete the following steps.

61 In **PartView** double click on **Cutoff1**.

62 Then **double click** on the tool selected.

63 Select **Holder** and change the **Cut Depth** value to **55mm** as shown.

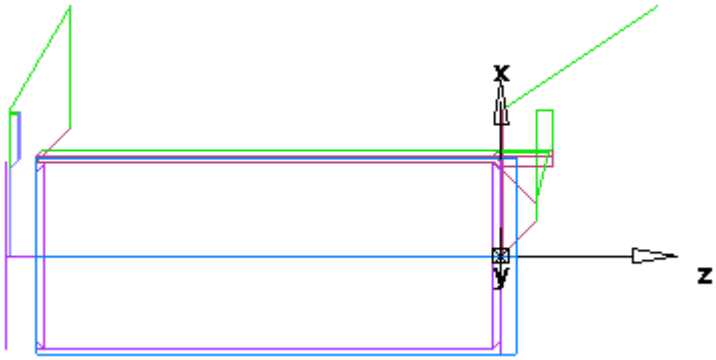
© Copyright 2016 Autodesk, Inc. All rights reserved

Results

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

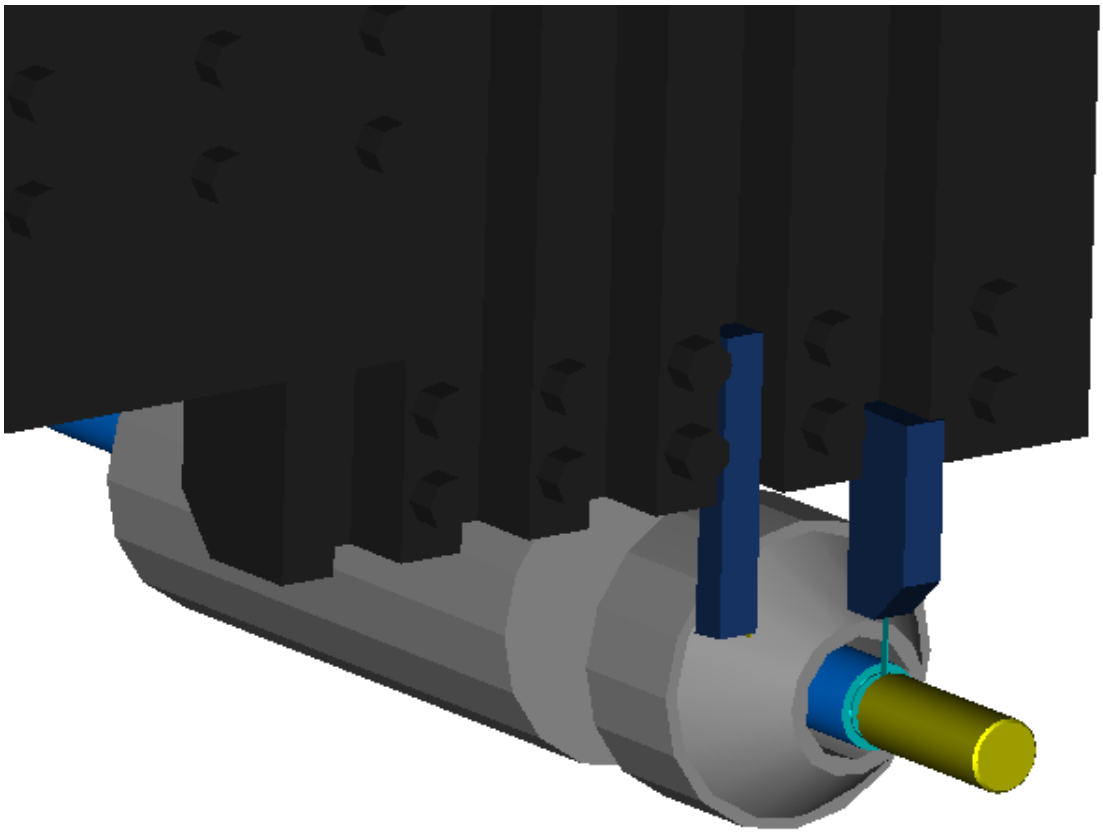
*The sample part is complete. View the **2D Centreline Simulation** followed by a **3D simulation** to view the **finished part.***

2D Simulation with Rapid moves



Machine Simulation

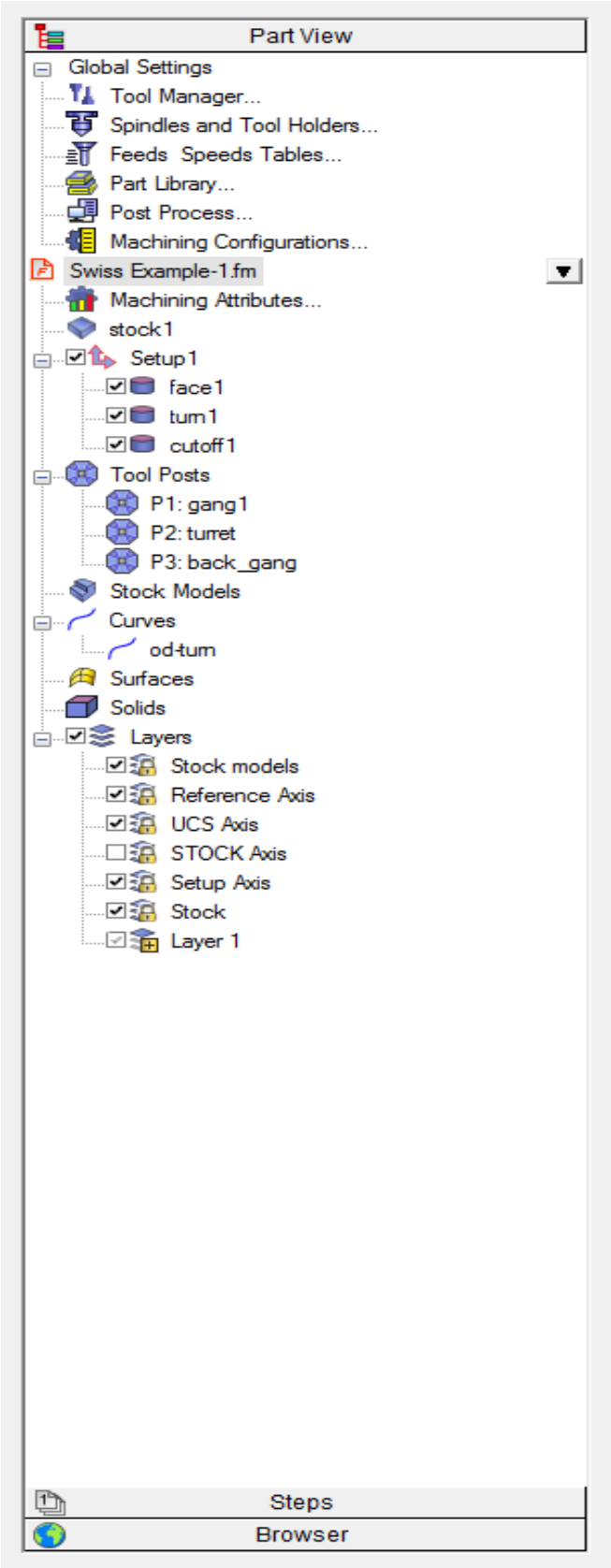
1 To show the machine simulation select the following icon



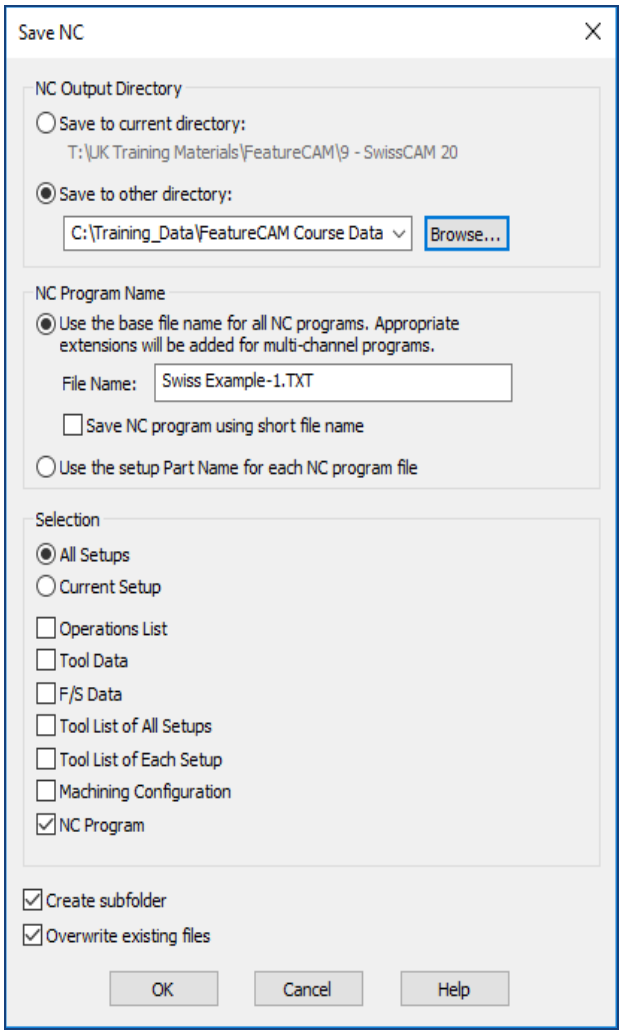
© Copyright 2016 Autodesk, Inc. All rights reserved

2 To output the code to a known location select **File>Save NC** and select **Save to other directory.**

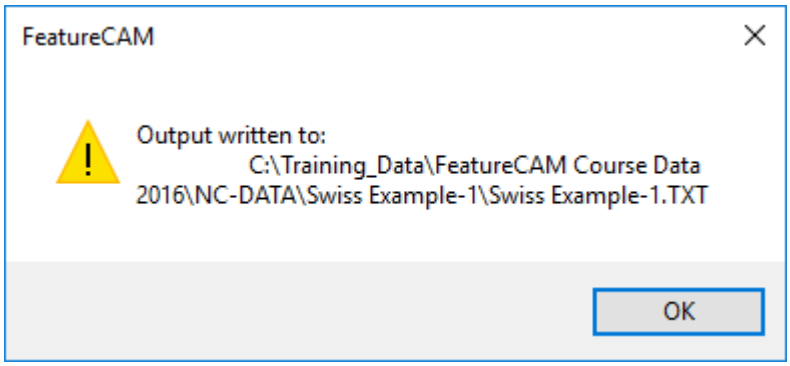
Part View All Operations



© Copyright 2016 Autodesk, Inc. All rights reserved



NC Code output example

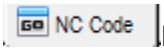


Part Documentation

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.

© Copyright 2016 Autodesk, Inc. All rights reserved

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.
4. **NC Code**
5. 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.
6. Before you can generate NC code for a part, you have to run a Simulation to calculate toolpaths. Simulate your part now.
7. Click the **NC** *code* step in the Steps toolbar.
8. Click the **Display the NC code** 

button in the dialog box to generate.

```
%
O1001 (  FM1  )
$1
( BEGIN CHANNEL 1 PROGRAM )

(  Citizen L20E-IX  )
(  DATE: 11-1-2016  )
(  TIME: 15:11:23  )

G99 M97 M87
G113 M9
G99
M6
/M52
#500=1.6 (DIST FROM TOOL ZERO TO FARTHEST SIDE OF INSERT)
#501=1.6 (CUTOFF TOOL WIDTH)
#502=0.5 (FACE ALLOWANCE)
G50 Z[#500-#501-#502]
G0 Z-.05
M3 S1=2500
G600 (WAIT1)
( MAIN SPINDLE,  TURNING )
(  FINISH  FACE  FACE1  )
T0200 (SW_sTurn_80m_RH)
```


© Copyright 2016 Autodesk, Inc. All rights reserved

```
33 G50 S6000
34 G96 S487
35 G99 G18 G0 Z-125.0 T02
36 X400.0
37 X19.975 Z0.
38 G1 X0.  F0.152
39 X4.243 Z-2.121
40
41 ( MAIN SPINDLE,  TURNING )
42 (  ROUGH  TURN  TURN1  )
43 G50 S6000
44 G96 S365
45 G0 X20.0 Z-2.121 T02
46 G0 Z-3.071
47 X12.2
48 G1 Z30.0  F0.381
49 X14.0
50 X14.707 Z29.646
51 G0 Z-3.071
52 G1 X11.141
53 Z-0.071
54 X12.141 Z0.429
55 G2 X12.2 Z0.5 R0.1
56 G1 X12.907 Z0.146
57 G0 Z-3.0
58
59 ( MAIN SPINDLE,  TURNING )
60 (  FINISH  TURN  TURN1  )
61 G50 S6000
62 G96 S487
63 G0 X6.757 Z-2.121 T02
64 G1 X12.0 Z0.5  F0.152
65 Z30.0
66 X16.243 Z27.879
67 G0 X400.0
68 T0
69
70 ( MAIN SPINDLE,  TURNING )
```


© Copyright 2016 Autodesk, Inc. All rights reserved

```
71 (    CHAMFER  CUTOFF1   )
72 T0100 (SW_sCut_160m_RH)
73 G50 S6000
74 G96 S365
75 G99 G0 Z27.929 T01
76 X400.0
77 X19.9 Z32.0
78 X18.2
79 G1 X11.0 F0.381
80 G4 U2.0
81 G0 X18.2
82 Z30.986
83 G1 X12.171  F0.381
84 X11.0 Z31.571
85 G0 X18.2
86 Z32.0
87 G1 X11.0
88 Z31.571
89 G4 U2.0
90 G0 X19.9
91 G50 S6000
92 G96 S487
93 X18.0
94 G1 X10.9  F0.152
95 G0 X18.0
96 X18.112 Z31.015
97 G1 X12.112
98 X10.971 Z31.586
99 G2 X10.9 Z31.6 R0.05
100 G1 X-0.1
101 T0
102 M5
103 G600 (WAIT2)
104 M53
105 M5
106 M7
107 G0 X0. Z[#500-#501-#502] T0
108 M56
```


© Copyright 2016 Autodesk, Inc. All rights reserved

```
109 G999
110 N999
111 M2
112 M99
113
114 $2
115 ( BEGIN CHANNEL 2 PROGRAM )
116
117 G50 Z0
118 M15
119 G44
120 T3000
121 G600 (WAIT1)
122 M88
123 G600 (WAIT2)
124 M141
125 M25
126 G999
127 N999
128 M2
129 M99
130 POST MESSAGE: #990 not set; default value will be used
131 $0
132 A2-LE-1-20-9-P-M
133 #814=0000140000
134 #815=0000000500
135 #816=0000001000
136 #817=0002500000
137 #822=0000000010
138 #824=0000000000
139 #818=0000546002
140 #819=0000001000
141 #820=0000000000
142 #821=0000000000
143 #990=0015728644
144 #991=0000051000
145 #992=0000060000
146 #893=0000000000
```


© Copyright 2016 Autodesk, Inc. All rights reserved

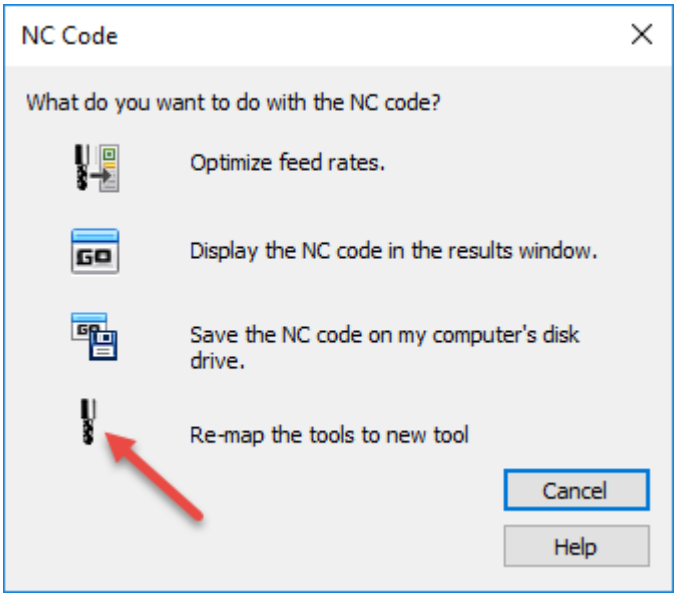
```
#25119=0000000000
#25115=0000000000
%
```

Tool Mapping

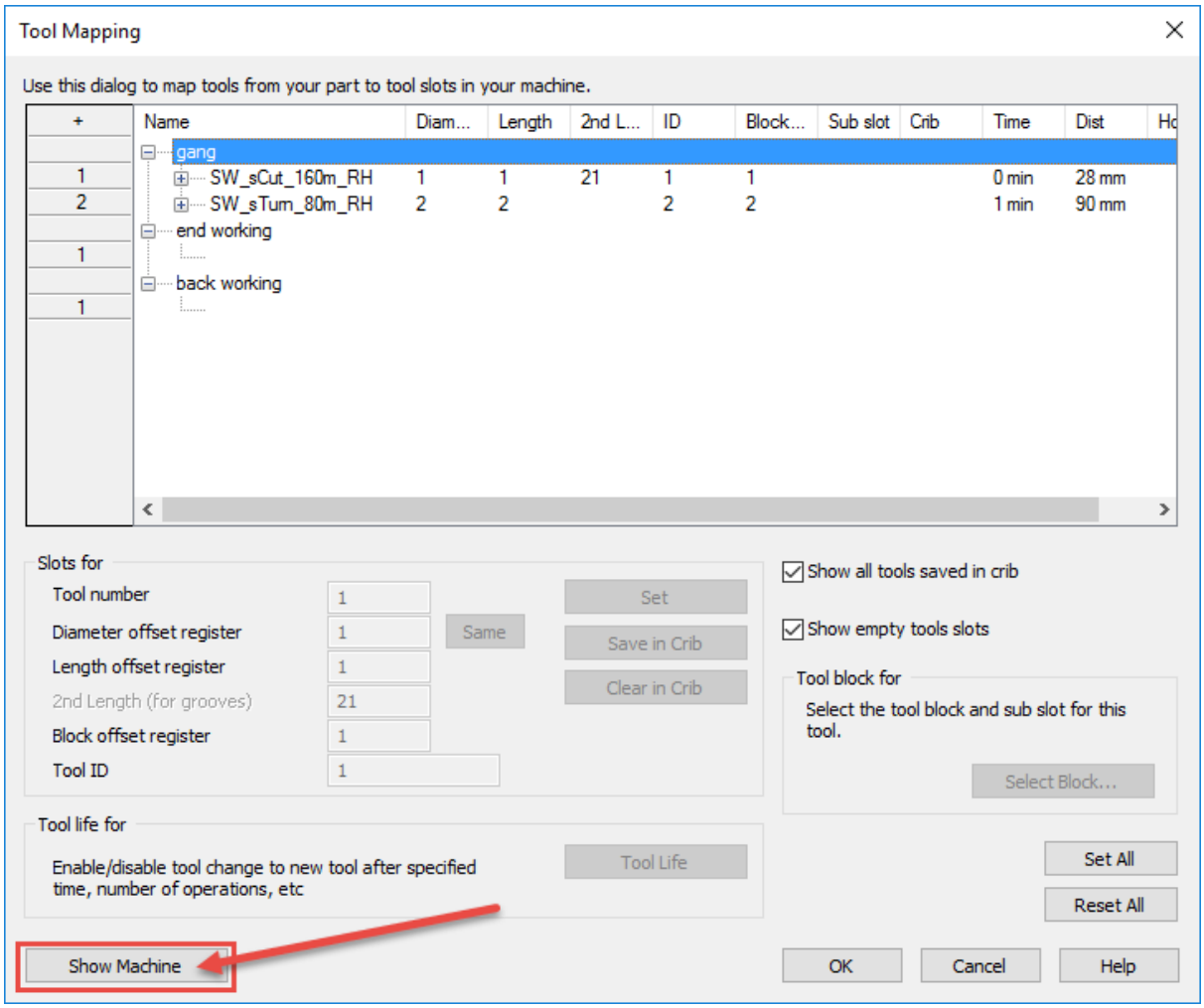
You can open the Tool Mapping dialog in one of these ways:

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

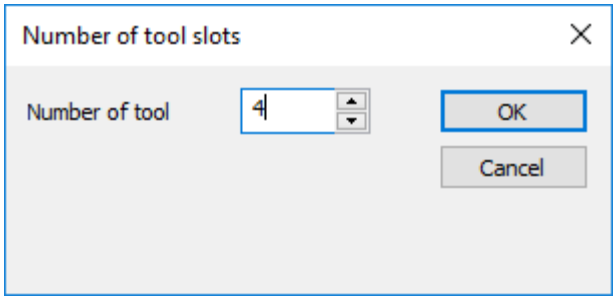
151 Click NC Code in the Steps panel, then click Re-map the tools to new tool slots in the NC Code dialog.



© Copyright 2016 Autodesk, Inc. All rights reserved



- **Show Machine** — In Swiss Turning documents, click this button to show a preview of the active MD file on the right of the dialog with the tool slot numbers displayed.
- 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:

152 Use the number assigned in the **Tool Mapping** dialog.

This numbering is in effect for the current part only.

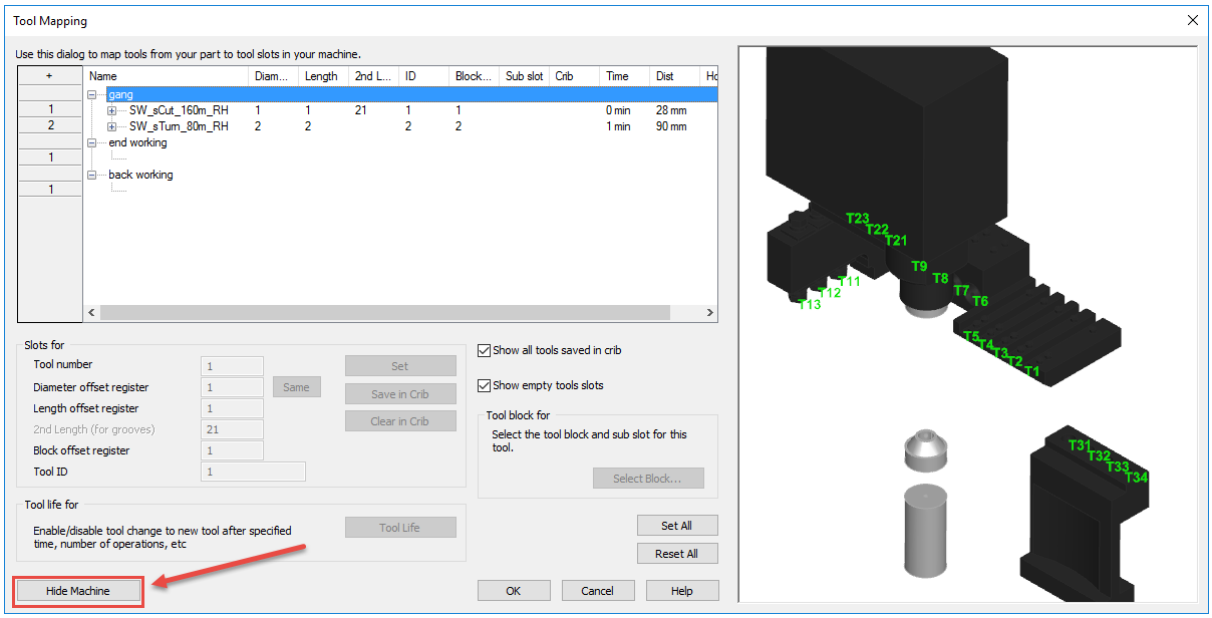
*If no number has been assigned via tool mapping, then the number assigned to the tool in the crib (the **Tool number** field for milling tools or the **Tool slot** for turning tools), is used as the tool number. If two tools have the same permanent number in the crib, the first tool used is assigned the pre-set number and the other tool is given a new number.*

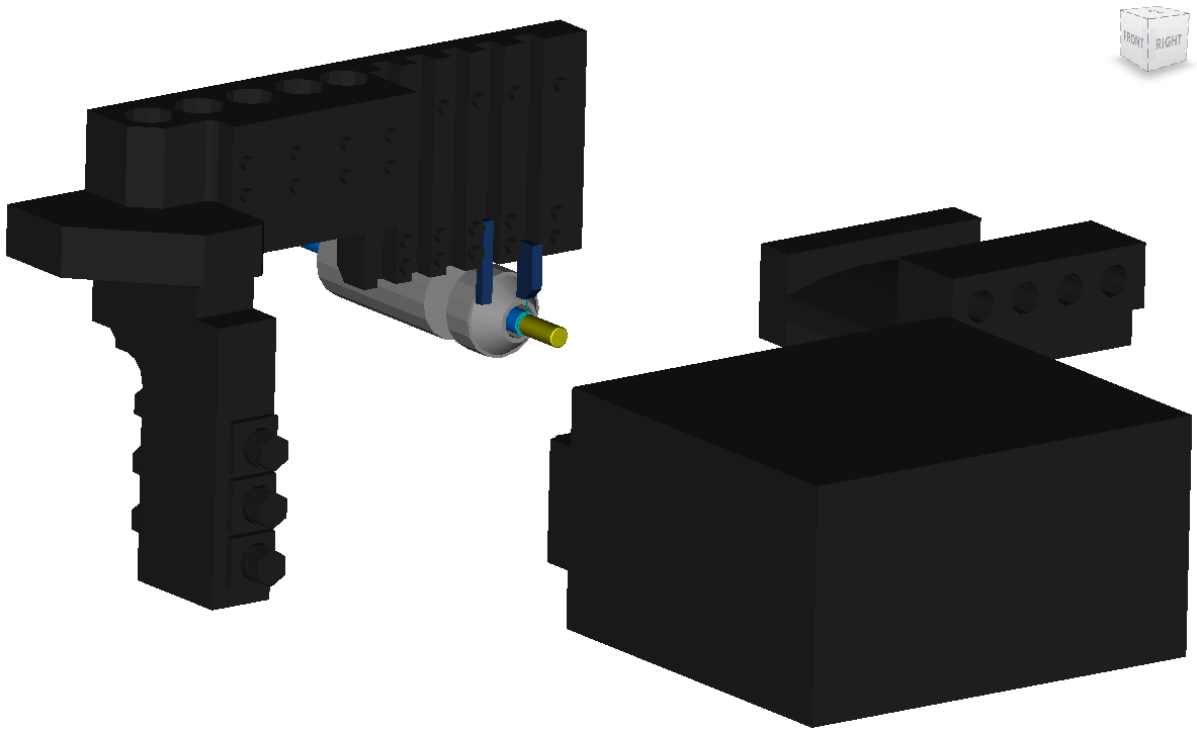
If no number has been assigned via tool mapping or in the crib, FeatureCAM assigns a tool number.

Show Machine in Tool Mapping

- **Show and Hide Machine** - In Swiss Turning documents, click this button to show a preview of the active MD file on the right of the dialog with the tool slot numbers displayed.
- To control the view of the machine preview, right-click the image and select Pan and Zoom or Rotate, then click and drag the image.
  Click Hide Machine to hide the machine preview.

© Copyright 2016 Autodesk, Inc. All rights reserved





© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Swiss 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
Post Variables 21
Misc. tab (Turn) 22
Turret location 25
Operations 26
Base priority 27
Use Rules 28
Use Template 29
What are Machining Attributes? 30
Creating Machining Configurations 32
The Attributes 33
Drilling Tab 33
Pecking Tab 33
Turn/Bore 33
Threading 33
Grooving 33
Cut-off and Bar Feed 33
Miscellaneous 34
Operations 34
Help Contents 34

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss Turning Attributes / Configurations

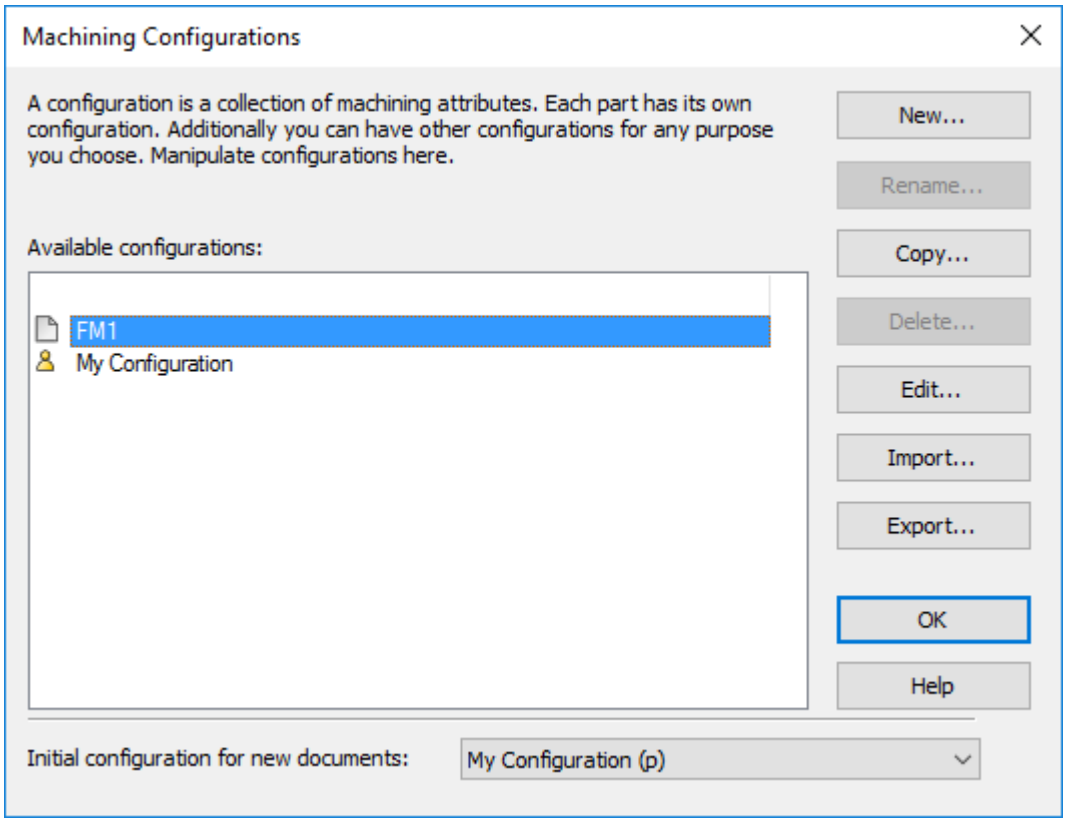
Introduction

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

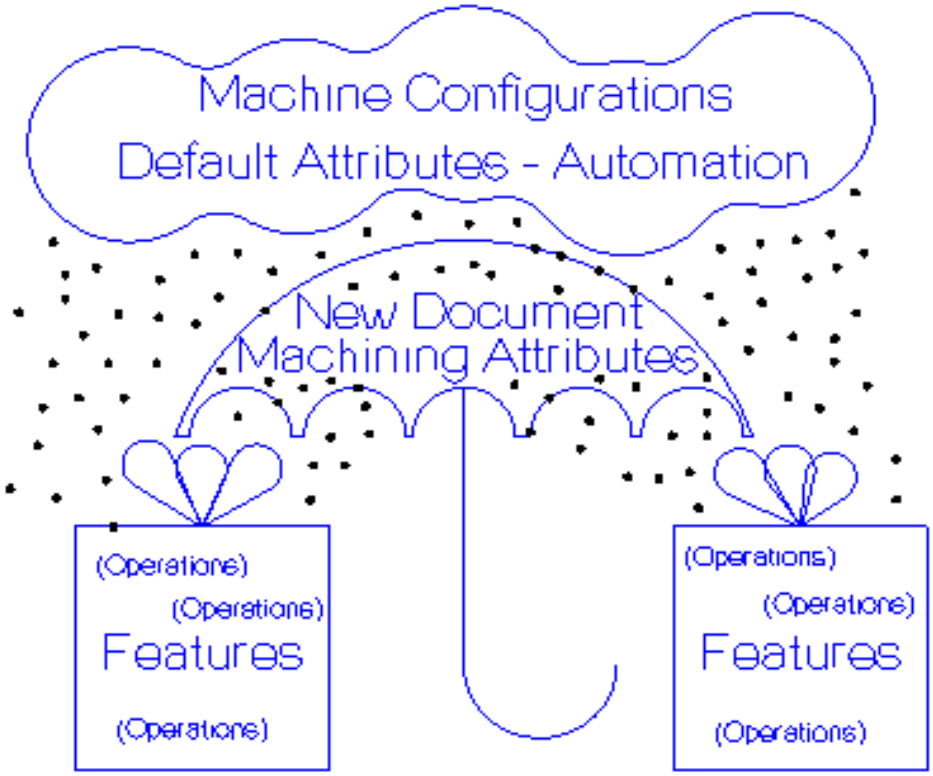
- The **Machining Configurations** can be accessed from the **Manufacturing** drop down menu.
- Machining Configurations are predetermined default functions that are selected, and used in all features that are created in the document.
- A Machining Configuration contains approximately 100 settings.
- 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

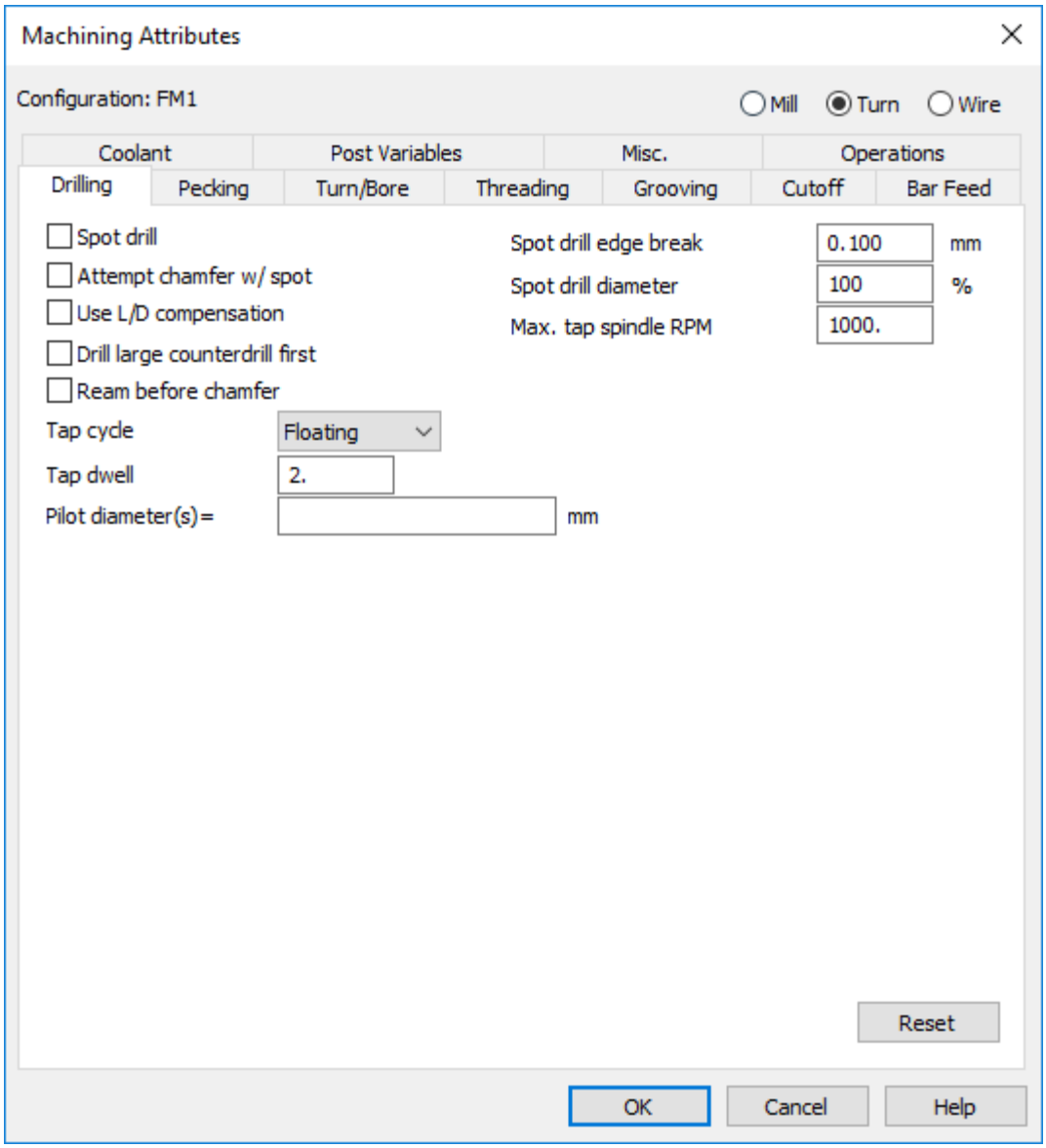
- When a Machining Configuration is selected prior to programming, the attributes, so to speak, rain down into the new document just opened. These are used as the default settings. They are divorced from the Machining Configurations and become Machining Attributes, assigned to, and a permanent part of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

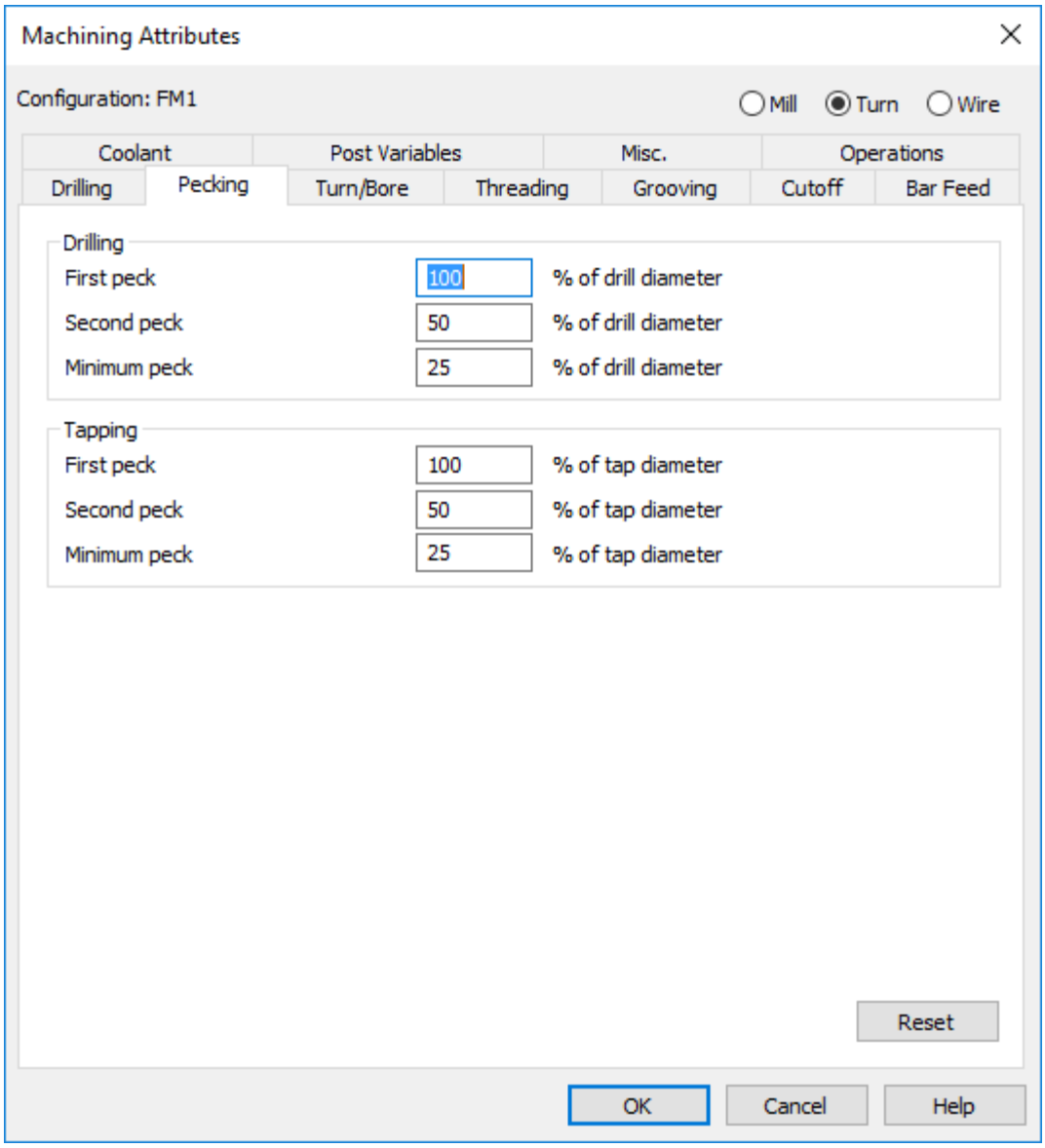
Pilot diameter(s): - This enables and sets a list of drill sizes used to drill pilot holes. Enter a comma-separated list of drill diameters. For example, entering **12, 25, 40** in metric, causes holes to be pilot drilled with the 12mm drill for final hole sizes up to 25mm. A hole in excess of 40mm is 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

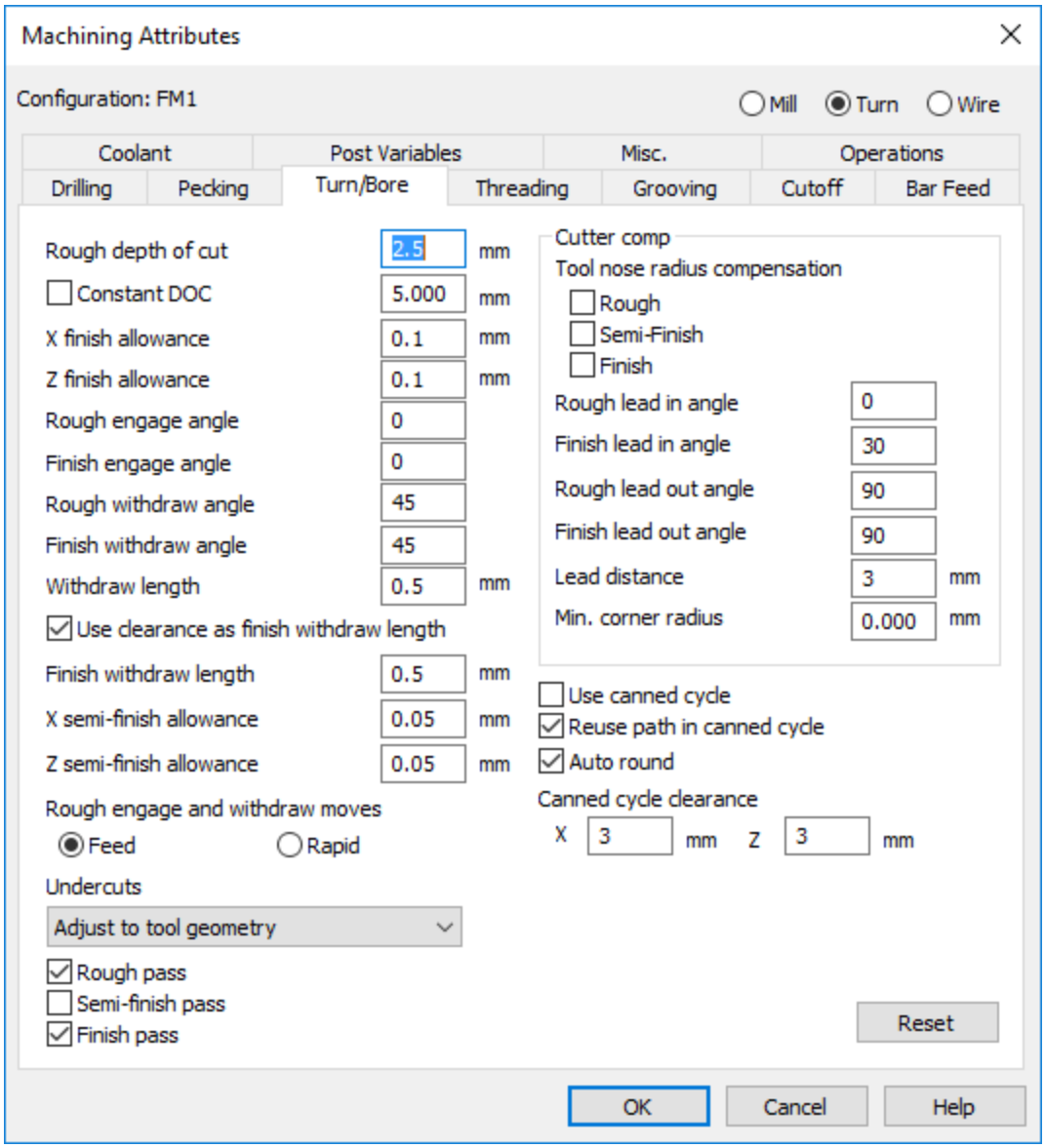
Pecking applies to Deep Hole, Chip Break, and Tap operations. FeatureCAM supports four styles of pecking. These styles are listed in the post processor. Three different attributes control the pecking and they are used differently depending on the style of pecking. FeatureCAM checks the pecking type in 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.

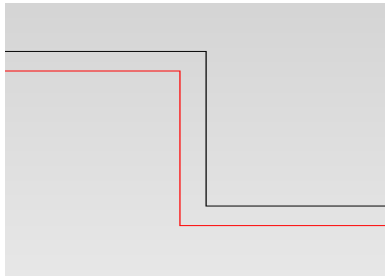
Use Clearance as finish with draw length: - When selected, the **Clearance** value is used as the approach and withdraw length for finishing moves. When deselected, the **Withdraw length** value is used.

Auto round

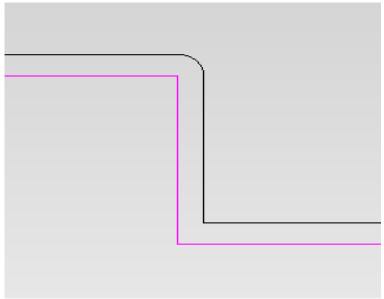
This turning attribute applies to both rough and finish passes. When **Auto round** is enabled, **FeatureCAM** automatically inserts arc moves to connect two non-tangent elements. The effects are:

- 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.

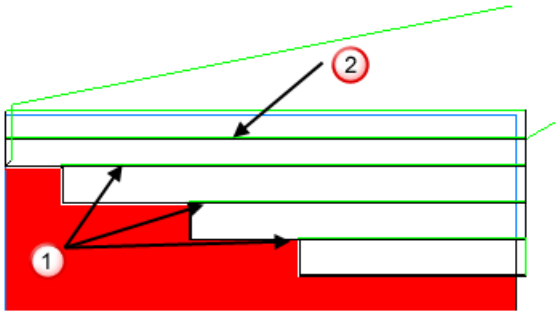
Canned cycle clearance X and Z: - These attributes control the tool location before the start of a turning canned cycle. The tool location is obtained by applying the X and Z clearances to the start point of the curve.

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**.

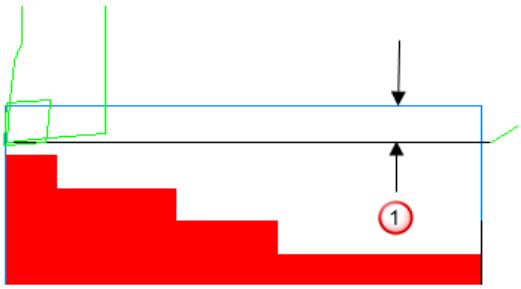
In the example below, the bottom three cutter paths are aligned with the steps. The top region is divided into two equal regions with shallow depths of cut.



