TURN Reference Guide

RhinoCAM-TURN 2026

Published: March 2026

MecSoft Corpotation

Table of Contents

Welcome to TURN

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

The RhinoCAM-TURN Module is a unique turning product that is used for offline programming of 2 Axis CNC Lathes. This product is a module of RhinoCAM, which is completely integrated inside of Rhino. This integration allows for the seamless generation of toolpaths and cut material simulation/verifications inside of Rhino for programming turning machines.

For purposes of brevity, RhinoCAM will be referred to as TURN in all subsequent references. Also, Rhino refers to both Rhinoceros 7 or Rhino 8.

The RhinoCAM-TURN module also comes with numerous post-processors to output the programmed G-code to some of the most popular CNC machine controllers on the market today. Additionally, the RhinoCAM-TURN module also includes a post-processor generator that has the capability to configure entirely new post-processors. These features make programming CNC Turning easy and affordable.

The RhinoCAM-TURN module supports turning and hole making operations such as Turn Roughing, Turn Finishing, Groove Roughing, Groove Finishing, Threading, Parting and Hole making operations such as Drilling, Tapping, Boring, Reverse Boring and Threading.

1.1 Features of the TURN module

The list below summarizes the toolpath generation features found in the TURN module.

Data Import:

Supports file formats that can be imported in Rhinoceros

Turn Machining Methods:

Roughing, Finishing, Groove Roughing, Groove Finishing, Follow Curve, Threading and Parting.

Hole Machining Methods:

Drilling, Tapping, Boring, Reverse Boring

Tool Types:

Turning Inserts - Diamond, Triangular, Circular, Trigon, Parallelogram, Groove.

Hole Making Tools - Drill, CenterDrill, Reamer, Tap, Bore, ReverseBore.

User customizable tool library:

Create and save tools to library. Saves feeds and speeds to tool library.

Feeds and Speeds:

Customizable feeds and speeds table, Feeds and Speeds calculator

Cut Material Simulation:

Fast and fully integrated Material Cutting Simulation. Create true 3D in-process stock model. Advanced cut material simulation includes tool holder collision and part stock comparison.

Post-Processors:

Standard set of post-processors bundled with the product. Choose from our large set of standard post processors.

Post-Processor Generator:

Customize your Post Processor or create your own Post Processor from scratch. This post processor generator supports 3 axis posts.

1.2 Understanding the TURN Module

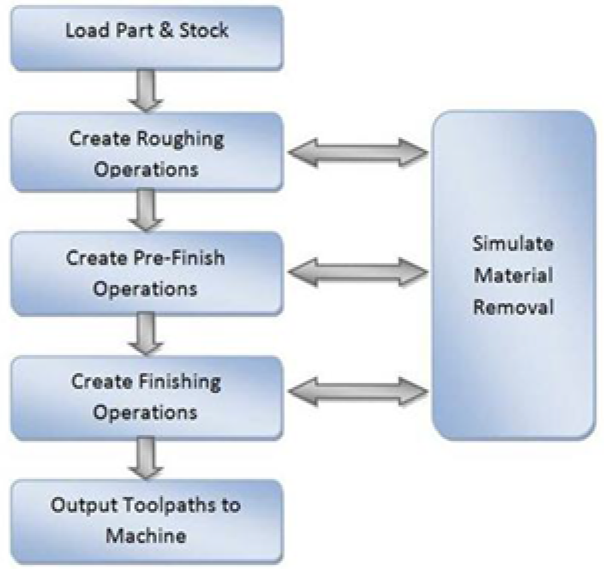
The manufacturing process aims to successively reduce material from the stock model until it reaches the final shape of the designed part model. To accomplish this, the machinist or programmer utilizes a 3-part machining strategy.

3-Part Machining Strategy

1. A typical machining strategy employed in the manufacturing industry is to use larger cutters to perform bulk removal of material early on in the manufacturing process. These operations are called roughing operations.
2. This is then followed by operations employing successively smaller cutters removing proportionately smaller amounts of material from the stock model. This is done until the part has a uniform amount of stock left. These operations are called pre-finishing operations.
3. This is then followed by finish operations. Here the uniform stock remaining on the part is removed by using a small cutter removing a constant amount of material with every motion to produce the net shape. The standard work

flow of TURN Module mimics this process and can be represented by the flow chart shown below.

Basic TURN Module Work Flow Chart



TURN Module Work Flow Chart

Basic TURN Module Work Flow Steps

1. Part geometry is imported into TURN Module via the various data interfaces provided in Rhino. A stock model representing the raw stock from which the part needs to be manufactured can then either be created using the various tools provided in TURN Module or imported.
2. You then determine the machining strategy to be used in manufacturing the part. This can be done by loading a previously saved manufacturing operation sequence or by creating a new one. This manufacturing strategy is represented by a sequence of machining operations in Turning.

   To create a new machining strategy, simply selects the tools and the machining operations in sequence and generates toolpaths. The system automatically records this sequence. This record can be archived as an operation list that can be retrieved for later use.
3. You can also simulate material removal to visualize how the stock model will look at any time during the process. This provides valuable feedback that can help you choose the most appropriate machining strategy.
4. To create a new machining operation, select a tool followed by the type of toolpath to be created. You then selects the parameters to use for machining and then generates the toolpath.

5. Rough machining can be done by Roughing operations, using a turning tool with a relatively large nose radius. These rough operations can be followed by subsequent roughing operations, either using the same tool or a smaller tool.
6. Final finishing of the part can then be performed by using one or more Finishing operations. Finishing operations typically use tools with smaller nose radius so as to obtain a better surface finish and tighter tolerance levels.
7. Depending on the geometry of the part and/or machining operations desired, Groove Roughing, Groove Finishing, Follow Curve, Threading and the Hole-Making operations can be considered. After completing all the machining operations, the final part is cut off from the rest of the bar stock by using the Part-Off operation.
8. Once all of the operations are completed you can then go back and review the operation sequence, re-order operations if desired and output the toolpath for post-processing. The "Machining Operations Browser" can be used to manage these operations.

1.3 Programming Work-flow

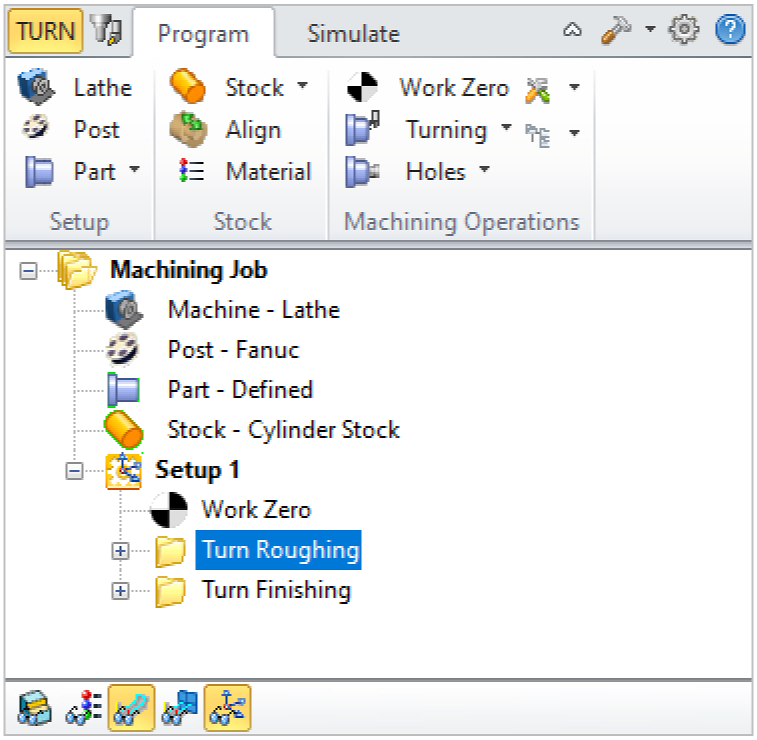
Once the part is loaded, the typical work flow is reflected in the layout of the tabs of the Machining Browser window. The work flow is designed to allow you to work starting from the left most tab and ending at the right most tab. As each tab is accessed, a ribbon toolbar with functions specific to the tab chosen will be displayed just below the tab. The functions in each of the toolbars corresponding to each tab are also best accessed in order from left to right.

Thus you typically would start with the Program tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, you will then proceed to the Machining Operations group to commence programming the part. Once machining operations are completed you can simulate the operations by selecting the Simulate tab before finally sending the operation codes to the machine tool.

Step 1: Program Tab

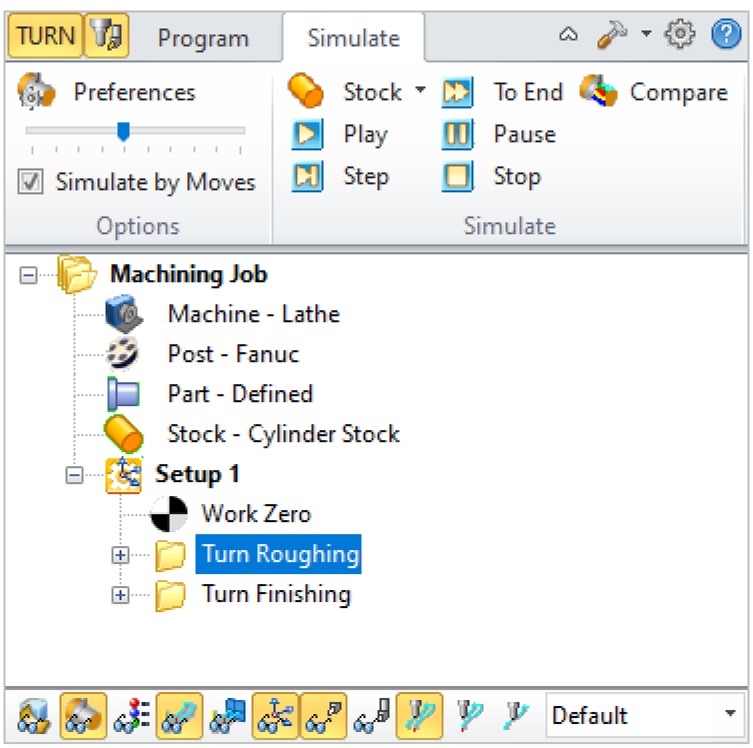
Define Machine Setup and Stock before programming

Create Machining Operations



Step 1: Program Tab

Step 2: Simulate machining operations



Step 2: Simulate machining operations

1.4 Post-Processing

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool.

Quick Start

RhinoCAM®

TURN Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

Quick Start Guides for each RhinoCAM module are available in both PDF and Video format. Refer to the following information to access these guides:

What's New!

What's New in RhinoCAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

How to Access the Quick Start Guide Documents

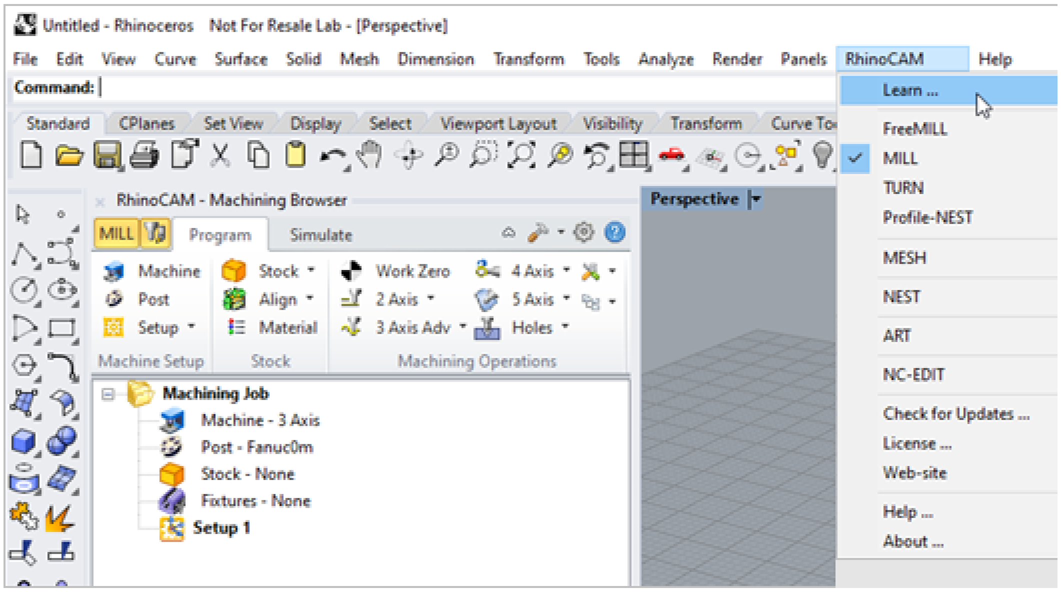
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

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:

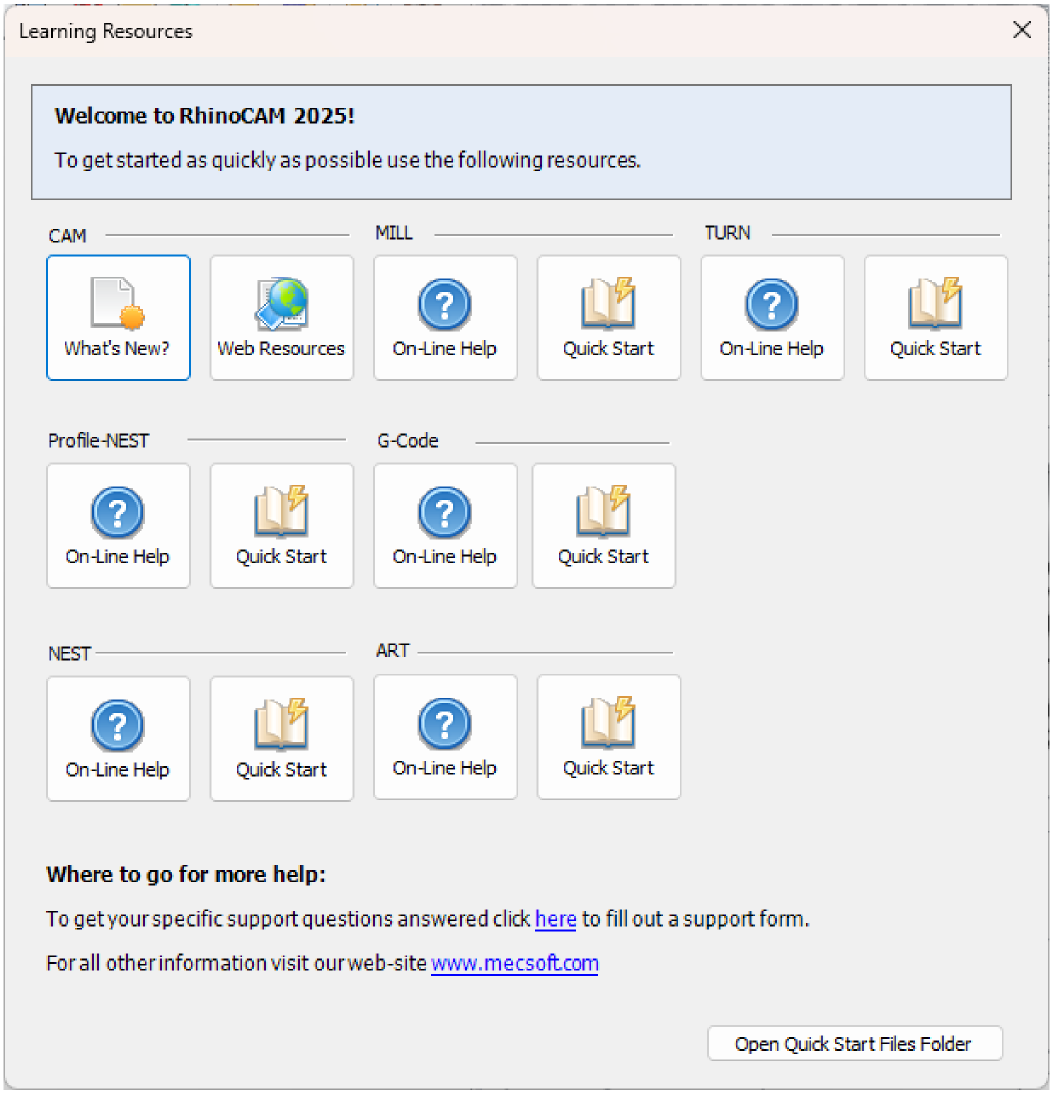
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog 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.

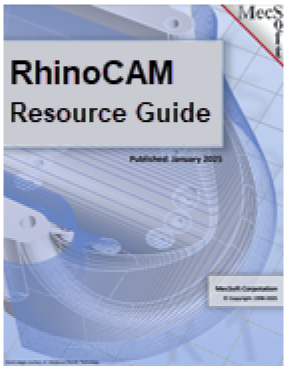


Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM 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!

