

TURN Quick Start Guide

RhinoCAM 2024

Published: February 2024



MecSoft Corporation

© Copyright 1998-2020

Table of Contents

Quick Start	5
Resource Guide	8
About this Guide	9
1 Useful Tips.....	9
2 About the TURN Module.....	9
3 Using this Guide.....	10
4 Watch the Video!.....	11
Getting Ready	12
1 Running RhinoCAM 2024.....	12
2 About the Rhino Display.....	12
3 Launch the TURN Module.....	13
4 Load the Part Model.....	16
5 Machining Strategy.....	18
6 Main Programming Steps.....	19
7 Define the Machine Tool.....	19
8 Select the Post Processor.....	20
Defining the Machine Setup	23
1 Define the Part Geometry.....	23
2 Create the Stock Geometry.....	27
3 Align the Part and Stock.....	29
4 Specify the Material.....	31
5 The Machining Setup.....	33
6 Set the Work Zero.....	34
Create a Cutting Tool	38
Outer Diameter Turn Roughing	42
1 Select the Cutting Tool.....	43
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
Outer Diameter Turn Finishing	58
1 Set Finishing Parameters.....	58
2 Simulate the Toolpath.....	64
3 Post-Process the Toolpath.....	66
Generate Reports	69
1 Machining Information.....	69
2 Shop Documentation.....	70
Where to go for more help	73
Index	75

Quick Start

RHINOCAM 2024



[Prefer Printed Documentation? Check Here!](#)

[Quick Start Guides](#) for each [RhinoCAM 2024](#) 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 RhinoCAM 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

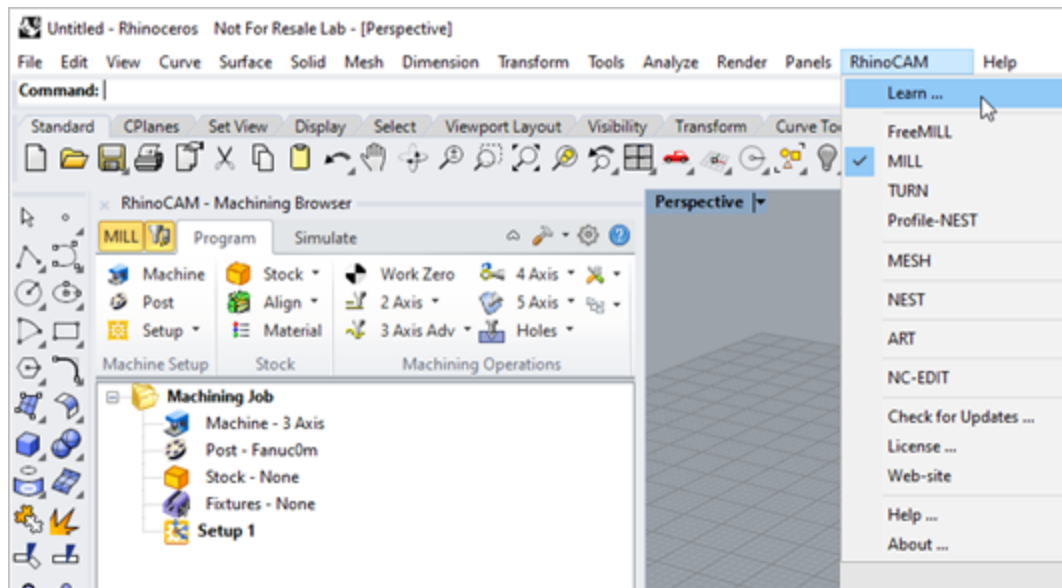
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM 2024 Learning Resources](#) dialog.

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:

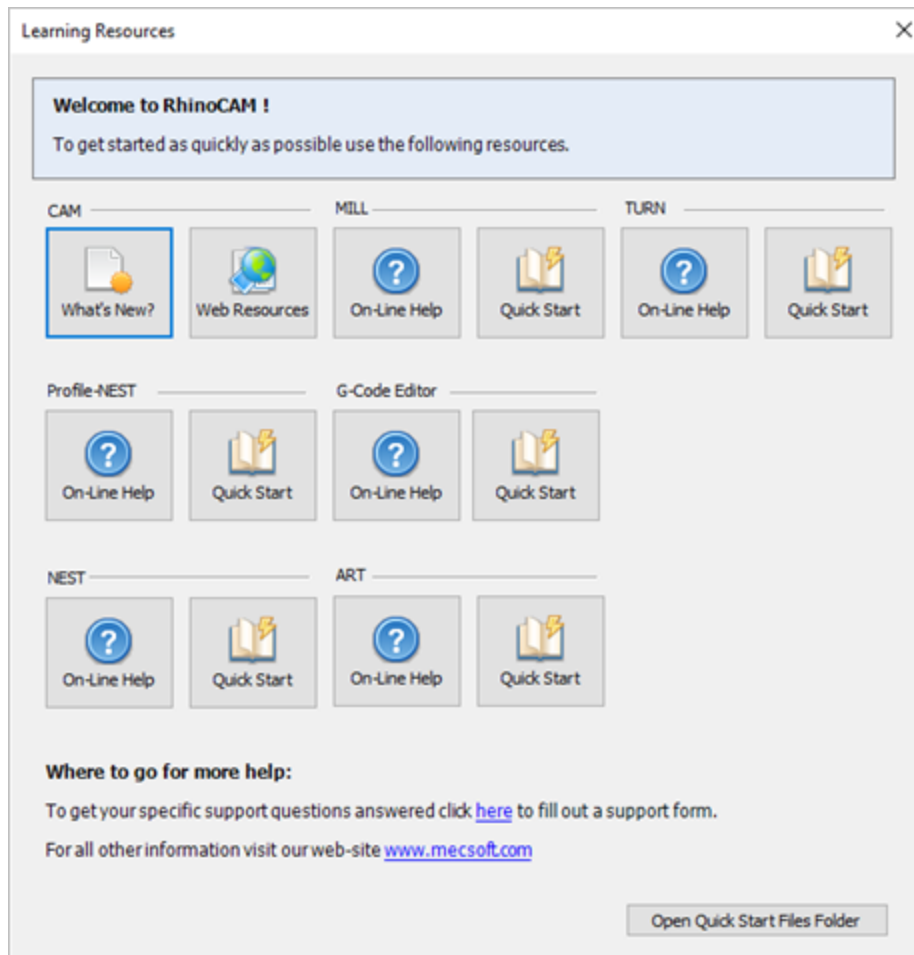
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog in RhinoCAM

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.



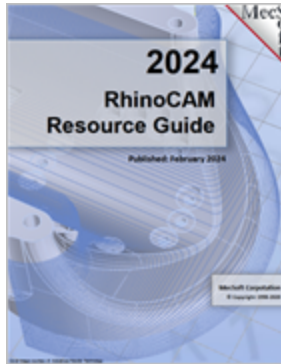
Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available [RhinoCAM 2024 Resources](#).



2024 RhinoCAM 2024 Resource Guide



The 2024 RhinoCAM 2024 Resource Guide!

18 Pages

Lists PDF downloads and Online resources including [Quick Start Guides](#), [Reference Guides](#), [Exercise Guides](#), [Tutorials](#) and [More](#).

[Prefer Printed Documentation? Check Here!](#)

About this Guide

3.1 Useful Tips

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

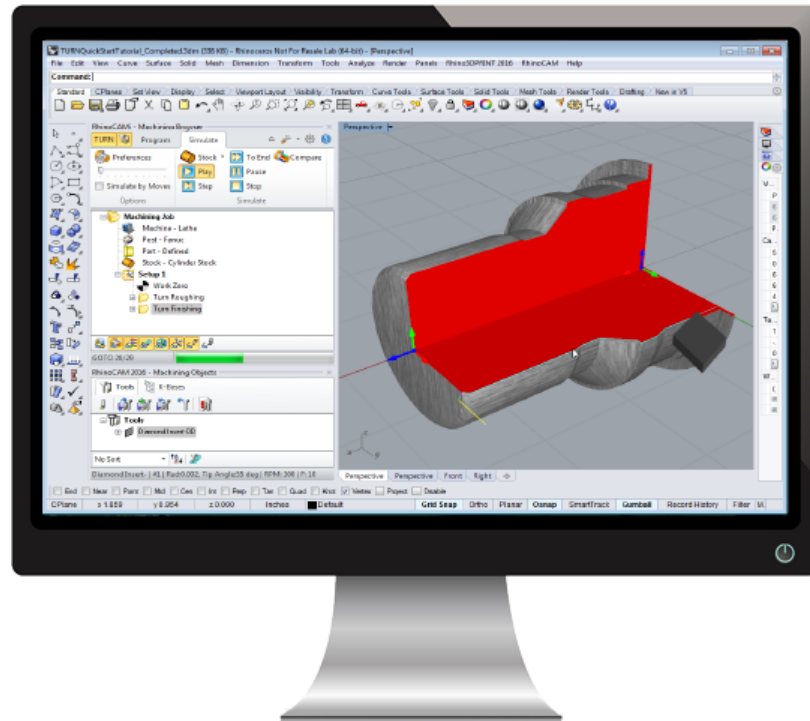
1. For purposes of brevity, [Rhino](#) refers to both [Rhinoceros 6](#) or [Rhino 7](#).
2. Copy the tutorial part files in a location other than the installation folder to make sure you have read/write privileges to the files.
3. Once you start working with the tutorial file, save your work periodically!
4. 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.
5. Most of all have fun!

3.2 About the TURN Module

The [RhinoCAM 2024 TURN](#) module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities for programming 2 Axis CNC Lathes, running inside [Rhino](#). This integration allows for seamless generation of toolpath and cut material simulation/verification inside [Rhino](#) 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 [Rhino](#) data as well as any of the data and file types that can be imported into [Rhino](#) for solids, surfaces and mesh objects. Then you can use the [RhinoCAM 2024 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.



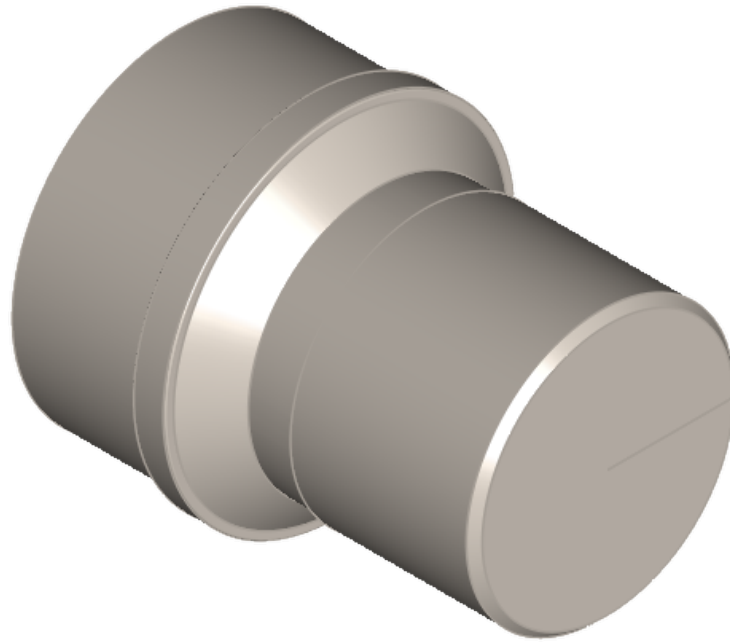
RhinoCAM's TURN Module Quick Start Guide

3.3 Using this Guide

If you have installed [RhinoCAM 2024](#) 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 [RhinoCAM 2024 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 [Rhino](#) 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.3dm

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 RhinoCAM 2024

Locate the [RhinoCeros 6](#) (or [RhinoCeros 7](#)) 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 [RhinoCeros 6](#) (or [RhinoCeros 7](#)). (The name of this program group will usually be called [RhinoCeros](#), unless you specified otherwise during setup.)

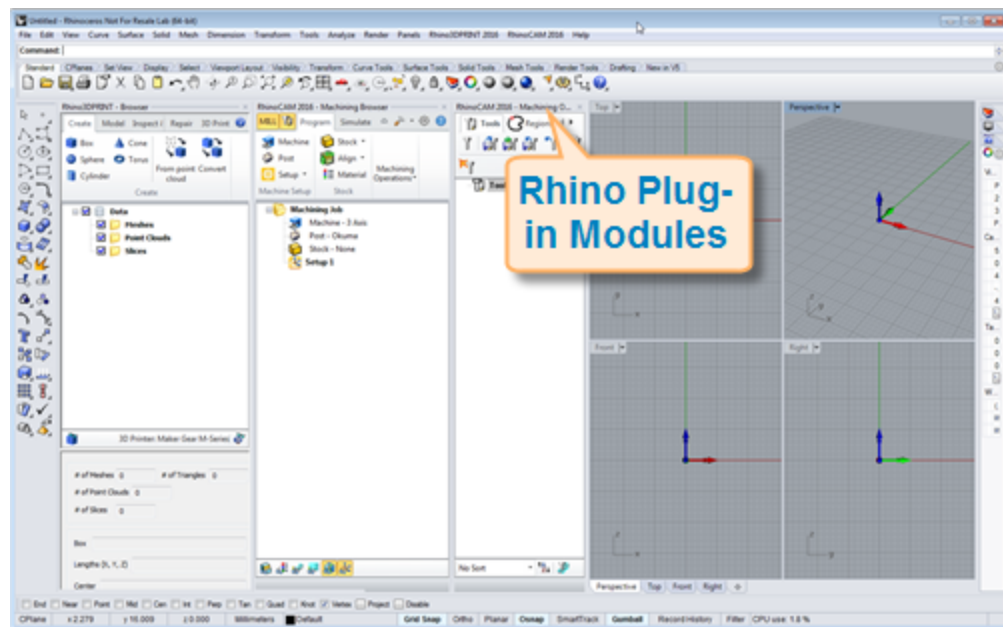
Once you locate the program group, select it and then select [RhinoCeros](#) to launch the application.

If the installation was successful, upon launching of [Rhino](#) you should observe a menu entry called [RhinoCAM 2023](#) in the main menu bar of [Rhino](#).

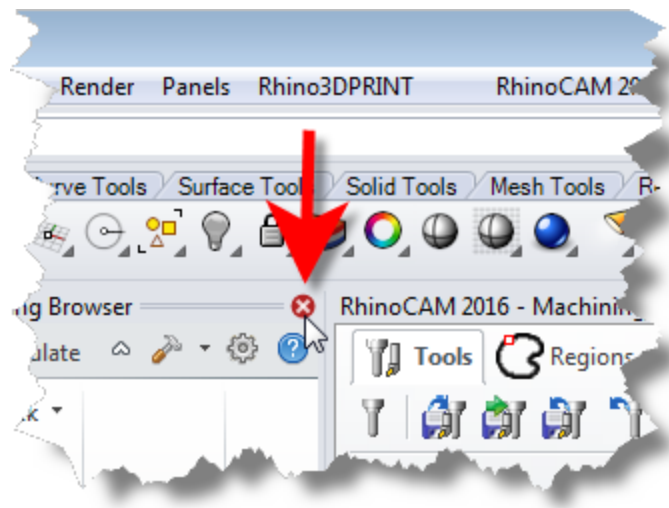
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the Rhino Display

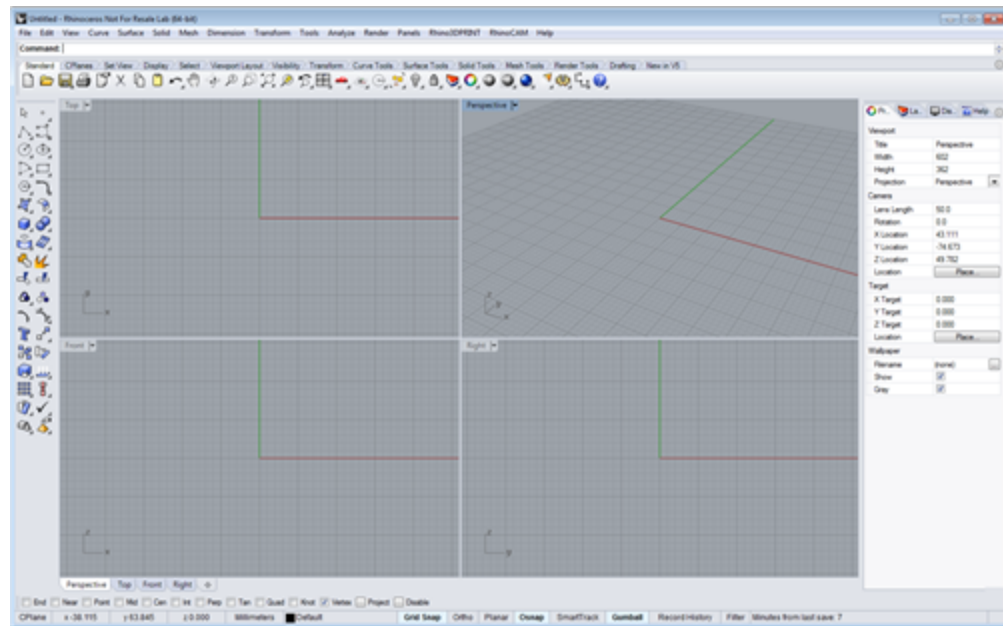
Before we begin, let's talk a bit about the [RhinoCAM](#) display. When you run [RhinoCAM](#) for the very first time, your screen may look this.



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



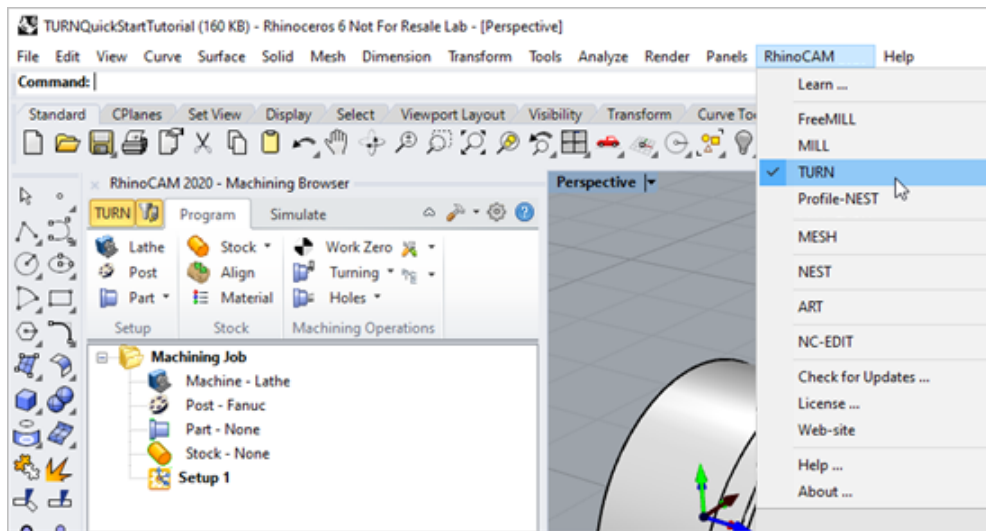
With all plug-in modules closed your screen will look like this:



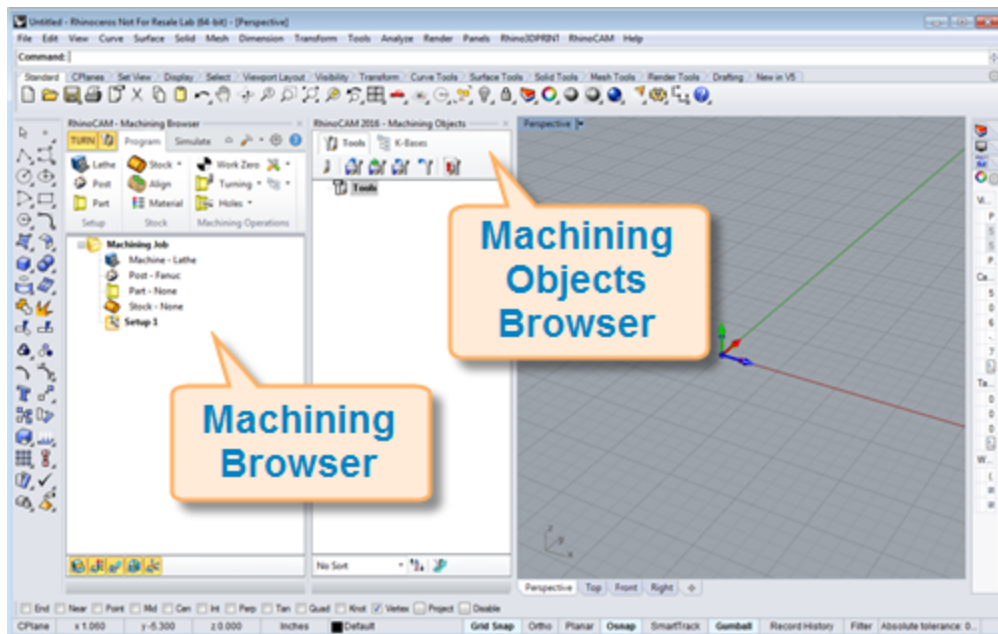
4.3 Launch the TURN Module

Now, let's begin by launching the [RhinoCAM 2024 TURN](#) module.