1 Aligned with Steps

2 Shallow Cut

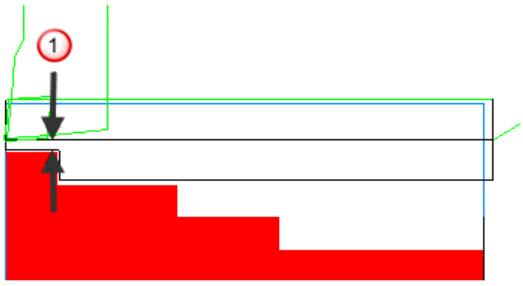
If Constant DOC is selected, each pass is performed at a consistent depth of cut. If required, the pass also climbs and cleans up previous steps. In the figure below, the first cut is performed straight across at the specified Depth of cut.



1 Depth of cut

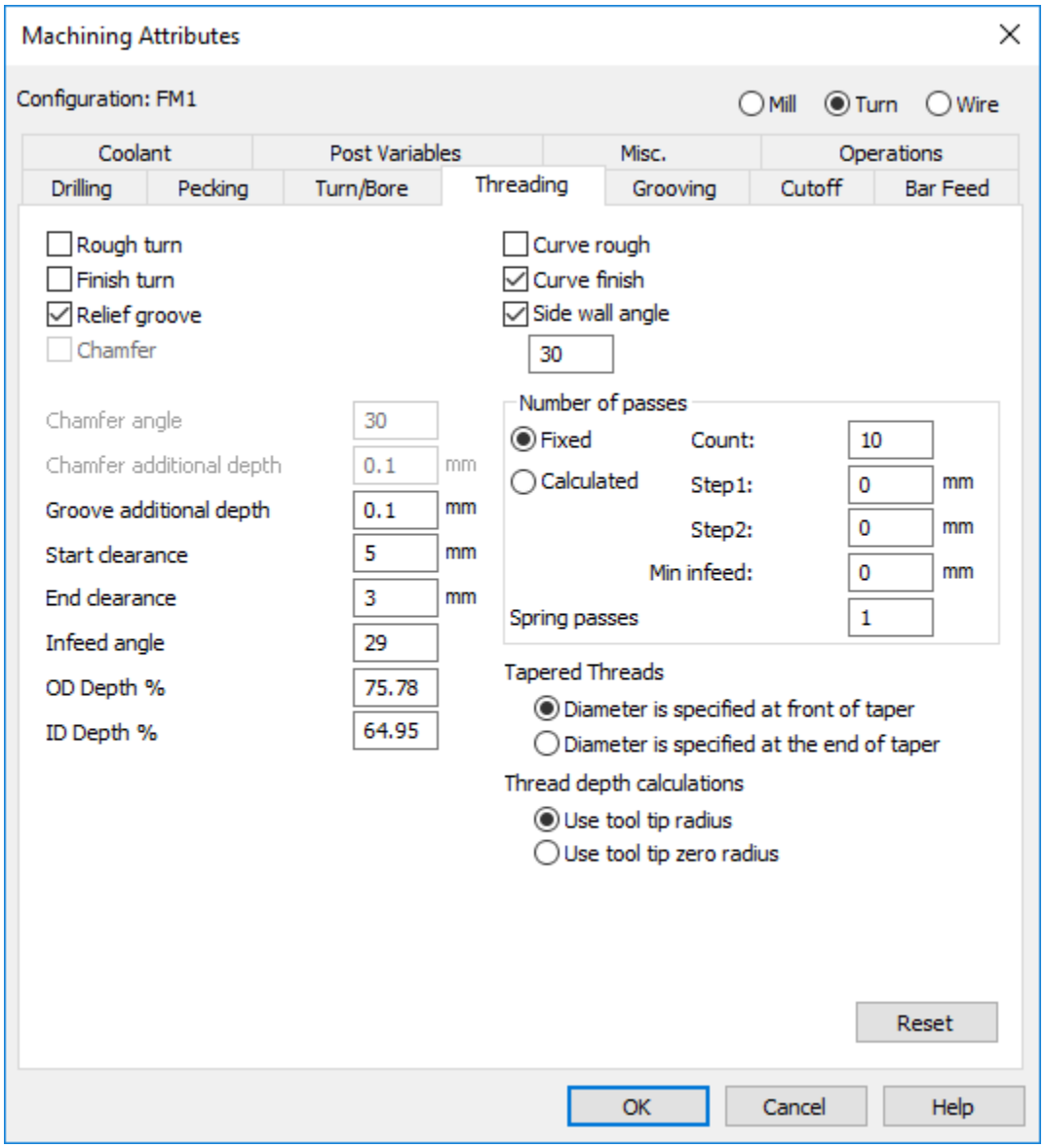
In the next pass, the tool cuts across at the specified depth of cut and then the tool climbs up the wall and the previous step is cut to with the finish allowance.

© Copyright 2016 Autodesk, Inc. All rights reserved



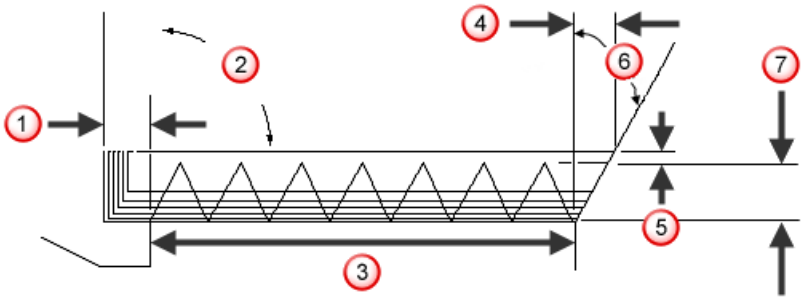
1 Clean up of previous steps

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

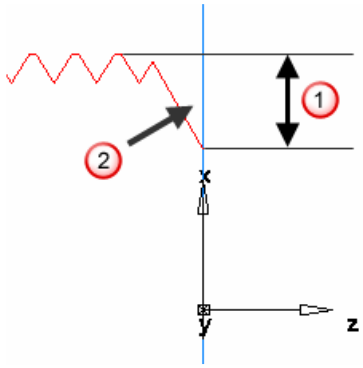
5 Clearance

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.

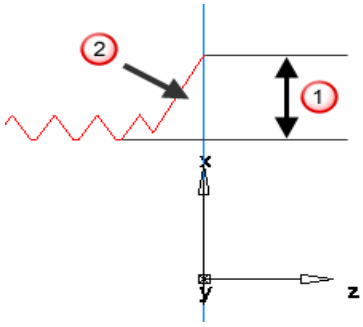
OD Thread Feature



1 (Chamfer addi.) **Depth**

2 (Chamfer) **Angle**

ID Thread feature



© Copyright 2016 Autodesk, Inc. All rights reserved

1 (Chamfer addi.) **Depth**

2 (Chamfer) **Angle**

(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.

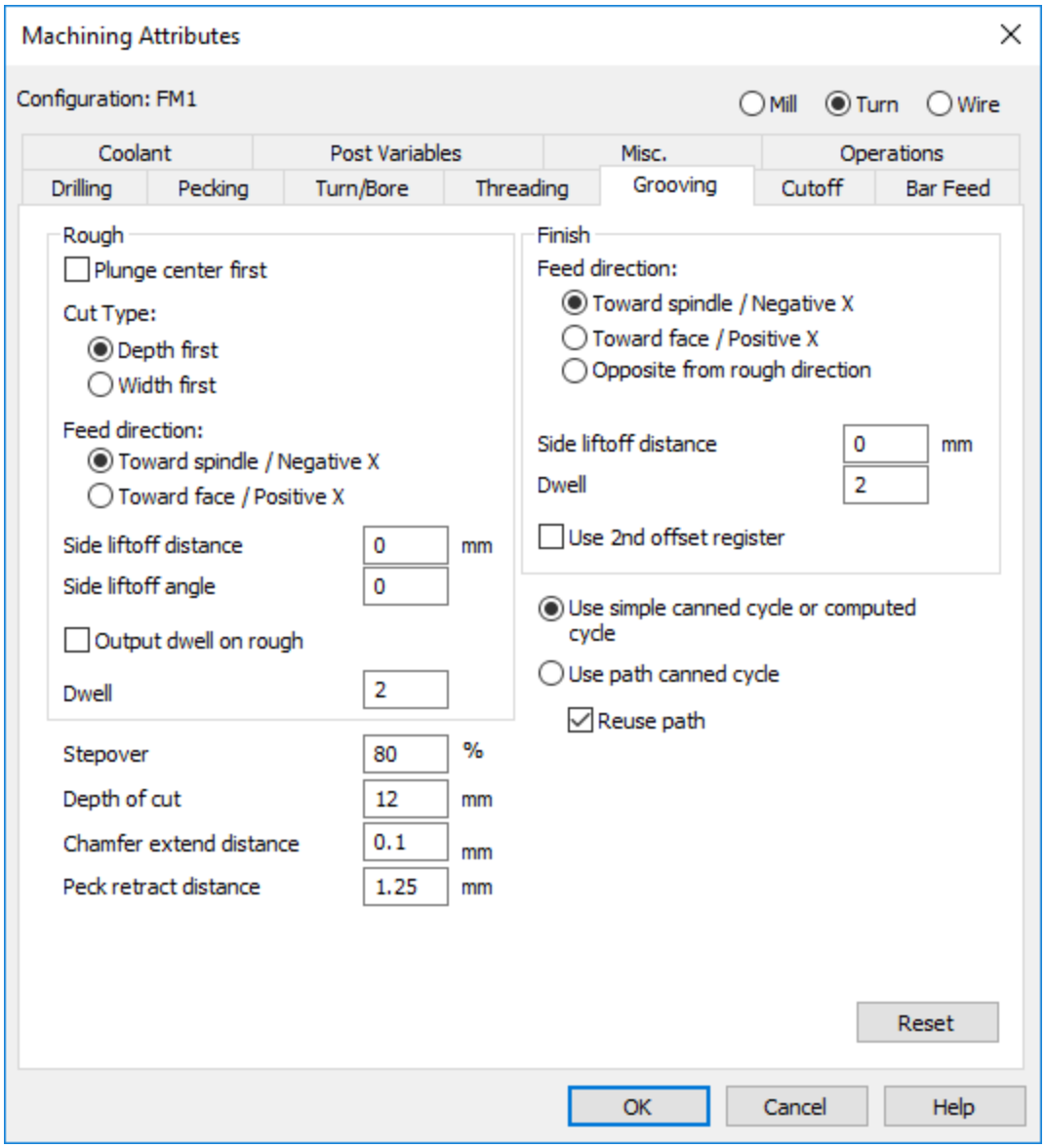
Spring passes: - A *spring pass* is a duplicate of the final threading pass. Enter the number of **spring passes** that you want.

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

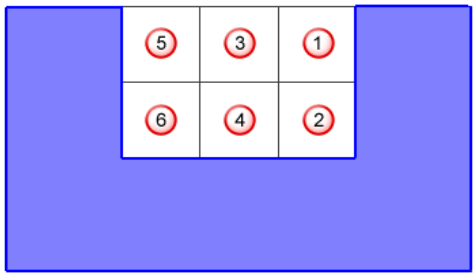
Grooving tab (Turn)



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

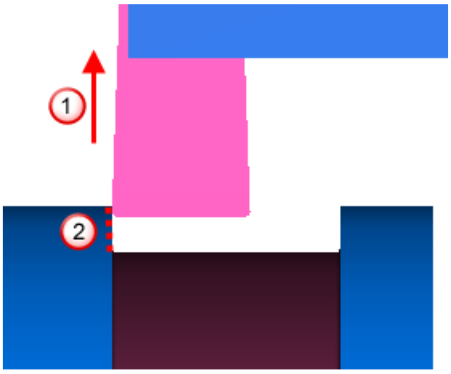
If **Depth First** and **Plunge Center First** are both enabled, then the groove is cut in the following order: 3, 4, 1, 2, 5, 6. If the groove were wider, the subsequent cuts would alternate from one side of the groove until the other until the entire groove is cut.

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**.

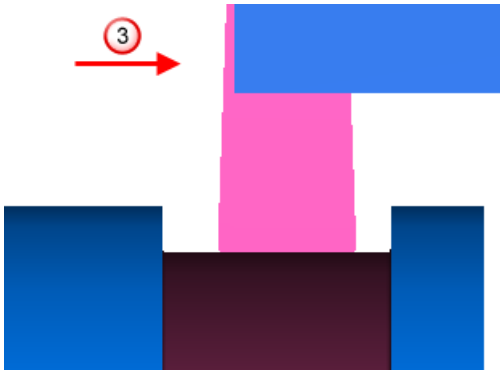
Side lift off dist: - Enter the distance to move the tool after a plunge cut, in the direction opposite to the cutting direction. This increases the tool's life and leaves a better finish on the part. This applies to a Groove feature. See also **Side liftoff angle**.

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.

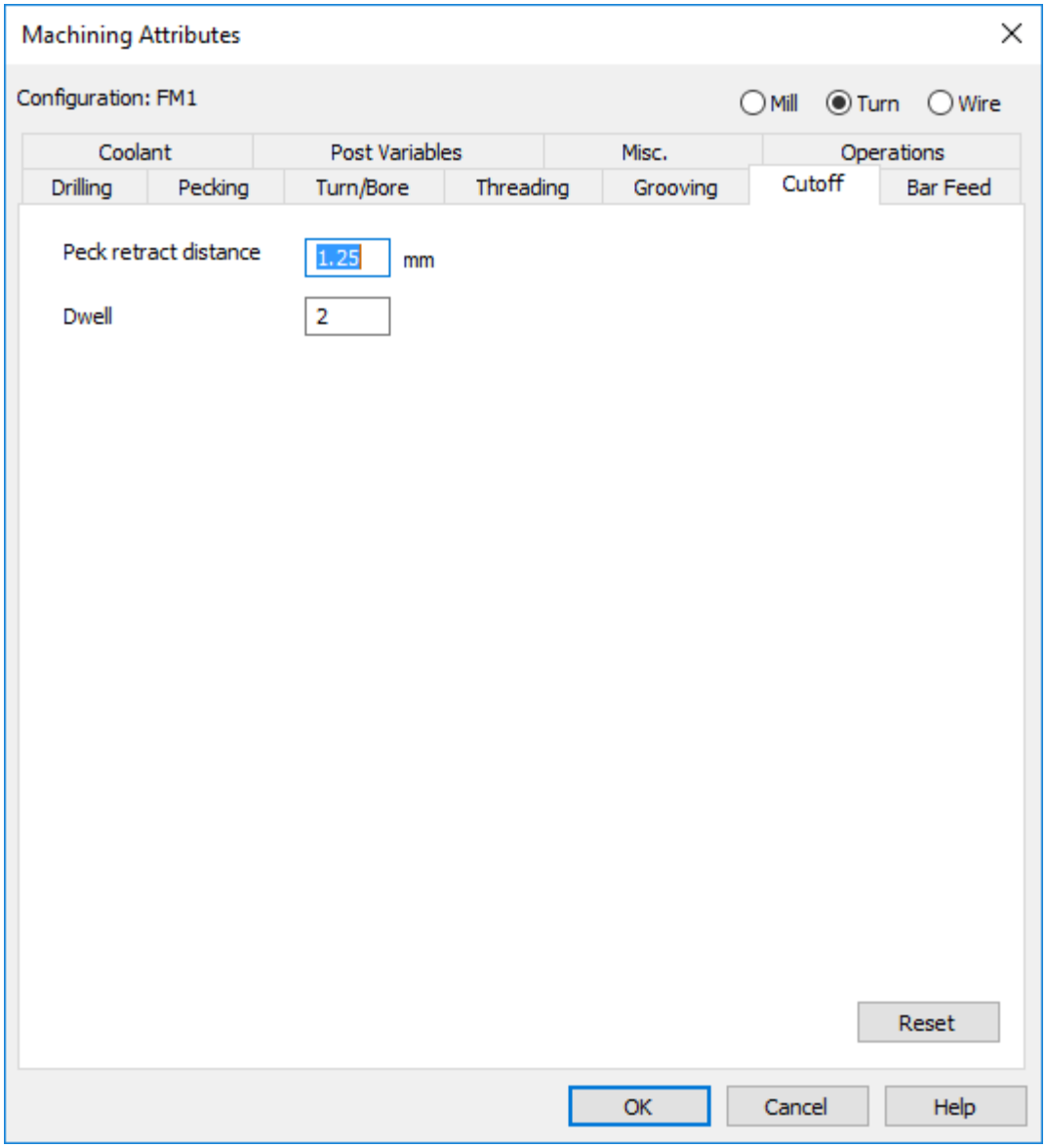
Peck retract dist.: - For Cutoff and Groove features, **Peck retract dist.** is the distance the tool retracts between plunges.

© 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.

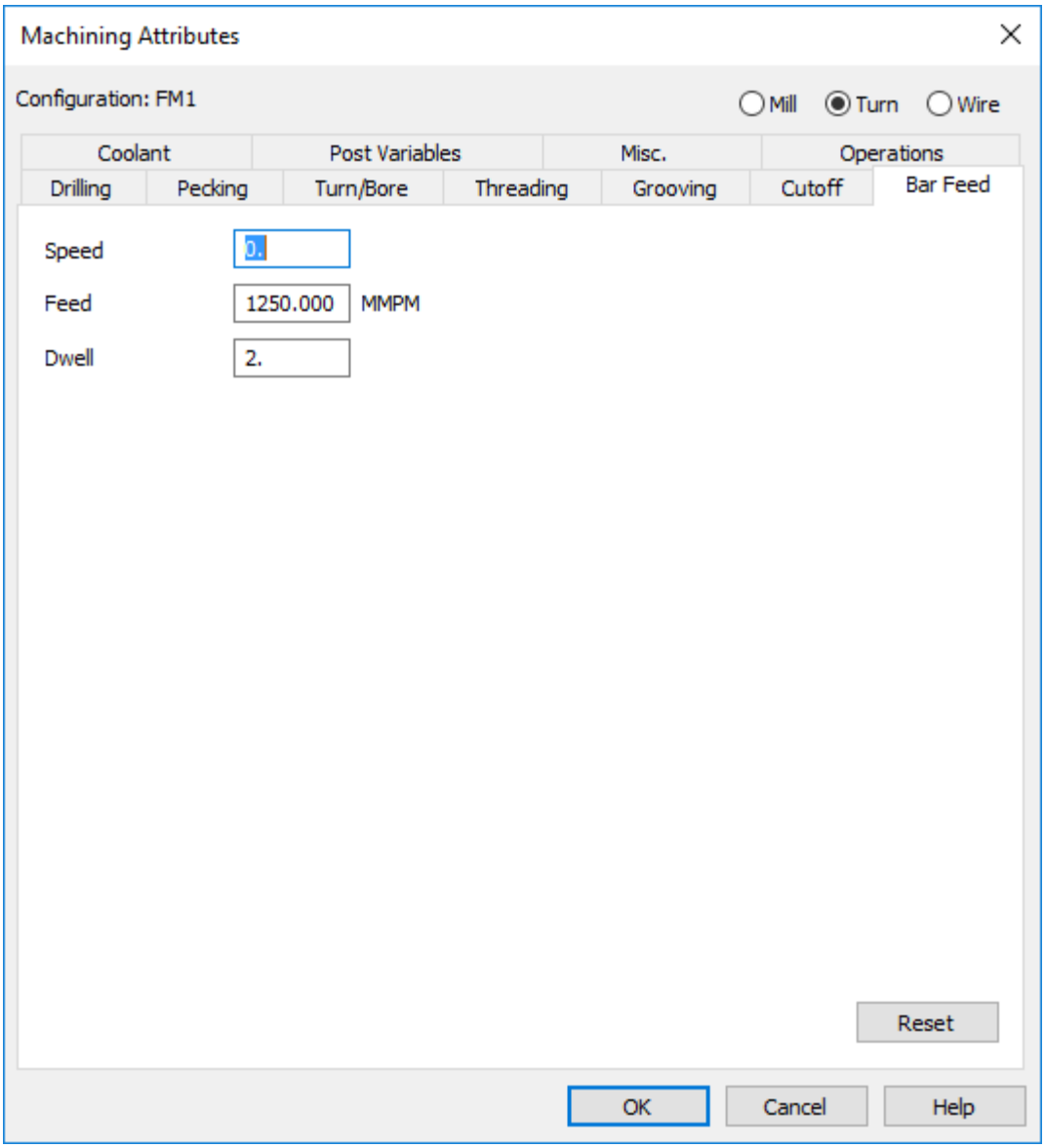
Dwell: - The number of seconds the tool dwells after plunging during a groove roughing pass. It also applies to the roughing of the Cutoff chamfer.



*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

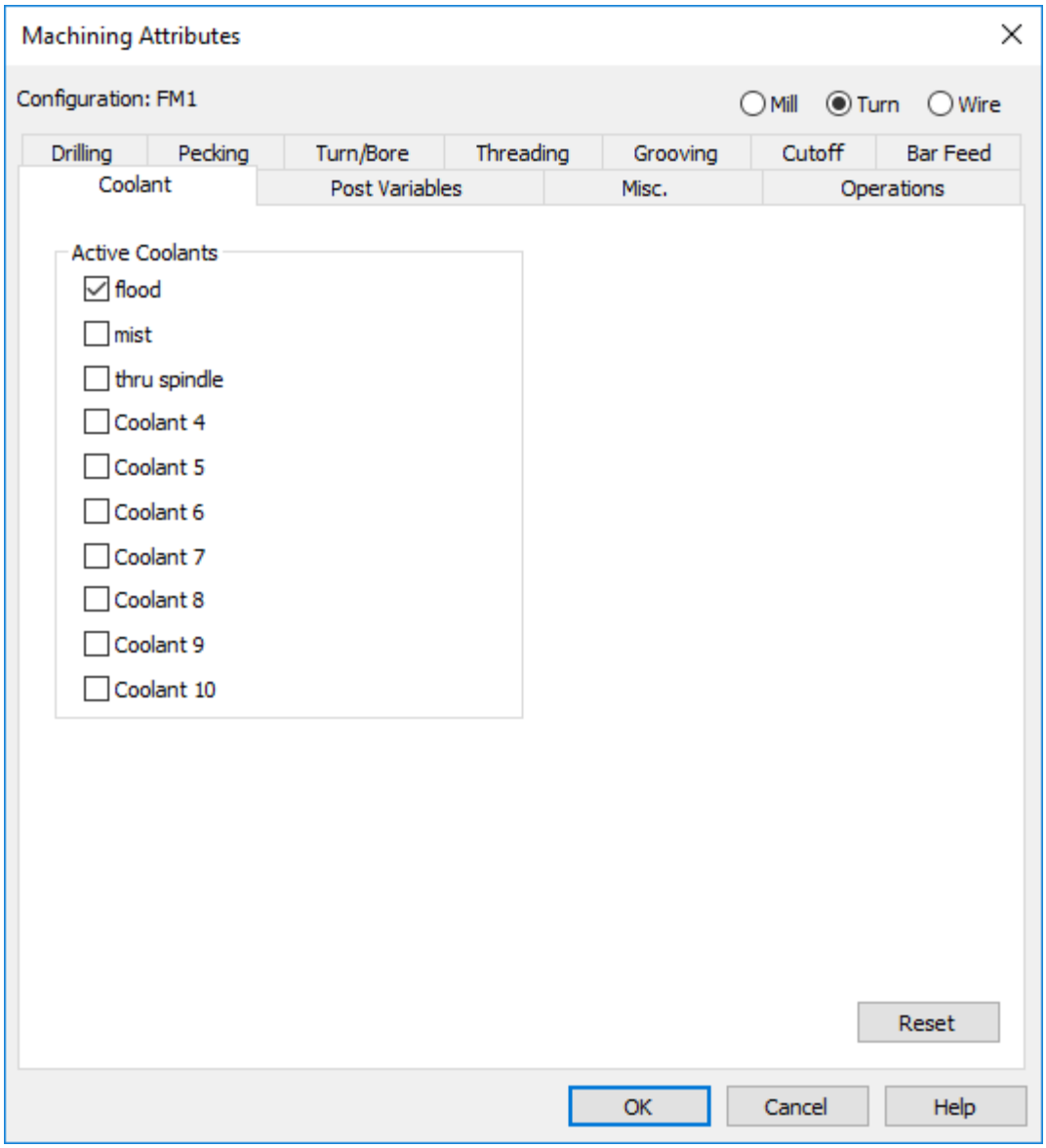
Bar Feed tab (Turn)



***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

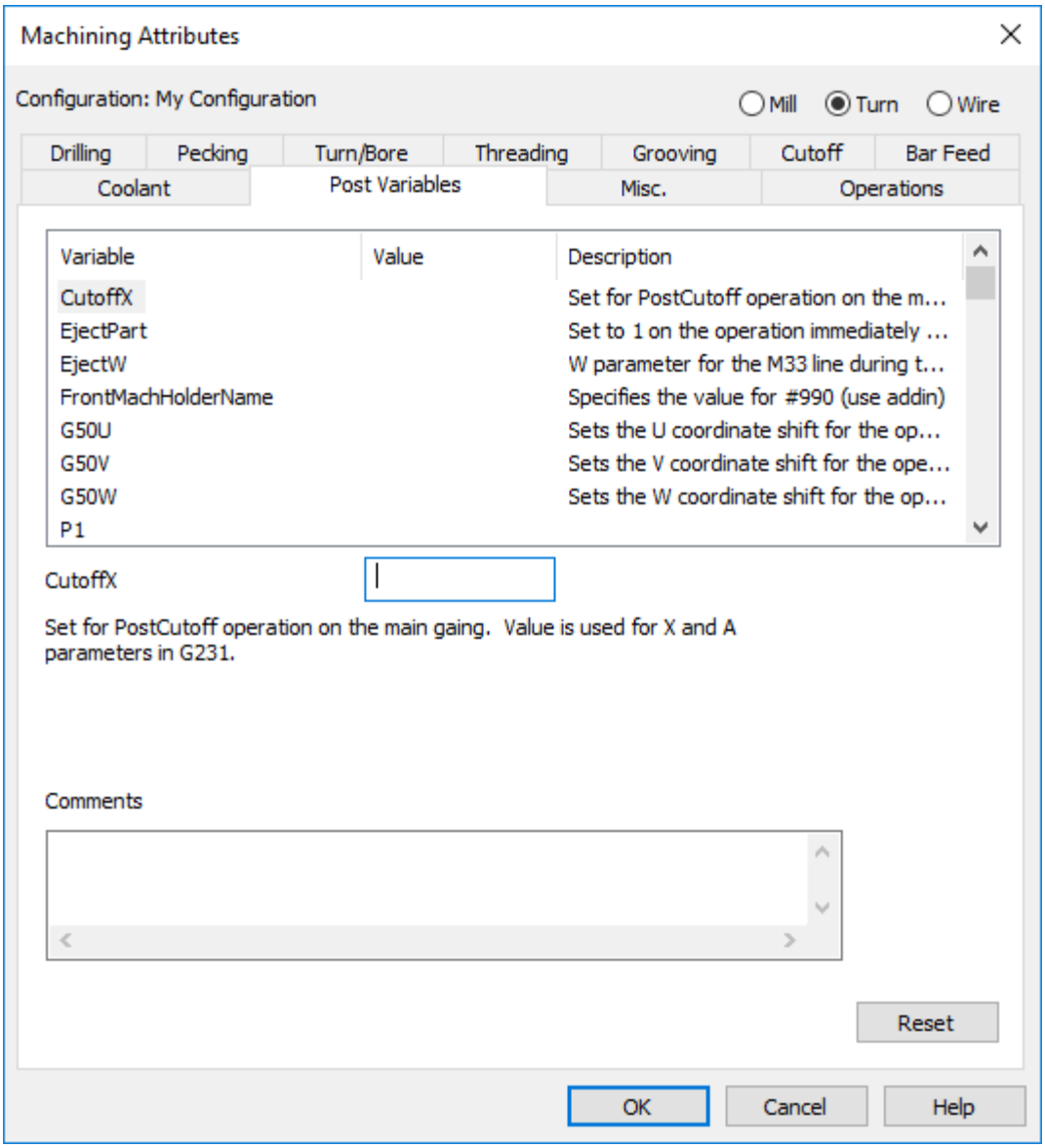
Coolant



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

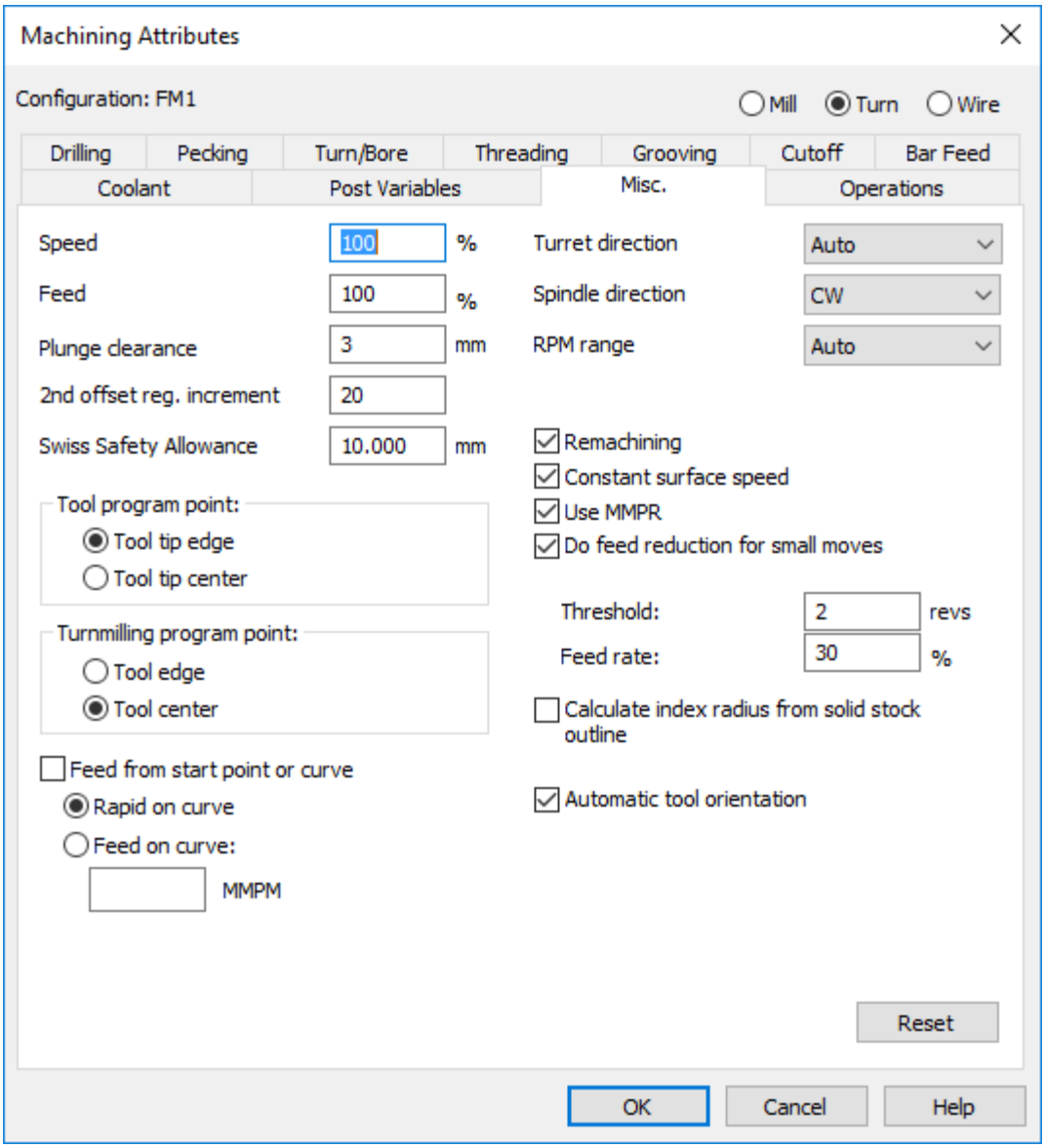
© Copyright 2016 Autodesk, Inc. All rights reserved

Post Variables



© 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

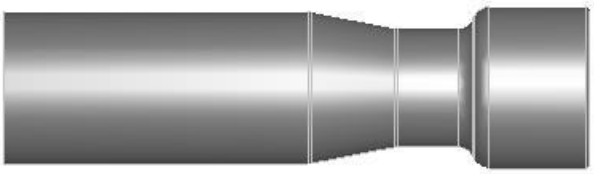
Feed from start point or curve — Select this option to use a feed move from the **Start point** or the end of the **Start curve**, to the beginning of the toolpath. If you are using a start curve, you have two further options:

- **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.

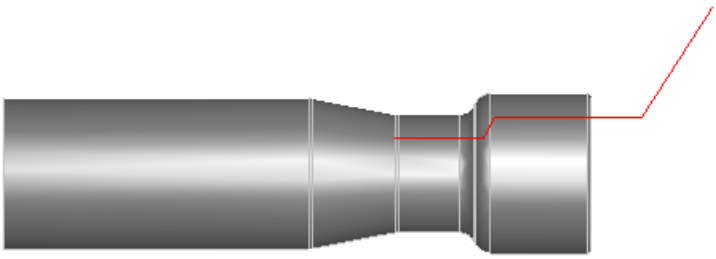
This example turned part has a Turn, Hole, and Groove feature:



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.

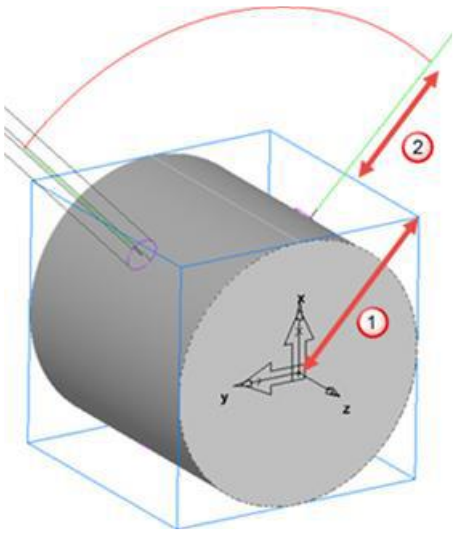
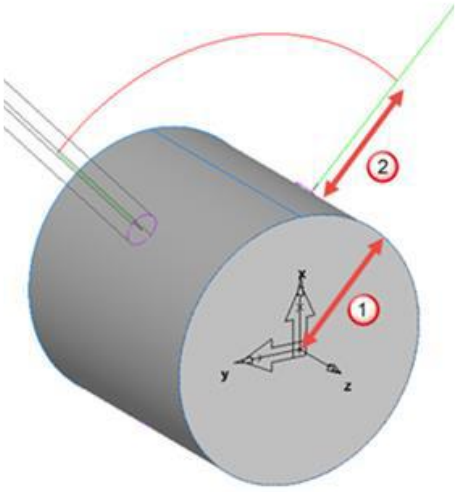
Use IPR/MMPR: - The default feed units are **IPM** (inches per minute) or **MMPM** (mm per minute). Select this option to use **IPR** (inches per revolution) or **MMPR** (mm per revolution).

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:

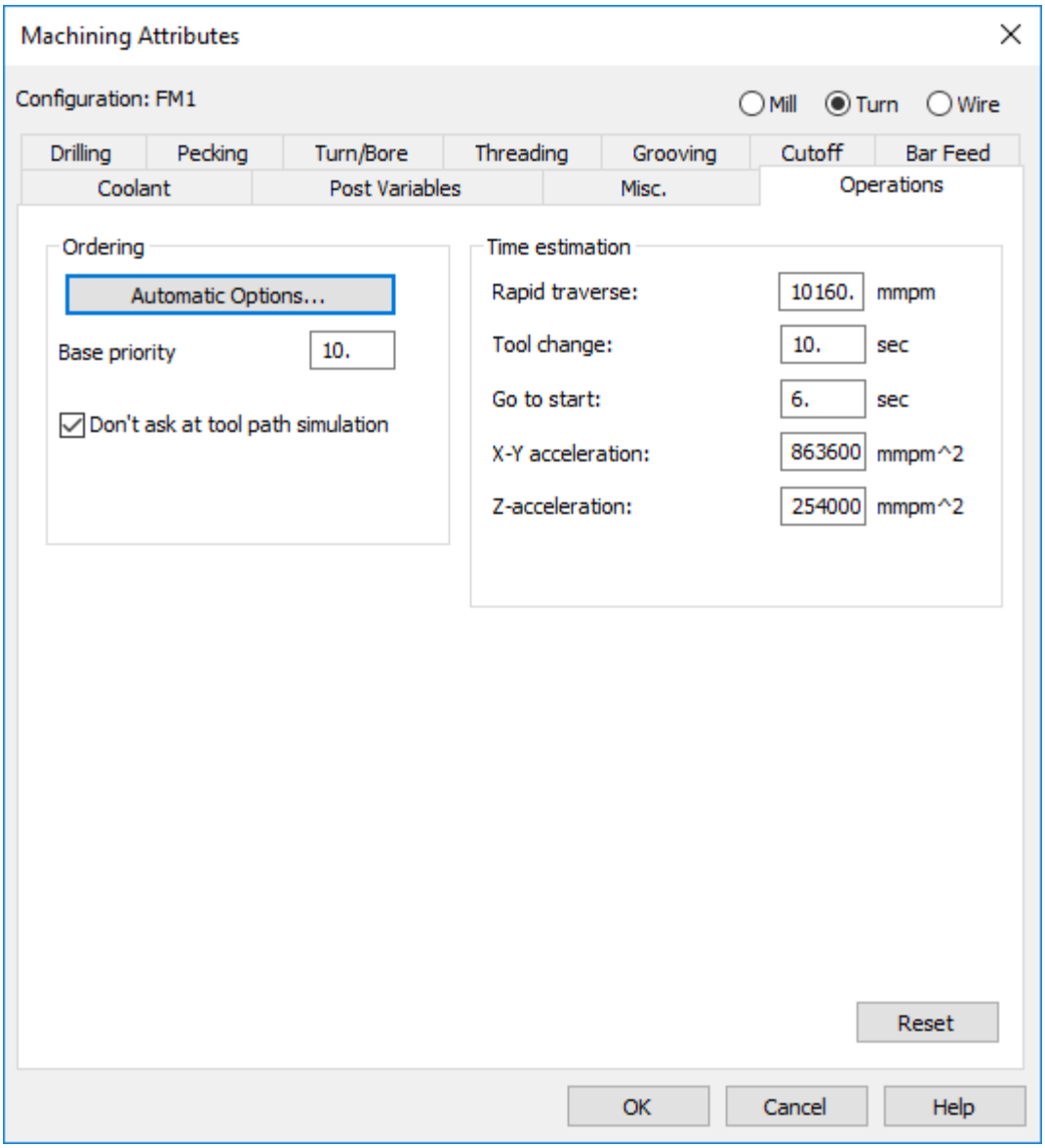
Select Manufacturing > Post Process from the menu.

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.

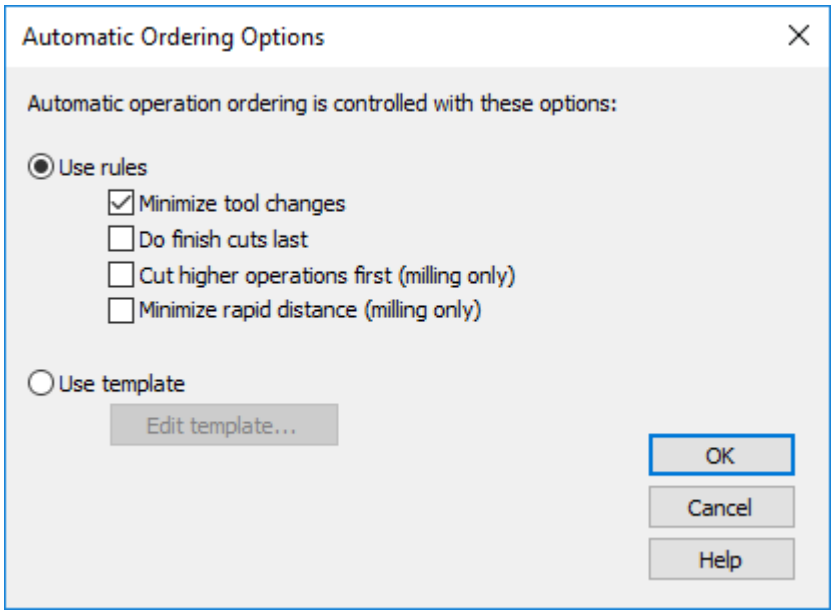
X-Y acceleration: - This is used in the formula to calculate the time for a particular tool move.

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:

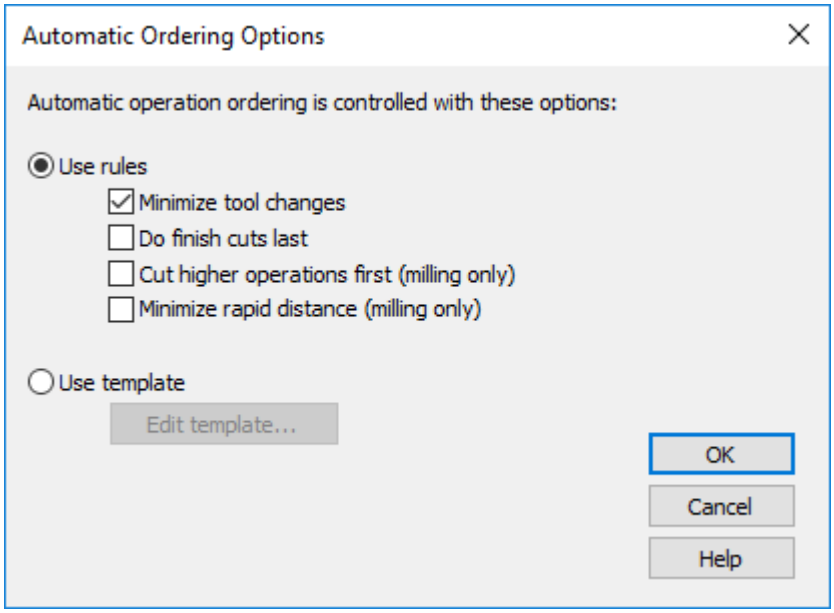
The operations are sorted by their top Z coordinate.

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

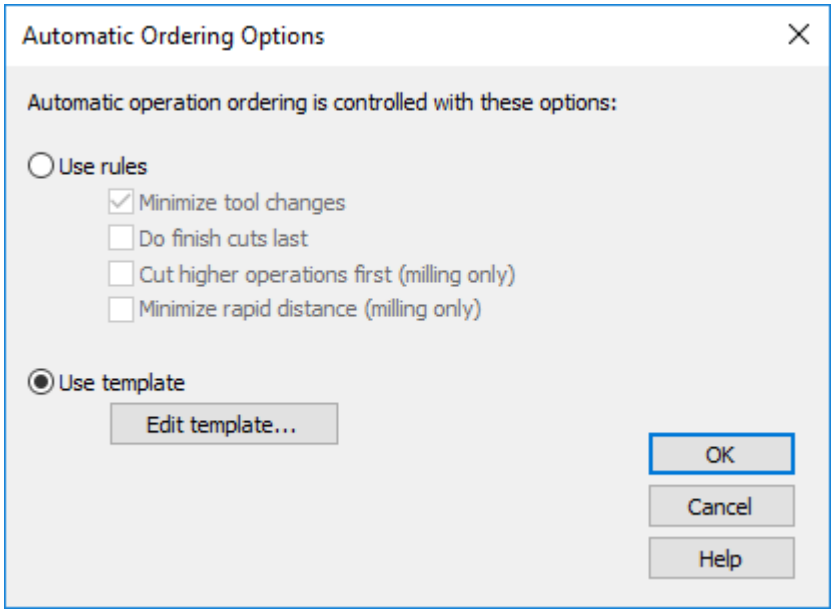
© Copyright 2016 Autodesk, Inc. All rights reserved

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

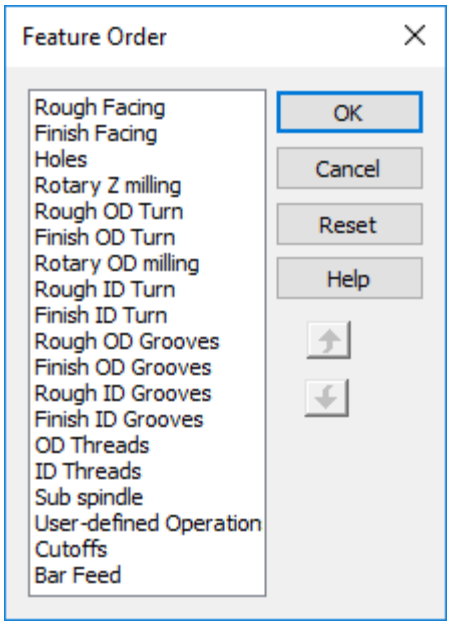
Use Template



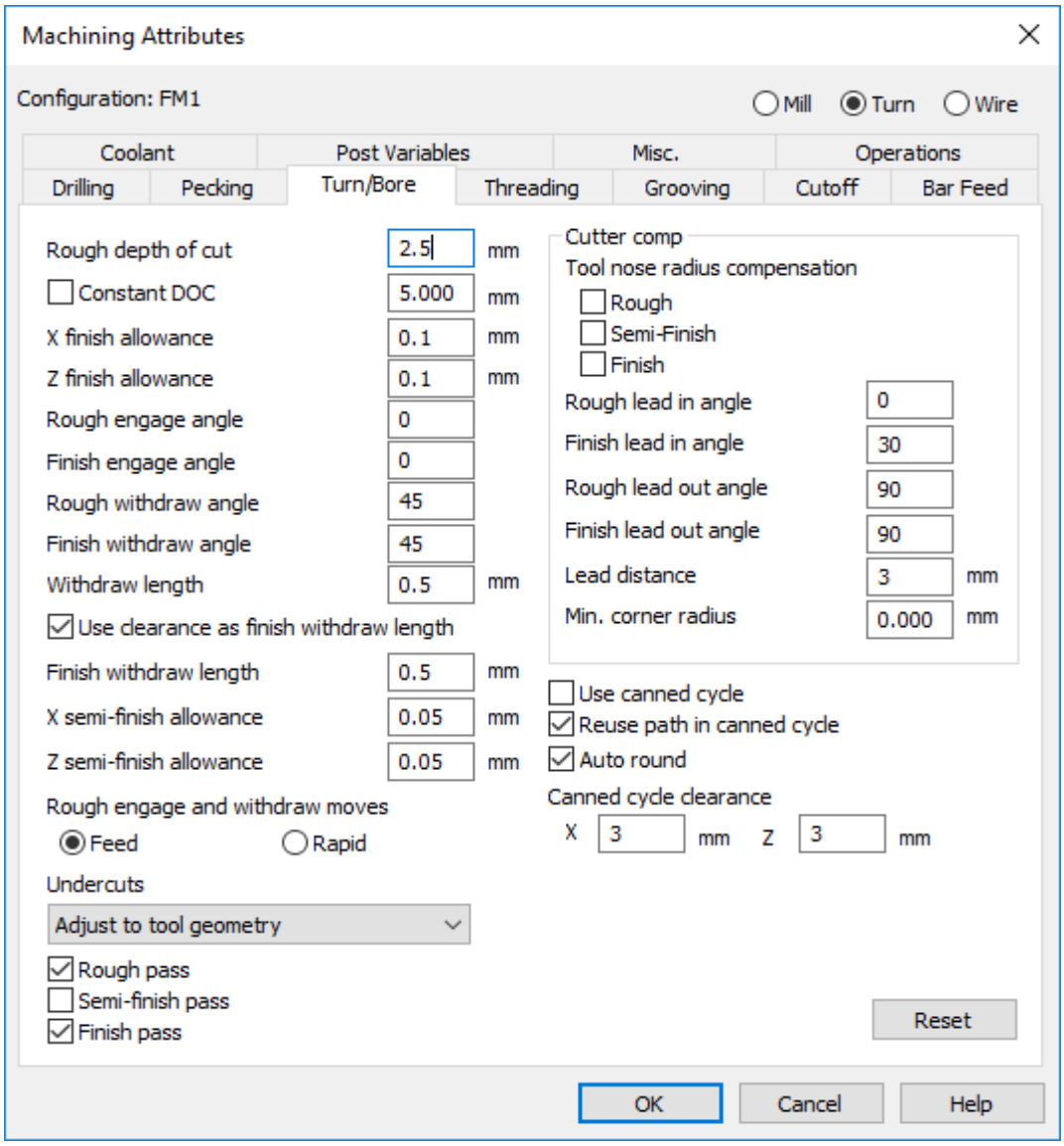
*This attribute is applicable only to turning setups. If you select **Use template** then the order of operations are determined by the outline of operations listed in the Feature Order dialog. Click **Edit template** to open the Feature Order dialog. See turn operation order.*



© Copyright 2016 Autodesk, Inc. All rights reserved

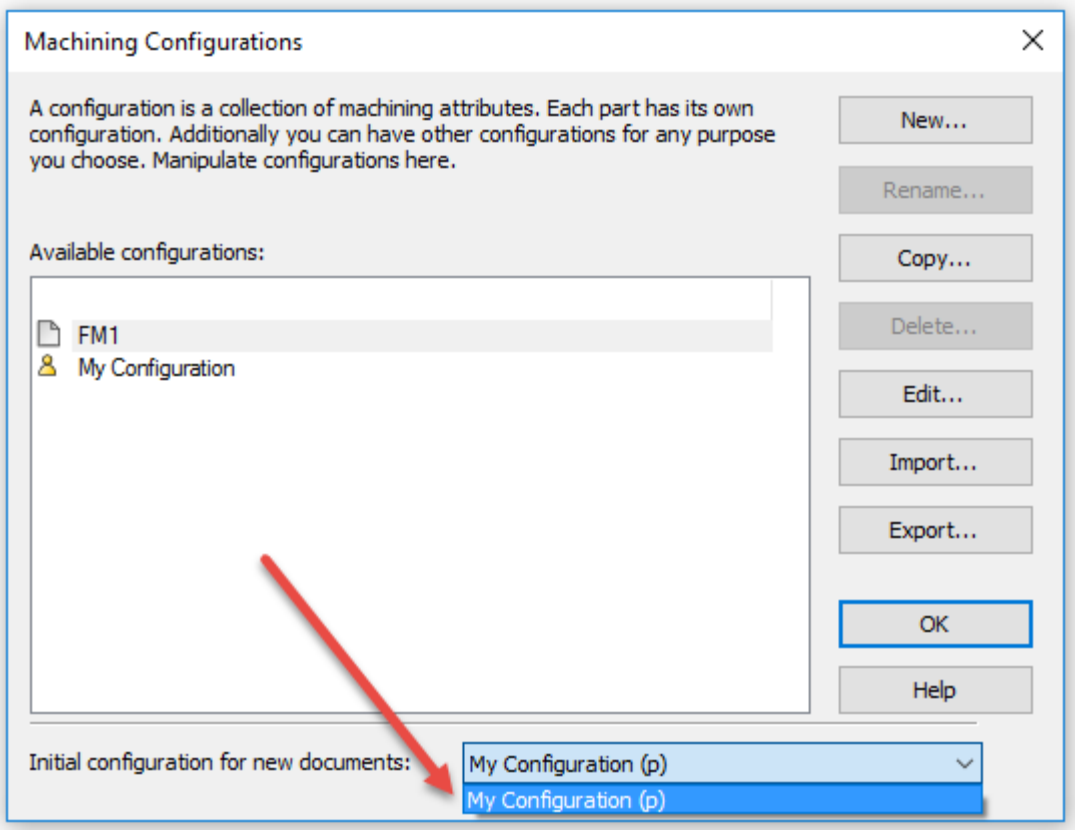


What are Machining Attributes?



