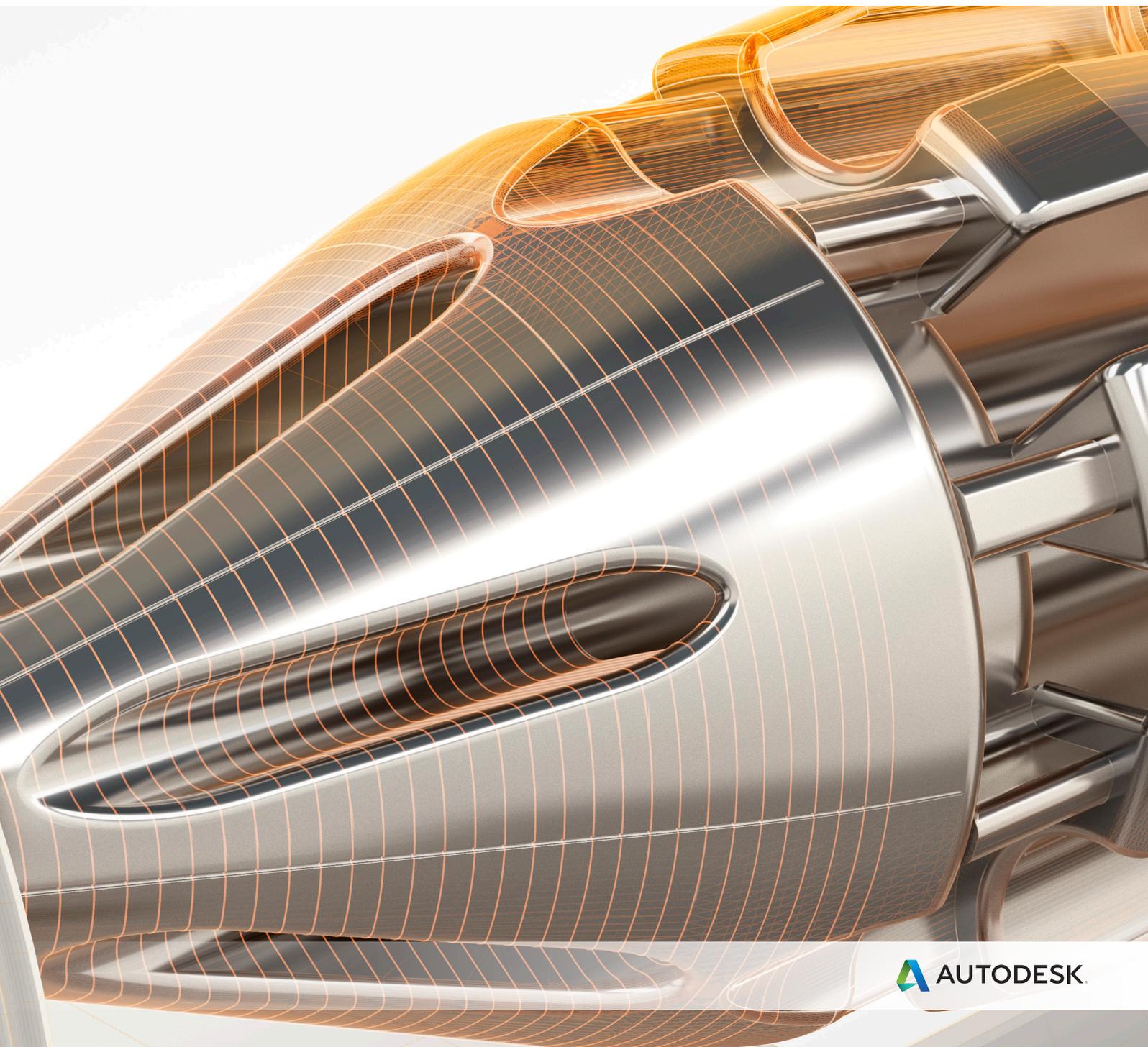


Automated to make parts faster

Training Course



Contents

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

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

AUTODESK® FEATURECAM®

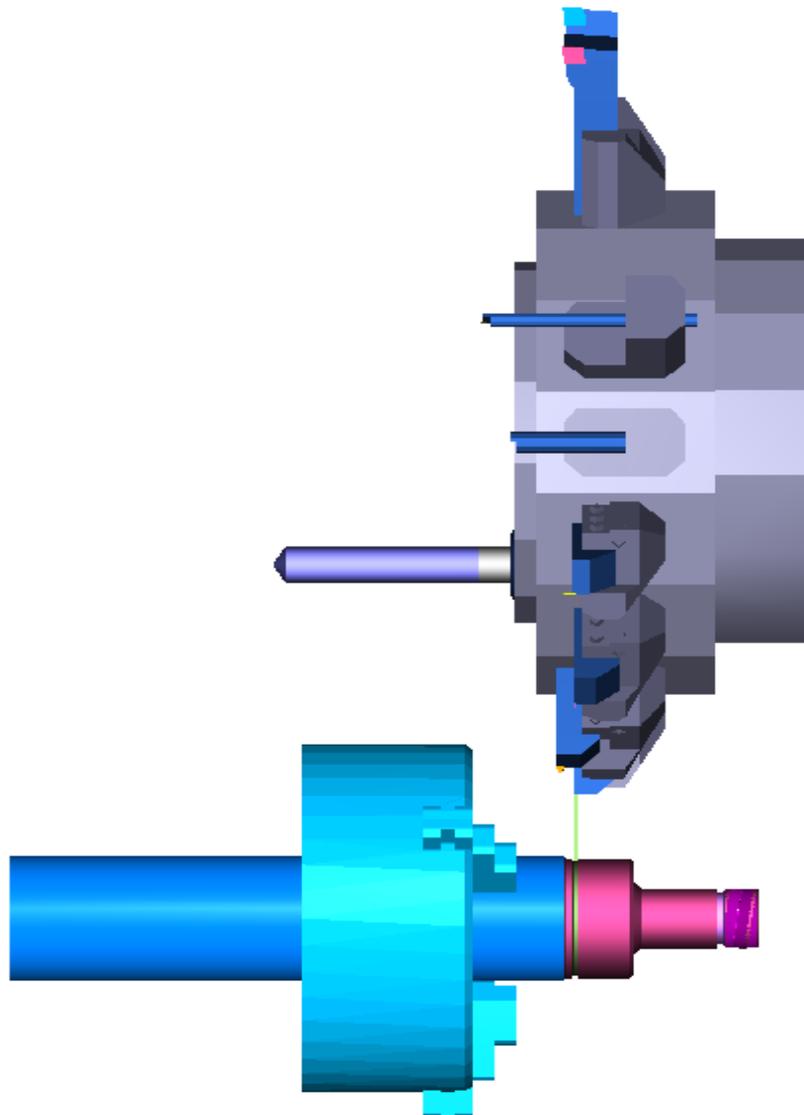
Introduction to FeatureCAM Turning



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



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





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



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

CAM Software Strategies



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



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



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



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

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

Feature Based CAM 	Operations-Based CAM	Process-Based CAM
<ol style="list-style-type: none"> 1. Draw or Import Your Part 2. Identify Features 3. Click Simulation <p>You Are Done!</p> <p>FeatureCAM Automatically:</p> <ul style="list-style-type: none"> • Determines Rough and Finish Operations • Selects ALL Tools Sizes • Calculates Feeds & Speeds • Determines Stepmover & Z Increments • Generates Toolpath • Creates NC Code 	<ol style="list-style-type: none"> 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 	<ol style="list-style-type: none"> 1. Import or draw part. 2. Identify features. 3. Select machining processes. 4. Click simulation. 5. Create NC code. <p>???Are You Done? No.</p> <p>You must analyse and verify:</p> <p>Does the process have the correct operations?</p> <p>Does the process use the correct size tools?</p> <p>Are the correct feeds and speeds used?</p> <p>Does the process have the correct stepover and Z increments?</p>

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

Feature-Based Machining

Has many advantages such as:

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



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

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

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

Built in intelligence



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



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

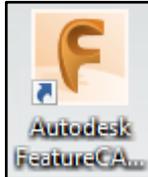


By providing a set of standard machining rules already in place, it's easier for you to customize the software to reflect the machining preferences of your Machine shop.



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

Starting FeatureCAM



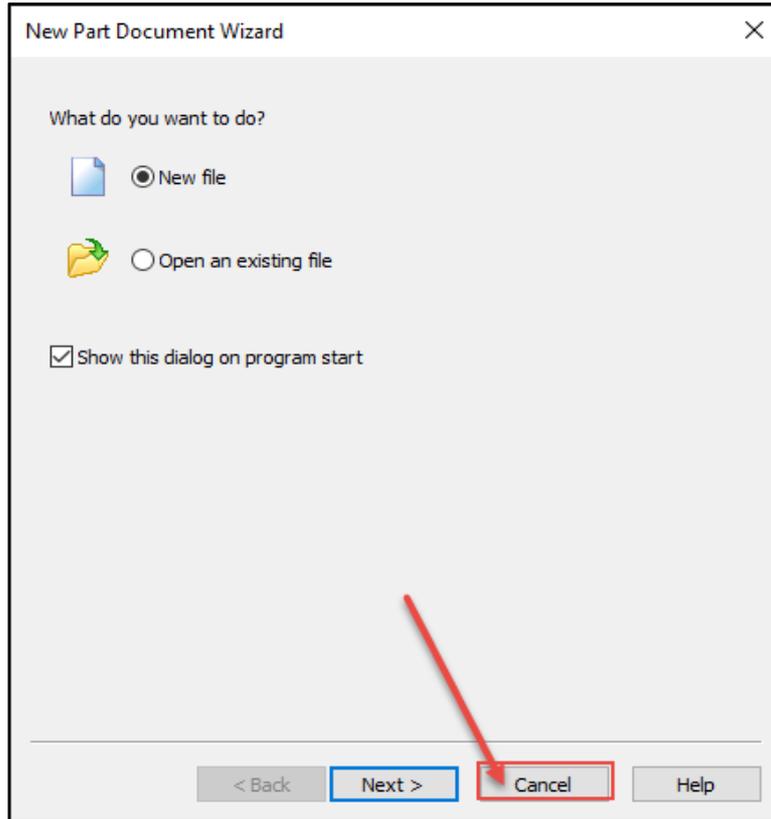
- 1 Select the FeatureCAM icon from your desktop and select **New File** and **Milling Setup** and **Millimetre**. Select **Finish** twice and then **Ok** to close the Stock menus. The stock menu will be covered later on in the training notes.



Evaluation Options



- 2 Click on the shortcut that is on the Desktop. This will start FeatureCAM.
- 3 When it has finished **cancel** the new **Part document wizard** that is on the screen.



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

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



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



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

Evaluation Options ✕

Select the FeatureCAM product components you would like to activate. OK

Cancel

Product component

Standard
 Premium
 Ultimate

2.5D Milling

2.5D and 3D HSM
2.5D and 3D MX

2.5D and 3D Lite

Turning

Turn/Mill

FeatureRECOGNITION

Tombstone Machining
5 Axis Positioning
Catia Import
Solid Modeling

Wire EDM

Step Import
Catia V5 Import
ProE Import
NX Import
Machine Simulation
Advanced Turn/Mill (MTT)

Network Database

5 Axis Simultaneous
Probing
Vericut verification
NCSIMUL verification
CAMplete verification
Swiss Turning

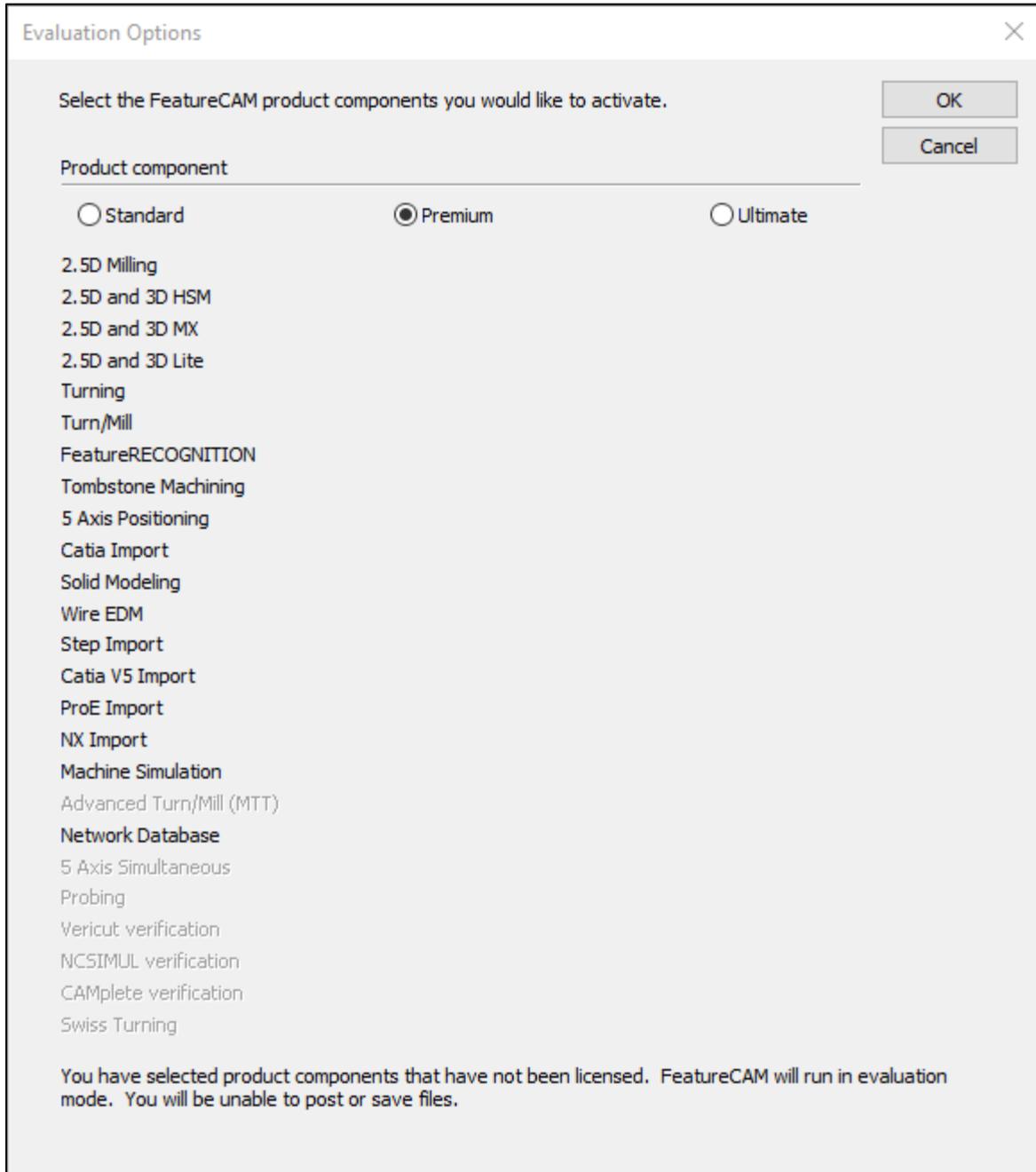
You have selected product components that have not been licensed. FeatureCAM will run in evaluation mode. You will be unable to post or save files.



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



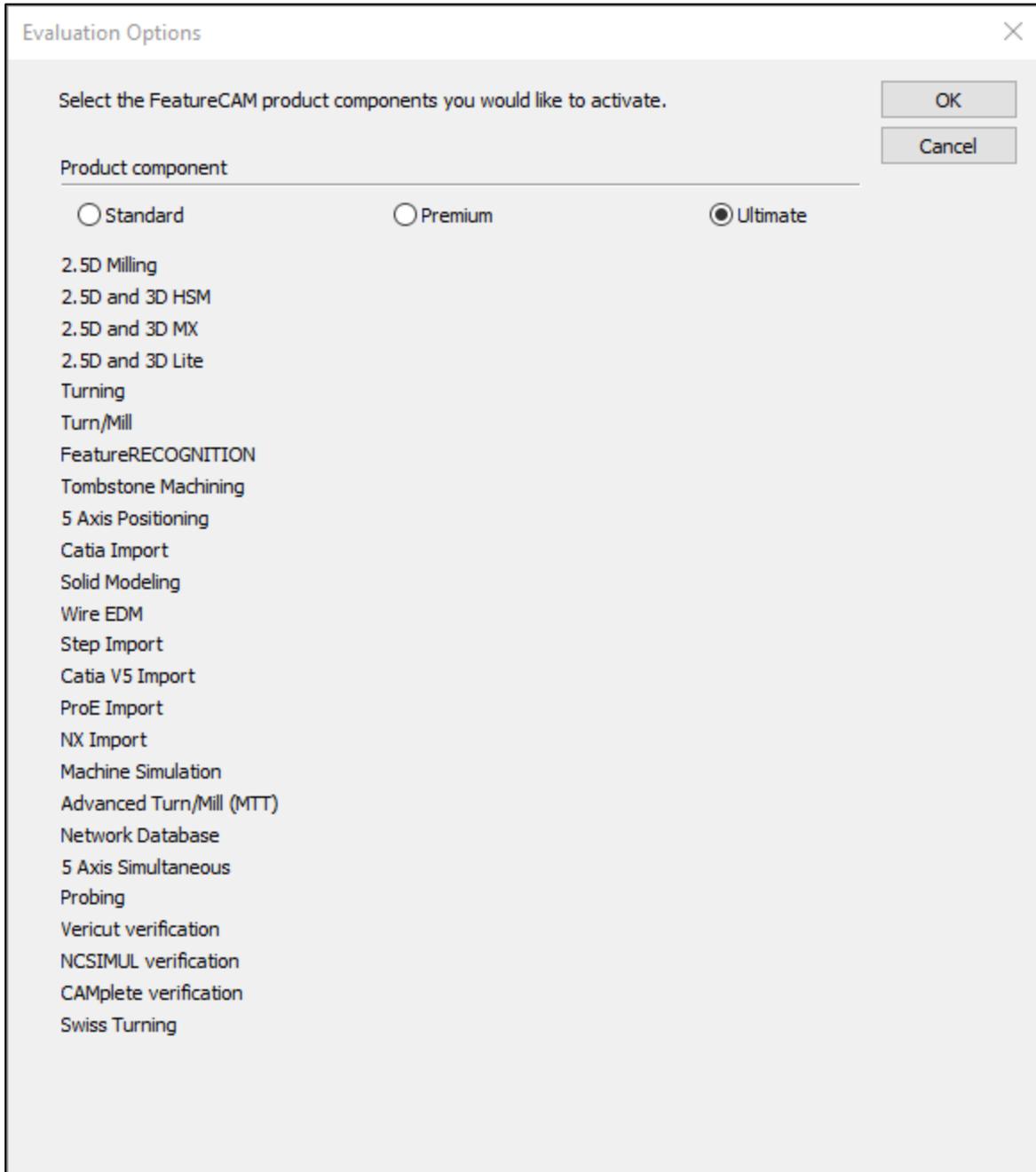
The image below show the available options in **Premium FeatureCAM**.



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



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



Evaluation Options

Select the FeatureCAM product components you would like to activate.

OK

Cancel

Product component

Standard Premium Ultimate

2.5D Milling
2.5D and 3D HSM
2.5D and 3D MX
2.5D and 3D Lite
Turning
Turn/Mill
FeatureRECOGNITION
Tombstone Machining
5 Axis Positioning
Catia Import
Solid Modeling
Wire EDM
Step Import
Catia V5 Import
ProE Import
NX Import
Machine Simulation
Advanced Turn/Mill (MTT)
Network Database
5 Axis Simultaneous
Probing
Vericut verification
NCSIMUL verification
CAMplete verification
Swiss Turning



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

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

Autodesk support

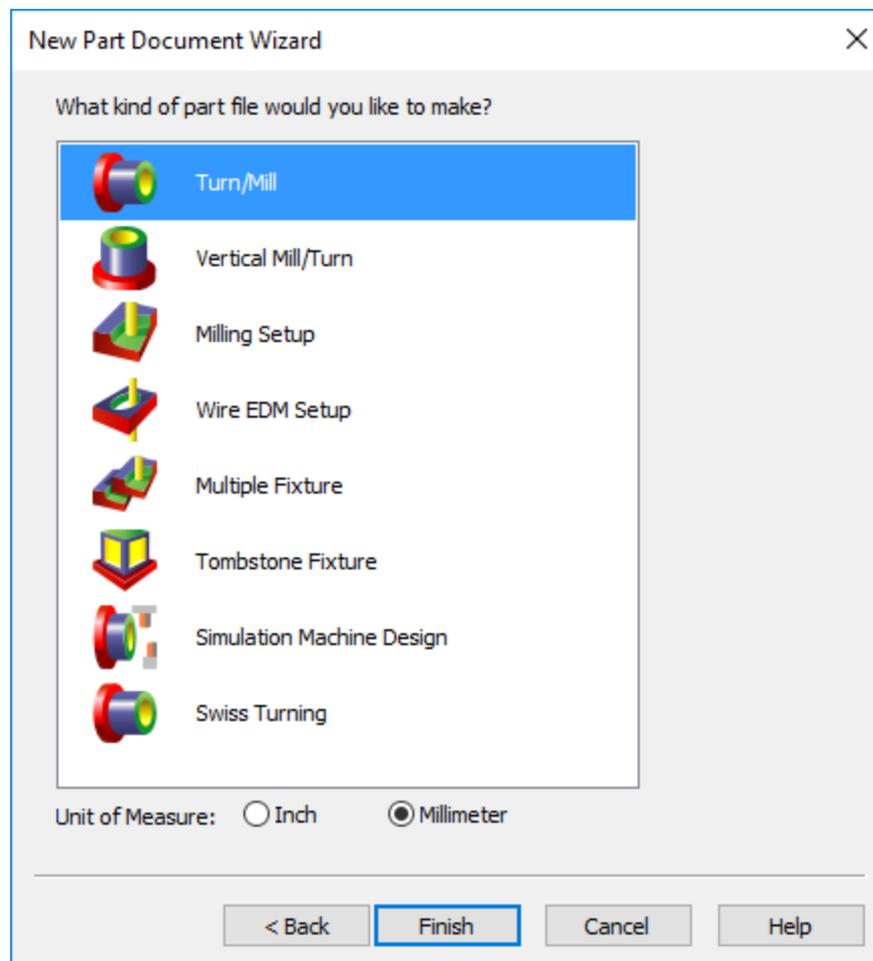


Let us help you to resolve issues quickly. You will have access to:

- 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.

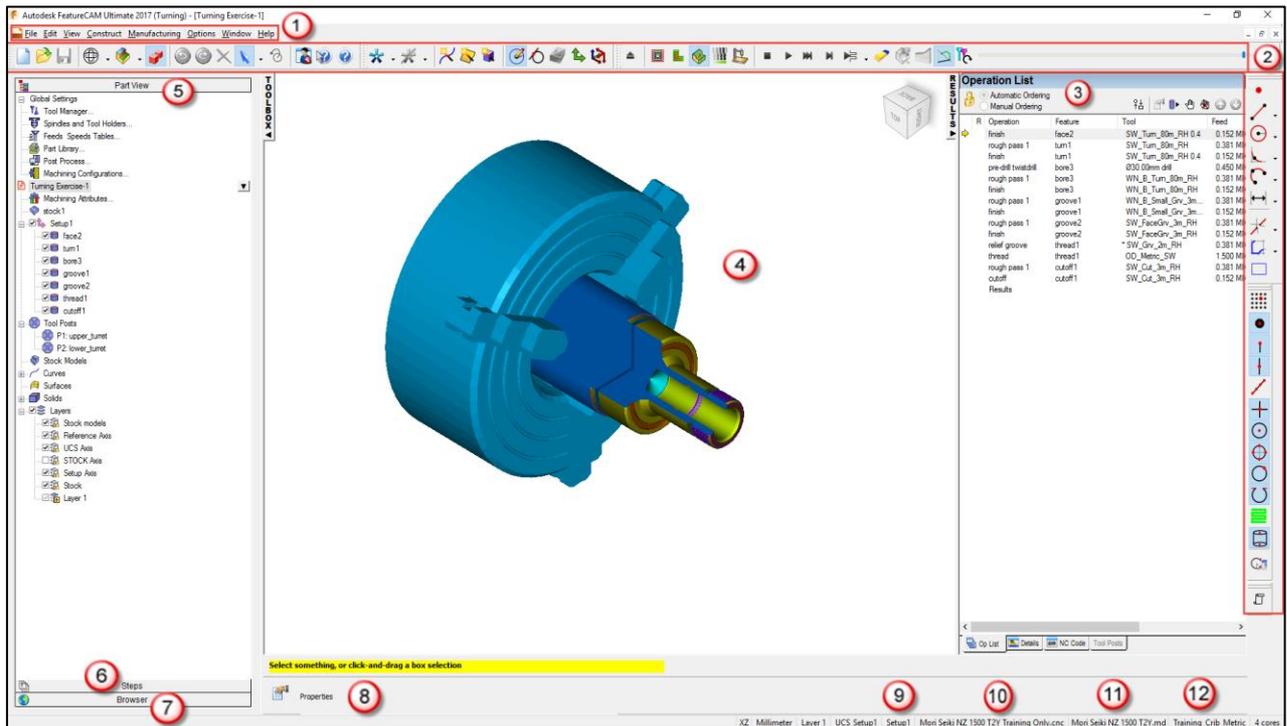
- Select **File>New** from the menu or select **Ctrl + N** keys.



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.

Screen Layout

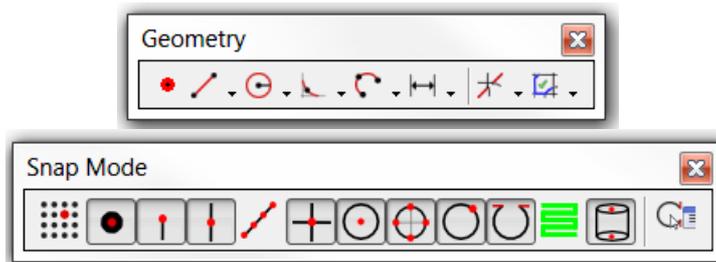


1 Menu Bar - Located at the top. Like many other Windows based software, packages **FeatureCAM** includes a series of drop down menus. These menus are unique to **FeatureCAM** and provide one or more functions used to facilitate the constructing, manufacturing, viewing and editing of your part program.

File Edit View Construct Manufacturing Options Window Help

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:



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

R	Operation	Feature	Tool	Feed	Speed	Depth
	finish	face2	SW_Tum_80m_RH 0.4	0.152 MMPR	488 SMM CW	52.500 mm
	rough pass 1	tum1	SW_Tum_80m_RH	0.381 MMPR	366 SMM CW	27.000 mm
	finish	tum1	SW_Tum_80m_RH 0.4	0.152 MMPR	488 SMM CW	27.000 mm
	pre-drill twistdrill	bore3	Ø30.00mm drill	0.450 MMPR	809 RPM CW	105.013 mm
	rough pass 1	bore3	WN_B_Tum_80m_RH	0.381 MMPR	366 SMM CW	2.000 mm
	finish	bore3	WN_B_Tum_80m_RH	0.152 MMPR	488 SMM CW	2.000 mm
	rough pass 1	groove1	WN_B_Small_Grv_3m...	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove1	WN_B_Small_Grv_3m...	0.152 MMPR	488 SMM CW	3.000 mm
	rough pass 1	groove2	SW_FaceGrv_3m_RH	0.381 MMPR	366 SMM CW	2.900 mm
	finish	groove2	SW_FaceGrv_3m_RH	0.152 MMPR	488 SMM CW	3.000 mm
	relief groove	thread1	* SW_Grv_2m_RH	0.381 MMPR	366 SMM CW	3.000 mm
	thread	thread1	OD_Metric_SW	1.500 MMPR	2330 RPM CW	1.137 mm
	rough pass 1	cutoff1	SW_Cut_3m_RH	0.381 MMPR	366 SMM CW	2.000 mm
	cutoff	cutoff1	SW_Cut_3m_RH	0.152 MMPR	488 SMM CW	50.000 mm
	Results					

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



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



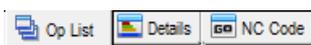
*At the bottom of the screen is the **Assistance Bars** for **Geometry** and **Status**. The **Assistance Bar** is where you are prompted when creating **Geometry**.*



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

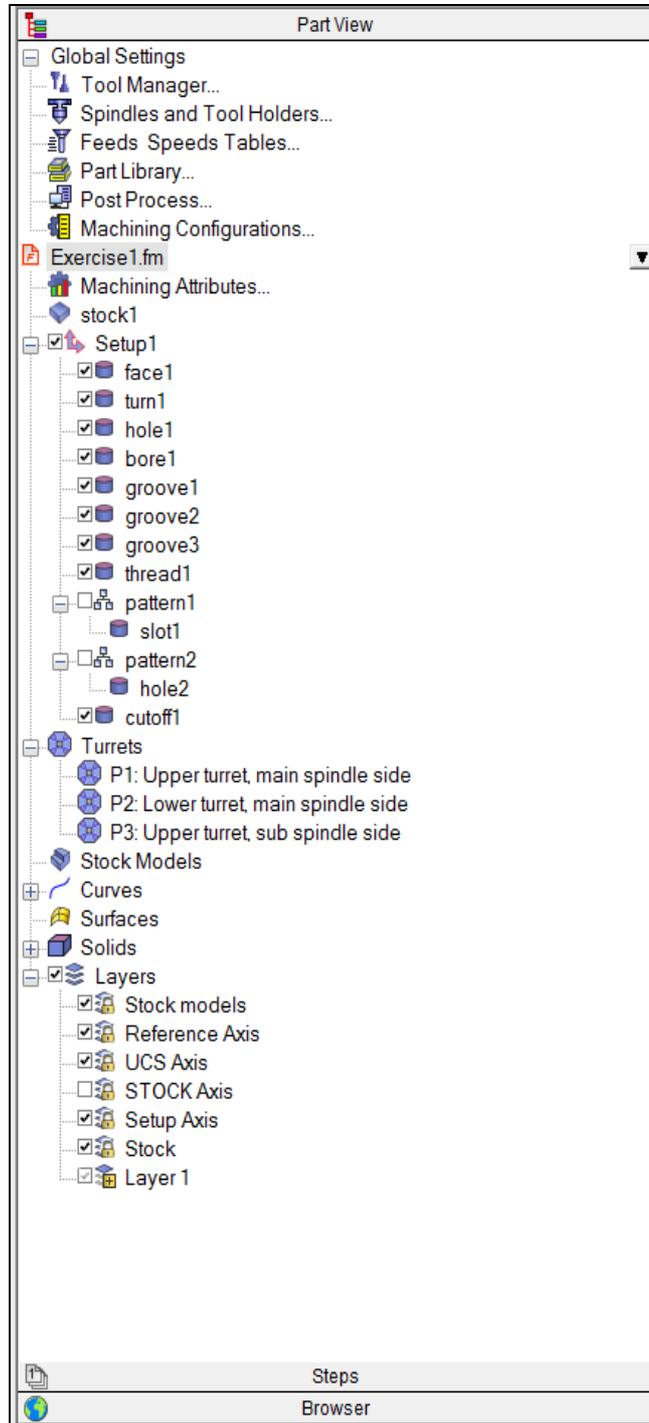


There are three tabs available on the results window at the bottom.



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

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.

Steps Menu



Steps to take to create a part in FeatureCAM

Drawing a part in FeatureCAM

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

Importing a file into FeatureCAM

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



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

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



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



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

Stock



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



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



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



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



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



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



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



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



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

Steps (See menu in PartView)

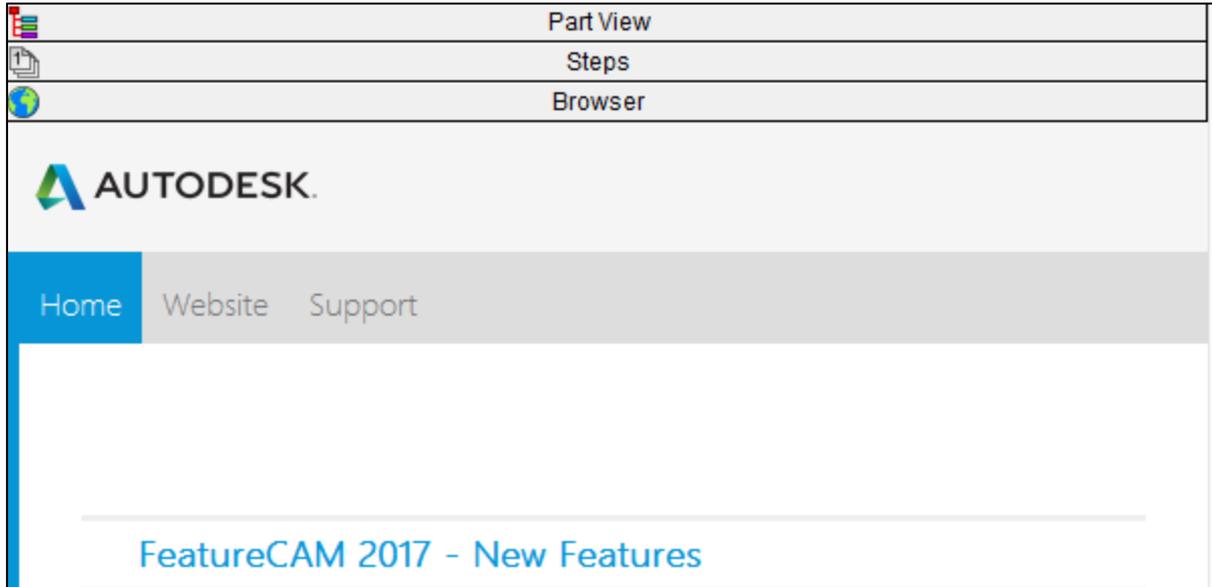


The following is a guide to help you choose the correct way in which to machine a component in **FeatureCAM**.

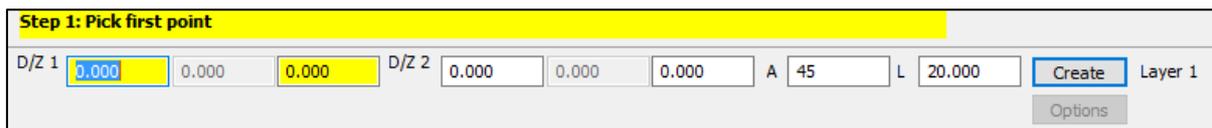
- 1 Open **FeatureCAM** and select **New File** (Default). Select **Next**. Choose your module of choice and your units. Select **Finish**. Cancel the dimensions wizard. Hide the stock by selecting **Stock** in **PartView**, right click and select **Hide Stock**. Draw you part using geometry creation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



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



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



Metric

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



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

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

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

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

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

Customize Manufacturing.

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



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: Turning Exercise-1
 Setup: Setup1 (1 of 1)
 Date: Thursday, September 08, 2016 13:40:18

Crib: Training_Crib_Metric
 Summary:
 Slot 1: SW_Turn_80m_RH 0.4
 Slot 2: SW_Turn_80m_RH
 Slot 3: Ø30.00mm drill D 30.000 mm L 143.000 mm
 Slot 4: WN_B_Turn_80m_RH
 Slot 5: WN_B_Small_Grv_3m_RH
 Slot 6: SW_FaceGrv_3m_RH
 Slot 7: SW_Grv_2m_RH
 Slot 8: OD_Metric_SW
 Slot 9: SW_Cut_3m_RH

NC Code

P1: Upper turret

```

%
OExercise1( FILENAME = Exercise1)
N20 G21 G40 G90 G80
N25 G28 U0
N30 G28 W250.0
( OPERATION: ROUGH FACE FACE1 )
N40 T101
N45 M8
N50 G92 S4500
N55 G96 S365
N60 G0 X113.779 Z28.237 M4
N65 X111.0 Z13.0
N70 G72 W2.0 R0.5
N75 G72 P80 Q85 U0. W0.25 F0.
N80 G1 Z0. F0.
N85 X-0.8
N90 G0 Z13.0
( OPERATION: FINISH FACE FACE1 )
N100 X113.779 Z28.237 T101
N105 G92 S4500
N110 G96 S487
N115 G70 P80 Q85
N120 G0 Z13.0
N125 G28 U0
N130 G28 W250.0
        
```

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

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

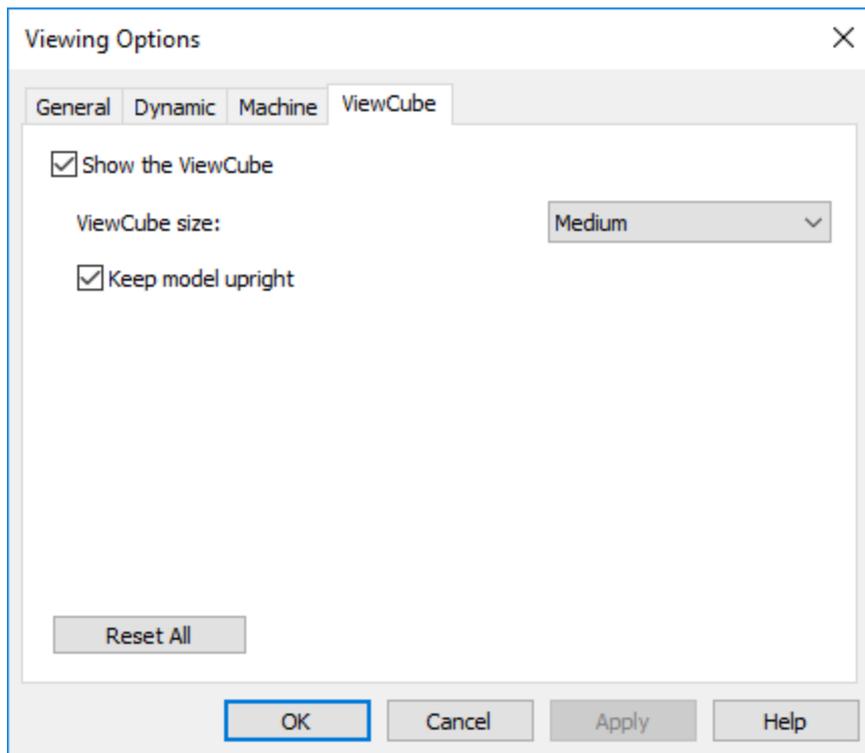
When you select a single-face view and move the cursor near the cube, control icons are displayed:

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.



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**.



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.

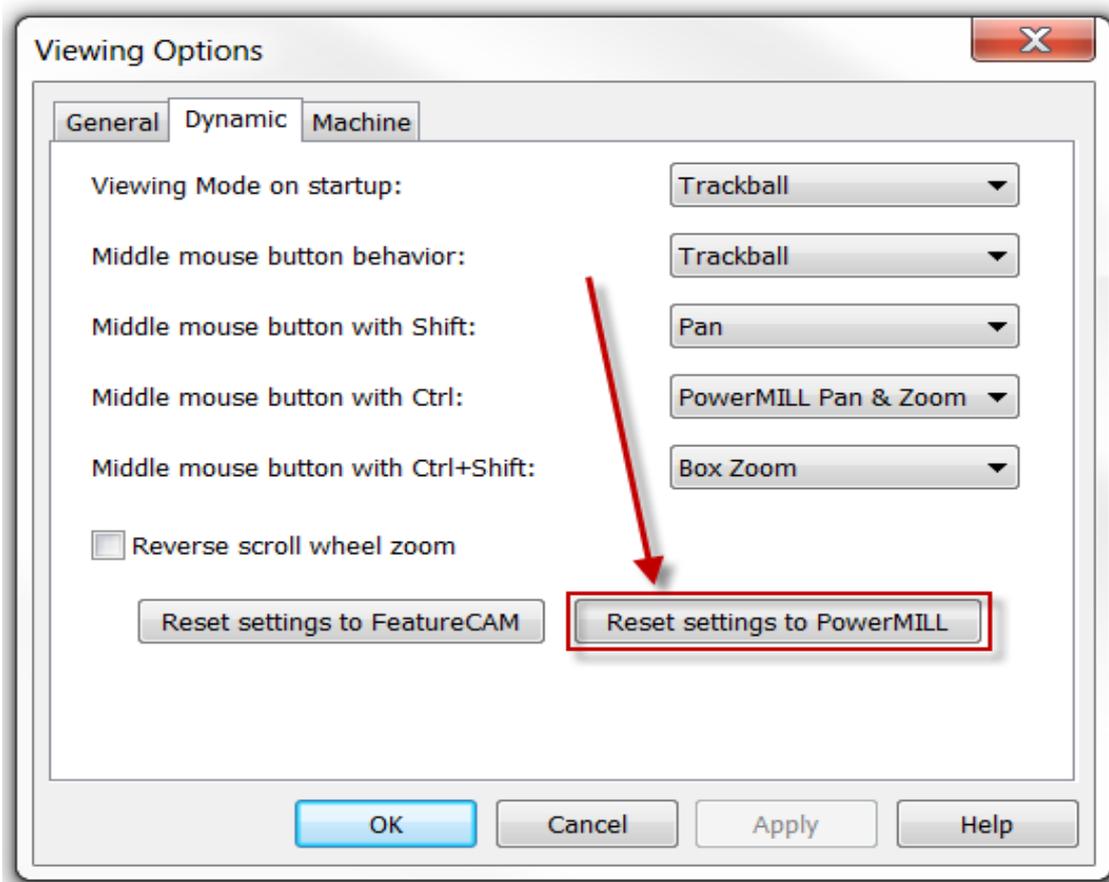
Right mouse button Special Menus & FeatureCAM Options.



- 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**

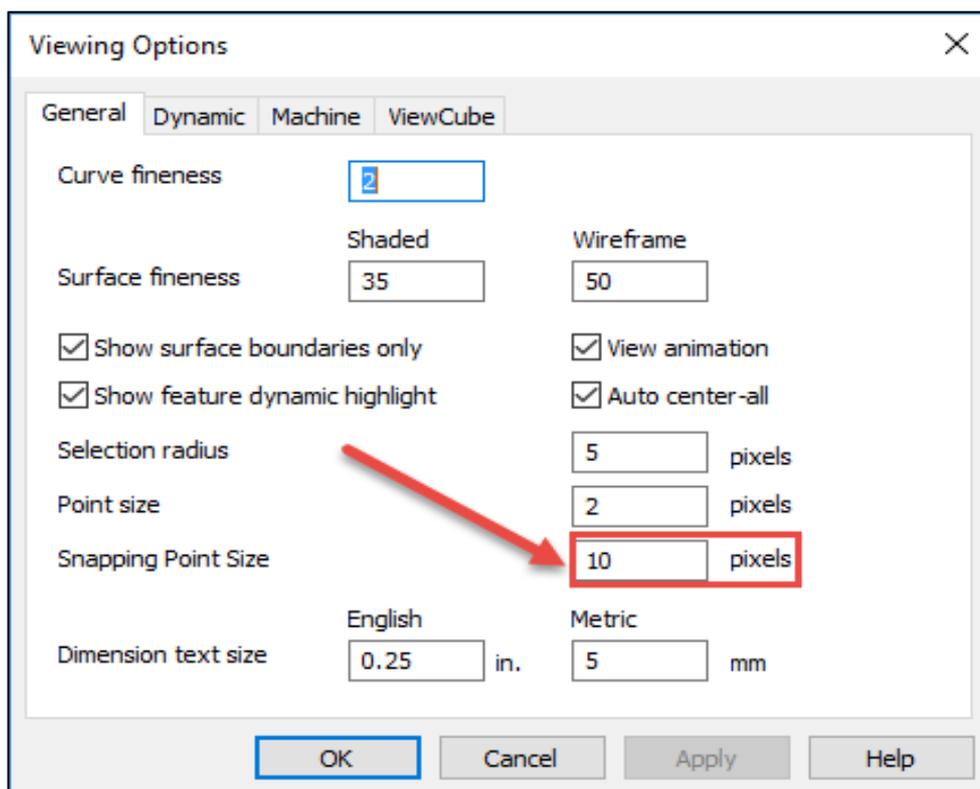
Reset to PowerMILL Settings

- 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

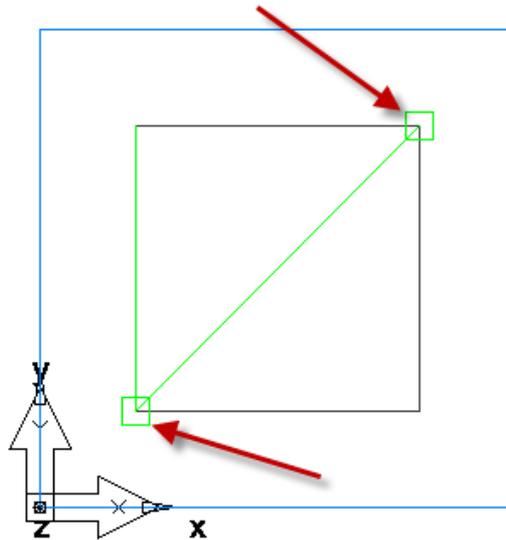


In the General Viewing Options we now have the ability to change the size of the Snapping point. By default this size is normally 4 pixels. In order to create larger points, change this to 8 pixels.





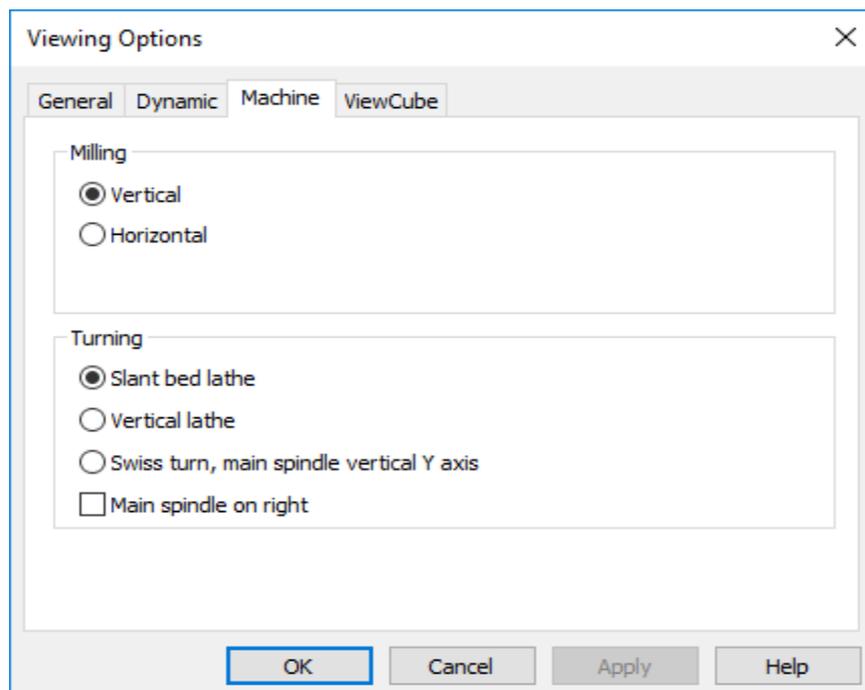
An example of this is shown below



Viewing Options Machine

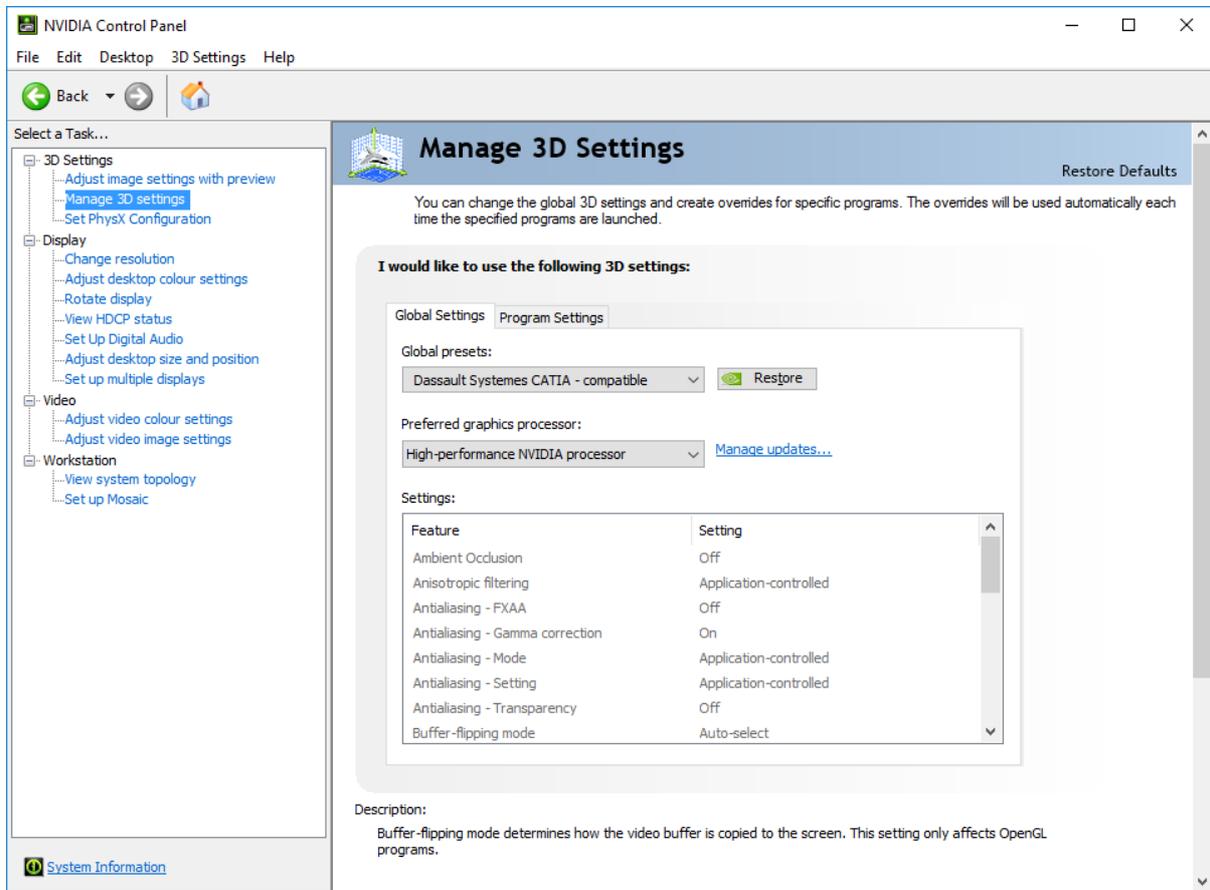
When viewing machine axis, by default the view is shown vertical: If you have a Horizontal machine or a Mill/Turn machine (Vertical turret lathe) use this option to change the view.

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



Display options

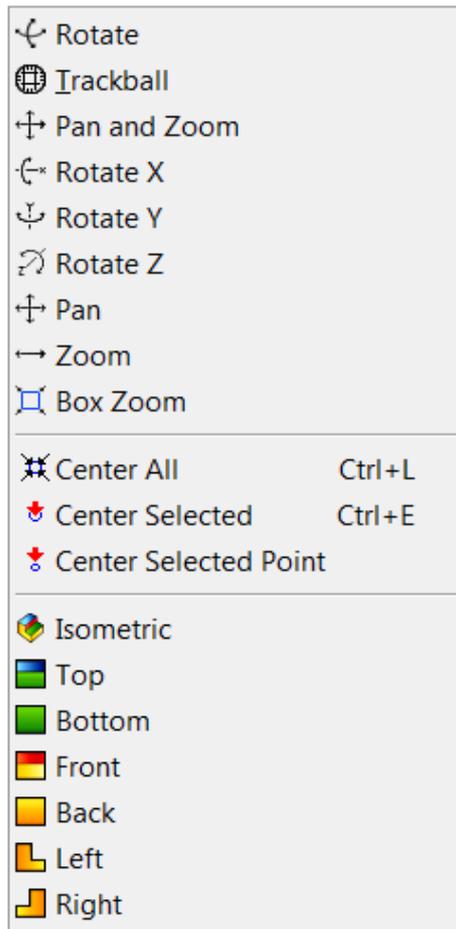
- For the best **Display Options** for your computer set the **N-Vidia® Control Panel** options to the following.



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.



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**.

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

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

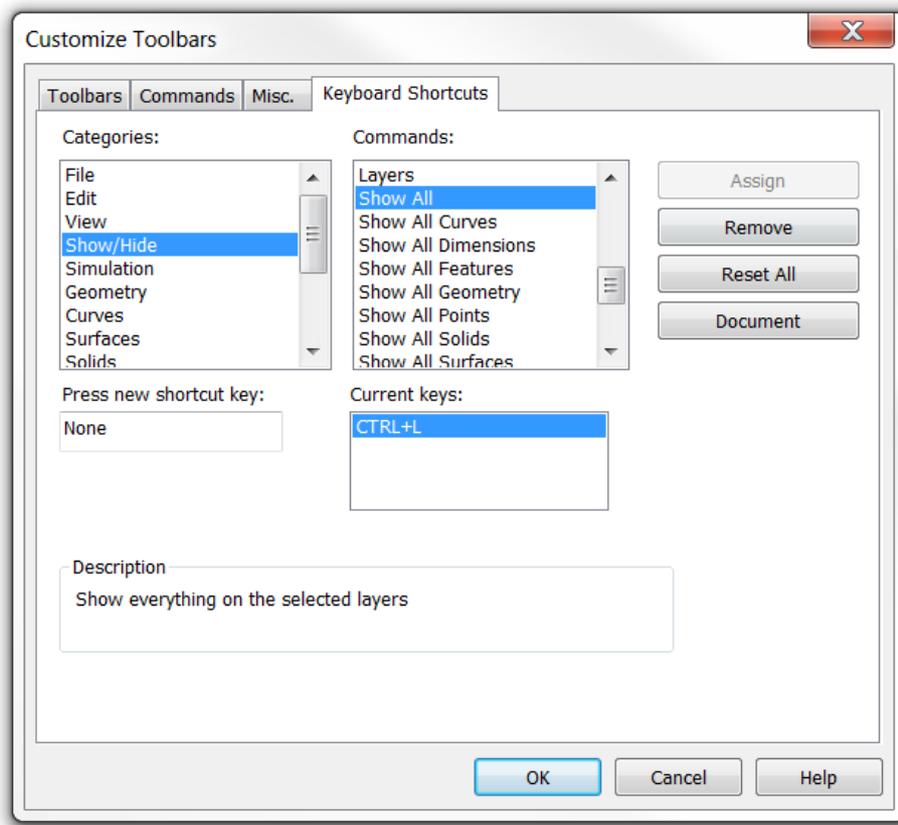
Commonly used Ctrl Keys

- **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**.

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

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

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

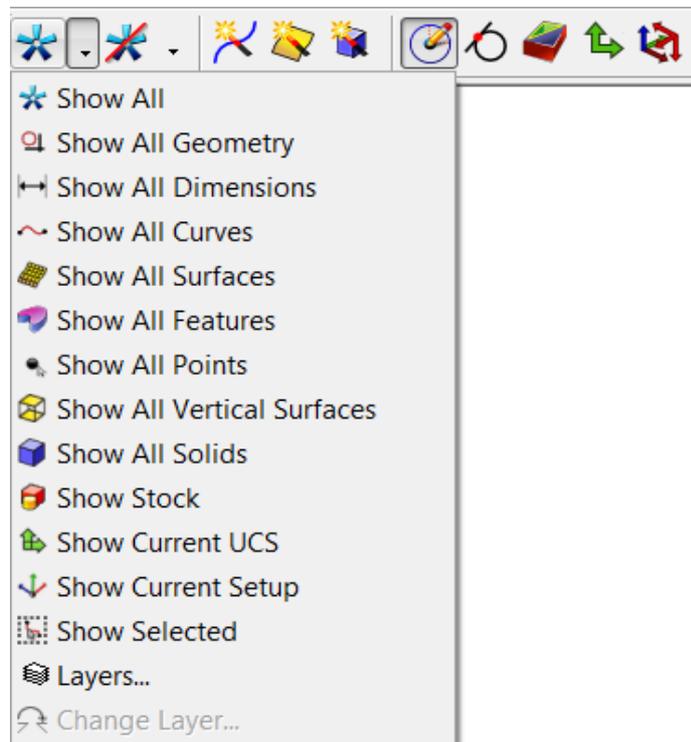
- **Remove** — Click this button to remove the shortcut assigned to the selected command.
- **Reset All** — Click this button to reset all shortcuts to the **FeatureCAM** defaults.

Blanking model entities

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

Show menu

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

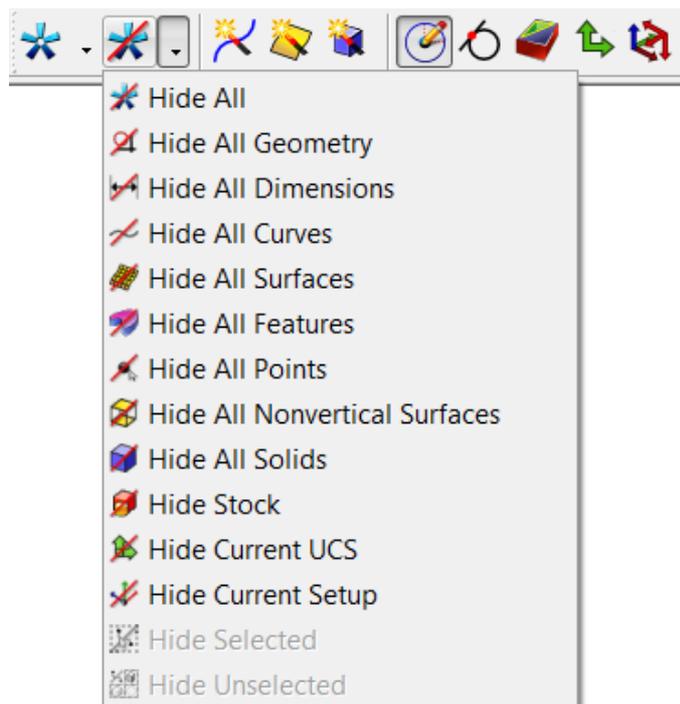


- **Show All** — Use this menu option to Show everything in the part model.

- 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

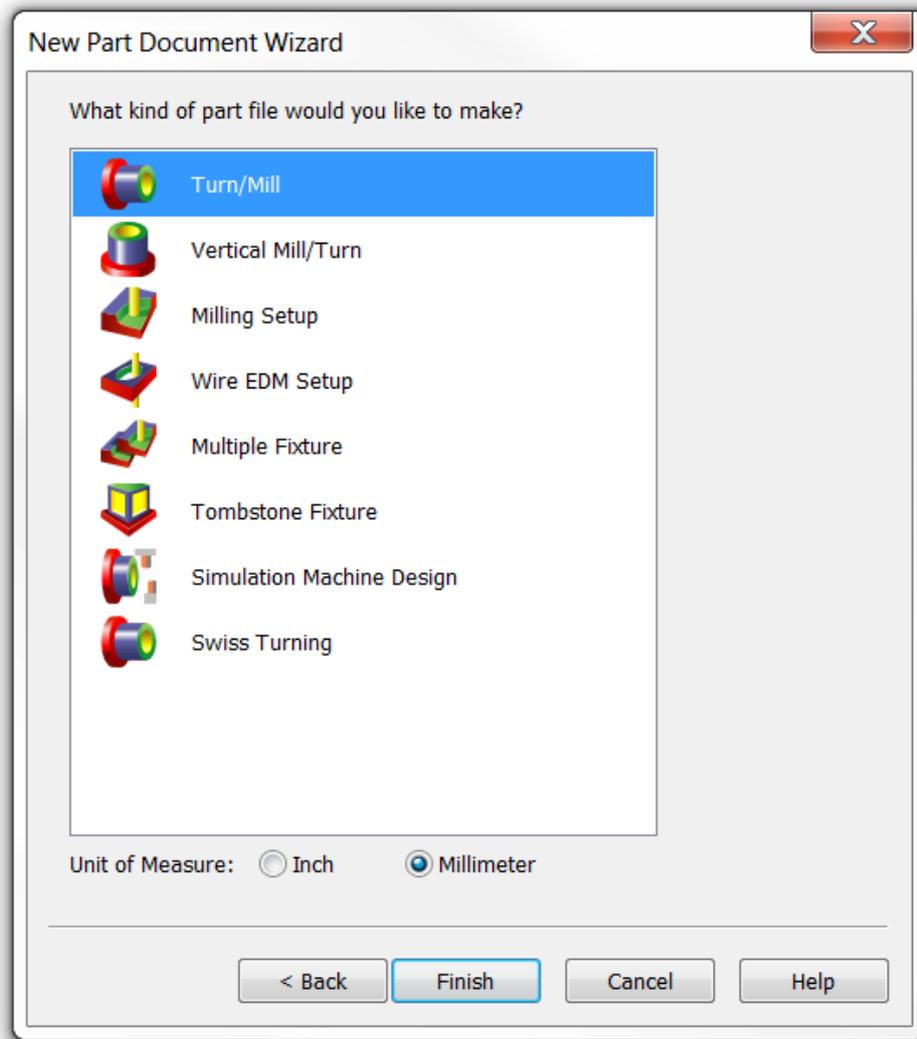
- Hide  controls what is displayed at any given time. This is useful as you place and model intricate features in a complex part. Besides the display factors, you can't snap, select or build curves from hidden entities. The Hide functions are not exclusive. You can click different buttons sequentially, 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:



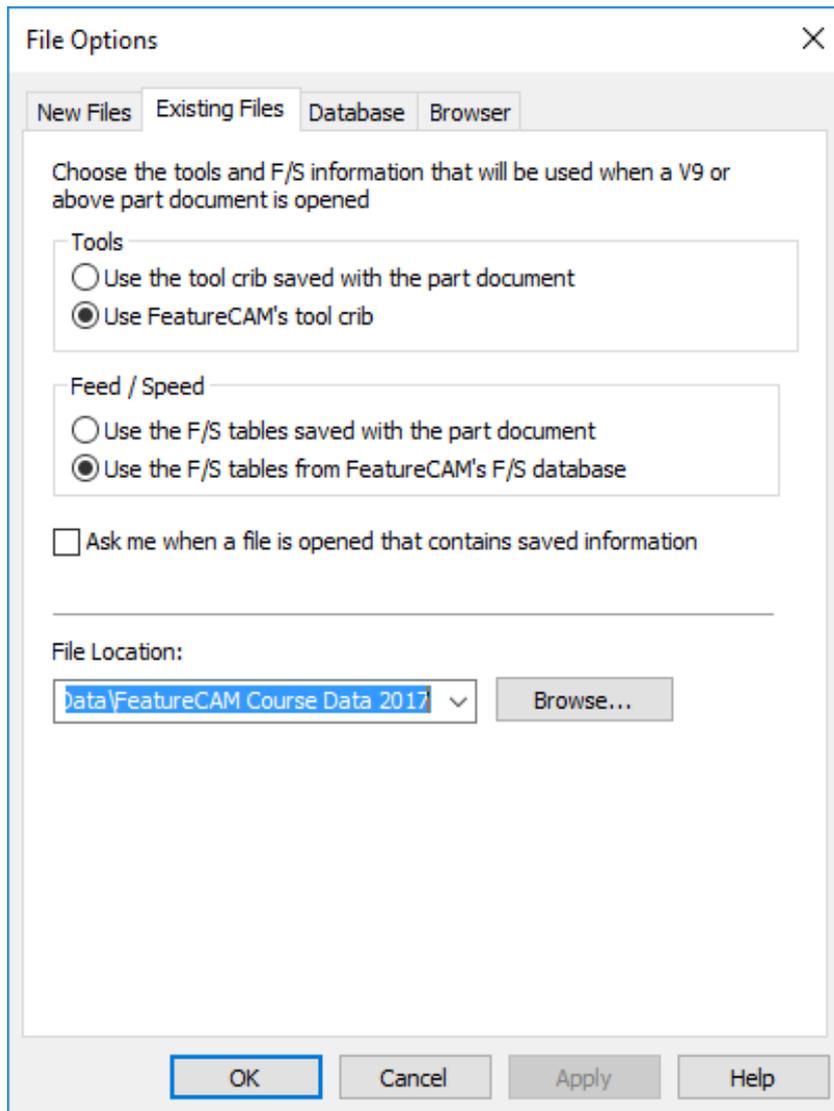
- Hide **All** — Use this menu option to hide all geometry, curves, Features. The stock and axis icon remain visible. A common procedure is to Hide **All**, then **Show** only one type of entity, for example features.
- Hide **All Geometry** — Use this menu option to hide all geometry. Other entities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

Setting file location options

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



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



File Options

New Files Existing Files Database Browser

Choose the tools and F/S information that will be used when a V9 or above part document is opened

Tools

Use the tool crib saved with the part document

Use FeatureCAM's tool crib

Feed / Speed

Use the F/S tables saved with the part document

Use the F/S tables from FeatureCAM's F/S database

Ask me when a file is opened that contains saved information

File Location:

Data\FeatureCAM Course Data 2017

Browse...

OK Cancel Apply Help

- 4 Select Browse. Then select the following location.
C:\Training_Data\FeatureCAM Course Data 2017.
- 5 Select **Apply** and **Ok** to close the form. When we save or import files **FeatureCAM** will take us to this location.
- 6 Please save all files in **Coursework**.
- 7 Tool Crib = **BasicMetric**.
- 8 Post processor used = **Fanuc 21i – T.cnc (2 Axis)**

What is a Feature?

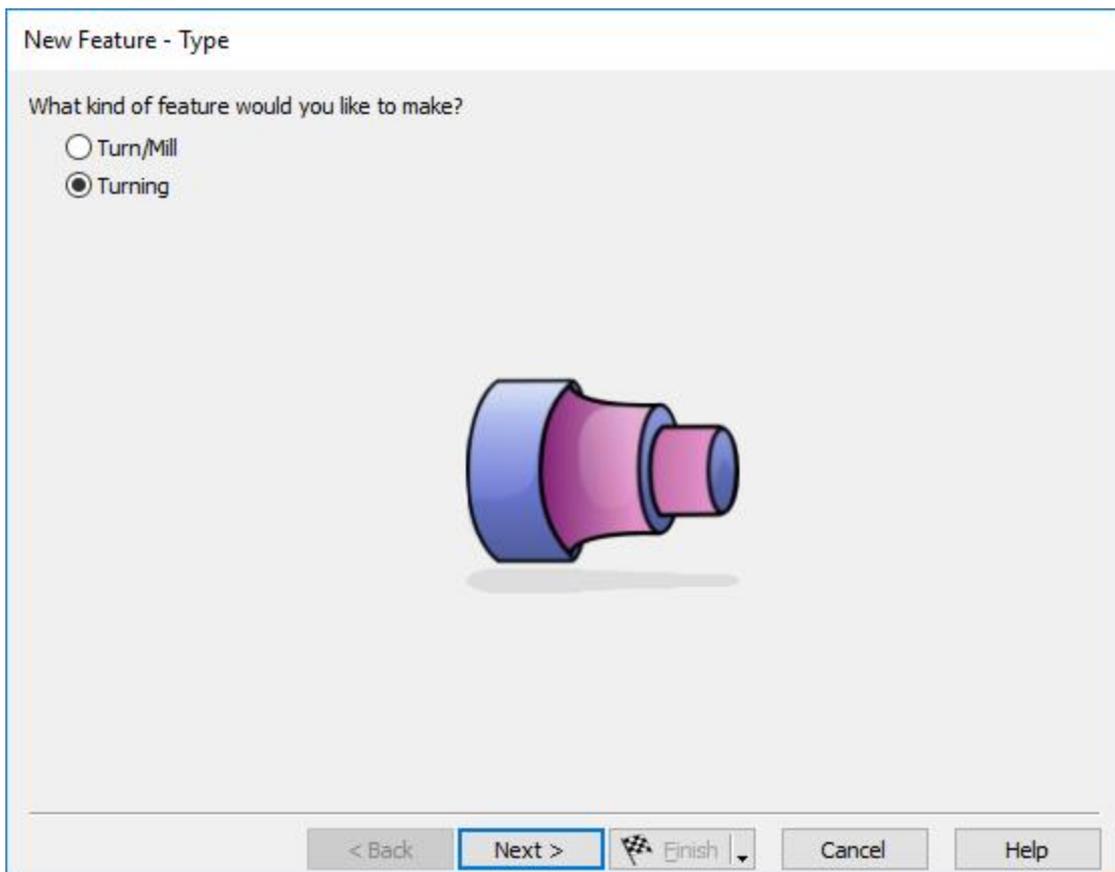
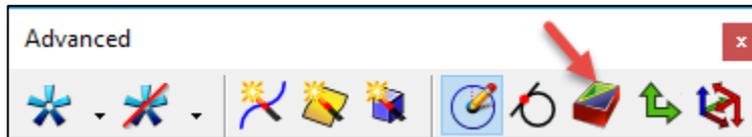
- If we look at a Part such as that shown on the next few pages and consider how we would manufacture it we can see that it can be broken down into a series of **Machining Operations**. For example we would probably want to start by **Facing** off the top of the part, then **Turn** the outside profile,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

- **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.



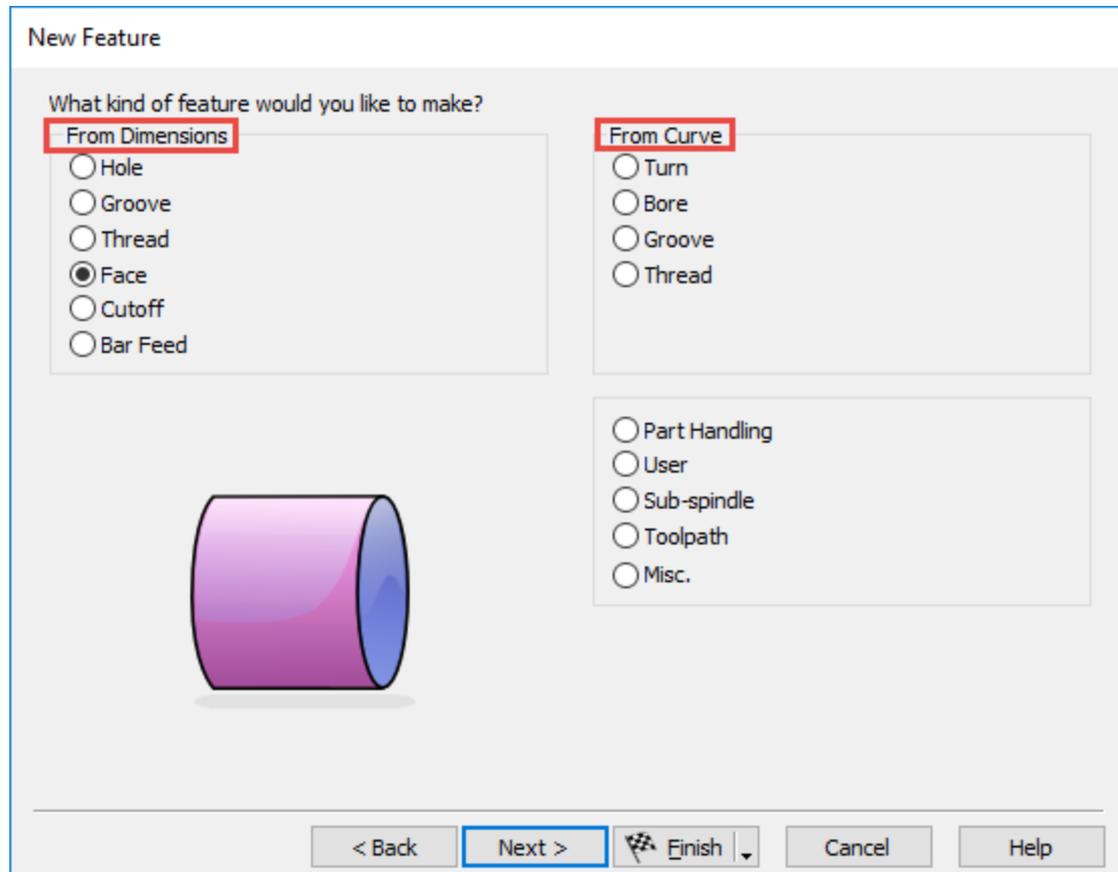
There are four different ways to activate the **New Features** menu.

- 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**.



How is a Feature Created?

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

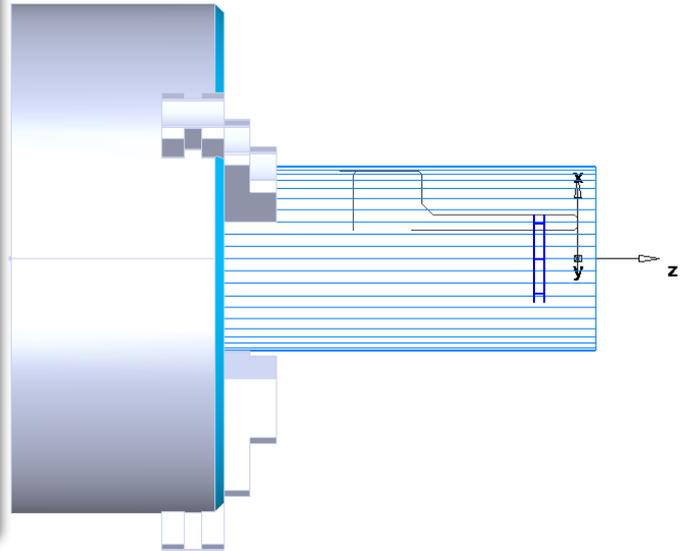
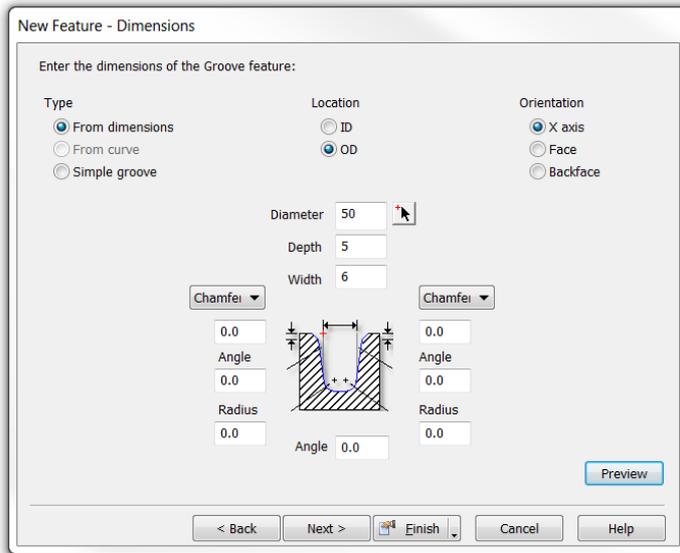


From Dimensions



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

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

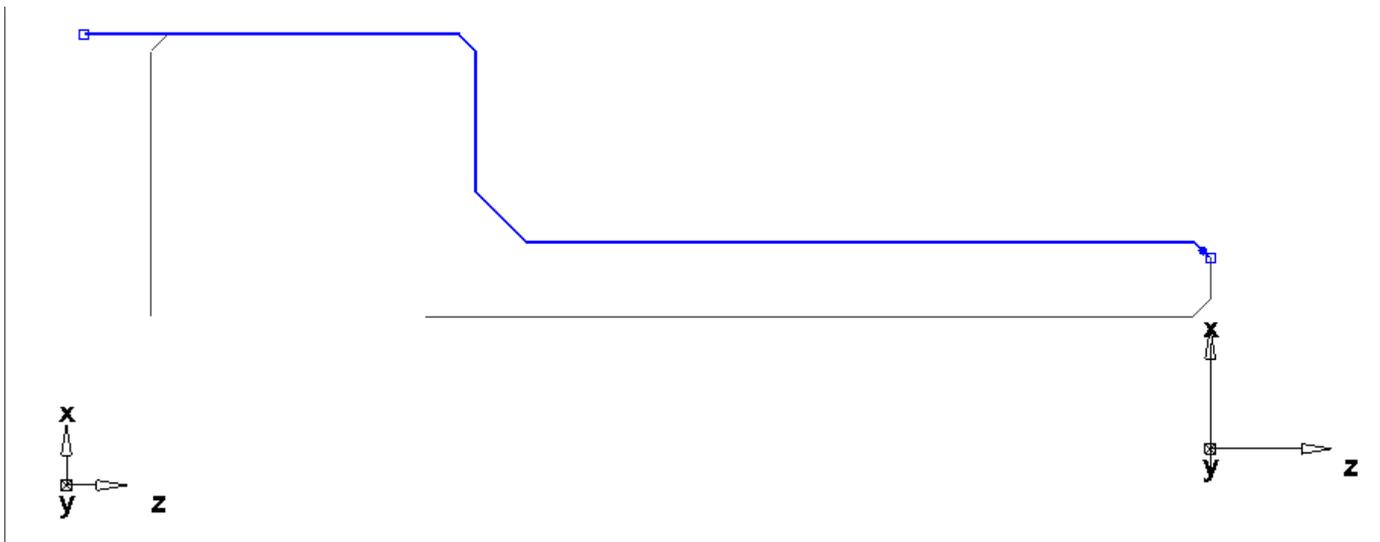


From Curve



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



Step 2: Pick next geometry segment, press ESC to select new anchor

Name: curve2

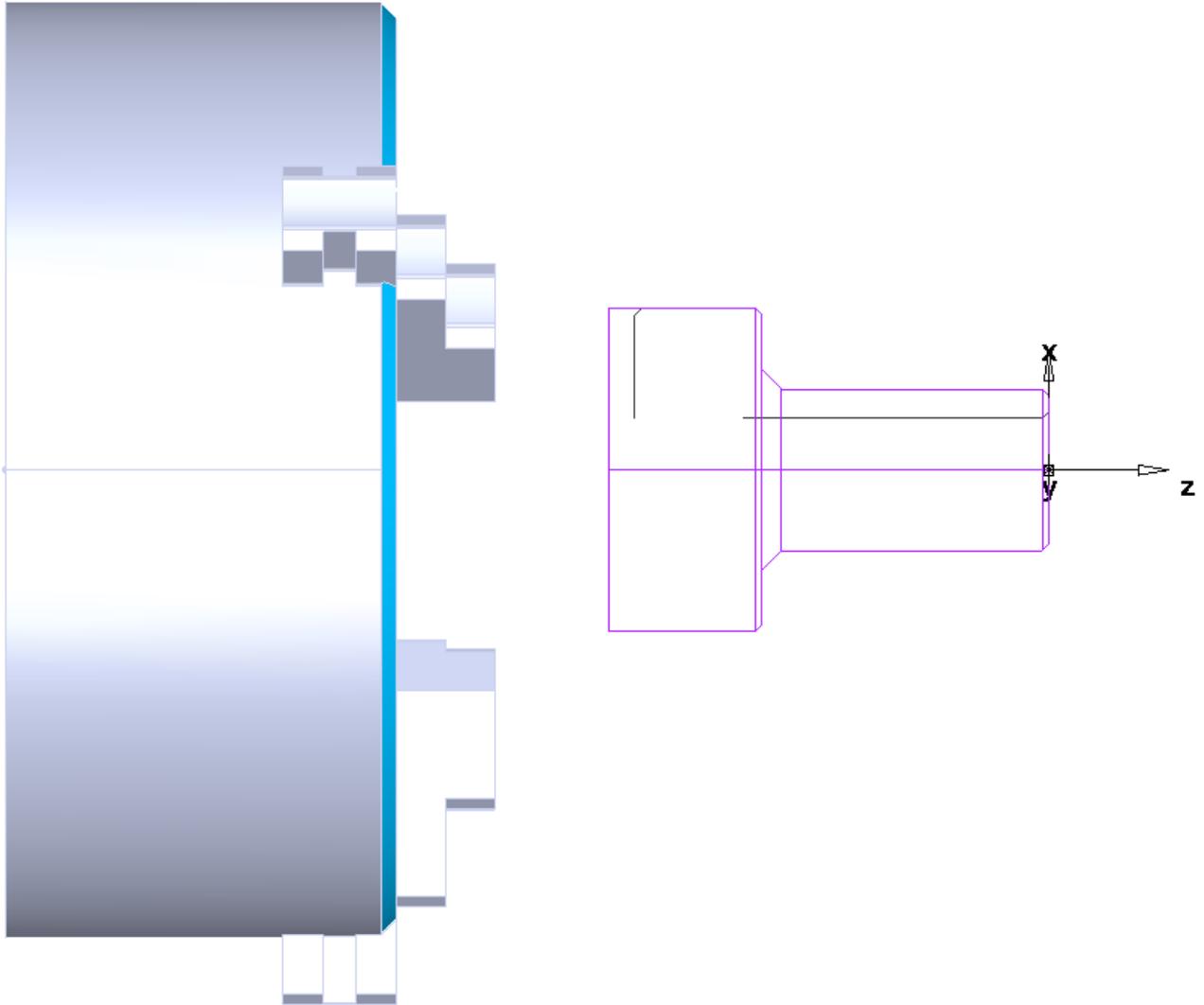


Clear Pieces

Create Layer 1

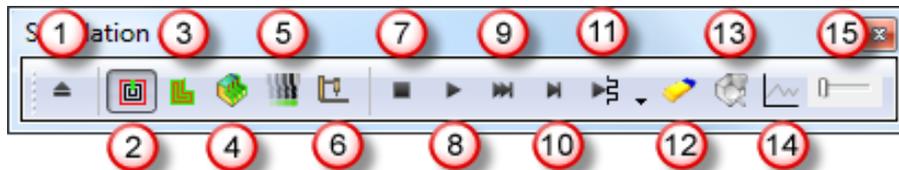
Plane: Grid UCS Setup Unrestricted

Options



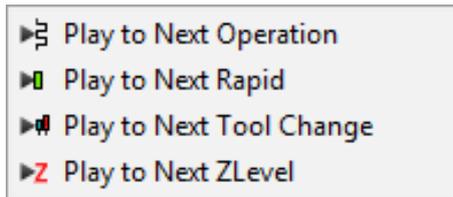
Select **3D Simulation** and then press **Play** **8** on the **Simulation Toolbar**.

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.
- 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.

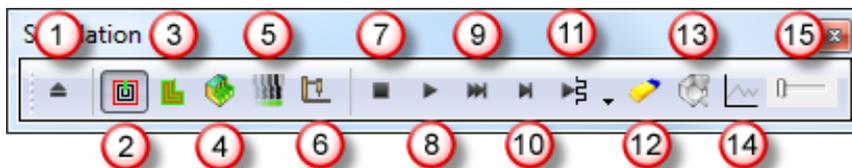


Next Rapid simulates until the next rapid tool move.

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.



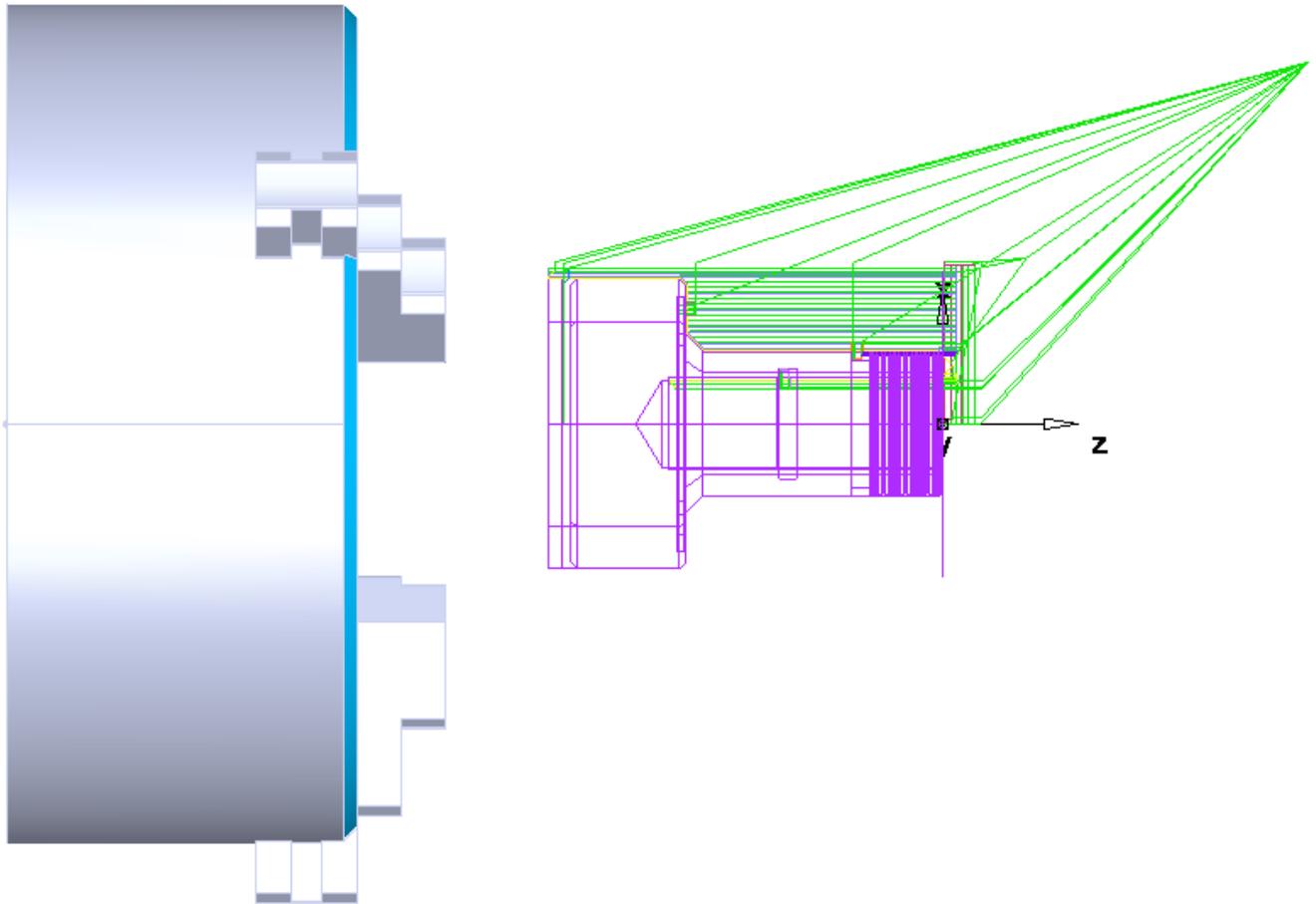
- 15 **Sim Speed:** To adjust the speed of a **Centreline, 2D or 3D Simulation**, use the **Slider** on the right-hand side of the controls.



*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.*



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.



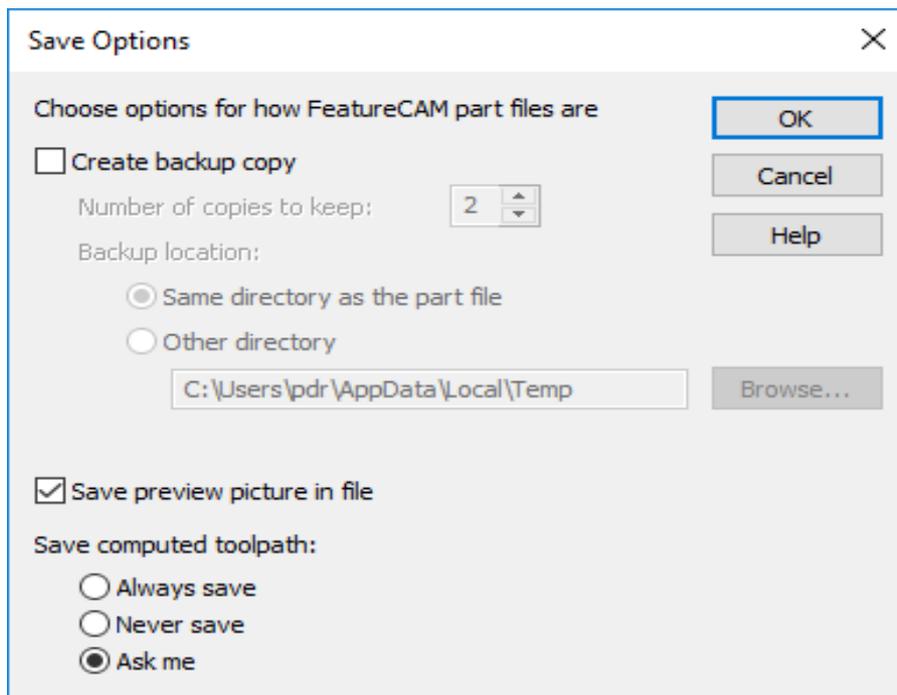
Code can only be output after the Simulation has finished. If the simulation fails, NC code will not be output.

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.

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.



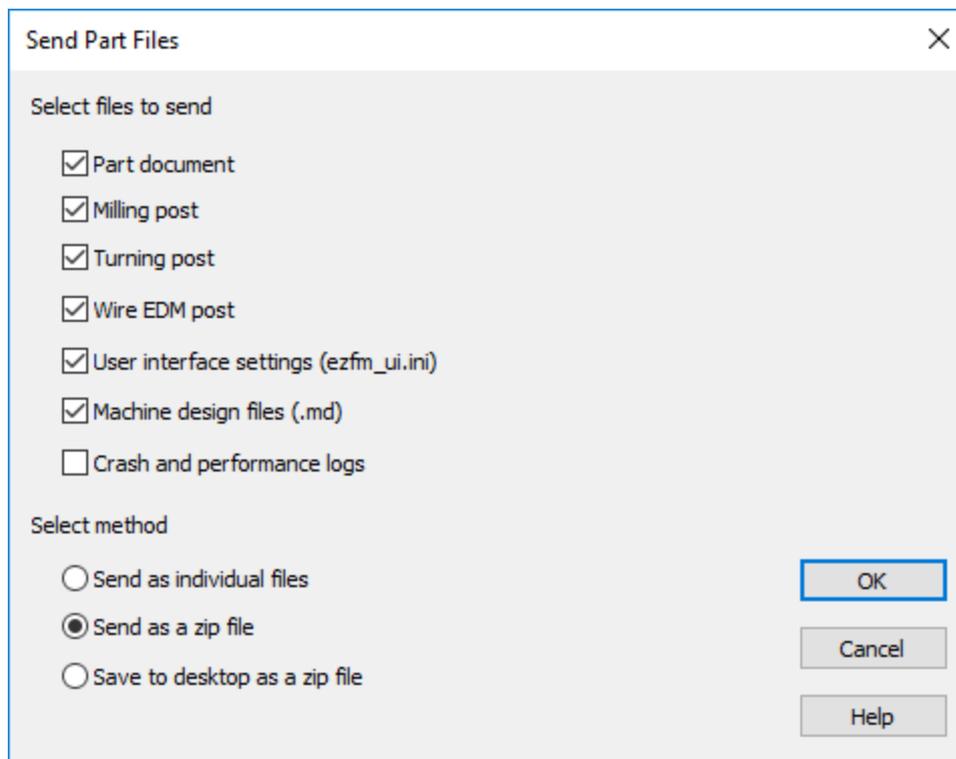
Cancel: - Click the **Cancel** button to close the dialog without saving any changes.



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

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.

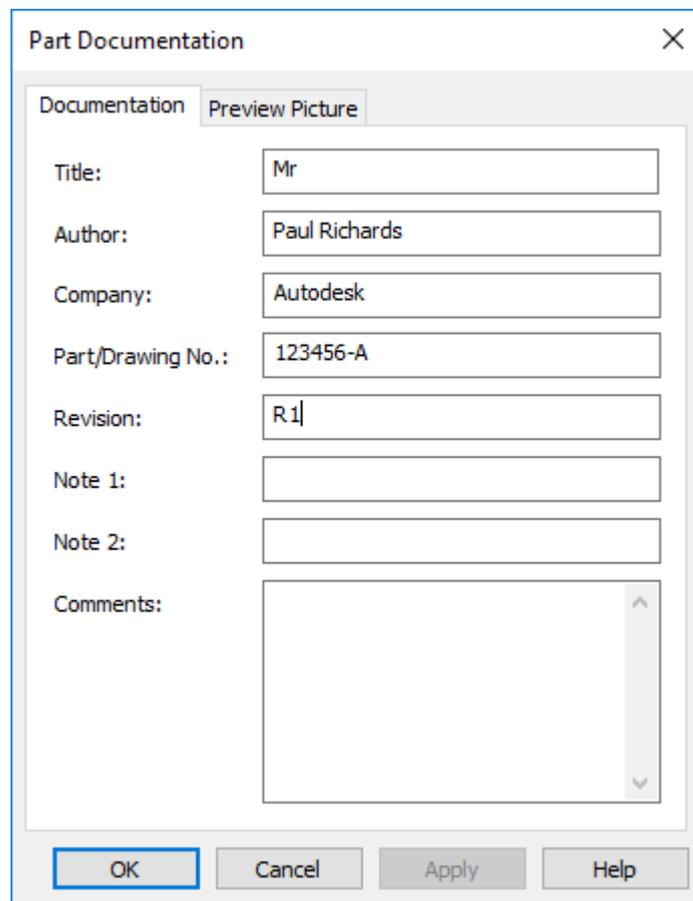
The **Part Documentation** dialog contains two tabs:

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.




