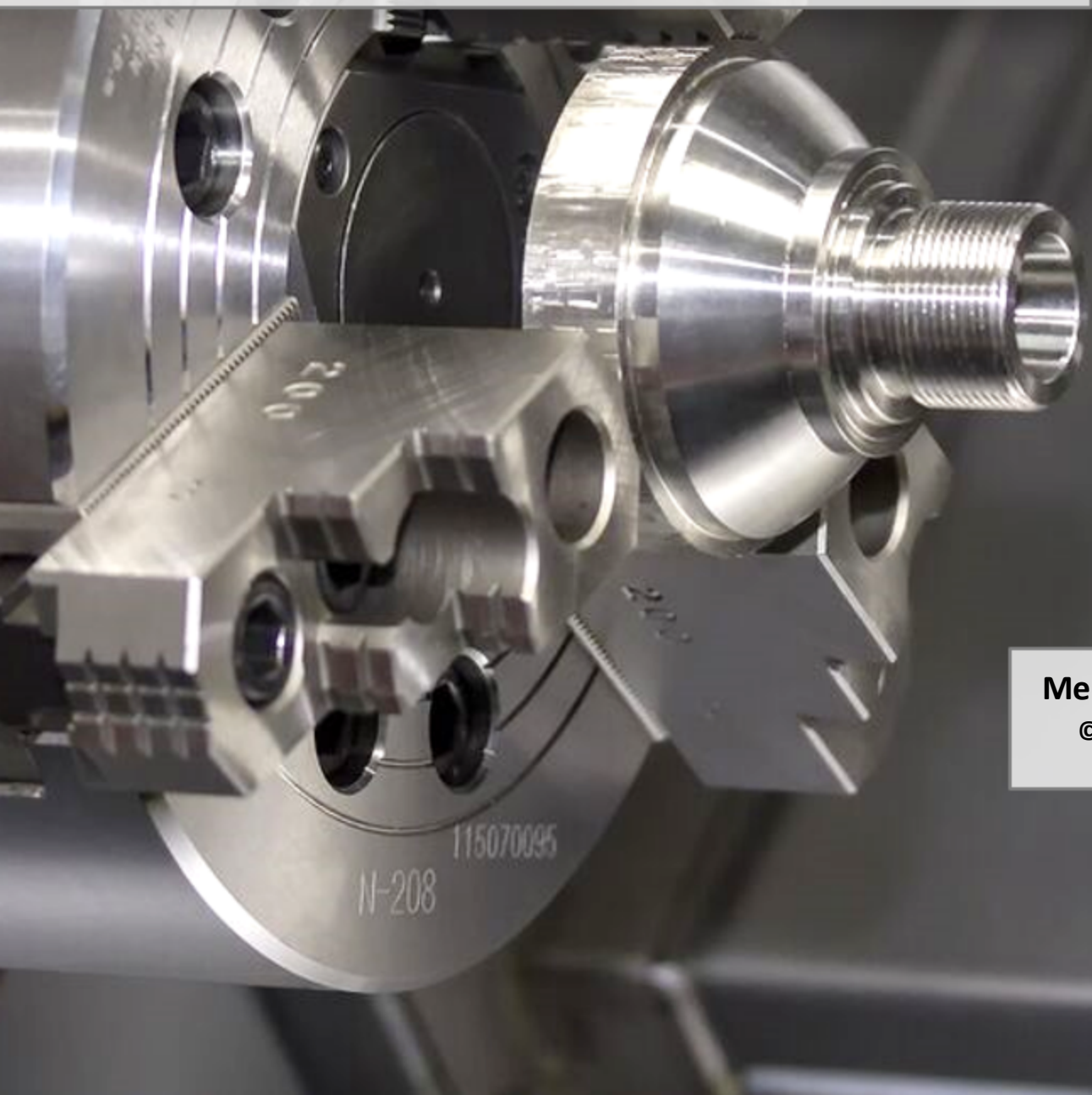
TURN
Quick Start Guide

VisualCAD/CAM 2024

Published: February 2024



MecSoft Corpotation

VisualCAD/CAM-TURN 2024 Quick Start

by MecSoft Corporation

User Notes:

Table of Contents

Quick Start

VisualCAD/CAM 2024

Prefer Printed Documentation? Click Here!

Quick Start Guides for each VisualCAD/CAM module are available in both PDF and Video format. Refer to the following information to access these resources:

What's New!

What's New in VisualCAD/CAM 2024

Watch the What's New in 2024 Webinar!

The Complete Quick Start Video Play List

Here is a link to the complete 2024 Video Play List

How to Access the Quick Start Guide Documents

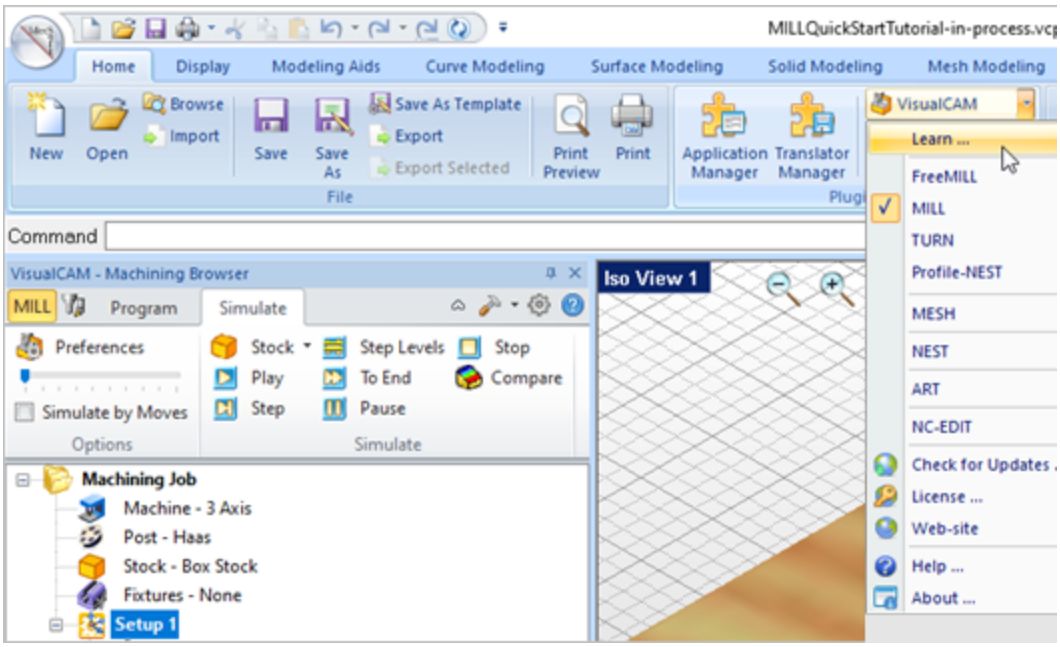
To help you quickly get started in working with each module, select one of the Help buttons located on the VisualCAD/CAM Learning Resources dialog.

You will find:

- Quick Start Guides
- What's New documents
- Online Help links

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

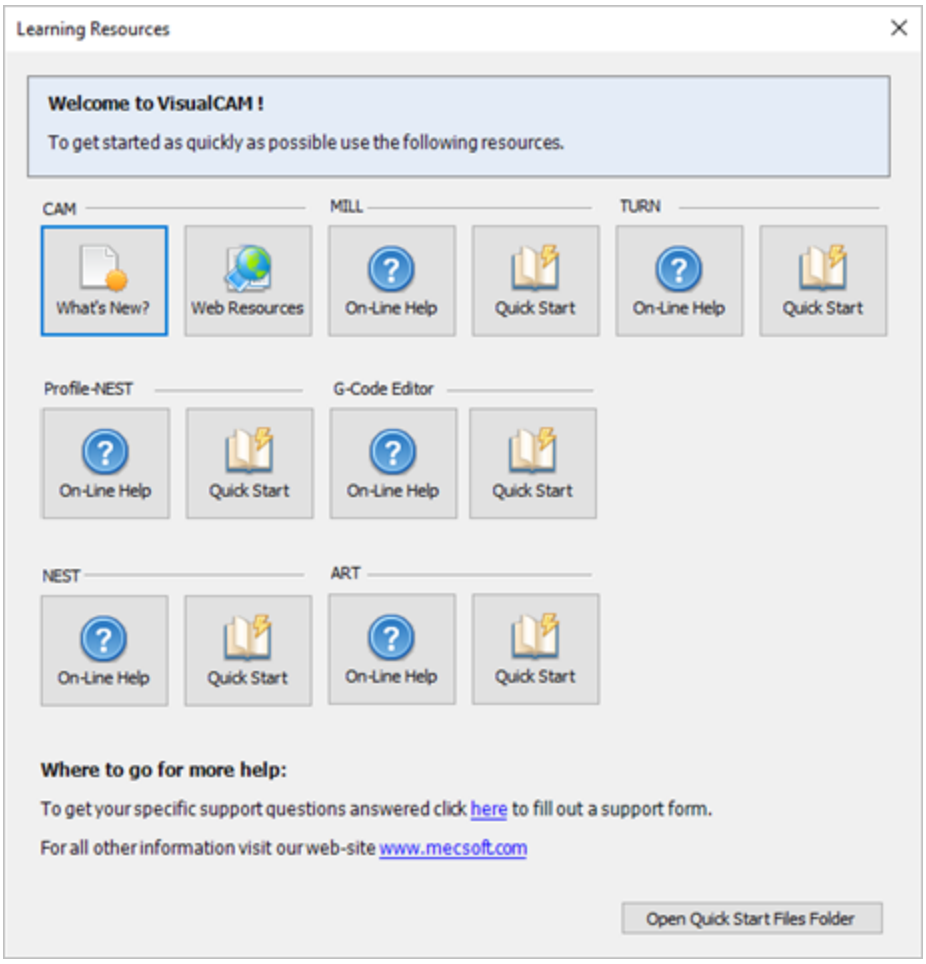
1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in VisualCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

   You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.

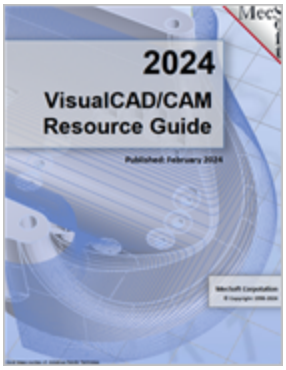


Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available VisualCAD/CAM Resources.

2024 VisualCAD/CAM Resource Guide



The 2024 VisualCAD/CAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Click Here!

About this Guide

3.1 Useful Tips

Here are some useful tips that will help you use this guide effectively.

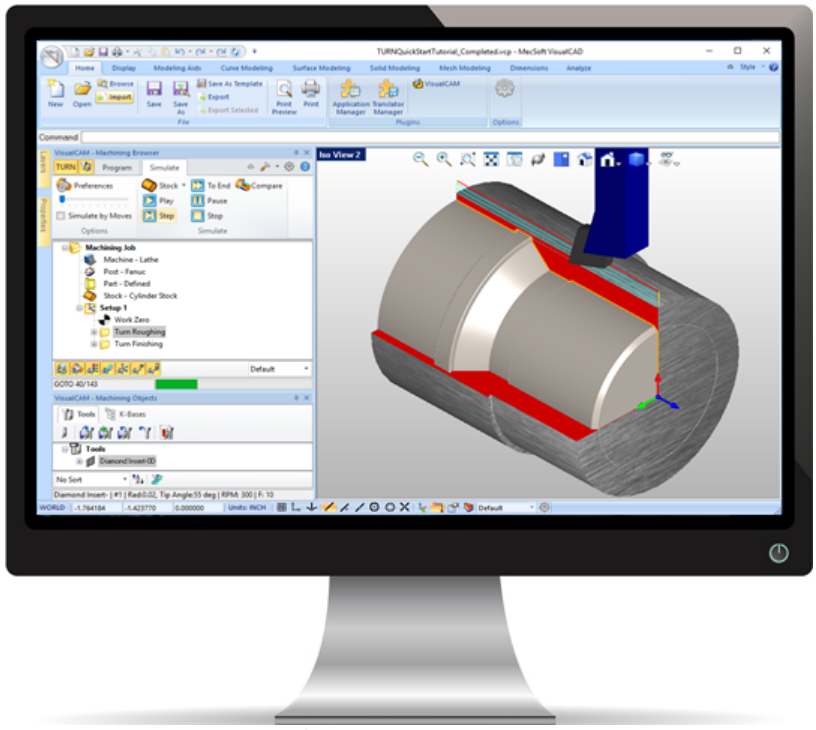
1. Copy the tutorial part files in a location other than the installation folder to make sure you have read/write privileges to the files.
2. Once you start working with the tutorial file, save your work periodically!
3. Don't stress out too much if you are having trouble with the tutorial. Call us or send us email and we can help you out.
4. Most of all have fun!

3.2 About the TURN Module

The VisualCAD/CAM TURN module (VisualTURN) offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities for programming 2 Axis CNC Lathes, running inside VisualCAM. This integration allows for seamless generation of toolpath and cut material simulation/verification inside VisualCAM for programming CNC lathes that support 2 axis machining.

The module also comes with numerous post-processors to output the programmed G-code to some of the most popular machines on the market. A simple and well thought-out user interface makes this system one of the most intuitive and easy to use.

You can work with native VisualCAD data as well as any of the data and file types that can be imported into VisualCAD for solids, surfaces and mesh objects. Then you can use the VisualCAD/CAM TURN module with its wide selection of tools and tool path strategies to create machining operations and associated tool paths for 2 Axis Lathes. These tool paths can be simulated, verified, and finally post-processed to the controller of your choice.



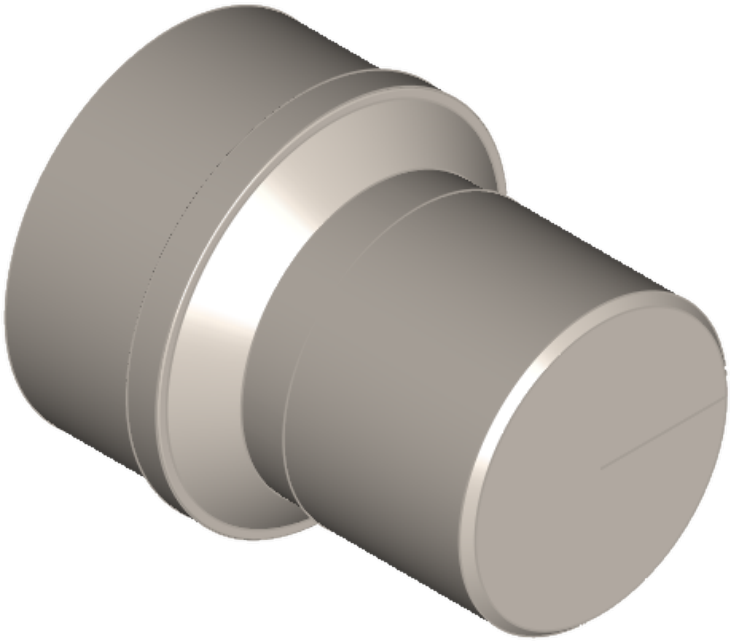
VisualCAM's TURN Module Quick Start Guide

3.3 Using this Guide

If you have installed VisualCAD/CAM successfully on your computer and are now looking at the blank screen and wondering what to do next, this is the guide for you. This guide will explain how to get started in using the VisualCAD/CAM TURN module to program a simple part through an example.

This guide will illustrate how to machine a part using Turn Roughing and Finishing operations. Even though we are using a 3D model, it will become apparent as we go that we could also machine this part using just a 2-D curve. Since all parts that can be created in a 2-Axis lathe are solids or surface of revolutions, it is enough to just describe the profile that needs to be revolved to create this shape.

This guide has two associated VisualCAD files that you can find located in the QuickStart folder under the installation folder. The first is a finished file that contains all of the completed toolpaths and machining operations and represents the file that you should end up with after working through this guide. The other is a starter file that contains only the geometry. Use the completed file as a reference. Copy the starter file and use it to begin the guide.



TURNQuickStartTutorial.vcp

3.4 Watch the Video!

Want to see a video demonstration of this quick start guide? Just click on the play list below and watch the TURN Quick Start Guide video.

Here is a link to the complete 2023 Video Play List

Getting Ready

4.1 Running VisualCAD/CAM

Locate the VisualCAD/CAM 2023 shortcut on your desktop and double click to launch the application.

Alternatively you can also click on the Windows Start button and select All Programs. Go to the program group containing VisualCAD/CAM 2023. (The name of this program group will usually be called VisualCAD/CAM 2023, unless you specified otherwise during setup.)

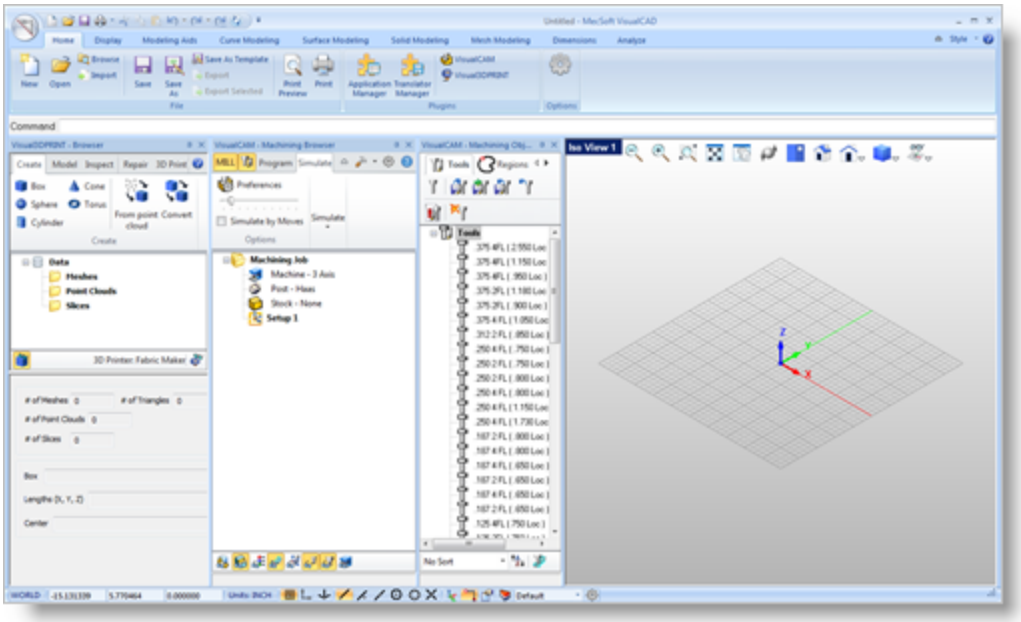
Once you locate the program group, select it and then select VisualCAD/CAM 2023 to launch the application.

If the installation was successful, upon launching of VisualCAD/CAM 2023 you should observe a menu entry called VisualCAM 2023 on the Home Ribbon Bar menu of VisualCAD.

If you do not see this menu entry then please check the On Line Help document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the VisualCAD Display

Before we begin, let's talk a bit about the VisualCAD display. When you run VisualCAD for the very first time, your screen may look this.



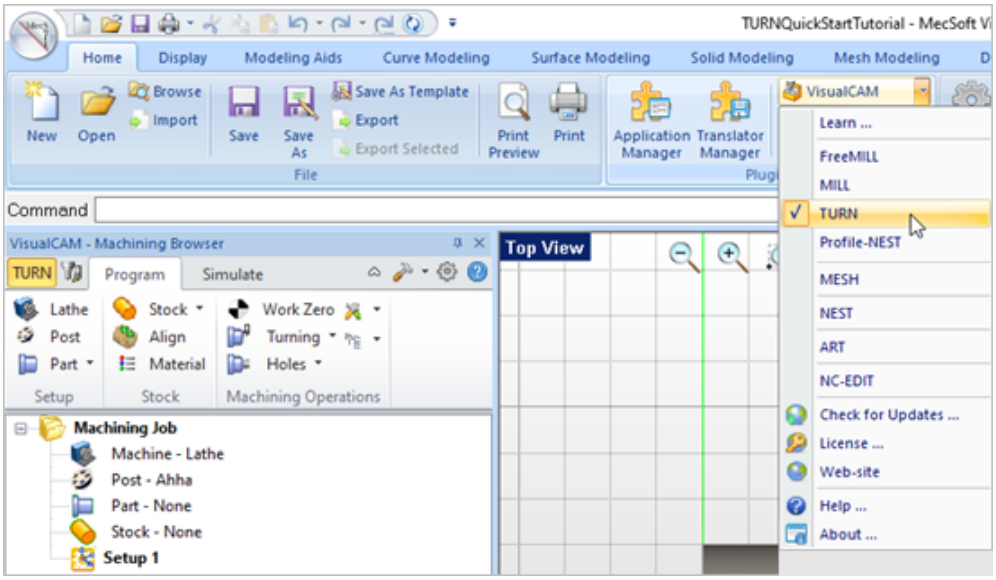
These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.

With all plug-in modules closed your screen will look like this:

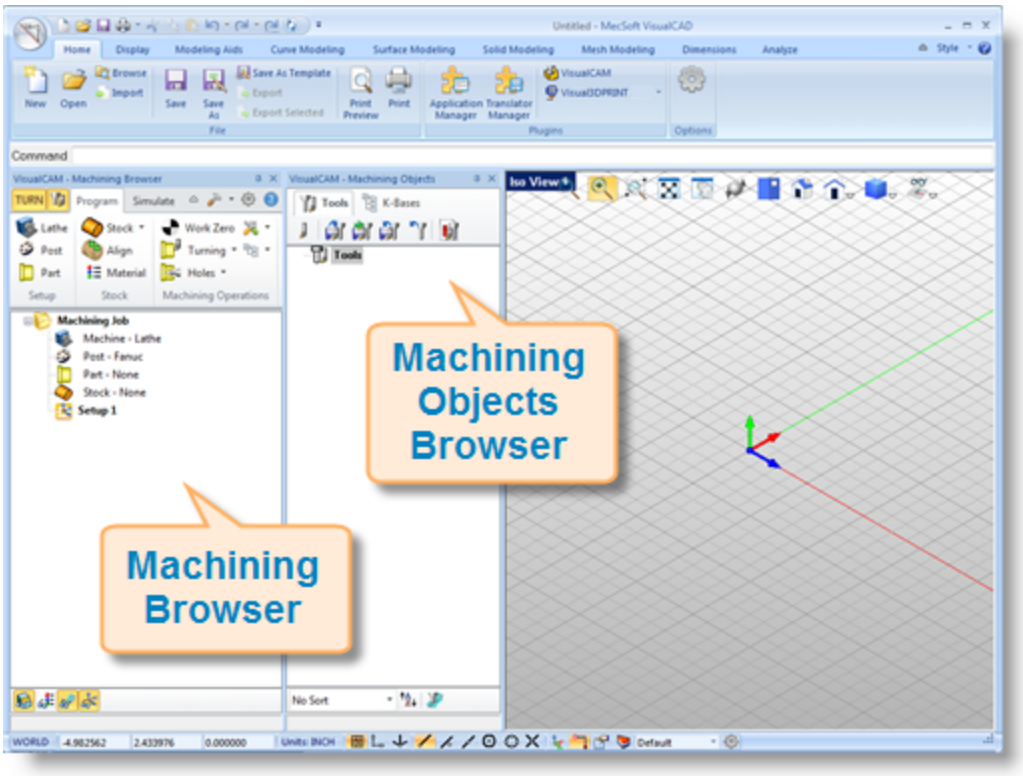
4.3 Launch the TURN Module

Now, let's begin by launching the VisualCAD/CAM TURN module.