© 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.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

Swiss Turn - 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
Swiss - Bore Tool Properties example. 20
Holder Information 20
Boring Orientation 22
Using an insert drill to drill and bore in the same program 23
Swiss - Bore Programming Point 24
Swiss - Groove tool Properties example 24
Swiss - Cutoff Tool Properties example 25
Swiss - Groove/Cutoff Holder tool Properties example 26
Swiss - Thread Tool Properties example. 27
Tool Post configuration 28

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss Turn - Tool Reference Guide

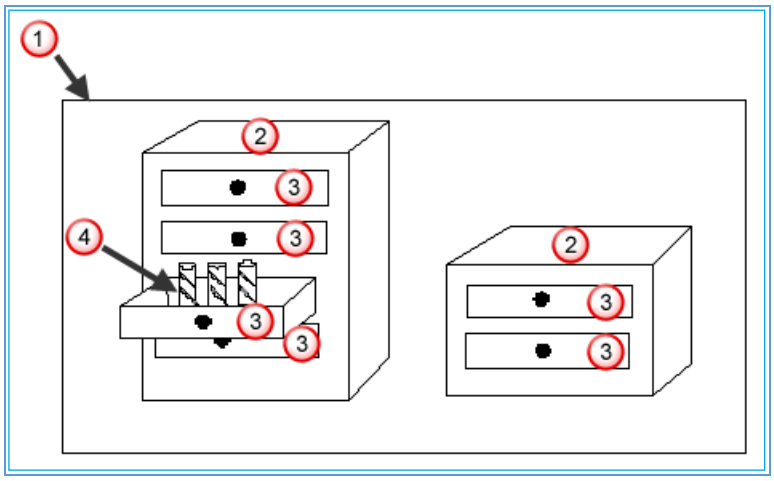
It is possible to create Tool libraries in **FeatureCAM**. These are called Tool **Cribs.** The main reason for creating Tool Cribs is because in any machine shop there is normally a finite amount of tooling available and by creating a custom Tool Crib, **FeatureCAM** will know what tooling is available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

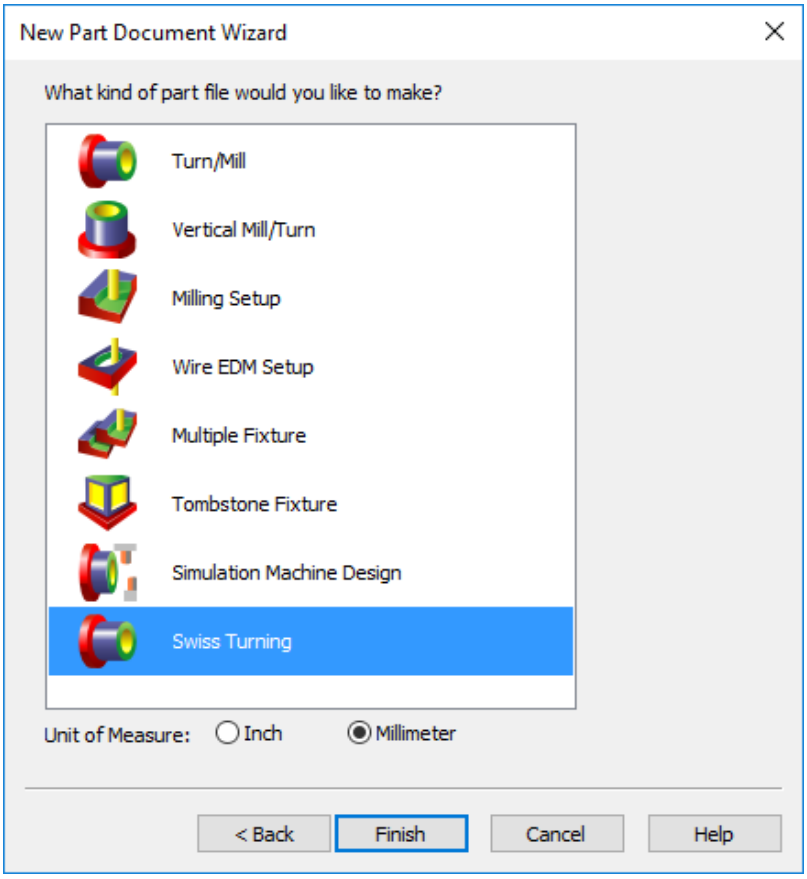
Tooling database

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.

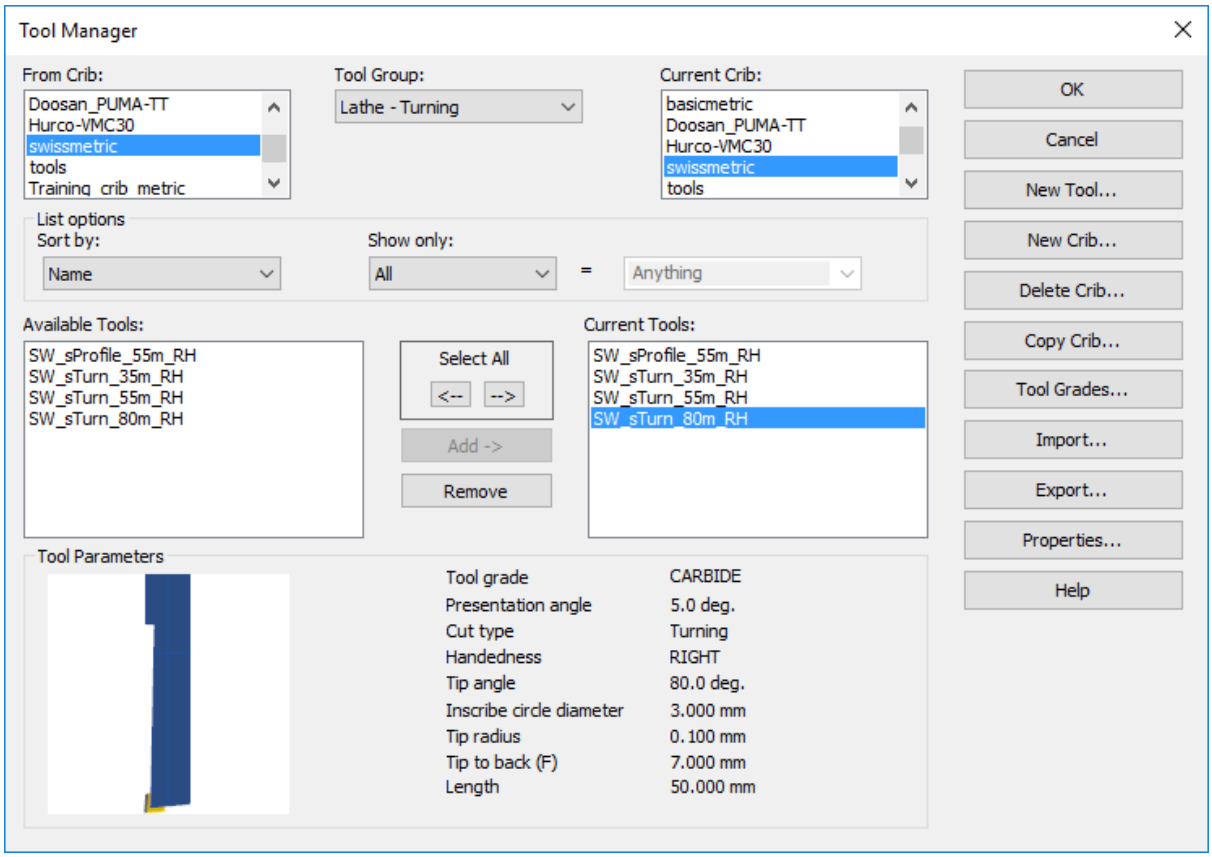
1. **Open** a new **FeatureCAM** part by selecting **File, New.**
2. The following Menu shown on the next page will appear. Select **Turn/Mill** Set-up and click **OK.**



© 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.*

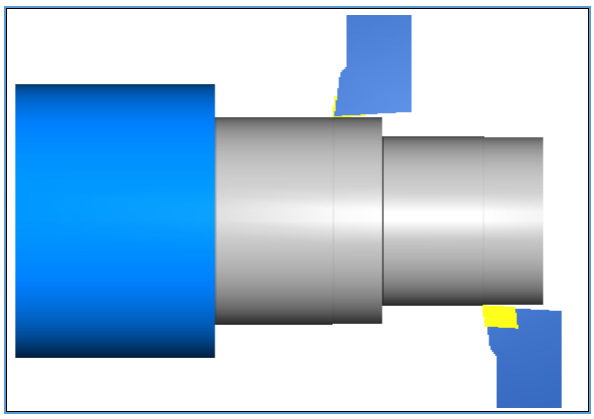
The window is divided into separate areas and at first glance this can look a little intimidating, but by applying some simple rules it becomes much easier to work with.

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.

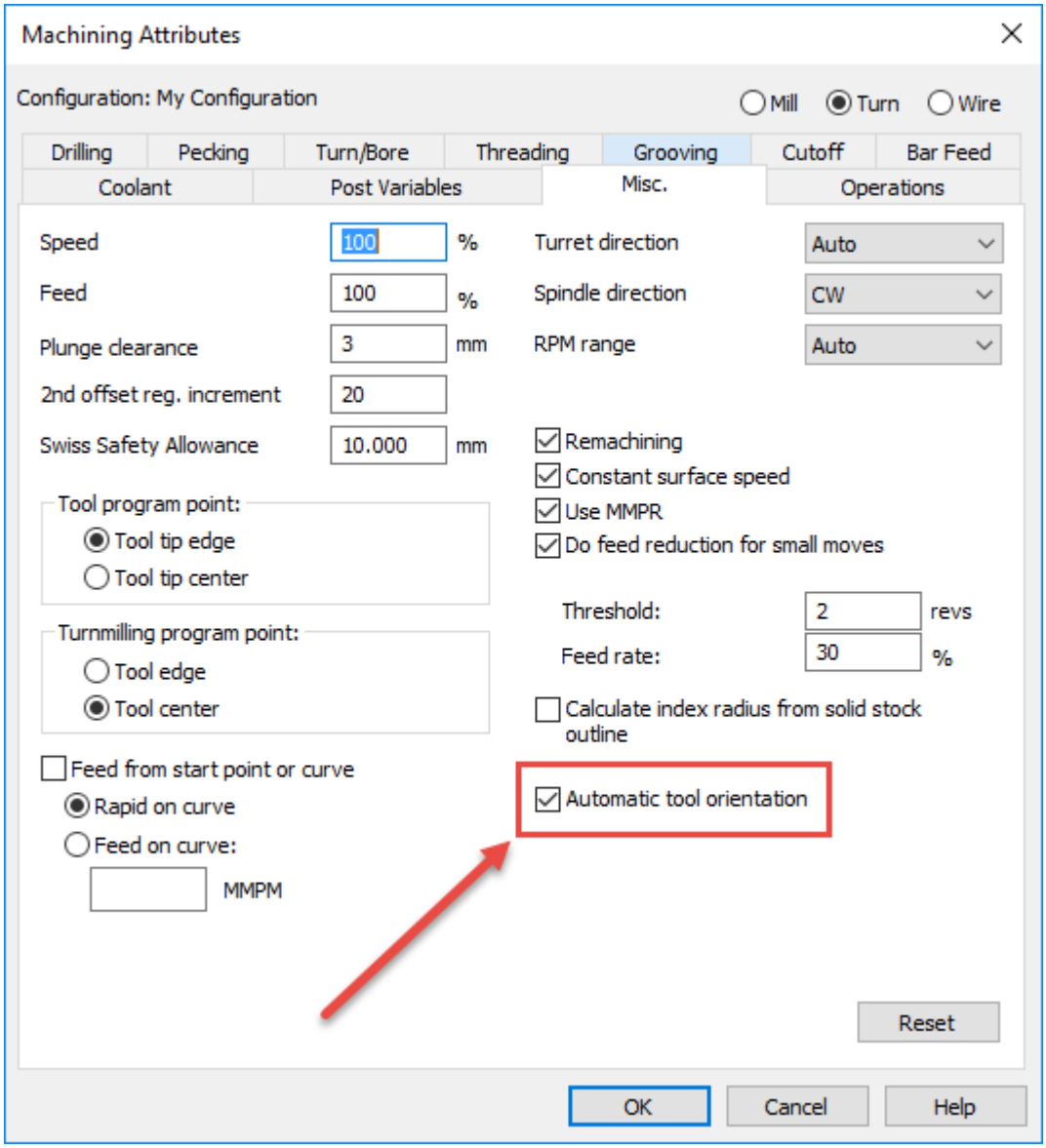
© Copyright 2016 Autodesk, Inc. All rights reserved

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.



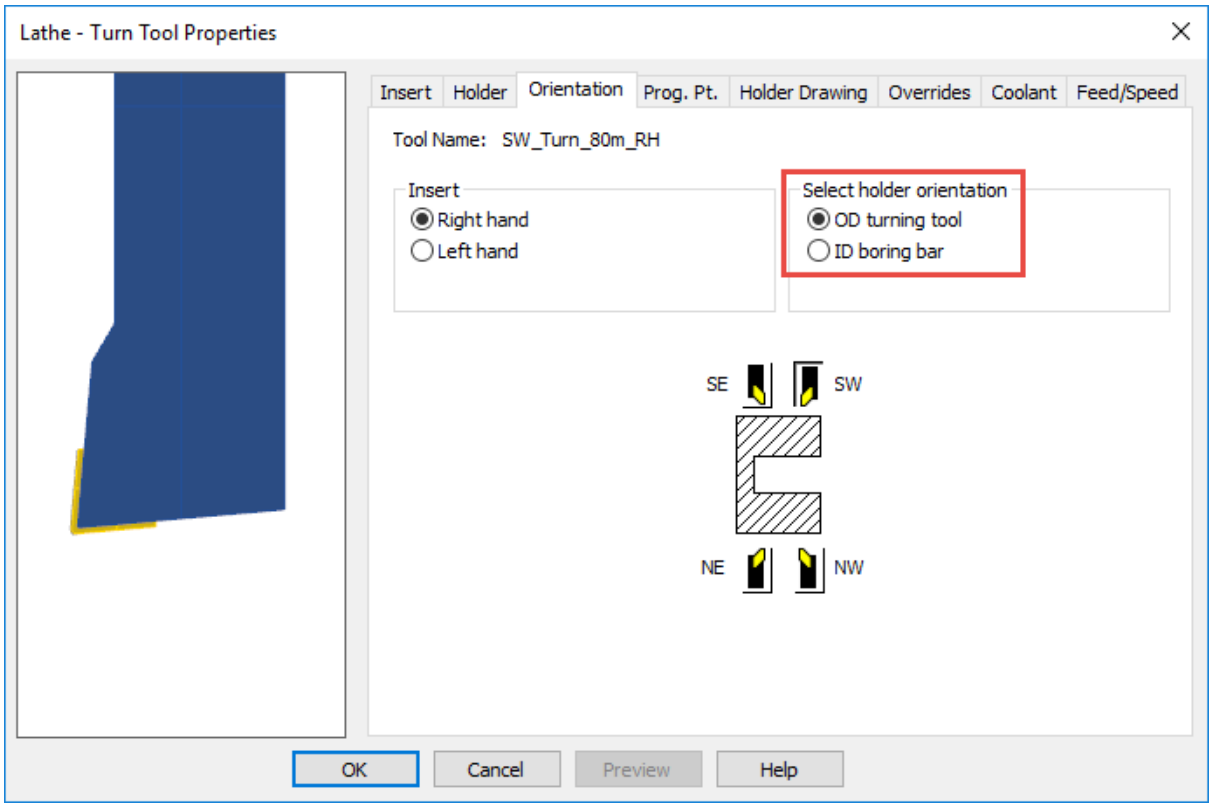
To activate this option please set the following in your configuration file.

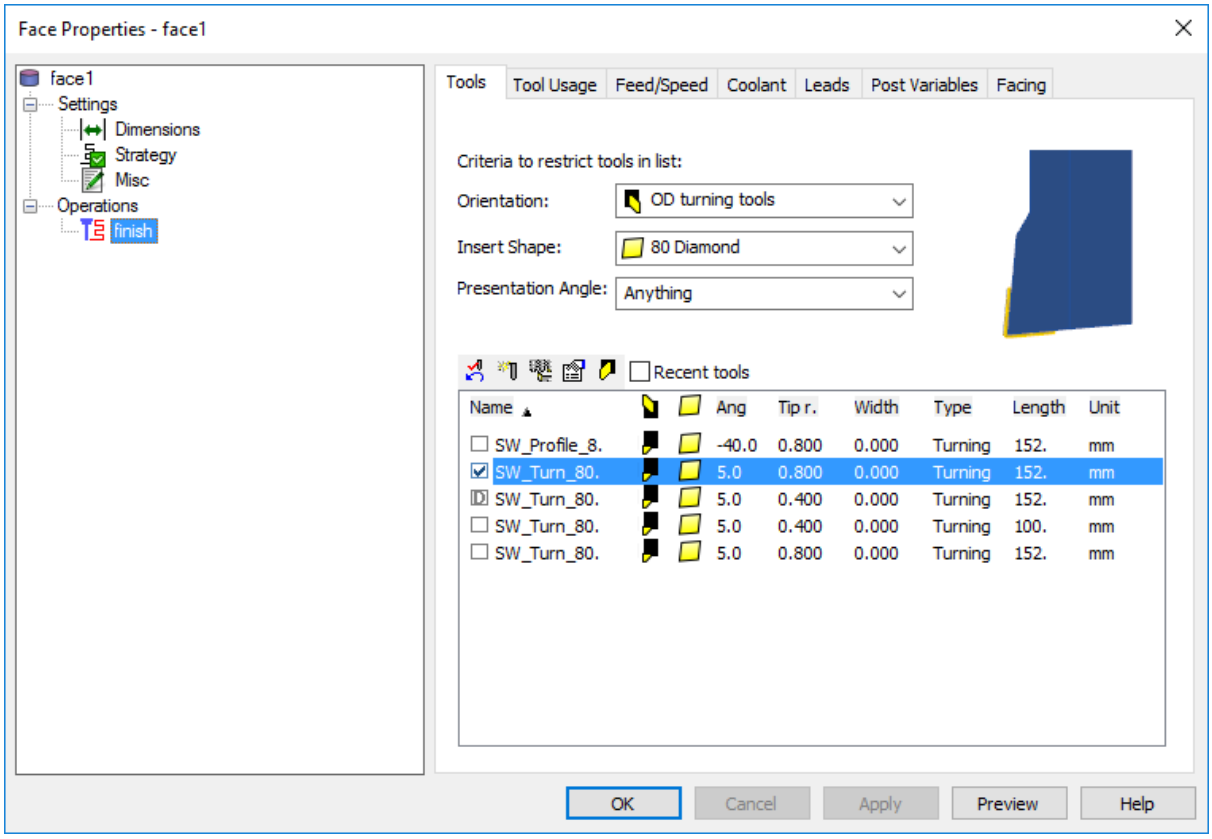


© Copyright 2016 Autodesk, Inc. All rights reserved

The tool orientation is split into two types.

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

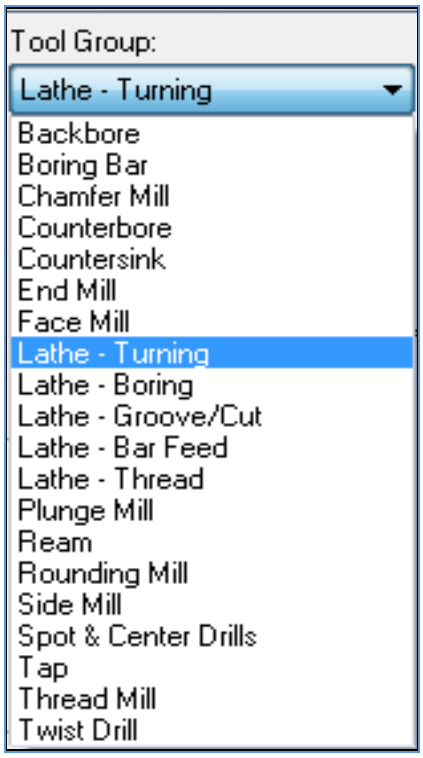




© Copyright 2016 Autodesk, Inc. All rights reserved

Tool Groups.

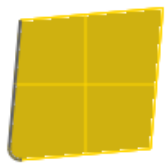
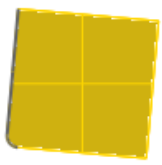
The tools can be grouped together depending upon their type or use.



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**

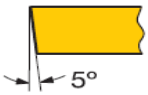
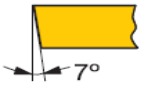
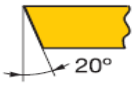
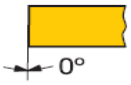
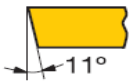
© Copyright 2016 Autodesk, Inc. All rights reserved

Inserts for General Turning

1 Insert shape

C		D	
K		R	
S		T	
V		W	

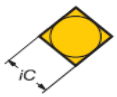
2 Insert clearance angle

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	Tolerance class 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

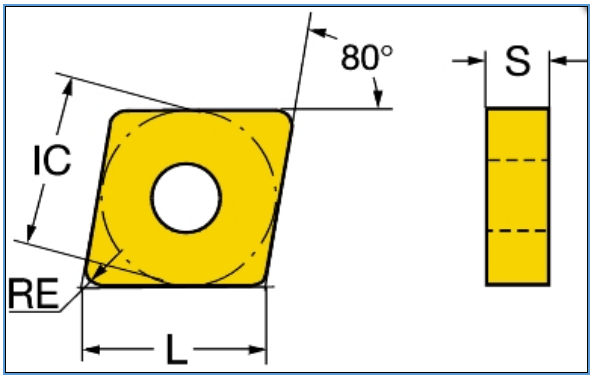
CNMG Insert

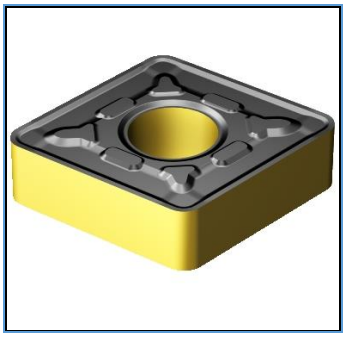
Tip Radius is the radius of the cutting tip of the insert.

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

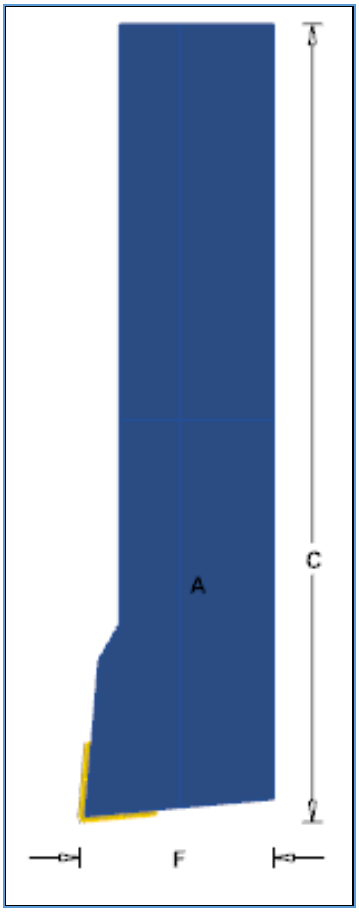




S = Insert Thickness

RE = Corner Radius

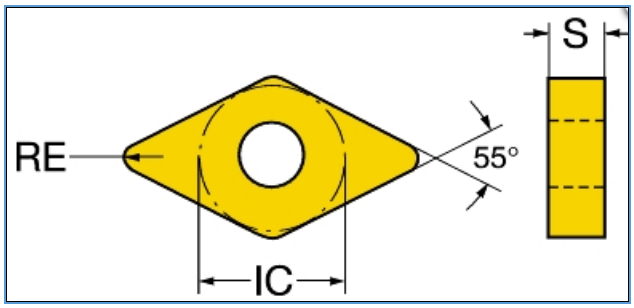
IC = Inscribed Circle Diameter

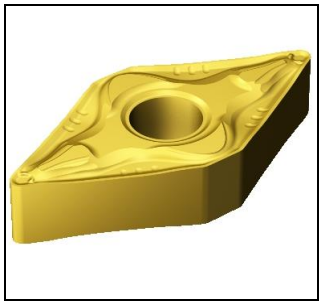


FeatureCAM tool designation = **SW_Turn_80m_RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

DNMG Insert





S = Insert Thickness

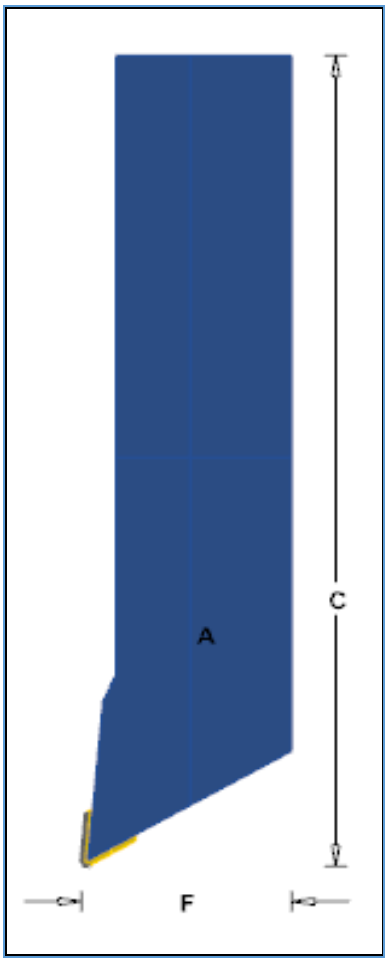
RE = Corner Radius

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.

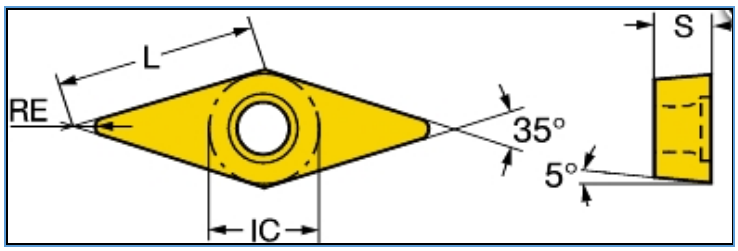
Inscribe Circle Diameter is the diameter of a circle that fits inside the insert shape.

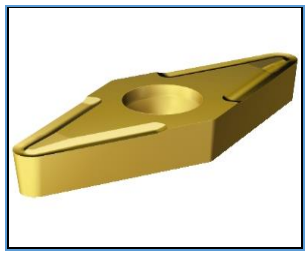


FeatureCAM tool designation = **SW_Turn_55m_RH**

© Copyright 2016 Autodesk, Inc. All rights reserved

VBMT Insert





S = Insert Thickness

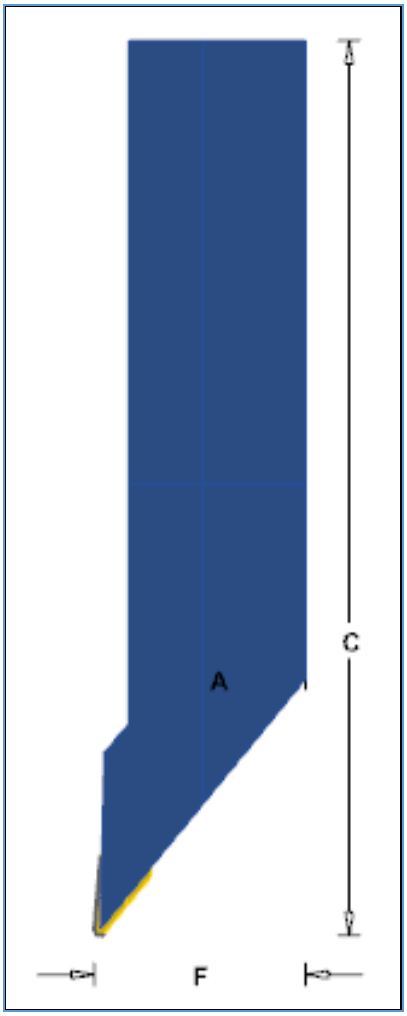
RE = Corner Radius

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.

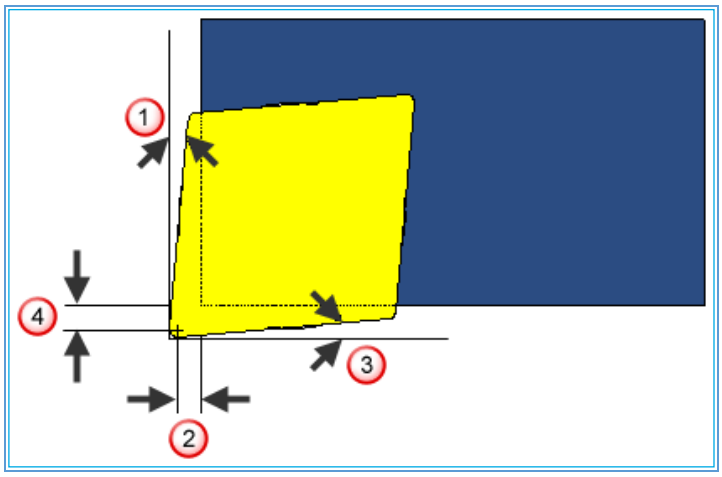


FeatureCAM tool designation = **SW_Turn_35m_RH**

© 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.

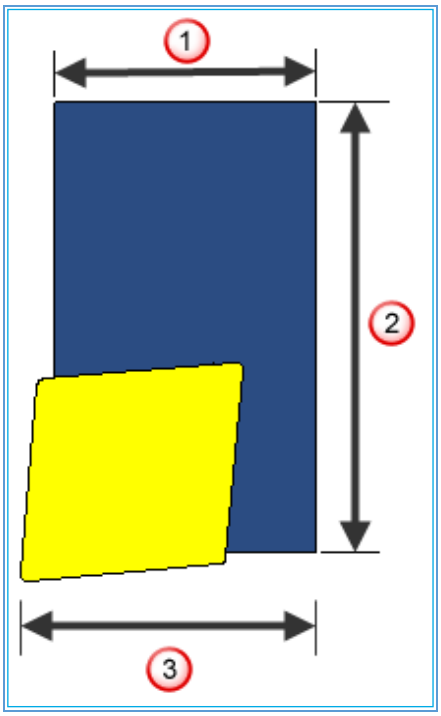


1 End Angle

2 End Clearance

3 Side Angle

4 Side Clearance



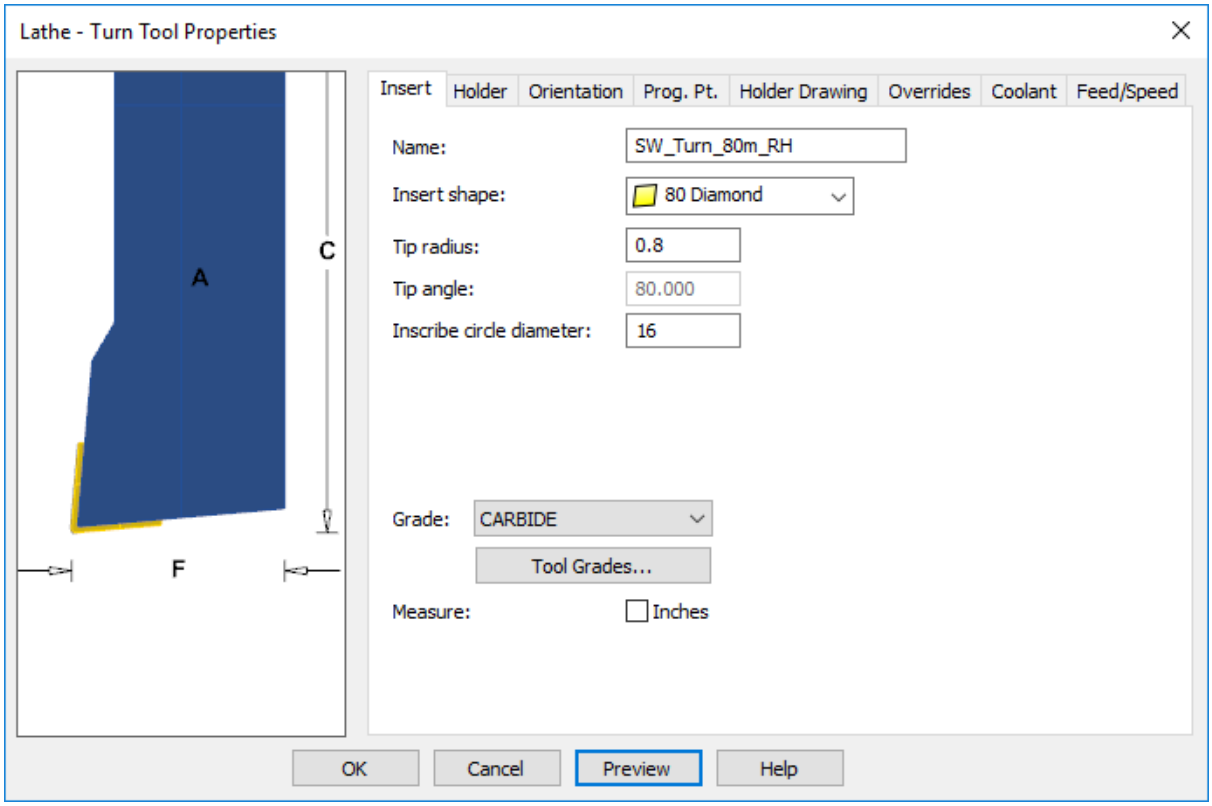
1 Shaft Width (A)

2 Length (C)

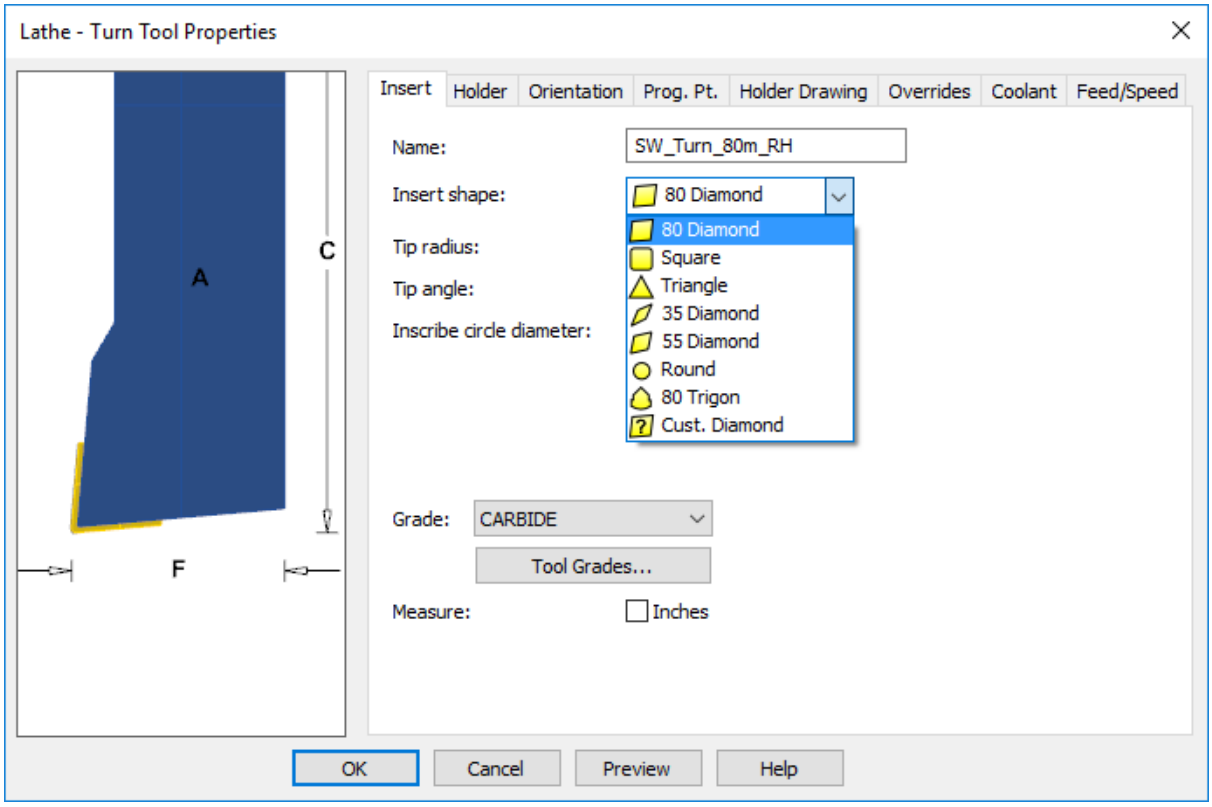
3 Tip to Back (F)

© 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.