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

Preview Picture tab

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

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

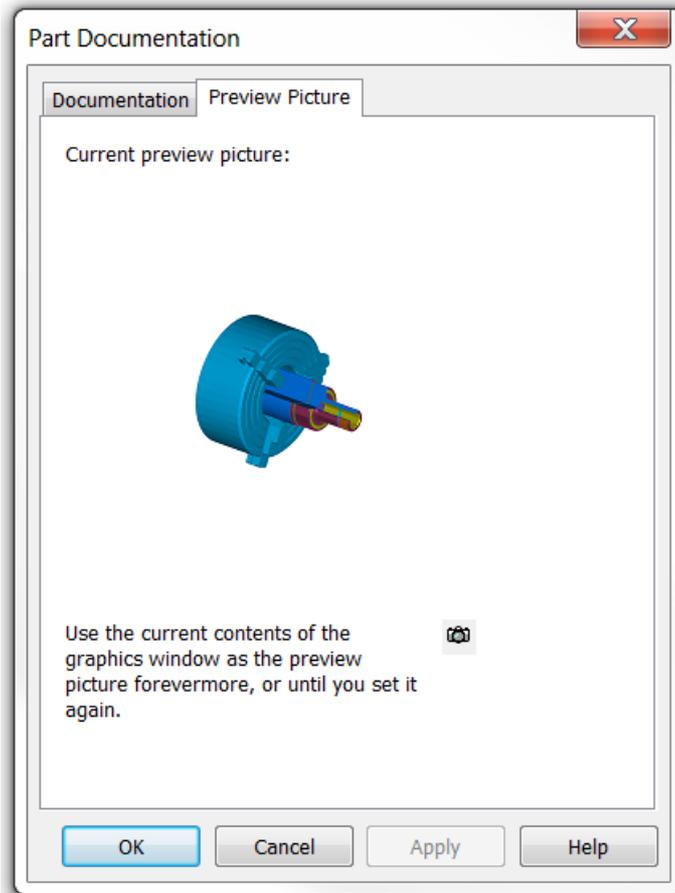
- 9 Create the view of the part you want to store.



- 10 Click Update preview picture



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



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

Saving your settings

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

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



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

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

Import/Export

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

Import/Export Options

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

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

The dialog has three tabs:

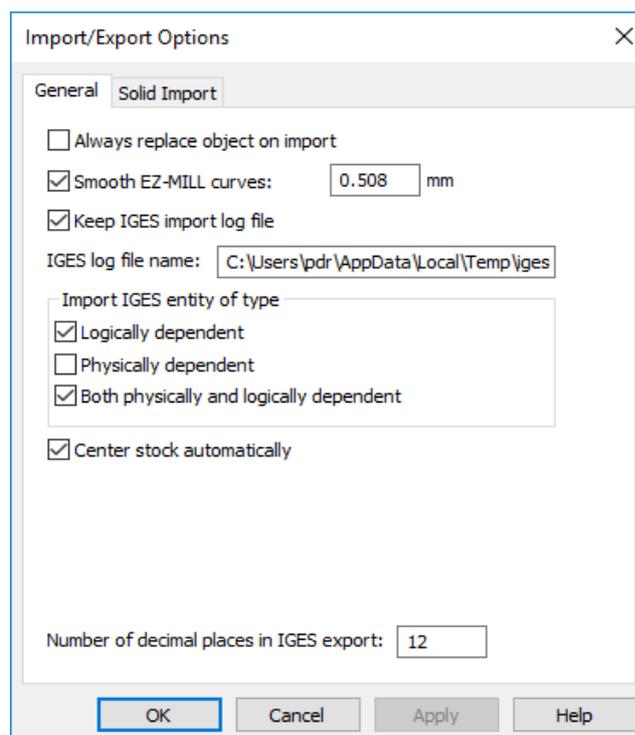
General

Solid Import

Digitized Data

General tab

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



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

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

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

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

Import IGES entity of type:

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

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

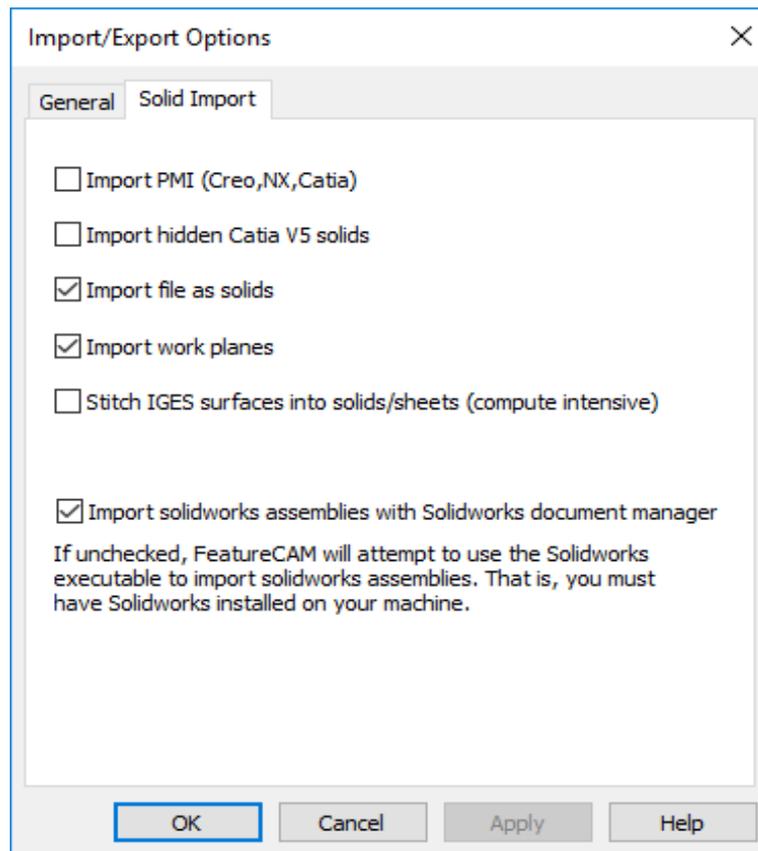
Both physically and logically dependent: - meets both logically and physically dependent criteria (is referenced by at least two other entities: - the parent for the logical link cannot be the parent for the physical as well).

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

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

Solid import tab

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



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

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

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

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

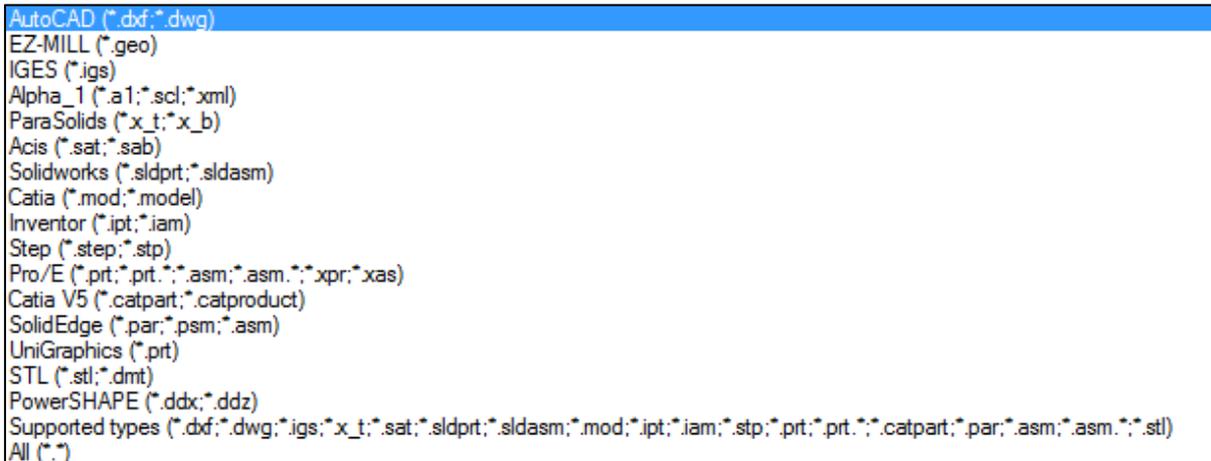
Import work planes: - When selected, planar surfaces in solids are imported. When deselected, planar surfaces in solids are ignored.

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

Importing Files

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

- 3 Open a new or existing part file. You must have a part open to import geometry.
- 4 If you want to set or change the import options, select **File > Import/Export Options**.
- 5 Select **File > Import** from the menu. The **Import** dialog is displayed.
- 6 Select the file you want to import, and click **Open**.
- 7 A message asks if you want to review the log file: click **Yes** to display the results, or click **No** to continue. The **Import Results** wizard is displayed.
- 8 Follow the instruction in the wizard to complete the process.



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



If this is the case then use the **Import using exchange** 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.

Contents

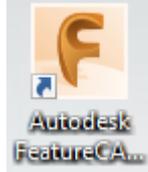
FeatureCAM Basic Turning.....	3
Introduction	3
Creating a New Turn/Mill Part Document.....	4
Basic Turning Exercise	6
Turning Input Modes	8
Geometry Creation Techniques	9
Curve Creation.....	12
Creating a Face Feature	14
Creating a Turn Feature.....	23
Multi –Turret options.	25
Creating a Bore Feature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

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

FeatureCAM Basic Turning

Introduction



Double click on the **FeatureCAM** Icon  this will start FeatureCAM and allows you to create a new Part document.



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

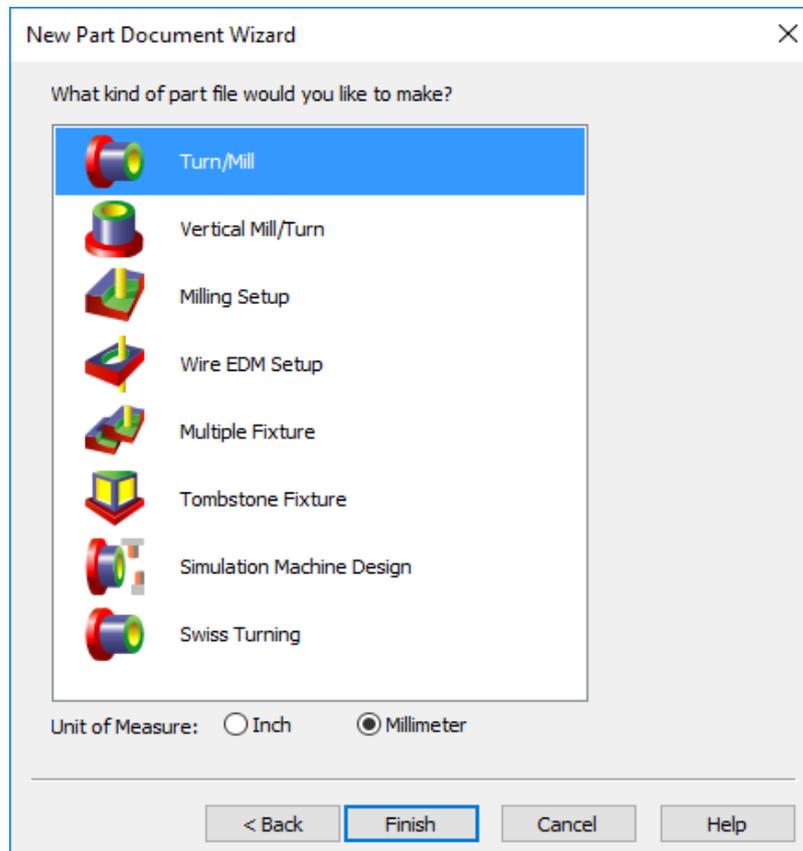
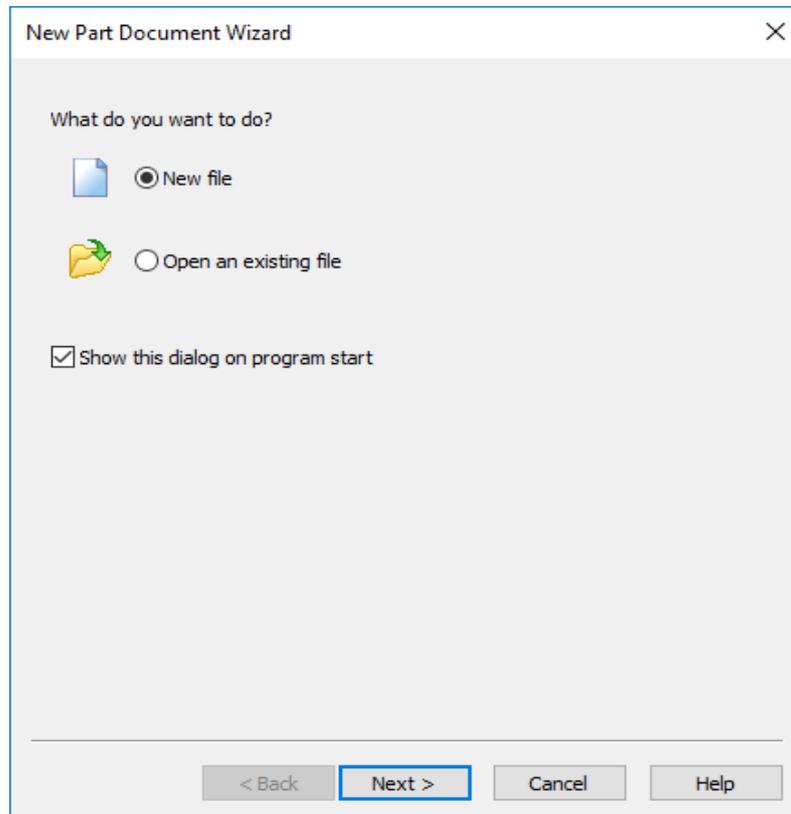
X and Z turning, boring, and grooving along with the basic user interface and different stock types. At the end of the module the user will be able to generate a program for a simple 2 axis part.



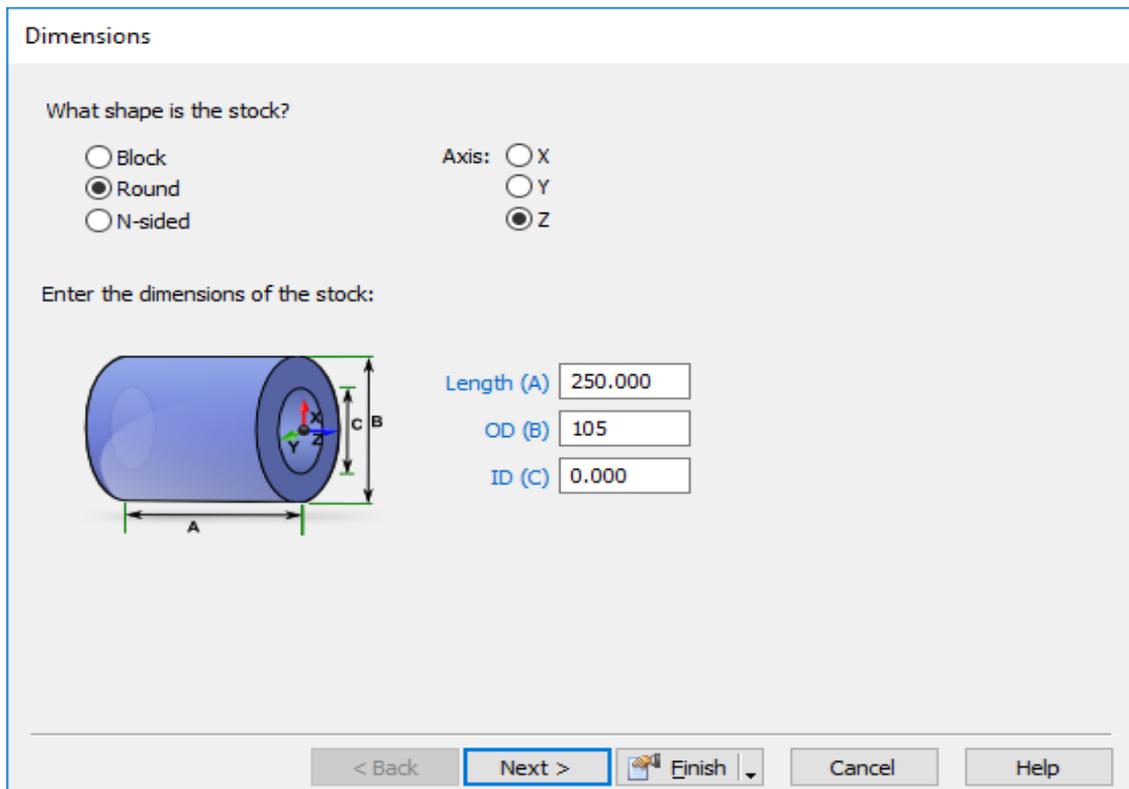
This chapter will show the user the basic steps required to create a finished **FeatureTURN** Part. The following Tutorial is designed for **Metric** use.

Creating a New Turn/Mill Part Document

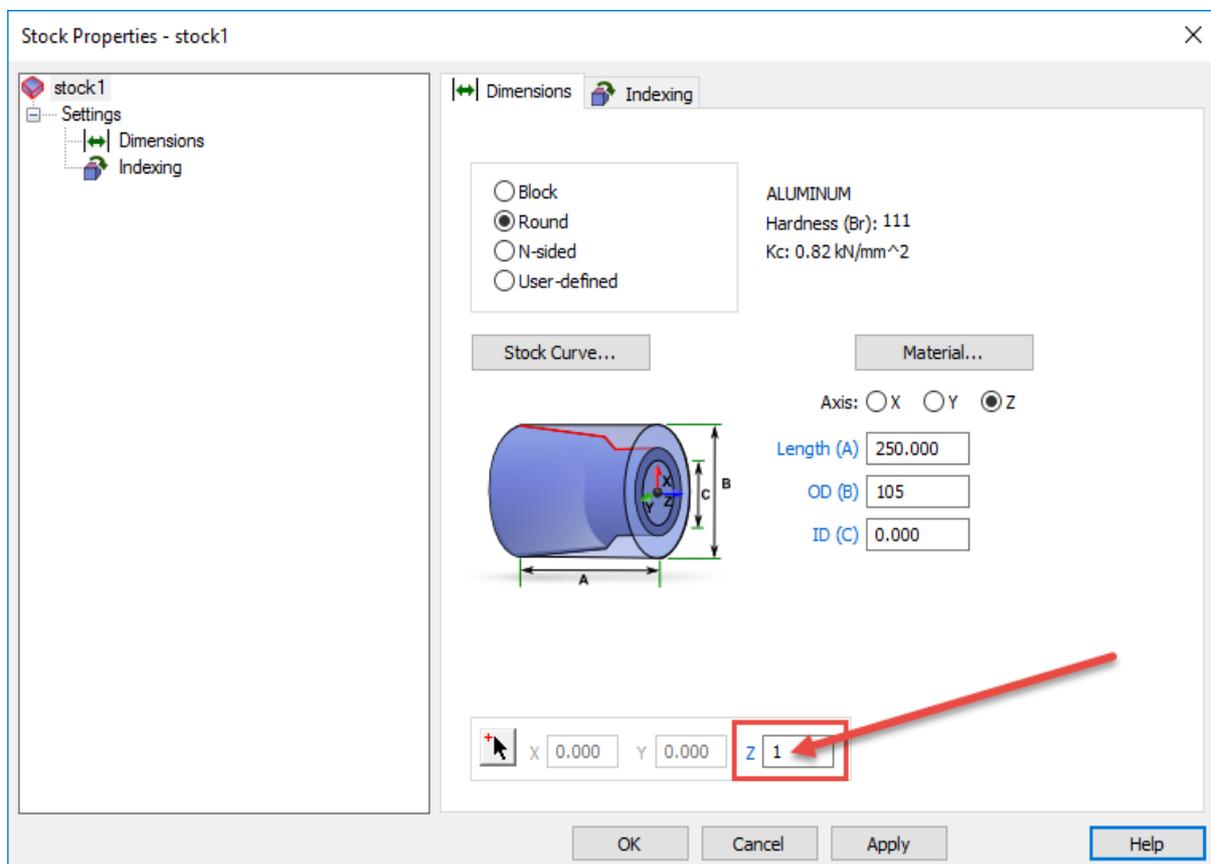
- Open a **New file > Turn/Mill** Part Document, the following menu will appear. Select **Next**.



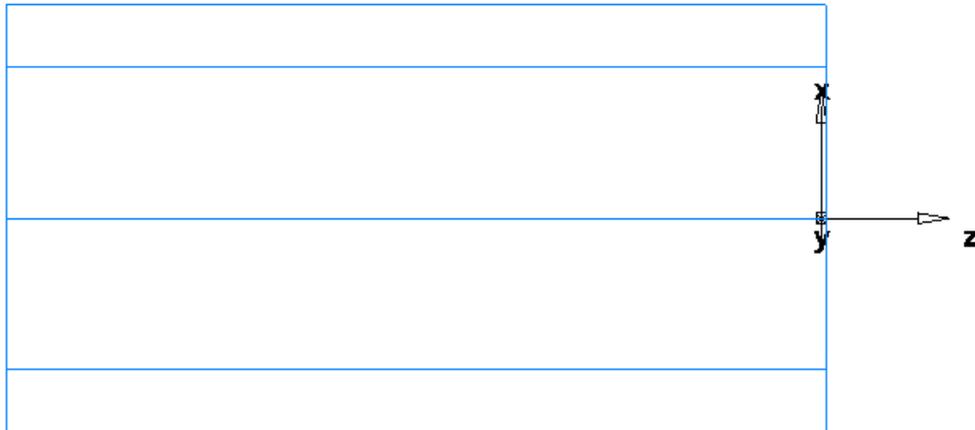
- 1 Select **Finish**.
- 2 For your Stock enter **Length 250mm** and **OD 105mm** then Select **Finish**.



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



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



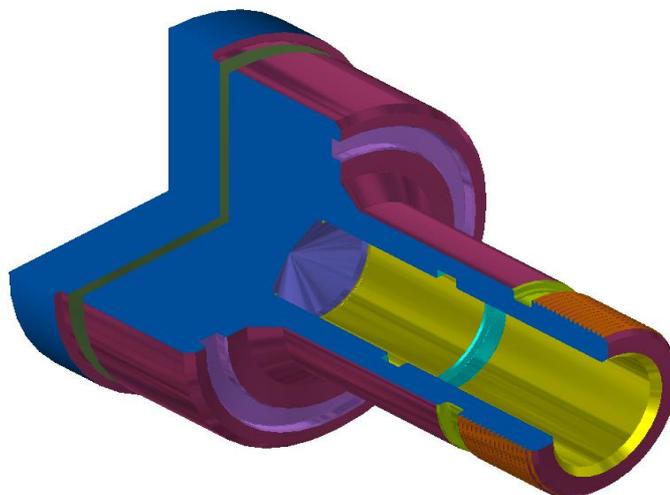
- 5 Hide stock by selecting the **Stock1** in **PartView** and Right click – Select **Hide Stock**.
- 6 In the **Z Offset** enter **-1mm** this will move the Offset **1mm** to the left of the material face leaving stock to be turned off later.

Basic Turning Exercise

- 1 Before we start please select **Options/File Options/Existing Files**.
- 2 **Select Browse. Then select the following location. C:\Training_Data\FeatureCAM Course Data 2017.**
- 3 Select **Apply**. When we save or import files FeatureCAM will take us to this location.
- 4 Please save all files in **Coursework**.
- 5 Tool Crib = **Training_Crib_Metric**.
- 6 Post processor used = **Fanuc21i-T.cnc**



The part we are going to machine is shown below.





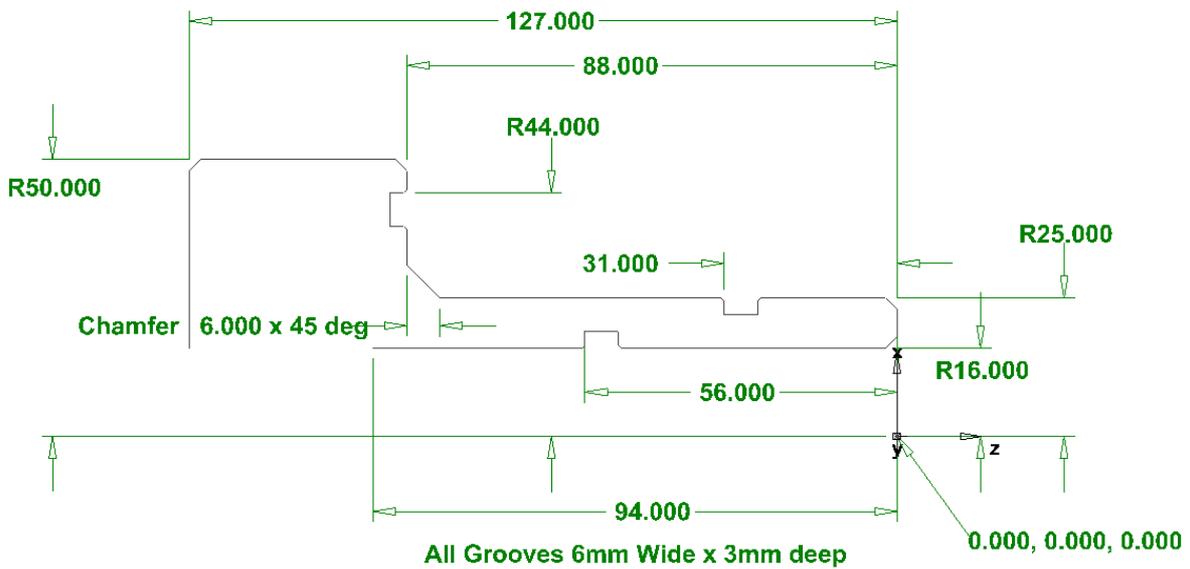
The example shown is a simple turned part designed to introduce you to several of the drawing and machining tools available in **FeatureTURN**. The tutorial walks you through creating the part and introduces you to **FeatureTURN's** automation and ease of use.



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

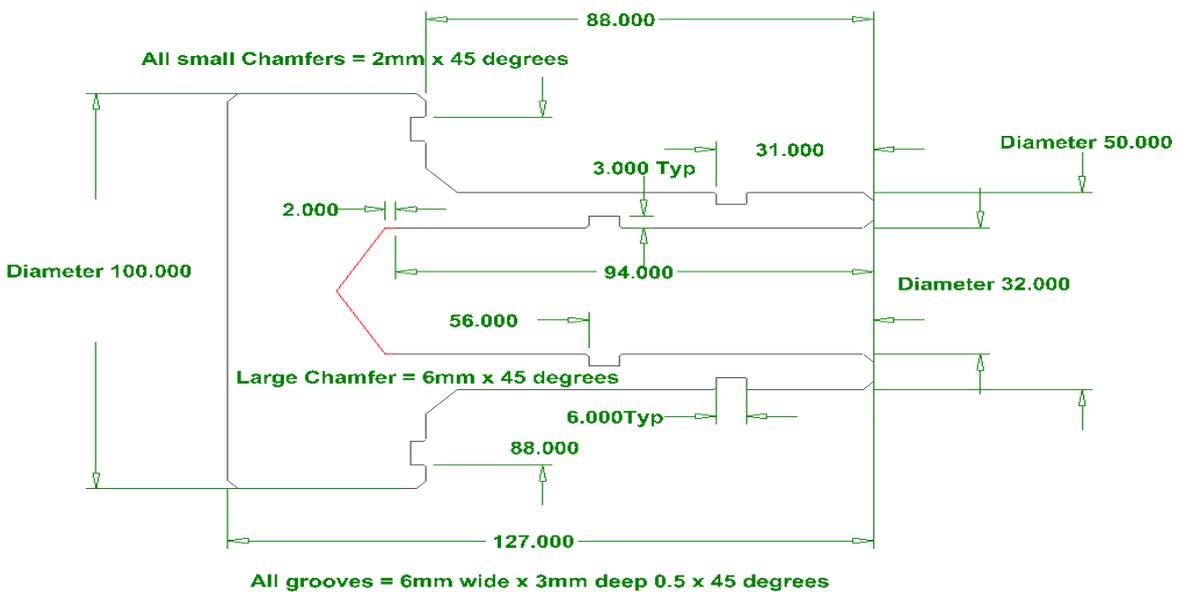
Radius (RZ)

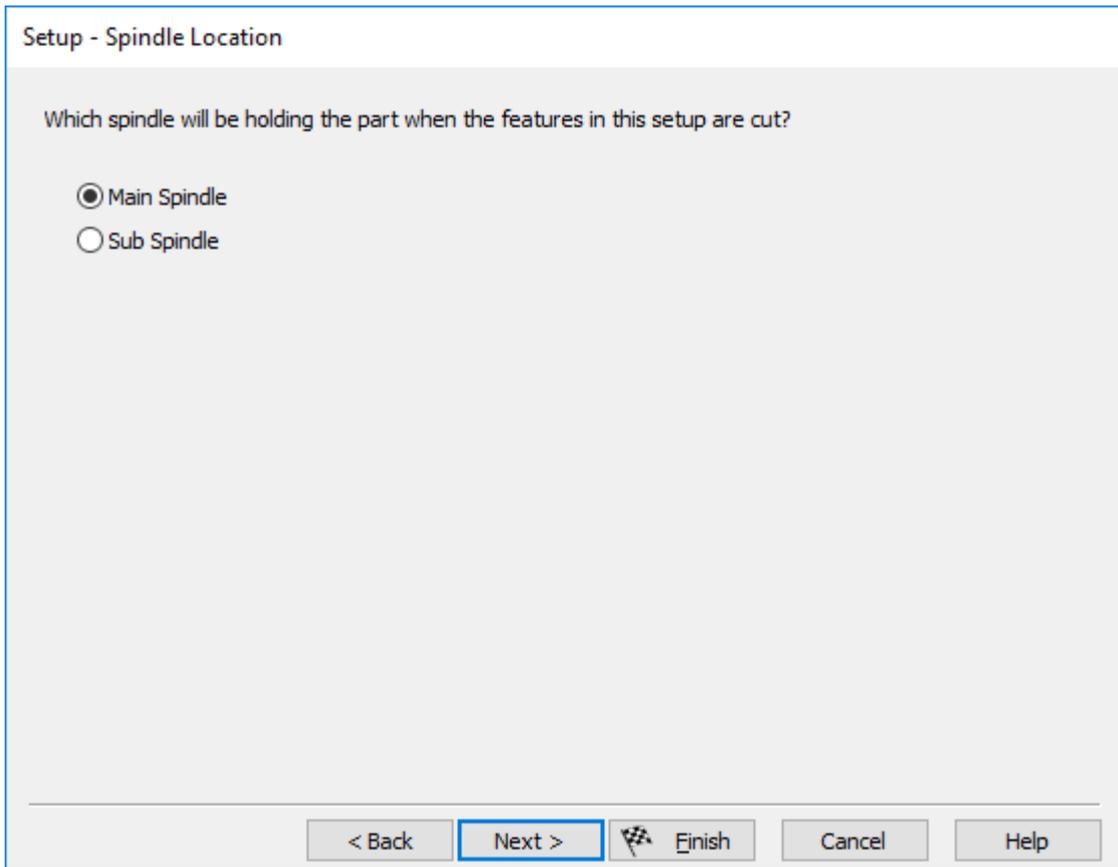
Chamfers 2mm x 45 deg Typ 4 Places



Groove Chamfers 0.5mm x 45 deg Typ 6 places

Diameter (DZ)





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



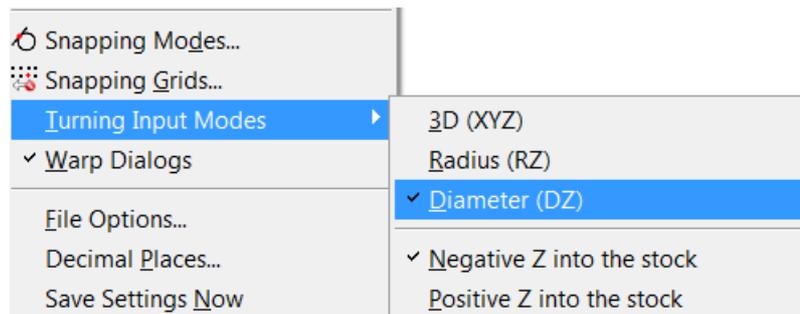
A sub spindle is a second spindle at the tailstock end of the machine opposing the main spindle and used to run the second side of the part. Machine dependant.

- 9 Select **Finish**.

Turning Input Modes

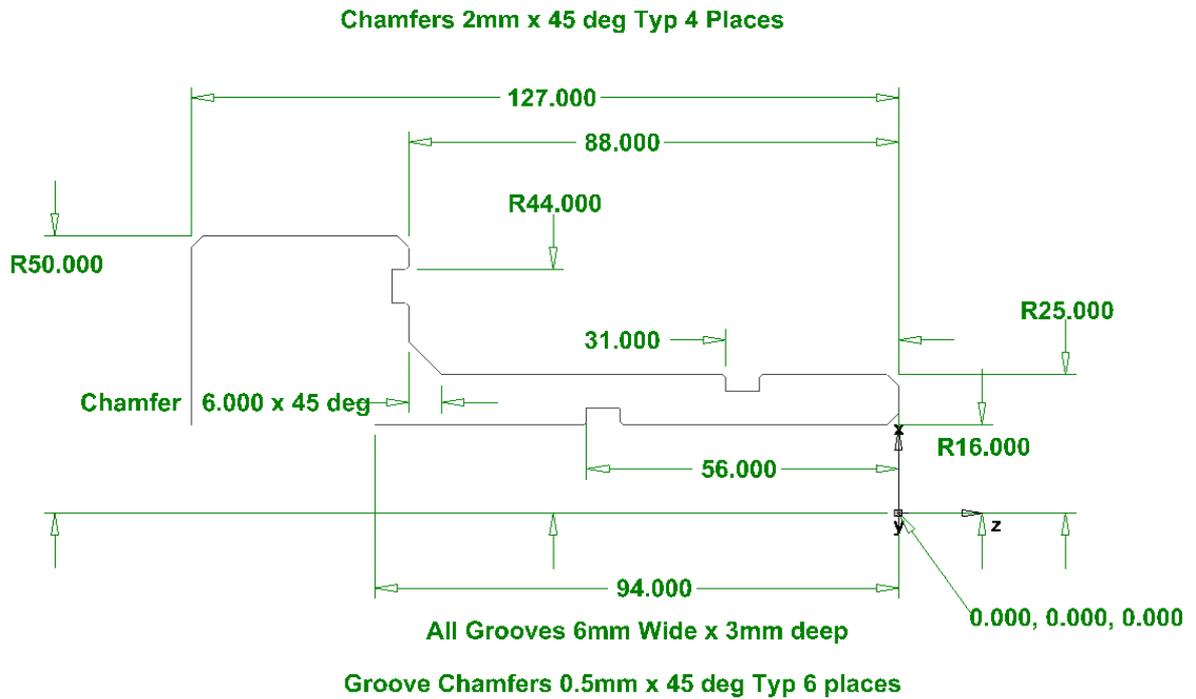


*Before we start to create out geometry we need to select **Options>Turning input modes** and select the following **Radius (RZ)** or **Diameter (DZ)** input modes. Our drawing is dimensioned in **Radius (RZ)** and **Diameter (DZ)** values. You choose your preferred values.*

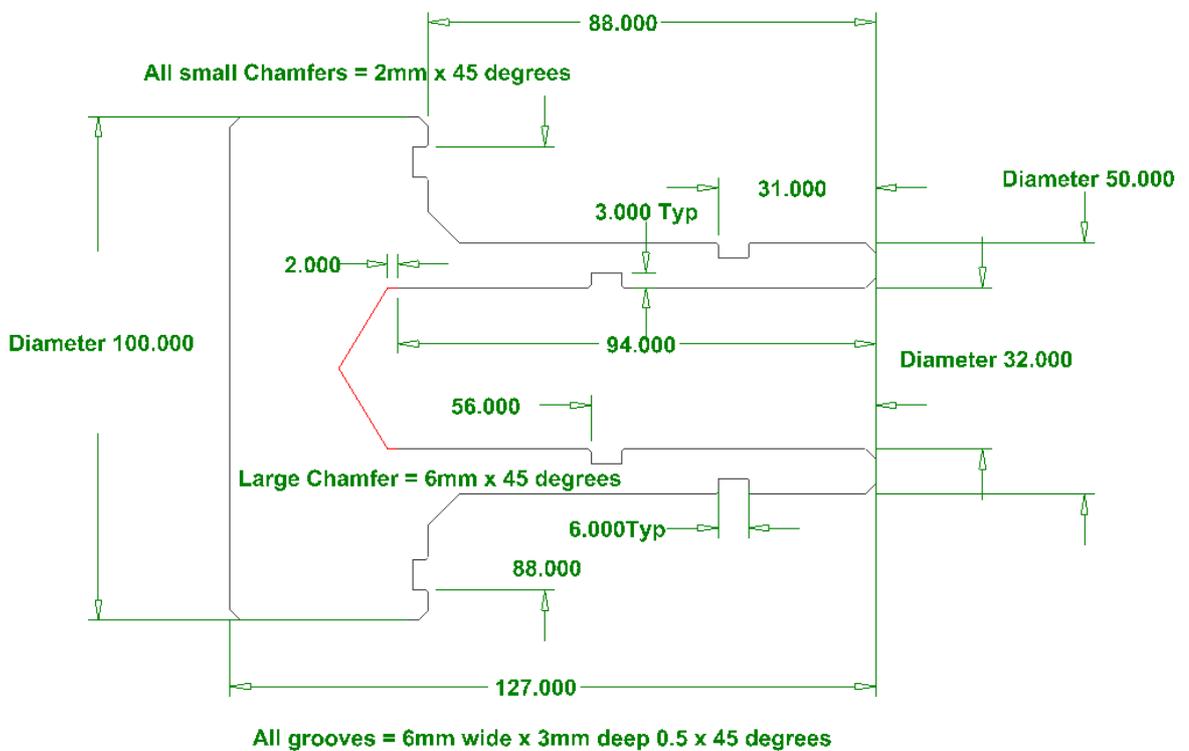


Geometry Creation Techniques

Radius (RZ)



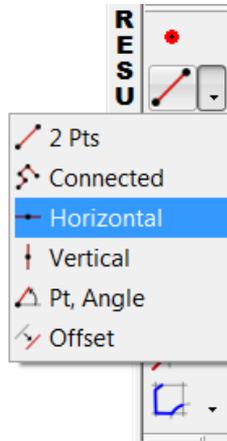
Diameter (DZ)





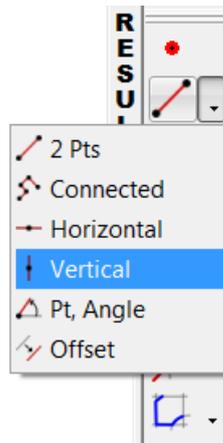
There are many ways to draw this job but the following method is by far the easiest. We are going to use Diameter values.

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



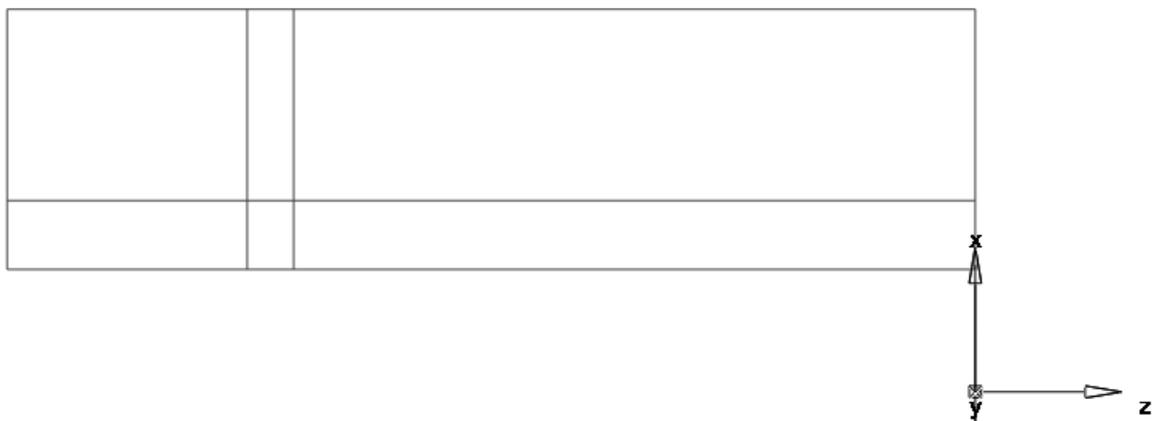
- 11 Input the following diameter Values **32mm, 50mm, 100mm.**

- 12 Then Select the Line **vertical** and input the following values



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

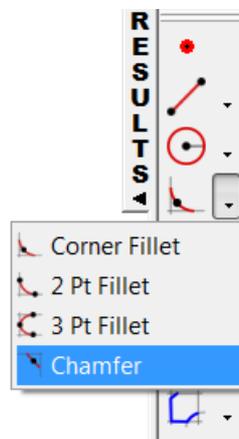
- 14 Clip  the surrounding lines as shown.



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



- 16 Now add the Chamfers by using the Chamfer command.

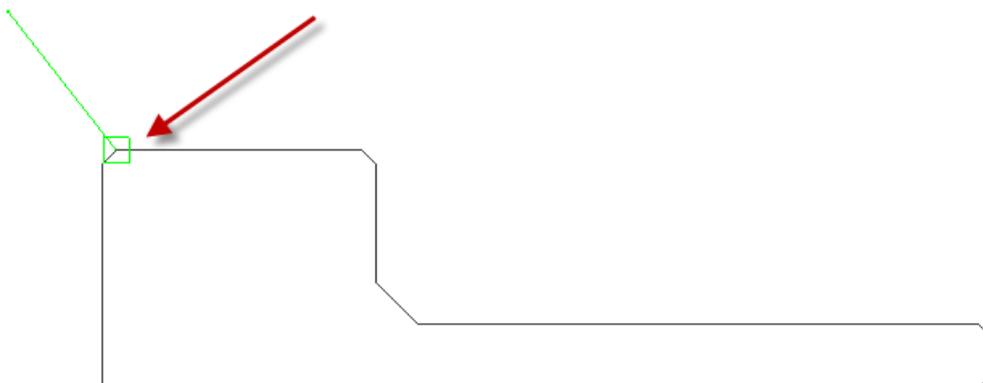


- 17 The large chamfer is **6mm x 6mm** all other chamfers are **2mm x 2mm**. We have one more item to change.



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

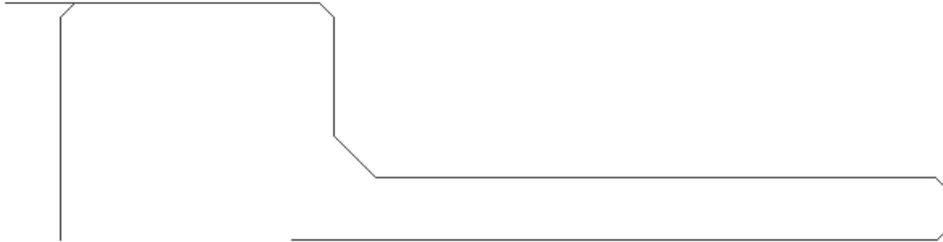
- 18 To do this, select a **Line from two points** from the **Geometry** menu. Snap to the chamfer as shown.



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

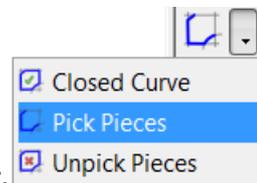


20 That is it the Geometry is now complete as shown.

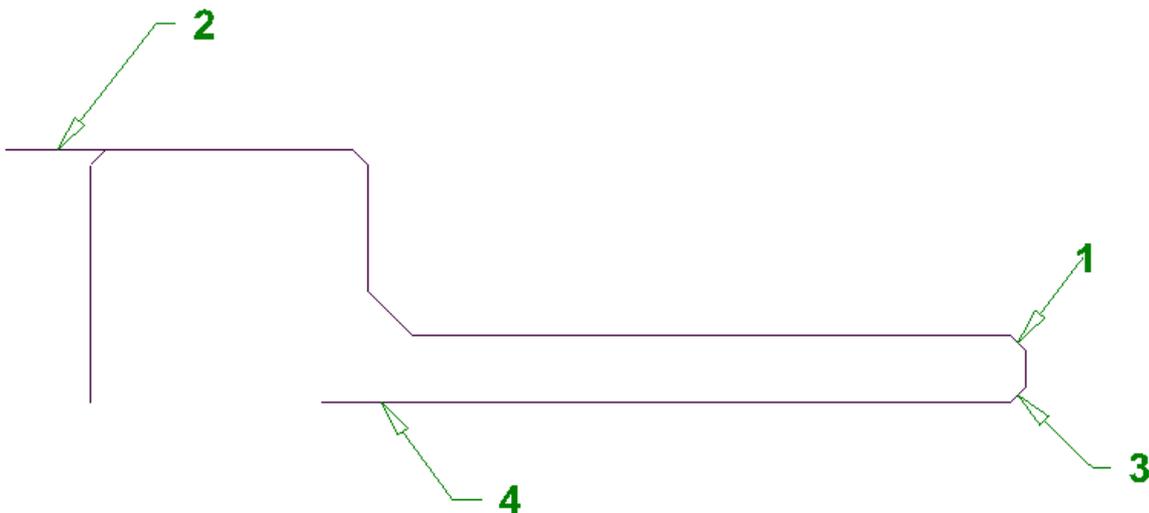


Curve Creation

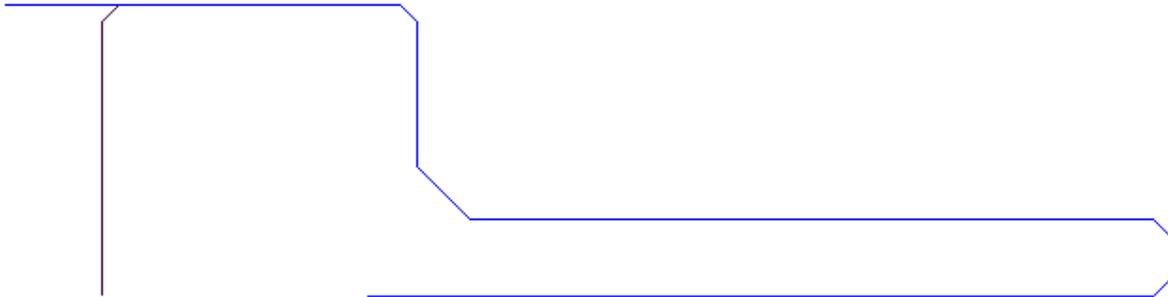
21 Select **Pick Pieces** from the Geometry Toolbar.



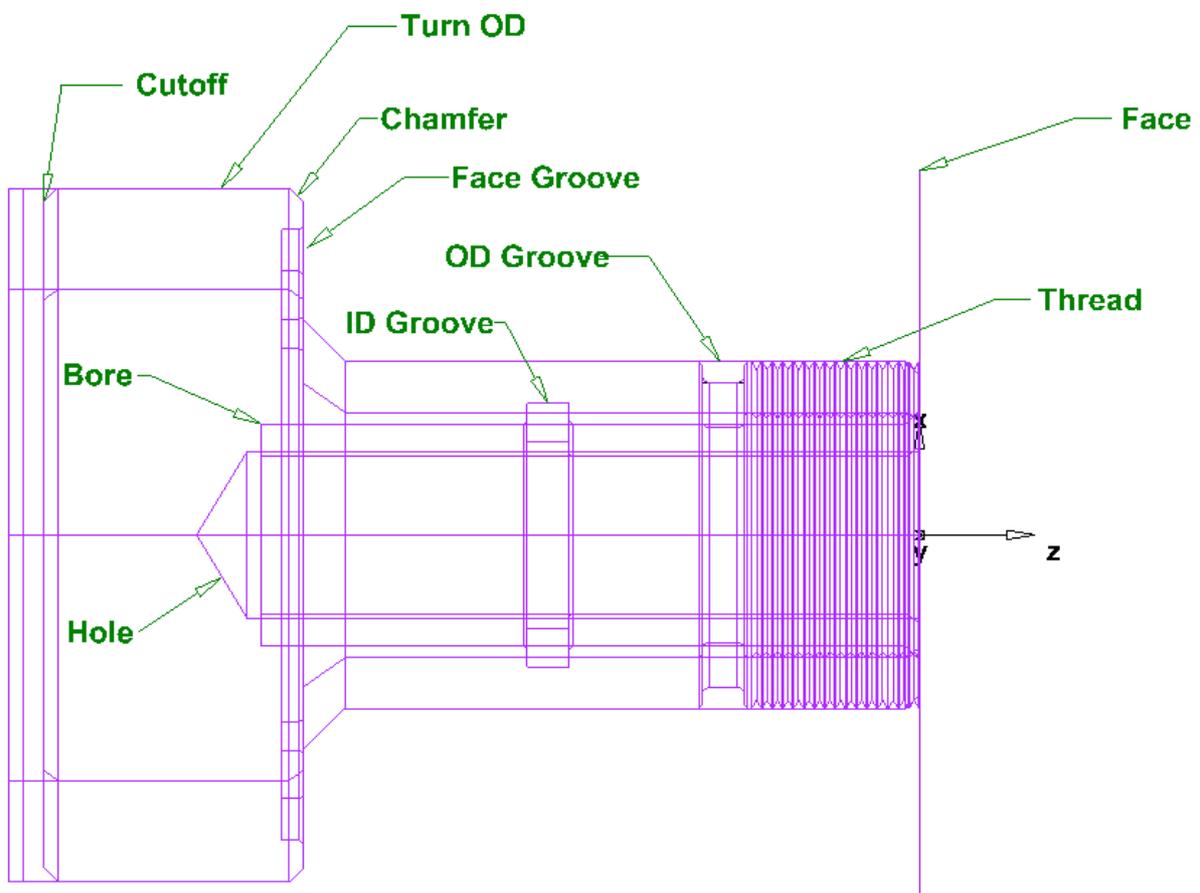
22 Select the **Pick Pieces** Icon  Click on locations **1, 2**, as shown in the image below. Each line segment will change colour when it is selected. Name this curve "**od profile**" and press the **ENTER** key. **Chain** the Bore **ID** curve by clicking on locations 3 and 4. Label it "**bore profile**"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



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

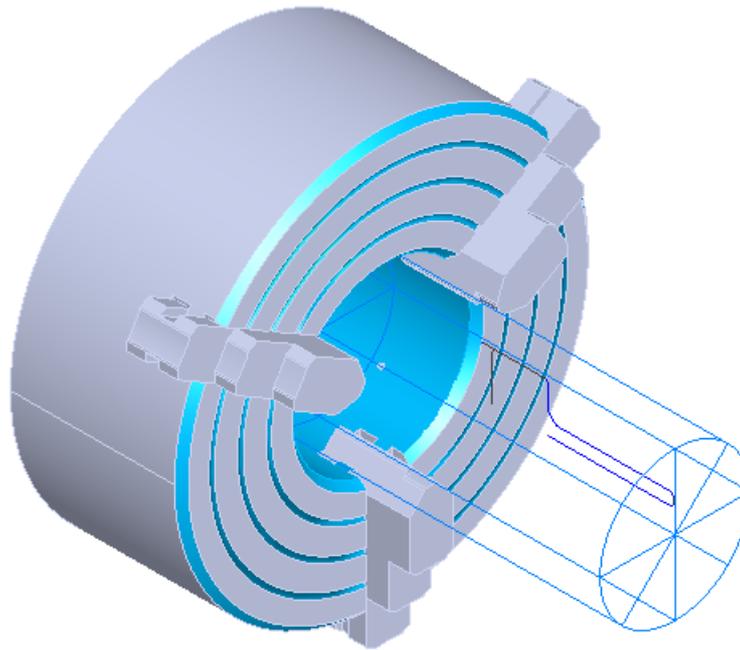


- 23** You should now have **two Curves** that will be shown in the **Part View**.
- 24** There is no need to create the **ID**, **OD** and **Face** grooves in the geometry as we will use **Feature from dimensions** from the **New Feature** menu.



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

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

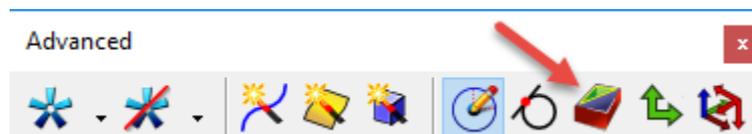


- 2 Please **cancel** the **import wizard**.
- 3 In **PartView** please select **all solids** by using **group select** and right click on the solids, from the menu select **Use Solids as Clamps**. This will then allow you to visualise the chuck when you run the **3D Simulation**.

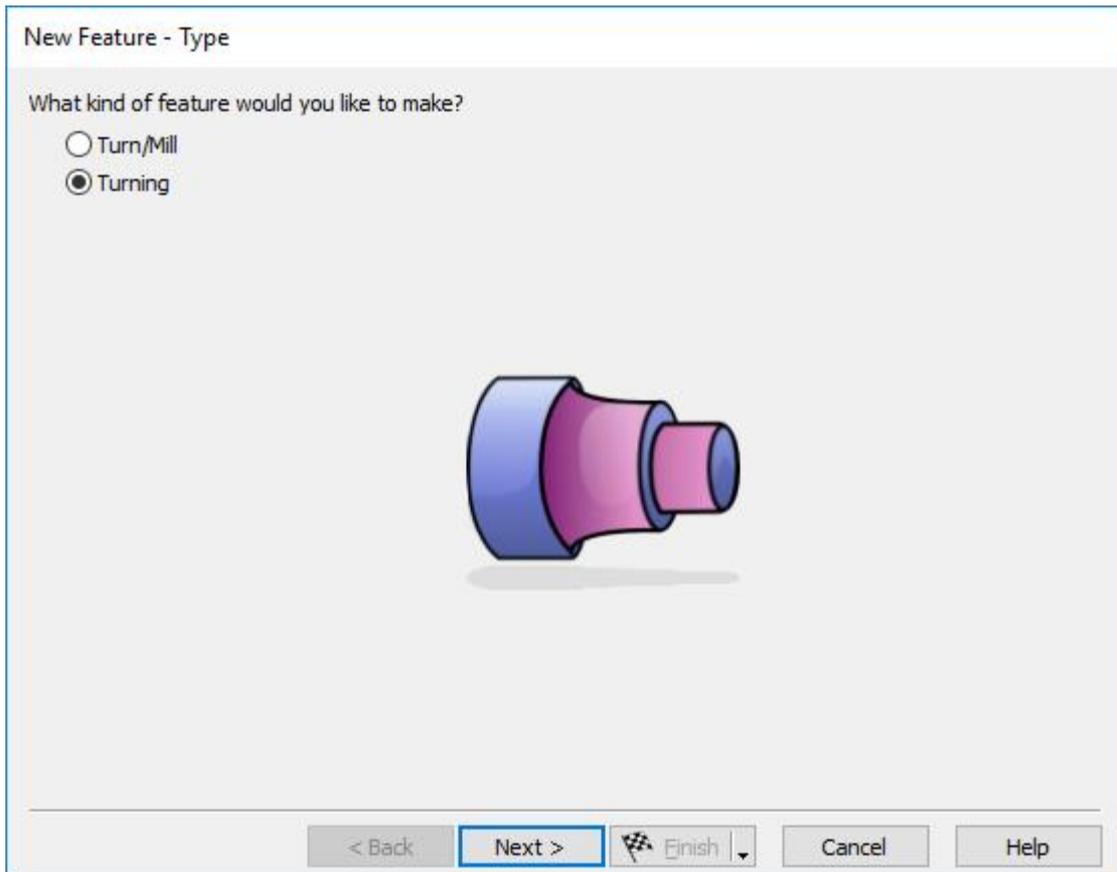
Creating a Face Feature



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

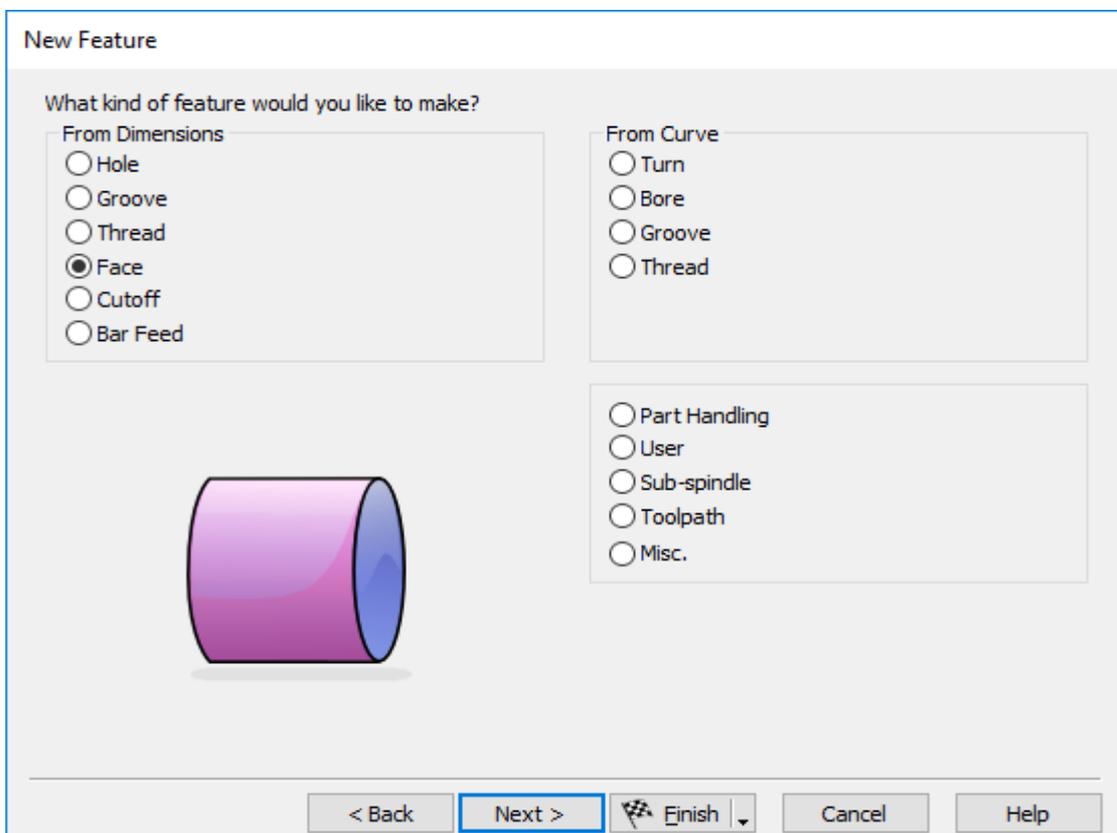


- 4 Select **Turning**.

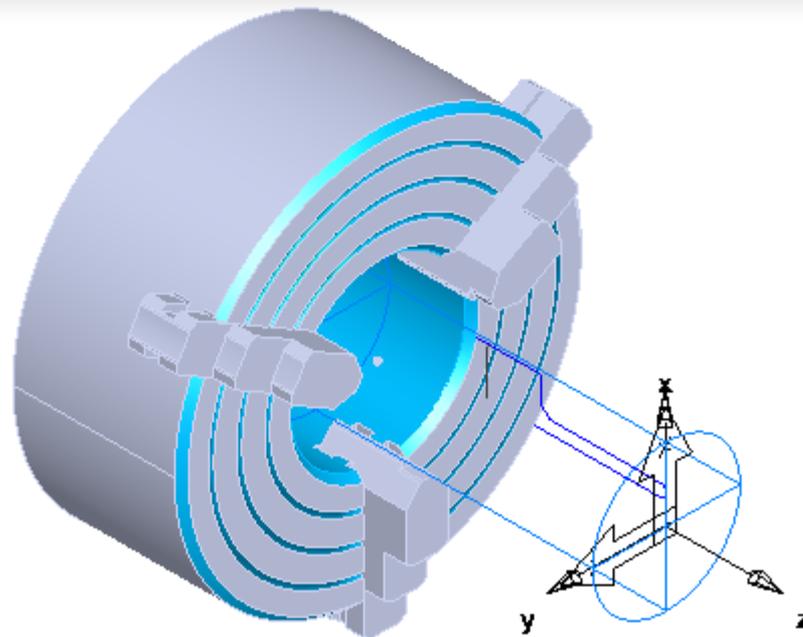
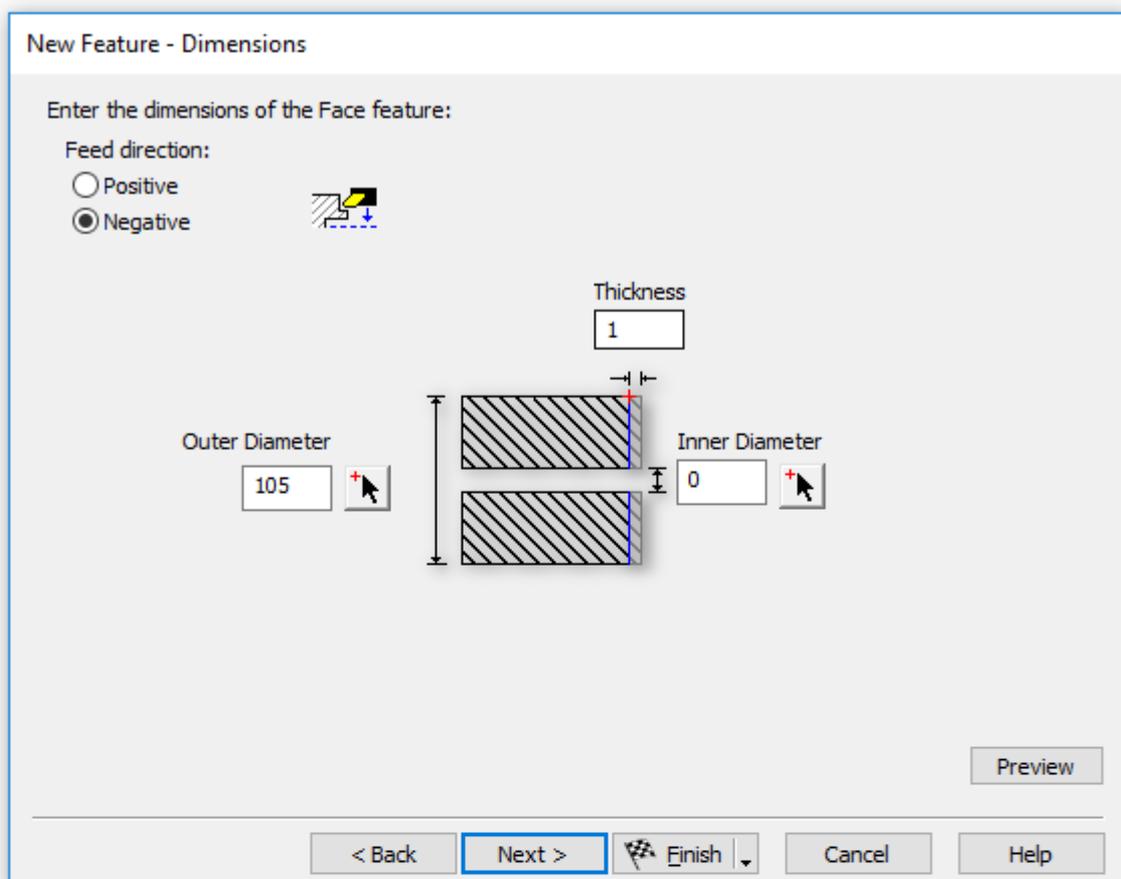


5 Select **Next**.

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

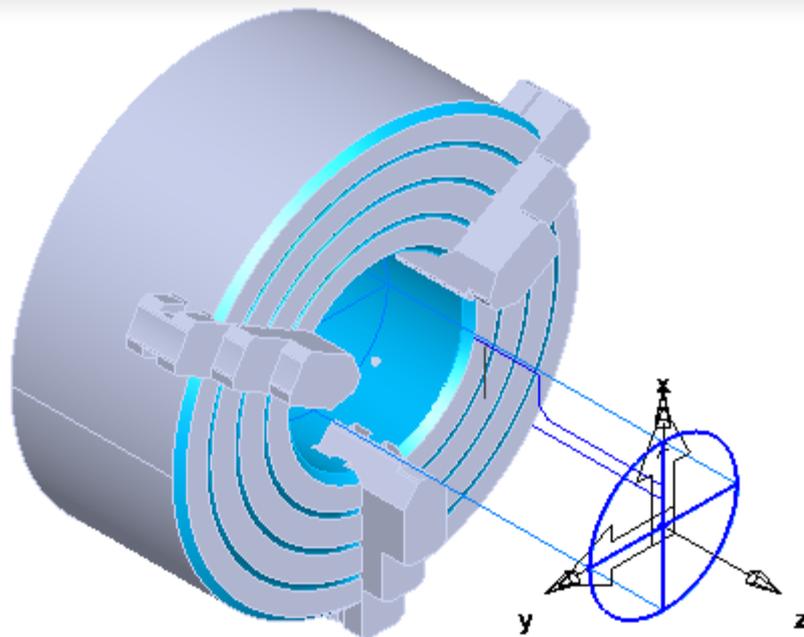
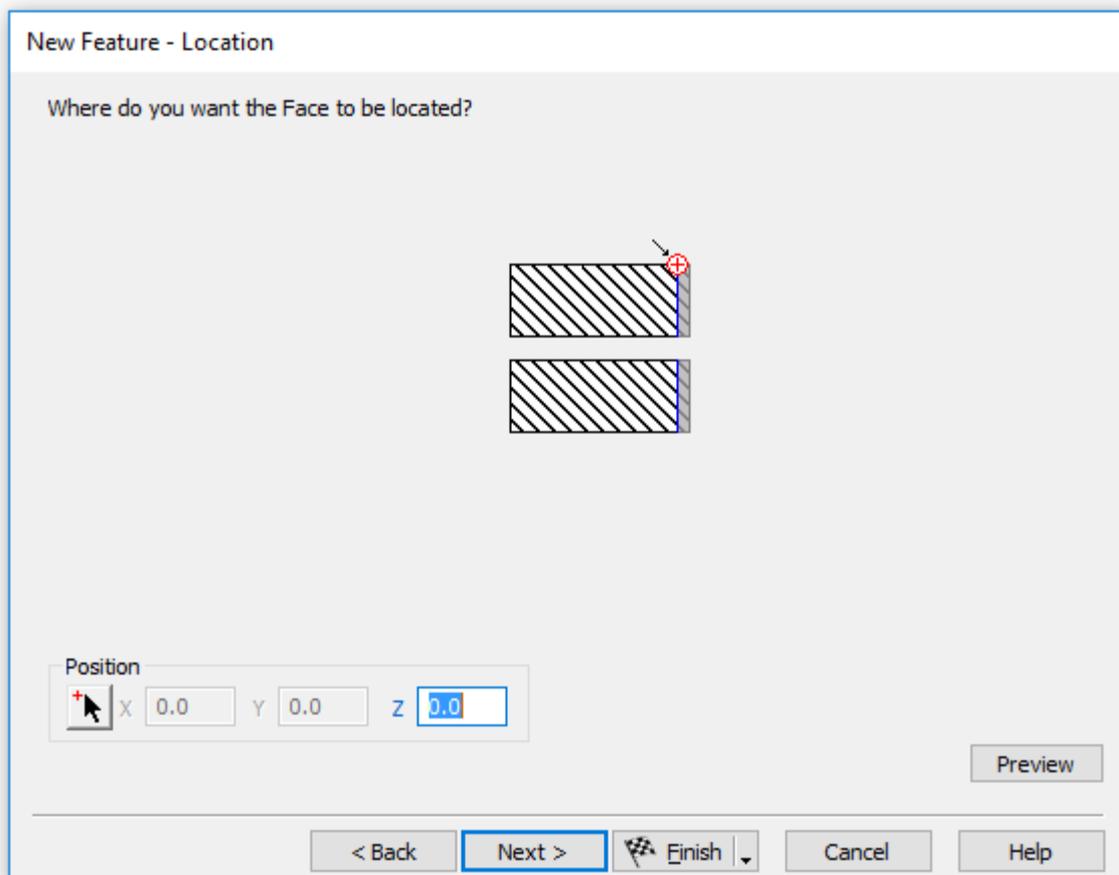


- 7 Select **Next**.
- 8 The following menu will appear.



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

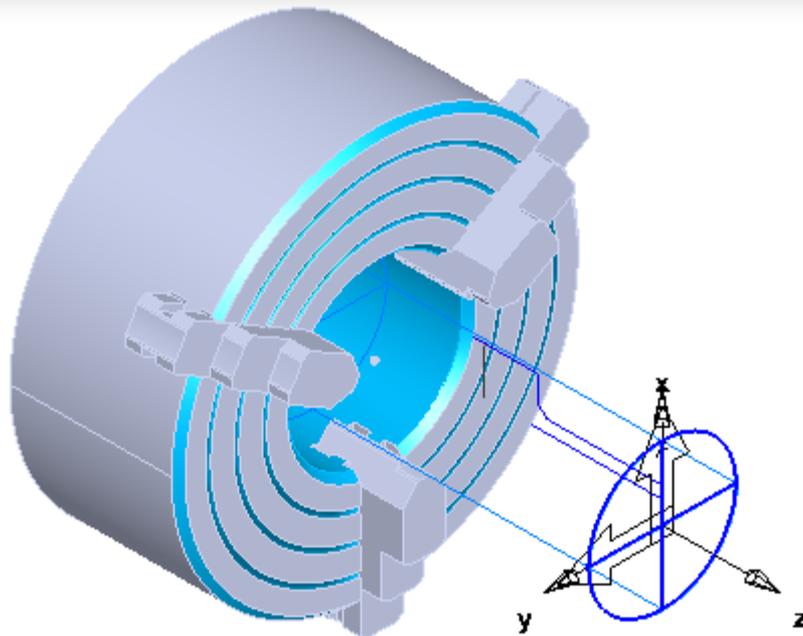
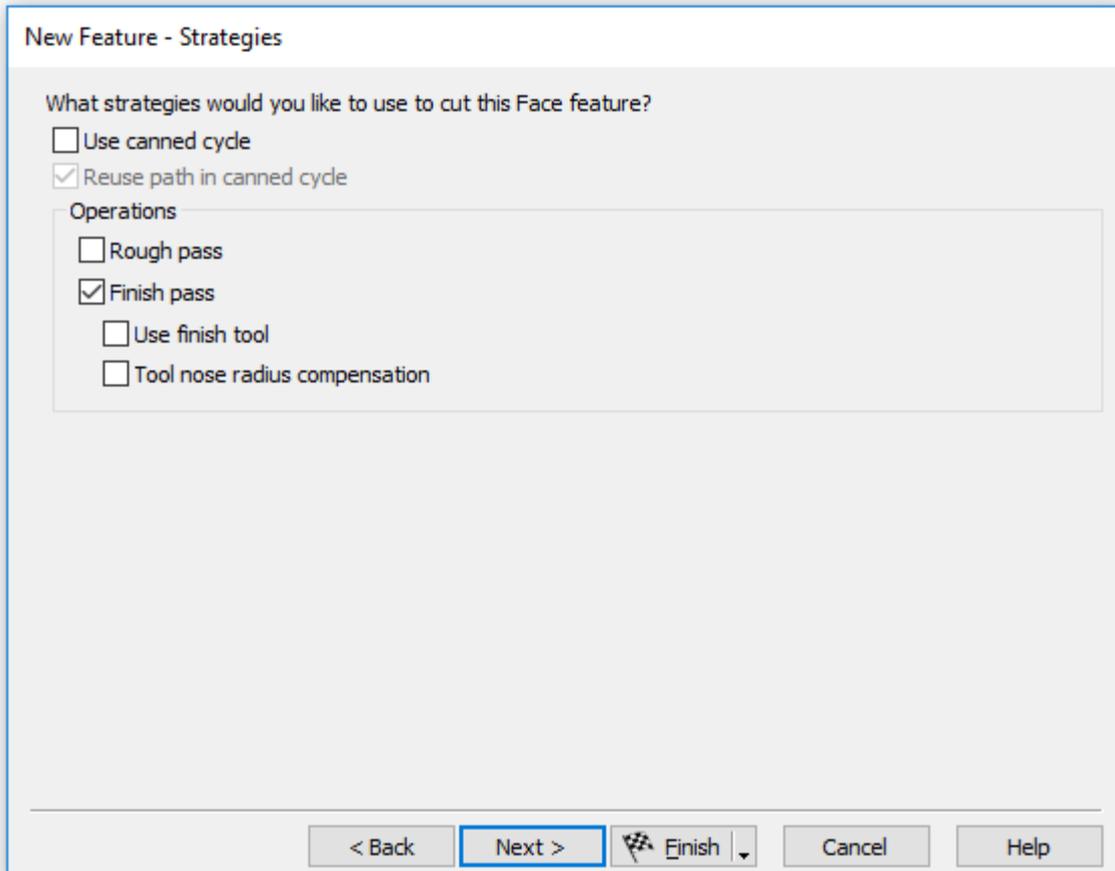
11 Select **Next**. The following menu will appear.



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

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

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



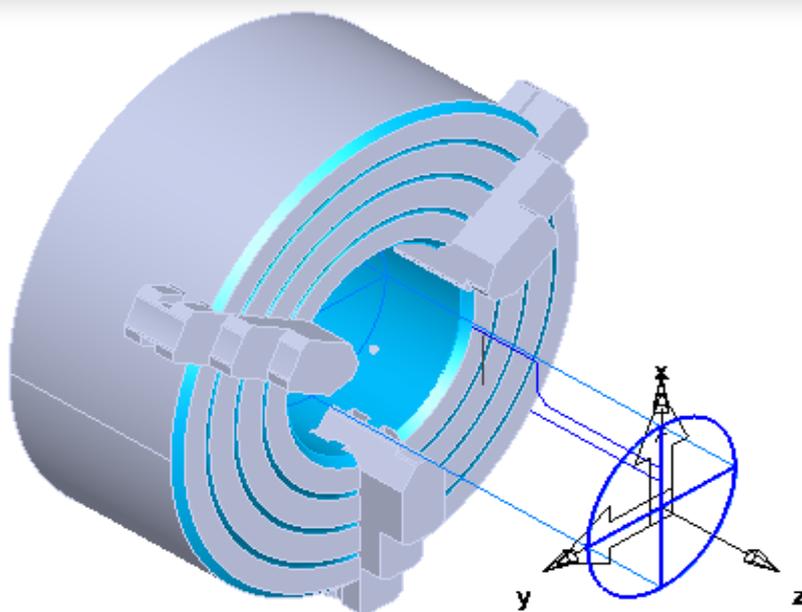
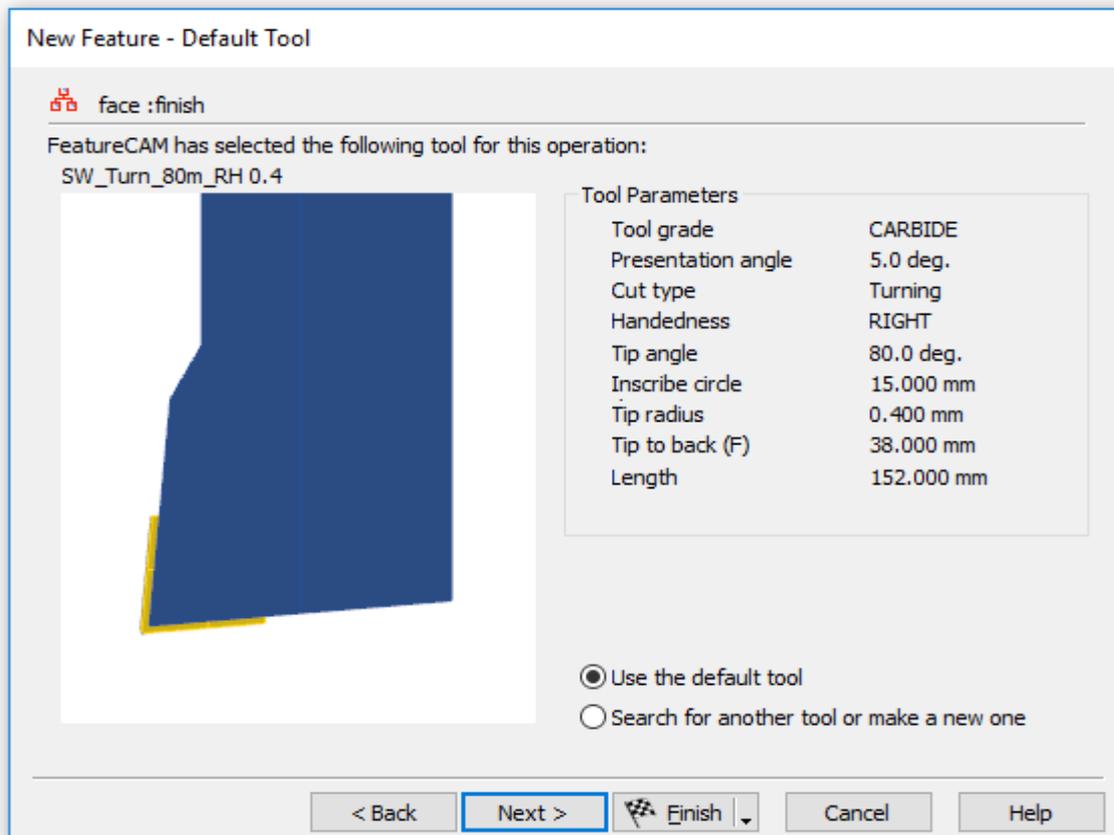
- 15 Notice that we have the ability to use **Canned Cycles**. The canned cycle will be output through the post. (If supported) This gives greater flexibility to the operator. He can then change this on the machine instead of having to go to the **FeatureCAM** programmer (Production Engineer) and re-output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



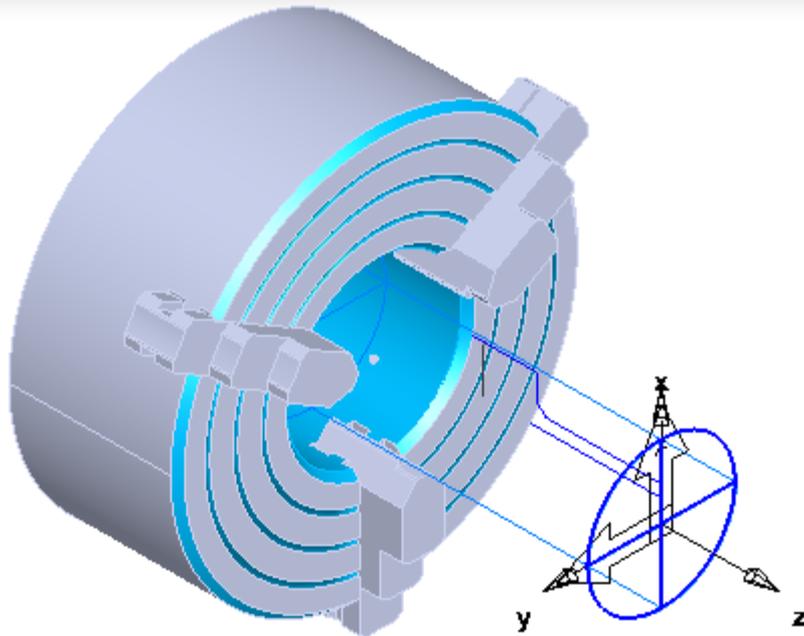
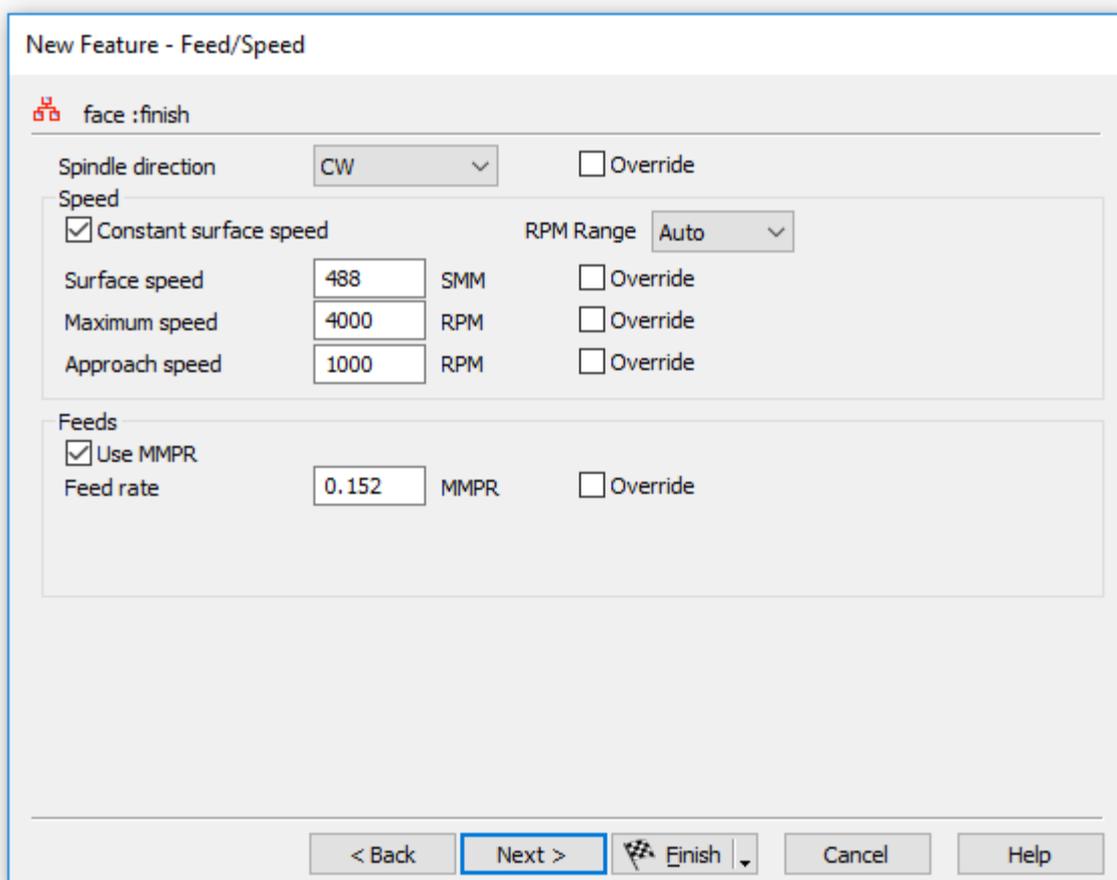
If you have excess stock on the front face, select the **Rough pass** option in the menu. If you do so, you will have the ability to have separate roughing and finishing tools that can be separated in the operations list.

16 Select **Next** twice.

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

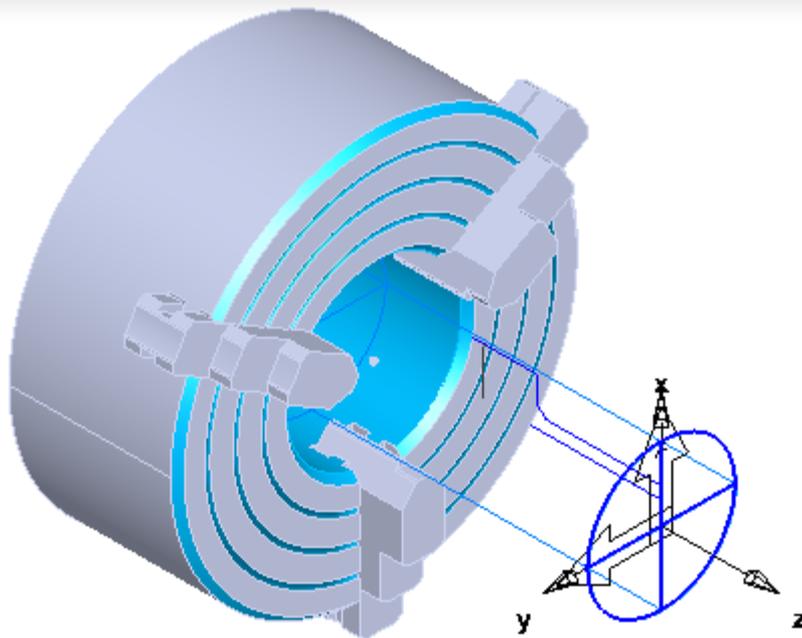
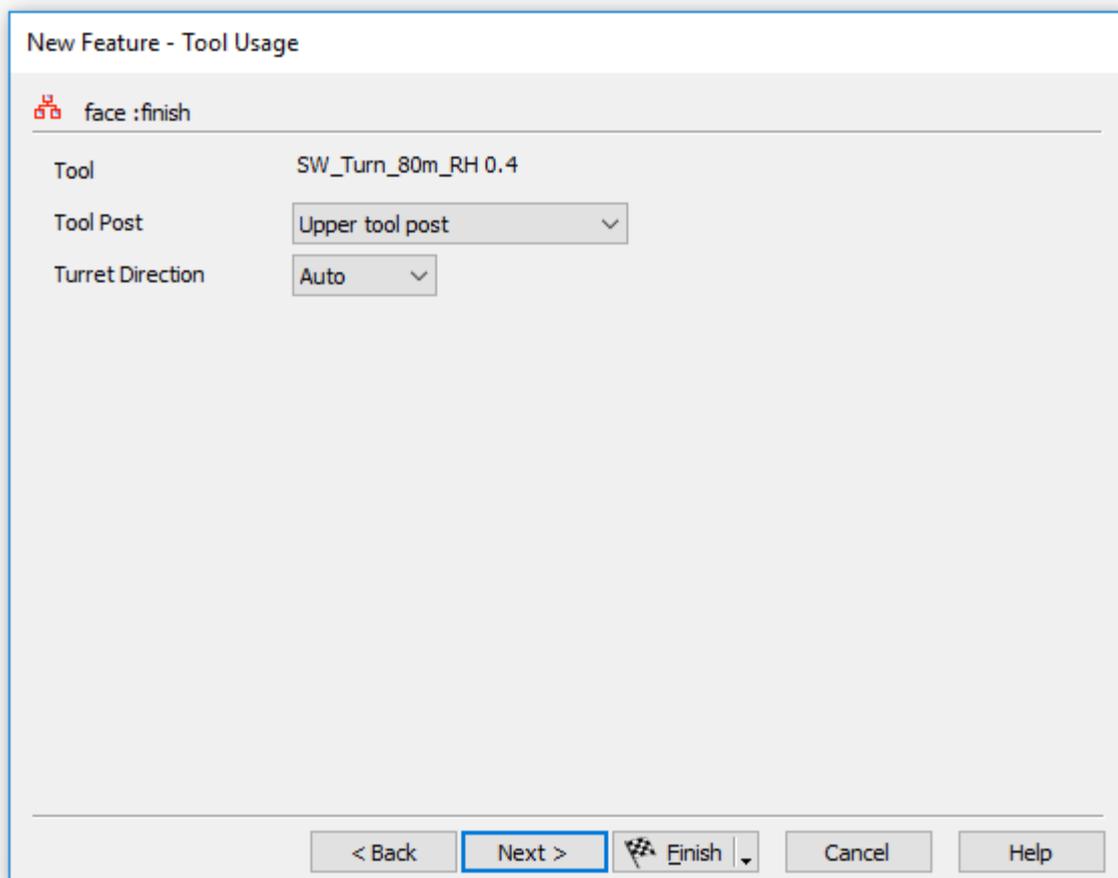


- 18 Select **Next** twice.
- 19 **New Feature – Feeds/Speeds.** This menu gives the ability to change the feeds and speeds etc.

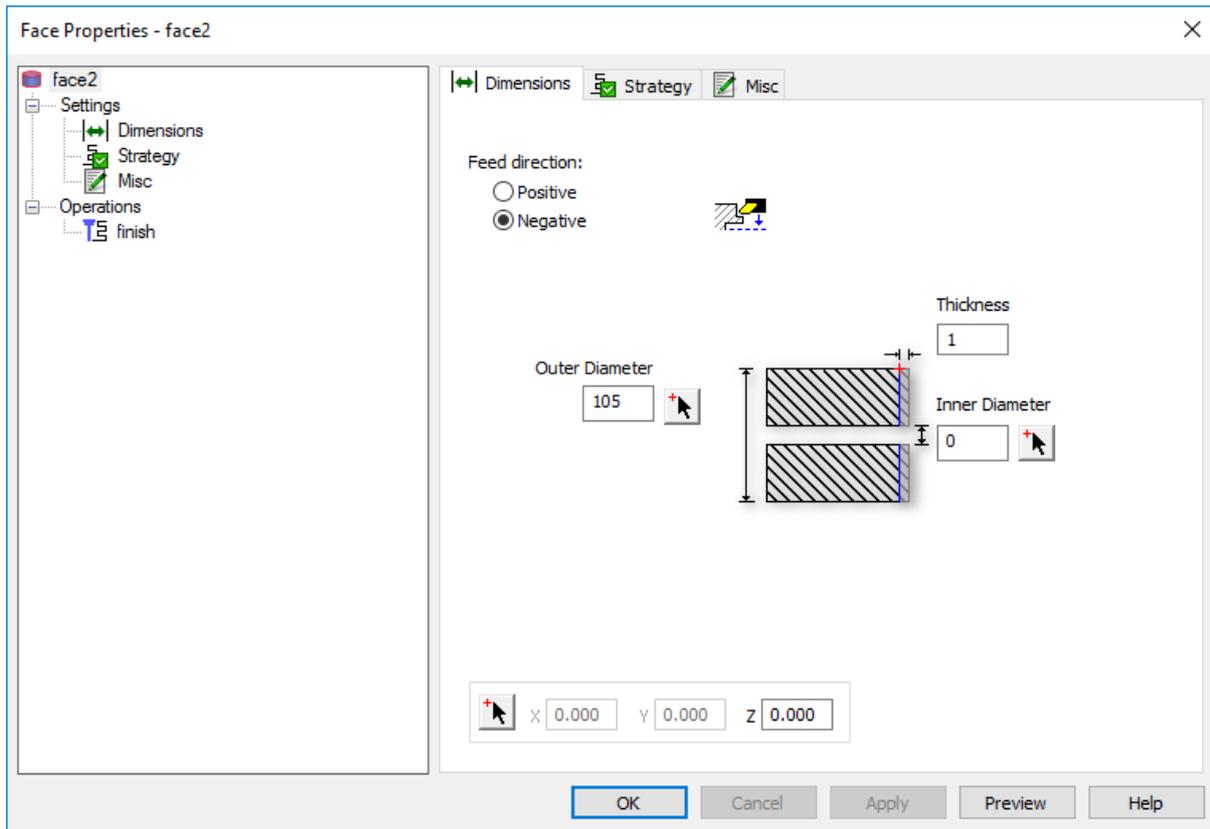


- 20 Select **Next**.

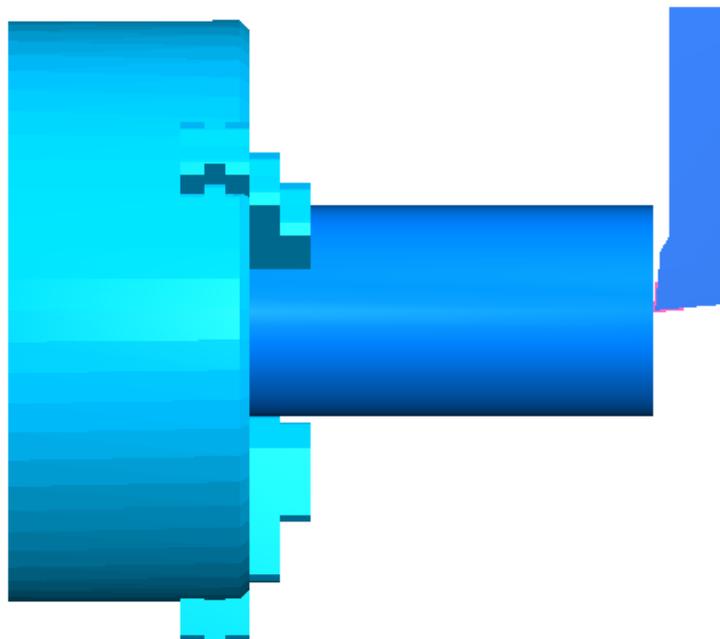
- 21 **New Feature – Tool Usage.** This menu allows you to select what coolant type also Turret type etc. Select **Next** for a summary of all the options.



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



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



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

Creating a Turn Feature

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

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

New Feature

What kind of feature would you like to make?