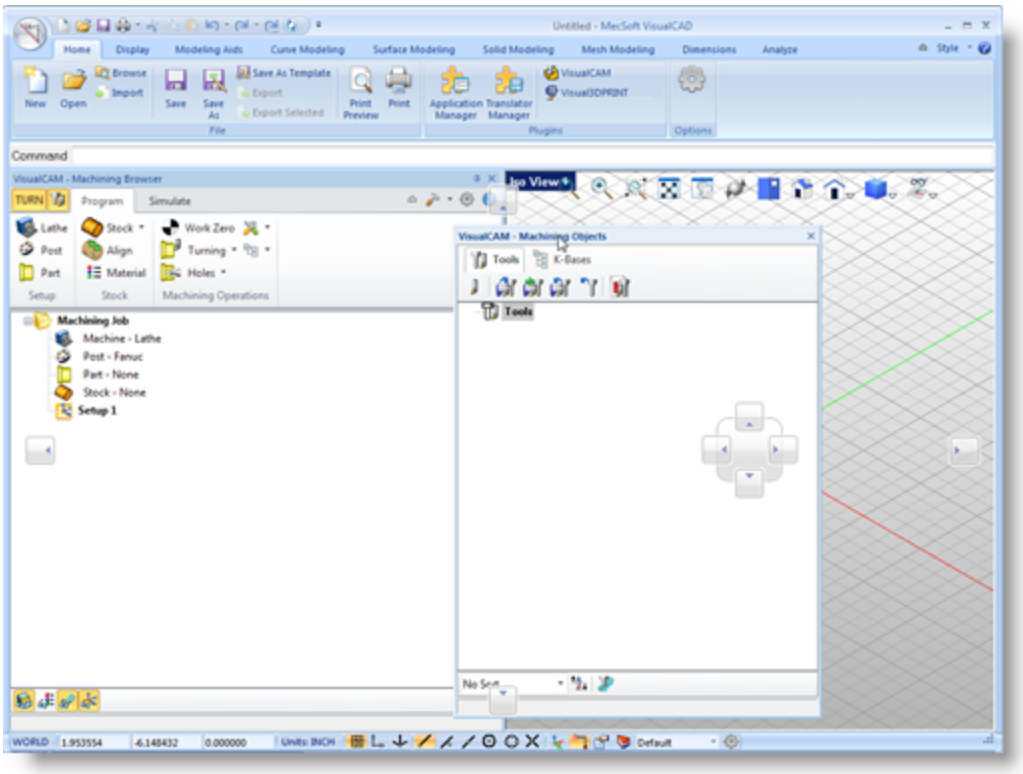
1. From the Plugins pane of VisualCAD's Home Ribbon Bar, you will see the VisualCAM 2023 main menu item.
2. Drop-down the menu and pick TURN to load the TURN module.



3. Docked on the left you will see the Machining Browser and the Machining Objects Browser. When you first run VisualCAD/CAM, these two browsers my be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



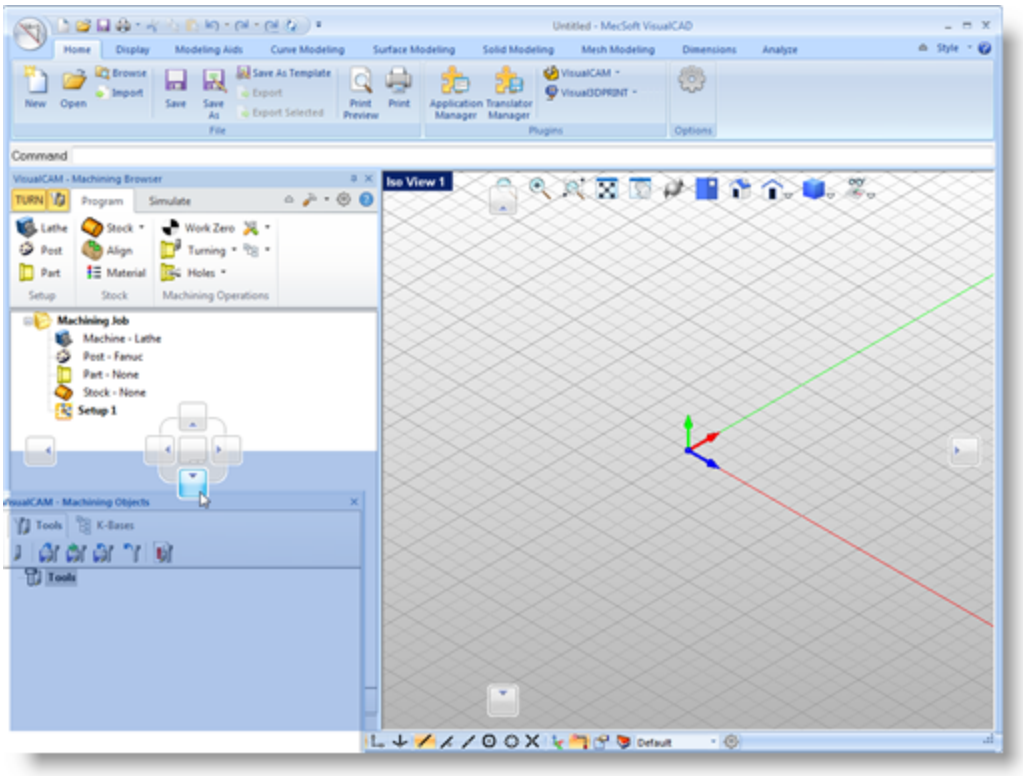
4. For example, let's move the Machining Objects Browser so that it displays under the Machining Browser on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



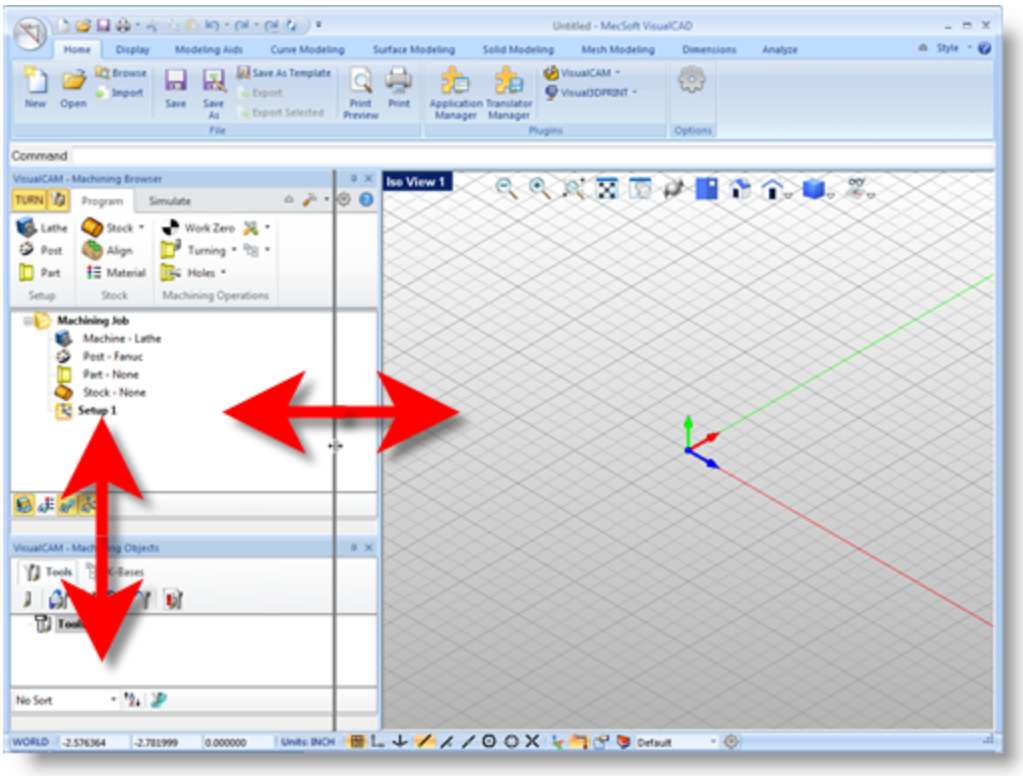
While doing so, you will see the docking widget display in the background with directional buttons allowing you to choose screen locations relative to the active window.

5. We'll drag the Machining Objects Browser over the base of the Machining Browser until the cursor activates the bottom directional button.

   When the preview of the new location displays, let go of the right-mouse button and the browser will move to that location.



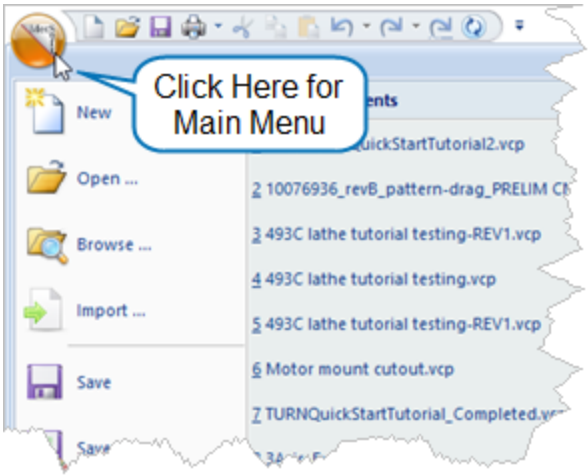
6. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



4.4 Load the Part Model

"Part" refers to the geometry that represents the final manufactured product. You can create parts within VisualCAD or import geometry created in another CAD system.

1. From VisualCAD's Main Menu, select Open.

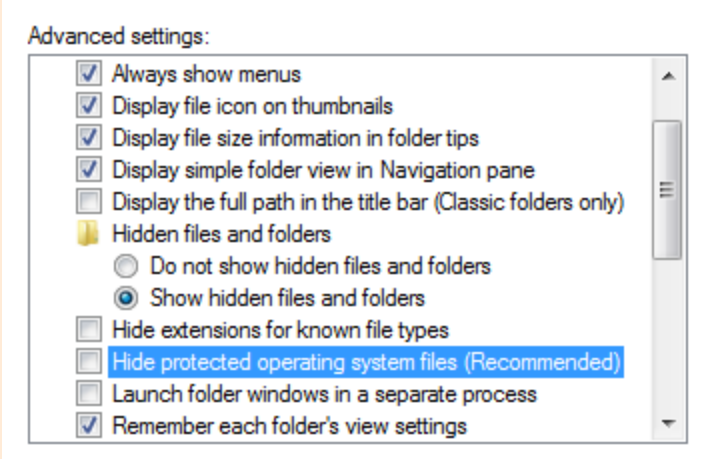


2. From the Open dialog box, select the TURNQuickStartTutorial.vcp file from the C:\ProgramData\MecSoft Corporation\VisualCAM 2023\QuickStart\ folder.

As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.

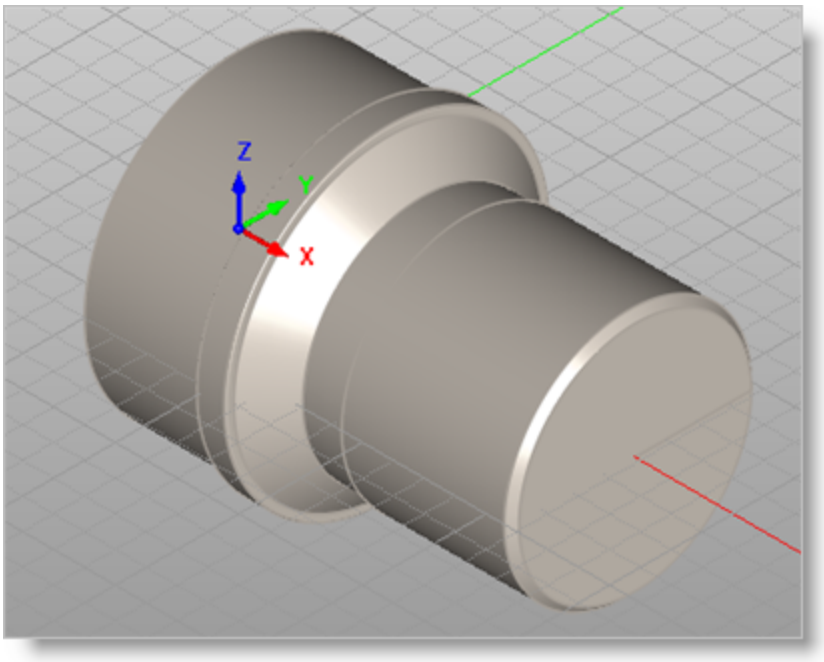
By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows 8 users: Go to Control Panel > Appearance and Personalization > Folder Options.
   For Windows10 users: Go to Control Panel > Appearance and Personalization > File Explorer Options.
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:
   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)



3. Click Apply and OK.

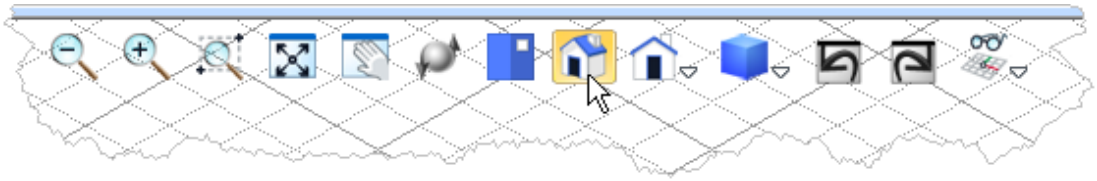
The part appears as shown below



MILLQuickStartTutorial.vcp

You can import solid models, Stereo-Lithography (both ASCII and binary) format files. Surfaces and Solids can be imported from IGES, STEP, Rhino (*.3dm), Parasolids (*.x_t, *.X_b), SAT and DXF / DWG files. Faceted (triangulated) models can be imported from STL, VRML, Raw Triangle, or Rhino Mesh.

3. From the View toolbar, select the Isometric View to work in.



4.5 Machining Strategy

Based on the type of geometry of this part, we will machine this model out of a cylindrical aluminum blank that is 3 inches diameter and a minimum length of 3¼ inches. As the part has only features on the outer diameter (OD) to be machined, we will machine this out by using a Turn Roughing and a Finishing operation. We will also use just a single diamond insert with a 20 degree relief angle and 0.02 inch tip radius with 0.5 inch inscribed circle radius for performing all machining. We will also assume that the cylinder blank will be held on the chuck over to the left side on the CNC lathe.

4.6 Main Programming Steps

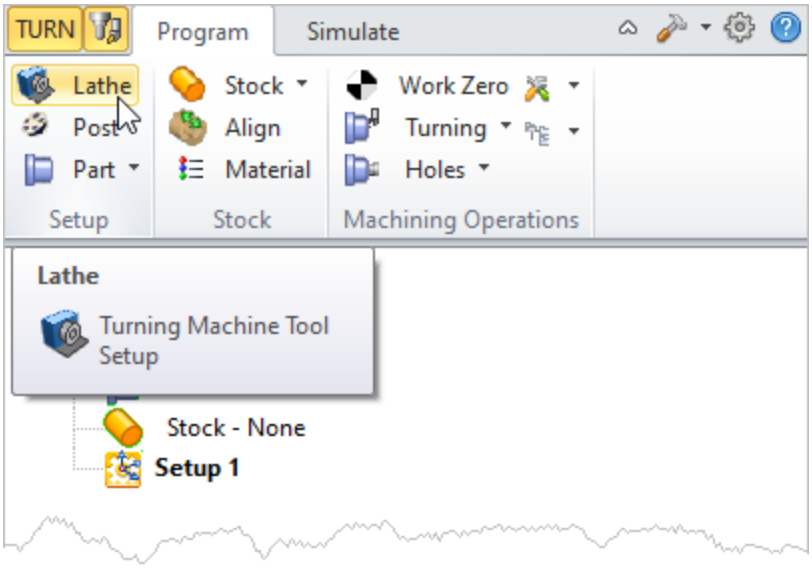
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

1. Define the Machine and Post Processor to use.
2. Define the Part and Stock Geometry, Material and Work Zero.
3. Create and Select a Tool to use for machining
4. Create the Machining Operations including the Feeds and Speeds, the Clearance Geometry and other Cutting Parameters.
5. Generate the toolpaths.
6. Simulate the toolpaths.
7. Post Process the toolpaths.
8. Generate Shop documentation.

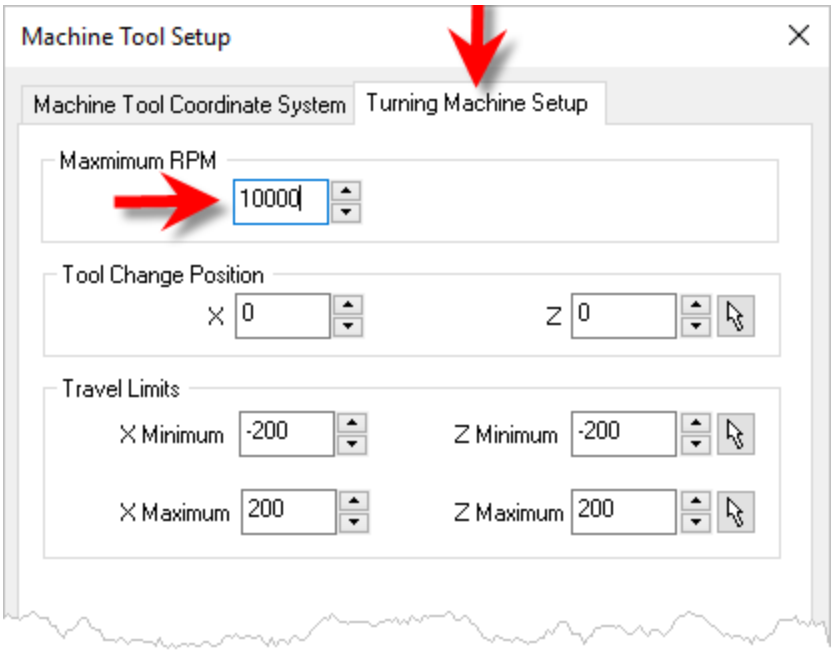
4.7 Define the Machine Tool

Let's start by defining the Machine to use for this job.

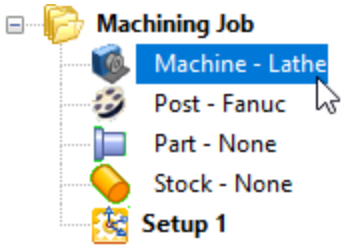
1. From the Program Tab select Lathe to display the dialog box.



2. Select the Turning Machine Setup tab and then set Maximum RPM to 10000.



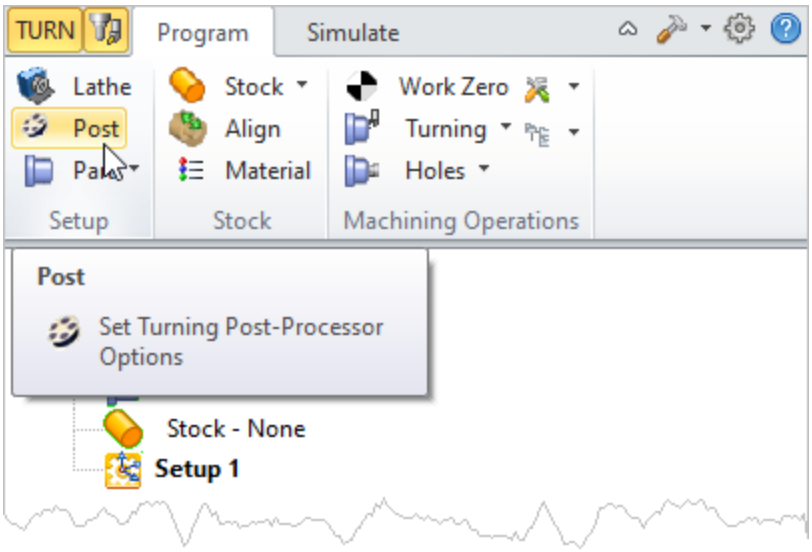
3. Pick OK and notice that the Machine - Lathe now appears under Machining Job in the Machining Browser.



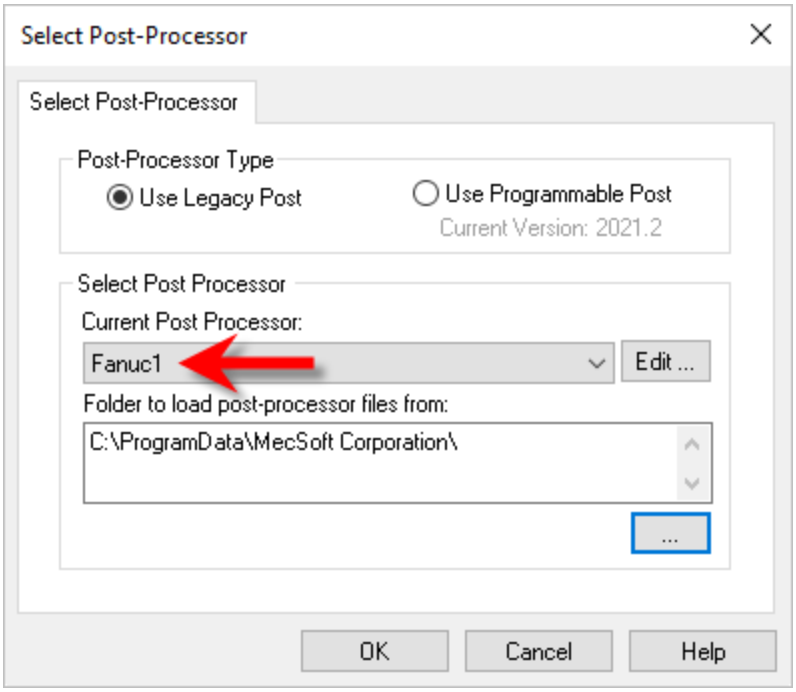
4.8 Select the Post Processor

Next, we'll define the Post Processor.

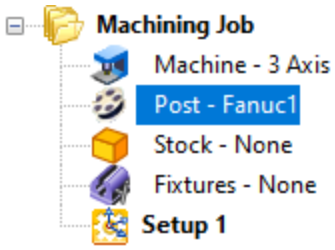
1. From the Program tab select Post to display the dialog.



2. For the Current Post Processor, select Fanuc1 from the list of available posts.
3. Then set the Posted File Extension to .nc. Other file extensions are available depending on your machine requirements.



4. Pick OK and notice that the Post type is now defined under the Machining Job tree in the Machining Browser.

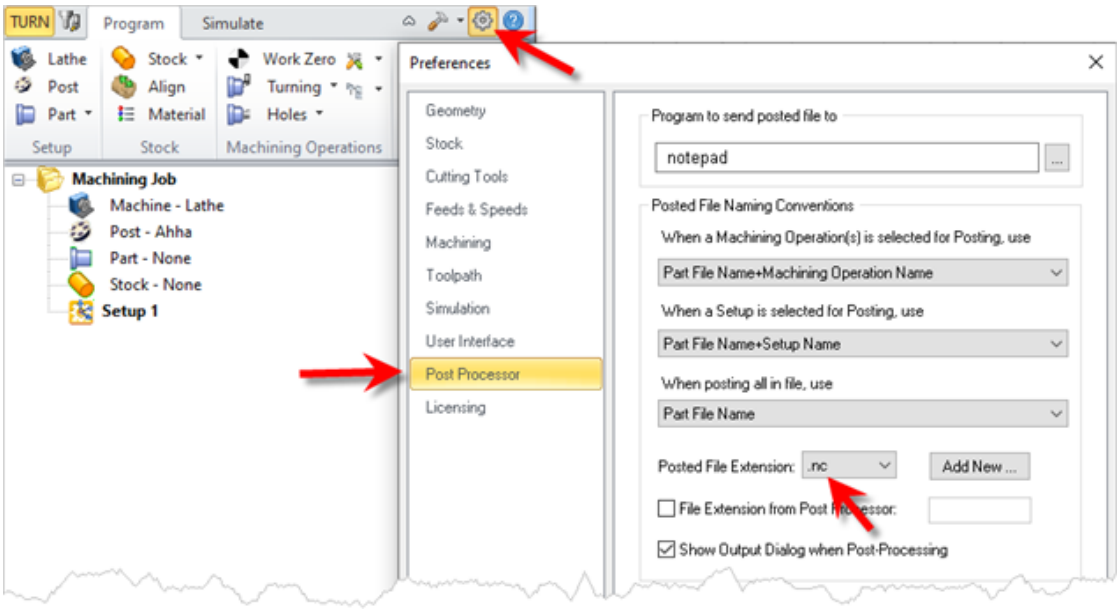


By default, post processor files are located under:

C:\ProgramData\MecSoft Corporation\VisualCAM 2023\Posts\MILL\SPM

The program to send the posted output is set to notepad.