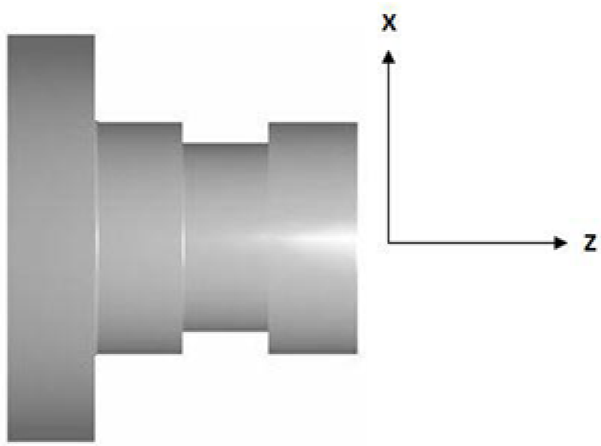
What's New | Quick Start Play List

Key Concepts

Before attempting to use the TURN Module there are a few key concepts that need to be understood. Some of these concepts will be familiar to lathe programmers and are explained here because they are essential for the proper use of TURN Module.

4.1 Turning Coordinate System

CNC turning centers use the Cartesian coordinate system for programmed coordinates but they are typically different from that used in milling. Turning centers follow the convention that axis of rotation that is aligned with the spindle is designated as the Z axis. Secondly the axis perpendicular to this axis along which the tool travels to cut into the stock is designated the X axis. Thus the part is rotated about the Z-axis of the lathe machine. Moving the tool along the Z-axis provides the direction of feed and moving it along the X-axis provides the depth of cut. This is shown below.



Turning Coordinate System

4.2 TURN Module Default View

TURN Module uses the Top view as the default view. The coordinate system of this top view is setup to be aligned with the turning coordinate system. That is the origin of the screen is located at the center of the screen and the Z axis goes from left to right and the X axis goes from bottom to top. This display setup is not typical in design systems where the Top view is aligned with the XY axes of the world coordinate system.

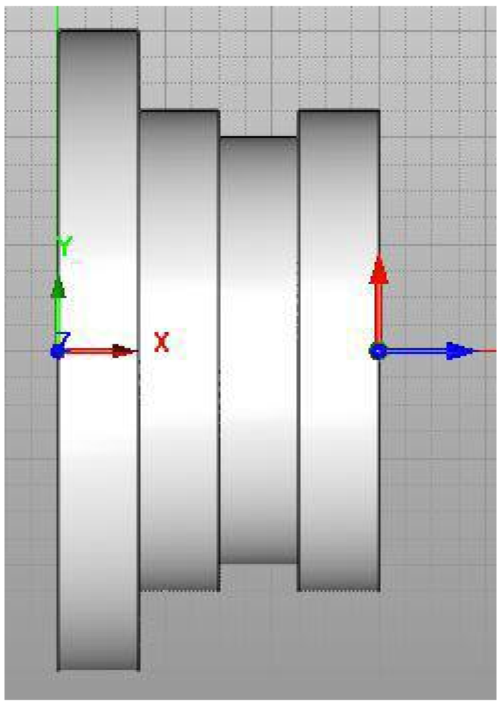
This view setup is used in TURN Module to allow the turning center programmer to work in turning center coordinates rather than in the XY coordinates of the design system. The world coordinate system in VisualCAD is in XY coordinates. All part modeling would still be done in XY coordinates. Loading the turn browser sets up the turn coordinate system.

It should be noted that this convention might sometimes be disorienting for users who are used to visualizing their design parts in the normal XY aligned display rather than the ZX aligned display.

TURN Module's Top view is by default aligned with the ZX turning center coordinate system.

TURN Coordinate System Example

The picture below shows the turn coordinate system with the work zero set the right face of the part and the world coordinates at the origin. The **Red** arrow in the coordinate system represents X coordinate axis, **Blue** represents Z coordinate axis and **Green** represents Y coordinate axis. The triad display for the world coordinate system includes the coordinate axis names to help you differentiate between world and machine coordinates.



4.3 Part Geometry

TURN Module requires Solid, Surfaces, Polygon Meshes, Regions/Curves that define the part geometry. Since all parts that can be created in a 2-Axis turning machine are solids of revolutions, it is enough to describe the profile that needs to be revolved to create this shape. The profile can be created in VisualCAD as a region or curve. TURN Module automatically creates the 2D silhouette of this part region when 3D Solid or Surface geometry is selected as TURN part geometry.

Basic steps

1. Make sure you have either a 2D profile or a 3D surface model visible.
2. From the TURN Program tab, select Part and then pick Select Turn Part.

3. Define the Turn part using the Select Part Geometry for Turning dialog,

The First Quadrant, XY Plane Restriction

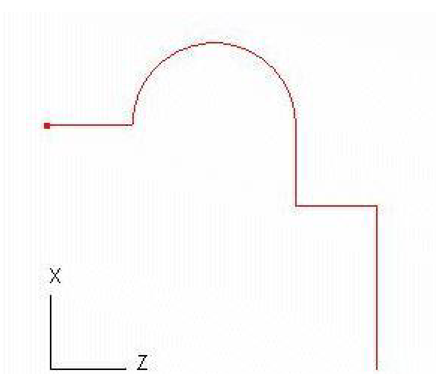
Furthermore, TURN Module places a further restriction that these part geometries need to be constrained to lay ***only*** in the first quadrant of the ZX plane in turn coordinate system. This would essentially be XY plane (Top view) in world coordinate system in VisualCAD.

If Geometry Falls Outside the First Quadrant of the XY Plane

TURN Module will be unable to process a part region that fall outside the first quadrant. If the selected part region is outside the first quadrant, TURN Module will trim this to the first quadrant.

Example of Curves Correctly Positioned

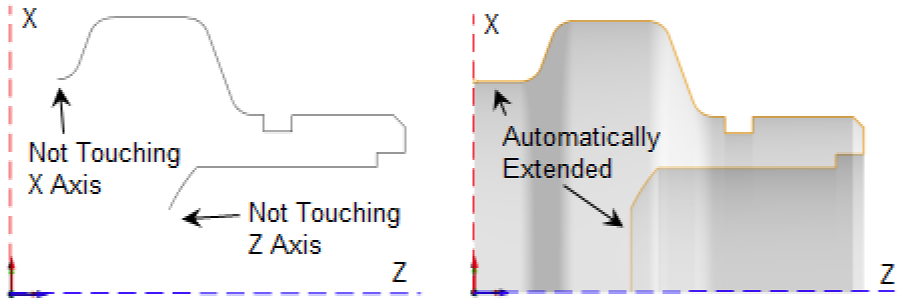
Example below shows curve region correctly positioned in the first ZX quadrant of the turn coordinate system touching both the X and Z axis. This would essentially be XY plane (Top view) in world coordinate system in VisualCAD.



Example of Curves Correctly Positioned within the First Quatrant of the XY Plane

Example of Curves NOT Touching the X or Y Axis

Example below shows region not touching the X axis and/or Z axis



Part Profiles are Automatically Extended to the X and Z axis

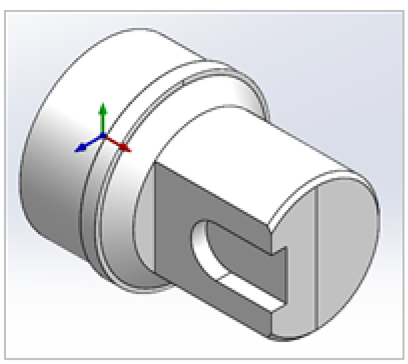
In such cases, TURN Module automatically extends the part regions to the X and Z axis when the regions do not touch the X axis and/or Z axis.

Part regions need to be constrained to the first quadrant of the ZX coordinate system.

Parts can be imported or can be created within VisualCAD using the Geometry creation and editing tools found under the CAD ribbon bar.

TURN Geometry with MILL features

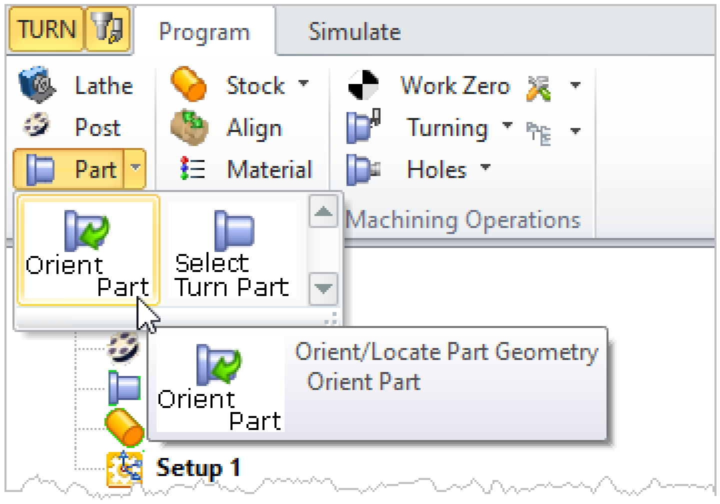
You can select 3D geometry (surfaces, solids or meshes) for defining the TURN part that contains milling features. Such milling features are ignored while the turn part profile is being defined.



4.3.1 Orient Part Geometry

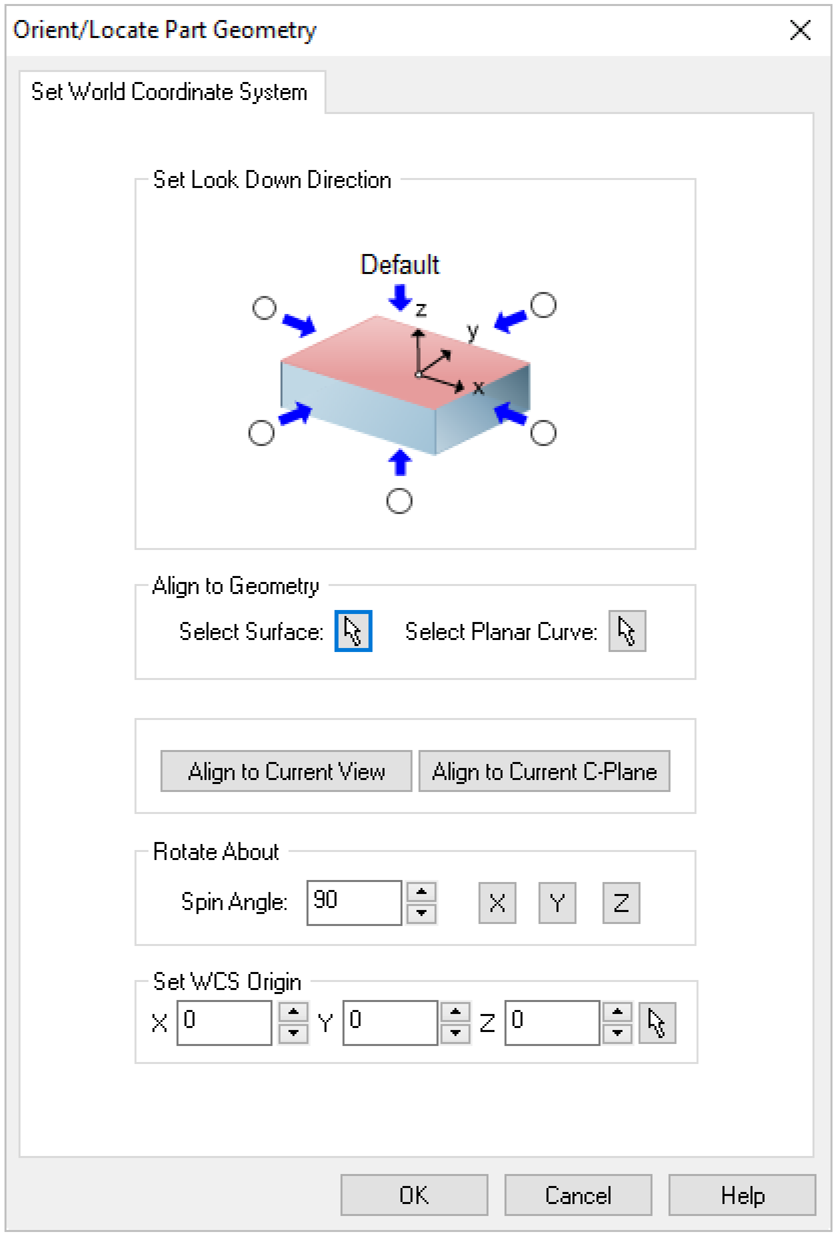
Once part geometry is loaded, you can use this command to change the orientation for turning. It can be invoked by selecting Part and Orient Part from Program tab under the Machining Browser. **Note**: This command changes the WCS orientation of your part. To change the MCS orientation use the Setup command.

Machining Browser: Part, Orient Part



Machining Browser: Part, Orient Part

Dialog Box: Orient/Locate Part Geometry



Dialog Box: Orient Geometry

Set Look Down Direction

Select the radio button representing the orthographic view of your part that you wish to machine. For example, select the right side radio button (on the positive X side in the dialog image) will rotate your part so that the positive X direction becomes the positive Z direction.

Align to

If your part is not aligned orthographically, you can use these options to align your part to selected geometry or active display element. Select from one of the options that will orient the part that you wish to machine:

Select Surface

Choose the Pick button and then select a surface of your part to orient to. The part will be aligned such that the surface normal direction is aligned with the -Z axis.

Select Planar Curve

Choose the Pick button and then select a Planar Curve of your part to orient to. The part will be aligned such that the curve will be parallel to the XY plane (i.e., normal to the Z axis).

Align to Current View

Pick this button to align the part such as the Current View direction is aligned with the Z axis (i.e., you are looking in the -Z direction).

Align to Current C-Plane

Pick this button to align the part so that the Current C-Plane becomes parallel with the XY plane.

Rotate About

Use this option to Rotate About one of the principal XY or Z axis. Enter an angle and then select the button representing the axis you wish to Rotate About.

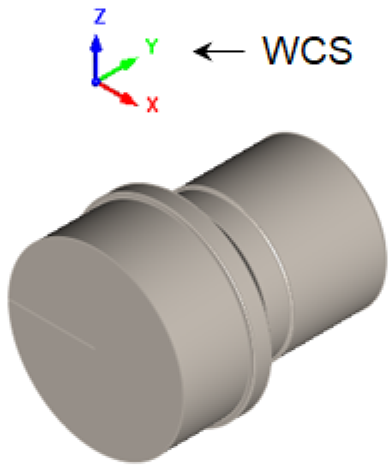
Set WCS Origin

You can also independently set the coordinate location for the WCS. So for example, you can orient the part normal to a surface and then also choose a point on the surface to become the new WCS origin. In this case the part would be oriented to the surface while the point on the surface remains at the WCS.

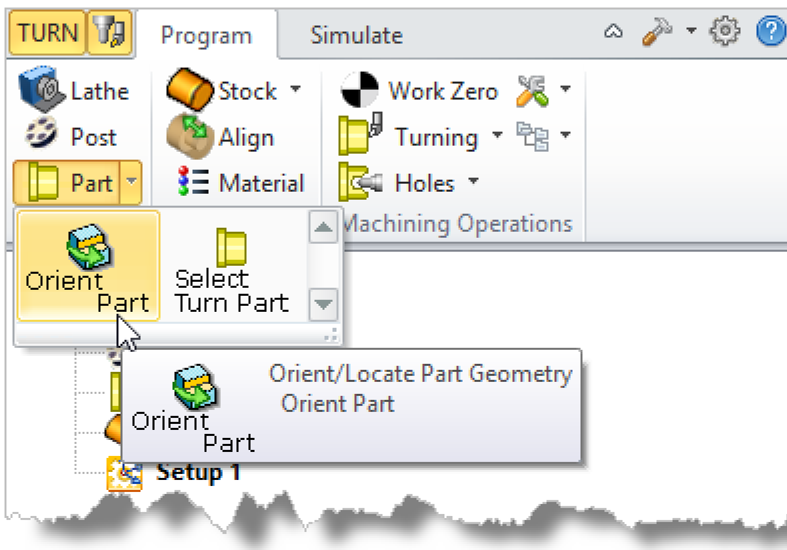
Example: Steps to orient your part for Turning

In TURN, the default rotational axis is along the X Axis of the World Coordinate System or WCS. **Note:** Once you select a direction and pick OK from the dialog, all of the geometry will be moved and/or rotated. In the example part shown below we want to orient the part so that the rotation axis is along the X Axis of the WCS. We also want the WCS origin to be located at the center of the back face of the part. Look at the steps below to see how it's done.

