

VisualCAM 2018 TURN Quick Start

© 2017 MecSoft Corporation

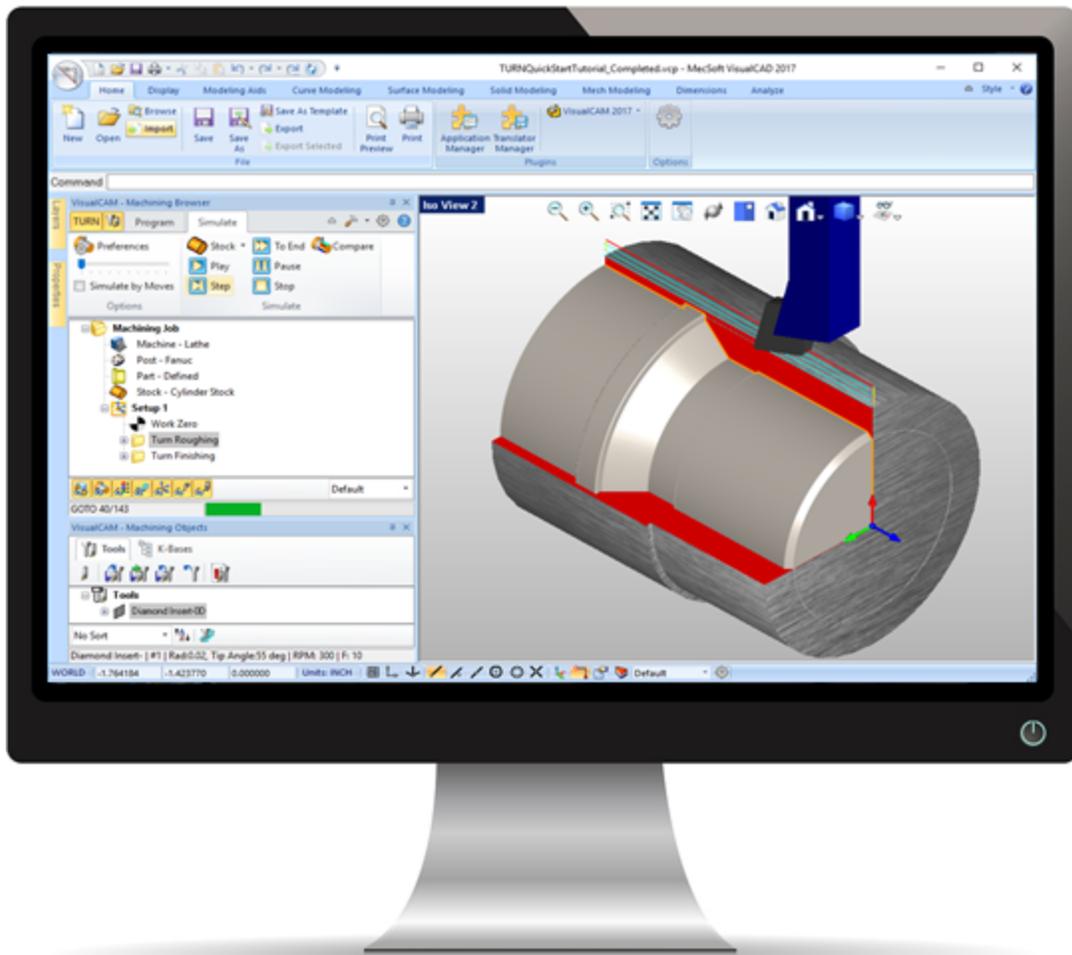


Table of Contents

About this Guide	4
1 About the TURN Module	4
2 Using this Guide	4
3 Useful Tips.....	5
Getting Ready	7
1 Running VisualCAM 2018.....	7
2 About the VisualCAD Display.....	7
3 Launch the TURN Module.....	8
4 Load the Part Model.....	12
5 Machining Strategy.....	14
6 Main Programming Steps.....	15
7 Define the Machine Tool.....	15
8 Select the Post Processor.....	16
Defining the Machine Setup	20
1 Define the Part Geometry.....	20
2 Create the Stock Geometry.....	23
3 Align the Part and Stock.....	27
4 Specify the Material.....	29
5 The Machining Setup.....	32
6 Set the Work Zero.....	33
Create a Cutting Tool	37
Roughing the Outer Diameter	41
1 Select the Cutting Tool.....	42
2 Set Feeds and Speeds.....	44
3 Set Clearance Geometry.....	45
4 Set Global Parameters.....	47
5 Set Roughing Parameters.....	48
6 Set Entry/Exit Parameters.....	49
7 Simulate the Toolpath.....	53
Finishing Outer Diameter	59
1 Set Finishing Parameters.....	59

2 Simulate the Toolpath.....	65
3 Post-Process the Toolpath.....	67
Generate Reports	70
1 Machining Information.....	70
2 Shop Documentation.....	71
Where to go for more help	75
Index	76

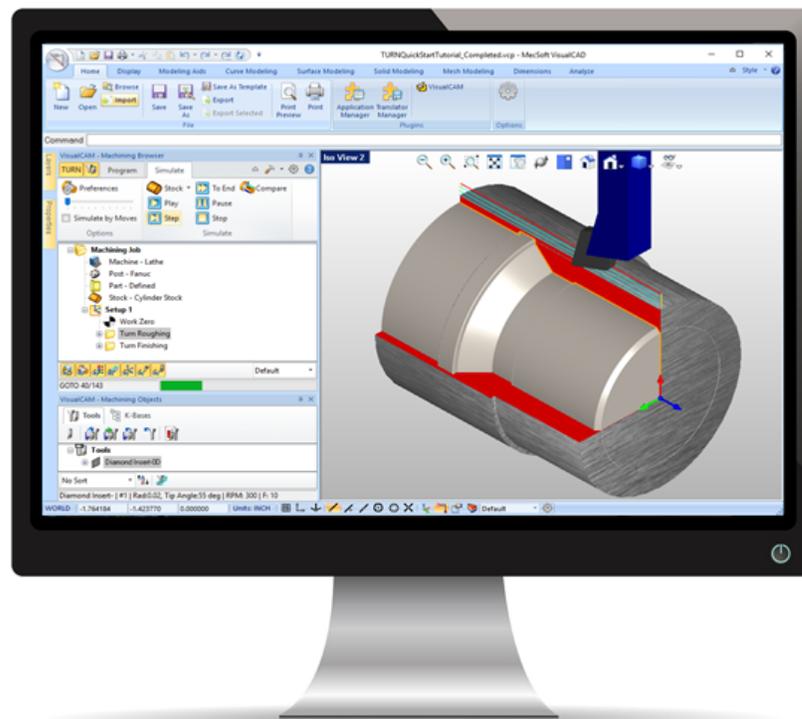
About this Guide

1.1 About the TURN Module

The **VisualCAM 2018 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 **VisualCAM 2018 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

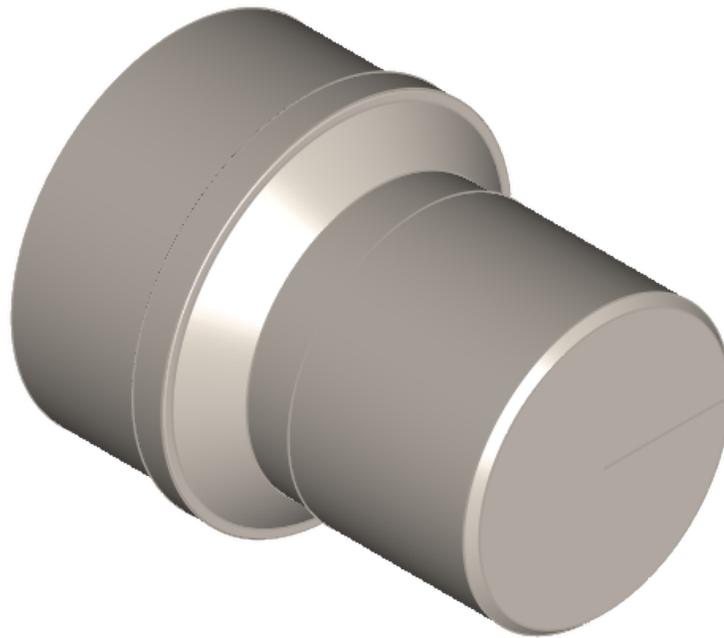
1.2 Using this Guide

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 [VisualCAM 2018 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

1.3 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!

Getting Ready

2.1 Running VisualCAM 2018

Locate the [VisualCADCAM 2018](#) 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 [VisualCADCAM 2018](#). (The name of this program group will usually be called [VisualCADCAM 2018](#), unless you specified otherwise during setup.)

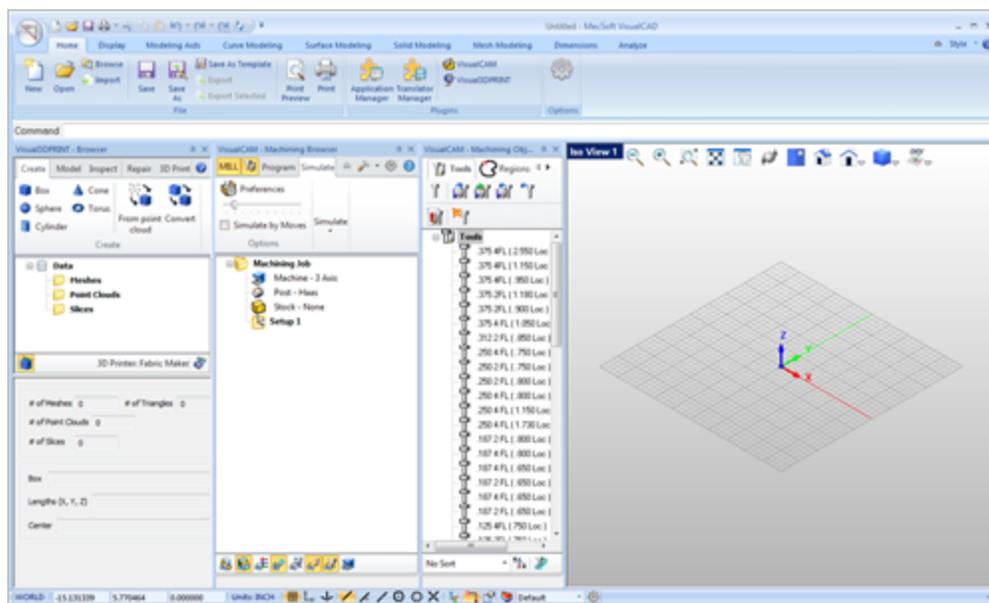
Once you locate the program group, select it and then select [VisualCADCAM 2018](#) to launch the application.

If the installation was successful, upon launching of [VisualCADCAM 2018](#) you should observe a menu entry called [VisualCAM 2018](#) on the [Home Ribbon Bar](#) menu of [VisualCAD](#).

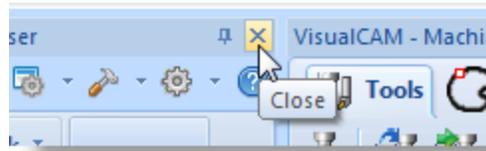
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

2.2 About the VisualCAD Display

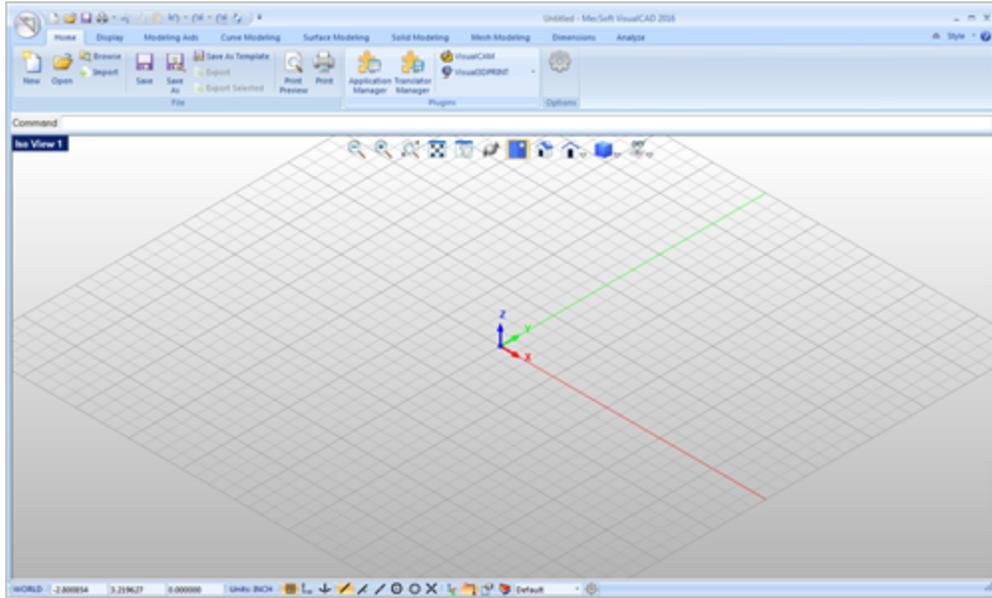
Before we begin, let's talk a bit about the [VisualCAD](#) display. When you run [VisualCAD](#) for the very first time, your screen may look this.



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



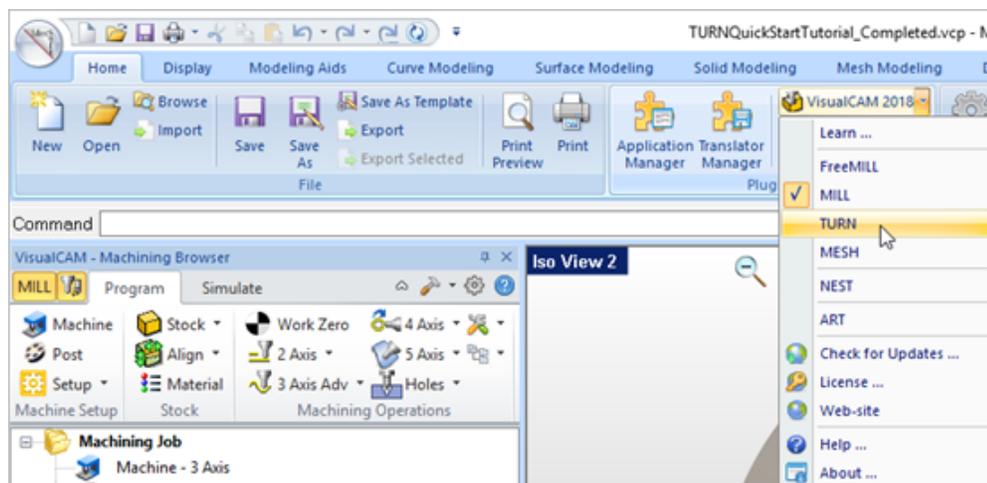
With all plug-in modules closed your screen will look like this:



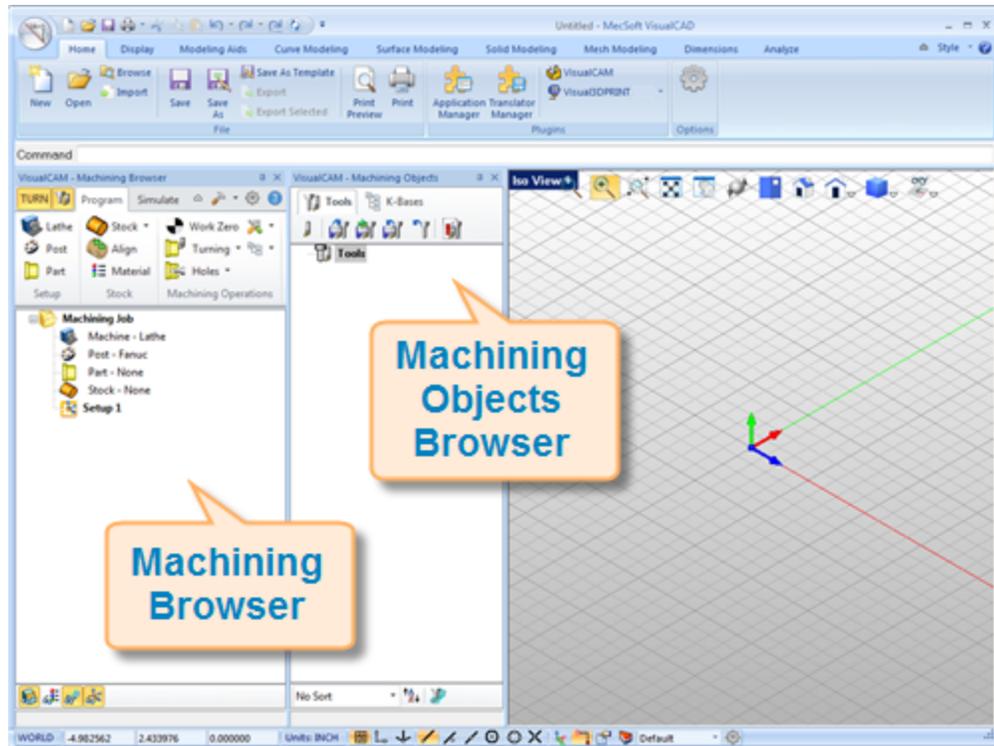
2.3 Launch the TURN Module

Now, let's begin by launching the VisualCAM 2018 TURN module.

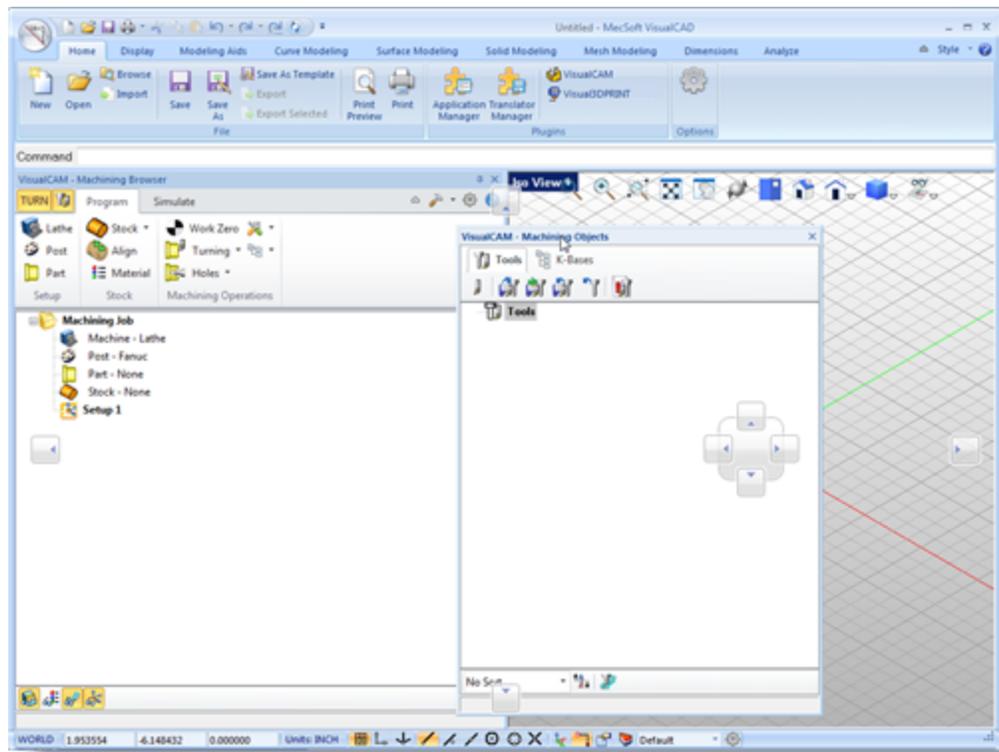
1. From the **Plugins** pane of VisualCAD's Home Ribbon Bar, you will see the VisualCAM 2018 main menu item.
2. Drop-down the menu and pick **TURN** to load the TURN module.



3. Docked on the left you will see the **Machining Browser** and the **Machining Objects Browser**. When you first run **VisualCAM 2018**, these two browsers may be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



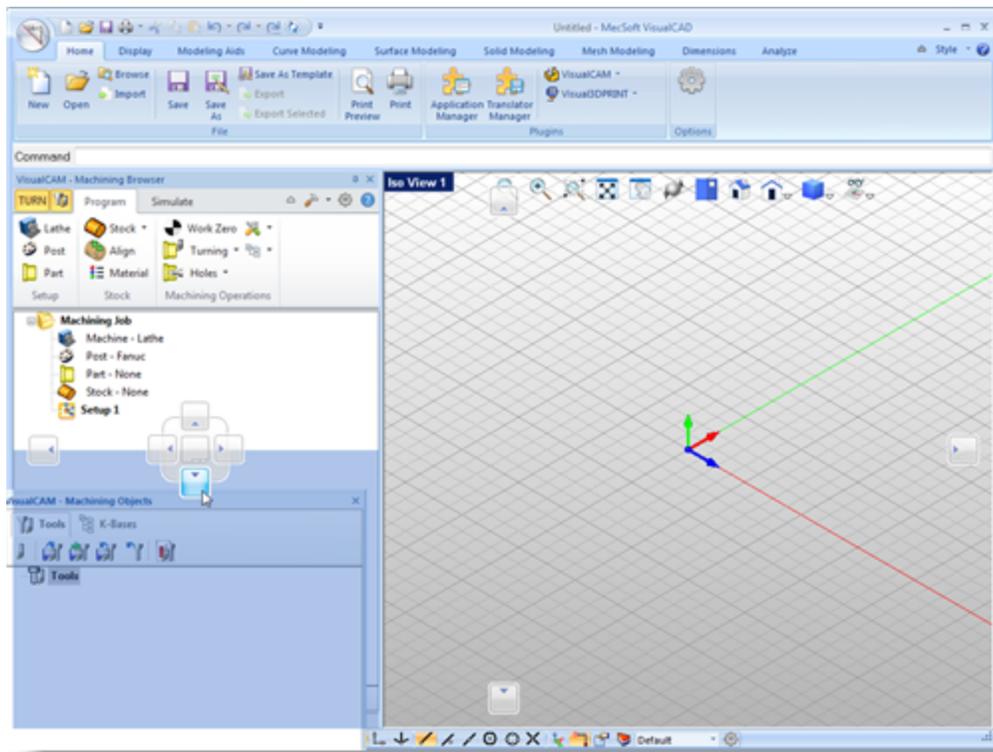
4. For example, let's move the **Machining Objects Browser** so that it displays under the **Machining Browser** on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



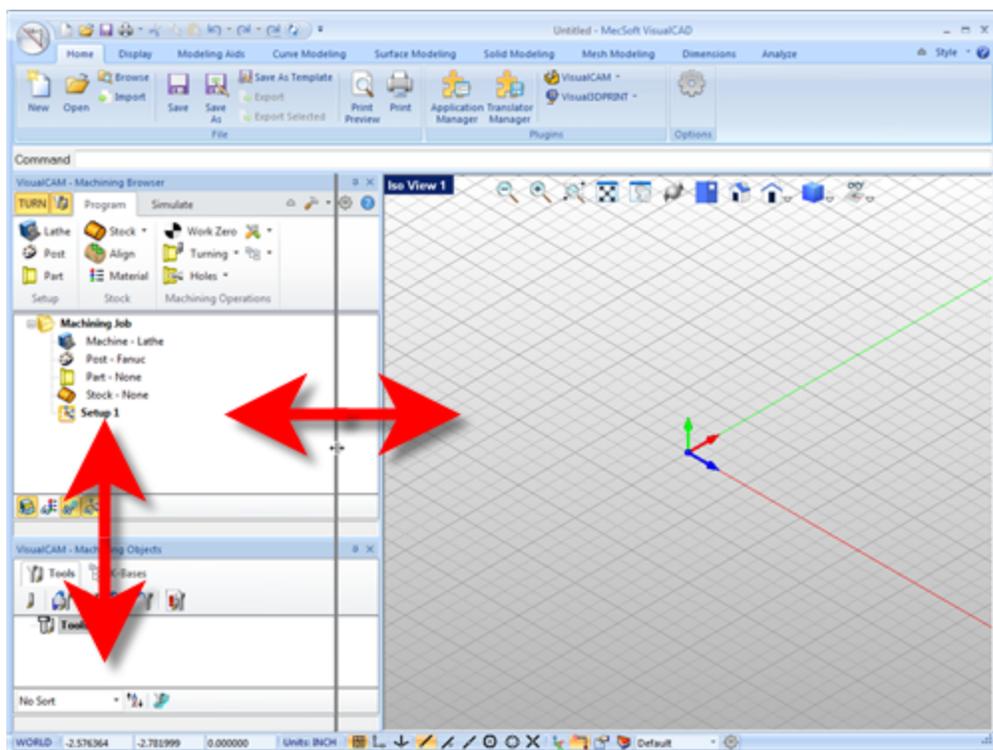
While doing so, you will see the docking widget display in the background with directional buttons allowing you to choose screen locations relative to the active window.