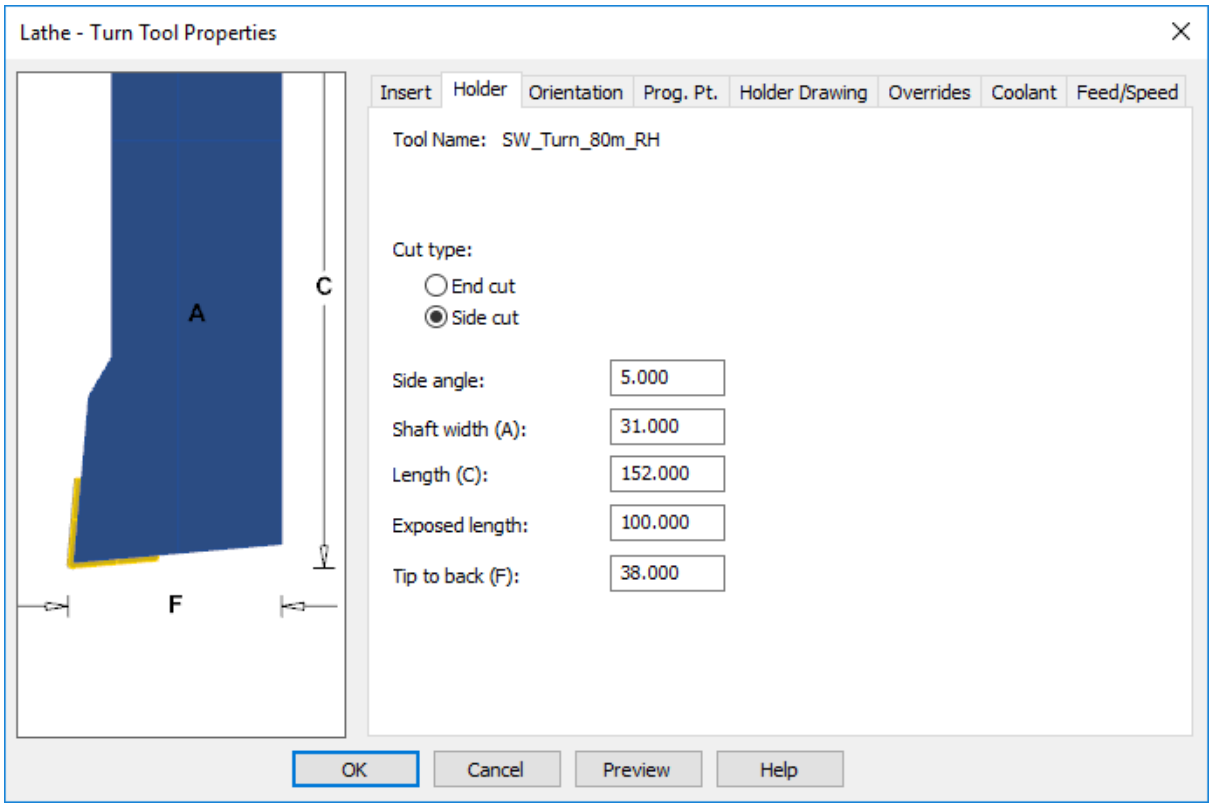


Insert Shape

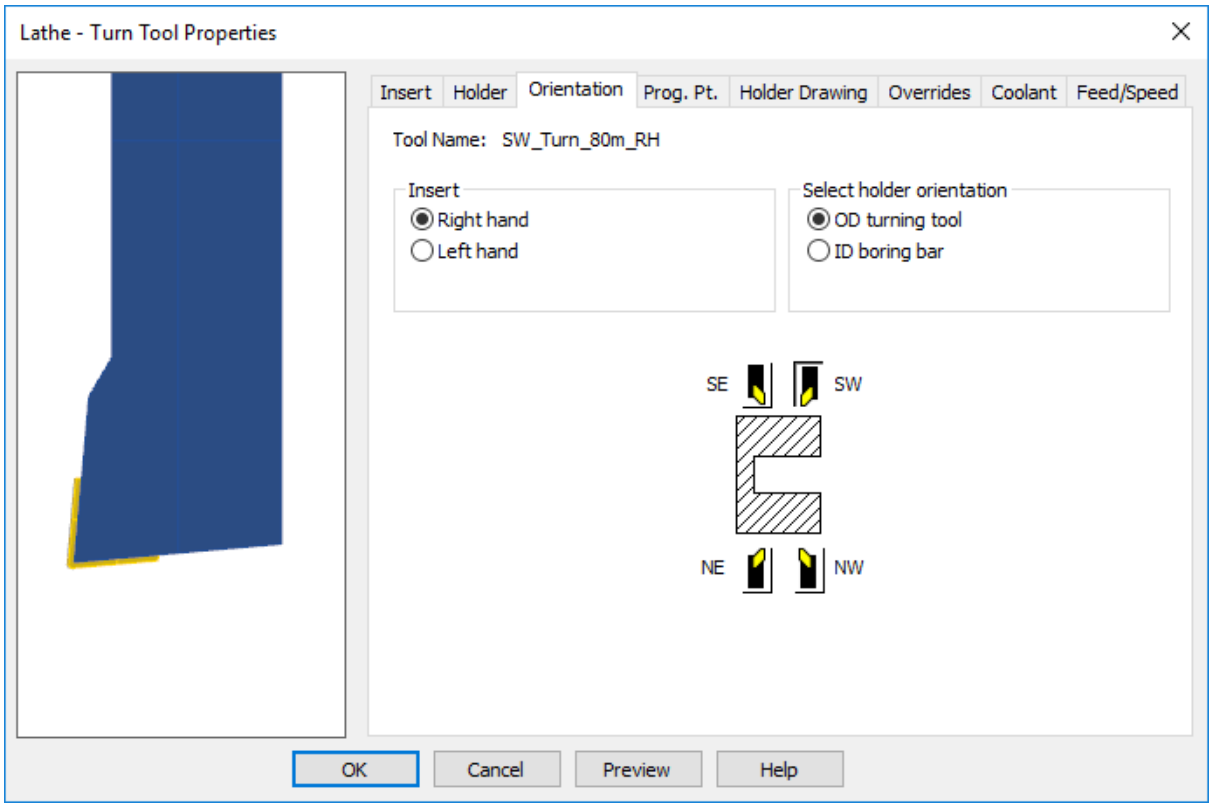


© Copyright 2016 Autodesk, Inc. All rights reserved

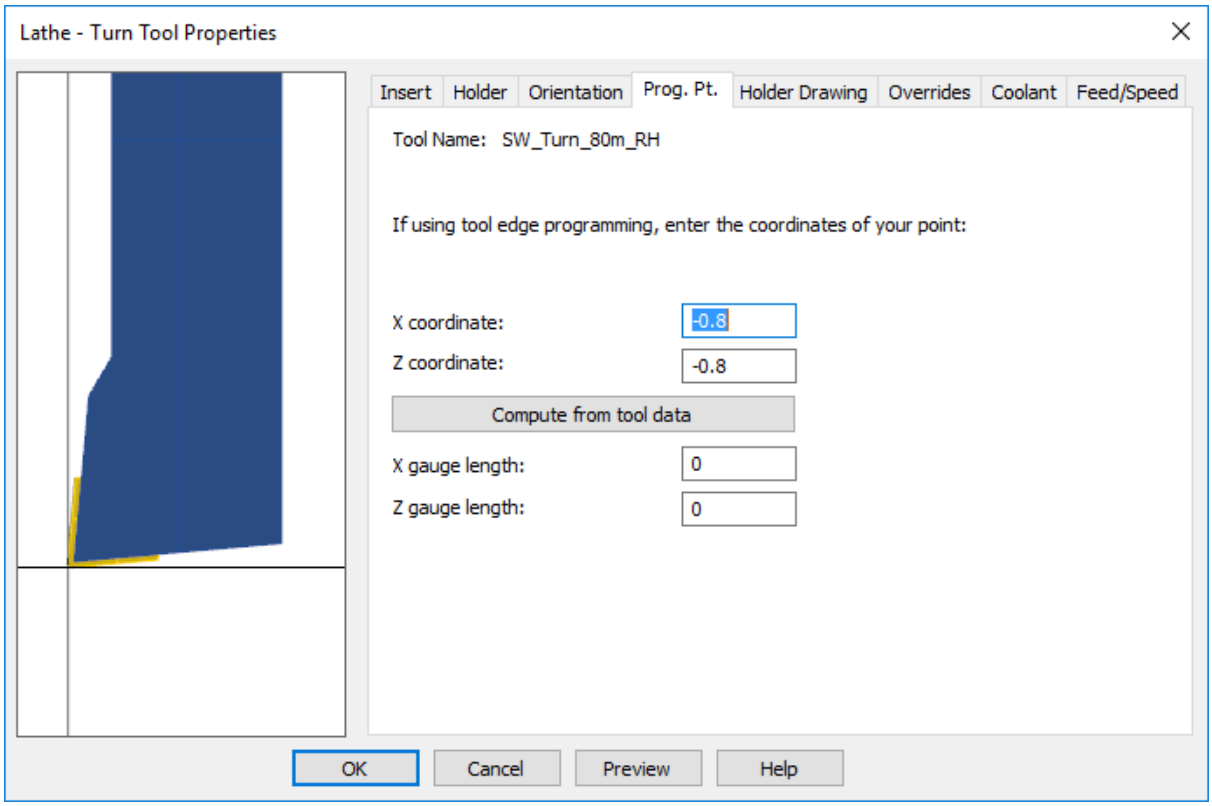
Holder



Orientation



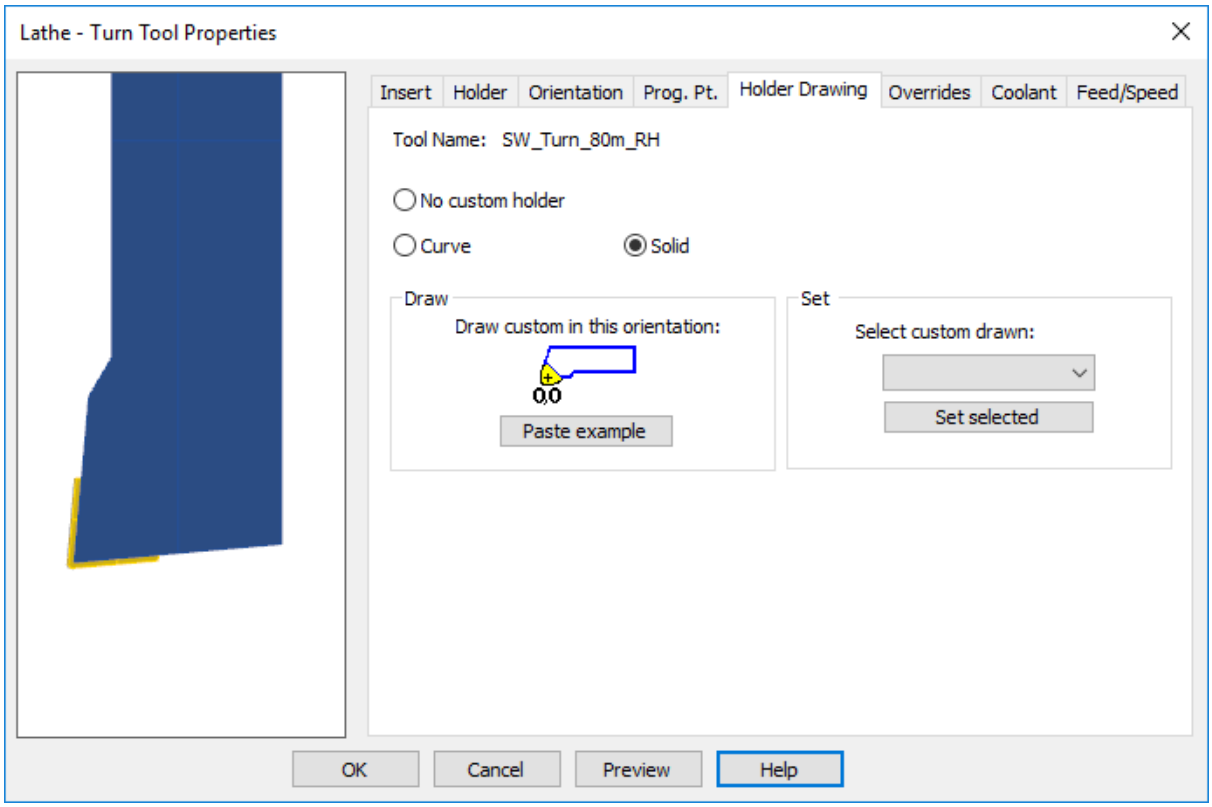
© Copyright 2016 Autodesk, Inc. All rights reserved



*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**.*

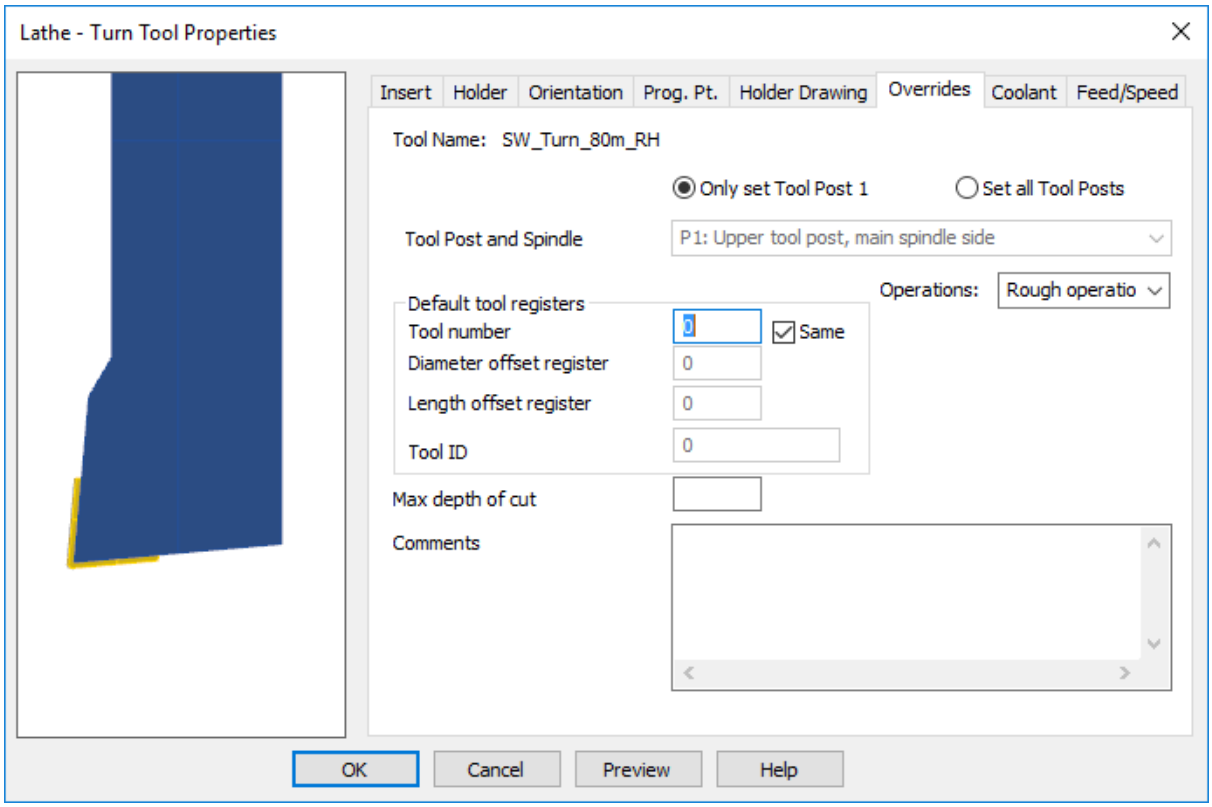
© Copyright 2016 Autodesk, Inc. All rights reserved

Holder Drawing



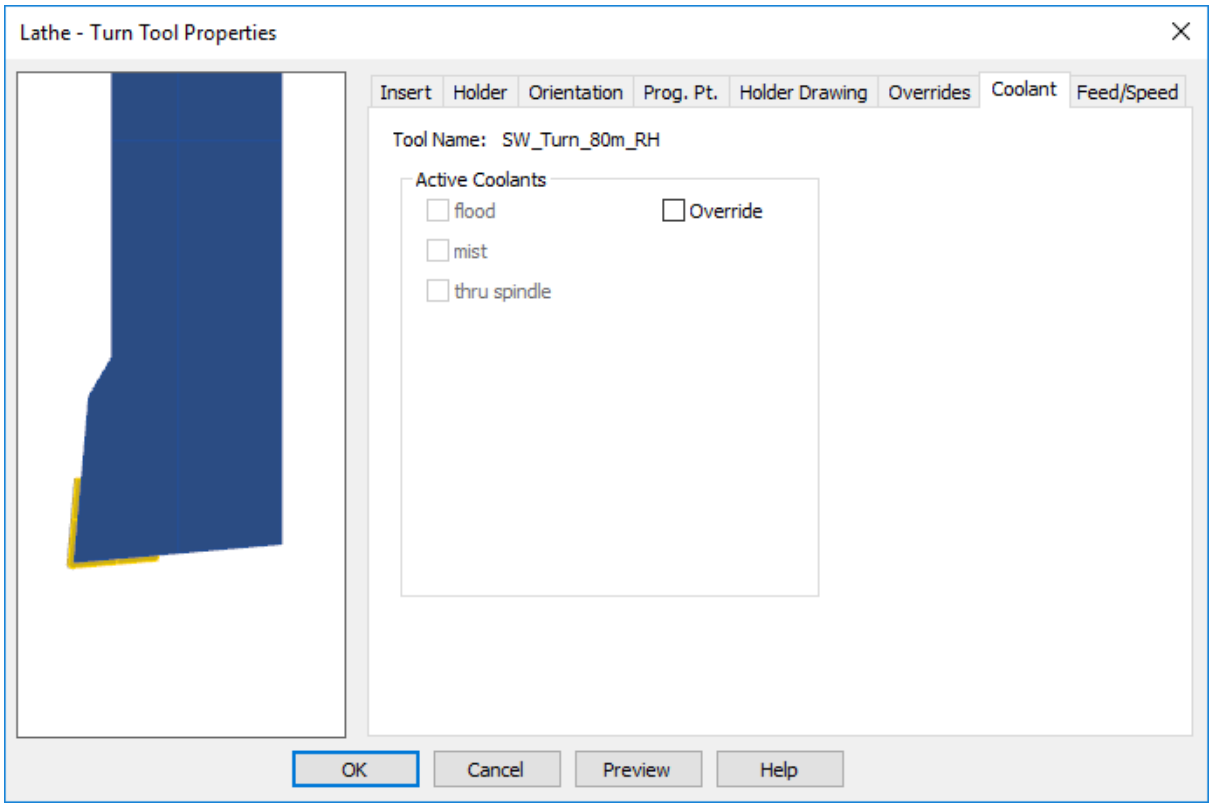
Solid — you can select a solid to define the shape of the Tool Holder. This is the most accurate way to represent the tool holder.

Overrides

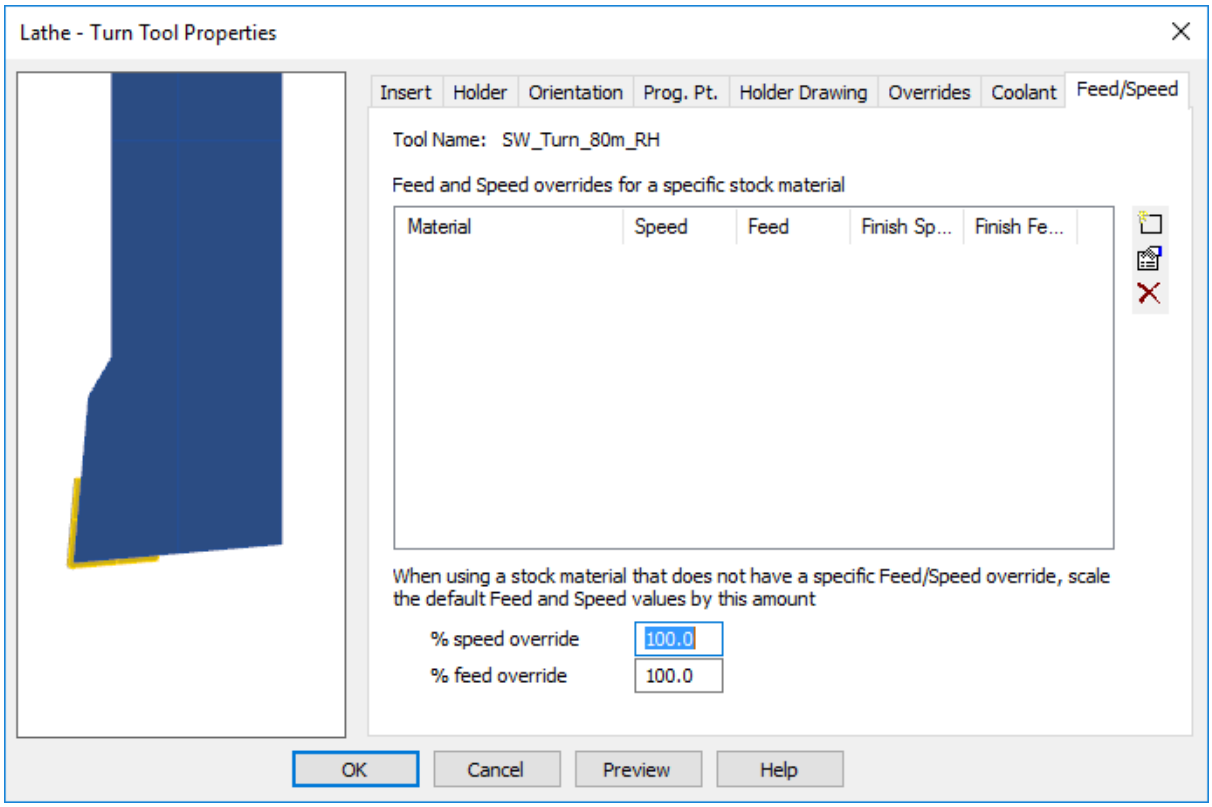


© Copyright 2016 Autodesk, Inc. All rights reserved

Coolant



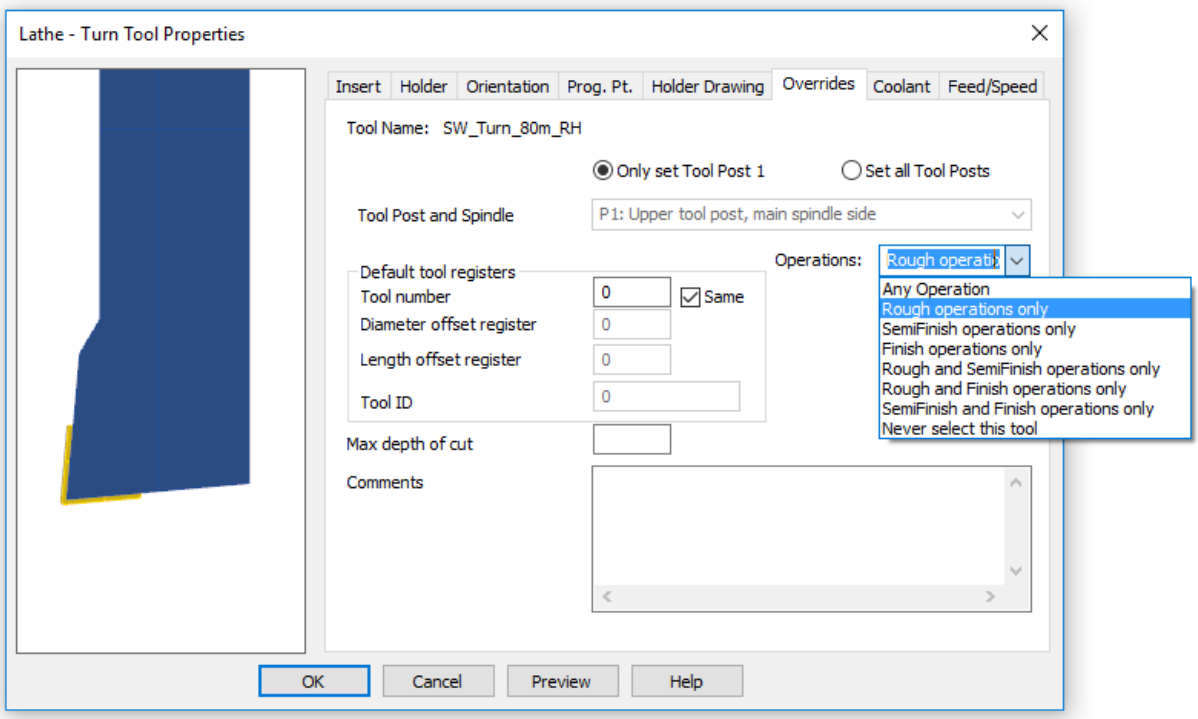
Feeds/Speeds



© 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.

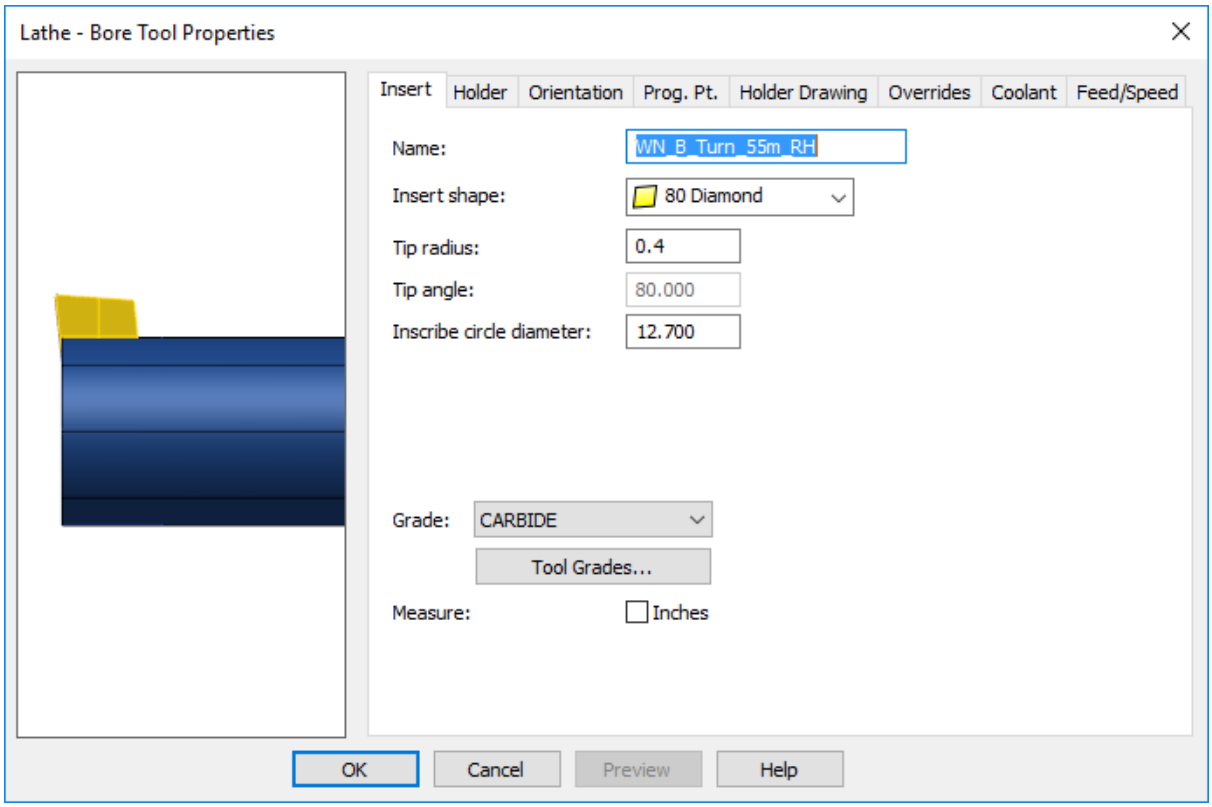
Remachining *- automatically sets the boundaries for subsequent operations that use the same curve. This minimizes air cutting and works between turn features, bore features and between holes and bore features. The same curve must be used in both features. ID features use the results of a previous turn 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).*

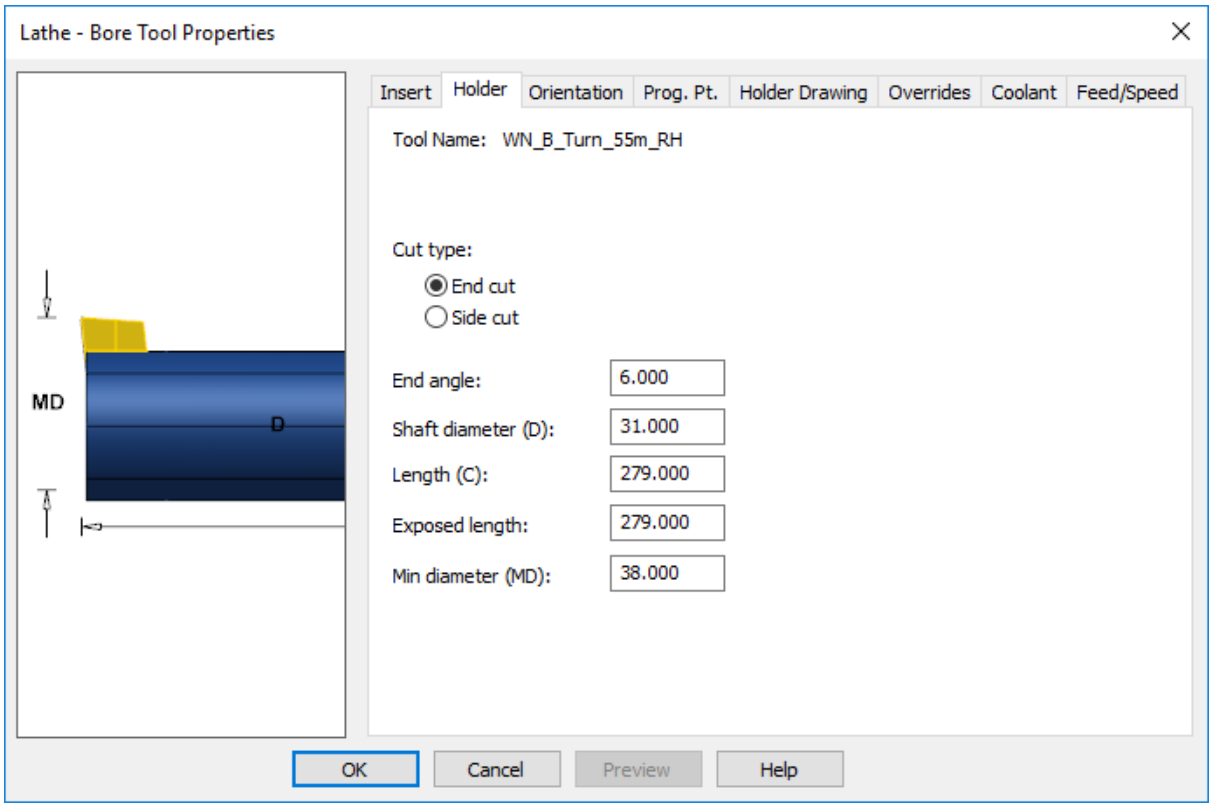
Do feed reduction for small moves *- This attribute helps FeatureCAM cut small features properly. It is typically applied to small chamfers or small radii but affects any small move. If Do feed reduction for small moves is selected, then any move with fewer revolutions than the Threshold, is reduced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

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss - Bore Tool Properties example.



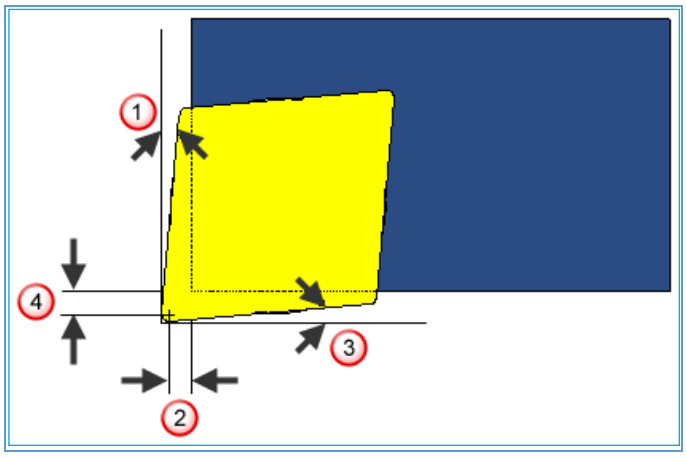
Holder Information



© 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.*

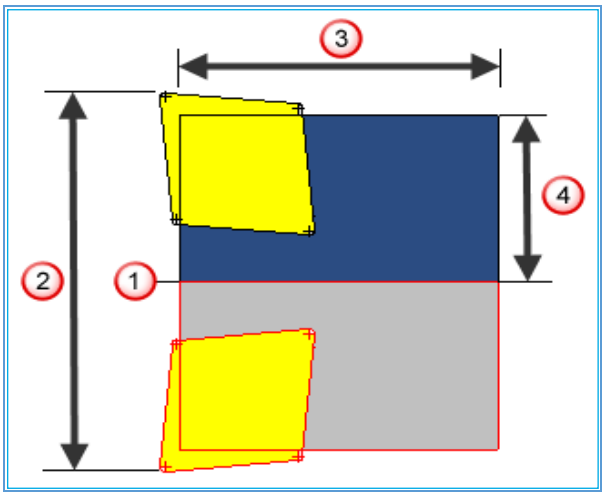


1 End Angle

2 End Clearance

3 Side Angle

4 Side Clearance

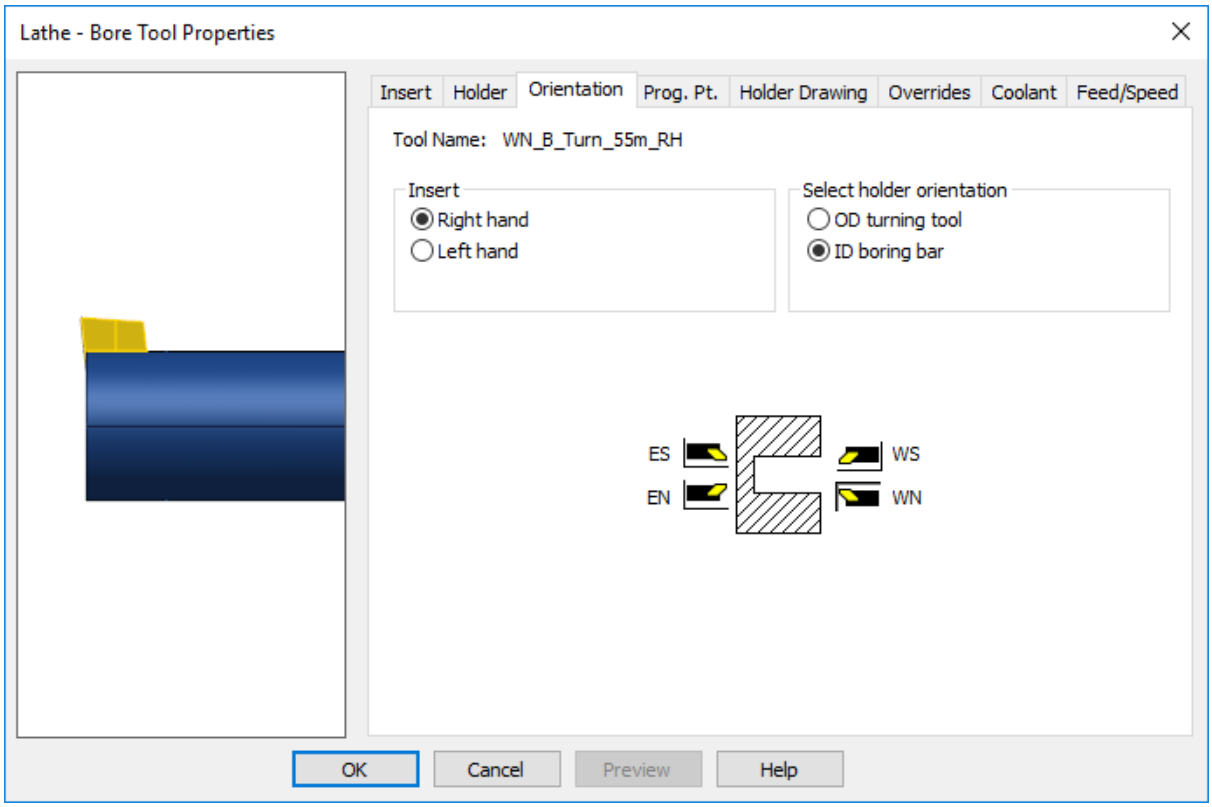


© Copyright 2016 Autodesk, Inc. All rights reserved

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.

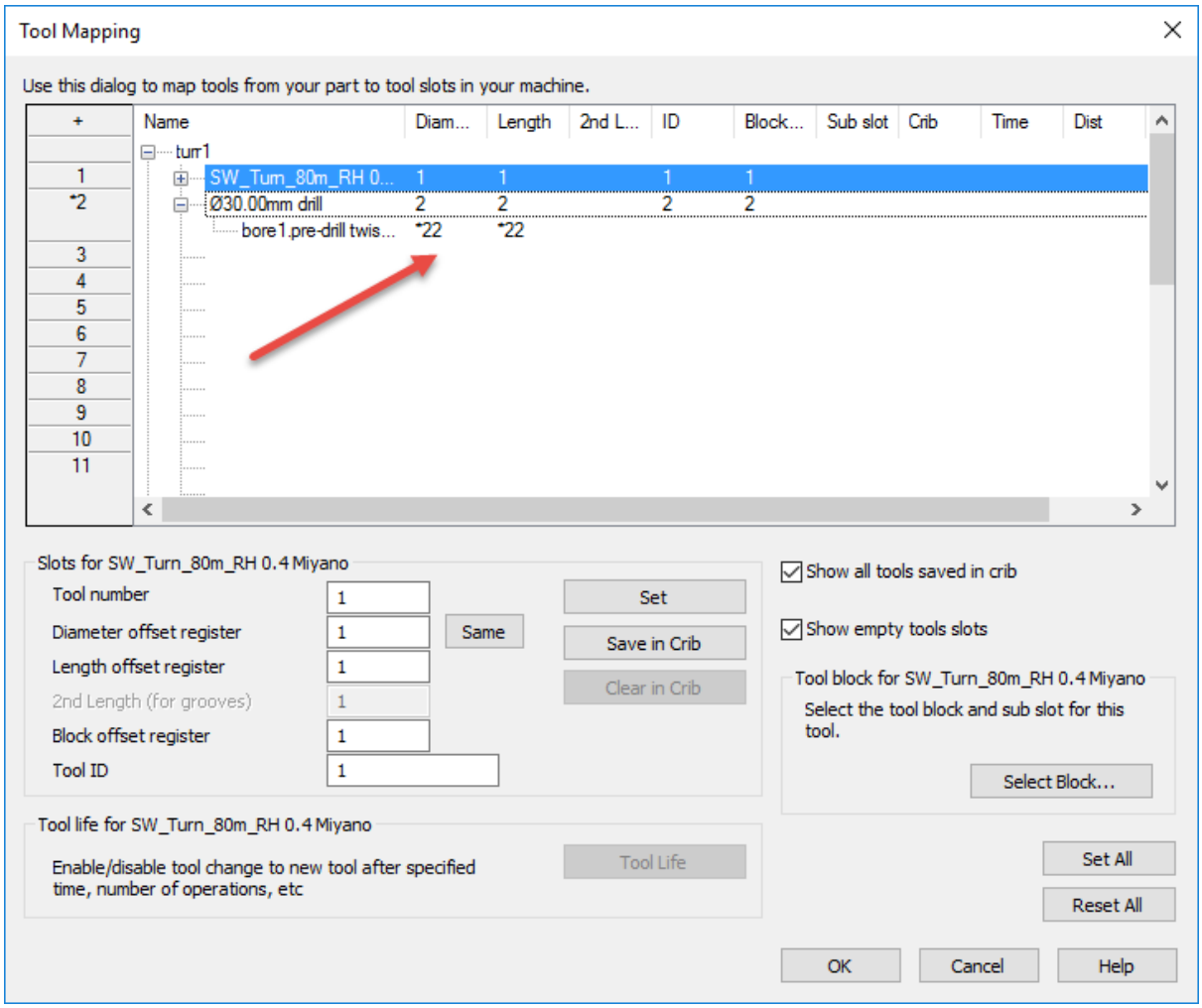
Boring Orientation



© Copyright 2016 Autodesk, Inc. All rights reserved

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

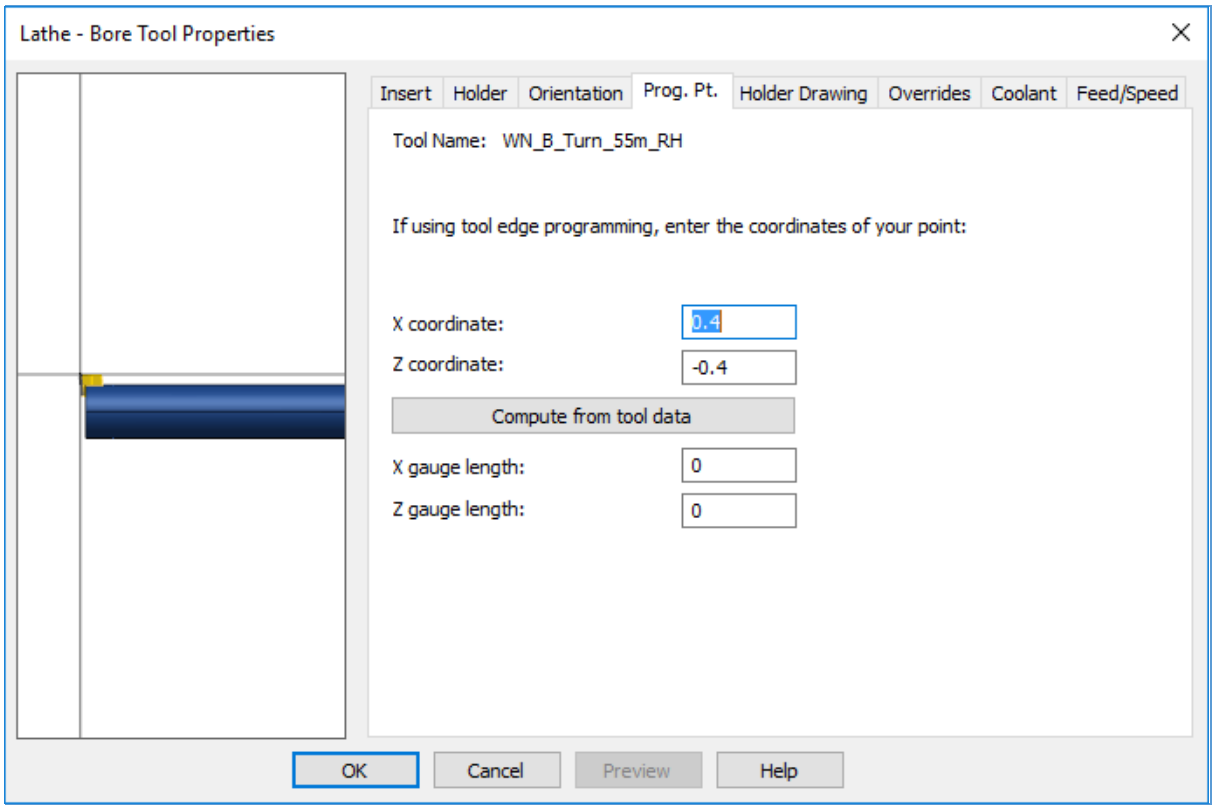
1. Create the Hole feature and override the tool for the drilling operation to be an insert drill.
2. Create the Bore feature and override the tool to be the same insert drill.
3. If you view the tools in the Tool mapping dialog you see that there are two drills listed in the same tool slot, but they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



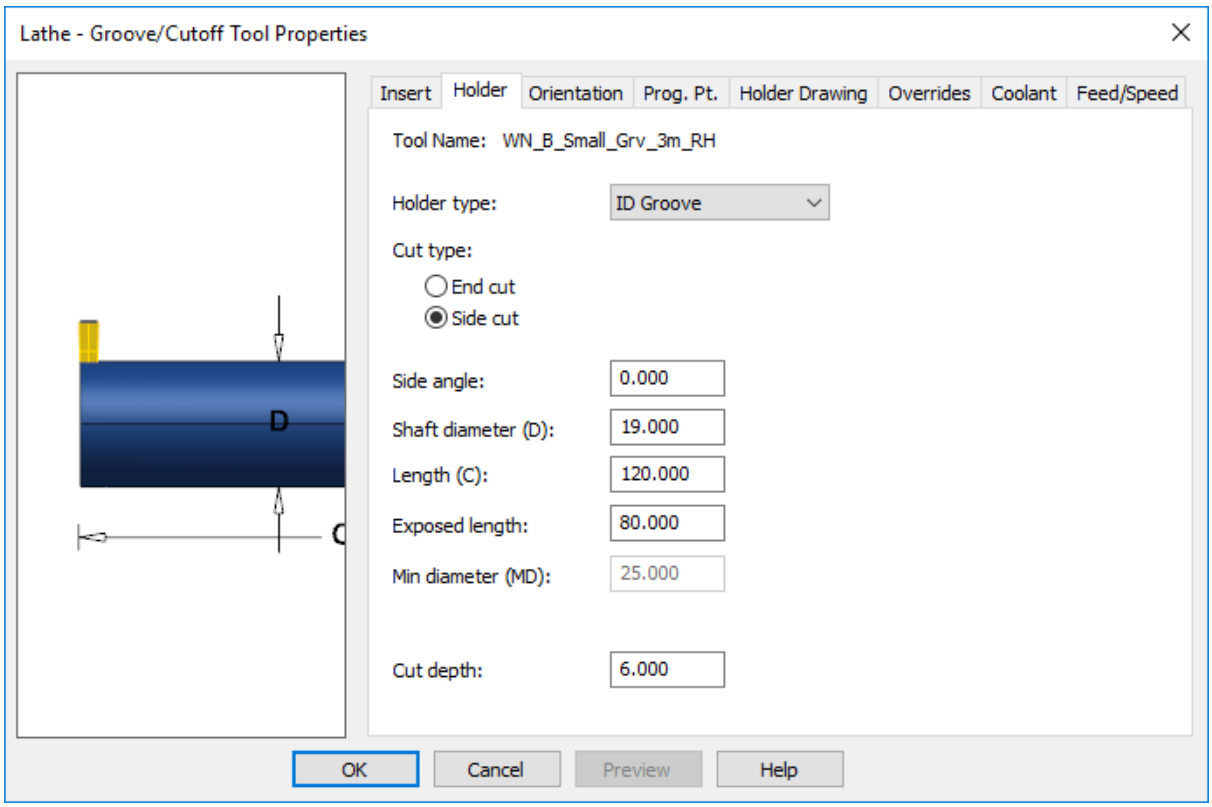
Please note that you need to reduce the retract distance to avoid a collision inside the bore. This needs to be applied to roughing and finishing operations.

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss - Bore Programming Point

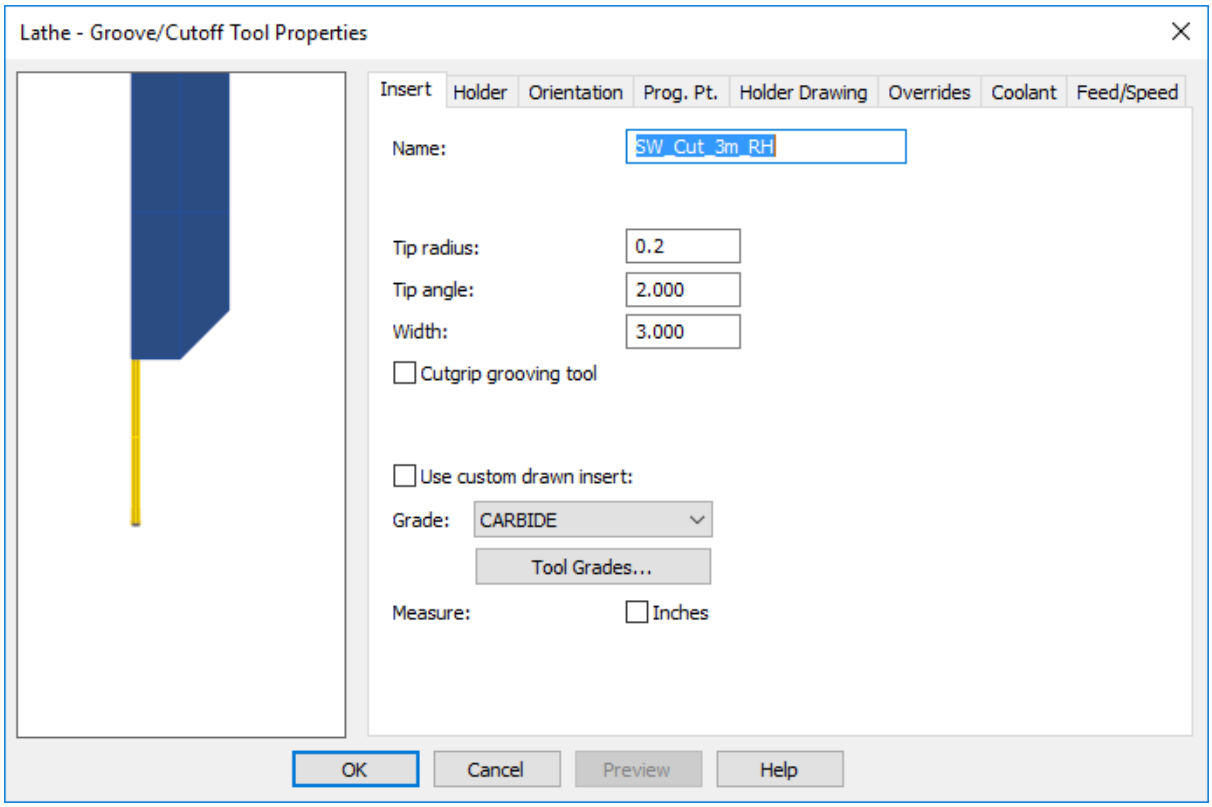


Swiss - Groove tool Properties example



© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss - Cutoff Tool Properties example

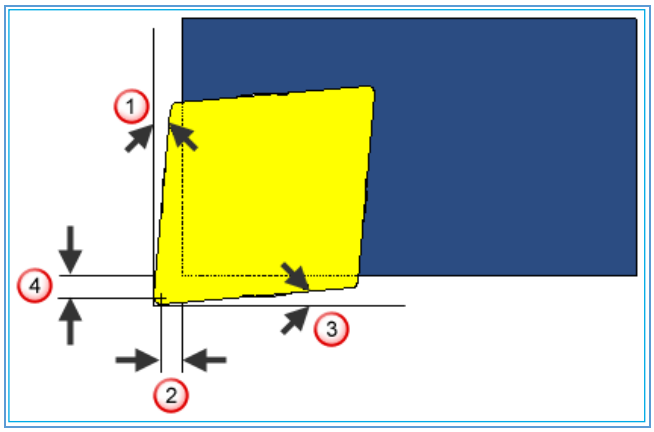


Holder Type: **-** For grooves possible holder types are:

- **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.

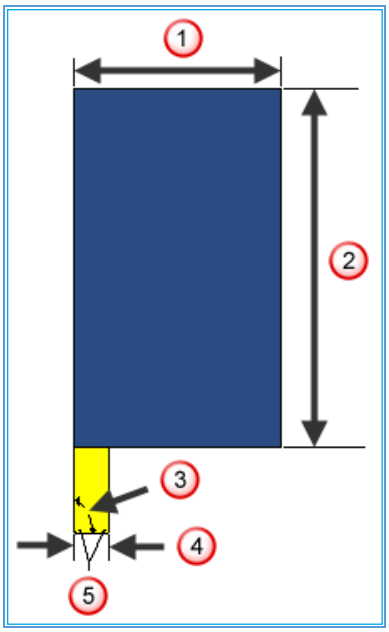
End Cut: **-** The tool cuts in a direction parallel with the length of the holder.