1. From the Program tab select Part and then Orient Part.



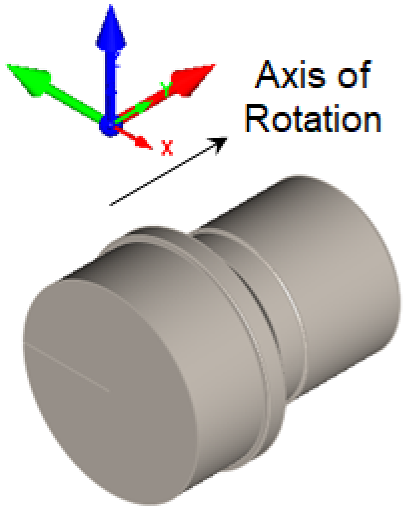
Incorrect Orientation



Machining Browser: Part, Orient Part

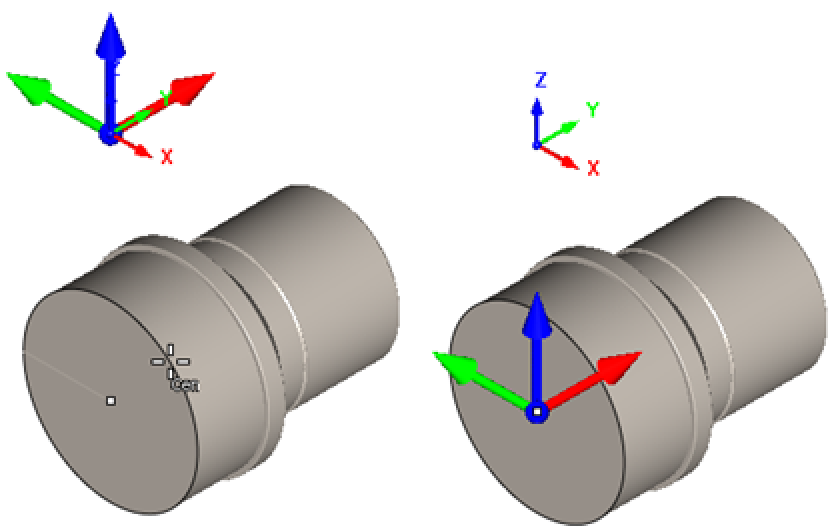
The dialog is displayed and the WCS is also displayed on the screen.

2. Under Rotate About, set the Spin Angle to 90 and then pick the Z button to rotate the WCS displayed on the screen. We want the X Axis of the WCS to point along the rotational axis of the part.



3. Now, let's locate the WCS origin. In the dialog under Set WCS Origin, select the Pick button.

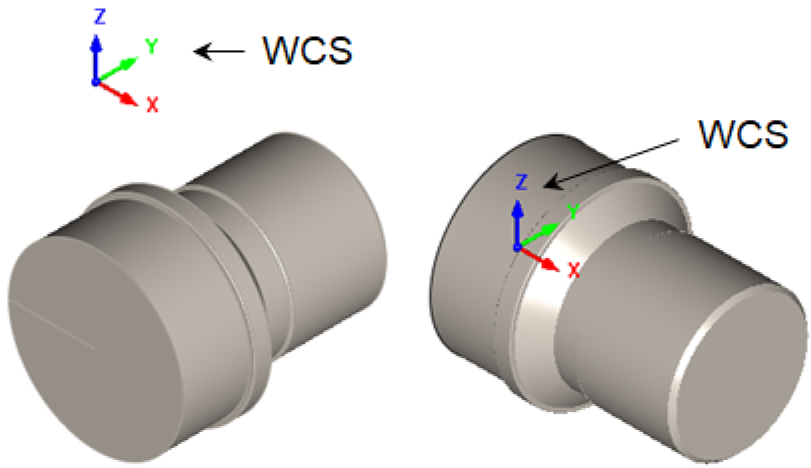
4. Select the center point of the circular face of the back of the part. This is the face that will be mounted on the spindle of the lathe. When you select the point, the WCS triad is move to that point.



Before

After

5. Now pick OK from the dialog and the part is oriented as desired.

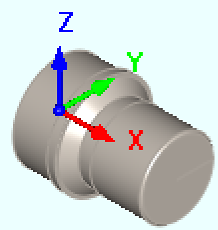


Before After

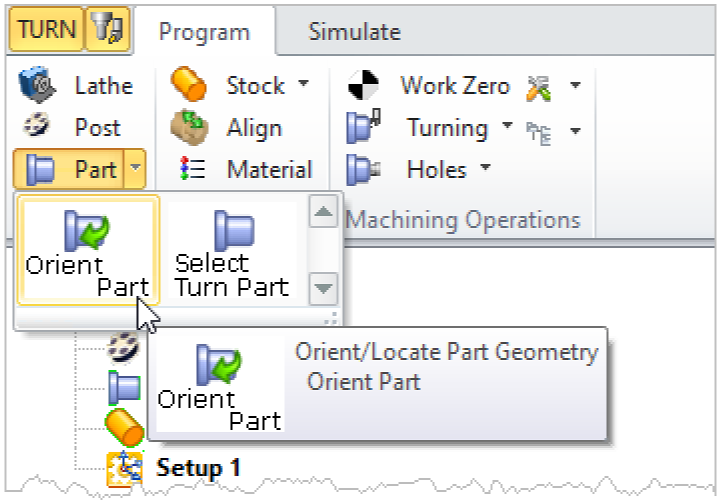
4.3.2 How to Define Part Geometry

Here are the basic steps to define part geometry:

BEFORE selecting part geometry for Turning, make sure it is oriented correctly. The image shows the correct orientation. The turning axis of the part is along the X Axis of the Word Coordinate System (WCS). Also, the WCS is located on the back rearmost face of the part. Orient Part can help you do this.

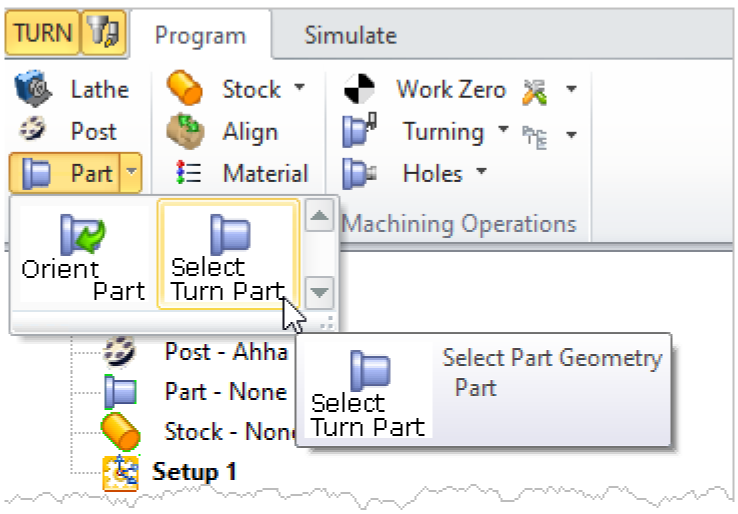


1. Orient your Part if needed.



Machining Browser: Part, Orient Part

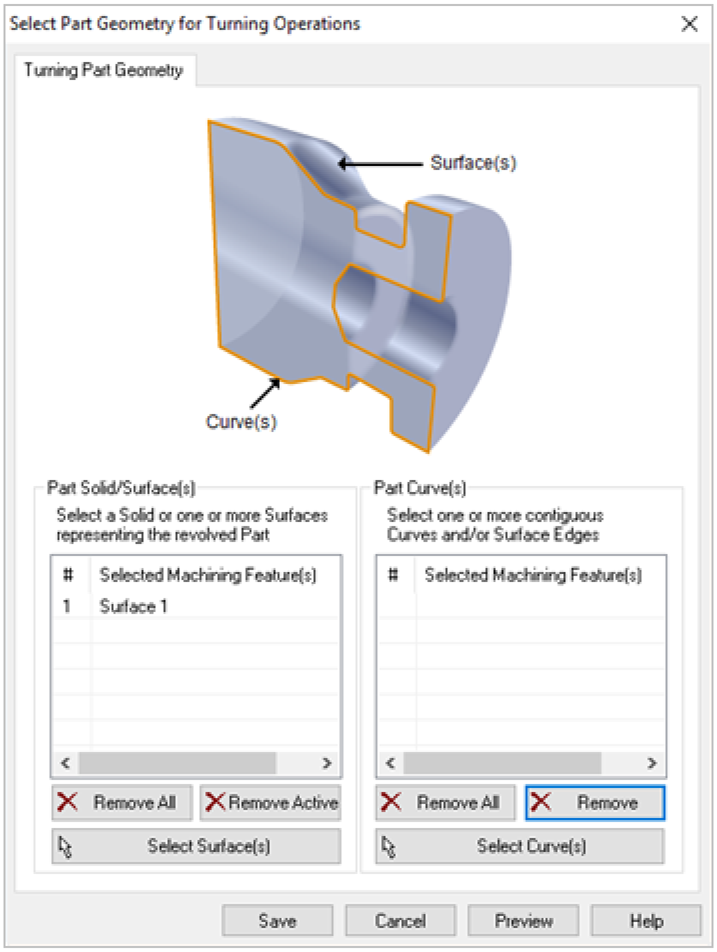
2. Select Turn Part in Machining Browser.



Select Part (Select Part Geometry) under Program tab in Machining Browser

3. Dialog Box: Select Part Geometry for Turning Operations

You can select surfaces or curves to define TURN part geometry.



Dialog Box: Select Part Geometry for Turning Operations

4. Define Part Geometry

There are two lists on this dialog. Use one or the other to define the Turn Part (not both).

If you have 3D geometry, pick the Select Surfaces button. You will be prompted to select part objects. Select the objects to define the part geometry and right-click or press Enter. If you only have a curve profile, make sure it is located in the positive XZ plane. See Part Geometry Types for more information.

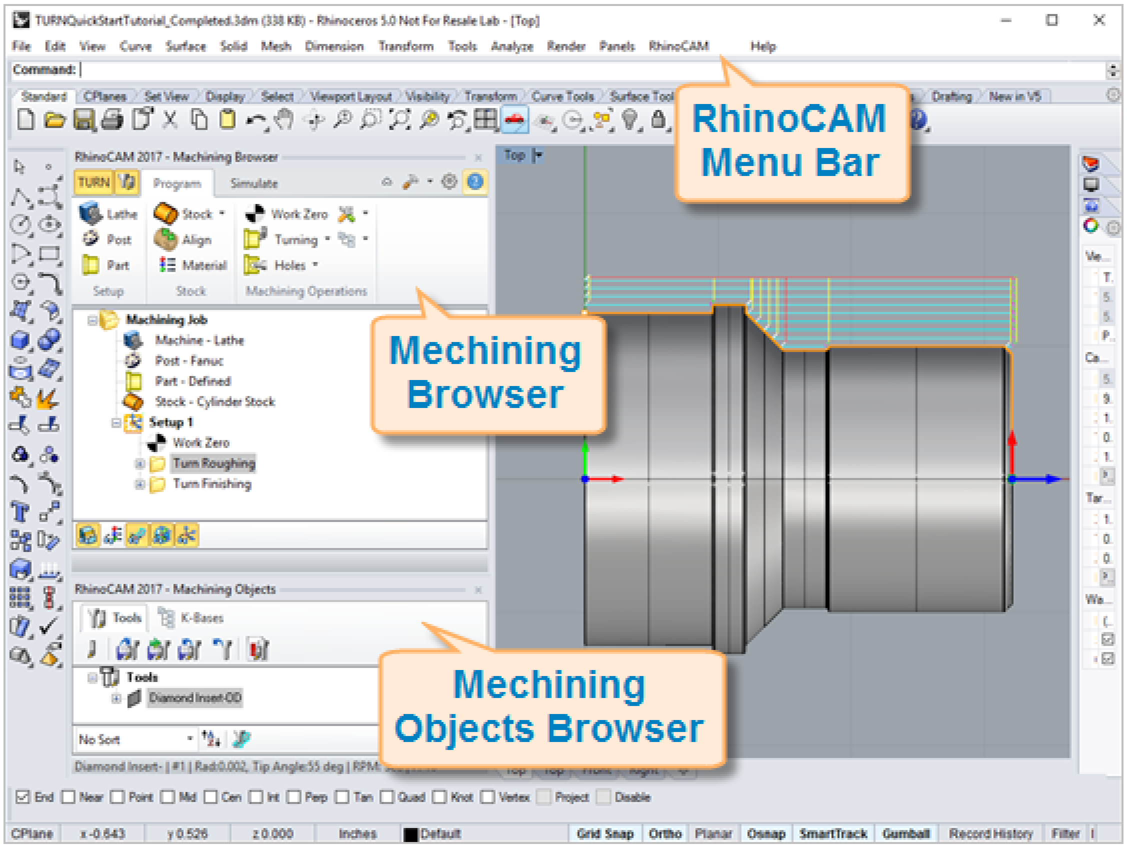
The selected geometry is listed under Selected Machining Feature(s) list.

5. Select Save

When the part is previewed, selecting Save defines the turn part geometry and displays Part-Defined under the machining browser.

User Interface

The RhinoCAM TURN Module adheres to the Windows standard for user interface design and integrated into the Rhino screen seamlessly. A screen shot of TURN Module running inside of Rhino is shown below:

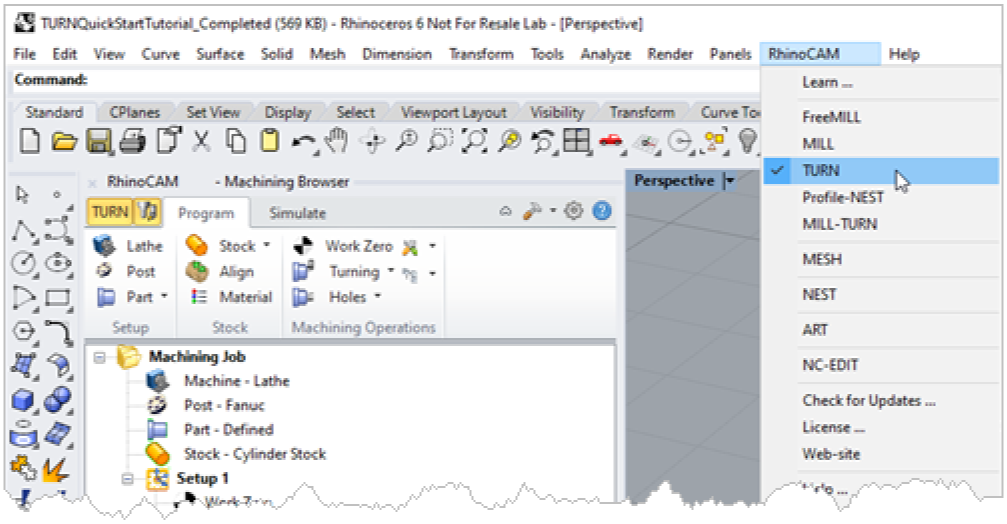


There are 3 main interface objects created when The RhinoCAM TURN Module is loaded.

1. RhinoCAM menu bar entry under Rhino menu bar.
2. Machining Browser window.
3. Machining Objects Browser window.

5.1 RhinoCAM Menu Item

When RhinoCAM is loaded it will add a menu bar item, titled RhinoCAM 2026 to the main Rhino menu bar. Selecting this menu bar item will create a drop down menu as shown below. To run TURN Module, select TURN from the RhinoCAM drop down menu. Selecting TURN toggles the display of the Turning Browser window from the Rhino user interface . If the FreeMILL, MILL, NEST or ART Browser is currently open selecting this will switch to the TURN Browser.

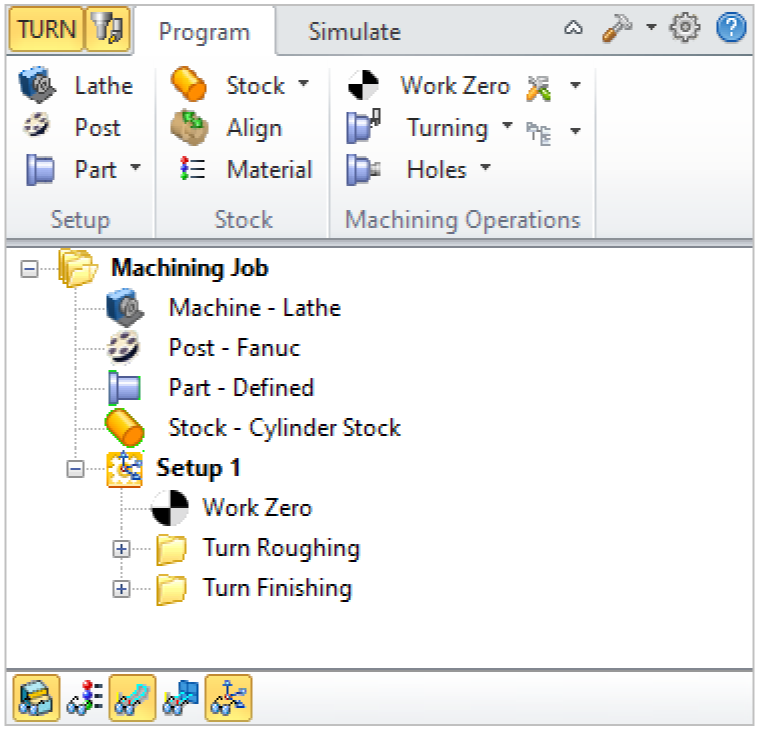


5.2 TURN Browsers

The Browser is a dock-able window that allows management of various entities or objects that can be created in TURN Module. There are 2 browsers in RhinoCAM – Machining Browser (Mops) and Machining Objects (Mobs).