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.



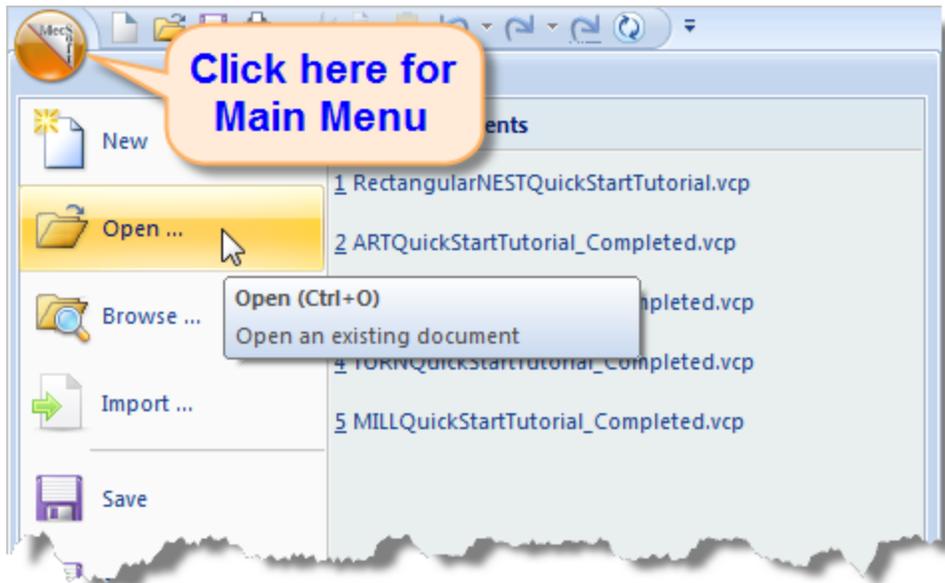
6. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



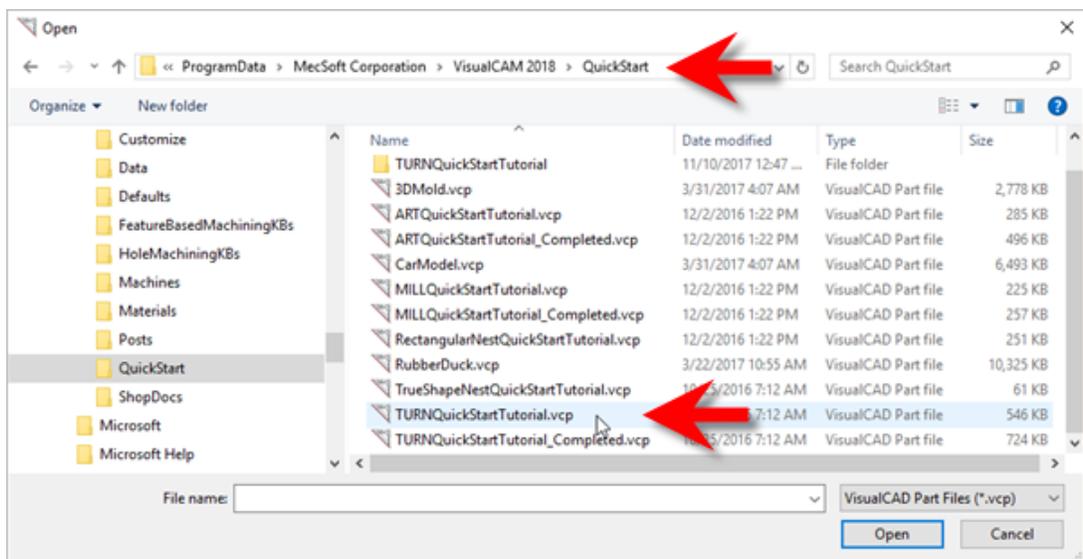
2.4 Load the Part Model

“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.

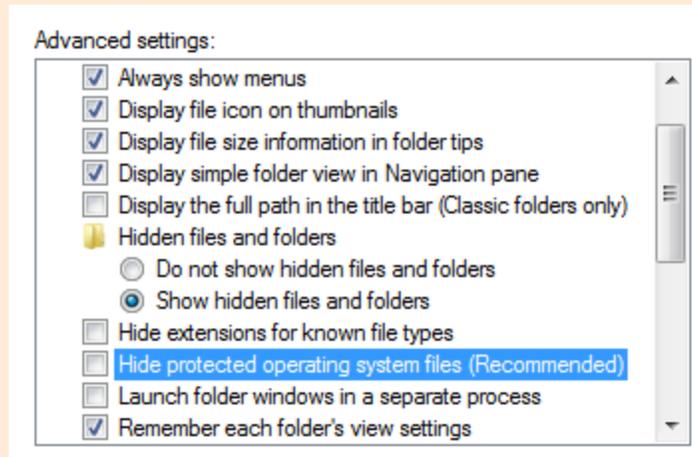


2. From the Open dialog box, select the TURNQuickStartTutorial.vcp file from the C:\ProgramData\MecSoft Corporation\VisualCAM 2018\QuickStart\ folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.



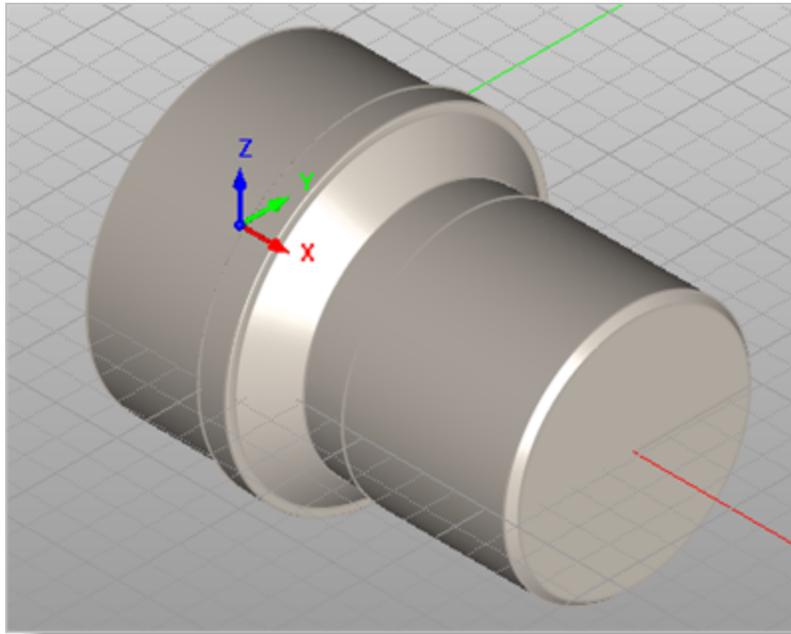
! By default, the `ProgramData` folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows 7/8 users: Go to **Control Panel > Appearance and Personalization > Folder 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**.

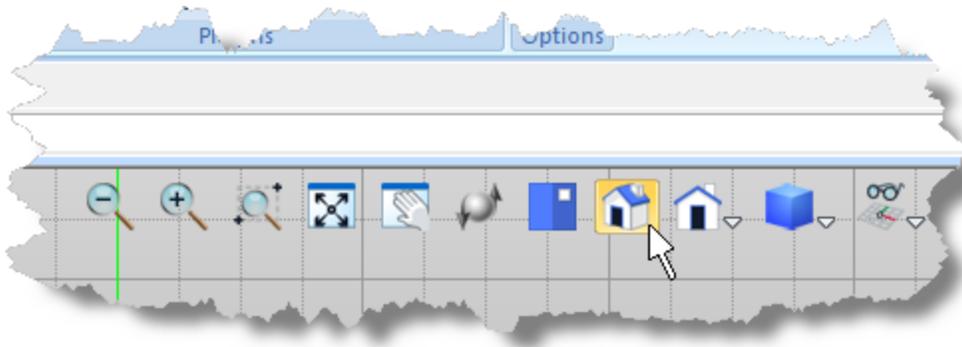
The part appears as shown below



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.



2.5 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 $3 \frac{1}{4}$ 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.

2.6 Main Programming Steps

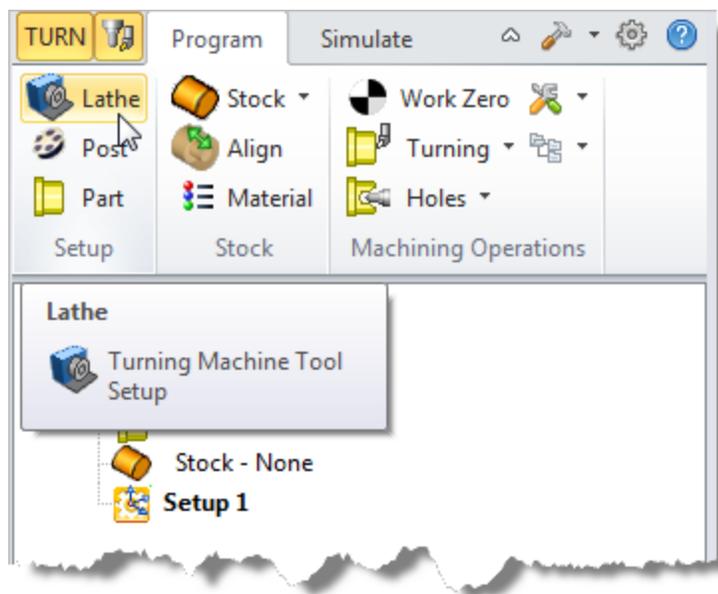
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

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.

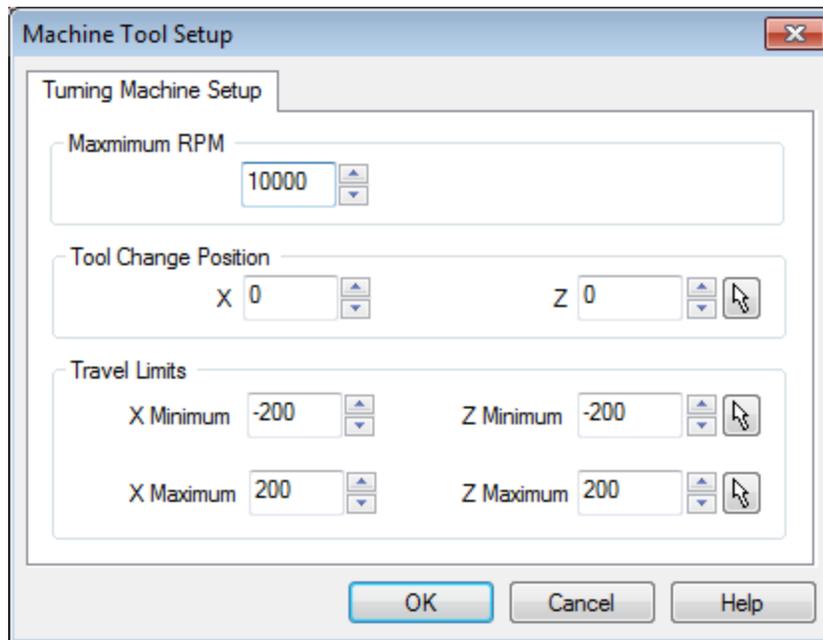
2.7 Define the Machine Tool

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

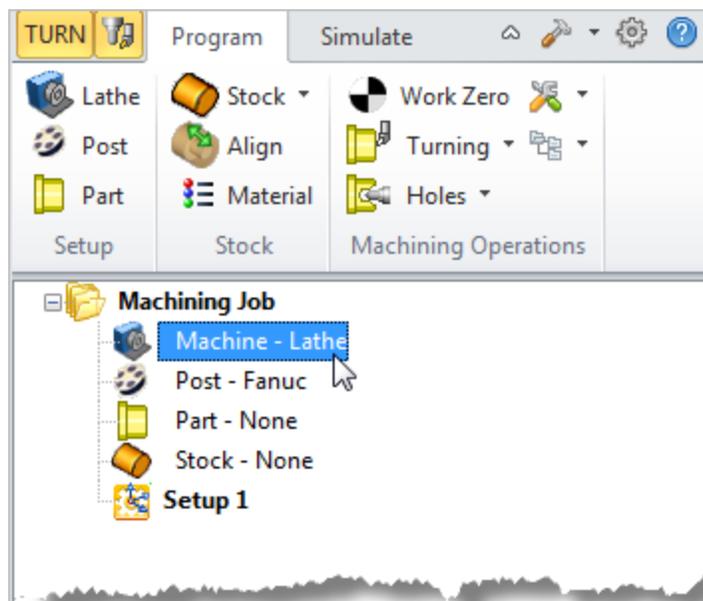
1. From the Program Tab select Lathe to display the dialog box.



2. Set Maximum RPM to 10000.



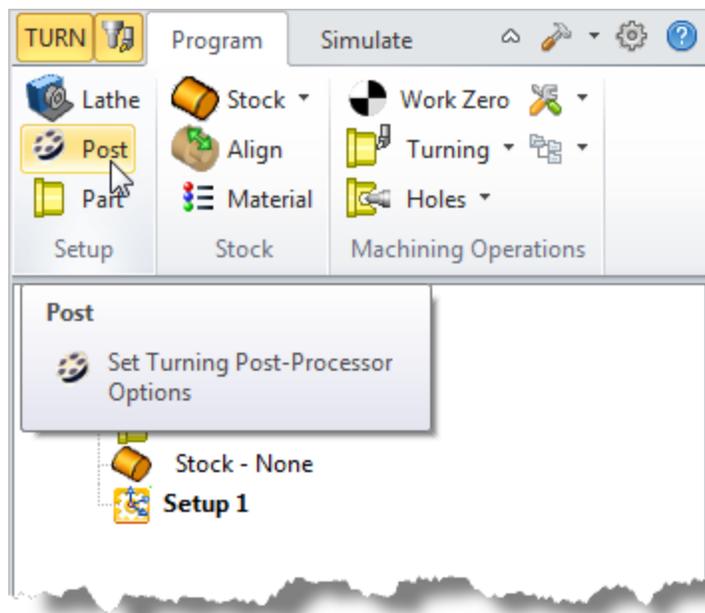
3. Pick OK and notice that the Machine - Lathe now appears under Machining Job in the Machining Browser.



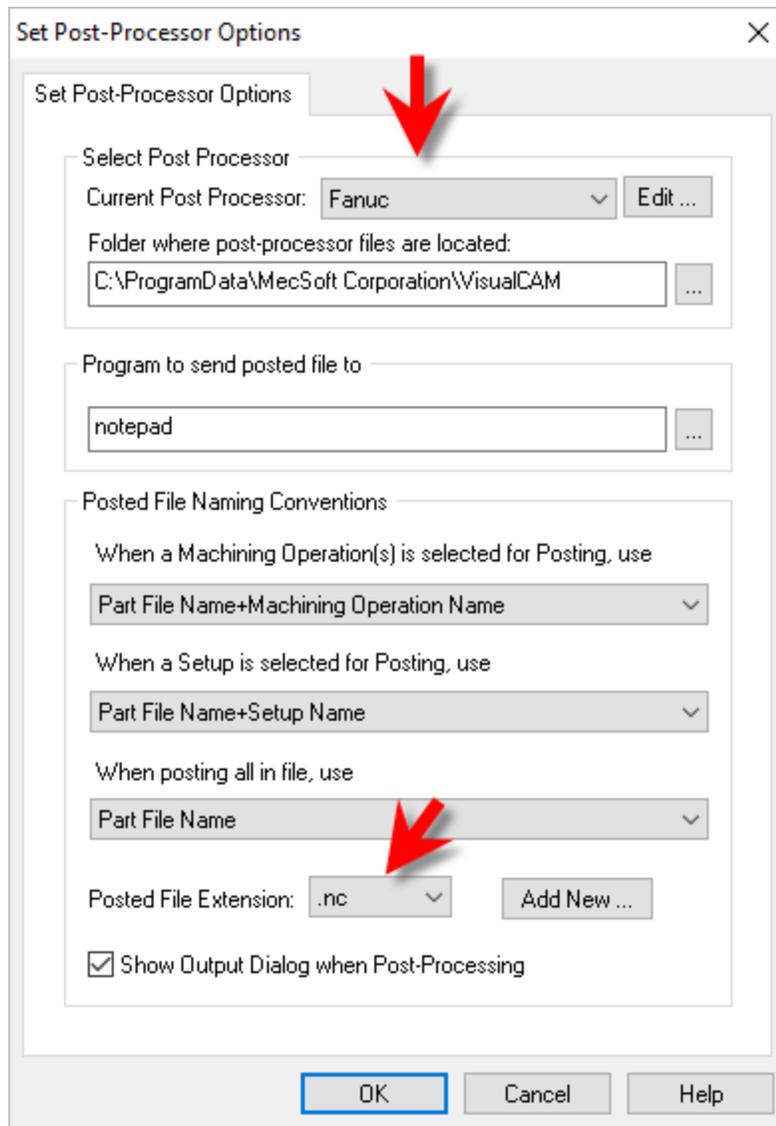
2.8 Select the Post Processor

Next, we'll define the Post Processor.

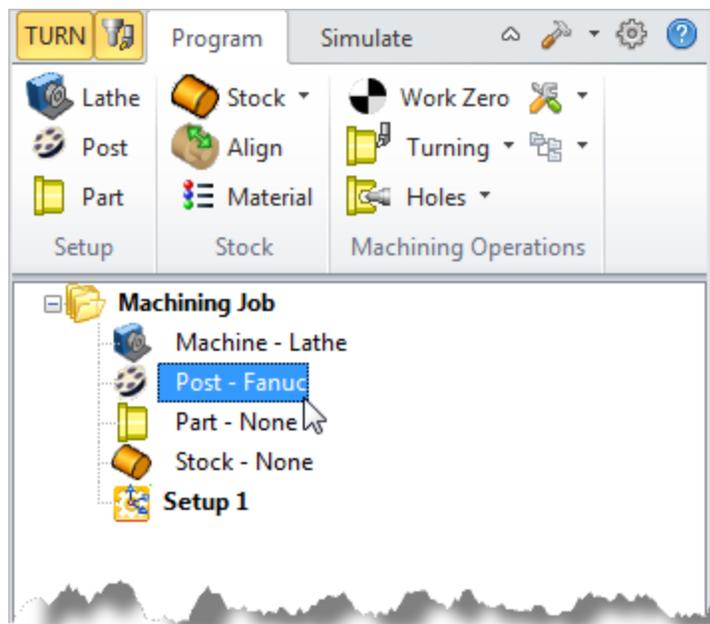
1. From the Program tab select Post to display the dialog.



2. For the **Current Post Processor**, select **Fanuc** 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**.



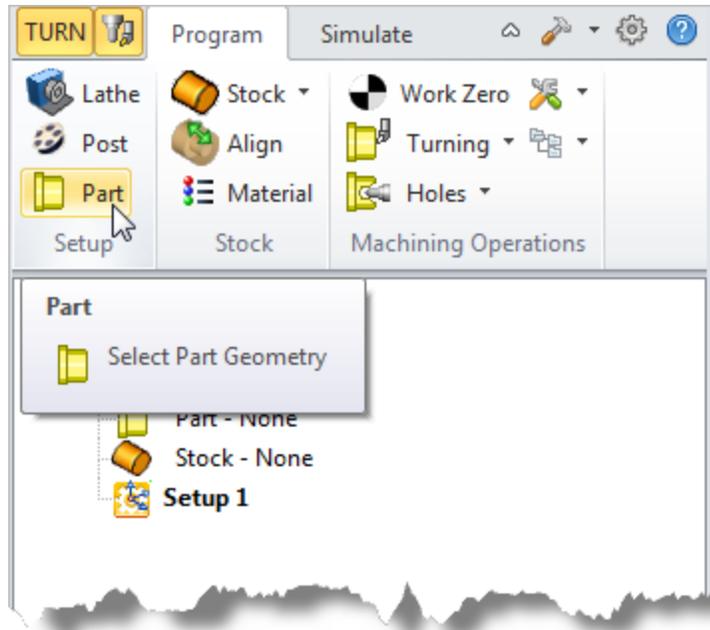
- ! By default, post processor files are located under:
C:\ProgramData\MecSoft Corporation\VisualCAM
2018\Posts\TURN\
The program to send the posted output is set to notepad.