From Dimensions

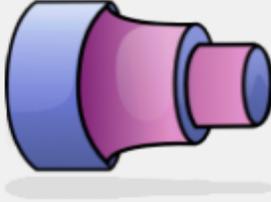
- Hole
- Groove
- Thread
- Face
- Cutoff
- Bar Feed

From Curve

- Turn
- Bore
- Groove
- Thread

Part Handling

- User
- Sub-spindle
- Toolpath
- Misc.



< Back Next > Finish Cancel Help

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

New Feature - Curve

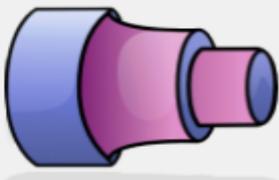
The Turn feature requires a curve to define the boundaries of the feature. If you have already created a curve for your feature, you may select it from the dropdown list.

Curve:

 od curve

If you have not yet created a curve, you need to do so before creating your feature.

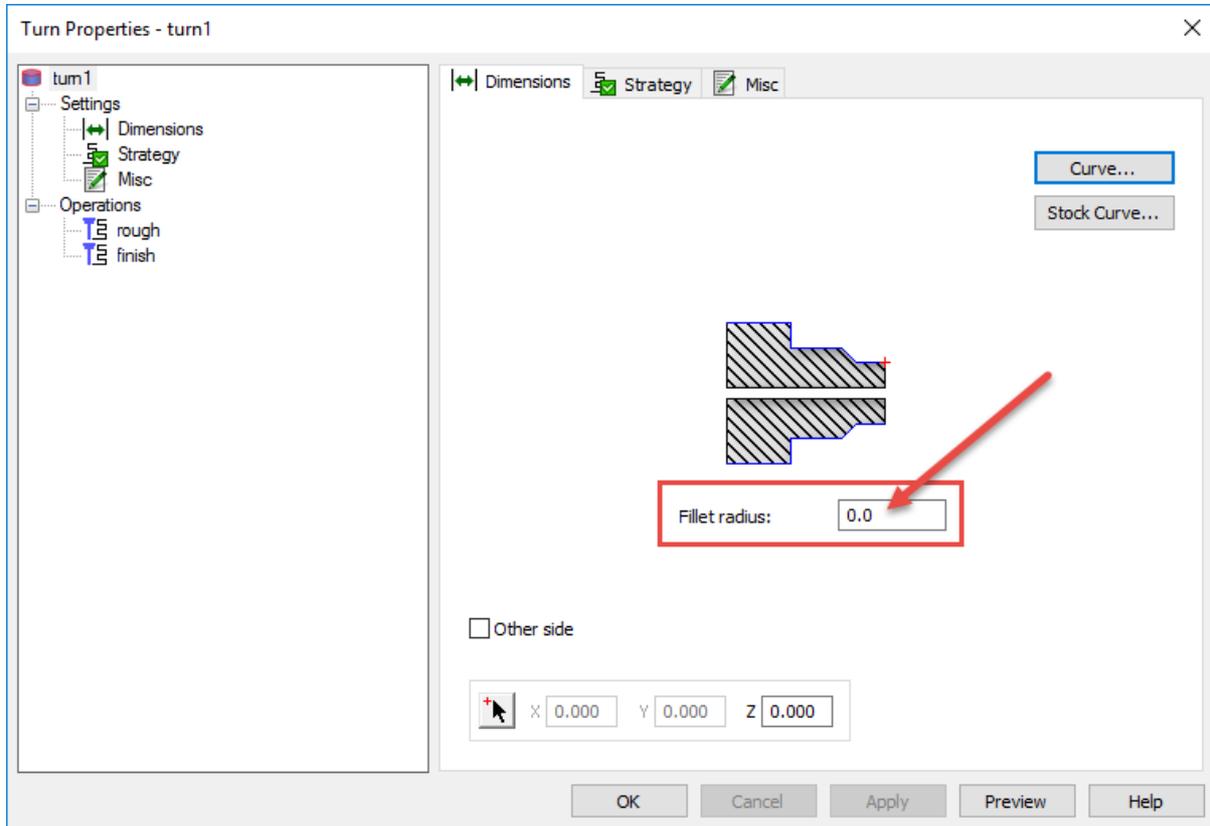
1. Use the geometry constructors to create the outline of the feature.
2. Then use chaining mode to connect the geometry segments together to make a curve.



Curve chaining...

< Back Next > Finish Cancel Help

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

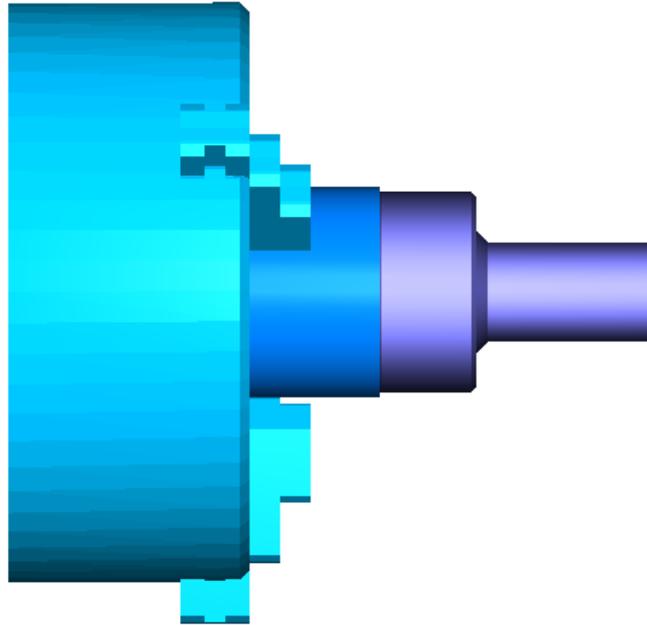


- 29 We can now automatically Fillet external or internal sharp corners. For internal bore fillet corners use **(Other side)**



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

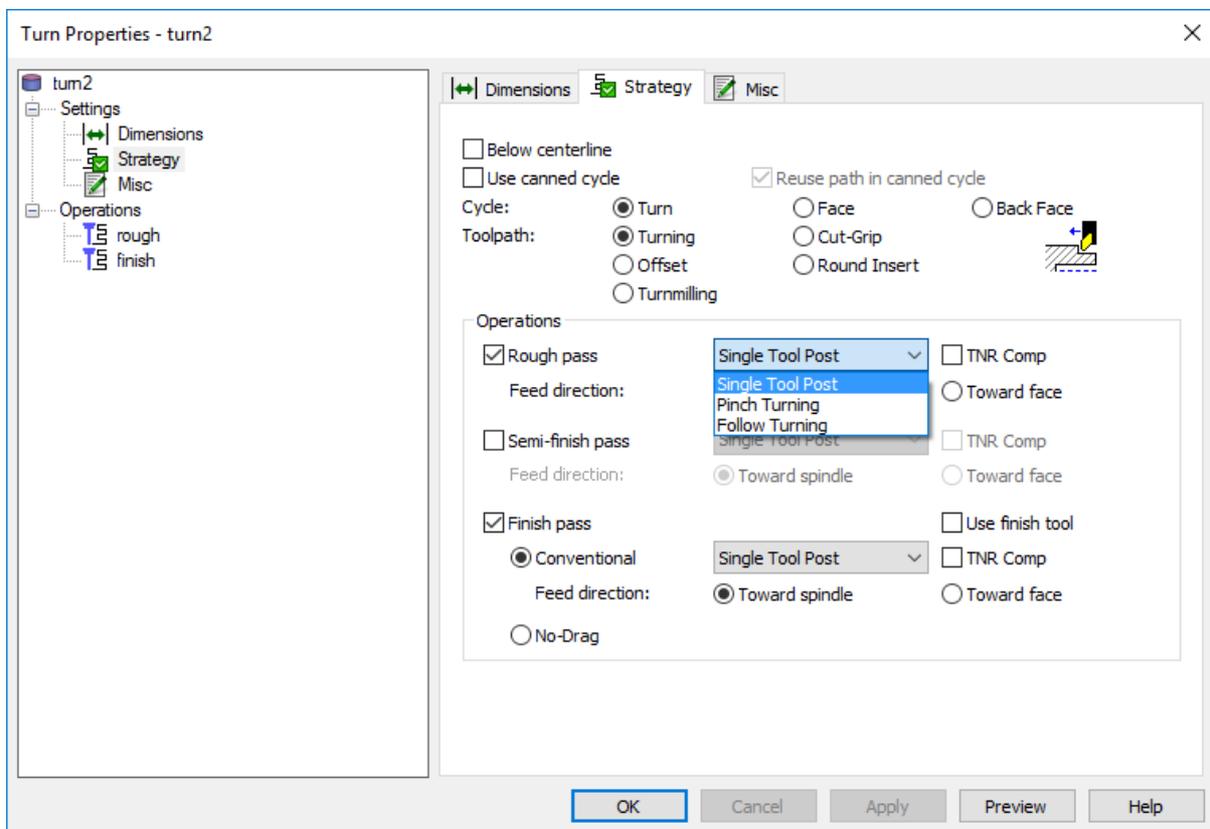




31 Select the **Eject** button to return to the drawing view.

Multi –Turret options.

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



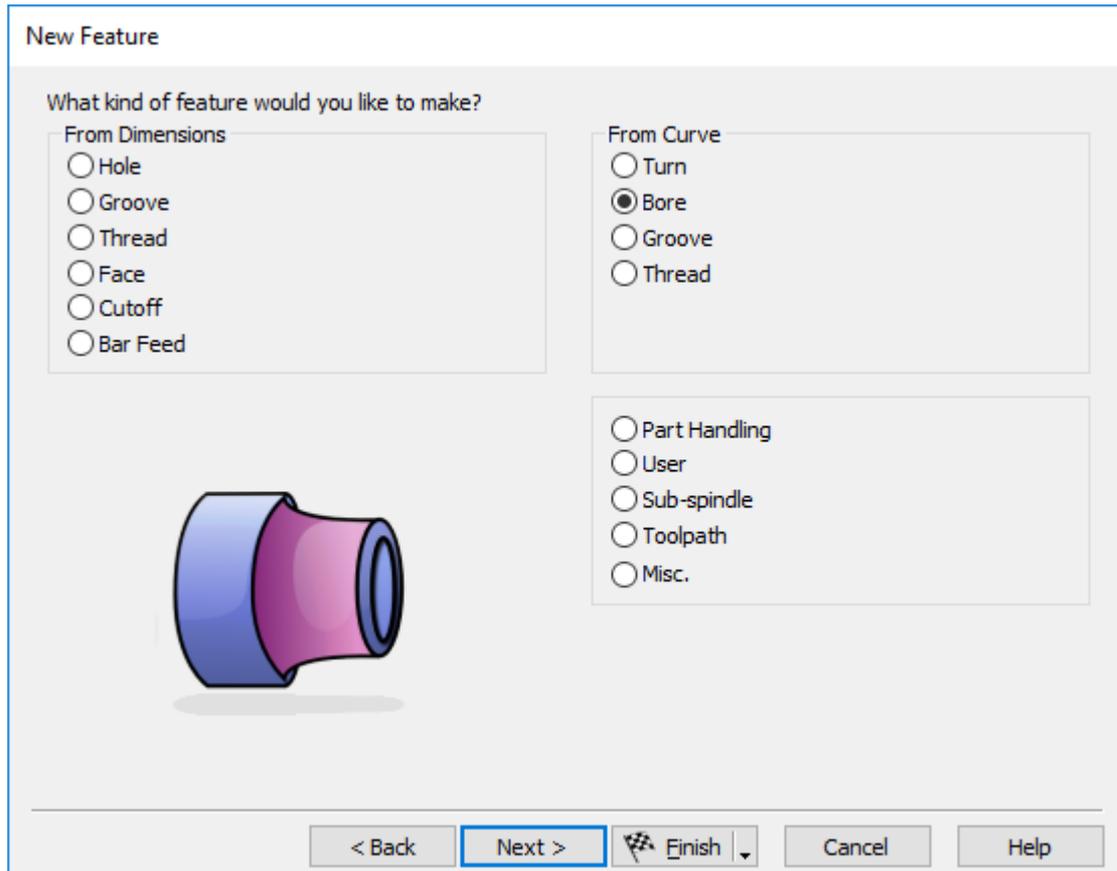
Creating a Bore Feature

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

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



With this option we have the ability to specify a pre-drilled hole. This then created a drill cycle and inserts this before the boring bar.



34 Select **Next**.

New Feature - Curve

The Bore feature requires a curve to define the boundaries of the feature. If you have already created a curve for your feature, you may select it from the dropdown list.

Curve:

 id curve

If you have not yet created a curve, you need to do so before creating your feature.

1. Use the geometry constructors to create the outline of the feature.
2. Then use chaining mode to connect the geometry segments together to make a curve.



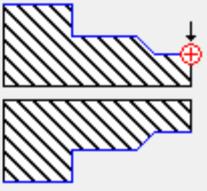
35 Select **Next** twice.

New Feature - Location

The top of the feature will be aligned to the Z location of the curve. Do you want to offset the top of the feature from the curve?

Offset from curve Z location:

 0.0



New Feature - Strategies

What strategies would you like to use to cut this Bore feature?

Use canned cycle Reuse profile in canned cycle

Cycle
 Bore Face Back Face

Toolpath
 Turning Cut-Grip
 Offset Round Insert

Operations

<input checked="" type="checkbox"/> Pre-Drill	Dia	30	Depth	96	Z	0.000
---	-----	----	-------	----	---	-------

Rough pass
 Feed direction: Toward spindle Toward face

Semi-finish pass
 Feed direction: Toward spindle Toward face

Finish pass
 Conventional No-Drag
 Feed direction: Toward spindle Toward face

TNR Comp

TNR Comp

Use finish tool

TNR Comp

< Back **Next >** Finish Cancel Help



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

36 Select Next.

New Feature - Operations

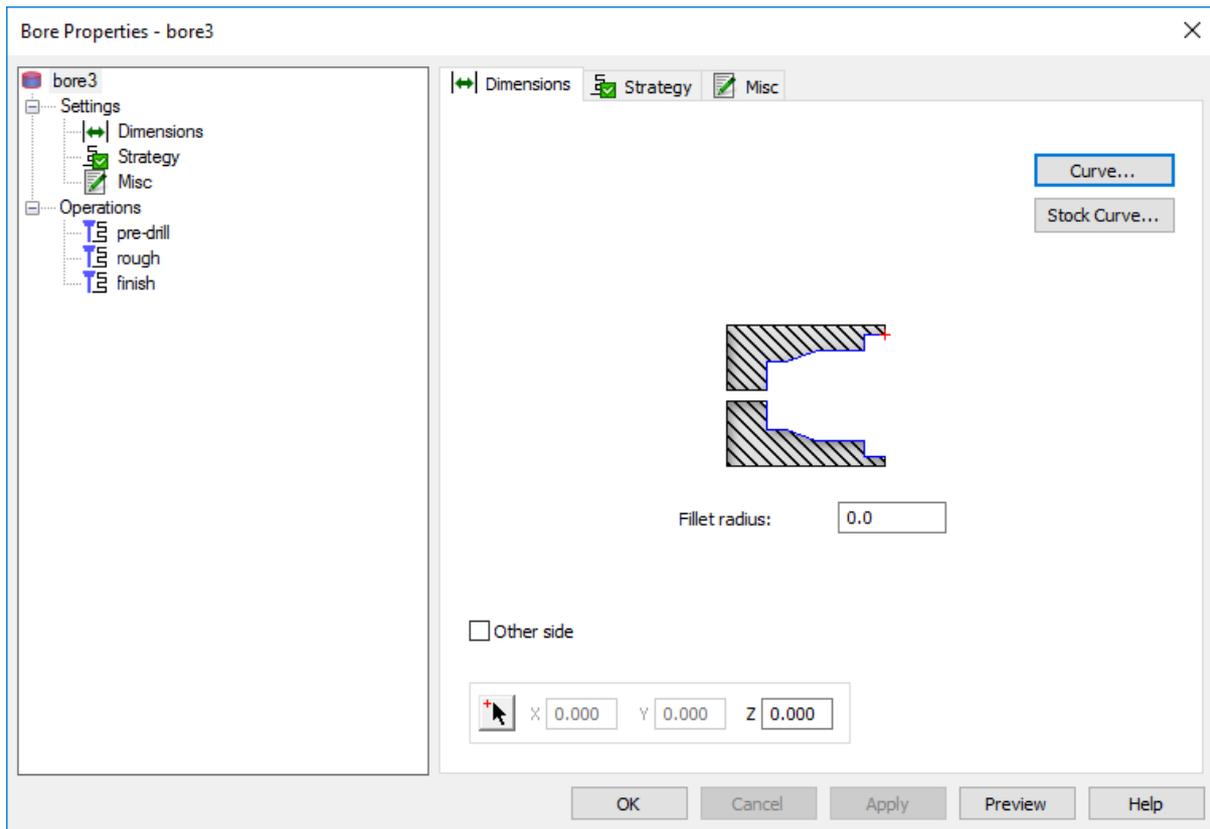
The following operations will be created to machine this Bore feature:

Operation	Tool	Feed	Speed
bore			
Operations			
pre...	Ø30.00mm drill	0.450 M...	808 RPM
rou...	WN_B_Tum_...	0.381 M...	365 SMM
finish	WN_B_Tum_...	0.152 M...	487 SMM

Click 'Finish' to create the feature. Click 'Next' to change tools and/or feeds and speeds for each operation.

< Back **Next >** Finish Cancel Help

37 Select **Finish**.



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

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



-  Play selected.
-  Fast forward to End Selected.
-  Single Step selected.
-  Play to Next operation selected.

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

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

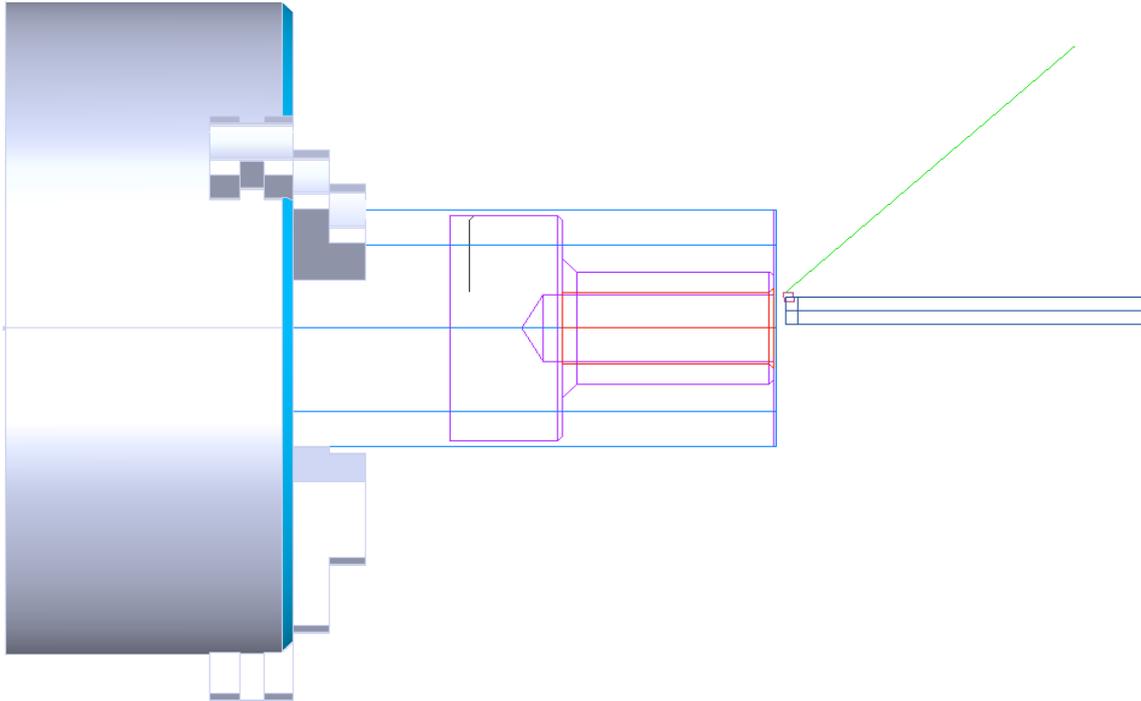
42 Double click on **Bore1** in **PartView**.

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

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

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

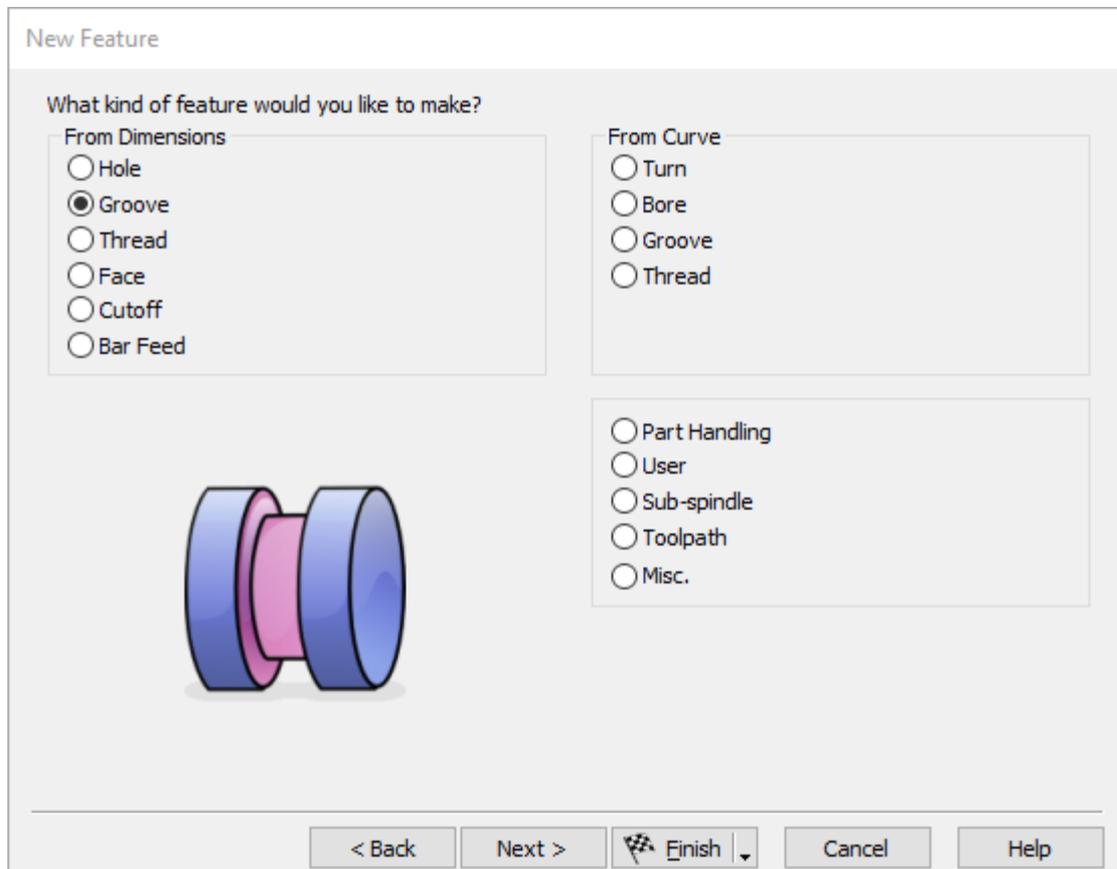
46 Select **Esc** to close the **Preview** Simulation.



Creating a Groove Feature

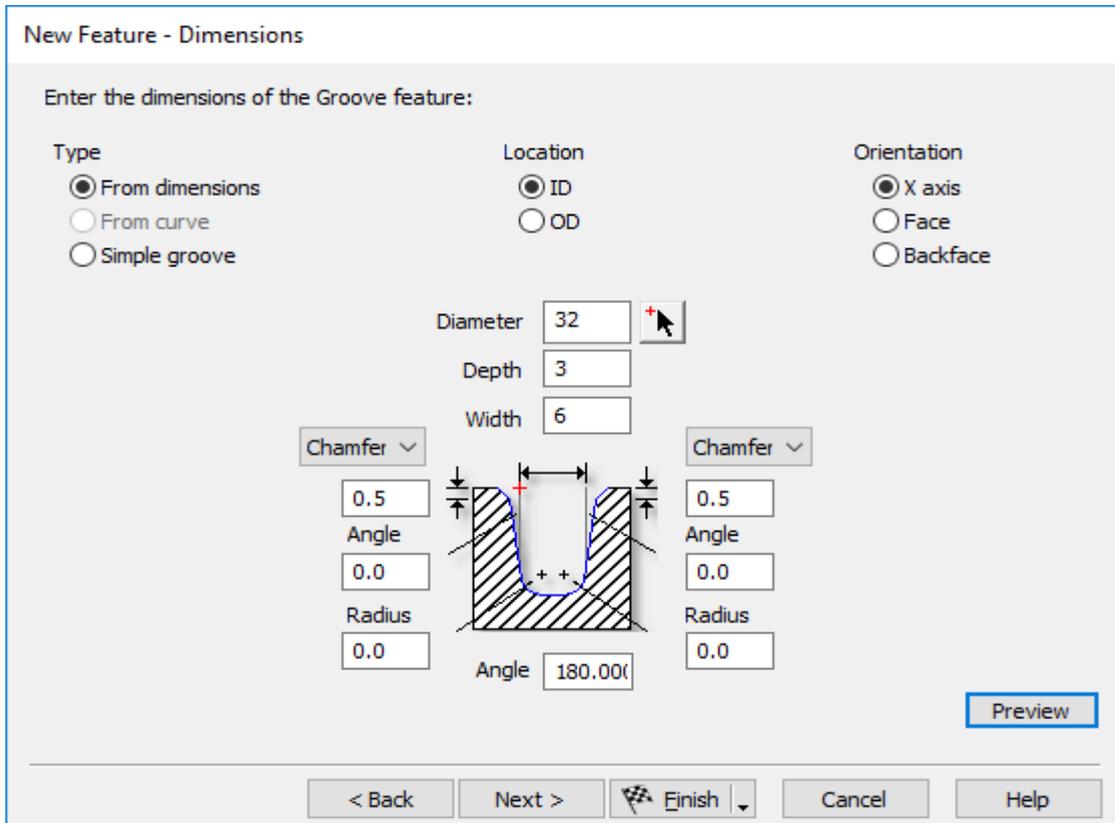
47 Select New Feature  or **Ctrl+R**.

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

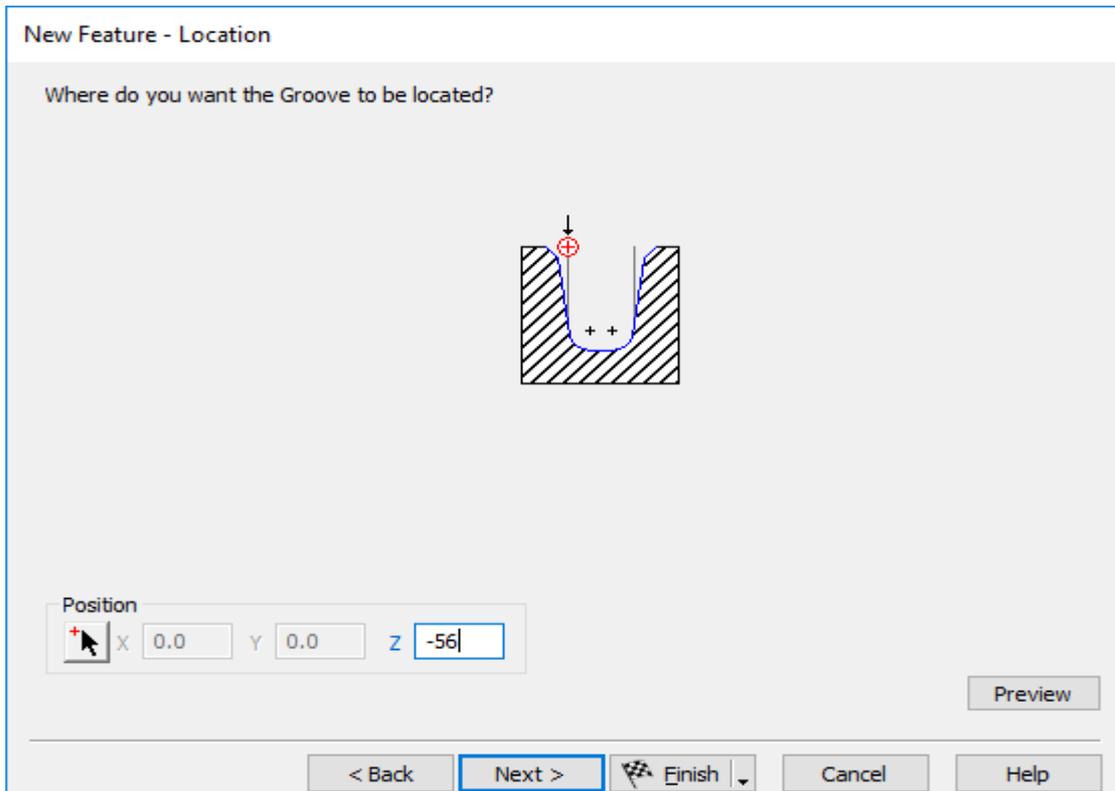


49 Select **Next**.

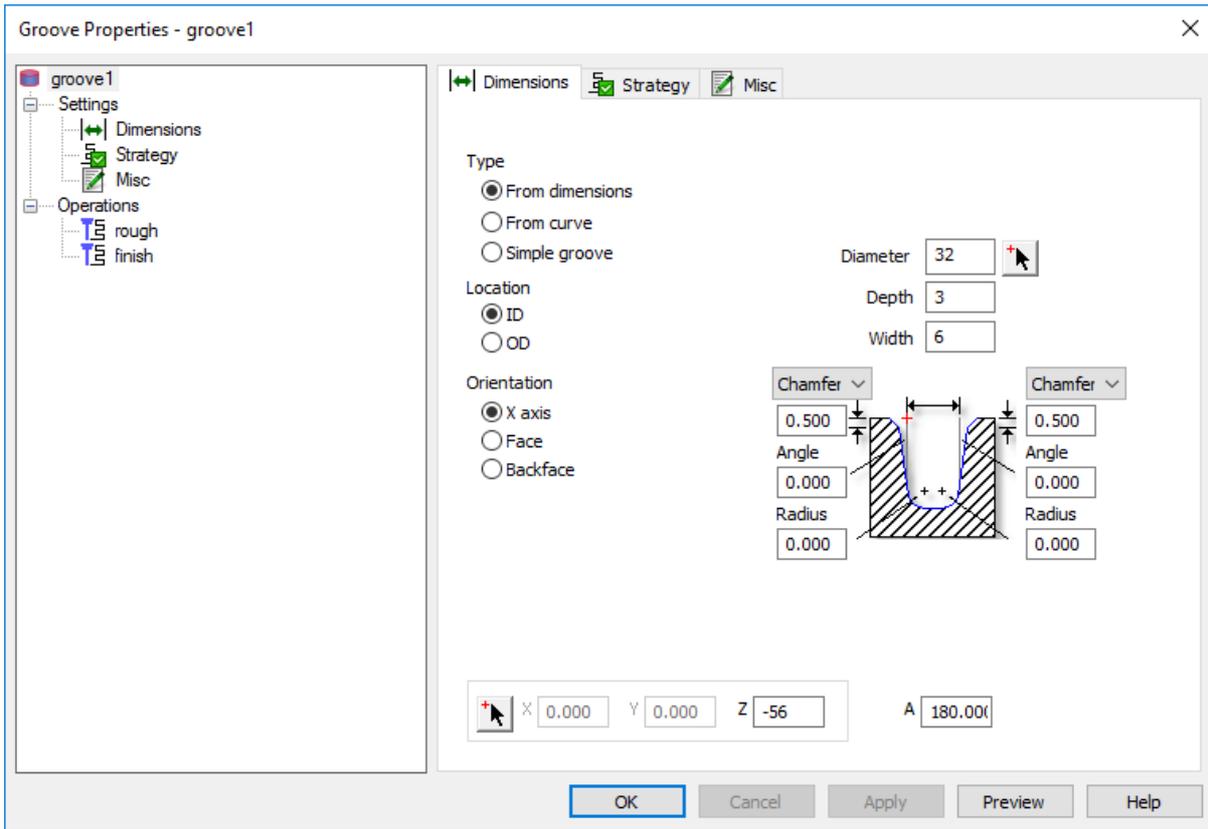
50 On the Dimensions page set Location to **ID**, Orientation to **X-axis**, and Diameter to **32mm**, and Depth to **3mm**, and Width to **6mm**. Chamfer settings at **0.5mm**.



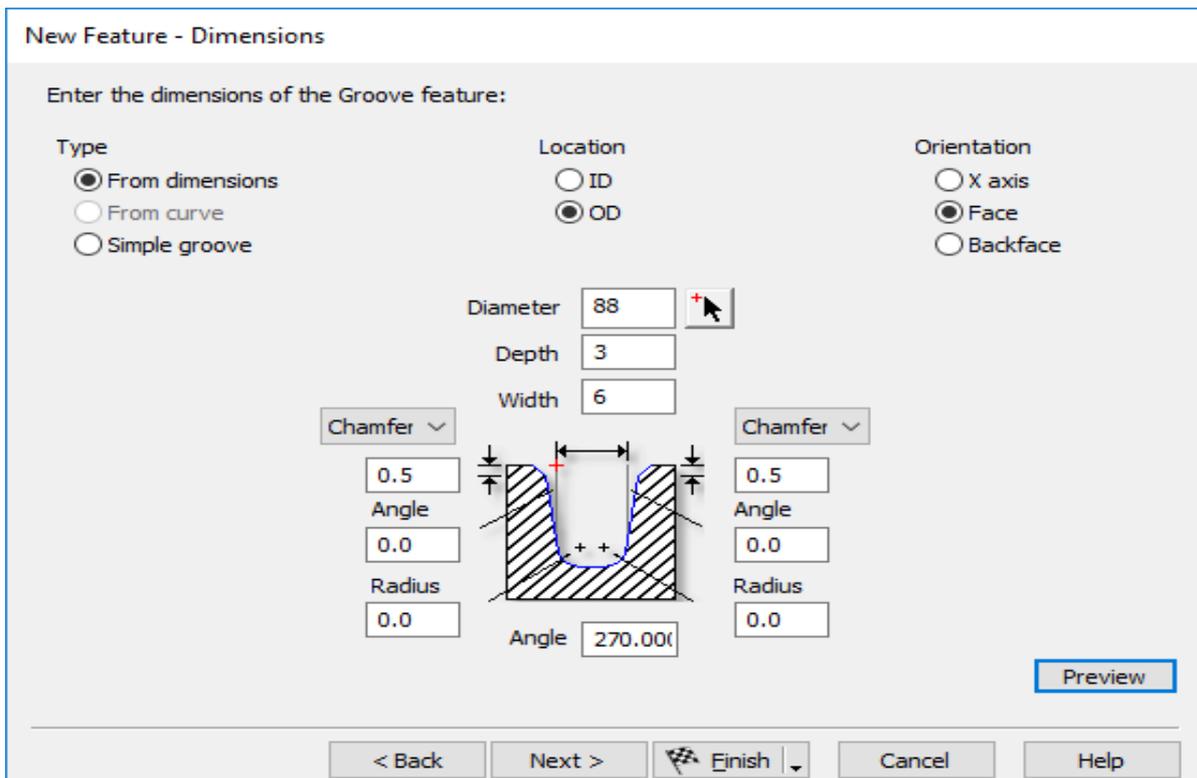
51 Click **Next** and set **Z-56mm**. Click **Finish**.



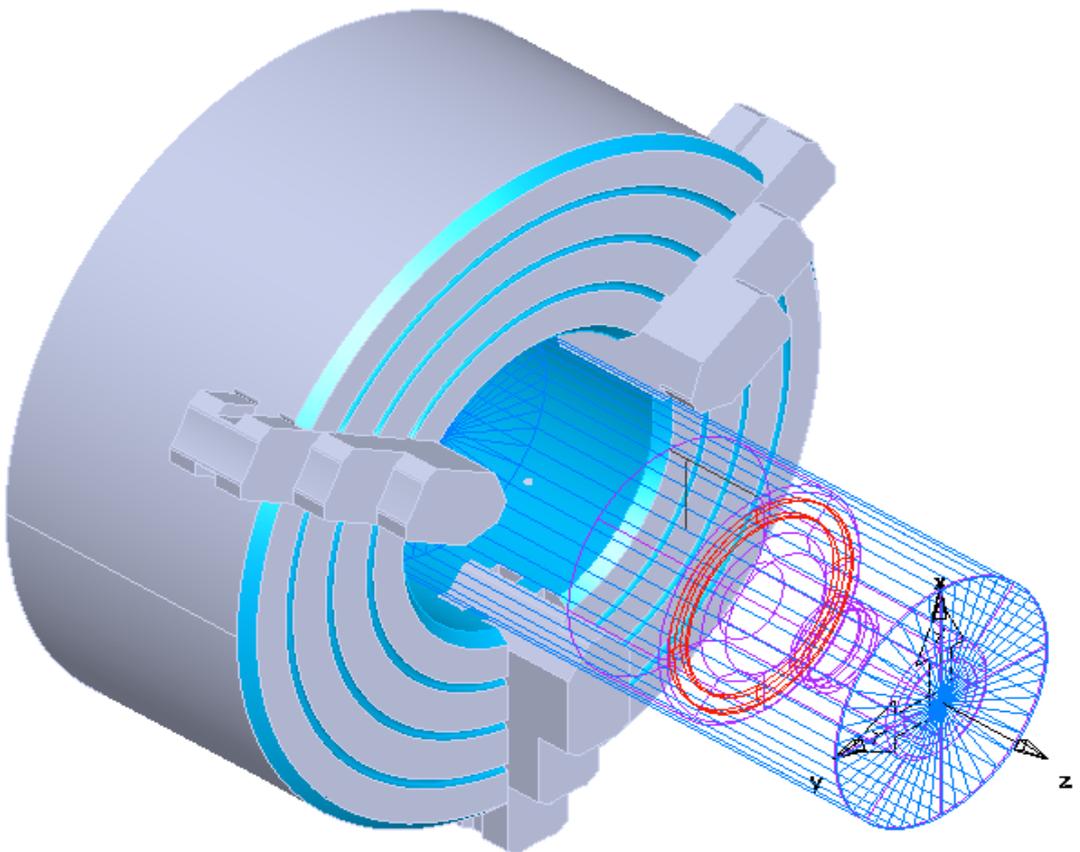
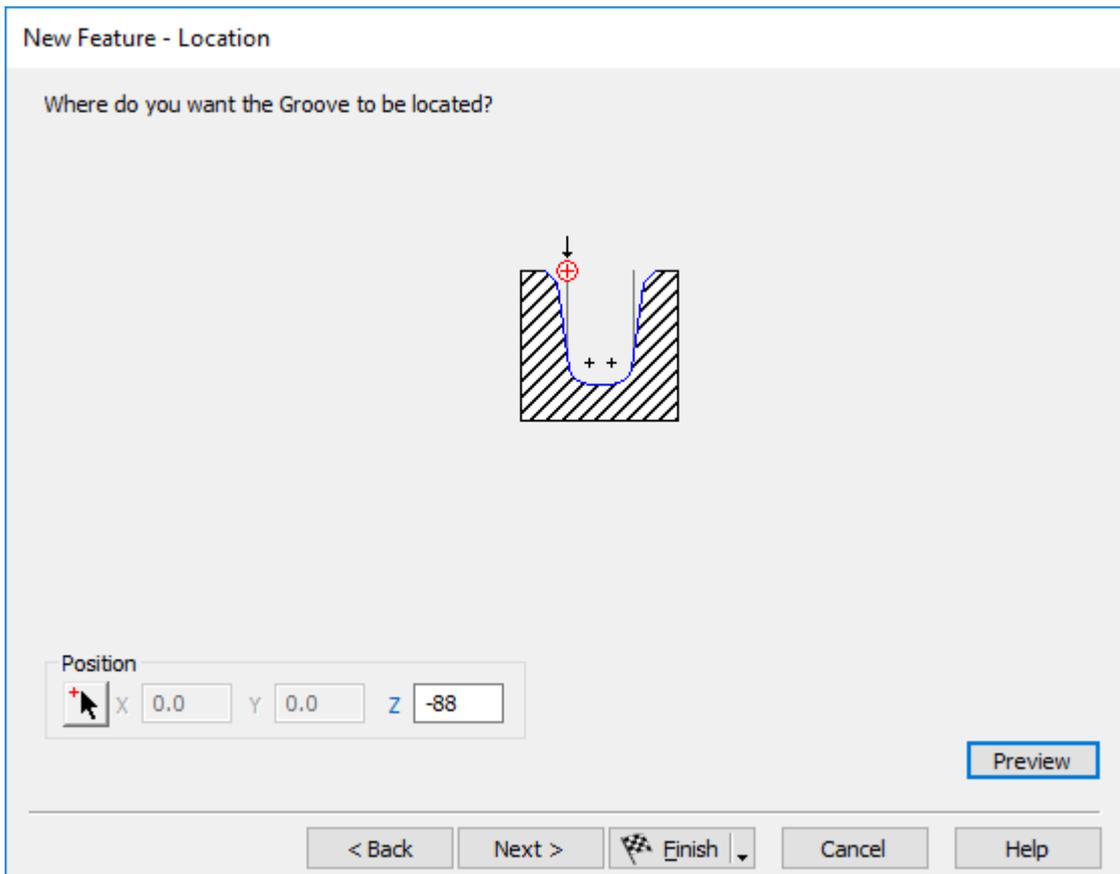
52 Select **Finish**. We will now create a **Face Groove**.



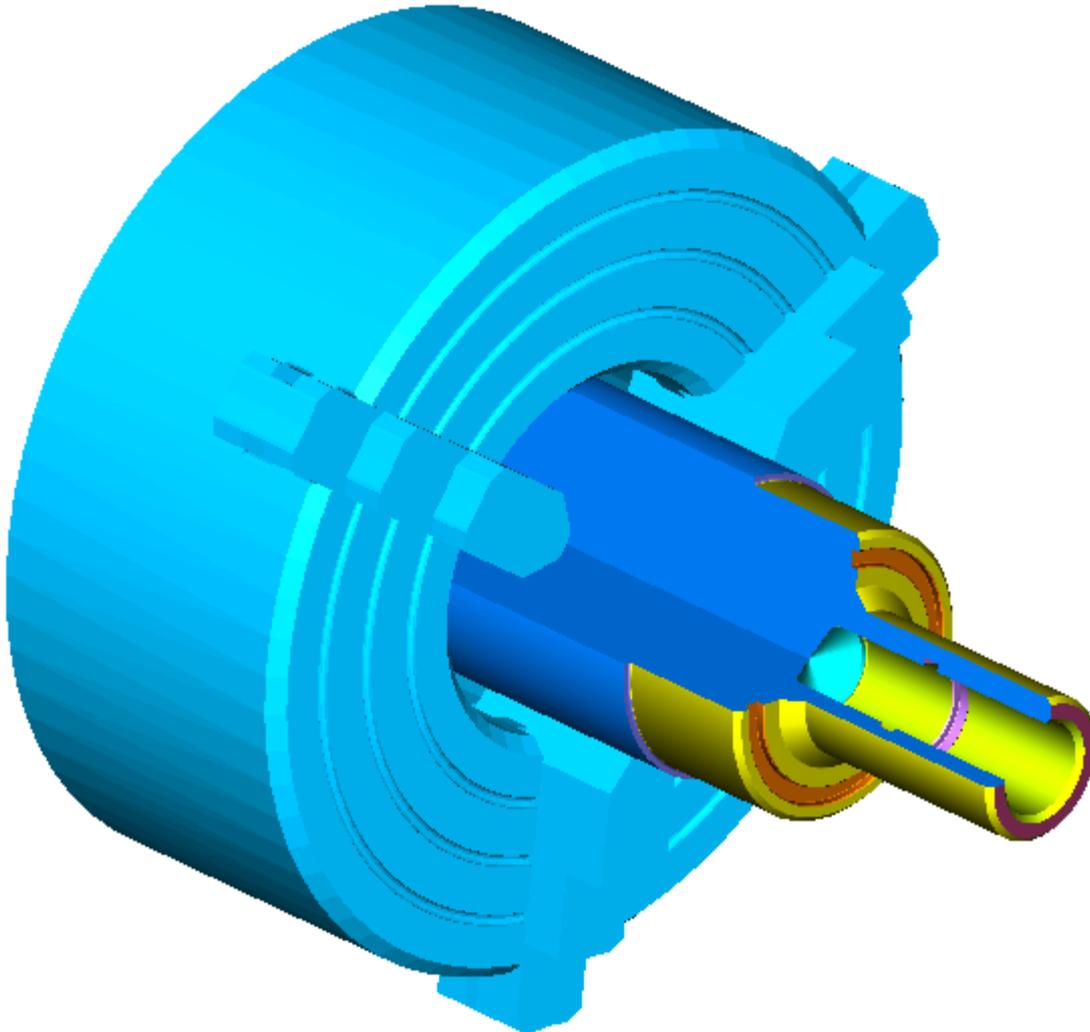
53 Select **Ctrl+R** New Feature, then select **Turning** and then **Next**. We need to select the **Groove** radio button in the **From Dimensions** group. On the Dimensions page set Location to **OD**, Orientation to **Face**, Diameter to **88mm**, Depth to **3mm**, and Width to **6 mm**. Chamfer settings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



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



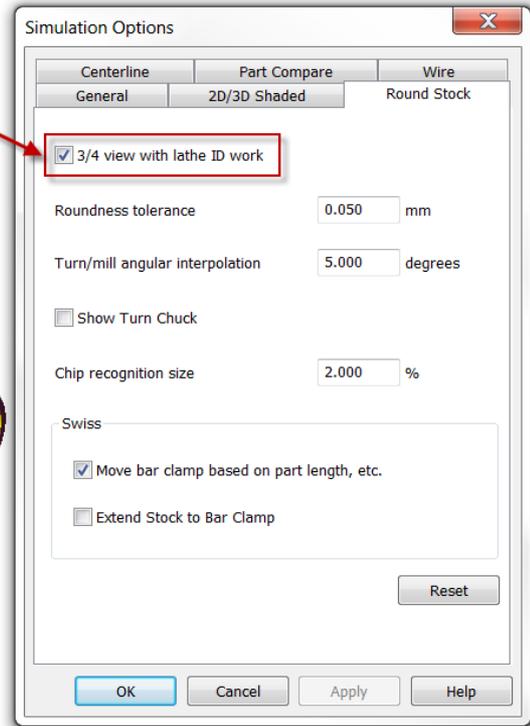
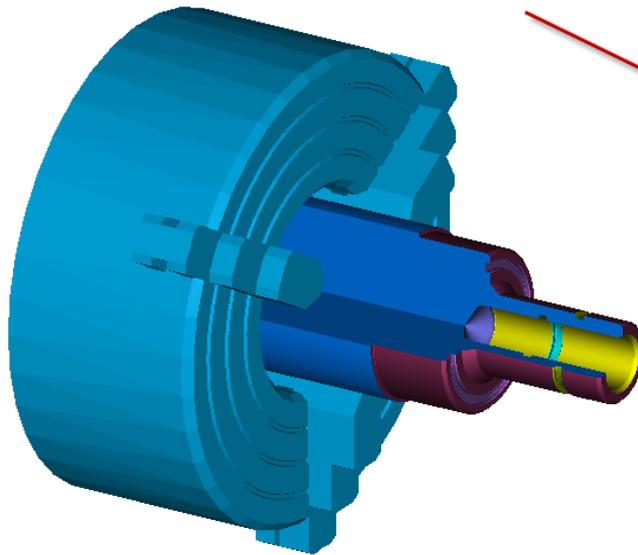
Run a **3D Simulation**  to see the results below.



To View the simulation in a $\frac{3}{4}$ **view** select **Options>Simulation>Round Stock – $\frac{3}{4}$ View with lathe ID work.**

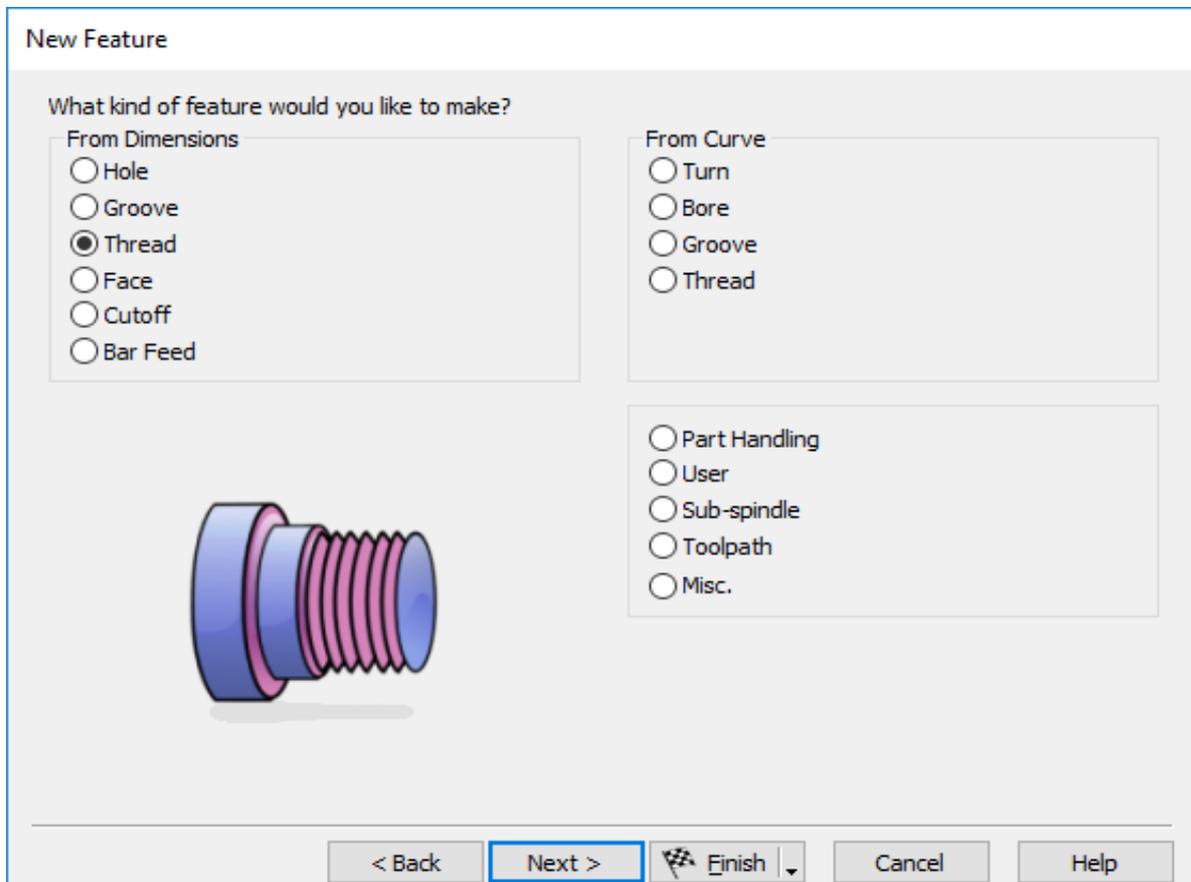


See image on next page.



Creating a Thread Feature

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



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

New Feature - Dimensions

Enter the dimensions of the Thread feature:

Type ID OD Custom Standard Thread
M50 x 1.5

Thread Length: 25.0 Pitch: 1.500
Thread Height: 1.137 Major Diameter: 49.850
 Tapered Angle: 0.000
Thread: Left hand Right hand

Preview

< Back Next > Finish Cancel Help

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

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

New Feature - Strategies

What strategies would you like to use to cut this Thread feature?

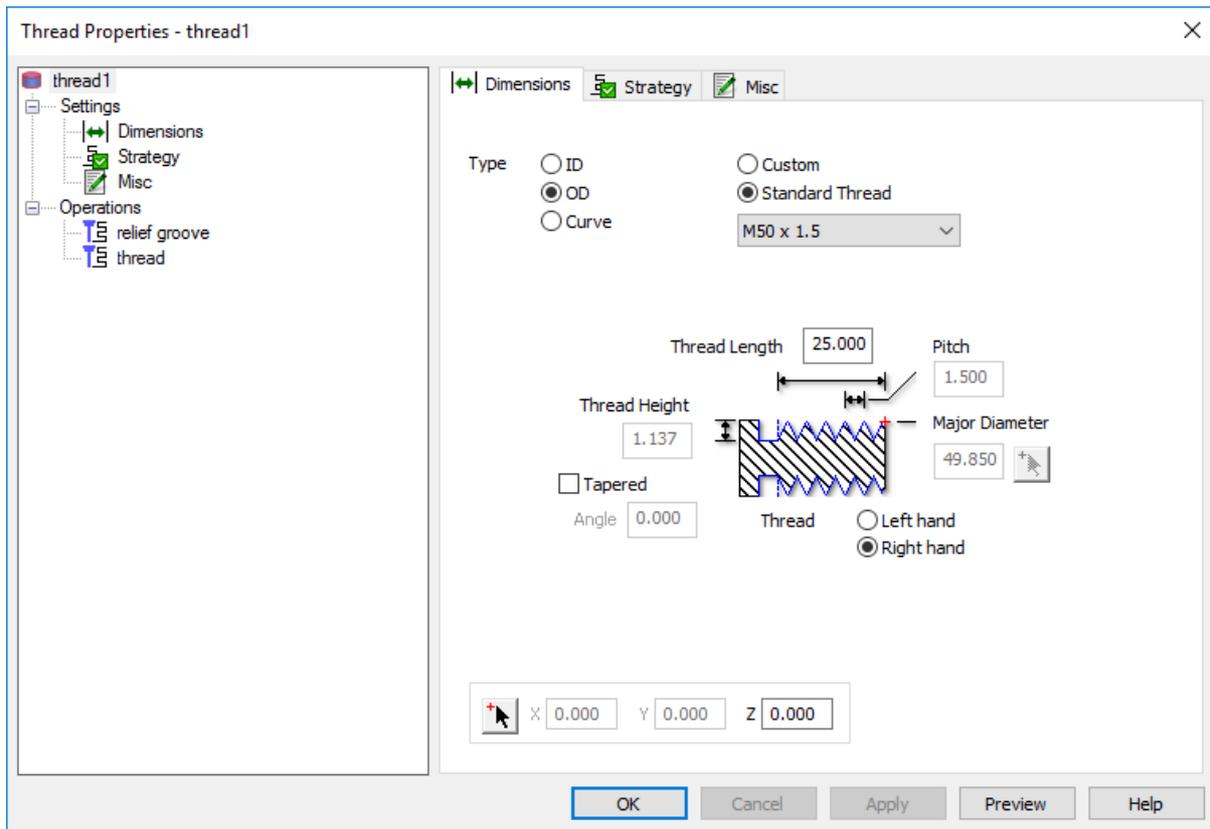
Operations

Turn to diameter: Rough Finish

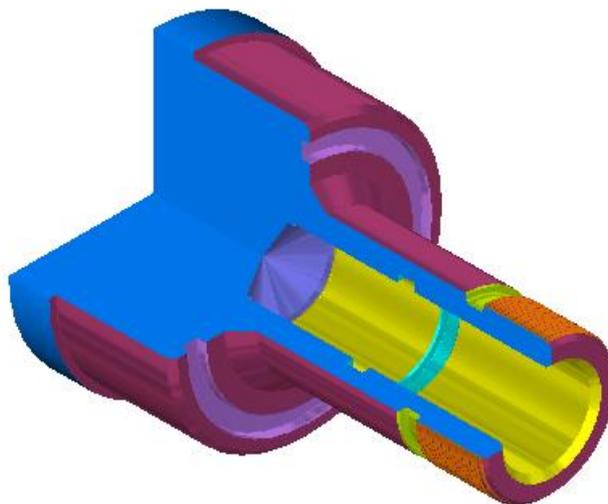
Chamfer Depth: 1.237 Angle: 30.000
 Relief groove Width: 4.500 Depth: 3
 Side wall angle: 30.000
 Thread
Feed direction: Toward spindle Toward face
Passes: Fixed Count: 5
 Calculated
Step1: 0.114 Step2: 0.114 Min infeed: 0.114
Spring passes: 1

< Back Next > Finish Cancel Help

59 Select **Finish**.



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

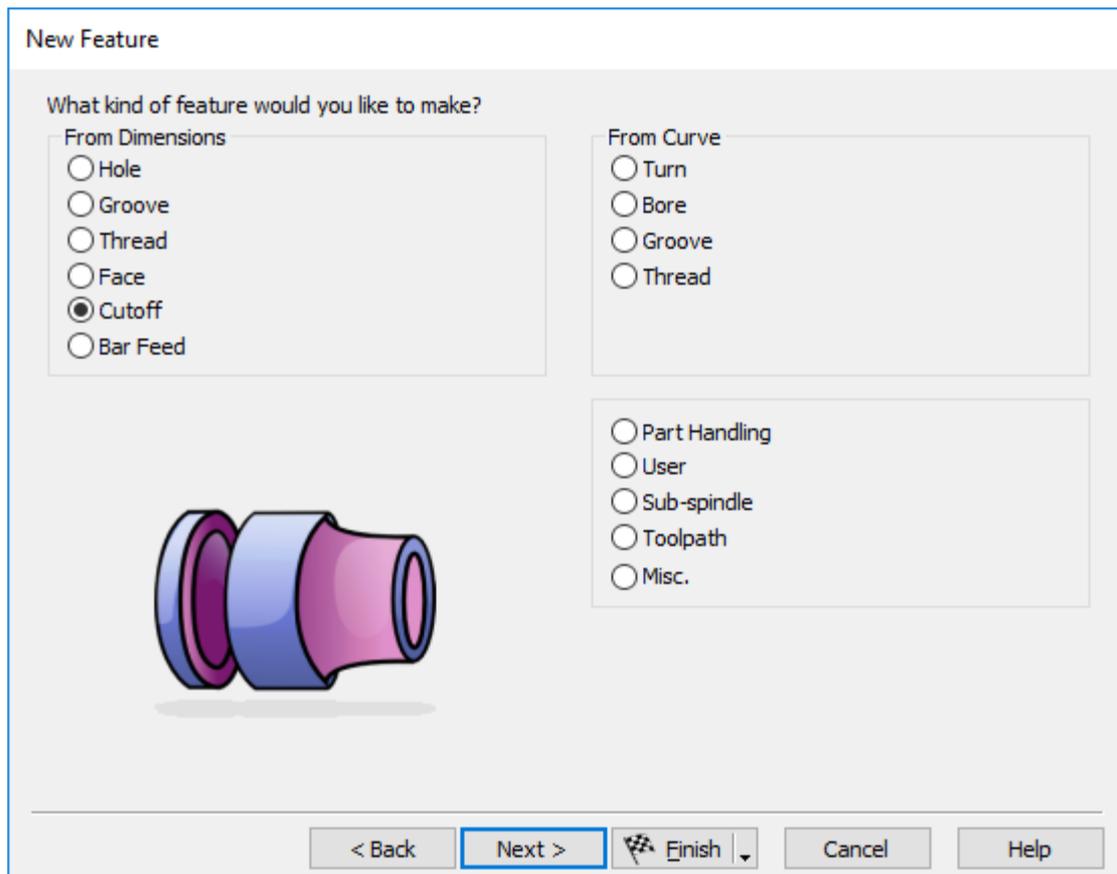


61 Remember to select the **Eject** button to clear the screen.

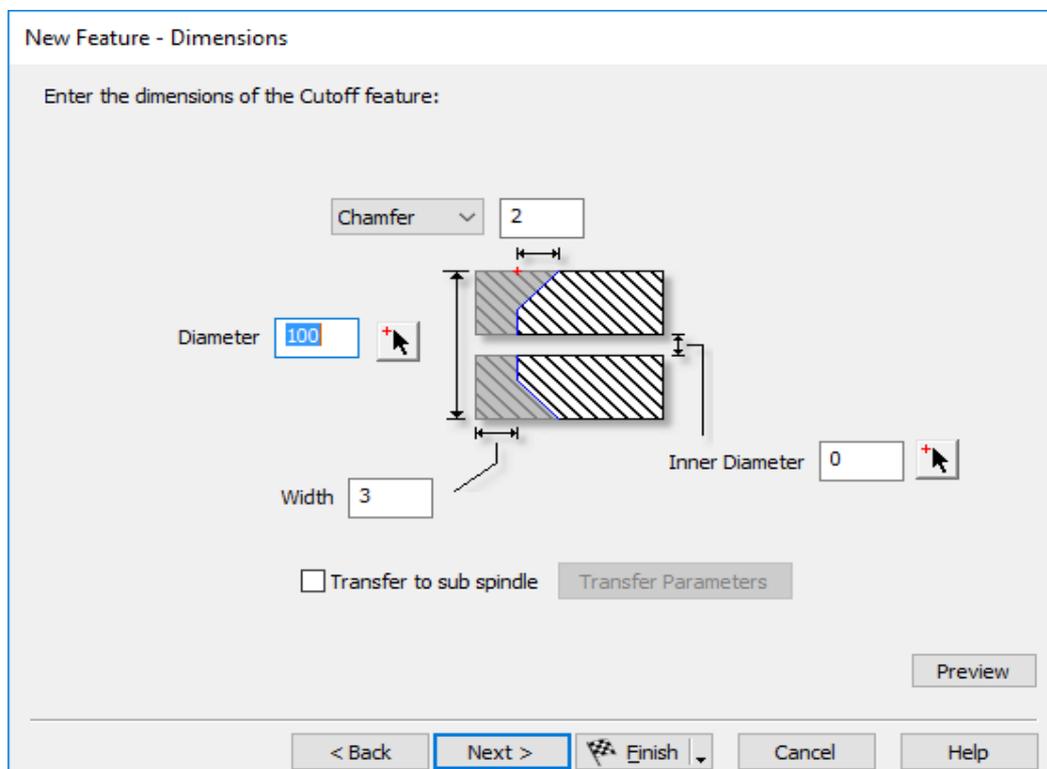


62 Select **Ctrl+R** New Feature and select **Turning**. Select **Cutoff**.

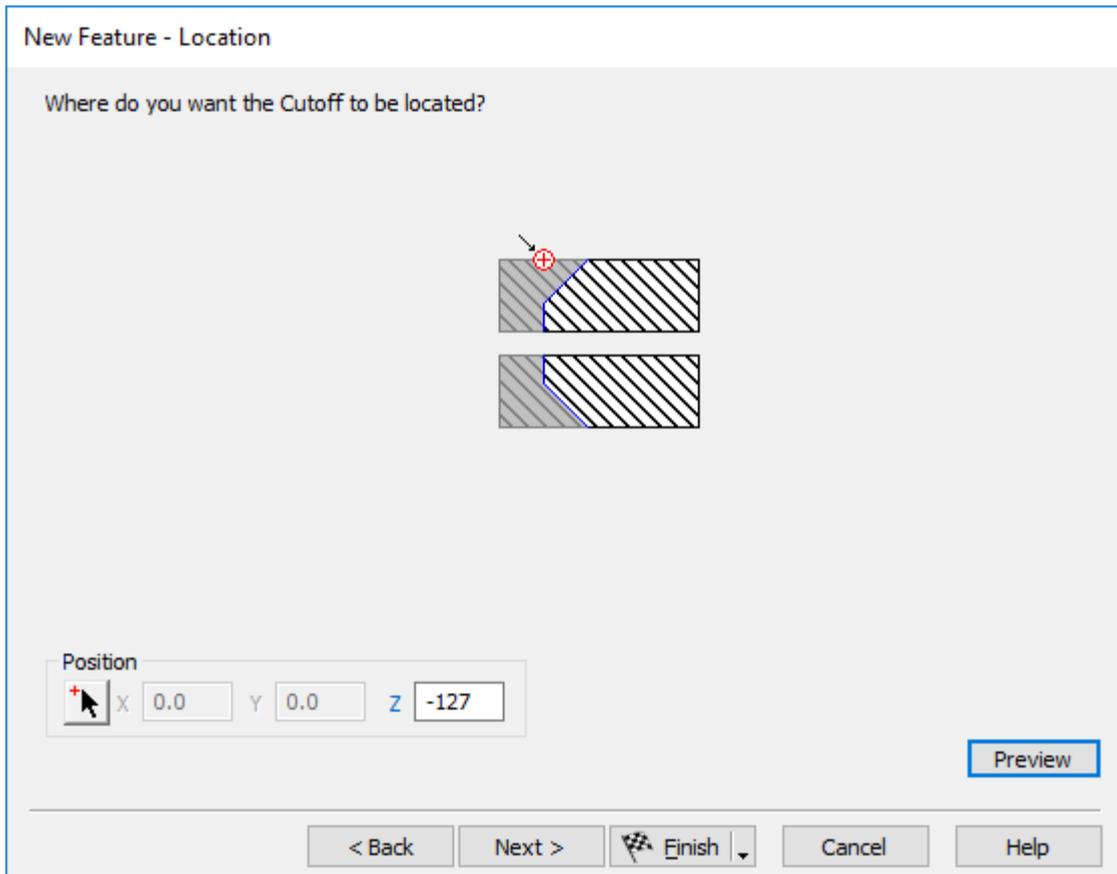




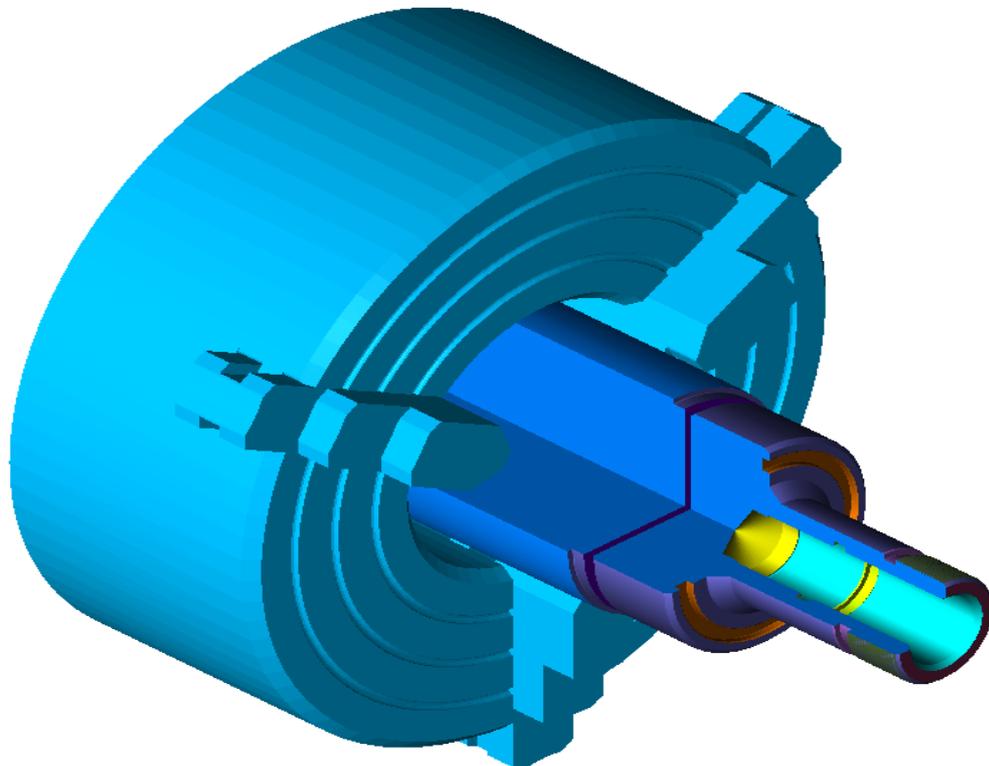
63 On the Dimensions page set Chamfer to **2** then set **Diameter** to **100mm** and **Inner Diameter** to **0** and **Width** to **3.0**



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



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



Results

Operation List							
<input checked="" type="radio"/> Automatic Ordering <input type="radio"/> Manual Ordering							
F	Operation	Feature	Tool	Feed	Speed		Depth
→	finish	face1	CNMG-0.4	0.152 MMPR	488 SMM CW		52.500 mm
	rough pass 1	turn1	CNMG-0.4	0.381 MMPR	366 SMM CW		27.000 mm
	finish	turn1	CNMG-0.4	0.152 MMPR	488 SMM CW		27.000 mm
	drill	hole1	30mm drill	0.450 MMPR	809 RPM CW		105.013 mm
	rough pass 1	bore1	WN_B_Sma...	0.381 MMPR	366 SMM CW		2.000 mm
	finish	bore1	WN_B_Sma...	0.152 MMPR	488 SMM CW		2.000 mm
	rough pass 1	groove1	WN_B_Sma...	0.381 MMPR	366 SMM CW		2.900 mm
	finish	groove1	WN_B_Sma...	0.152 MMPR	488 SMM CW		3.000 mm
	rough pass 1	groove2	SW_Grv_3...	0.381 MMPR	366 SMM CW		2.900 mm
	finish	groove2	SW_Grv_3...	0.152 MMPR	488 SMM CW		3.000 mm
	rough pass 1	groove3	SW_FaceGr...	0.381 MMPR	366 SMM CW		2.900 mm
	finish	groove3	SW_FaceGr...	0.152 MMPR	488 SMM CW		3.000 mm
	thread	thread1	OD_Metric...	1.500 MMPR	2330 RPM CW		1.137 mm
	rough pass 1	cutoff1	SW_Cut_3m...	0.381 MMPR	366 SMM CW		2.000 mm
	cutoff	cutoff1	SW_Cut_3m...	0.152 MMPR	488 SMM CW		50.000 mm
	Results						

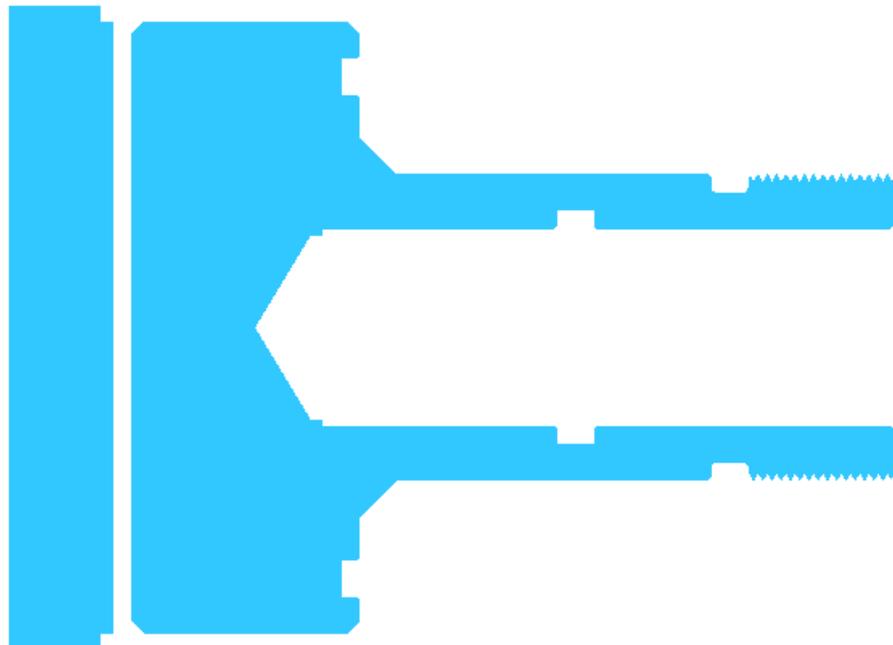


NC Code can only be output after a complete simulation has been performed. If any errors are found, the Simulation will not continue.

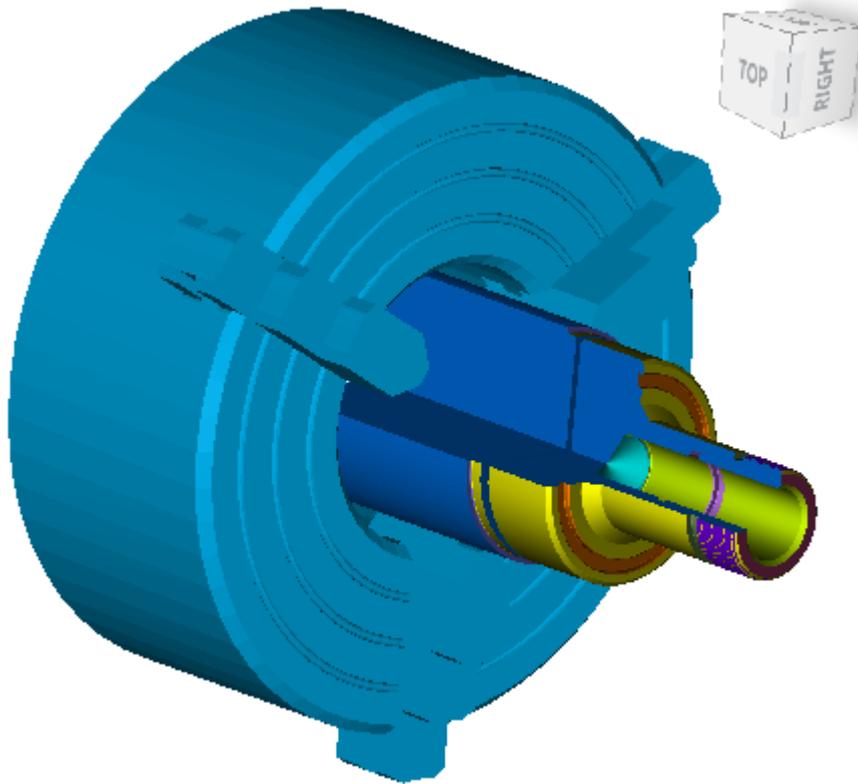


The sample part is complete. View the **2D Centreline Simulation** followed by a **3D simulation** to view the finished part.

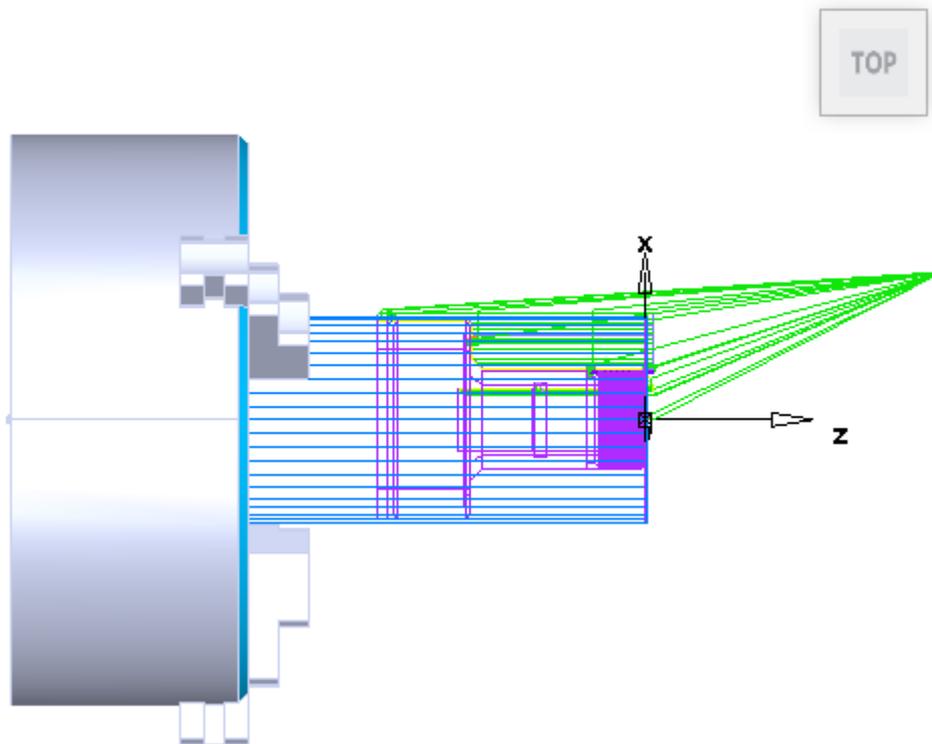
2D Simulation



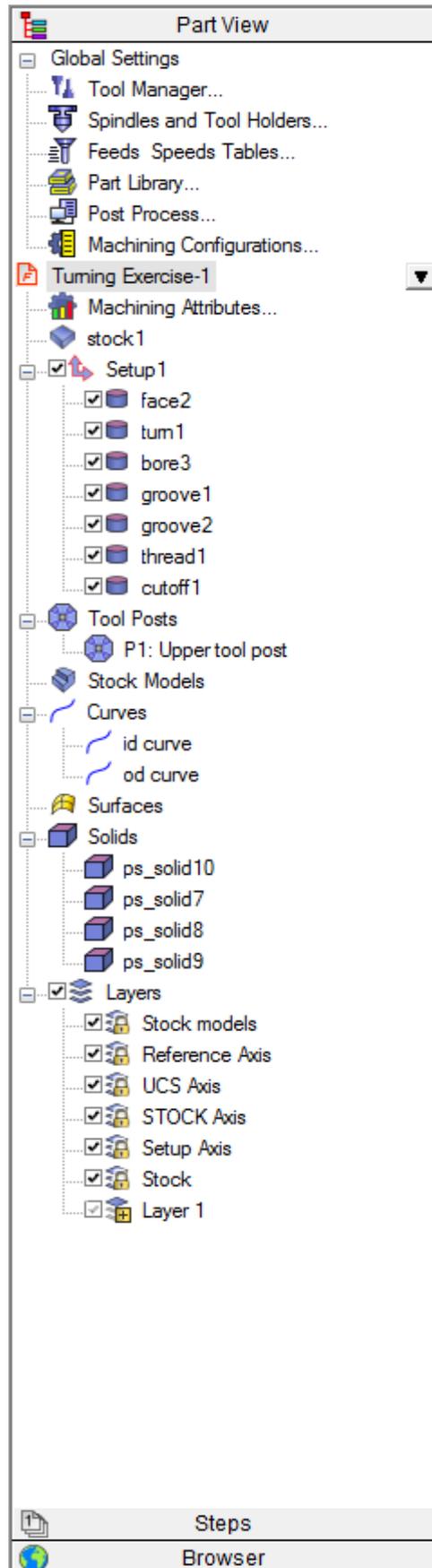
3D Simulation



Rapid moves to home position



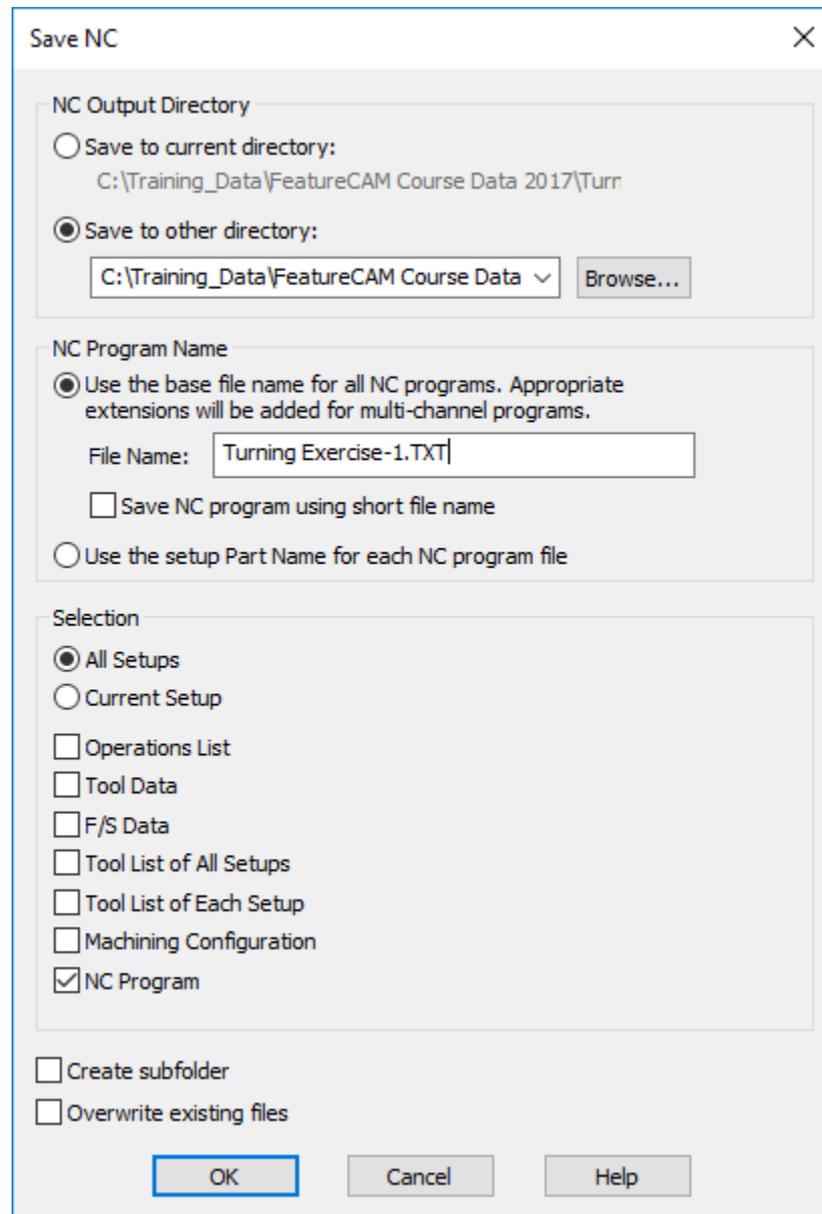
Part View All Operations





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

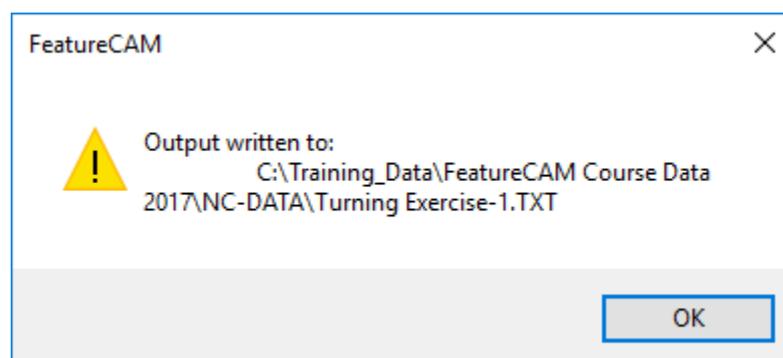
- 1 **Run the 3D Simulation** to check for errors etc.
- 2 From the **File** menu **select SaveNC**.



The 'Save NC' dialog box is shown with the following settings:

- NC Output Directory:**
 - Save to current directory:
C:\Training_Data\FeatureCAM Course Data 2017\Turn
 - Save to other directory:
C:\Training_Data\FeatureCAM Course Data (selected in dropdown) Browse...
- NC Program Name:**
 - Use the base file name for all NC programs. Appropriate extensions will be added for multi-channel programs.
File Name:
 - Save NC program using short file name
 - Use the setup Part Name for each NC program file
- Selection:**
 - All Setups
 - Current Setup
 - Operations List
 - Tool Data
 - F/S Data
 - Tool List of All Setups
 - Tool List of Each Setup
 - Machining Configuration
 - NC Program
- Create subfolder
- Overwrite existing files

Buttons: OK, Cancel, Help



The 'FeatureCAM' message box displays a yellow warning icon and the following text:

Output written to:
C:\Training_Data\FeatureCAM Course Data
2017\NC-DATA\Turning Exercise-1.TXT

Button: OK

NC Code output example

NC Code

P1: Upper tool post

```

%
O0001(TURNING EXERCISE-1)
( FEATURECAM - TRAINING POST FOR B AXIS TURNING )
( 07/09/2016 - 08:57:15 )

N1 (MAIN SPINDLE, TURNING)
( FINISH FACE FACE2 )
G18 G21 G40 G54 G80 G99
M69
M46
M303
G0 T1001 ( SW_TURN_80M_RH 0.4)
G361 B0. D1.
G43 H01.
T1002 (Next Tool)
G50 S4000
G97 S1397 M3
G0 Y0. Z0. M8
X111.0
G96 S487
G1 X-0.8 F0.152
X4.008 Z2.404
M9
G28 U0. V0.
G28 W0.

N2 (MAIN SPINDLE, TURNING)
( ROUGH TURN TURN1 )
G18 G40 G54 G80 G99
M69
M46
M303
G0 T1002 ( SW_TURN_80M_RH)
G361 B0. D1.
G43 H02.
T1001 (Next Tool)
G50 S4000
G97 S1160 M3
G0 Y0. Z3.636 M8
X100.2
G96 S365
G1 Z-135.8 F0.381
X105.0
X105.707 Z-135.446
G0 Z3.636
G1 X95.2
Z-R7 994
    
```

Op List Details NC Code Tool Posts

Part Documentation

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

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



- 6 Click the NC code step  in the Steps toolbar.

- 7 Click the Display the NC code  button in the dialog box to generate.

```

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

```

NC code.

Changing Post Processors and Saving NC Code

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

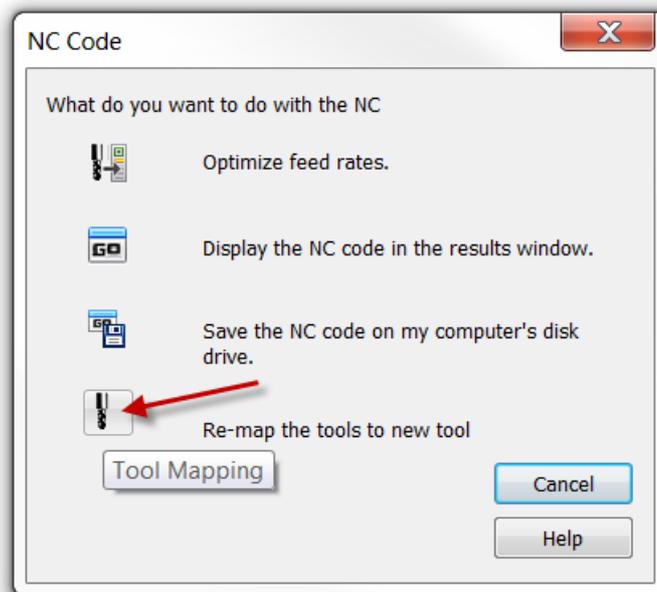
- 5 Clicking Close will not change your post processor.
- 6 Click the NC code step  in the Steps toolbar.
- 7 Click the Save NC button  from the dialog box.
- 8 Accept the default filename and directory and click OK.
- 9 Load your file onto your cnc machine via RS232 or Memory stick.
- 10 You are ready to machine the part.
- 11 Save the FeatureTURN Part
- 12 Select the Save button  from the Standard toolbar.
- 13 Select the directory and then type your part name and click OK.

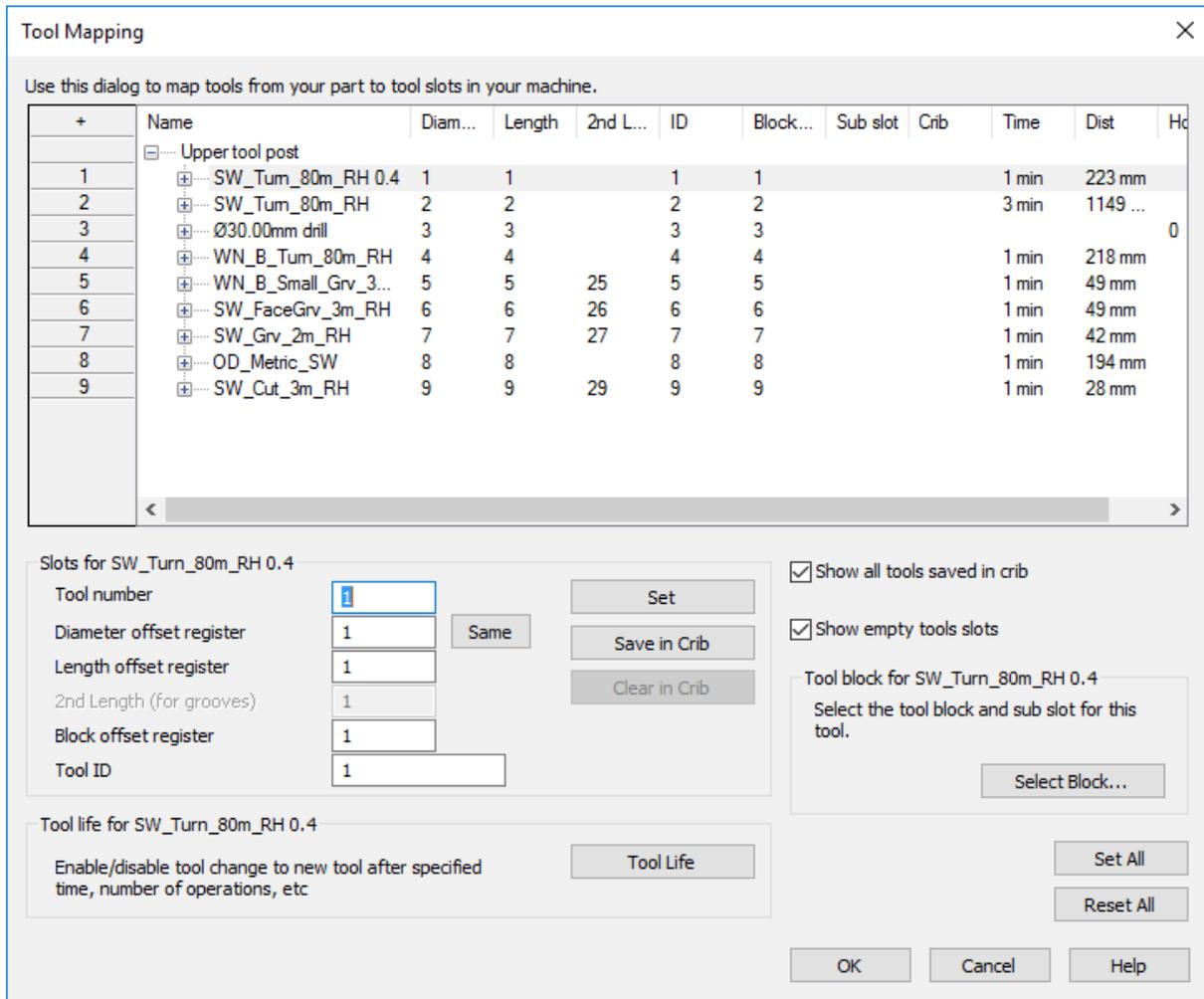
Tool Mapping



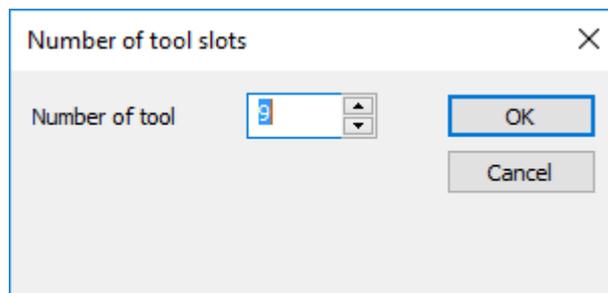
You can open the Tool Mapping dialog in one of these ways:

- 14 Select **Manufacturing > Tool Mapping** from the menu.
- 15 Click NC Code  in the Steps panel, then click Re-map the tools to new tool slots in the NC Code dialog.





- 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.



- **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.