5.3 Machining Browser (Mops)

The Machining Browser (Mops) has two main modes of operation represented by tabs at the top of the window. These are Program and Simulate. Each tabbed view also incorporates a ribbon toolbar at the top. These toolbars group all of the functions associated with the type of object in the tab.

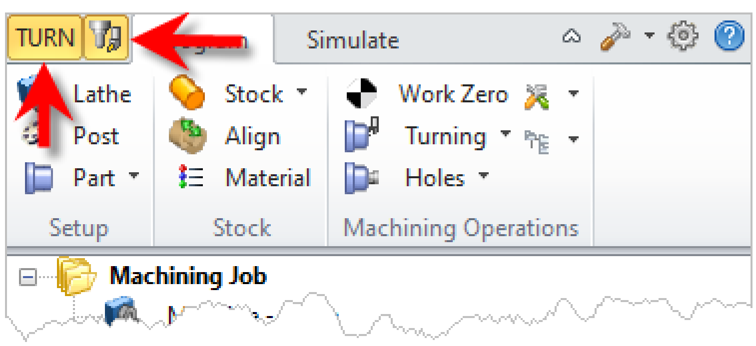


Machining Operations Browser

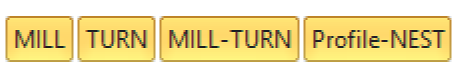
5.3.1 Browser Toggle Tabs

The tabs available on the Machining Browser allow you to toggle the display of both the Machining Browser and the Machining Objects Browser.

Browser Toggle Tabs



Locating the Browser Toggle Tabs



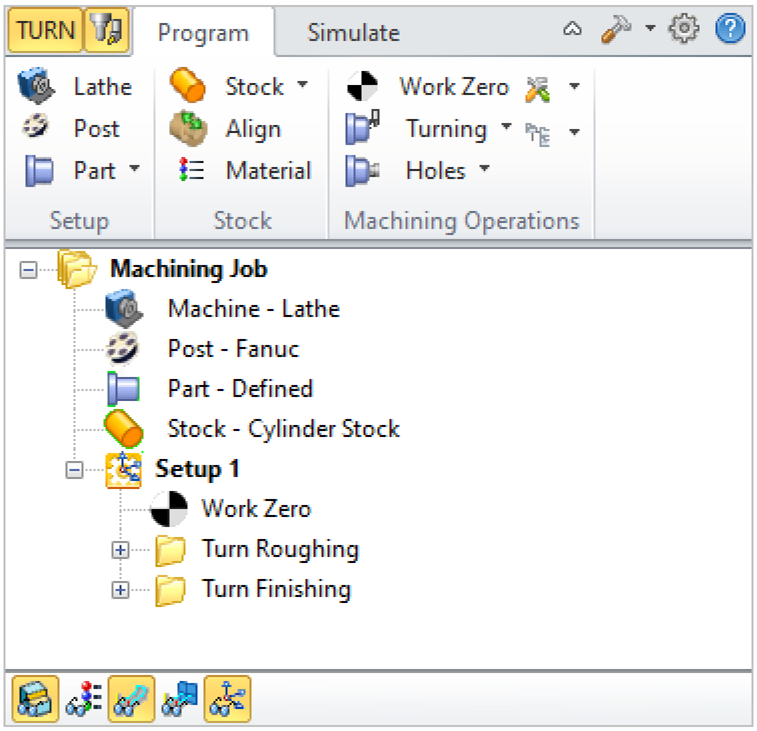
Selecting this tab toggles between the MILL , TURN, MILL-TURN and Profile-NEST Machining Browser.

Select this tab to toggle the display of the Machining Objects Browser.

5.3.2 Program tab

Selecting the Program tab in the Mops Browser shows 3 groups of menus that provide access for specifying Machine, Stock and defining Machining Operations.

Machining Browser, Program Tab



Machine Setup Group

Sets the Machine for Turning. Allows user input to set Tool change position, maximum RPM and travel limits.

Allows you to set the Current Post Processor, posted file naming conventions, posted file extension, program to display the posted file.

Provides access to commands that allow you to Orient and Define your Part geometry for TURN programming. See Orient Part Geometry and Part Geometry

Machining Stock Group

Allows you to create Stock geometry. User can also delete a Stock geometry by selecting Delete Stock.

Aligns stock model to part. This function is especially useful when the part model and the stock model are created without regard to their respective positional locations.

Allows you to select a material from the material list.

Machining Operations Group

This section allows user to create machining operations. TURN Module allows you to create multiple machining operations in a part file. This is a powerful feature that allows you to create an entire sequence of machining operations that is necessary to create the part model from the stock model. This set of operations can additionally be archived with the part file and retrieved at a later time with no loss of information.

Allows you to set the Work Coordinate zero (Origin) for the part being programmed.

Provides access to Turn Roughing & Finishing, Groove Roughing & Finishing, Threading and Parting Off machining methods

Provides access to Drilling, Tapping Boring and Reverse Boring Machining Methods

Allows you to create Machining operation sets, Machine Control Cycles.

Allows saving and loading of Machining operations to and from a knowledge base.

Provides access to G Code Editor and Post process generator.

Provides access to specify Color, User Interface, Machining, Simulation and Feeds Speeds Preferences.

Minimizes the Ribbon Bar.

Maximizes the Ribbon Bar.

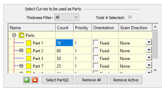
Open On-line Help document.

The Status bar on the Program tab

The toolbar above status bar on the Program tab has the following controls

 Turn on/off Stock Model

 Turn on/off material Texture Visibility

 Turn on/off toolpath display

 Hidden Toolpath Visibility: Turn the hidden portions of toolpaths on/off.

 Turns on/off of World Coordinate System display.

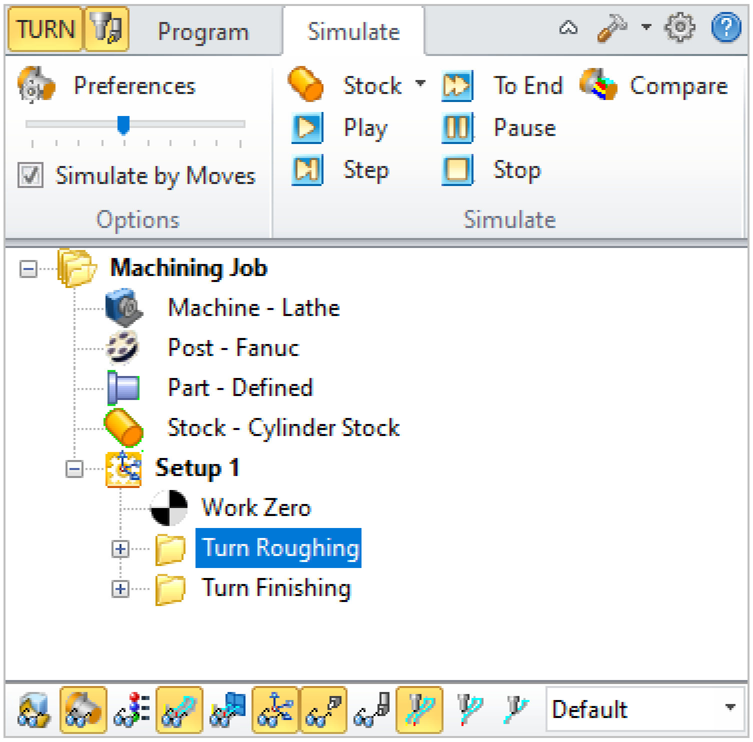
 Turns on/off of Machine Coordinate System display.

The status bar displays the currently selected tool, spindle speed and cut feedrate.

5.3.3 Simulate tab

Selecting the Simulate tab allows you to run the material simulation and toolpath animation. This tab also provides controls to vary simulation speed and set simulation preferences.

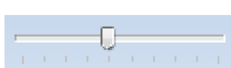
Show me the Machining Browser, Simulate Tab



Machining Browser, Simulate Tab

The following controls are available on the Simulate tab.

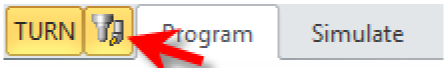
Simulate Tab Functions

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Stock	✓	✓	✓	✓	✓
	Create Stock Model: Allows you to create Stock geometry. You can also delete a Stock geometry by selecting Delete Stock.				
Play	✓	✓	✓	✓	✓
	Perform Toolpath Simulation or Animation: Allows you to perform cut material simulation with tool animation.				
Step	✓	✓	✓	✓	✓
	Simulate Next Toolpath Block: Simulation is performed in steps as defined by the display interval in the simulation preferences.				
Step Levels	✓	✓	✓	✓	✓
	Simulate Next Toolpath Z Levels: Simulation is performed in separate Z levels.				
To End	✓	✓	✓	✓	✓
	Simulate to End: Simulation is performed without updating the display until the end of the toolpath.				
Pause	✓	✓	✓	✓	✓
	Pause Toolpath Simulation: Pause/Stop the simulation.				
Compare			✓	✓	✓
	Part Stock Compare*: Compare the simulated model with the part geometry. The part geometry must contain surface/solid/mesh geometry.				
Stop	✓	✓	✓	✓	✓
	Stop Toolpath Simulation: Exits Simulation Mode. Pause simulation before exiting simulation mode.				
	✓	✓	✓	✓	✓
	Simulation Speed: Varies simulation speed				
Preferences	✓	✓	✓	✓	✓
	Set Simulation Preferences: Provides access to simulation preferences.				
Simulate by Moves	✓	✓	✓	✓	✓
	Simulate by Moves: Switches from Simulate by Distance to				

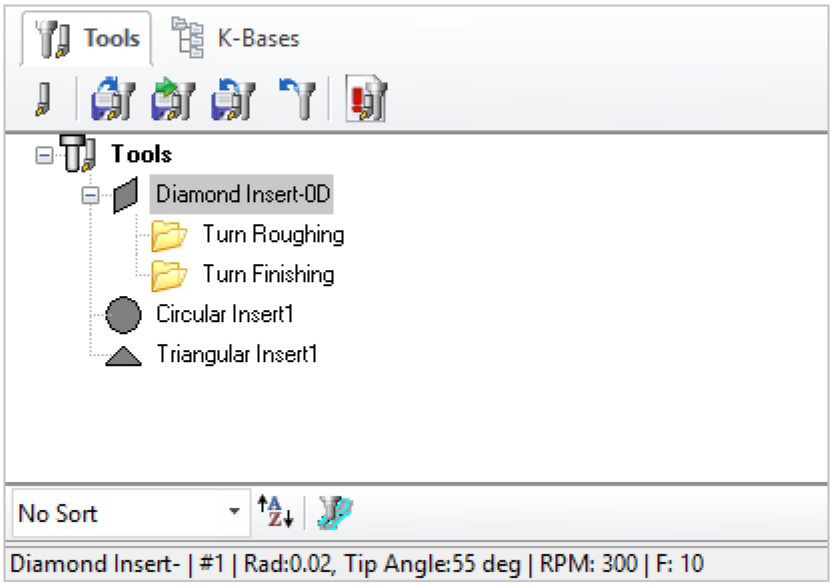
Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
	✓	✓	✓	✓	✓
	Part Model Visibility: Turn on/off part model display during simulation.				
	✓	✓	✓	✓	✓
	Stock Model Visibility: Turn on/off stock model				
	✓	✓	✓	✓	✓
	Material Texture Visibility: Turn on/off material texture visibility				
	-	✓	✓	✓	✓
	Fixture Model Visibility: Turn on/off stock model				
	✓	✓	✓	✓	✓
	Toolpath Visibility: Turn on/off toolpath display				
	✓	✓	✓	✓	✓
	Hidden Toolpath Visibility: Turn the hidden portions of toolpaths on/off.				
	✓	✓	✓	✓	✓
	World CSYS Visibility: Turns on/off of World Coordinate System display.				
	✓	✓	✓	✓	✓
	Machine CSYS Visibility: Turns on/off of Machine Coordinate System display.				
	✓	✓	✓	✓	✓
	Tool Visibility: Turn on/off tool display during simulation.				
	✓	✓	✓	✓	✓
	Holder Visibility: Turn on/off tool holder display during simulation				
	-	-	-	✓	✓
	Machine Tool Visibility**: Turn on/off machine tool display during simulation				
	✓	✓	✓	✓	✓
	Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes.				
	✓	✓	✓	✓	✓
	Trace Toolpath Display: The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the				

5.4 Machining Objects (Mobs) Browser

Machining Objects (Mobs) browser has two main modes of operation represented by tabs at the top of the window. These are Tools and K-Bases. Each tabbed view also incorporates a toolbar at the top. These toolbars group all of the functions associated with the type of object in the tab.



To toggle the display the Machining Objects Browser select Tools, Machining Objects Browser button located on the Machining Browser just to the left of the Program tab.

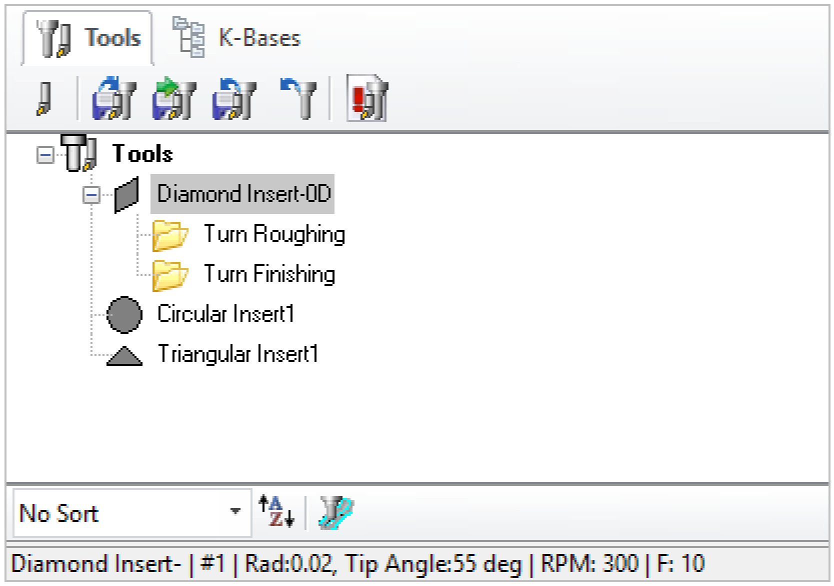


Machining Objects (MObs) Browser

5.4.1 Tools tab

Selecting the Tools tab under the Machining Objects Browser brings up the tool manager. The tool manager lists all of the tools currently defined as well as the tools that are in use in machining operations. Users can edit a tool by double clicking the tool button in the browser. A tool can be deleted by selecting the tool from the Tools browser, right click cut or use the delete key from the keyboard.

Show me the Machining Objects Browser, Tools Tab



Machining Objects (MObs) Browser

RhinoCAM supports 2 types of tool library file format *.vkb and *.csv.

- *.vkb format saves and loads tools with the feeds and speeds assigned for each tool.
- *.csv format saves and loads tools without the feeds/speeds assigned for each tool.

Tools Tab Functions

Create/Edit Tools

This button brings up the tool dialog that enables the creation and saving of the desired tool. All turning inserts and drilling tools can be created here. Refer to the Tool section for a detailed description on creating tools and defining tool parameters.