Defining the Machine Setup

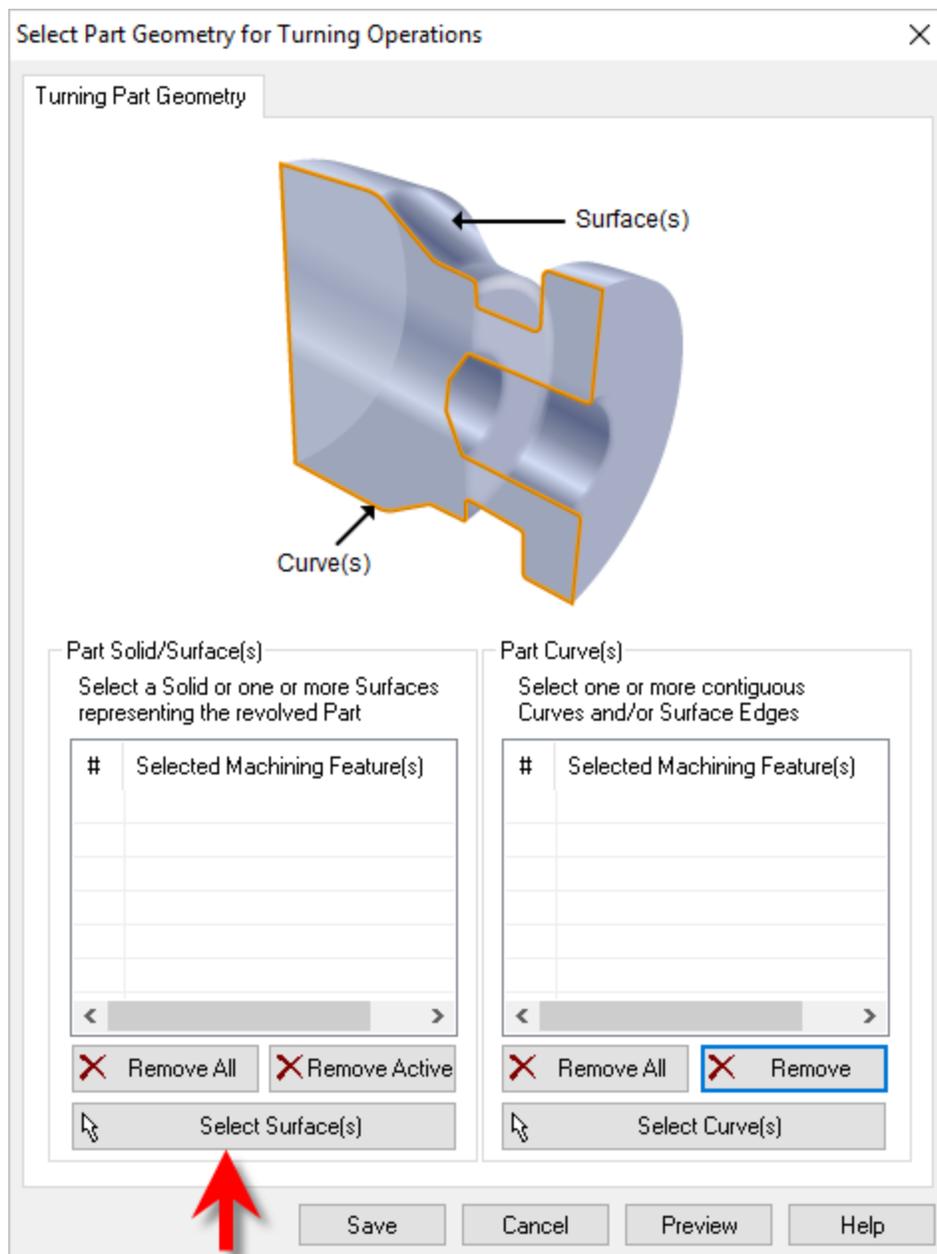
3.1 Define the Part Geometry

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.

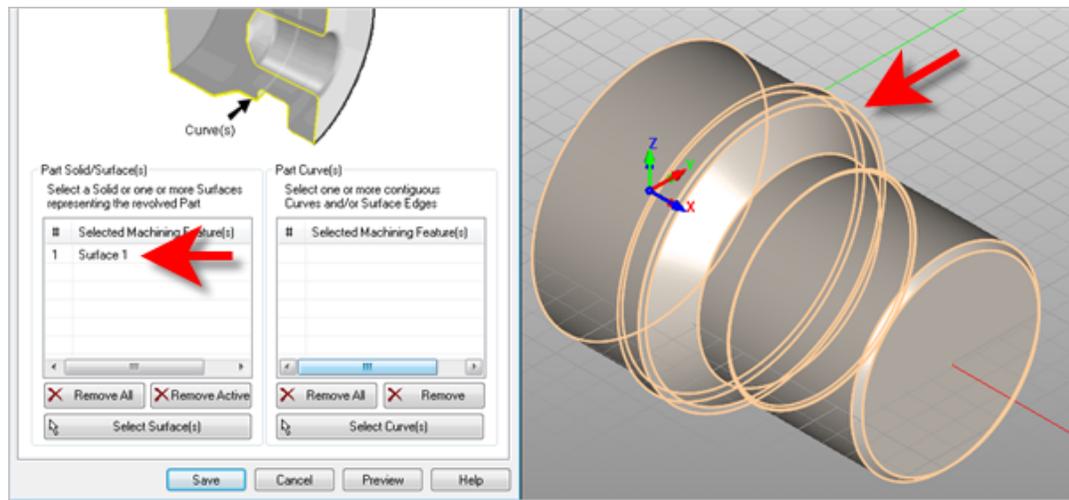
1. From the **Program** tab select **Part** to display the dialog.



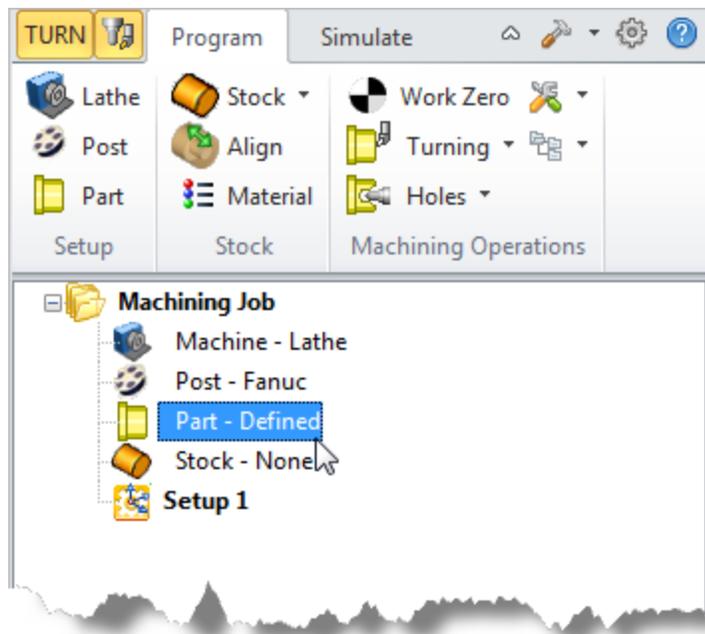
2. Pick the **Select Surfaces(s)** button.



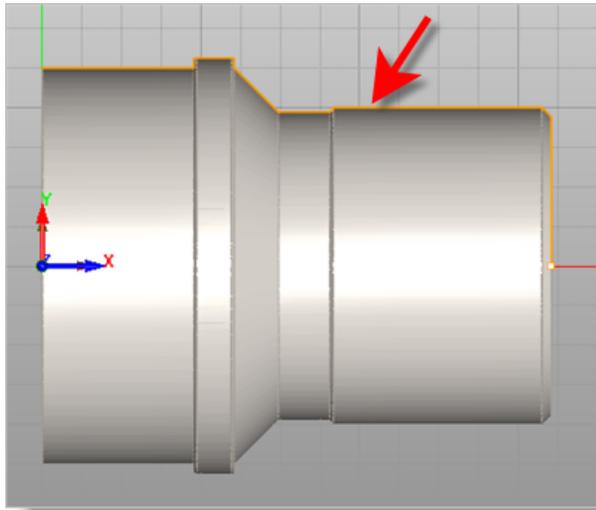
3. The dialog is minimized and allows us to select [Part Objects](#).
4. Window-select the entire 3D solid and then [right-click](#) or press [Enter](#) to accept the selection.
5. The dialog reappears and lists the selected surfaces under [Selected Machining Features](#).



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.



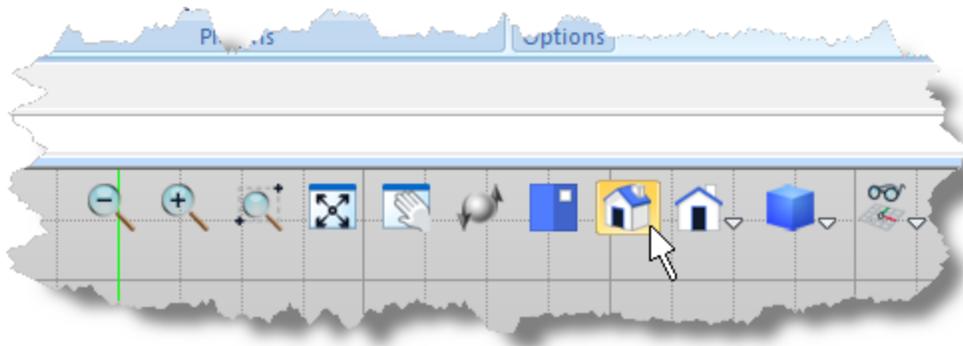
! In the future, if surfaces are selected as part geometry, the system will slice the selected surfaces with the ZX plane of the **Machine Coordinate System (MCS)** and use the resultant curves as the actual profile to be machined on the lathe.

Alternatively you can perform the slicing using CAD tools and select just the resultant curve as the 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**.

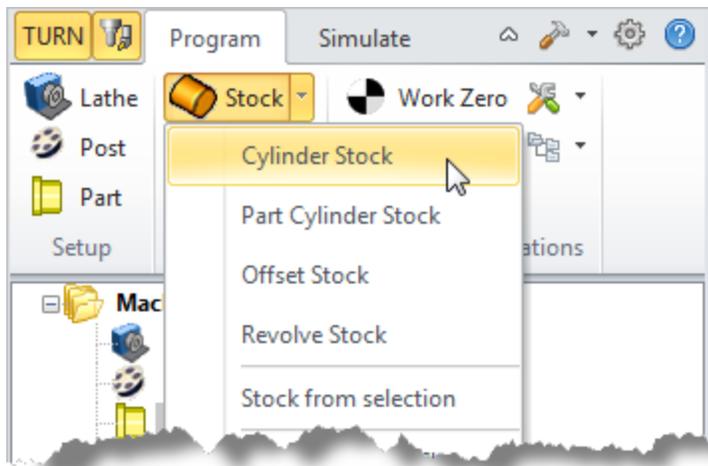
3.2 Create the Stock Geometry

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

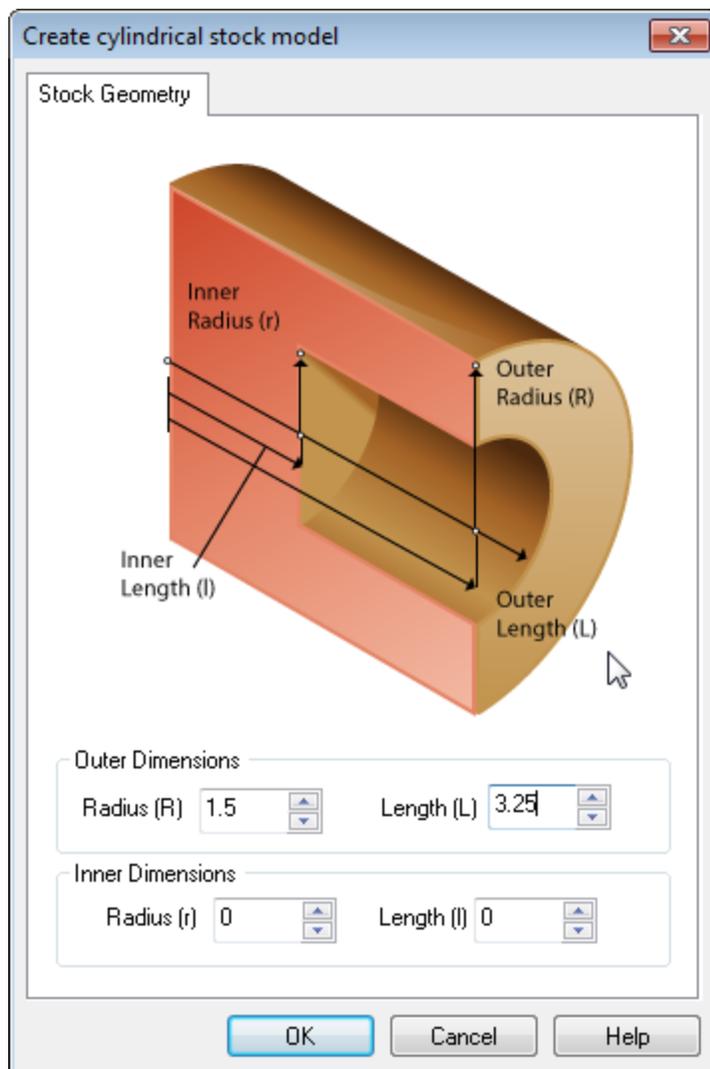
1. First, switch back to the **Isometric View**.



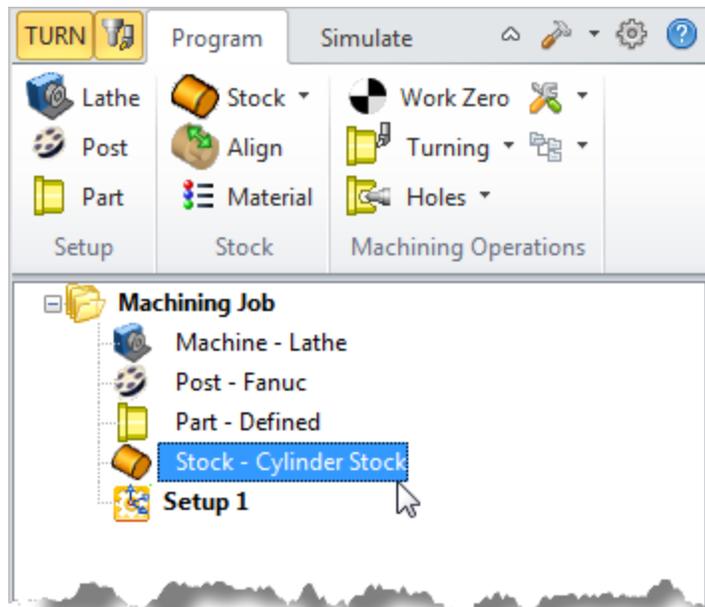
2. From the **Program** tab select **Stock** and then select **Cylinder Stock** from the menu to display the dialog.



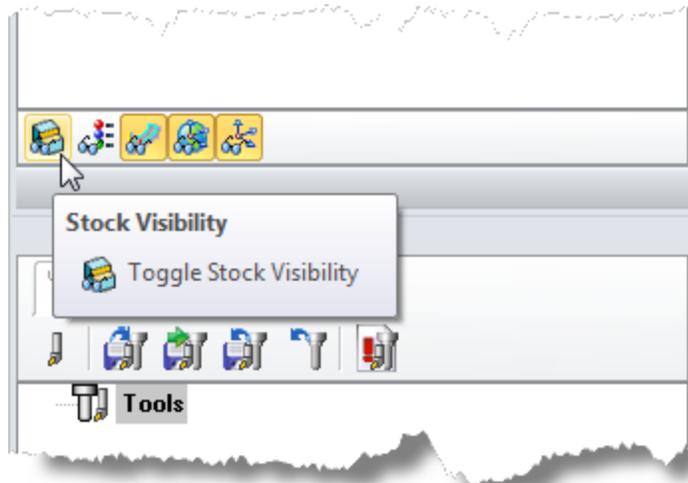
3. Under **Outer Dimensions**, set **Radius (R)** to 1.5 and **Length (L)** to 3.25.

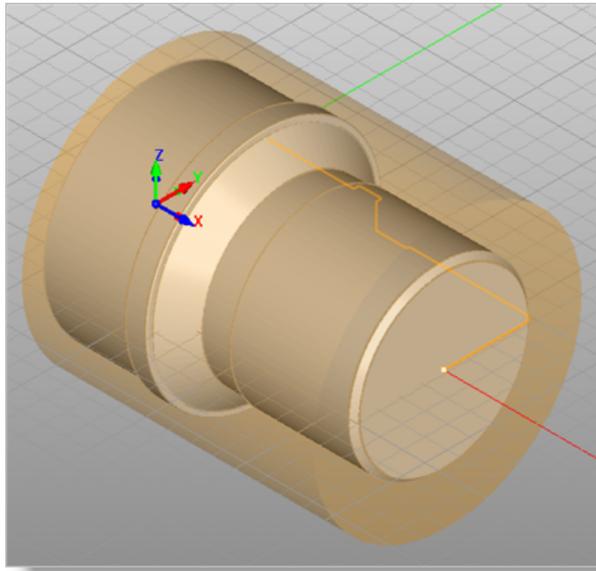


4. Pick **OK** and notice that the **Stock** type now appears under **Machining Job** in the **Machining Browser**.



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**.

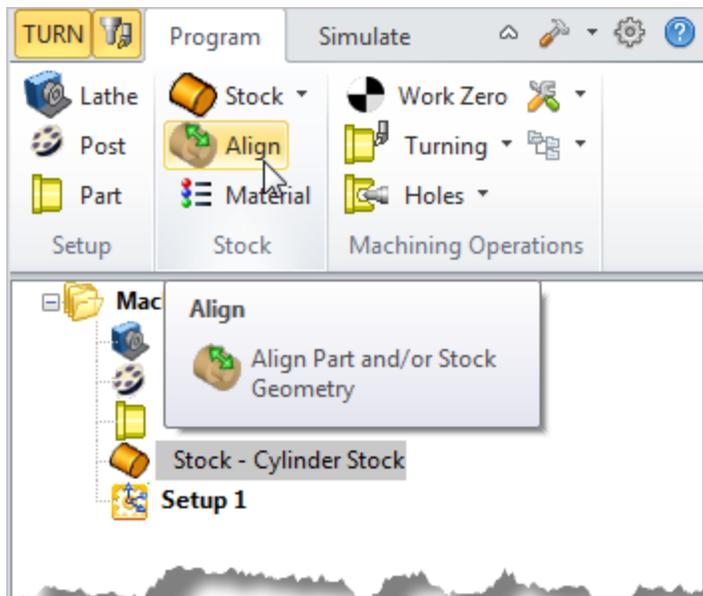




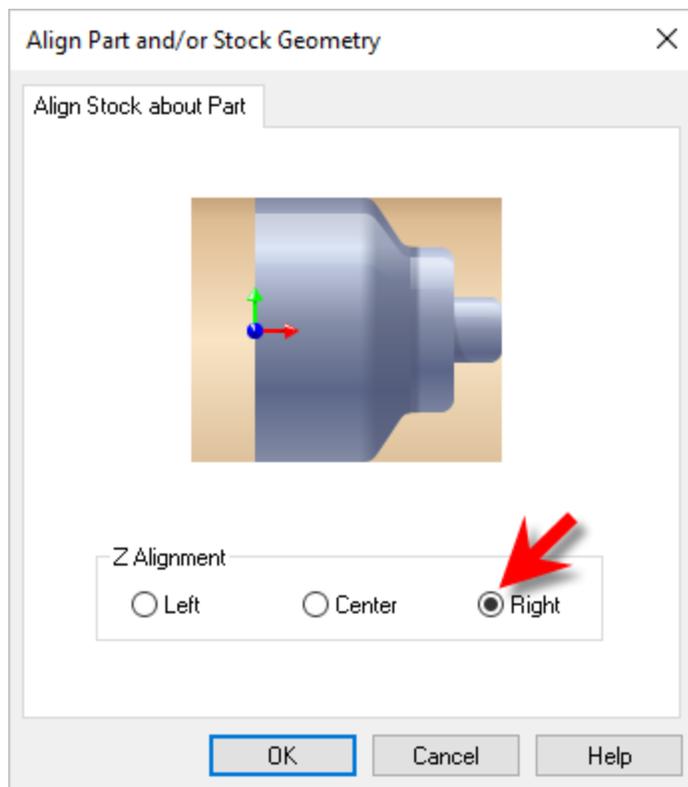
3.3 Align the Part and Stock

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

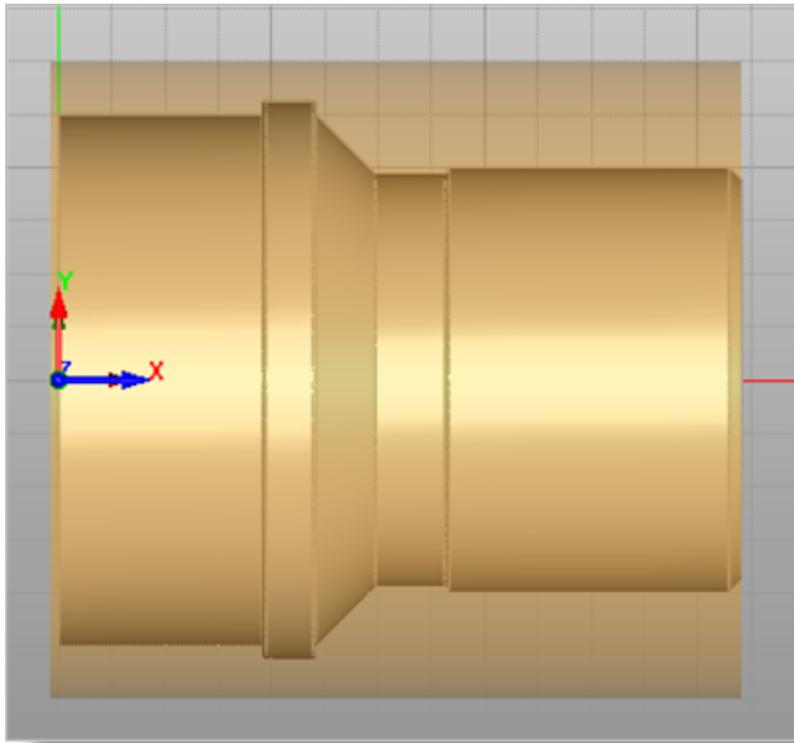
1. From the **Program** tab select **Align**.