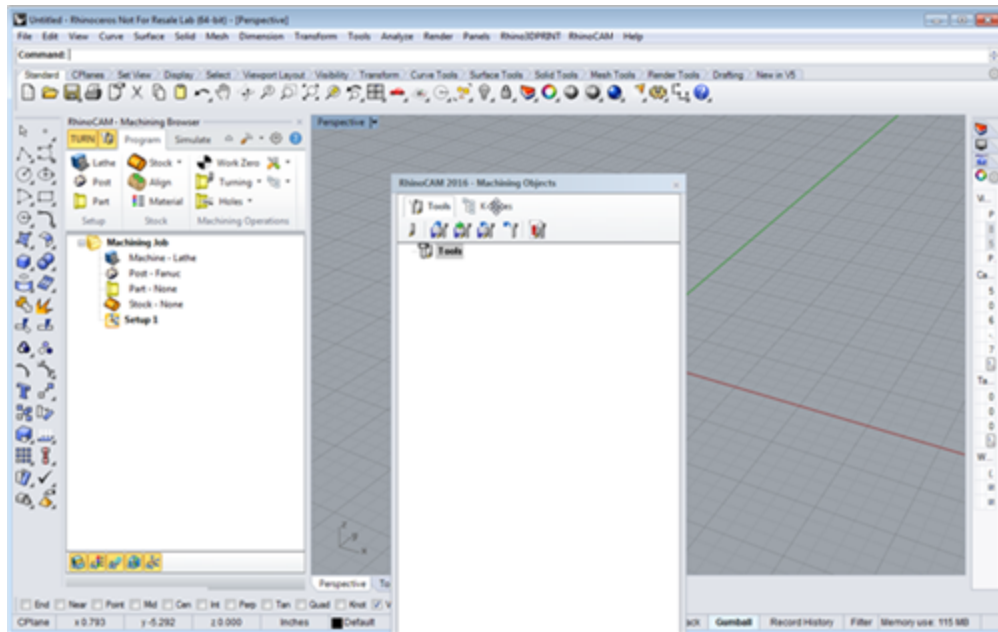
1. From the [Rhino](#) main menu bar, you will see the [RhinoCAM](#) menu item.
2. Drop-down the menu and pick [TURN](#) to load the [TURN](#) module.



3. Docked on the left you will see the [Machining Browser](#) and the [Machining Objects Browser](#). When you first run [RhinoCAM 2024](#), these two browsers may be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



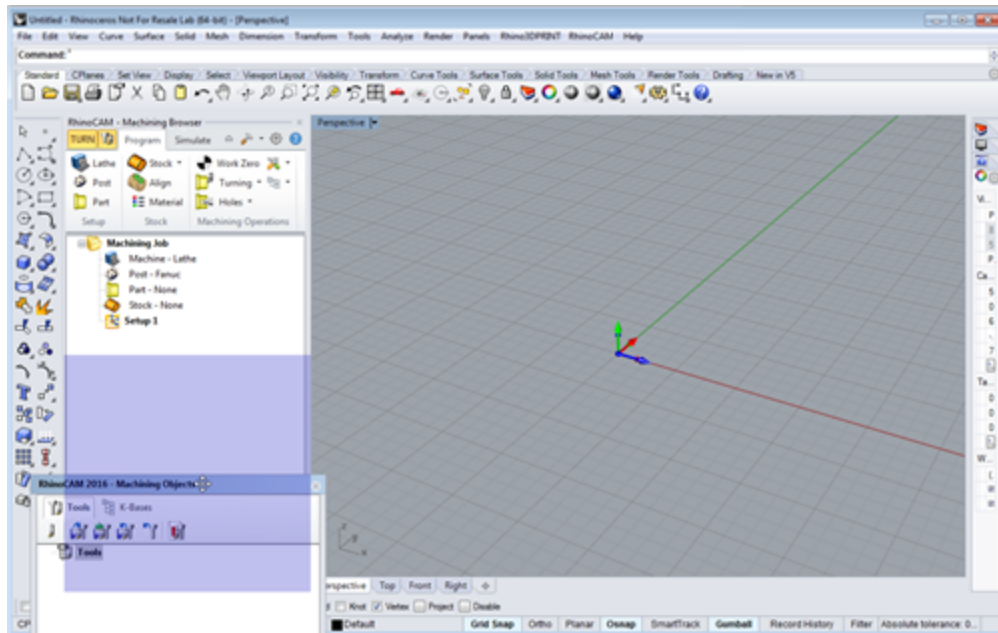
4. For example, let's move the [Machining Objects Browser](#) so that it displays under the [Machining Browser](#) on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



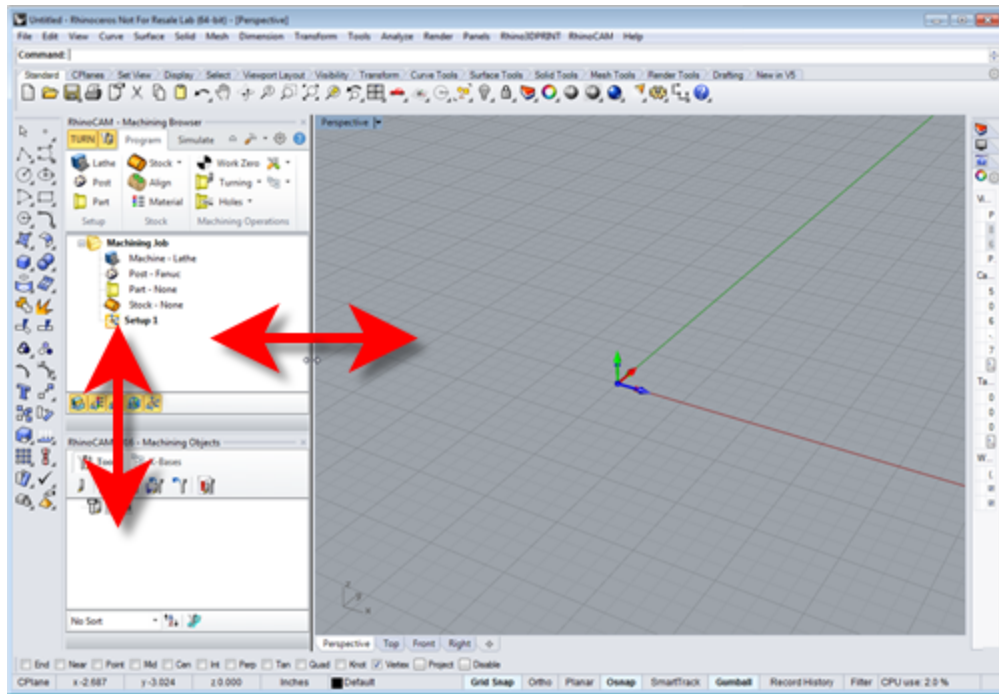
While doing so you will see possible docking location highlight on the display.

5. We'll drag the [Machining Objects Browser](#) over the base of the [Machining Browser](#) until the cursor activates the the bottom docking location as shown below.

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



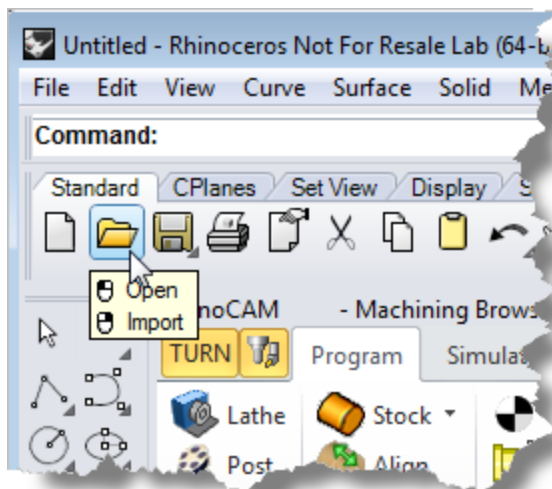
- You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



4.4 Load the Part Model


“Part” refers to the geometry that represents the final manufactured product. You can create parts within [Rhinceros](#) or import geometry created in another [CAD](#) system.

- Select **File / Open** from the **Main Menu** bar or click the **Open** icon from the **Standard** bar.

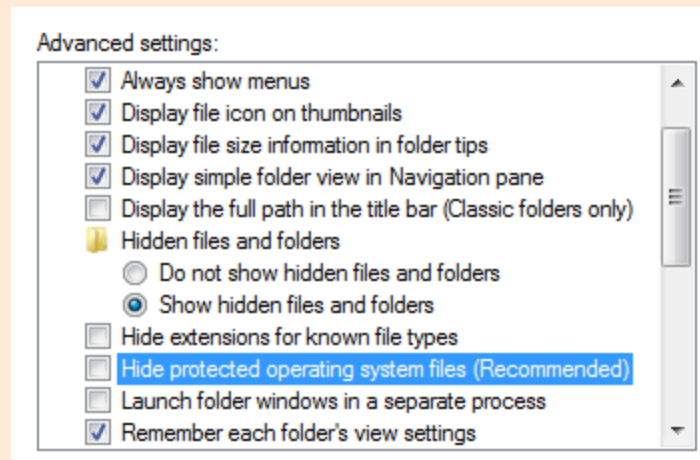


- From the **Open** dialog box, select the **TURNQuickStartTutorial.3dm** file from the **C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart** folder.

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.

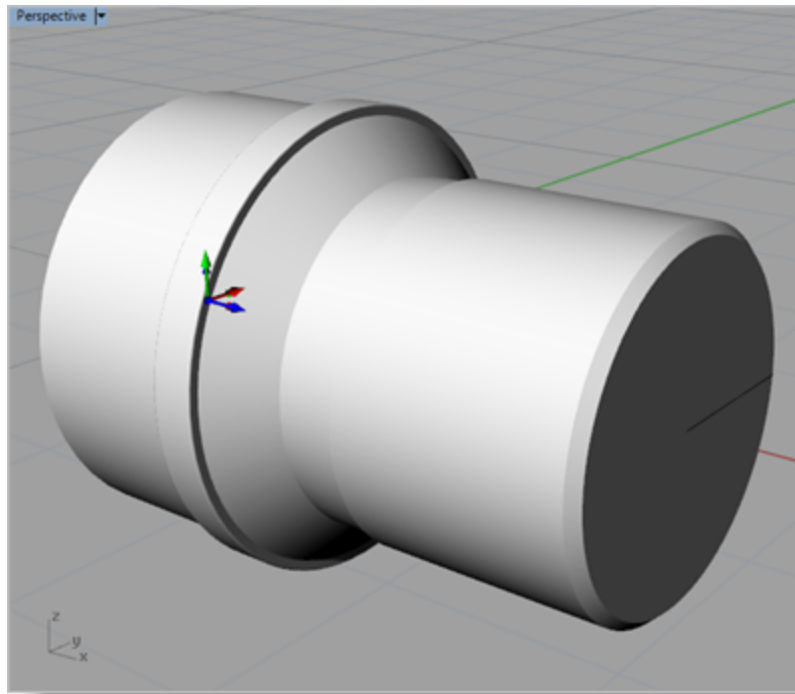
 By default, the [ProgramData](#) folder is "hidden" from view. Here are the steps to Show hidden files and folders:

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](#).

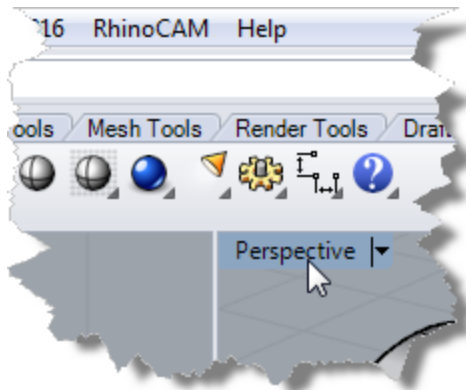
The part appears as shown below



MILLQuickStartTutorial.3dm

 You can import 2D drawings, Solid, Surface and Mesh models that are supported in Rhinoceros.

3. From the **Rhino** display, **double-left-click** on the **Perspective View** tab to maximize it.



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

4.6 Main Programming Steps

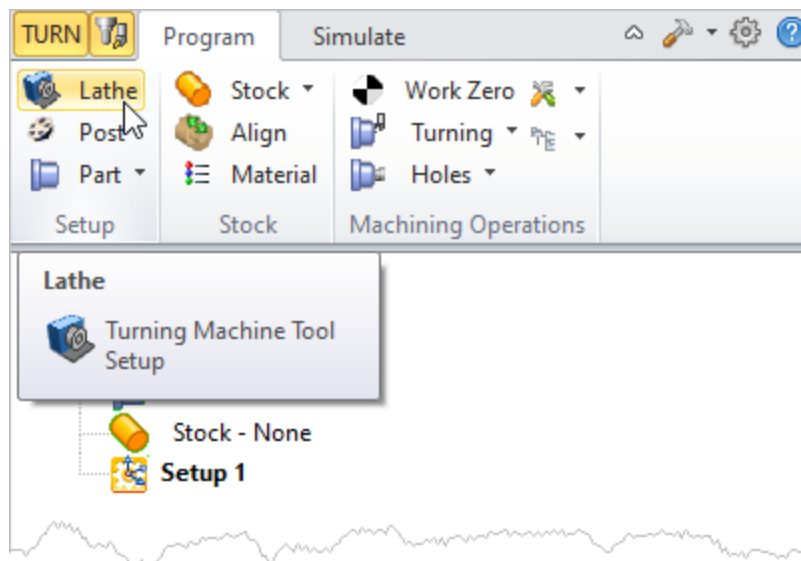
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

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.

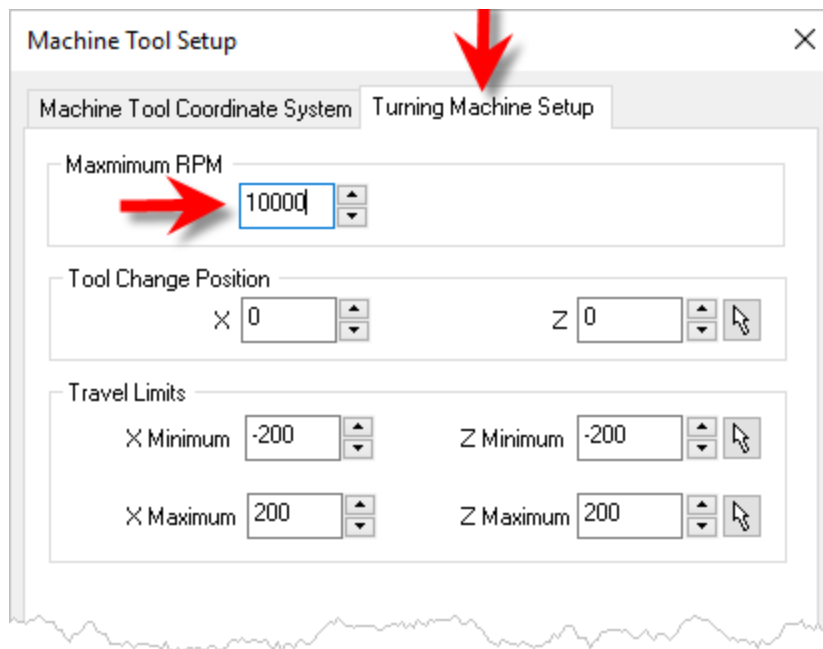
4.7 Define the Machine Tool

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

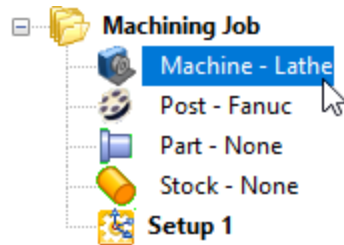
1. From the **Program** Tab select **Lathe** to display the dialog box.



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



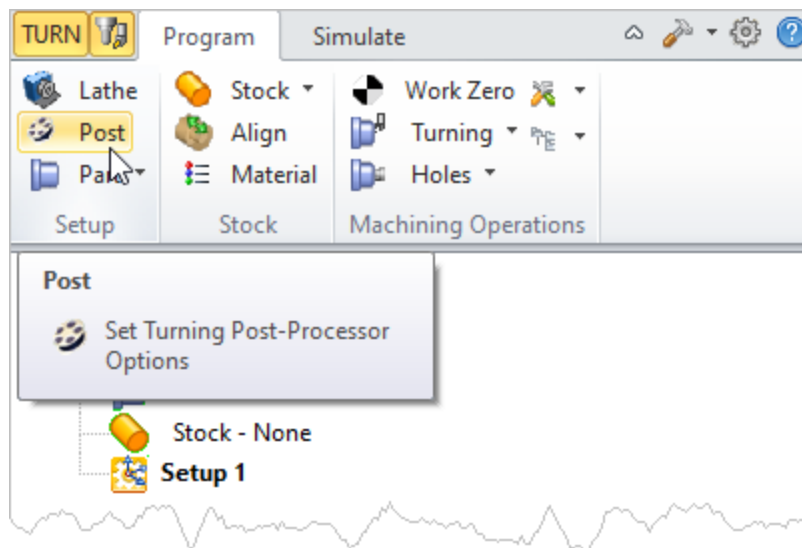
3. Pick OK and notice that the **Machine - Lathe** now appears under **Machining Job** in the **Machining Browser**.



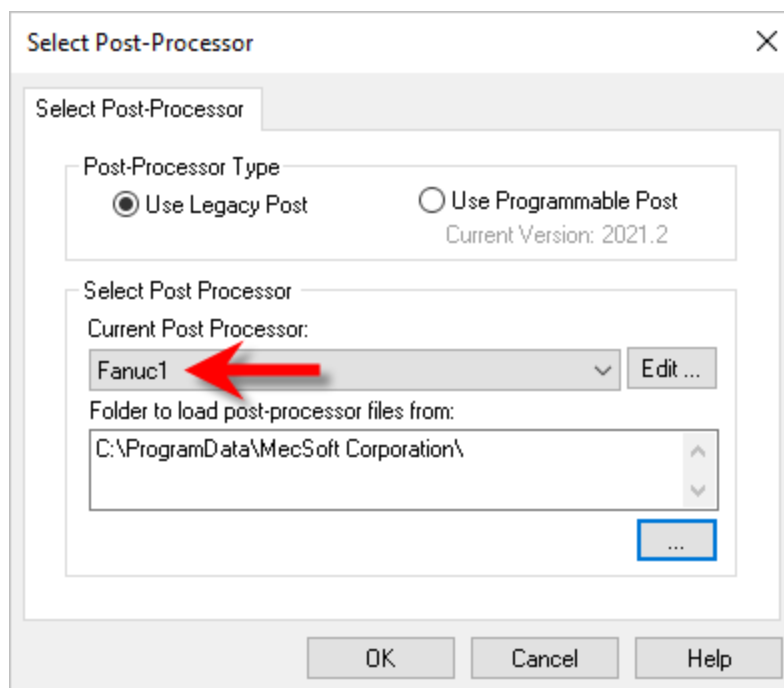
4.8 Select the Post Processor

Next, we'll define the **Post Processor**.

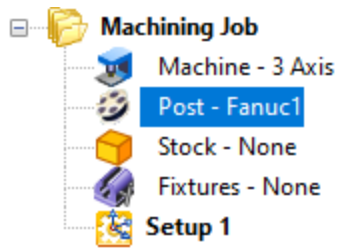
1. From the **Program** tab select **Post** to display the dialog.



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](#).

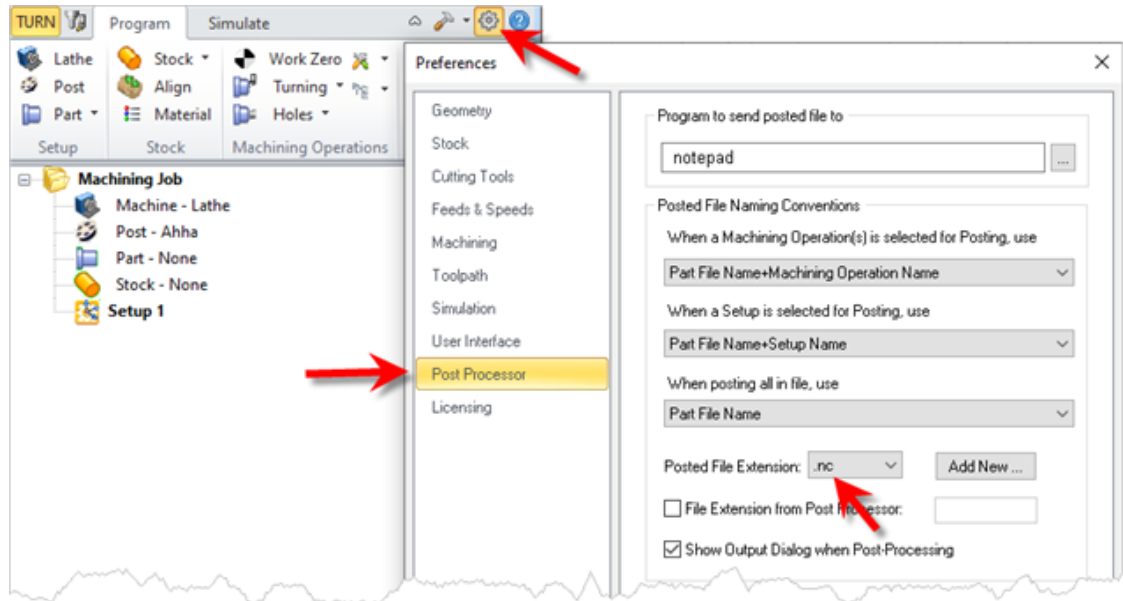


! By default, post processor files are located under:

C:\ProgramData\MecSoft Corporation\Rhinoceros x.x\Plugins\RhinoCAM 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.