Load Tool Library

The load tool library button enables the loading of a previously saved tool library. Refer to the following section for additional information - Load Tool Library

Select Tools from Library

The select tool library button enables users to select tools from a previously saved tool library. Refer to the following section for additional information - Select Tools from Library

Save Tool Library

This button enables the created tools to be saved in a tool library file. The file can be saved in the desired directory and read in when required. Refer to the following section for additional information - Save Tool Library

Unload Tool Library

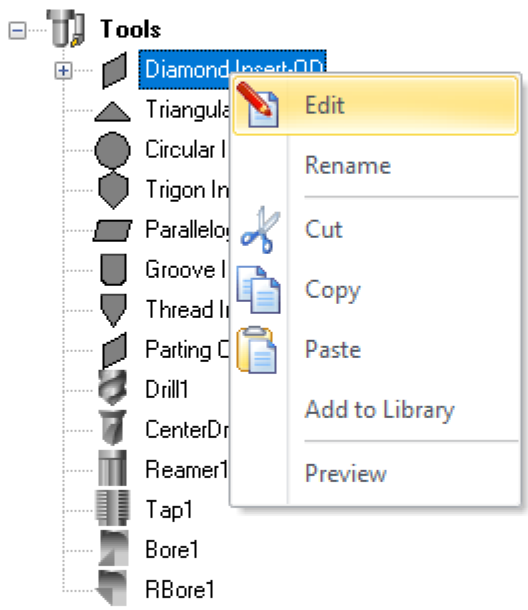
This button will unload the current Tool Library. Refer to the following section for additional information - Load Tool Library

List Tools

The button brings up all the tool properties associated with the tools currently recorded in the current TURN Module session. Refer to the following section for additional information - List Tools

Right-click Options on Tools

You can right-click on a Tool listed in the Mobs Browser to perform various functions. These are listed below:



Right-click Options on a Tool

Edit

Displays the Create/Edit Tool dialog allowing you to edit the Tool parameters.

Rename

Allows you to Rename the selected tool.

Cut / Copy / Paste

These options allow you to Cut or Copy the selected Tool to the Windows Clipboard and then Paste it back to the Tools list to create a new tool using the previous tool as a template.

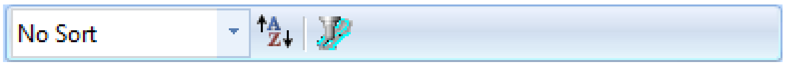
Add to Library

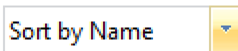
This allows you to Add the selected Tool to an exiting Tool Library *.csv data file.

Preview

This will display a Preview of the selected Tool in the Graphics Window similar to how the Tool displays during Simulation.

Tools Toolbar Functions

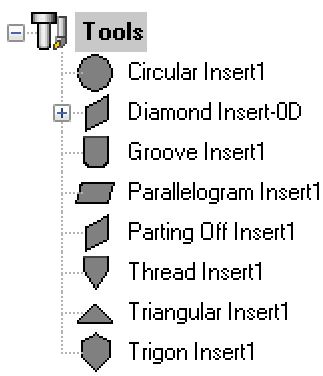




Sorting Selector: This allows you to sort the tool list. You can select No Sort or you can sort by Name, Number, Type and Diameter.

 Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.

 List on the Tool used in Machining Operations: Toggle this icon to list ONLY the tools currently assigned to an operation. **Note**: You must Generate an operation for the assigned tool to be listed.

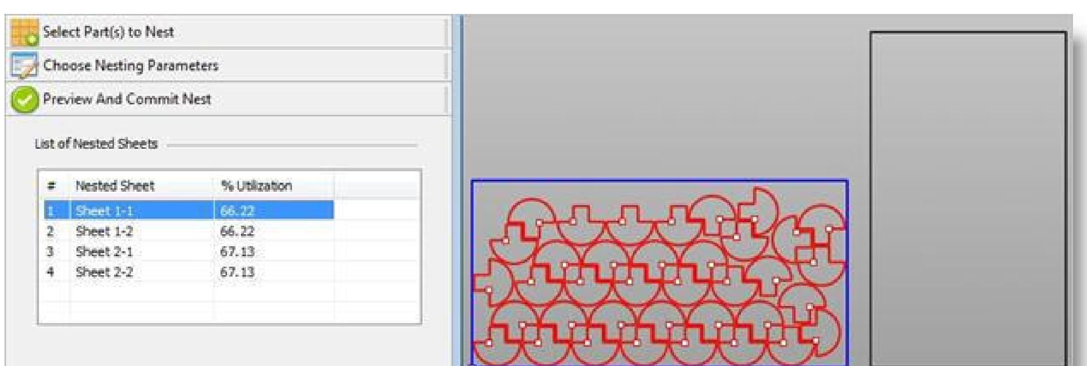


Tools Sorted by Name

Tools Sorted by Used

Tools Status Bar

The status bar displays the currently selected tool, tool tip radius & angle, spindle speed and cut feedrate.

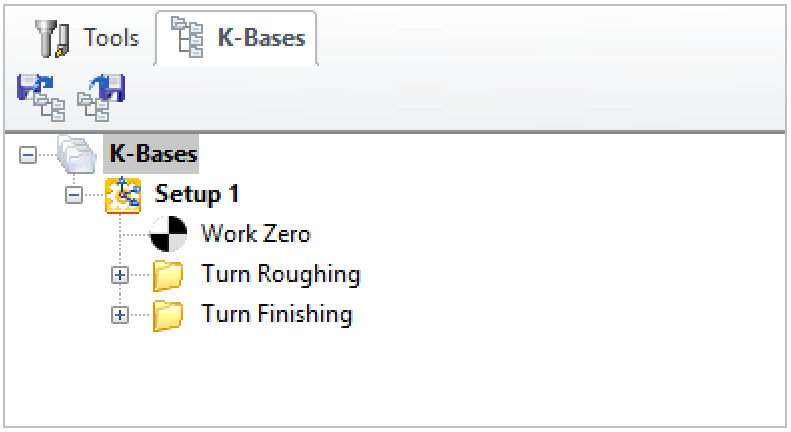


Right mouse button click on Tool in Tools tab provides the following options Edit, Rename, Cut, Copy, Paste and Preview tool.

5.4.2 K-Bases tab

Selecting the K-Bases tab under the Machining Objects Browser displays the Knowledge Base manager.

The Machining Objects (Mobs) Browser, K-Bases Tab



The Machining Objects (MObs) Browser, K-Bases Tab

K-Bases Tab Functions

 Load Knowledge Base: Allows you to select a machining operations knowledge base to load.

 Save Knowledge Base: Allows saving of knowledge bases which can be archived and used across other files.

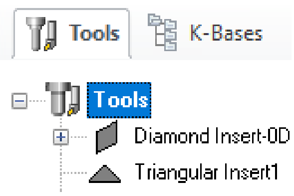
5.5 Right-Click Commands

You can perform a variety of commands by right-clicking on items within the Machining Browser and Machining Job. The tables below lists each of the available commands.

from the Machining Operations (MOps) Browser

RhinoCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Job Machining Job Machine - Lathe Post - Fanuc Part - Defined Stock - Part Cylinder Stock **Setup 1** Work Zero Turn Roughing Turn Finishing	Regenerate All
	Post All
	Simulate All
	Save to Knowledge Base
	Information
	Shop Documentation
	Delete All
Machine - Lathe	Machine Coordinate System
Post - Fanuc	Set Post Options Dialog
Stock - Part Cylinder Stock	Cylinder Stock Dialog
	Part Cylinder Stock
	Offset Stock
	Revolve Stock
	Stock from Selection
	Export Stock to STL
	Delete Stock
	Remove Simulations
Setup 1	Regenerate
	Post
	[illegible]
	Suppress
	Cut
	Copy
	Paste
	Clone
	Save to Knowledge Base
	Save As Defaults
	Properties

from the Tools tab of the Machining Objects (MObs) Browser

RhinoCAM Right-Click Commands - MILL Module	
Machining Objects Browser / Tools Tab 	Simulate
	Simulate Until Cut
	Paste
	Tool List
	Edit
	Rename
	Cut
	Copy
	Paste
	Add to Library
	Preview

5.6 Docking Browsers

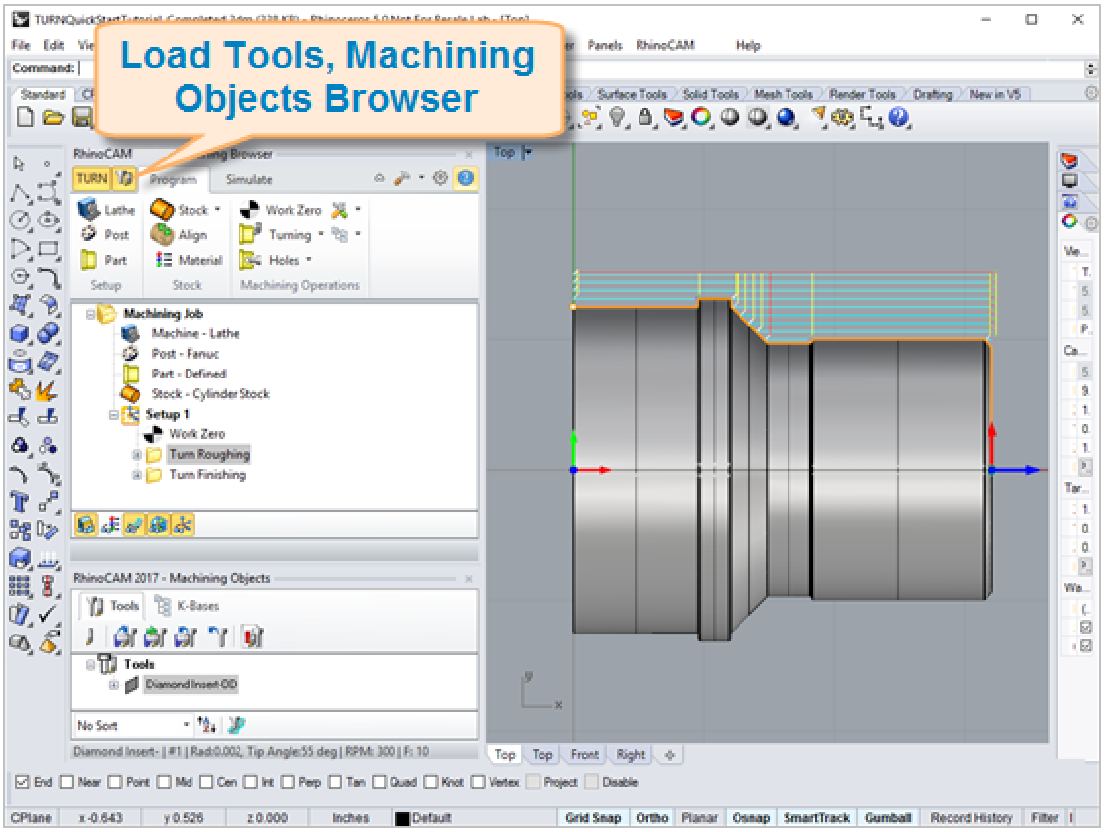
Both Machining Operations Browser and Objects Browser windows are dockable windows. This means these windows can be docked in any position in Rhino. This section describes the procedure to be used to dock both of these windows such that they are stacked vertically.

Step 1: Launch the TURN Browser

Select RhinoCAM from the menu bar and click TURN Browser. This displays the machining operations browser and by default is docked to the left half of the application window next to the view bar.

Step 2: Display the Tools, Machining Objects Browser

Select Tools, Machining Objects Browser button located on the Machining Operations Browser just to the left of the Program tab. This displays the Machining Objects Browser next to the operations browser.



Display the Tools, Machining Objects Browser

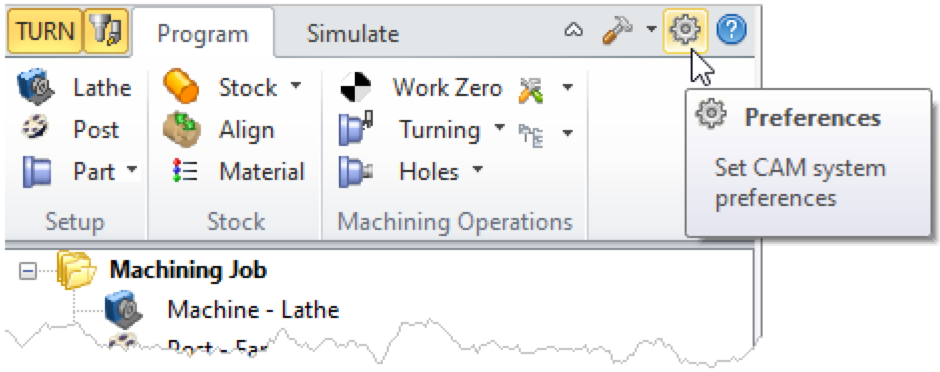
Step 3: Drag & Drop the Browser

Selecting the title bar and holding the left mouse button down and dragging the browser window allows you to dock the browser to desired location. You can dock a browser inside of another browser or have them docked side by side.

5.7 CAM Preferences

You can set various CAM Preferences that will be saved even after you exit the program. Select the Preferences icon from the Machining Browser. When you install a new RhinoCAM update you are choose to import your CAM Preferences from one version to the next.

The CAM Preferences Icon



The available Preferences include:

User Interface
Includes General, Stock Information and Ribbon Style preferences. Show the dialog

Geometry
Includes color preferences for Regions and Surfaces. preference.

Stock
These include stock colors, stock edge display and stock transparency. Show the dialog.

Cutting Tools
Includes Tool colors, Tool display states and the default Tool Library preferences. Show the dialog.

Feeds & Speeds
Includes Feeds & Speed preferences such as default values and other options. Show the dialog.

Machining
Includes Arc Output, Drill Cycle Output, Toolpath Resolution and the default machining Knowledge Base preferences. Show the dialog

Toolpath
Includes Toolpath Colors and Toolpath Display preferences. Show the dialog

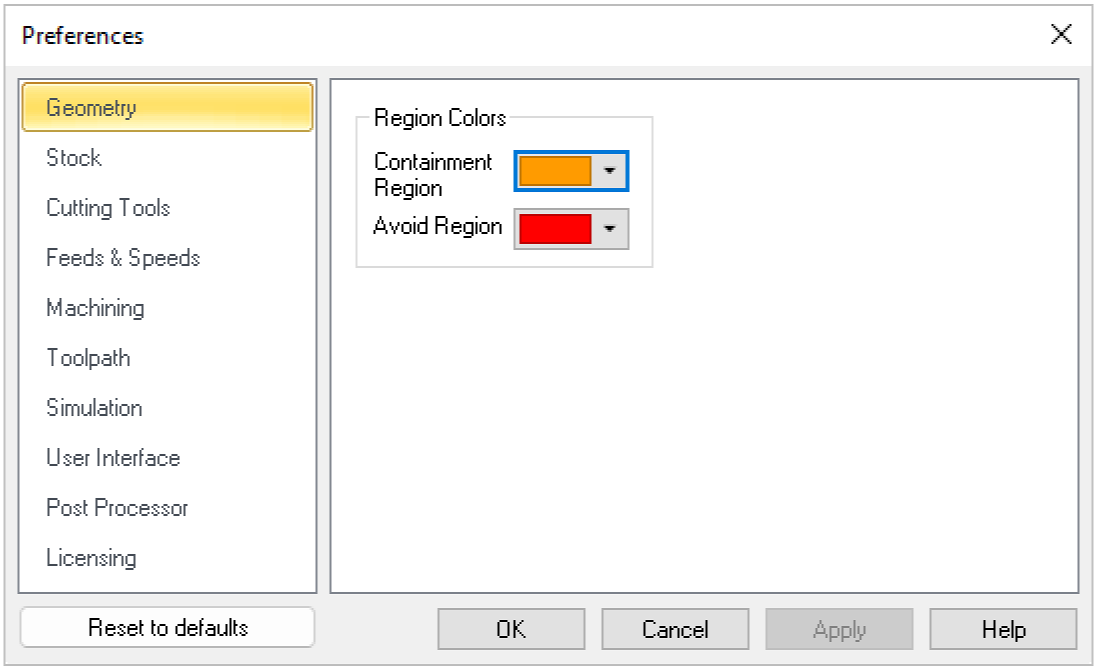
Simulation
Includes Simulation Type, Mode, Accuracy, Transparency and other preferences. Show the dialog

Licensing
Includes network licensing preferences. Show the dialog

5.7.1 Geometry

You can set the colors to display various objects using this dialog. To change each of the color settings in this dialog select the colored button next to the item of interest. This will bring up the color selection dialog, which can be used to choose the color needed. Once a color has been selected the button will change its color to the selected one. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Geometry Dialog