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



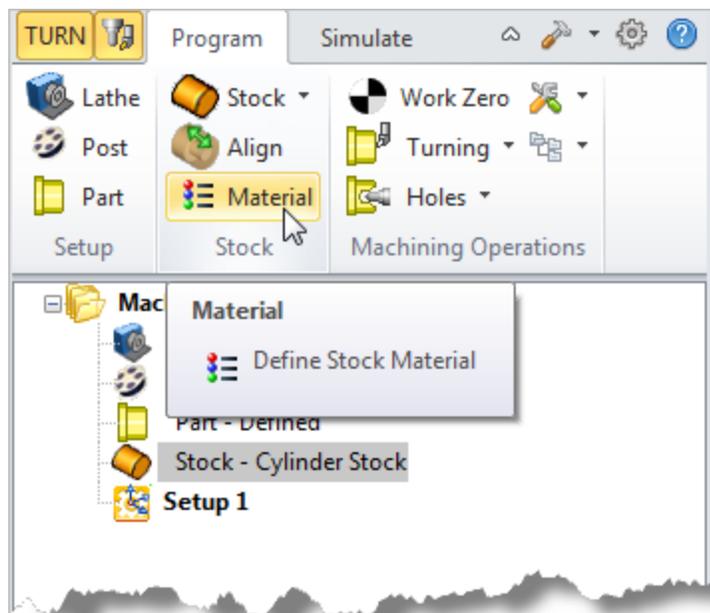
3. If you switch to the [Front View](#), you see that the stock is now aligned to the right side face of the part geometry in the Z axis of the lathe.



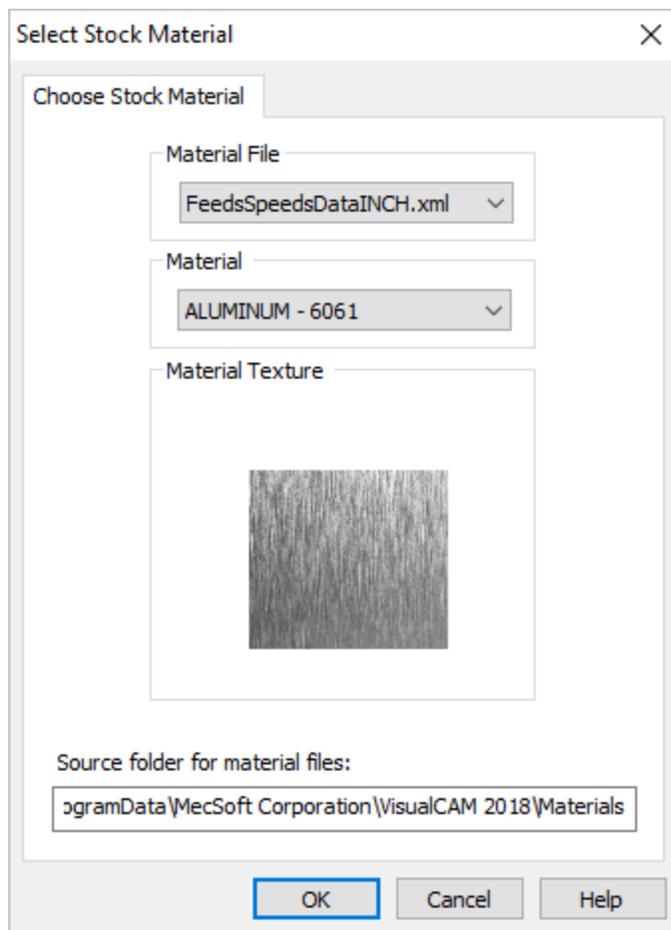
3.4 Specify the Material

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.

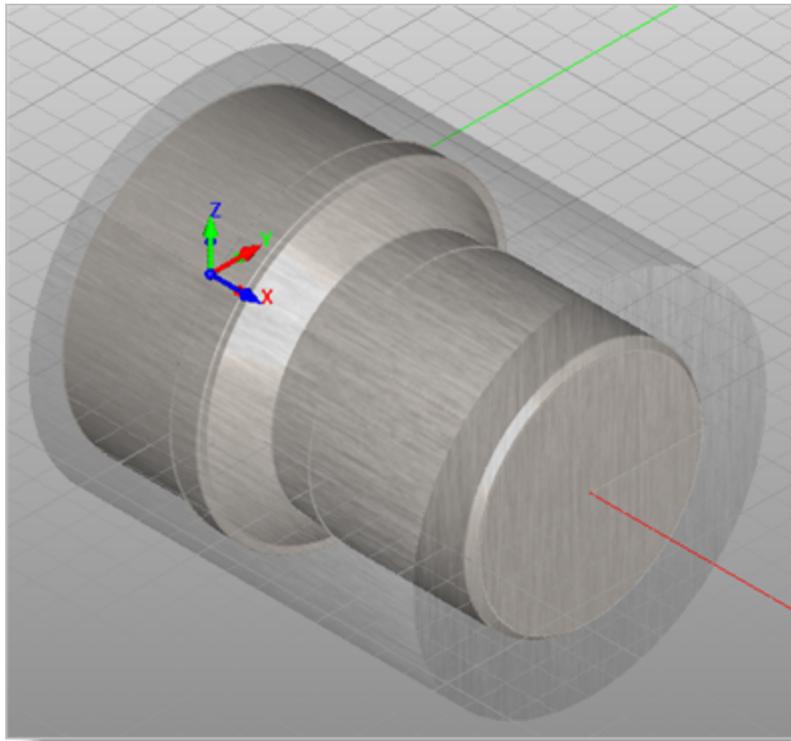
1. From the [Program](#) tab select [Material](#) to display the dialog.



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](#).



3.5 The Machining Setup

Now let's discuss the [Machining Setup](#).

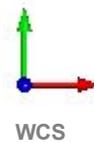
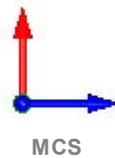
The [Setup](#) icon ( **Setup 1**) displayed in the [Machining Job](#) tree defines the [Turn Machine Coordinate System](#) or (MCS) and is defined automatically. CNC Turning centers or Lathes use the [Cartesian](#) coordinate system for programmed coordinates.

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			

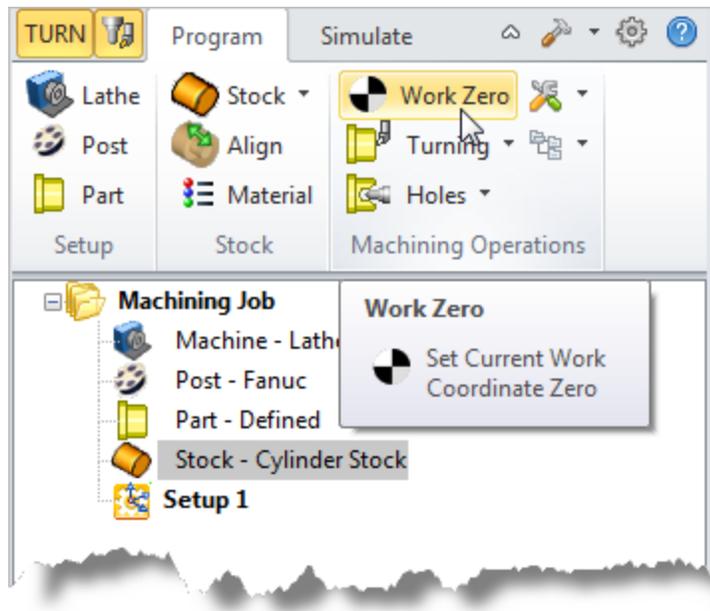
! By default Setup 1 is created when a new part is loaded. The MCS of this setup is oriented as described above. That is, the world XY plane is the same as the Lathe ZX plane. This cannot be changed. In the future, when you create a part profile for turning, create it in the XY plane of the WCS.

3.6 Set the Work Zero

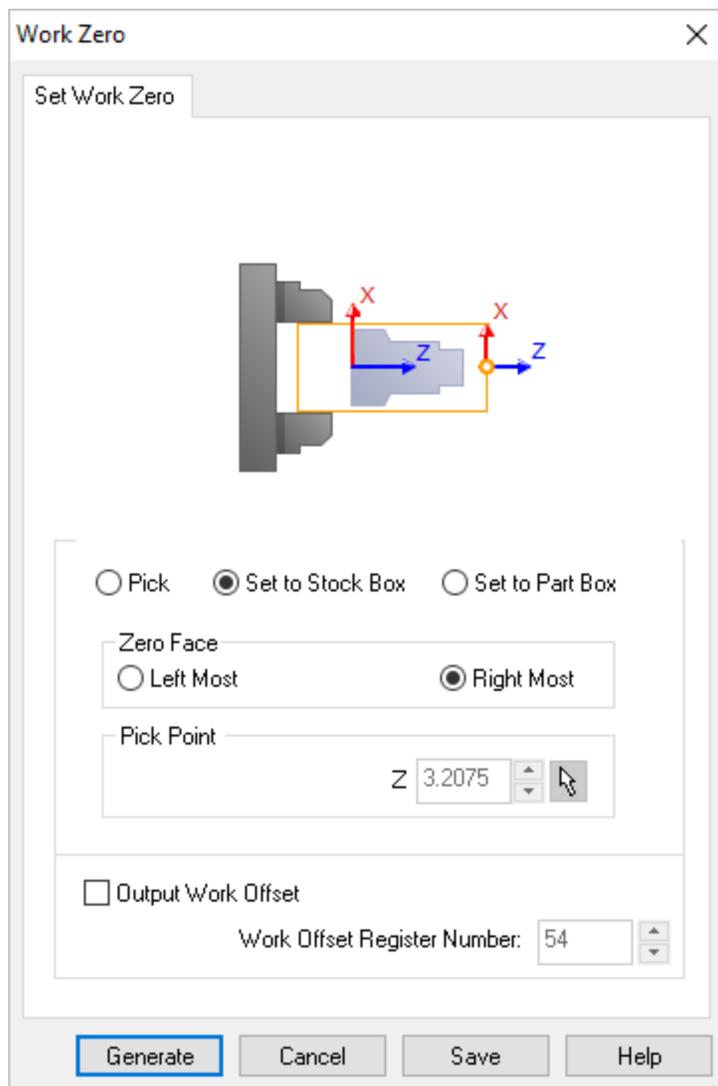
The Work Zero is used to define the work-piece origin. The Work Zero translates the Machine Coordinate System (MCS) origin from the origin defined in the Setup to the desired Work Zero location. This can be set to any location along the lathe Z axis. Typically this is set to the right most face of the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

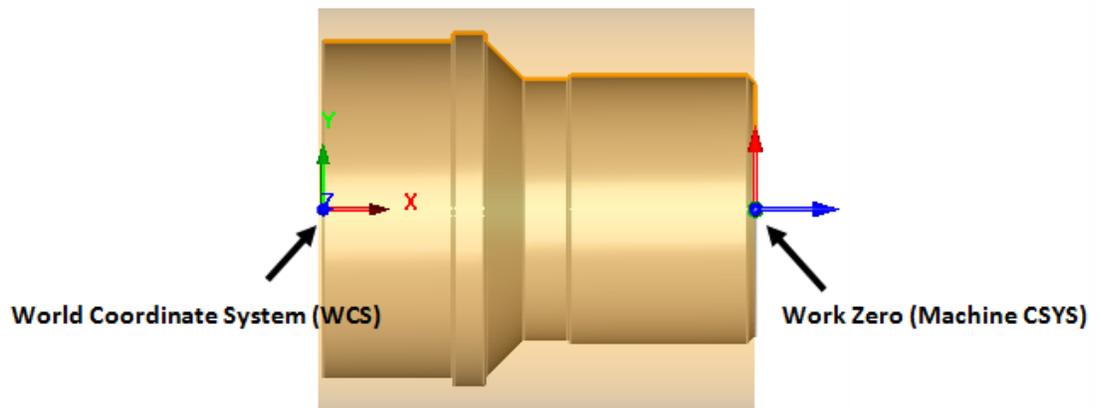
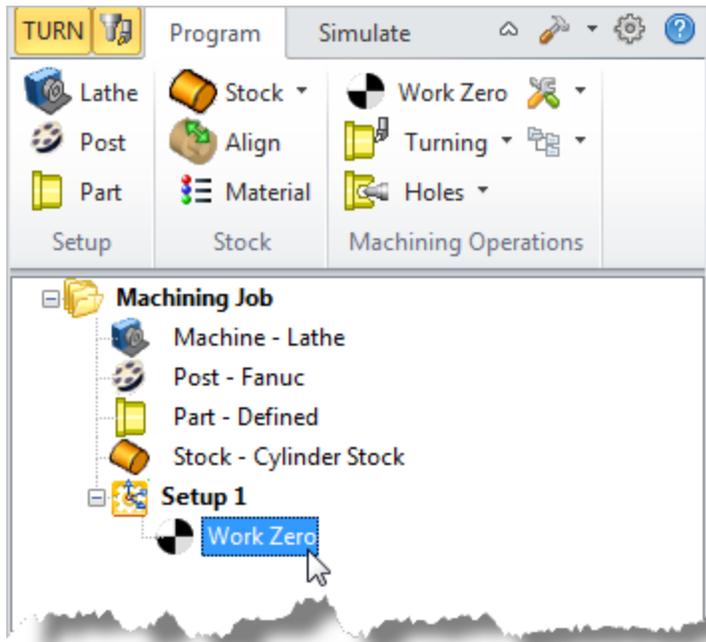
1. From the Program tab select Work Zero to display the dialog.



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**.



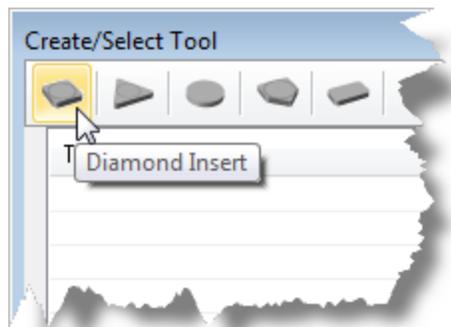
Create a Cutting Tool

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.

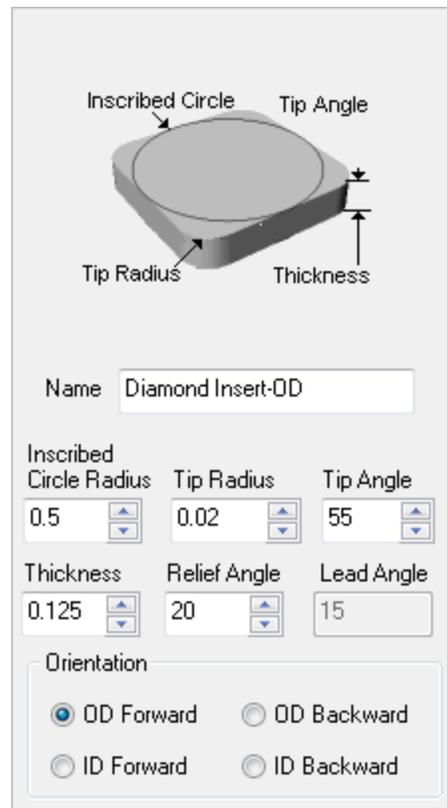


1. This will display the [Create/Select Tool](#) dialog. Select the [Diamond Insert](#) tool icon from the top-left side of the dialog.



2. Now set the following parameters in the dialog:

- Set Name as [Diamond Insert-OD](#).
- [Inscribed Circle Radius](#) to 0.5
- [Tip Radius](#) to 0.02
- [Tip Angle](#) to 55
- [Relief Angle](#) to 20
- [Thickness](#) to 0.125
- [Orientation](#) to OD Forward

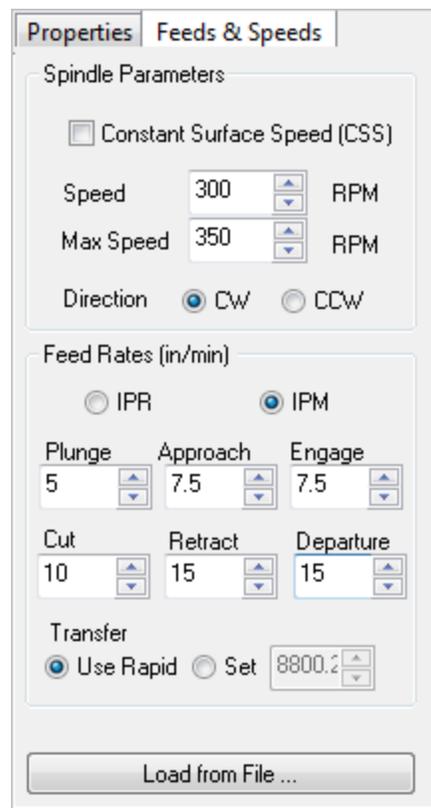


4. Next, we'll switch to the **Feeds and Speeds** tab and use the following values. For **Spindle Parameters** we'll set:

- Speed to 300 RPM
- Max Speed to 350 RPM

For **Feedrates** we'll select **IPM** and set:

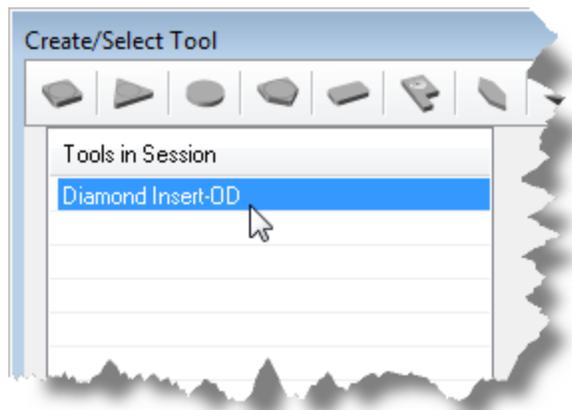
- Plunge to 5
- Approach to 7.5
- Engage to 7.5
- Cut to 10
- Retract to 15
- Departure to 15
- Transfer to Use Rapid



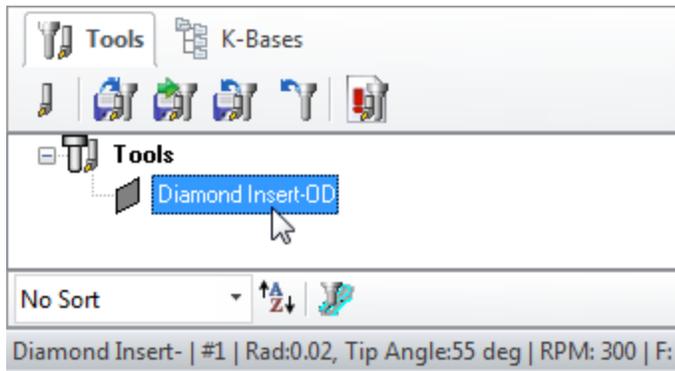
5. Now, we'll pick [Save as New Tool](#).

! 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. Now 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](#).

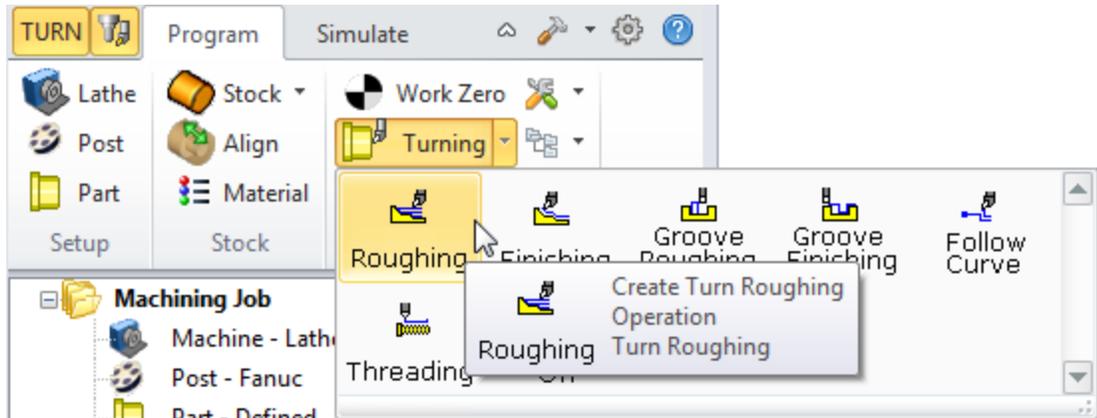


! In the future, to save Tools to a library, click **Save Tool library** under the **Tools** tab in **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box.

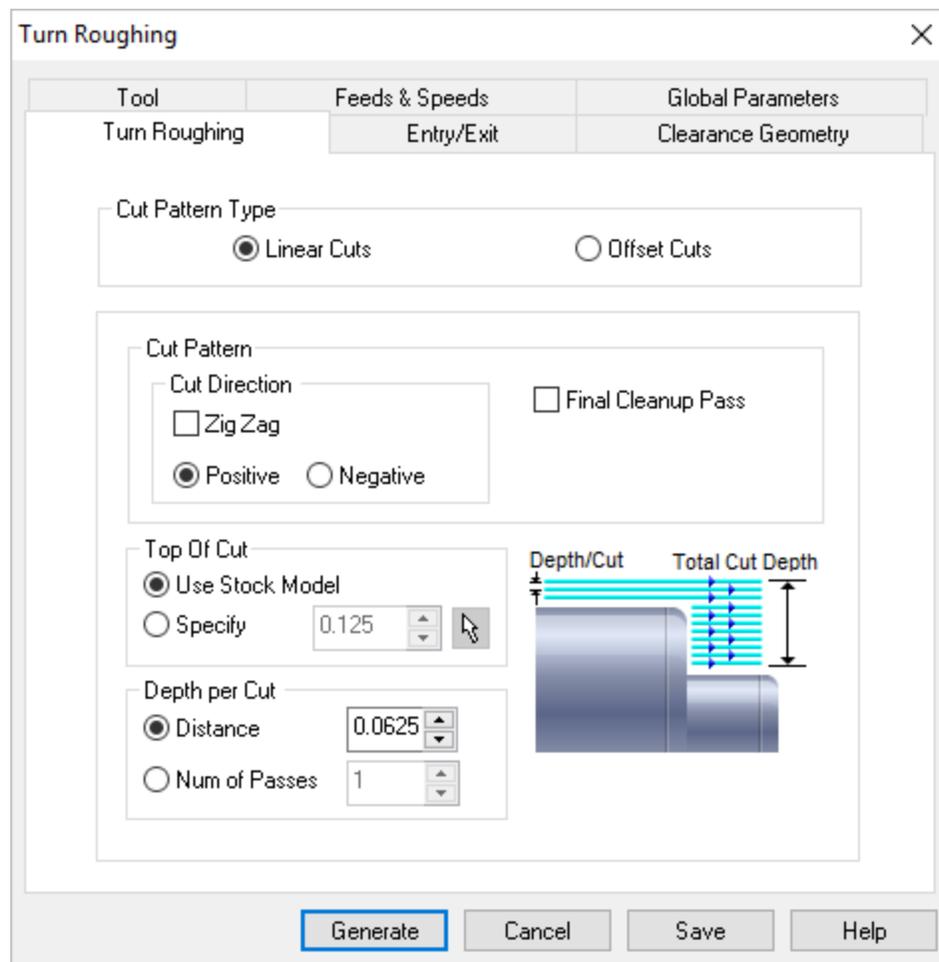
Roughing the Outer Diameter

Now we're 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.