5. Now let's have a look at the Post related Preferences. Pick the CAM Preferences icon at the top left of the Program tab and then select the Post-Processor tab as shown below.



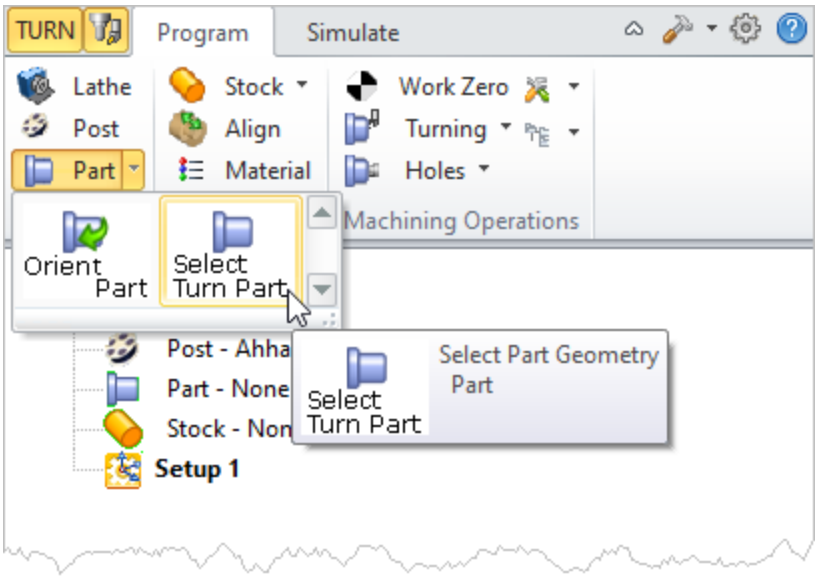
For Post File Extension select .nc from the dropdown list. If you need a different extension, pick the Add New button and enter your file extension and pick OK. The posted file extension looks like this: my-gcode-file.nc

Defining the Machine Setup

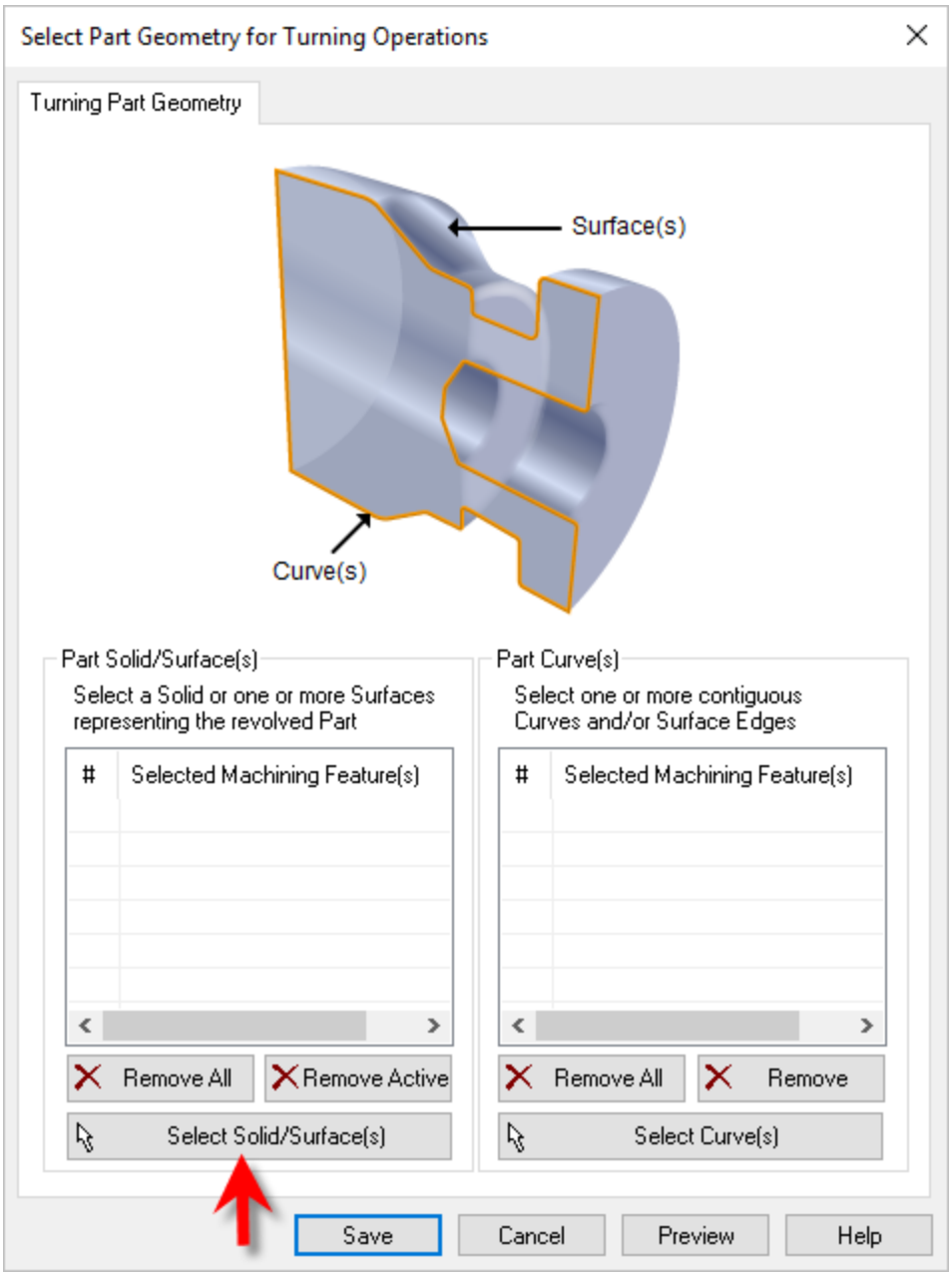
5.1 Define the Part Geometry

Part Geometry constitutes the end product of the manufacturing operation. The TURN module requires you to select solid/surfaces/polygon meshes or curves that defines the part geometry. Once selected this part geometry will be used for all machining operations.

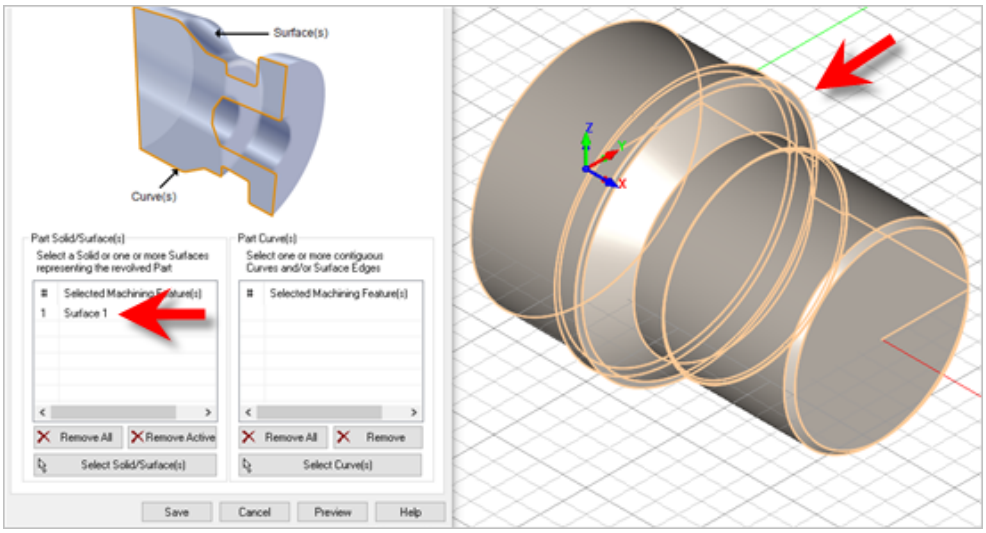
1. From the Program tab select Part and then Select Turn Part from the menu to display the dialog.



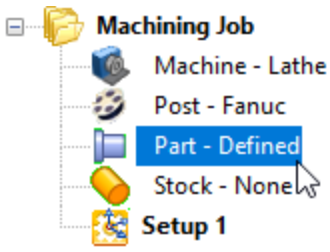
2. Pick the Select Solid/Surfaces(s) button.



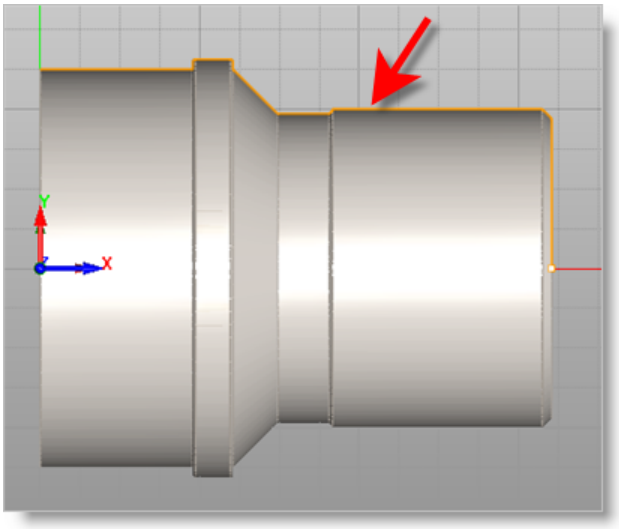
3. The dialog is minimized and allows us to select Part Objects.
4. Select the 3D solid and then right-click or press Enter to accept the selection.
5. The dialog reappears and lists the selected surfaces under Selected Machining Features.



6. Now pick Save.
7. The Part is now defined and listed under the Machining Job in the Machining Browser.



8. Now select Part - Defined from the Machining Job in the Machining Browser. Switch to the Top View. You can see the actual 2D profile that was created to use in toolpath computations.



In the future, if surfaces are selected as part geometry, the system will slice the selected surfaces with the ZX plane of the Machine Coordinate System (MCS) and use the resultant curves as the actual profile to be machined on the lathe.

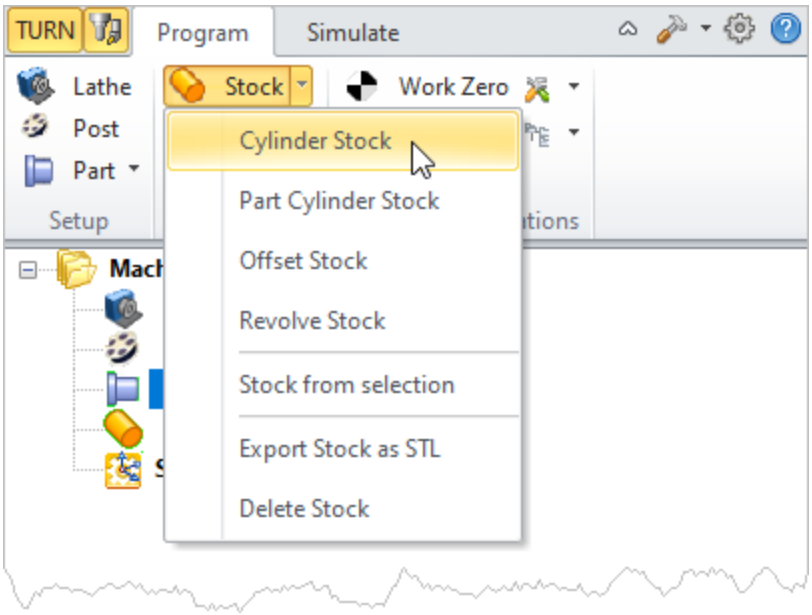
Alternatively you can perform the slicing using CAD tools and select just the resultant curve as the part profile. If you select a curve to represent the Part, make sure the curve is a 2D curve that lies in the ZX plane of the MCS. By default the ZX plane is the same as the XY plane of the World Coordinate System.

5.2 Create the Stock Geometry

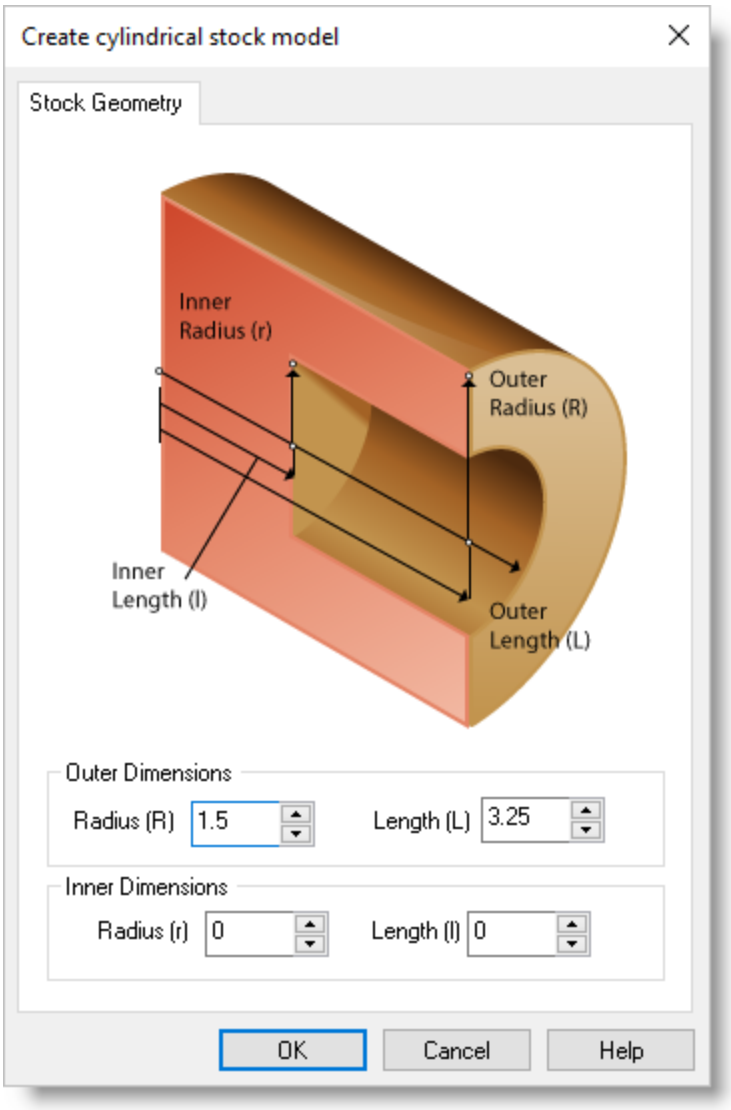
In this step we'll define the raw stock from which to cut the part.

1. First, switch back to the Isometric View.

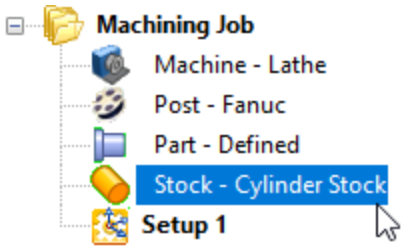
2. From the Program tab select Stock and then select Cylinder Stock from the menu to display the dialog.



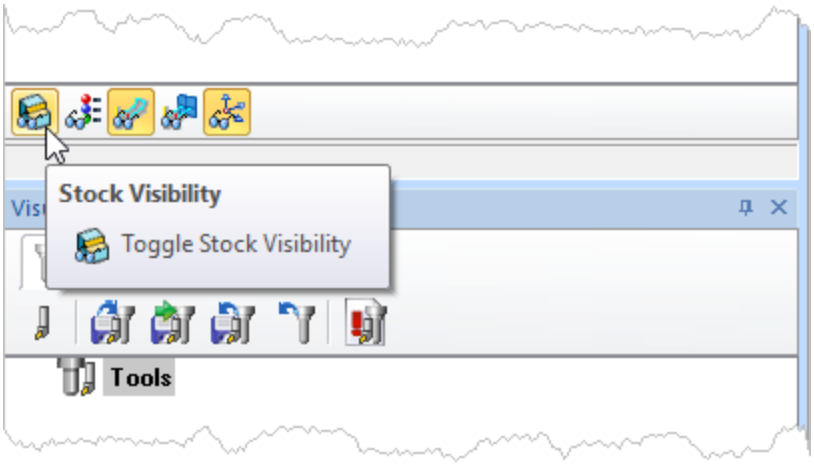
3. Under Outer Dimensions, set Radius (R) to 1.5 and Length (L) to 3.25.

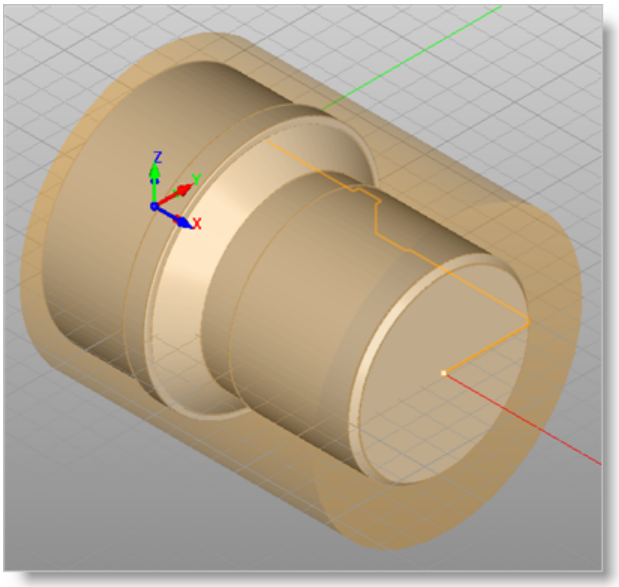


4. Pick OK and notice that the Stock type now appears under Machining Job in the Machining Browser.



5. If the stock does not display on the screen when selecting Stock - Cylinder Stock from the Machining Browser, select the Stock Visibility icon located at the base of the Machining Browser.

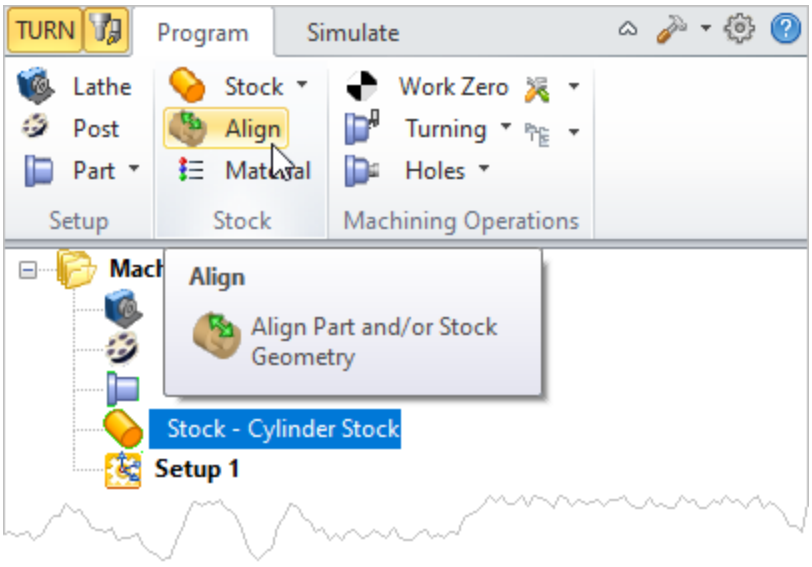




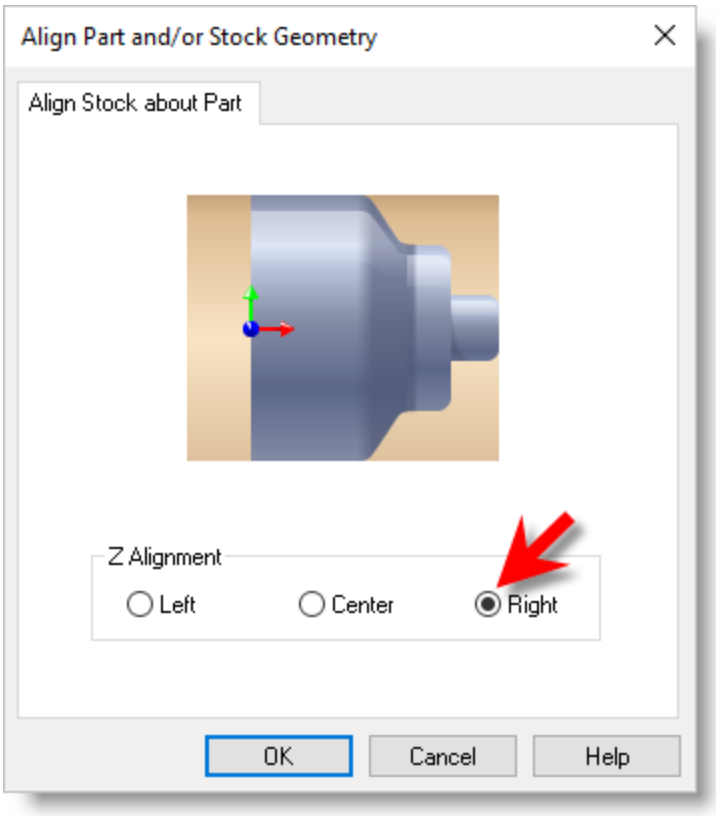
5.3 Align the Part and Stock

Once the stock model is created you can move it in alignment with the part if needed.

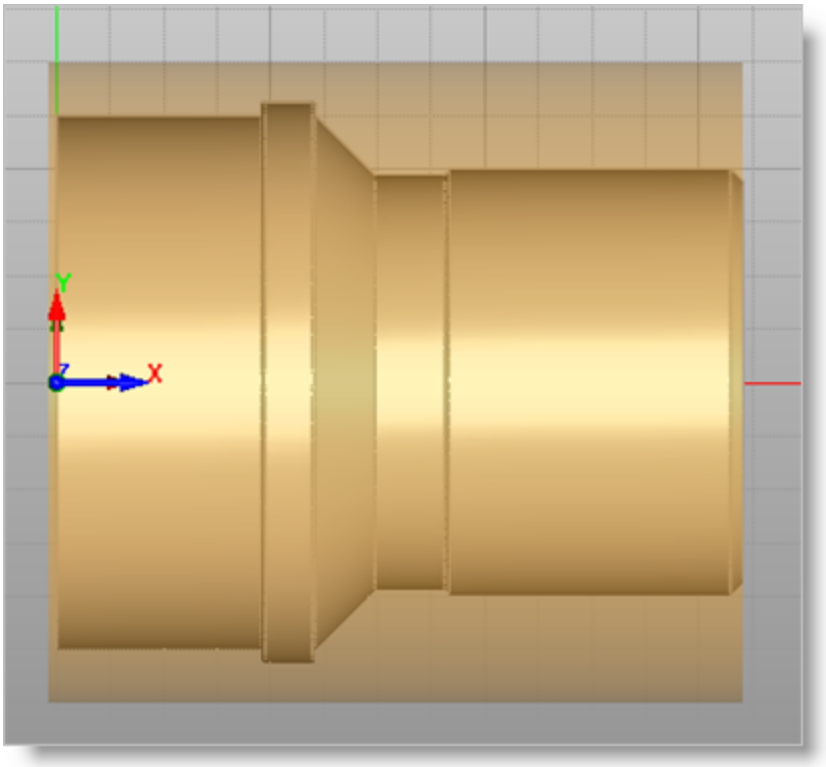
1. From the Program tab select Align.