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

Tool Numbering

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

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

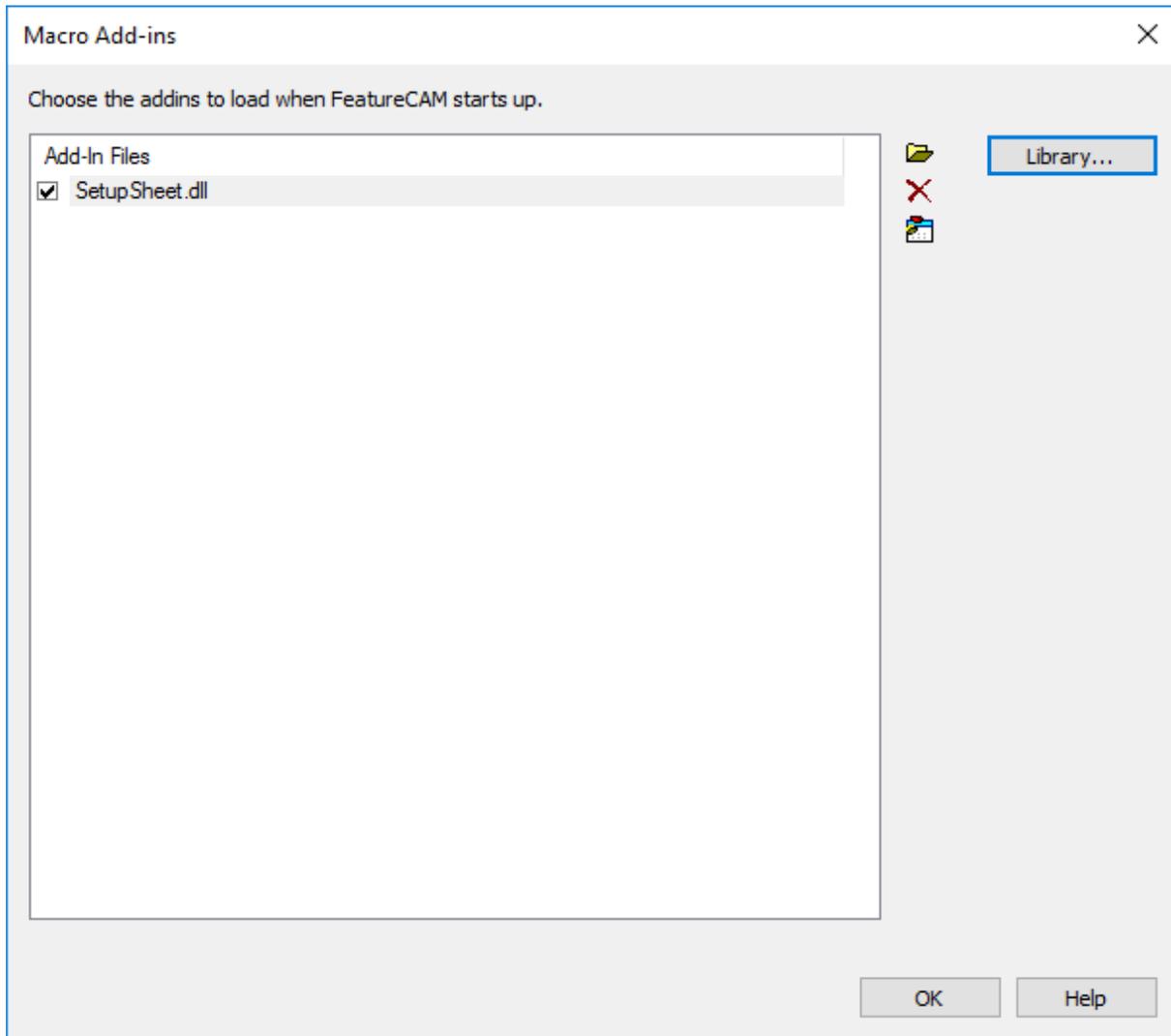
Custom Setup Sheet Add-In

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

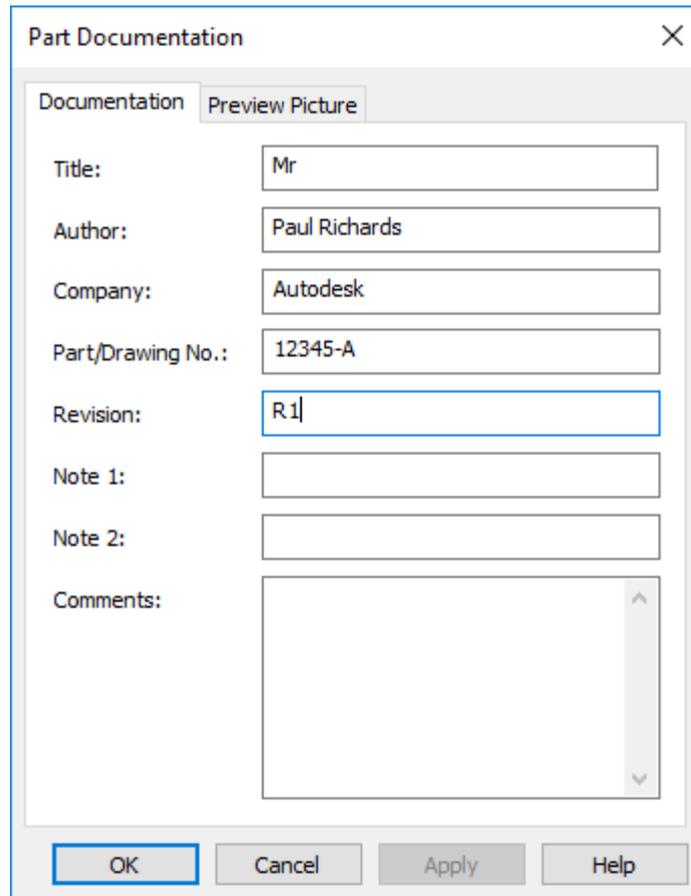
The add-in extracts information from the **Part Documentation** dialog and enables you to take images for each Setup.

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

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



- 22 Click the Browse  button and browse to the SetupSheet.dll file. If you installed FeatureCAM in the default location, the file is at **C:\ProgramFiles\Autodesk\FeatureCAM\Addins\SetupSheet\SetupSheet.dll**.
- 23 In the Macro Add-ins dialog, in the Add-In Files list, ensure that the check box to the left of the SetupSheet.dll file address is selected.
- 24 Click **OK**. The Utilities toolbar is displayed, containing the SetupSheet button .
- 25 Click the SetupSheet button to run the add-in. The Setup Sheet Options dialog is displayed:
- 26 Remember to set the Part Document information in **File>Part Documentation**.
- 27 The Title, Author, Note 1, Note 2, Company, Part/Drawing No., Revision, and Comments values are copied from the Documentation tab of the Part Documentation dialog and you can edit them there.



The image shows a 'Part Documentation' dialog box with a close button (X) in the top right corner. It has two tabs: 'Documentation' (selected) and 'Preview Picture'. The 'Documentation' tab contains several text input fields: 'Title:' with 'Mr', 'Author:' with 'Paul Richards', 'Company:' with 'Autodesk', 'Part/Drawing No.:' with '12345-A', and 'Revision:' with 'R1'. Below these are three empty text boxes for 'Note 1:', 'Note 2:', and a larger text area for 'Comments:'. At the bottom, there are four buttons: 'OK' (highlighted with a blue border), 'Cancel', 'Apply', and 'Help'.

- 28 Select Ok to close the dialog.
- 29 Click **OK**. The Utilities toolbar is displayed, containing the SetupSheet button 
- 30 You can use this dialog to capture an image for each Setup in the current document, as well as an image to represent the whole document.
- 31 To capture an image for a Setup, select the Setup name in the Setups list and click the Capture Setup Image button to capture the current contents of the graphics window.

Setup Sheet Options
✕

Title

Author

Note 1

Note 2

Company

Part/Drawing No.

Revision

This information is copied from the File > Part Documentation dialog and you can edit it there.

Comments

Setups

Setup 1

Capture Setup Image

Capture Document Image

Create Setup Sheets

Browse Template...

Output Type

HTML

Excel

Create Sub Folder for Output

OK

Help



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



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

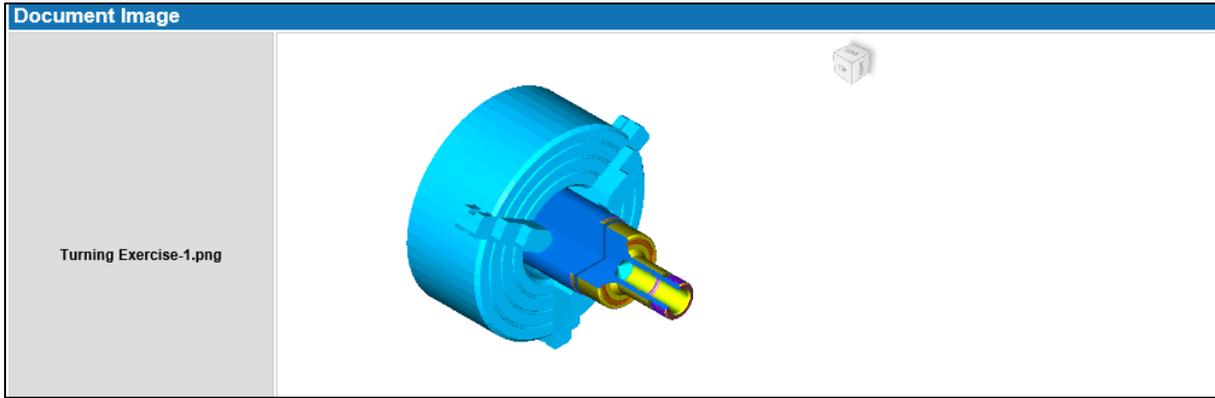
- 32 To create the setup sheets, click the **Create Setup Sheets** button. The part is simulated to generate toolpaths and the setup sheet is displayed in your web browser. You can save the HTML file from your browser.
- 33 To open the setup sheet in the FeatureCAM Browser, select **File > Open** from the menu and browse to the setup sheet **HTML** file.

Turning Exercise-1				FeatureCAM SetupSheet	
Title	Mr	Company	Autodesk	Comments Info	
Author	Paul Richards	Part/Drawing No.	12345-A		
Note 1		Revision	R1		
Note 2					

Stock Info				
Type	Material	OD	ID	Z
Round	ALUMINUM	105.000000	0.000000	250.000000

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

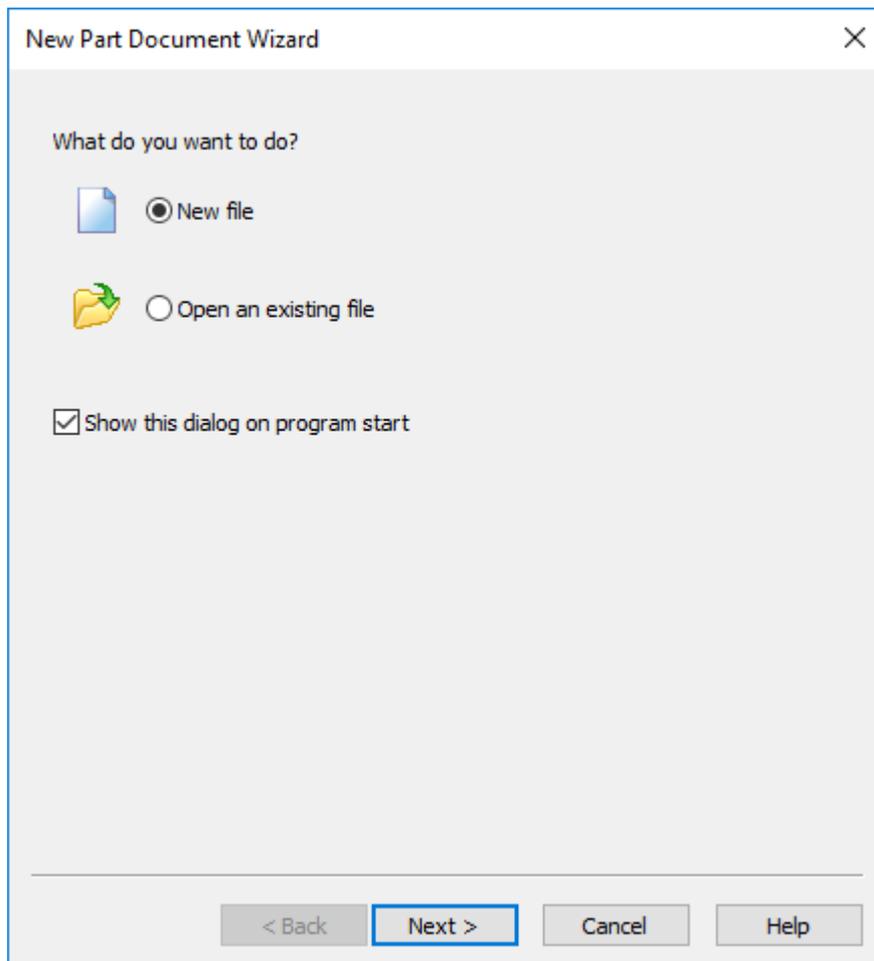
Setup : Setup1									
Setup Origin	0.0000, 0.0000, 0.0000								
Feature Name	Feature Type	Operation	Tool Number	Tool Name	Operation Feed	Operation Speed	Operation Time	Operation Comment	
face2	Face Turn	Finish	1	SW_Turn_80m_RH 0.4	0.1524	487.6800	0:00:19.4		
turn1	Turn Bore	Rough	2	SW_Turn_80m_RH	0.3810	365.7600			
turn1	Turn Bore	Finish	1	SW_Turn_80m_RH 0.4	0.1524	487.6800			
bore3	Turn Bore	Spiral	3	Ø30.00mm drill	0.4500	808.5071			
bore3	Turn Bore	Rough	4	WN_B_Turn_80m_RH	0.3810	365.7600			
bore3	Turn Bore	Finish	4	WN_B_Turn_80m_RH	0.1524	487.6800			
groove1	Groove Turn	Rough	5	WN_B_Small_Grv_3m_RH	0.3810	365.7600			
groove1	Groove Turn	Finish	5	WN_B_Small_Grv_3m_RH	0.1524	487.6800			
groove2	Groove Turn	Rough	6	SW_FaceGrv_3m_RH	0.3810	365.7600			
groove2	Groove Turn	Finish	6	SW_FaceGrv_3m_RH	0.1524	487.6800			
thread1	Thread	Rough	7	SW_Grv_2m_RH	0.3810	365.7600			
thread1	Thread		8	OD_Metric_SW	1.5000	2330.0000			
cutoff1	Cutoff	Rough	9	SW_Cut_3m_RH	0.3810	365.7600			
cutoff1	Cutoff	Rough	9	SW_Cut_3m_RH	0.1524	487.6800			
Total Setup Time									



Save your File and then close this document.

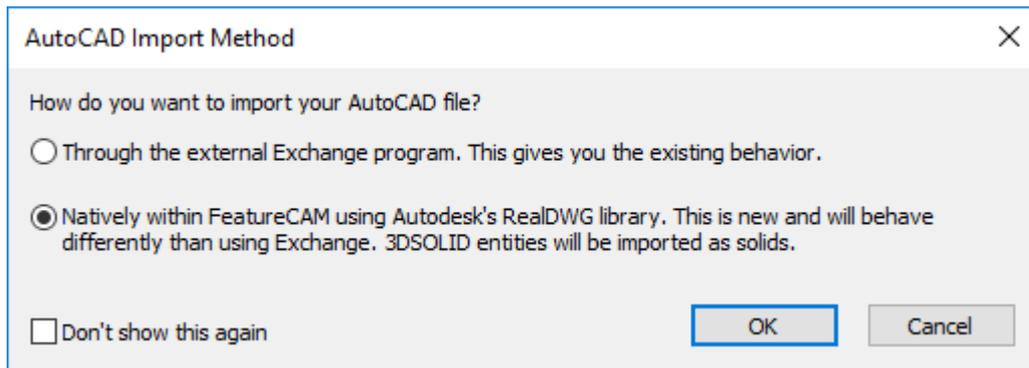
Exercise 2

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

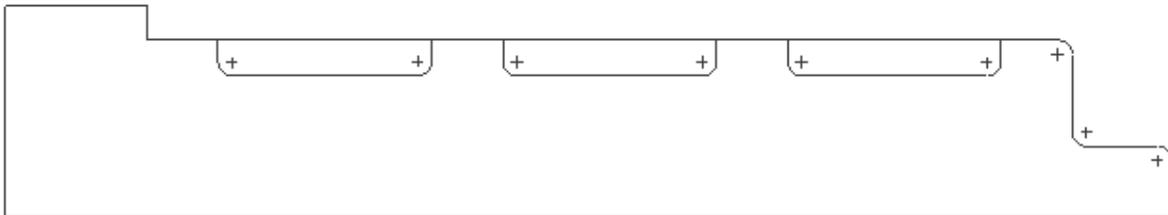


- 1 Cancel the **Stock Wizard**.
- 2 Import – **Turning Example 2.dxf** select **Open** to import the file into FeatureTURN. (Class Exercise)

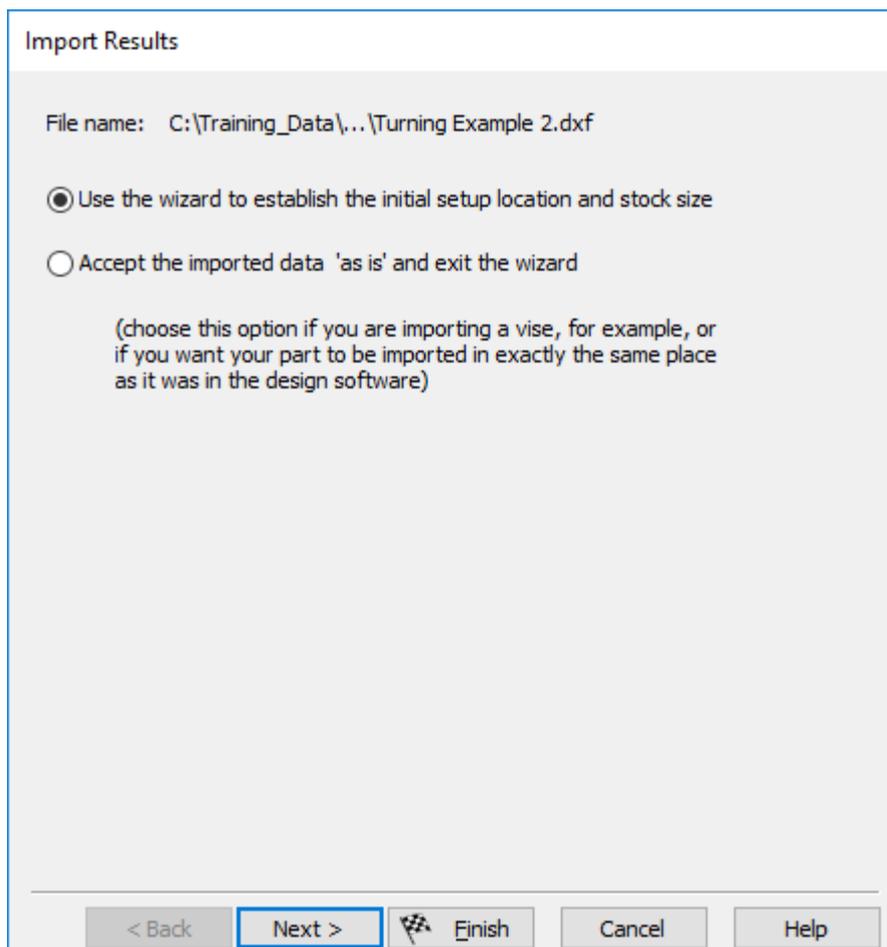
- 3 We are going to use the RealDWG Library file (see below)



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



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

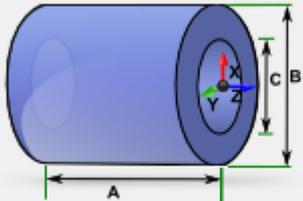


- 6 Keep selecting **Next** until you get to the **Stock Dimensions**.
- 7 Remember to use the option **Compute stock size from the size of the part**.

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

Stock Dimensions

Enter specific stock dimensions
 Compute stock size from the size of the part



	Imported Data	Extra stock size	Stock size
Length:	410.000	Front <input type="text" value="1"/> mm.	= 411.000
		Back <input type="text" value="0.000"/> mm.	
OD	75.000	OD <input type="text" value="0.000"/> mm.	= 75.000

Preview

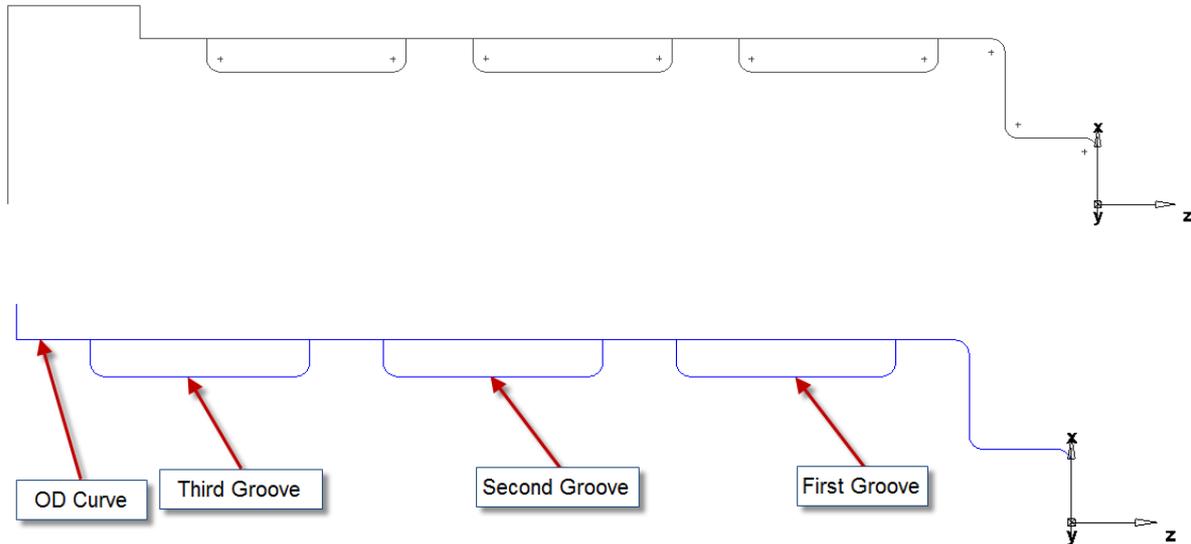
9 Select **Finish**.



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



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



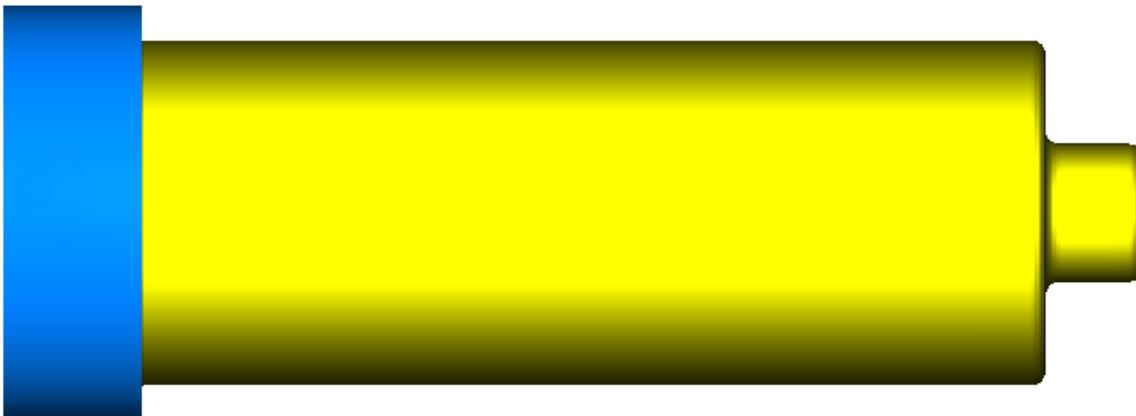
- 10 On the illustration above, the Curves shown are the **OD Curve, First Groove, Second Groove, and Third Groove.**

Turn Feature

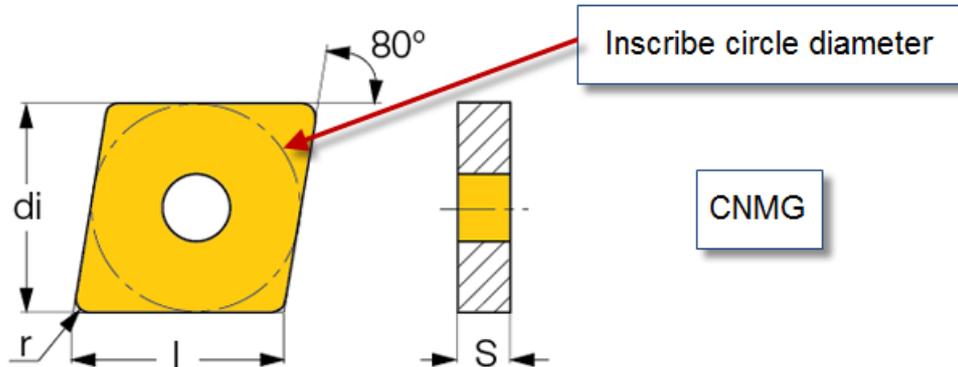
- 11 Cancel this command by pressing **ESC**.
- 12 Hide the Geometry by selecting **View>Hide>Hide all Geometry**.
- 13 Select **Ctrl+R New Feature** Select **Turning**, select **next**, select **Face** Select **Finish**.
- 14 Select **Ctrl+R New Feature** and select **Turning> Turn** select next then select **odcurve** from the **New Feature Curve** menu. Select **Finish**.



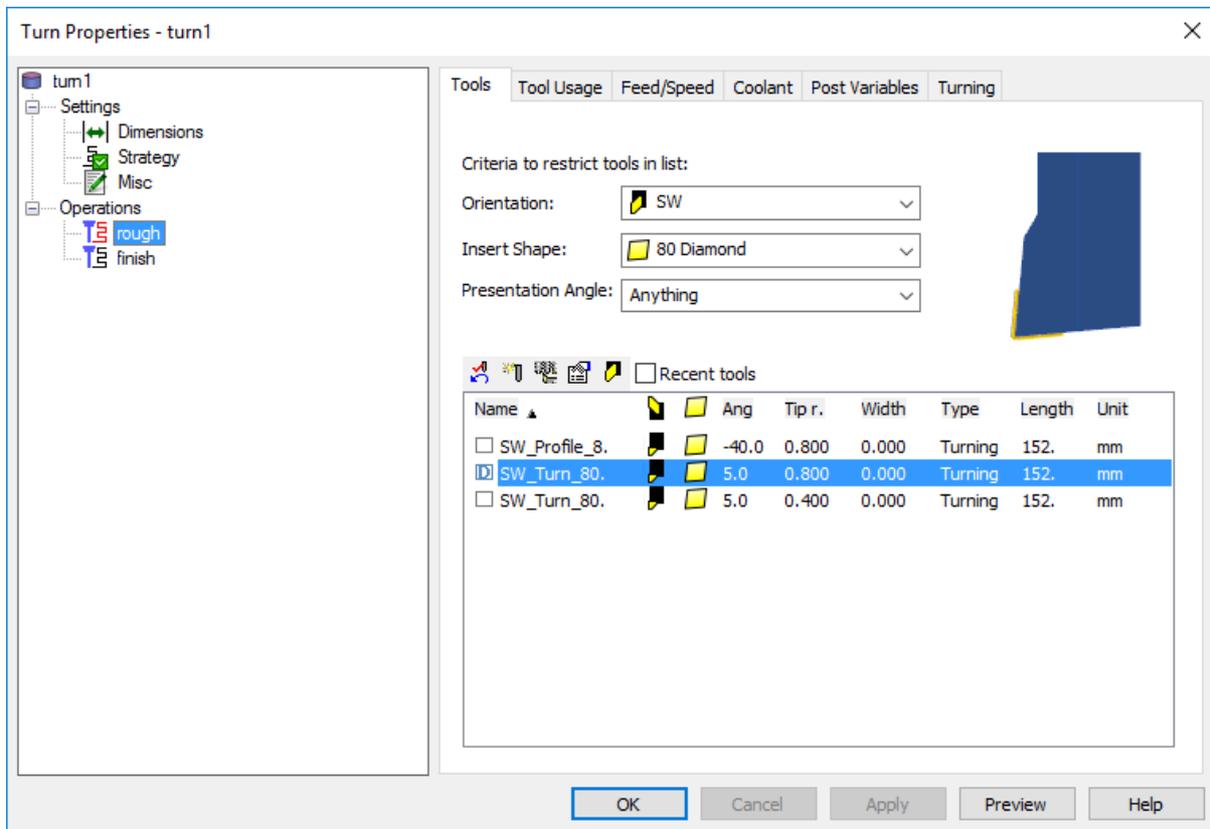
New to **FeatureCAM** we can automatically Fillet external or internal sharp corners. For internal bore fillet corners use **Other side**.



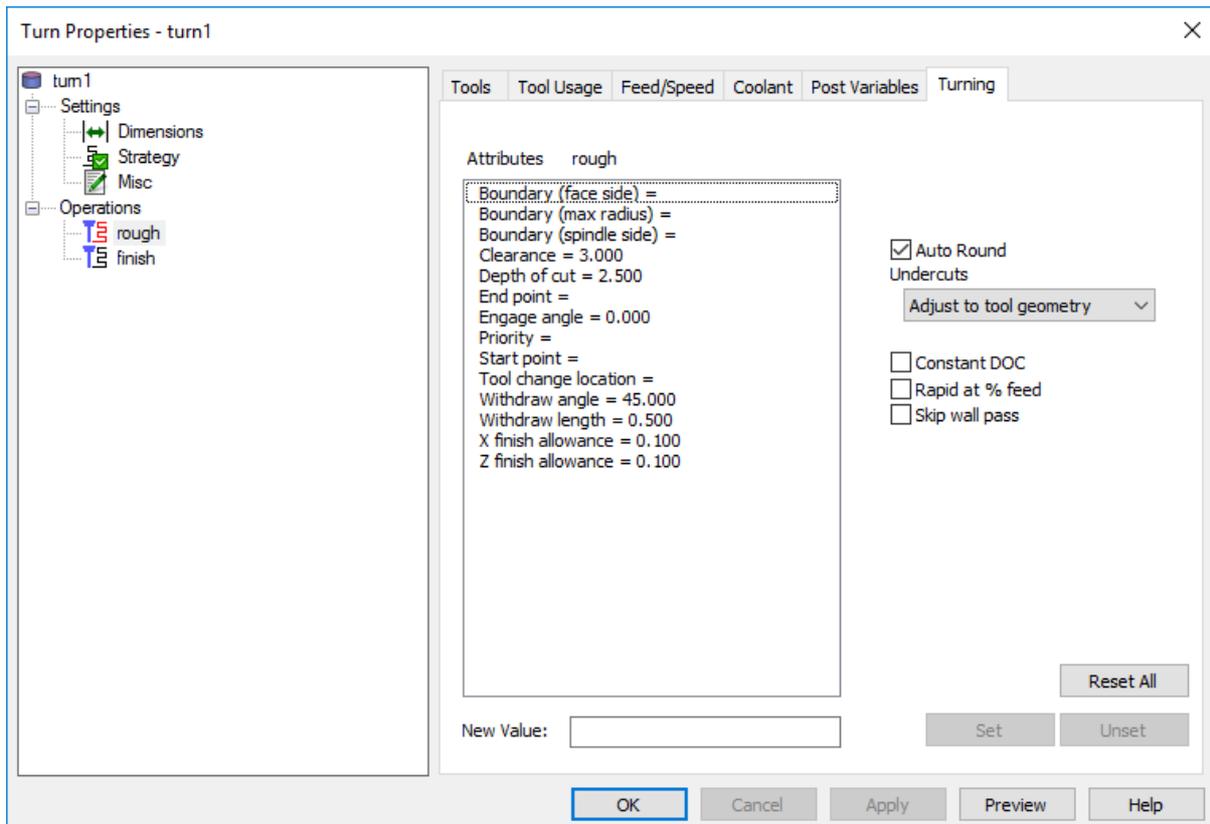
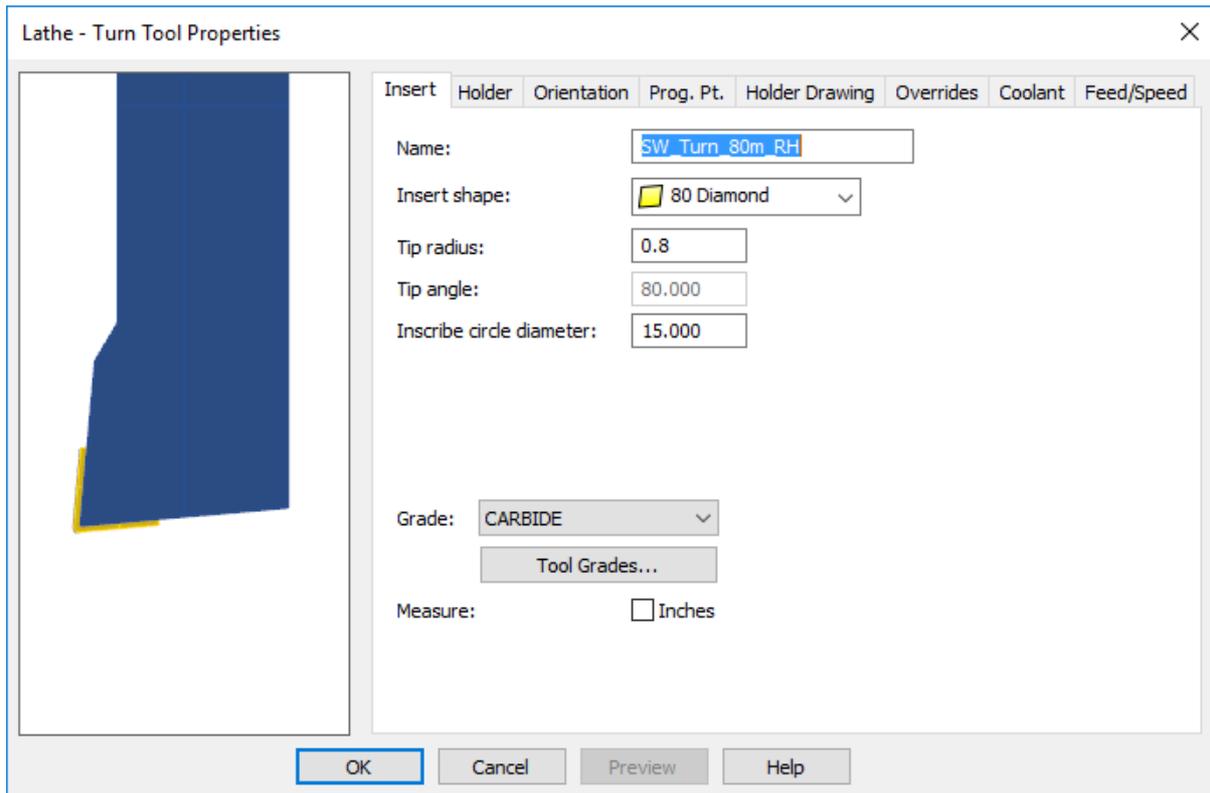
In the Properties, please select **Rough** and then the **Tools** tab. This is a listing of the tool that **FeatureCAM** picked as a default tool with a **(D)** **Default** in the check box and other tools that may be chosen to replace the **80 degree tool**.



Typical inscribed circle diameters 9.52, 12.7, 15.88, 19.05

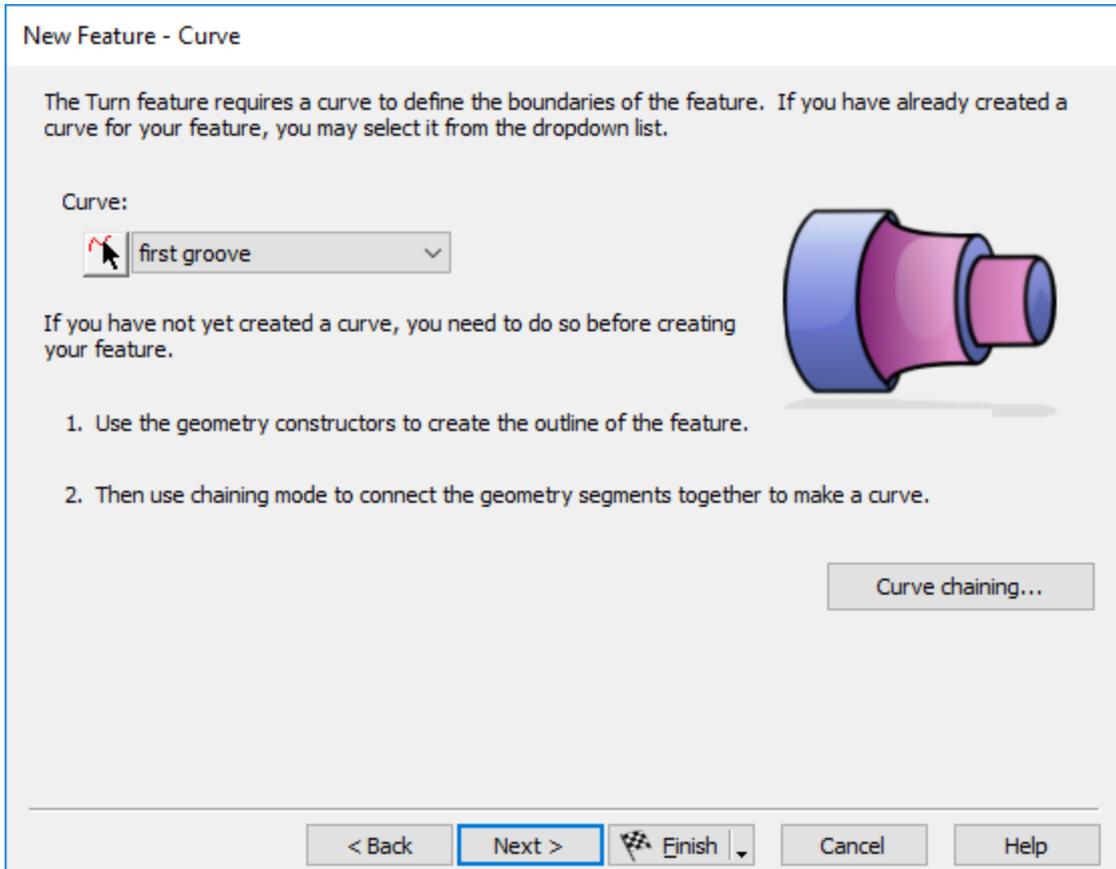


The **CNMG** 80 degree diamond tool may be the most used style in the industry today. It is strong and allows for turning and facing with the same tool because of the 5 degree relief on two sides of the insert.

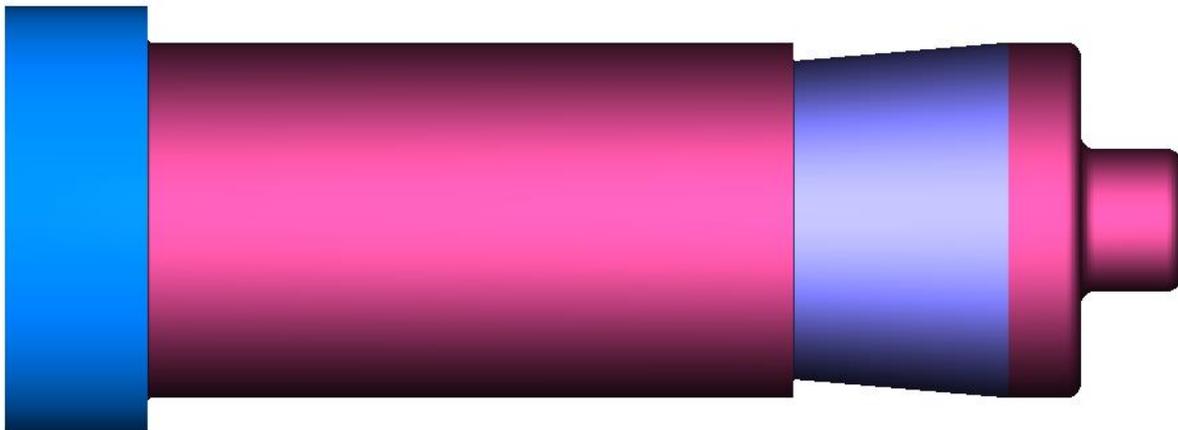


For the next operation we are going to machine the first groove using a turning operation and then create a back turning operation to finish the groove.

- 15 Create a new feature. Select **Ctrl+R** and select **Turning and Turn**. Select **Next**.
- 16 Select the **first groove** curve. Then select **Finish**.



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

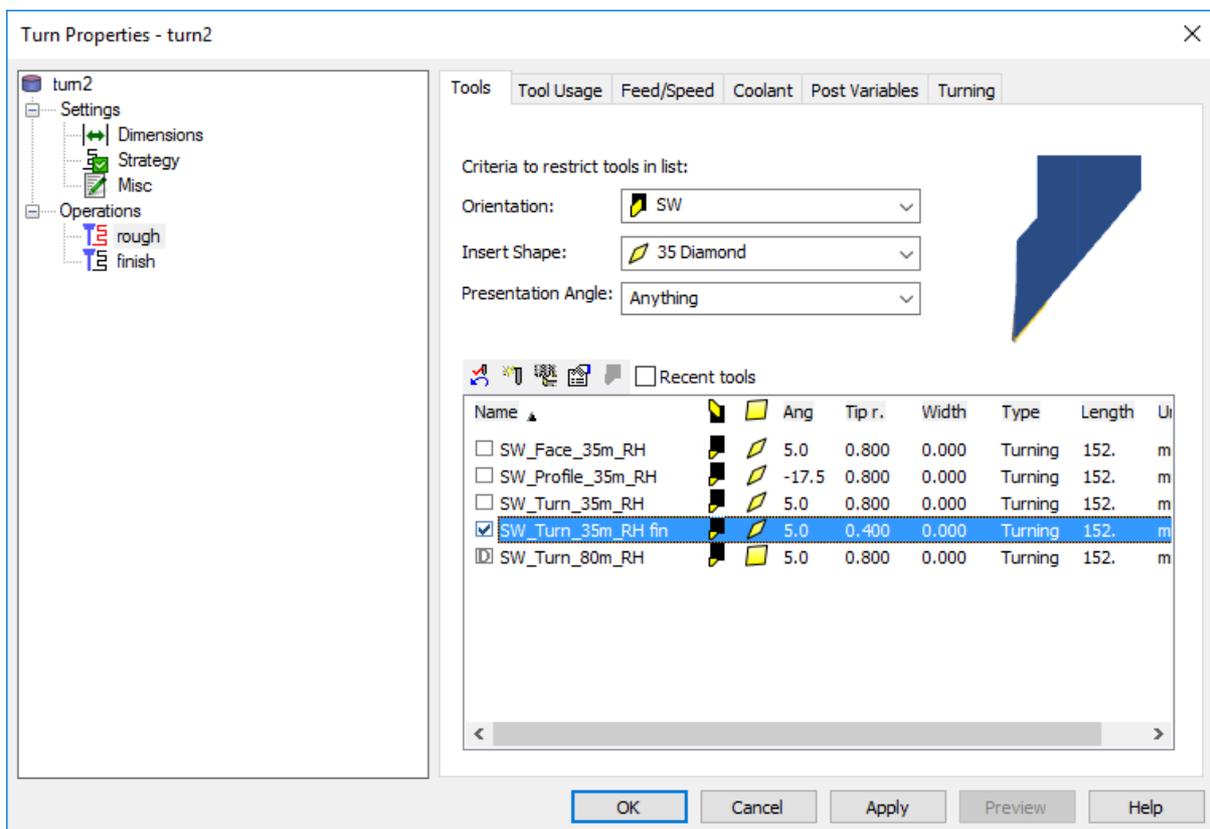


***FeatureCAM** defaults to an 80 degree diamond insert tool. A very common turning tool but in this case with the undercut in the **OD**. You can see that the integrity of the shape (curve) that you are cutting is protected so that the trailing side of the tool does not gouge the undercut. **FeatureCAM**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



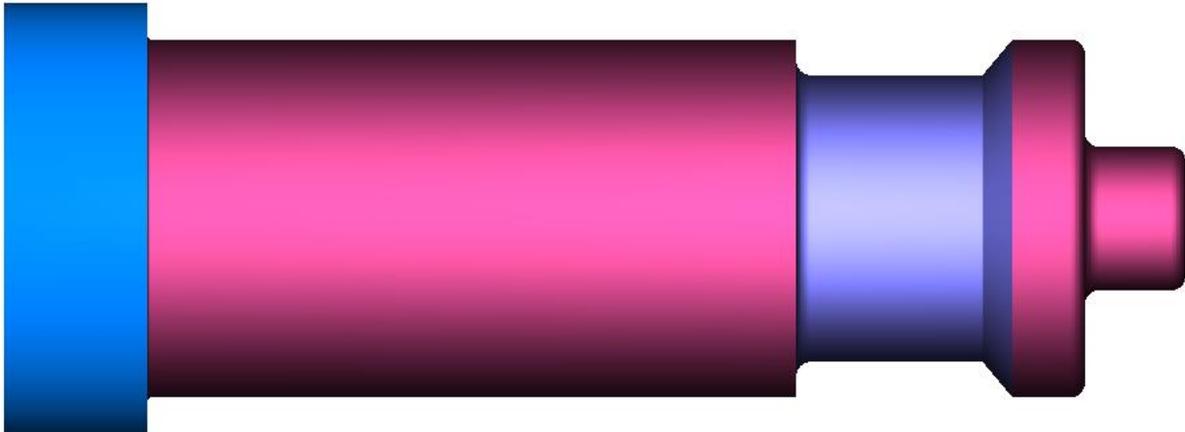
FeatureCAM allows for alternative tools to be selected from the tool crib so we can replace the 80 degree tool with a tool with more relief on the back side and this can be done on the fly as you create the feature. We will exchange this tool for a 35 degree tool with much more clearance.

- 17 First, in the Orientation box the tool is described as **SW** or **South West**. This is the orientation of the current tool and will be the same for the replacement.
- 18 Drop the menu down in the **Insert Shape** box and select **35 Diamond**.
- 19 The presentation angle will remain the same.
- 20 When you Clicked on the **35 Diamond**, **FeatureCAM** shows you the **35 Diamond tools** but this does not select one, you must do that manually.
- 21 Click on each tool description until you find the right tool displayed for the application. You must check the box to the left of the tool to select the new tool.



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

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



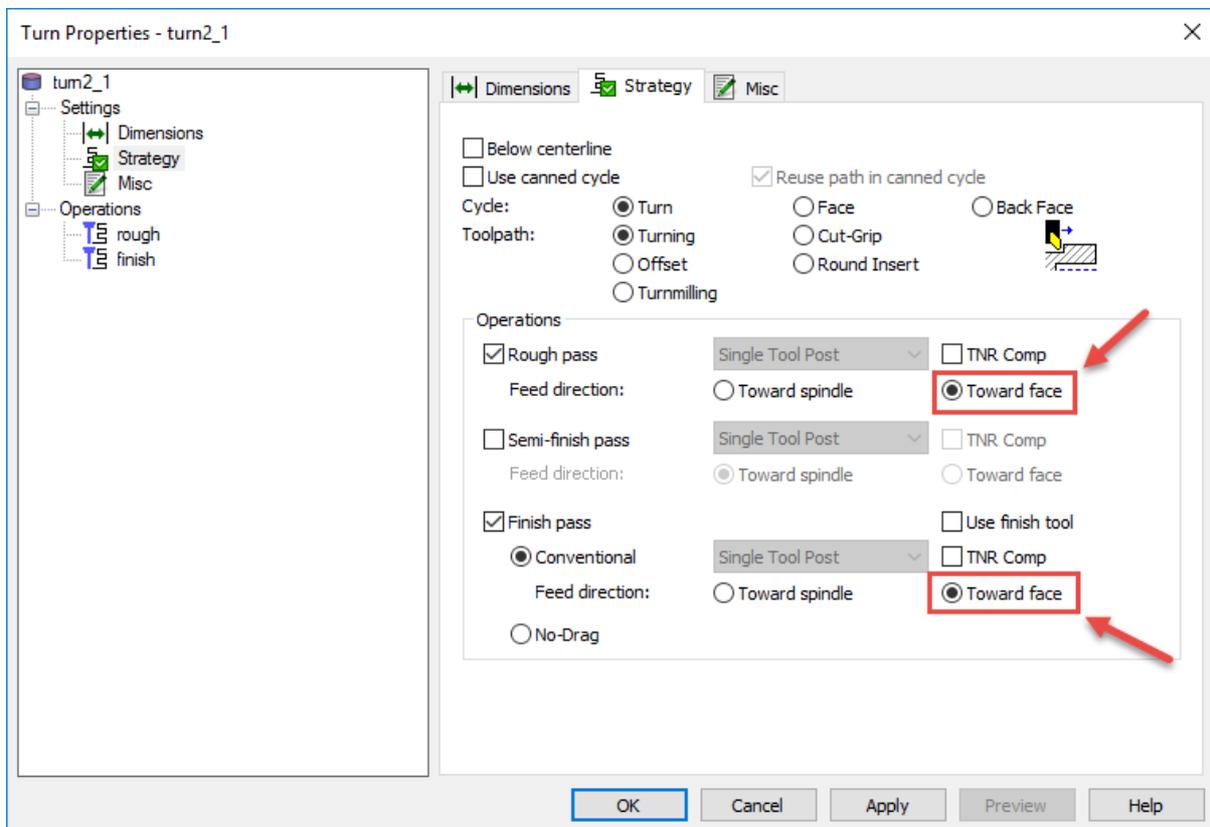
You will notice that when FeatureCAM starts it cuts fresh air at the start.

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



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

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

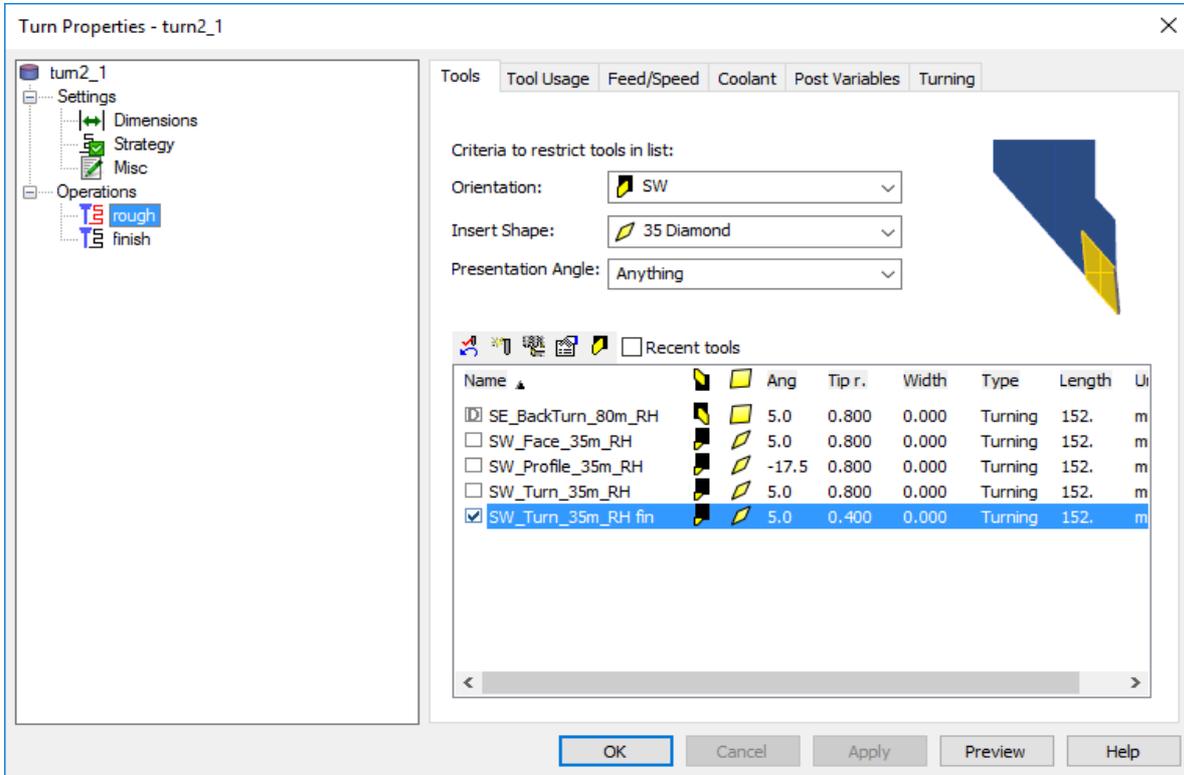




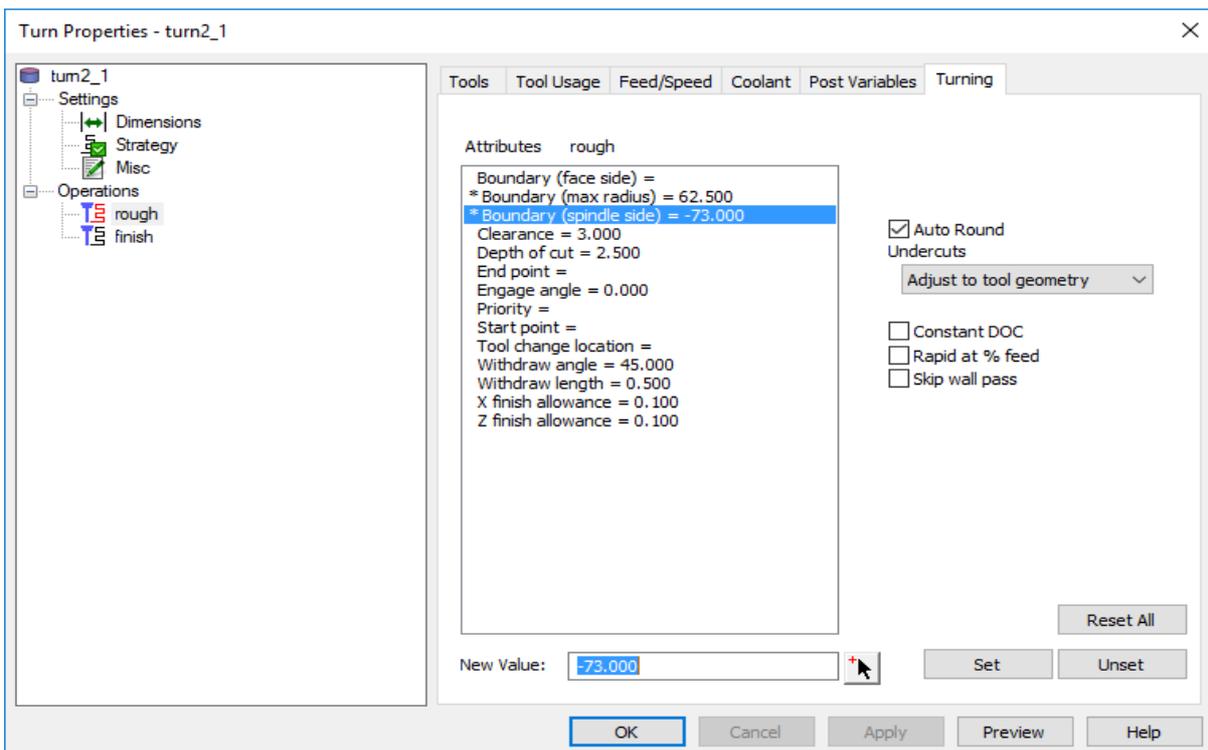
By creating a copy of the original Feature the tooling and maximum radius boundary will be maintained.



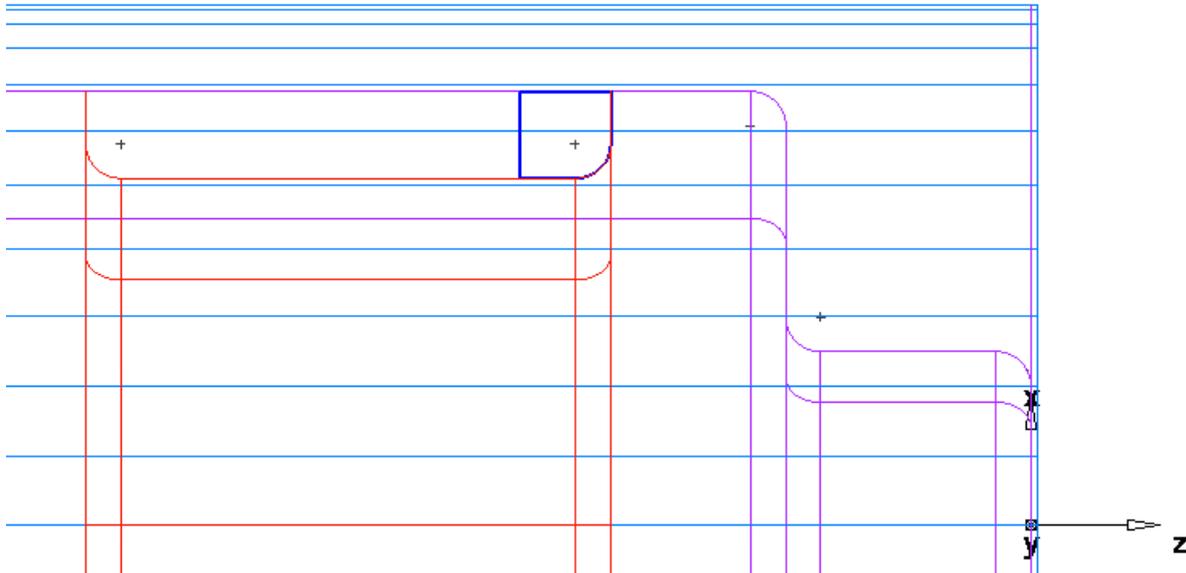
Double click on the copy of the groove feature in PartView



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

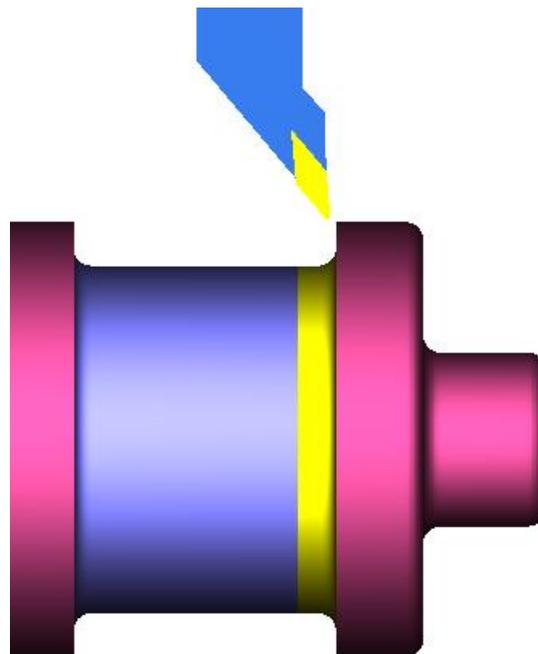


- 26 Click on **boundary (Spindle side) =** and either Click on the **Arrow** next to the New Value box at the bottom and then click the location on the part in the Graphics Window where you want the finish cut to start or enter a dimension using a negative number measuring from the Left end of the Curve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



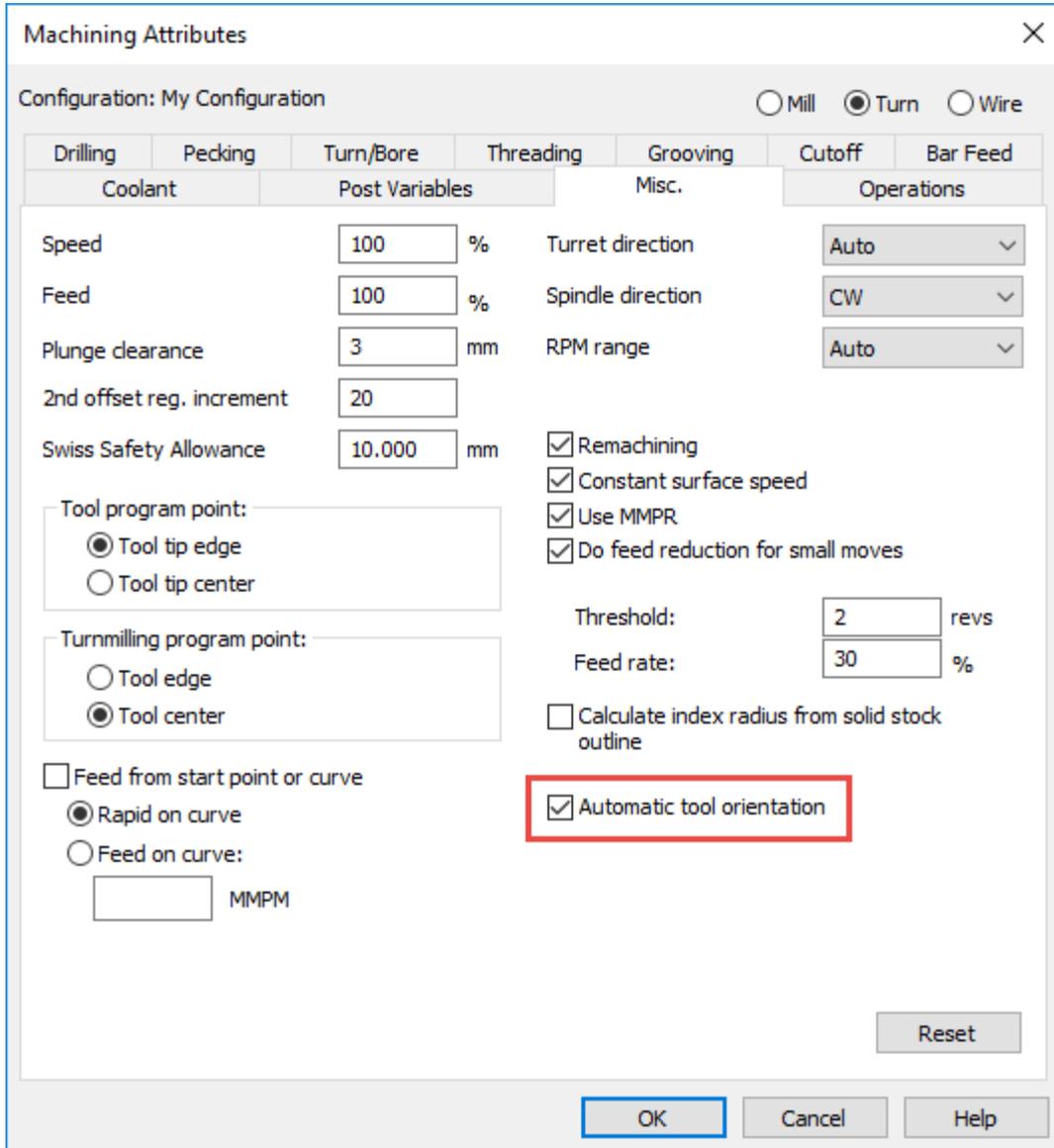
Boundary (Spindle side) as shown above. Select **Apply** and **Finish**.

- 27 You may wish to set the **Right boundary** position. Activate the same menu again and set a value into the **Boundary (face side)** position. This will limit the amount the tool has to travel.
- 28 Run **3D Simulation** to see results.





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



Machining Attributes

Configuration: My Configuration Mill Turn Wire

Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed
Coolant		Post Variables		Misc.	Operations	
Speed	<input type="text" value="100"/>	%	Turret direction	<input type="text" value="Auto"/>		
Feed	<input type="text" value="100"/>	%	Spindle direction	<input type="text" value="CW"/>		
Plunge clearance	<input type="text" value="3"/>	mm	RPM range	<input type="text" value="Auto"/>		
2nd offset reg. increment	<input type="text" value="20"/>					
Swiss Safety Allowance	<input type="text" value="10.000"/>	mm	<input checked="" type="checkbox"/> Remachining			
Tool program point:			<input checked="" type="checkbox"/> Constant surface speed			
<input checked="" type="radio"/> Tool tip edge			<input checked="" type="checkbox"/> Use MPMR			
<input type="radio"/> Tool tip center			<input checked="" type="checkbox"/> Do feed reduction for small moves			
Turnmilling program point:			Threshold: <input type="text" value="2"/> revs			
<input type="radio"/> Tool edge			Feed rate: <input type="text" value="30"/> %			
<input checked="" type="radio"/> Tool center			<input type="checkbox"/> Calculate index radius from solid stock outline			
<input type="checkbox"/> Feed from start point or curve			<input checked="" type="checkbox"/> Automatic tool orientation			
<input checked="" type="radio"/> Rapid on curve						
<input type="radio"/> Feed on curve:						
<input type="text" value=""/>			MMPM			

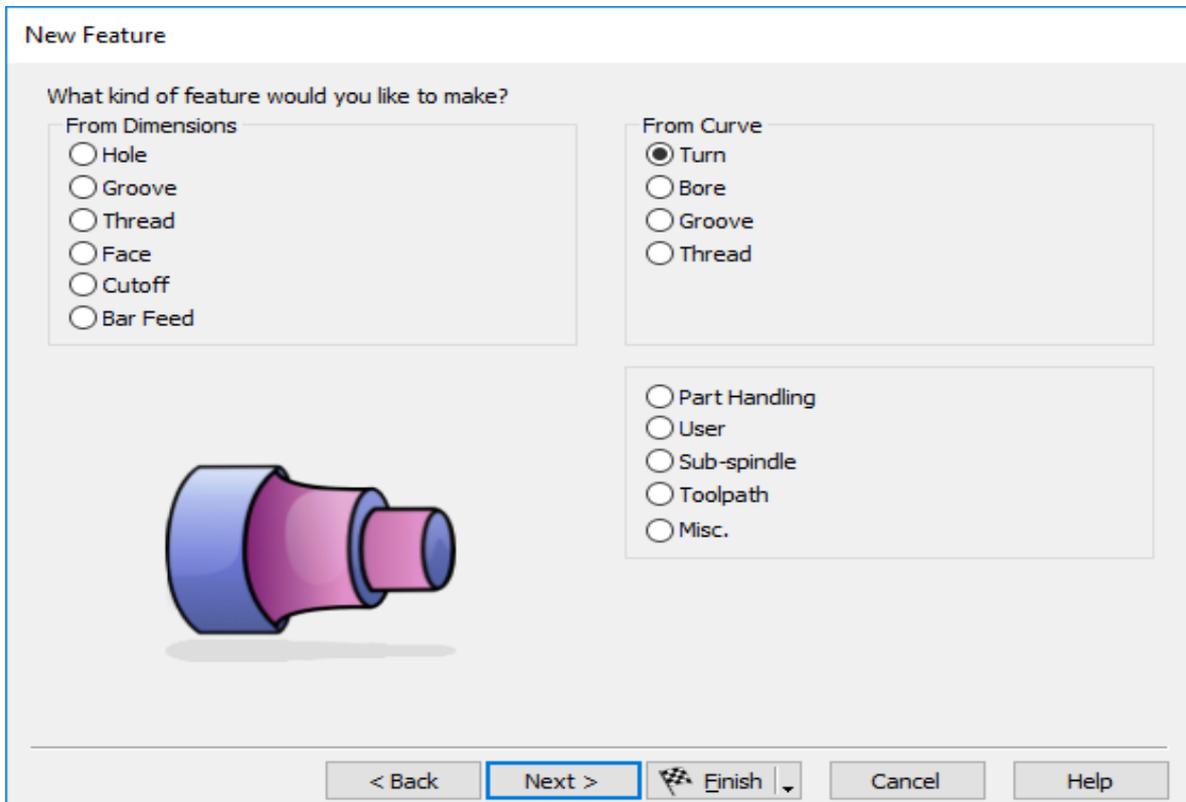
Cut-Grip



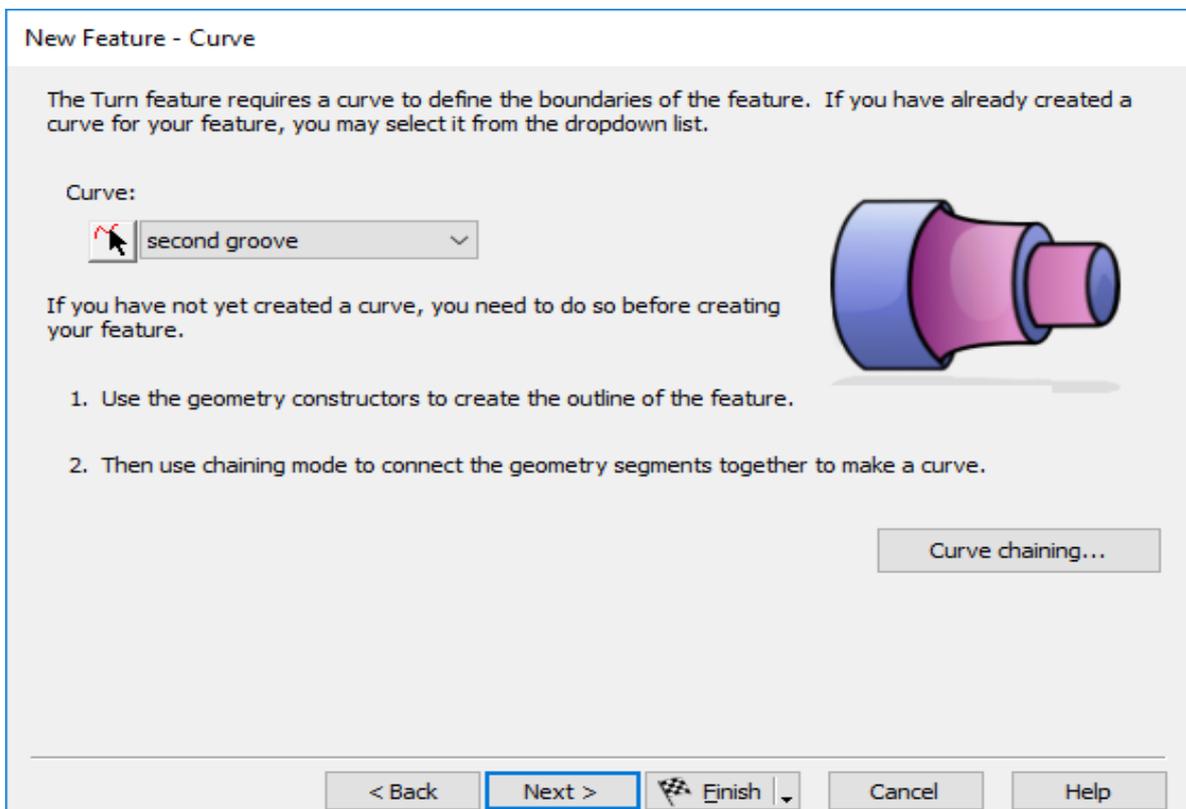
Cut-Grip is a specially designed strategy for turning (also grooving) and looks much like a standard grooving tool but because of the geometry of the insert and the design of the holder it can cut parallel to the Z axis almost as fast as a standard turning tool. In large quantity production machining,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

Second groove Feature using Cut-Grip

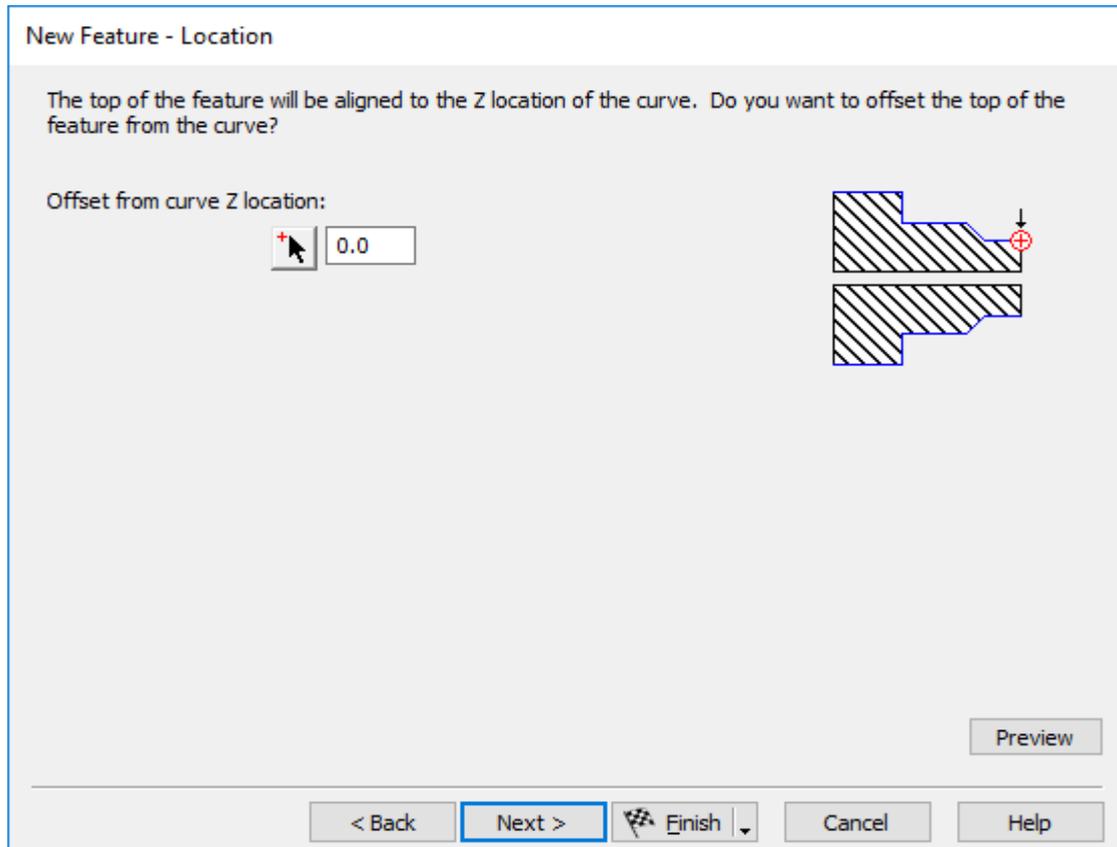
29 The Next feature to consider is **Cut-Grip**. Click on **New Feature** and select **Turn** from Curve.



30 Select **Next**.

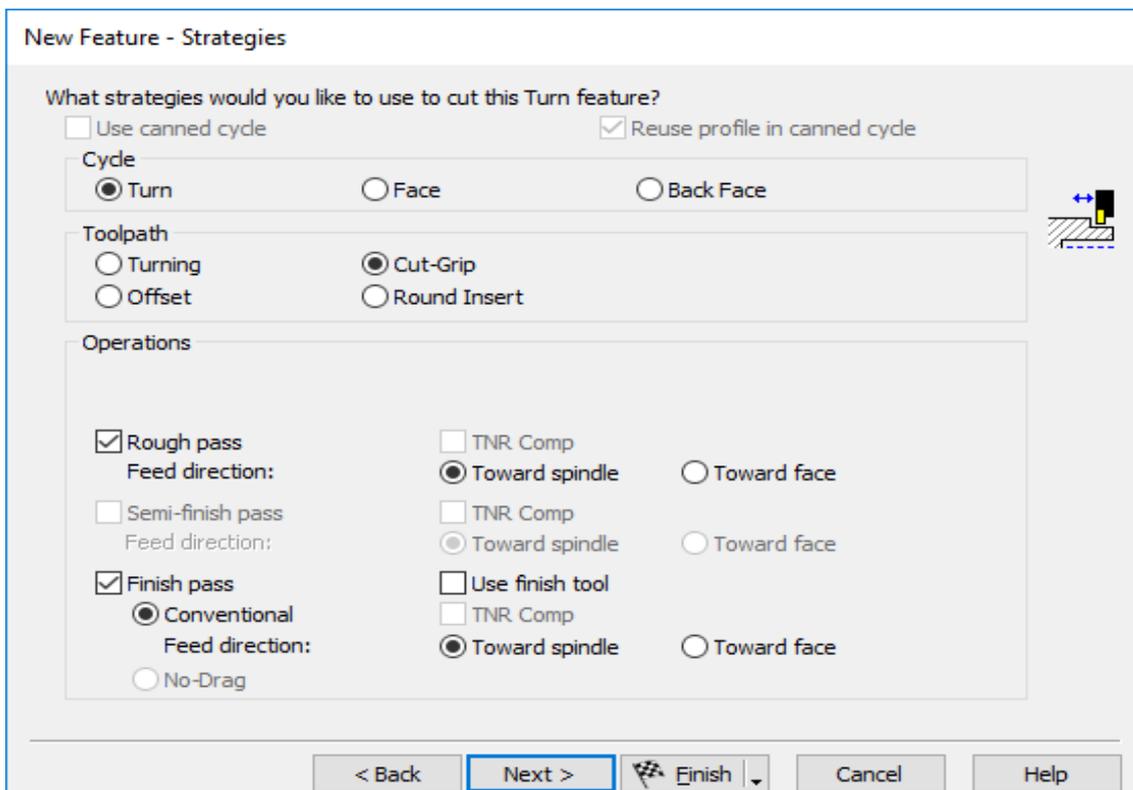


- 31 Select **second groove**. Your Curve may have a different name.
- 32 Select **Next**. Accept the location.

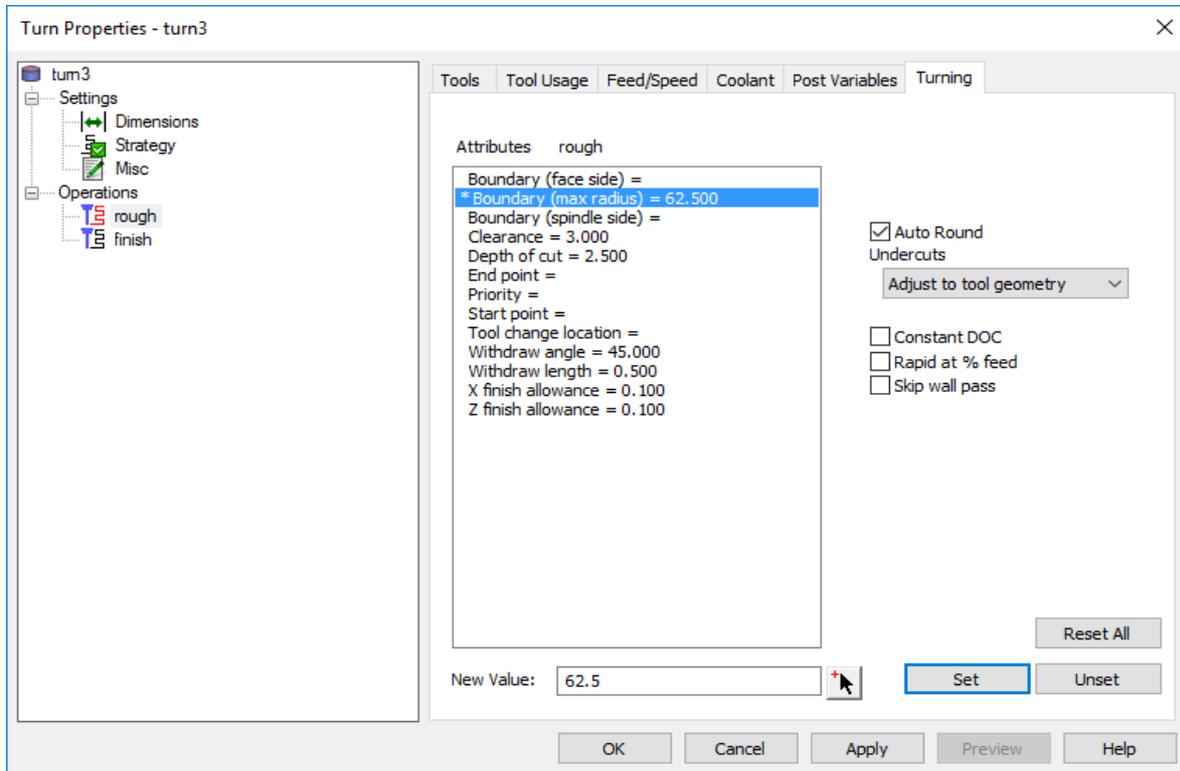


- 33 Select **Next**.

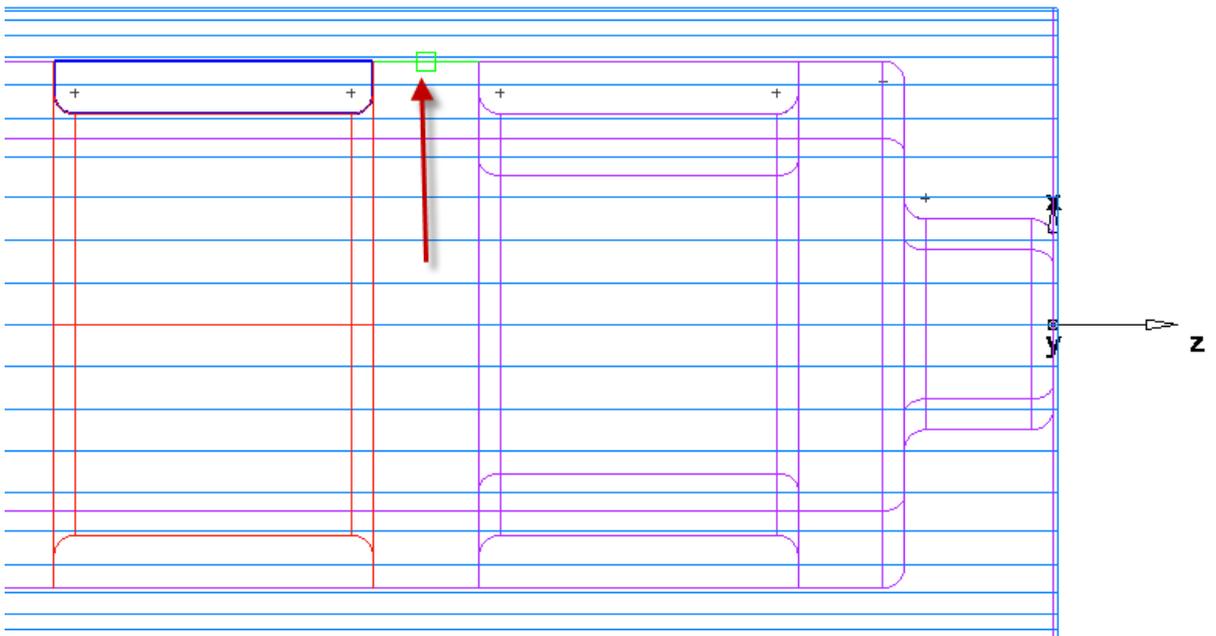
34 New Feature Strategies



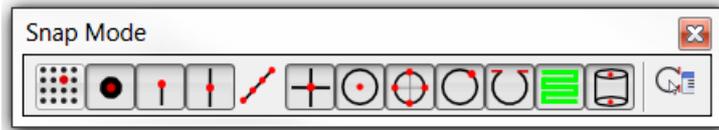
- 35 Select **Cut-Grip** then select **Finish**.
- 36 Select **Rough and Turning**
- 37 Select **Maximum Radius Boundary** and **pick X location** or enter **62.5mm** into the New value field.



- 38 Pick Boundary edge.



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

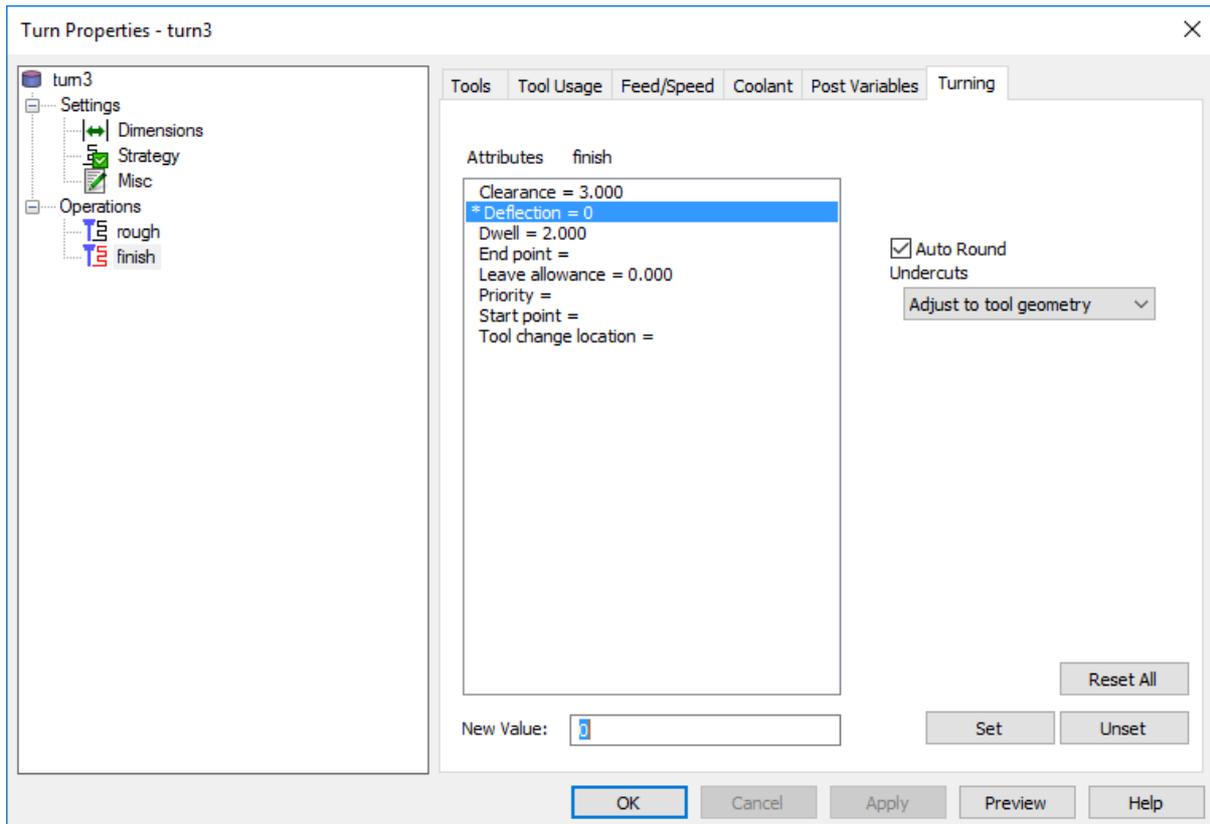


39 The Boundary is now adjusted to **62.5mm** Radius **125mm** Diameter.

40 Shown below is the Boundary Adjusted.



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



41 **Cut-Grip** is now completed

Groove from Curve

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

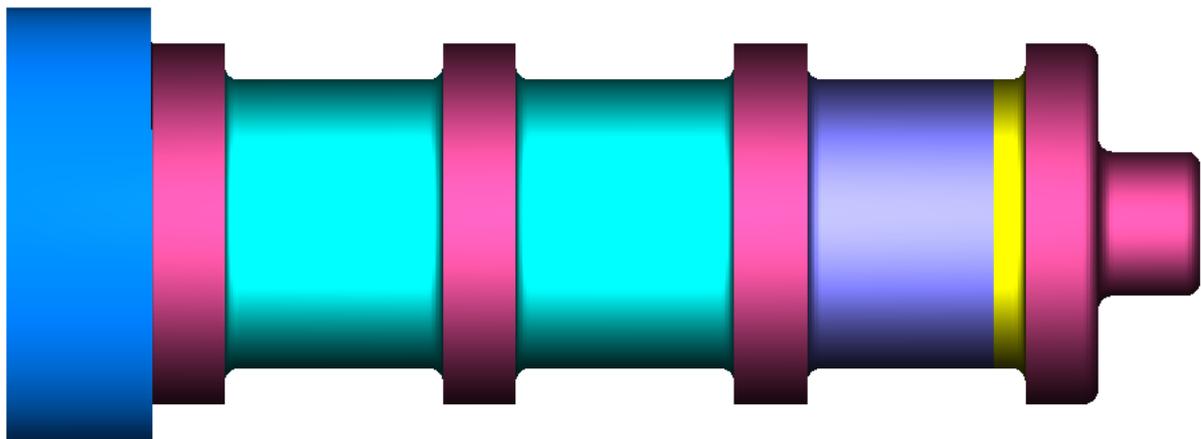
- 42 **Eject** the **Simulation**.
- 43 Click on **New Feature** and select **Groove** from Curve. Select **Next**.
- 44 Select the **Third Groove** Curve.
- 45 On the New Feature – Dimension menu make sure you select **X-axis**.
- 46 Select **Finish**. Please select the **strategy** page.



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



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



Face Boring

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

Offset Boring

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

Boring below Centreline

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

Boring with Canned Cycles

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

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

Tool Nose Radius Compensation

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

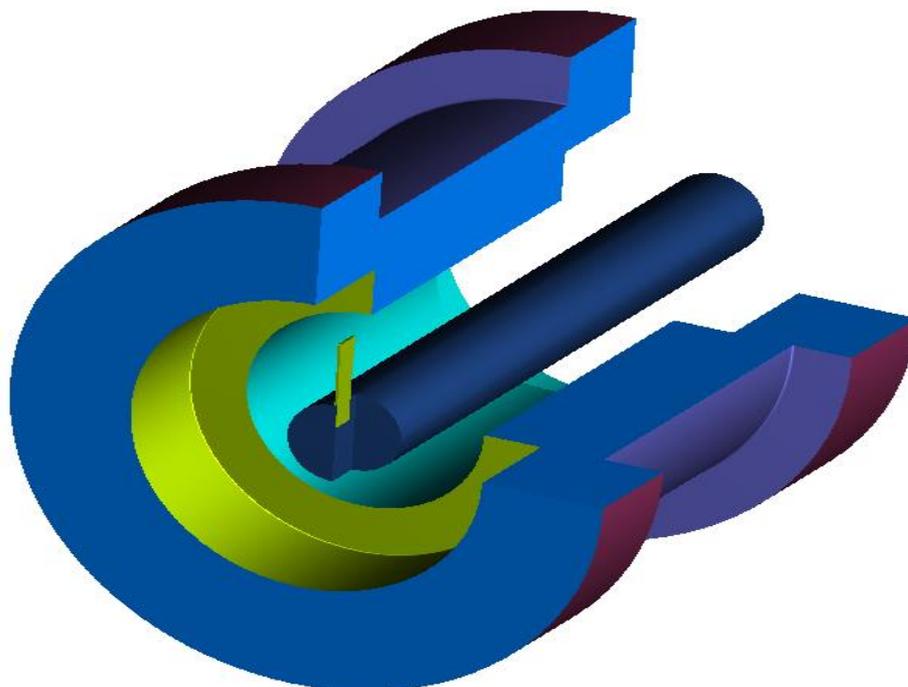
Back Face Boring

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



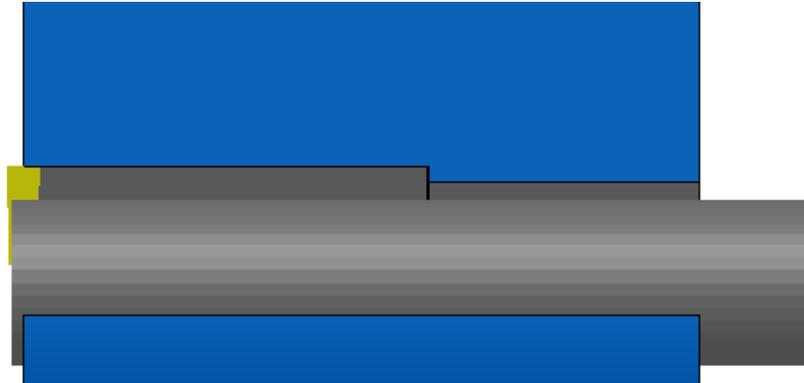
Back Boring

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

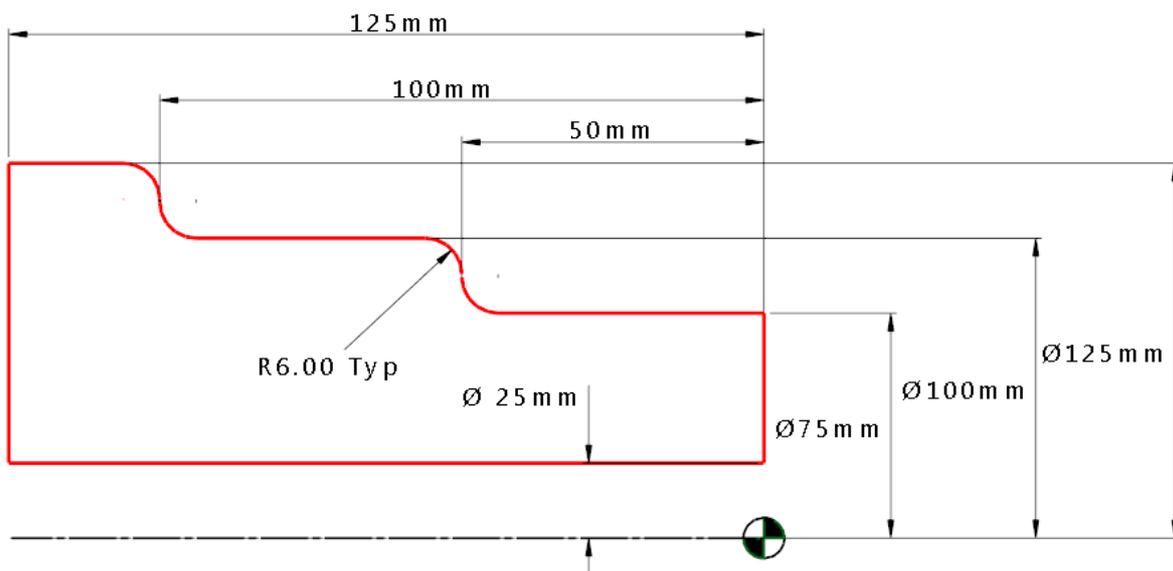


Boring with a groove tool

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



Turning Exercise 3



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

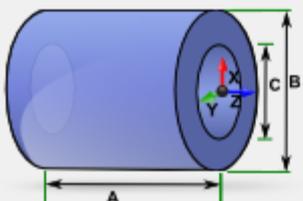
Dimensions

What shape is the stock?