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



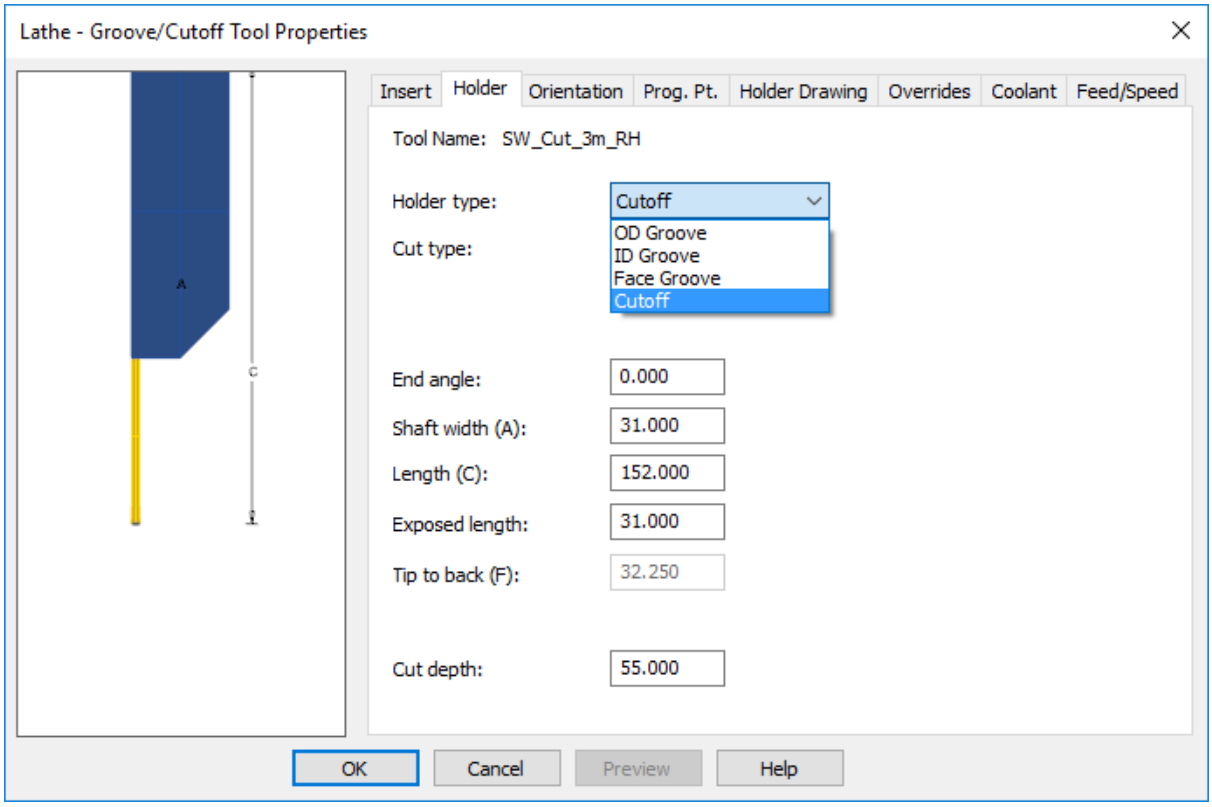
1 End Angle 2 End Clearance 3 Side Angle 4 Side Clearance

© Copyright 2016 Autodesk, Inc. All rights reserved



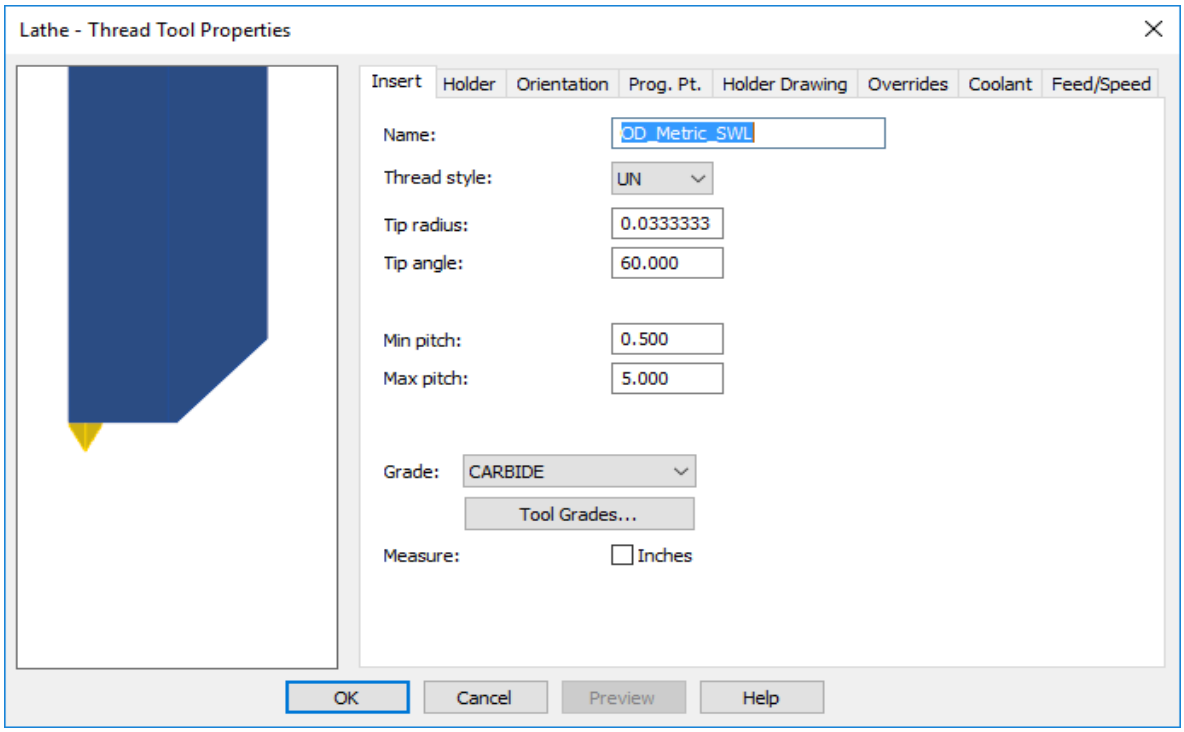
1 Shaft Diameter (D) 2 Length (C) 3 Tip Angle 4 Width 5 Tip Radius.

Swiss - Groove/Cutoff Holder tool Properties example



© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss - Thread Tool Properties example.

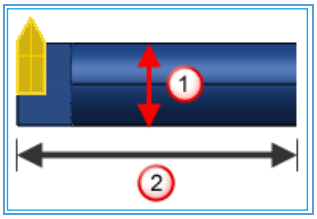


Name *- is a string that identifies the tool. The name must be unique among all the tools in the crib.* ***Tip Radius*** *is the radius of the cutting tip of the insert. See diagram.*

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*

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

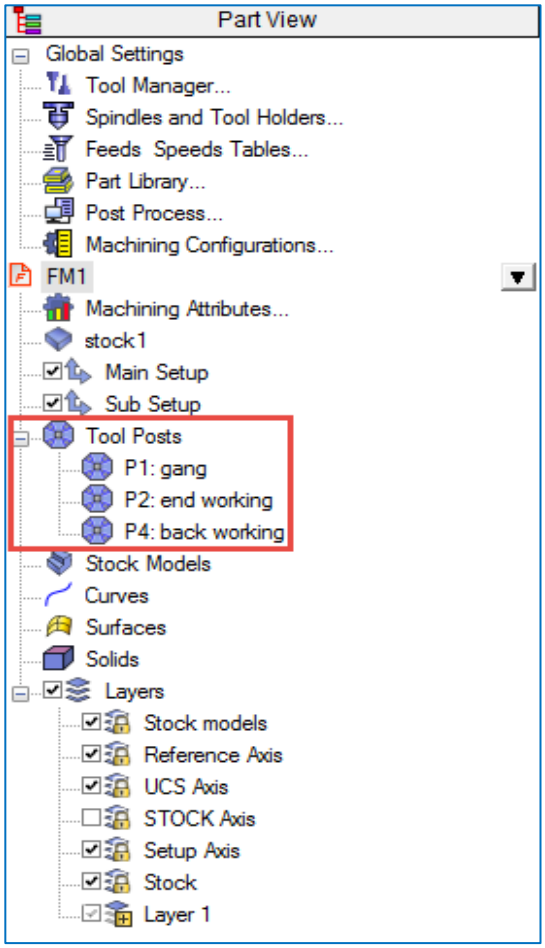
Side Cut *- The tool cuts in a direction perpendicular with the length of the holder.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Tool Post configuration

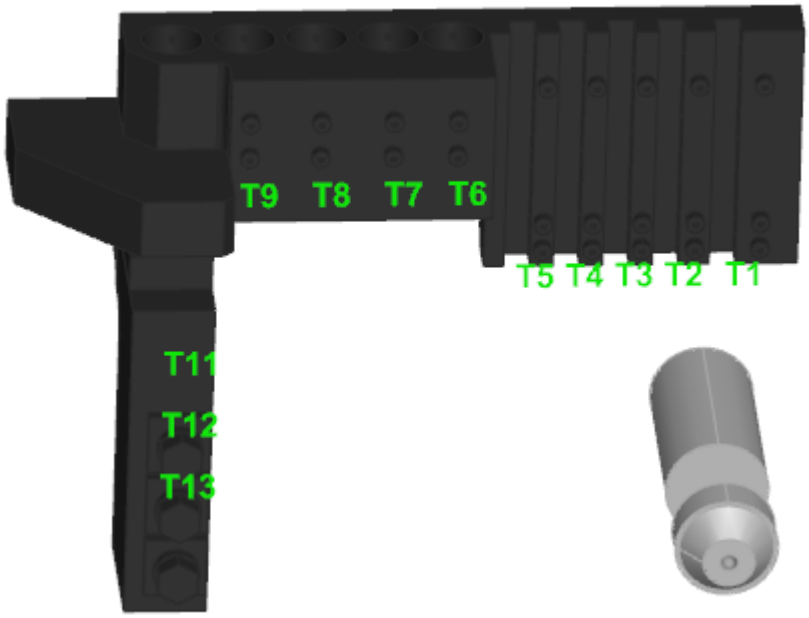
The example machine within FeatureCAM for Swiss Turning is a Citizen L20E-IX.cnc machine. Other machines are available as the product develops.

This is the Tool post configuration inside FeatureCAM for this machine.



© Copyright 2016 Autodesk, Inc. All rights reserved

- **Gang** Tool post supports tool numbers **T1 T2 T3 T4 T5** Turning Tools for **Main spindle**
- **T6 T7 T8 T9** 4 tools for milling drilling around the diameter.
- **T11 T12 T13** 3 Collet chucks for end working on the **(Main – spindle)**



- **End working** tool numbers **T21 T22 T23** 3 Collet chucks for **end working** on the **(Main – spindle)**

- **Back working (Sub – Spindle)** tool numbers **T31 T32 T33 T34**

© Copyright 2016 Autodesk, Inc. All rights reserved

This page is intentionally left blank

© Copyright 2016 Autodesk, Inc. All rights reserved

Contents

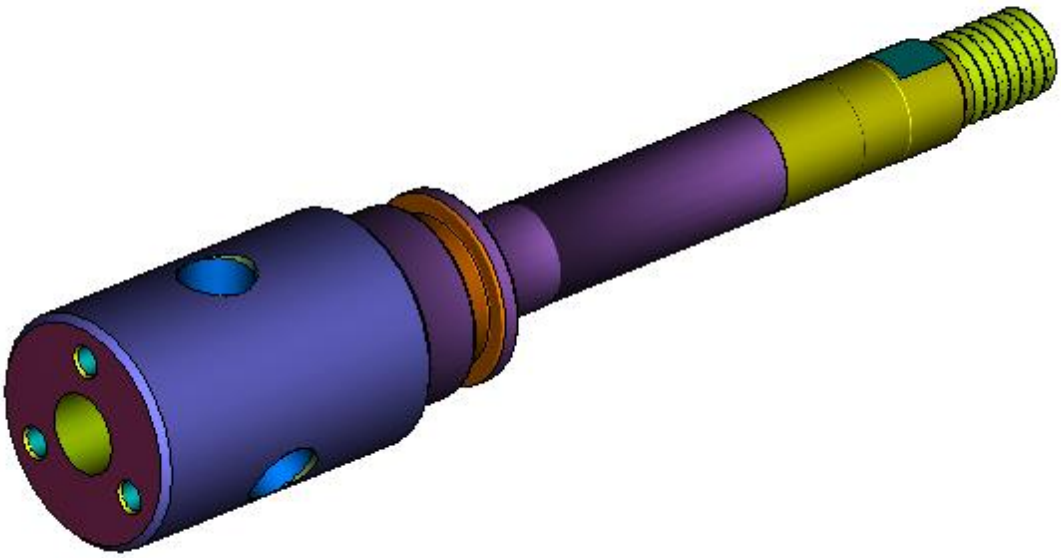
Swiss AFR example - 1 2
Getting Started Part-1 3
Turned Profile Extraction 3
Creating Curves 5
Main Spindle Turning 5
Creating a Side Feature 8
Create a Pattern of the Side Feature. 11
Creating a Thread 12
Turning up to large diameter 15
Creating a Groove feature from Curve 16
Creating Hole Features around the diameter 18
Creating a Cutoff feature 18
Sub – Spindle Turning 19
Sub – Spindle Back holes 20
Output NC 21
Swiss AFR example - 2 22
Getting Started Part-2 23
Turned Profile Extraction 23
Main Spindle Turning 25
Drill 10.4mm diameter centre hole 20mm deep 27
Create a Bore feature. 27
Create the 2 long slots along full length. 28
Large open side feature x 3 face planes. 31
Create a Group and then a Pattern of all three side Features 34
Extract 2mm diameter holes from around the diameter 35
We will now machine the 2 slots on the end 35
Cutoff and transfer to Sub-Spindle. 36
Sub-Spindle Face feature 37
Holes on Sub-Spindle 37
Ordering the tool post operations 38

© Copyright 2016 Autodesk, Inc. All rights reserved

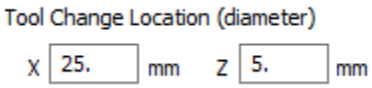
Swiss AFR example - 1

A Swiss-type machine uses a sliding headstock that feeds barstock through a guide bush and past a tool during an OD turning operation. The guide bush offers support for the barstock very near the point of the cut, preventing workpiece distortion. This makes Swiss-types machines particularly effective for producing long, slender parts.

The following example works through the fundamental machining principles of a simple Swiss turned and milled component using Feature recognition.



- The following information is located in folder.
- **C:\Training_Data\FeatureCAM Course Data 2017**
- Post Processor = **Citizen L20E-IX.cnc**
- Machine Design file = **Citizen L20E-IX - 16mm.md**
- Crib = **Swissmetric**
- Solid model **= FeatureCAM- Swiss-Example-1.x_t**
- **Tool Change position** from post Options



-
- Remember to select **Automatic tool orientation** in **Machining Attributes** or **Machining Configuration.**

© Copyright 2016 Autodesk, Inc. All rights reserved

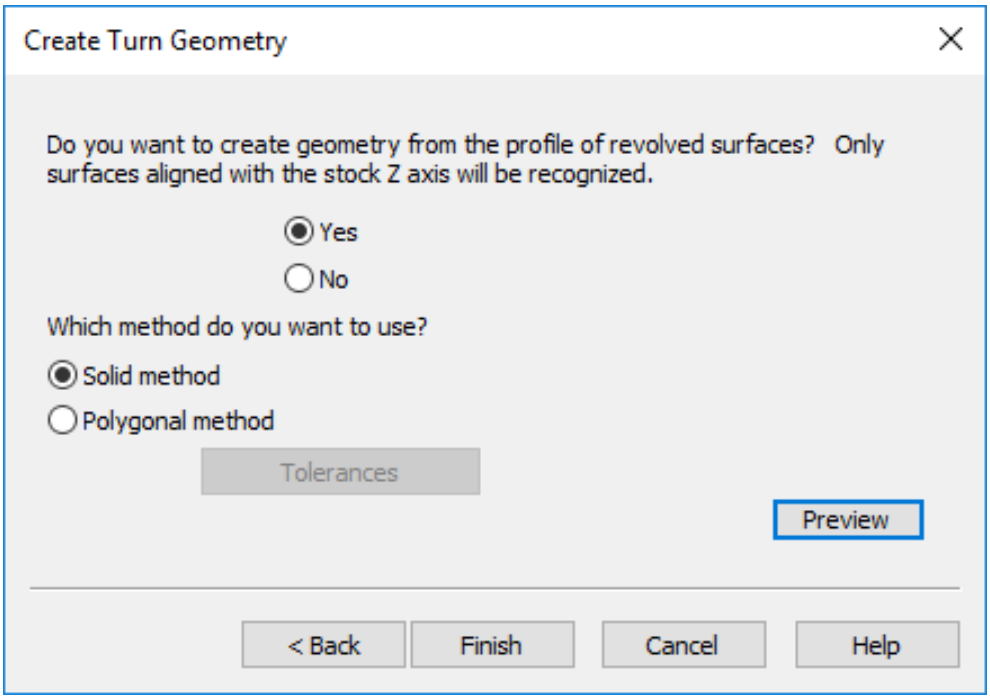
Getting Started Part-1

1. Open a new session of FeatureCAM and select **Swiss Turning** as the machining module. We are using **Metric** Units.
2. Cancel the **Stock dimensions** wizard.
3. Import the Solid model **FeatureCAM- Swiss-Example-1.x_t** from **Swiss Turning files to import.**
4. Use the Wizard to align the part and to establish the initial setup location and stock size.
5. Select **Next** 5 times until you get to the **Stock Dimensions** page.
6. Select **Compute stock from the size of the part**.
7. Enter **1mm Front**, **3mm Back** and **1mm** on the **Outside Diameter**.
8. Select **Next**. Then select the **Right-hand Hand** and then put **-1mm** into the Z location field.
9. Select **Finish**.

Turned Profile Extraction

*We will use the Wizard to extract the turned profile only. Select No and enter until you get to the **Create Turn Geometry** page.*

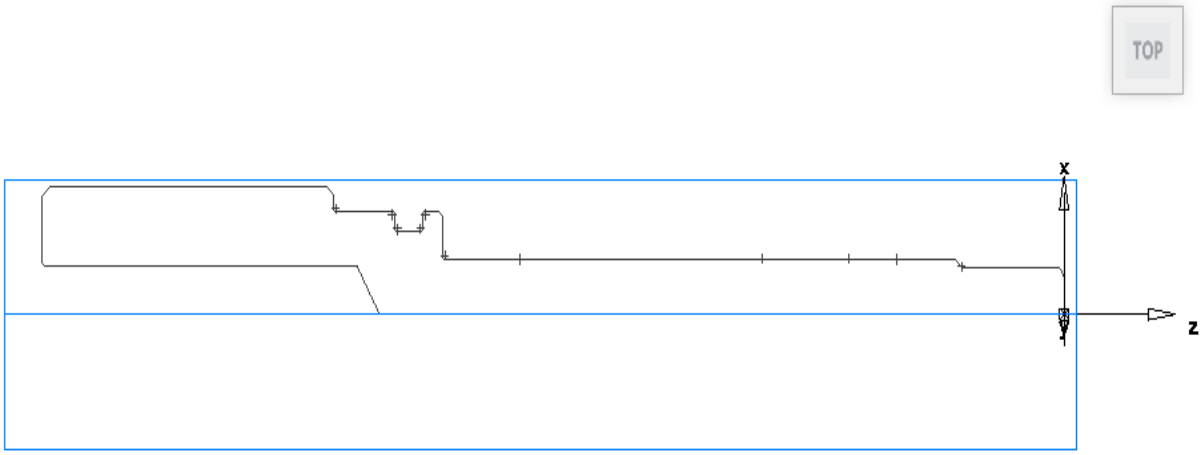
Select the following options.



10. Select **Finish**.
11. **Hide** the Solid model.

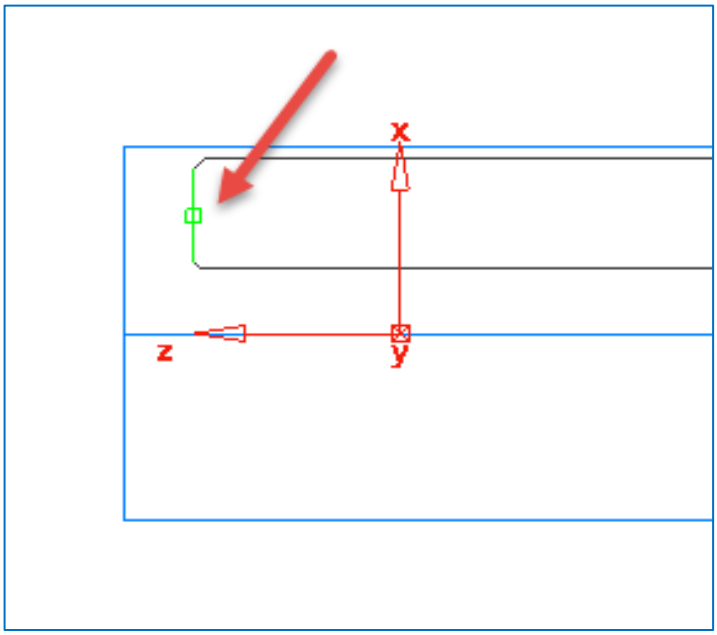
The following image shows the Turned Geometry.

© Copyright 2016 Autodesk, Inc. All rights reserved



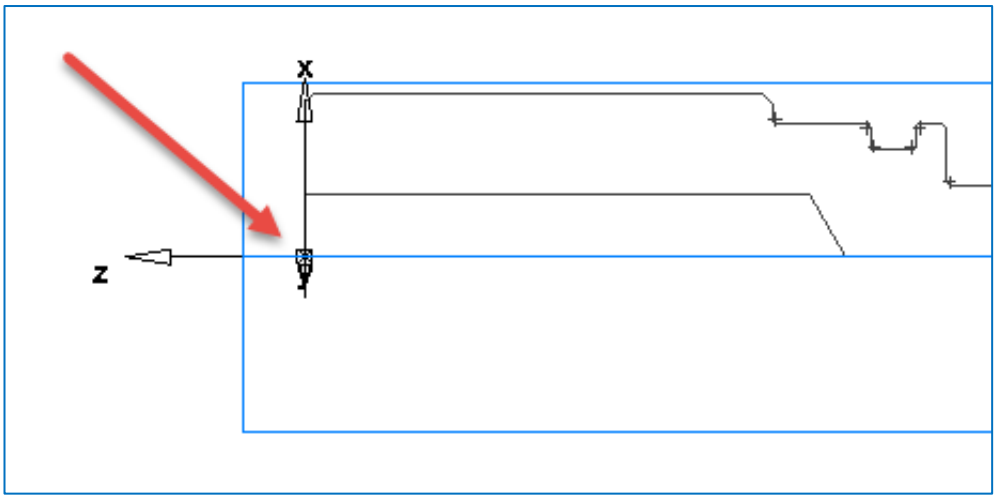
*We also have to setup the **Sub Setup location**. We can do this by **double clicking** on the **Sub Setup** in **PartView**.*

12 Double click on **Sub Setup** and select **Edit**. Select **Next** until you get to the option Align to Stock Face. Select this option and select **Next.** Then select **Pick location**. Pick the line as shown below.



13 Select **Finish** to action the command. The **Sub Setup** will then move to the correct position as shown on the next page.

© Copyright 2016 Autodesk, Inc. All rights reserved



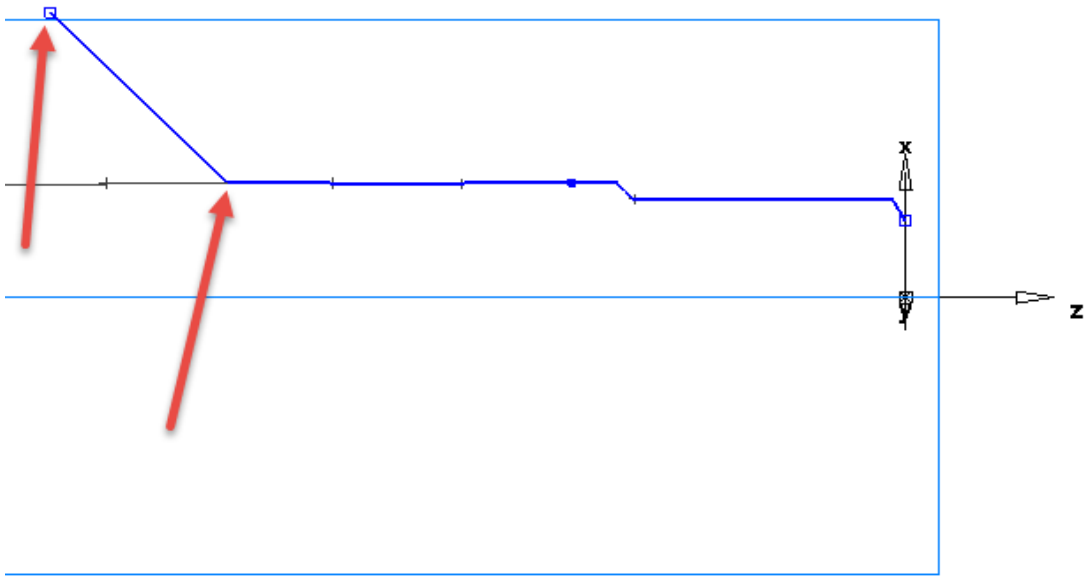
14 Please select **Main Setup.**

One of the main problems with using Swiss type machining centres is that they use a guide bush that holds the stock in position. In order for the part to remain in the bush we have to machine in 20mm sections. Before moving onto the next operation. The following example shows you this method of machining.

Creating Curves

A good idea when creating curves for Swiss turning is to Offset a line from Z Zero 20mm back from the Setup. We can then use this line to prevent the Curve from traveling past this point. I have also created a line a 45 degrees up to the stock material. See image on next page.

15 Create your Curve as shown on the next page.

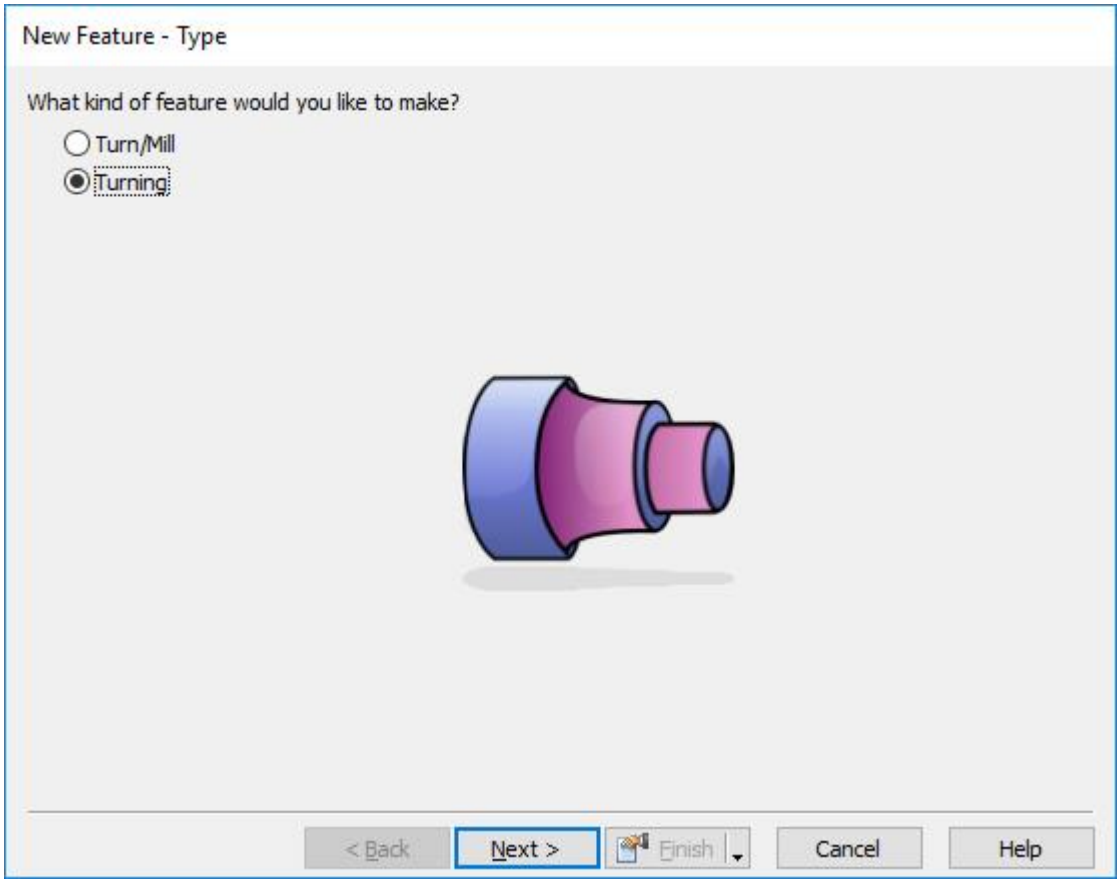


Main Spindle Turning

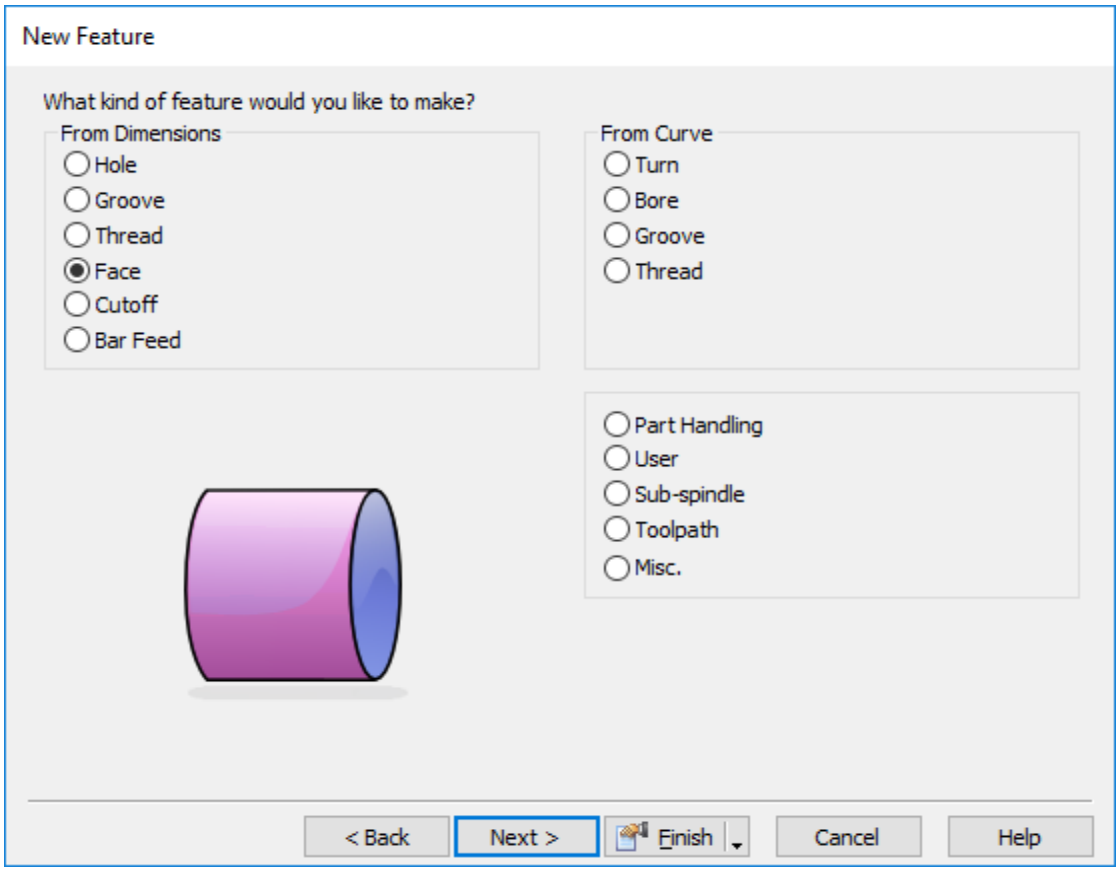
16 Create a **Facing** operation and then a **Turn** operation.

© Copyright 2016 Autodesk, Inc. All rights reserved

17 Select **CTRL + R** New Feature and select **Turning**.

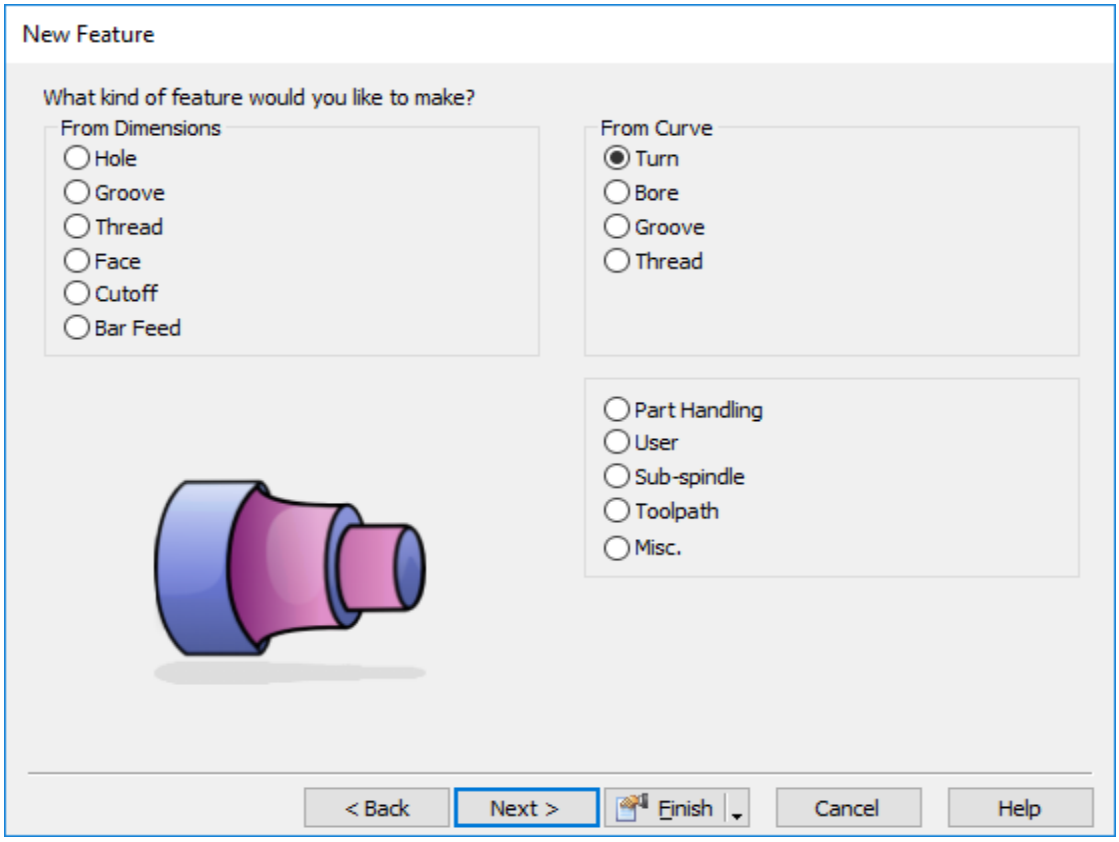


18 Select **Next**. Then Select **Face**.

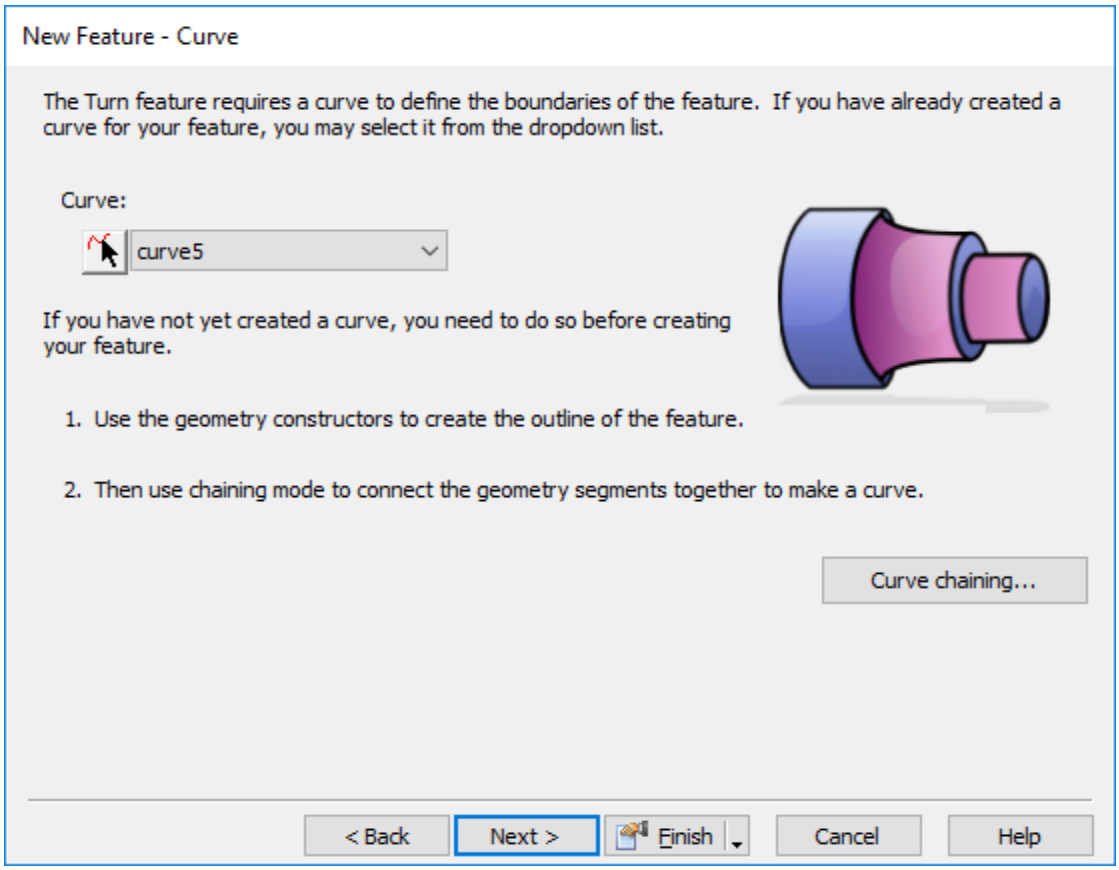


© Copyright 2016 Autodesk, Inc. All rights reserved

19 Select **Finish**. Then create a new Turning feature and select the curve you have just created. Make sure you use a **35 degree** tool to machine the undercuts.

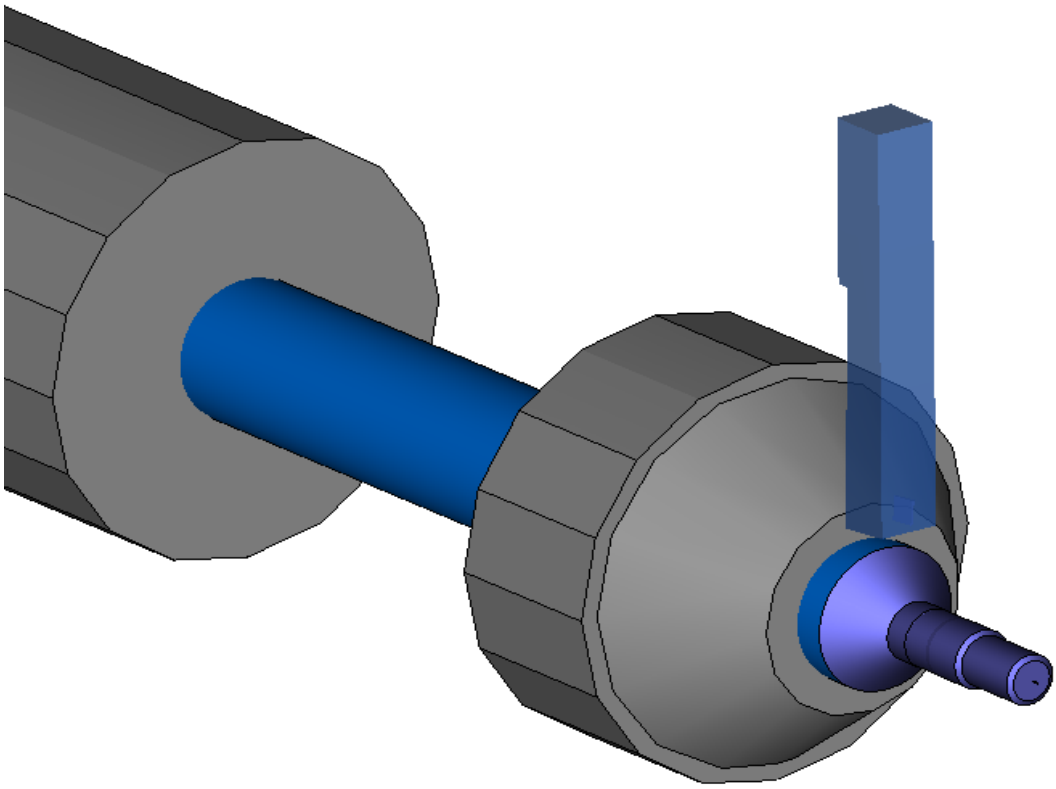


20 Select **Next** and pick your **Curve** and then select **Finish**.



© Copyright 2016 Autodesk, Inc. All rights reserved

21 Run a Machine Simulation.



Creating a Side Feature

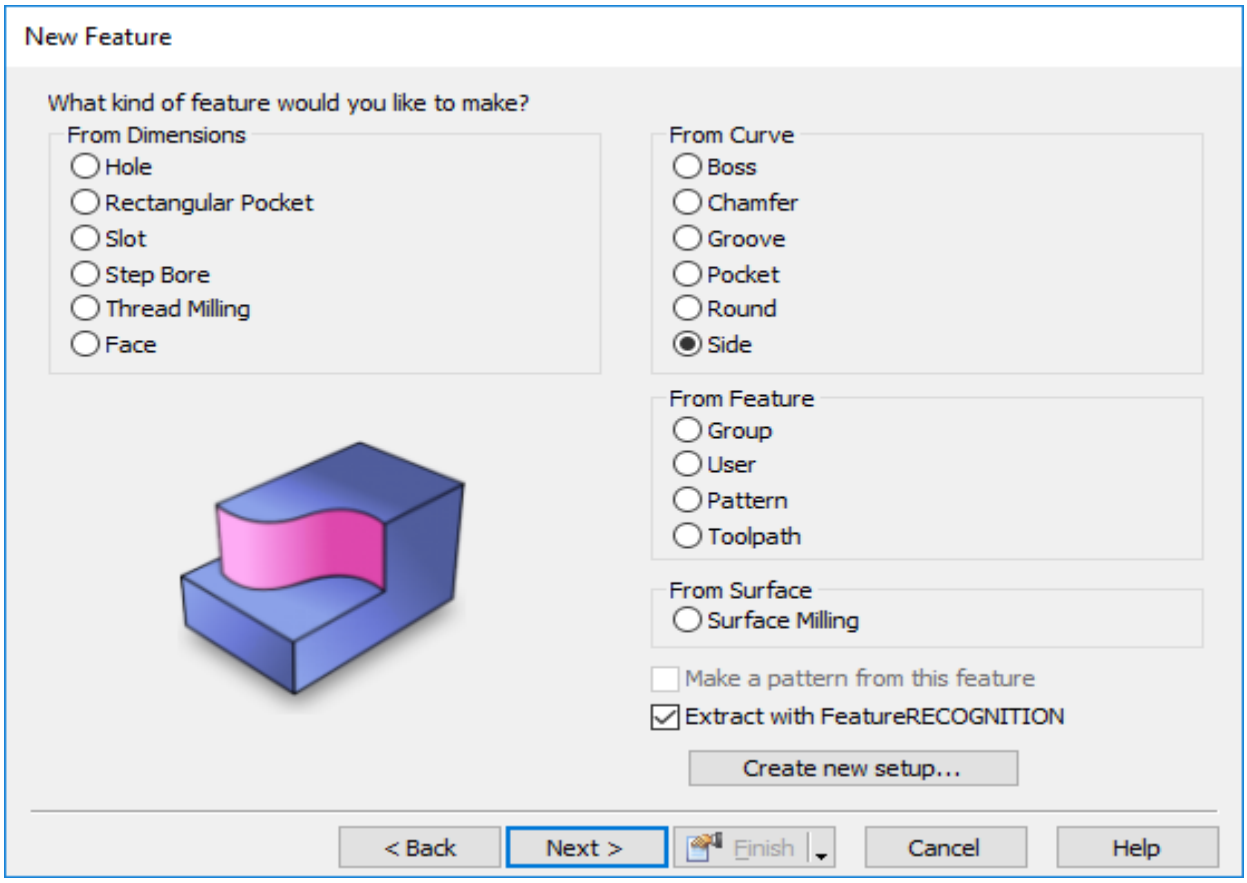
Before we turn the other part of the shaft we will have to machine the two flats on and thread the part. By doing this the part will not pull out of the guide bush.

22 **Eject** the machine simulation.

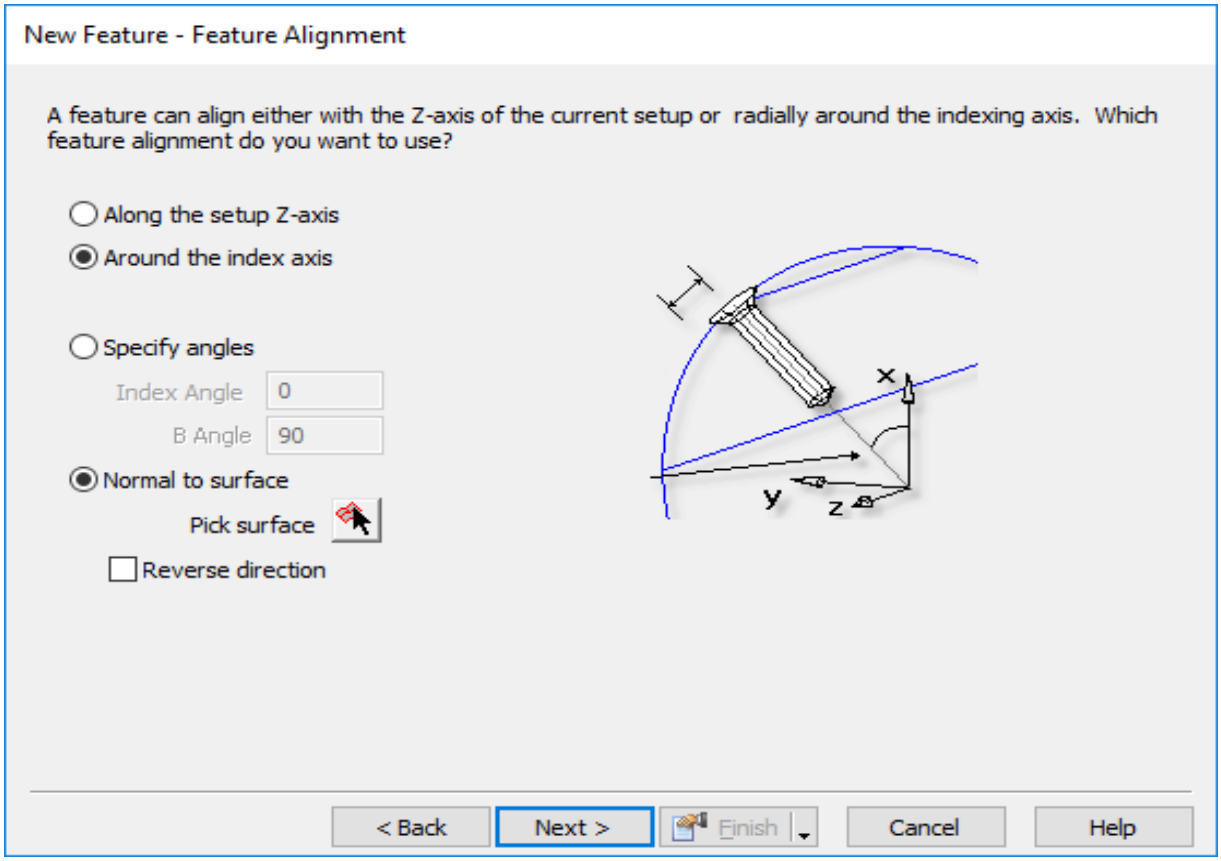
23 Then **show** the **solid model**.