2. For Z Alignment select Right and then pick OK.



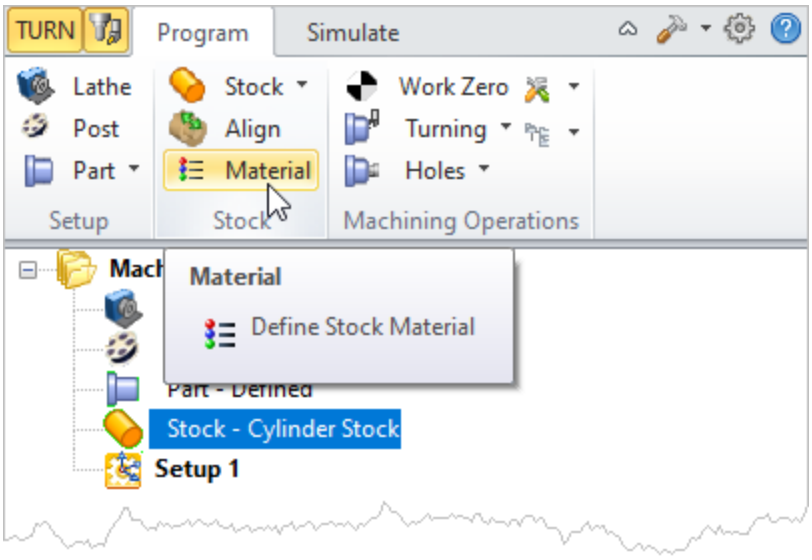
3. If you switch to the Front View, you see that the stock is now aligned to the right side face of the part geometry in the Z axis of the lathe.



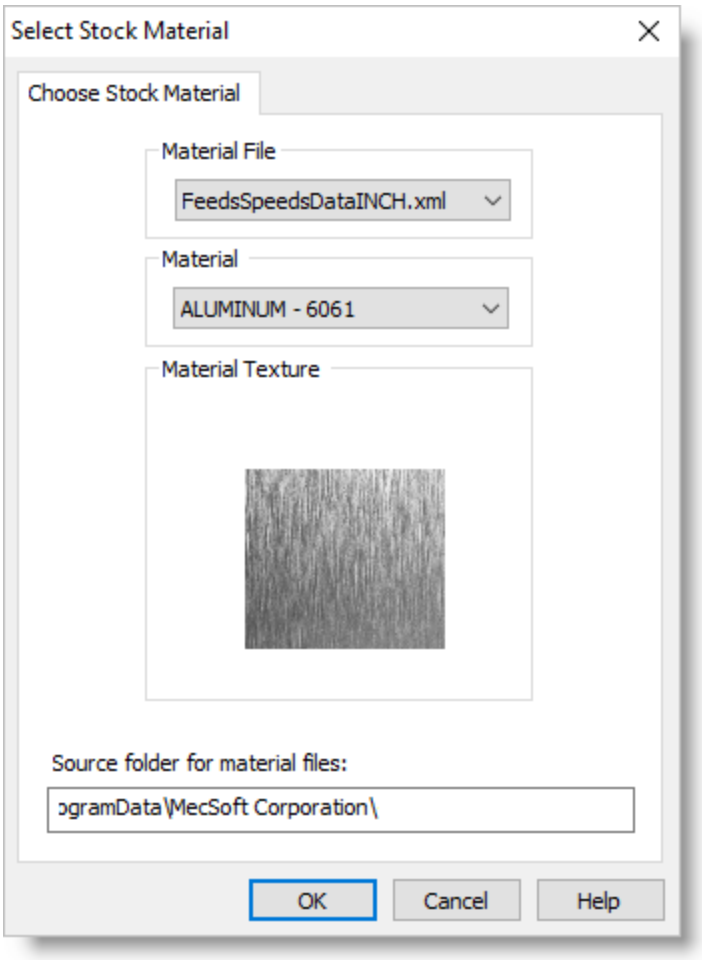
5.4 Specify the Material

We will now set the material for the stock geometry. The material definition can be used for display purposes as well as to compute feeds and speeds values for machining.

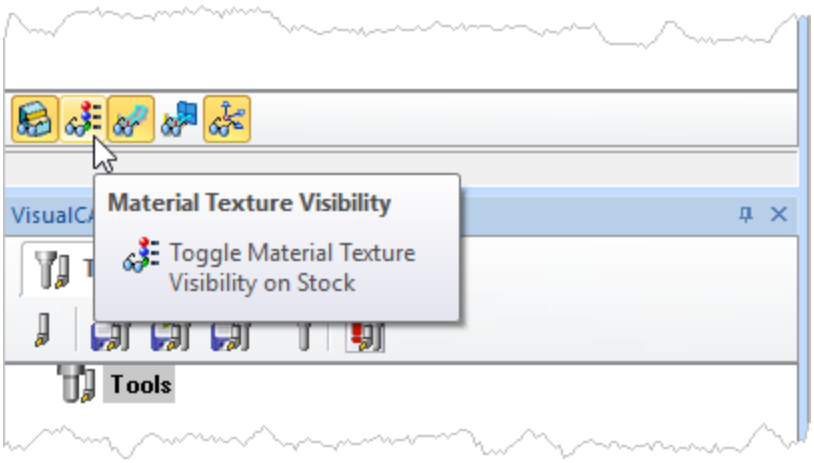
1. From the Program tab select Material to display the dialog.

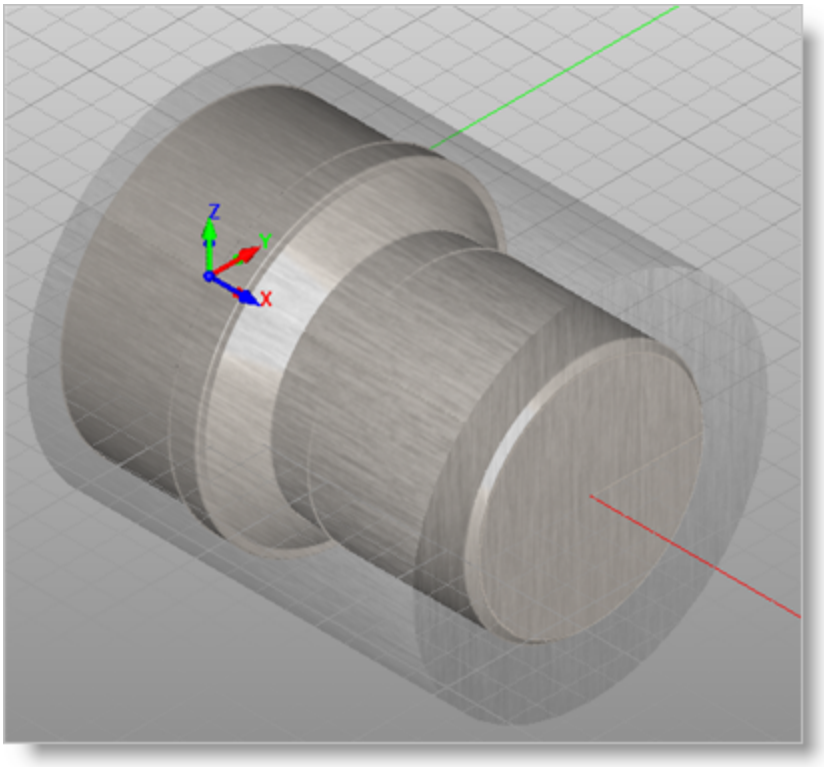


2. For Material, select Aluminum-6061 from the list of available materials and then pick OK.



3. If the material texture does not display on the stock, select the Material Texture Visibility icon located at the base of the Machining Browser.





5.5 The Machining Setup

Now let's discuss the Machining Setup.

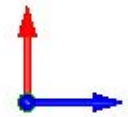
The Setup icon (Setup 1) displayed in the Machining Job tree defines the Turn Machine Coordinate System or (MCS) and is defined automatically. CNC Turning centers or Lathes use the Cartesian coordinate system for programmed coordinates.

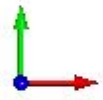
They follow the convention that the spindle axis of rotation is designated as the Z axis. The axis perpendicular to this axis along which the tool travels to cut into the stock is designated as the X axis.

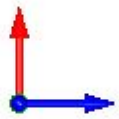
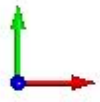
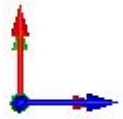
So the part and spindle rotate about the Z-axis and moving the tool along the Z-axis provides the direction of feed and moving it along the X-axis provides the depth of cut.

By default, in the TURN module, the lathe Z axis is aligned with the World X axis and the lathe X axis is aligned with the World Y axis. (The Lathe Y axis points in the same direction of the World Z axis.)

The Turn Machine Coordinate System (MCS) is displayed as a triad with a **Blue** Z-axis, a **Red** X-axis and a **Green** Y-axis. The World Coordinate System or (WCS) is displayed the same way but with XYZ coordinates labeled on top of it.





Orientation Parallel to	Triad Display States		
	MCS Visibility ON WCS Visibility OFF	MCS Visibility OFF WCS Visibility ON	MCS Visibility ON WCS Visibility ON
Front View			

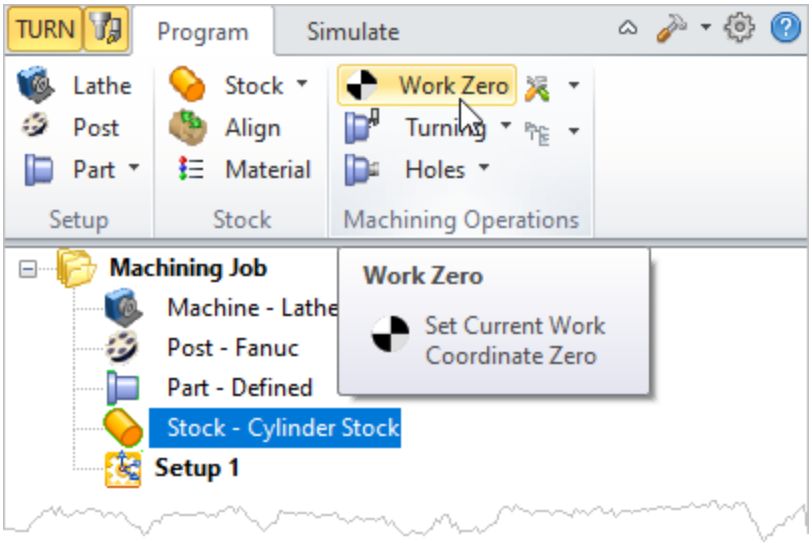
By default Setup 1 is created when a new part is loaded. The MCS of this setup is oriented as described above. That is, the world XY plane is the same as the Lathe ZX plane. This cannot be changed. In the future, when you create a part profile for turning, create it in the XY plane of the WCS.

5.6 Set the Work Zero

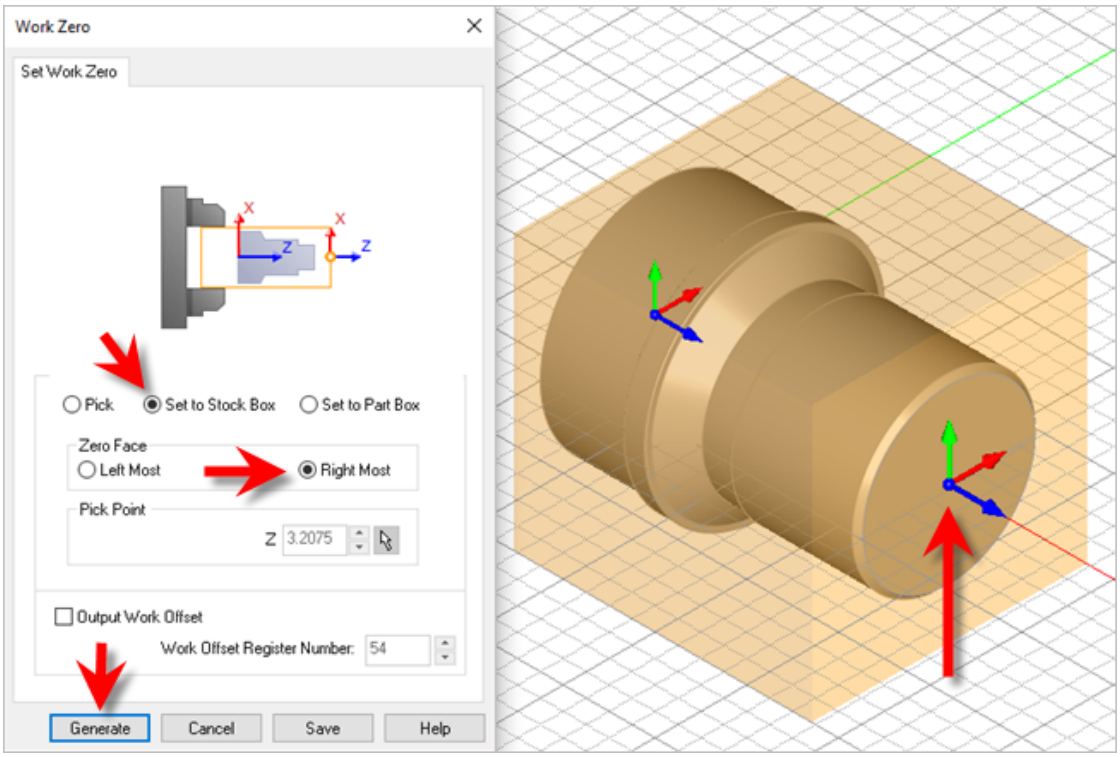
The Work Zero is used to define the work-piece origin. The Work Zero translates the Machine Coordinate System (MCS) origin from the origin defined in the Setup to the desired Work Zero location. This can be set to any location along the lathe Z axis. Typically this is set to the right most face of the part or stock geometry on the lathe Z axis.

It is important to understand that the Work Zero defines the zero point from which all toolpath points are interpreted by the controller. This would normally be the same as the tool touch off point on the actual work-piece on your machine. So care should be taken to make sure that this Work Zero point matches the tool zero point used on the actual work piece located in your machine.

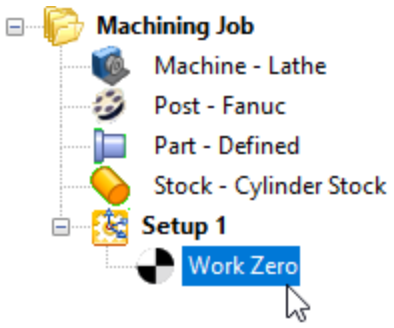
1. From the Program tab select Work Zero to display the dialog.

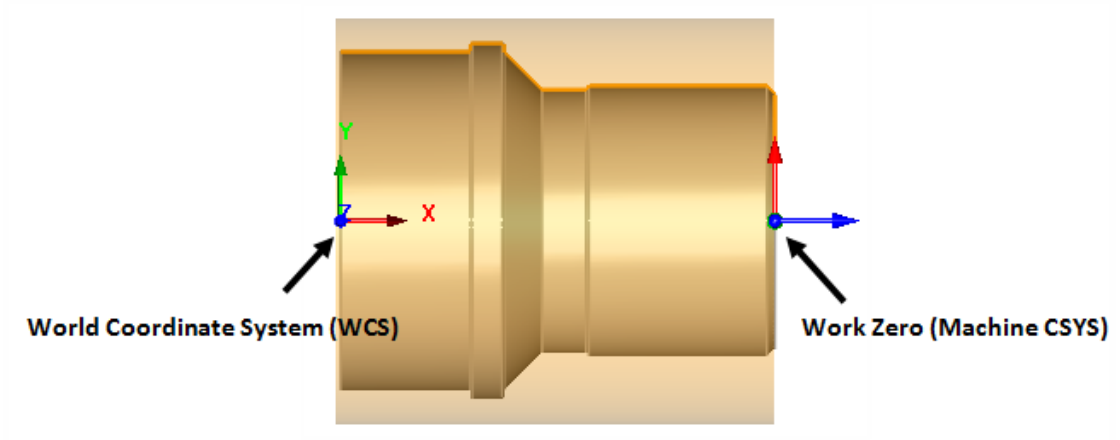


2. Select Set to Stock Box and set the Zero Face to Right Most. This locates the machine origin point to the right most face of the stock geometry along the lathe Z axis.



3. Pick Generate and notice that the MCS is translated and that the Work Zero now appears under Setup 1 in the Machining Browser.



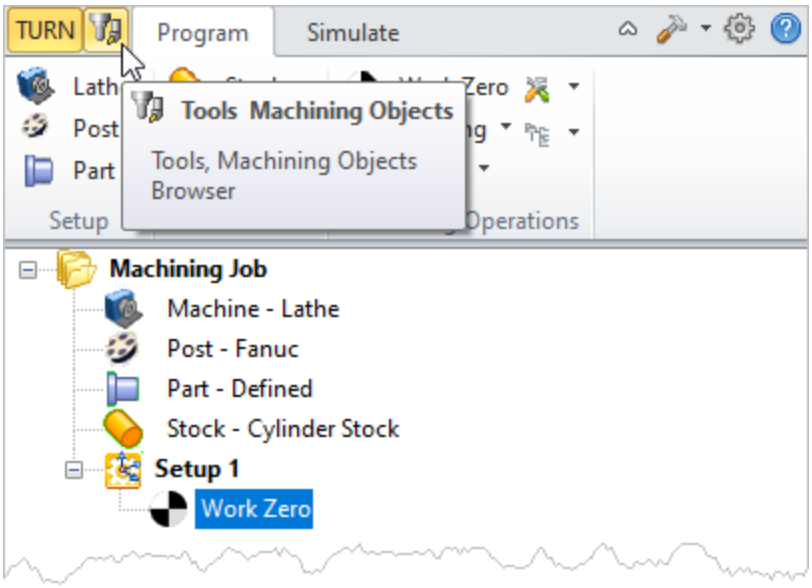


Create a Cutting Tool

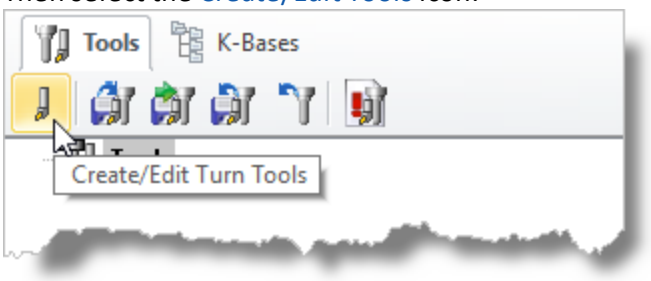
To machine our part we will create a diamond insert with a 20 degree relief angle, a 0.02 inch tip radius and with 0.5 inch inscribed circle radius.

1. Next to the Program tab at the top of the Machining Browser, locate and select the Tools Machining Objects button. Selecting this button toggles the Machining Objects lower portion of the browser On and Off. Then locate the Tools tab and pick the Create/Edit Tools icon.

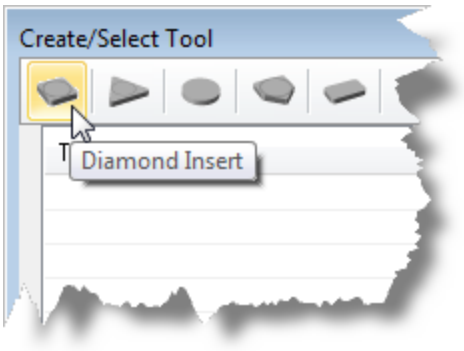
   These buttons and icons are shown in the menus below.



   Then select the Create/Edit Tools icon:

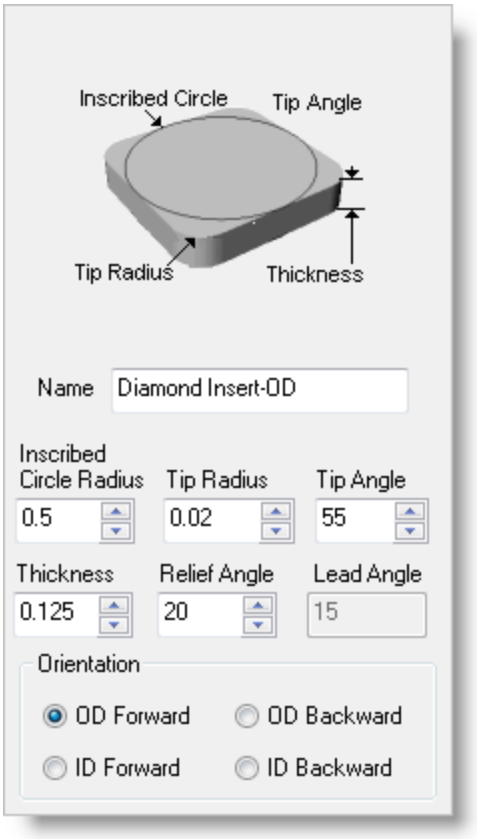


2. This will display the Create/Select Tool dialog. Select the Diamond Insert tool icon from the top-left side of the dialog.



3. Now set the following parameters in the dialog:

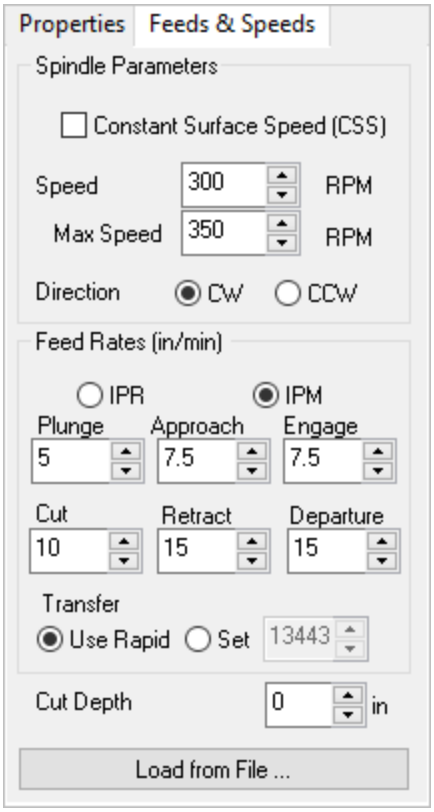
   Name: Diamond Insert-OD
   Inscribed Circle Radius: 0.5
   Tip Radius: 0.02
   Tip Angle: 55
   Relief Angle: 20
   Thickness: 0.125
   Orientation: OD Forward



4. Next, we'll switch to the Feeds and Speeds tab and use the following values.

For Spindle Parameters we'll set:
Speed: 300 RPM
Max Speed: 350 RPM

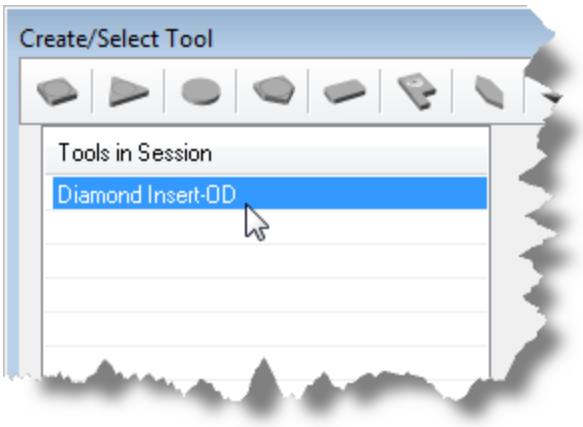
For Feedrates we'll select IPM and set:
Plunge: 5
Approach: 7.5
Engage: 7.5
Cut: 10
Retract: 15
Departure: 15
Transfer: Use Rapid
Cut Depth: 0



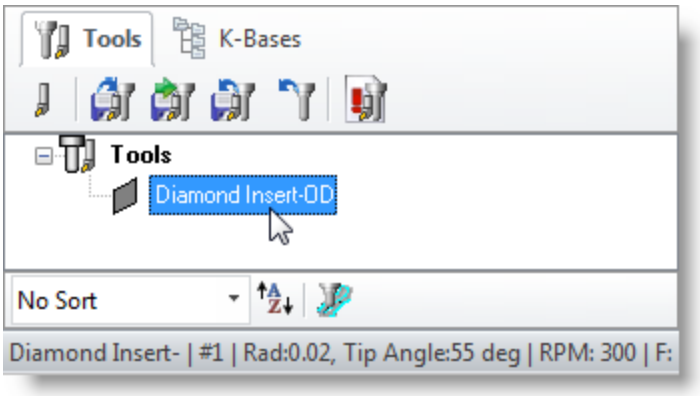
5. Now, we'll pick Save as New Tool.