Block
 Round
 N-sided

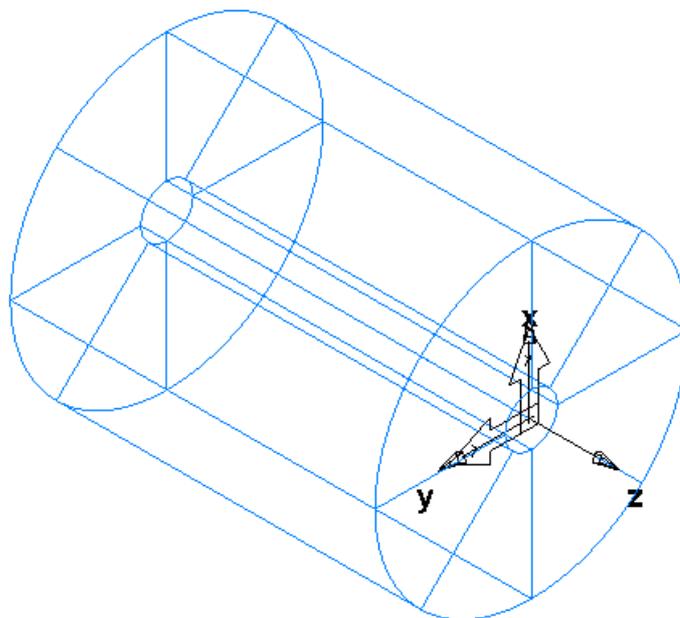
Axis:
 X
 Y
 Z

Enter the dimensions of the stock:



Length (A)
 OD (B)
 ID (C)

- 3 Select **Next**.
- 4 Select **Stainless-304L Material**. Select **Next**.
- 5 No Multi-Axis. Select **Next twice**.
- 6 Setup, Use current location. Select Next.
- 7 Main Spindle. Select **Next**.
- 8 Select **Finish**.



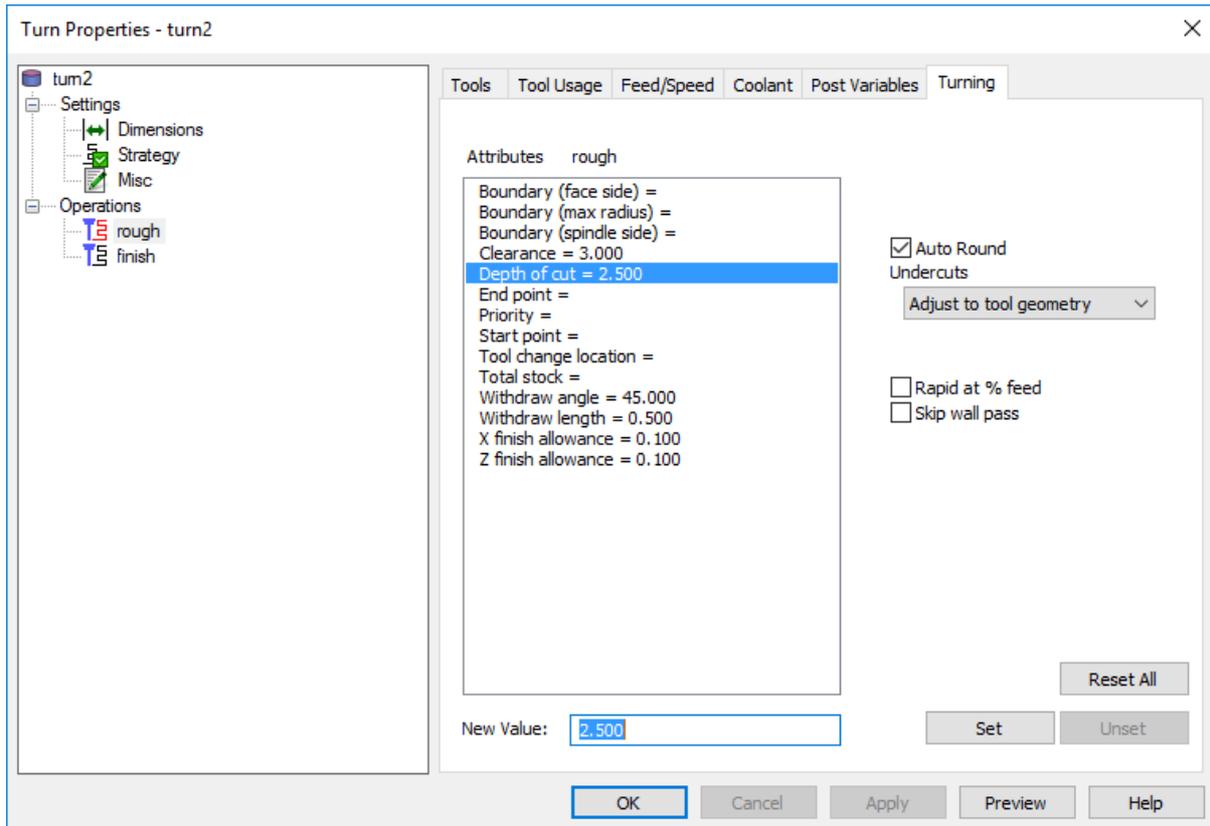
Exercise 3 in detail

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



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

- 10 Select **New Feature** or **CTRL + R** and select **Face**. Select **Finish**.
- 11 Select **New Feature** or **CTRL + R** and select **Turn from curve**.
- 12 Select **Next**, Select the **curve** then select **Finish**.



Turning Properties – Rough - Turning

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

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

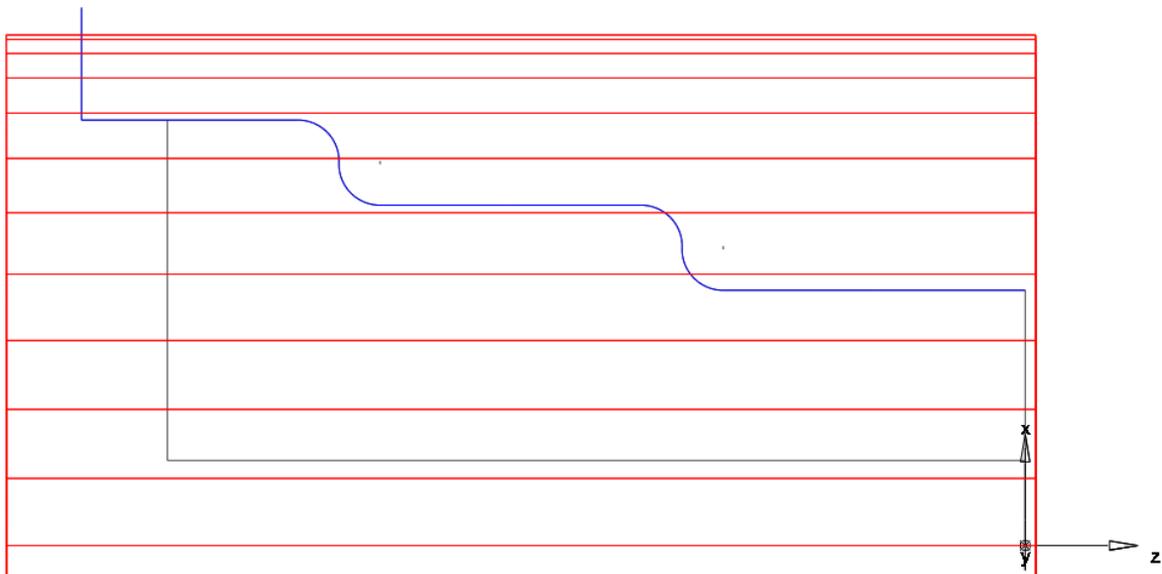
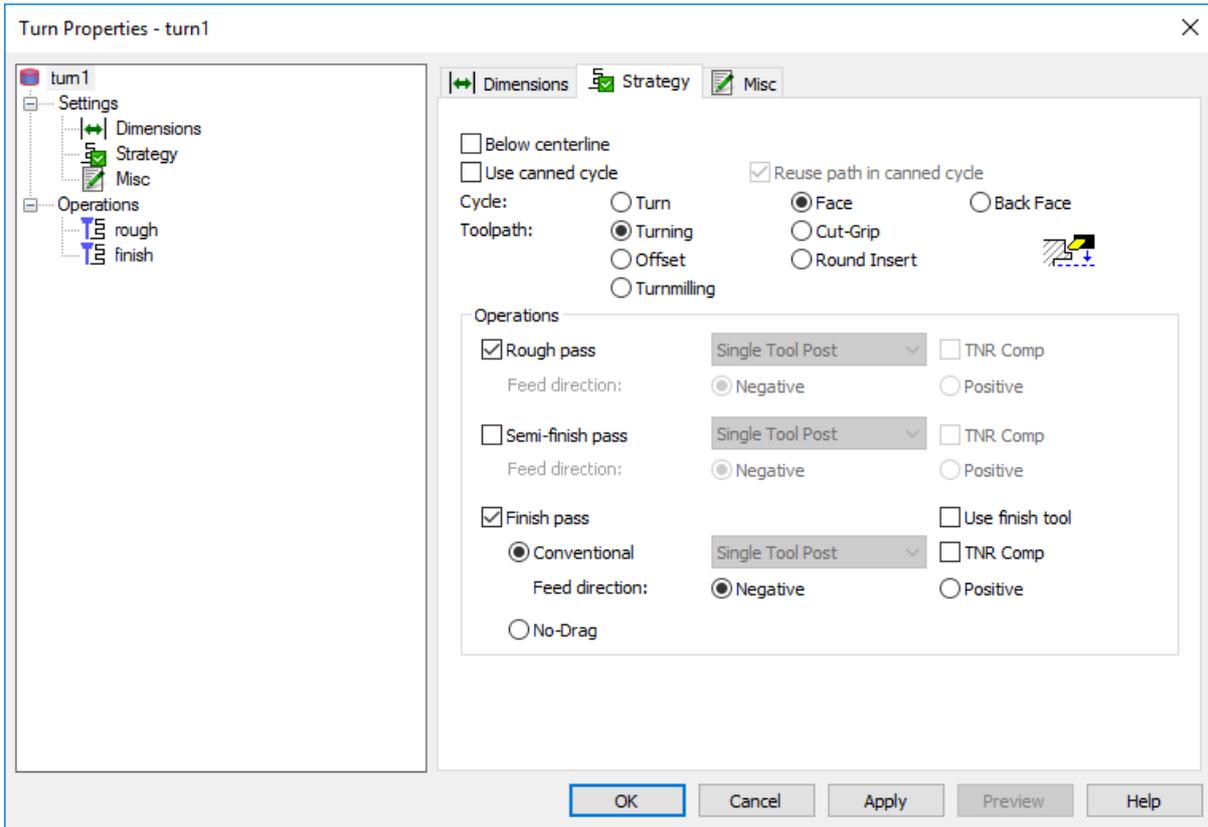
Face Turning

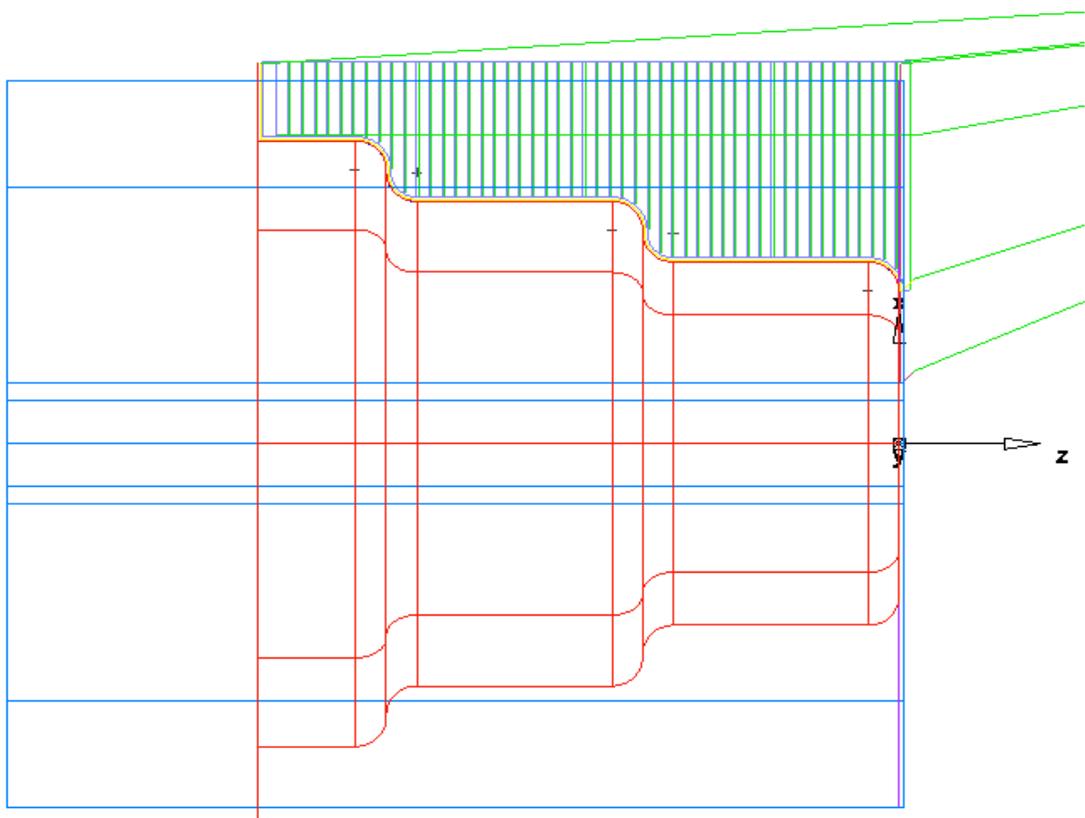
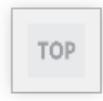


Face turning roughs the part in facing moves starting from the outside and cuts toward the curve in the negative X axis direction instead of turning parallel to the Z axis. If the last face does not go to the stock boundary in X you may need to use the Max Radius Boundary set to the stock diameter dimension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



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





Turn Properties - turn2 ✕

Dimensions
Strategy
Misc

tum2

- Settings
 - Dimensions
 - Strategy
 - Misc
- Operations
 - rough
 - finish

Below centerline
 Use canned cycle
 Reuse path in canned cycle

Cycle: Turn Face Back Face
 Toolpath: Turning Cut-Grip Round Insert
 Offset Turnmilling

Operations

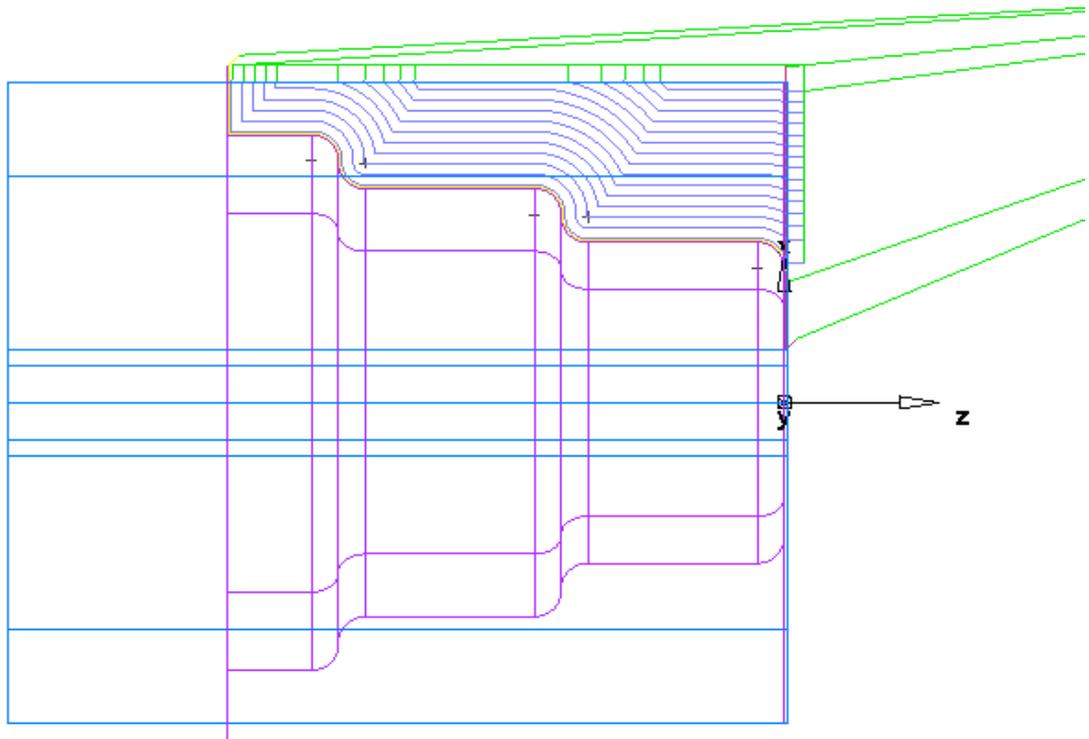
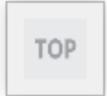
Rough pass Single Tool Post TNR Comp
 Feed direction: Toward spindle Toward face

Semi-finish pass Single Tool Post TNR Comp
 Feed direction: Toward spindle Toward face

Finish pass Single Tool Post Use finish tool
 Conventional TNR Comp
 Feed direction: Toward spindle Toward face
 No-Drag



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



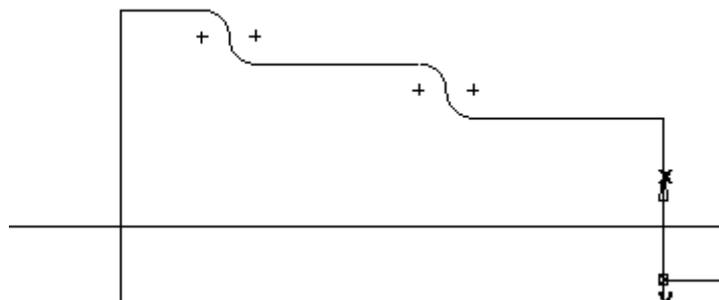
Stock Curve



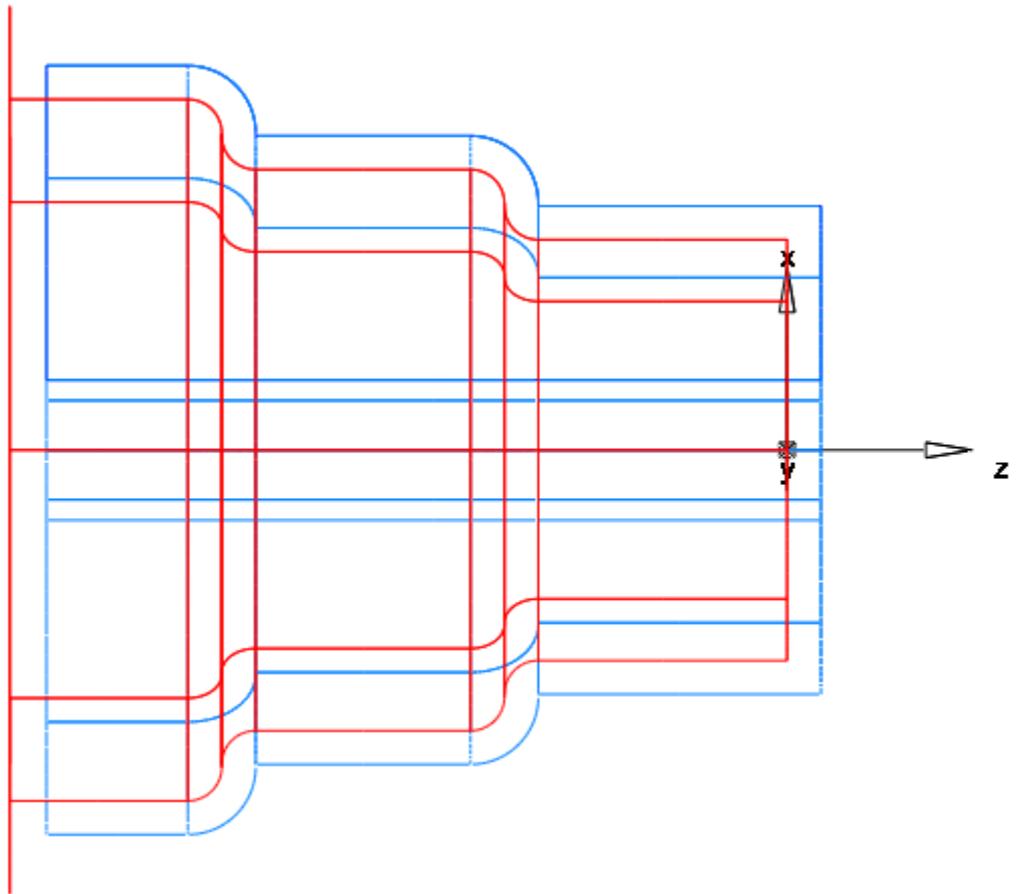
A stock curve can be used to represent a specific shape of the stock rather than bar stock. An example is when a forging or casting is used for the stock. The shape of the material is near the finished part shape with enough extra material to be turned down to the finish part size and shape.



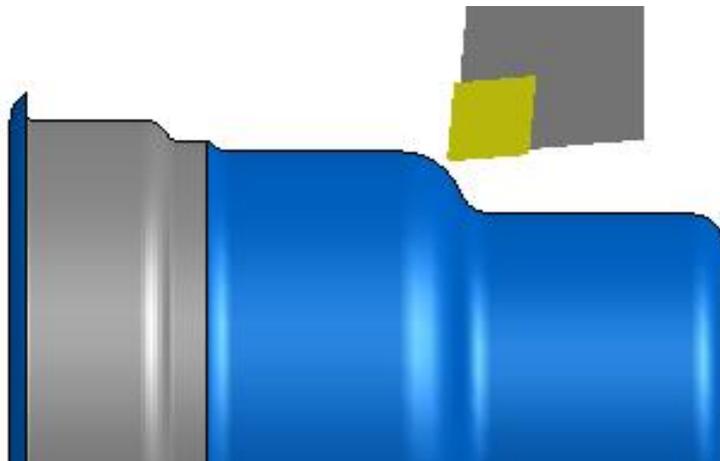
To draw a stock shape and then chain a curve, hide all features, curves and stock. Draw a horizontal line at 25mm diameter



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



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



Turning Setups Explained



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

How do setups relate to UCSs?

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



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



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

Setups dialog



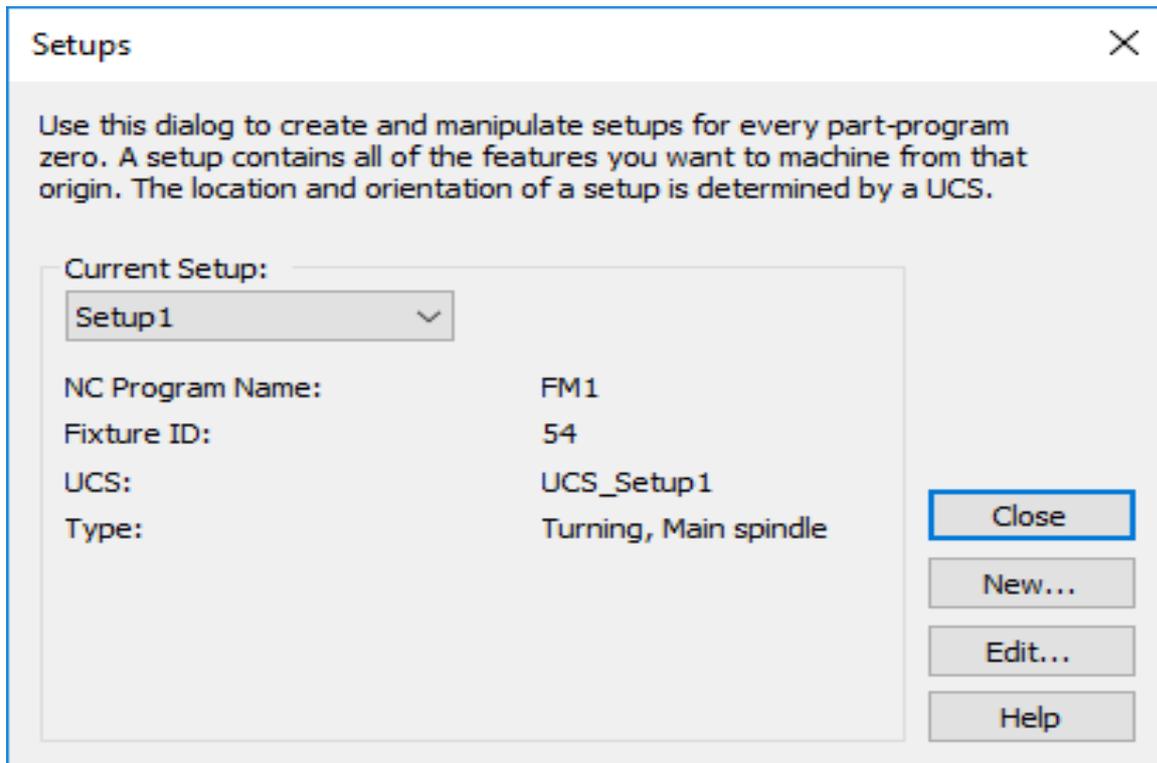
*You use the **Setups** dialog to set the active setup, create new setups and edit existing setups.*



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

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

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



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



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



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



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

XZ Millimeter Layer 1 UCS_Setup1 Setup1 Naka WT-300 Fanuc18i w Sub & Y.CNC basicmetric 2 cores

Setup – Definition

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

Setup - Definition

Please enter the setup name and fixture ID:

Setup name: Setup1

Fixture ID: 54

Part name: FM1

Setup type: Turning, Main spindle

UCS: UCS_Setup1

< Back Next > Finish Cancel Help

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

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



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

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



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

A few quick notes about Part Name



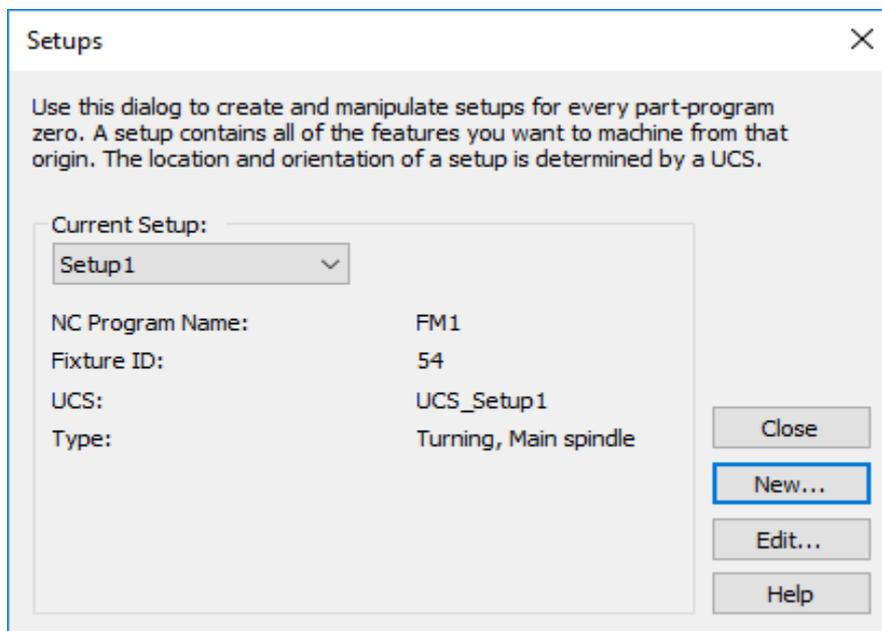
The Part Name are inserted in and must be what is recognized in the NC code **(00001)**. **The fixture ID** comes from the post processor and the call is applied to the NC code, such as **G-54**. Also, the **Part Name** is used to identify the program text file when you "**Save NC**" (Use Setup Part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

Creating a new Setup2



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

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



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

Setup - Definition

Please enter the setup name and fixture ID:

Setup name:

Fixture ID:

Part name:

What type of setup do you want to create?

Milling
 Turning
 Wire EDM



< Back Next >  Finish Cancel Help

Setup - Part Program Zero

What method do you want to use to define the part program zero location for Setup1?

Align to Stock Face
 Align to Index axis
 Align with existing UCS
 Align to part geometry
 Use current location

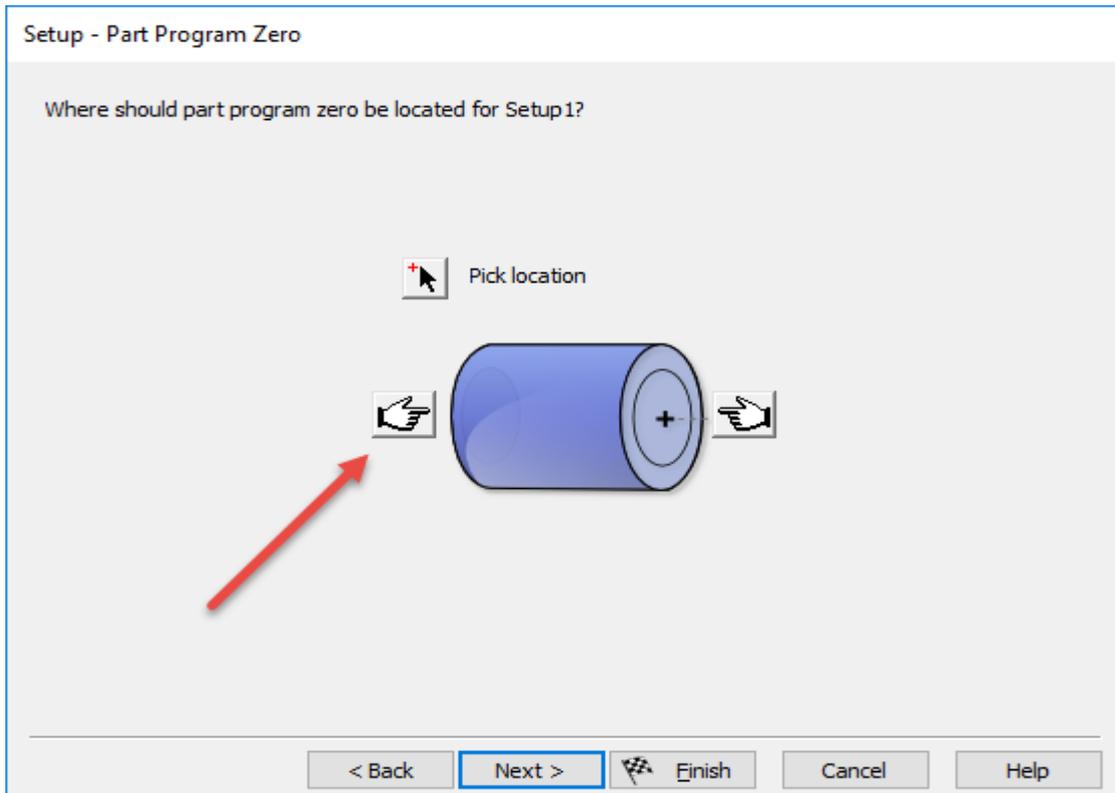


< Back Next >  Finish Cancel Help

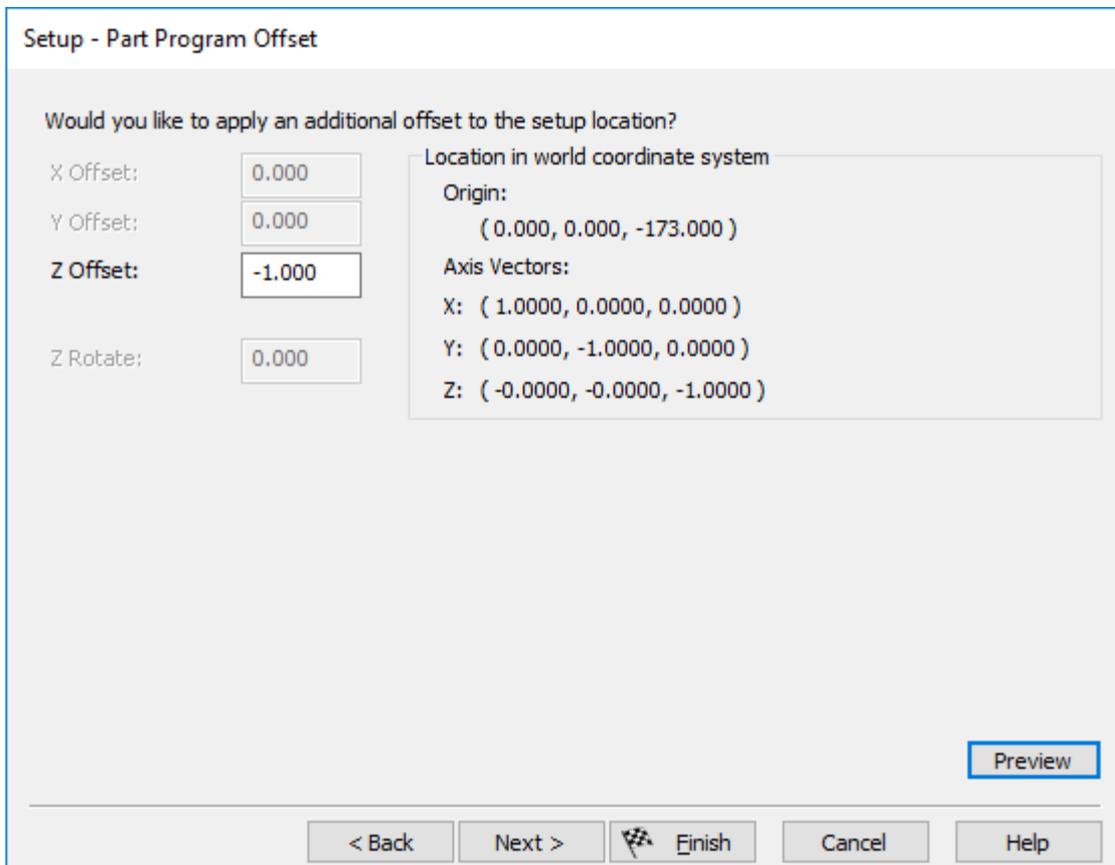
- 3 Choose **Align to Stock Face**
- 4 Select **Next**

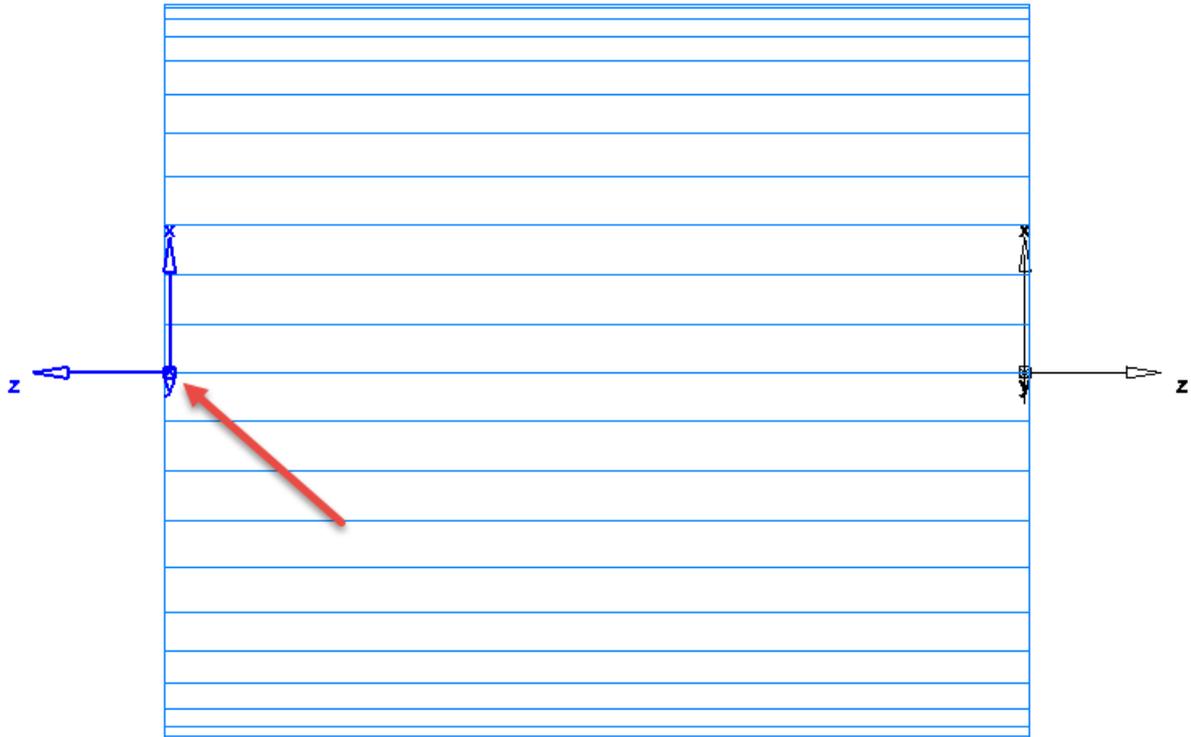


Choose the Left hand Icon as shown on the next image.

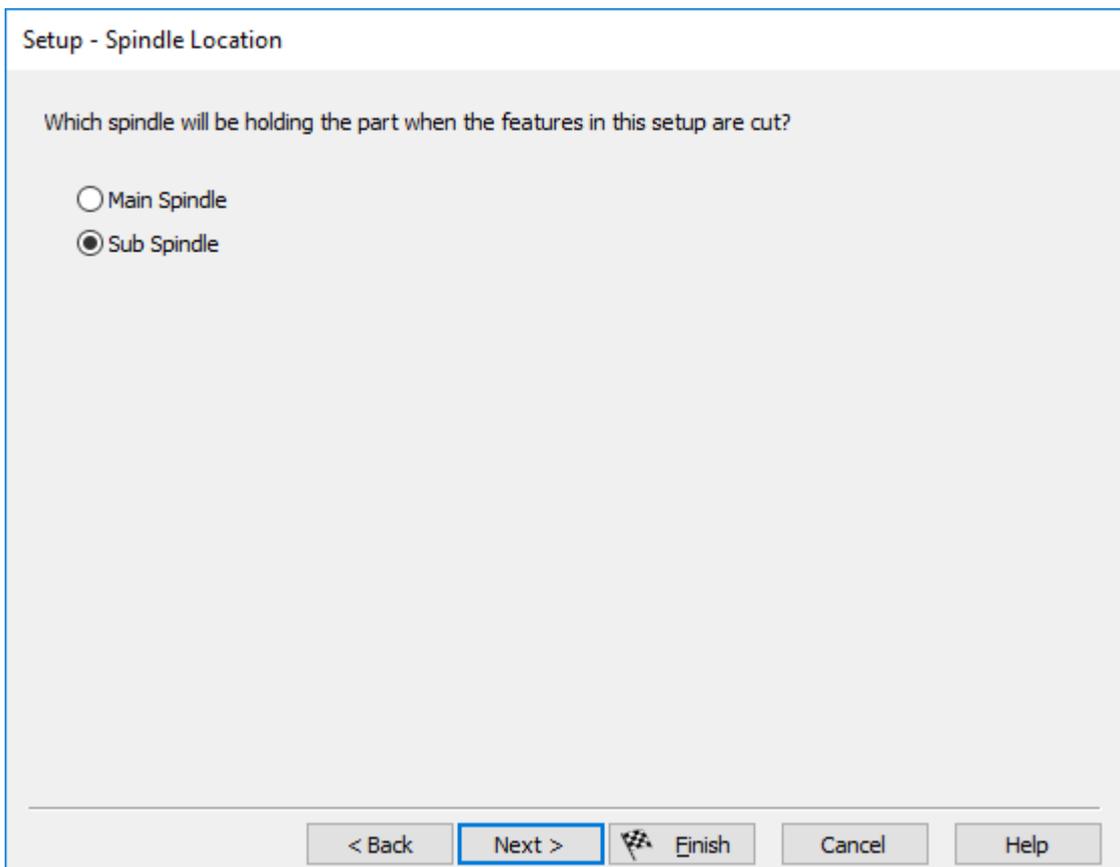


5 Select **Next**. I have entered -1 to move the **Setup2** into the work piece.





6 Select **Next**.



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

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

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

Specifying the machine simulation file

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

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



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

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

Setting up a part for multi-spindle turning

Setups

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



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

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



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

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

Associating features with setups

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

Moving the part

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

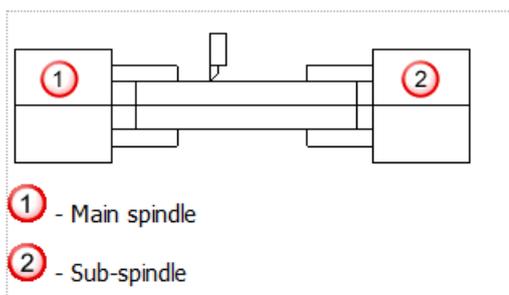
Machine Simulation

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

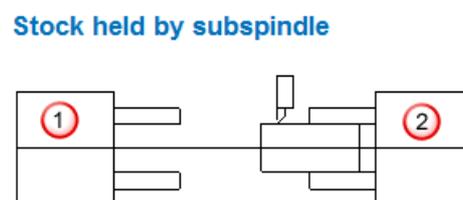
Sub spindle overview

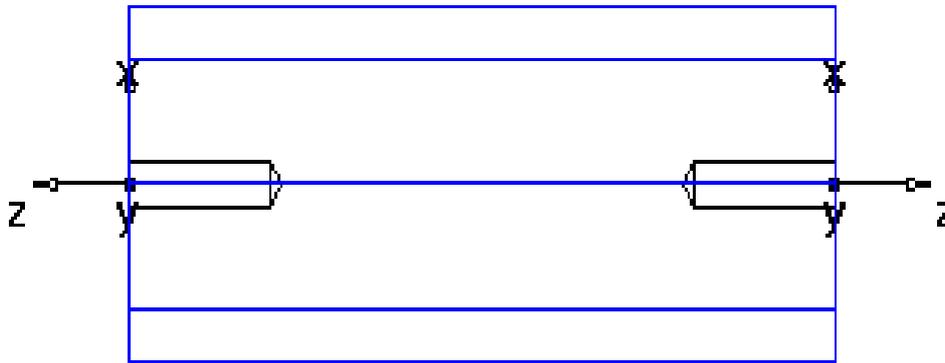
- Sub spindles can be used to support the part from both ends of the stock, or to change which spindle is used to hold the stock. To cut features on two different ends of your stock, you must first create setups at each end of your part with the Z directions pointing out from the stock, as shown below.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

Stock held by main spindle



Stock held by sub spindle





Sub-Spindle

Main spindle

Creating a Sub spindle feature

New Feature

What kind of feature would you like to make?

From Dimensions

Hole

Groove

Thread

Face

Cutoff

Bar Feed

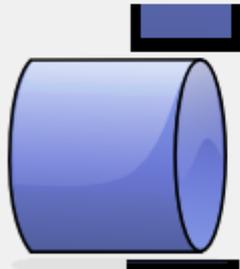
From Curve

Turn

Bore

Groove

Thread



Part Handling

User

Sub-spindle

Toolpath

Misc.

< Back **Next >** Finish ▾ Cancel Help

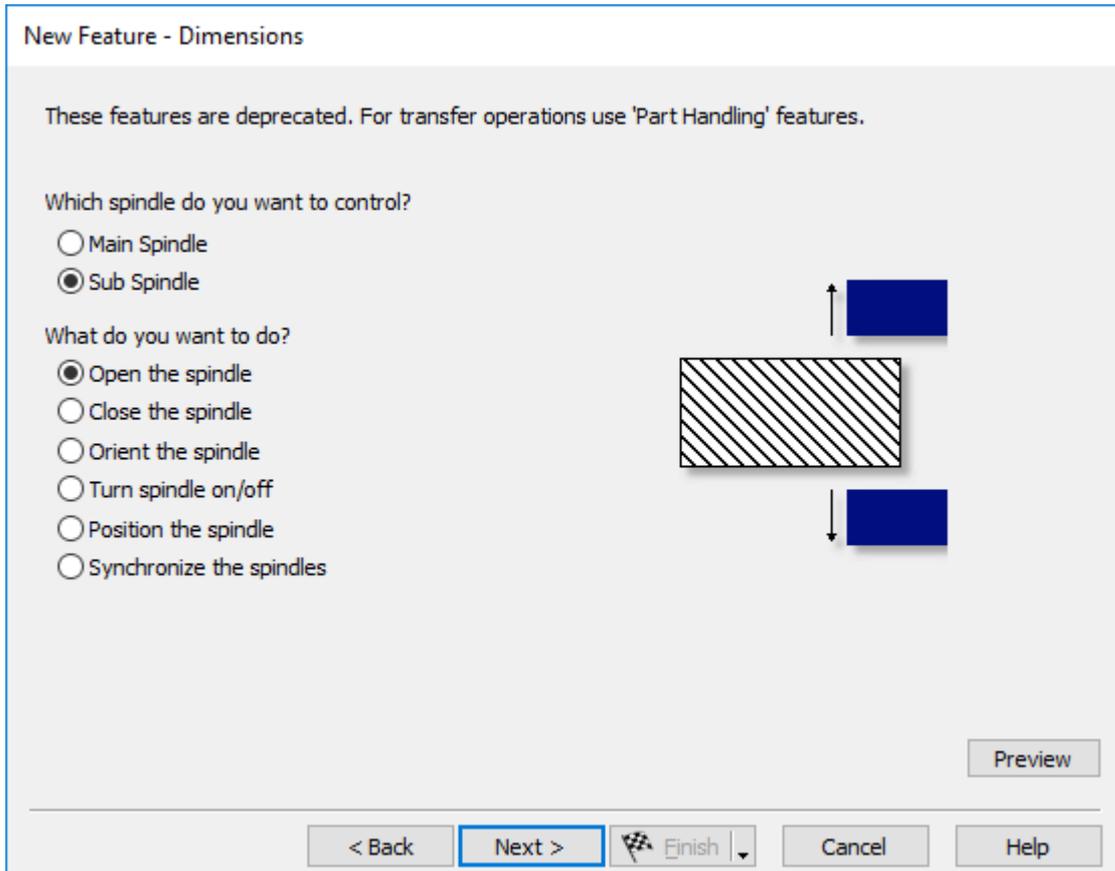
- To create a sub spindle feature: Select **Features** in the **Steps** panel.



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

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

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



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

New Feature - Strategies

Operations

Extend part catcher before opening spindle

Dwell after opening spindle:

< Back Next >  Finish Cancel Help

- Click **Finish**.

Setting up a part for multi-turret turning



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



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



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

New Feature

What kind of feature would you like to make?

From Dimensions

- Hole
- Groove
- Thread
- Face
- Cutoff
- Bar Feed

From Curve

- Turn
- Bore
- Groove
- Thread

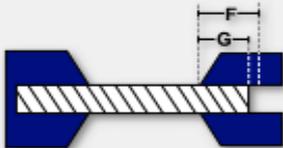
- Part Handling
- User
- Sub-spindle
- Toolpath
- Misc.

< Back
Next >
Finish v
Cancel
Help

New Feature - Dimensions

What do you want to do?

- Slug Transfer
- Reverse Slug Transfer
- Bar Pull
- Part Support On
- Part Support Off
- Cutoff



Grab distance (G)
 Feed distance (F)

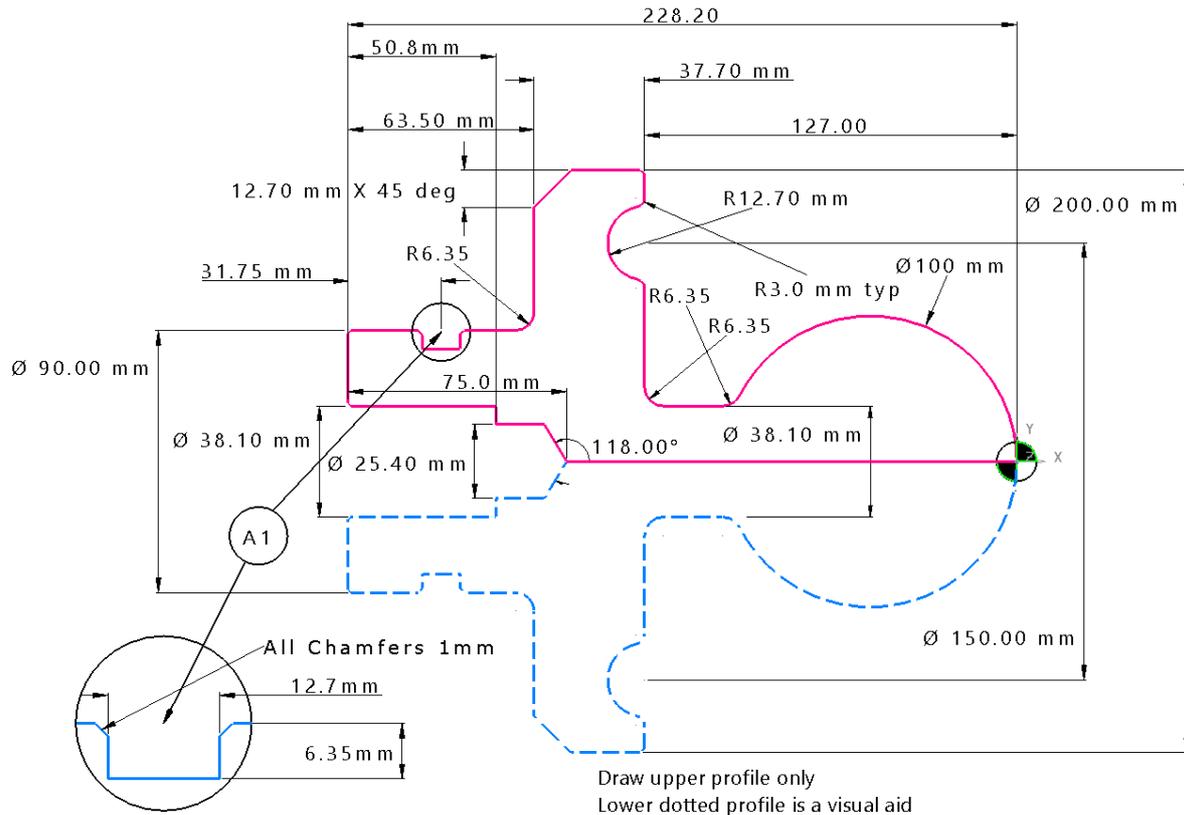
Preview

< Back
Next >
Finish v
Cancel
Help

Turning Exercise 4 Lightbulb (Class exercise)



Work through this Exercise on your own.



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



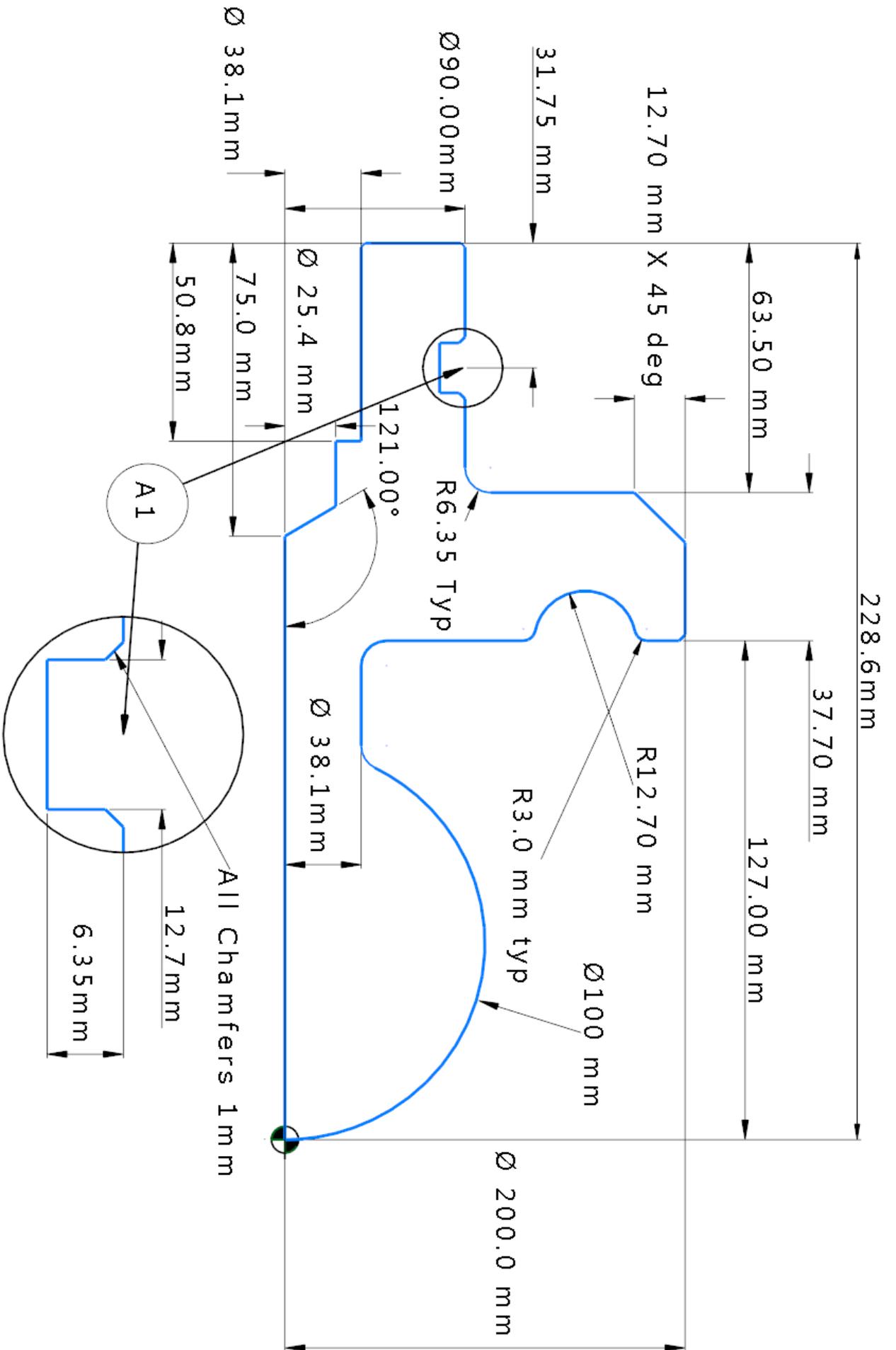
Remember to create a Curve we use pick pieces. All Curves for Turning are open Curves.

Lightbulb drawing exercise in detail



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

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



Round Insert Strategy

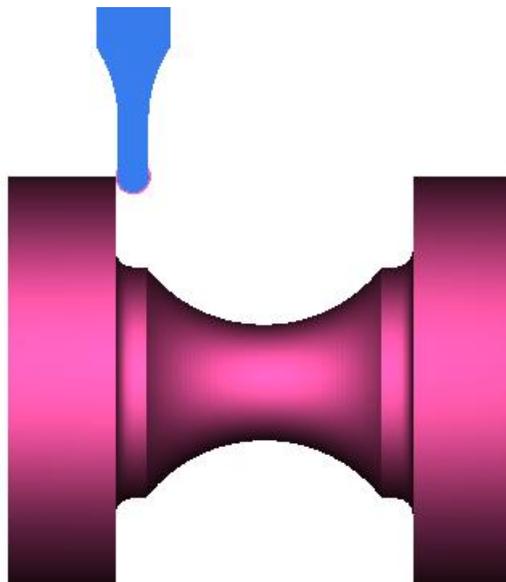
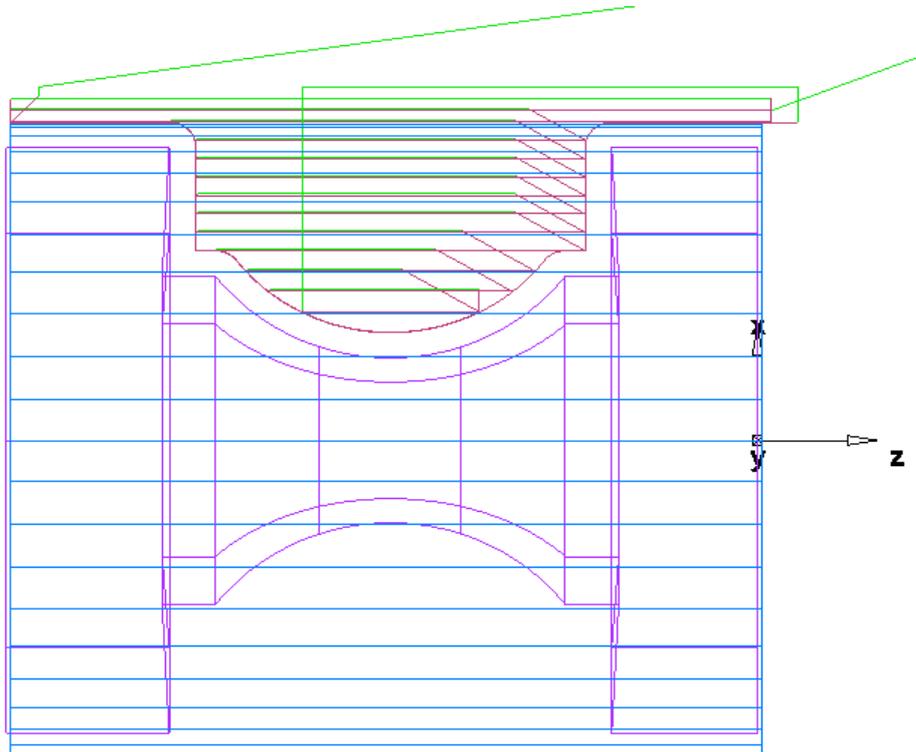
Round Insert— Round tool roughing toolpaths are designed to ease the tool more gently into a groove shape using a Button Style insert.

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

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



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



This Page Intentionally Left Blank

Contents

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

Basic Turning Attributes / Configurations

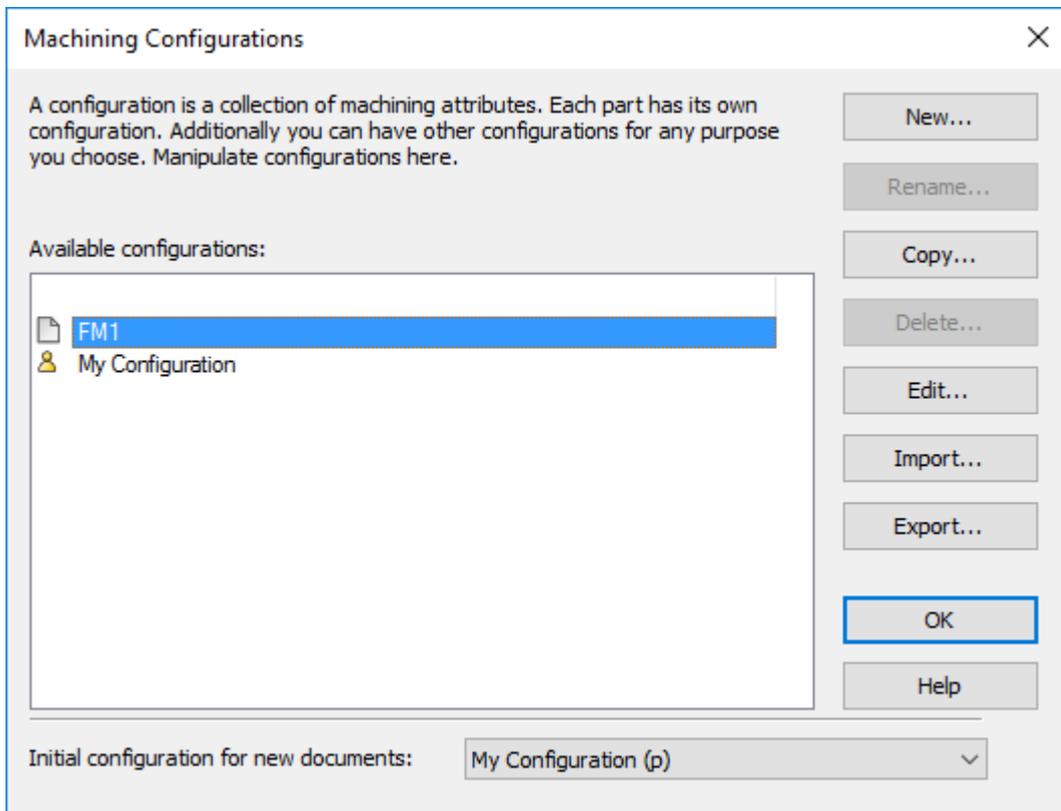
Introduction

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

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

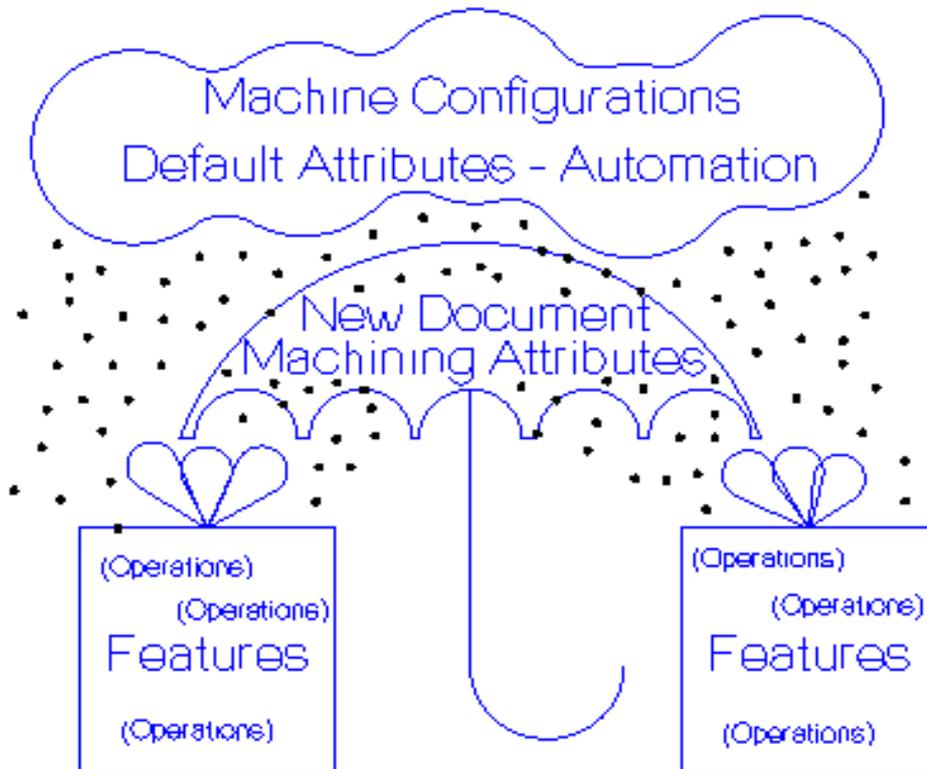
Machining Configurations

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



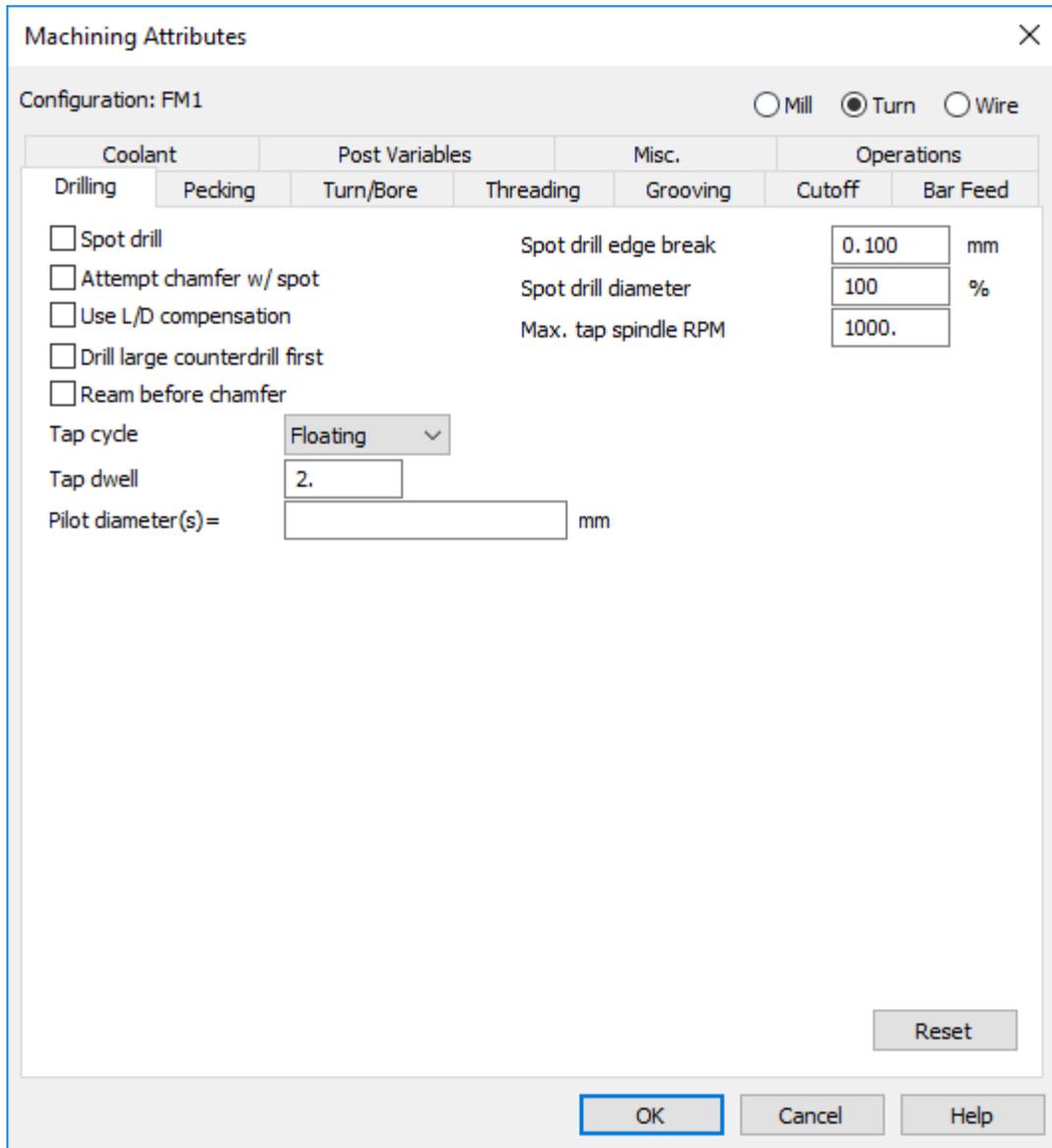
- "Initial Configuration for new documents", is in the drop down menu at the bottom.

- 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.

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.

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.

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:

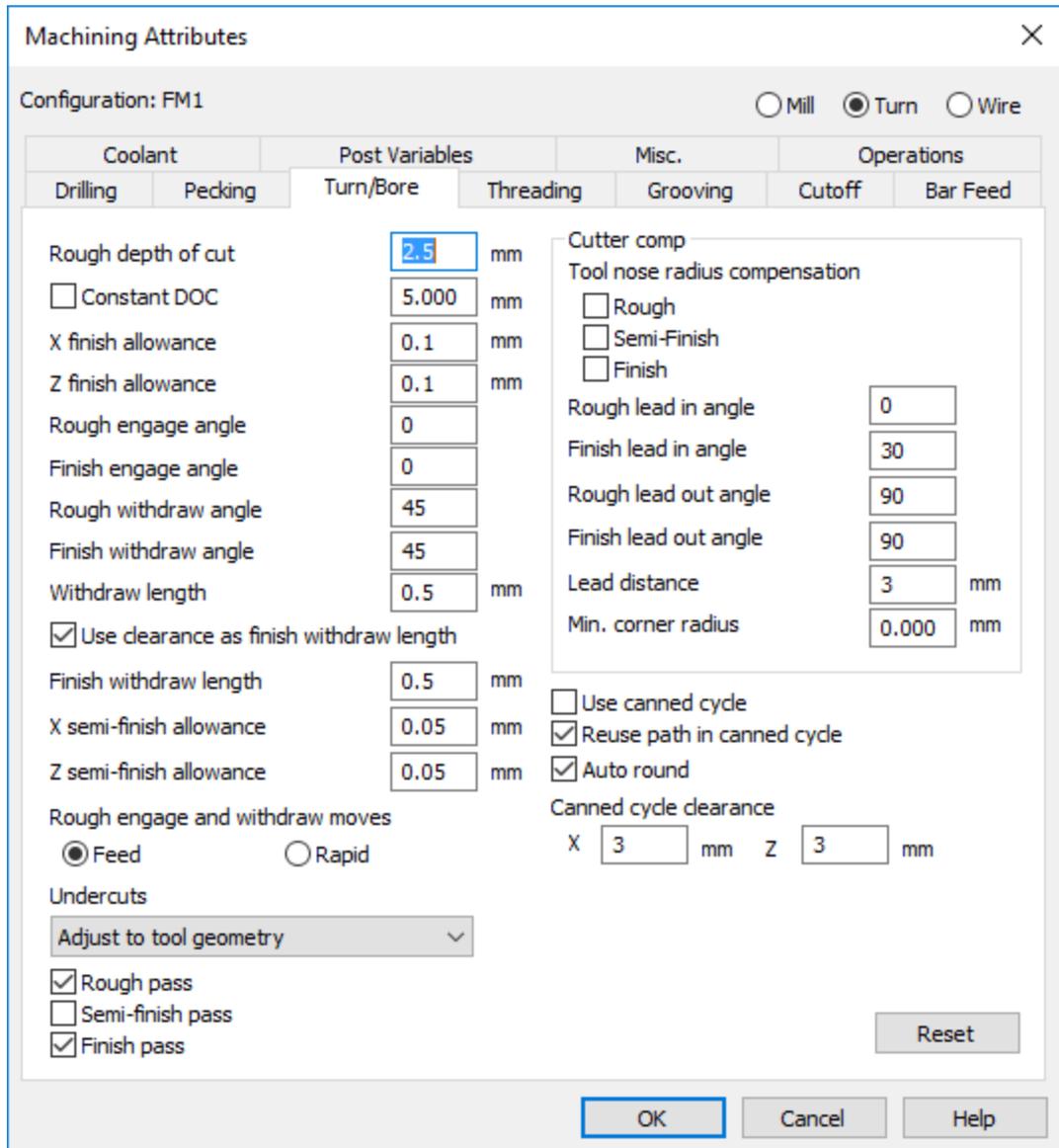
- **First peck:** - This is the depth of the first peck of a drilling/tapping operation specified as a percentage of tool diameters. If the depth of the hole is less than First peck, the hole is drilled in a single peck.
- **Second peck:** - This is the depth of the second peck of a drilling/tapping 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.

Machining Attributes

Configuration: FM1 Mill Turn Wire

Coolant	Post Variables	Misc.	Operations
Drilling	Pecking	Turn/Bore	Threading
Drilling	Pecking	Grooving	Cutoff
Drilling	Pecking	Bar Feed	
Drilling			
First peck	<input type="text" value="100"/>	% of drill diameter	
Second peck	<input type="text" value="50"/>	% of drill diameter	
Minimum peck	<input type="text" value="25"/>	% of drill diameter	
Tapping			
First peck	<input type="text" value="100"/>	% of tap diameter	
Second peck	<input type="text" value="50"/>	% of tap diameter	
Minimum peck	<input type="text" value="25"/>	% of tap diameter	

Turn/Bore tab (Turn)



Machining Attributes Configuration: FM1 ○ Mill ● Turn ○ Wire

Coolant		Post Variables		Misc.		Operations	
Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed	
Rough depth of cut		<input type="text" value="2.5"/>	mm	Cutter comp			
<input type="checkbox"/> Constant DOC		<input type="text" value="5.000"/>	mm	Tool nose radius compensation			
X finish allowance		<input type="text" value="0.1"/>	mm	<input type="checkbox"/> Rough			
Z finish allowance		<input type="text" value="0.1"/>	mm	<input type="checkbox"/> Semi-Finish			
Rough engage angle		<input type="text" value="0"/>		<input type="checkbox"/> Finish			
Finish engage angle		<input type="text" value="0"/>		Rough lead in angle		<input type="text" value="0"/>	
Rough withdraw angle		<input type="text" value="45"/>		Finish lead in angle		<input type="text" value="30"/>	
Finish withdraw angle		<input type="text" value="45"/>		Rough lead out angle		<input type="text" value="90"/>	
Withdraw length		<input type="text" value="0.5"/>	mm	Finish lead out angle		<input type="text" value="90"/>	
<input checked="" type="checkbox"/> Use clearance as finish withdraw length				Lead distance		<input type="text" value="3"/>	mm
Finish withdraw length		<input type="text" value="0.5"/>	mm	Min. corner radius		<input type="text" value="0.000"/>	mm
X semi-finish allowance		<input type="text" value="0.05"/>	mm	<input type="checkbox"/> Use canned cycle			
Z semi-finish allowance		<input type="text" value="0.05"/>	mm	<input checked="" type="checkbox"/> Reuse path in canned cycle			
Rough engage and withdraw moves				<input checked="" type="checkbox"/> Auto round			
<input checked="" type="radio"/> Feed		<input type="radio"/> Rapid		Canned cycle clearance			
Undercuts				X		<input type="text" value="3"/>	mm
Adjust to tool geometry				Z		<input type="text" value="3"/>	mm
<input checked="" type="checkbox"/> Rough pass							
<input type="checkbox"/> Semi-finish pass							
<input checked="" type="checkbox"/> Finish pass							

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

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.

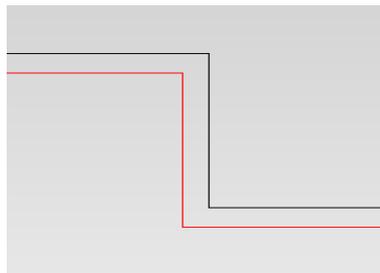
Use Clearance as finish with draw length: - When selected, the **Clearance** value is used as the approach and withdraw length for finishing moves. When deselected, the **Withdraw length** value is used.

Auto round

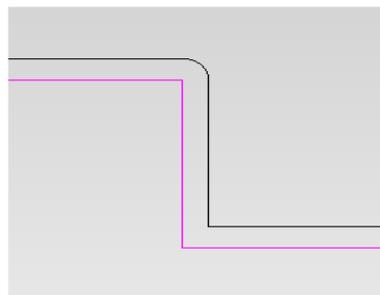
This turning attribute applies to both rough and finish passes. When **Auto round** is enabled, **FeatureCAM** automatically inserts arc moves to connect two non-tangent elements. The effects are:

- 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:



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.

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.

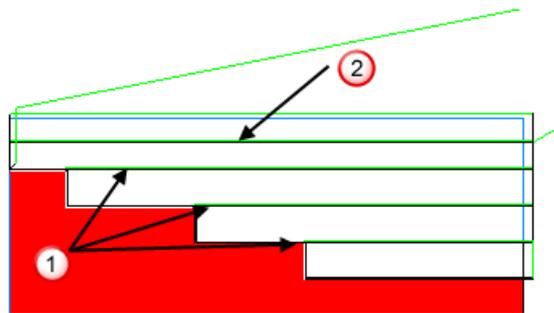
Canned cycle clearance X and Z: - These attributes control the tool location before the start of a turning canned cycle. The tool location is obtained by applying the X and Z clearances to the start point of the curve.

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.



① Clean up of previous steps

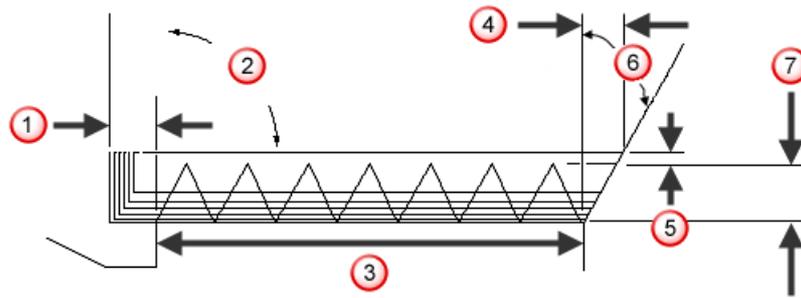
Threading tab (Turn)

Machining Attributes ✕

Configuration: FM1
 Mill
 Turn
 Wire

Coolant		Post Variables		Misc.	Operations	
Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed
<input type="checkbox"/> Rough turn			<input type="checkbox"/> Curve rough			
<input type="checkbox"/> Finish turn			<input checked="" type="checkbox"/> Curve finish			
<input checked="" type="checkbox"/> Relief groove			<input checked="" type="checkbox"/> Side wall angle			
<input type="checkbox"/> Chamfer			<input type="text" value="30"/>			
Chamfer angle	<input type="text" value="30"/>		Number of passes <input checked="" type="radio"/> Fixed Count: <input type="text" value="10"/> <input type="radio"/> Calculated Step1: <input type="text" value="0"/> mm Step2: <input type="text" value="0"/> mm Min infeed: <input type="text" value="0"/> mm Spring passes <input type="text" value="1"/>			
Chamfer additional depth	<input type="text" value="0.1"/> mm		Tapered Threads <input checked="" type="radio"/> Diameter is specified at front of taper <input type="radio"/> Diameter is specified at the end of taper			
Groove additional depth	<input type="text" value="0.1"/> mm		Thread depth calculations <input checked="" type="radio"/> Use tool tip radius <input type="radio"/> Use tool tip zero radius			
Start clearance	<input type="text" value="5"/> mm		<input type="button" value="Reset"/>			
End clearance	<input type="text" value="3"/> mm					
Infeed angle	<input type="text" value="29"/>					
OD Depth %	<input type="text" value="75.78"/>					
ID Depth %	<input type="text" value="64.95"/>					

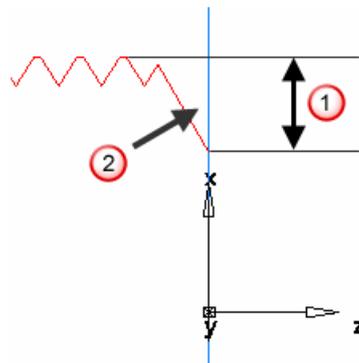
Some of the Threading attributes are shown in the diagram below:-



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

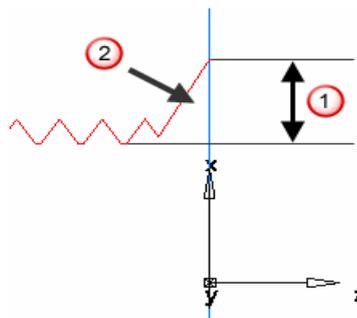
Rough turn and **Finish turn** — Select these options to automatically turn the piece down to the thread diameter. See How a thread feature is manufactured.

OD Thread Feature



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

ID Thread feature



① (Chamfer addl.) **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.

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

Grooving tab (Turn)

Machining Attributes ✕

Configuration: FM1 ○ Mill Turn ○ Wire

Coolant		Post Variables		Misc.	Operations	
Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed

Rough

Plunge center first

Cut Type:

Depth first

Width first

Feed direction:

Toward spindle / Negative X

Toward face / Positive X

Side liftoff distance mm

Side liftoff angle

Output dwell on rough

Dwell

Stepover %

Depth of cut mm

Chamfer extend distance mm

Peck retract distance mm

Finish

Feed direction:

Toward spindle / Negative X

Toward face / Positive X

Opposite from rough direction

Side liftoff distance mm

Dwell

Use 2nd offset register

Use simple canned cycle or computed cycle

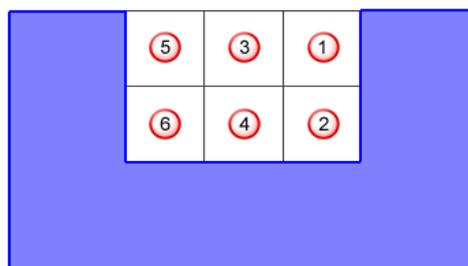
Use path canned cycle

Reuse path

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.



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.

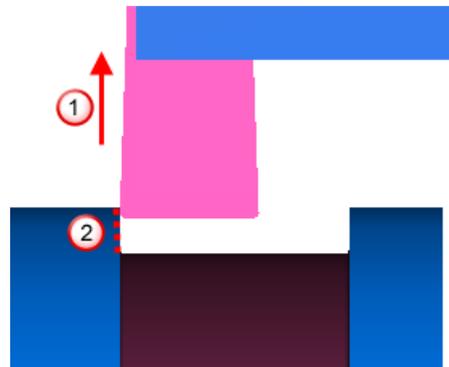
Feed dir: - This is the direction the tool feeds. The choices are either **Neg** (negative, -Z direction) or **Pos** (positive, +Z direction). For finish, you can also set this to **Opposite from rough dir**.

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**.

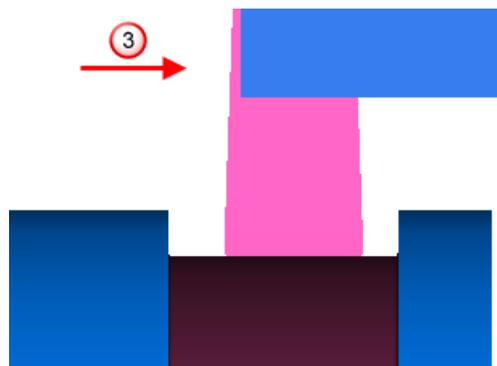
This part has a Groove feature, shown in pink:



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.



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.

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.

Machining Attributes

Configuration: FM1 Mill Turn Wire

Coolant		Post Variables		Misc.		Operations	
Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed	
Peck retract distance		1.25		mm			
Dwell		2					

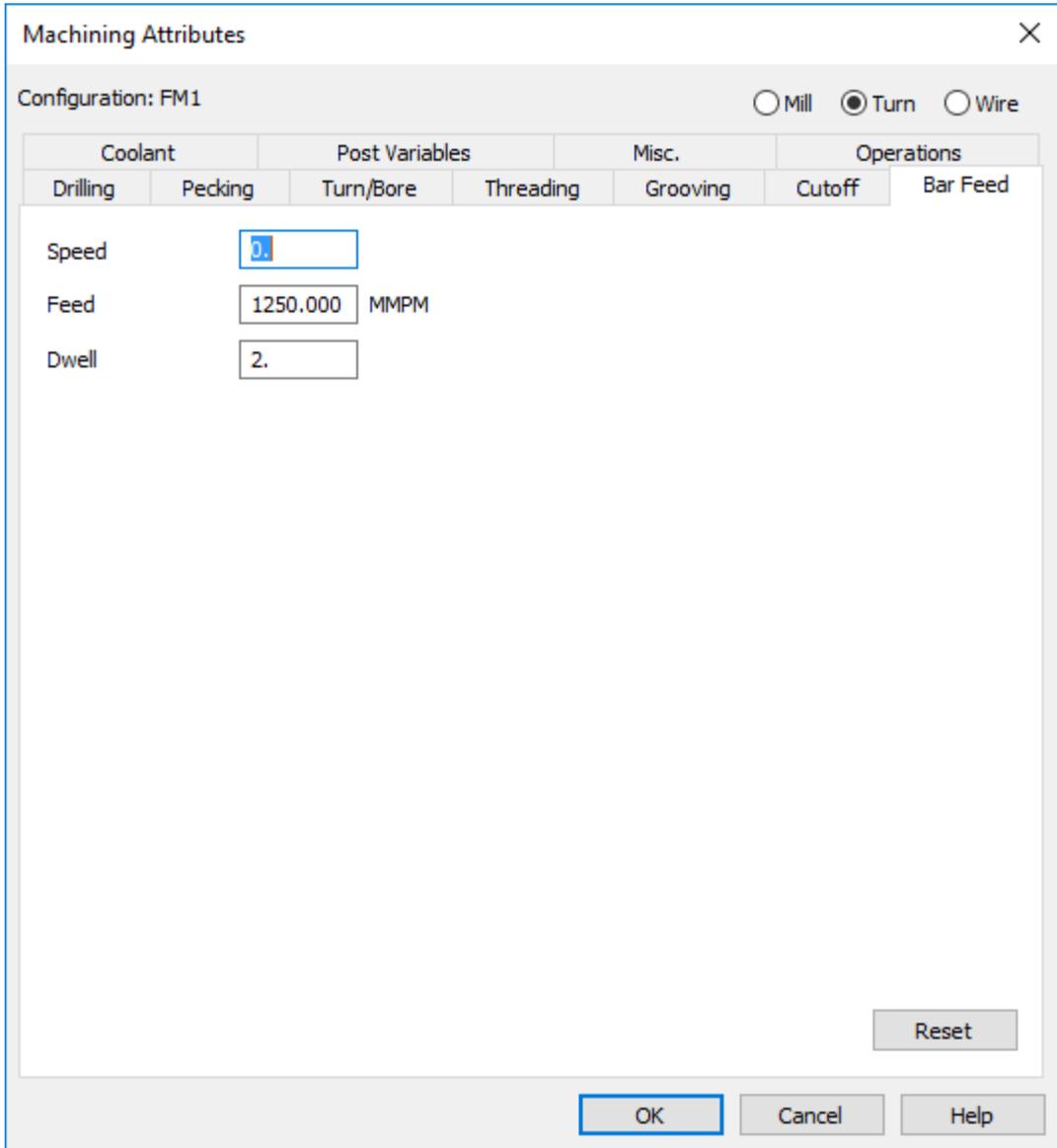
Reset

OK Cancel Help



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

Bar Feed tab (Turn)



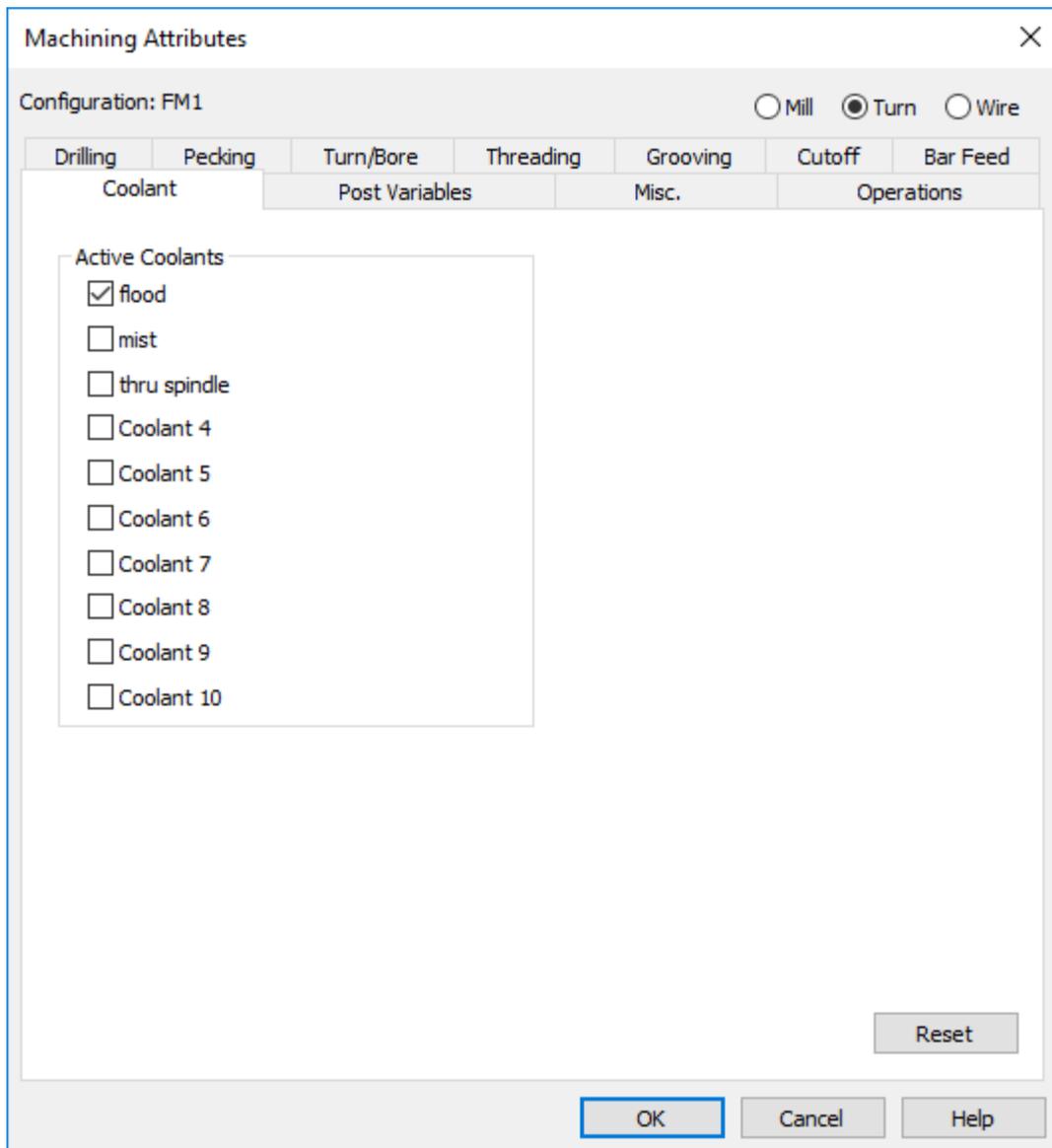
The screenshot shows the 'Machining Attributes' dialog box with the 'Bar Feed' tab selected. The configuration is set to 'FM1' and the operation type is 'Turn'. The 'Bar Feed' tab is active, showing input fields for Speed, Feed, and Dwell. The Speed field is empty, the Feed field contains '1250.000' with 'MMPM' as the unit, and the Dwell field contains '2.'. There are 'OK', 'Cancel', 'Help', and 'Reset' buttons at the bottom.

Coolant		Post Variables		Misc.		Operations	
Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed	
Speed	<input type="text" value=""/>						
Feed	<input type="text" value="1250.000"/>	MMPM					
Dwell	<input type="text" value="2."/>						



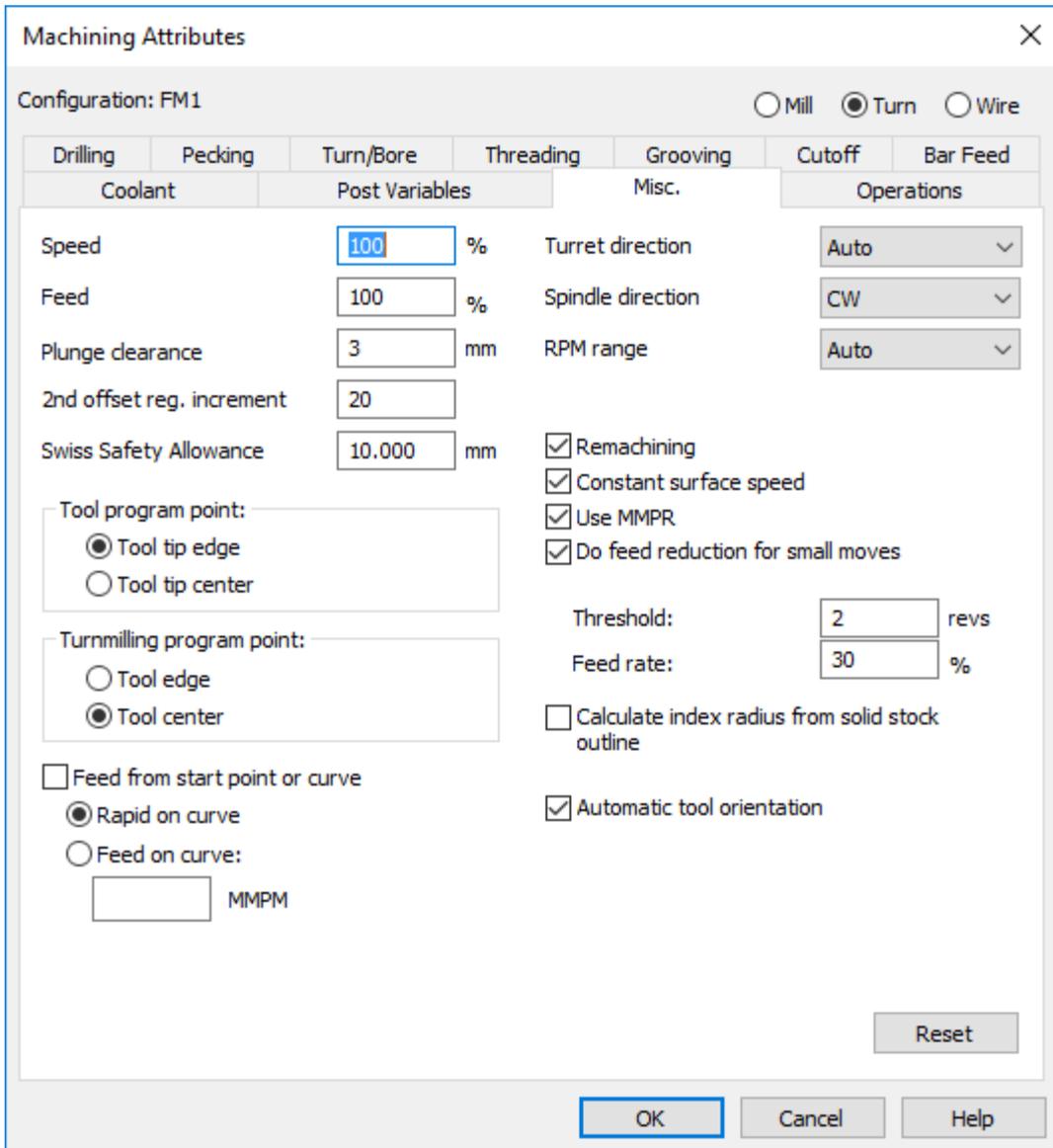
Dwell: - The number of seconds you want the tool to dwell after plunging during a groove roughing pass. It also applies to the roughing of the Cutoff chamfer.