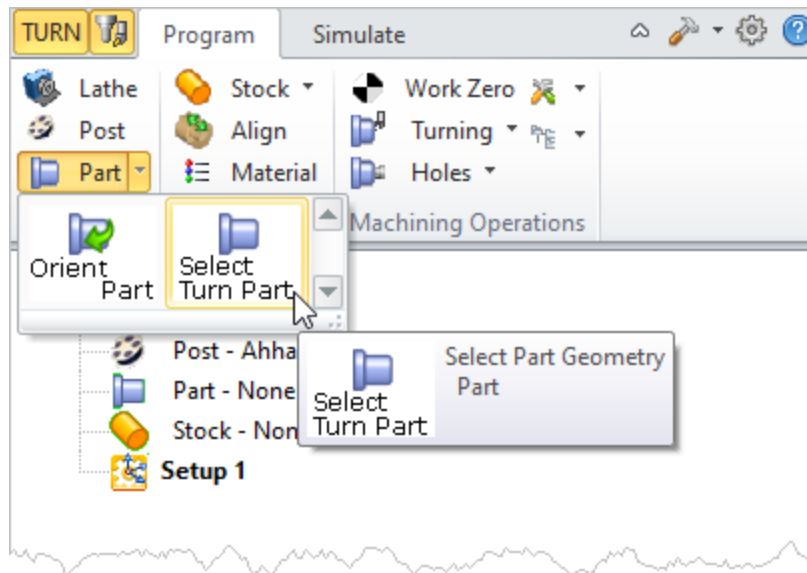
For **Post File Extension** select **.nc** from the dropdown list. If you need a different extension, pick the **Add New** button and enter your file extension and pick **OK**. The posted file extension looks like this: **my-gcode-file.nc**

Defining the Machine Setup

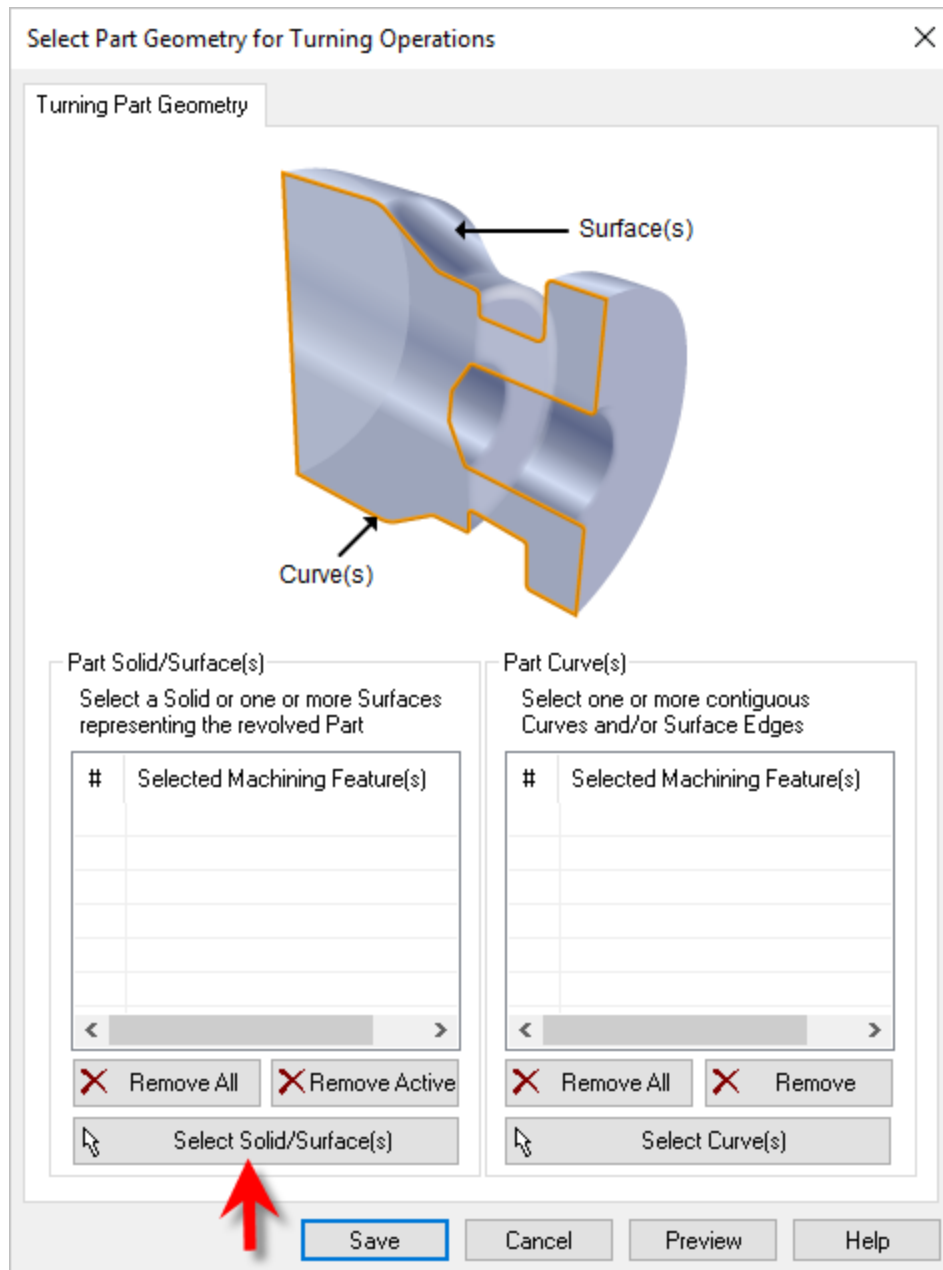
5.1 Define the Part Geometry

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.

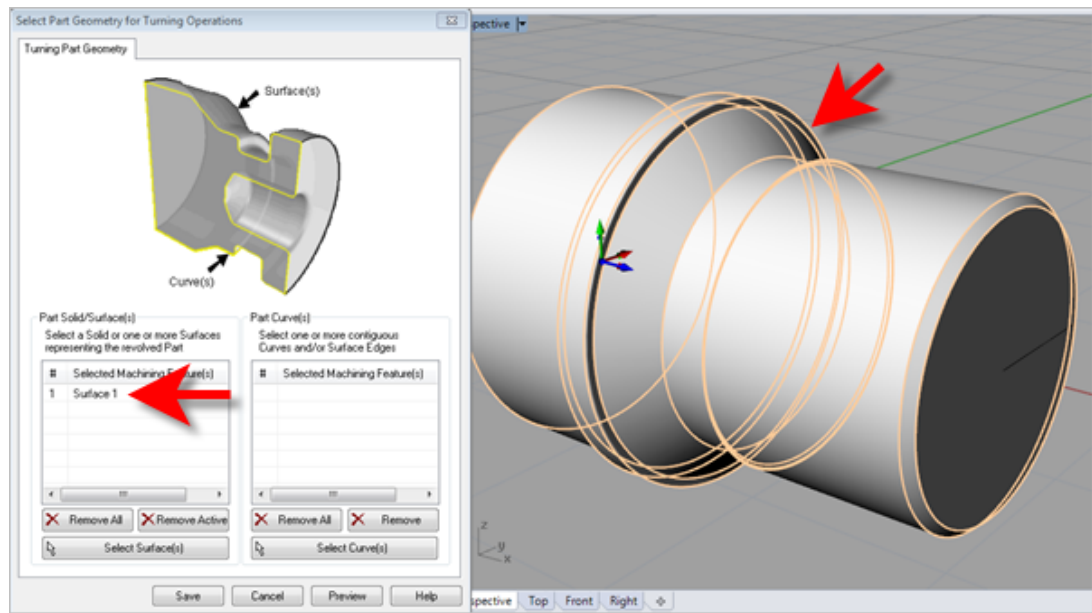
1. From the **Program** tab select **Part** and then **Select Turn Part** from the menu to display the dialog.



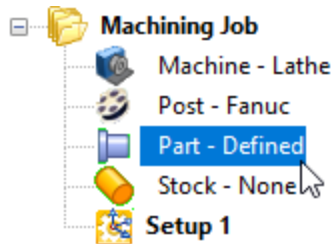
2. Pick the **Select Solid/Surfaces(s)** button.



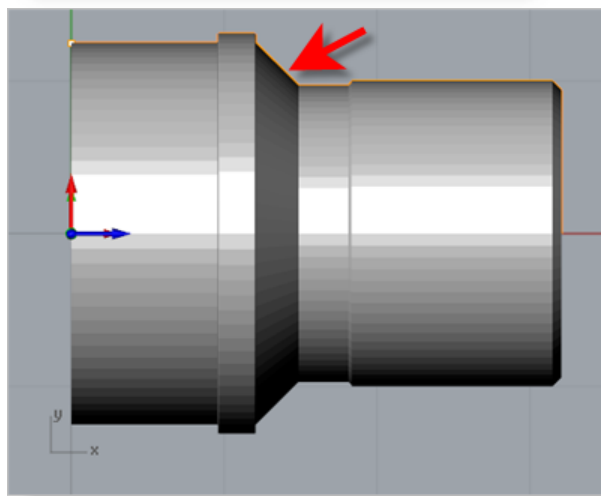
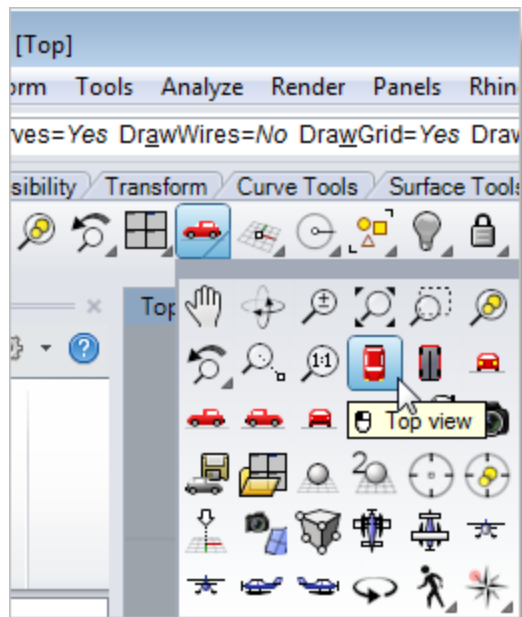
3. The dialog is minimized and allows us to select **Part Objects**.
4. Select the 3D solid and then **right-click** or press **Enter** to accept the selection.
5. The dialog reappears and lists the selected surfaces under **Selected Machining Features**.



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.



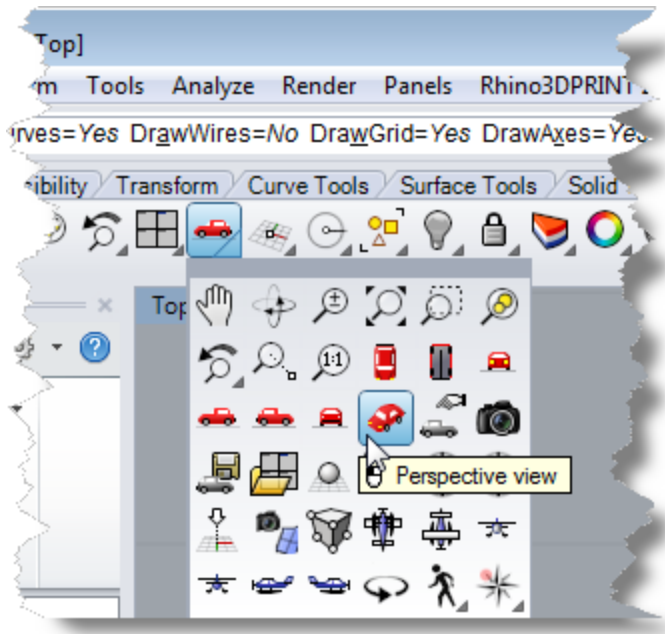
! In the future, if surfaces are selected as part geometry, the system will slice the selected surfaces with the ZX plane of the **Machine Coordinate System (MCS)** and use the resultant curves as the actual profile to be machined on the lathe.

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**.

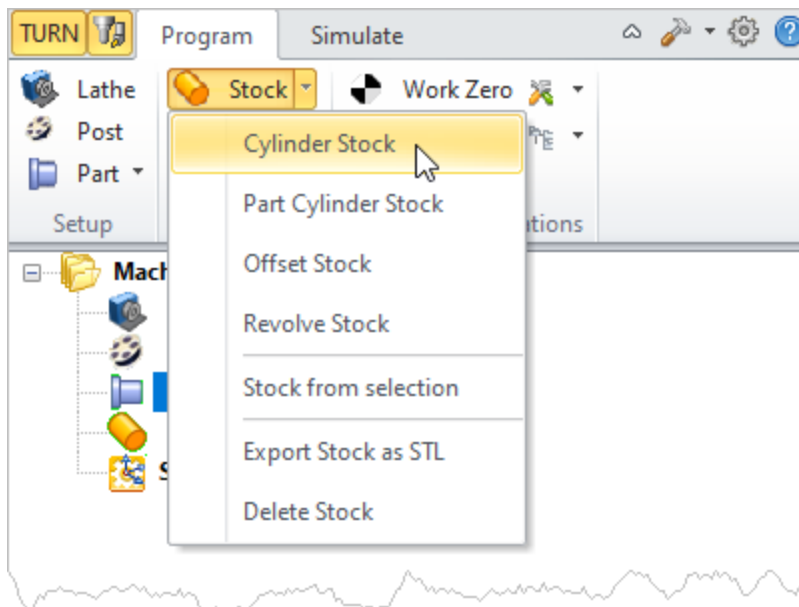
5.2 Create the Stock Geometry

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

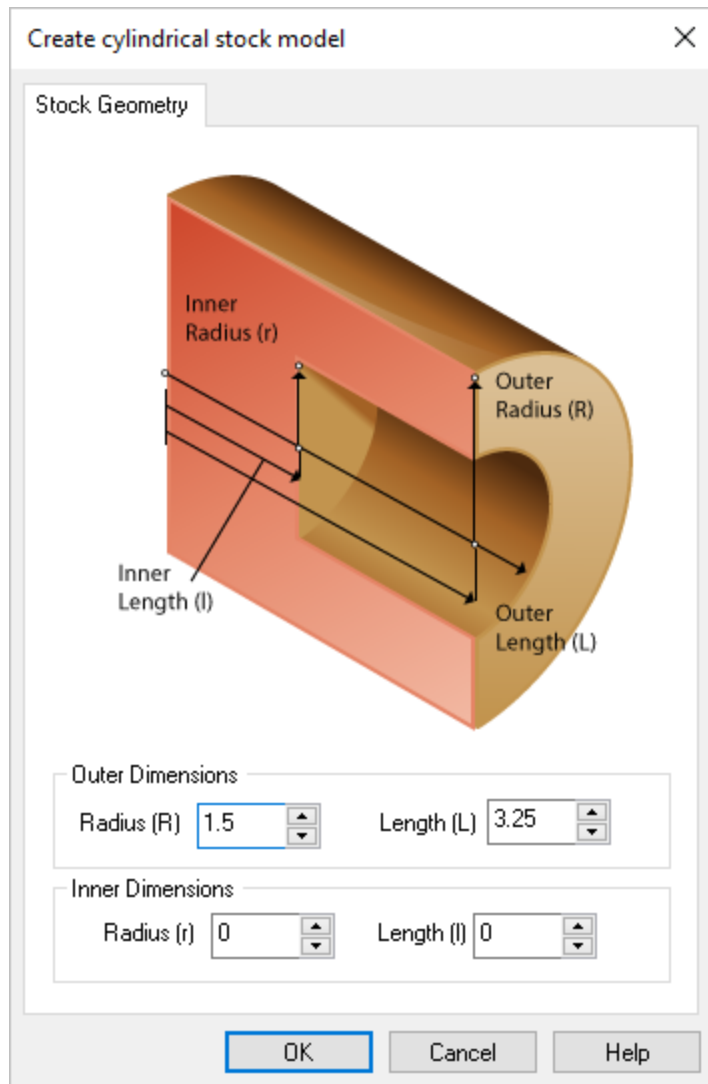
1. First, switch back to the [Perspective View](#).



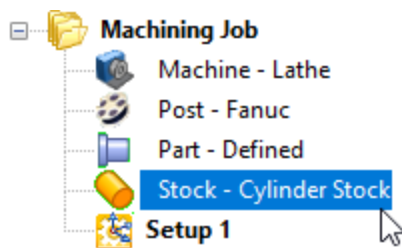
2. From the [Program](#) tab select [Stock](#) and then select [Cylinder Stock](#) from the menu to display the dialog.



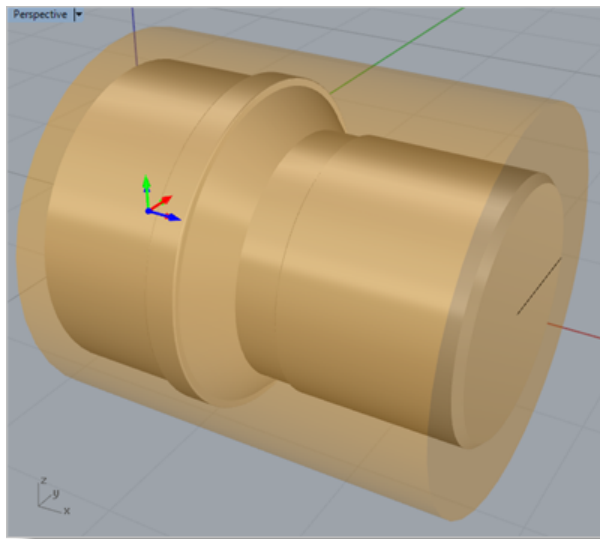
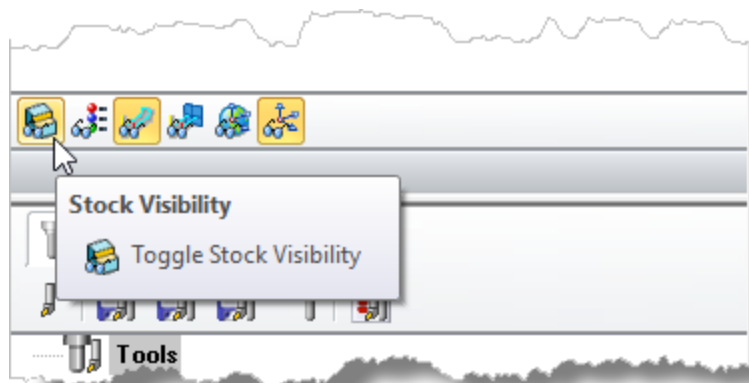
3. Under [Outer Dimensions](#), set [Radius \(R\)](#) to 1.5 and [Length \(L\)](#) to 3.25.



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



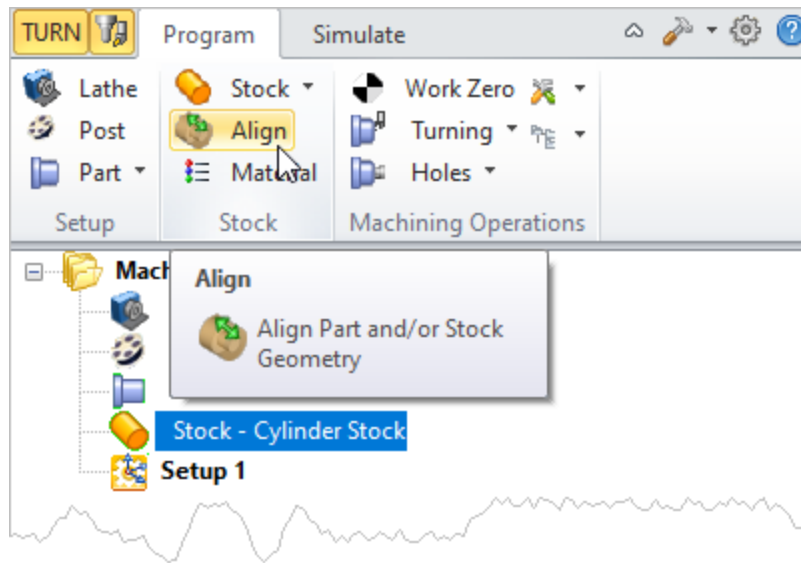
- If the stock does not display on the screen when selecting **Stock - Cylinder Stock** from the **Machining Browser**, select the **Stock Visibility** icon located at the base of the **Machining Browser**.



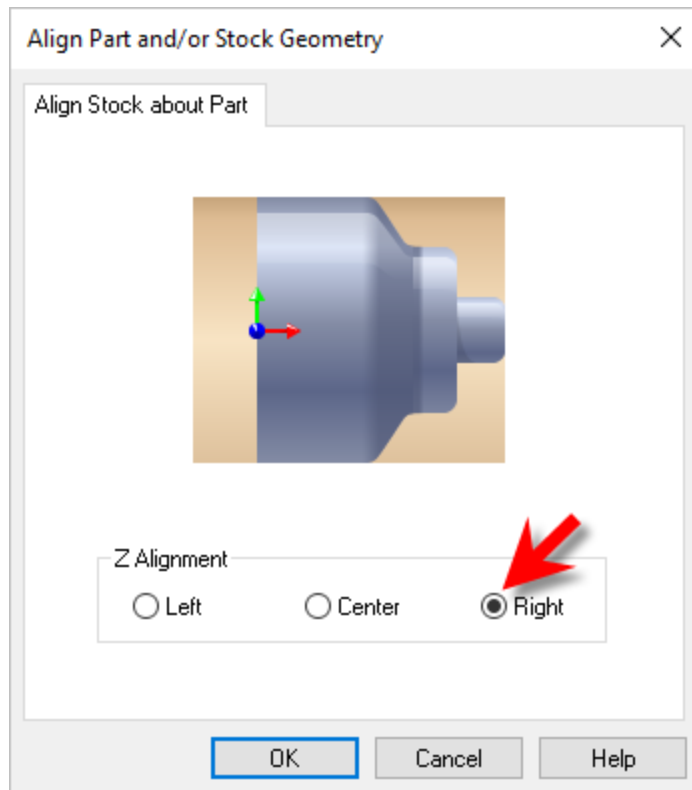
5.3 Align the Part and Stock

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

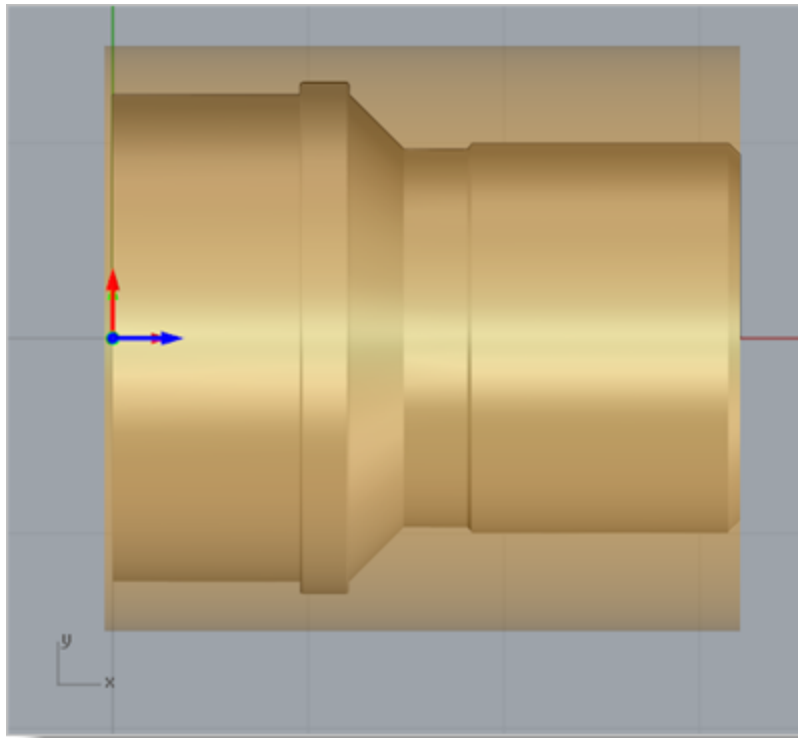
1. From the [Program](#) tab select [Align](#).