24 Select **Ctrl +R** and create a new **Turn/Mill Side** Feature. Please **select Extract with FeatureRECOGNITION.**

© Copyright 2016 Autodesk, Inc. All rights reserved

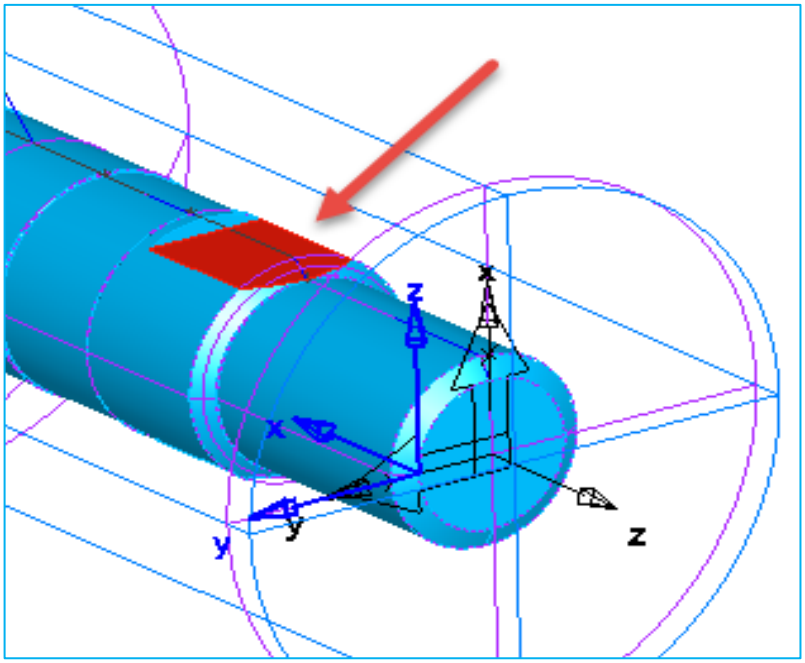


25 Select **Next**. We will have to select the option - **Around the index axis** and **Normal to surface**. See image on next page.



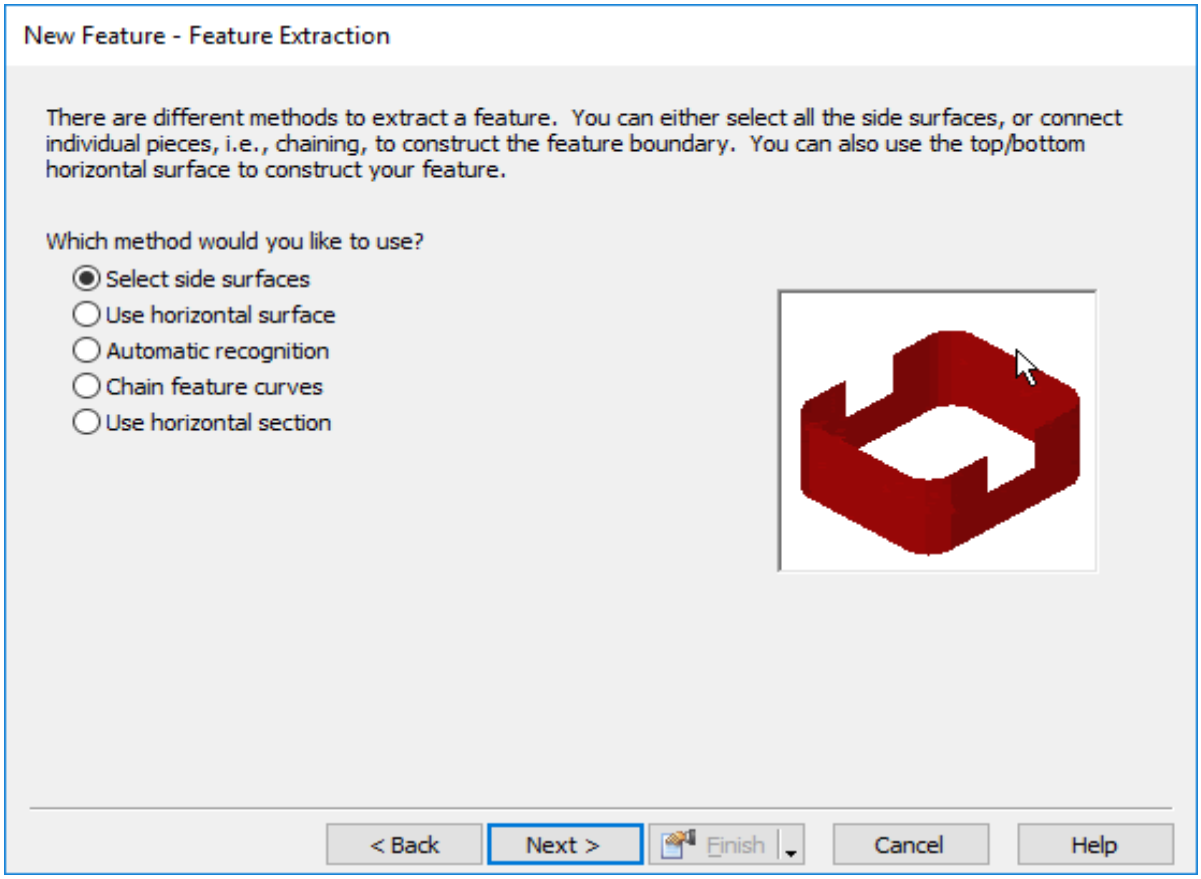
© Copyright 2016 Autodesk, Inc. All rights reserved

26 Use the Pick Icon to select your surface. Pick the face as shown.



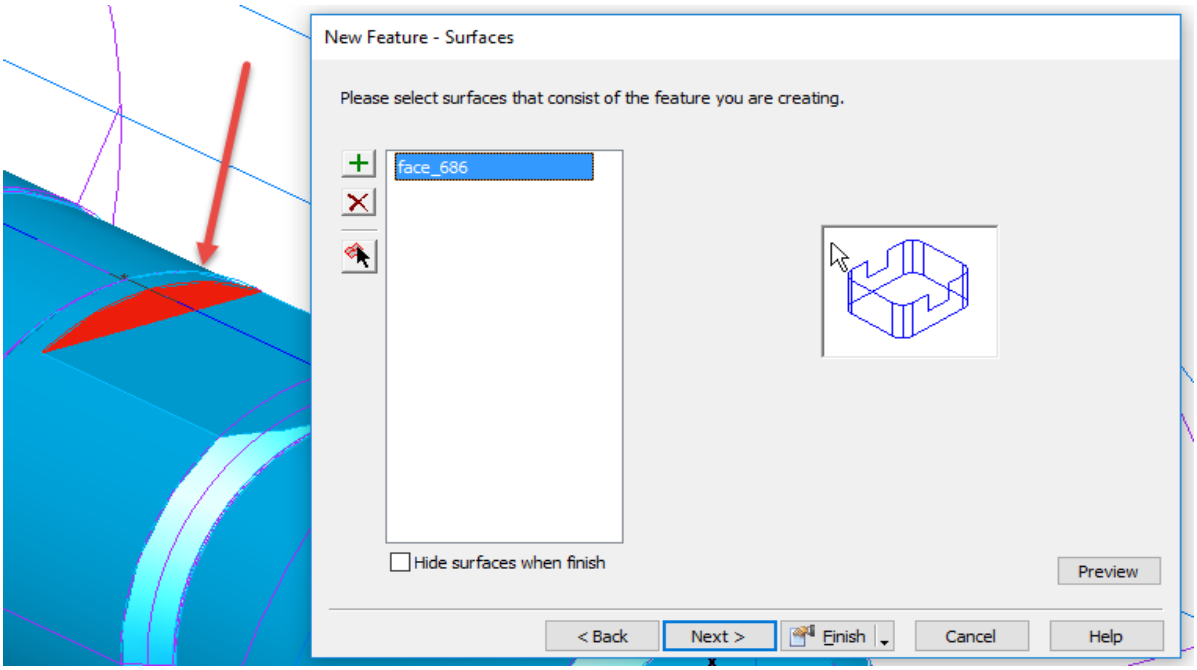
27 Please select Next.

28 Then click on **Select side surface**.



29 Pick the following **side** surface.

© Copyright 2016 Autodesk, Inc. All rights reserved



30 Select Next and make sure the arrow is pointing towards the **Main Setup**.

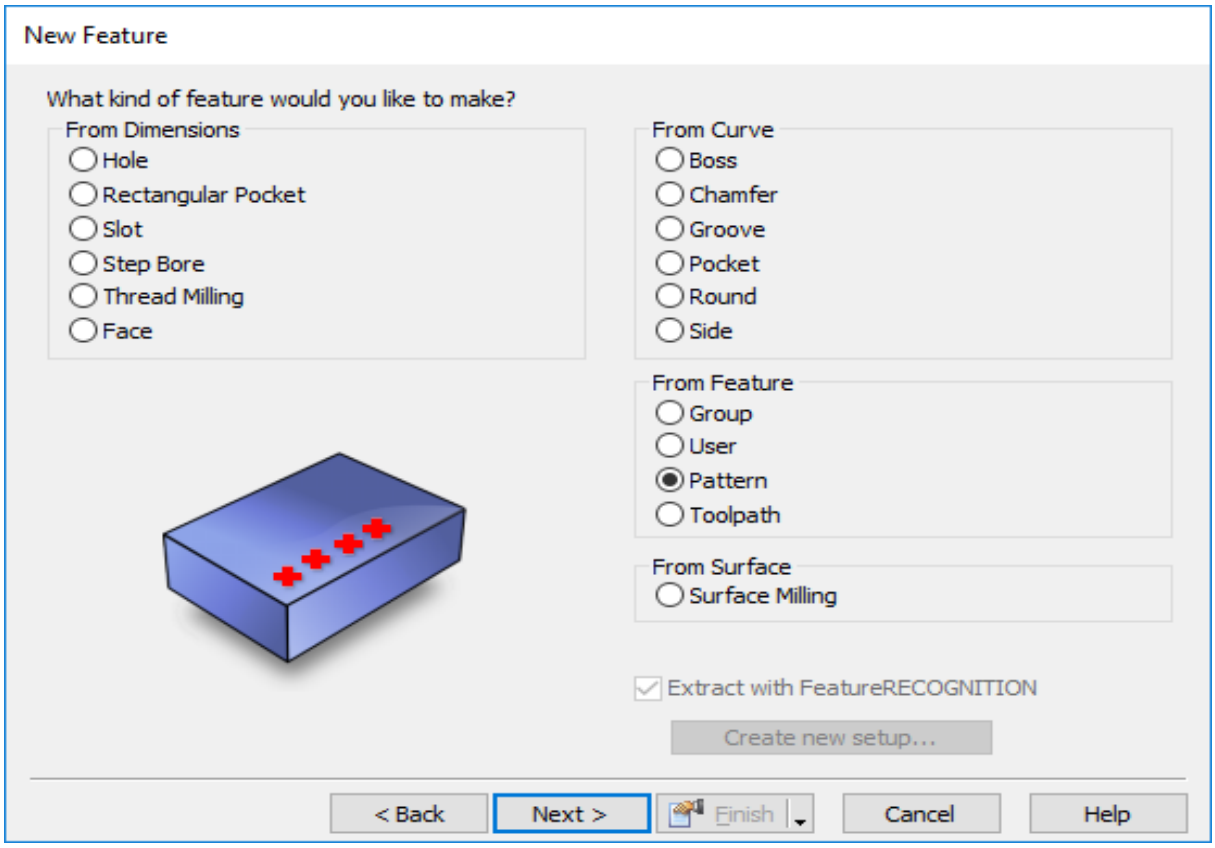
31 Select **Next.** The height is correct as it has taken this off the model.

32 Select **Finish**.

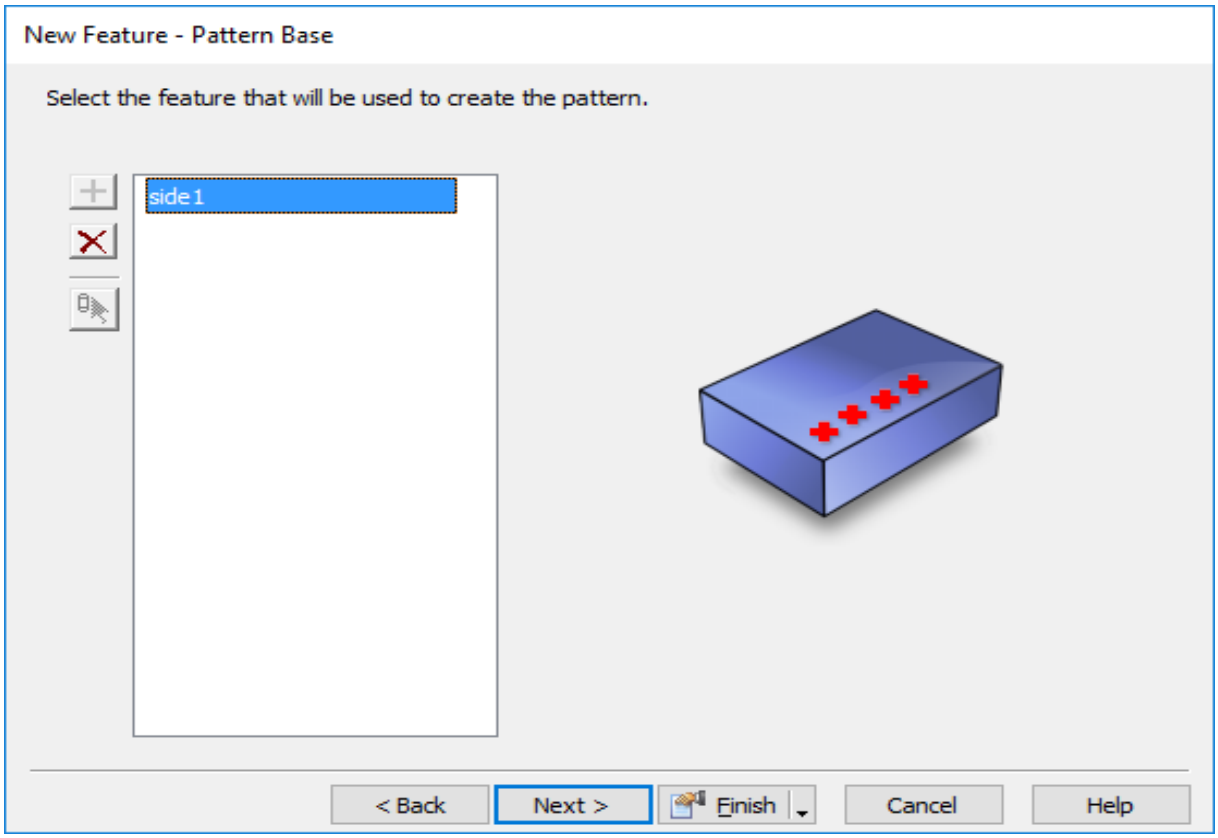
Create a Pattern of the Side Feature.

33 Select **Ctrl + R** and select **Turn/Mill**.

34 Select **Next**. Select **Pattern**. Select **Next**. Then select **Side1** from **PartView**.



© Copyright 2016 Autodesk, Inc. All rights reserved

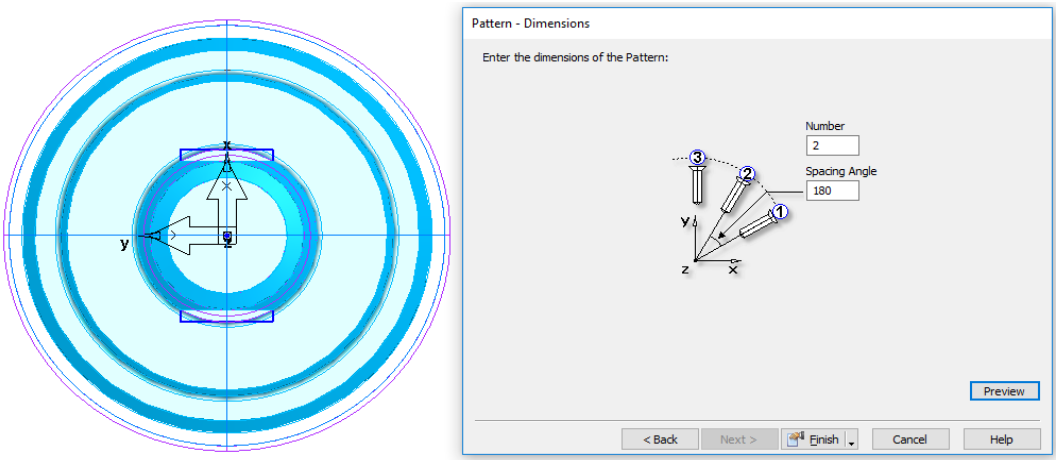


35 Select **Next**.

36 Then select **Radial around index axis**.

37 Select **Next**.

38 We want **2** which includes the original. **180 degrees** apart.



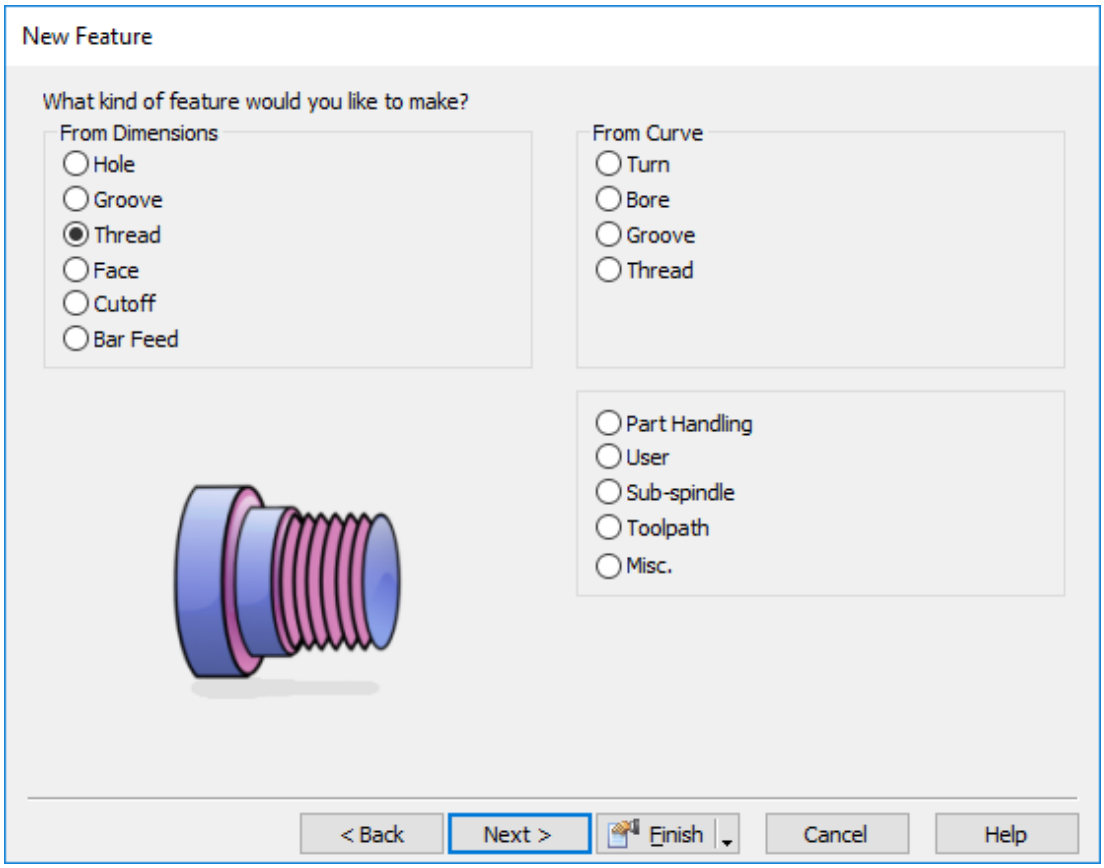
39 Select **Finish**.

Creating a Thread

What we need to do now is thread the beginning of the shaft.

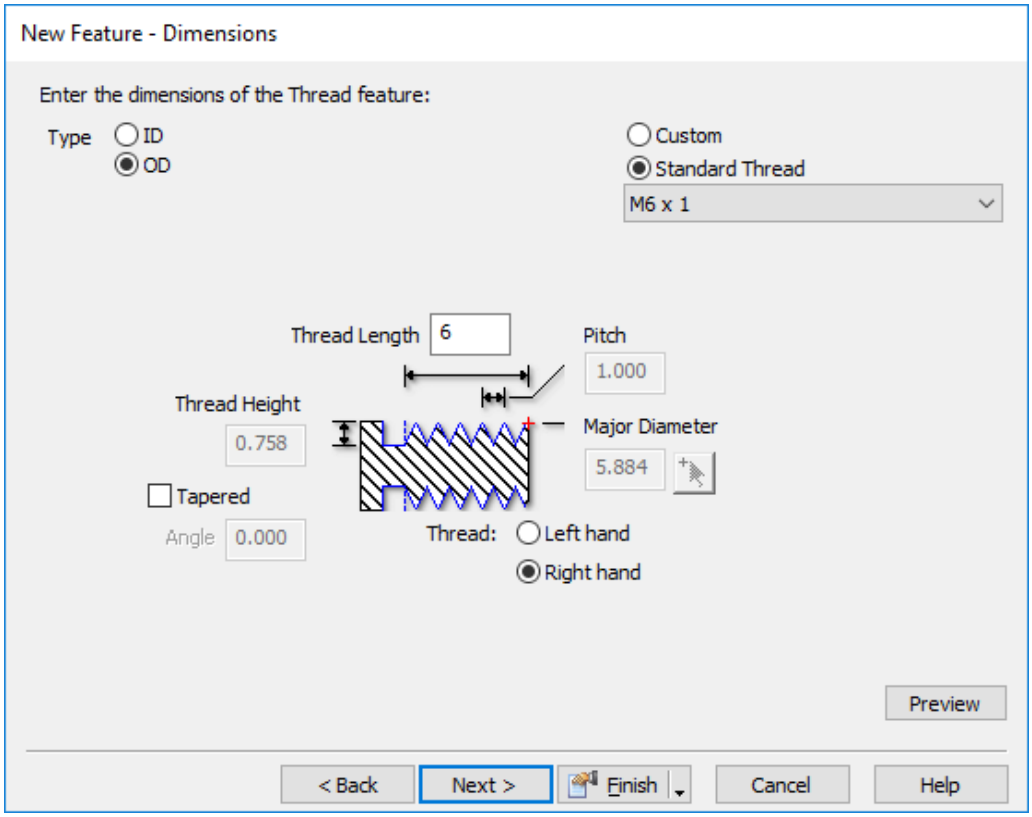
40 Create a new **Thread** Feature. Select **Ctrl + R**. Select **Turning** and then **Thread from Dimensions**.

© Copyright 2016 Autodesk, Inc. All rights reserved



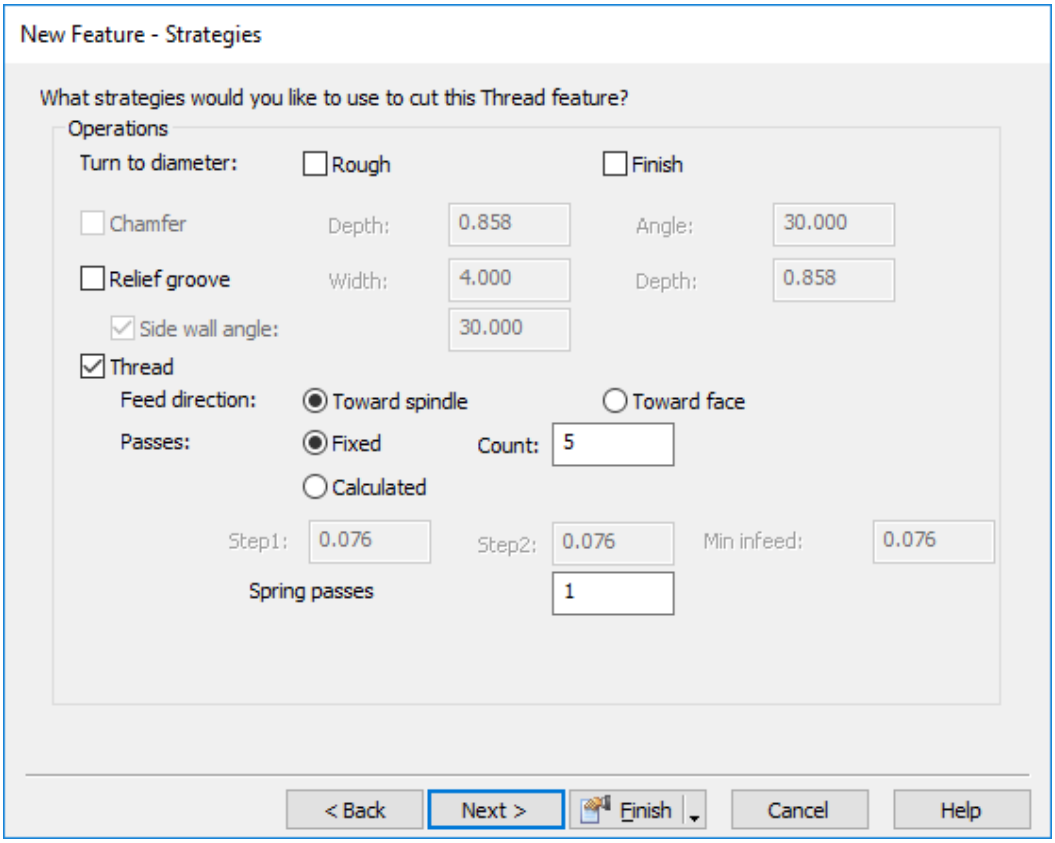
41 Select **Next**.

Set your menu the same as the one below.

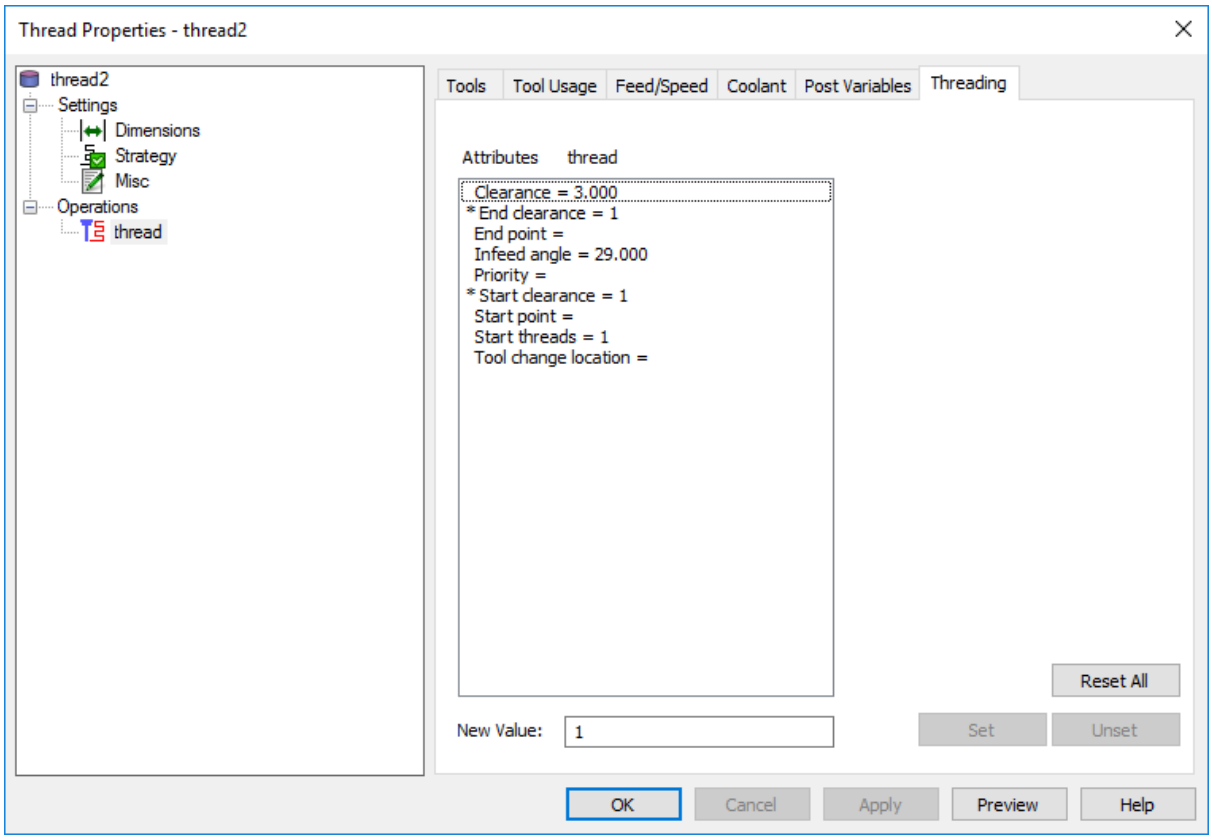


© Copyright 2016 Autodesk, Inc. All rights reserved

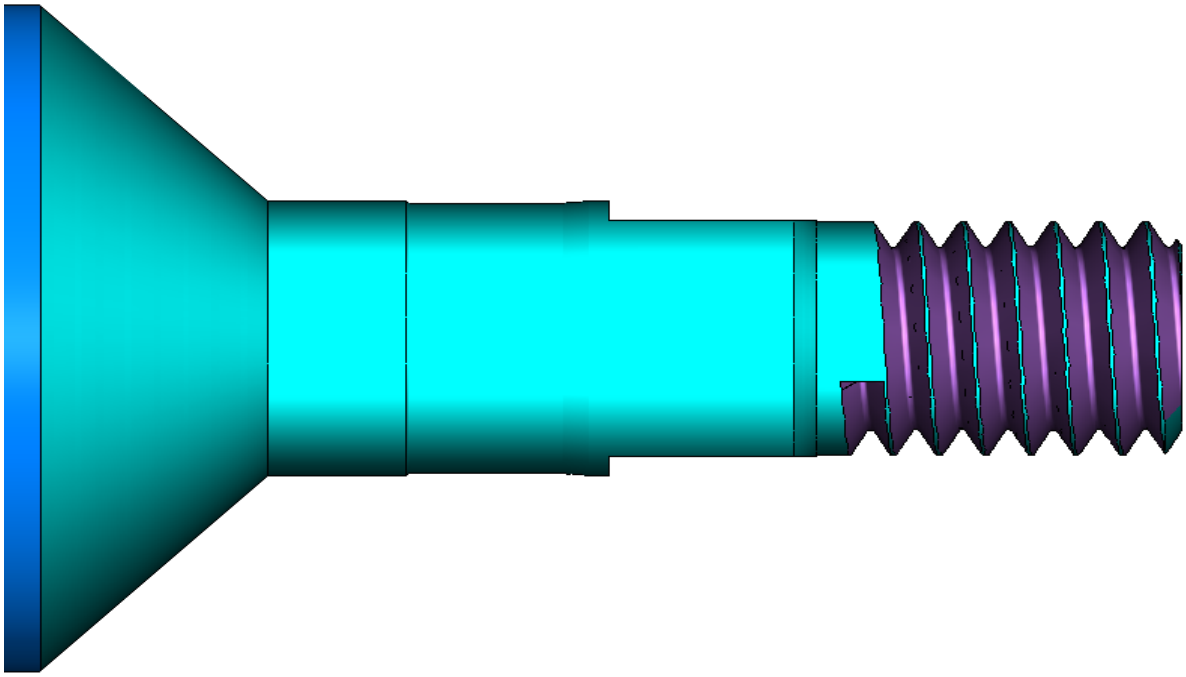
42 Select **Next twice** and un-tick the relief groove and change the **count** to **5**



*Double click on the threading operation in **PartView**. Then change the thread attributes to the following setting.*



© Copyright 2016 Autodesk, Inc. All rights reserved



We are now ready to turn the shaft up to 40mm long as this will still have enough material in the guide bush.

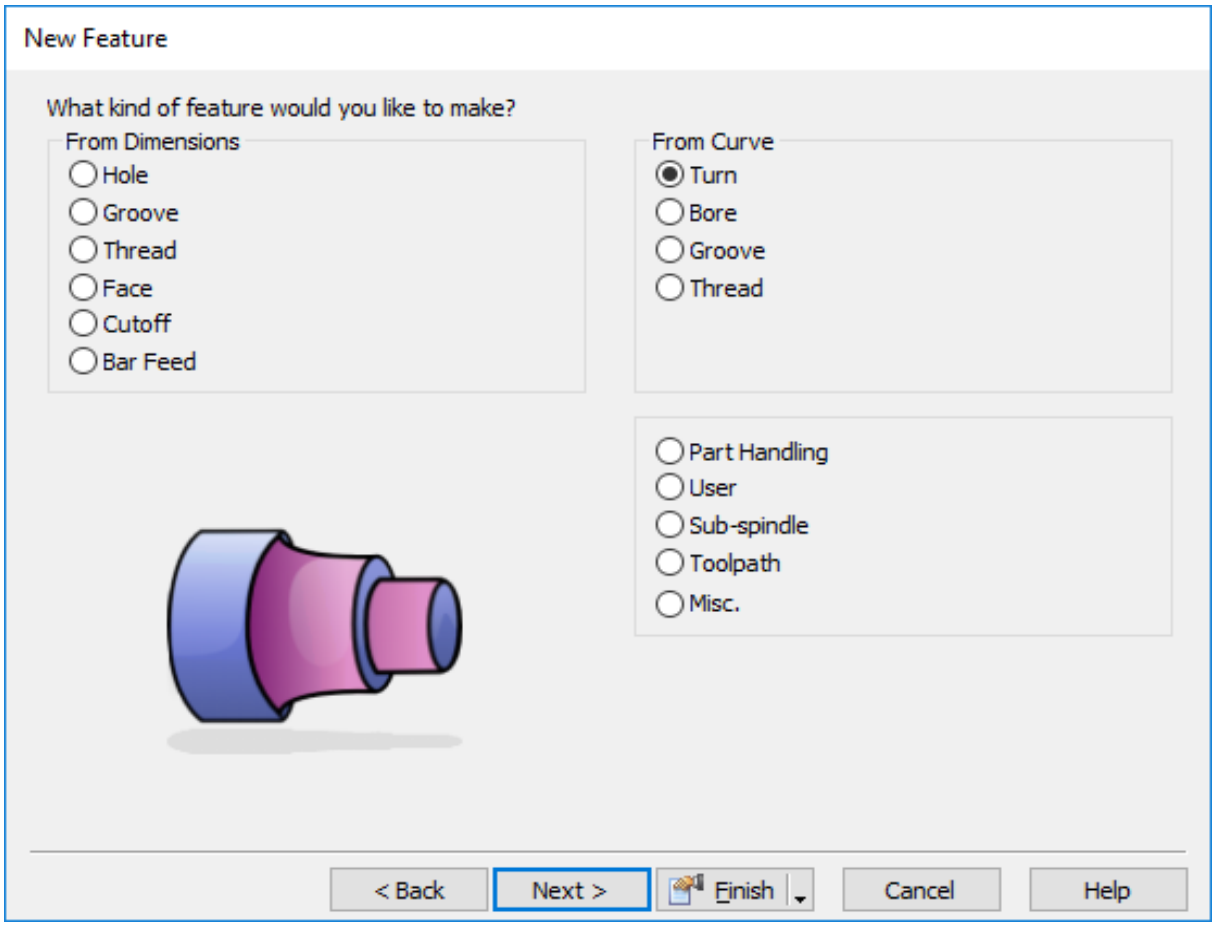
Turning up to large diameter

*We must change the order in which we machine this part. Select each individual operation in **PartView** and go to the **Misc** Tab and change the **Base priority** in the order in which you want to machine your part. Change this to 1 then 2 and so on in the order you want.*

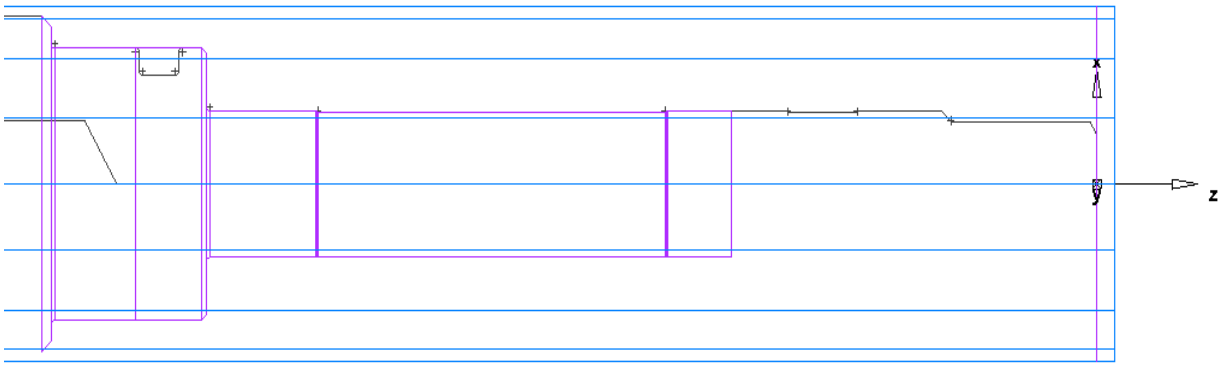
Because we want to machine up to the chamfer on the larger diameter we need to create a line across the groove. Draw a line from two points to cover over the groove.

43 Select **CTRL + R New Feature** and select **Turning**.

© Copyright 2016 Autodesk, Inc. All rights reserved



Then create a curve from the previous curve endpoint and finish at the end of the chamfer on the large diameter.

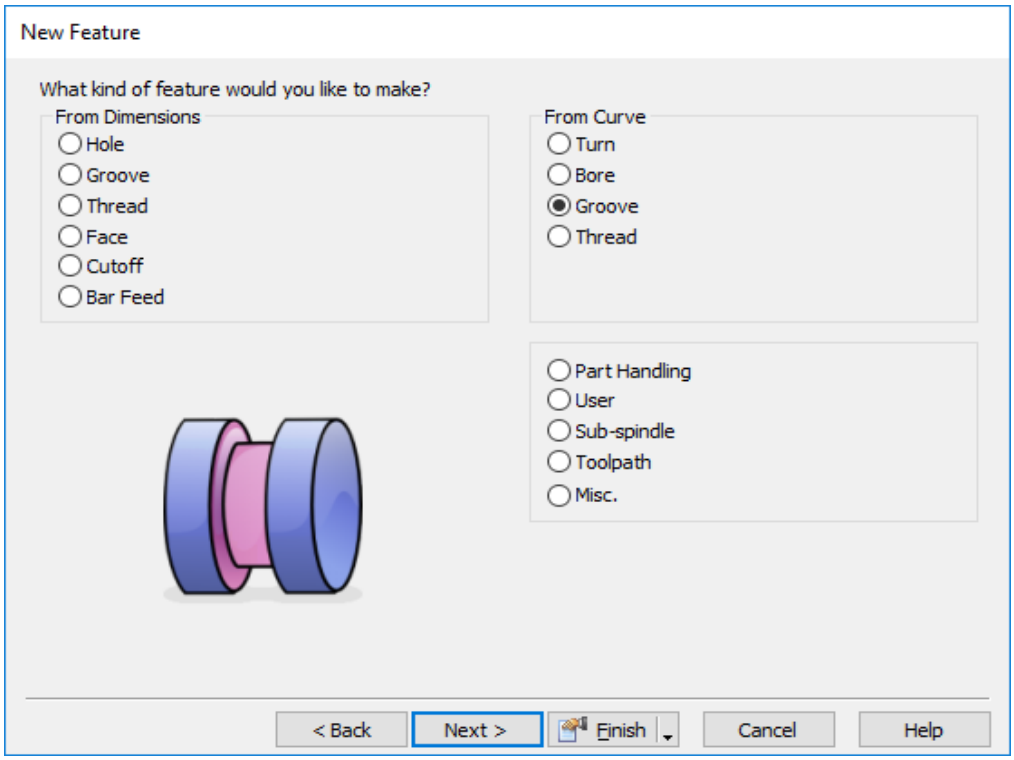


Creating a Groove feature from Curve

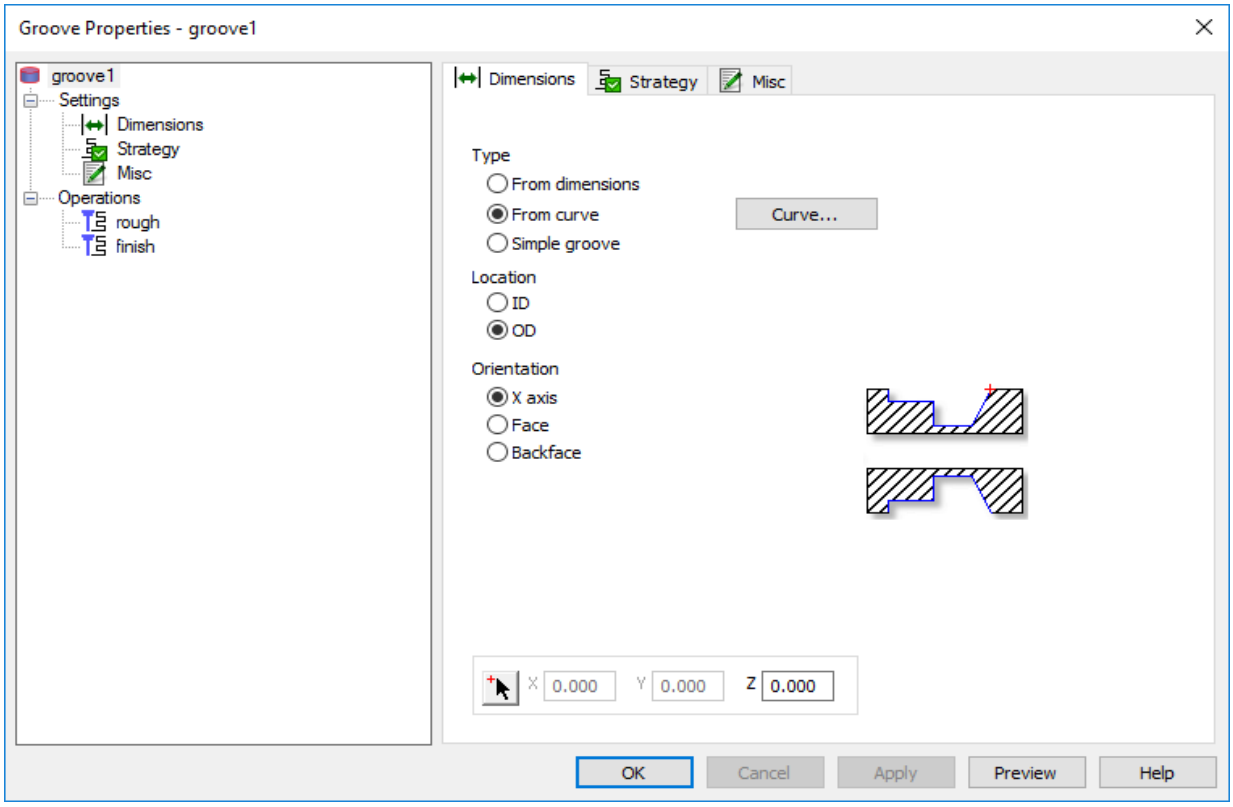
Create a Curve from the Groove Geometry and then create Groove from Curve.

44 Select **Ctrl + R** New Feature. Select **Groove from Curve**.

© Copyright 2016 Autodesk, Inc. All rights reserved



45 Select the curve you have just created. Select **Finish**.



*Use a **1mm wide grooving tool with 0.2 corner rads.***

© Copyright 2016 Autodesk, Inc. All rights reserved

Creating Hole Features around the diameter

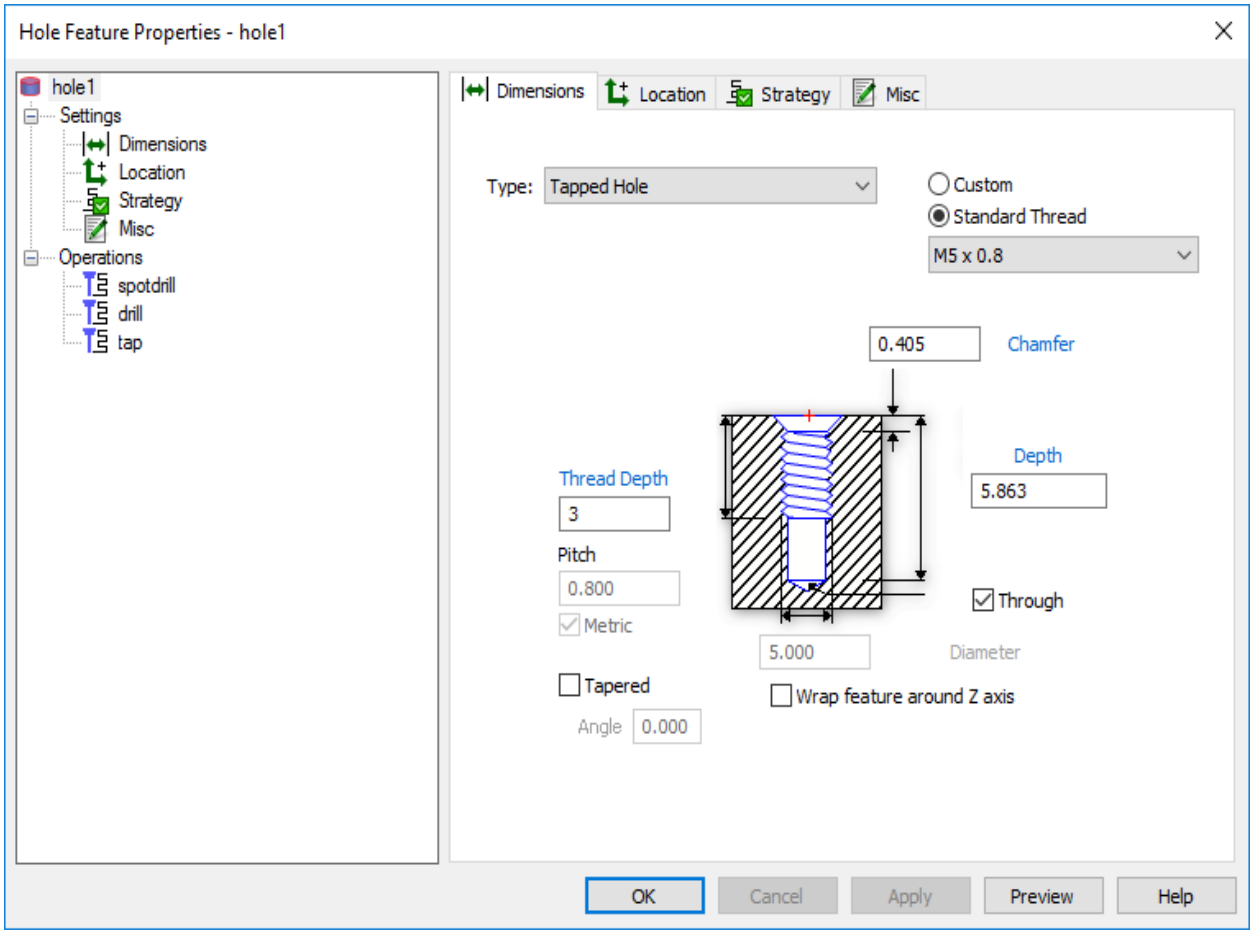
We are now going to drill and tap the 3 holes around the diameter. We will then turn the larger diameter on the sub spindle.