> In the future you can edit tool parameters and click Save Edits to Tool to save the changes. You can create additional tools by assigning a different Name and tool parameters.

6. The tool is created and listed under Tools in Session on the left.



7. Pick OK to close the dialog and notice that the new tool is also listed under the Tools Tab of the Machining Objects Browser.

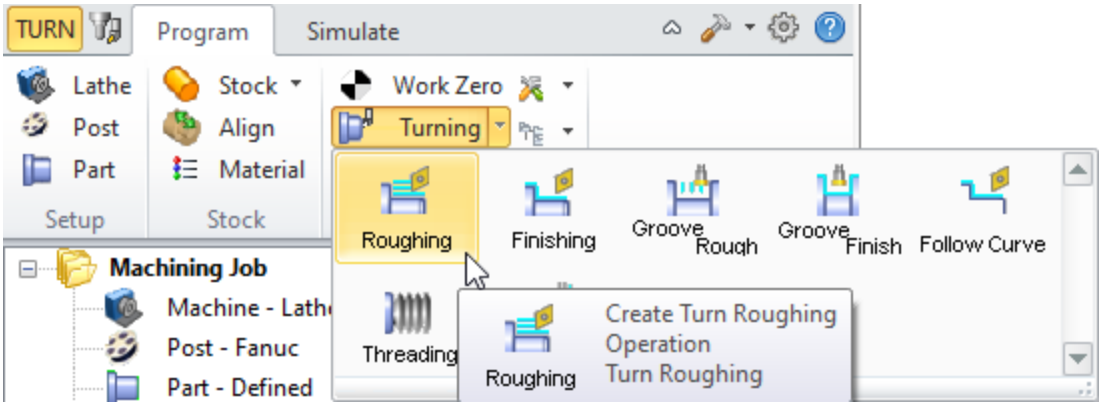


In the future, to save Tools to a library, click Save Tool library under the Tools tab in Machining Objects Browser and specify a folder location and file name in the Save as dialog box.

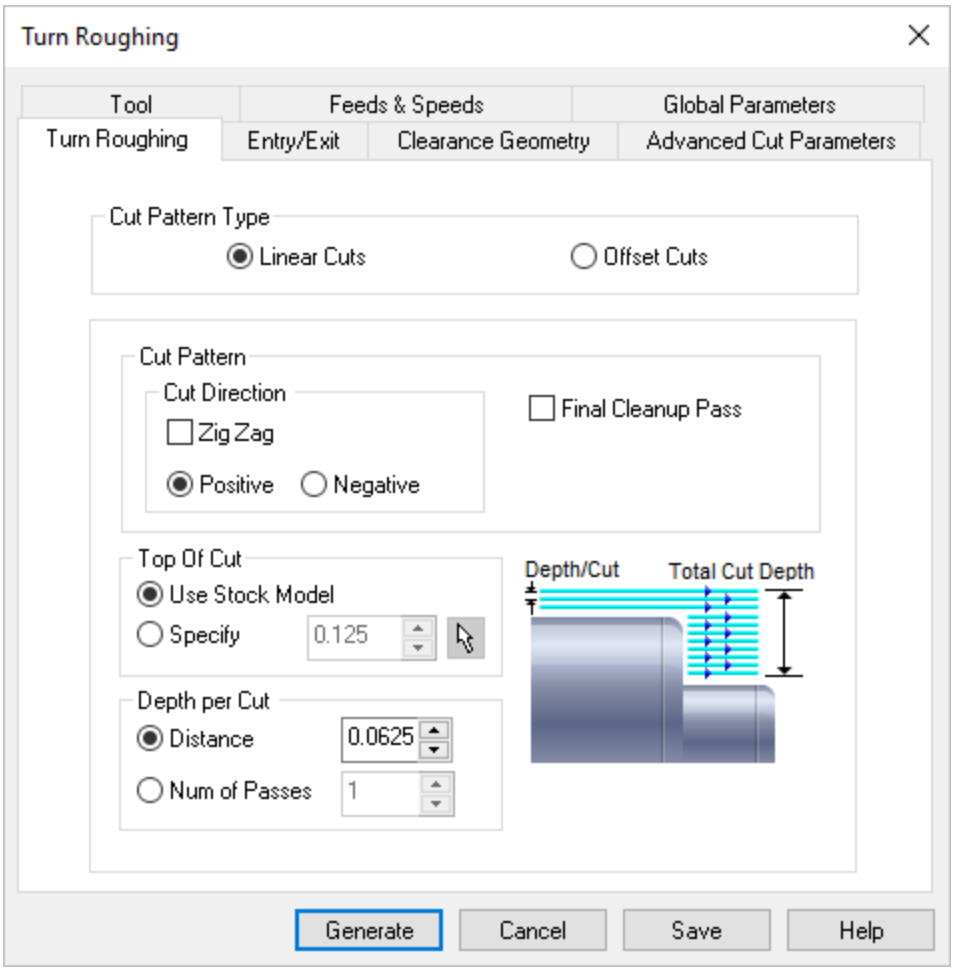
Outer Diameter Turn Roughing

Now were ready to create our Turn Roughing operation for machining the part.

1. From the Program tab select Turning and then Roughing from the menu of operations.



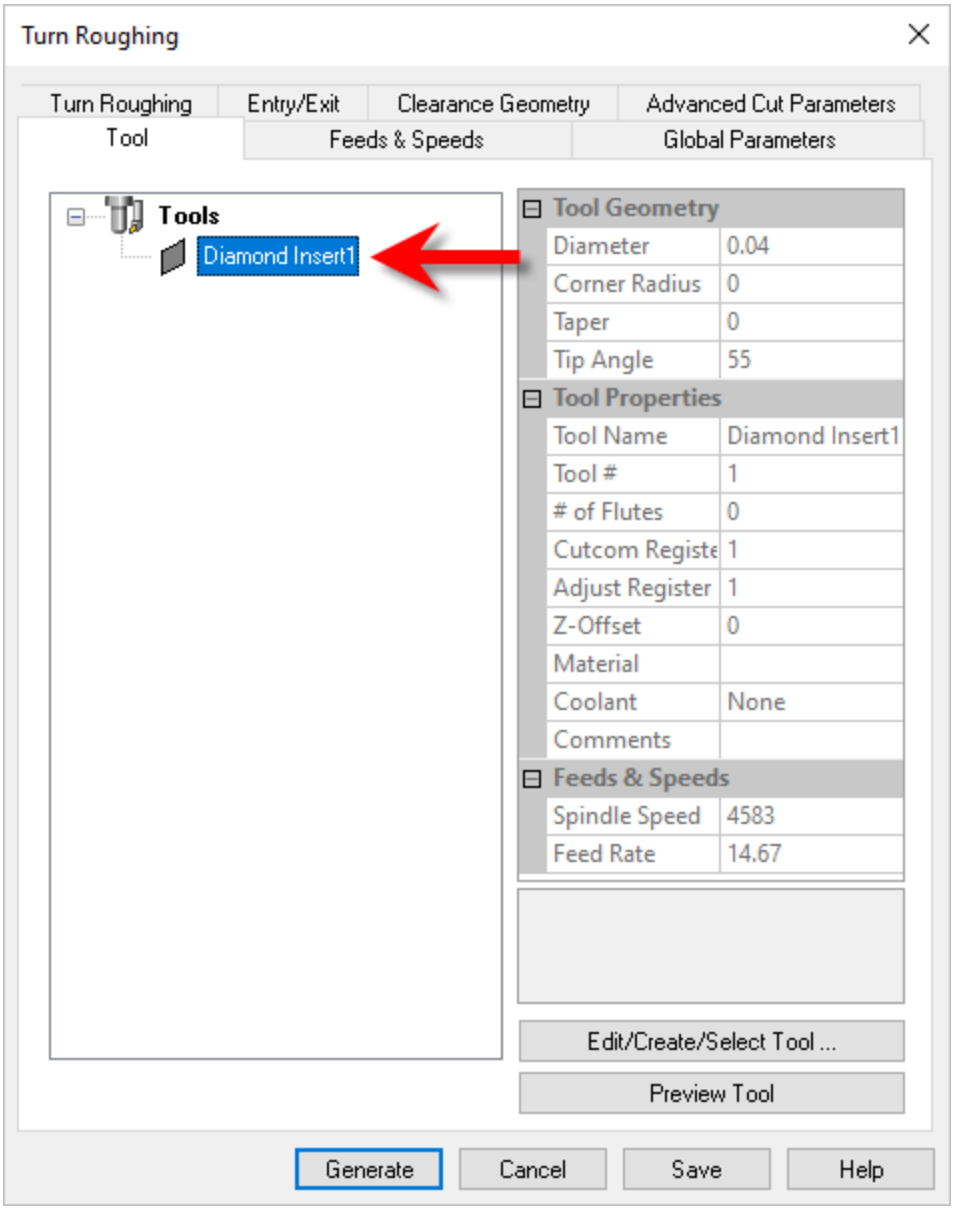
2. This will display the Turn Roughing operation dialog.



7.1 Select the Cutting Tool

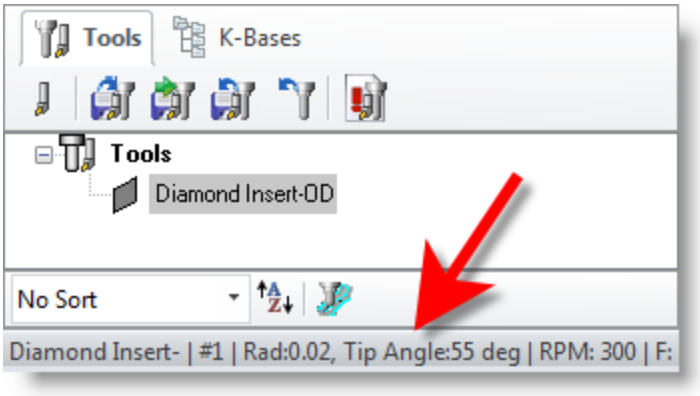
Next we'll select the cutting tool for the Turn Roughing operation.

1. From the Tool tab we'll select the Diamond Insert-OD tool we just created as the active tool.



2. The diamond insert is now selected as the active tool.

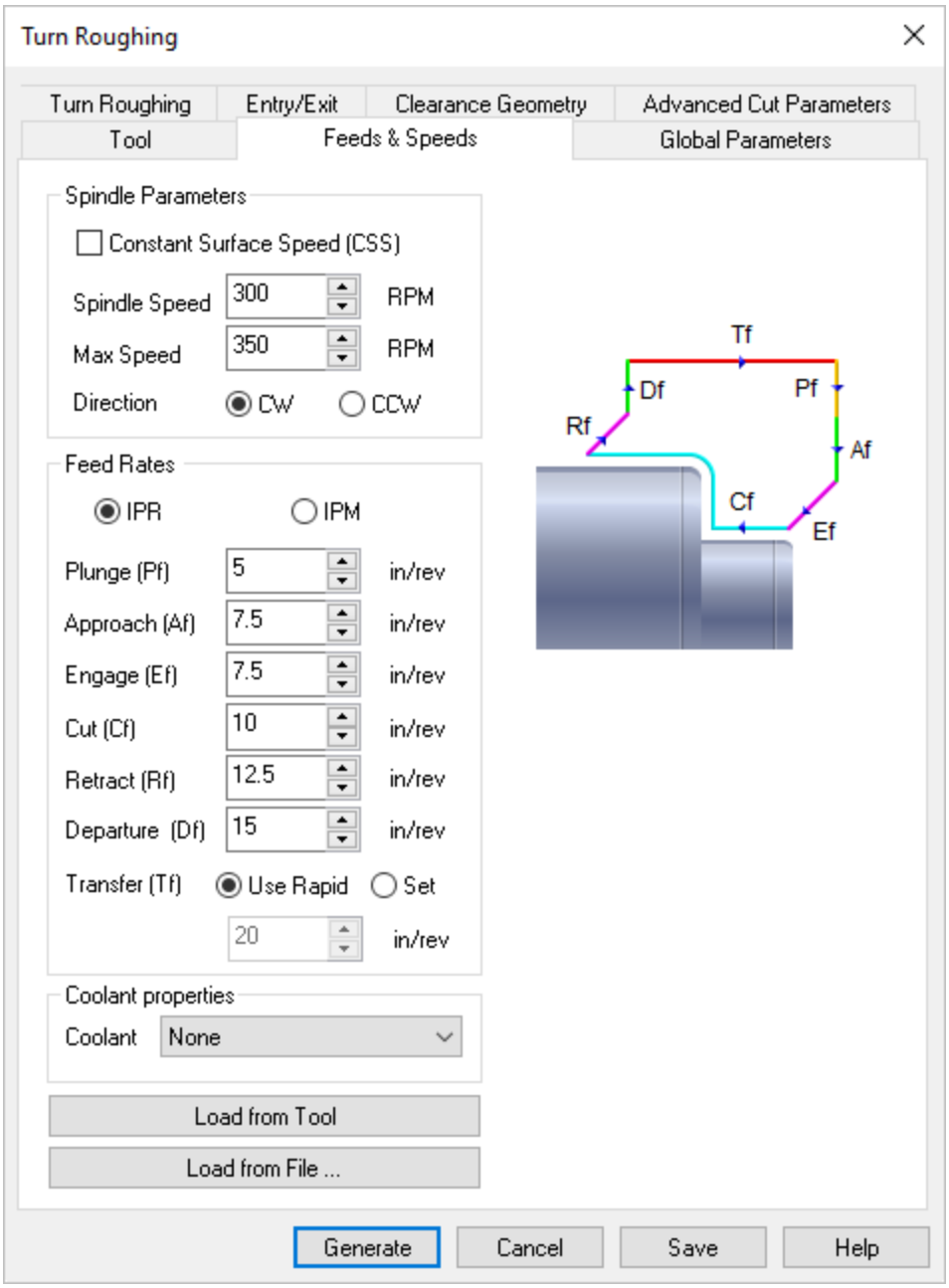
> The Tool parameters of the currently active tool are always displayed in the status bar at the bottom of the Machining Objects browser as shown below.



7.2 Set Feeds and Speeds

Next we'll set the Feeds and Speeds for the Turn Roughing operation.

1. Pick the Feeds & Speeds tab of the dialog.



2. Then pick the Load from Tool button. The system will retrieve the feed and speed parameters that we set when we created the tool and associate them with the current operation.

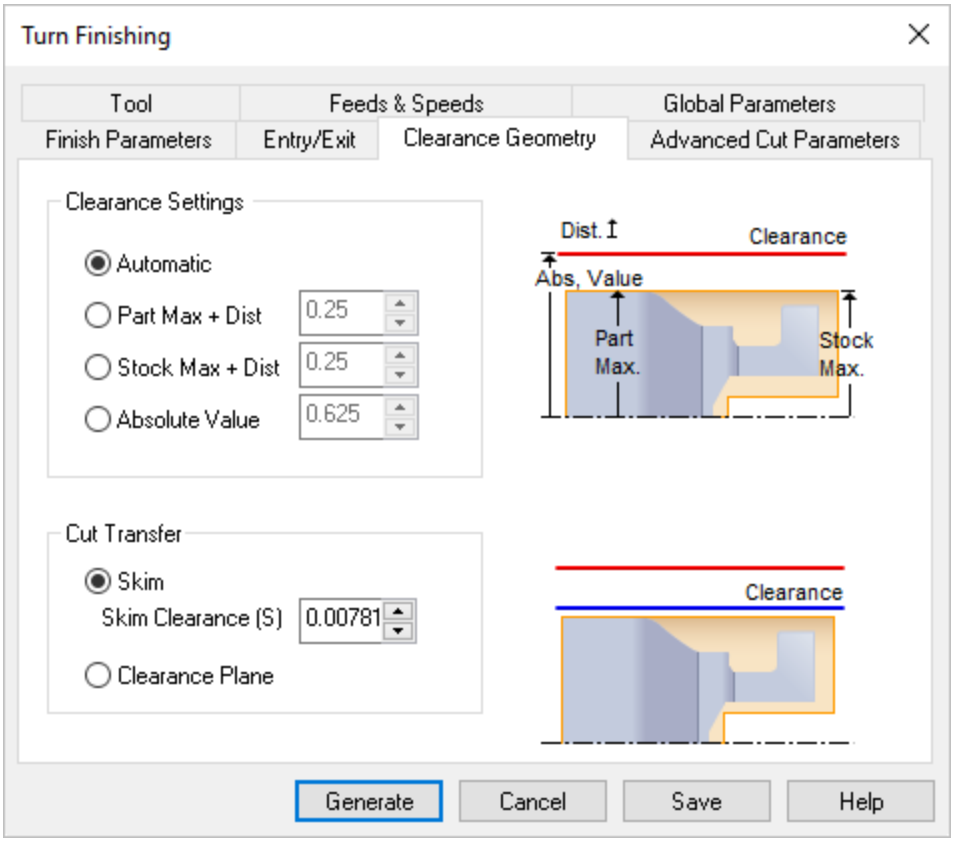
7.3 Set Clearance Geometry

Next we'll set the Clearance Geometry for the Turn Roughing operation.

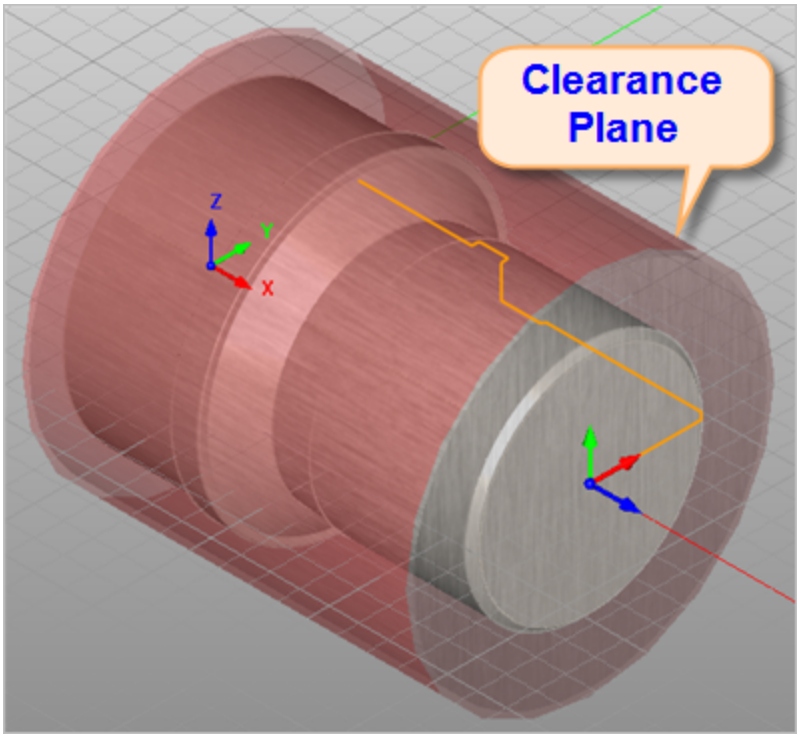
1. Select the Clearance Geometry tab of the dialog.

2. Here, we'll set Clearance Settings to Automatic and Cut Transfer to Clearance Plane.

   In the Automatic mode, the system will determine a safe height for locating the clearance plane. Setting Cut Transfer to Clearance Plane will force all transfer moves to be performed in this computed clearance plane.



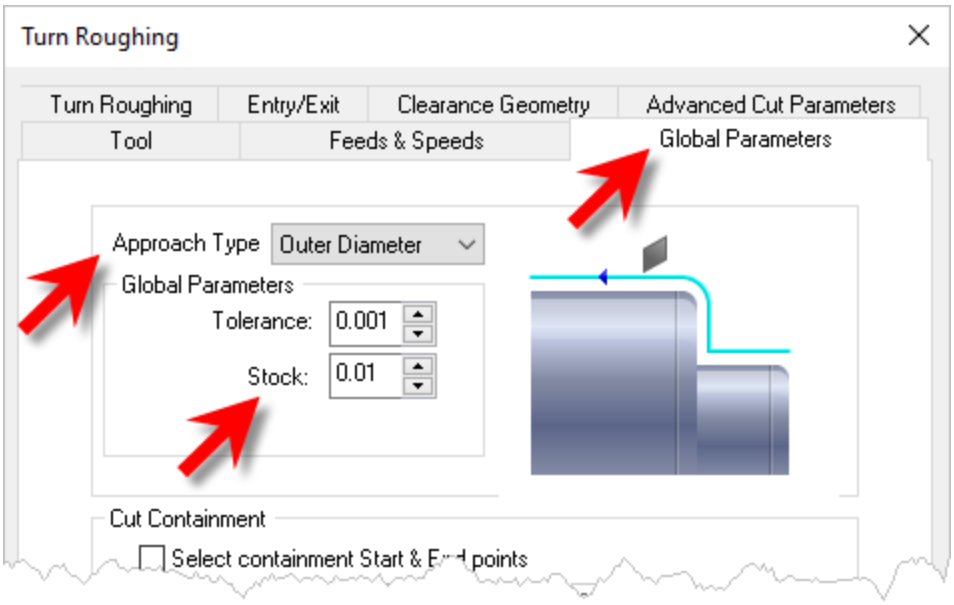
3. When this dialog is active, the clearance plane is shown on the graphics screen.



7.4 Set Global Parameters

Next we'll set the Global Parameters for the Turn Roughing operation to specify parameters to control the cutting.

1. Switch to the Global Parameters tab of the dialog.
2. Here, we'll set the Approach Type to Outer Diameter.
3. We'll set Stock to 0.01. This means that we'll be leaving a 0.01" thickness on the part after machining.