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



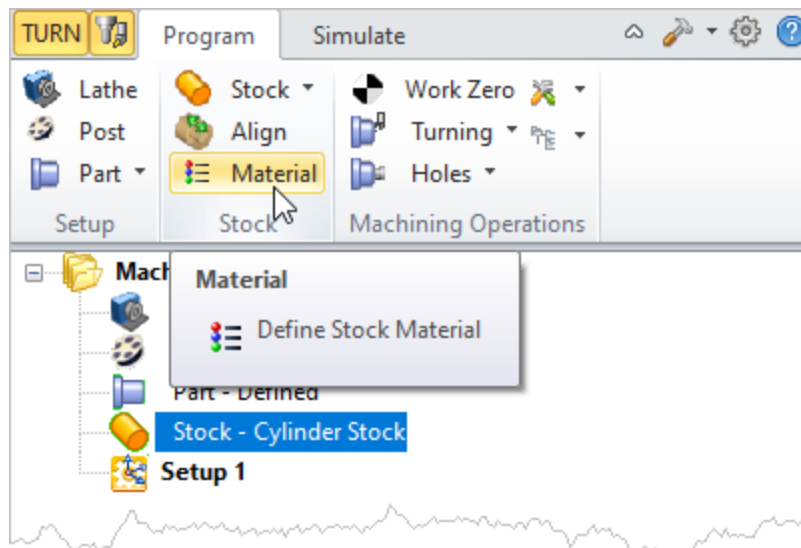
3. If you switch to the **Front View**, you see that the stock is now aligned to the right side face of the part geometry in the Z axis of the lathe.



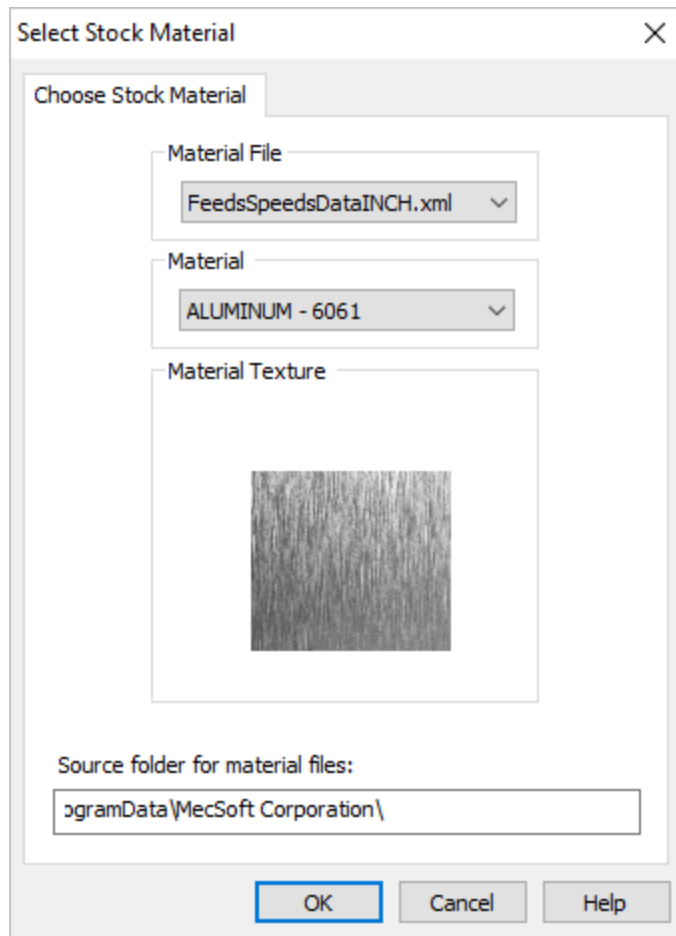
5.4 Specify the Material

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.

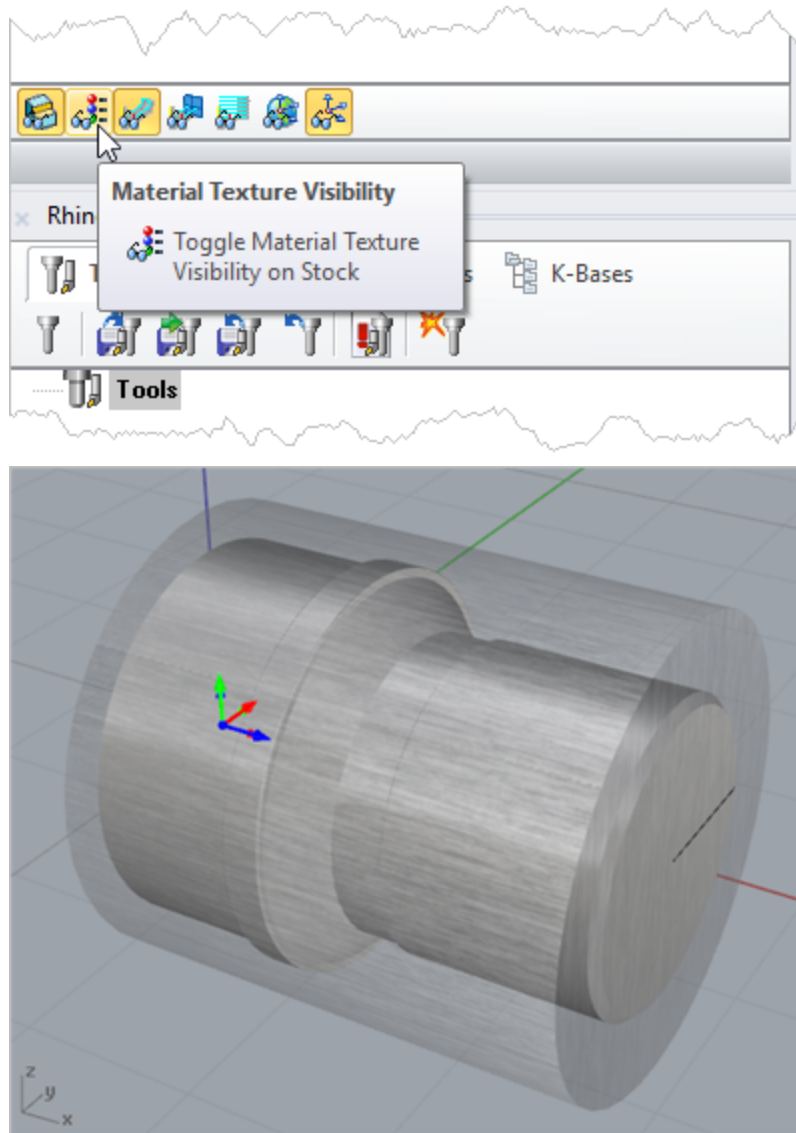
1. From the **Program** tab select **Material** to display the dialog.



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](#).

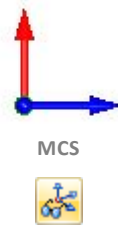
The [Setup](#) icon ( **Setup 1**) displayed in the [Machining Job](#) tree defines the [Turn Machine Coordinate System](#) or ([MCS](#)) and is defined automatically. CNC Turning centers or [Lathes](#) use the [Cartesian](#) coordinate system for programmed coordinates.

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



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	 MCS	 WCS	 WCS/MCS

! 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**.

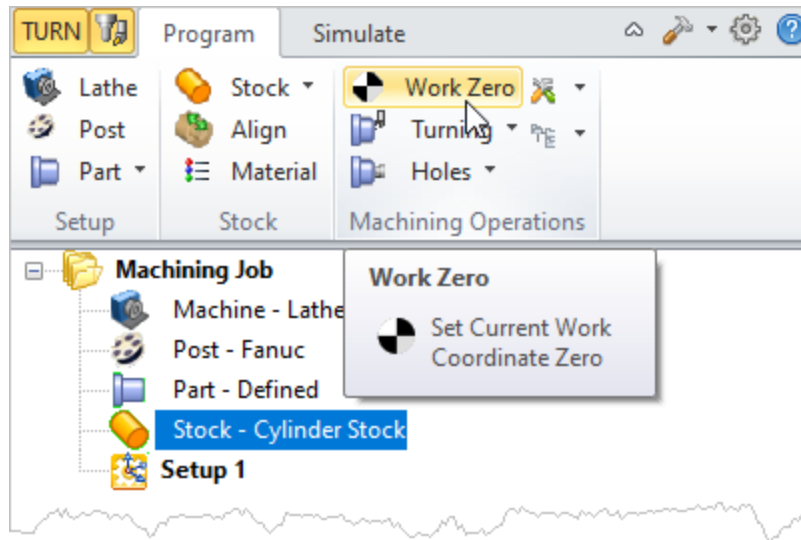
5.6 Set the Work Zero

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**.

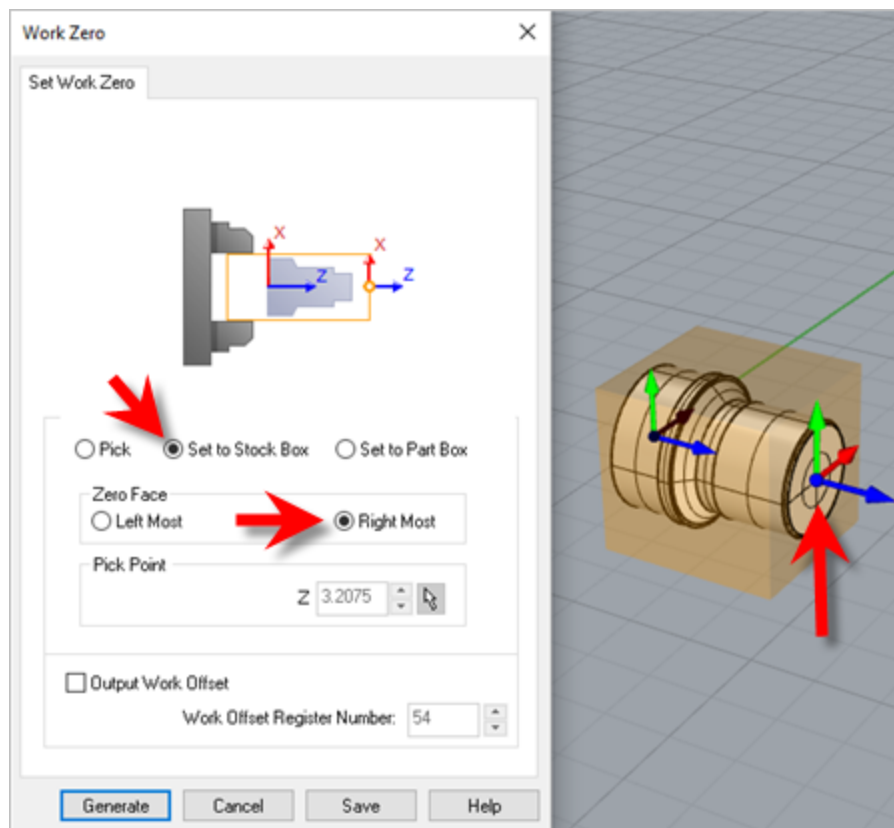
! It is important to understand that the **Work Zero** defines the zero point from which all toolpath points are interpreted by the controller. This would normally be the same as the tool touch off

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.

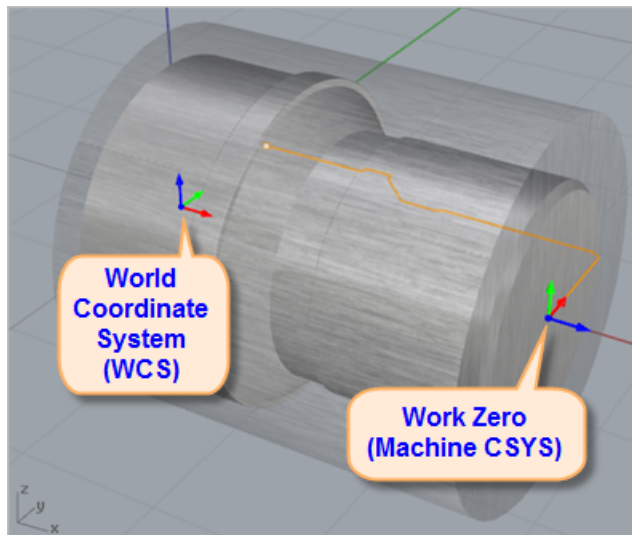
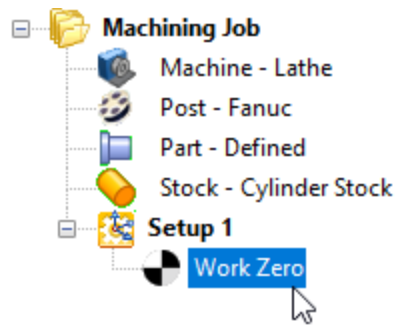
1. From the **Program** tab select **Work Zero** to display the dialog.




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](#).



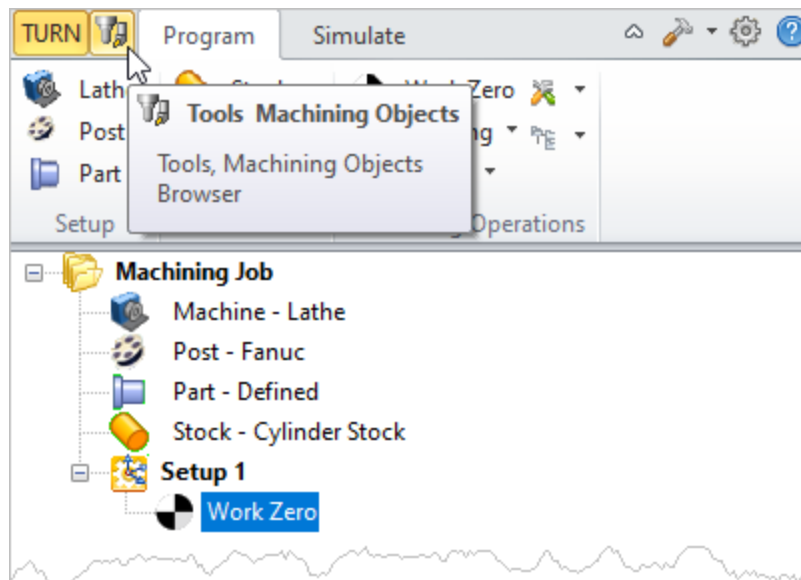
Note: When the Machine Coordinate System (Work Zero) is moved, the WCS is displayed in its place. If you do not see the WCS, select the Toggle World CSYS icon  to display it.

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.

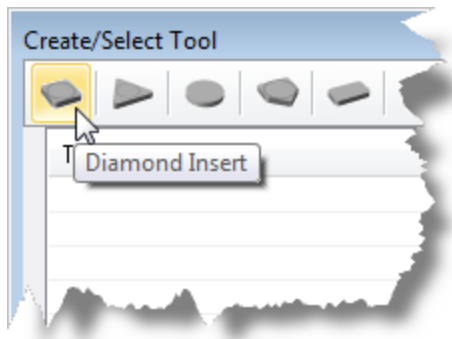
These buttons and icons are shown in the menus below.



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.



- Now set the following parameters in the dialog:

Name: Diamond Insert-OD

Inscribed Circle Radius: 0.5

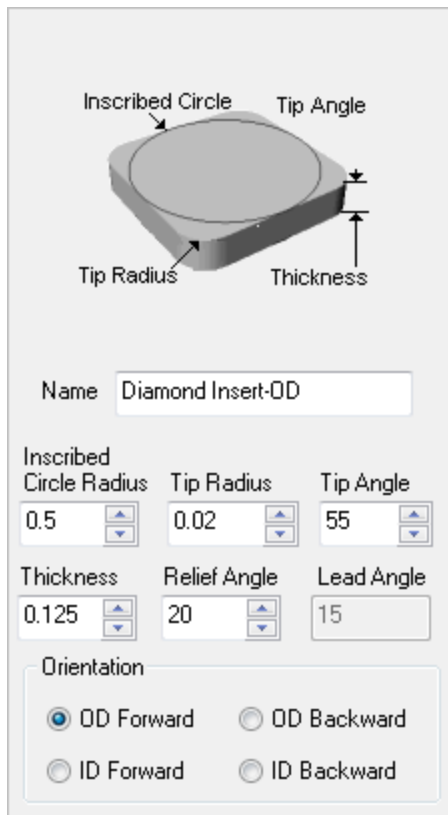
Tip Radius: 0.02

Tip Angle: 55

Relief Angle: 20

Thickness: 0.125

Orientation: OD Forward



- 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

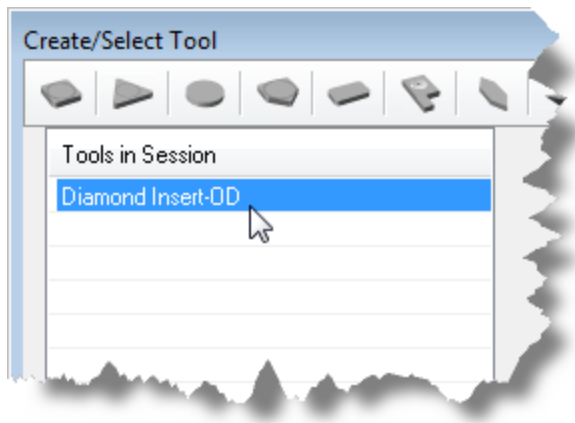
[Cut Depth](#): 0

The screenshot shows the 'Feeds & Speeds' dialog box in RhinoCAM 2024. It is divided into two main sections: 'Spindle Parameters' and 'Feed Rates (in/min)'.
In the 'Spindle Parameters' section, the 'Constant Surface Speed (CSS)' checkbox is unchecked. The 'Speed' is set to 300 RPM and the 'Max Speed' is set to 350 RPM. The 'Direction' is set to CW (Clockwise).
In the 'Feed Rates (in/min)' section, the 'IPM' (Inches Per Minute) radio button is selected. The 'Plunge' feed rate is 5, 'Approach' is 7.5, and 'Engage' is 7.5. The 'Cut' feed rate is 10, 'Retract' is 15, and 'Departure' is 15. The 'Transfer' section has the 'Use Rapid' radio button selected, with a numerical value of 13443. The 'Cut Depth' is set to 0 in. At the bottom of the dialog is a 'Load from File ...' button.

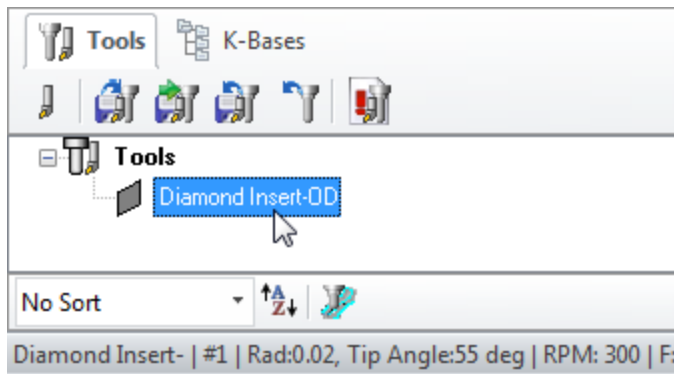
5. Now, we'll pick [Save as New Tool](#).

 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**.

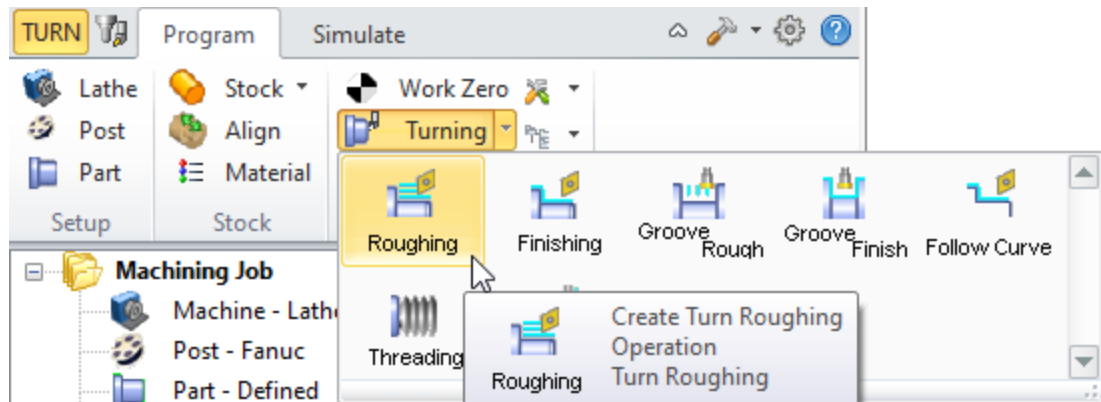


! In the future, to save Tools to a library, click **Save Tool library** under the **Tools** tab in **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box.