46 Create a new **Hole** Feature. Select **Ctrl+R Turn/Mill** and select **Extract with FeatureRECOGNITION.**

47 Select **Next**. Then select **Around the index axis** and **Automatic**.

48 This will automatically extract the holes from the solid model.

*The only thing you have to do is change the hole type to Tapped Hole. Change the thread depth to 3mm and select the Standard thread to **M5 X 0.8** as shown on the next page.*



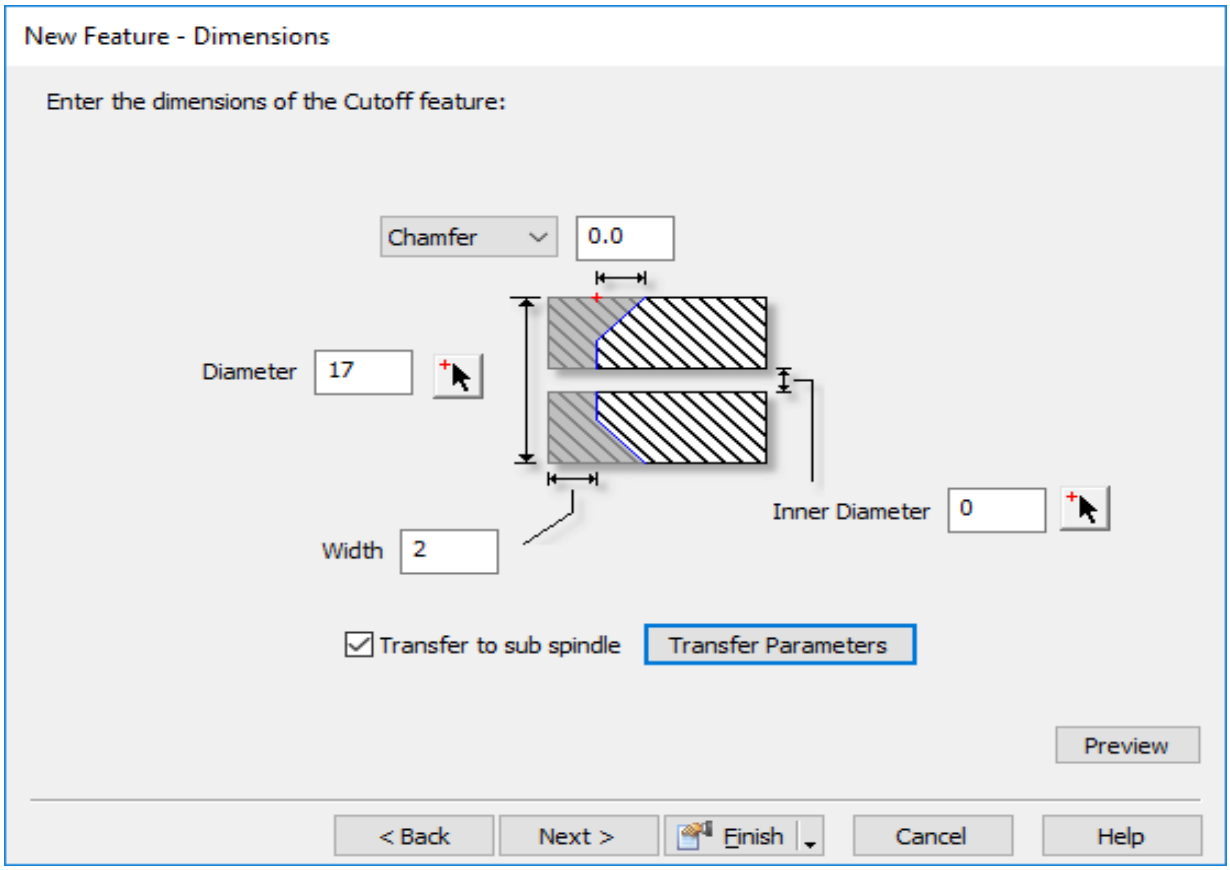
Creating a Cutoff feature

What we are going to do now is to Cut-off the part. But at the same time we need to use the Sub-Spindle to grab the part ready for the cut-off. This option is available in the Cut-off sequence menu.

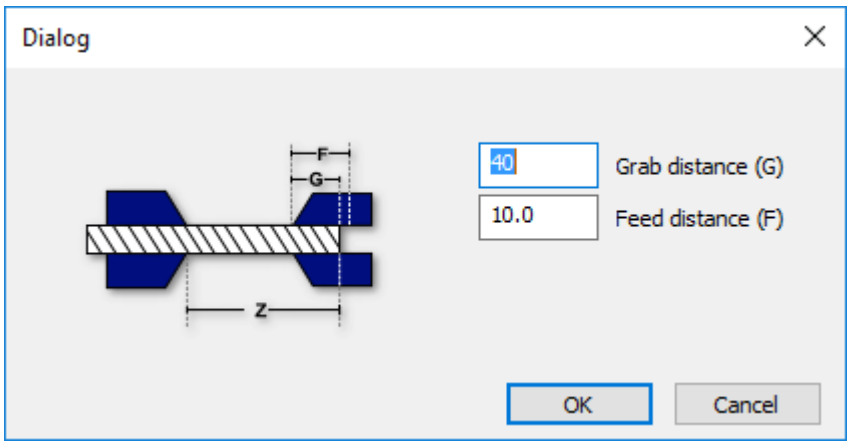
49 Create a new Feature **Ctrl+R New Feature**. Select **Turning**.

50 Select **Next**. Then select **Cutoff** from the menu. Select **Next.**

© Copyright 2016 Autodesk, Inc. All rights reserved



51 Select **Transfer to Sub-Spindle**. Then select the **Transfer Parameters**. Change this to the **following. Grab distance=40mm**



52 Select **OK** and then **Finish**.

Sub – Spindle Turning

53 Select **Sub – Setup** in **PartView.**

54 Extend the geometry for the OD Sub Turning so it goes slightly past the Chamfer.

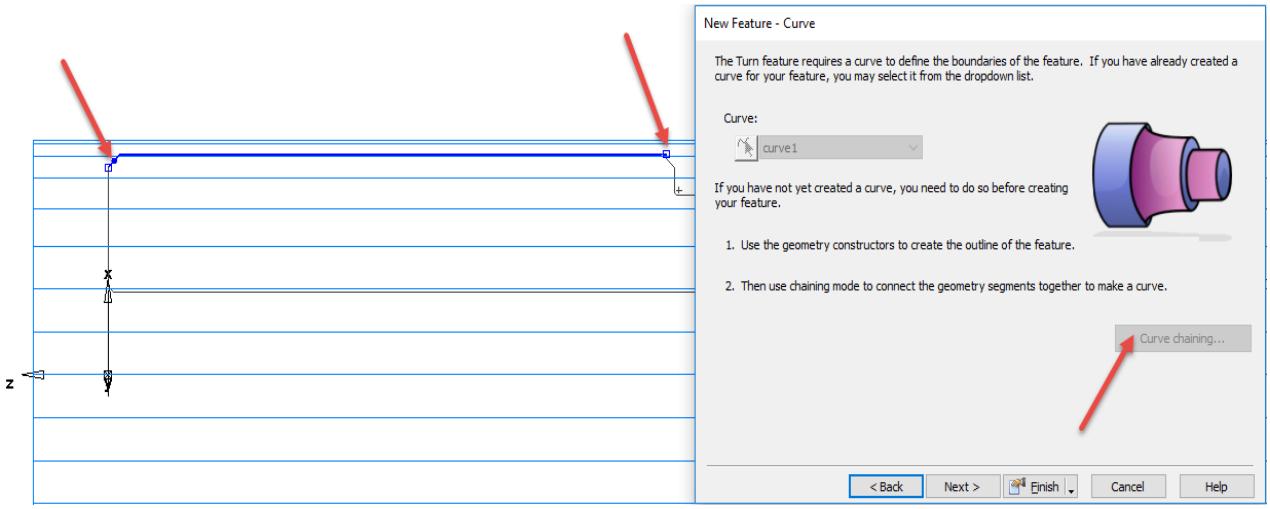
Create a Turning operation

55 Create a new Feature **Ctrl+R New Feature**. Select **Turn.**

56 Select **Next.**

© Copyright 2016 Autodesk, Inc. All rights reserved

57 We have not created a **Curve** yet but we can do this using **Curve Chaining**.



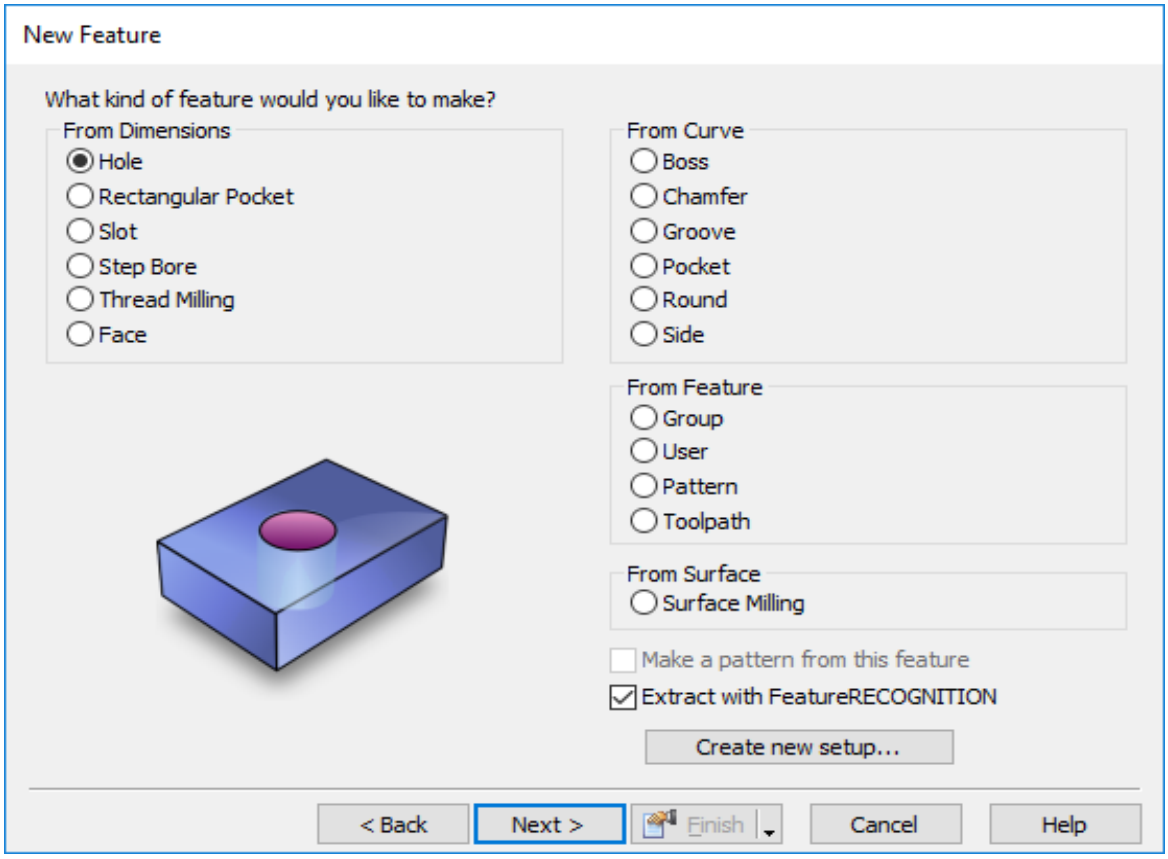
58 Select the points shown. If the wrong tool is selected then you may have to select the Automatic tool orientation option in your Machine Configuration or Machining Attributes. This will automatically orientate the tool depending on which face you are working on.

Sub – Spindle Back holes

The next operation will be to extract the holes using the Back Tool Post on the Sub Spindle.

59 Create a new Feature **Ctrl+R New Feature**. Select **Turn/Mill**

60 Select Hole with Feature RECOGNITION.



© Copyright 2016 Autodesk, Inc. All rights reserved

61 Select **Next** and then select the **Along the Setup Z-axis**.

62 Select **Next.**

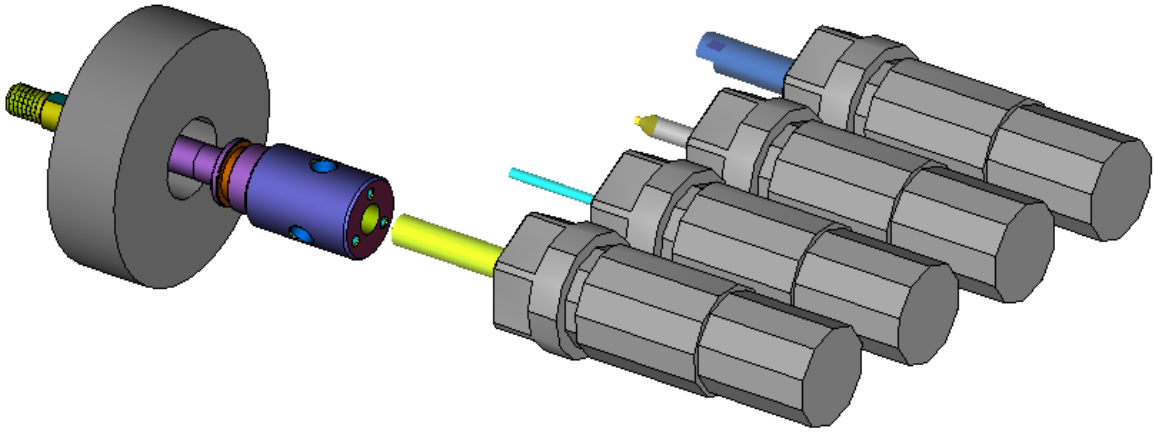
63 The default option will be to **Recognise and Construct multiple holes**.

This will extract all holes and Chamfers

64 Select **Finish** to accept the current selection.

65 Or **Click** on **Select All** to select all the holes that have been extracted.

See finished Part below.

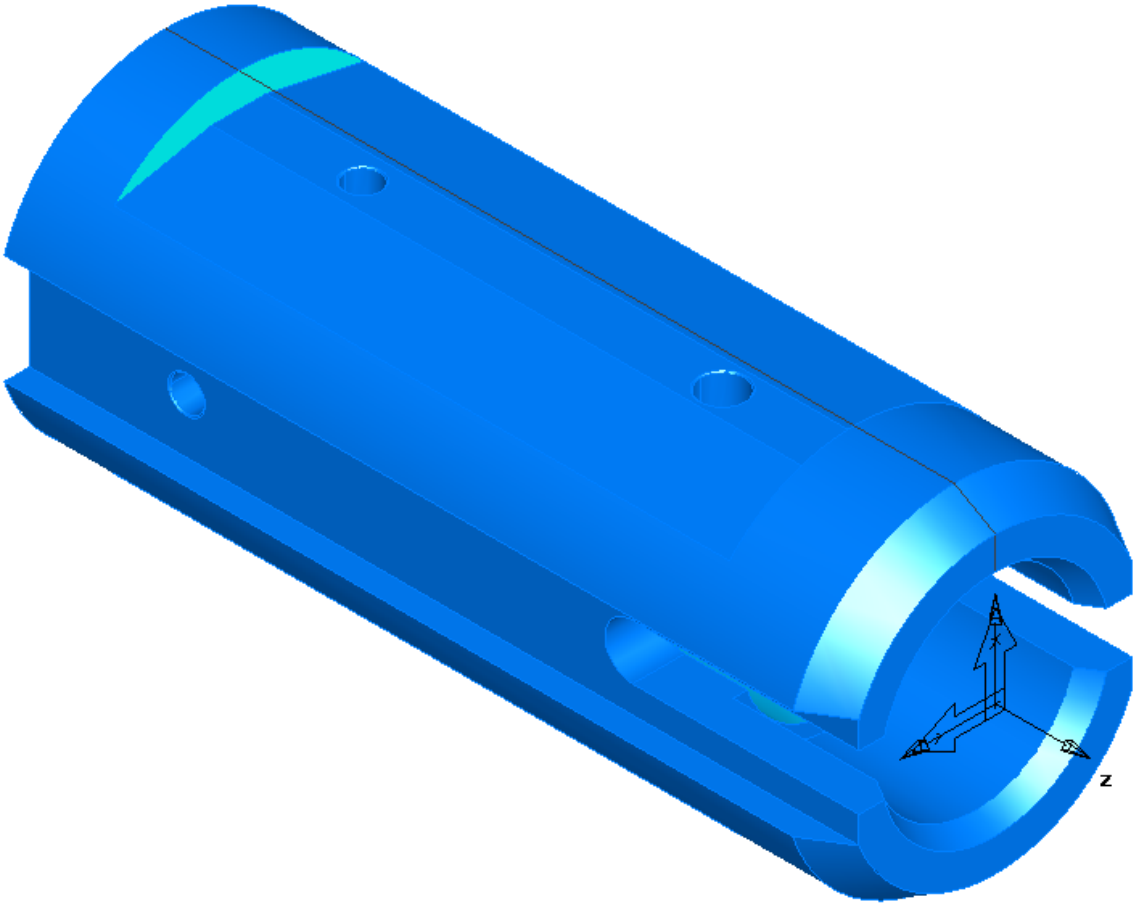


Output NC

*To output code remember to select **Save NC** from the **File** Menu.*

© Copyright 2016 Autodesk, Inc. All rights reserved

Swiss AFR example - 2



For this example the outside diameter is finished as we are using 20mm diameter bar.

- The following information is located in folder.
- C:\Training_Data\FeatureCAM Course Data 2017
- Post Processor = Citizen L20E-IX.cnc
- Machine Design file = Citizen L20E-IX - 16mm.md
- Crib = Swissmetric
- Solid model = FeatureCAM- Swiss-Example-2.x_t
- Tool Change position from post Options

  Tool Change Location (diameter)

  X 25. mm Z 5. mm

-
- Remember to select Automatic tool orientation in Machining Attributes or Machining Configuration.

© Copyright 2016 Autodesk, Inc. All rights reserved

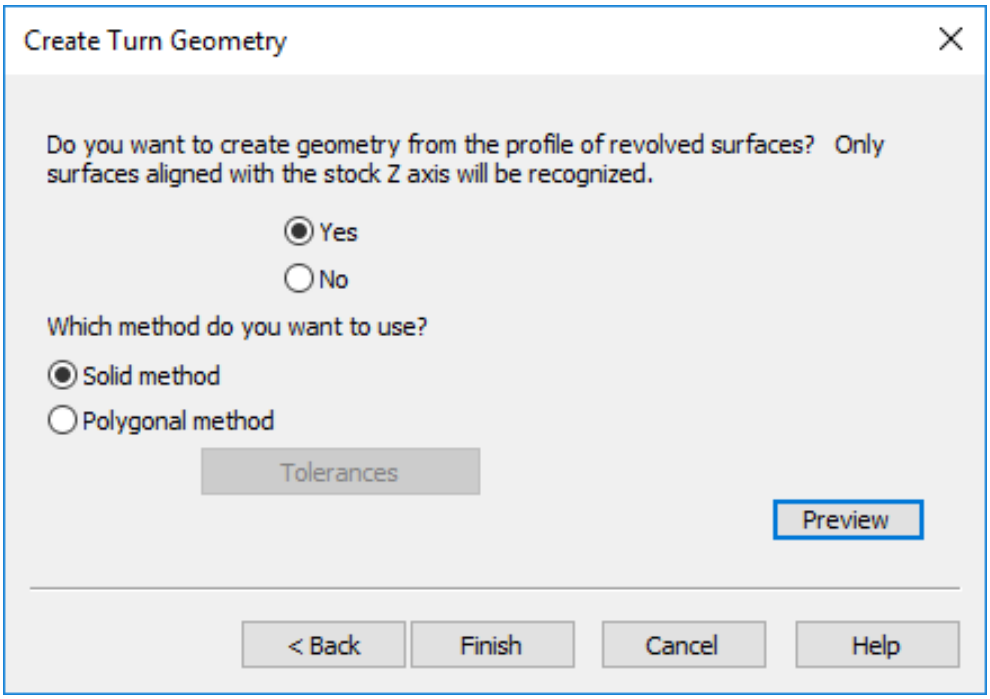
Getting Started Part-2

1. Open a new session of FeatureCAM and select **Swiss Turning** as the machining module. We are using **Metric** Units.
2. Cancel the **Stock dimensions** wizard.
3. Import the Solid model **FeatureCAM- Swiss-Example-2.x_t** from **Swiss Turning files to import.**
4. Use the Wizard to align the part and to establish the initial setup location and stock size.
5. Select **Next** 5 times until you get to the **Stock Dimensions** page.
6. Select **Compute stock from the size of the part**.
7. Enter **1mm Front**, **3mm Back** and **1mm** on the **Outside Diameter**.
8. Select **Next**. Then select the **Right-hand Hand** and then put **-1mm** into the Z location field.
9. Select **Finish**.

Turned Profile Extraction

*We will use the Wizard to extract the turned profile only. Select No and enter until you get to the **Create Turn Geometry** page.*

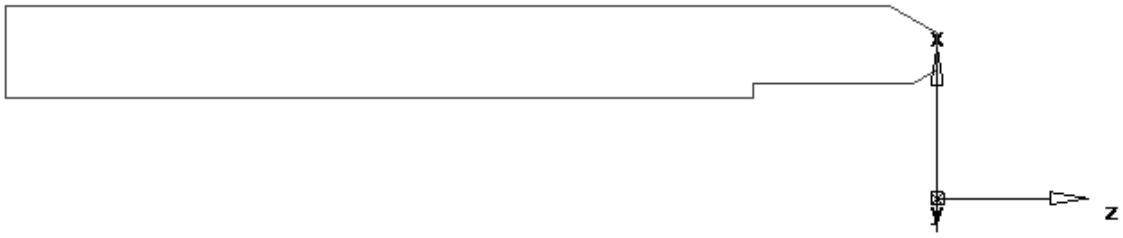
Select the following options.



10. Select **Finish**.
11. **Hide** the Solid model.

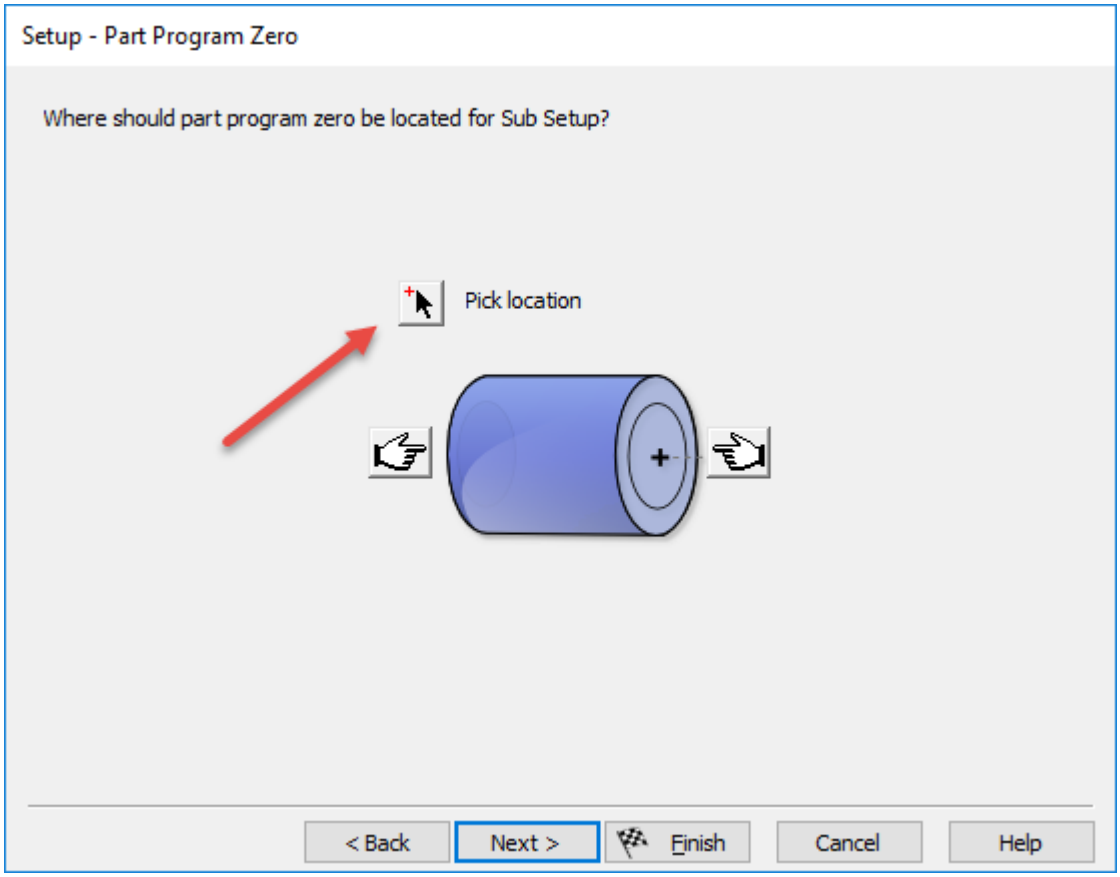
The following image shows the Turned Geometry.

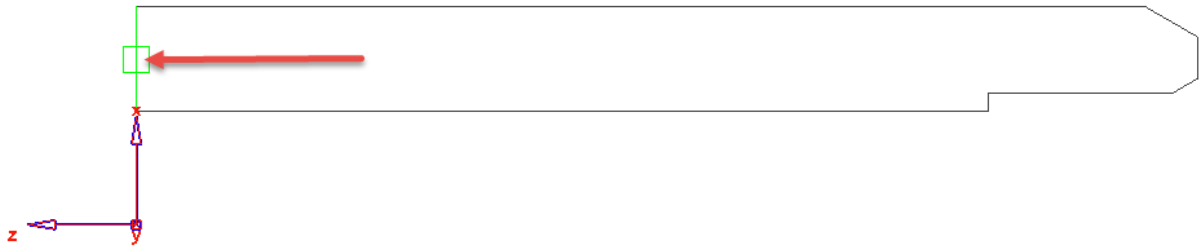
© Copyright 2016 Autodesk, Inc. All rights reserved



*We also have to setup the **Sub Setup location**. We can do this by **double clicking** on the **Sub Setup** in **PartView**.*

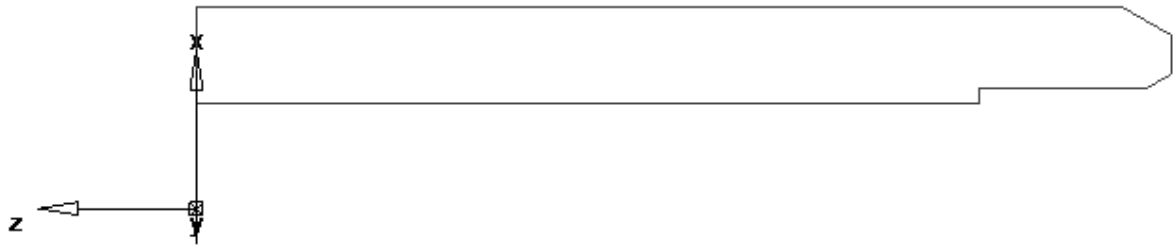
12 Double click on **Sub Setup** and select **Edit**. Select **Next** until you get to the option **Align to Stock Face.** Select this option and select **Next.** Then select **Pick location**. Pick the line as shown below.





© Copyright 2016 Autodesk, Inc. All rights reserved

13 Select **Finish** to action the command. The **Sub Setup** will then move to the correct position as shown below.

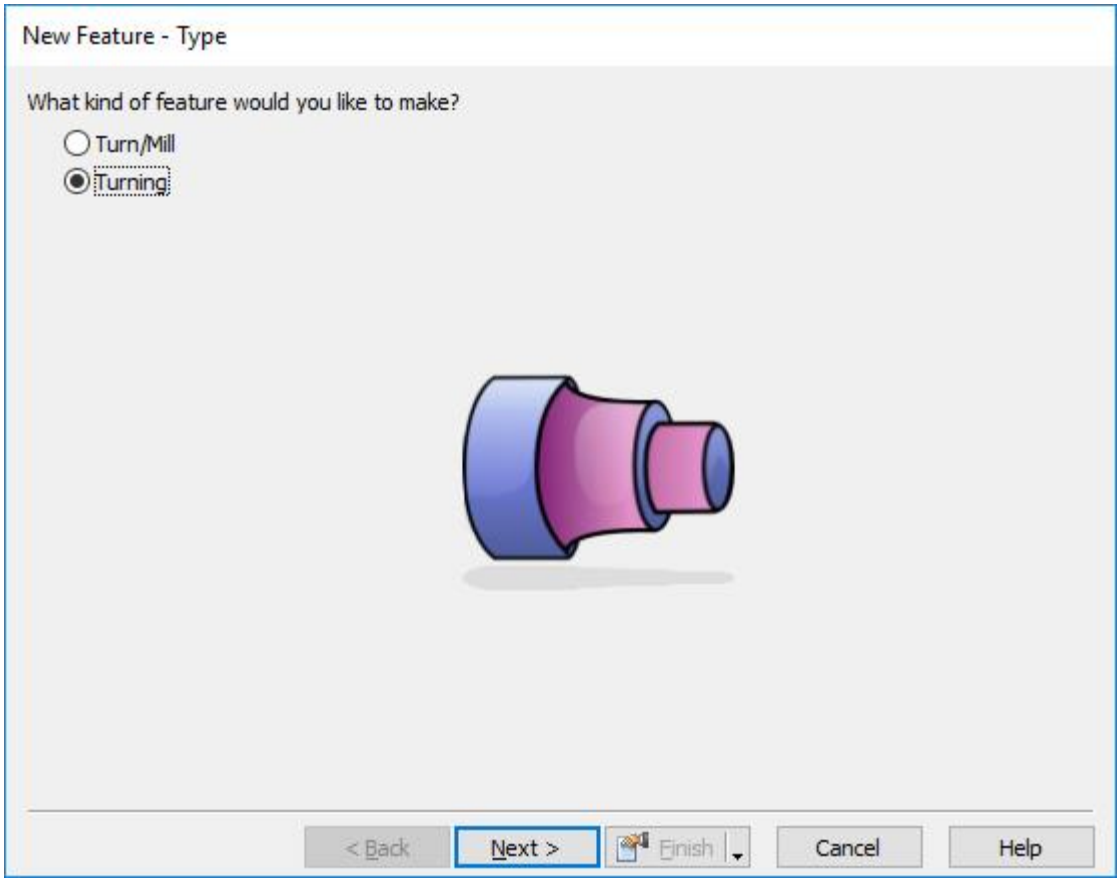


Main Spindle Turning

14 Select **Main Setup**.

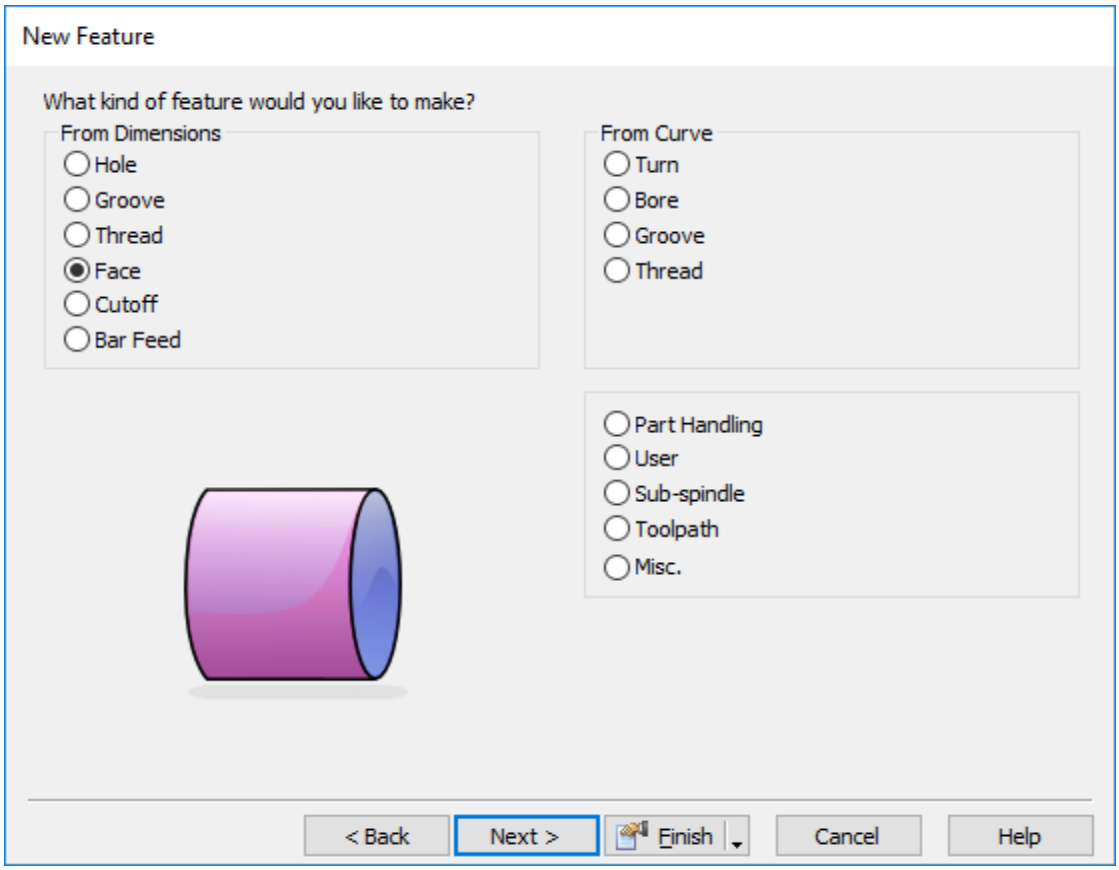
15 Create a **Facing** operation and then a **Turn** operation for the **chamfer**.

16 Select **CTRL + R** New Feature and select **Turning**.

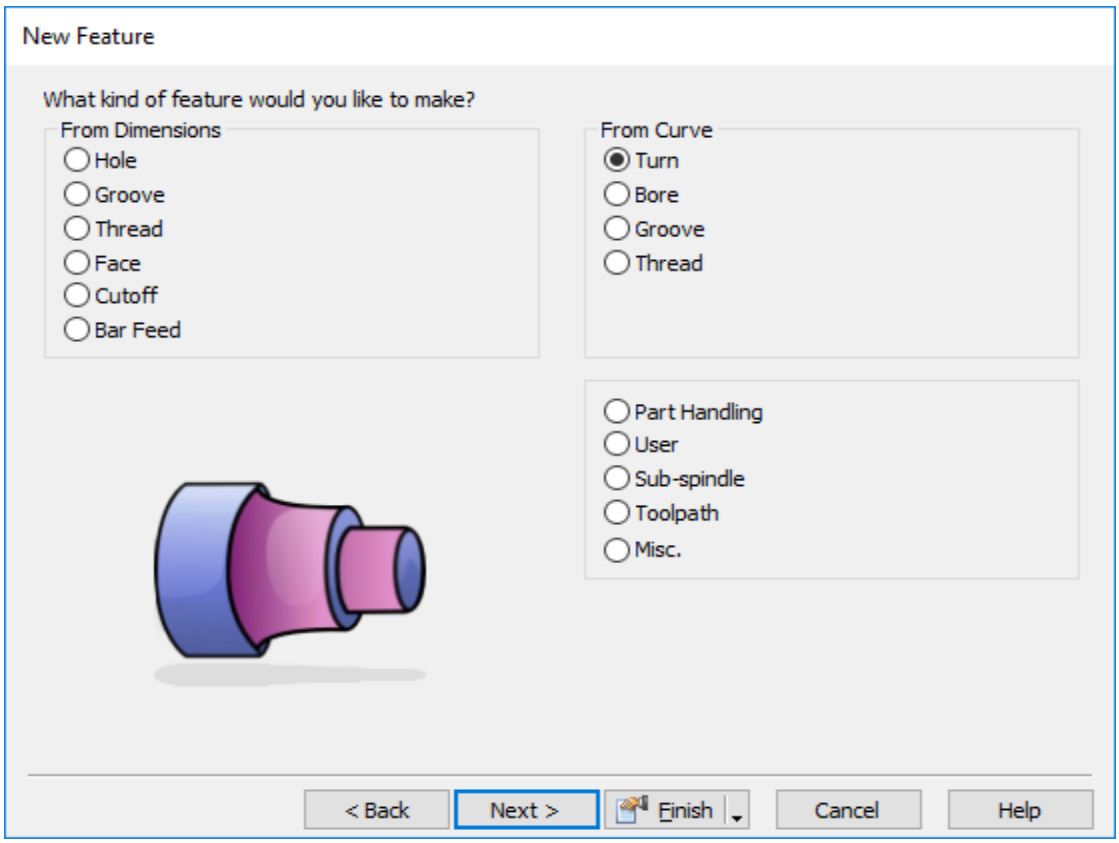


17 Select **Next**. Then Select **Face**.

© Copyright 2016 Autodesk, Inc. All rights reserved



18 Select **Finish**. Then create a new Turning feature and select the curve you have just created.



© Copyright 2016 Autodesk, Inc. All rights reserved

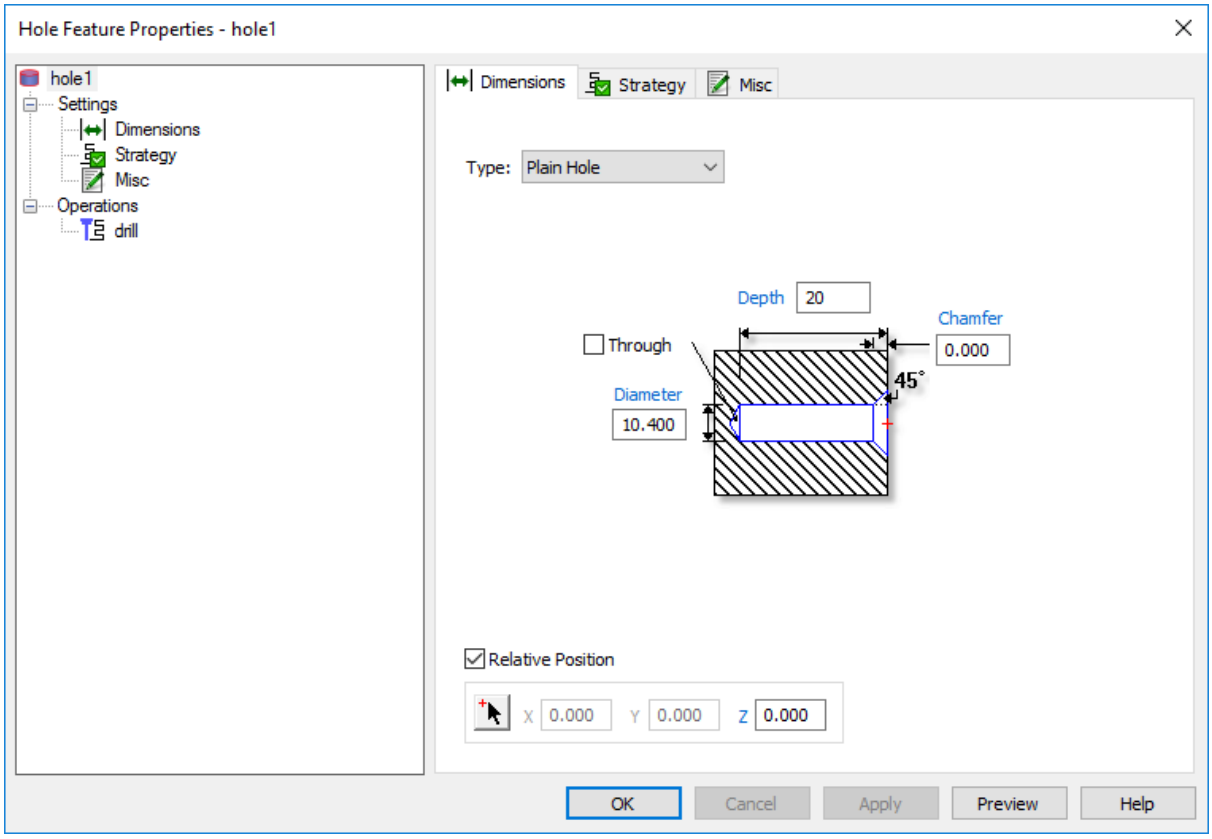
19 Select **Next** and pick **Curve chaining.** Create a **Curve** from the Front Chamfer Geometry and then select **Finish**.

20 Run a Machine Simulation

Drill 10.4mm diameter centre hole 20mm deep

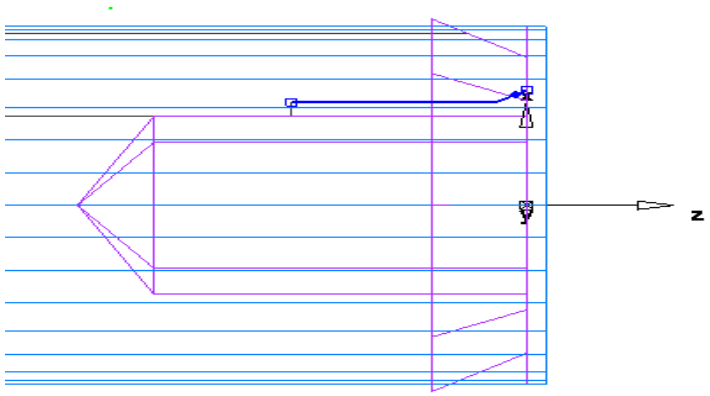
Because we are limited by the distance between the Chuck and Bush then the deepest we can drill is about 20mm. This is enough to machine the bore and relief for our slots.

21 Create a hole Feature and drill to **20mm** deep as shown below.

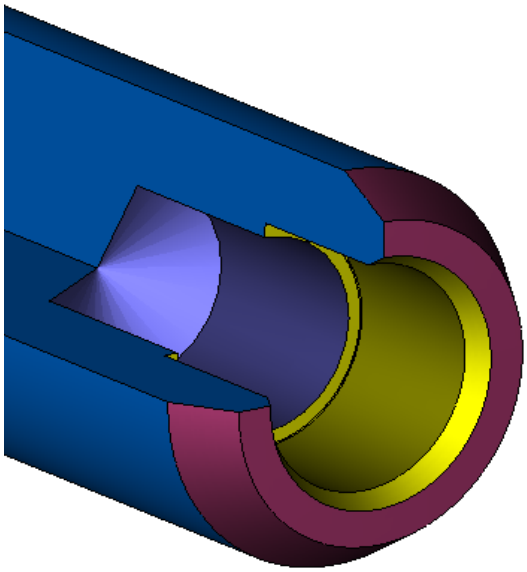


Create a Bore feature.

Use Curve chaining to create our profile.



© Copyright 2016 Autodesk, Inc. All rights reserved



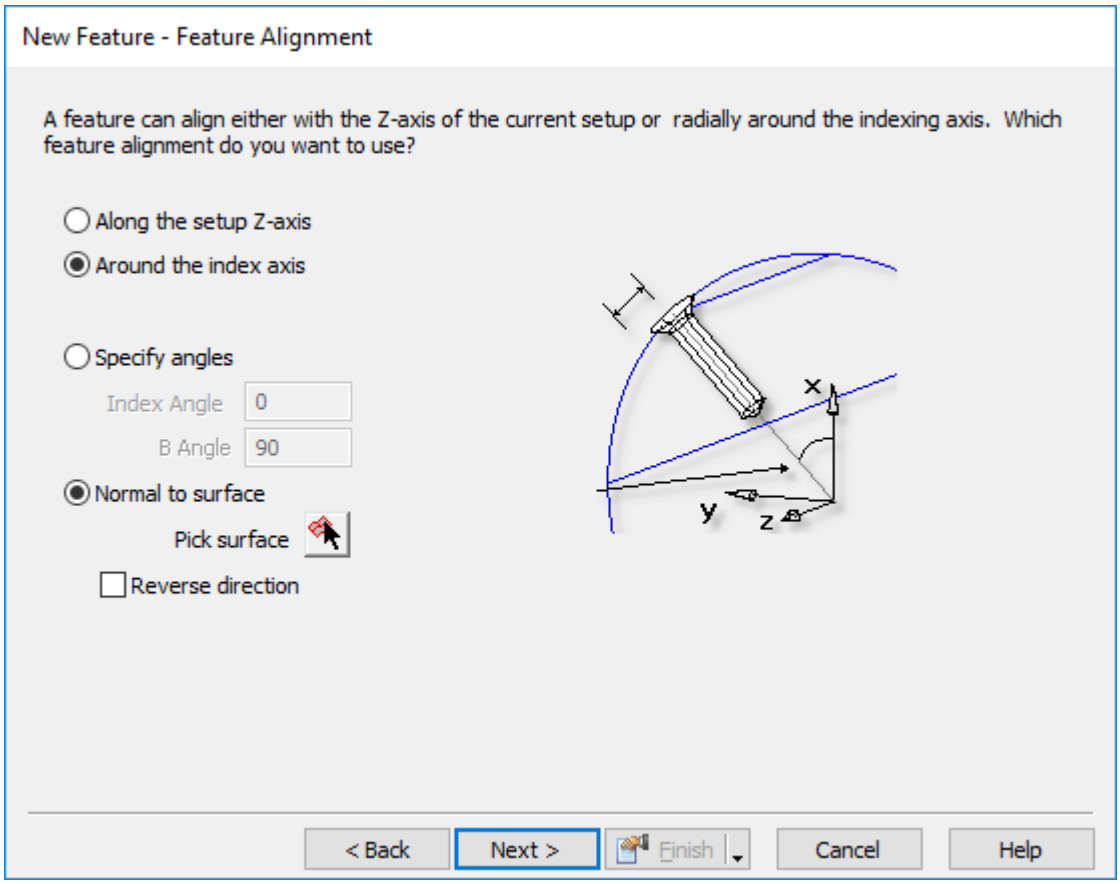
Create the 2 long slots along full length.

Create a Side Feature.

22 Select **Ctrl+R** and select **Turn/Milling** then select **Next**.

23 Select Side and Extract with Feature Recognition. Select **Next**.

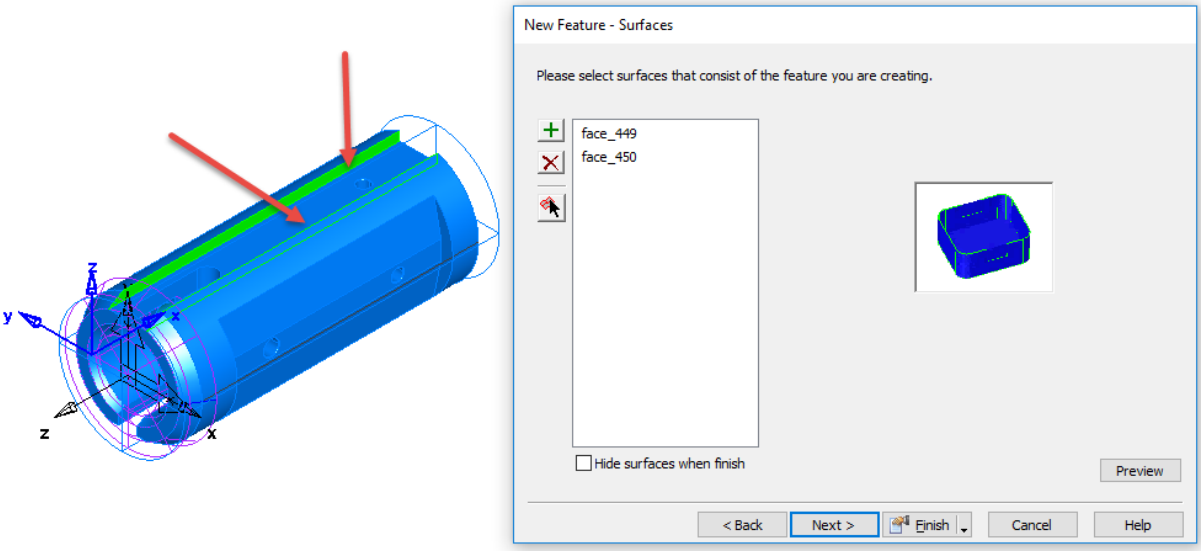
24 Please select **Around the index axis** and **Normal to surface**.