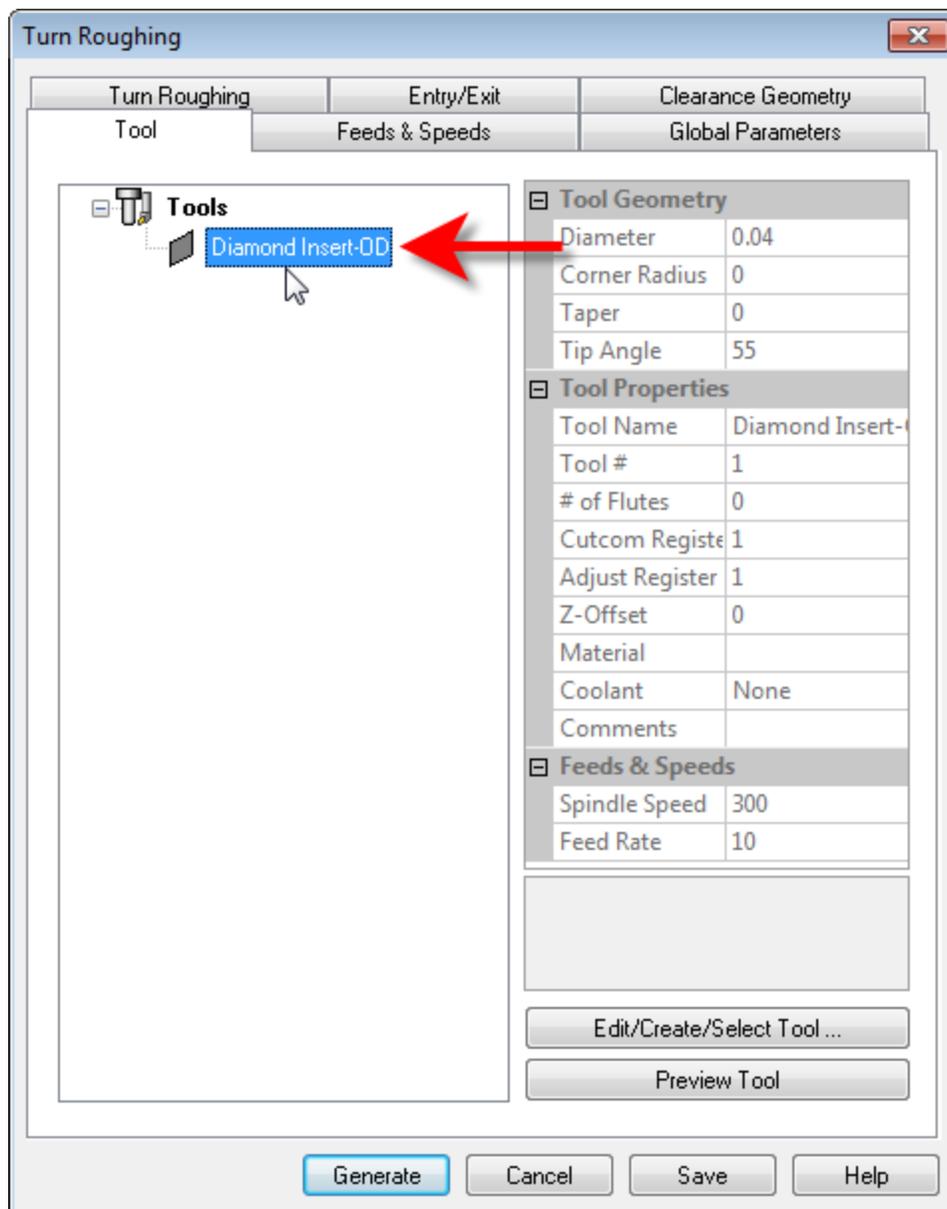
2. This will display the **Turn Roughing** operation dialog.



5.1 Select the Cutting Tool

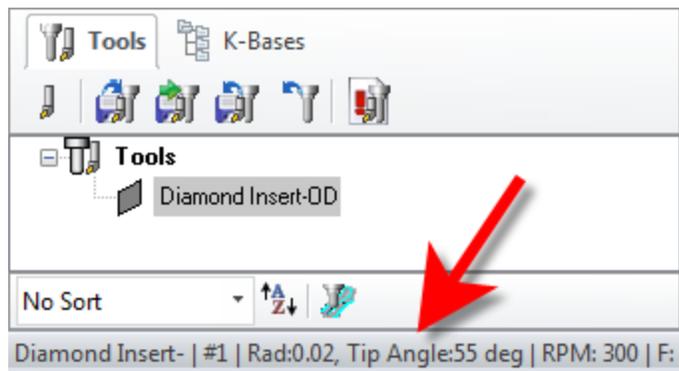
Next we'll select the cutting tool for the [Turn Roughing](#) operation.

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.

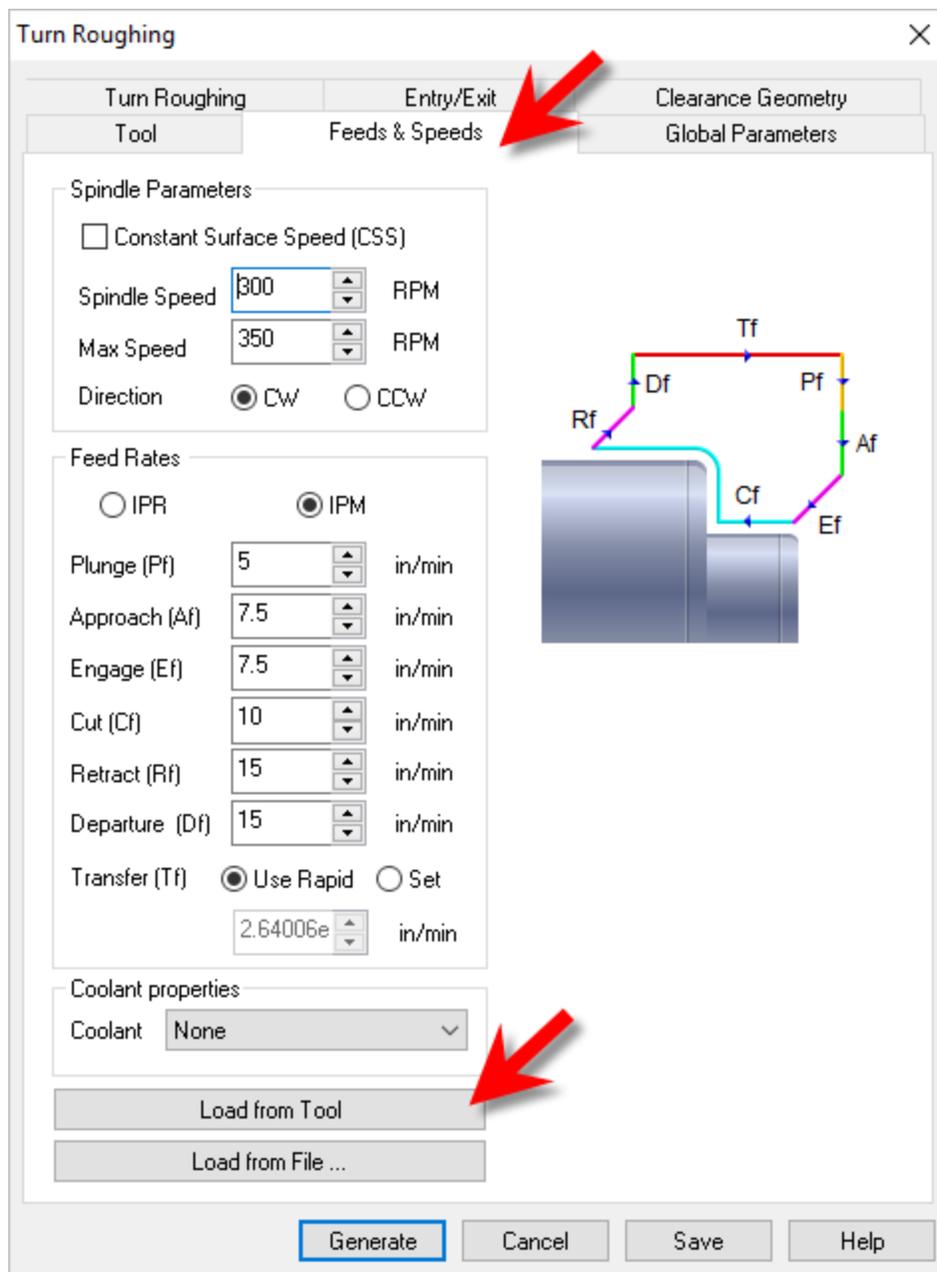
! The Tool parameters of the currently active tool are always displayed in the status bar at the bottom of the Machining Objects browser as shown below.



5.2 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.

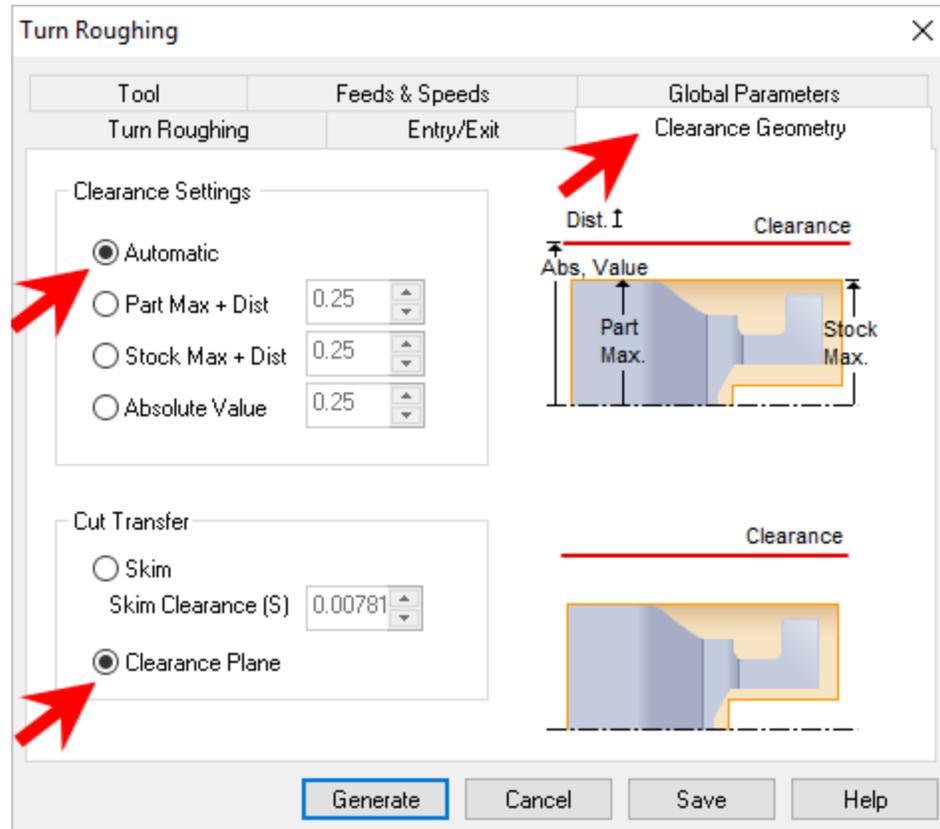
5.3 Set Clearance Geometry

Next we'll set the [Clearance Geometry](#) for the [Turn Roughing](#) operation.

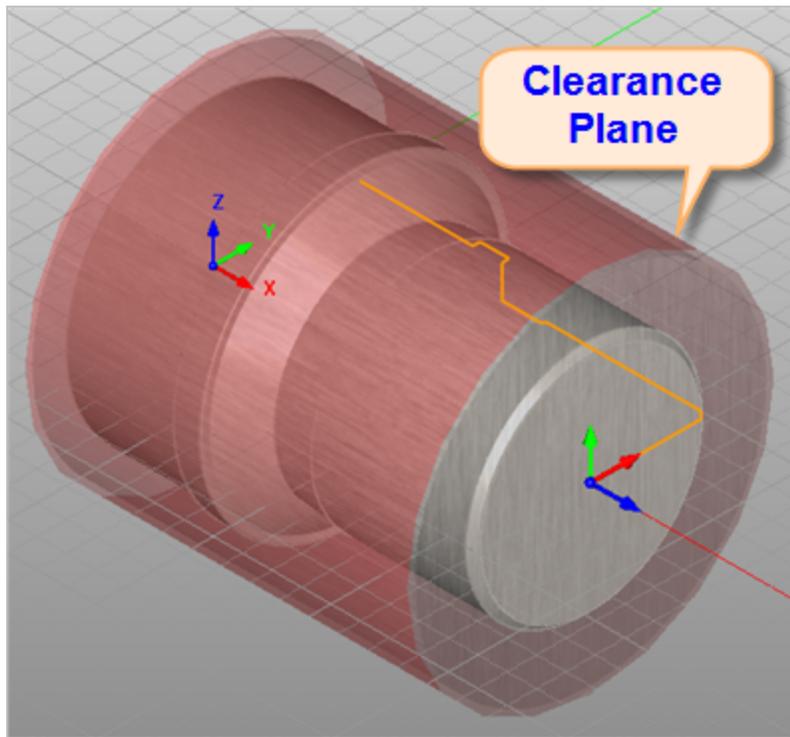
1. Select the [Clearance Geometry](#) tab of the dialog.

- 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.



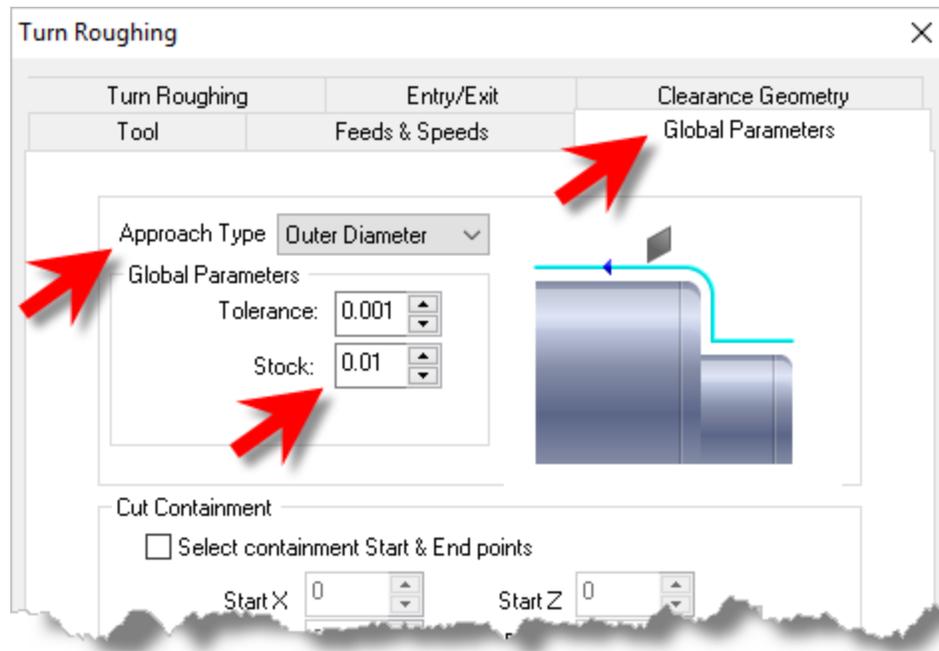
- When this dialog is active, the clearance plane is shown on the graphics screen.



5.4 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.

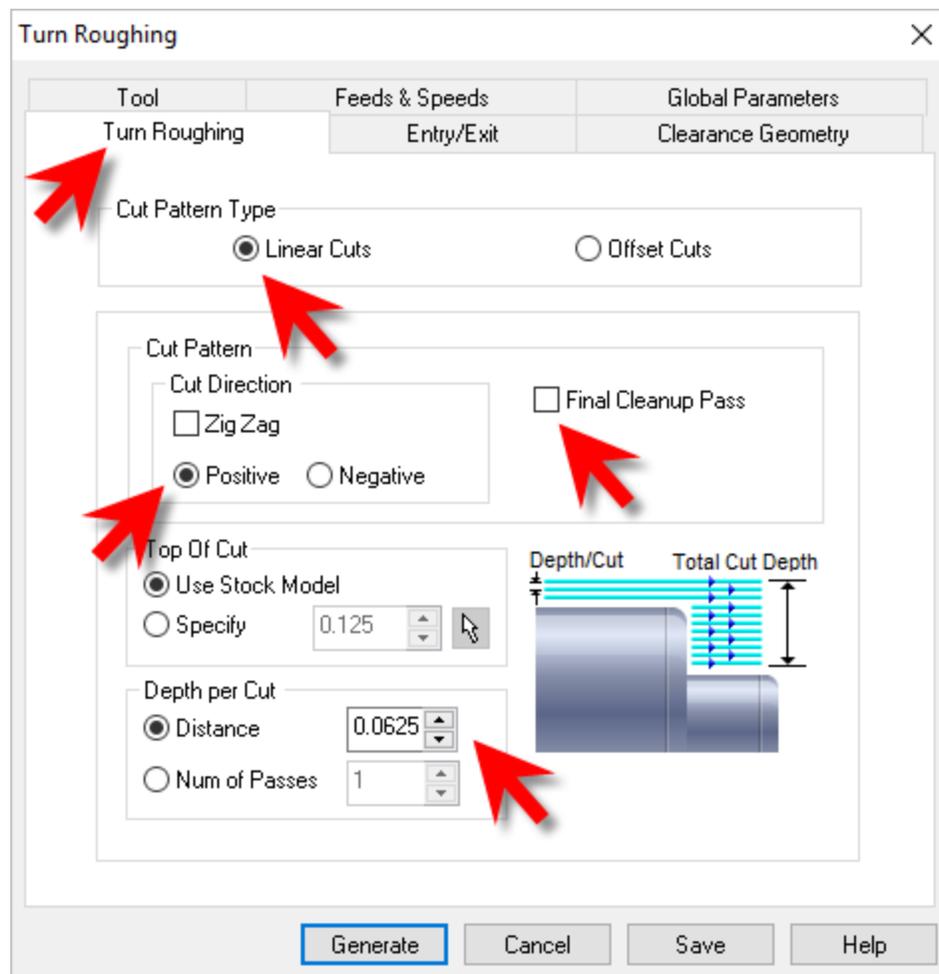


5.5 Set Roughing Parameters

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.

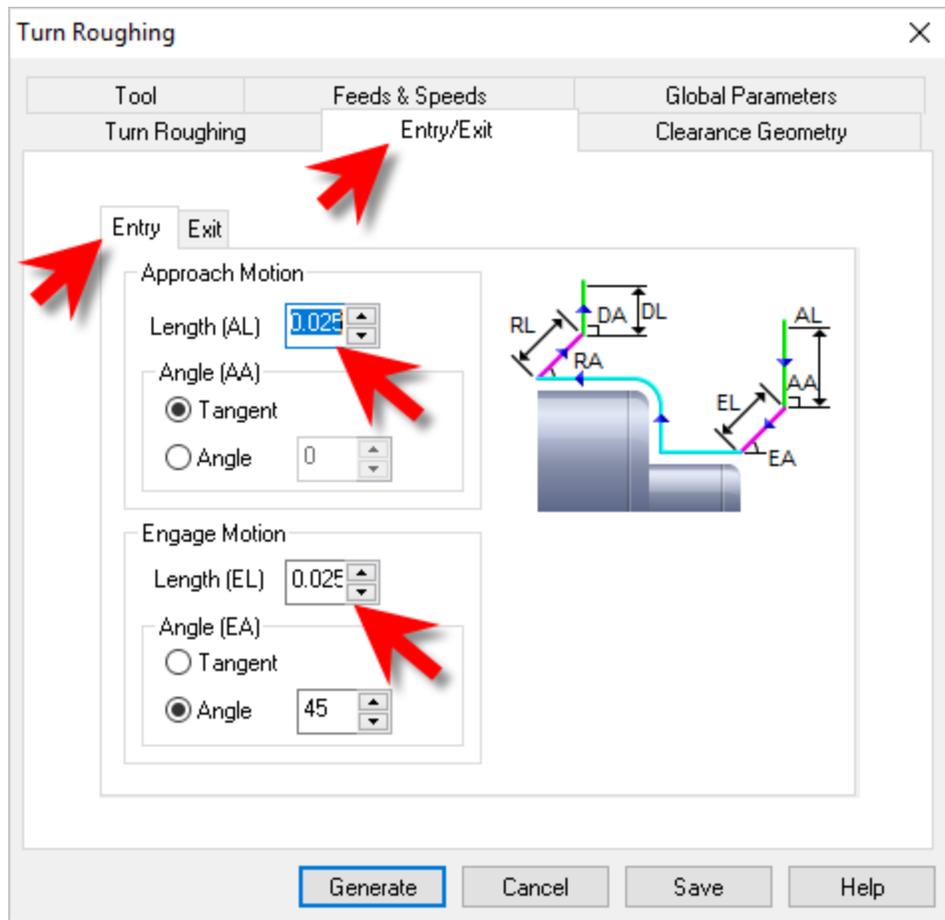
Note that [Depth per Cut](#) is always set to an absolute value.