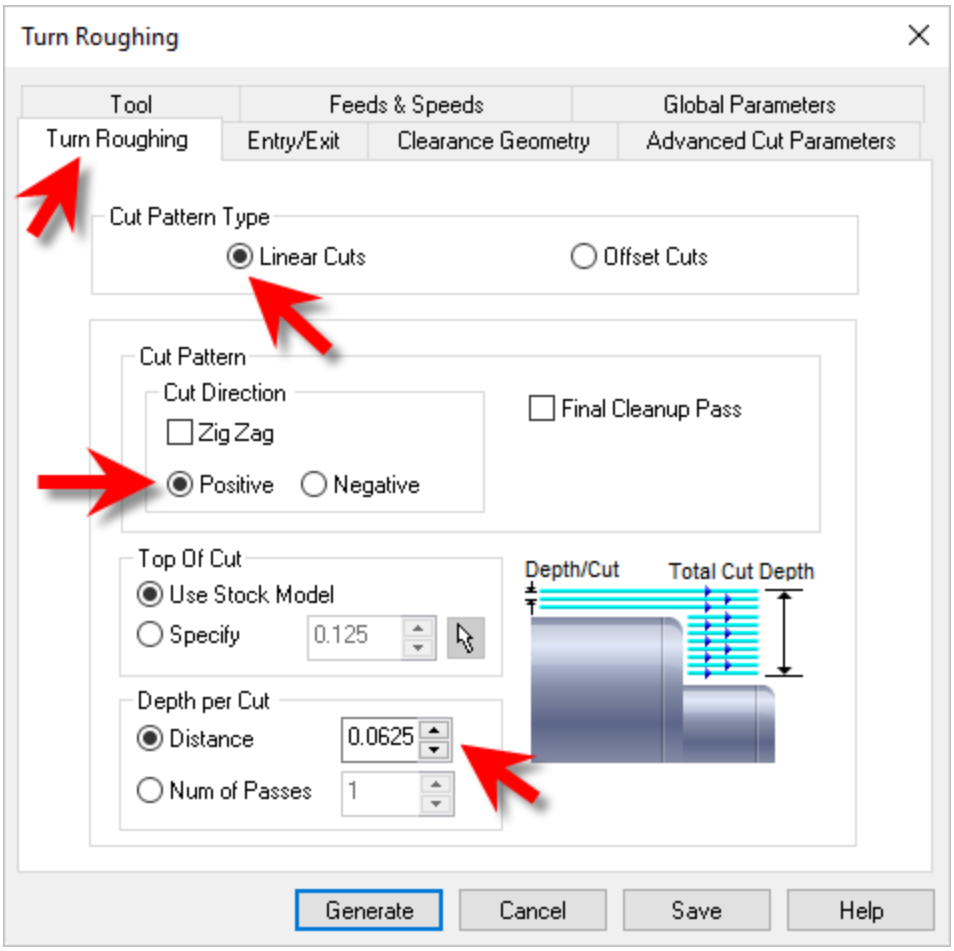


7.5 Set Roughing Parameters

Next we'll set Roughing Parameters for the Turn Roughing operation.

1. Select the Turn Roughing tab of the dialog.
2. Set Cut Pattern Type to Linear Cuts.
   This will create a cut pattern with straight line cuts. Offset cuts on the other hand will create cuts that are successive offsets of the TURN profile.
3. Set Cut Direction to Positive.
   This will ensure that the cut traverses along the positive Z axis of the Lathe coordinate system.
4. Then uncheck Final Cleanup Pass.
5. Then also set Depth per Cut to 0.0625.

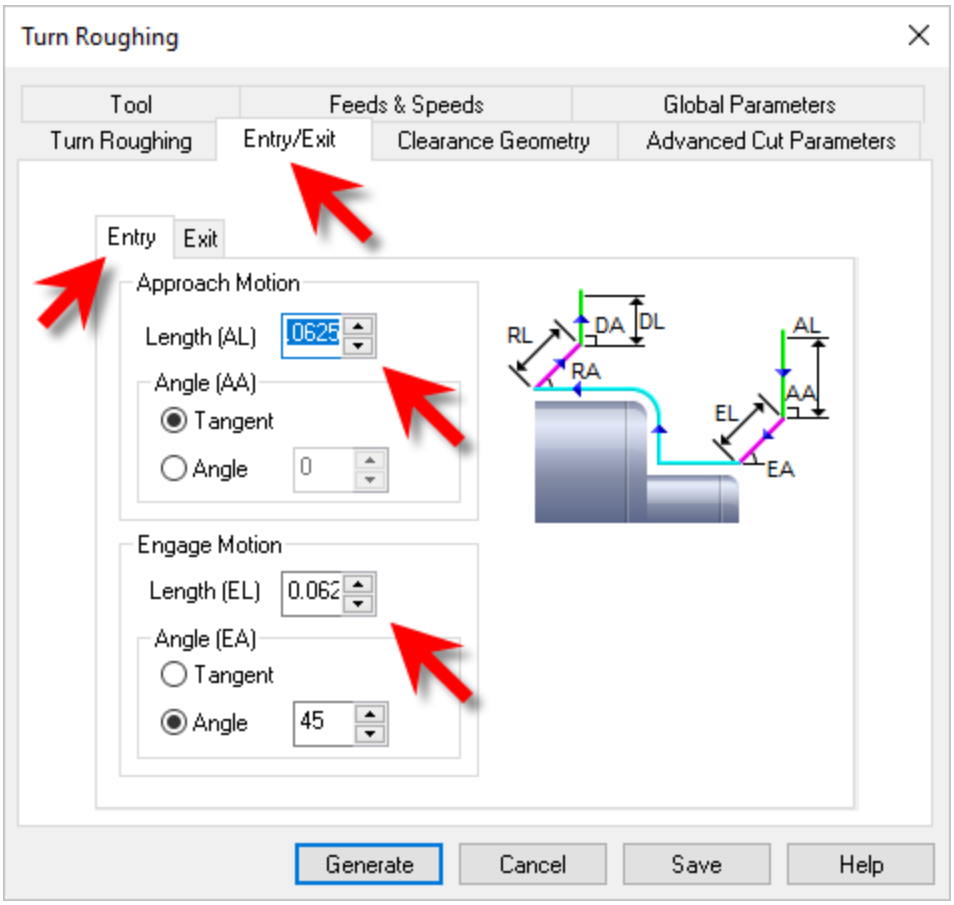
   Note that Depth per Cut is always set to an absolute value.



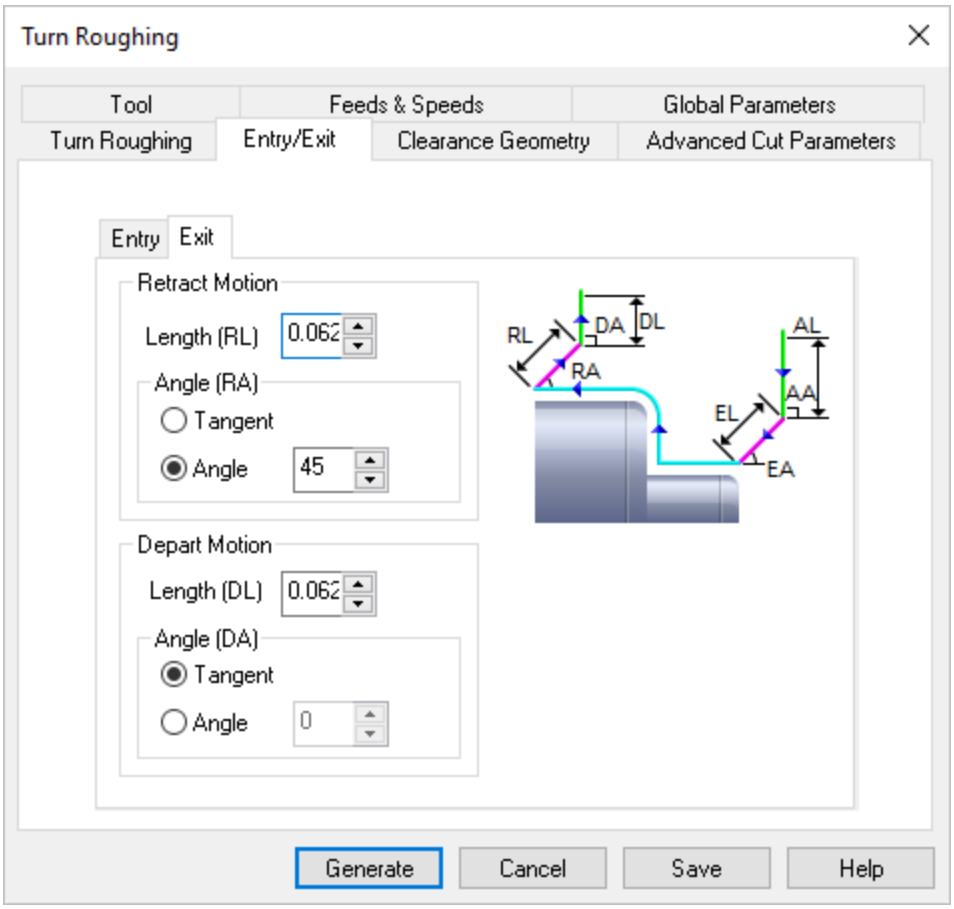
7.6 Set Entry/Exit Parameters

Next we'll set Entry and Exit Parameters for the Turn Roughing operation. Entry/Exit parameters control how the cutter will engage material as it begins cutting and how it will leave the material as it completes cutting.

1. Select the Entry/Exit tab of the dialog.
2. Select the Entry tab and set the Approach Motion Length (AL) to 0.025.
3. We'll then set the Engage Motion Length (EL) to 0.025 also.

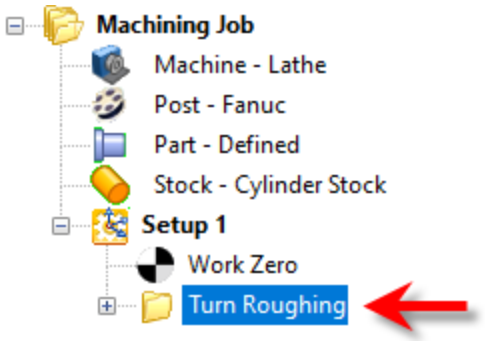


4. Next, we'll switch to Exit tab set the Retract Motion Length (RL) to 0.025 and do the same for the Depart Motion Length (DL).

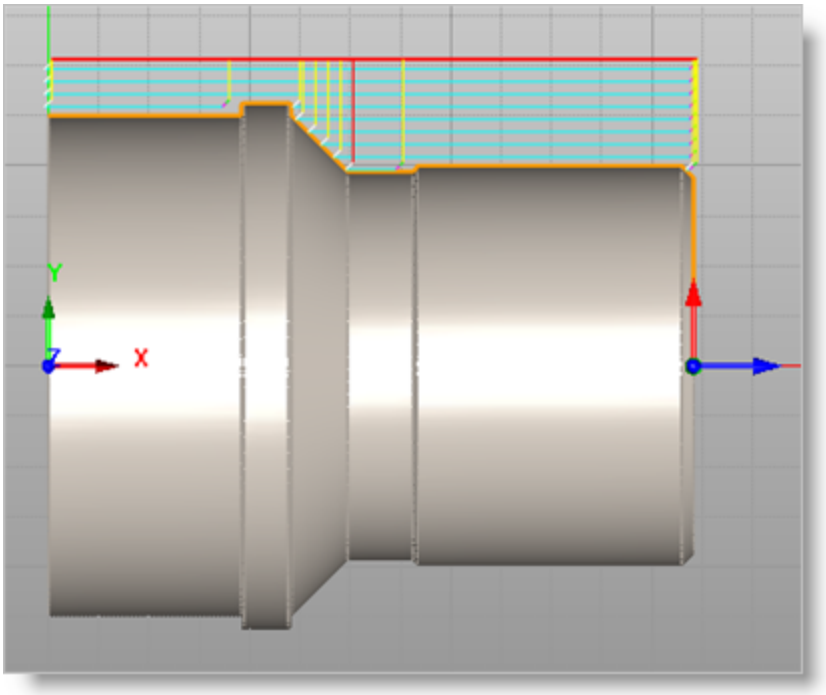


5. Now pick Generate.

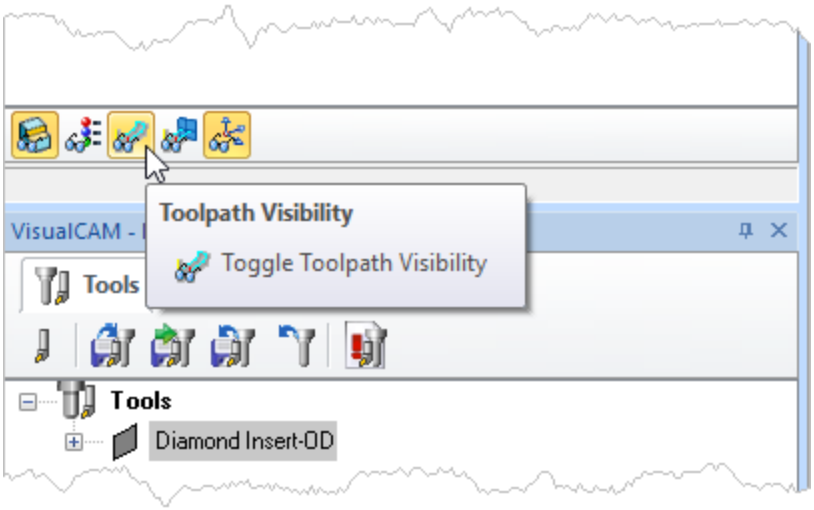
   The Turn Roughing toolpath is generated and the operation is listed under Setup 1 in the Machining Browser.



6. Now from the View toolbar in VisualCAD, select the Top View and you can see the toolpath displayed clearly in the graphics screen.



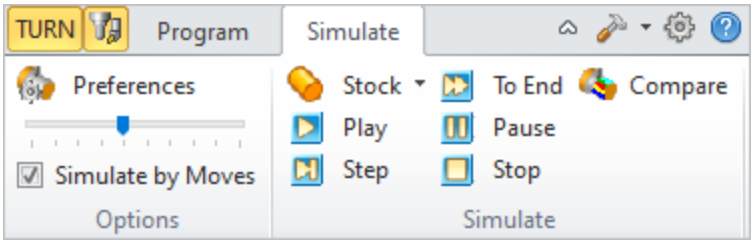
The display of the toolpath in the graphics screen can be turned on/off by selecting the Toolpath Visibility icon in the toolbar at the bottom of the Machining Browser.



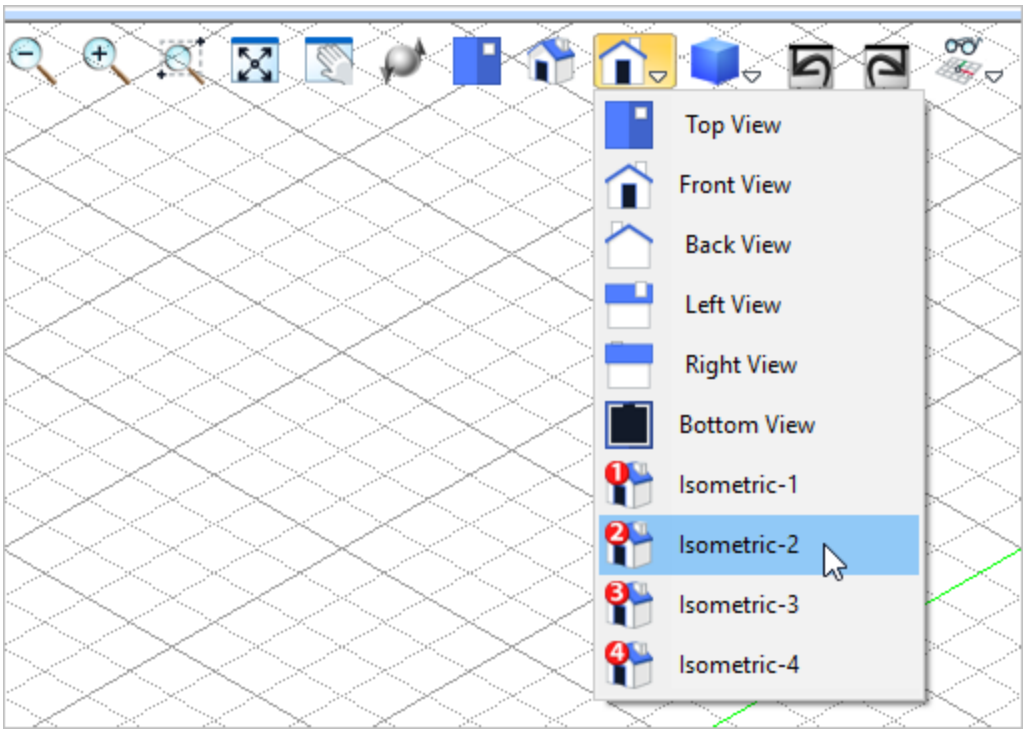
7.7 Simulate the Toolpath

The generated toolpath can now be simulated to display the in-process stock model by using the functions under the Simulate tab in the Machining browser.

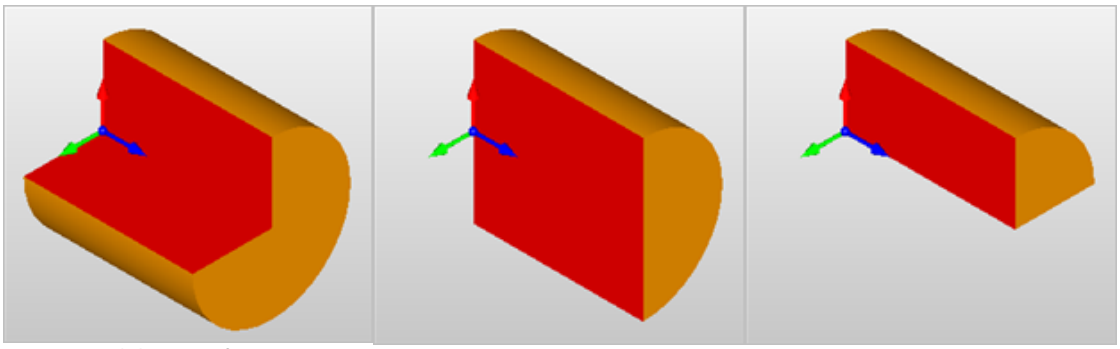
1. Switch to the Simulate tab at the top of the Machining Browser.



2. From the View toolbar, select the Isometric-2 icon

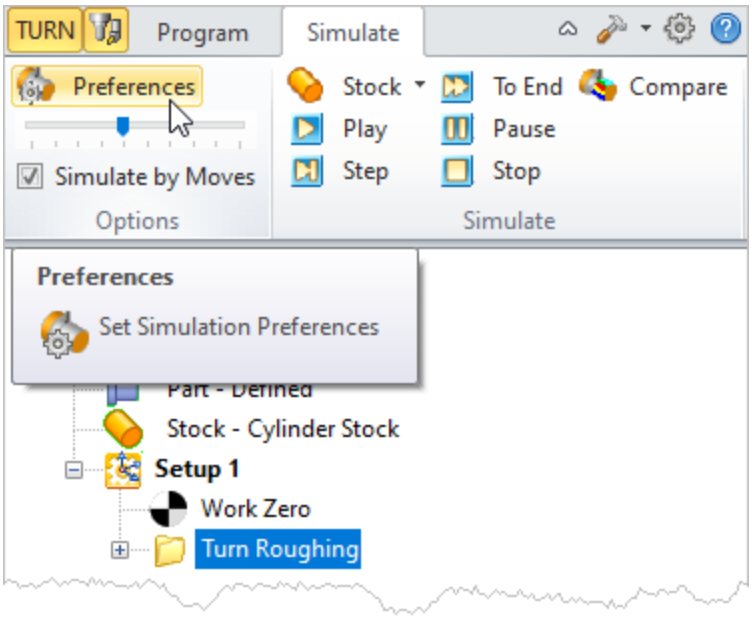


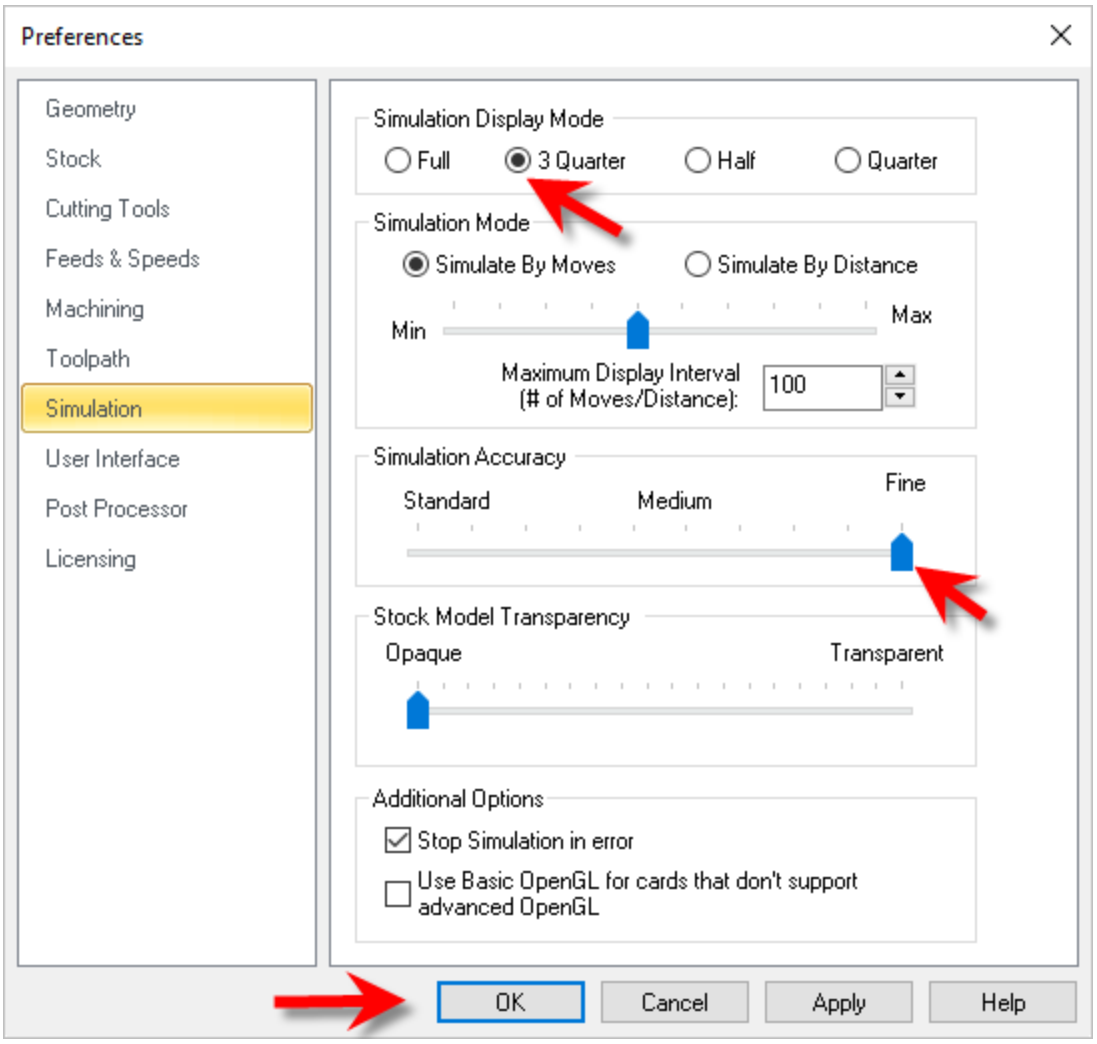
3. In the TURN module the rendering mode of the simulation model can be controlled for better visibility by using cut away section views. Three modes, in addition to the normal rendering mode, are available. These are 3 Quarter, Half and Quarter views. These display modes can be set in the Simulation Preferences dialog. They can be useful when you are machining parts with inner diameter features.