© Copyright 2016 Autodesk, Inc. All rights reserved

25 Please select the radio button **Select Side Surface** (Default)

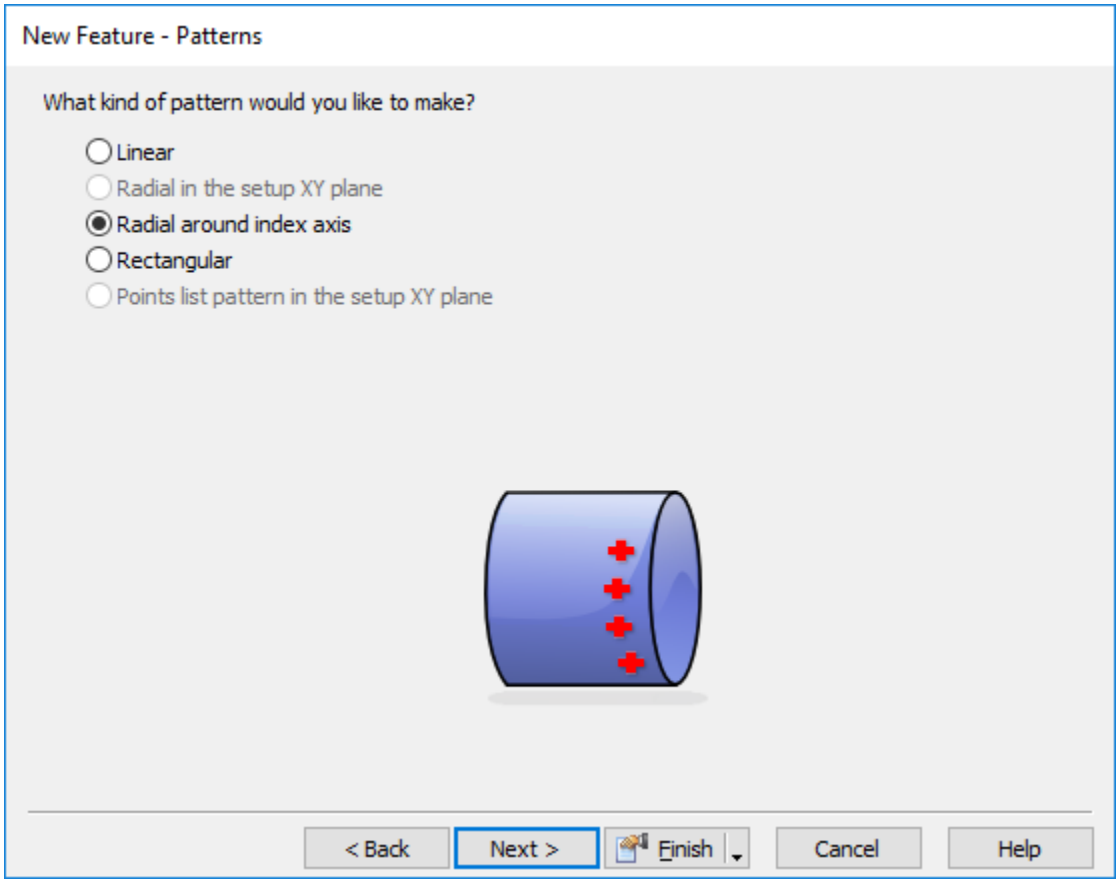
26 Select **Next**. Pick the side surface as shown below.



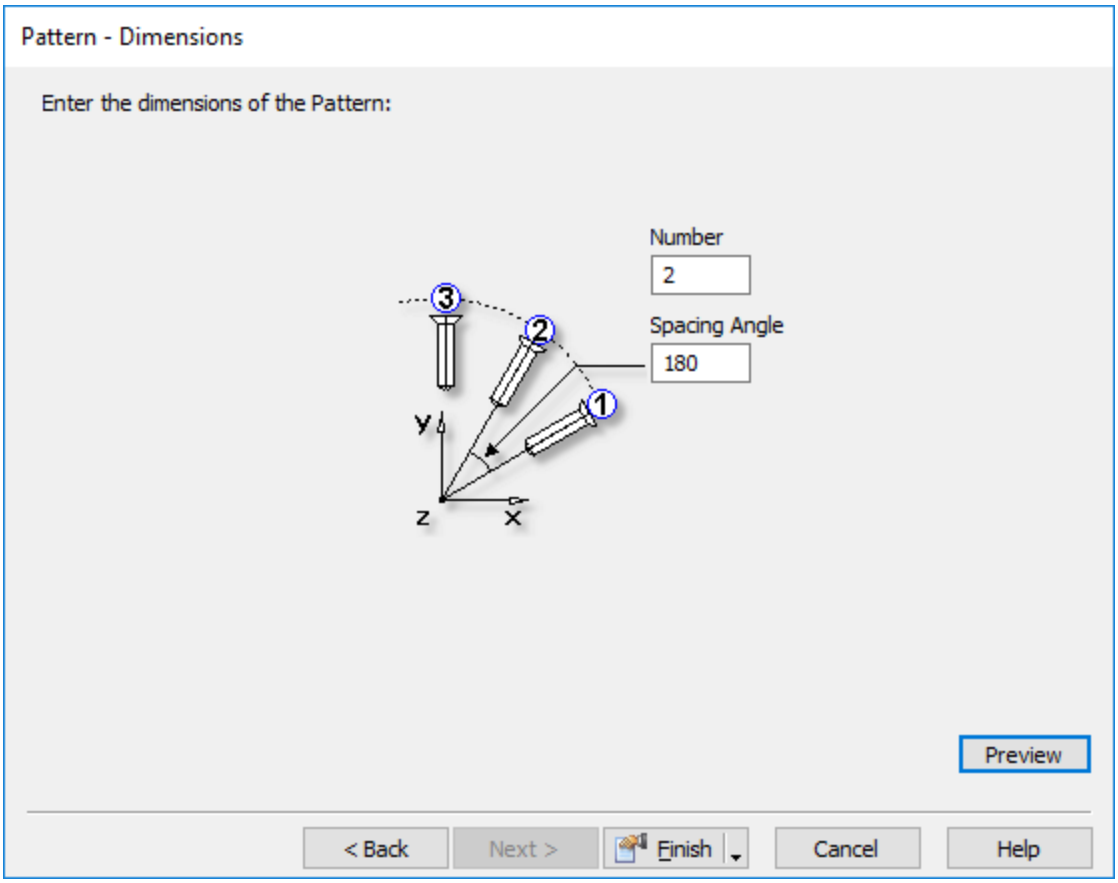
27 Select **Next** to check the side control of each face. Make sure they are both pointing inwards. Select **Finish**.

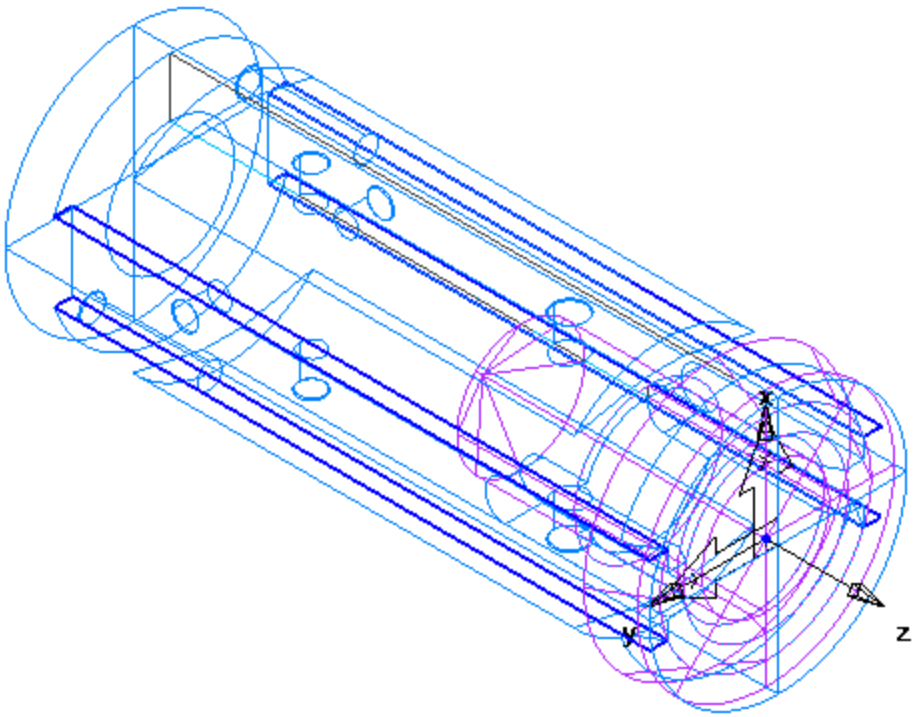
*By default the roughing tool may be too large. Make all tools the same and make them **5mm**. The width is 6.4mm. The default diameter can be changed in Machine configuration.*

*Once completed make a Pattern of the **Side1 Feature**.*

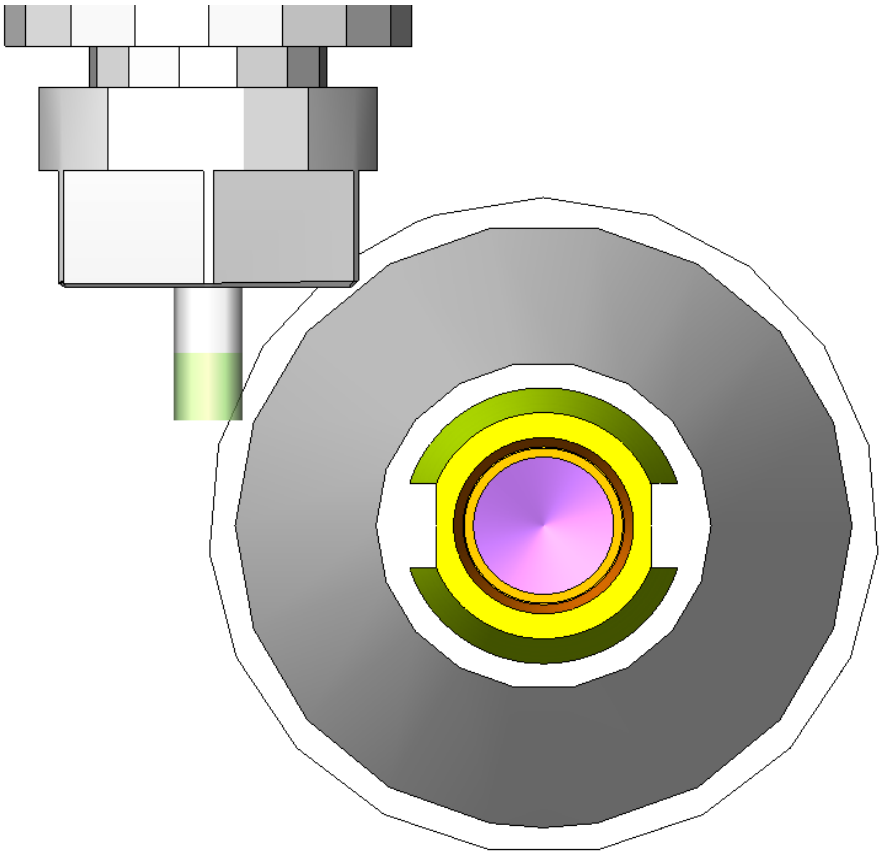


© Copyright 2016 Autodesk, Inc. All rights reserved





© Copyright 2016 Autodesk, Inc. All rights reserved



Large open side feature x 3 face planes.

We are now going to machine the 3 flats and then create a Pattern @180 degrees apart. Create another Side Feature.

You will have to create 3 Side features all using a different face plane and selecting Side Surfaces and selecting the side walls of the open slot. The following is the procedure of one Side Feature but adopt these principles to create the other two features.

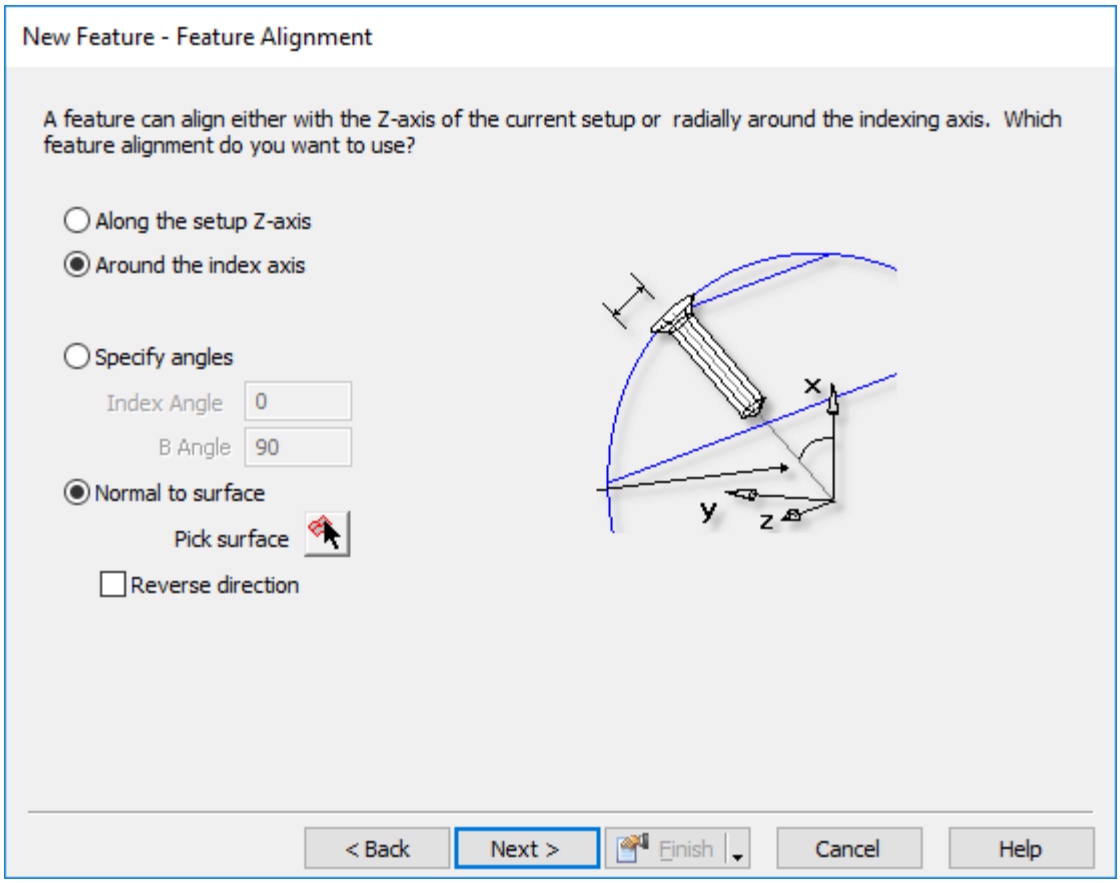
28 Select **Ctrl+R** and select **Turn/Milling** then select **Next**.

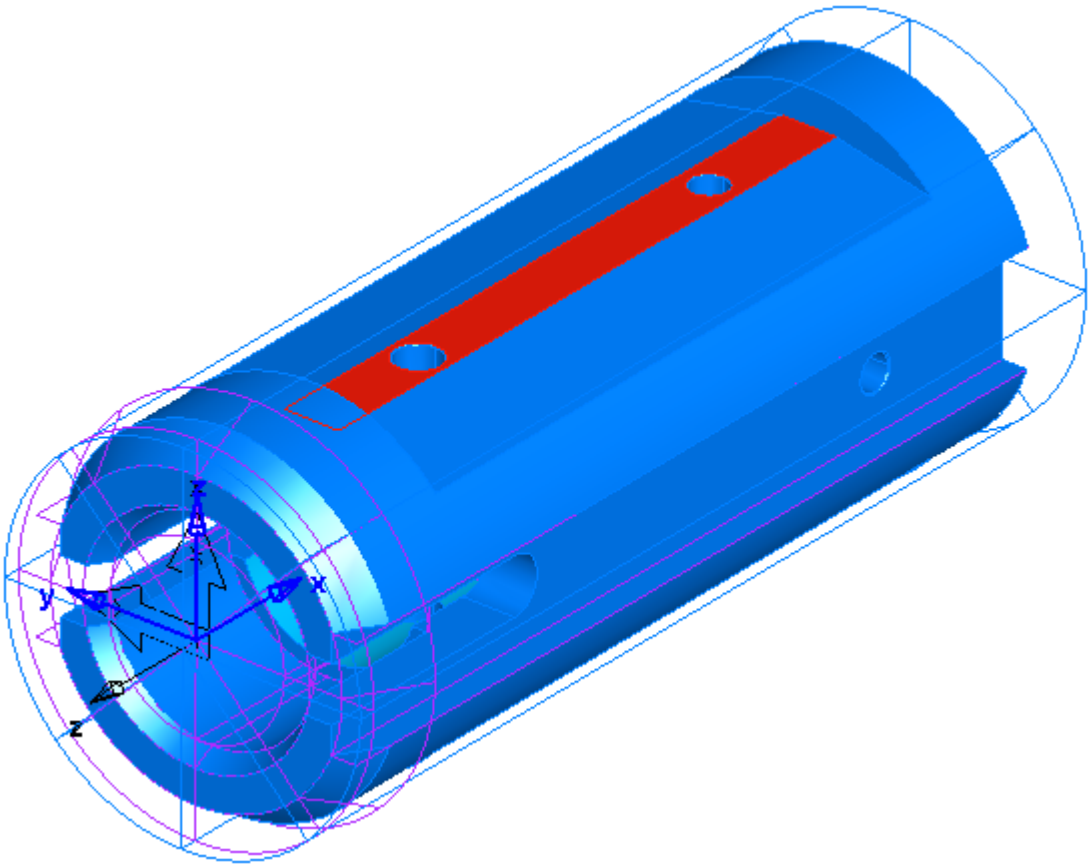
29 Select **Side** and **Extract with Feature Recognition**. Select **Next**.

30 Please select **Around the index axis** and **Normal to surface**.

31 Pick the flat surface with the two holes in from the large open slot.

© Copyright 2016 Autodesk, Inc. All rights reserved

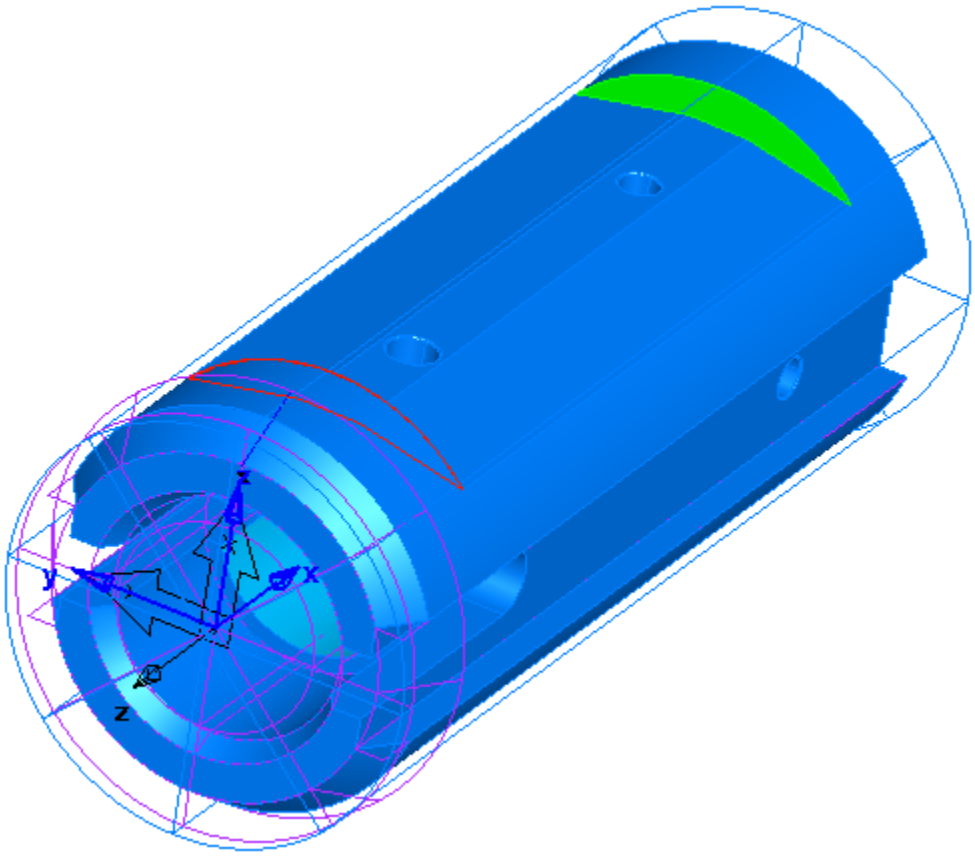


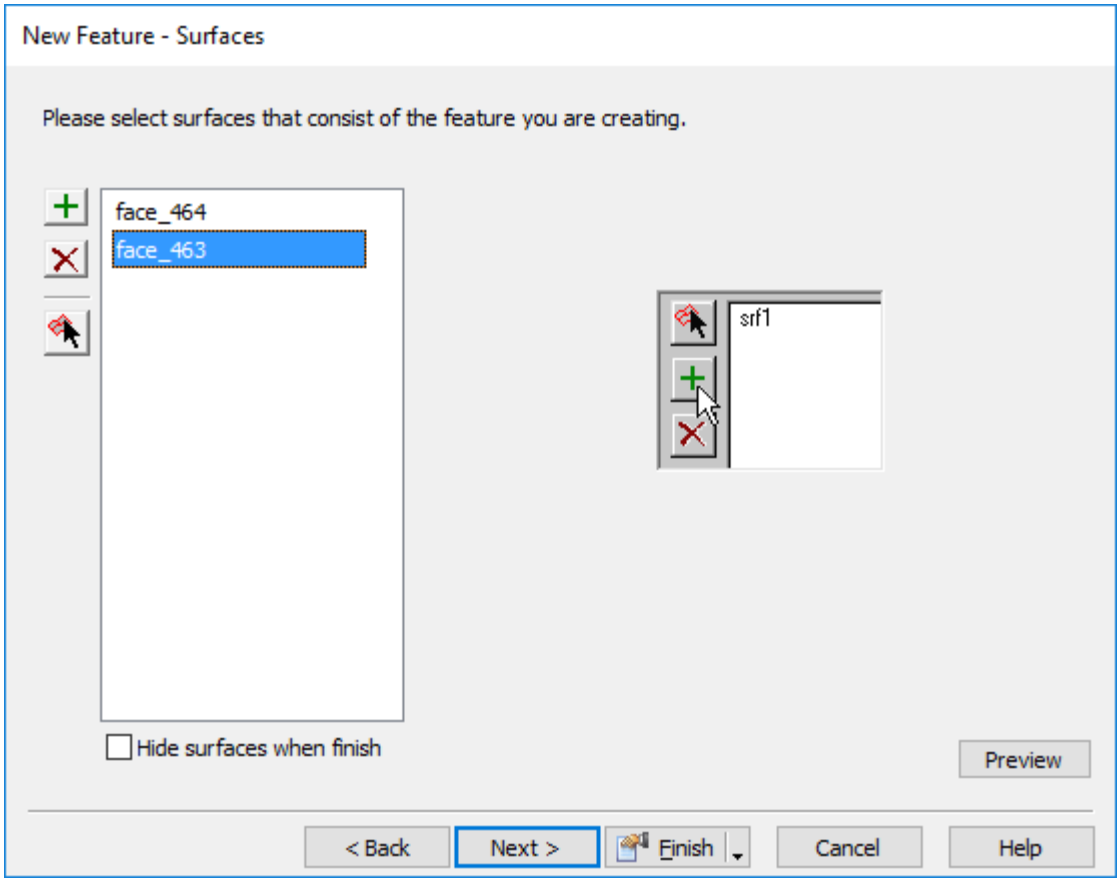


32 Select **Next**. Please select the radio button **Select Side Surface** (Default)

© Copyright 2016 Autodesk, Inc. All rights reserved

33 Select **Next**. Pick the **side surfaces** as shown below.





© Copyright 2016 Autodesk, Inc. All rights reserved

Once you have selected the surfaces add them by selecting the green + icon.

34 Select **Next**.

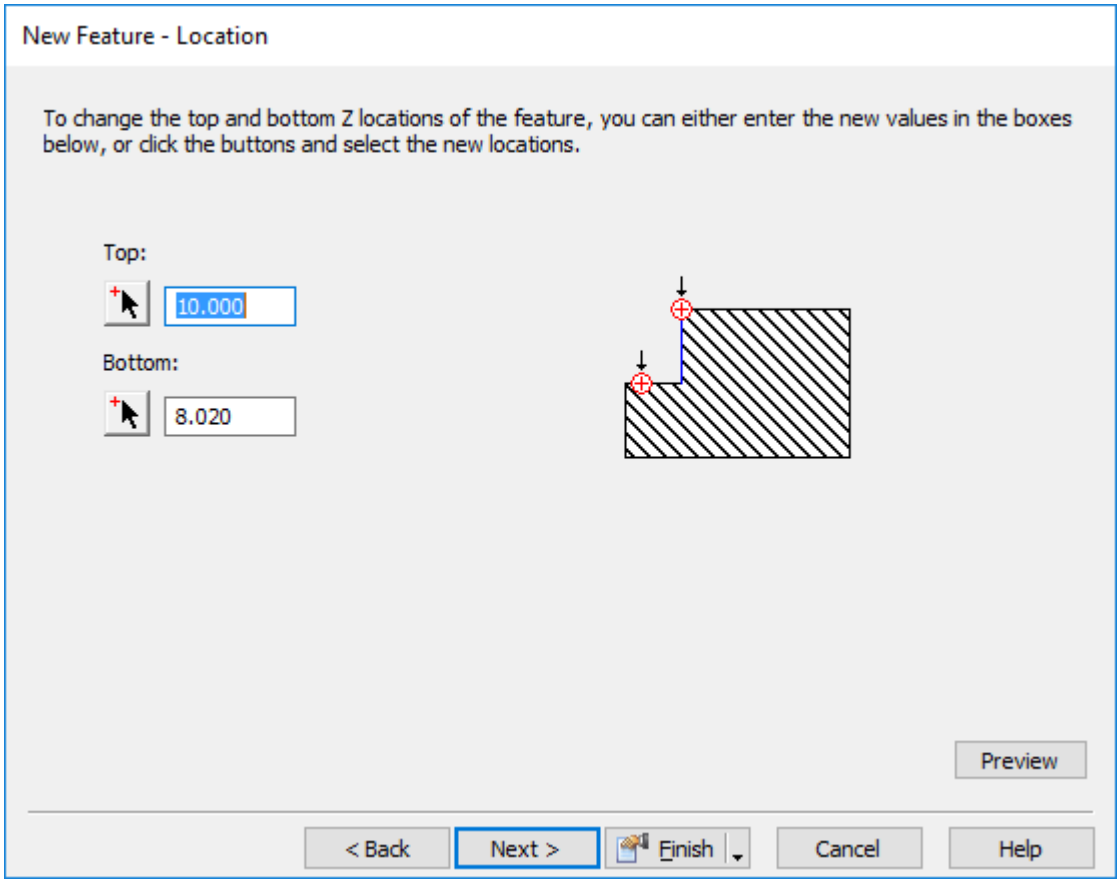
35 Check the direction by selecting machining Side. Make sure they are pointing inwards. In our case Normal is correct for both faces.

36 Select **Next**.

The location is incorrect.

37 Select the Bottom Icon and select the small flat face with the two holes in.

The correct location dimensions are shown below.



38 Select **Finish**.

Create a Group and then a Pattern of all three side Features

Adopt this technique for the other two faces inside the open Slot. Then create a Group for all three side features. Then make a Pattern of the group.

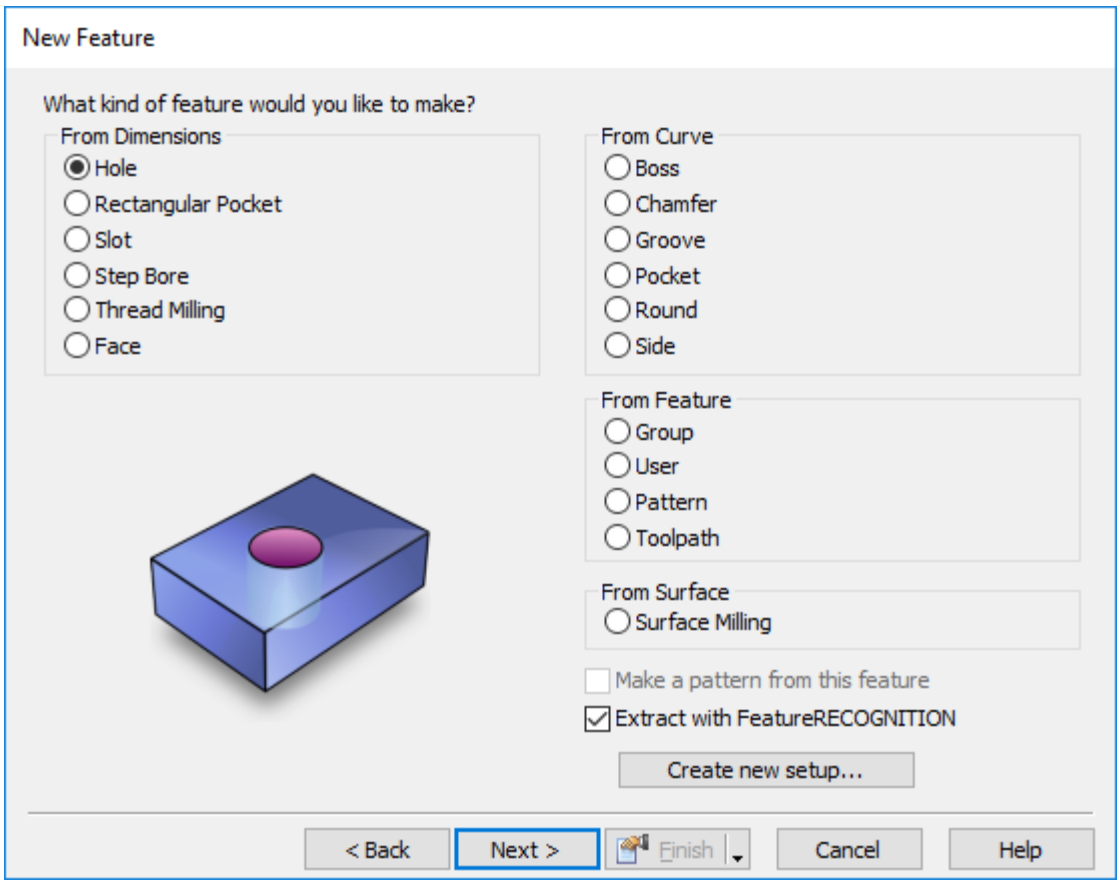
© Copyright 2016 Autodesk, Inc. All rights reserved

Extract 2mm diameter holes from around the diameter

Create a new hole feature using Feature Recognition.

39 Select Ctrl+R New Feature. Select Turn/Mill

40 Select **Next**. Select **Extract with Feature Recognition**.



41 Select **Next**.

42 Select **Around the index axis** and **Automatic**.

43 Then select **Next**.

44 FeatureCAM will default to **Recognize an construct multiple holes**.

45 Select **Next** and then **Select All**.

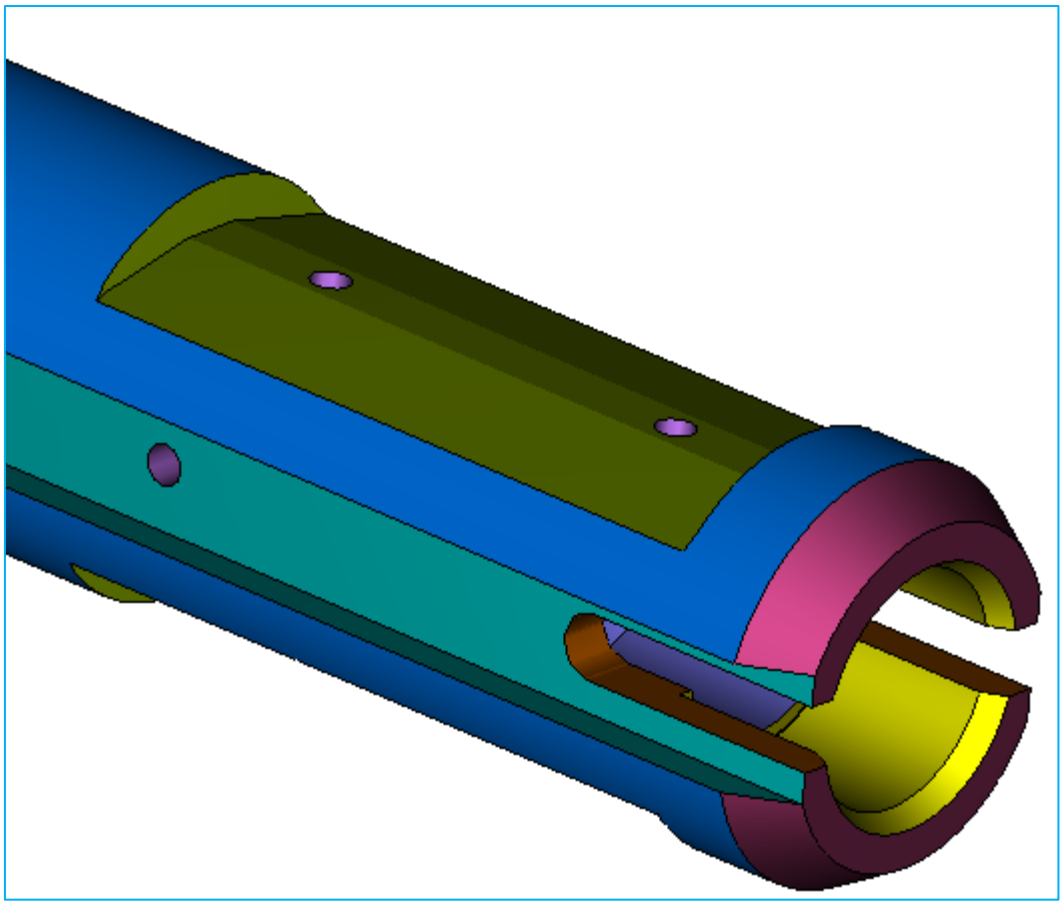
All of the holes will now be extracted.

I have removed the chamfers as it was using too many tools.

We will now machine the 2 slots on the end

Use the same principles that you used for machining the two long slots. Create one and then create a pattern.

© Copyright 2016 Autodesk, Inc. All rights reserved



Cutoff and transfer to Sub-Spindle.

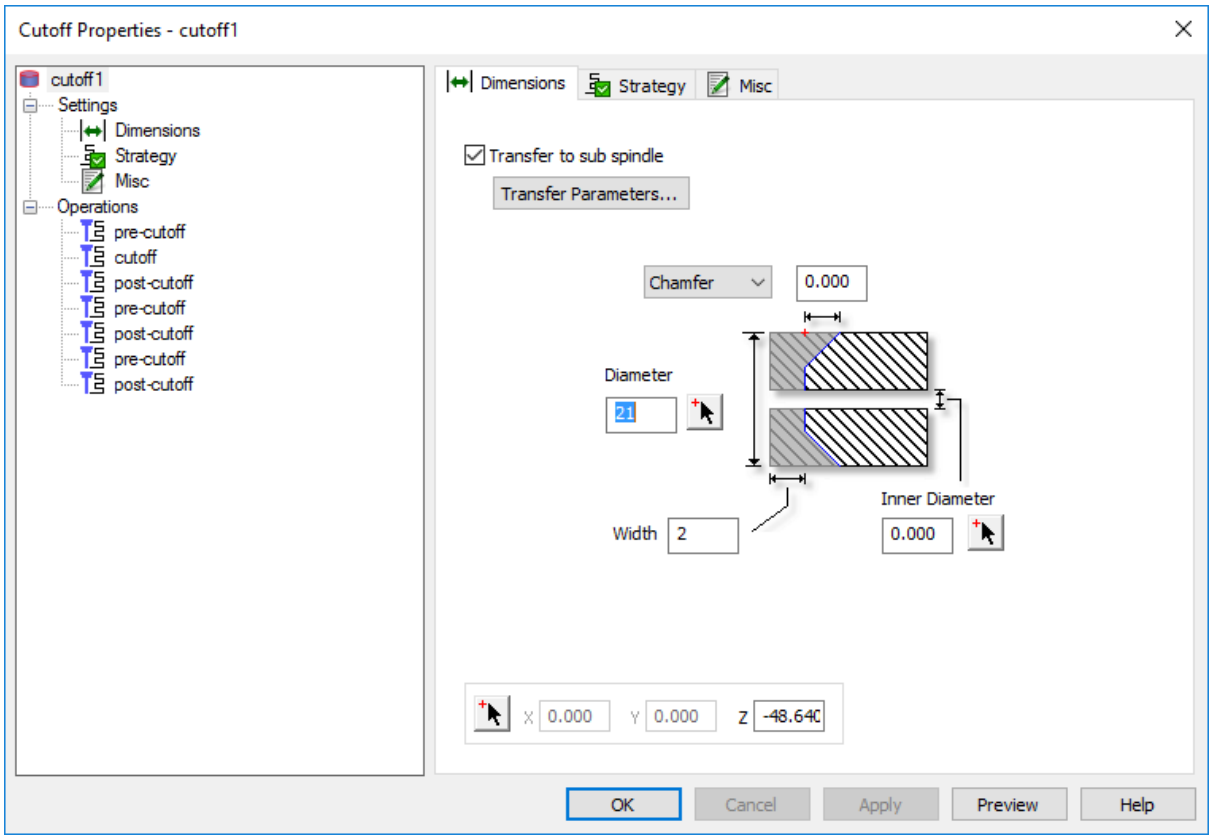
What we are going to do now is to Cut-off the part. But at the same time we need to use the Sub-Spindle to grab the part ready for the cut-off. This option is available in the Cut-off sequence menu.

46 Create a new Feature **Ctrl+R New Feature**. Select **Turning**.

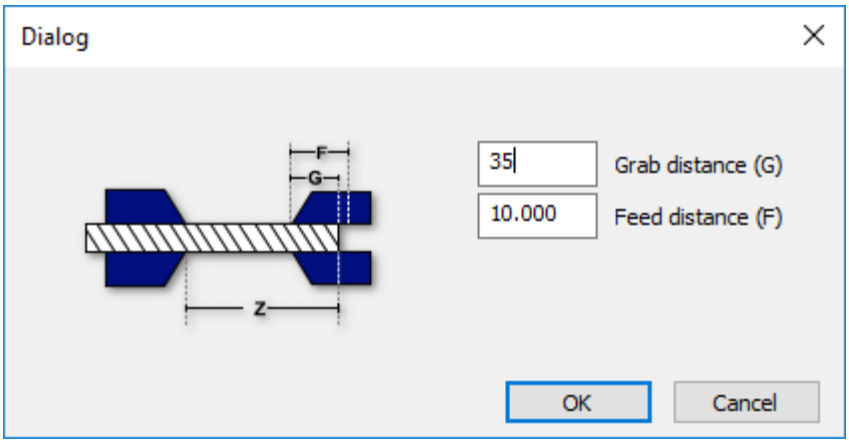
47 Select **Next**. Then select **Cutoff** from the menu. Select **Next.**

Remember to select transfer to Sub-Spindle

© Copyright 2016 Autodesk, Inc. All rights reserved



Transfer parameter Grab Distance.



Sub-Spindle Face feature

Create a facing feature to machine off the excess material.

48 Select **Sub Setup**

49 Create a Facing feature. Remember to add a Roughing pass to allow for the excess material.

Holes on Sub-Spindle

50 Create a Hole Feature using Automatic Feature Recognition.

© Copyright 2016 Autodesk, Inc. All rights reserved

This will extract all holes including the centre 10.4mm hole.

51 Select **Ctrl+R New Feature**. Select **Turn/Mill.**

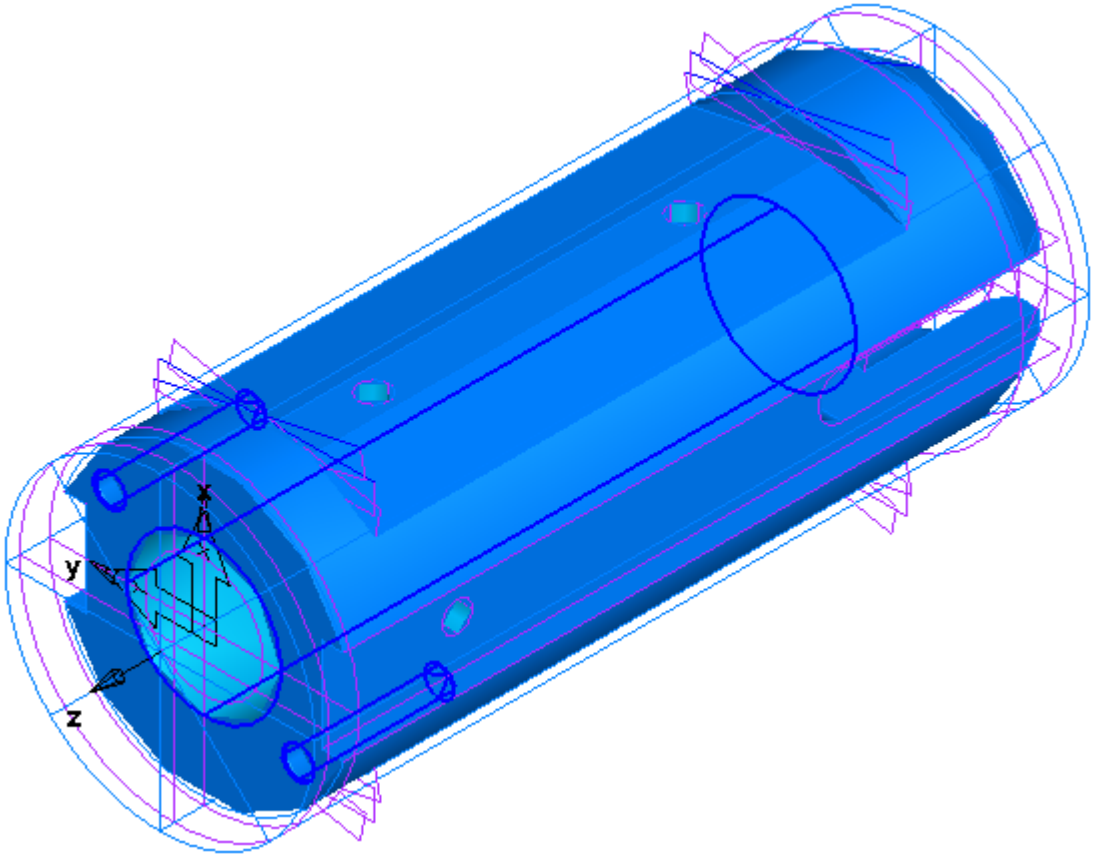
52 Select **Hole from Dimension** and **Extract with Feature Recognition**.

53 Select **Next.**

54 Select **Along the setup Z –axis.**

55 Select **Next**.

56 The default is **Recognise and construct multiple holes**.



57 Please pick **Select All.**

58 Select **Finish**.

Make sure that the Sub-Spindle operations are using P4 Back tool post. Otherwise you will get an error message.

The Tool post information is on the next page.

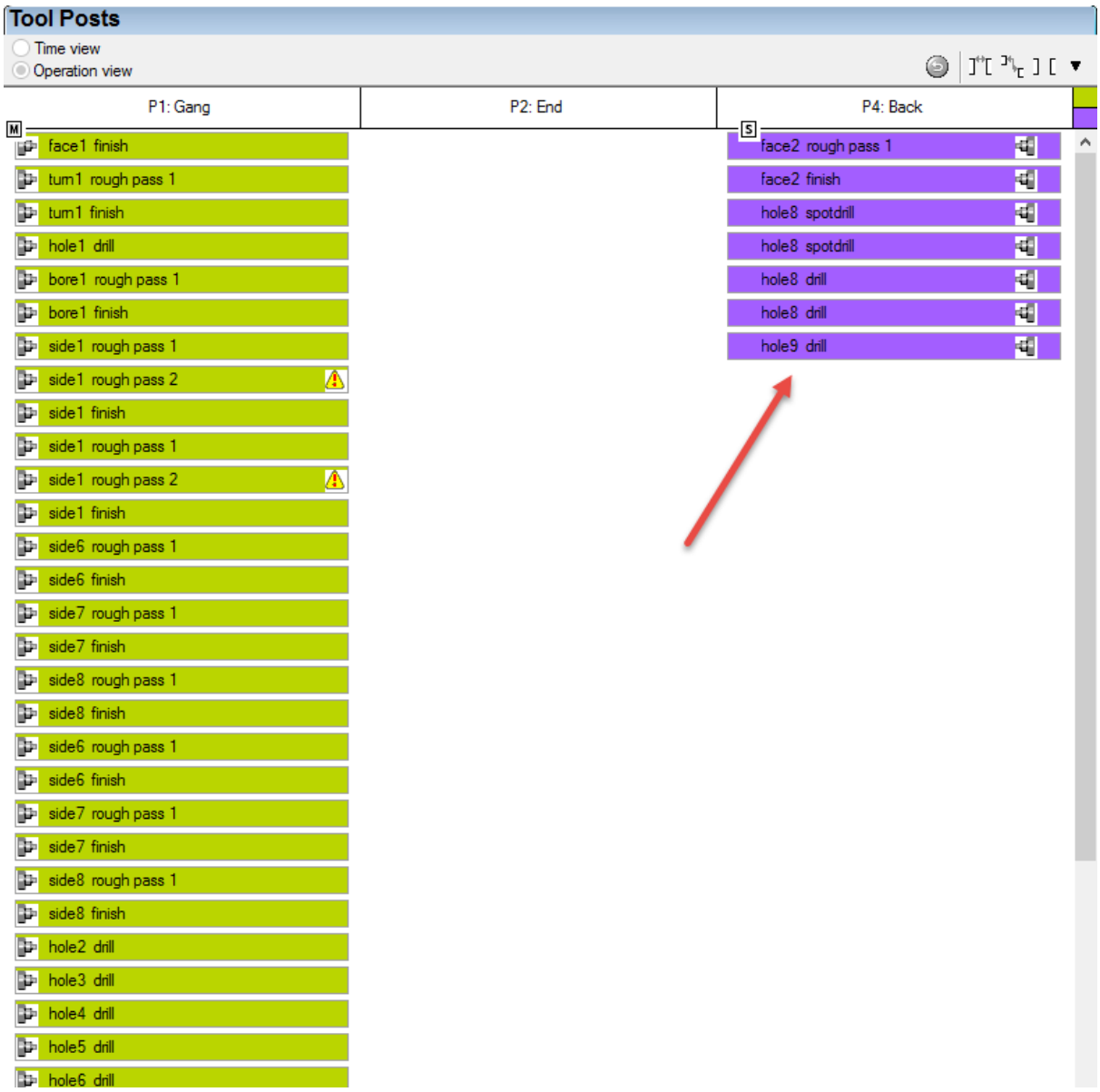
Ordering the tool post operations

See examples on the next page

© Copyright 2016 Autodesk, Inc. All rights reserved

© Copyright 2016 Autodesk, Inc. All rights reserved

In order to run this file in Automatic mode, drag and drop the Back operation to the start the tree structure. As shown below.



This concludes the training for the Citizen L20E-IX machine.

More examples will come available as the product develops.

© Copyright 2016 Autodesk, Inc. All rights reserved