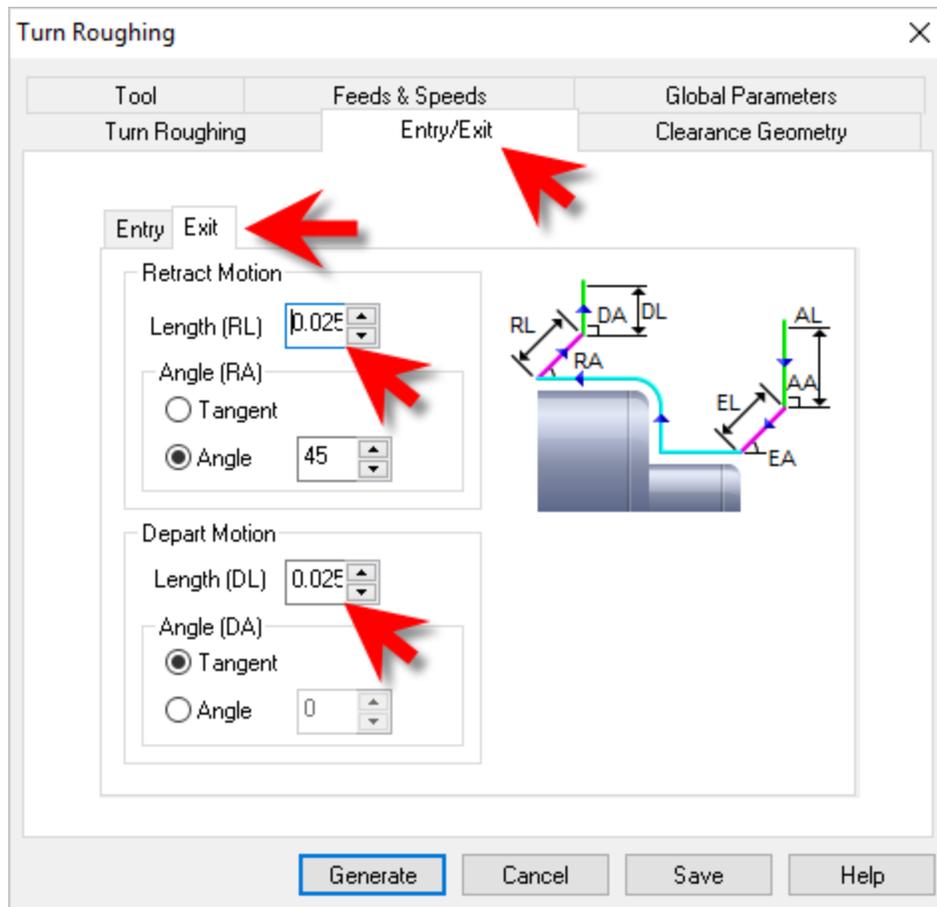
5.6 Set Entry/Exit Parameters

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.

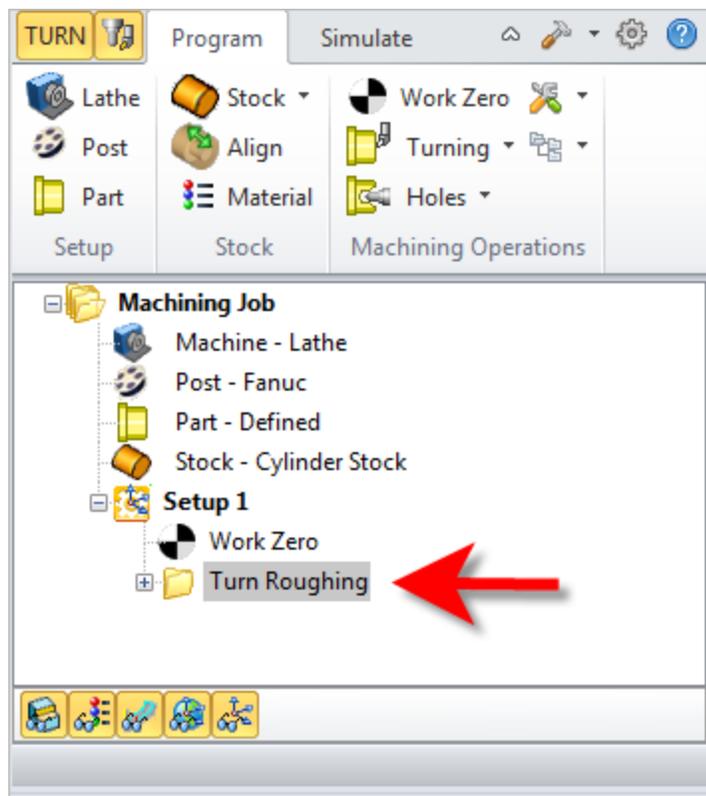


- Next, we'll switch to **Exit** tab set the **Retract Motion Length (RL)** to 0.025 and do the same for the **Depart Motion Length (DL)**.

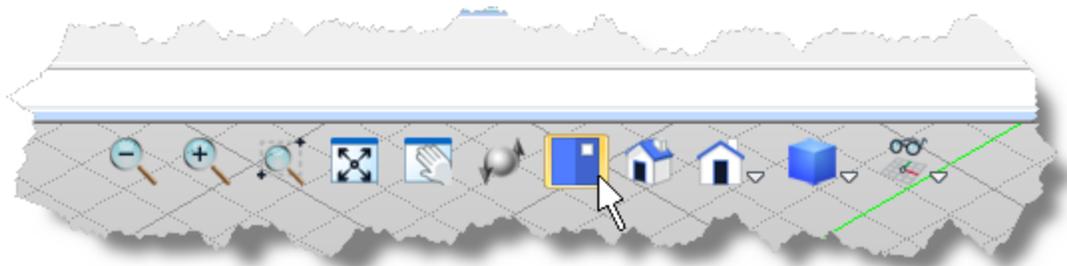


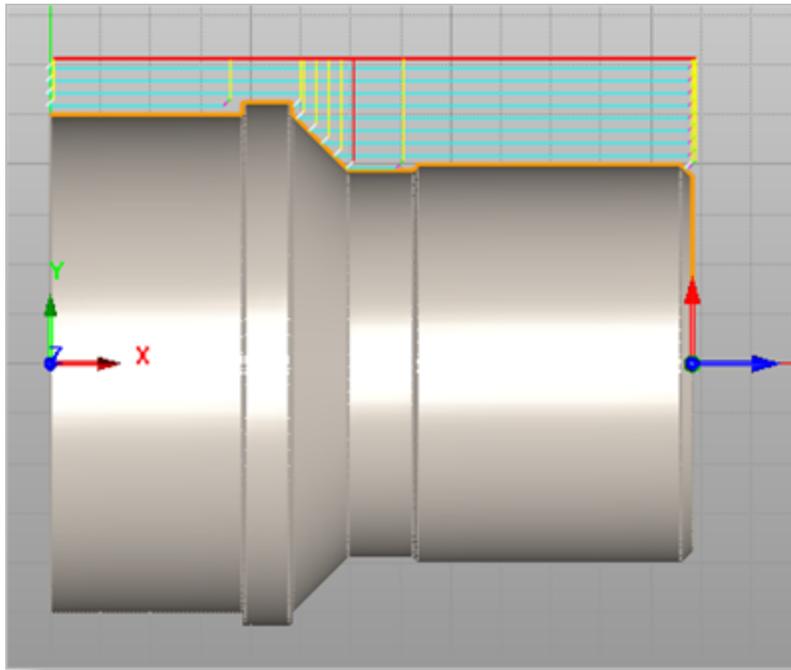
5. Now pick [Generate](#).

The [Turn Roughing](#) toolpath is generated and the operation is listed under [Setup 1](#) in the [Machining Browser](#).

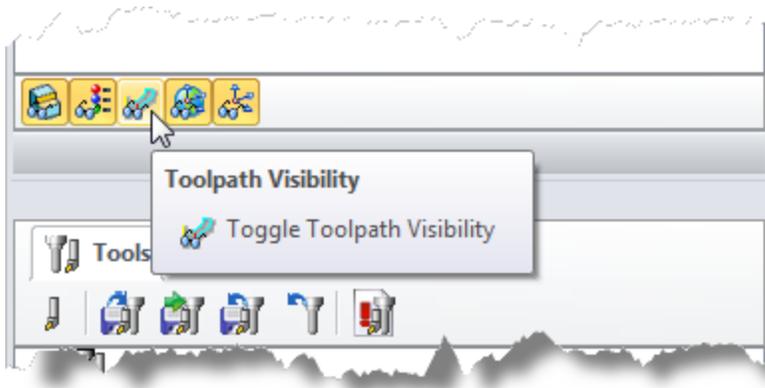


6. Now from the **View** toolbar in **VisualCAD**, select the **Top View** and you can see the toolpath displayed clearly in the graphics screen.





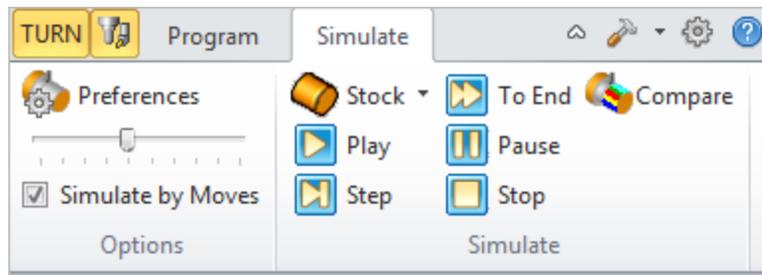
! The display of the toolpath in the graphics screen can be turned on/off by selecting the **Toolpath Visibility** icon in the toolbar at the bottom of the **Machining Browser**.



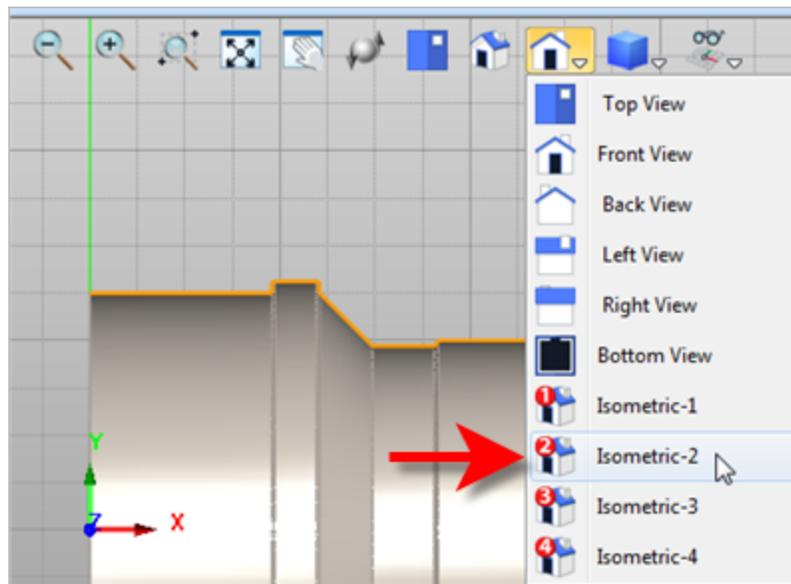
5.7 Simulate the Toolpath

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**.

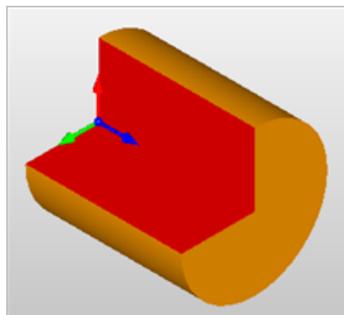
1. Switch to the **Simulate** tab at the top of the **Machining Browser**.



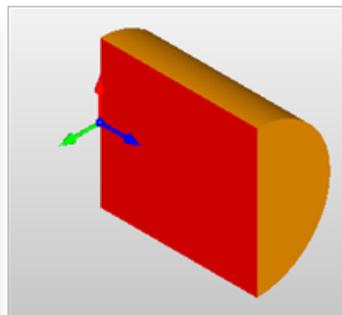
- From the **View** toolbar, select the **Isometric-2** icon



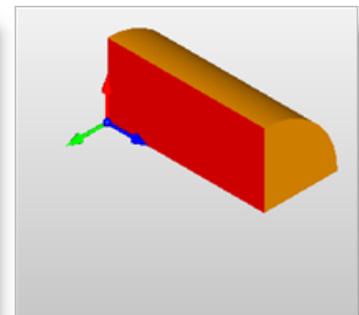
- In the **TURN** module the rendering mode of the simulation model can be controlled for better visibility by using cut away section views. Three modes, in addition to the normal rendering mode, are available. These are **3 Quarter**, **Half** and **Quarter** views. These display modes can be set in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



3 Quarter View

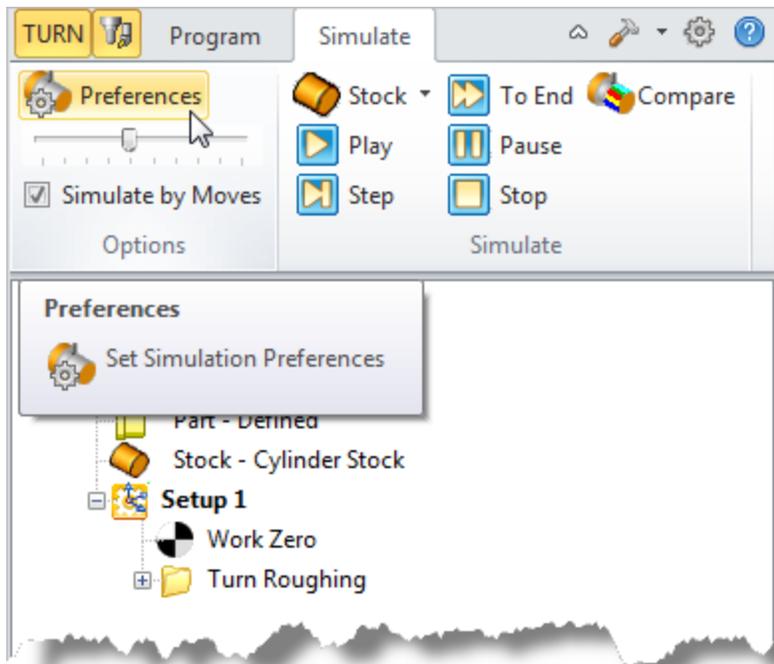


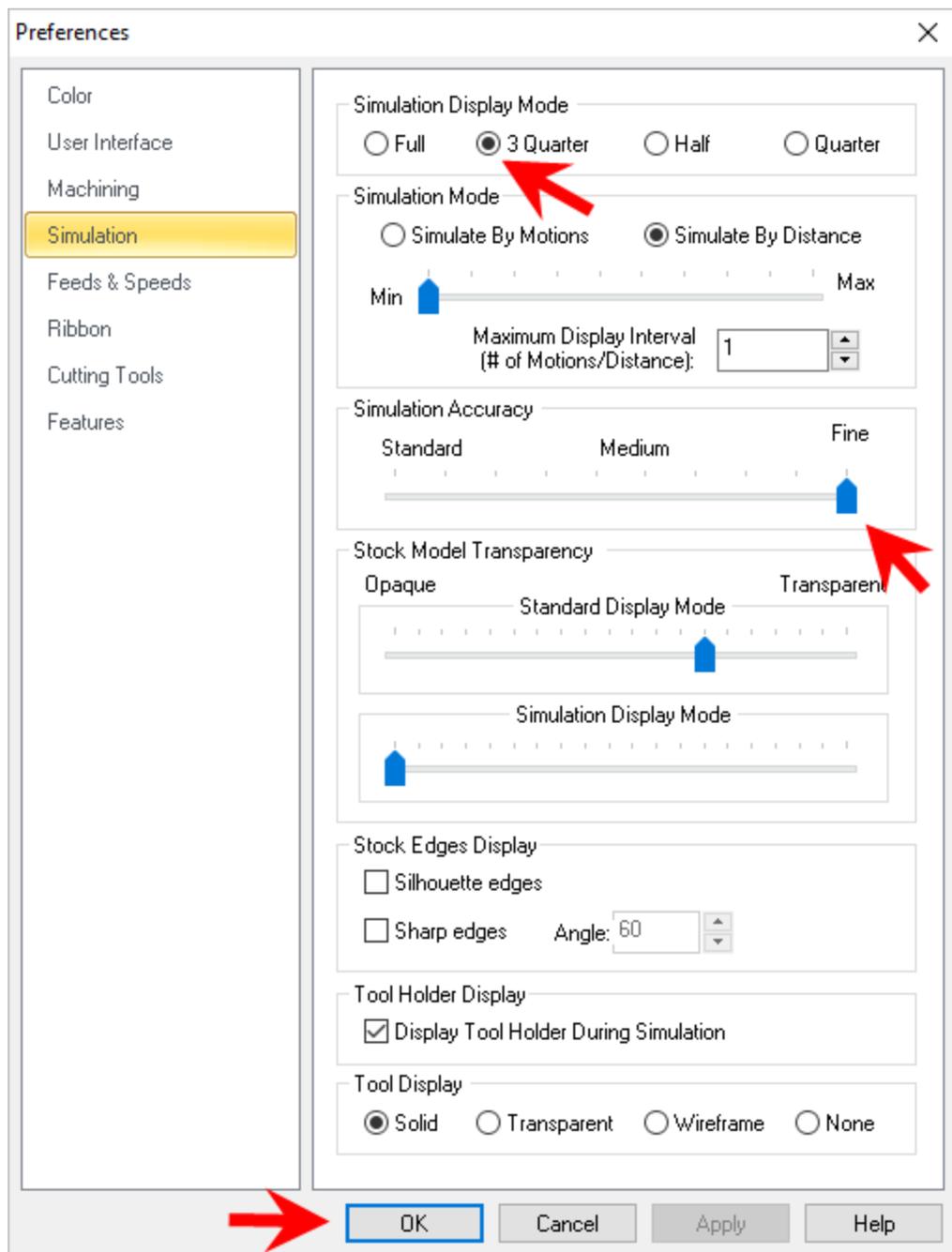
Half View



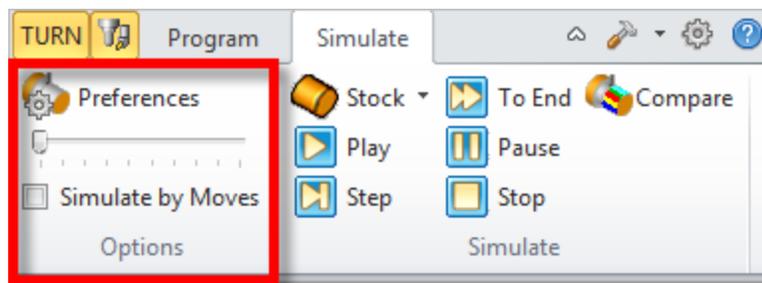
Quarter View

- 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**.

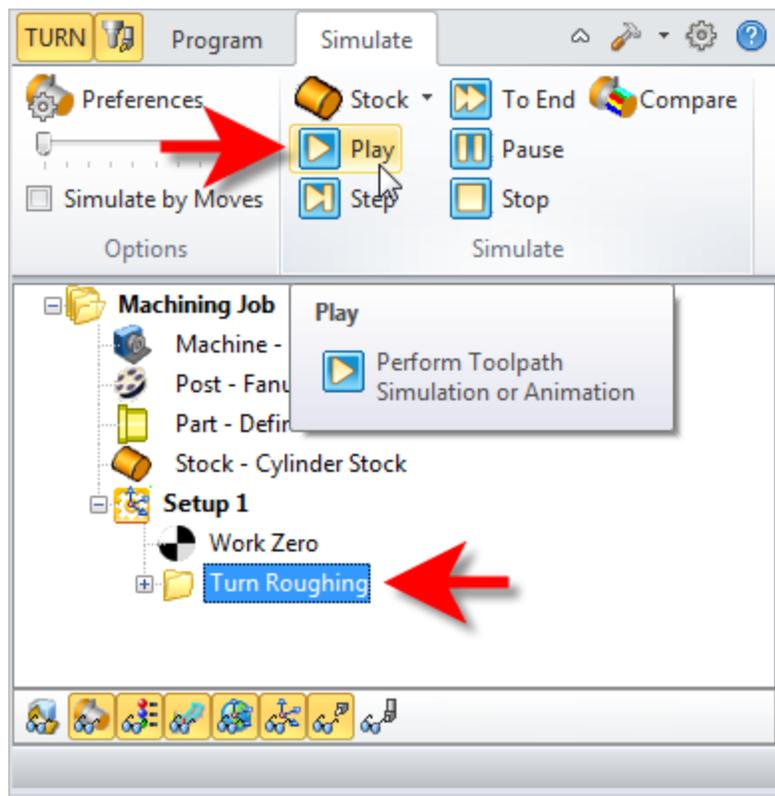




5. From the **Simulate** tab, uncheck **Simulate by Moves** and then move the slider to the left to slow down the simulation speed.

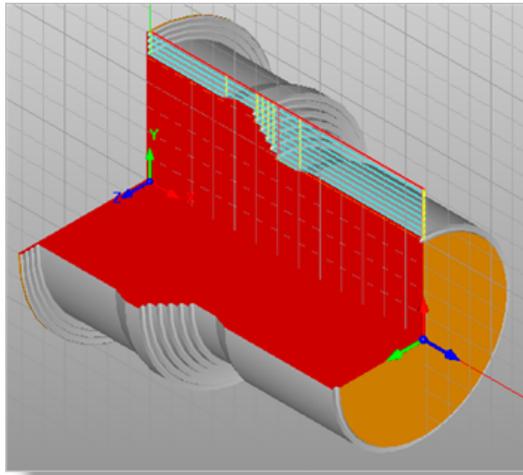


6. Now select the Turn Roughing operation under Setup 1 and then pick Play.

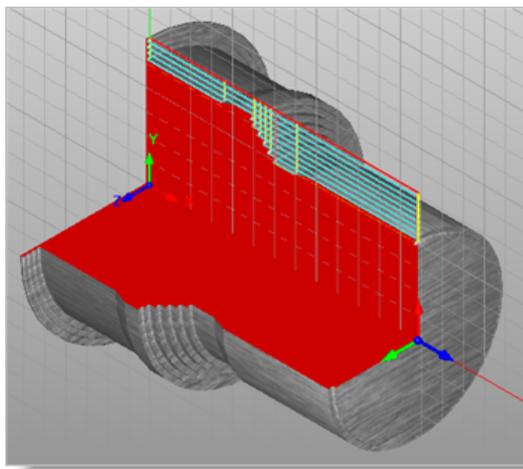
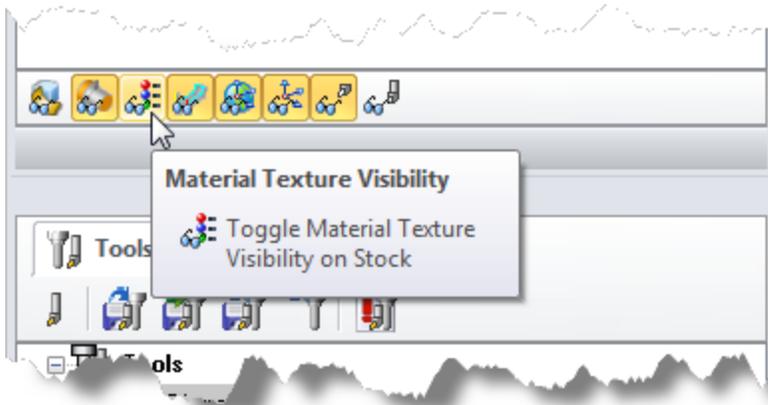


! You can stop the simulation at anytime by selecting the **Pause** button from the **Simulate** tab. After **Pausing**, you can choose either **Play** to continue or **Stop** to exit the simulation.