Coolant



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

Misc. tab (Turn)



Machining Attributes

Configuration: FM1 Mill Turn Wire

Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed
Coolant		Post Variables		Misc.		Operations

Speed: %

Feed: %

Plunge clearance: mm

2nd offset reg. increment:

Swiss Safety Allowance: mm

Tool program point:

Tool tip edge
 Tool tip center

Turnmilling program point:

Tool edge
 Tool center

Feed from start point or curve
 Rapid on curve
 Feed on curve:
 MPPM

Turret direction:

Spindle direction:

RPM range:

Remachining
 Constant surface speed
 Use MPMR
 Do feed reduction for small moves

Threshold: revs

Feed rate: %

Calculate index radius from solid stock outline

Automatic tool orientation

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.

- **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:

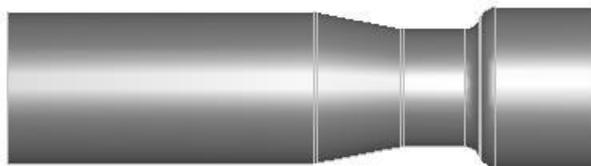
- **Rapid on curve:** - Select this option if you want to use a rapid move along the start curve.



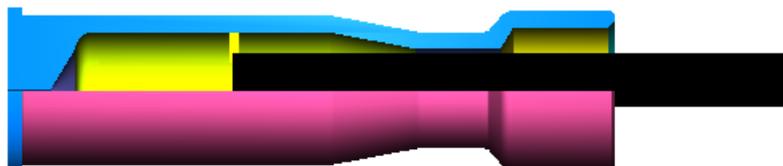
Selecting **Feed from start point or curve** and **Rapid on curve** is the equivalent of selecting the **Feed from start** option on the **Turning tab** in previous versions of FeatureCAM.

- **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.

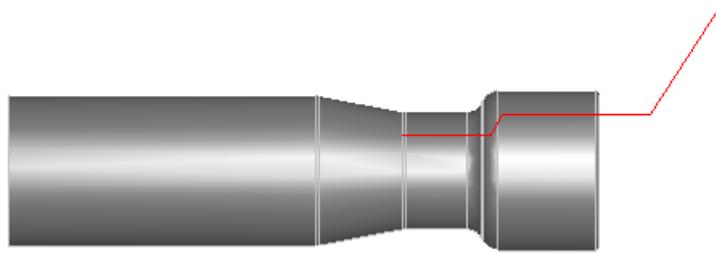
This example turned part has a Turn, Hole, and Groove feature:



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.



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 %**.

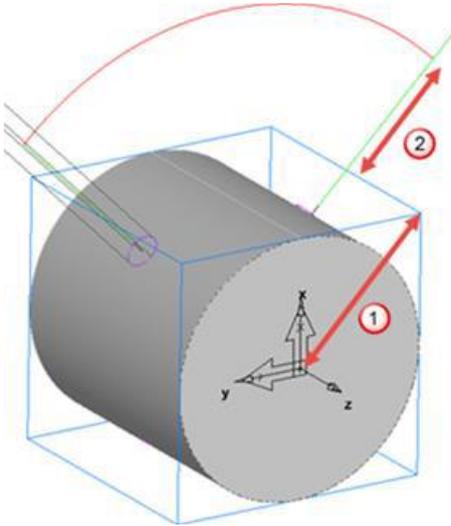
Automatic Tool Selection:- This option set the same tool in the appropriate direction for OD Turning and ID Turning, with respect to Main Spindle/Sub-Spindle.



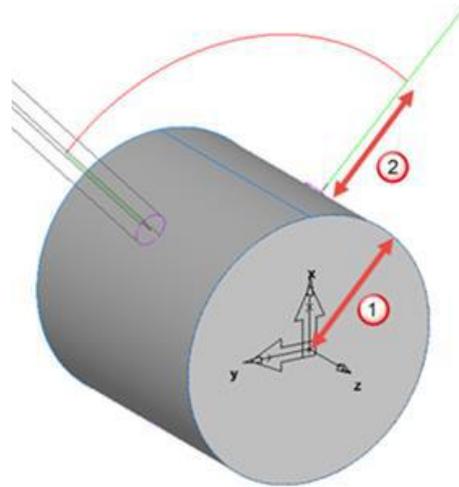
Calculate index radius from solid stock outline: - Select this option to determine the index height directly from the stock solid, instead of calculating it above a square bounding box.

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



- 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.

Operations

Machining Attributes ✕

Configuration: FM1 Mill Turn Wire

Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed
Coolant	Post Variables		Misc.		Operations	

Ordering

Automatic Options...

Base priority

Don't ask at tool path simulation

Time estimation

Rapid traverse: mmpm

Tool change: sec

Go to start: sec

X-Y acceleration: mmpm²

Z-acceleration: mmpm²

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).

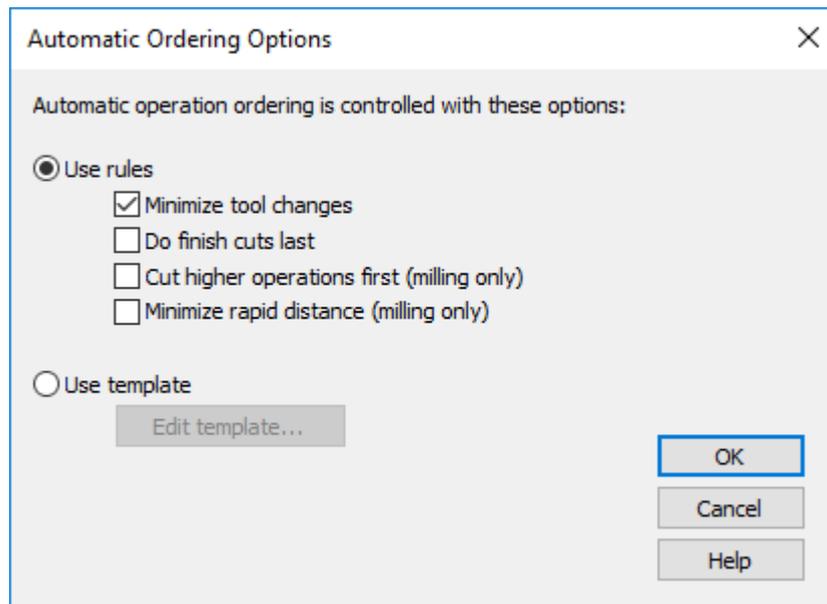
Go to start: - This is the time it takes for the tool head to move to the start location and the spindle or tool head to come to a stop.

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:**

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.

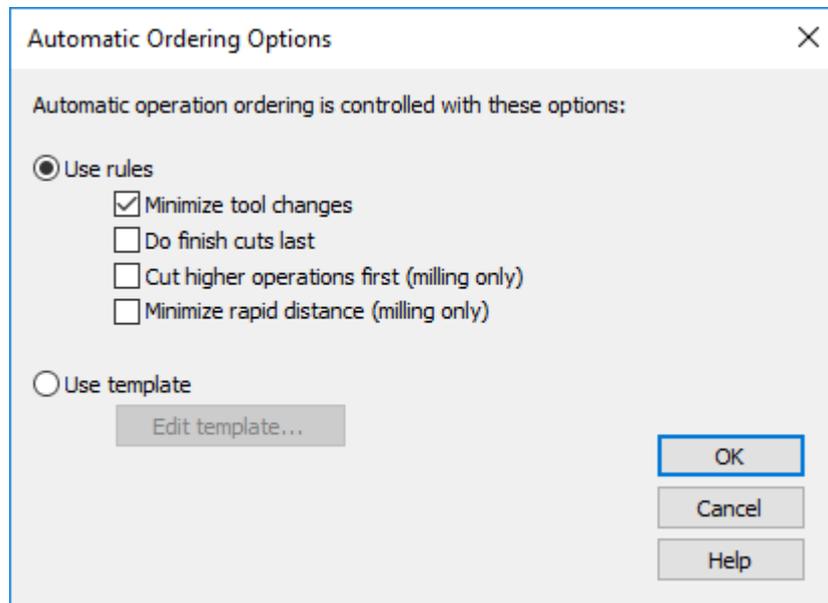
If you have selected all of the optimization options, the manufacturing order for operations that were derived from different features is determined like this:

The operations are sorted by their top Z coordinate.

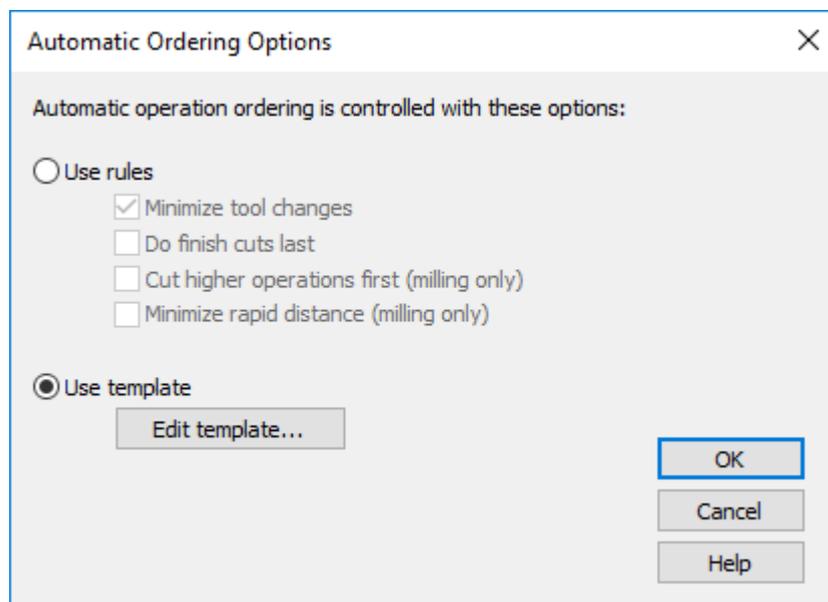
Among operations with the same top Z coordinate, operations are grouped by the tool with which they are cut.

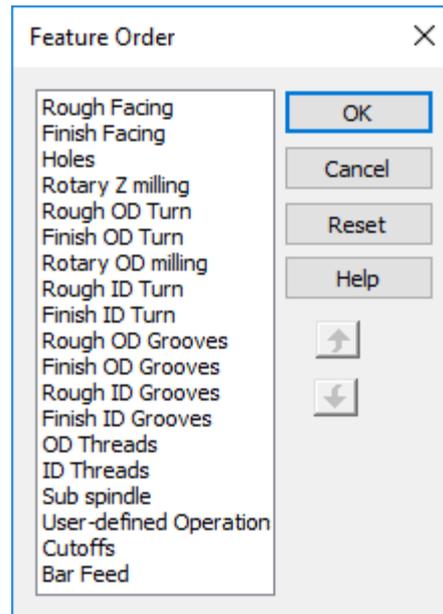
After an operation is cut, FeatureCAM moves to the next operation performed with the same tool that is the closest to the current operation.

Use Template

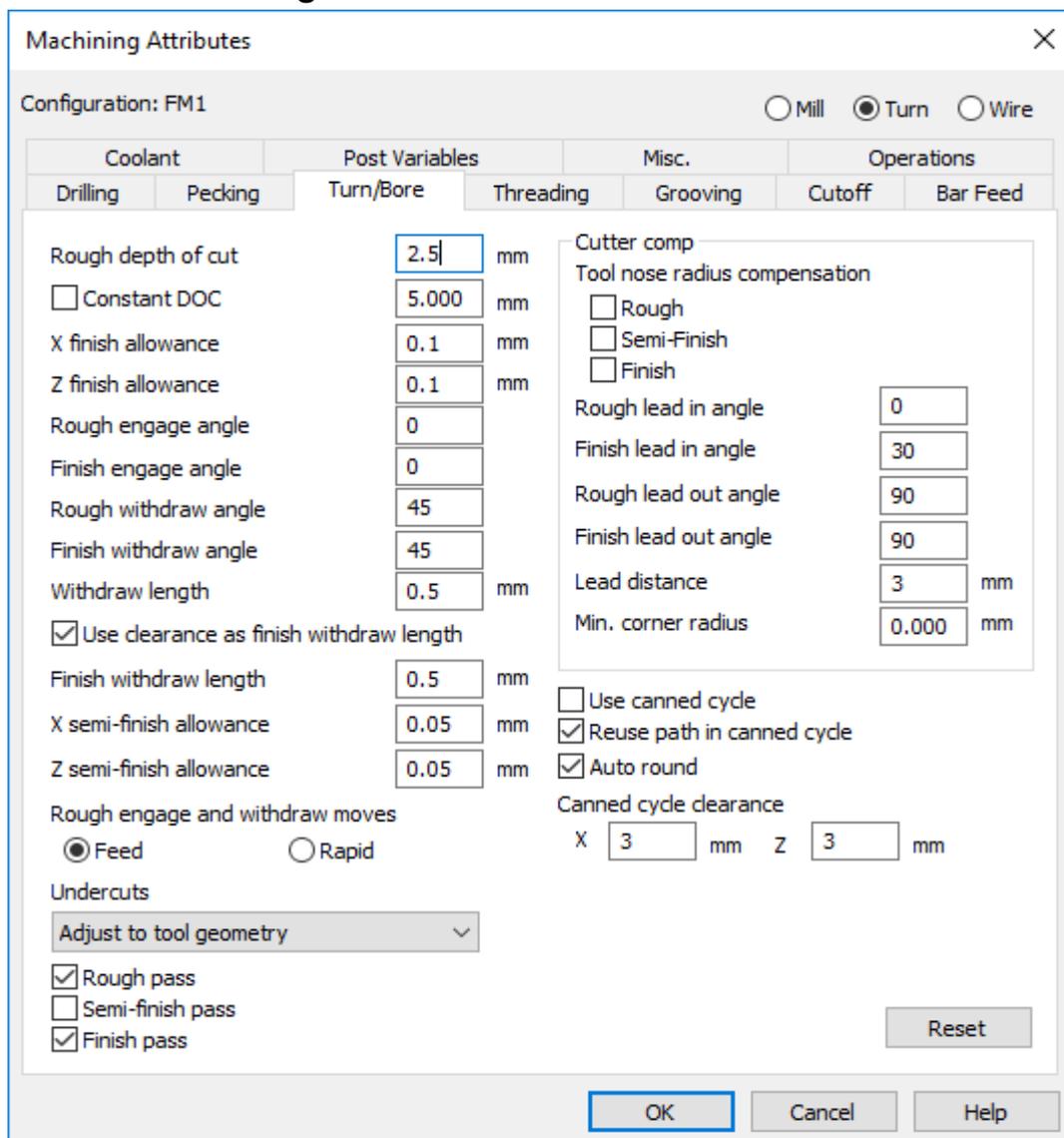


*This attribute is applicable only to turning setups. If you select **Use template** then the order of operations are determined by the outline of operations listed in the Feature Order dialog. Click **Edit template** to open the Feature Order dialog. See turn operation order.*

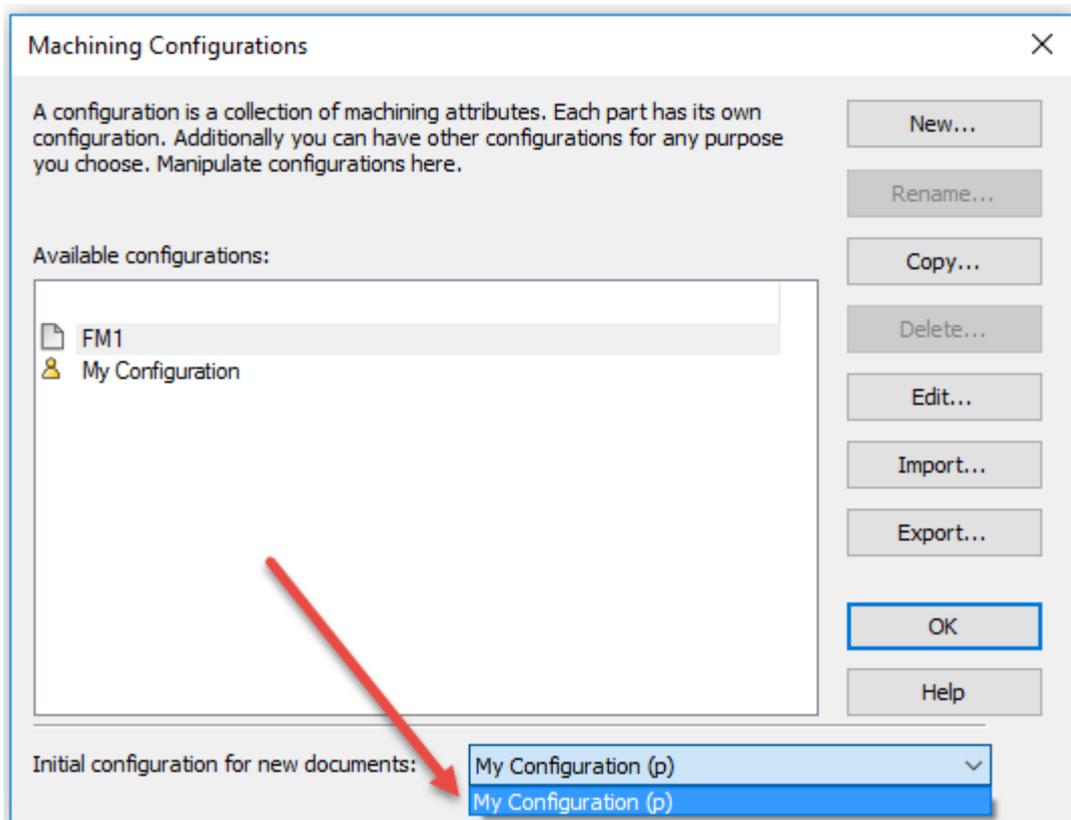




What are Machining Attributes?



- 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:

- 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



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.

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 MPR (Millimetres per revolution) by default are set here.

Remachining is addressed a little later on in this module.

Operations

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

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

Help Contents

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

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



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

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



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

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



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

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



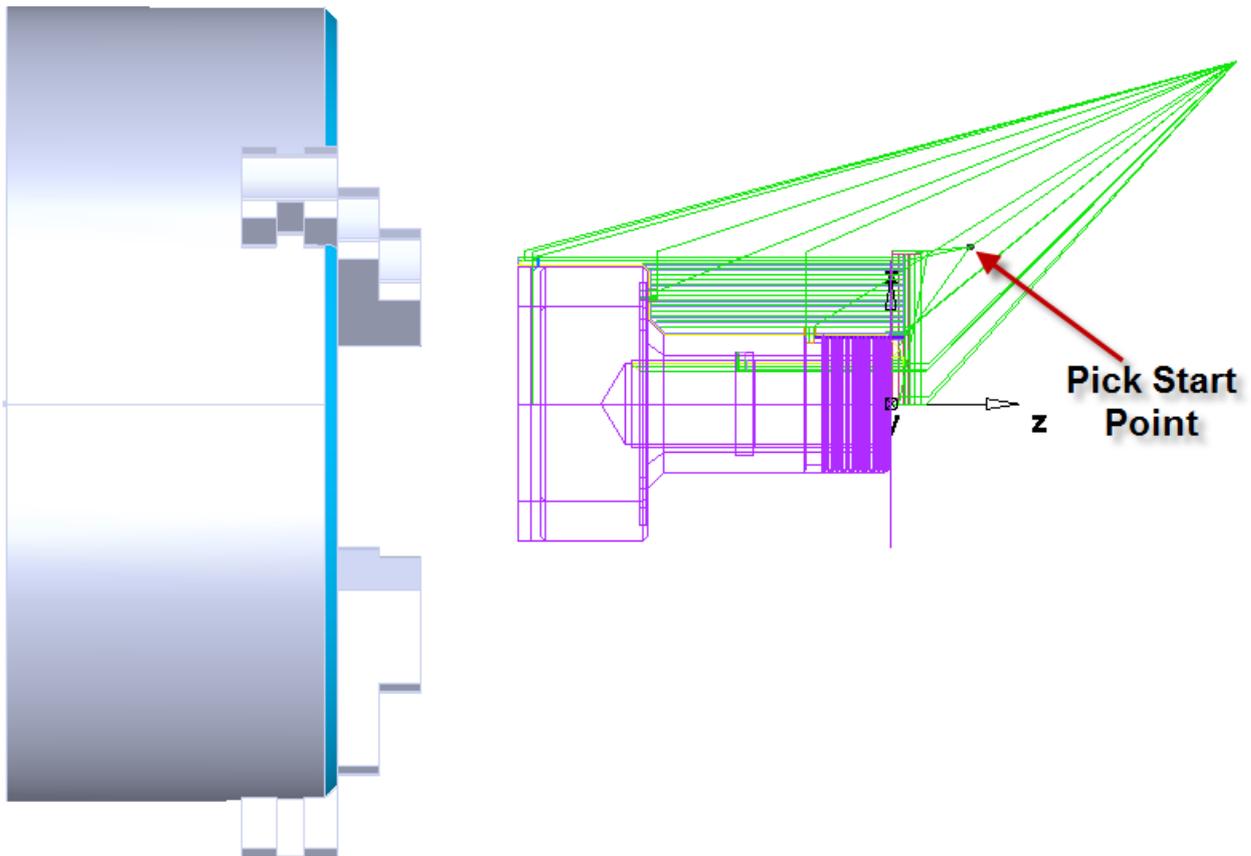
The Select topic list shows all the topics where the key words **Start** and/or **Point** appear. Since we know that setting exists in the Turning tab of our feature we will want to choose that listing.

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



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

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



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



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



Try a few other settings after referencing the Help file.

Contents

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

Turning Tool Reference Guide

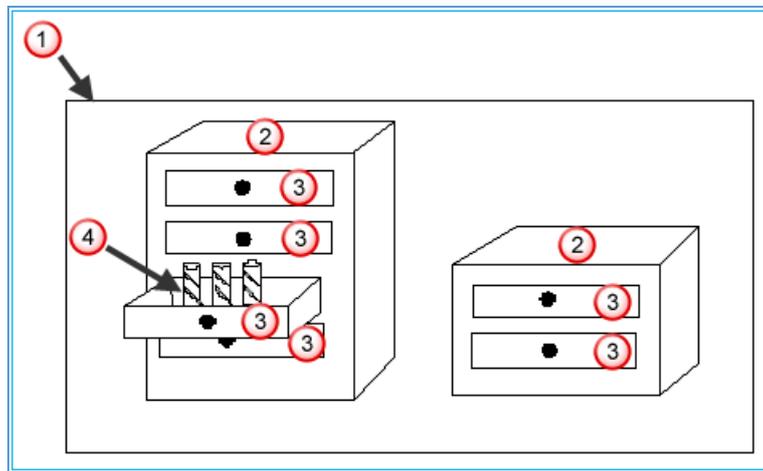
It is possible to create Tool libraries in **FeatureCAM**. These are called Tool **Cribs**. The main reason for creating Tool Cribs is because in any machine shop there is normally a finite amount of tooling available and by creating a custom Tool Crib, **FeatureCAM** will know what tooling is available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

Tooling

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

Overview of tooling

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



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

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

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

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

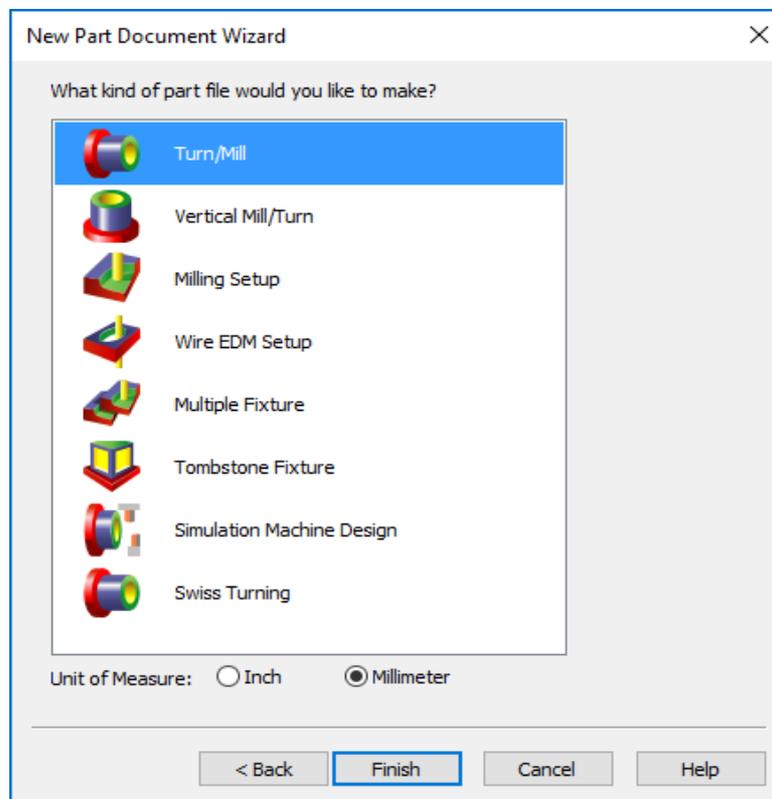
The tooling database defines the set of tools from which FeatureCAM selects tools to perform manufacturing operations. These tool sets are called cribs. FeatureCAM comes with two standard tool cribs, the **Tools** tool crib and the **Basic** tool crib. The **Tools** tool crib is a comprehensive tool 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.

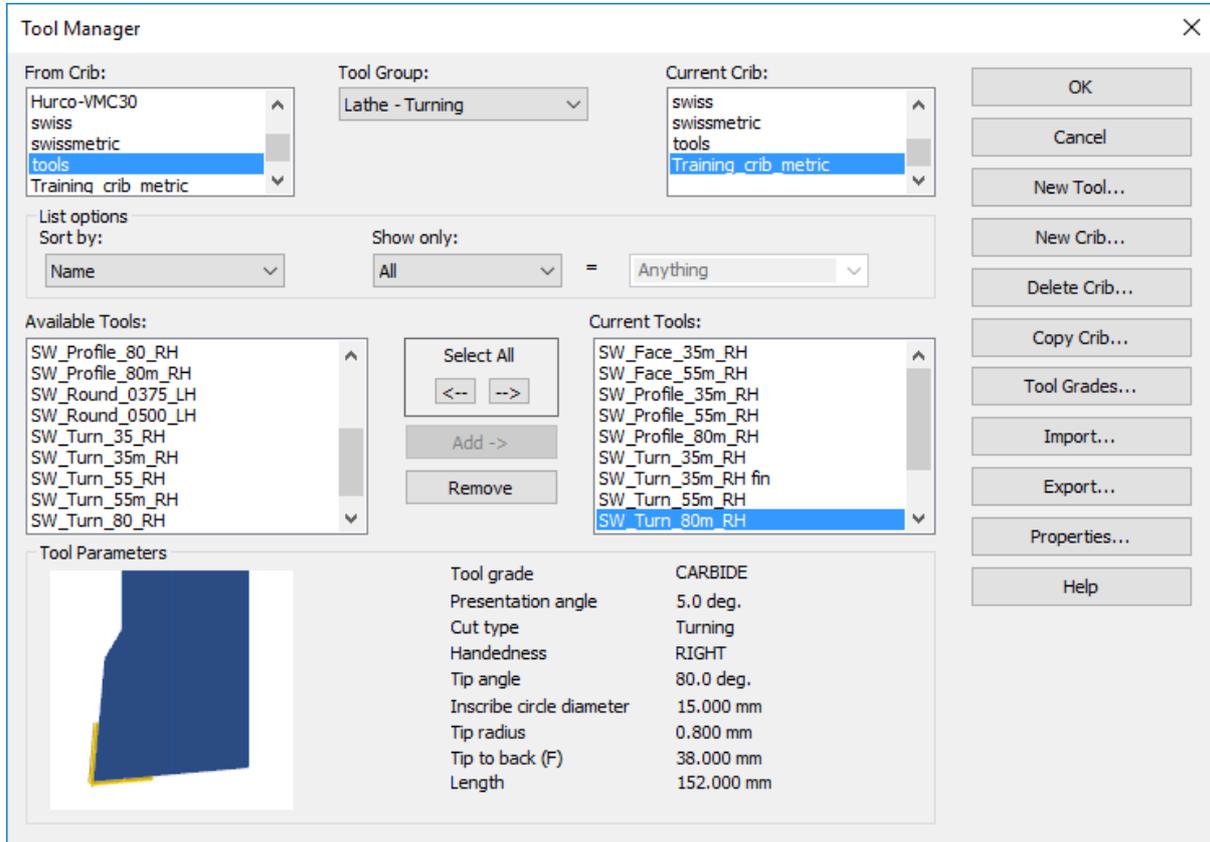
- 1 **Open** a new **FeatureCAM** part by selecting **File, New**.
- 2 The following Menu shown on the next page will appear. Select **Turn/Mill** Set-up and click **OK**.





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**

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



There are three tool Crib 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 Crib. 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**.



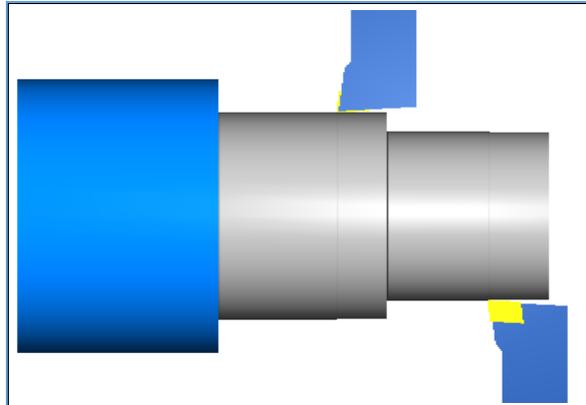
The window is divided into separate areas and at first glance this can look a little intimidating, but by applying some simple rules it becomes much easier to work with.



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.

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.

Machining Attributes [X]

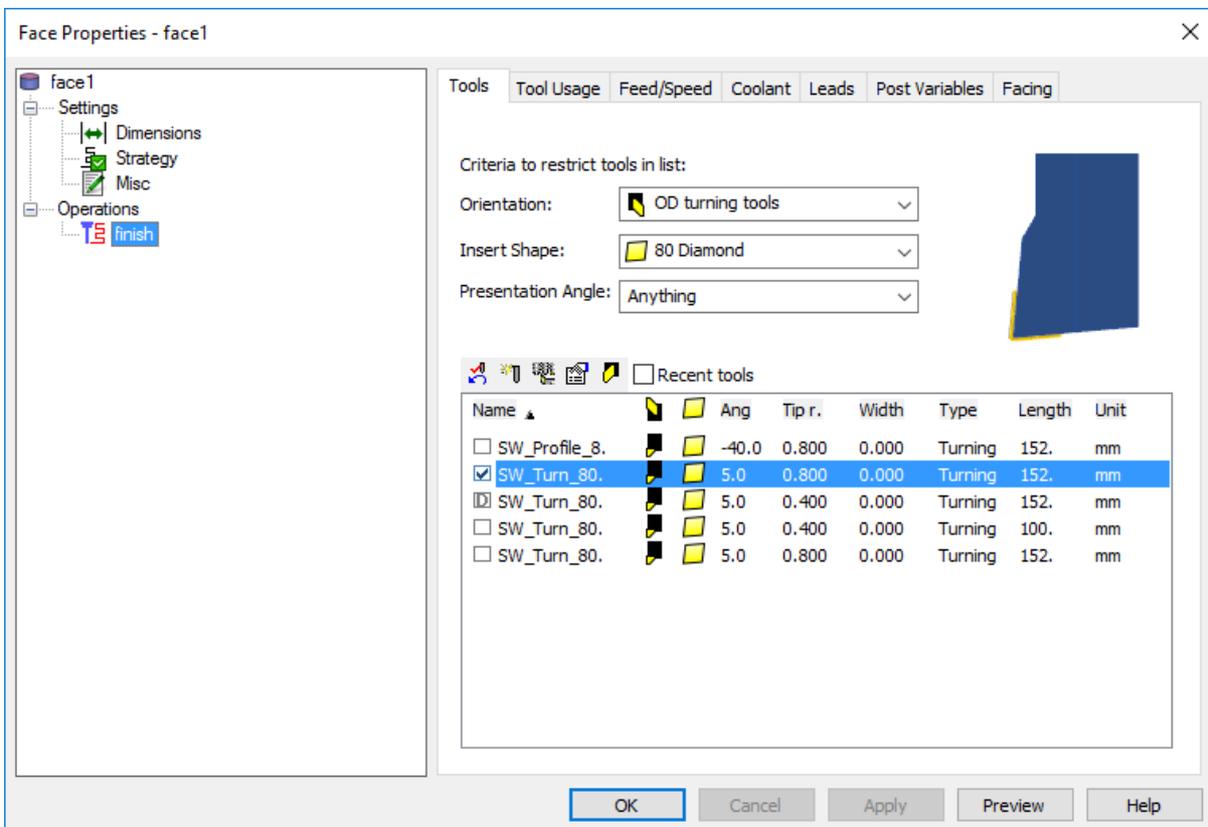
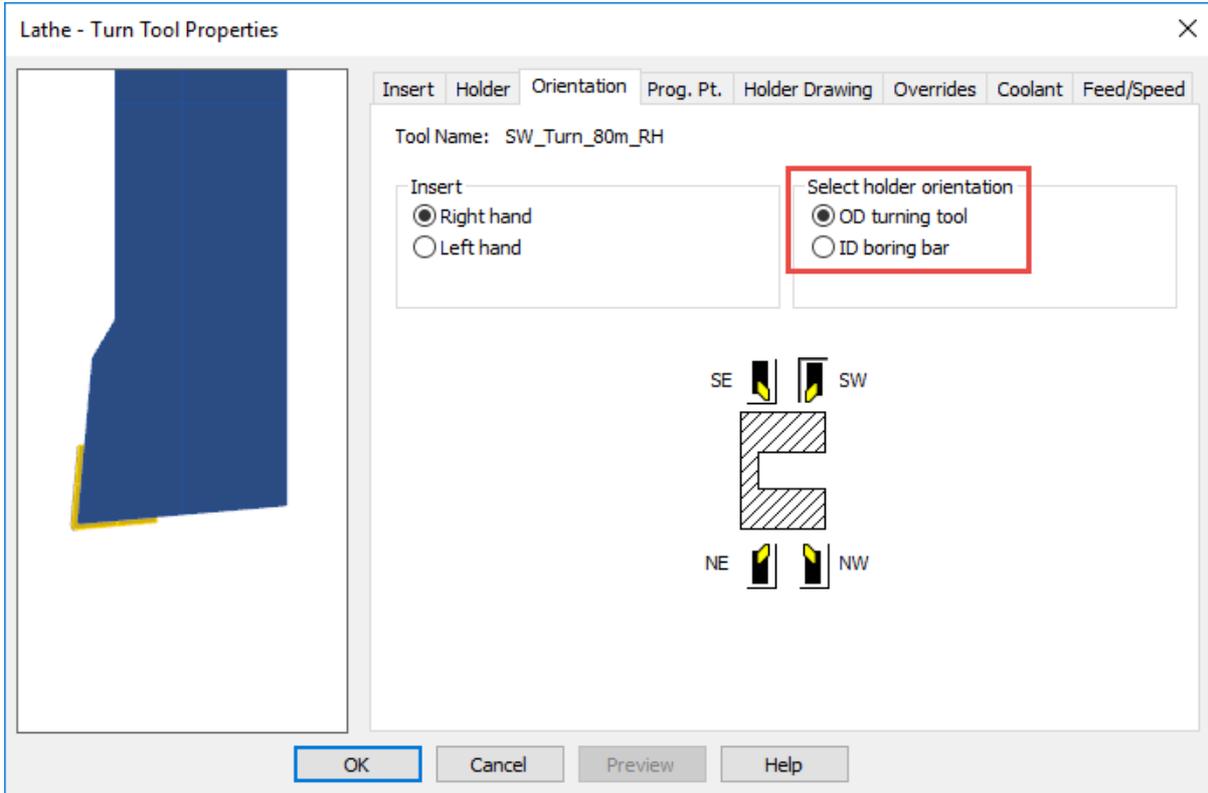
Configuration: My Configuration Mill Turn Wire

Drilling	Pecking	Turn/Bore	Threading	Grooving	Cutoff	Bar Feed
Coolant		Post Variables		Misc.	Operations	
Speed	<input type="text" value="100"/>	%	Turret direction	<input type="text" value="Auto"/>	<input type="text" value="Auto"/>	
Feed	<input type="text" value="100"/>	%	Spindle direction	<input type="text" value="CW"/>	<input type="text" value="Auto"/>	
Plunge clearance	<input type="text" value="3"/>	mm	RPM range			
2nd offset reg. increment	<input type="text" value="20"/>					
Swiss Safety Allowance	<input type="text" value="10.000"/>	mm	<input checked="" type="checkbox"/> Remachining			
Tool program point:			<input checked="" type="checkbox"/> Constant surface speed			
<input checked="" type="radio"/> Tool tip edge			<input checked="" type="checkbox"/> Use MPMR			
<input type="radio"/> Tool tip center			<input checked="" type="checkbox"/> Do feed reduction for small moves			
Turnmilling program point:			Threshold:	<input type="text" value="2"/>	revs	
<input type="radio"/> Tool edge			Feed rate:	<input type="text" value="30"/>	%	
<input checked="" type="radio"/> Tool center			<input type="checkbox"/> Calculate index radius from solid stock outline			
<input type="checkbox"/> Feed from start point or curve			<input checked="" type="checkbox"/> Automatic tool orientation			
<input checked="" type="radio"/> Rapid on curve						
<input type="radio"/> Feed on curve:						
<input type="text" value=""/>			MMPM			



The tool orientation is split into two types.

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



Tool Groups.



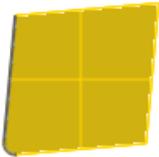
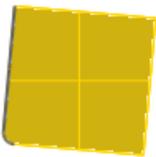
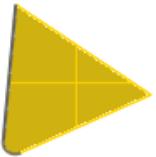
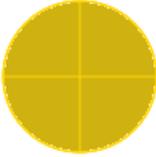
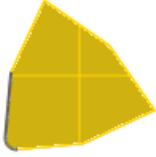
The tools can be grouped together depending upon their type or use.



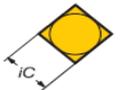
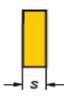
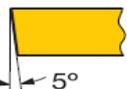
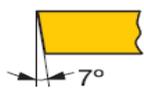
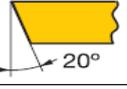
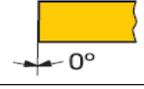
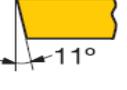
Insert Shapes

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

Inserts for General Turning

1 Insert shape		3 Tolerances, metric																					
C 	D 	<table border="1"> <thead> <tr> <th>Class</th> <th>s</th> <th colspan="2">iC / iW</th> </tr> </thead> <tbody> <tr> <td>G</td> <td>±0.13</td> <td colspan="2">±0.025</td> </tr> <tr> <td>M</td> <td>±0.13</td> <td colspan="2">±0.05 – ±0.15¹⁾</td> </tr> <tr> <td>U</td> <td>±0.13</td> <td colspan="2">±0.08 – ±0.25¹⁾</td> </tr> <tr> <td>E</td> <td>±0.025</td> <td colspan="2">±0.025</td> </tr> </tbody> </table> <p>¹⁾Varies depending on the size of iC. See below.</p>	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		    
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																					
K 	R 																						
S 	T 																						
V 	W 																						
2 Insert clearance angle		Inscribed circle iC mm		Tolerance class M	U																		
B 	C 	3.97																					
E 	N 	5.0																					
P 	O Specific description	5.56																					
		6.0	±0.05	±0.08																			
		6.35																					
		8.0																					
		9.525																					
		10.0																					
		12.0	±0.08	±0.13																			
		12.7																					
		15.875																					
		16.0	±0.10	±0.18																			
		19.05																					
		20.0																					
		25.0	±0.13	±0.25																			
		25.4																					
		31.75	±0.15	±0.25																			
		32.0																					
		For positive inserts iC is valid for a sharp corner. See cutting edge condition F. (Picture 8).																					

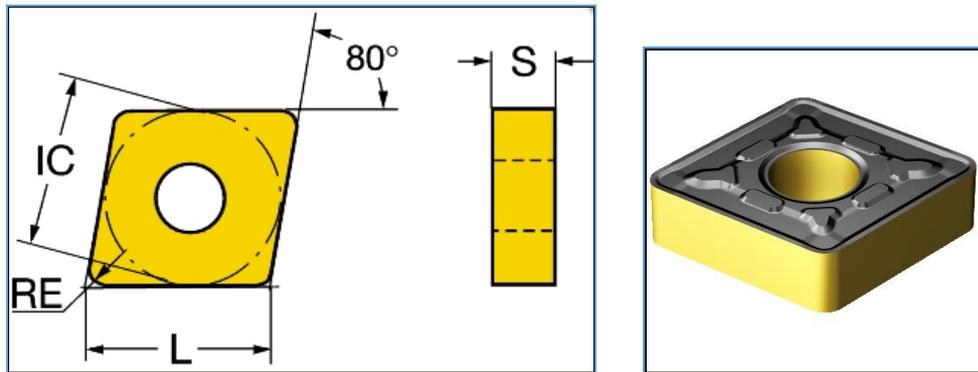
CNMG Insert

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.

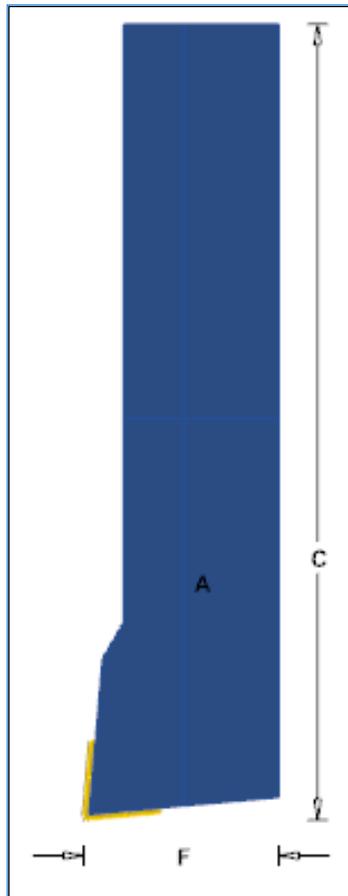
CNMG 12 04 08-PR 4225



S = Insert Thickness

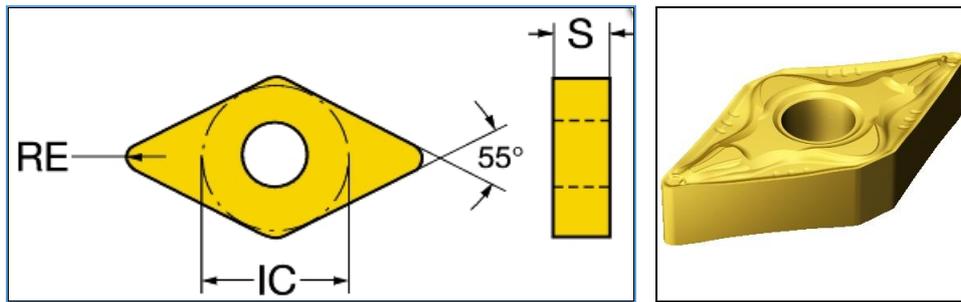
RE = Corner Radius

IC = Inscribed Circle Diameter



FeatureCAM tool designation = **SW_Turn_80m_RH**

DNMG Insert



S = Insert Thickness

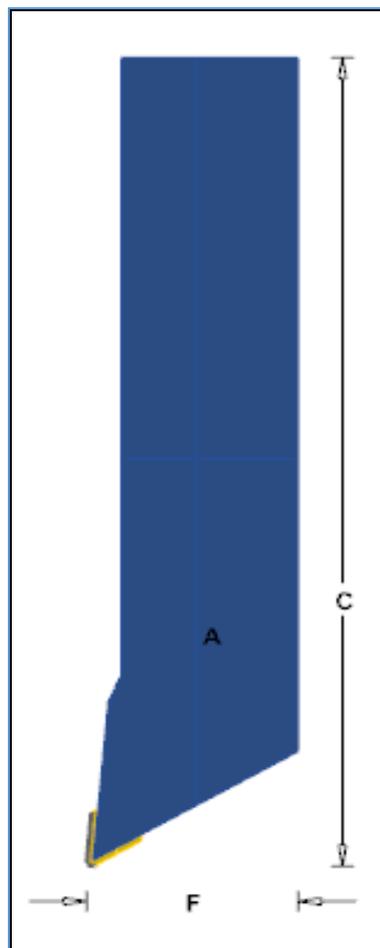
RE = Corner Radius

IC = Inscribed Circle Diameter

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

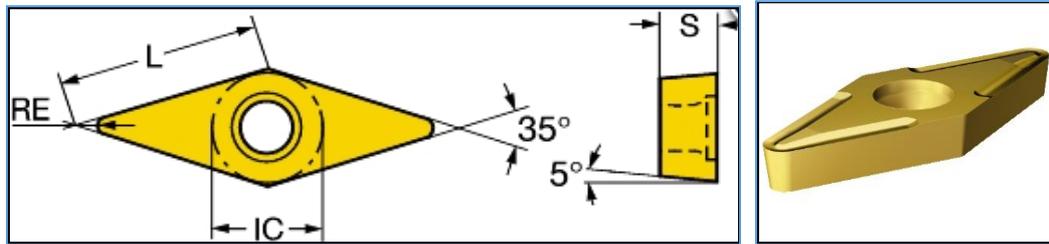
Tip Angle is the included angle of the insert.

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



FeatureCAM tool designation = **SW_Turn_55m_RH**

VBMT Insert



S = Insert Thickness

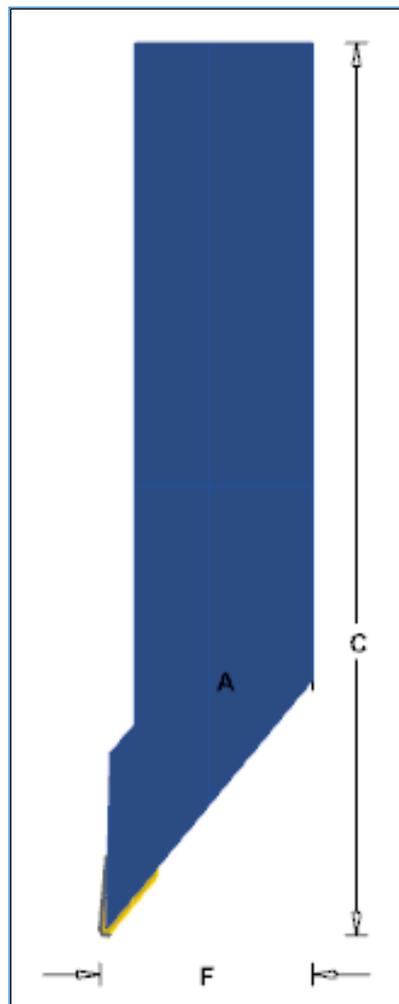
RE = Corner Radius

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.

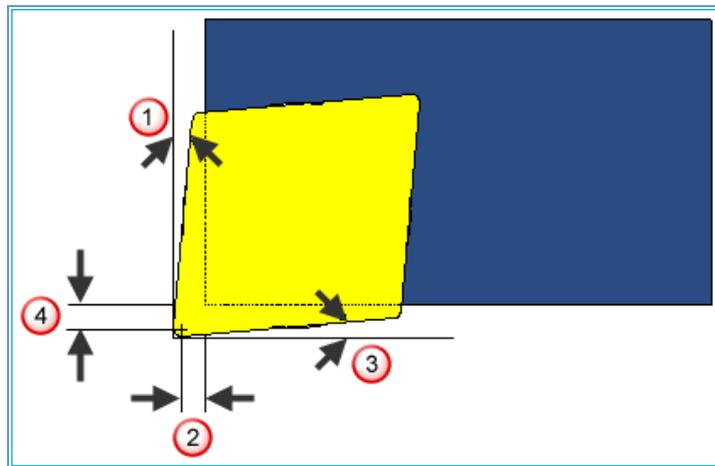
Inscribe Circle Diameter is the diameter of a circle that fits inside the insert shape.



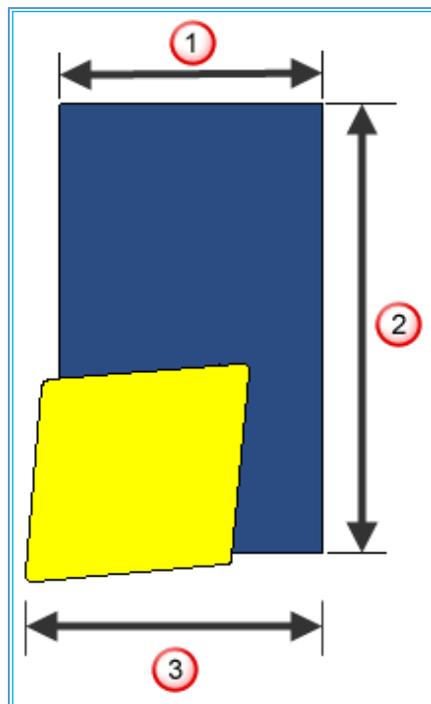
FeatureCAM tool designation = **SW_Turn_35m_RH**

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.

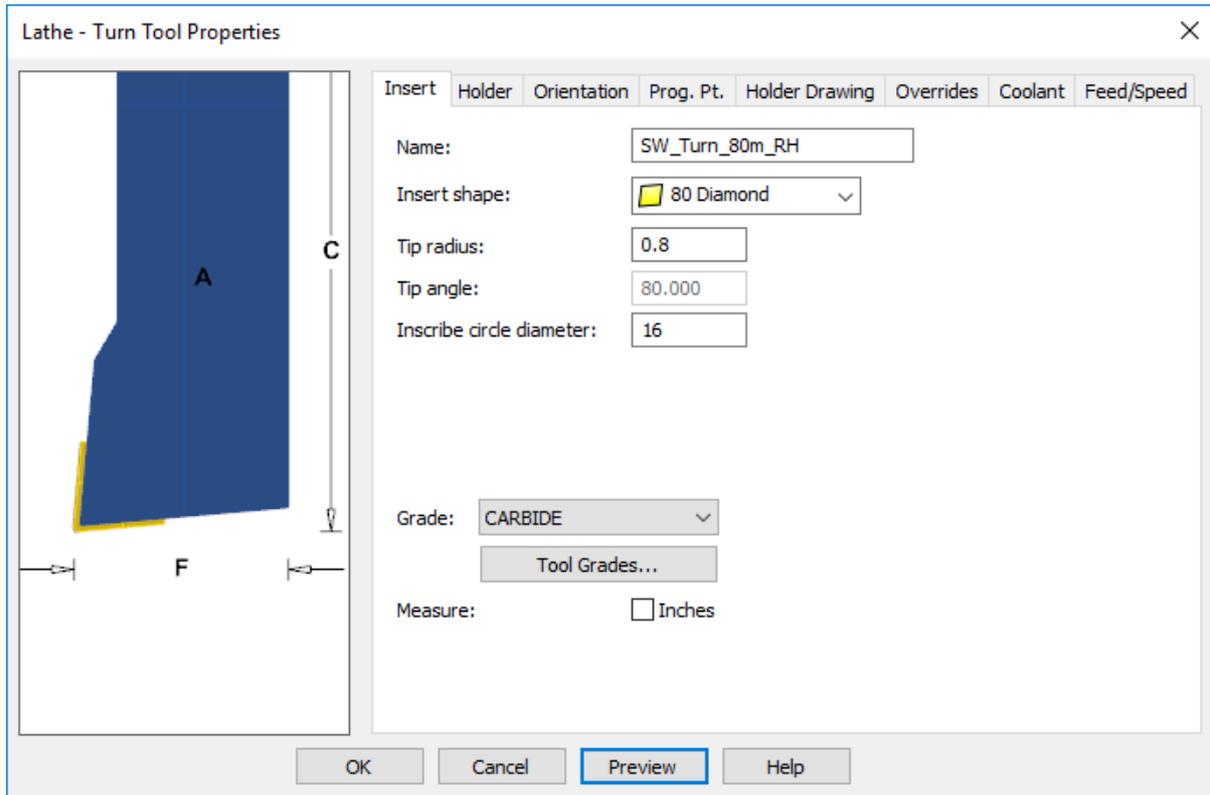


- ① End Angle
- ② End Clearance
- ③ Side Angle
- ④ Side Clearance

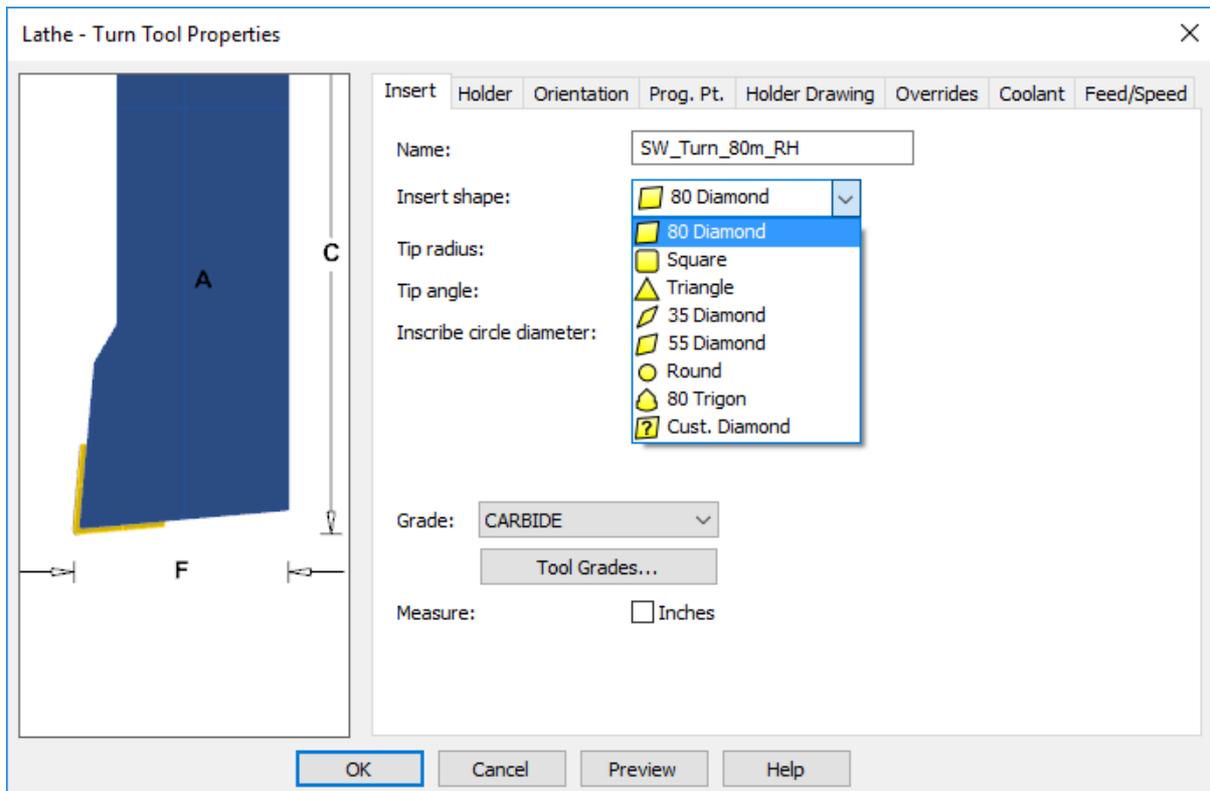


- ① Shaft Width (A)
- ② Length (C)
- ③ Tip to Back (F)

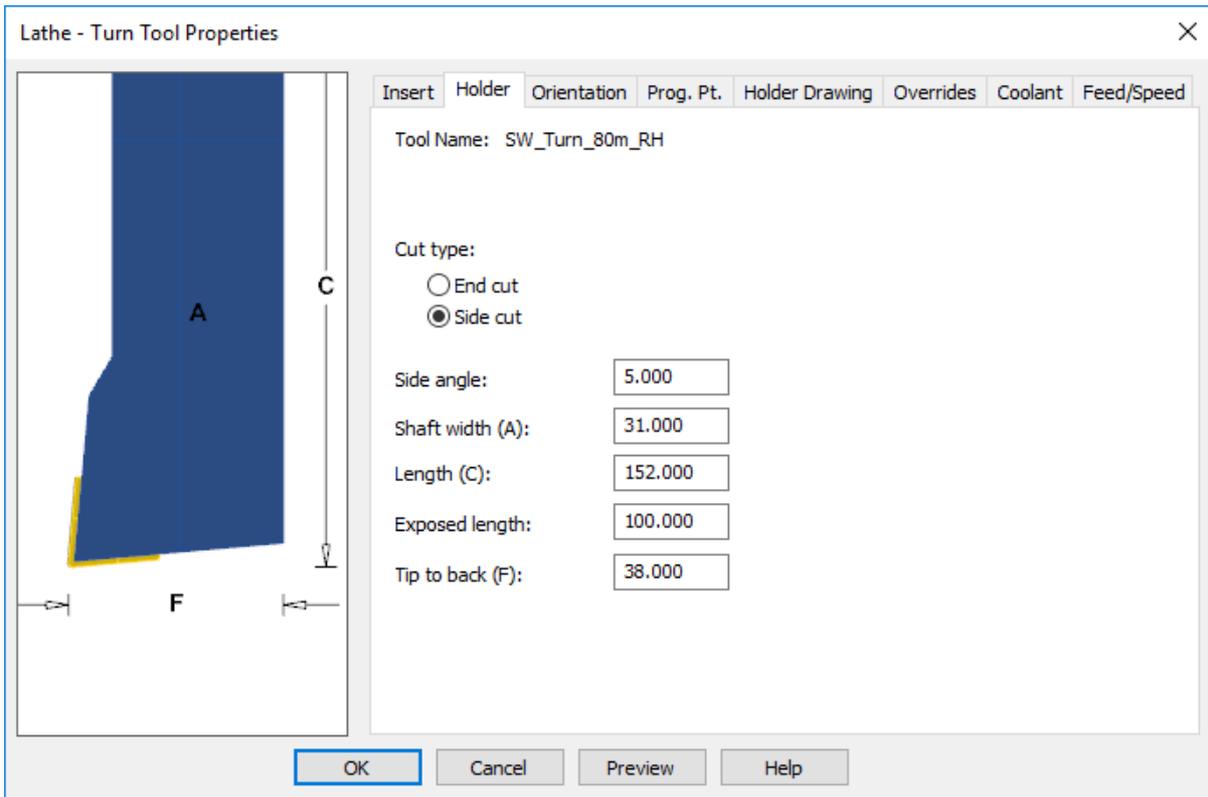
Exposed length: - This is the amount of the tool that sticks out of the Turret if the holder is simulated. Lathe - Turning Tool Properties example.



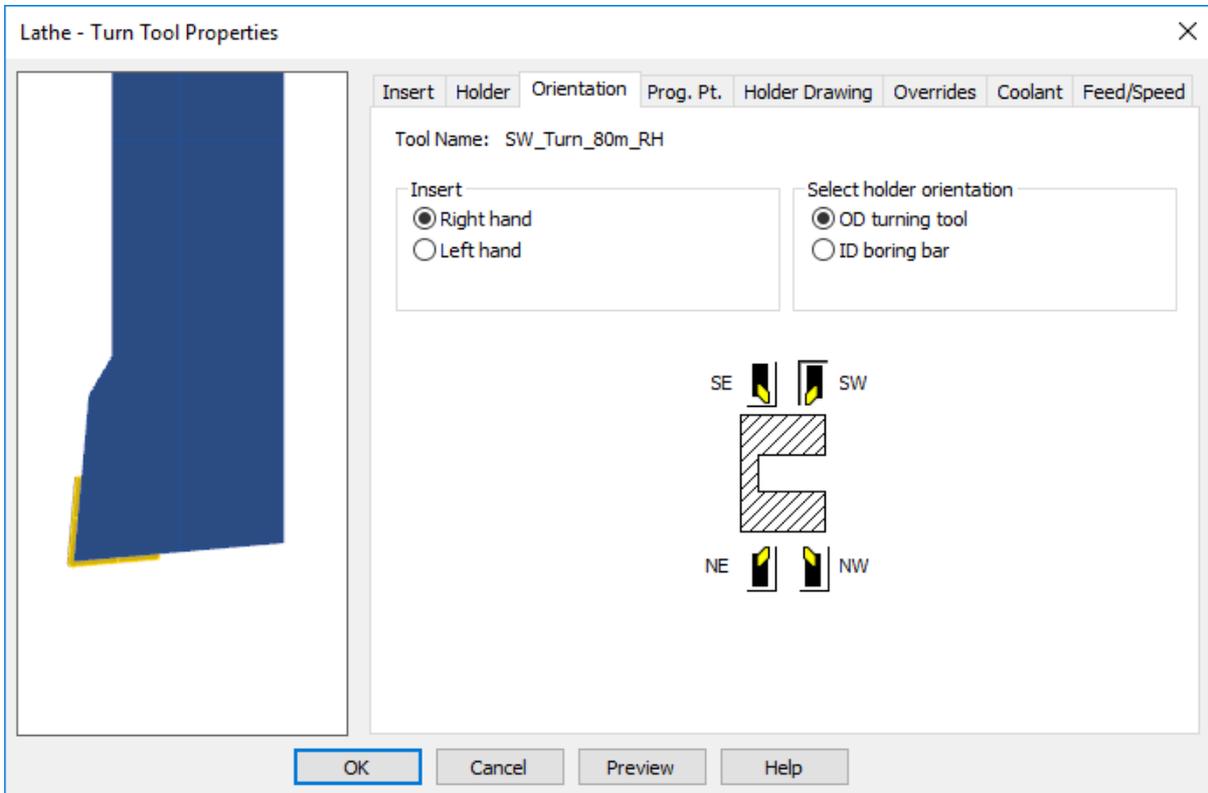
Insert Shape

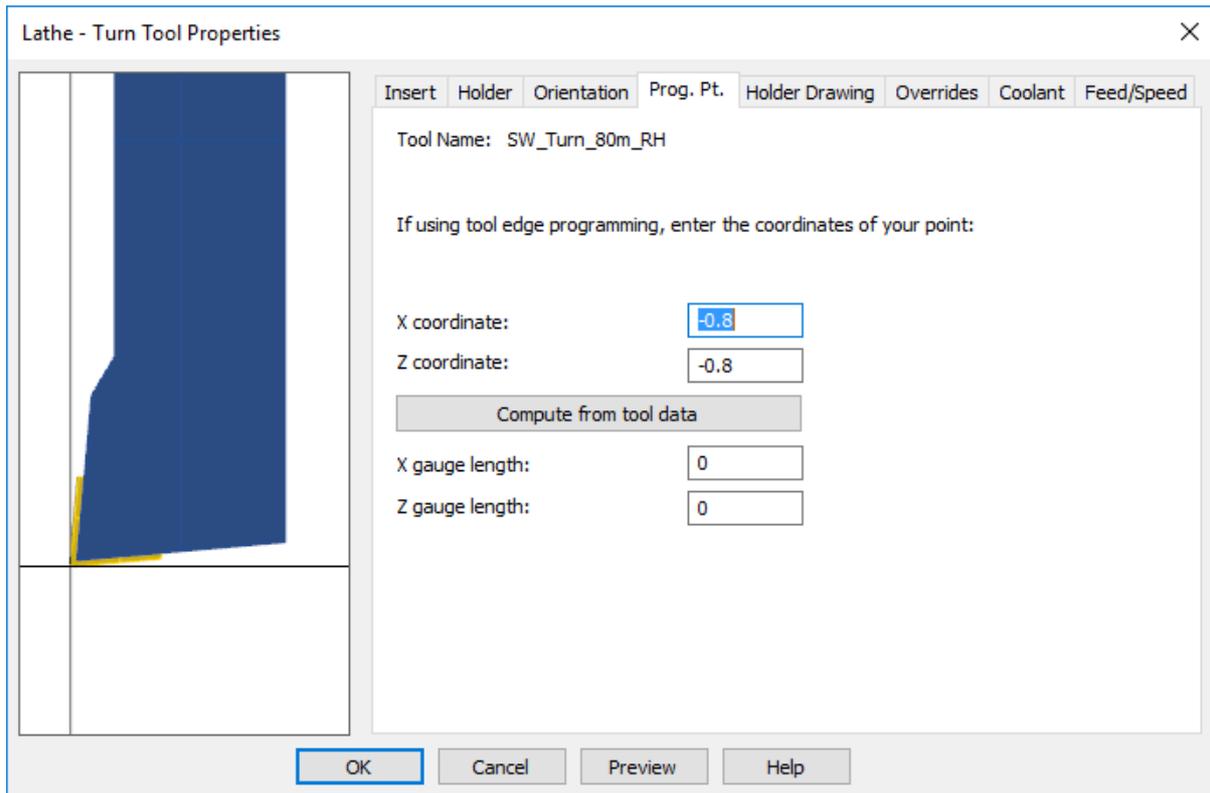


Holder



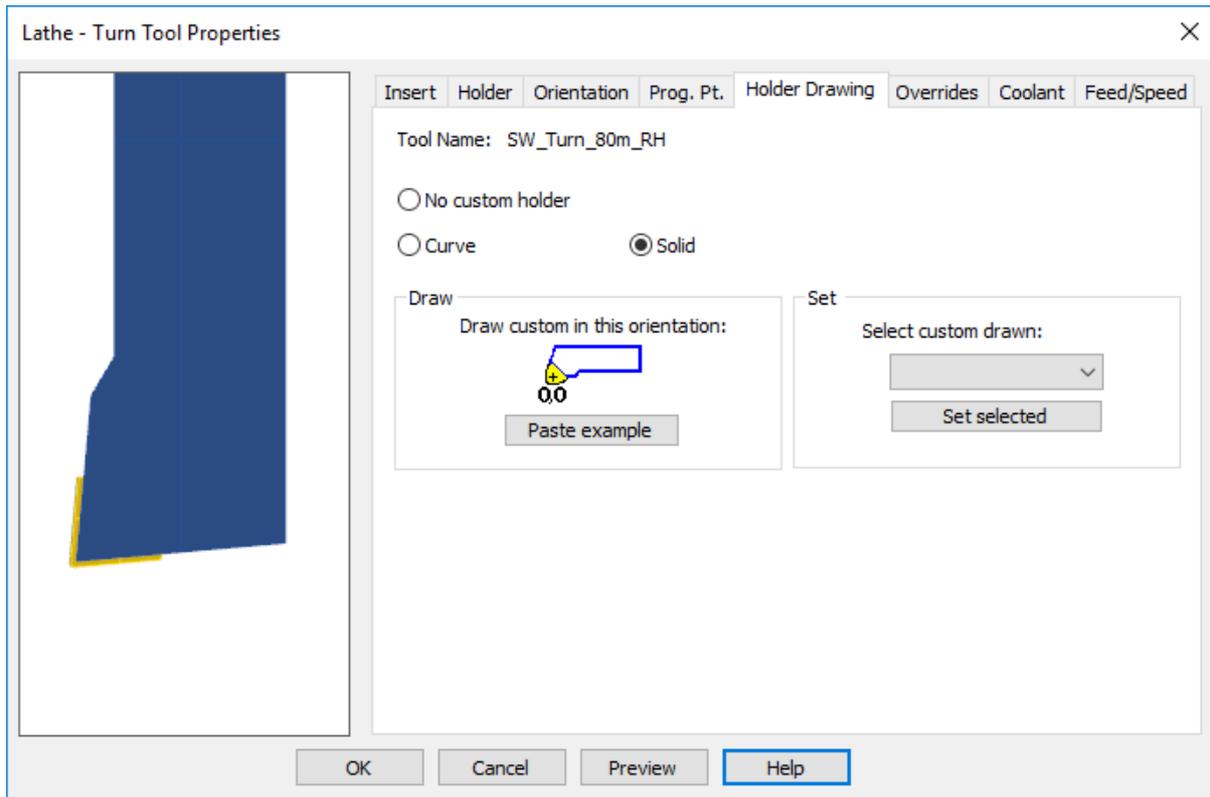
Orientation





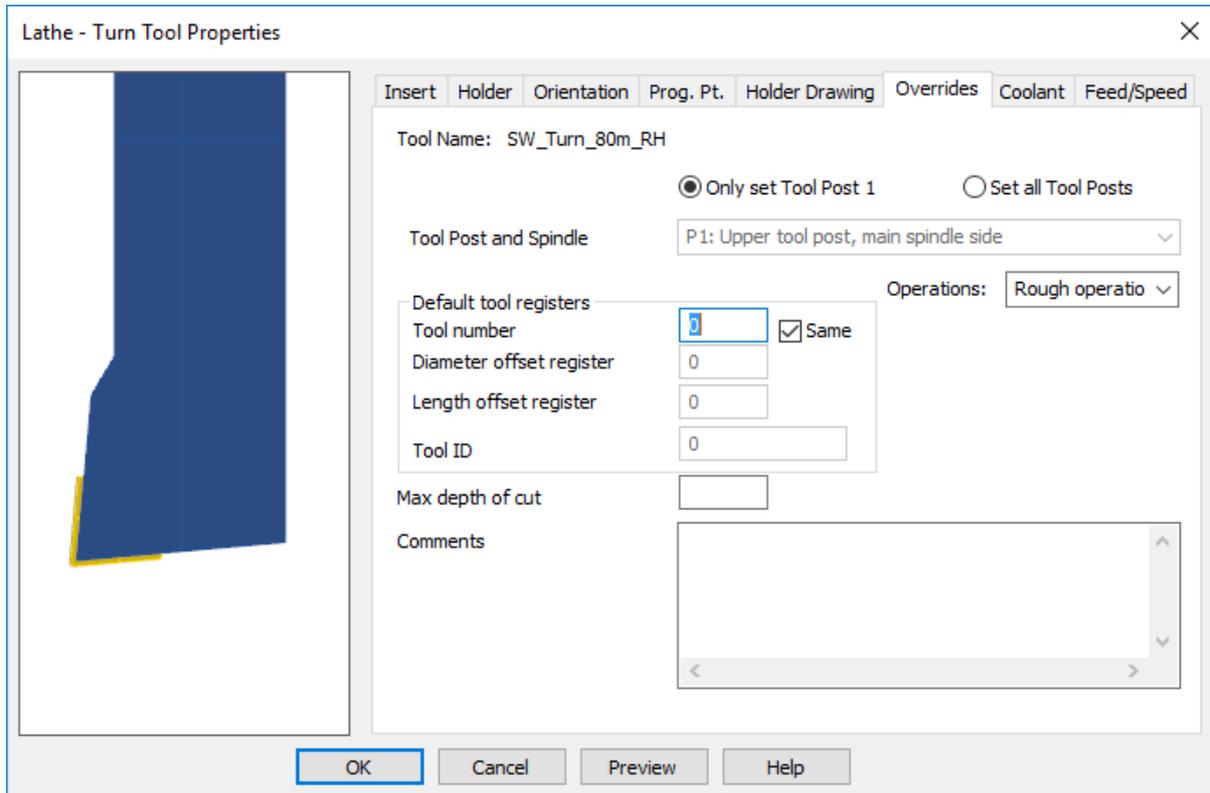
On the **Prog. Pt.** (program point) tab you set the point of the insert that is actually programmed. You can set both the **X Coordinate** and the **Z Coordinate**. **Tool tip centre** - If both **X Coordinate** and the **Z Coordinate** are set to **0.0** then the centre point of the tip arc is programmed. 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**.

Holder Drawing

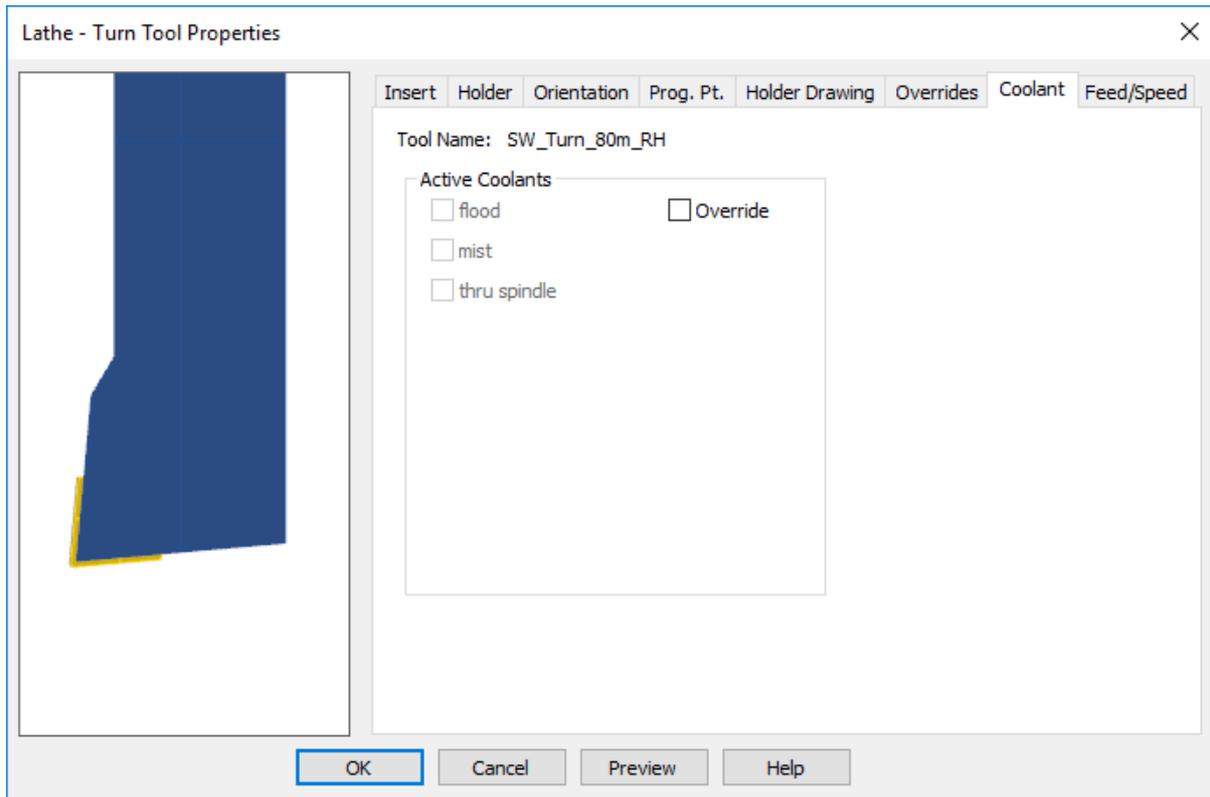


Solid — you can select a solid to define the shape of the Tool Holder. This is the most accurate way to represent the tool holder.

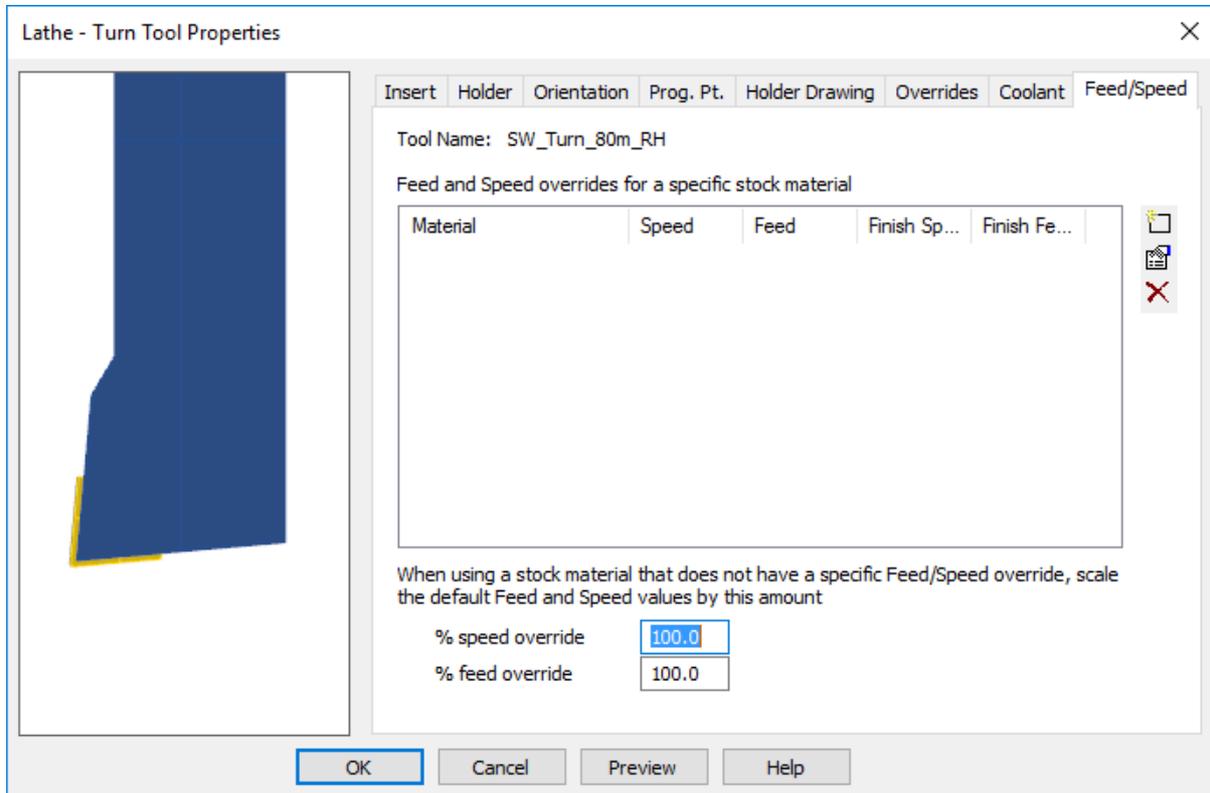
Overrides



Coolant

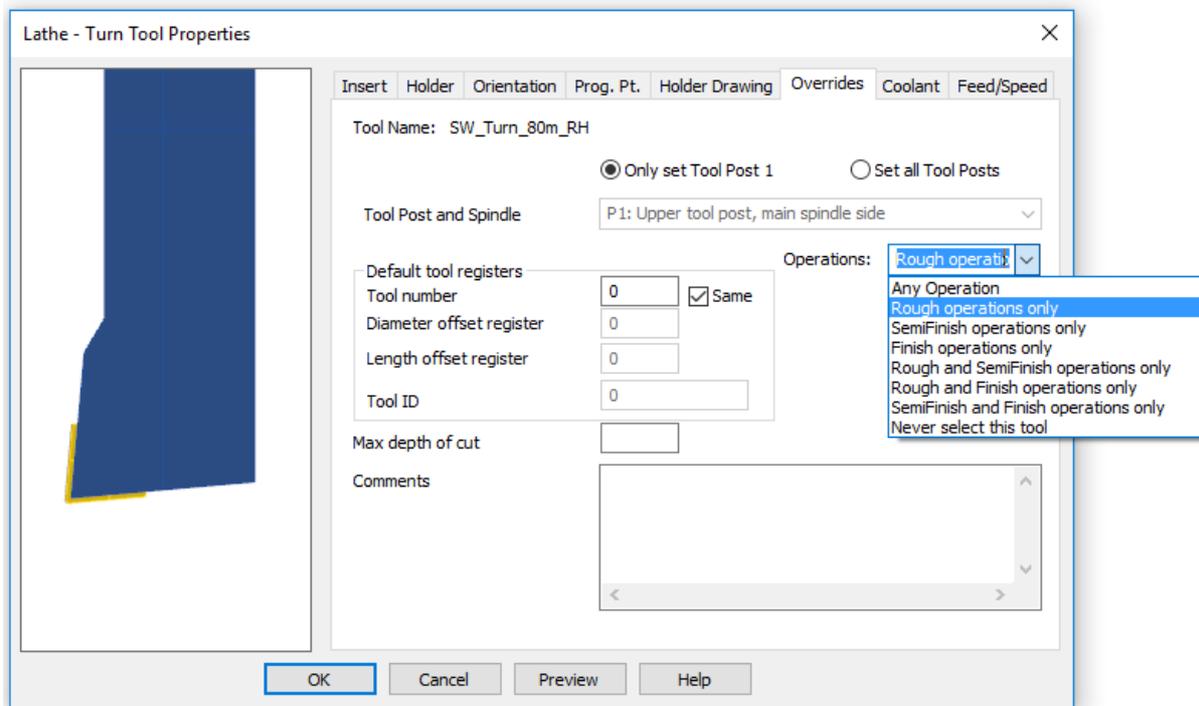


Feeds/Speeds



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).



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.

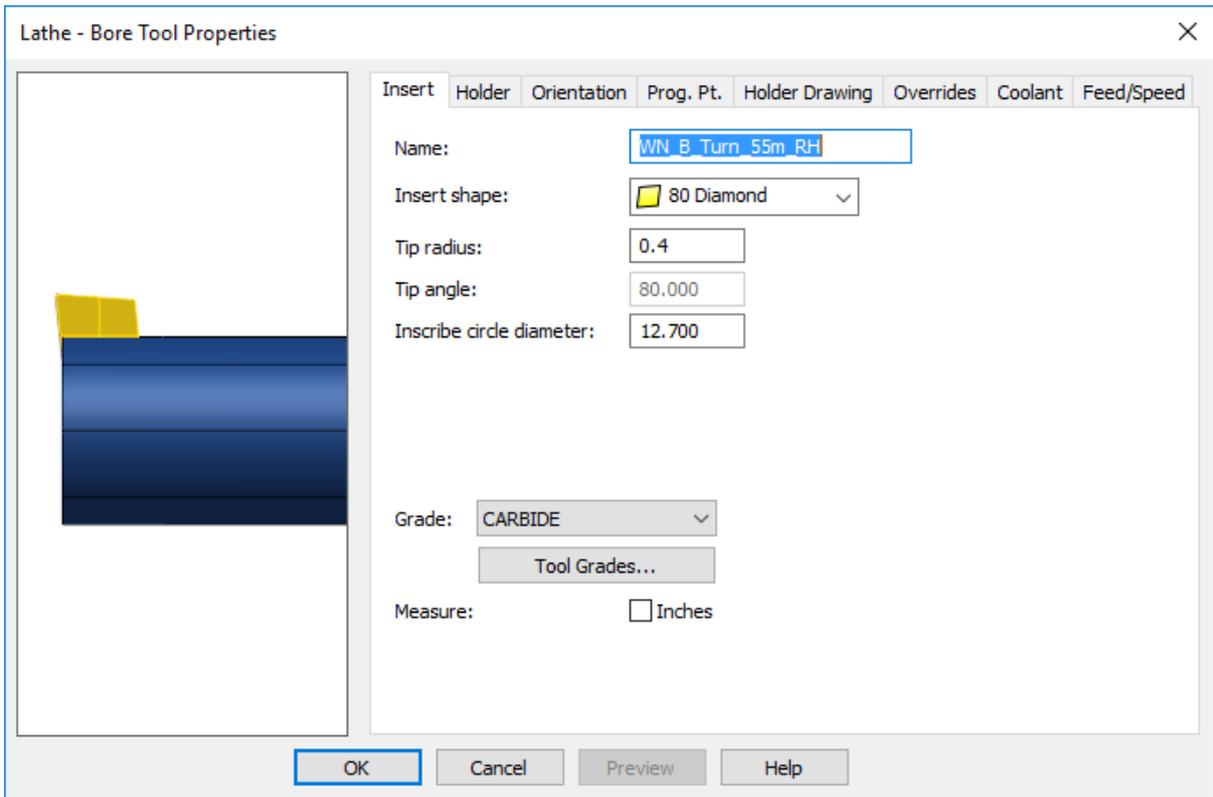


Use IPR/MMPR - The default feed units are IPM (inches per minute) or MPM (mm per minute). Select this option to use IPR (inches per revolution) or MMPR (mm per revolution).

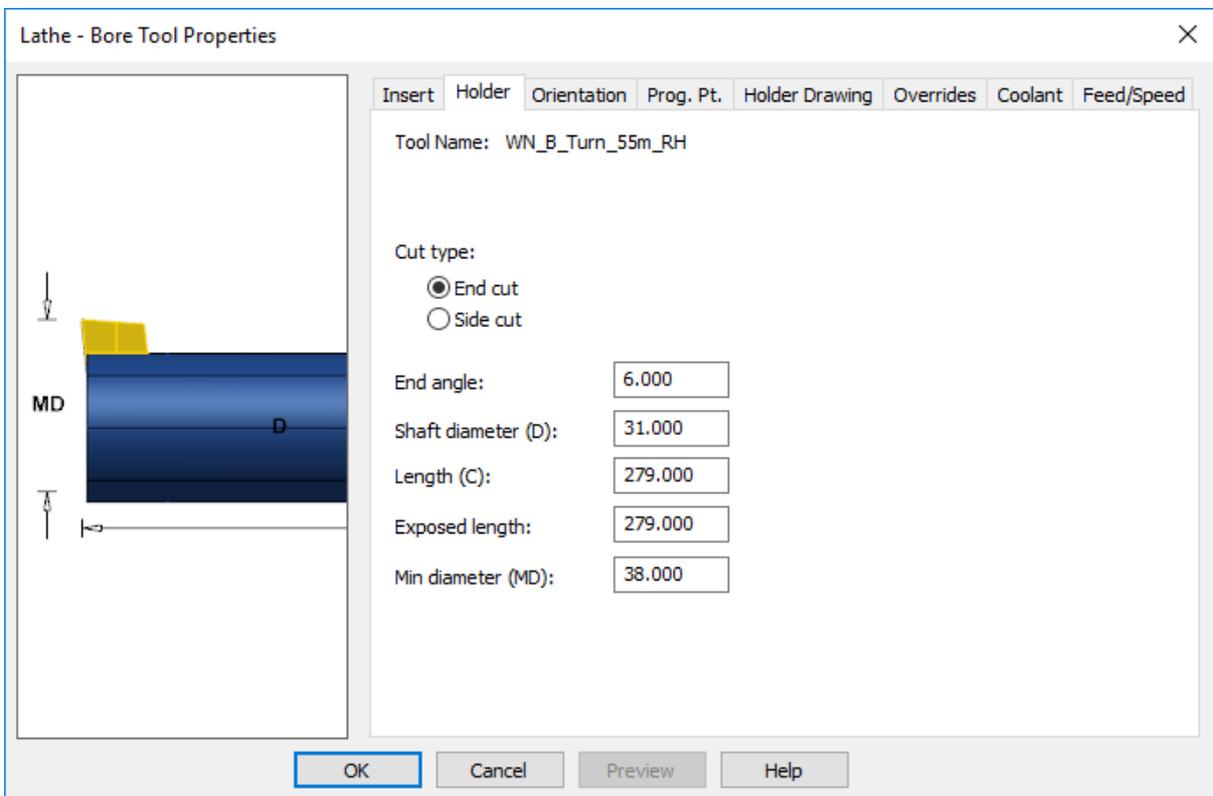


Do feed reduction for small moves - This attribute helps FeatureCAM cut small features properly. It is typically applied to small chamfers or small radii but affects any small move. If Do feed reduction for small moves is selected, then any move with fewer revolutions than the Threshold, is reduced by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

Lathe - Bore Tool Properties example.



Holder Information

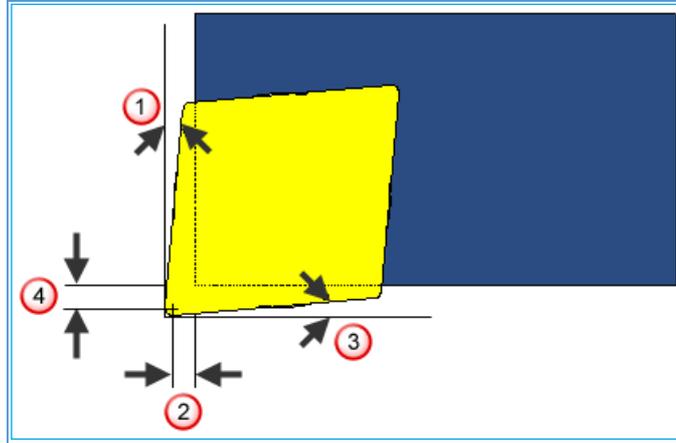




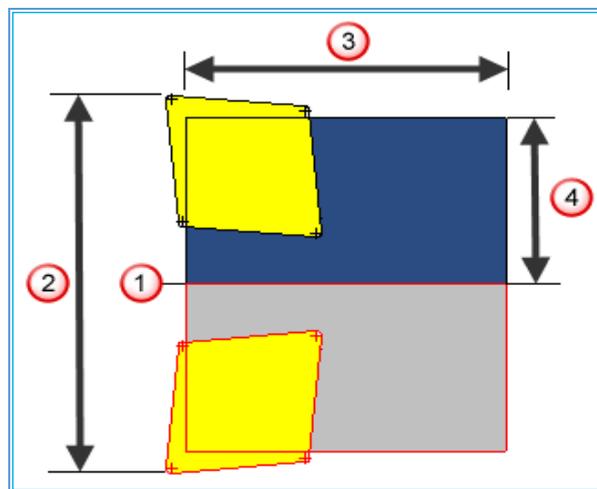
End Cut: - The tool cuts in a direction parallel with the length of the holder.