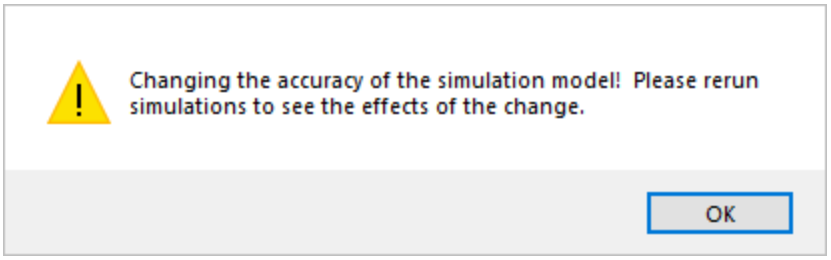
3 Quarter View | Half View | Quarter View

4. From the Simulate tab, select Preferences and set the Simulation display mode to 3 Quarter and the Simulation Accuracy to Fine and then pick OK.

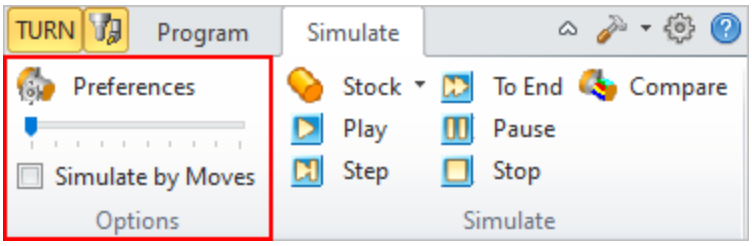




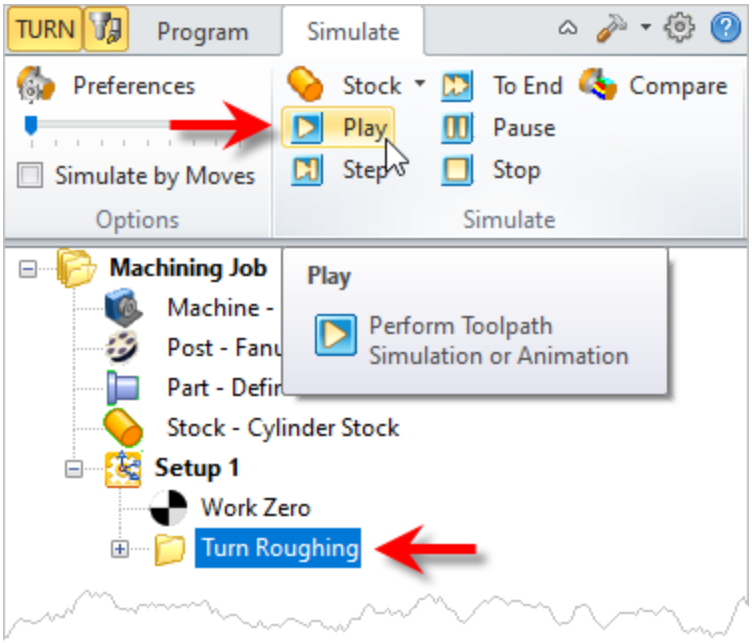
Pick OK from the message dialog that displays:



5. From the Simulate tab, uncheck Simulate by Moves and then move the slider to the left to slow down the simulation speed.

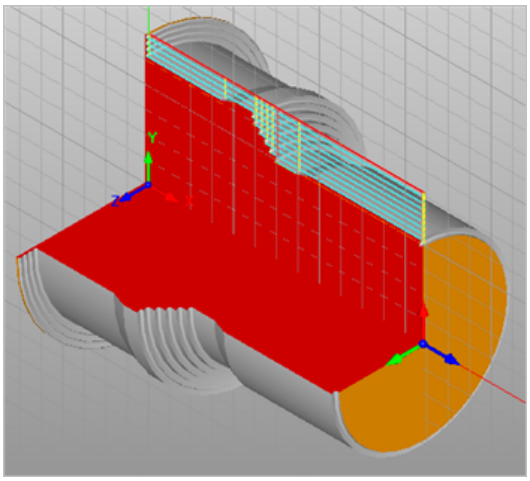


6. Now select the Turn Roughing operation under Setup 1 and then pick Play.

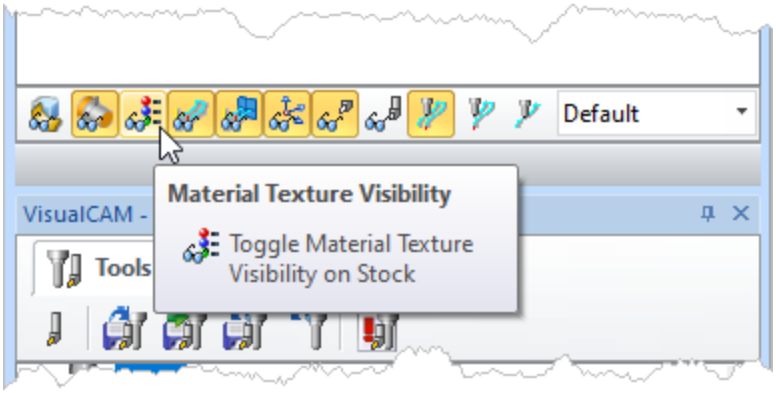


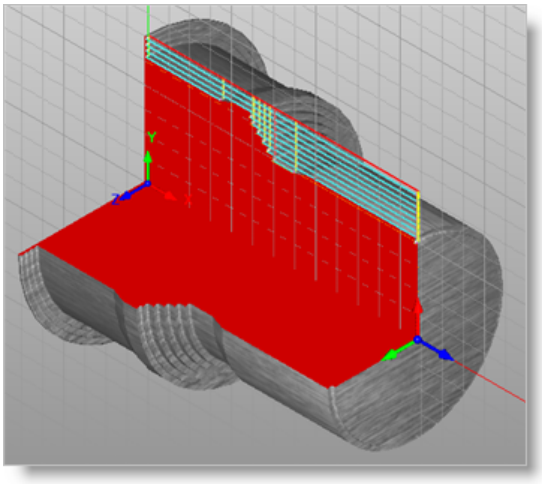
You can stop the simulation at anytime by selecting the Pause button from the Simulate tab. After Pausing, you can choose either Play to continue or Stop to exit the simulation.

7. Once the simulation is complete, the state of the stock model is displayed in the graphics screen.



8. To view the cut model with textures applied, select the Material Texture Visibility icon located at the base of the Machining Browser.

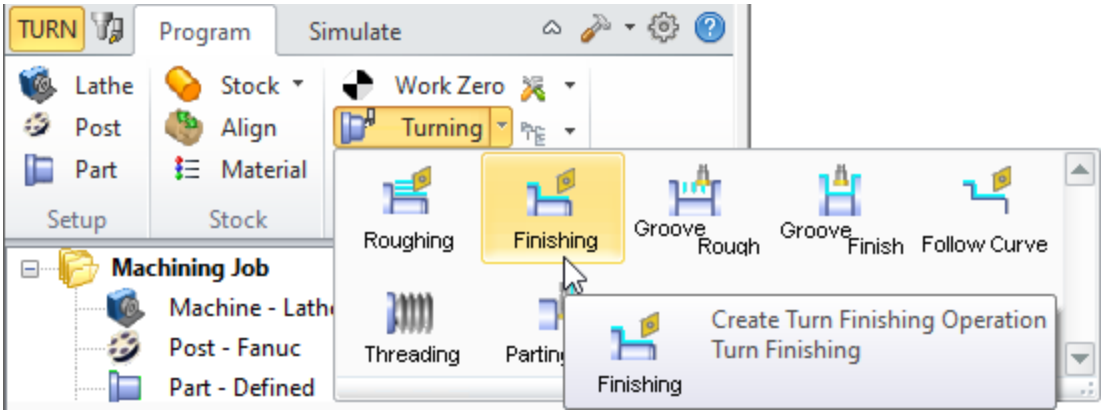




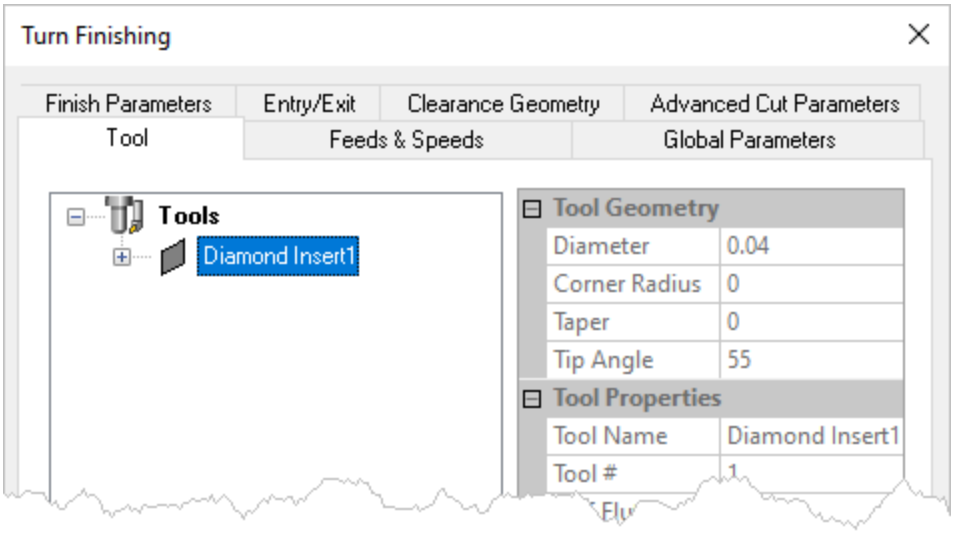
Outer Diameter Turn Finishing

Now we will turn our attention to finishing the outer diameter of the part using the same tool as we used for roughing.

1. Switch back to Program tab in the Machining Browser.
2. Select Turning and then Finishing from the menu.



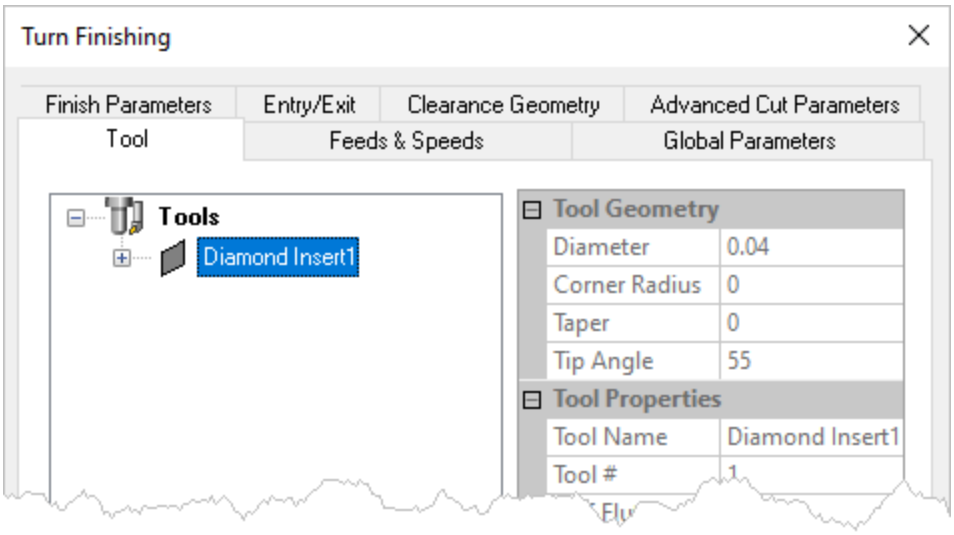
3. This will display the Turn Finishing operation dialog shown below.



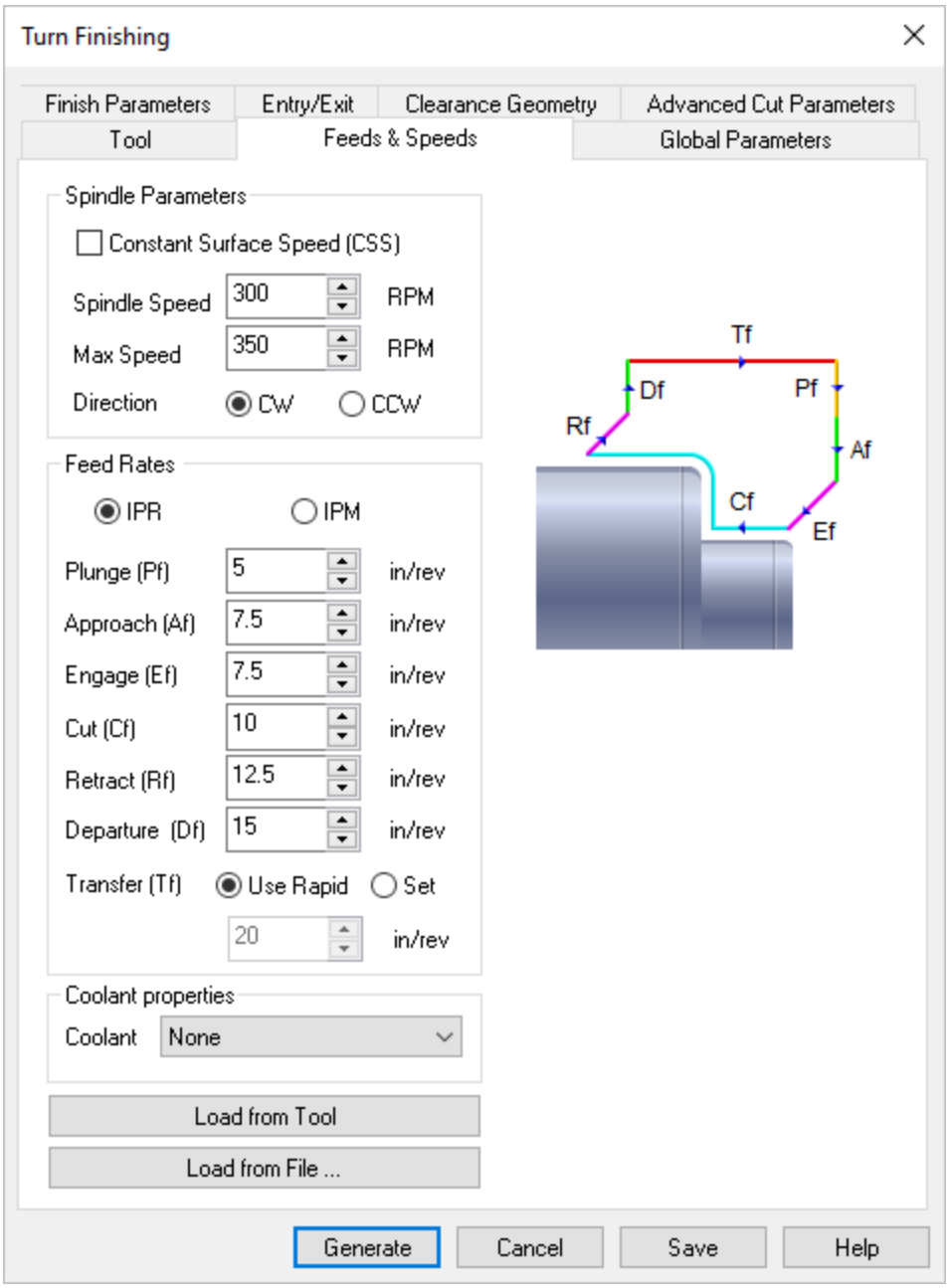
8.1 Set Finishing Parameters

Here we will use the Turn Finishing dialog to set our finishing parameters:

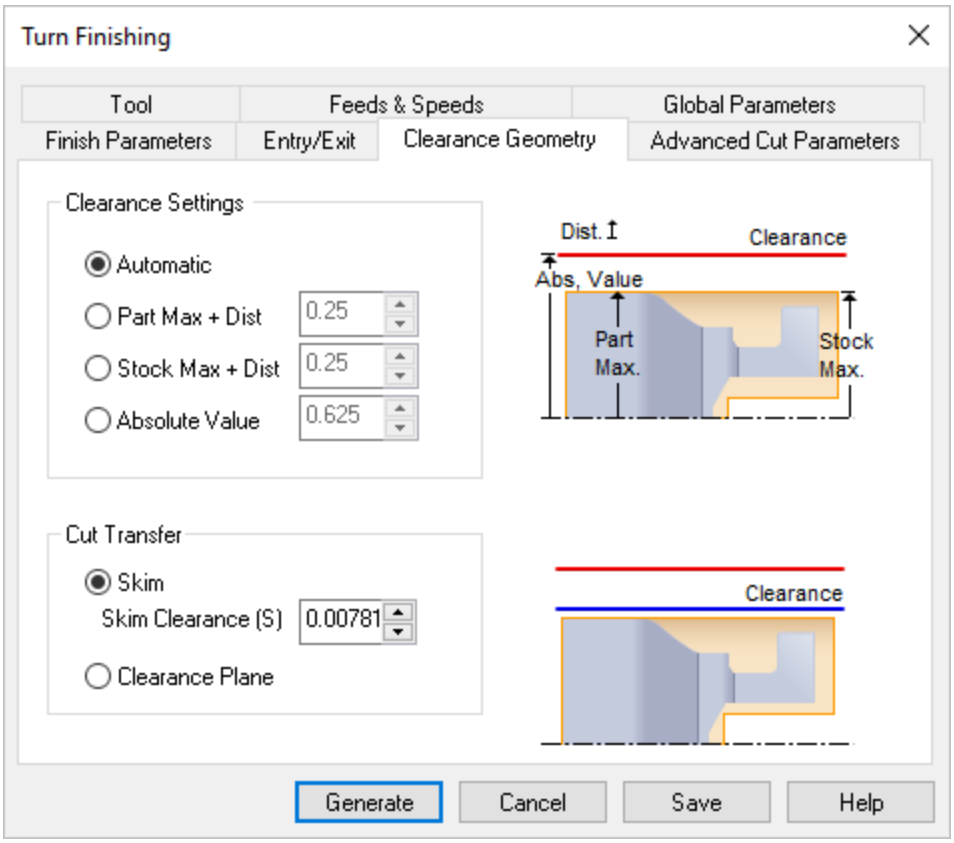
1. From the Tool tab in the Turn Finishing dialog select the Diamond Insert-OD tool.



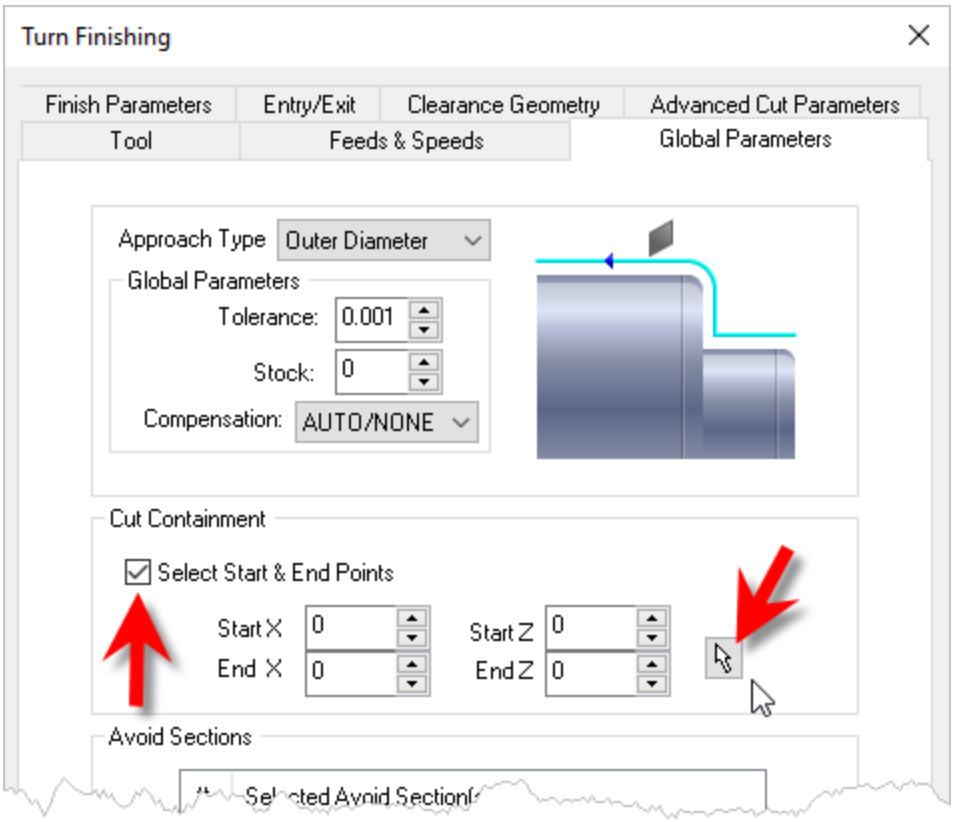
2. Now, pick the Feeds & Speeds tab and select the Load from Tool button. The system will retrieve the feeds and speeds parameters that was set when the tool was defined and associate them with the current operation.



3. Pick the Clearance Geometry tab and set the Clearance Settings to Automatic and Cut Transfer to Clearance Plane.

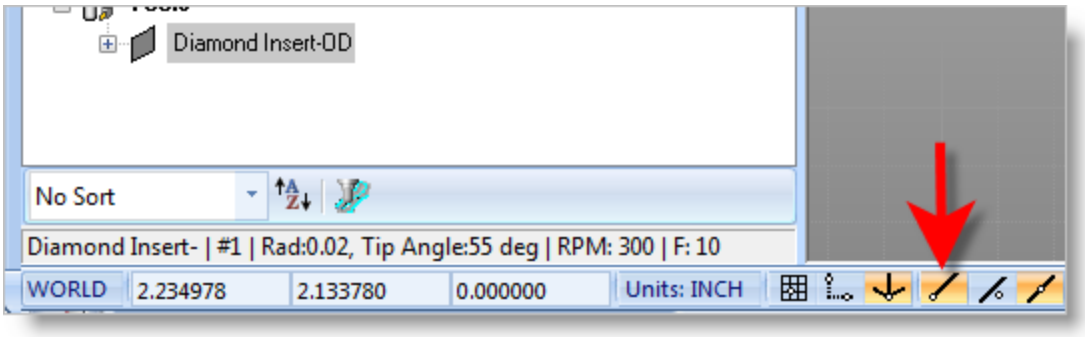


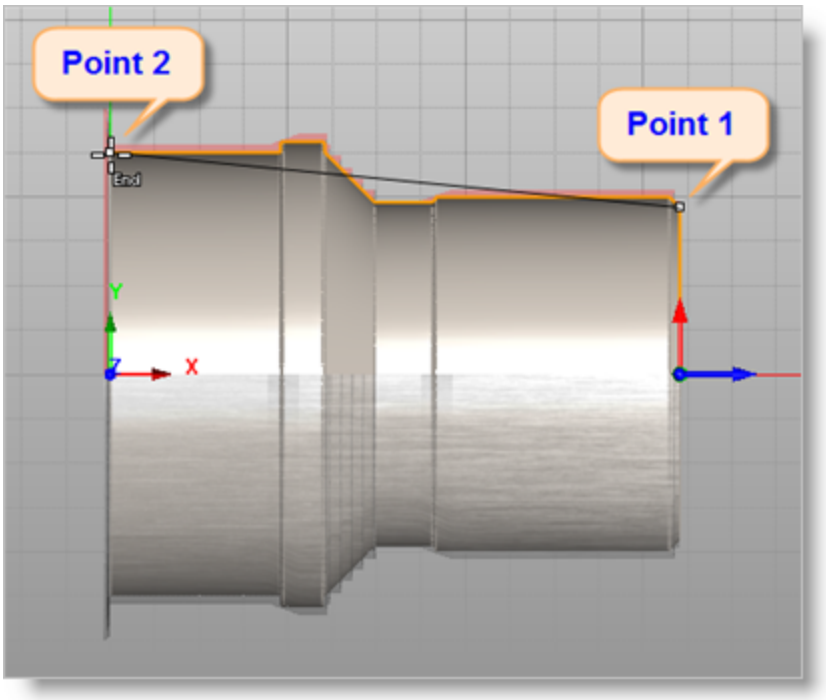
4. Now we'll switch to the Global Parameters tab to specify parameters to control the cutting.
5. Set the Approach Type to Outer Diameter and Stock = 0.
   We will not be leaving any thickness on the part after machining, effectively removing all stock left over from the previous roughing operation.
6. Now under Cut Containment, check the box for Select Start & End Points.
   This allows you to specify an area to contain the toolpath by selecting cut start and end points. This is useful in cases where only a section of the part needs to be machined.
7. In this guide, we will graphically select the start and end points from the part to specify cut containment. Click on the Pick button. This minimizes the dialog and prompts you to select start and end points.