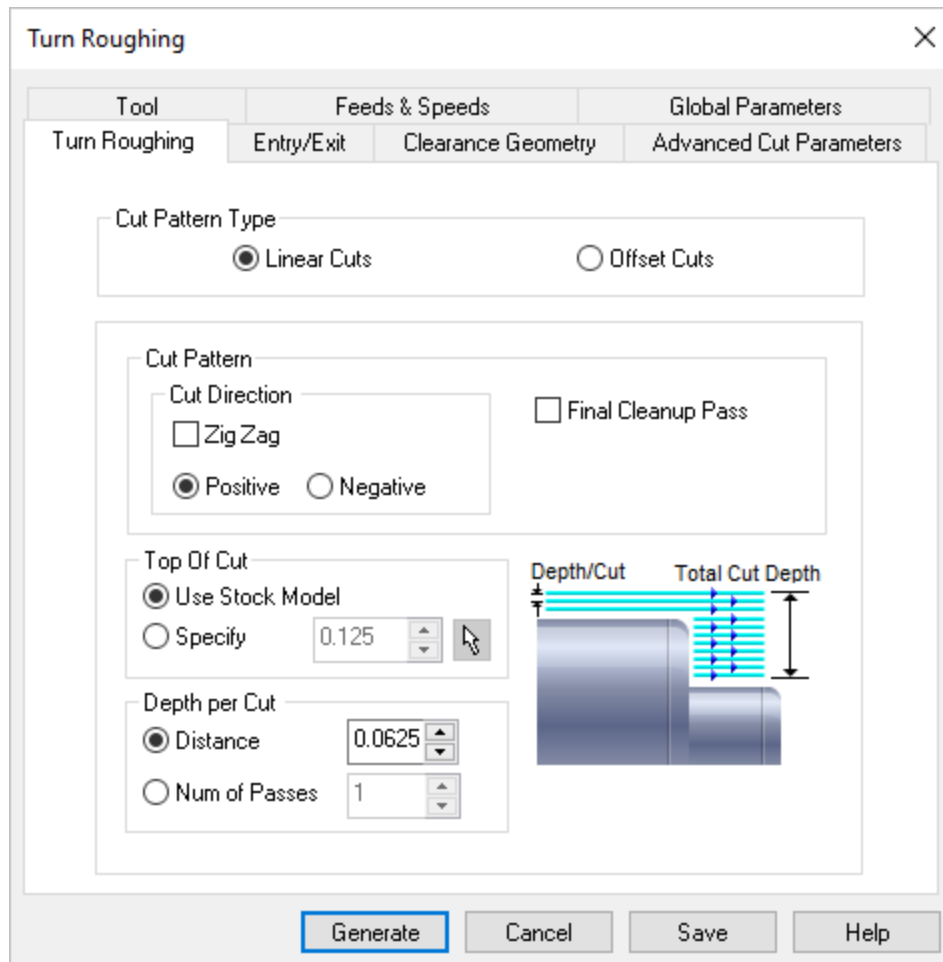
Outer Diameter Turn Roughing

Now 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.



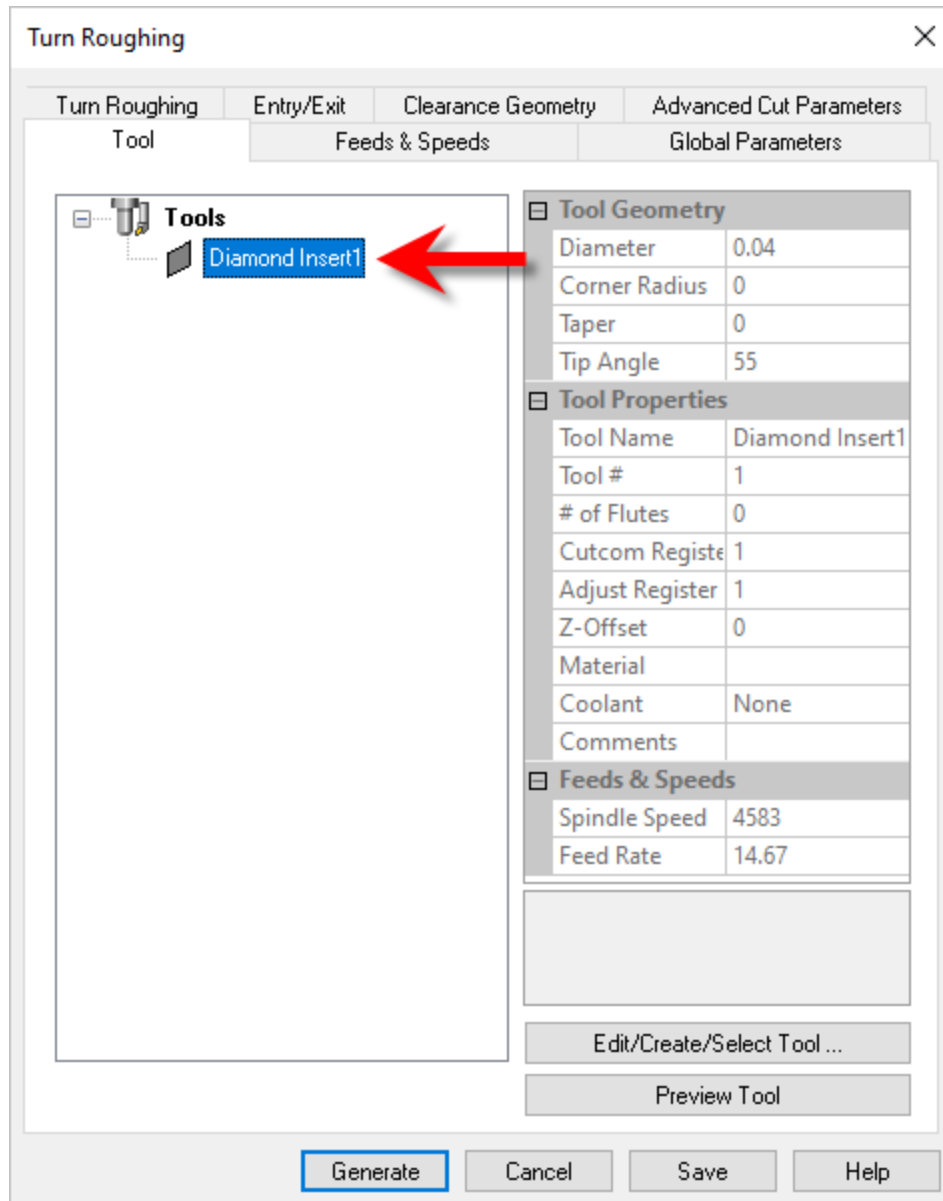
2. This will display the [Turn Roughing](#) operation dialog.




7.1 Select the Cutting Tool

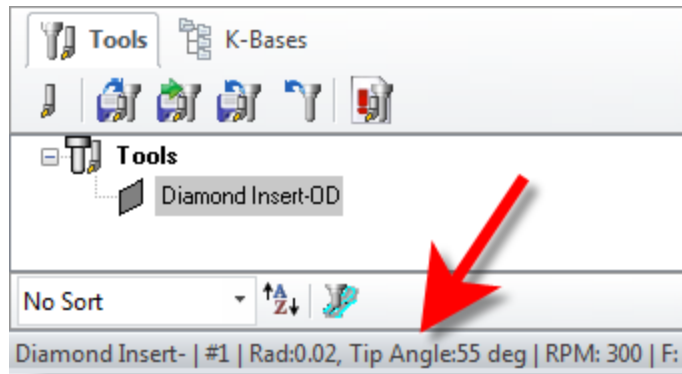
Next we'll select the cutting tool for the [Turn Roughing](#) operation.

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.

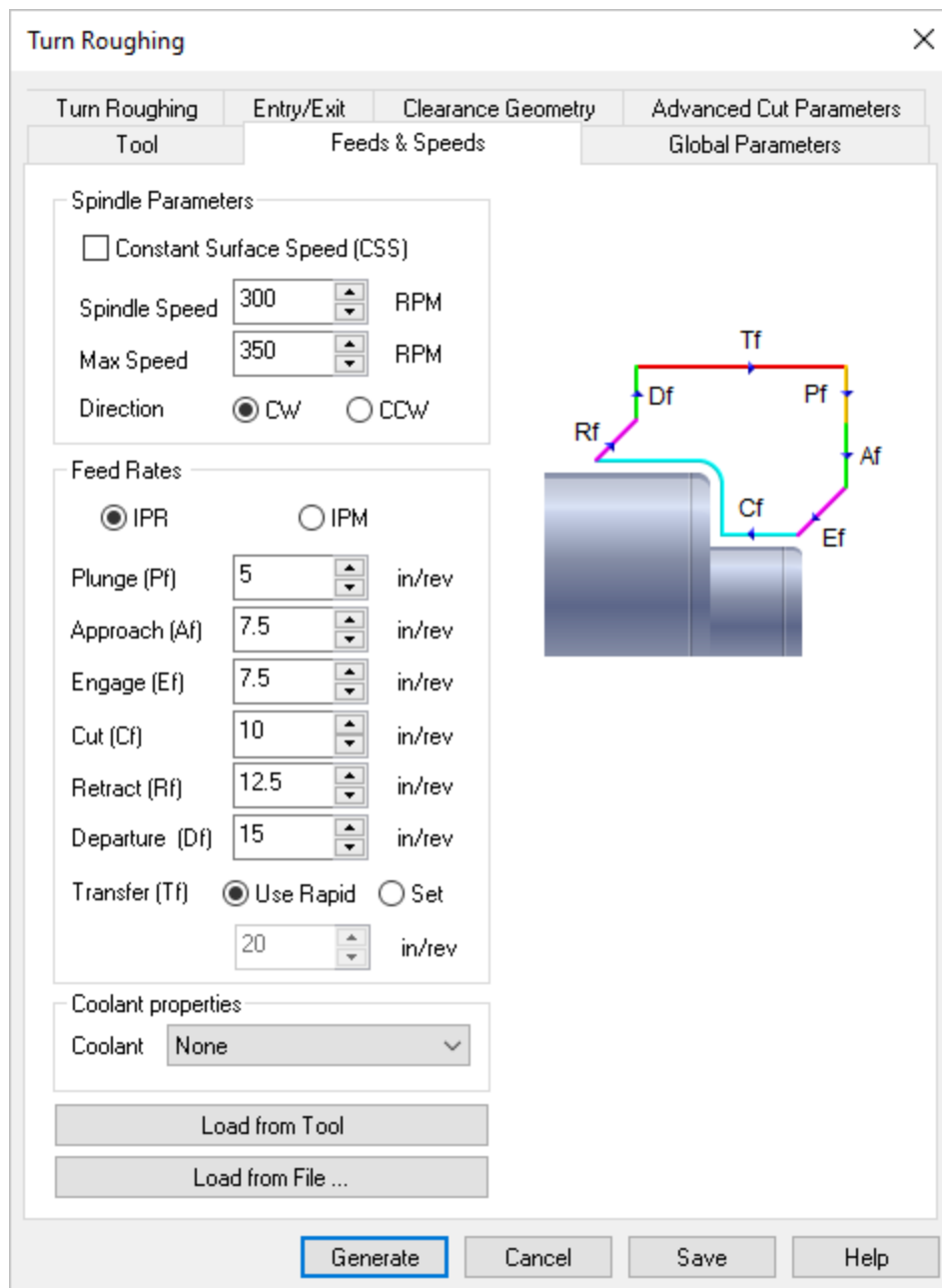
 The [Tool](#) parameters of the currently active tool are always displayed in the status bar at the bottom of the [Machining Objects browser](#) as shown below.



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.

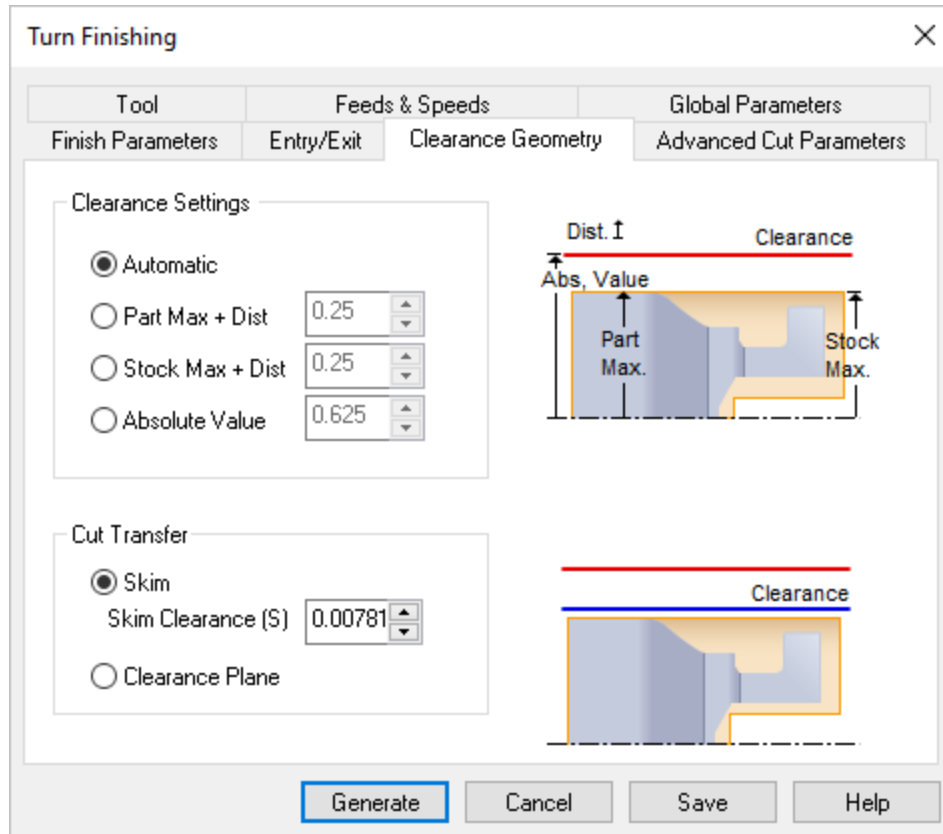
7.3 Set Clearance Geometry

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

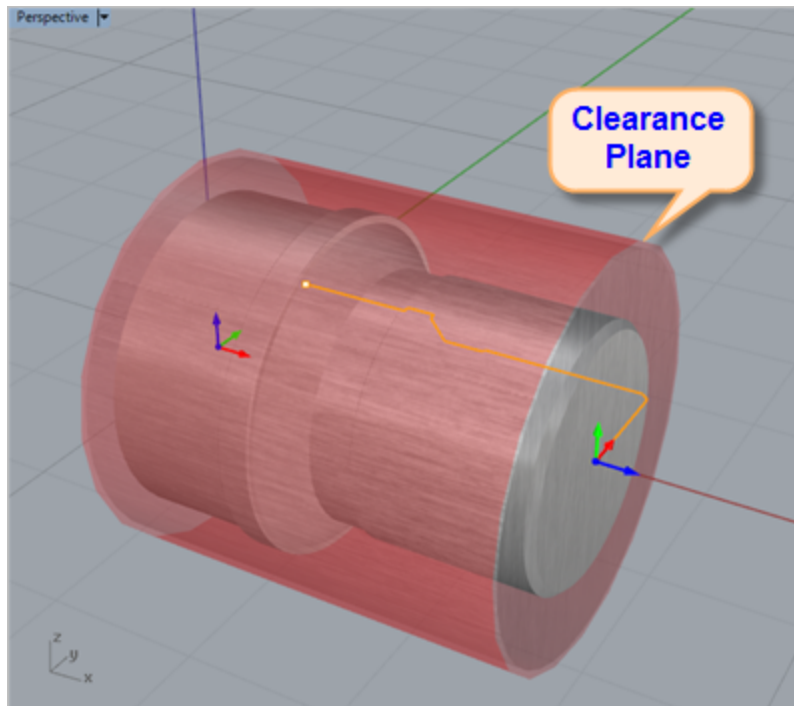
1. Select the [Clearance Geometry](#) tab of the dialog.

- 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.



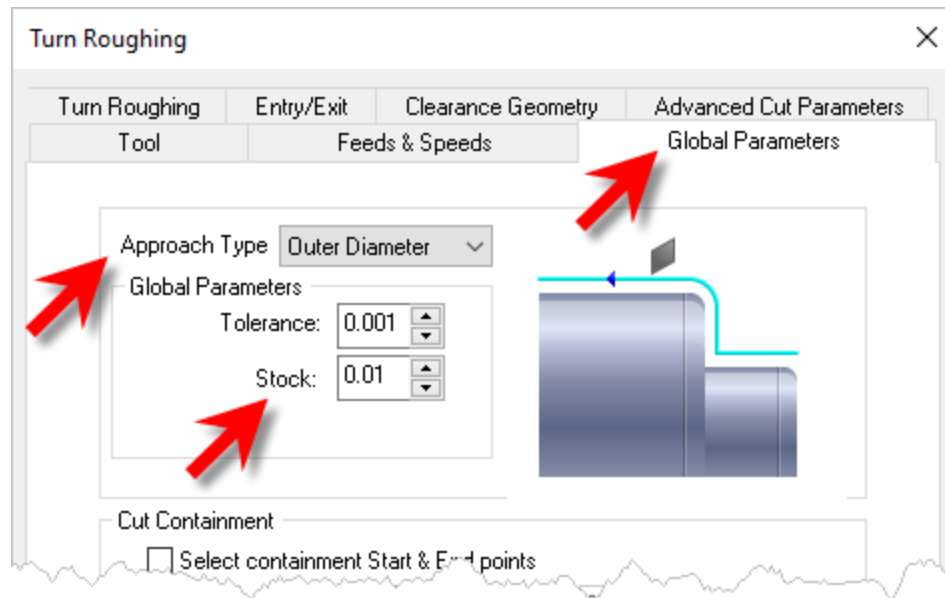
- When this dialog is active, the clearance plane is shown on the graphics screen.



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.

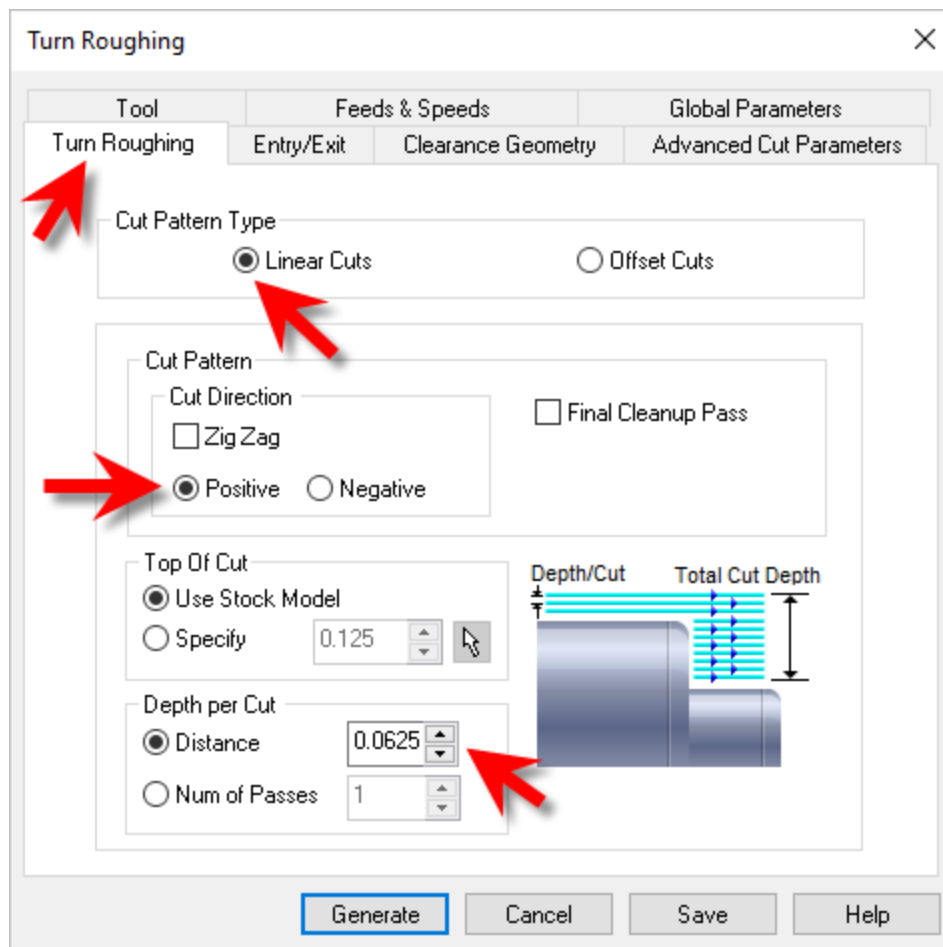


7.5 Set Roughing Parameters

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.

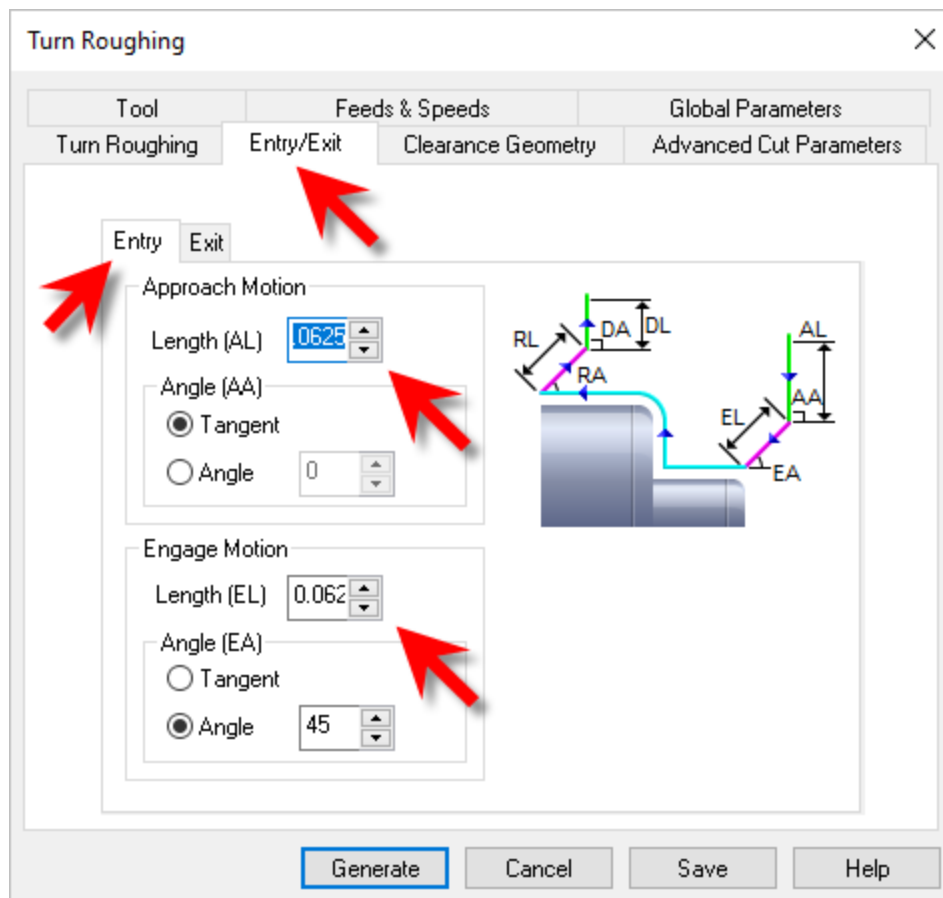
Note that [Depth per Cut](#) is always set to an absolute value.