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



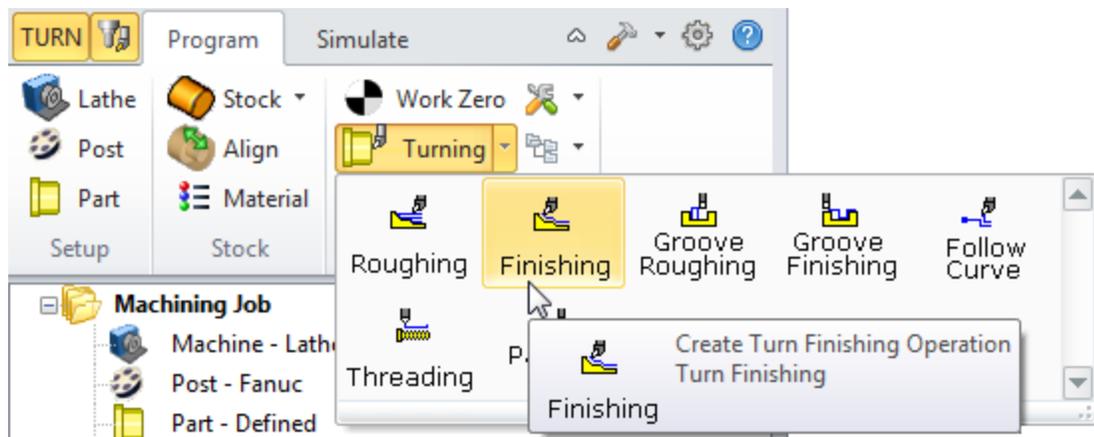
- To view the cut model with textures applied, select the **Material Texture Visibility** icon located at the base of the **Machining Browser**.



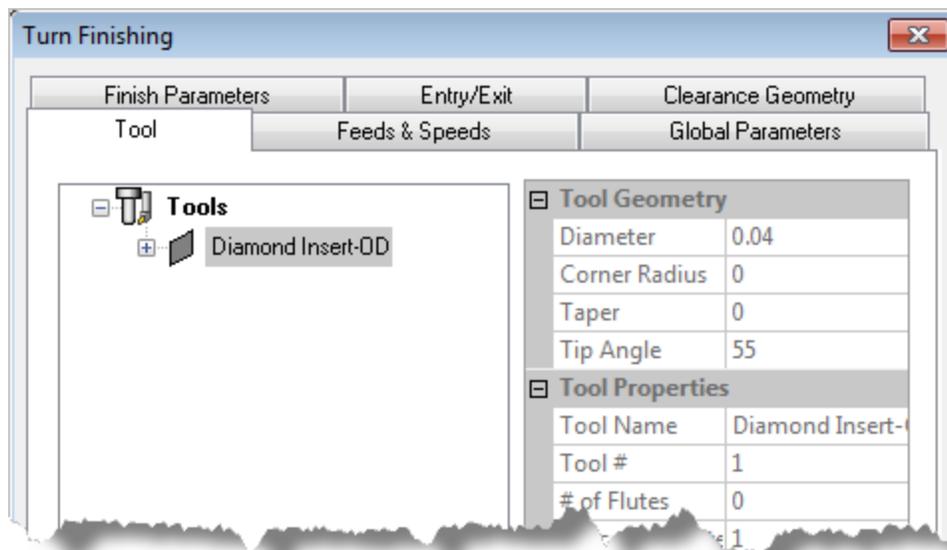
Finishing Outer Diameter

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.



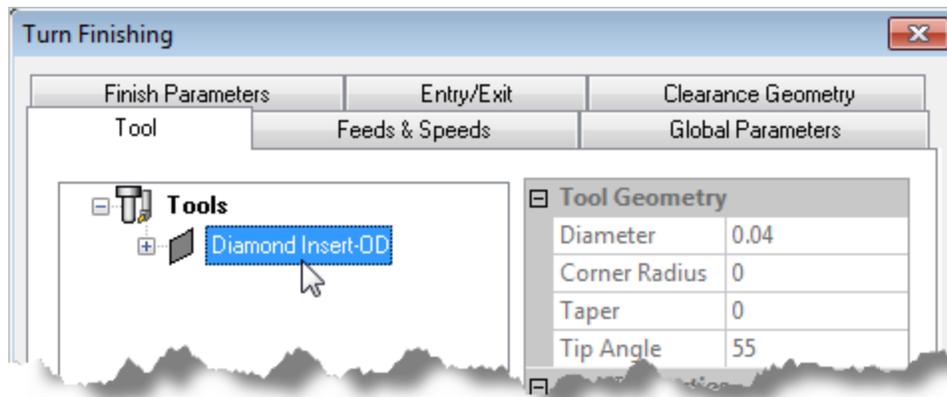
3. This will display the **Turn Finishing** operation dialog shown below.



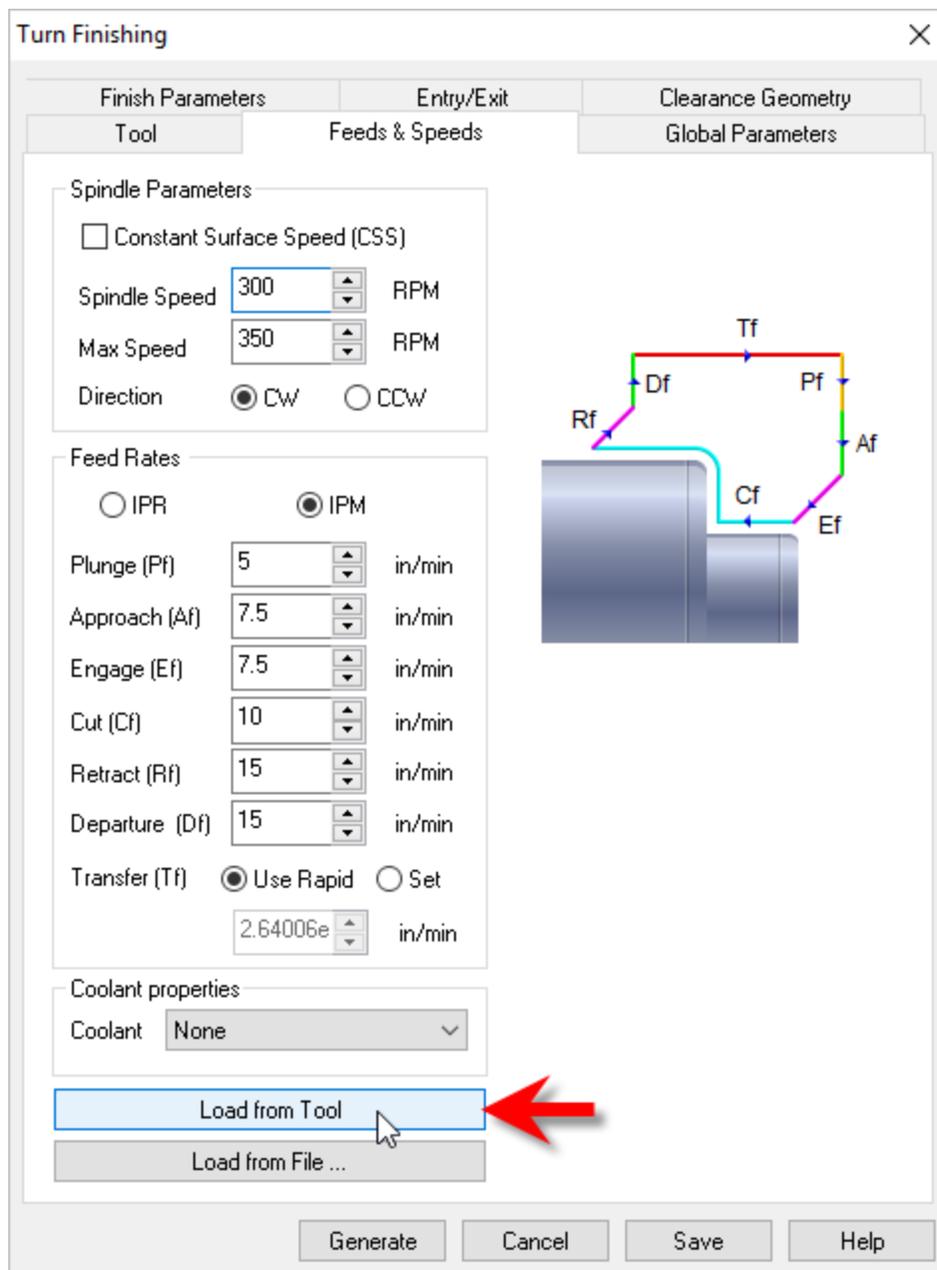
6.1 Set Finishing Parameters

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

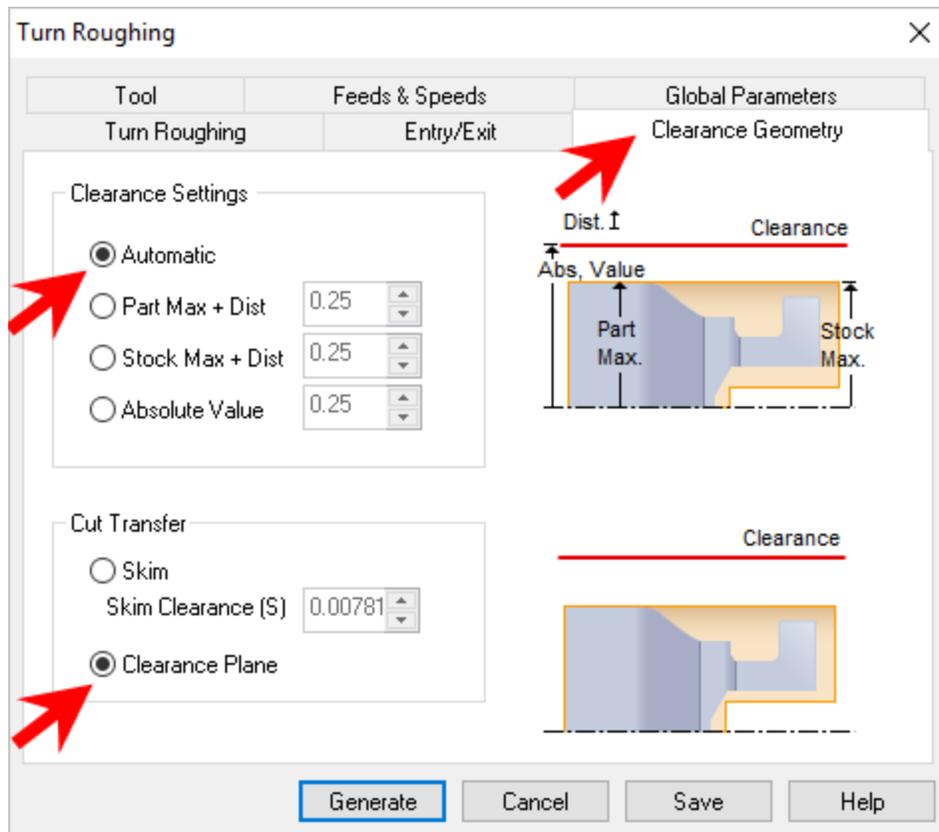
1. From the **Tool** tab in the **Turn Finishing** dialog select the **Diamond Insert-OD** tool.



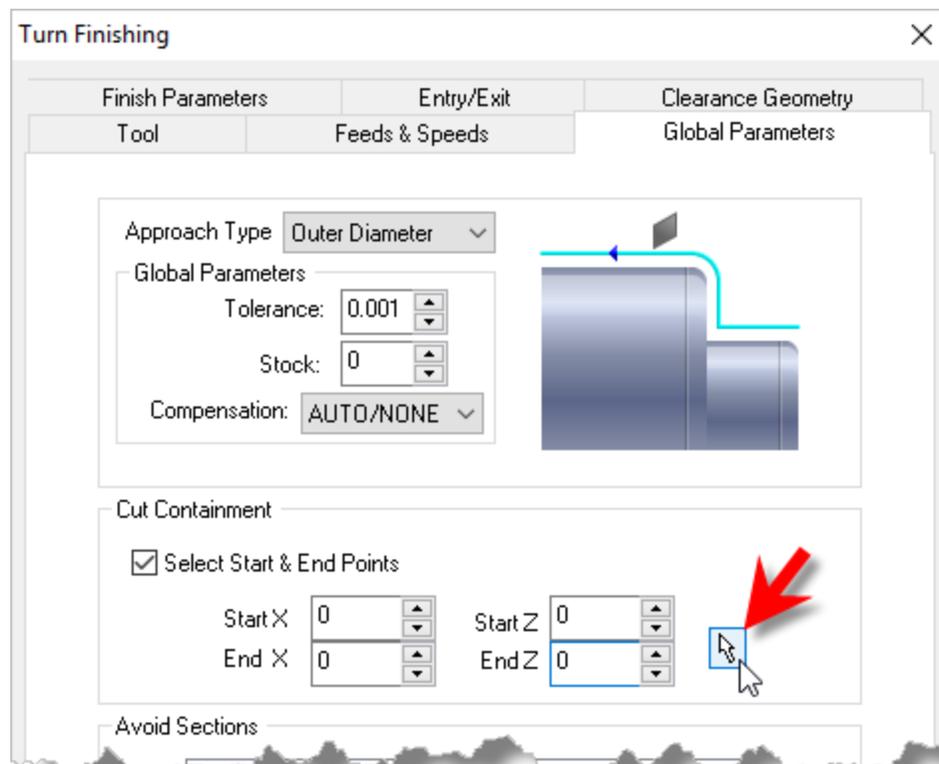
2. Now, pick the **Feeds & Speeds** tab and select the **Load from Tool** button. The system will retrieve the feeds and speeds parameters that was set when the tool was defined and associate them with the current operation.



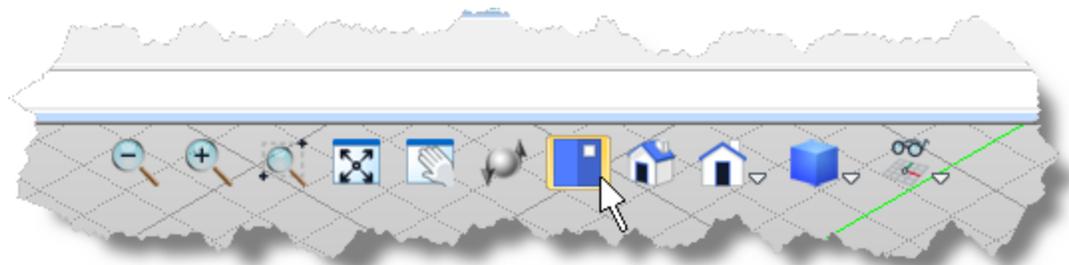
3. Pick the **Clearance Geometry** tab and set the **Clearance Settings** to **Automatic** and **Cut Transfer** to **Clearance Plane**.



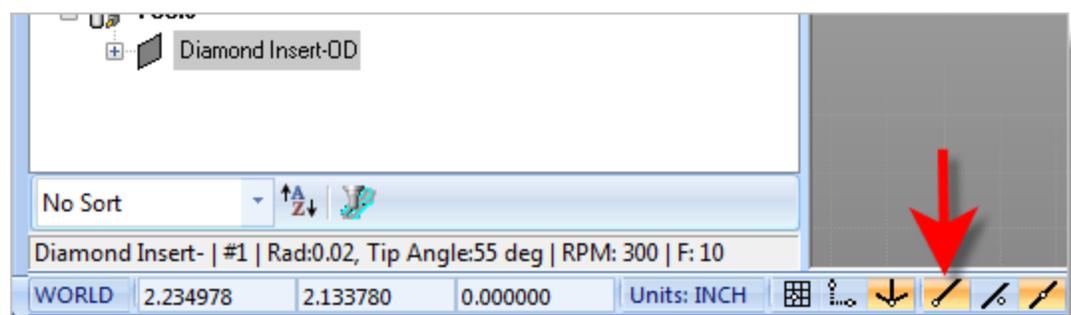
4. Now we'll switch to the **Global Parameters** tab to specify parameters to control the cutting.
5. Set the **Approach Type** to **Outer Diameter** and **Stock = 0**.
We will not be leaving any thickness on the part after machining, effectively removing all stock left over from the previous roughing 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.

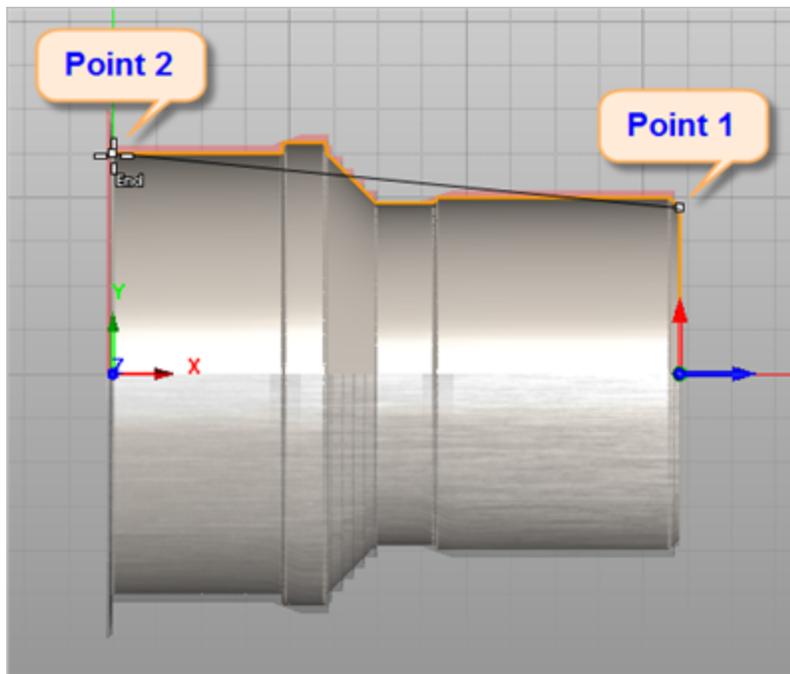


8. From the [View](#) toolbar, we'll switch to [Top View](#).

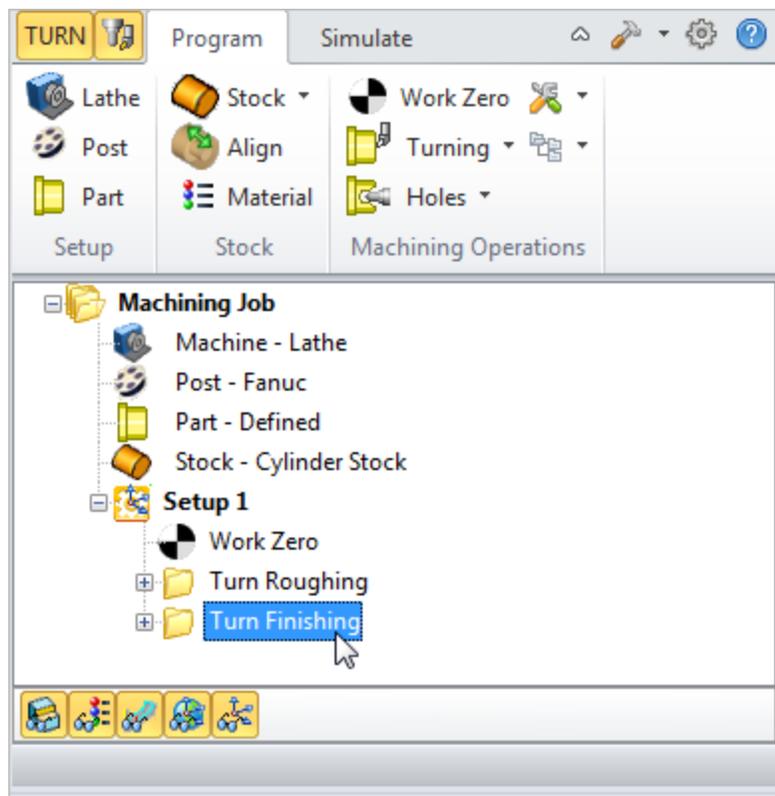


9. Now, with the [End Snap](#) toggle on, we'll select 2 points as shown below.





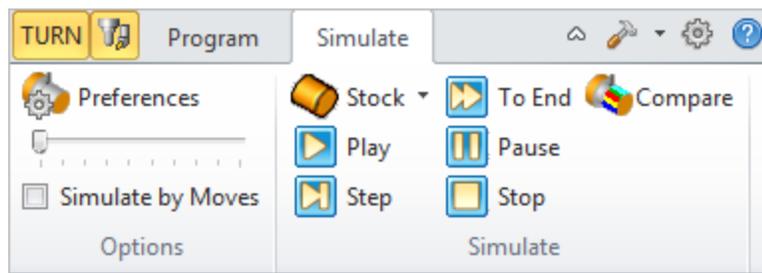
10. The **Turn Finishing** dialog reappears and displays **Start** and **End** point coordinate values for the cut containment.
11. Leaving all other parameters with default settings, we pick **Generate**. The operation is generated and added to the **Machining Job** in the **Machining Browser** below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



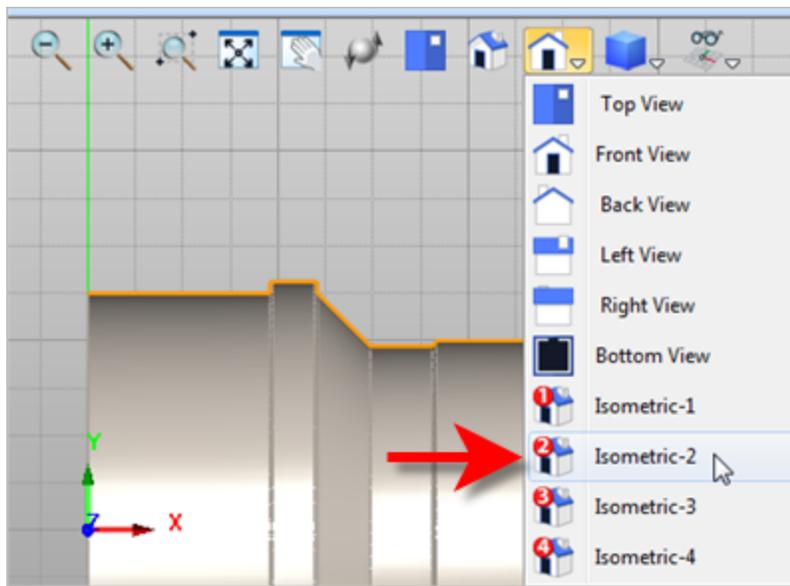
6.2 Simulate the Toolpath

We're now ready to [Simulate](#) the operation.