CAM Preferences > Geometry

Region Colors

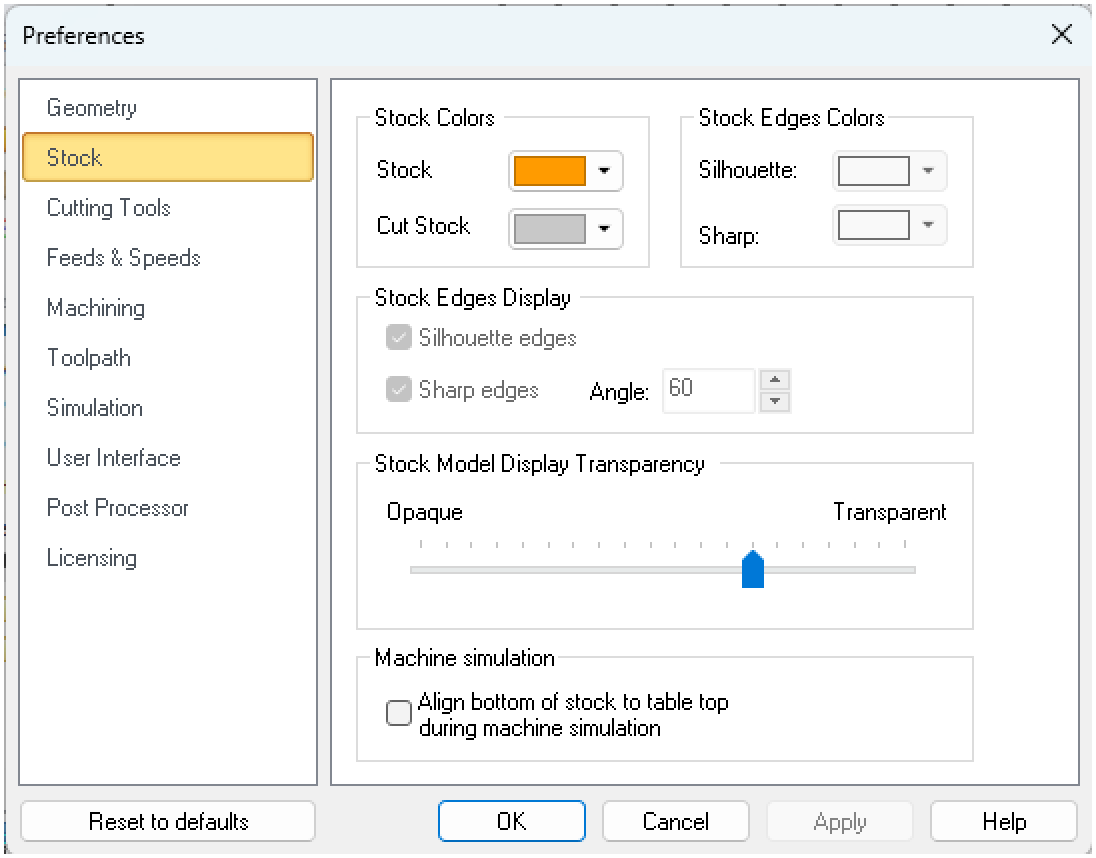
Sets the region highlight color for selection of containment and avoid regions.

5.7.2 Stock

You can set the simulation preferences using this dialog. **Note**: Some options are not available in XPR (Xpress) configuration. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > Stock

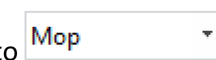
Users can set the simulation preferences using this dialog:



CAM Preferences > Stock

Stock Colors / Stock Edge Colors

Here you can set the Stock and Stock Edge colors. You can differentiate between cut and non-cut areas by specifying different colors for them here.

Note: If the Simulation Display State is set to 

then the Color assigned using the Machining Operation Properties is used to display the cut stock. Right-click on an operation in the Machining Job tree and select Properties to set this color.

Stock Edges Display

This section allows you to control the Stock Edges Display states. For example, you can check the boxes to display Silhouette Edges and Sharp Edges as well as the Angle to display for stock edges. Silhouette Edges and Sharp Edge colors are set using the Colors section of this dialog. Experimentation is advised until you are comfortable with the way your stock display.

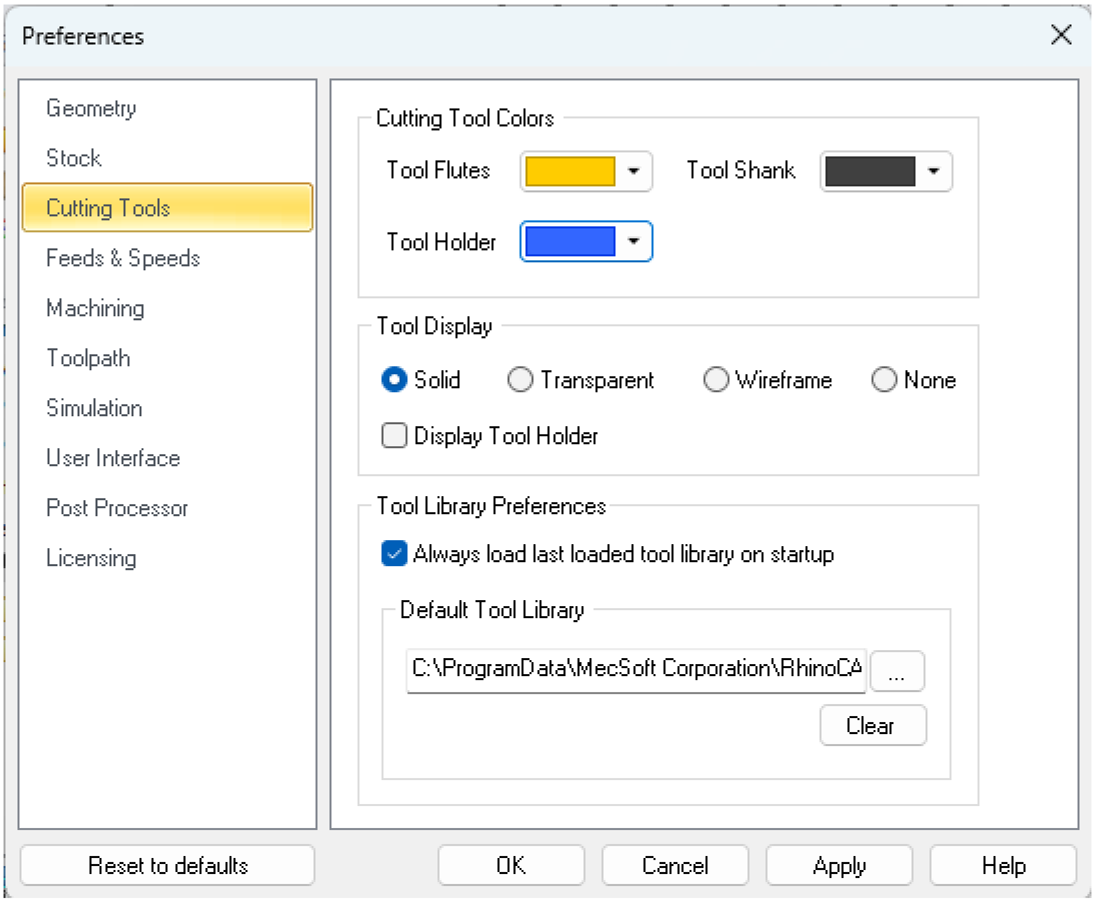
Stock Model Transparency

Use this slider to adjust the Stock Model Transparency when the Program tab is selected (i.e., when you are not simulating).

5.7.3 Cutting Tools

You can set the Tool Library to load on startup and also specify the location of your Tool Library files. You can use the Reset to defaults button if you want to revert to the default factory install settings.

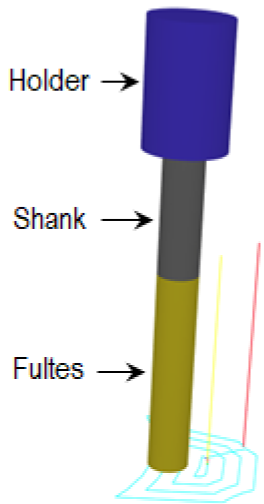
CAM Preferences > Cutting Tools



CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

Cutting Tool Colors

Use the color selectors to set the default display colors for the cutting tool. The Tool Flutes, Tool Shank and Tool Holder can each be assigned a different.



Tool Display

The cutting tool can be displayed as either Solid, Transparent, Wireframe or None by selecting the desired option. You can also toggle the display of the Tool Holder by checking or un checking the box provided.

Tool Library Preferences

This defines your Tool Library preferences:

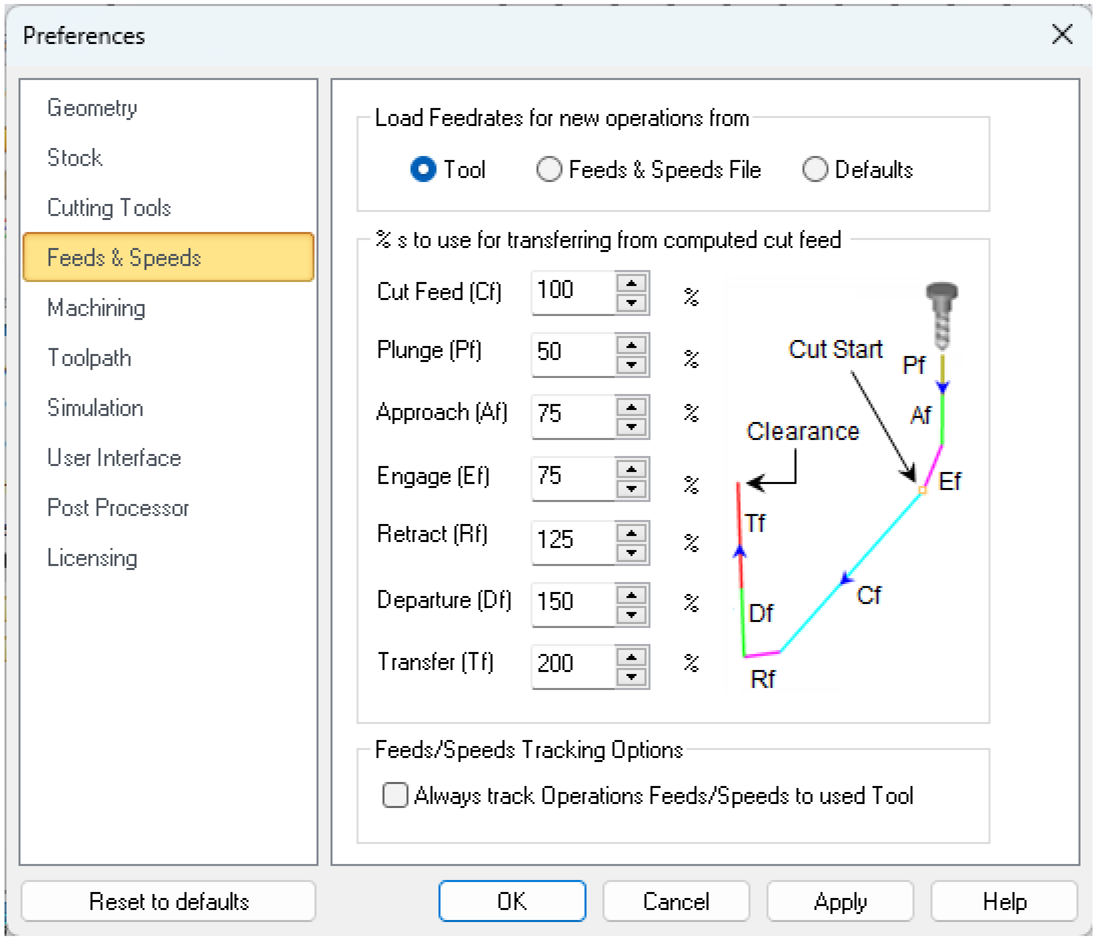
Always load last loaded tool library on startup
If you check this box, every time RhinoCAM loads, the last loaded Tool Library will be loaded automatically.

Default tool library path
Optionally you can specify the file path for your default tool library files. **Note**: It is recommended that you save your custom tool library files to a location outside of the RhinoCAM install path. This will keep them from being overwritten when you install new updates of RhinoCAM.

5.7.4 Feeds & Speeds

You can set the Feeds & Speeds preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Feeds & Speeds



CAM Preferences > Feeds & Speeds

Load Feedrates for operations from

This allows you to select a preference option for loading Feeds/Speeds from table or from tool or use defaults when creating a new operation.

Tool
Selecting this option loads the feeds/speeds saved with the tool when creating a new operation.

Table
Selecting this option loads the feeds/speeds based on the material selected when creating a new operation.

Defaults
Selecting this option loads the feeds/speeds from the default knowledge base when creating a new operation. If default knowledge base is set to undefined, the system defaults would be used for loading feeds and speeds.

% s to use for transfer from computed cut feed

These % values apply when using the Load from File option (i.e., commonly referred to as the Feeds & Speeds Calculator) from either the Create/Edit Tools dialog of from the Feeds & Speeds tab of any of the toolpath operation dialogs. 100% of the Cut Feed specified in this dialog is applied and a percentage of the Cut Feed is used to populate the remaining feedrates for Plunge, Approach, Engage, Retract, Departure and Transfer. You can set the % values to use here.

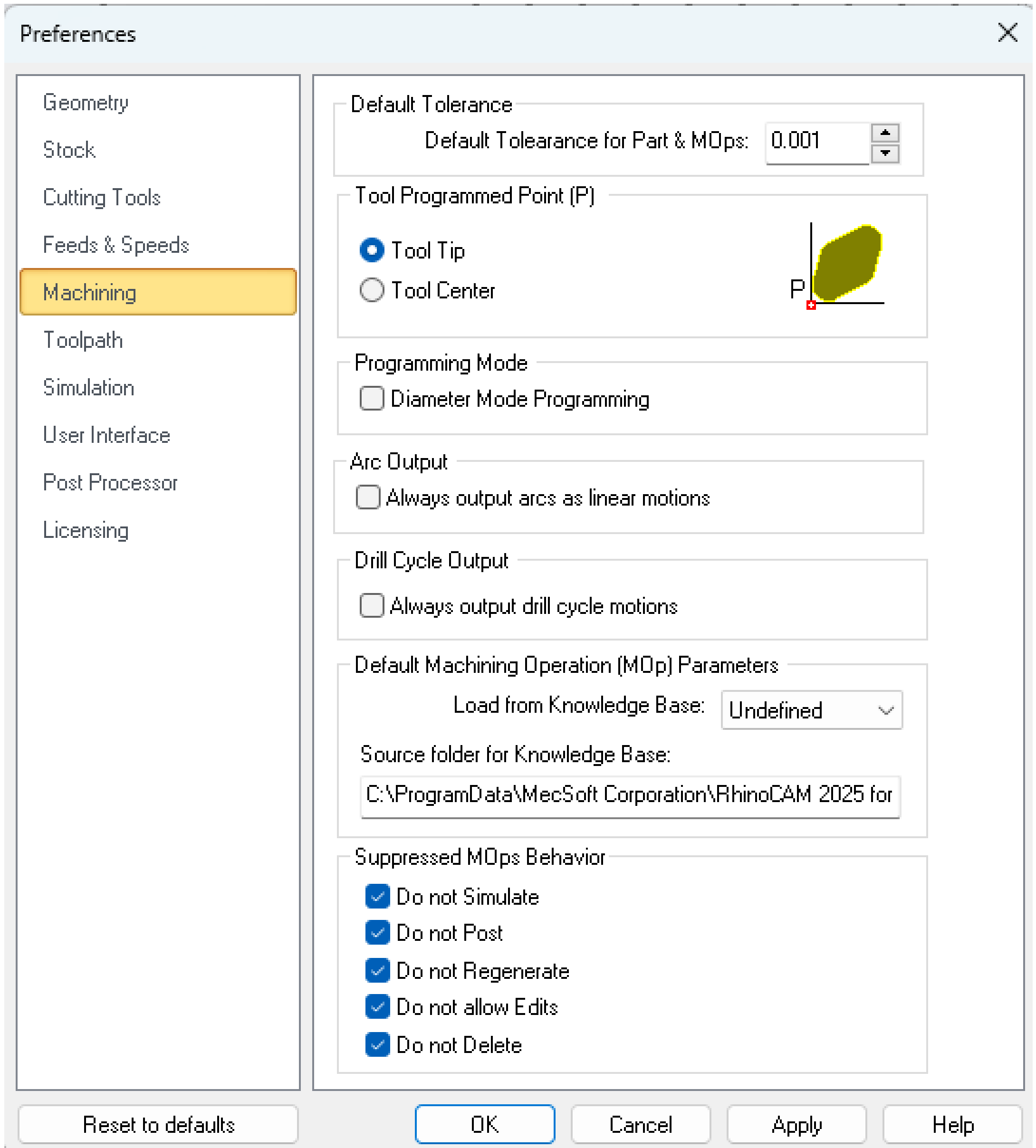
Feeds/Speeds Tracking Options

When you select the Load from Tool option from any of the toolpath operation dialogs, the Feeds & Speeds specified for the active tool are populated into the Feeds & Speeds tab of the operation's dialog. You can check this box to perform this automatically when new toolpath operations are created.