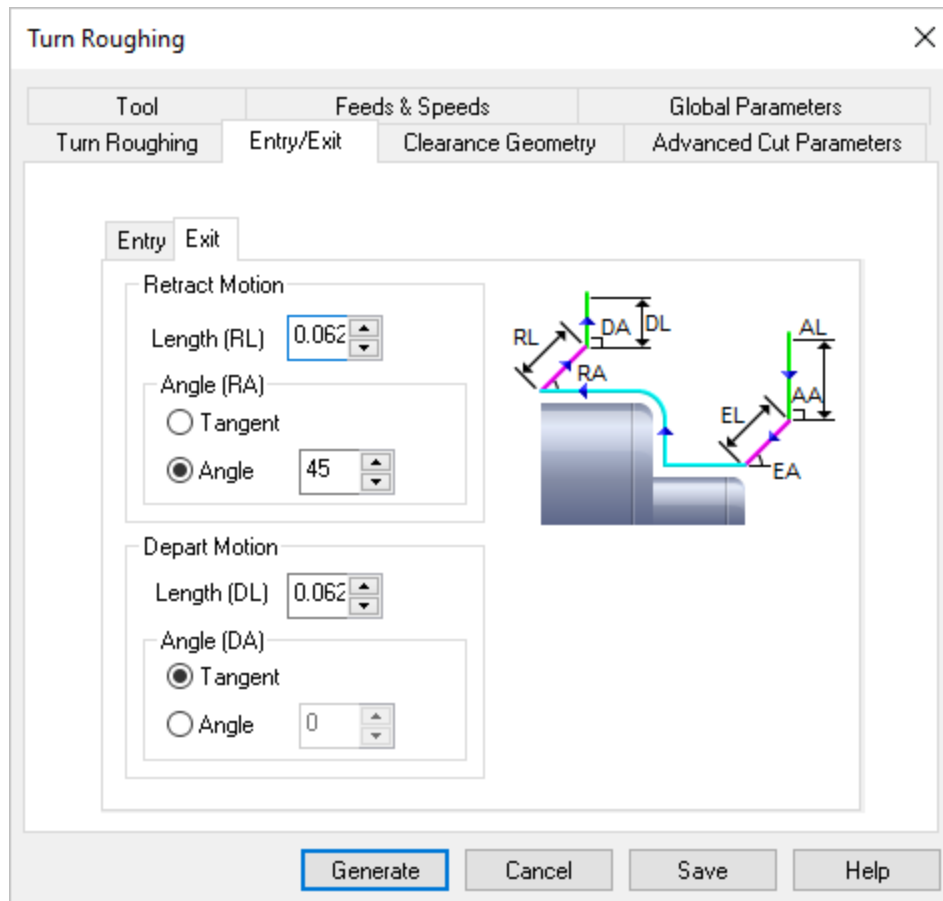
7.6 Set Entry/Exit Parameters

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.

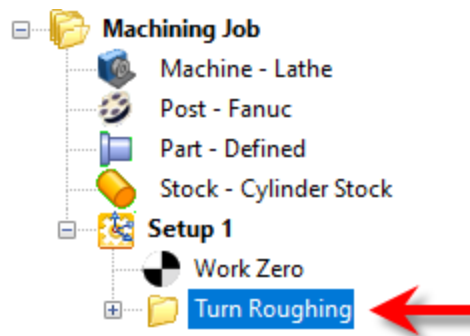


- Next, we'll switch to **Exit** tab set the **Retract Motion Length (RL)** to 0.025 and do the same for the **Depart Motion Length (DL)**.

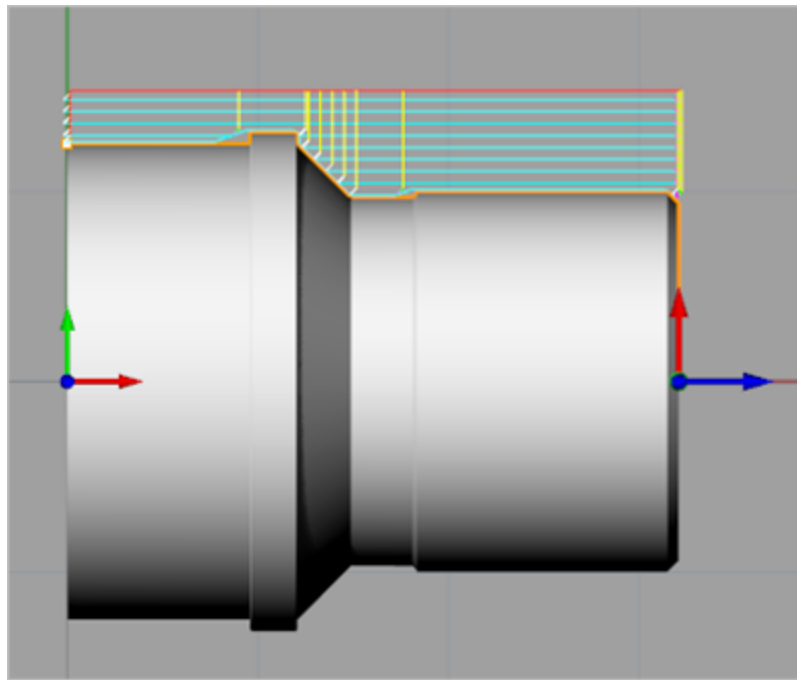
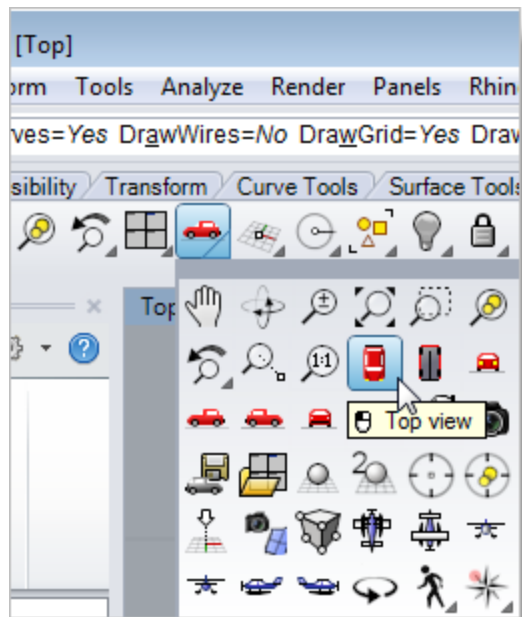



5. Now pick [Generate](#).

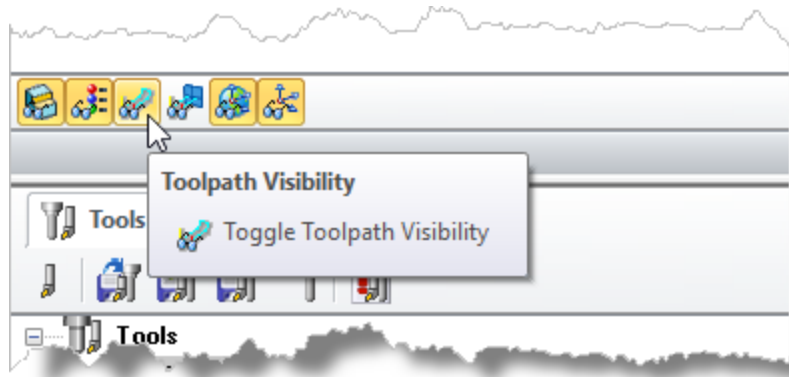
The [Turn Roughing](#) toolpath is generated and the operation is listed under [Setup 1](#) in the [Machining Browser](#).



6. Now from the [View](#) toolbar in [Rhino](#), select the [Top View](#) and you can see the toolpath displayed clearly in the graphics screen.



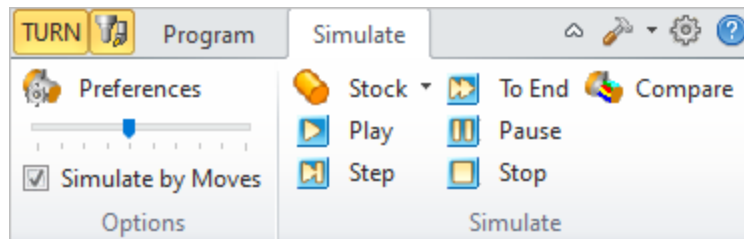
 The display of the toolpath in the graphics screen can be turned on/off by selecting the **Toolpath Visibility** icon in the toolbar at the bottom of the **Machining Browser**.



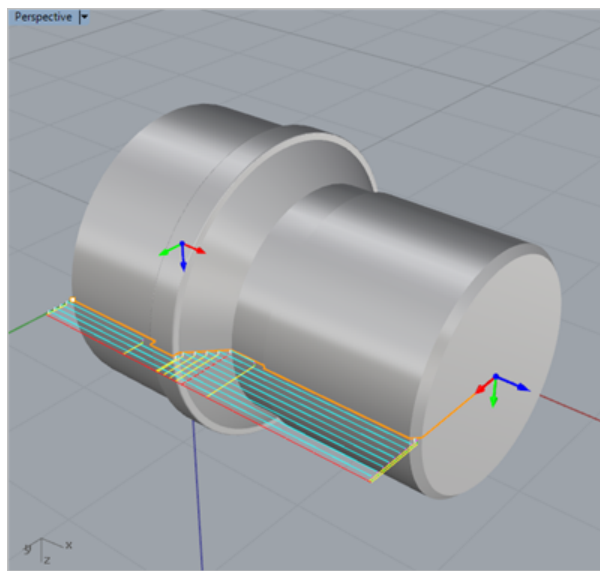
7.7 Simulate the Toolpath

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**.

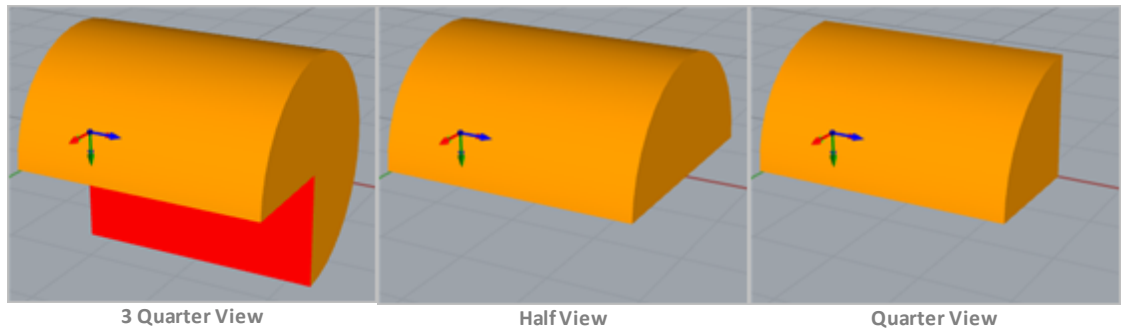
1. Switch to the **Simulate** tab at the top of the **Machining Browser**.



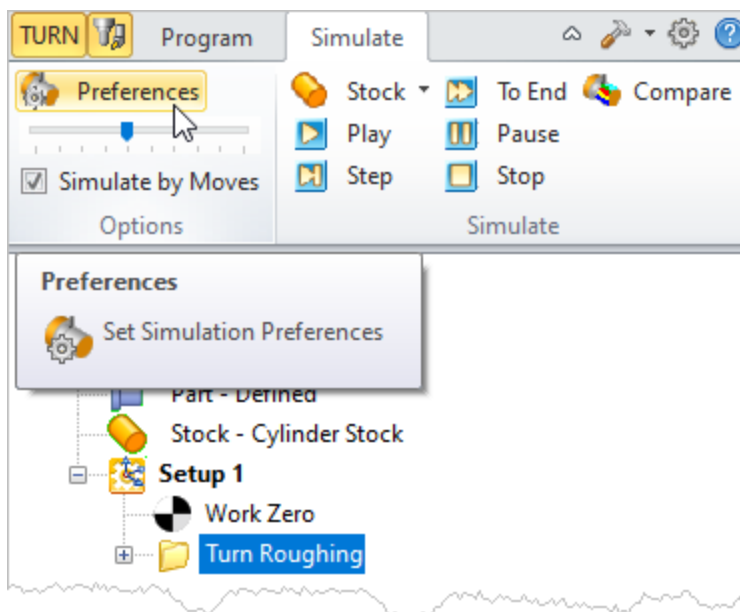
2. Press and hold the **right-mouse** button to rotate the model to a position suitable for viewing the new **Turn Roughing** toolpath as shown below

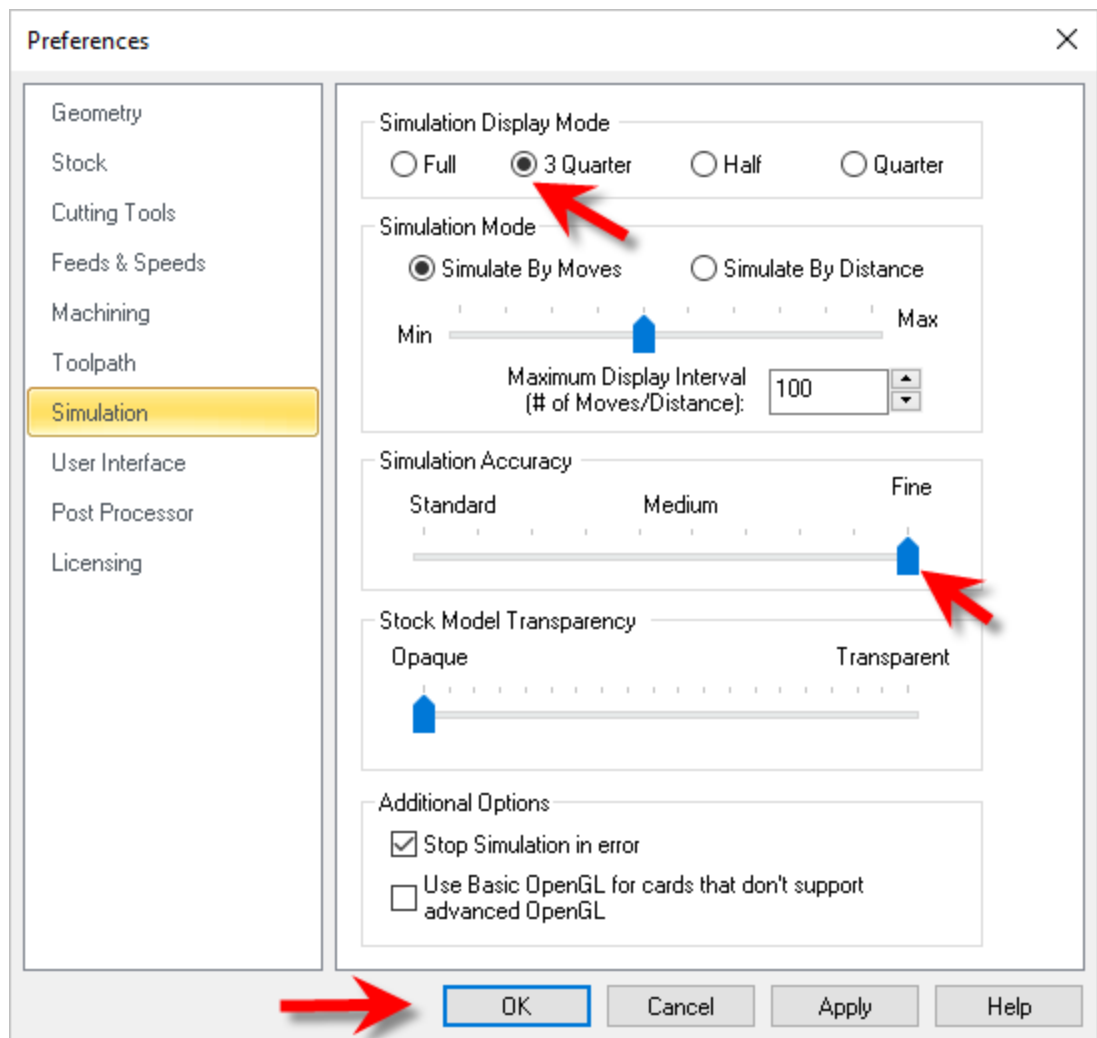


- In the **TURN** module the rendering mode of the simulation model can be controlled for better visibility by using cut away section views. Three modes, in addition to the normal rendering mode, are available. These are **3 Quarter**, **Half** and **Quarter** views. These display modes can be set in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

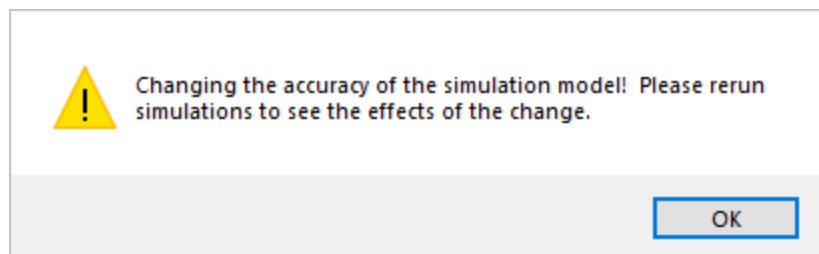


- From the **Simulate** tab, select **Preferences** and set the **Simulation** display mode to **3 Quarter** and the **Simulation Accuracy** to **Fine** and then pick **OK**.

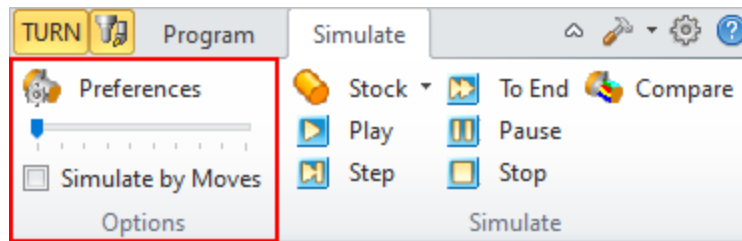




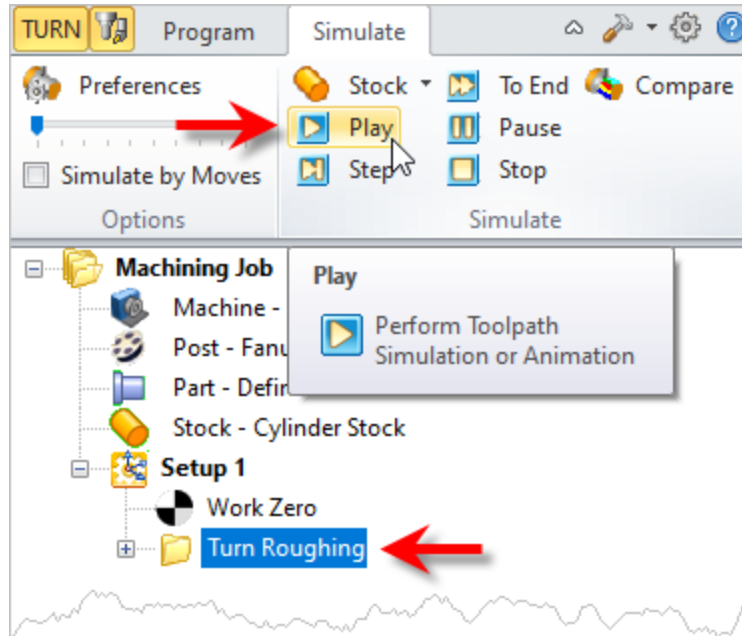
Pick **OK** from the message dialog that displays:



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

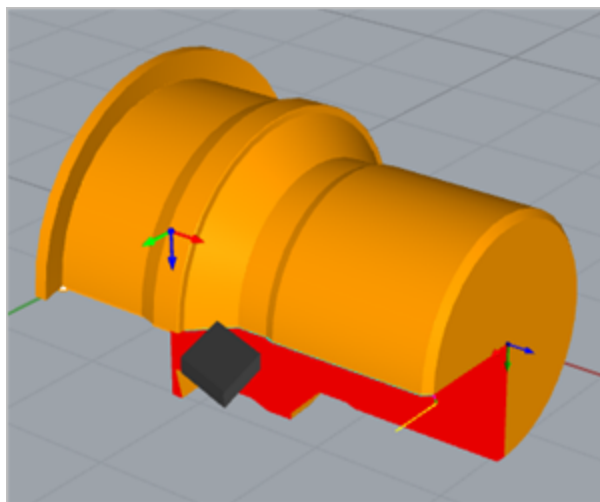


- Now select the **Turn Roughing** operation under **Setup 1** and then pick **Play**.