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



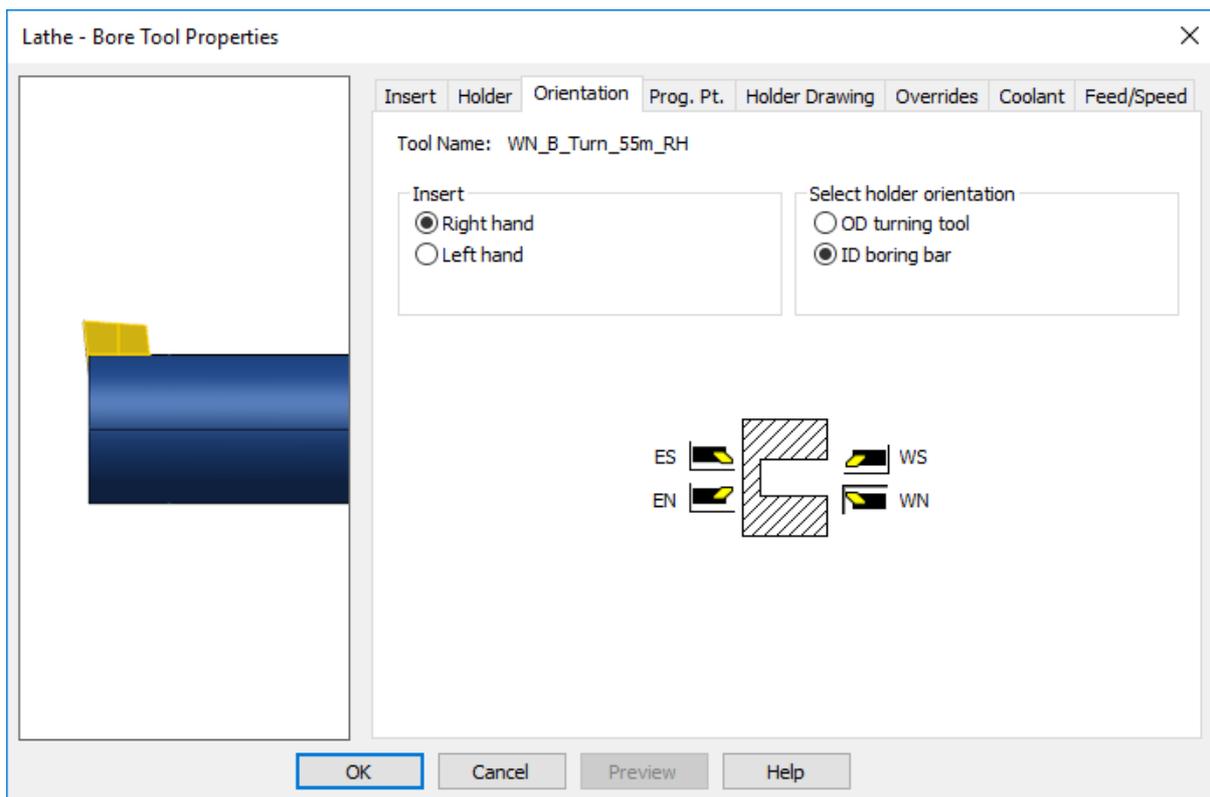
- ① End Angle
- ② End Clearance
- ③ Side Angle
- ④ Side Clearance



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

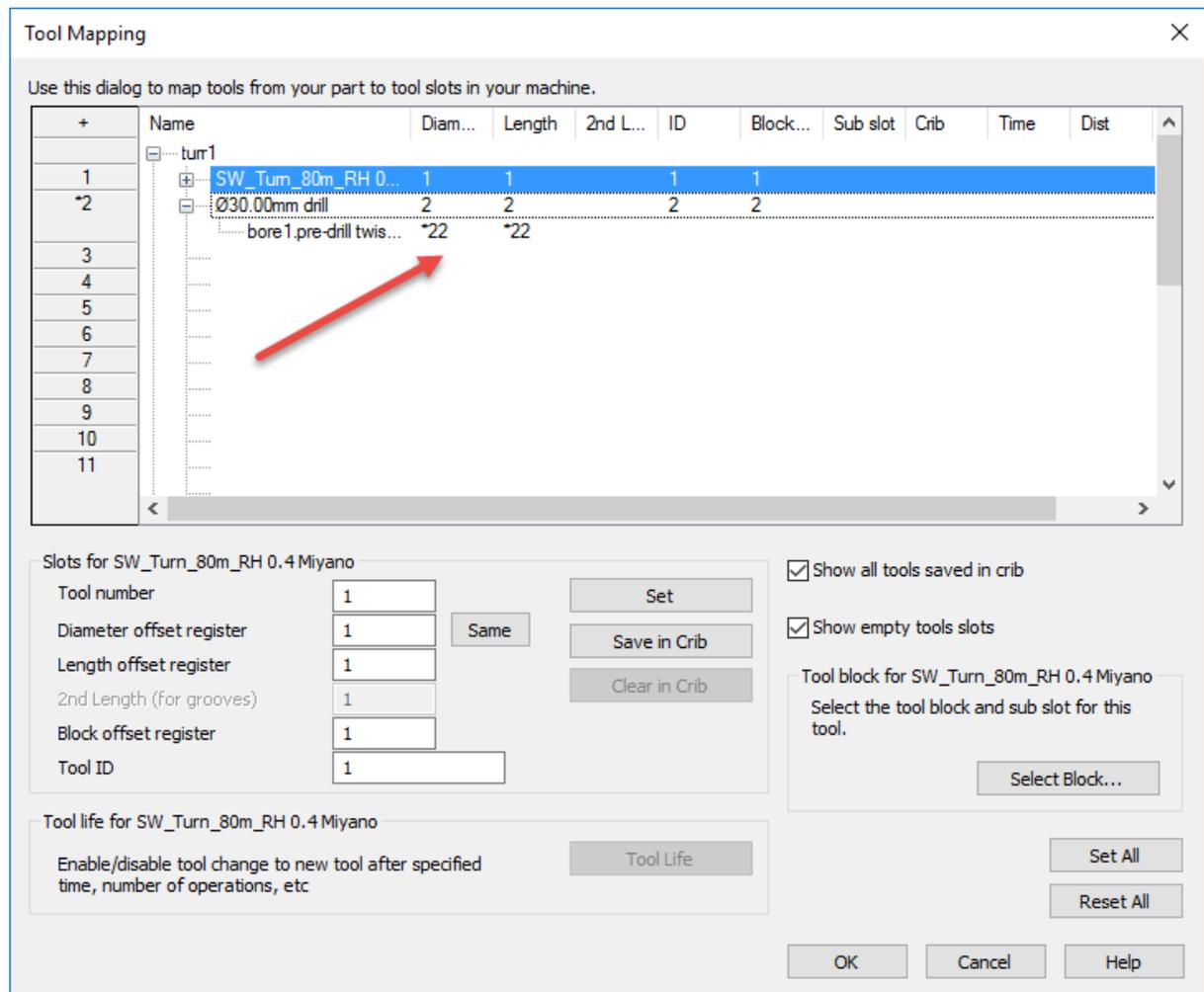
Exposed length: - This is the amount of the tool that sticks out of the Turret if the holder is simulated.

Boring Orientation



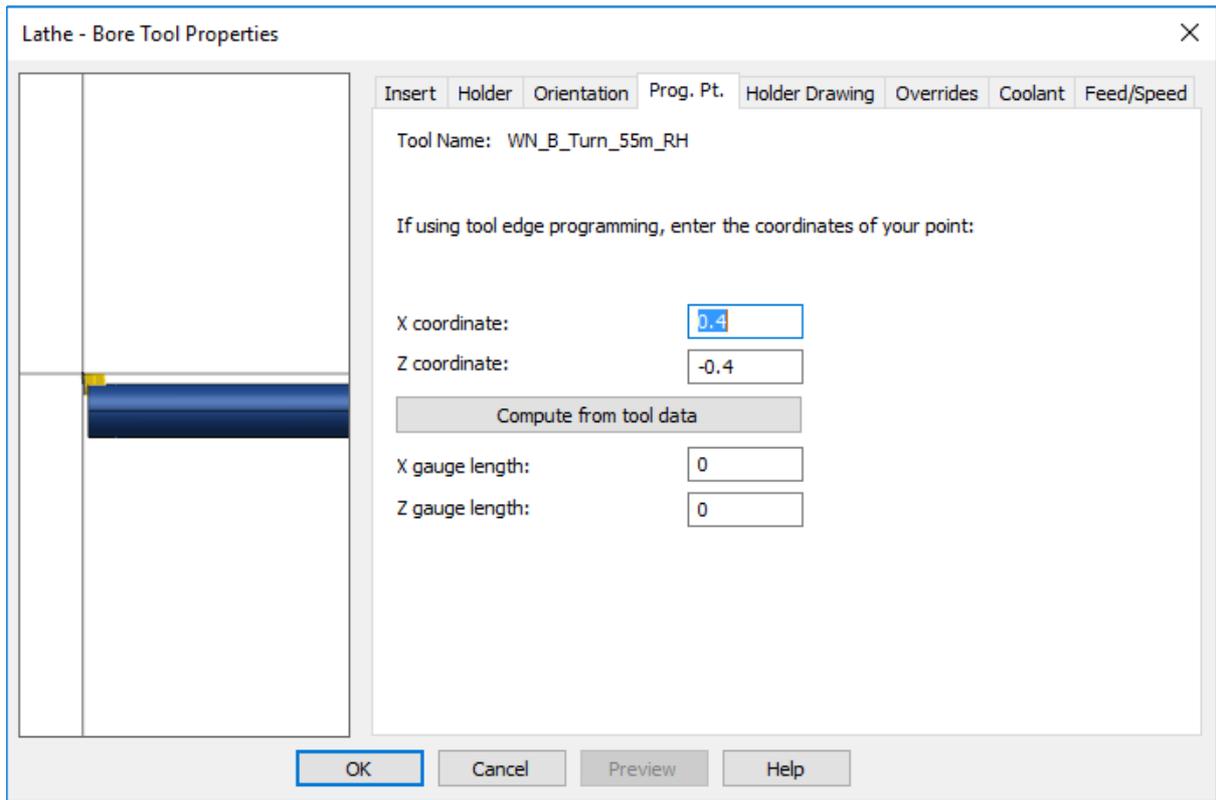
Using an insert drill to drill and bore in the same program

- 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.

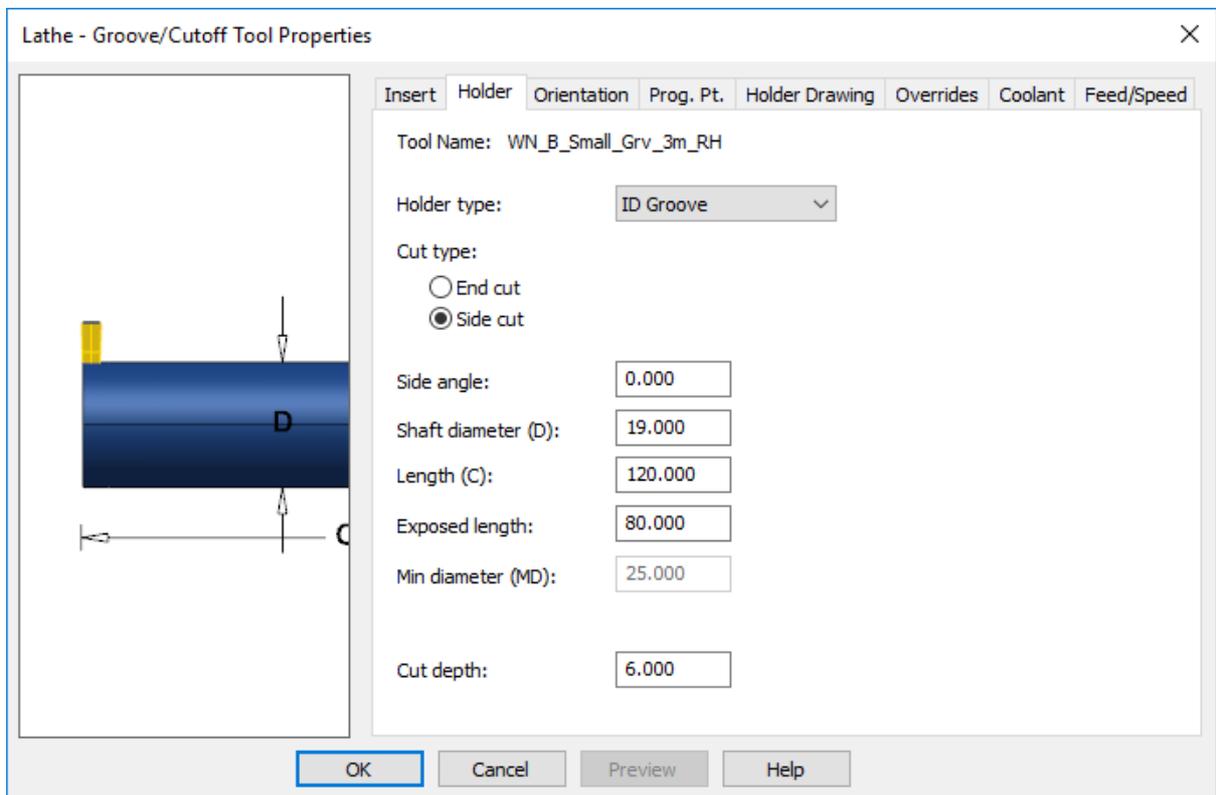


Please note that you need to reduce the retract distance to avoid a collision inside the bore. This needs to be applied to roughing and finishing operations.

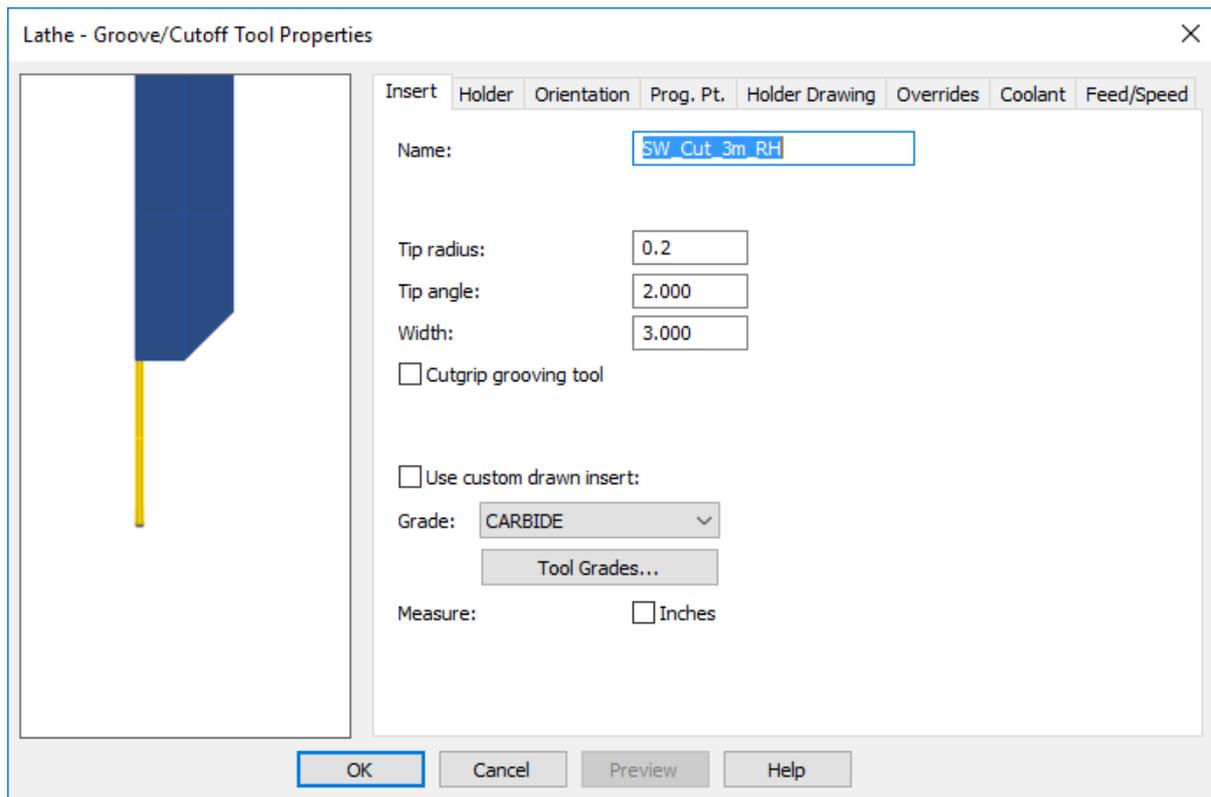
Lathe - Bore Programming Point



Lathe – Groove tool Properties example



Lathe - Cutoff Tool Properties example

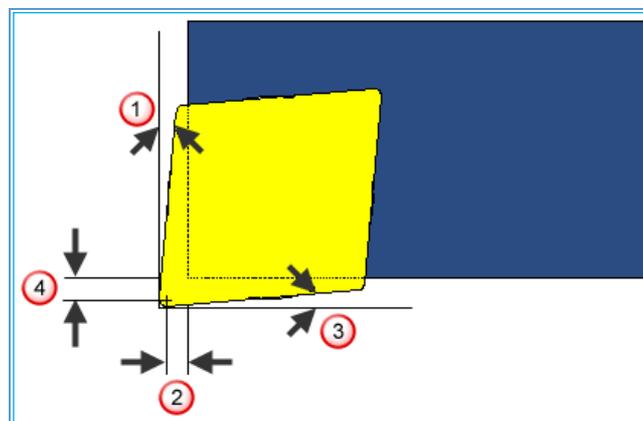


Holder Type: - For grooves possible holder types are:

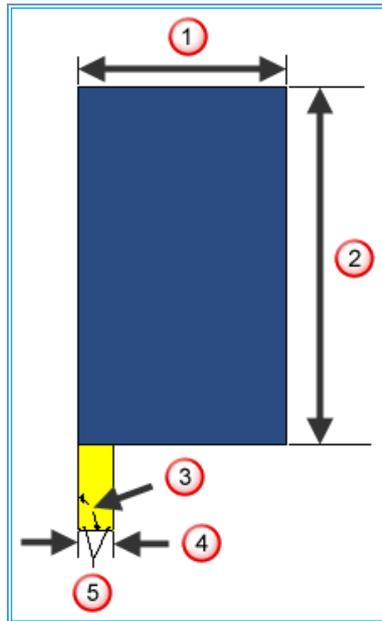
- **OD Groove:** - Select this option for outer diameter grooving tools.
- **ID Groove:** - Select this option for inner diameter grooving tools.
- **Face Groove:** - Select this option for face grooving tools.
- **Cutting:** - Select this option for Cutoff tools.

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

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

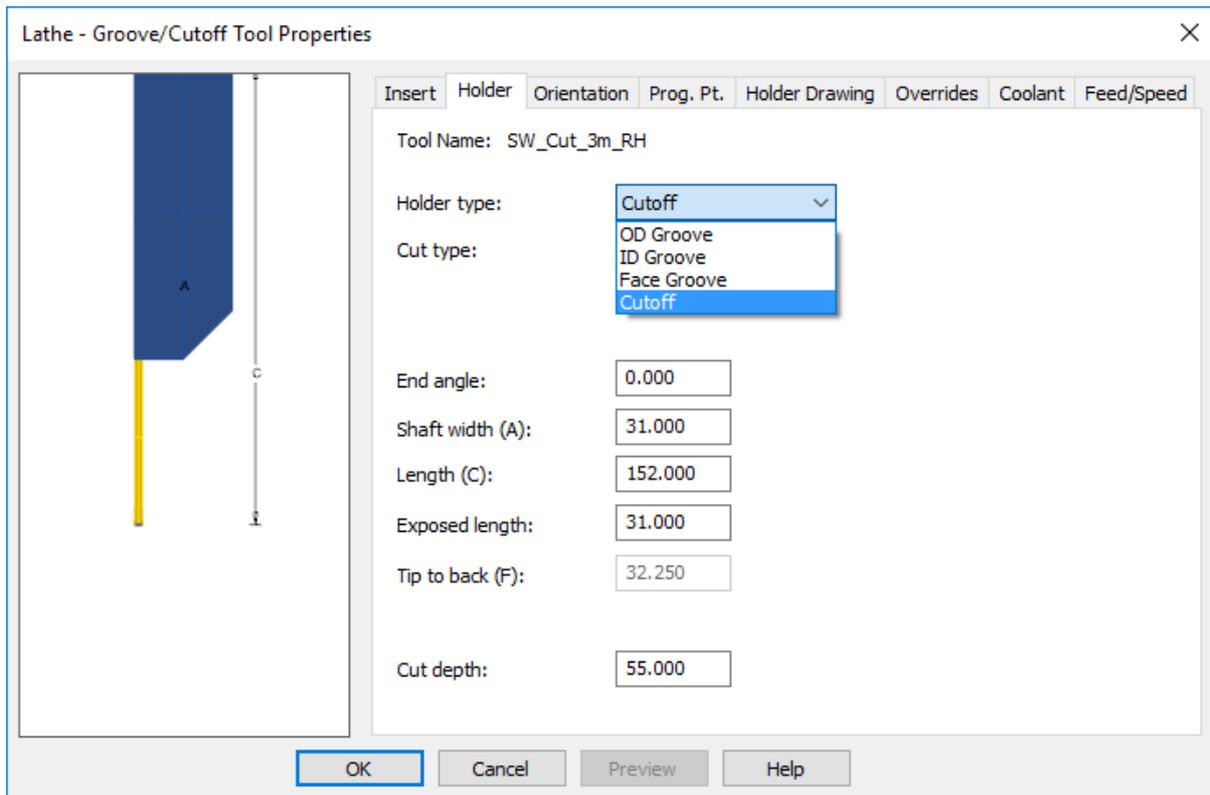


① End Angle ② End Clearance ③ Side Angle ④ Side Clearance

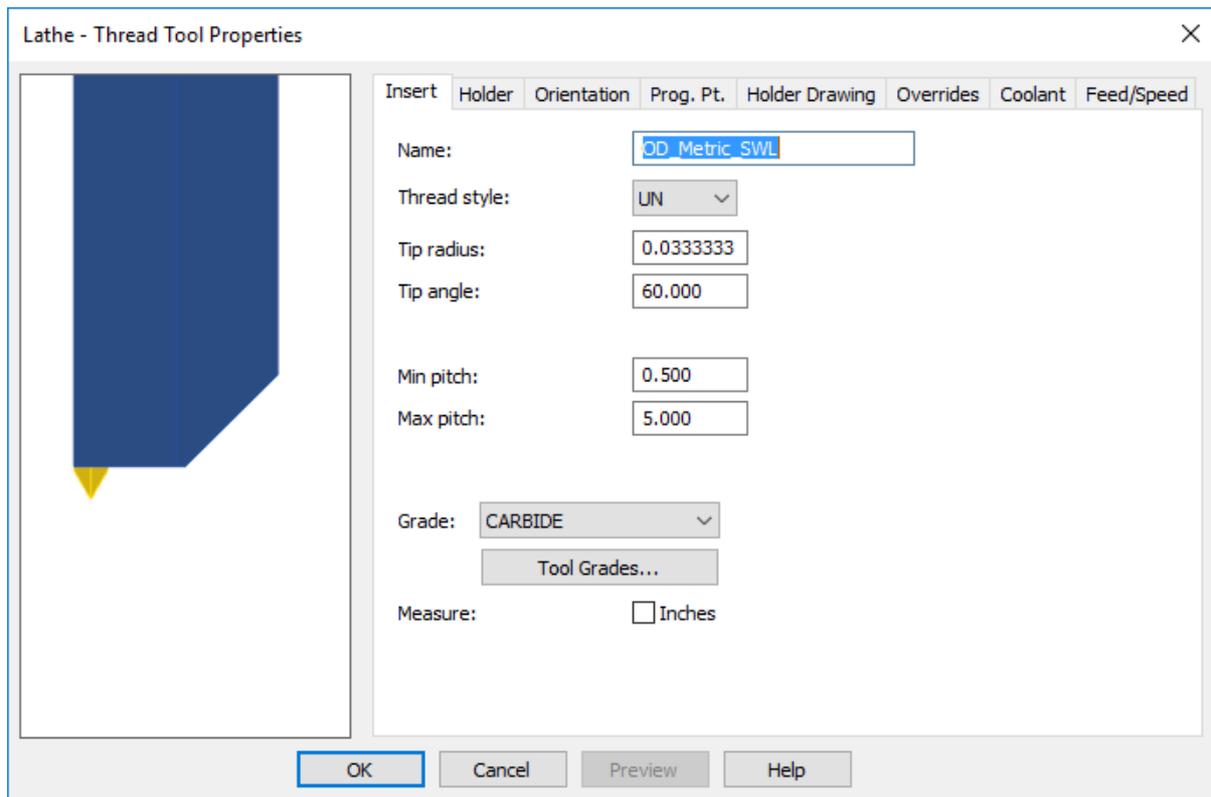


- ① Shaft Diameter (D) ② Length (C) ③ Tip Angle ④ Width ⑤ Tip Radius.

Lathe - Groove/Cutoff Holder tool Properties example



Lathe – Thread Tool Properties example.



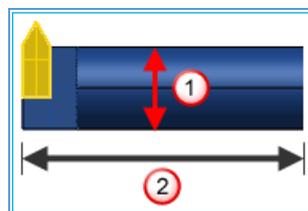
Name - is a string that identifies the tool. The name must be unique among all the tools in the crib. **Tip Radius** is the radius of the cutting tip of the insert. See diagram.

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

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



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



① **Shaft Diameter**

② **Length**

Holder Type - For threads possible holder types are:

OD Threads - Select this value for outer diameter threading tools

ID Threads - Select this value for inner diameter threading tools

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

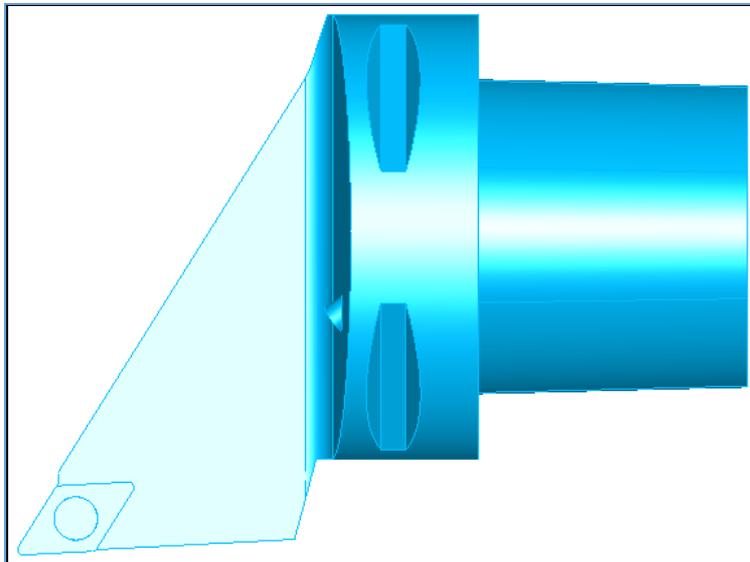
Side Cut - The tool cuts in a direction perpendicular with the length of the holder.

Adding a Solid Model Turning Tool

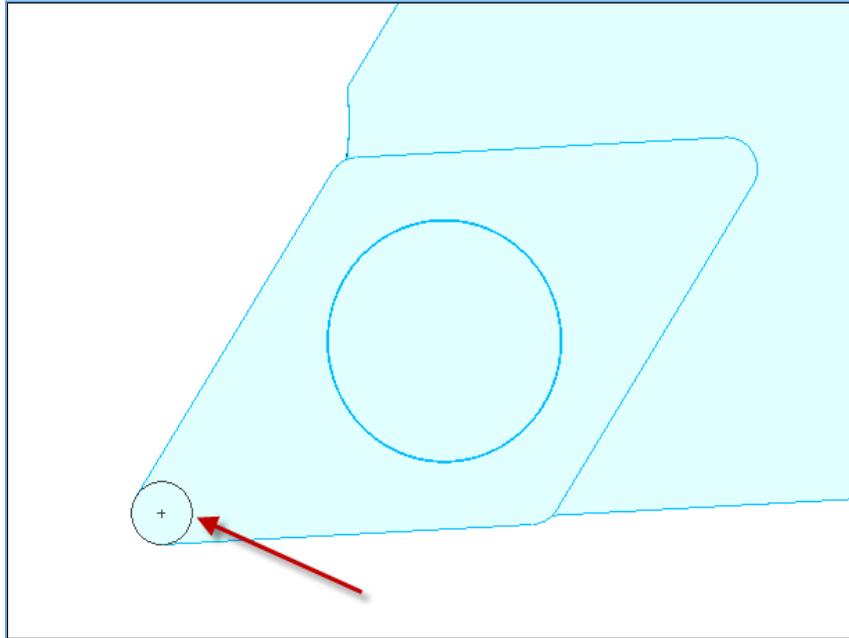
FeatureCAM has the ability to use Solid model Turning Tools and add them to the tooling database. The following steps, take you through the importing of the Solid Model tool and transforming it to the correct position. Then adding the tool to the tool database.

- 1 Open a new Turning document.

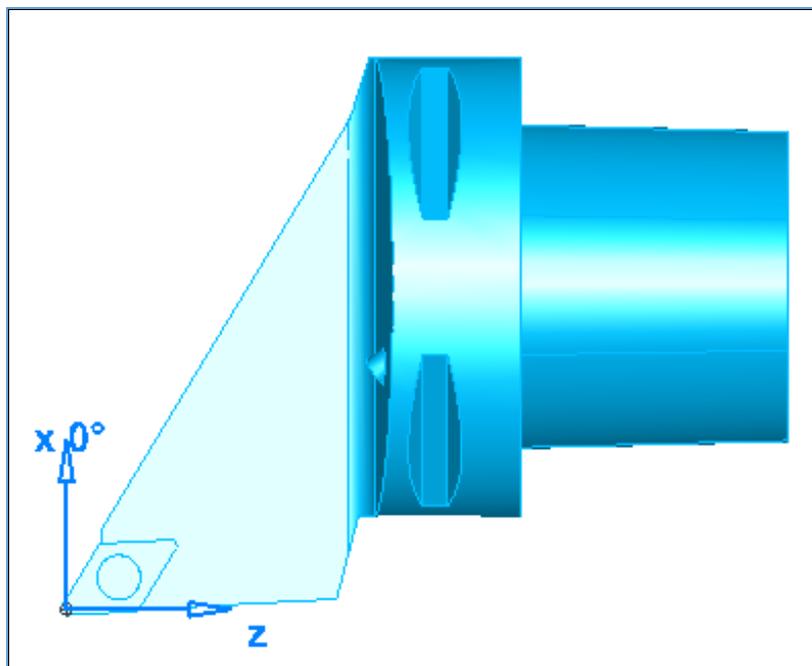
Import the solid model **from C:\Training_Data\FeatureCAM Course Data 2017\Tooling\Turning Tools C6-SDJCR-45065-11-635.x_t**



- 2 Switch on the **Stock axis** in **Part View**. You will notice that the tool tip radius centre point 0.8 is **not** located in the correct position. The centre point of the tip radius must be at X0, Y0, Z0. (Stock Axis).
- 3 Draw a circle on the tip radius. To do this use **Circle from 3 points**  from the geometry menu. Make sure you have snap to object  switched on.

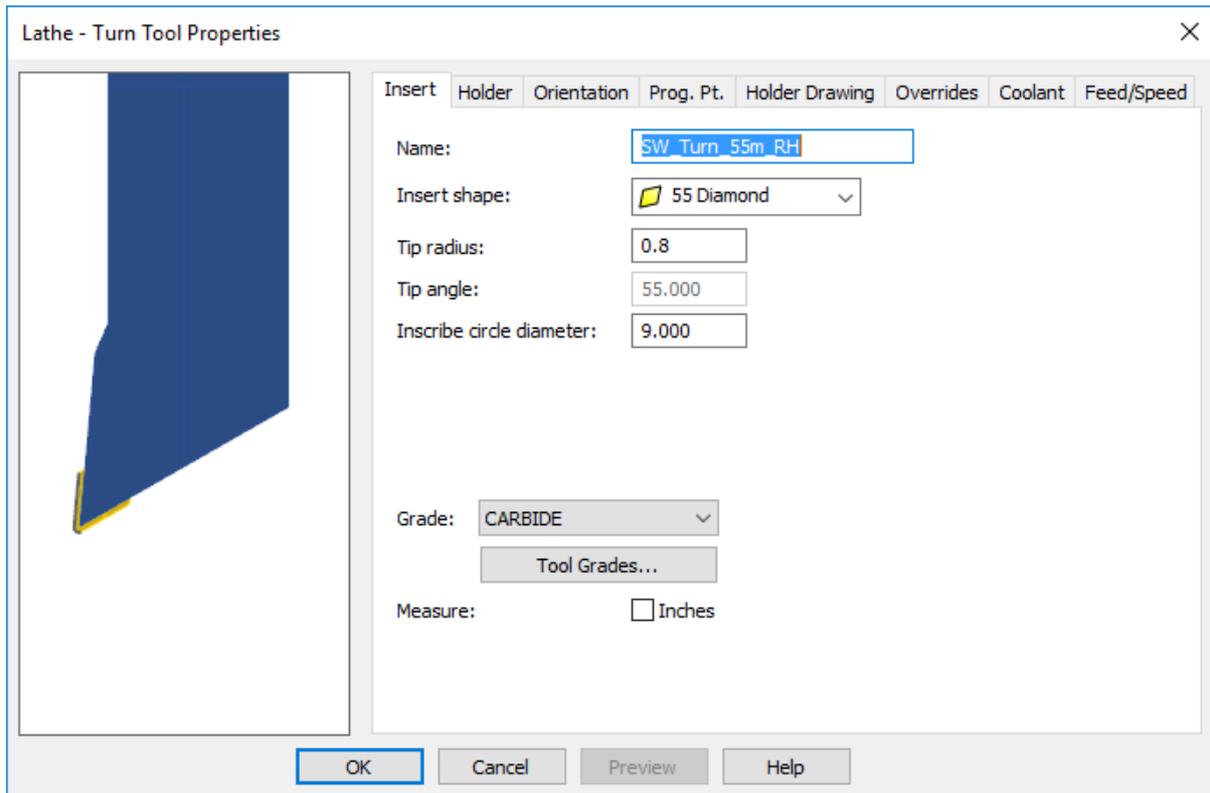


- 4 In **PartView** Select the solid model tool shape in the Solids directory. This will make it active. Then from the **Edit** menu select **Transform/Translate/**
- 5 **Move-From**. Snap to the circle centre **To X0, Y0, Z0**.
- 6 Select **Ok** to action the process.
- 7 Switch on the **Stock Axis**. You will see the following.

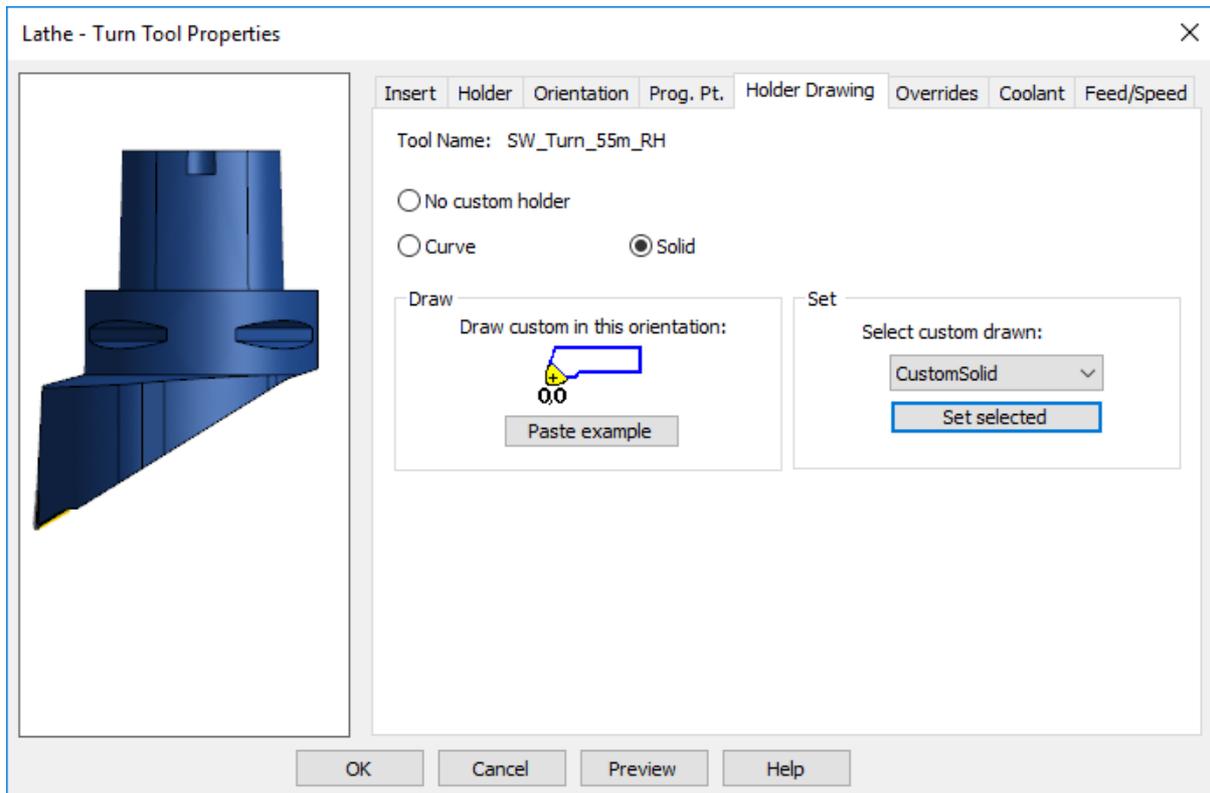


The bottom tool will be used for Back Turning.

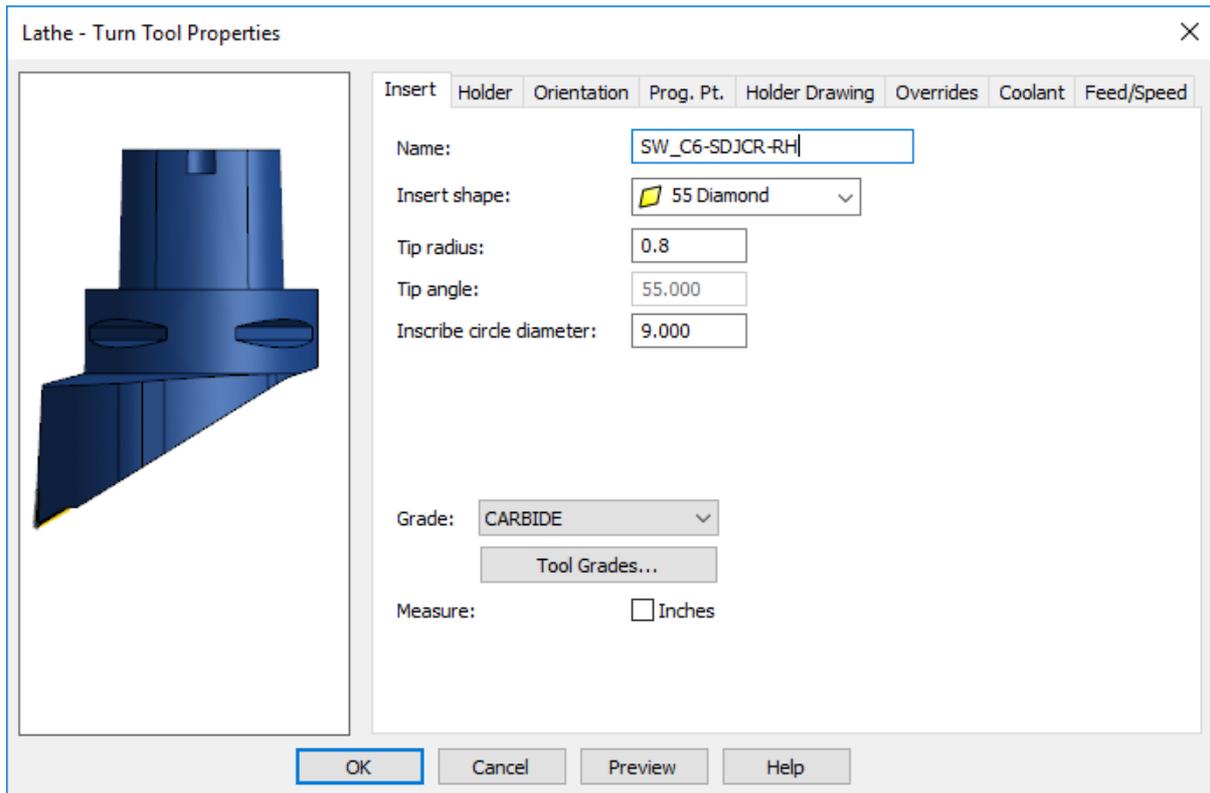
- 8 Open the Tool manager from the Manufacturing menu.
- 9 Select the **SW_Turn_55m_RH** turning tool by double clicking on that tool.



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



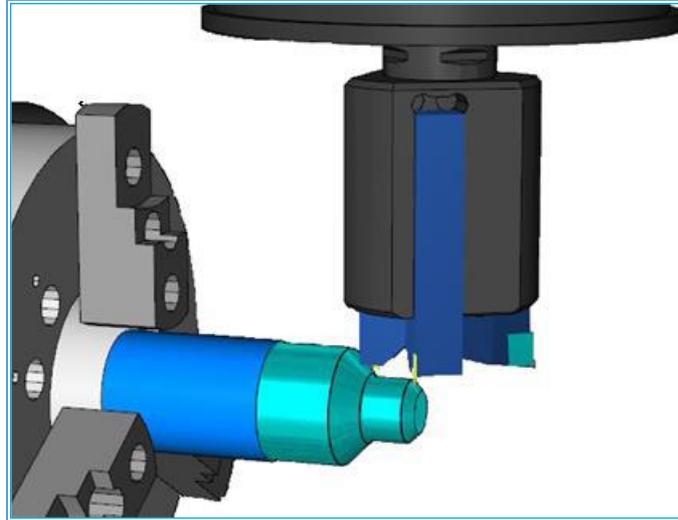
11 Change the name of the Tool to **SW_C6-SDJCR-RH**



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

Mini-turrets

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



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



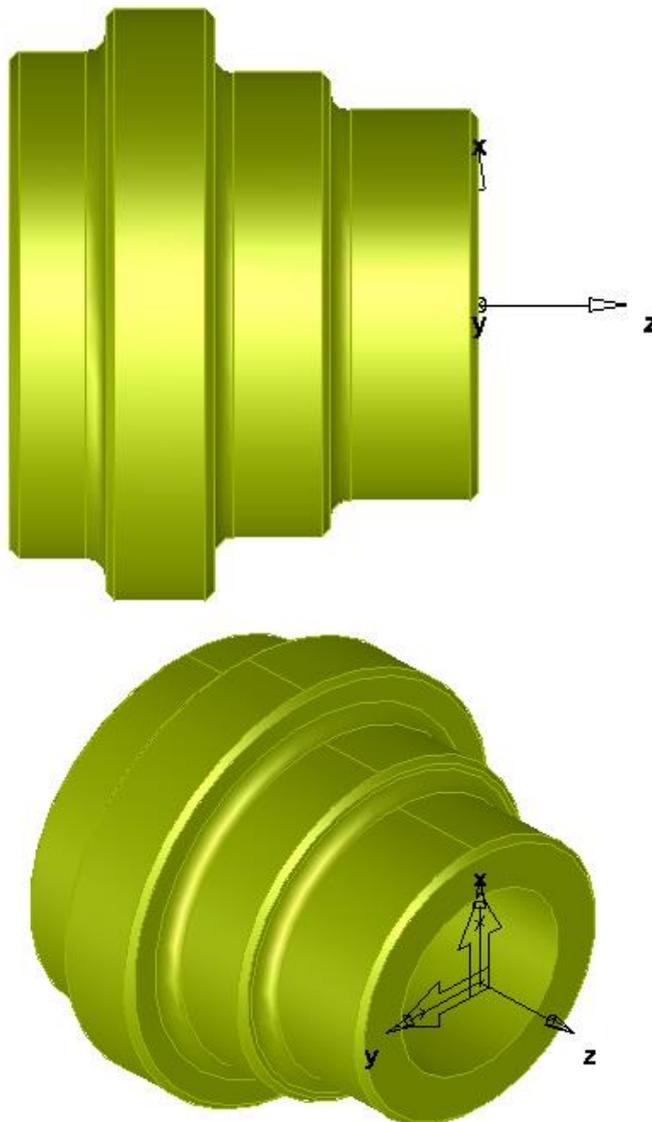
Contents

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

AFR Turning Example 1 - Part4 Turning

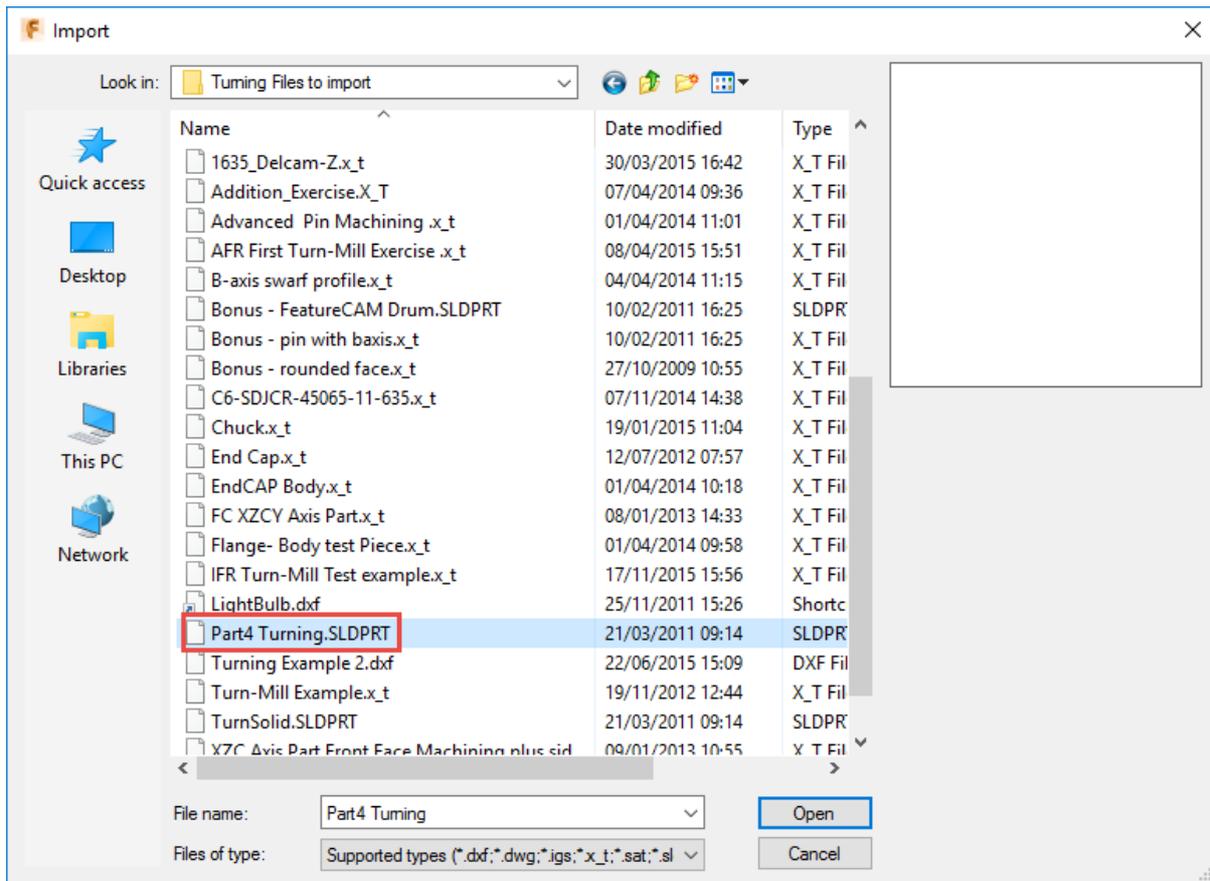
Automatic Feature Recognition recognises different features on solid models and automatically creates toolpaths for different setups. FeatureCAM will select the relevant tooling, but will warn you if it cannot find the correct tooling. Using the **AFR Wizard**, this will guide you through the process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

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

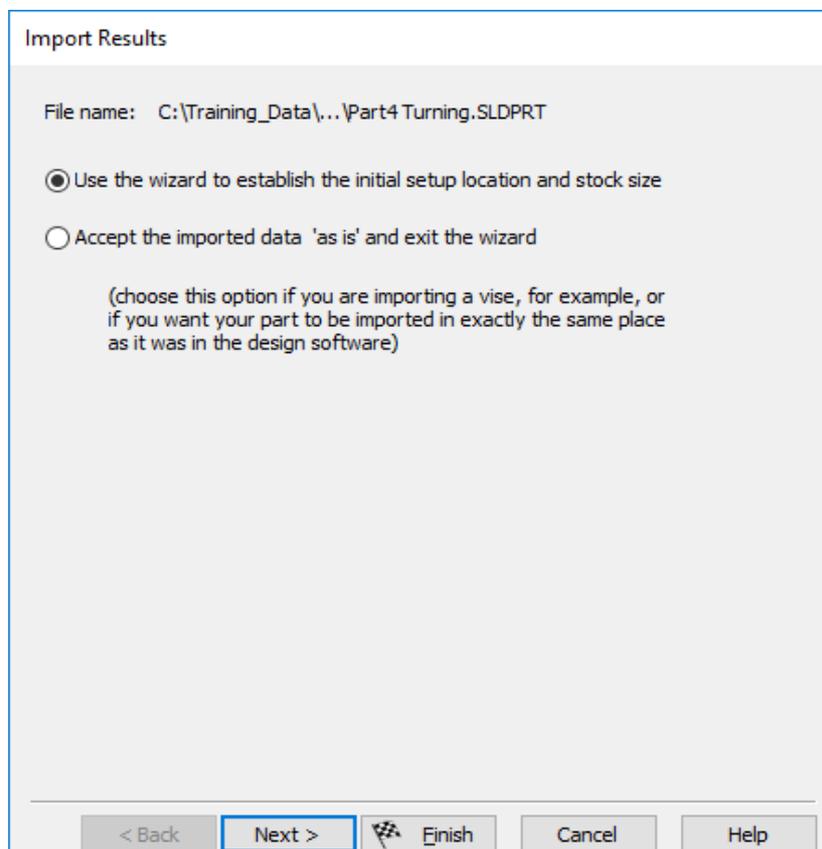


- 2 We will start by importing the model.
- 3 Go to **File>Import C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**.

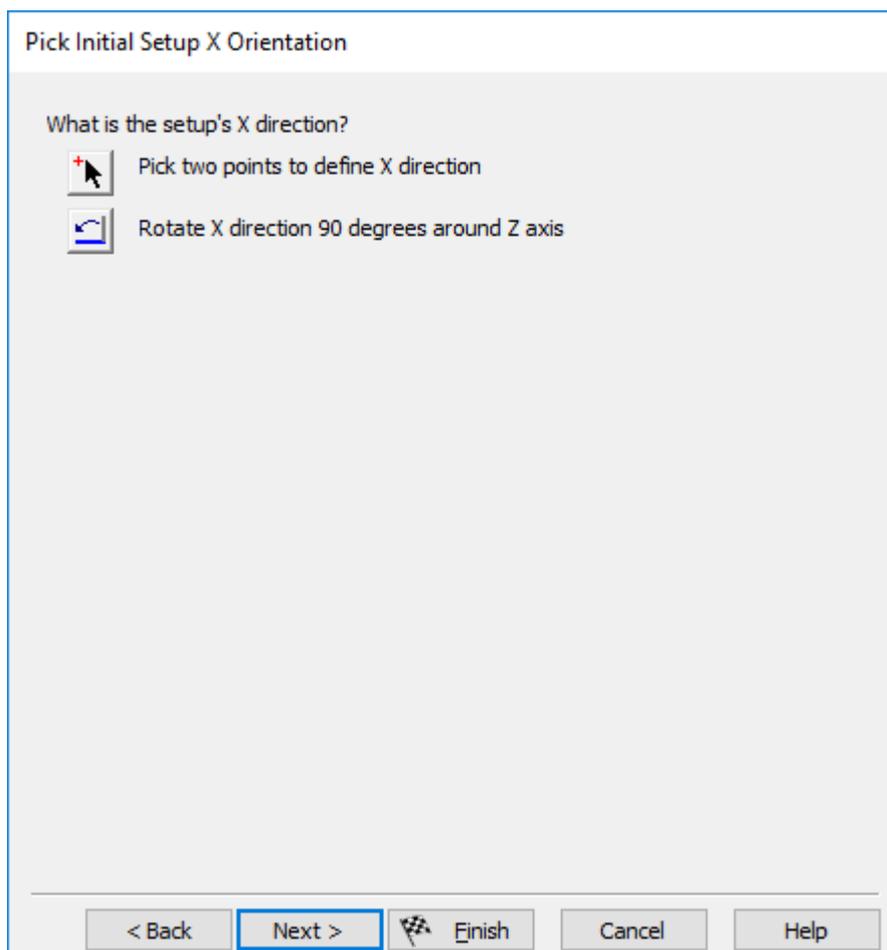
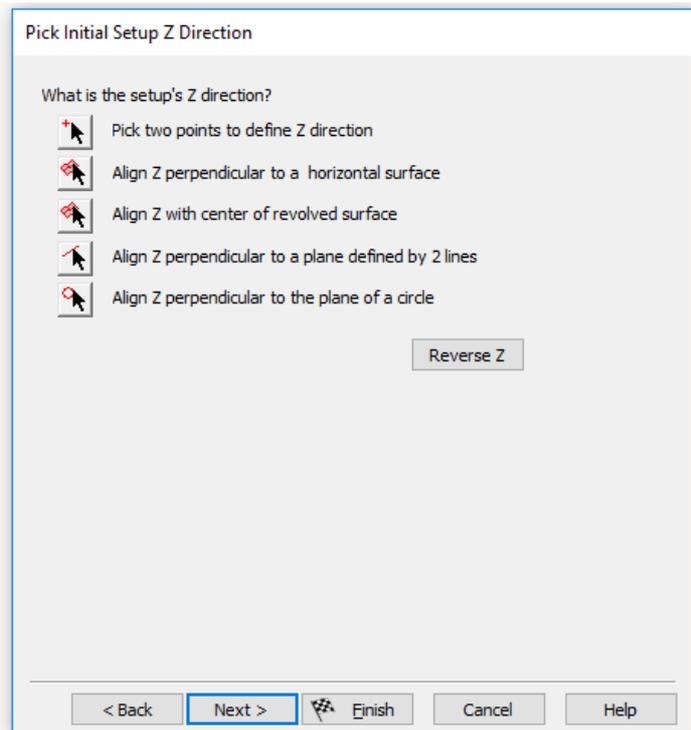
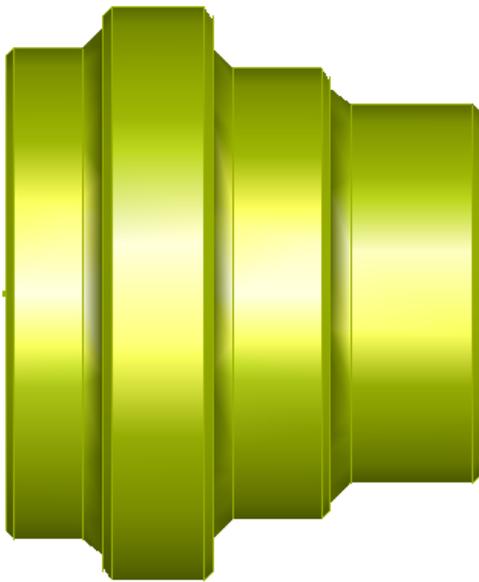
4 Import the following part. **Part4 Turning.SLDPRT**



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

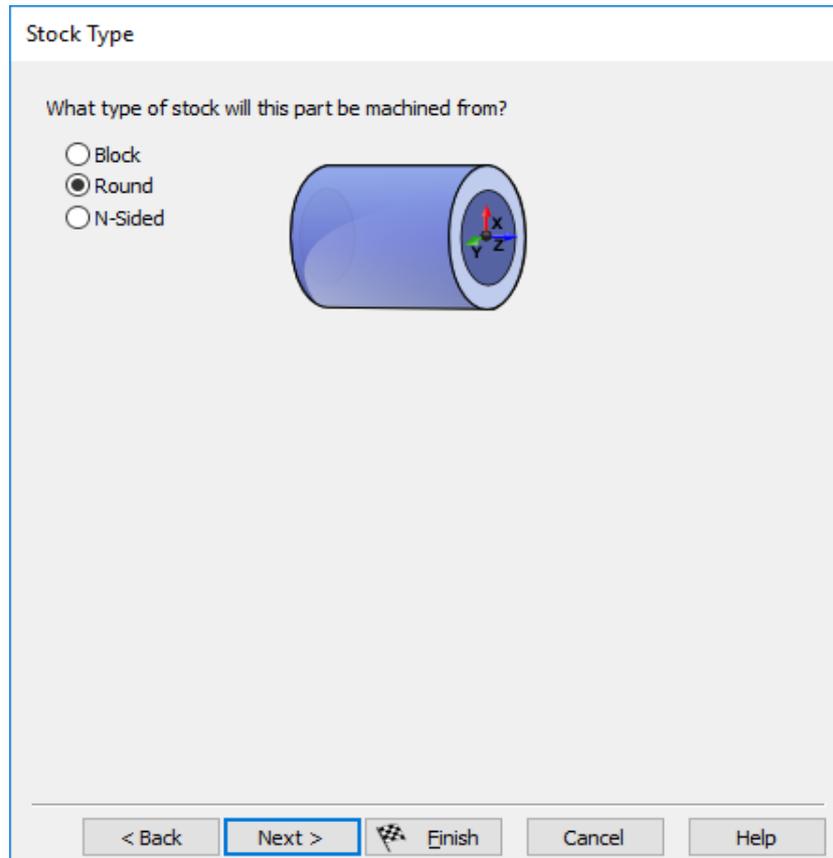


6 Is the part properly aligned with the Z axis? In this case the answer is **yes**.

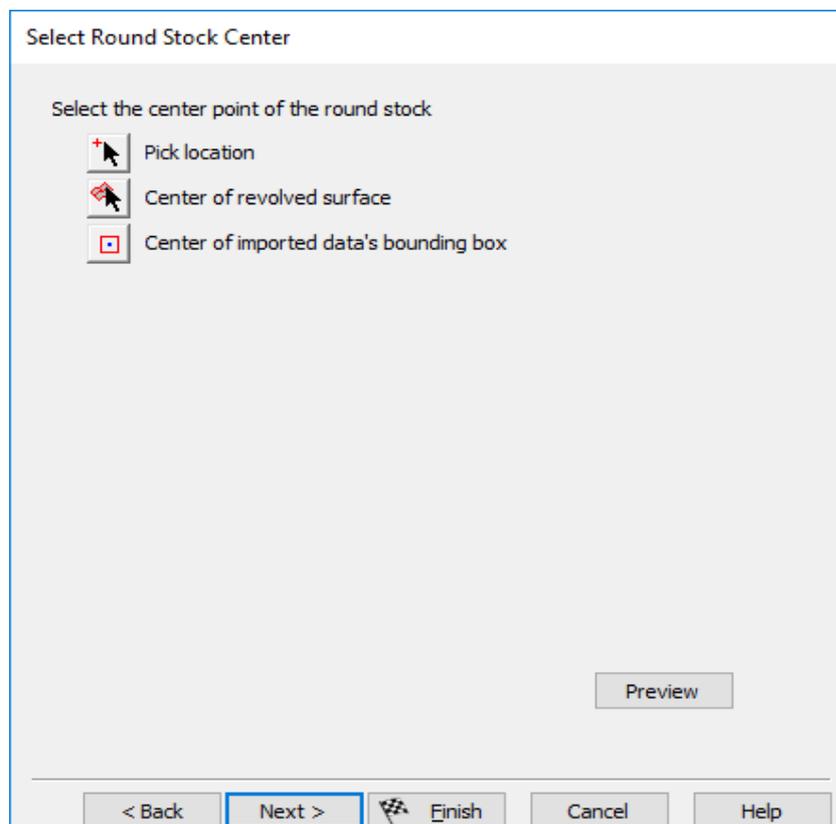


7 Select **Next**.

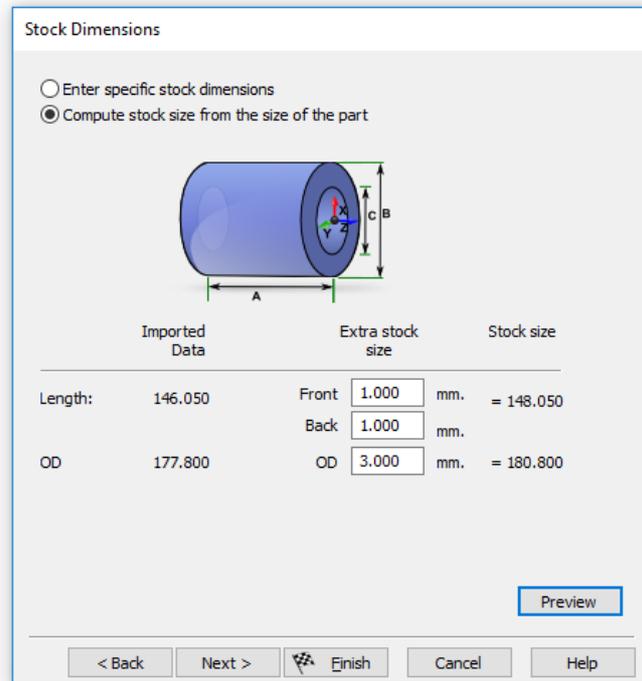
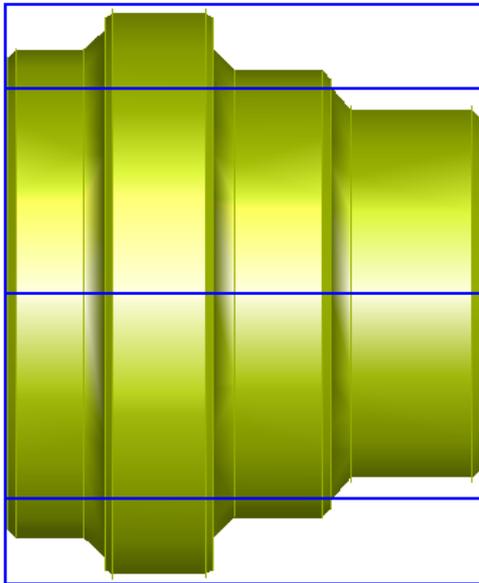
- 8 The next Menu gives you the option to select the Stock Type.
- 9 For our turned part select **Round** as shown.



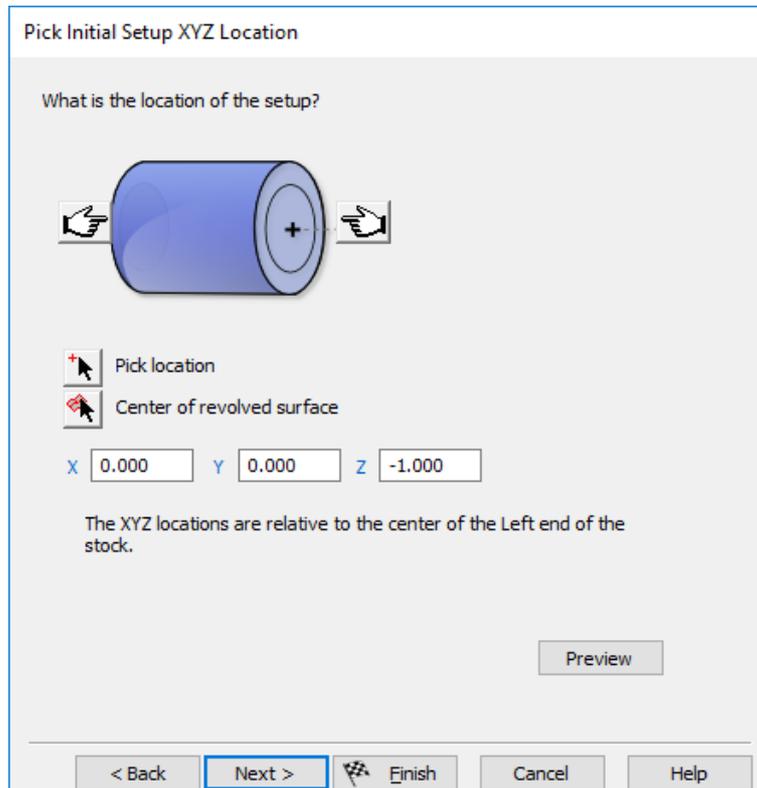
- 10 The next available menu will ask you to select the **Centre** of the round stock.

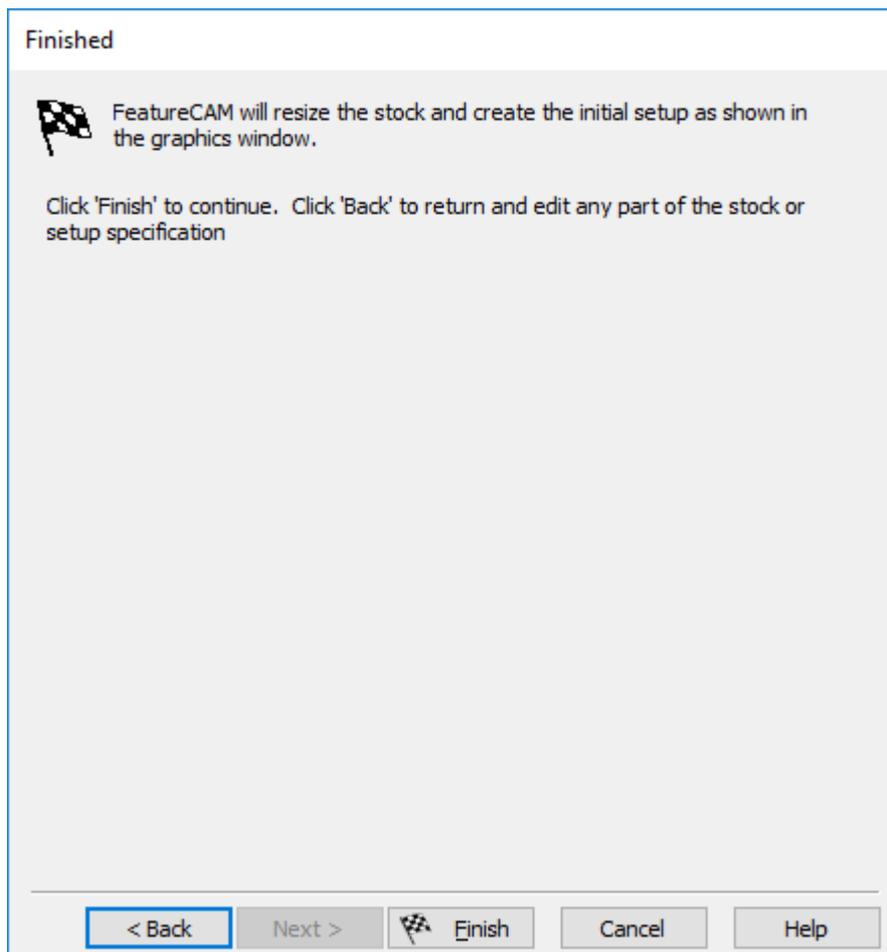
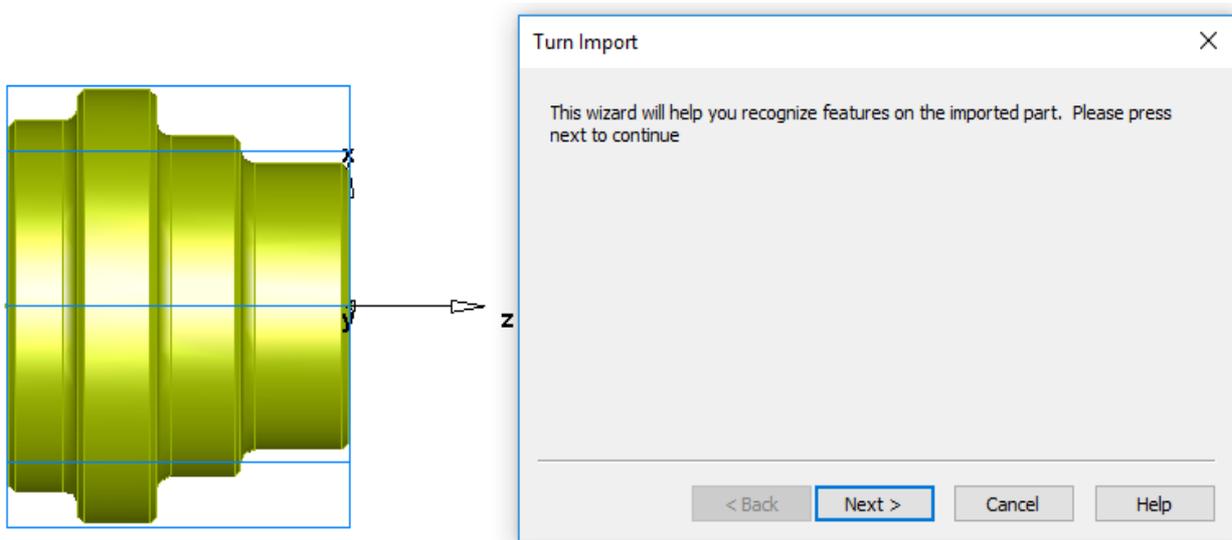


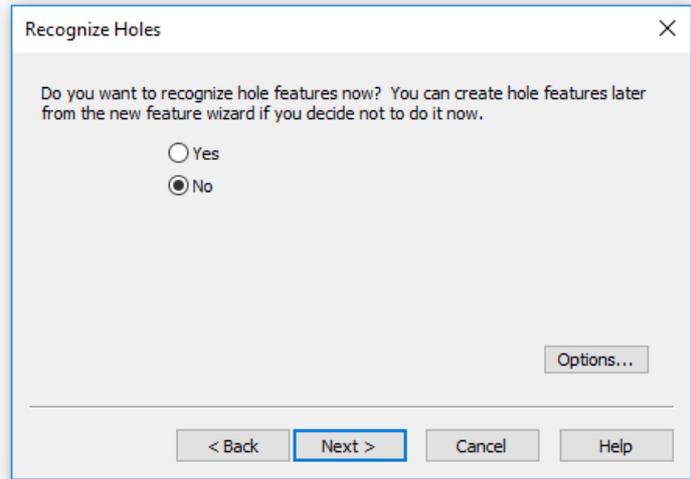
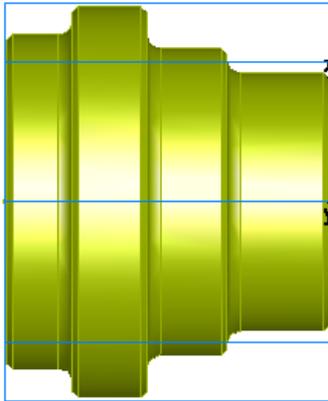
- 11 Select **Next**.
- 12 The next available menu will give you two options. **Option1- Enter specific stock dimensions** or **Option 2 – Compute stock size from the size of the part**. Please select **option 2**.



- 13 Enter the above coordinates. This allows **1mm** on the front and back faces plus **3mm** on diameter.
- 14 Select Next. You will notice that the Z datum shows **-1.000**. FeatureCAM knows how much material you have left on from the previous menu.

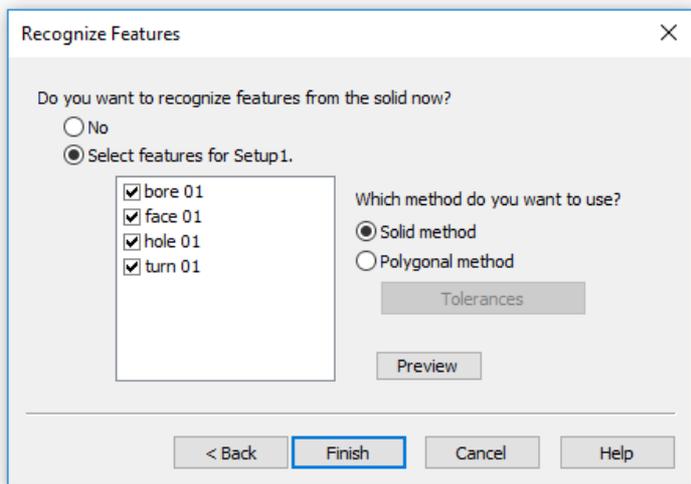
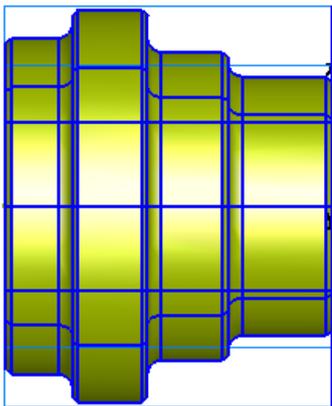


15 Select Next.**16** Select **Finish**. The following menu will be shown.**17** Select **Next**.**18** You will be asked if you want to **Recognise Hole Features now**. Select **No**.



19 Select **Next**.

20 Please select **Select features for Setup1**.



21 Select **Finish**.

22 In your operation list you will get a warning?

RESULTS

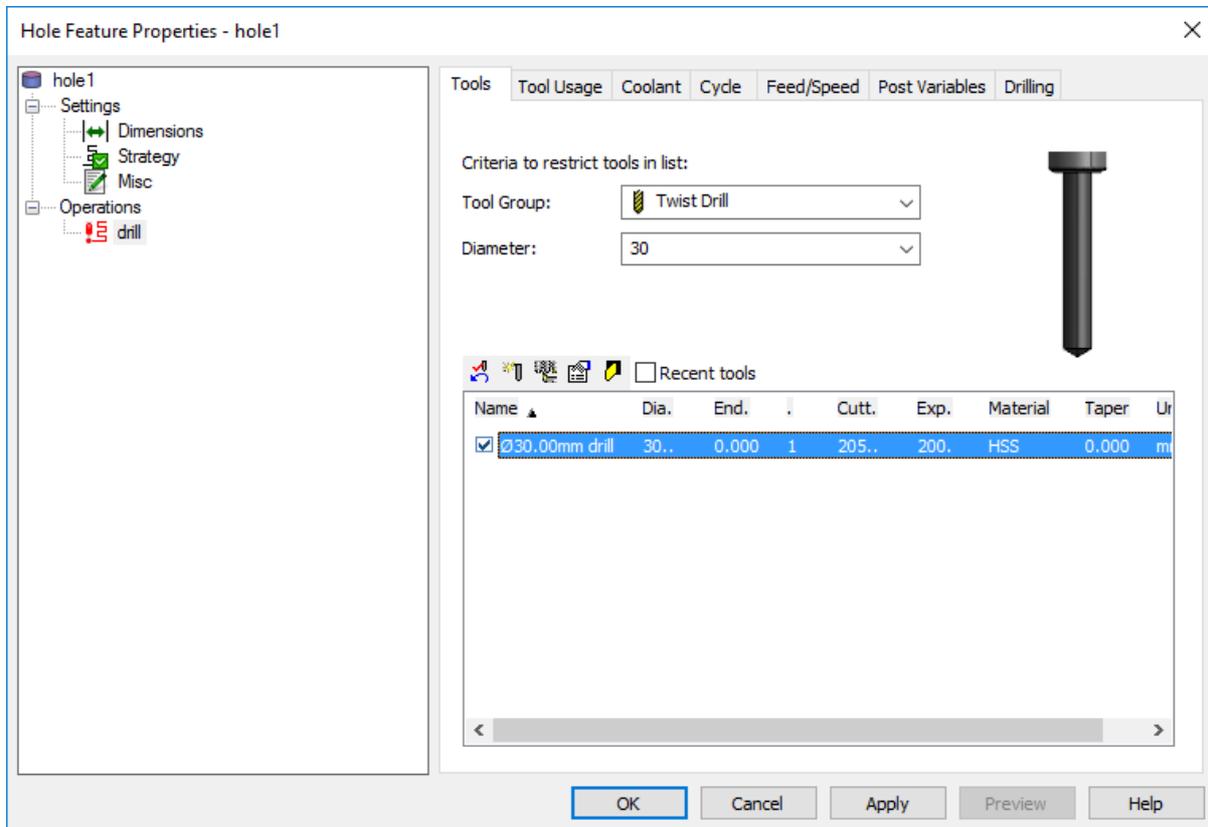
Operation List

Automatic Ordering
 Manual Ordering

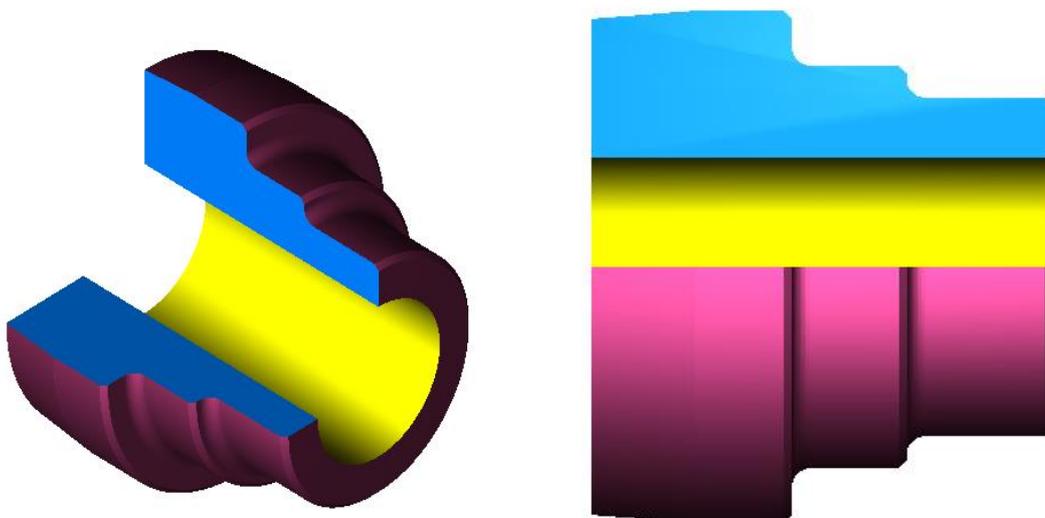
R	Operation	Feature	Tool	Feed	Speed	Depth
	rough pass 1	face1	SW_Tum_80m...	0.381 MMPR	366 SMM CW	90.400 mm
	rough pass 1	tum1	SW_Tum_80m...	0.381 MMPR	366 SMM CW	32.702 mm
	finish	face1	SW_Tum_80m...	0.152 MMPR	488 SMM CW	90.400 mm
	finish	tum1	SW_Tum_80m...	0.152 MMPR	488 SMM CW	32.702 mm
!	drill	hole1	*****	0.000 MMPR	0 RPM	146.050 mm
	rough pass 1	bore1	WN_B_Tum_5...	0.381 MMPR	366 SMM CW	
	finish	bore1	WN_B_Tum_5...	0.152 MMPR	488 SMM CW	

Results

- 23 For the **Hole1** Feature. Adjust the hole roughing tool to a **30mm Drill**. From the **Training_Crib_Metric** crib this will not exist, so create a new tool. Choose an existing drill of a smaller size and change the values. Remember to change the name so you can create a new tool. To do this double click the on the following ***** this will open the tooling form.



- 24 After changing the drill you may also have to change the **minimum Radius boundary** to **15mm** in the **Bore1** Feature. Change depth of cut to **2mm**. Run a **3D Simulation** to view results from the **Automatic Feature Recognition**.
- 25 Remember that a **Bore Feature** has a **Pre-Drill** option that combines the two strategies into one.

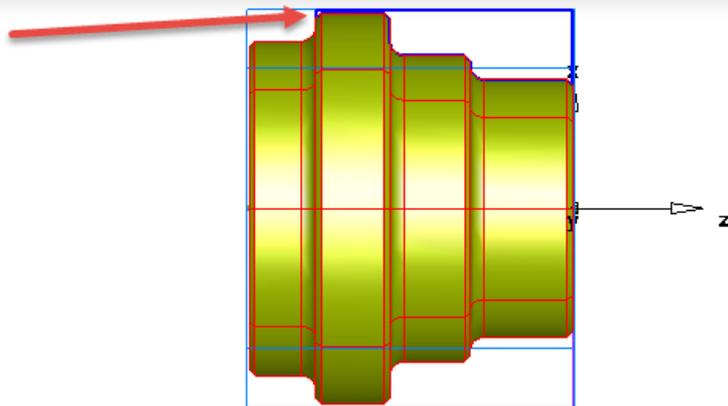
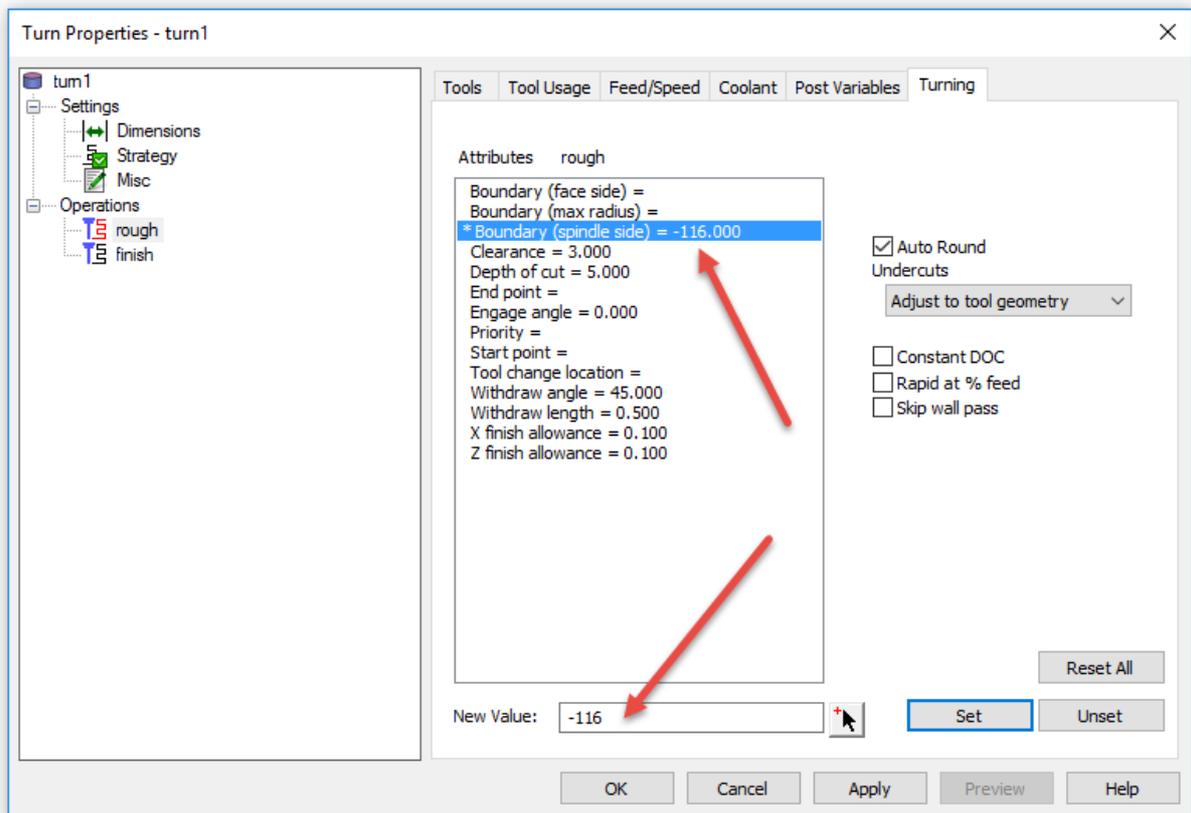




Notice how the tool will machine as much as it can, which depends on which tool is selected. (see image)

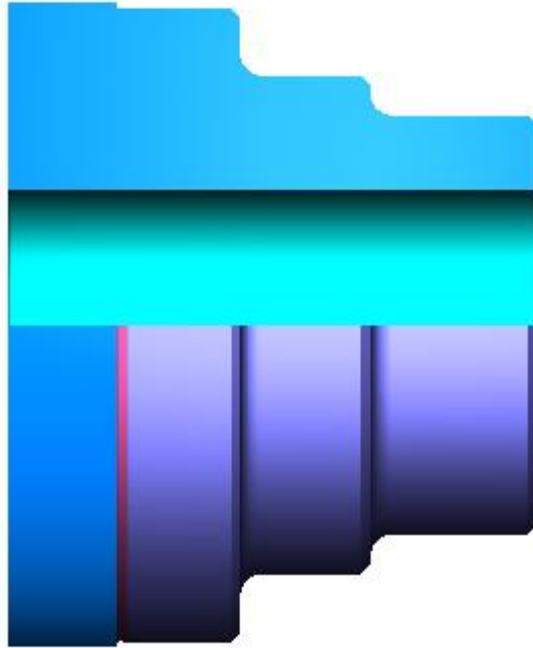
26 Try using **Select left boundary** and limit the length to the back shoulder. The chuck will be damaged if you have not left enough material at the back of the part. Select **Turn1** by double clicking in **Part View**. Select **Roughing**.

27 Select **Pick Z Location**.

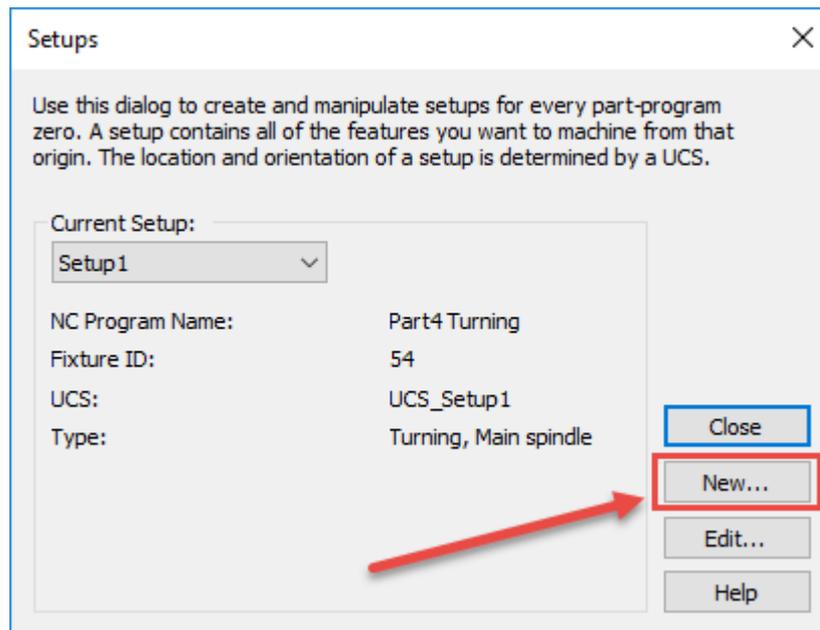


The Z Value is **-116.0** the * denotes that the value has changed.

28 Re-Run the **3D Simulation**.



We can now create **Setup2**. Double click on **Setup1** following menu will appear.



29 Select **New** and the following menu on the next page will be displayed.

Setup - Definition

Please enter the setup name and fixture ID:

Setup name:

Fixture ID:

Part name:

What type of setup do you want to create?

Milling
 Turning
 Wire EDM



< Back **Next >**  Finish Cancel Help

30 Select **Next**.

Setup - Part Program Zero

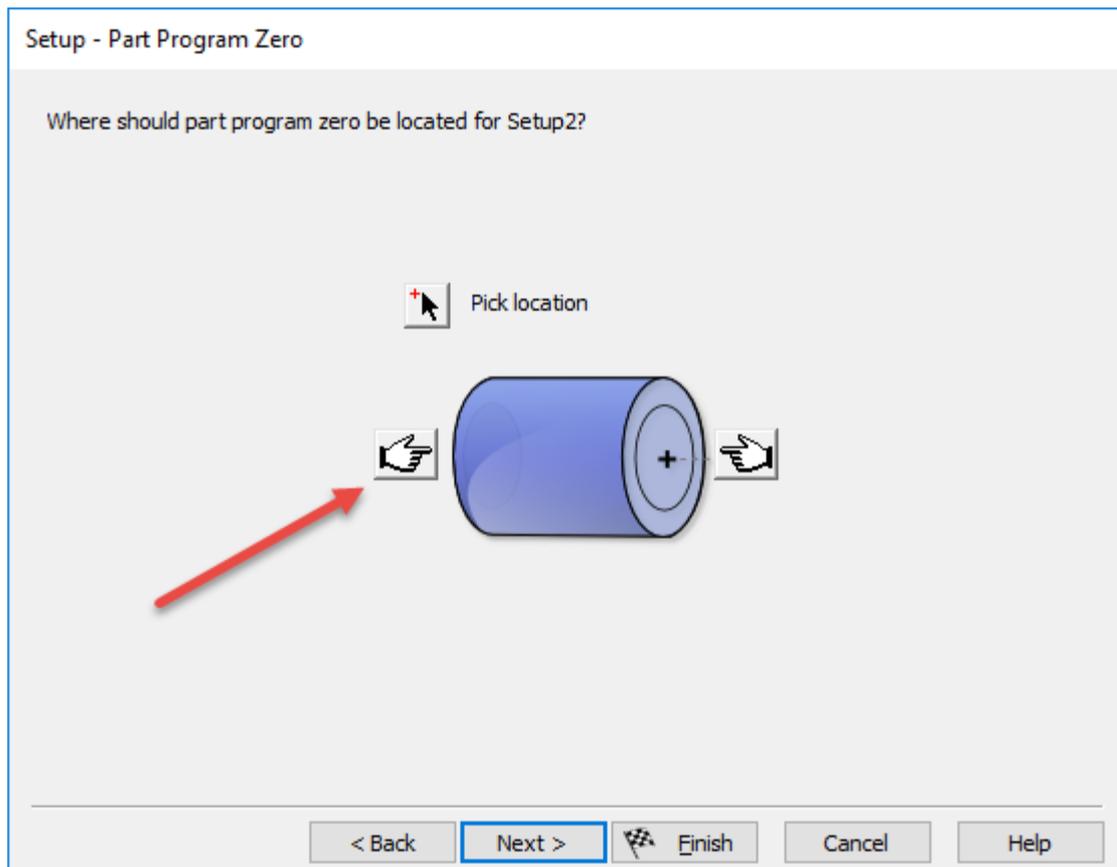
What method do you want to use to define the part program zero location for Setup2?

Align to Stock Face
 Align to Index axis
 Align with existing UCS
 Align to part geometry
 Use current location



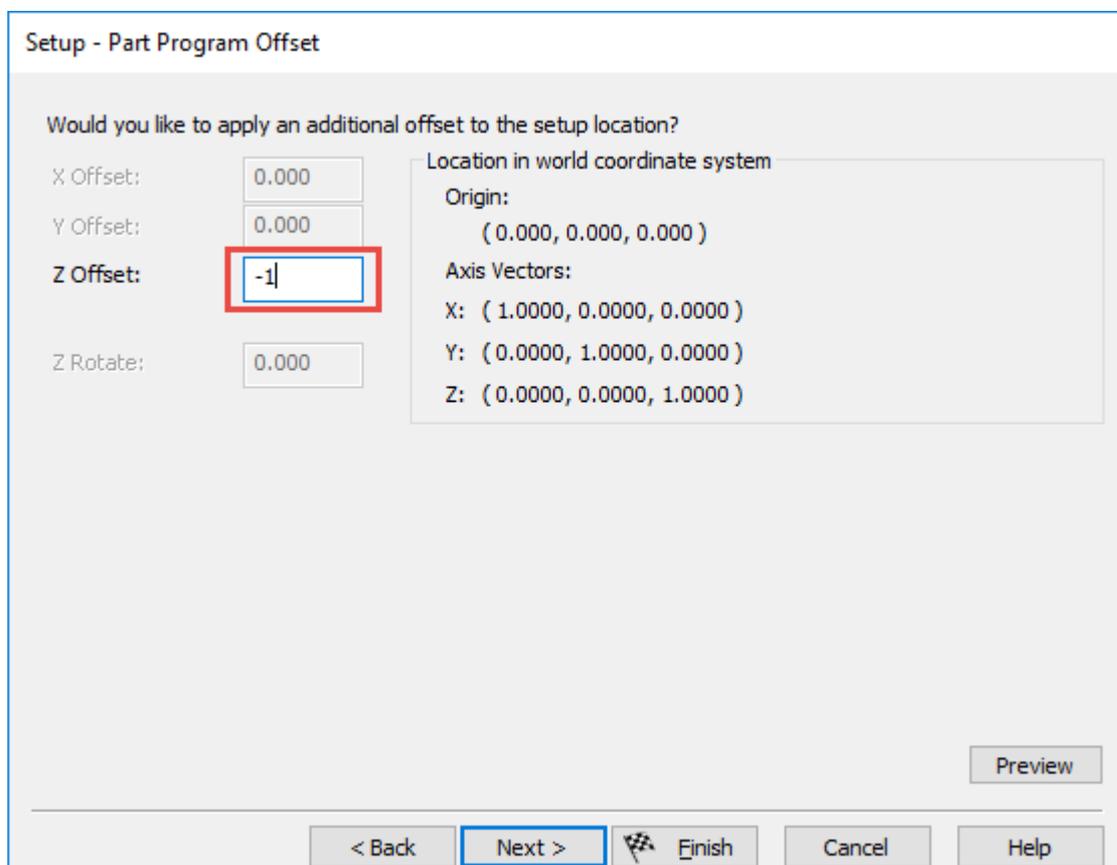
< Back **Next >**  Finish Cancel Help

31 Select **Next**.



32 Select the **left hand** as shown.

33 Select **Next**. Then set the **Z Offset** to **-1** as shown.

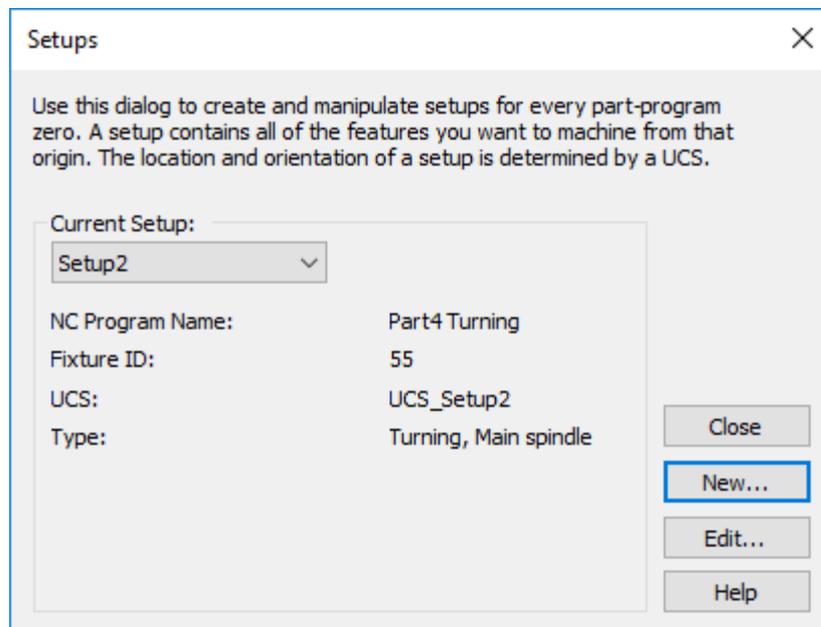
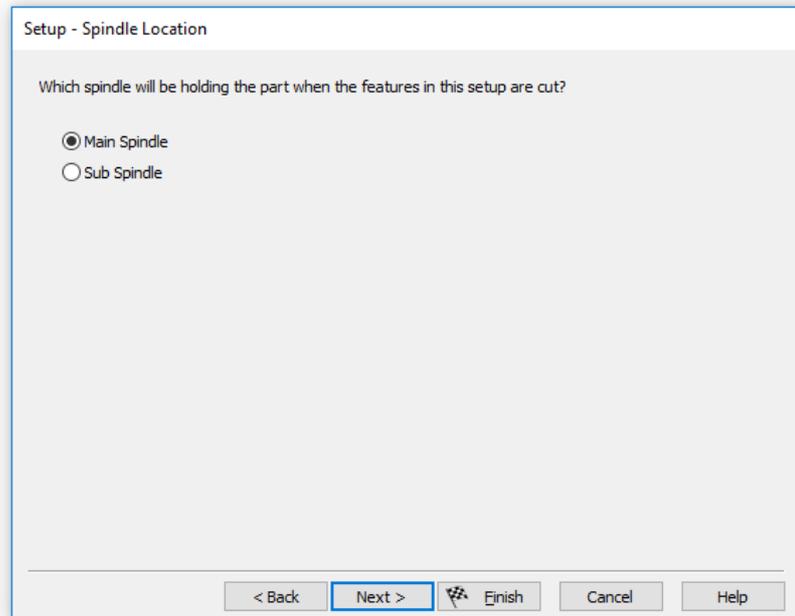
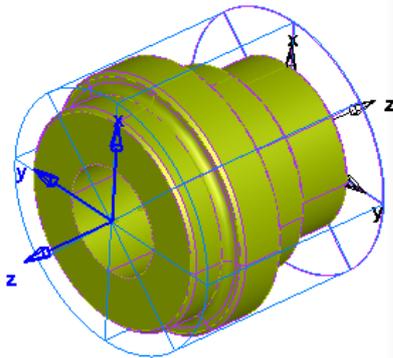




This will set the new datum to the opposite face. This is when you take the job out of the chuck and turn it around if you only have one spindle. If you have a Sub-Spindle please select this option. You will have to use Part Handling to transfer the part into the Sub- Spindle.