5.7.5 Machining

Users can set the machining preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: Set Machining Preferences



Dialog Box: Set Machining Preferences

Default Tolerance

Default Tolerance for Part & Mops

Enter the default tolerance to use for the TURN Part definition as well as for new machining operations. You can edit this parameter manually from the Cut Parameters tab of each machining operation (Mop) dialog.

Tool Programmed Point (P)

The toolpath can be output as the tool tip or the tool center. If the output is set to be the tool center, the toolpath will be offset by the difference in the height of the tool tip and tool center. The default value is the tool tip.

Programming Mode

Check the box to program in Diameter Mode for Turning. With this box checked, your posted g-code will output diameter values for the X axis. Make sure your Turn Machine Controller is set to the appropriate mode.

Arc Output

Some of the controllers do not handle arc outputs (for example G2, G3). For such type of controllers, the arcs that are generated in the TURN module toolpath can be output as linear segments using this option.

Drill Cycle Output

Check this box to always output drill cycles as linear motions. Left unchecked and the output is a canned drill cycle. Example g-code for the Fanuc post is shown below:

Checked

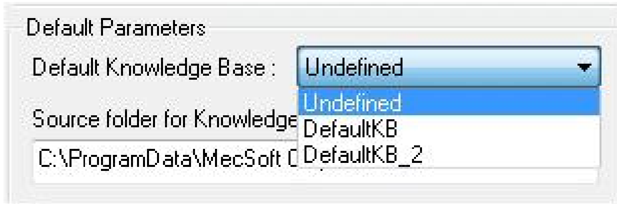
```
%
(Standard Drill )
T2
G54
G50 S300
G97 S300 M03
M08
G00 X0. Z0.25
X0. Z0.1
G01 X0. Z-0.5  G70 F10.
X0. Z0.1
G00 X0. Z0.25
G00G28W0.
G00G28U0.
M09
M05
M30
%
```

Unchecked

```
%
(Standard Drill )
T2
G54
G50 S300
G97 S300 M03
M08
G00 X0. Z0.25
G81 X0. Z-0.5 R0.1 F10.
G00 X0. Z0.25
G00G28W0.
G00G28U0.
M09
M05
M30
%
```

Default Machining Operation (Mop) Parameters

Allows you to select a Default Knowledge Base to load for creating machining operations. Selecting a knowledge base as Default loads the operation parameters when creating new operations. If no Default Knowledge Base is specified, the system defaults are used for machining operation parameters.



Default Knowledge Base

Refer to the following section for creating Default Knowledge Base

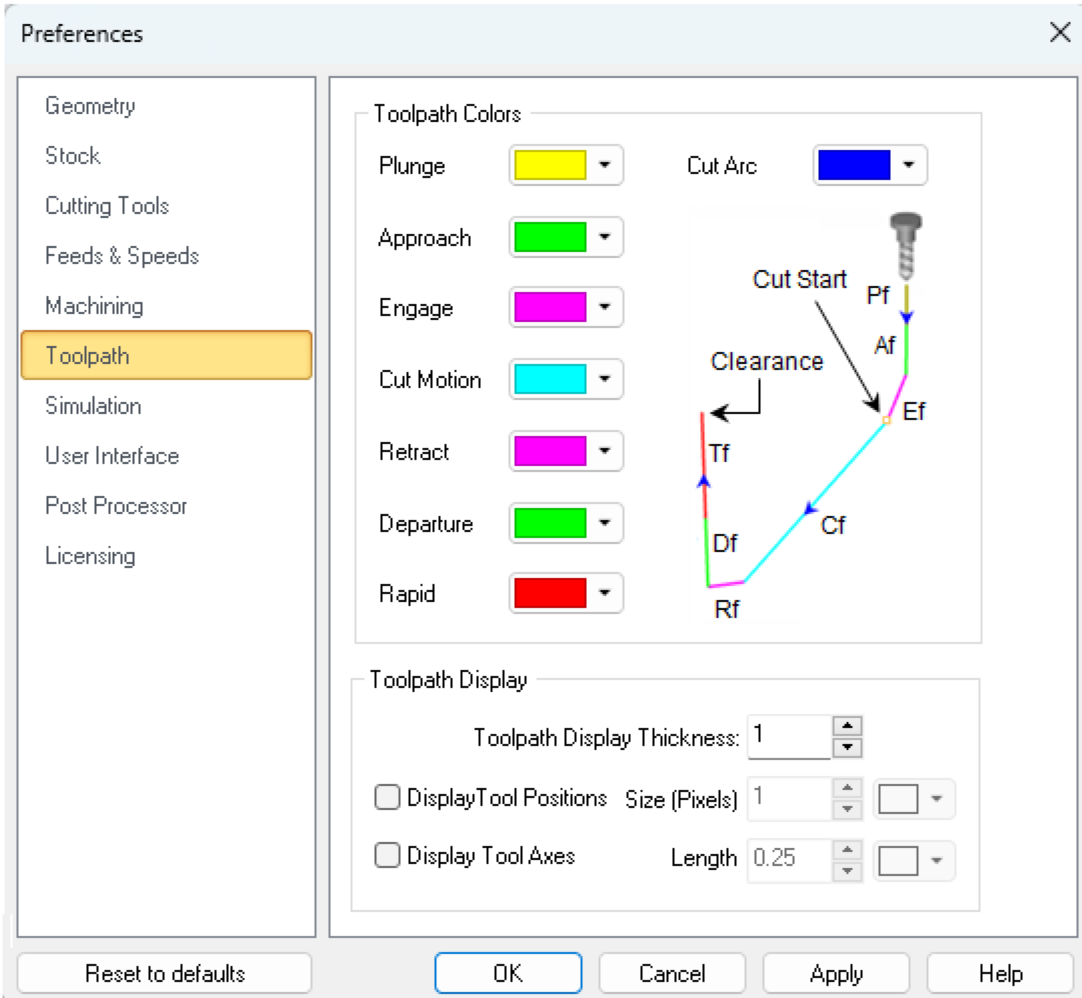
Suppressed Mops Behavior

When you Suppress Machining Operations (Mops) you can apply one or more of these conditions. Check each box to enable that condition and then pick OK to close this dialog.

5.7.6 Toolpath

These preferences relate to the graphical display of toolpath cut motions. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Toolpath



CAM Preferences > Toolpath

Toolpath Colors

Use the color selectors to define the display color for each motion in the toolpath. The following can be set: Cut Motion, Plunge, Approach, Engage, Retract, Departure, Rapid and Cut Arc.

Toolpath Display

These preferences control the display of the toolpath in the graphics window.

Toolpath Display

This refers to the graphical display of toolpaths. Enter a value to effect the size of the toolpath during display.

Display Tool Positions Size (Pixels)

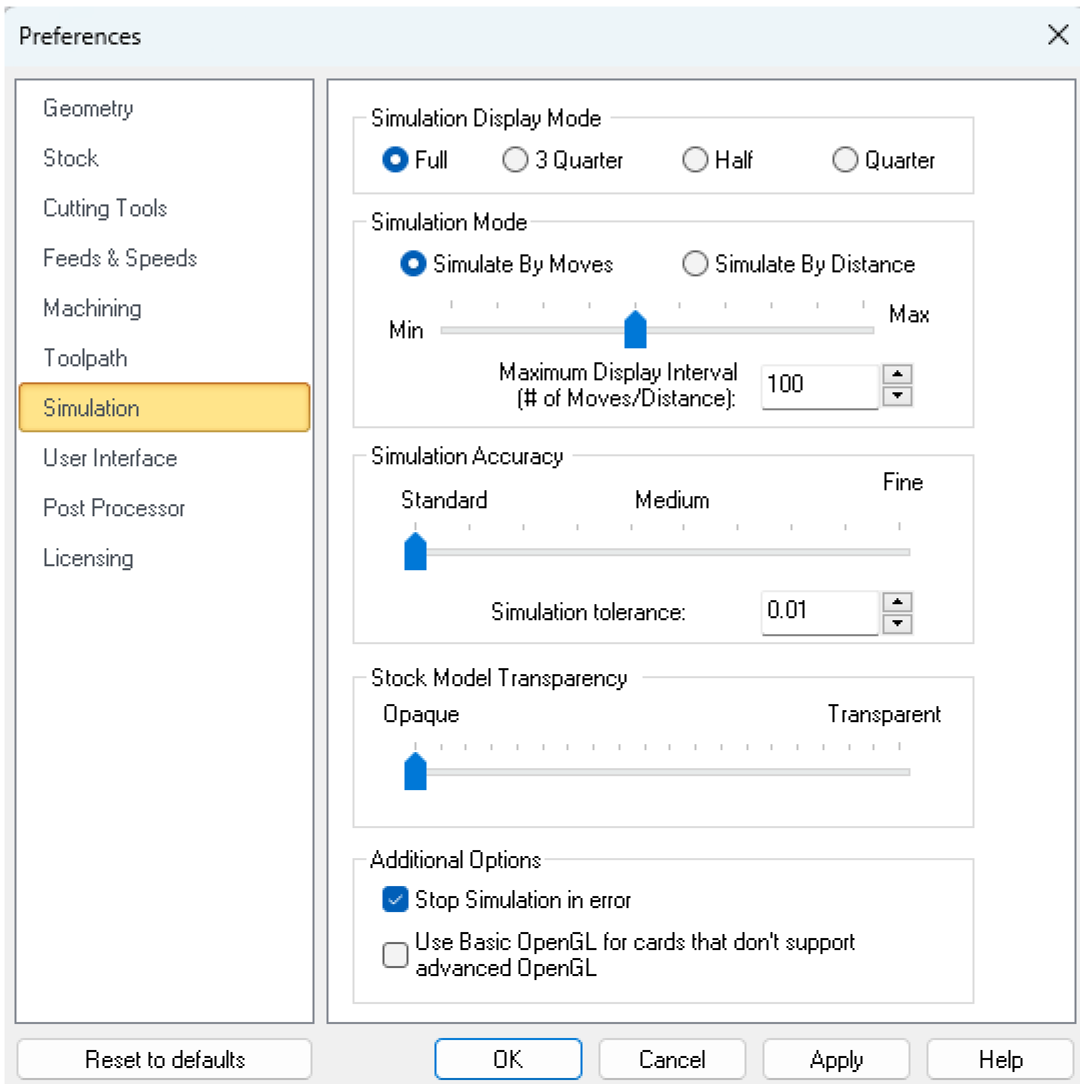
Check this box to display tool position locators. Each coordinate represents one tool position. Then enter the pixel size for the locator point as well as the color of the position points. You can also use the color selector to assign a color to these markers.

Display Tool Axis
Check this box to display the Tool Axis line. You can then enter a Length for the axis line and use the Color selector to assign it a color.

5.7.7 Simulation

Users can set the simulation preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

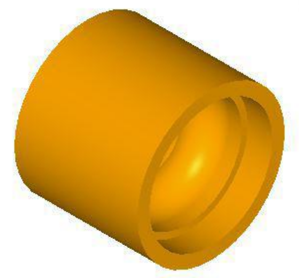
Dialog Box: Set Simulation Preferences



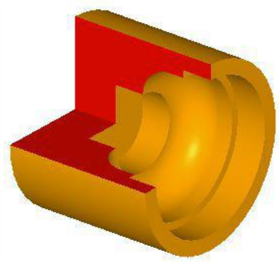
Dialog Box: Set Simulation Preferences

Simulation Display Mode

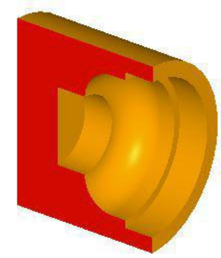
This parameter allows you to display the model in either Full, 3 Quarter, Half and Quarter. Simulations hidden inside the stock (Internal Boring, Drilling etc.), can be visualized better using the 3 Quarter or Half View.



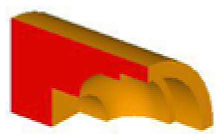
Full View



3Quarter View



Half View



Quarter View

Simulation Mode

User can set the simulation mode to Distance or by Motion. Simulate by Motion simulates the toolpath based on the number of go to motions in the generated toolpath. Simulate by Distance uses a distance based approach.

Simulation Speed

User can control the speed of the simulation using the slider bar and the Maximum display interval. When using Simulate by distance mode, the speed is determined as # of Motions / Distance.

Simulation Accuracy

This setting is used to control the accuracy of display of the simulated model. You can control the accuracy of the stock model by selecting from Standard, Medium or Fine. The finer the stock model accuracy results in slower performance and increases the simulation time.

Stock Model Transparency

User can control the stock model transparency under standard mode and under simulation mode.

Stock Edges Display

This section allows you to control the Stock Edges Display states. For example, you can check the boxes to display Silhouette Edges and Sharp Edges as well as the Angle to display for stock edges. Silhouette Edges and Sharp Edge colors are set using the Colors section of this dialog. Experimentation is advised until you are comfortable with the way your stock display.

Tool holder Display

User can turn on /off the display of tool holder during simulation.

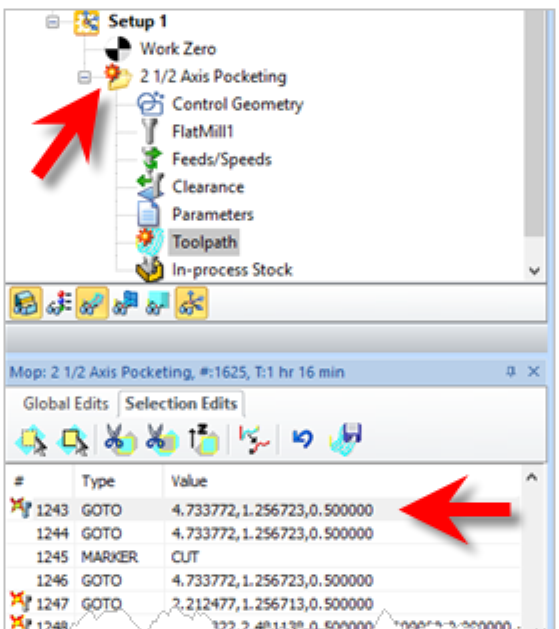
Tool Display

The cutting tool can be displayed either as a sold, Transparent, wireframe or can be turned off during simulation.

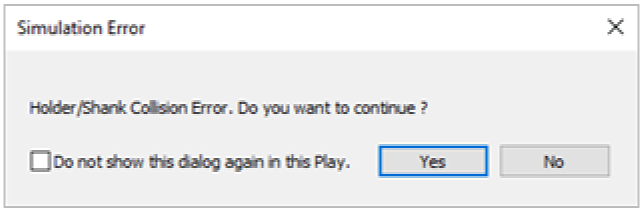
Additional Options

Use Basic OpenGL
Check this box only if you have an older graphics card adapter that does not support advanced OpenGL (i.e., OpenGL 2). Some older cards may only support OpenGL 1 for example. If you experience graphics instability checking this box may help resolve the issue.

Stop Simulation in Error
Check this box to pause the Simulation at each error flag. If enabled, a message will display asking if you wish to continue with the simulation. Selecting Play will simulate to the next error flag and then pause. etc.