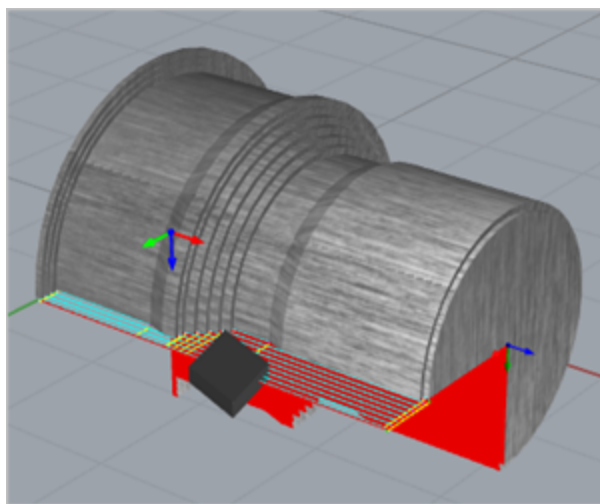
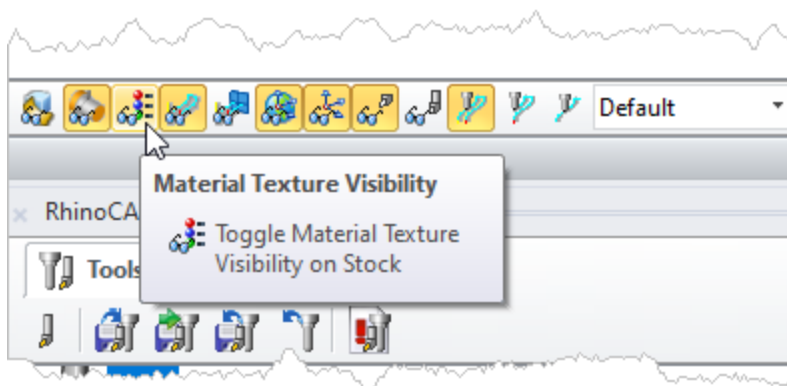


! You can stop the simulation at anytime by selecting the **Pause** button from the **Simulate** tab. After **Pausing**, you can choose either **Play** to continue or **Stop** to exit the simulation.

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



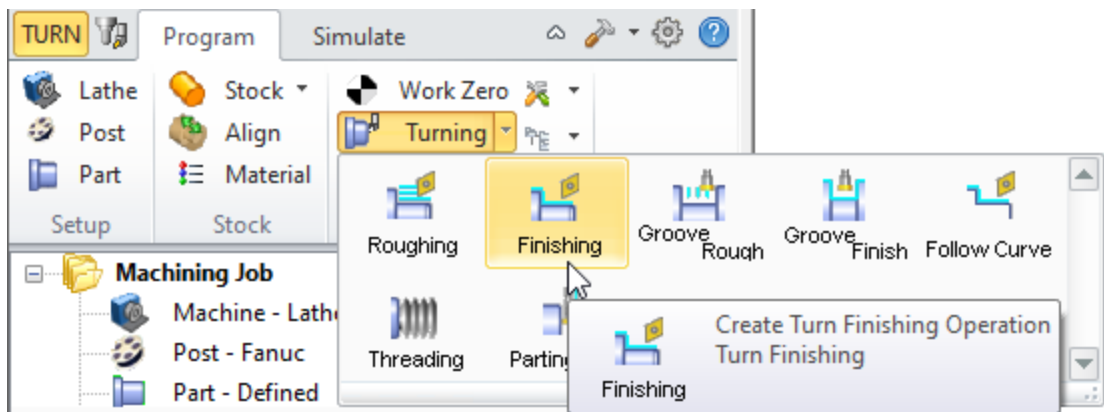
- To view the cut model with textures applied, select the [Material Texture Visibility](#) icon located at the base of the [Machining Browser](#).



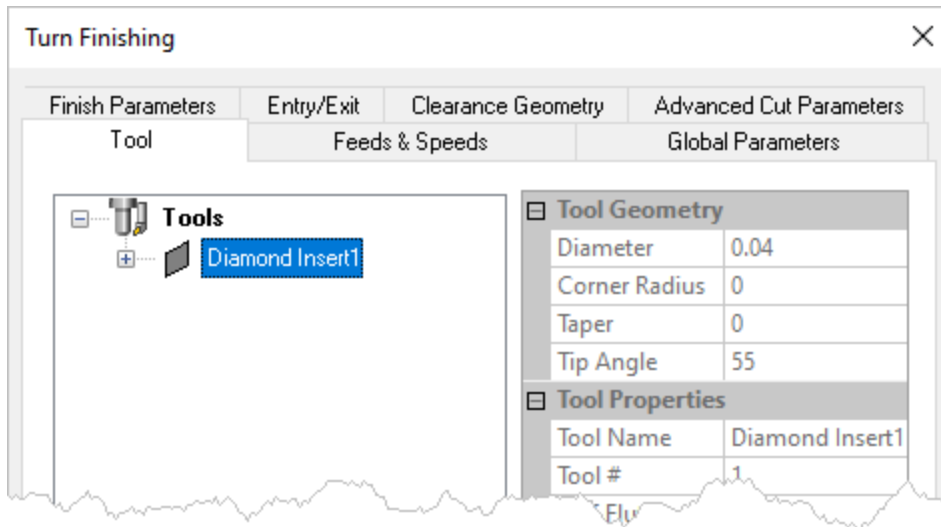
Outer Diameter Turn Finishing

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.



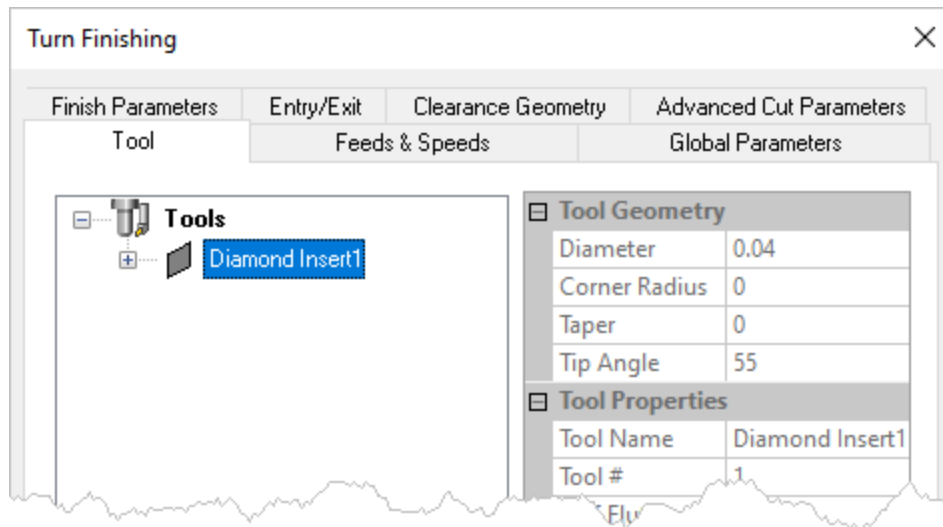
3. This will display the **Turn Finishing** operation dialog shown below.



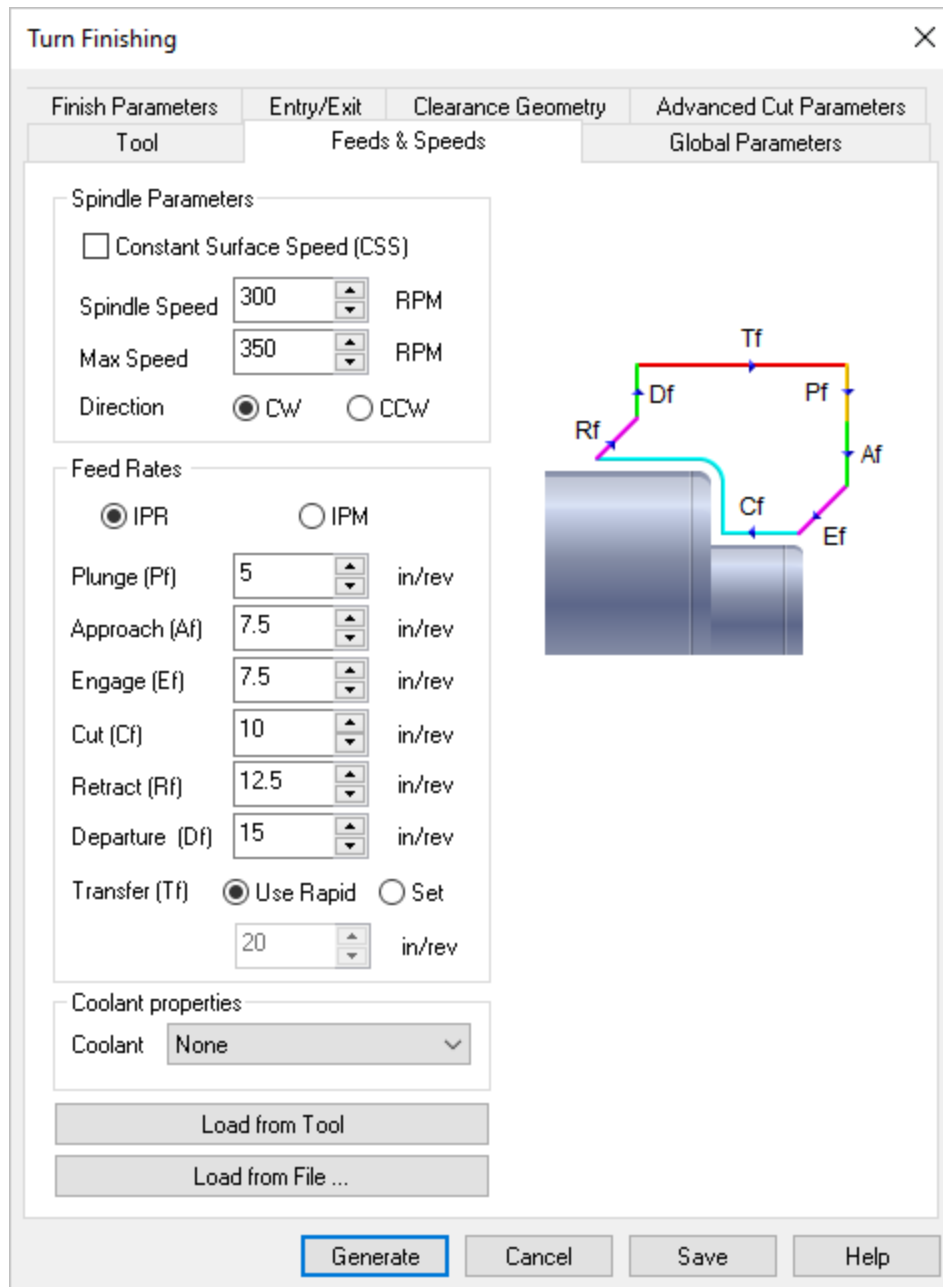
8.1 Set Finishing Parameters

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

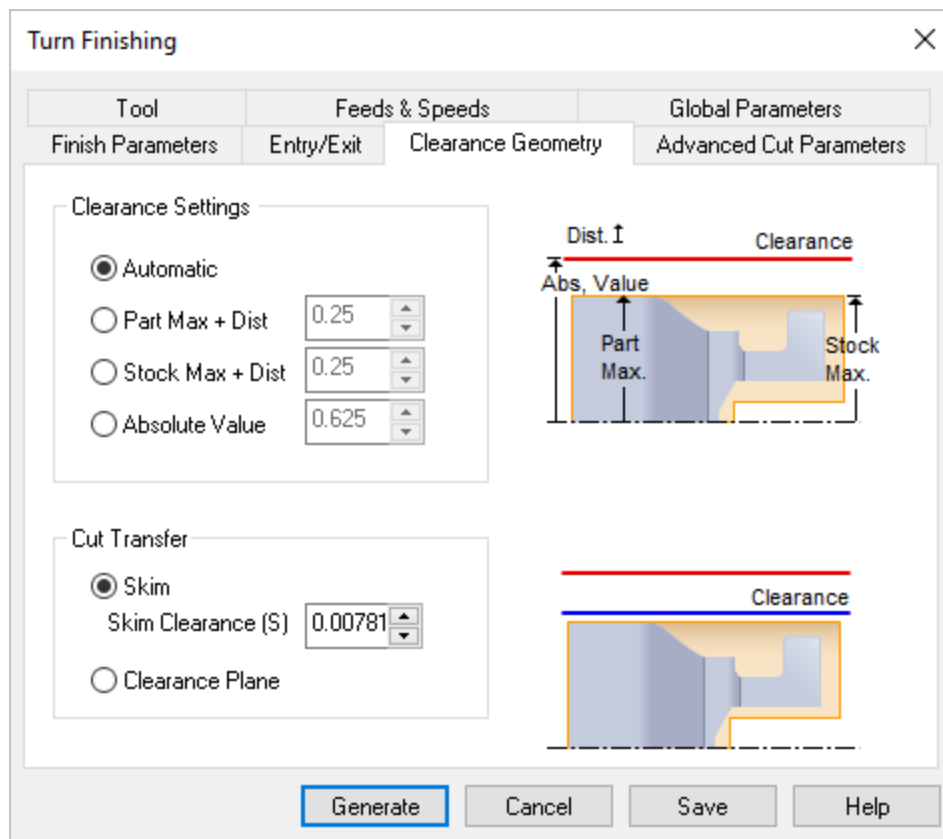
1. From the **Tool** tab in the **Turn Finishing** dialog select the **Diamond Insert-OD** tool.



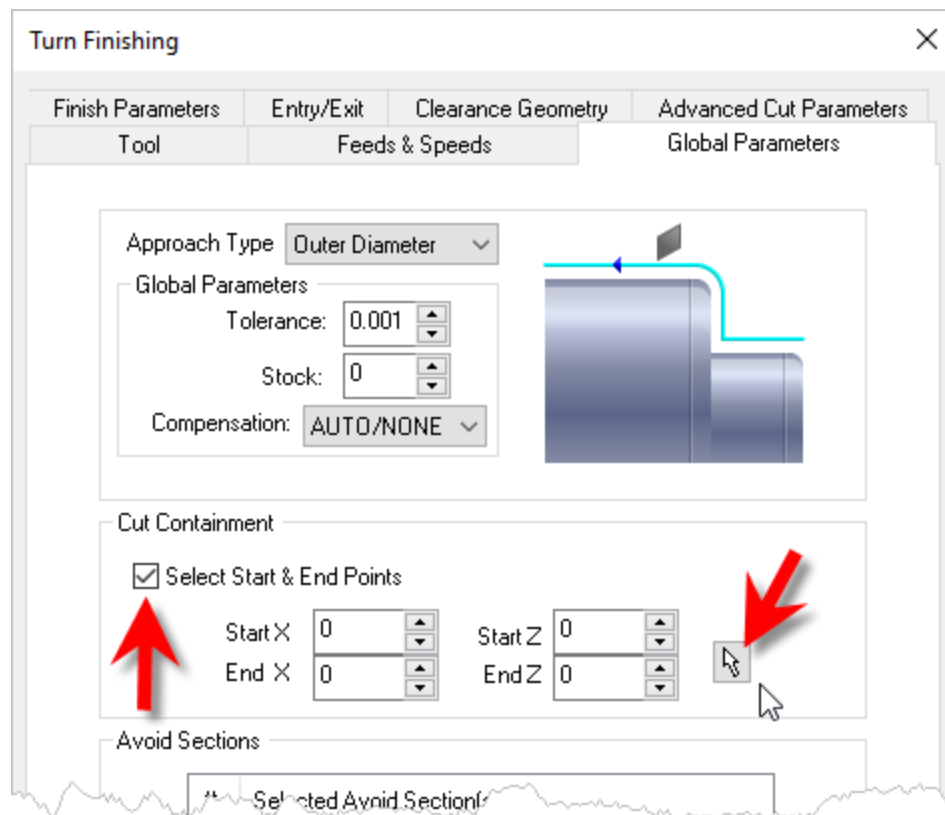
2. Now, pick the **Feeds & Speeds** tab and select the **Load from Tool** button. The system will retrieve the feeds and speeds parameters that was set when the tool was defined and associate them with the current operation.



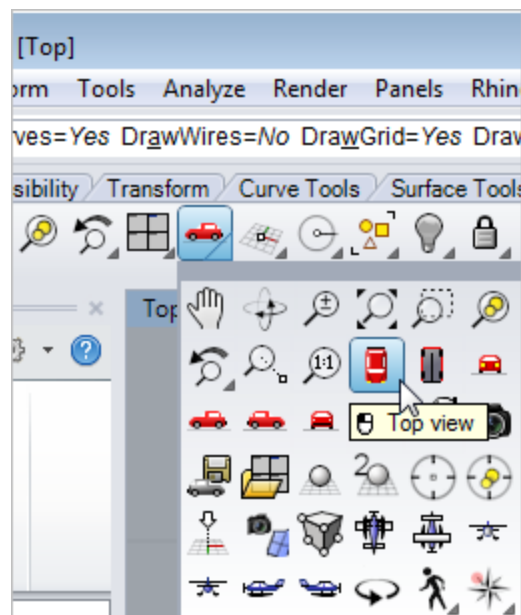
- Pick the [Clearance Geometry](#) tab and set the [Clearance Settings](#) to [Automatic](#) and [Cut Transfer](#) to [Clearance Plane](#).



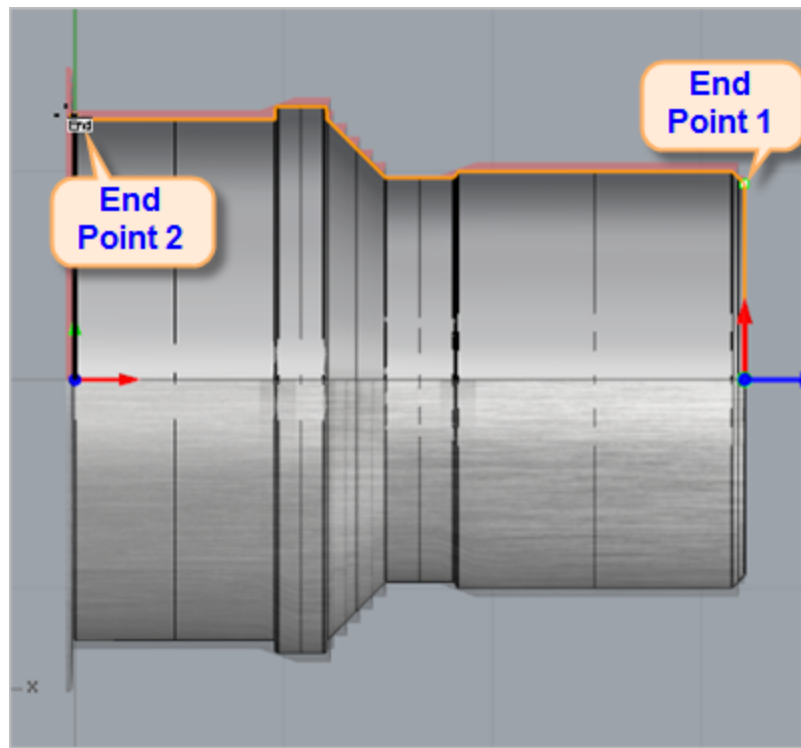
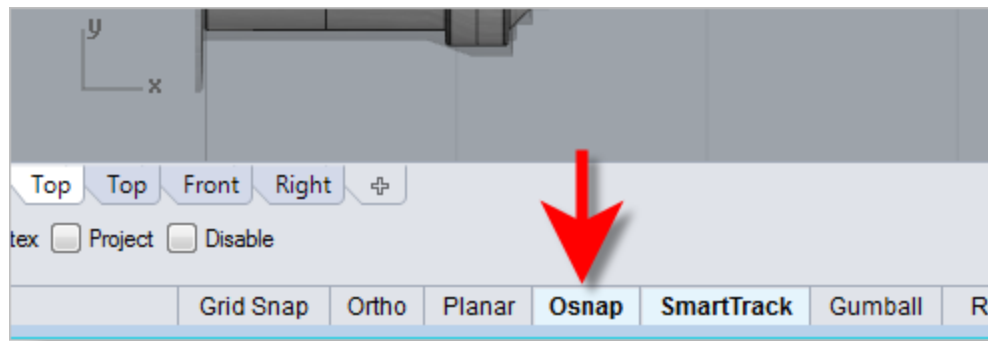
4. Now we'll switch to the [Global Parameters](#) tab to specify parameters to control the cutting.
5. Set the [Approach Type](#) to [Outer Diameter](#) and [Stock](#) = 0.
We will not be leaving any thickness on the part after machining, effectively removing all stock left over from the previous roughing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



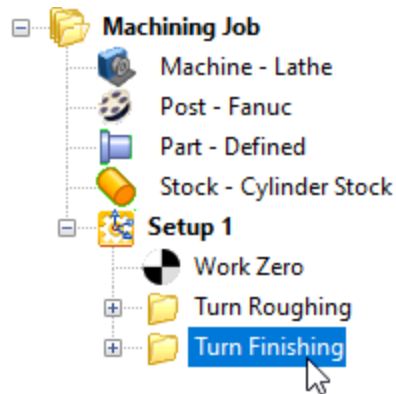
8. From the [View](#) toolbar, we'll switch to [Top View](#).