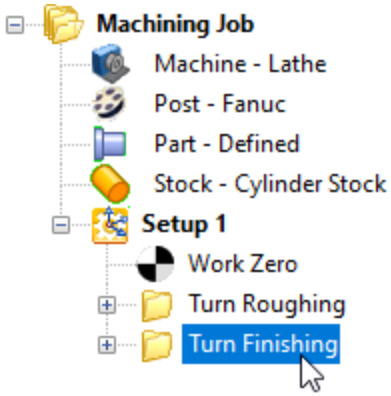
8. From the View toolbar, we'll switch to Top View.

9. Now, with the End Snap toggle on, we'll select 2 points as shown below.





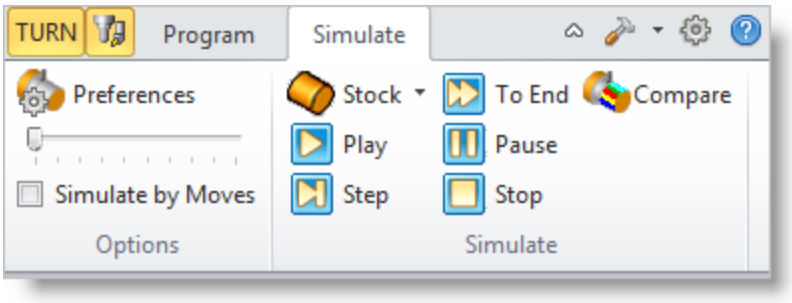
10. The Turn Finishing dialog reappears and displays Start and End point coordinate values for the cut containment.
11. Leaving all other parameters with default settings, we pick Generate. The operation is generated and added to the Machining Job in the Machining Browser below the Turn Roughing operation we previously created.



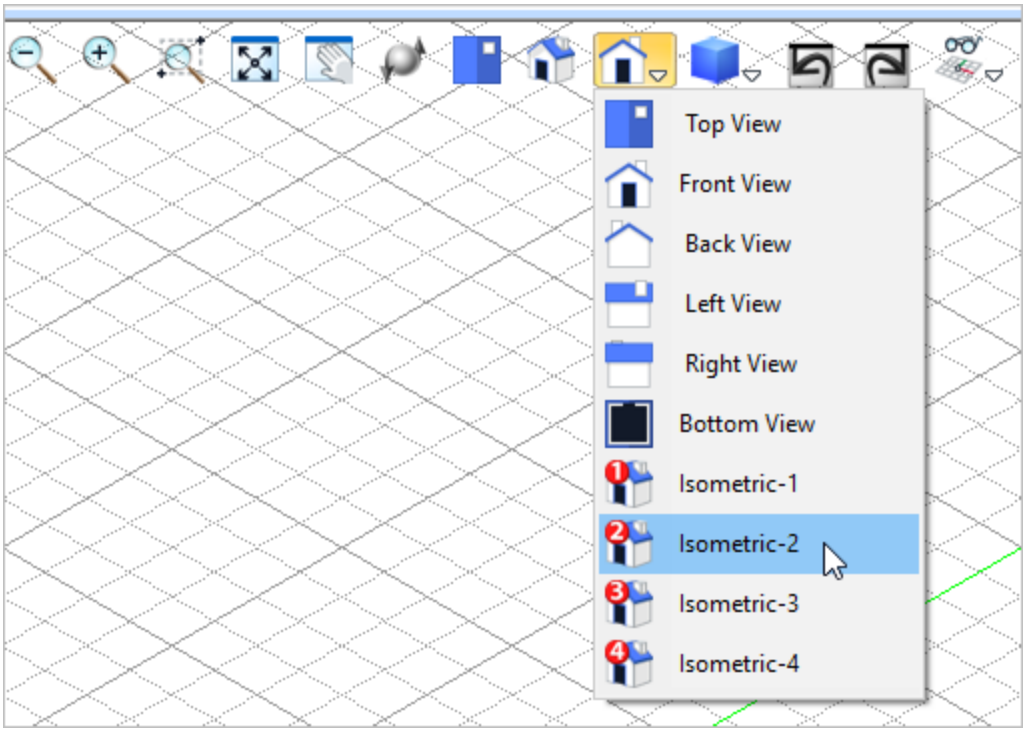
8.2 Simulate the Toolpath

We're now ready to Simulate the operation.

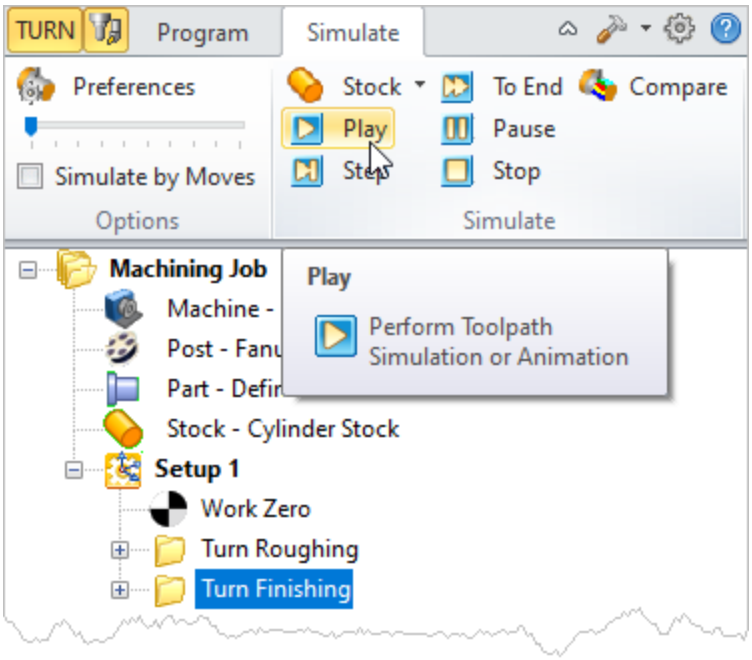
1. First make sure the Turn Finishing toolpath is selected from the Machining Job tree.
2. Select the Simulate tab.

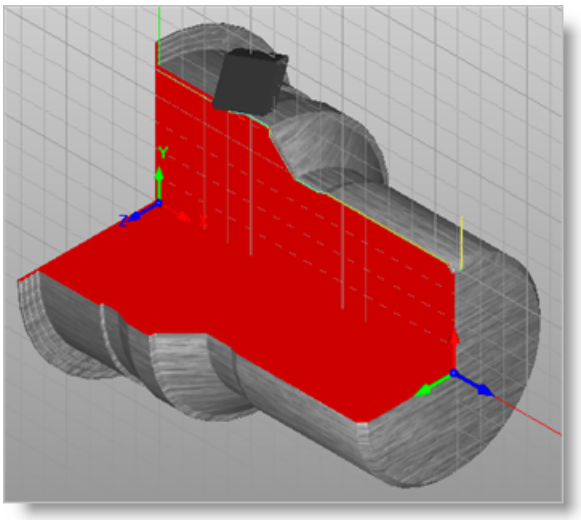


3. From the View toolbar, select Isometric 2.



4. Now pick Play to view the simulation.

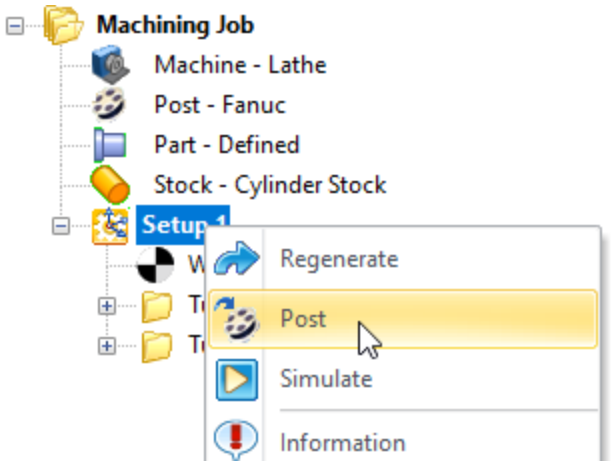




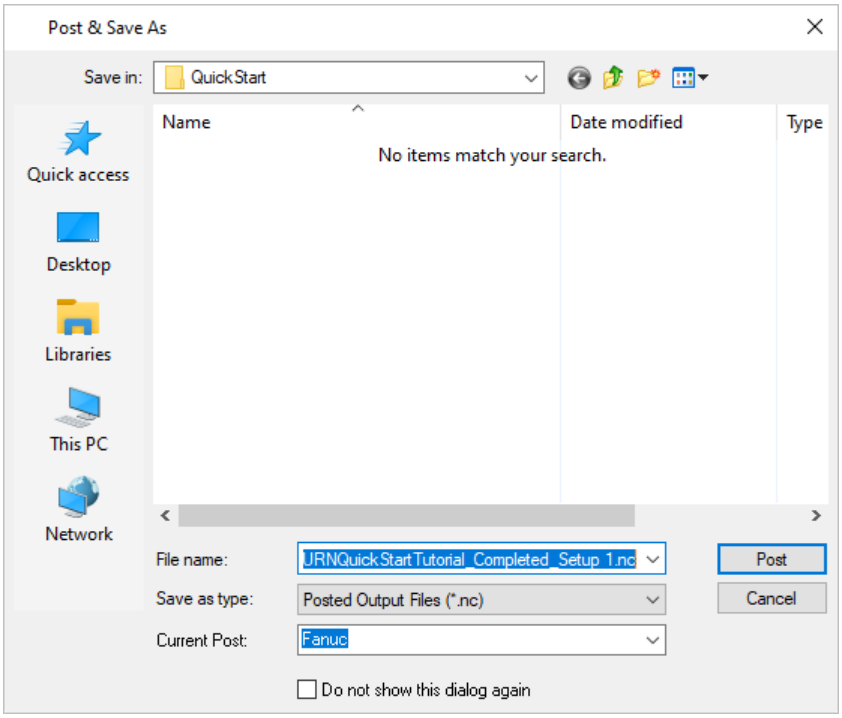
8.3 Post-Process the Toolpath

With the toolpaths complete we're ready to post-process to an output text file containing G-codes that can then be sent to the machine tool to actually machine the part.

1. Select Setup 1 from the Machining Browser, right-click and select Post.

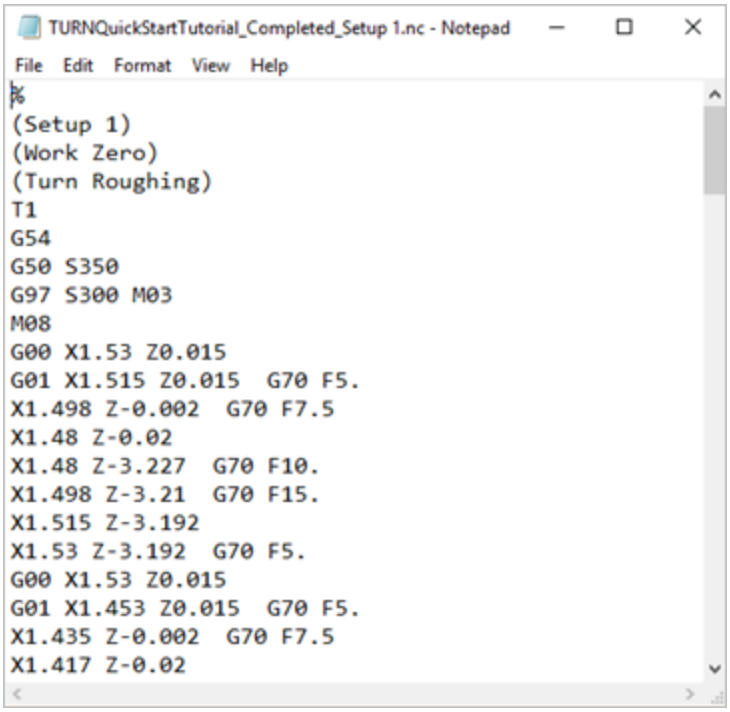


2. This will post-process all operations created under Setup 1.
3. By default, the Part file name and the Setup name are appended for the G-code file name. Also by default, the posted G-code file is saved to the folder where the part file is located.



The post by default is set to Fanuc as we specified under the Select the Post Processor section of this guide. You can change the post processor here by selecting a different one from the Current Post list. The posted g code by default will be saved to the folder where the part file is located.

4. Now pick Post and the G-code file is displayed in Notepad where it can be viewed or edited manually.



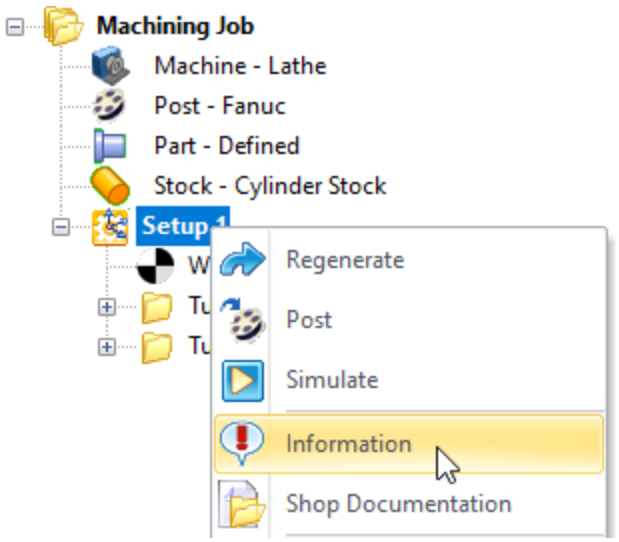
5. Now close Notepad.

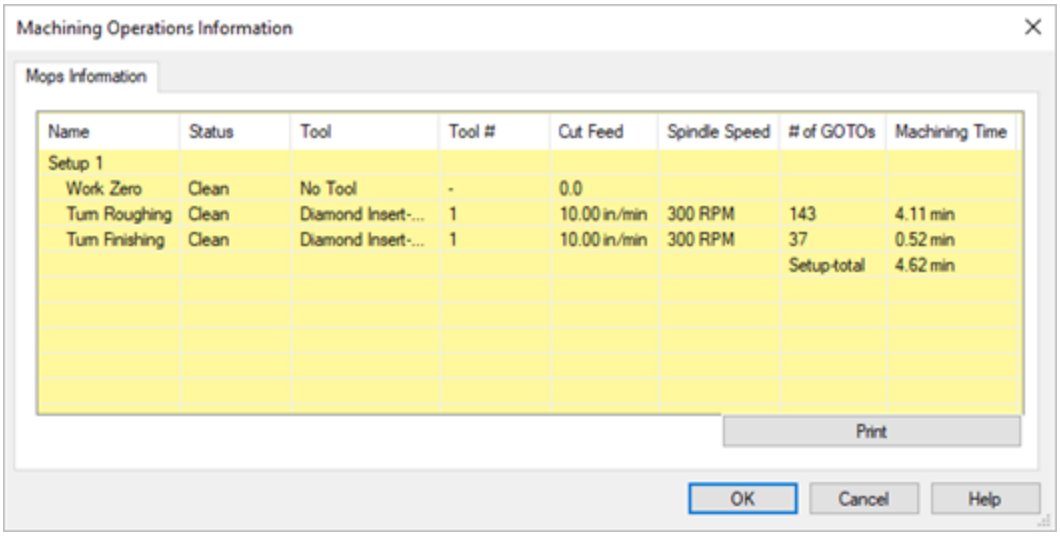
Generate Reports

9.1 Machining Information

At any time, you can create an Information Report of your Machining Operations.

1. From the Program tab, select Setup 1 under the Machining Job.
2. Right-click and select Information to display and Print the report.





This dialog provides an estimate of the machining time required for the operations in the Setup.

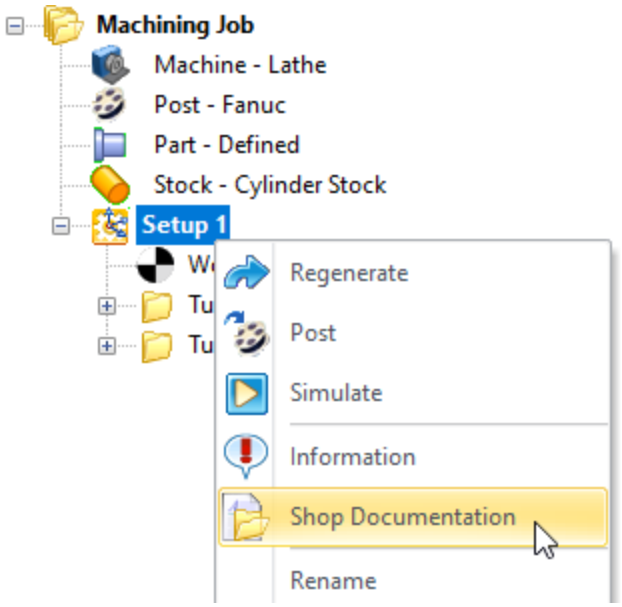
You can perform the same right-click sequence on the Machining Job to determine the estimated machining time for all Setups.

3. Now pick OK to close the Information dialog.

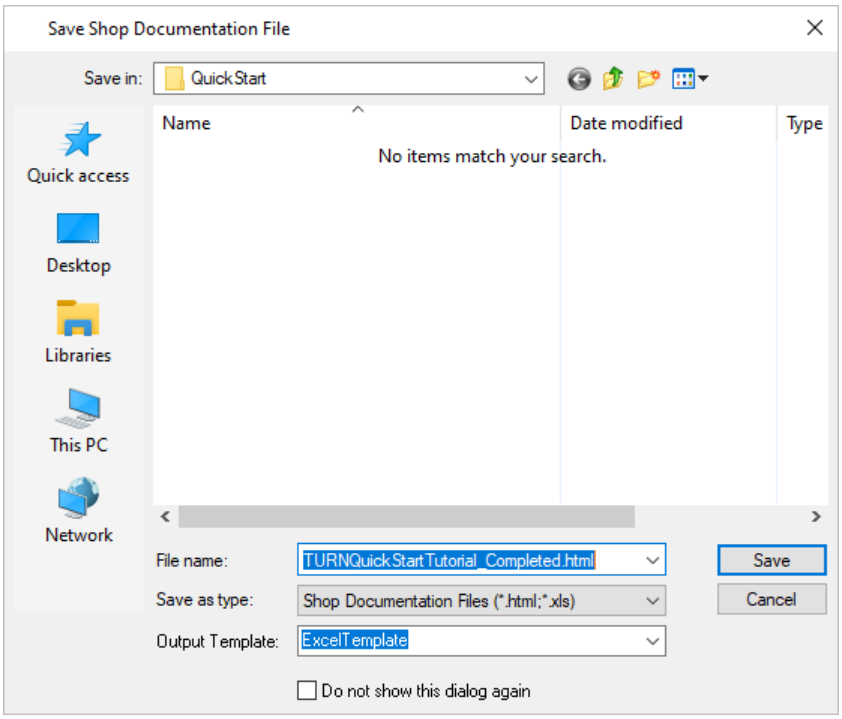
9.2 Shop Documentation

You can also create a Setup Sheet by generating a Shop Document. This is typically used to instruct machine operators on how to setup and machine the part on the CNC machine.

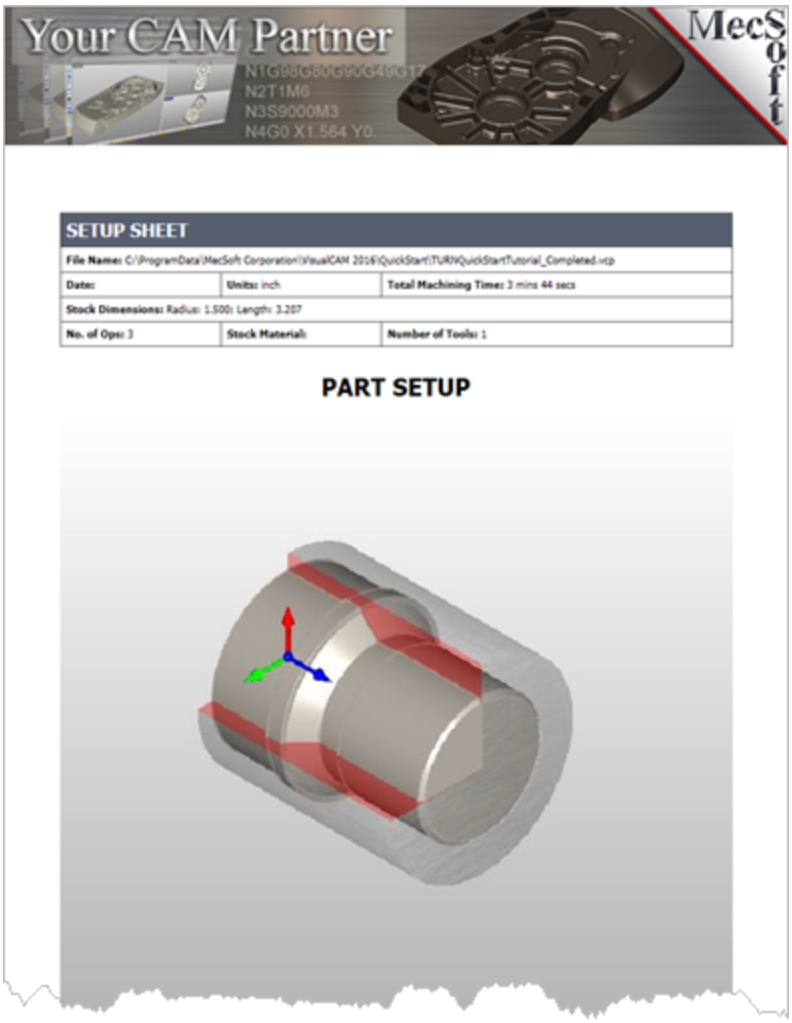
1. Select Setup1 under the Machining Job tree in the Machining Browser.
2. Right-click and select Shop Documentation.



3. Select Template1. You can also select from one of the additional HTML templates that are shipped with the product and generate shop documentation.



4. This creates an HTML based Shop Document. This file can then be printed and/or viewed in a web browser such as Internet Explorer.

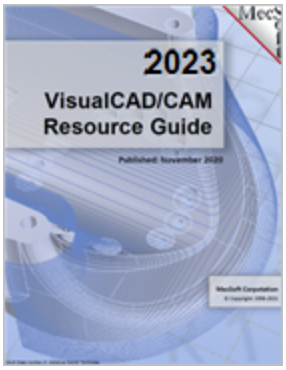


5. You can perform the same right-click sequence on the Machining Job to generate Shop Documentation for all Setups.

Where to go for more help

Download this PDF Guide for a list of the available VisualCAD/CAM Resources.

2023 VisualCAD/CAM Resource Guide



The 2023 VisualCAD/CAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Click Here!

Index

- A -

- C -

- D -

- E -

- F -

- L -

- M -

- P -

- R -

- S -

- T -

- U -

- V -

- W -