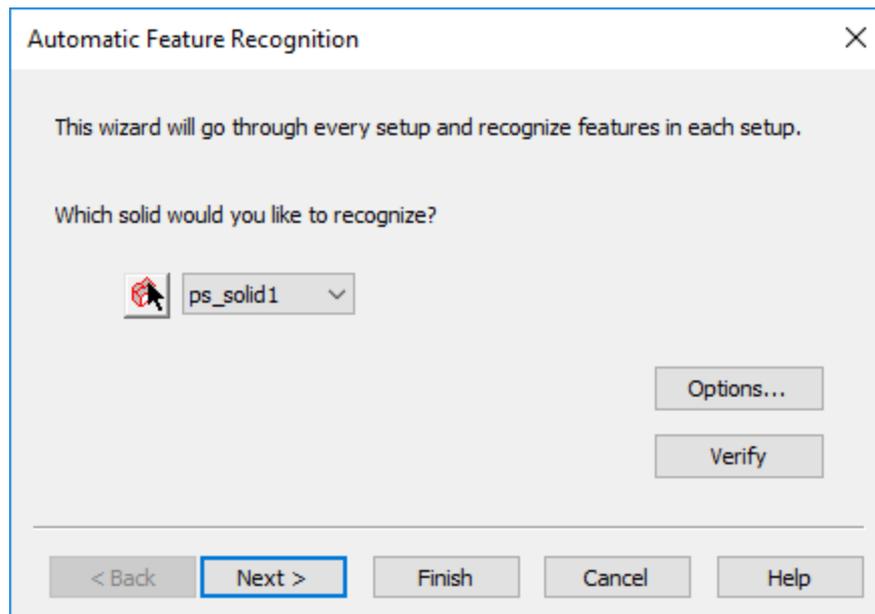


FeatureCAM will ask if you want the **main Spindle** or **Sub-Spindle**. See menu on the next page.



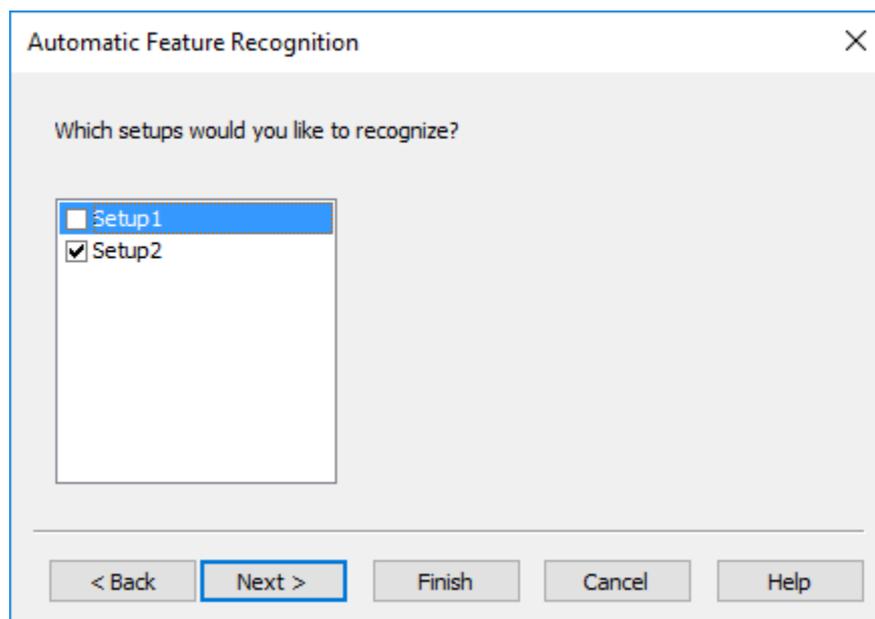
34 We are now ready to apply **AFR** for **setup2**. Select **STEPS** and select **AFR**.

35 From the **Construct** menu please select **Automatic Feature Recognition**.



36 Select **Next**.

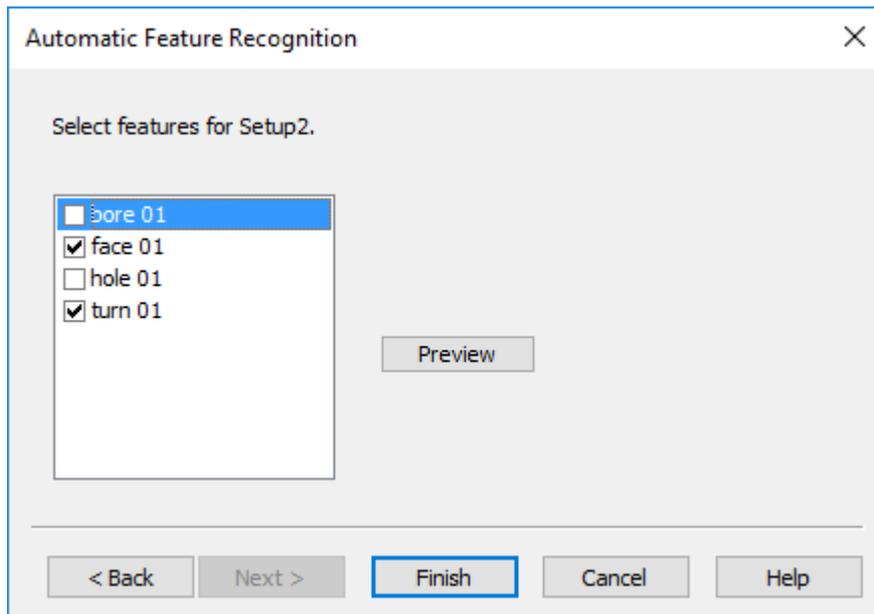
37 Unselect **setup1** so it will only recognise **setup2**



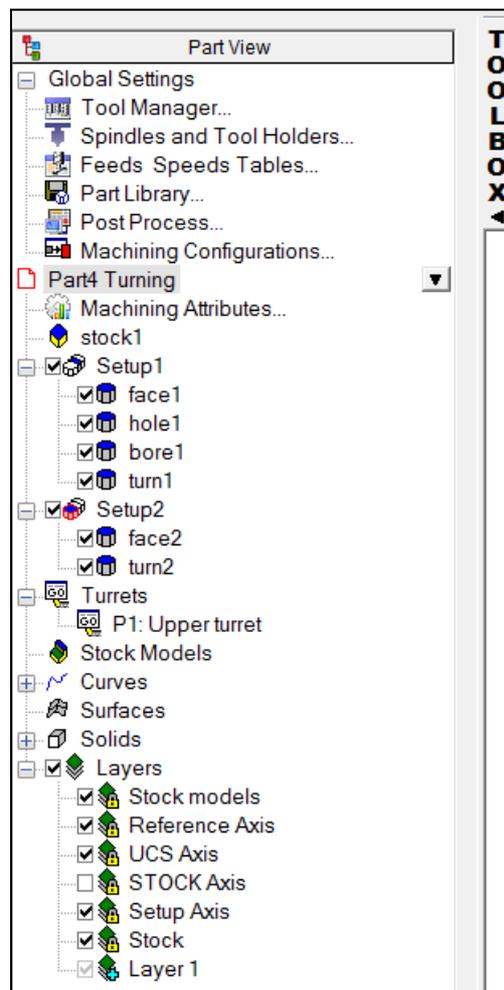
*If you select **Setup1** and **Setup2** this will duplicate operations.*

38 You will be given the option to select which Features are relevant to **Setup2**.

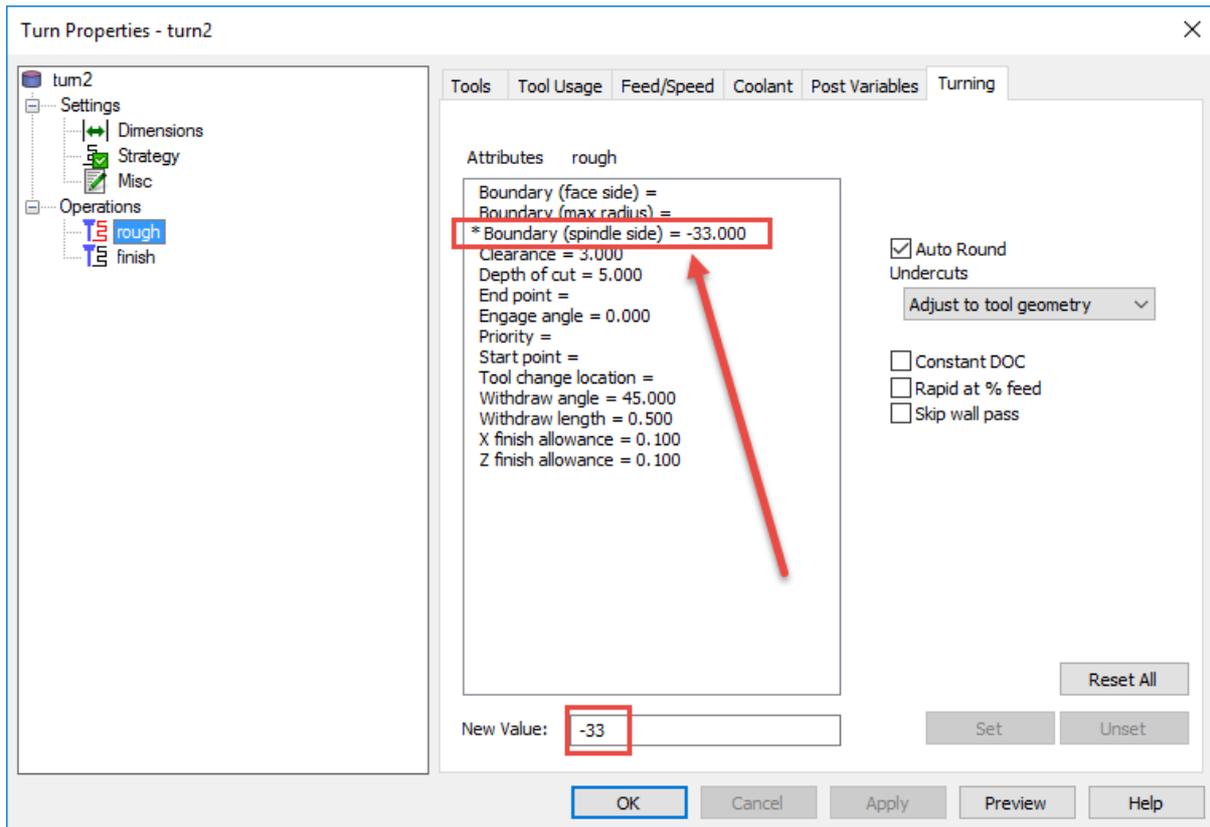
39 Please unselect **Bore 01** and **Hole 01** as shown on the next page.



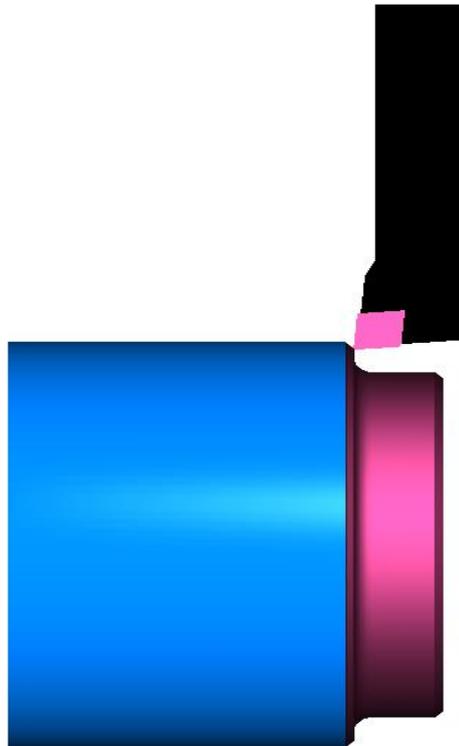
40 Select **Finish**.



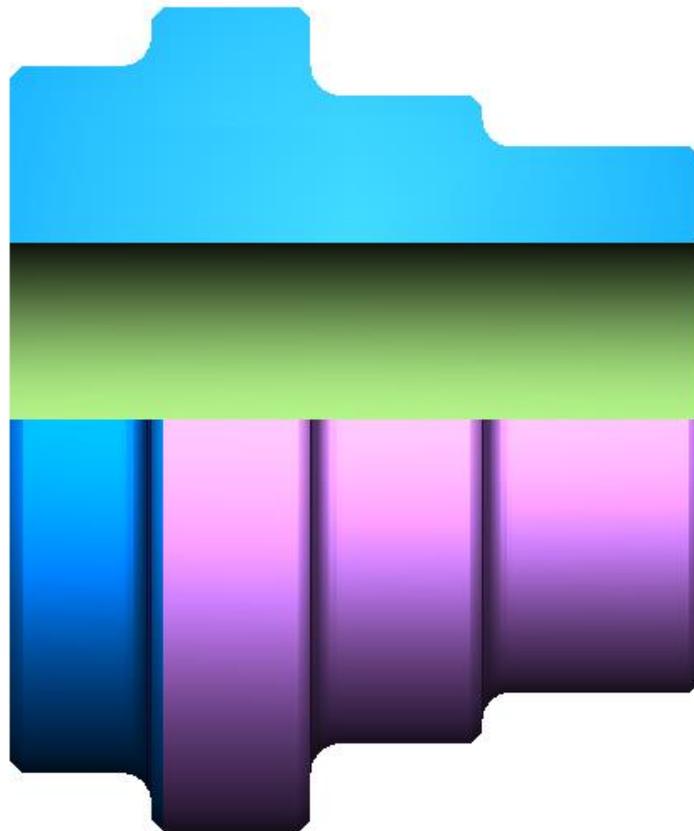
41 We need to adjust **turn2** in **Setup2**. Double click on **turn2** in **Setup2**. Select the Rough pass in the Operation list this will show the following menu.



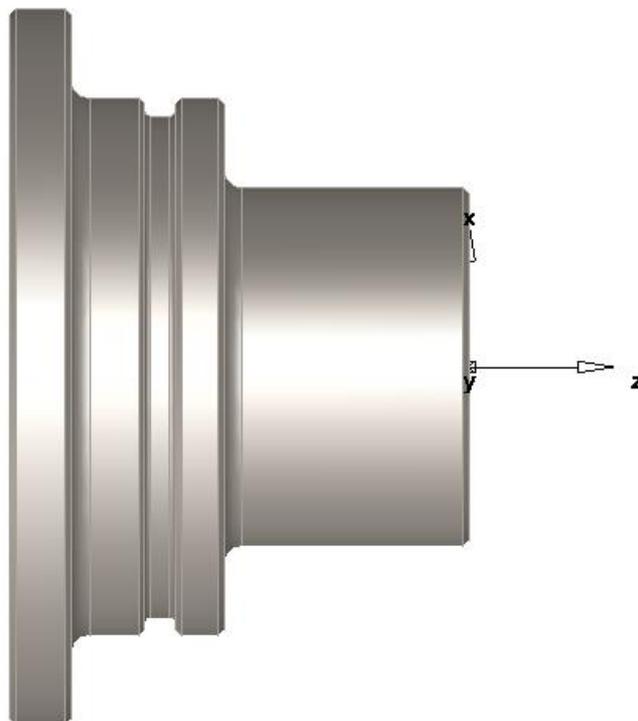
- 42** This will now only allow the machining up to the point shown. See **3D Simulation**.



- 43** The component is now finished see image of both operations on the next page. Change the Post Processor and look at the code output.



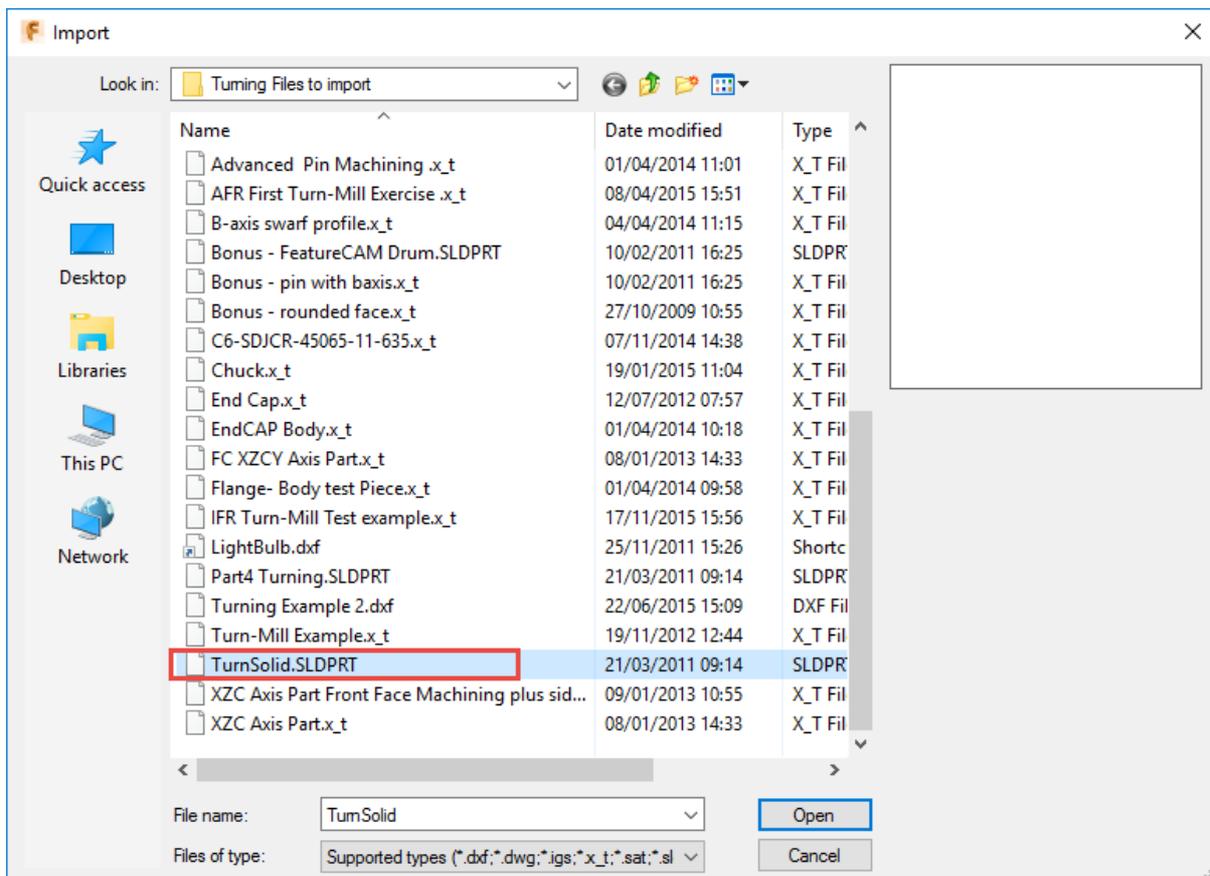
AFR Turning Example 2 - Turnsolid



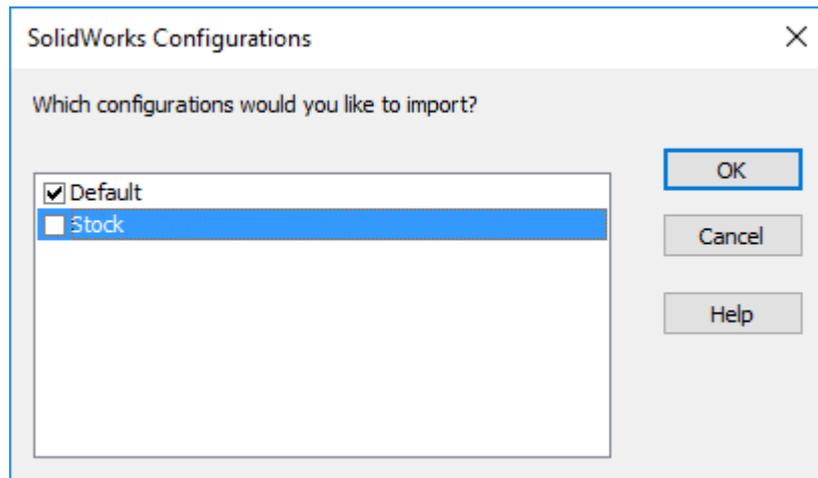
Turnsolid - Manipulating the model to the correct orientation ready for AFR.



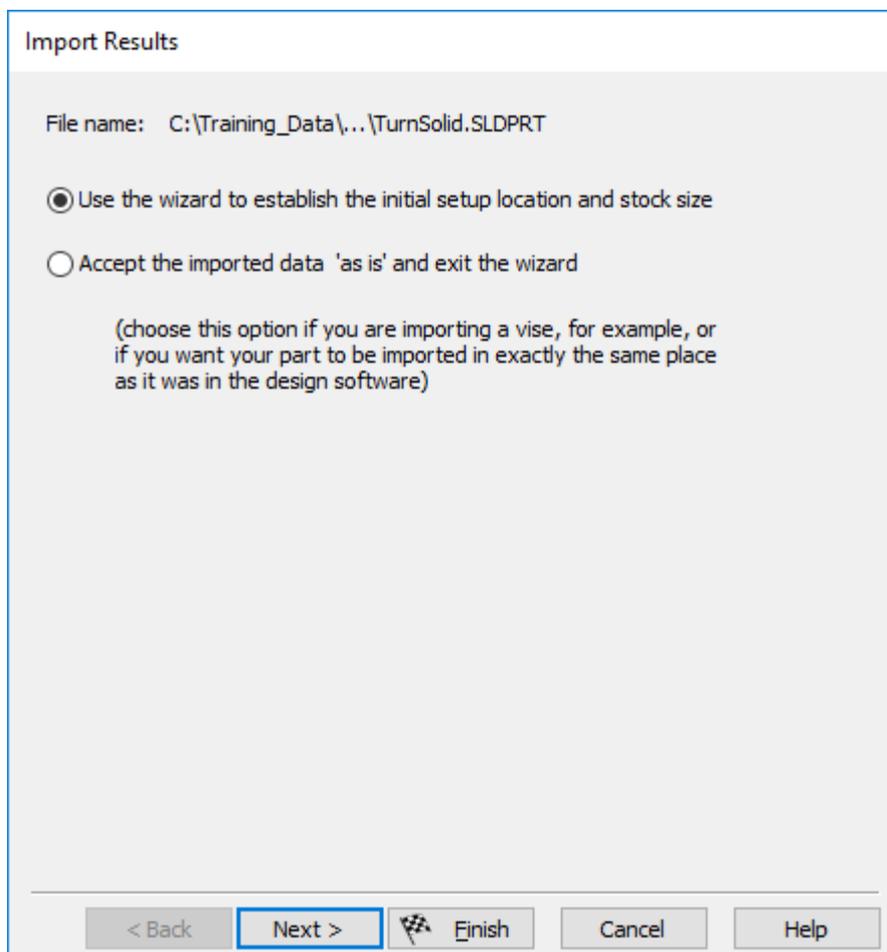
- 1 We will start by importing the model.
- 2 Go to **File>Import. C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import**



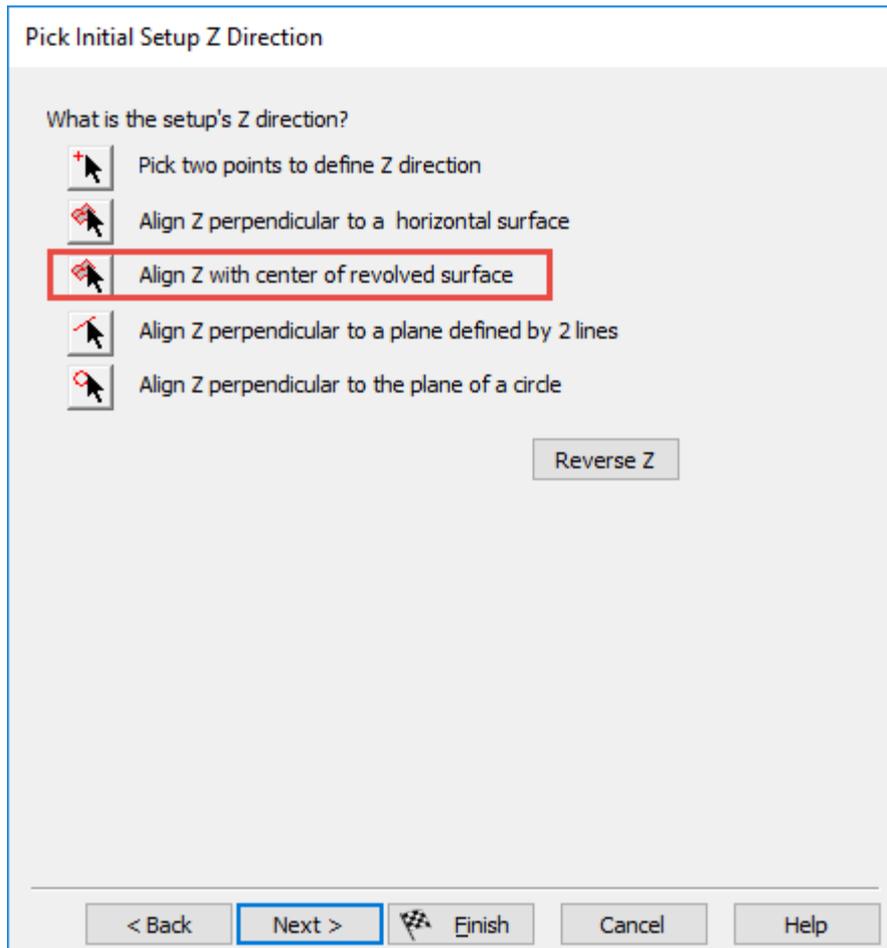
- 3 Select the file **TurnSolid.SLDPRT** then select **Open**. Please select the Default Configuration only.



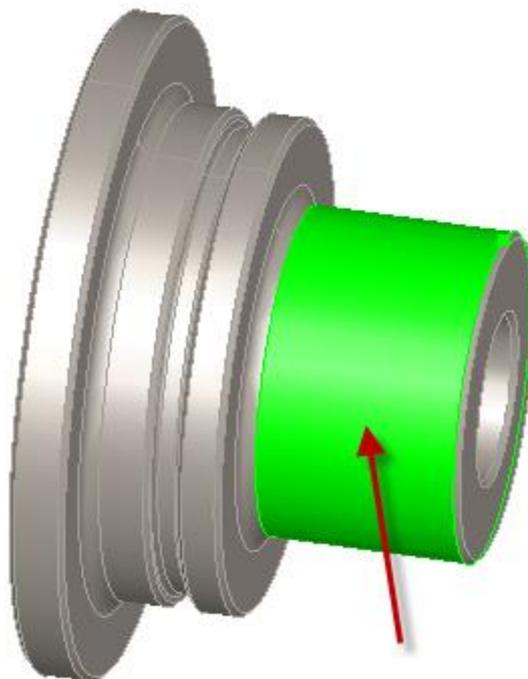
The part needs to be orientated, we can Use the **Wizard** to establish the initial **Setup location** and **stock size**. This example needs to be rotated.



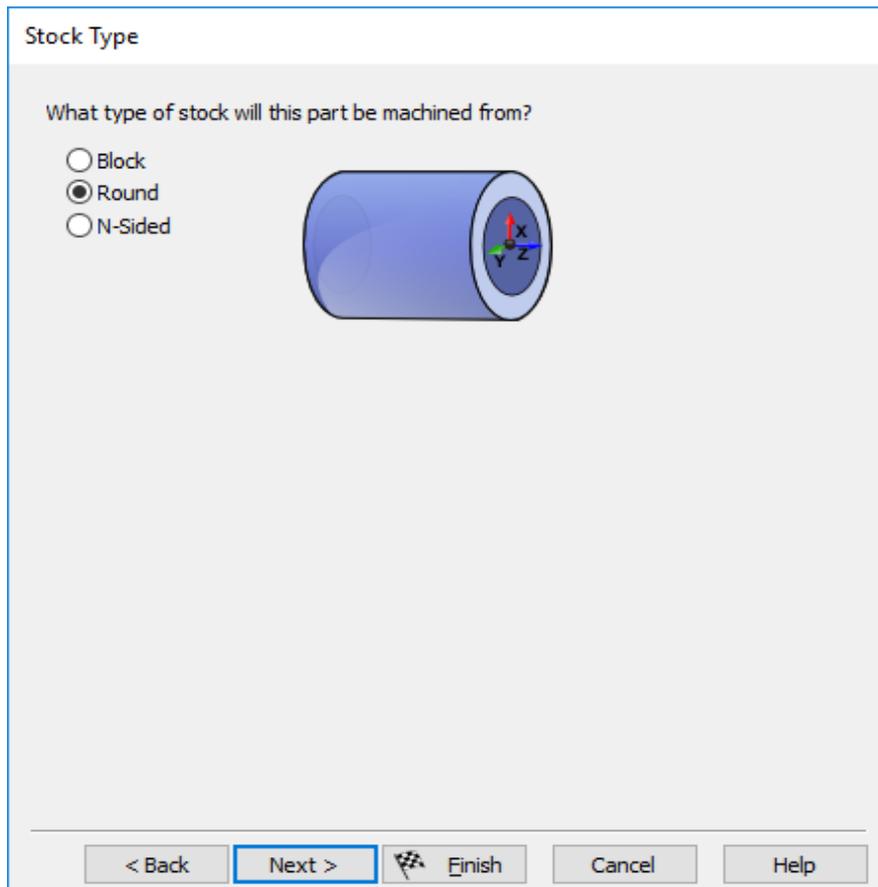
4 Select **Next**.



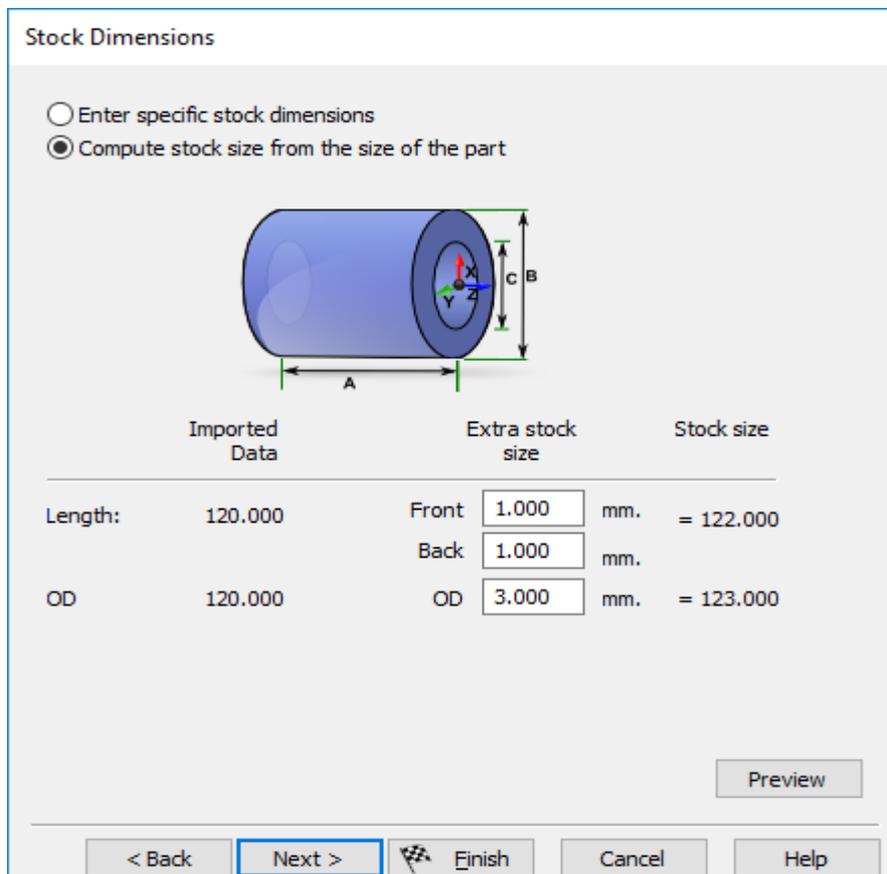
- 5 Use **Align Z with centre of Revolved Surface**.
- 6 Select a **revolved Surface**.



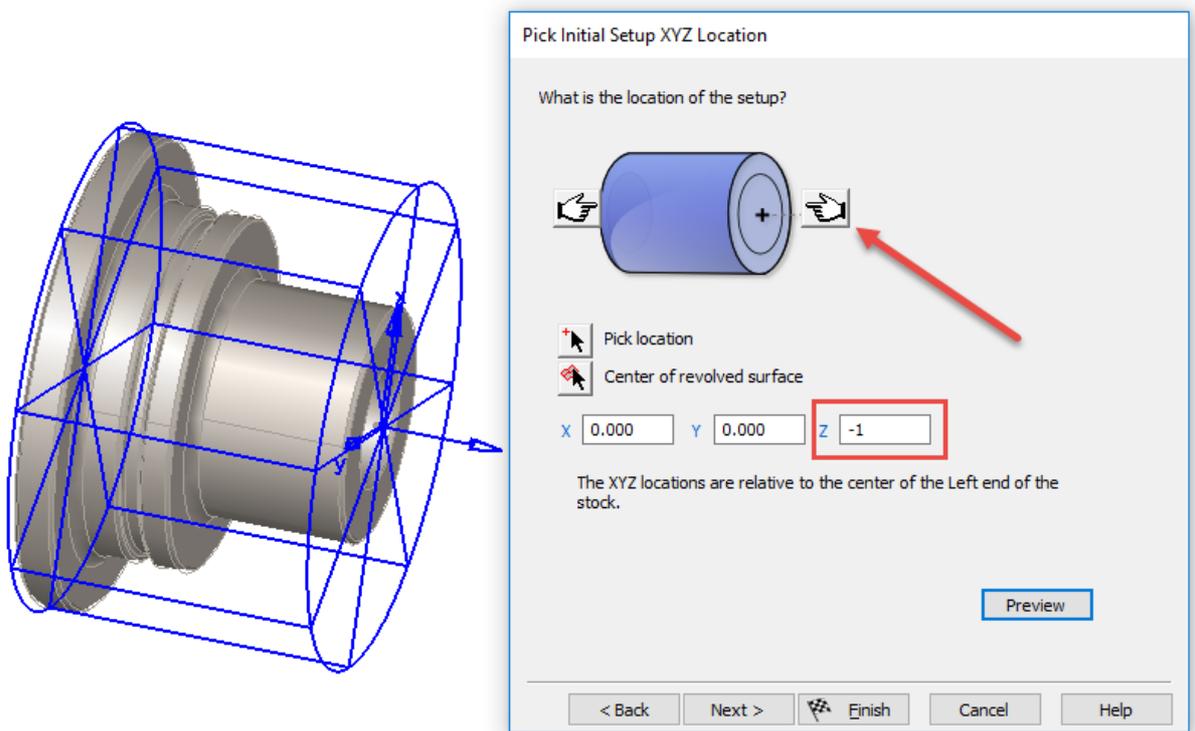
- 7 Select **Next twice** and then set the **Stock Type** to **Round** (Default)



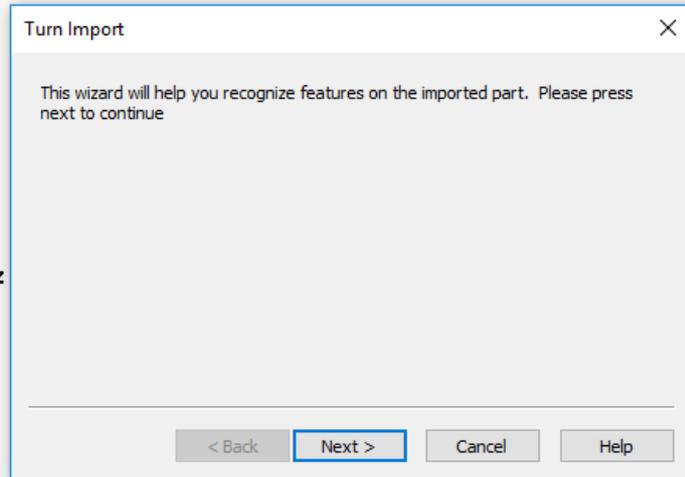
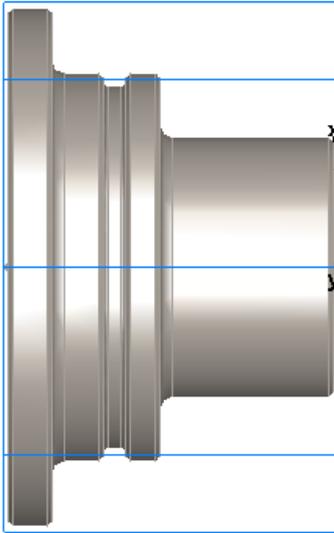
8 Select **Next** twice.



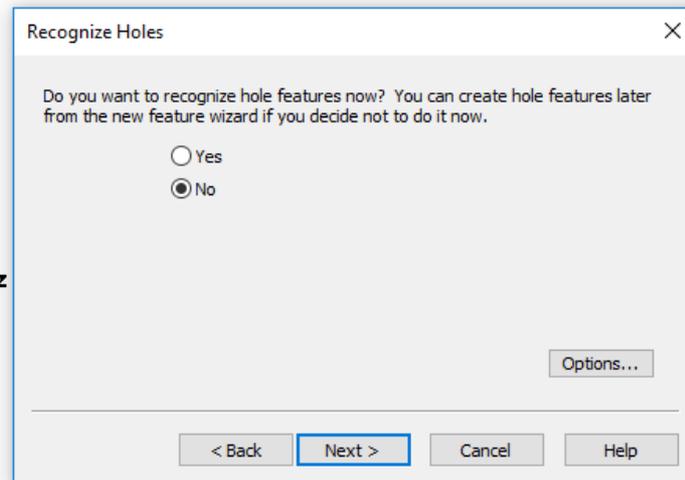
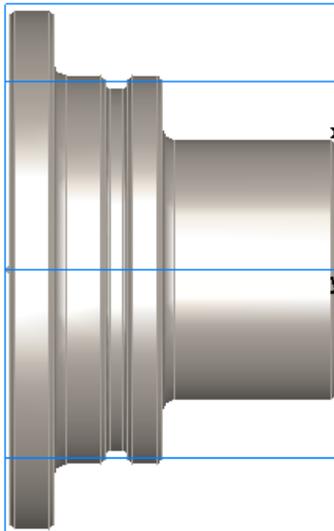
- 9 Select **Next** to set the Stock dimensions.
- 10 Select Stock Size from the size of the part.
- 11 Enter **1mm** Extra Stock **Front**
- 12 Enter **1mm** Extra Stock **Back**
- 13 Enter **3mm** Extra Stock **OD**
- 14 Select **Next** to align **Setup1** to the front of the part.



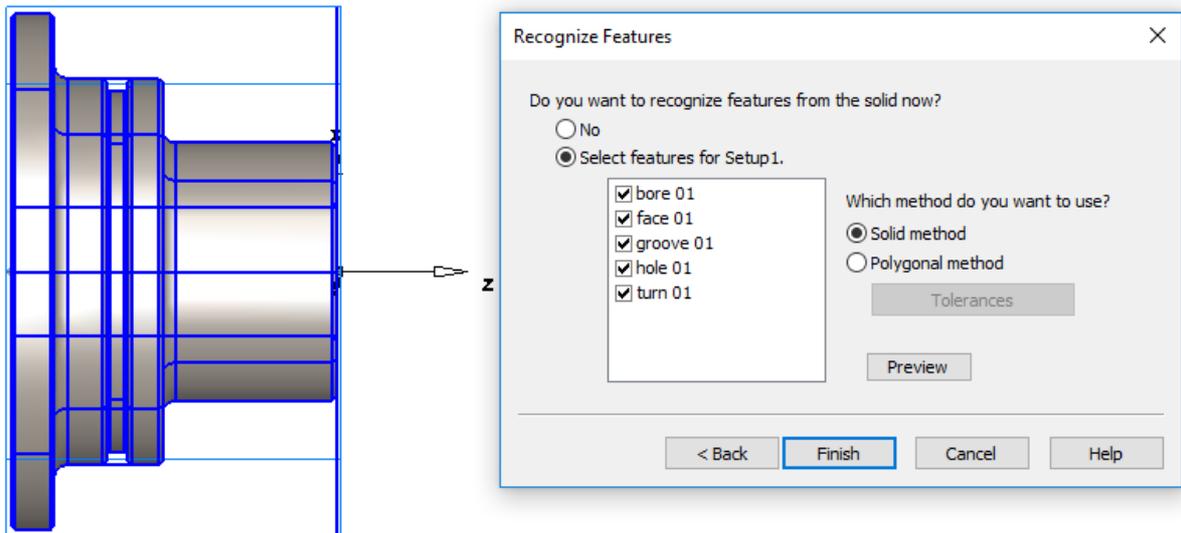
- 15 Select the **Right Hand** Icon; this will set the value to Zero. **Type** in **-1mm**
- 16 Select **Finish**.
- 17 The **Wizard** will guide you through the Feature extraction process.



18 Select **Next** to continue.



19 Select **Yes** and then **Next**. Accept the features created.

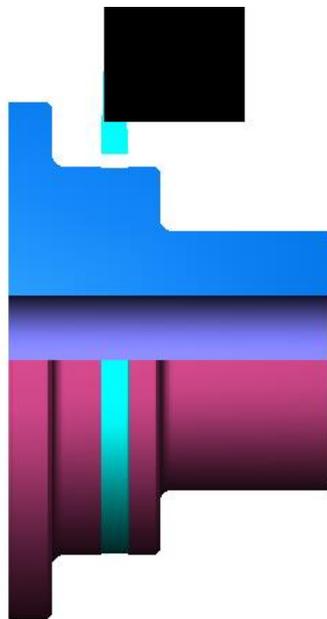


20 Select Finish and Run 3D Simulation.



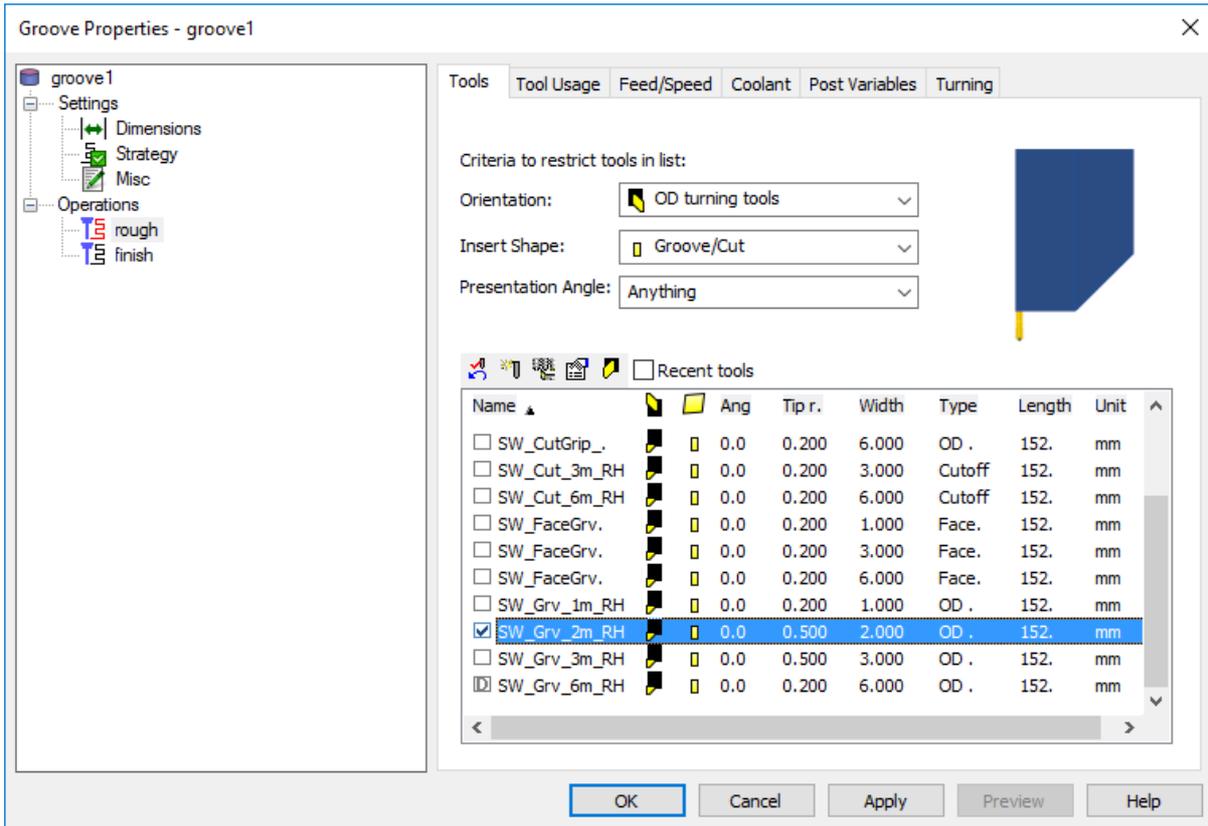
*If you run the **3D Simulation** we have two problems one the Tool that **FeatureCAM** has picked is too big for the actual groove plus in the process table we have two error messages see below. Which are related to the grooving tool*

Warning: TPL08W: Undercut detected. Unable to completely finish feature with this tool.

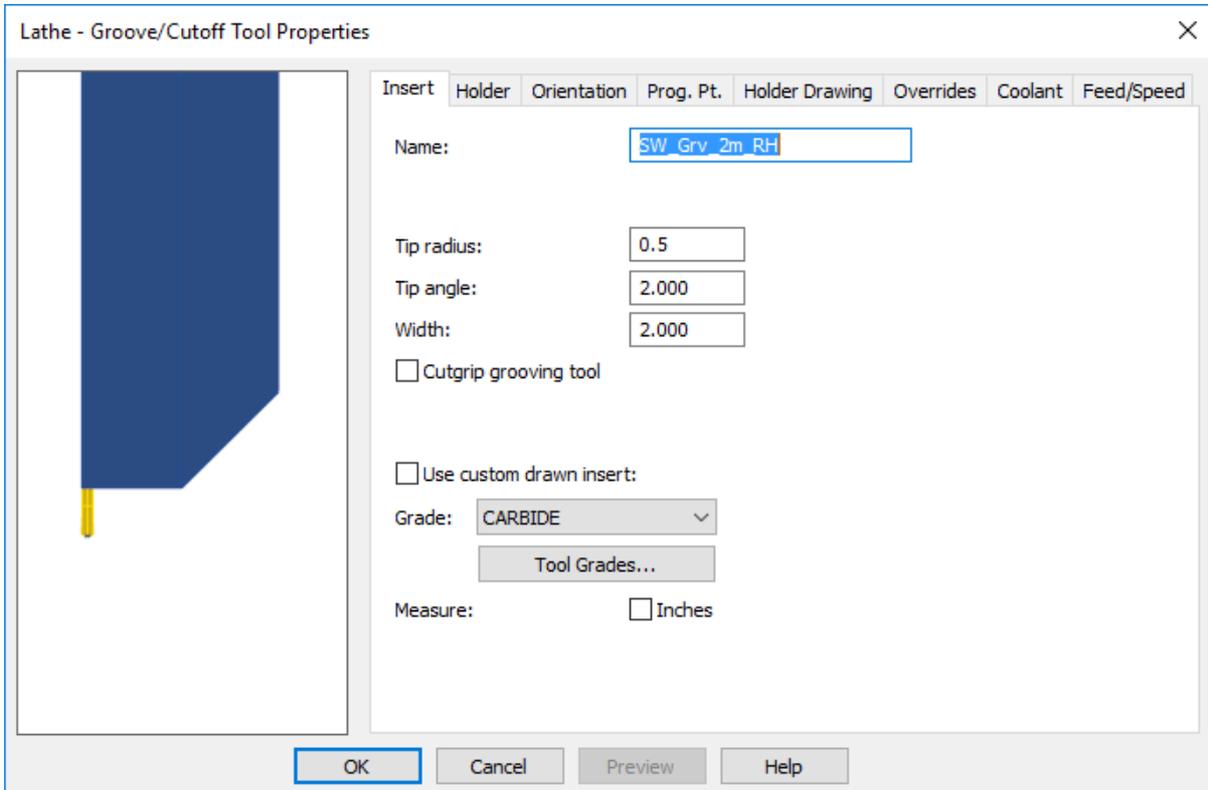


21 To resolve the problems first of all we need to create a grooving tool **2mm R0.5** this will allow the tool to finish the job.

22 **Eject** the **3D Simulation** and double click on the **groove 01** feature. The following menu will appear. Select **Rough**. Select Tools.

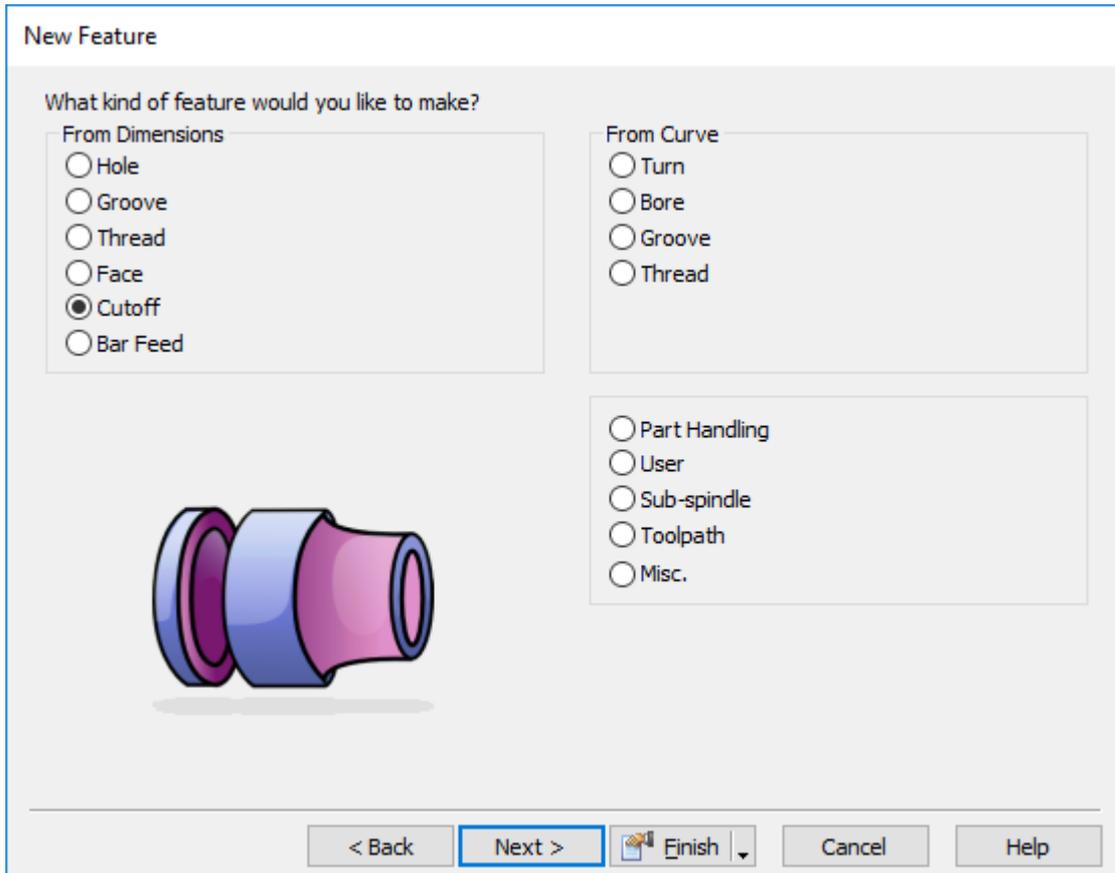


23 Double click on **SW_cut_2** and from this tool create the tool as shown on the next page. Remember to change the name.



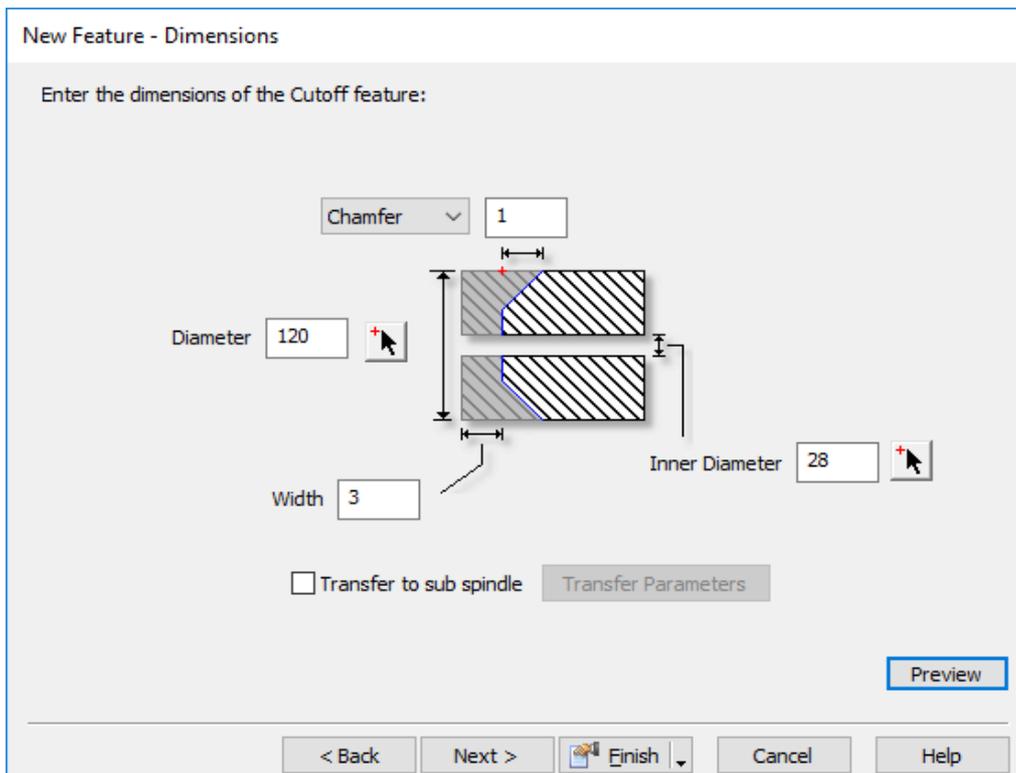
24 Change the tool to the one shown and hit apply. This will update the machining process automatically. Double click on the tool that shows the error. See menu. Select the tool listed below. Re-run the **3D Simulation**.

25 We now have to create a **Cut-Off** Feature manually to finish the part. Select **New Features** or **(Ctrl+R)** and then select **Cut-off**.



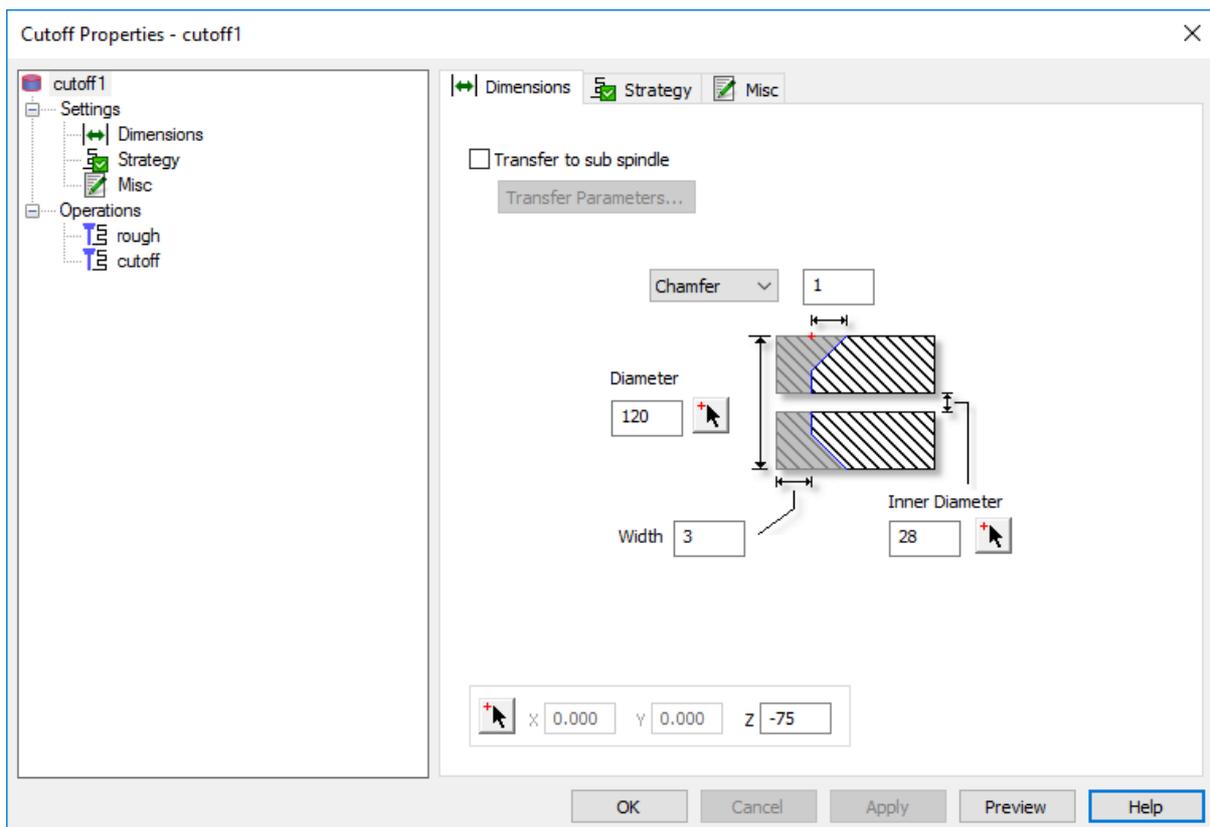
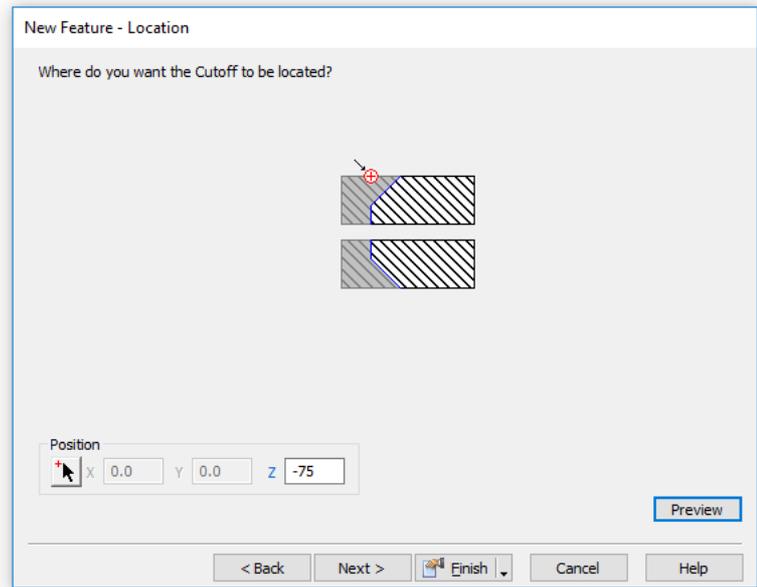
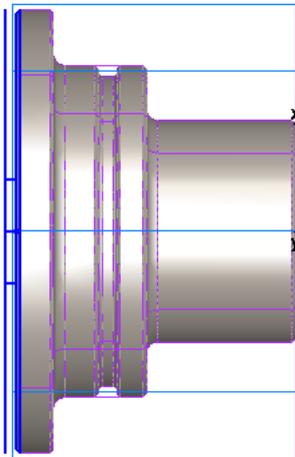
26 Select **Next**.

27 Enter the following information into the menu.

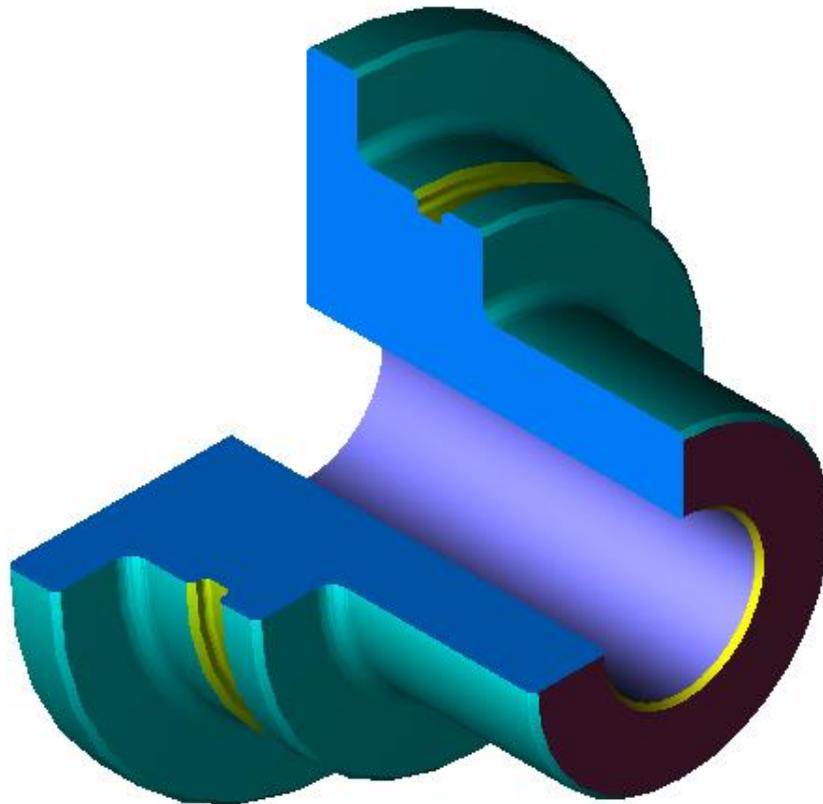


28 Select **Next**.

29 The overall length should finish at **-75mm**



30 This is the **finished part** shown below.



Using a Solid model for stock

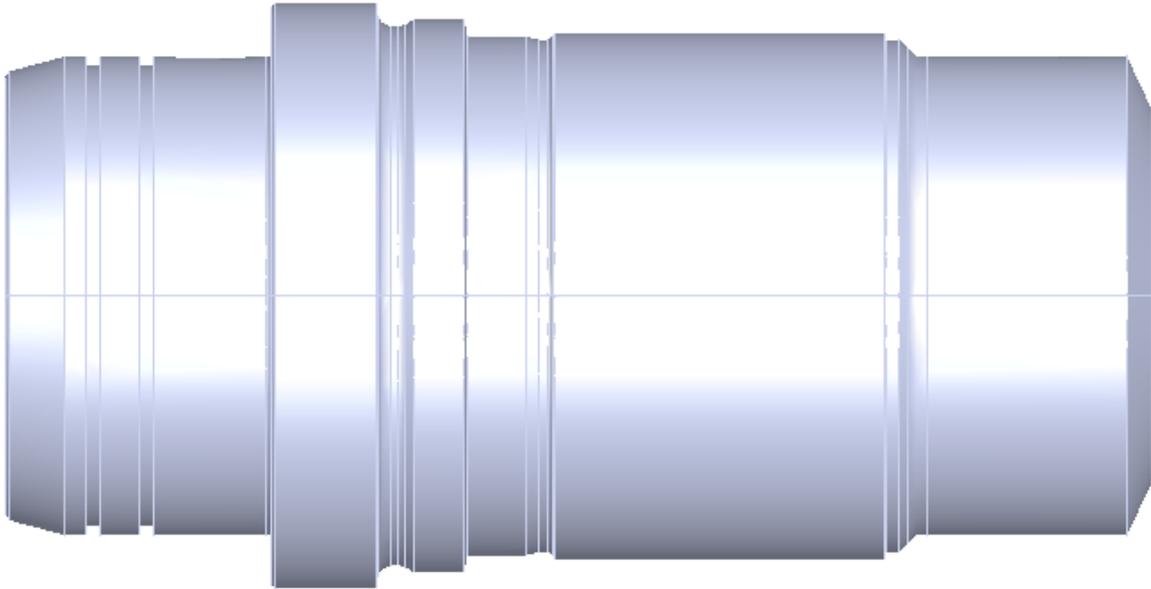


The next exercise uses a Solid model to define the stock material. This method minimises the amount of toolpaths required as it only machines where the stock is located.

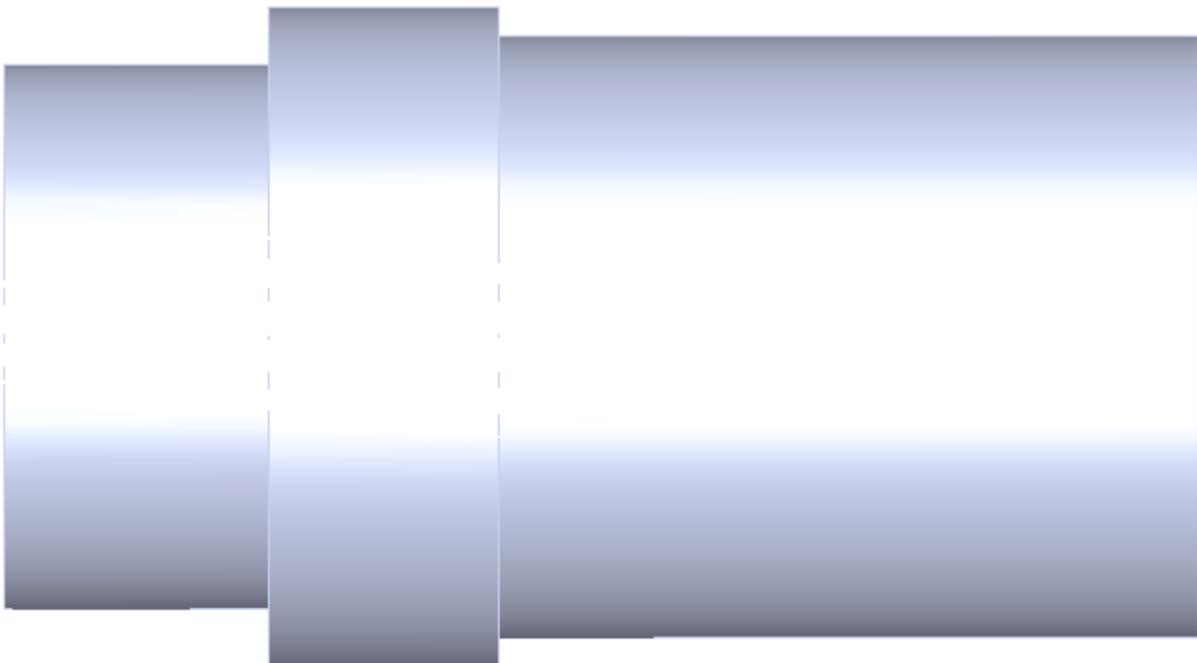
- 1 Import: - **Solid Stock-2 - All Models .x_t** from.
C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import\Solid Stock-2 - All Models .x_t



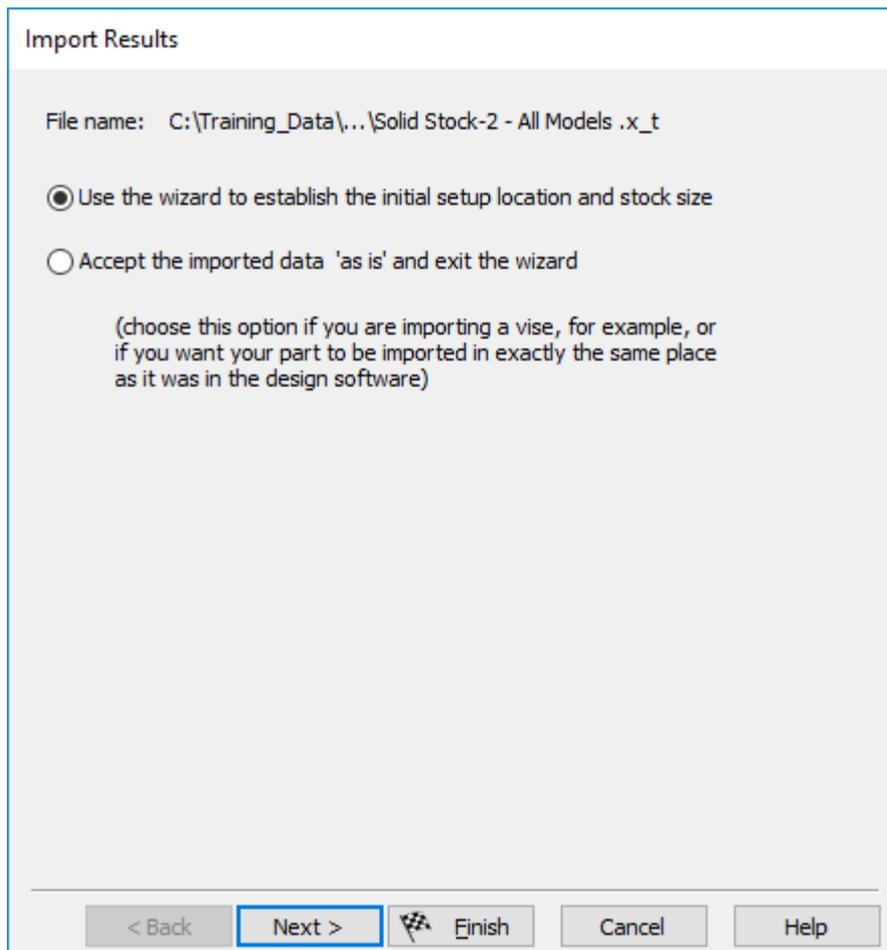
The following model is the **Part**



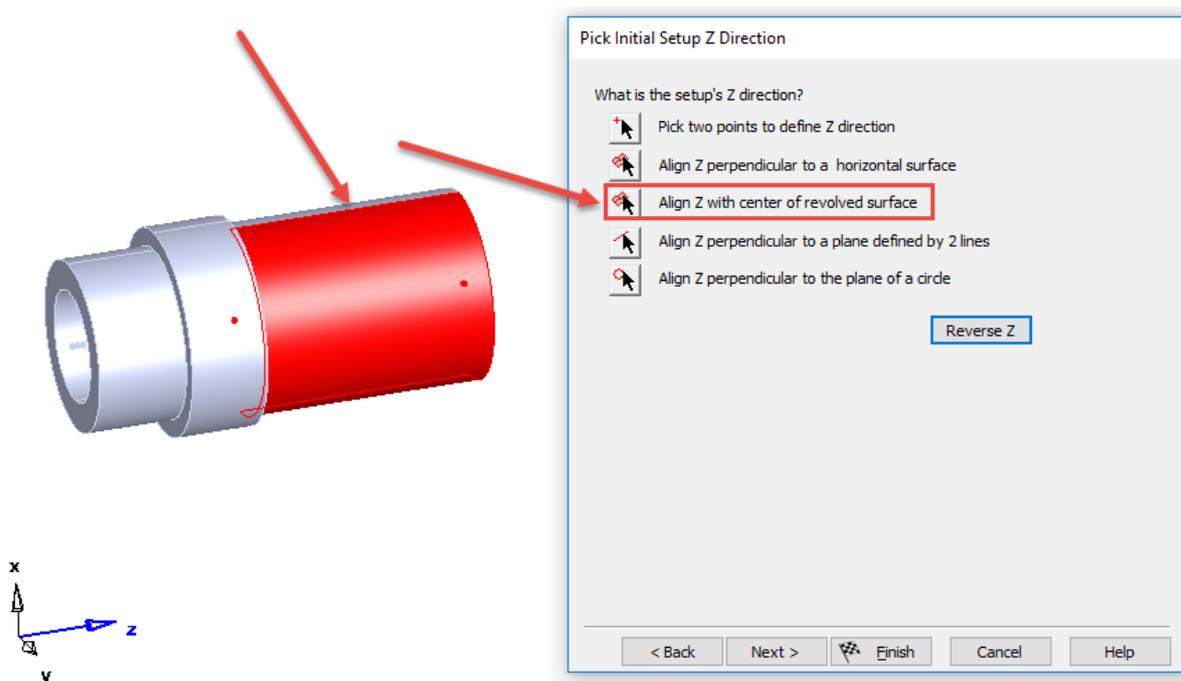
The following model is the **Stock**.



- 2 Work through the Wizard.

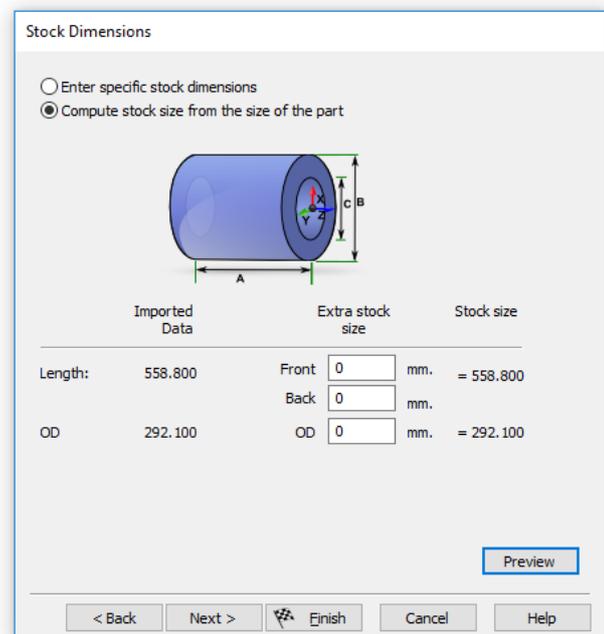
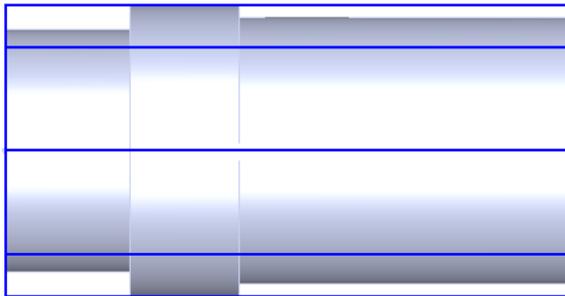


- 3 Select **Next**.
- 4 Use Align Z with centre of revolved surface.

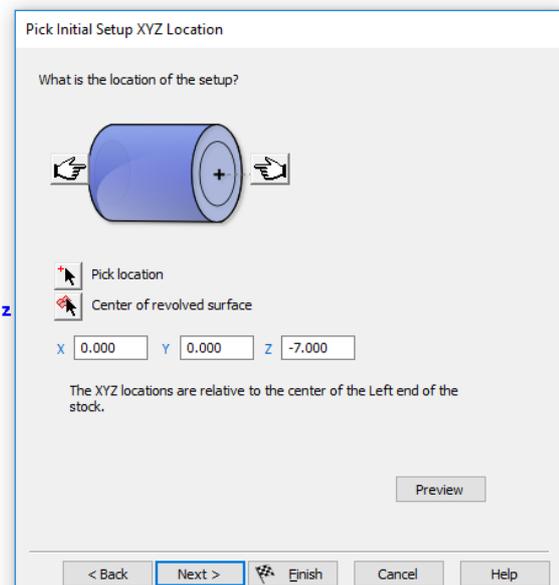
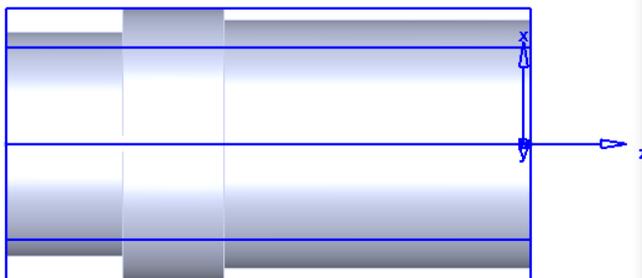


- 5 Select **Next** Twice.

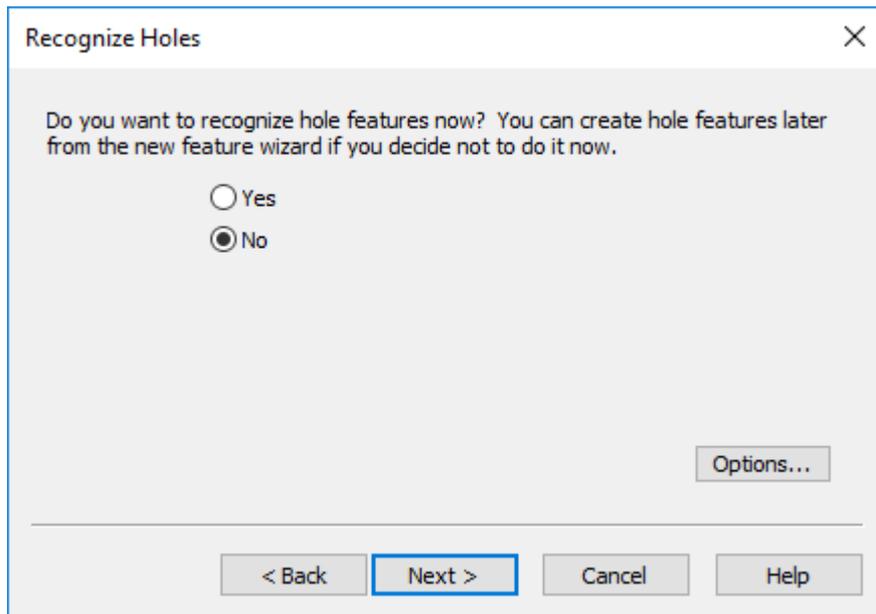
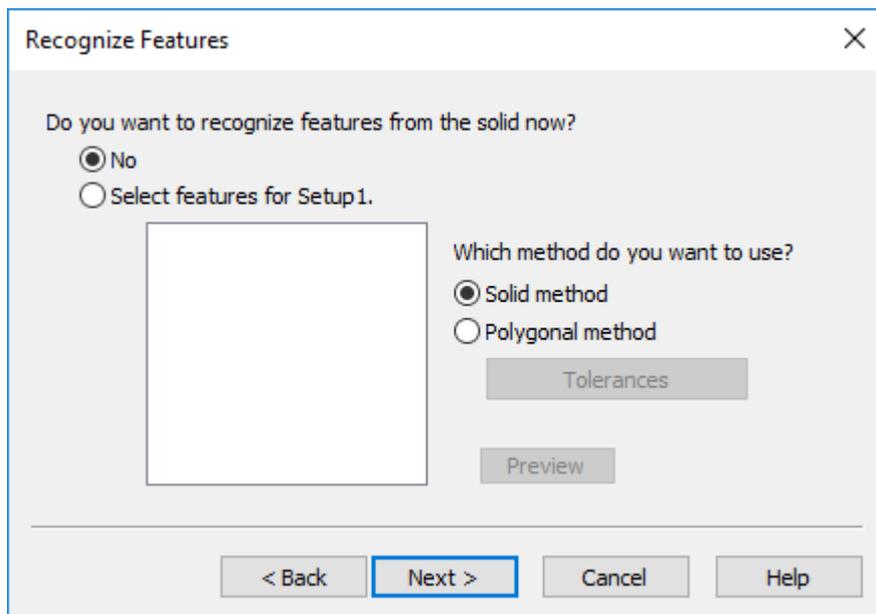
- 6 Stock Type = **Round**. Select **Next**.
- 7 Select **Next** Twice.
- 8 Select **Compute Stock size from the size of the part**. Set all data fields to zero.



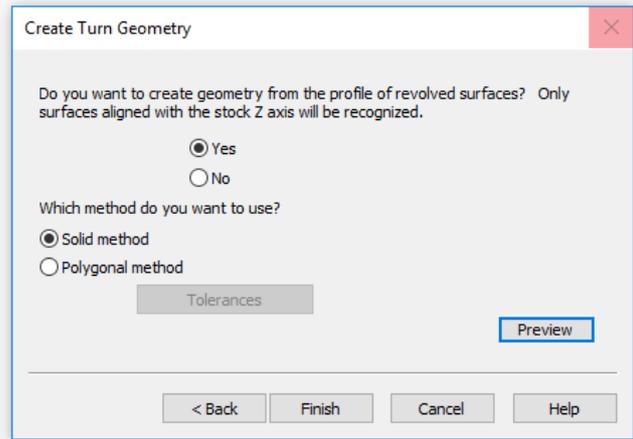
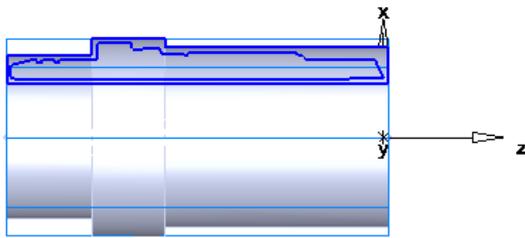
- 9 Select **Next**.



- 10 The datum will be -7mm. Leave this as it is as it will set to the Part which is inside the Stock solid.

11 Select **Finish**. Select **No** to **Hole Features**.**12** Select **Next**.**13** Say **No** to **Automatic Feature recognition**.**14** Select **Next**.**15** Select **Yes** to extract the turned profile.

Because we have two solid models. Both profiles are extracted.



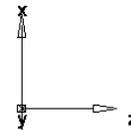
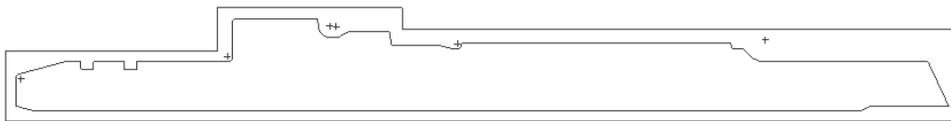
This will extract the 2D Geometry from the two solids we have imported.

16 We can use this Geometry to create Curves to create Stock and machine the Part complete.

17 Select **Finish**.



The geometry shown has been created from the Solid Models.



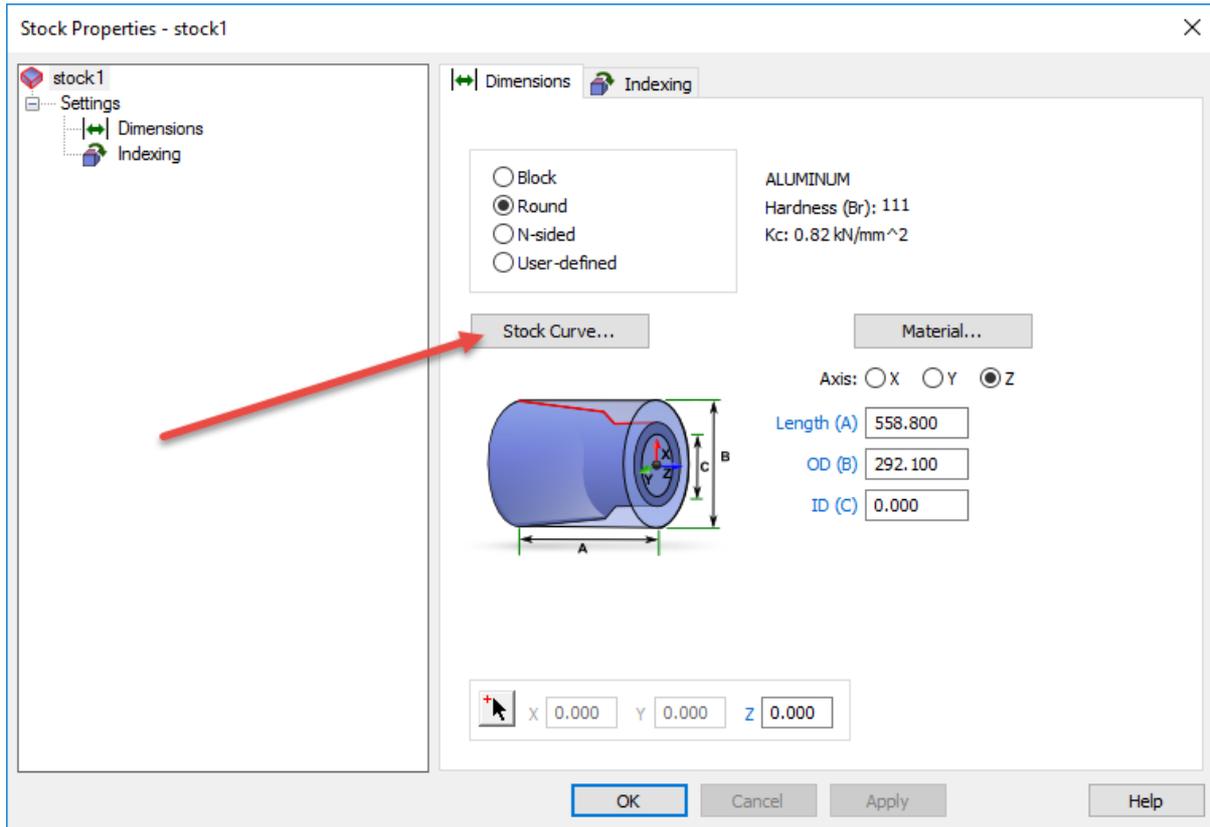
We have a number of ways to create Stock from the imported information.

18 We can use the larger Solid model and define this solid as stock or Create a Curve from the outer shape Geometry and define this as a Stock Curve.

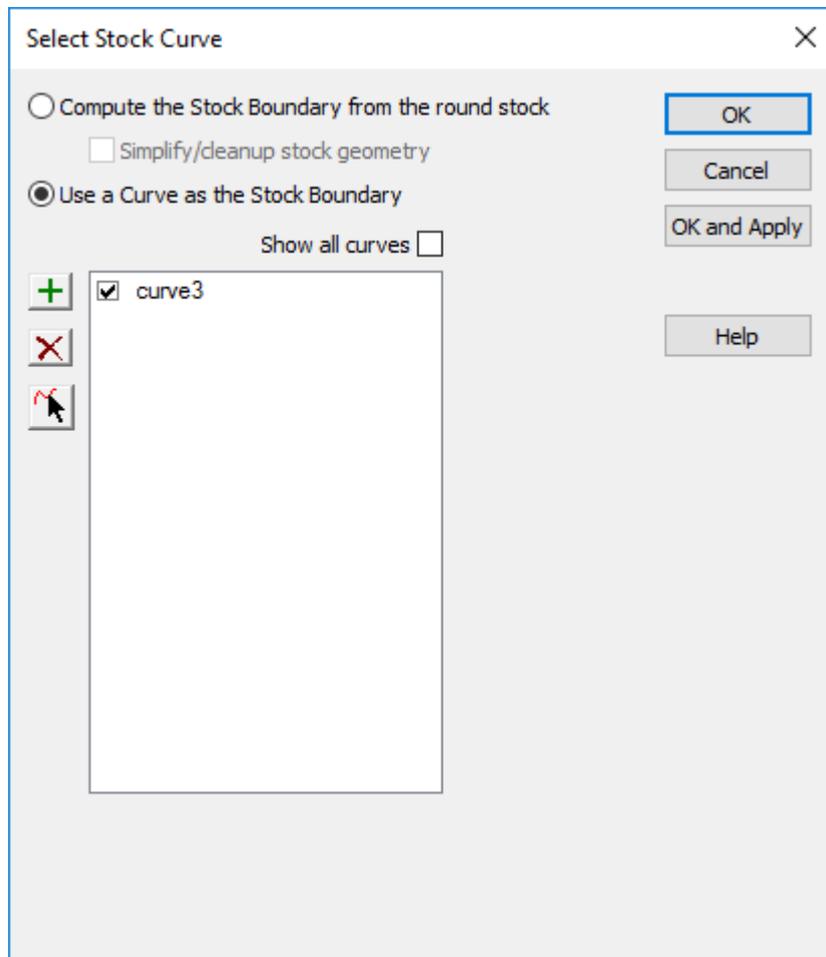
19 Remember to **hide** the solid models so you can see the Geometry.

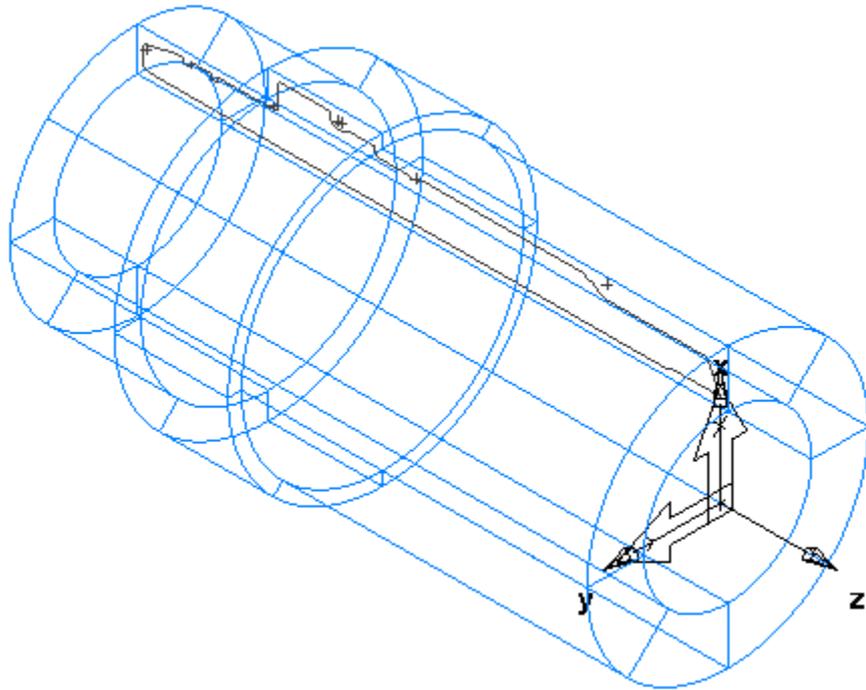
20 In this case my stock Curve is called **Curve5** your curve will be called a different Name.

21 Double click on **Stock1** in **Part View**.



22 Select Stock Curve.





- 23 Or alternatively use the User Defined button and select the **Stock Solid**.
- 24 From the Geometry you have extracted have a go at machining the Part.
- 25 Once completed have a go at **Solid Stock-3 - All Models .x_t**

Feature Recognise additional models

Location of files C:\Training_Data\FeatureCAM Course Data 2017\Turning Files to import

1036_Autodesk-A_01.x_t

1036_Autodesk-B_01.x_t

1036_Autodesk-C_01.x_t

1294_Autodesk-D_01.x_t