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



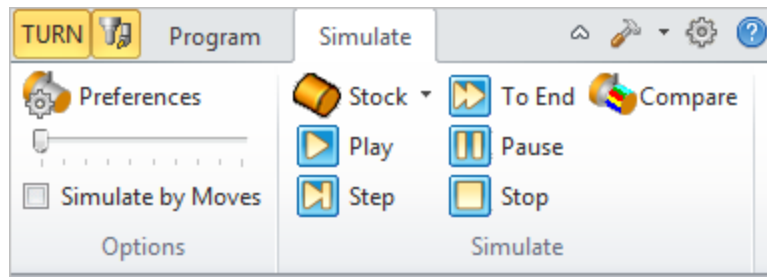
10. The **Turn Finishing** dialog reappears and displays **Start** and **End** point coordinate values for the cut containment.
11. Leaving all other parameters with default settings, we pick **Generate**. The operation is generated and added to the **Machining Job** in the **Machining Browser** below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



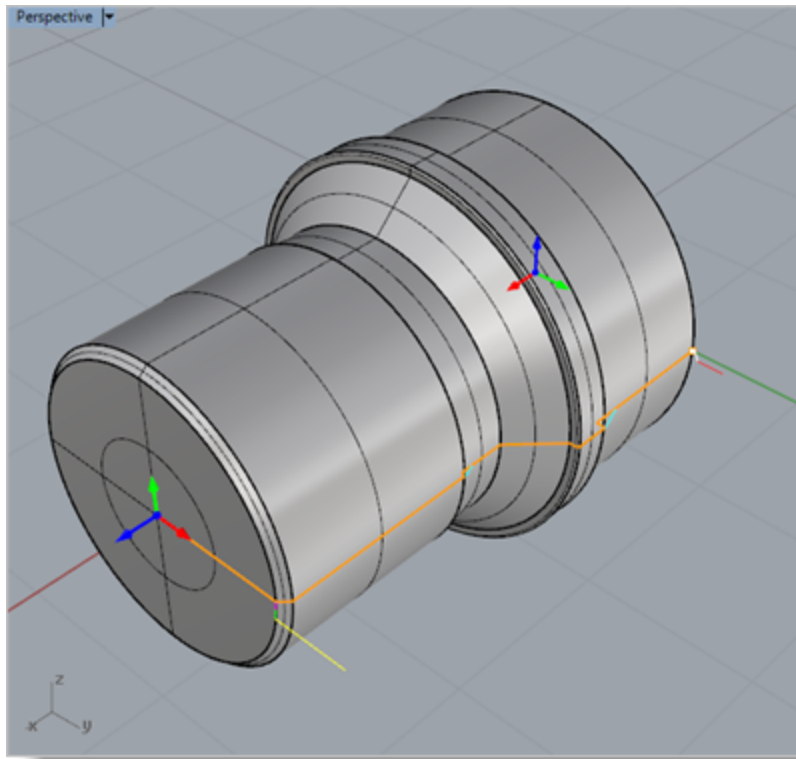
8.2 Simulate the Toolpath

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

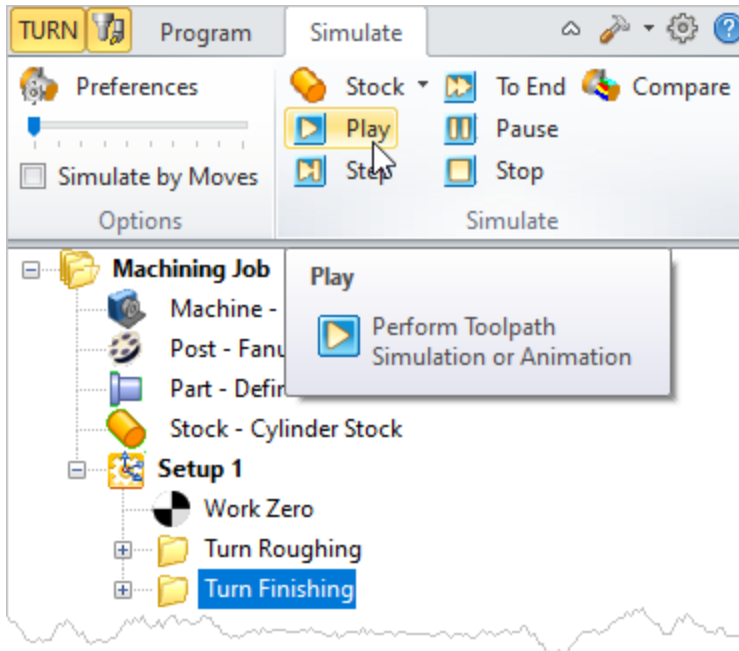
1. First make sure the [Turn Finishing](#) toolpath is selected from the [Machining Job](#) tree.
2. Select the [Simulate](#) tab.

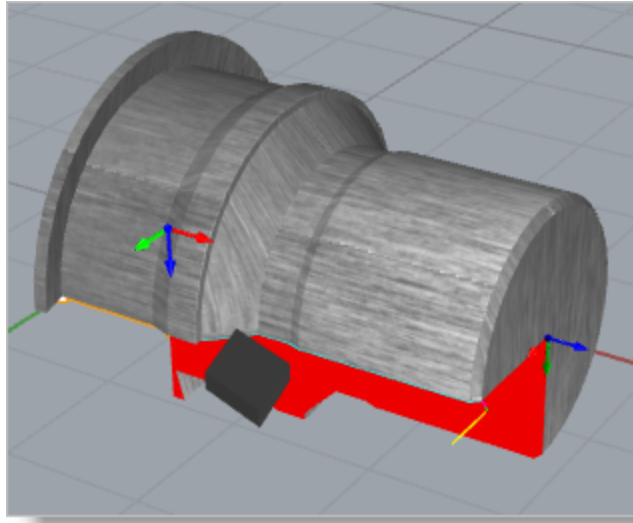


3. Press and hold the [right-mouse](#) button to rotate the model to a position suitable for viewing the new [Turn Roughing](#) toolpath as shown below.



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

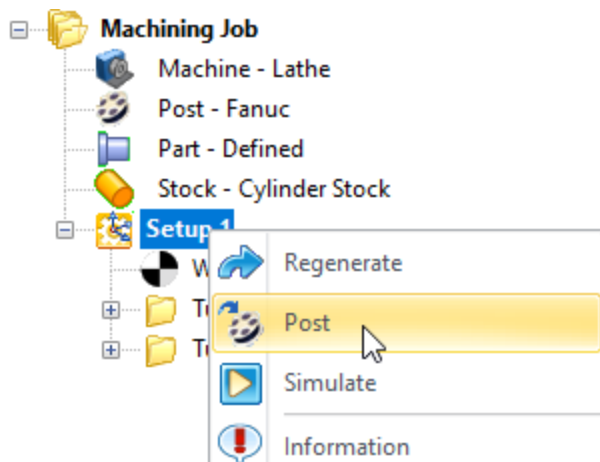




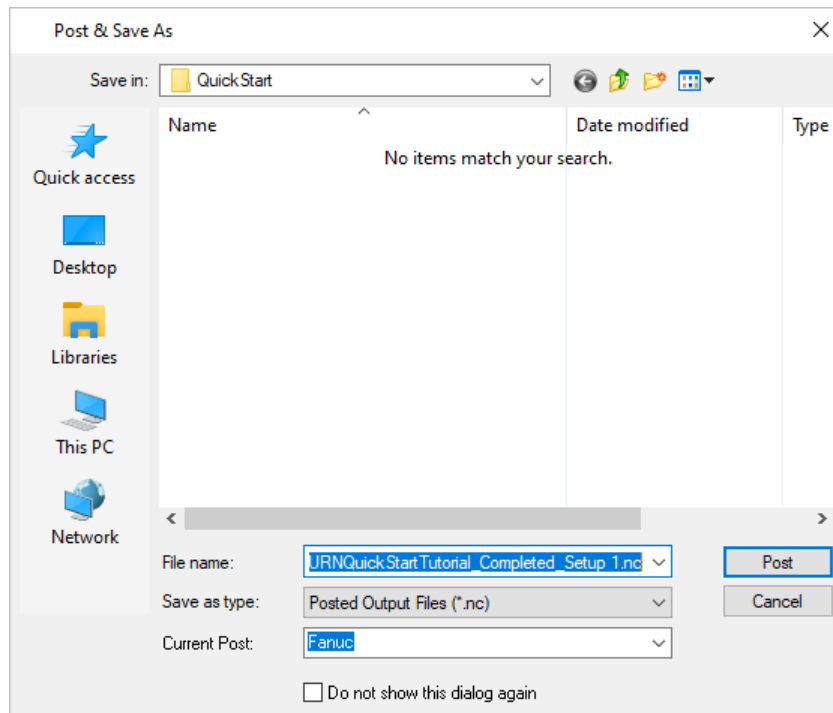
8.3 Post-Process the Toolpath

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**.

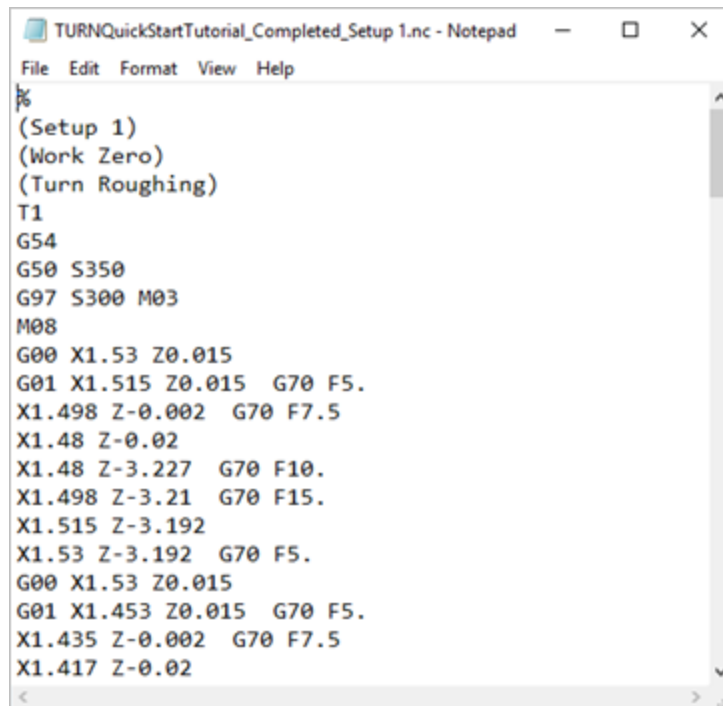


2. This will post-process all operations created under **Setup 1**.
3. By default, the **Part** file name and the **Setup** name are appended for the G-code file name. Also by default, the posted G-code file is saved to the folder where the part file is located.



! 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.



```
TURNQuickStartTutorial_Completed_Setup 1.nc - Notepad
File Edit Format View Help
%
(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.
X1.435 Z-0.002 G70 F7.5
X1.417 Z-0.02
```

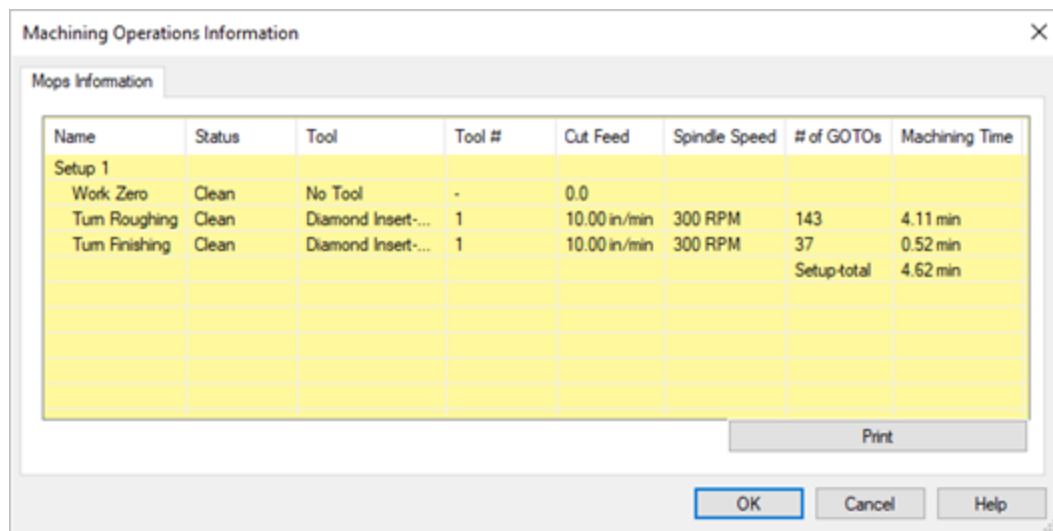
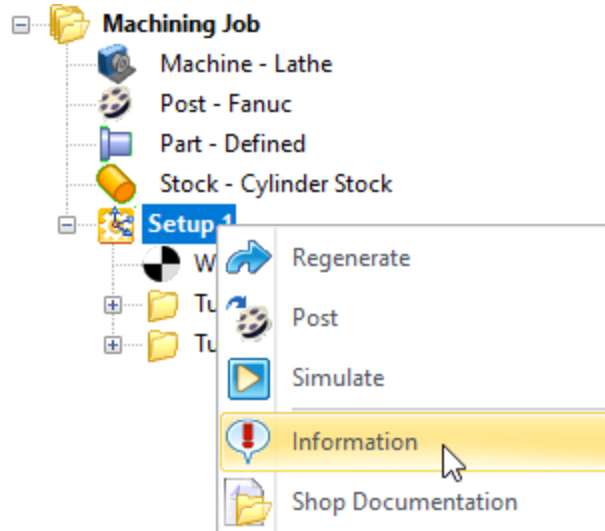
5. Now close [Notepad](#).

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](#).

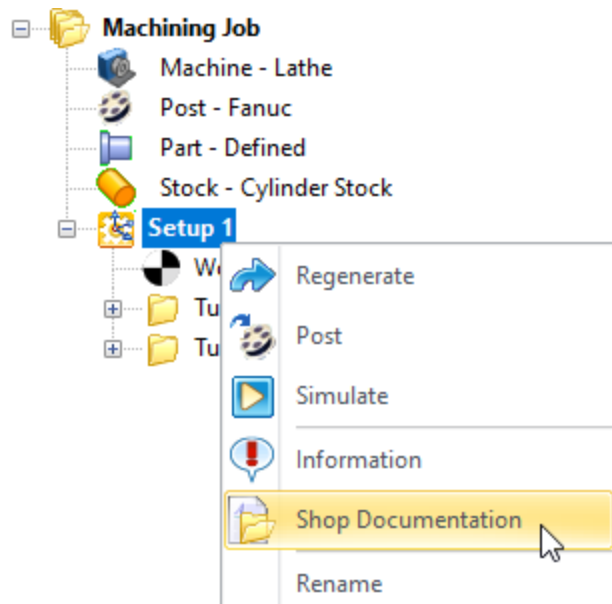
You can perform the same right-click sequence on the [Machining Job](#) to determine the estimated machining time for all [Setups](#).

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

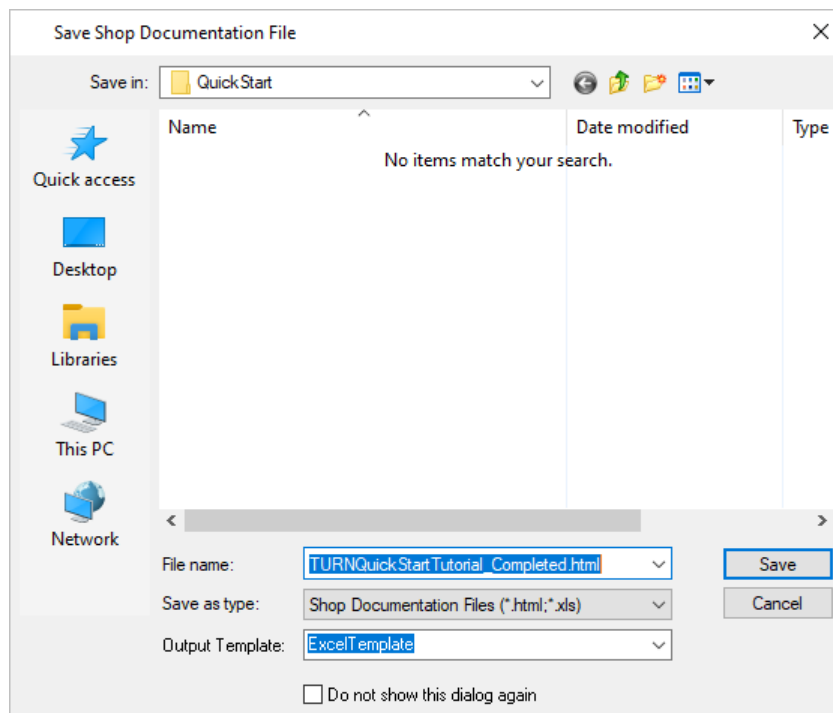
9.2 Shop Documentation

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.

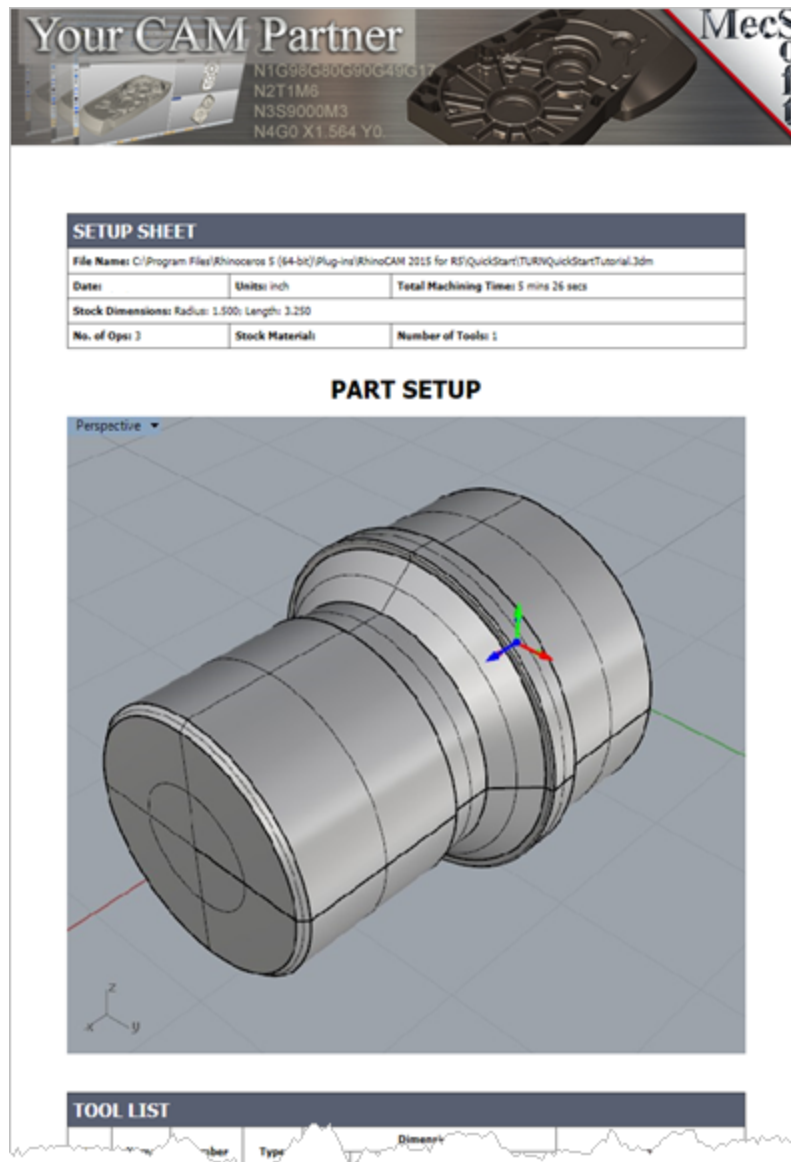
1. Select [Setup1](#) under the [Machining Job](#) tree in the [Machining Browser](#).
2. Right-click and select [Shop Documentation](#).



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](#).



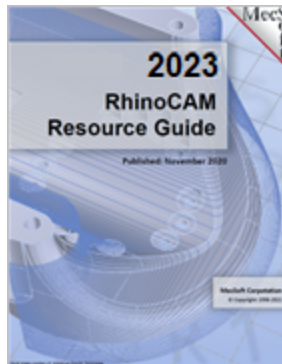
5. You can perform the same right-click sequence on the [Machining Job](#) to generate [Shop Documentation](#) for all [Setups](#).

Where to go for more help

Download this PDF Guide for a list of the available [RhinoCAM 2024 Resources](#).



2023 RhinoCAM 2024 Resource Guide



The 2023 RhinoCAM 2024 Resource Guide!

18 Pages

Lists PDF downloads and Online resources including [Quick Start Guides](#), [Reference Guides](#), [Exercise Guides](#), [Tutorials](#) and [More](#).

[Prefer Printed Documentation? Check Here!](#)

Index

- A -

- About
 - the RhinoCAM Display 12
 - the TURN Module 9
 - Using this Guide 10
- Align Part and Stock 29

- C -

- Clearance Geometry 45, 58
- Create Stock Geometry 27
- Create Tools 38

- D -

- Define the Machine Tool to use 19
- Define the Part Geometry 23

- E -

- Estimated Machining Time 69

- F -

- Feeds and Speeds
 - for Roughing 44
- Finishing
 - Select Cutting Tool 58
 - Set Clearance 58
 - Set Feeds and Speeds 58
 - Simulate the Toolpath 64
 - Specify Global Parameters 58
 - the Outer Diameter 58

- L -

- Launching the RhinoCAM 2024 TURN Module 13
- Loading the Part Model 16

- M -

- Machine
 - Define 19
 - Setup 33
- Machining Information 69
- Machining Strategy 18
- Main Programming Steps 19

- P -

- Part
 - Define Geometry 23
- Post
 - Create G-Code 66
 - Select 20
- Post Processing 66
- Print Media 8

- R -

- Reports
 - Estimated Machining Time 69
 - Machining Information 69
 - Setup Sheet 70
 - Shop Documentation 70
- Roughing
 - Entry/Exit Parameters 49
 - Select Cutting Tool 43
 - Set Clearance 45
 - Set Feeds and Speeds 44
 - Set Global Parameters 47
 - Set Roughing Parameters 48
 - Simulate Toolpath 53
 - the Outer Diameter 42
- Running RhinoCAM 2024 12

- S -

- Select Cutting Tool 43, 58
- Select the Post Processor to use 20
- Set Feeds and Speeds 44, 58
- Set Work Zero 34
- Setup 33
- Setup Sheet 70
- Shop Documentation 70

Simulate Roughing Toolpath	53
Simulate the Toolpath	64
Specify Entry/Exit Parameters	49
Specify Global Parameters	47, 58
Specify Material	31
Stock	
Align with Part	29
Define	27
Define Material	31

- T -

Tools	
Create	38
Select for Roughing	43
Turn Roughing	48

- U -

Useful Tips	9
Using this Guide	10

- V -

Videos & Guides	5
-----------------	---

- W -

Where to go for more help	73
Work Zero	34