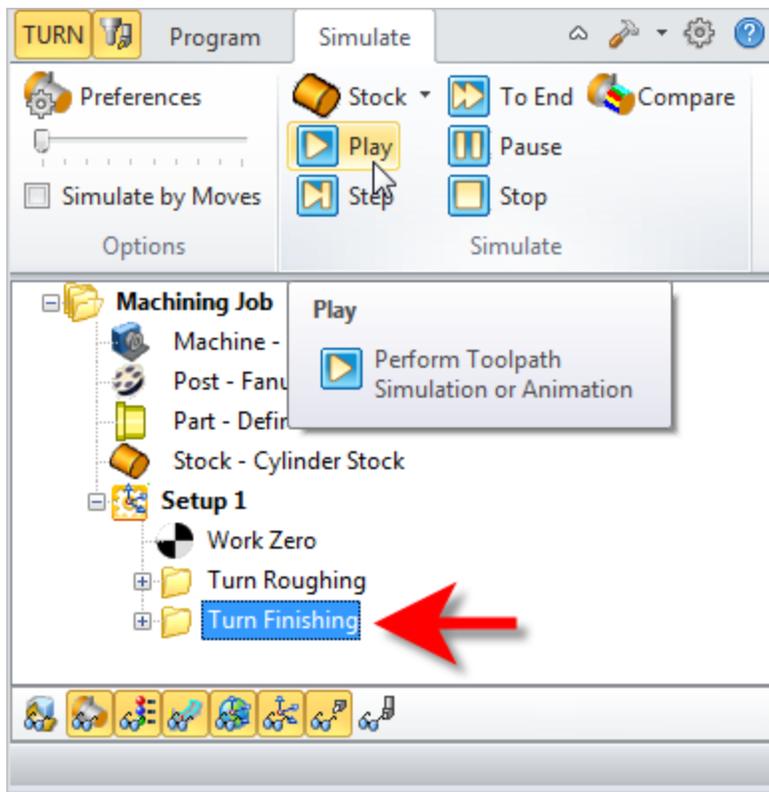
1. First make sure the [Turn Finishing](#) toolpath is selected from the [Machining Job](#) tree.
2. Select the [Simulate](#) tab.

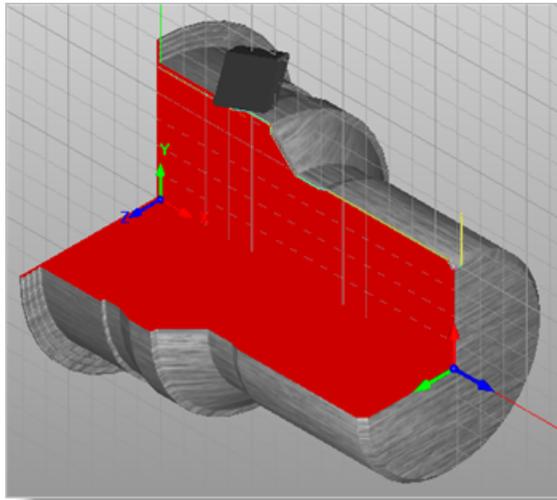


3. From the [View](#) toolbar, select [Isometric 2](#).



4. Now pick **Play** to view the simulation.

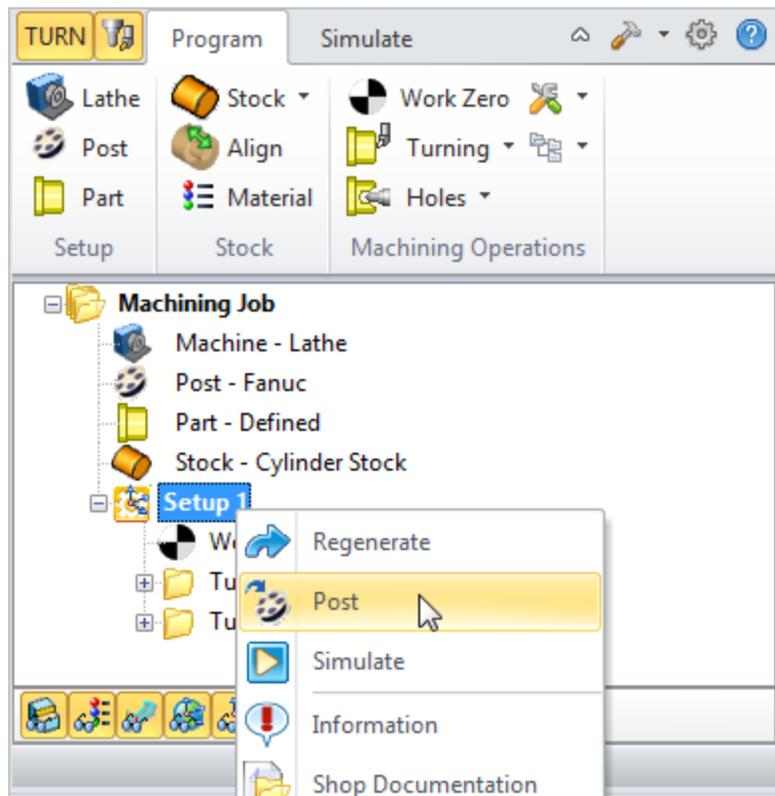




6.3 Post-Process the Toolpath

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**.



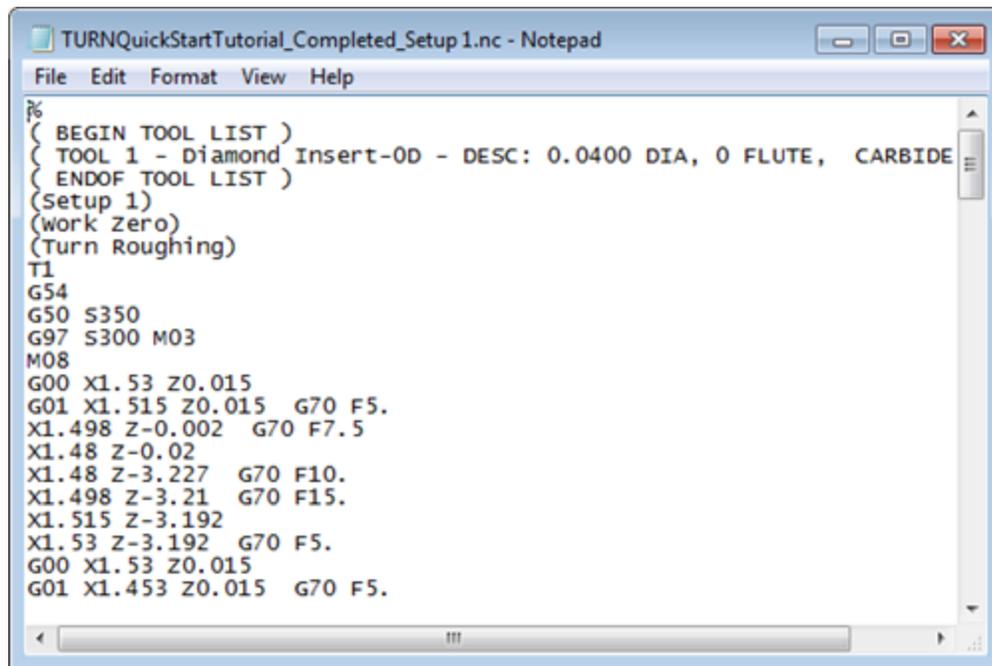
2. This will post-process all operations created under **Setup 1**.

- 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.

- Now pick [Post](#) and the G-code file is displayed in [Notepad](#) where it can be viewed or edited manually.



```
TURNQuickStartTutorial_Completed_Setup1.nc - Notepad
File Edit Format View Help
%
( BEGIN TOOL LIST )
( TOOL 1 - Diamond Insert-0D - DESC: 0.0400 DIA, 0 FLUTE, CARBIDE
( ENDOF TOOL LIST )
(Setup 1)
(work Zero)
(Turn Roughing)
T1
G54
G50 S350
G97 S300 M03
M08
G00 X1.53 Z0.015
G01 X1.515 Z0.015 G70 F5.
X1.498 Z-0.002 G70 F7.5
X1.48 Z-0.02
X1.48 Z-3.227 G70 F10.
X1.498 Z-3.21 G70 F15.
X1.515 Z-3.192
X1.53 Z-3.192 G70 F5.
G00 X1.53 Z0.015
G01 X1.453 Z0.015 G70 F5.
```

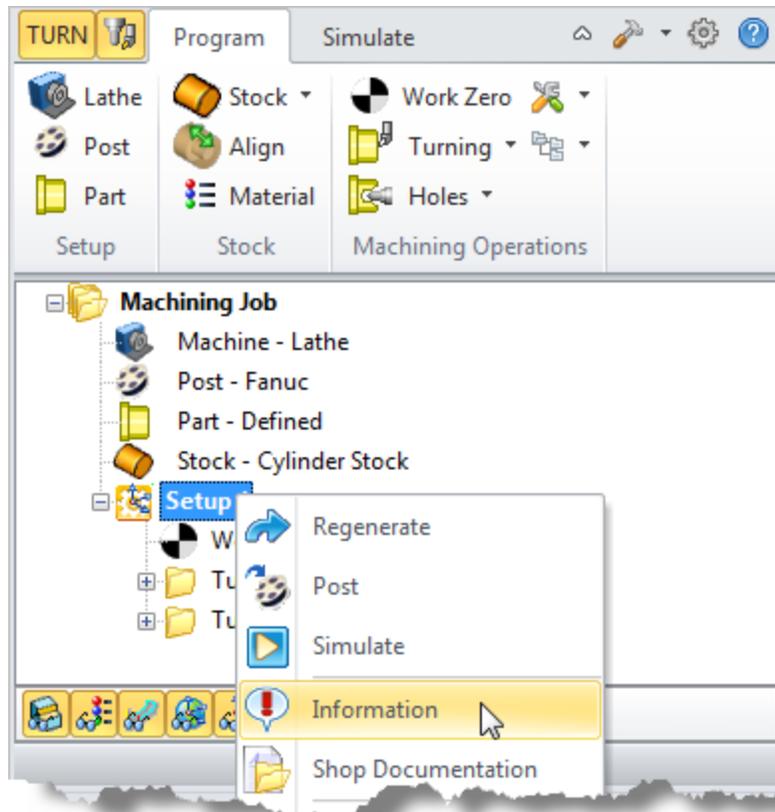
5. Now close Notepad.

Generate Reports

7.1 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.



Name	Status	Tool	Tool #	Cut Feed	# of GOTOs	Machine Time
Setup 1						
Work Zero	Clean	No Tool	-	0.0		
Tum Roughing	Clean	Diamond In...	1	10.00 in/min	143	5.00 min
Tum Finishing	Clean	Diamond In...	1	10.00 in/min	37	0.44 min
					Sub-total	5.44 min

This dialog provides an estimate of the machining time required for the operations in the [Setup](#).

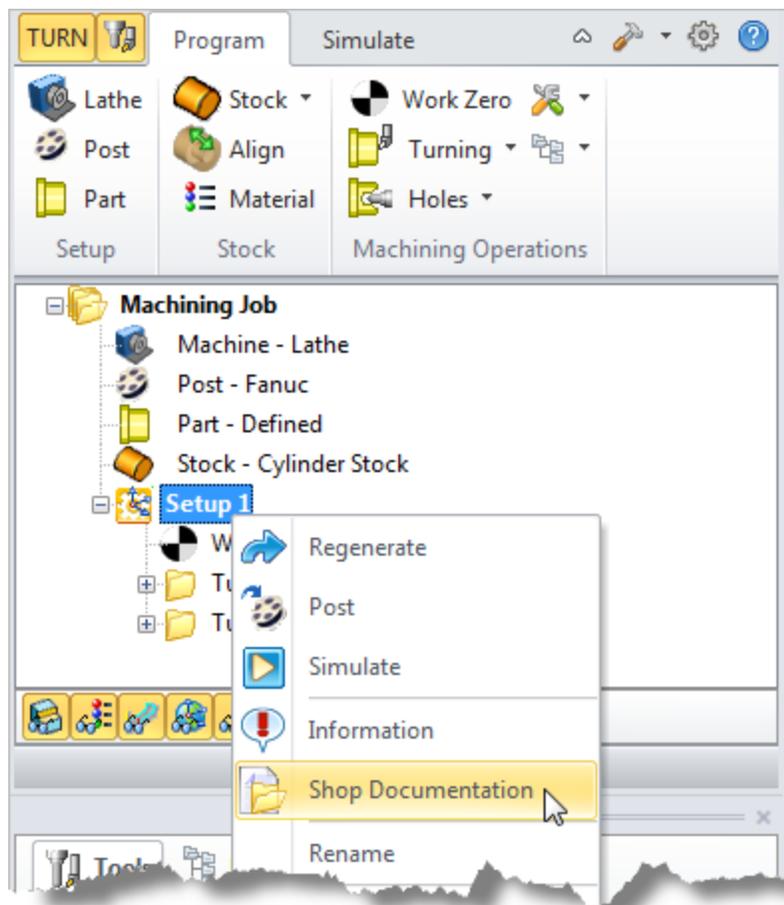
You can perform the same right-click sequence on the [Machining Job](#) to determine the estimated machining time for all [Setups](#).

3. Now pick [OK](#) to close the [Information](#) dialog.

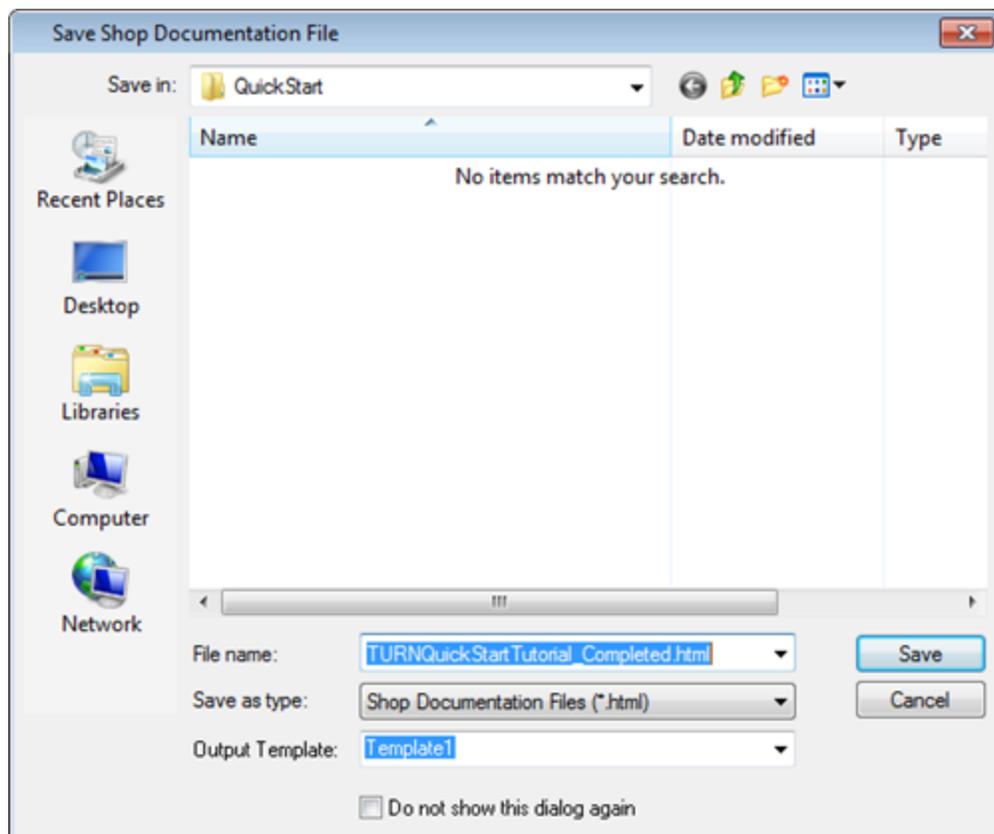
7.2 Shop Documentation

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.

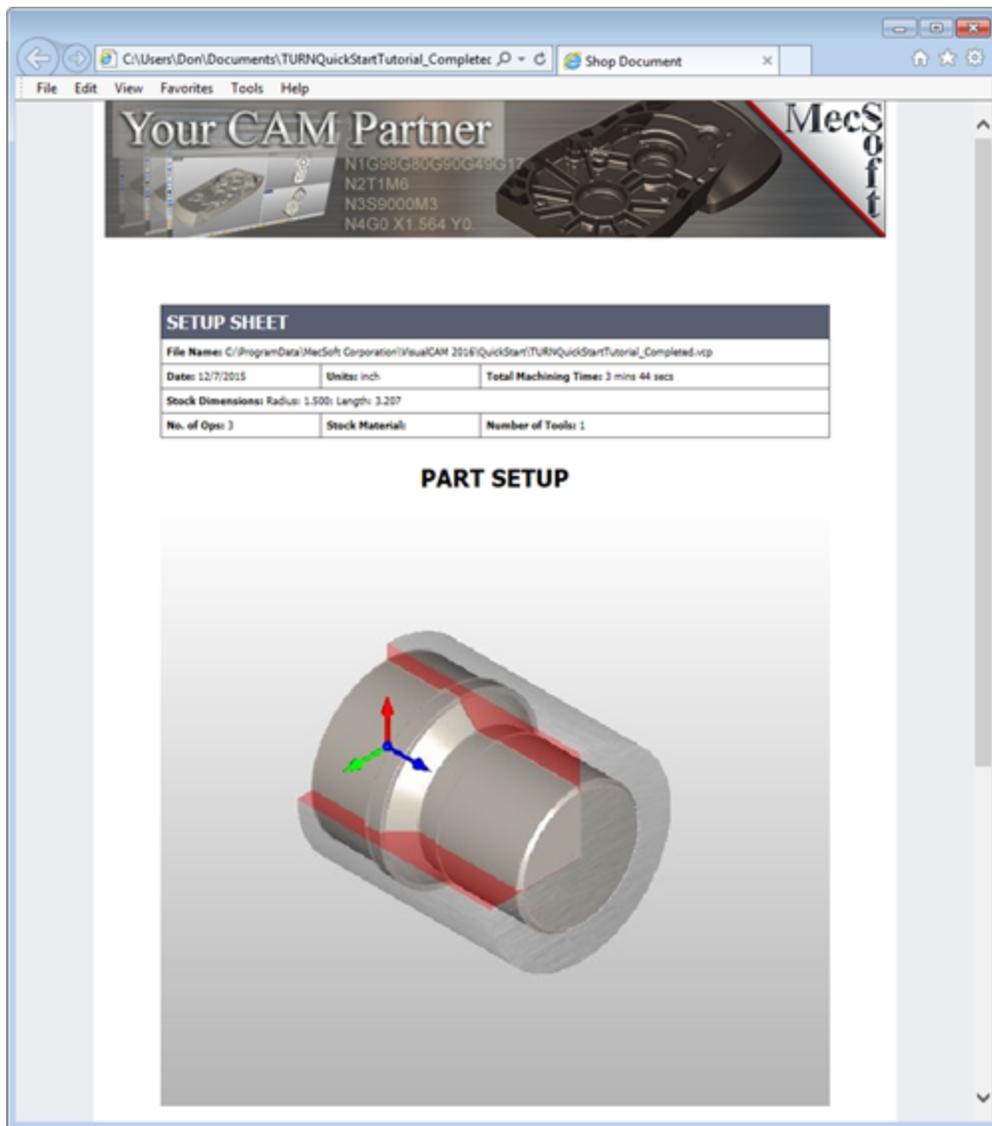
1. Select [Setup1](#) under the [Machining Job](#) tree in the [Machining Browser](#).
2. Right-click and select [Shop Documentation](#).



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

If you need additional help please take advantage of the following MecSoft resources:

1. **On-Line Help**

The on-line help distributed with the product is a great resource to find reference information on the various functions available

2. **MecSoft.com**

Apart from the on-line help system you can download other tutorials and projects from MecSoft Corporation's web site at www.mecsoft.com. This will help you get started with using VisualCAM 2018.

3. **Free Videos**

You can visit the [MecSoft Corporation YouTube Channel](#) to watch videos. Note that the functionality of MecSoft's CAM products is very similar across the different platforms that we support!

4. **MecSoft Blog**

You can visit the [MecSoft Blog](#) for short articles about using our products.

5. **Case Studies**

You can also visit our *real-world* [Case Studies page](#) to learn how others are using MecSoft products in their workshops.

6. **CAMJam Video Archive**

If you are an active AMS (Annual Maintenance Subscription) user, you have free access to our [CAMJam self-training video archive](#) and companion guide containing over 80 videos from our support staff on every aspect of VisualCAM 2018. If you are new or have recently signed up for AMS, [this document will show you how to access your CAMJam archive](#). Want to sign up for AMS? Just give a call at 949-654-8163 (select Option 1 for Sales).

7. **MecSoft Support**

If you need additional help, or if you have any questions regarding VisualCAM 2018, you may contact us via e-mail at support@mecsoft.com or our [online support page](#).

8. **On-Demand Training**

MecSoft offers [On-Demand Training](#) as well as personalized full day training sessions. Please look up our website or email us at sales@mecsoft.com for further details

9. **Product Page**

Please do continue to [visit the VisualCAM 2018 product page](#) to learn about the latest updates and additional help material.

Index

- A -

About the TURN Module 4
Align Part and Stock 27

- C -

Clearance Geometry 45, 59
Create Stock Geometry 23
Create Tools 37

- D -

Define the Machine Tool to use 15
Define the Part Geometry 20

- E -

Estimated Machining Time 70

- F -

Finishing the Outer Diameter 59

- L -

Launching the VisualCAM 2018 TURN Module 8
Loading the Part Model 12

- M -

Machining Information 70
Machining Strategy 14
Main Programming Steps 15

- P -

Post Processing 67

- R -

Reports
 Estimated Machining Time 70
 Machining Information 70
 Setup Sheet 71
 Shop Documentation 71
Roughing the Outer Diameter 41
Running VisualCAM 2018 7

- S -

Select Cutting Tool 42, 59
Select the Post Processor to use 16
Set Feeds and Speeds 44, 59
Set Work Zero 33
Setup 32
Setup Sheet 71
Shop Documentation 71
Simulate the Toolpath 65
Simulate Toolpath 53
Specify Entry/Exit Parameters 49
Specify Global Parameters 47, 59
Specify Material 29

- T -

Turn Roughing 48

- U -

Useful Tips 5
Using this Guide and Associated Part Files 4

- W -

Where to go for more help 75