Stop Simulation at Error Flag

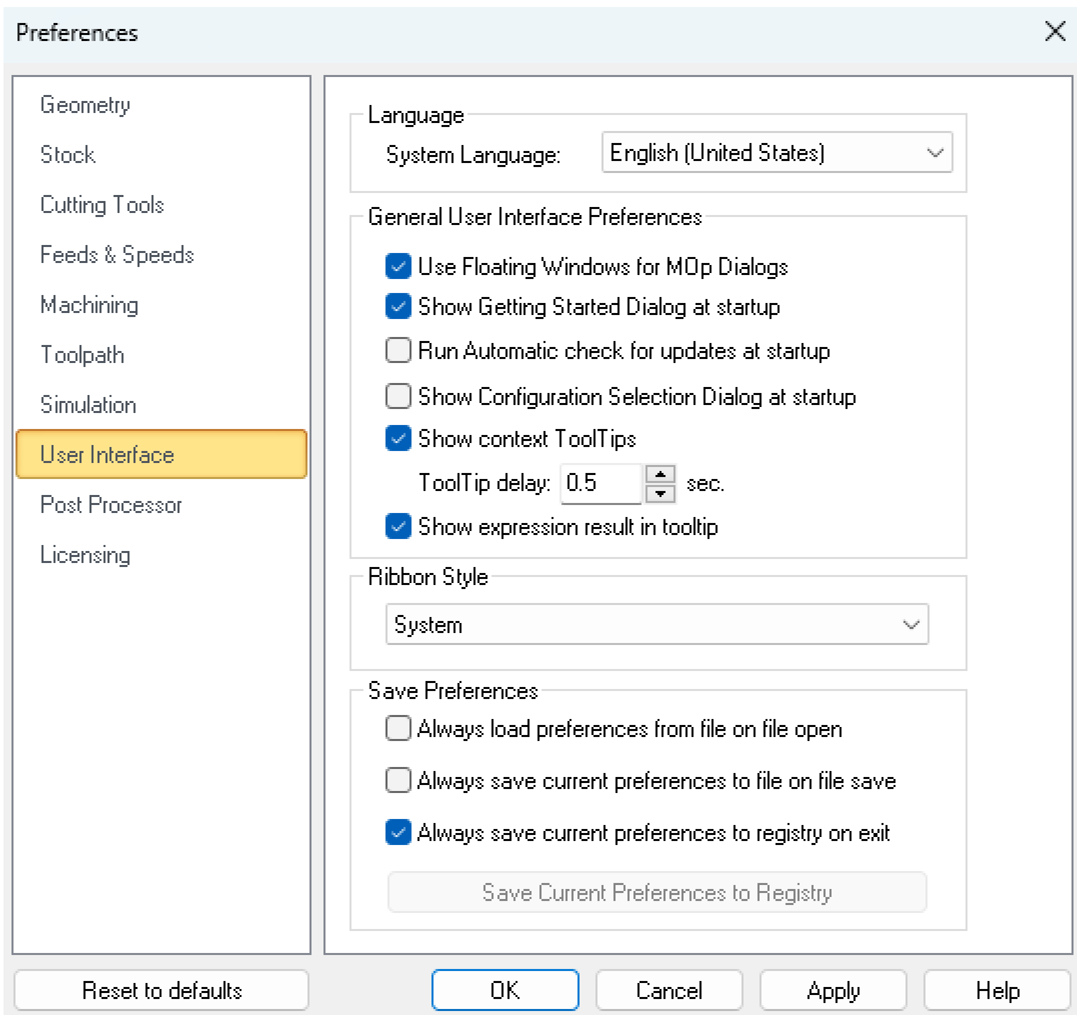


Stop Simulation at Error Flag message

5.7.8 User Interface

Users can set the various user interface options. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: Set User Interface Preferences



Dialog Box: Set User Interface Preferences

User Floating Windows for Mop Dialogs

Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

Show Getting Started Guide at startup

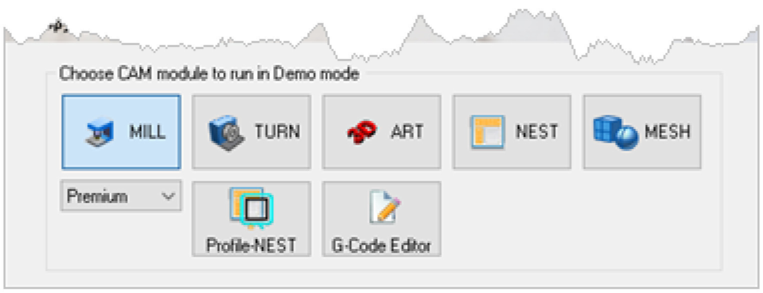
This displays Getting Started dialog at program startup every time the program is loaded. This dialog provides quick access to resources on MecSoft's website.

Run Automatic check for updates at startup

Selecting this option automatically checks for product updates when RhinoCAM is loaded. This requires access to internet on the computer running RhinoCAM.

Show Configuration Selection Dialog at startup

Selecting this option displays the product configuration dialog to run when the program is loaded. User can select from the following MILL modules - Standard, Expert, Professional and Premium.

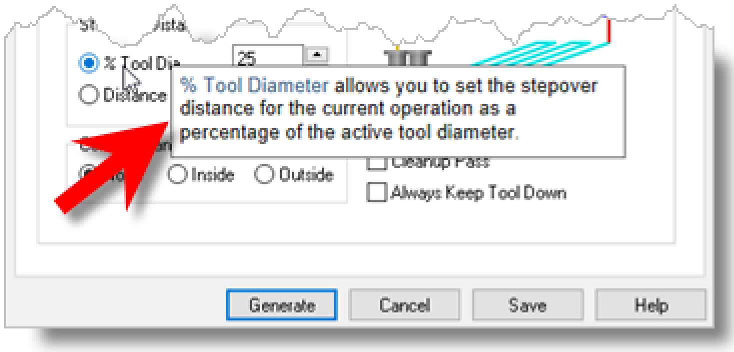


Configuration Selection Dialog at startup

This dialog appears at startup when RhinoCAM is running in demo mode. Selecting a configuration loads RhinoCAM and provides the features available in the selected configuration.

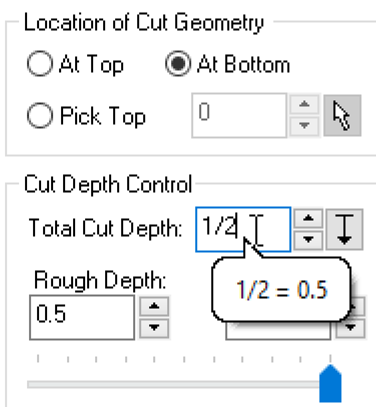
Show context ToolTips

Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in seconds. This is the amount of time it takes to display the Context ToolTip when the mouse activate it.



Show Expression Results in Tooltip

You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a ToolTip balloon. An example is shown below.



Show Expressions in ToolTip

Save Preferences

Always load preferences from file when opening a new file

Check this box if you wish to always load CAM Preferences from the file you are opening. Remember, however, that your current settings including your selected post is subject to be being changed.

Always save current preferences to file on file save

Check this box if you wish to always save the current CAM preferences to the file on file save. Remember, however, that your current settings including your currently selected post will replace those preferences that were in the current file originally.

Always save settings to registry on exit

Check this box if you wish to always save the current CAM preferences to the Windows registry when you exit your MecSoft CAM plugin. This will ensure that your current CAM settings will always be used when starting a new file.

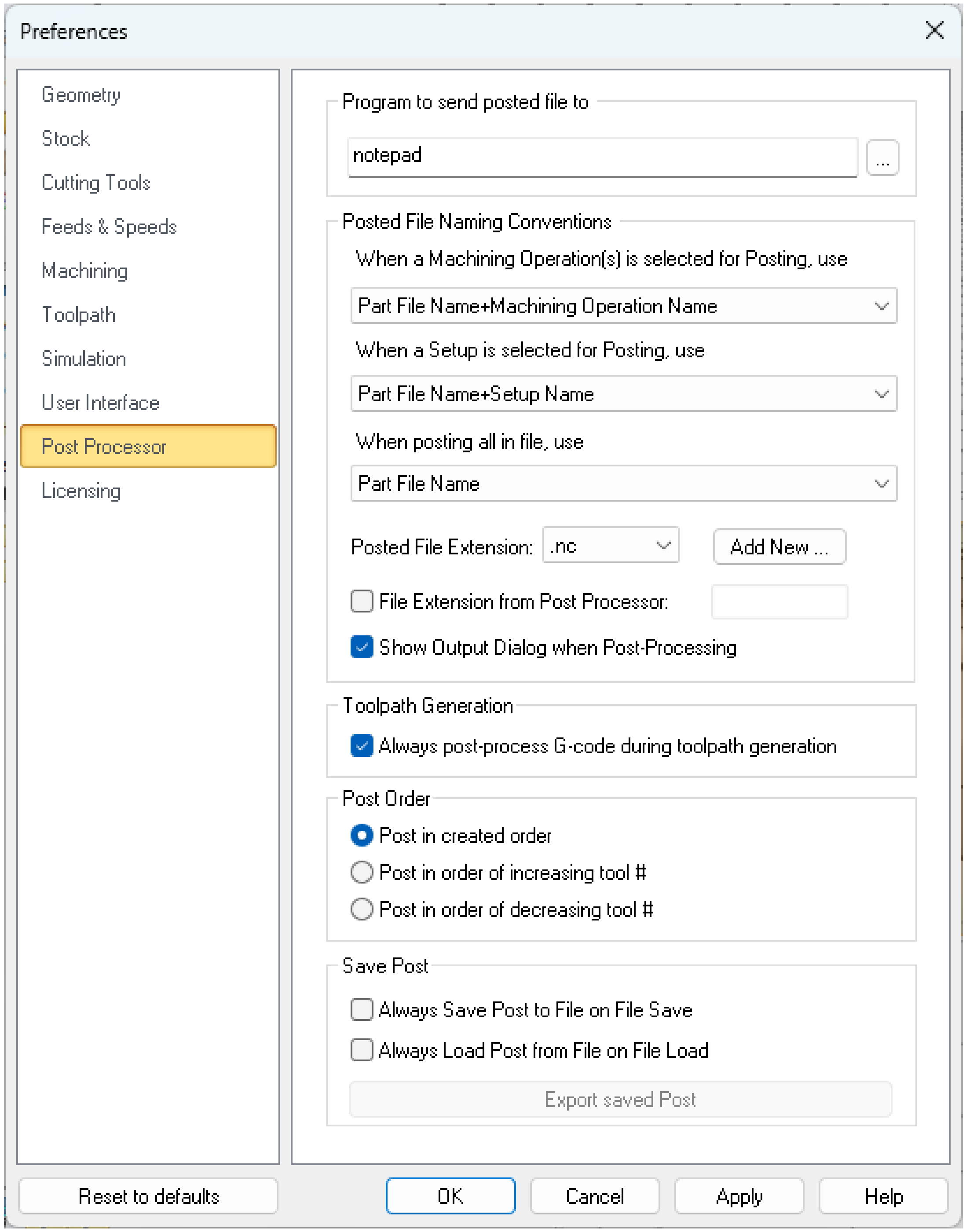
Save Current Preferences to Registry

If you have your preferences set the way you want them and do not want them top change, select this button to save the current preferences to your Windows registry. Doing this will force them to be loaded when you create new files.

5.7.9 Post Processor

These preferences relate to posting toolpath operations to gcode files. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Toolpath



CAM Preferences > Post

Program to send the Posted file to

This feature allows you to specify a program to display the posted file. This could be a NC editor or a text editor like Notepad.

You could also have this point to your control software's executable file and RhinoCAM will automatically launch this application when the machining operations are post processed.

Posted File Naming Conventions

This allows you to set rules for posted file name when post processing machining operations.

When a machining operation is selected for posting you can set the output file name from one of the following options.

- Part File Name + Machining Operation Name
- Part File Name + Setup Name + Machining Operation Name
- Setup Name + Machining Operation Name
- Machining Operation Name

When a setup is selected for posting you can set the output file name from one of the following options.

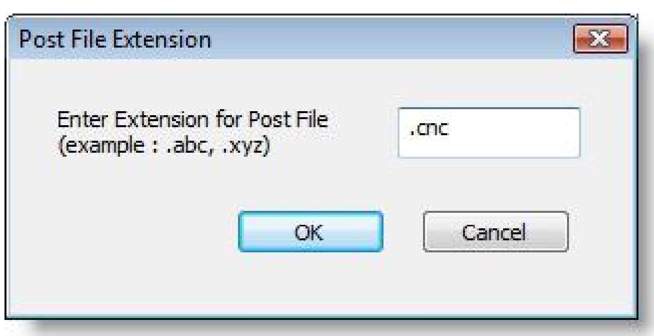
- Part File Name + Setup Name
- Setup Name

When Machining Job is selected to Post All, you can set the output file name from one of the following options.

- Part File Name
- Part File Name + First Setup Name
- First Setup Name

Posted File extension

You can select a posted file extension from the list or add an extension to the list by selecting Add new button. This displays the Post File Extension dialog shown below where you can specify a new file extension and click OK.



Dialog Box: Post File Extension

The new file extension is now set as your posted file extension automatically.

By default RhinoCAM performs interactive post-processing. That is, when you select a toolpath for post-processing, RhinoCAM launches the post-processor and waits for it to complete. You can also turn off the display of the output dialog (post and save dialog).

During interactive post-processing, RhinoCAM launches the NC editor to view the output file. You can specify a different NC editor to use. See Program to send the Posted file to above for doing this.

File Extension from Post Processor
Check this box to *"pull"* the posted g-code file extension from the Legacy Post-Processor (*.spm) file. This ensures that whichever post that you use, your posted g-code file will match the file extension defined in the active post. **Note**: You must edit your legacy post and set the Output File Extension value from the General tab in the Post-Processor Generator.

Show Output Dialog When Post Processing
Check this box to always display the Post & Save As file dialog when you select Post from an operation (Mop), Setup or Machining Job.

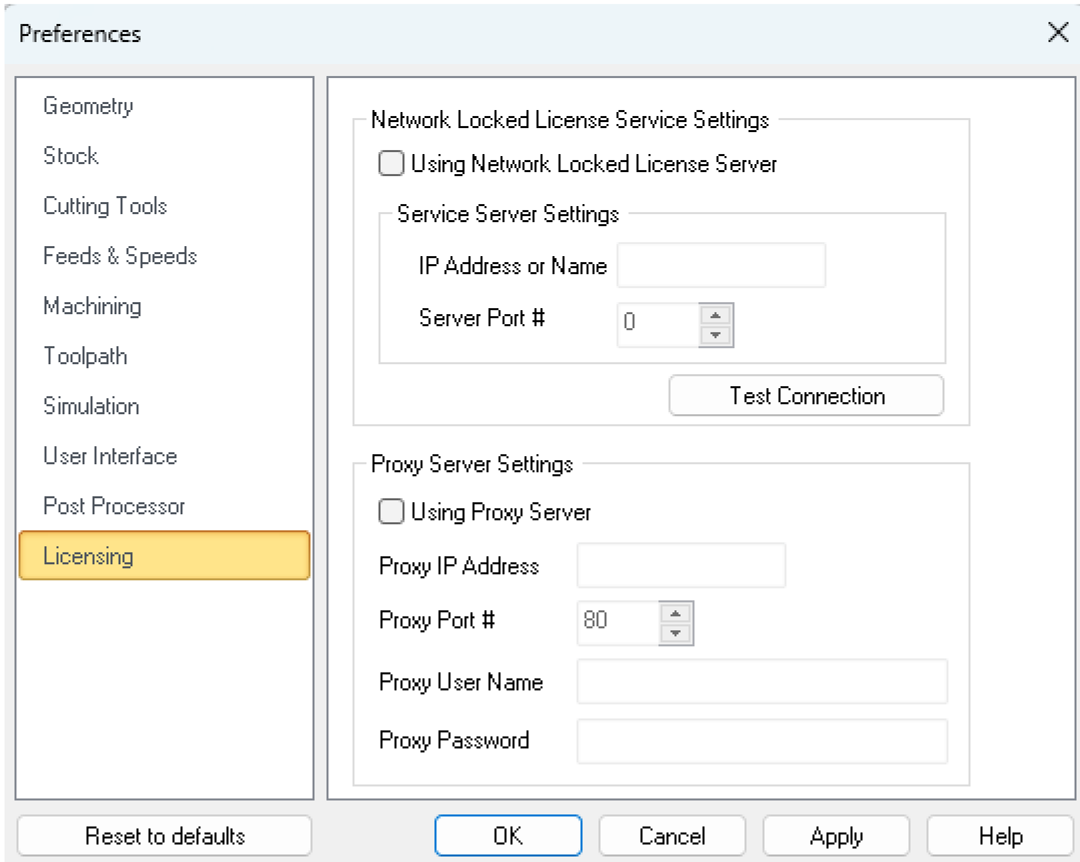
Toolpath Generation

Always post-process G-code during toolpath generation
Your G-code is stored with your CAM file so that the latest G-code is always available to you when you open a file. Uncheck this box if you do not want new G-code generated every time you generate a toolpath operation.

5.7.10 Licensing

This dialog allows you to set Licensing Preferences for using a Proxy Server and/or a LAN Daemon (for Network Licenses). This information would be provided by your network administrator. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: License Preferences



Dialog Box: License Preferences

Network Locked License Service Settings

Network Locked License is a security process required when a computer on a network tries to connect to the server in order to use its resources. This license type is locked to the specified license server and cannot operate without a secure connection to the specified license server. If the user's client machine has access to the license server and a license is available, the license will activate. A valid server IP Address and Server Port # are required.

Using Network Locked License Server
Check this box to enable Network Locked License Server. Then complete the Service Server Settings provided here.

Server IP Address
For Network Authentication, enter the Service Server's IP Address here.

Server Port #
For Network Authentication, enter the Service Server's Port # here.

Proxy Server Settings

Proxy Server Settings need to be set if your computer or network is behind a proxy. A proxy server is a computer that acts as an intermediary between the user's computer and the Internet. It allows client computers to make indirect network connections to other network services.

Using Proxy Server
Check this box to enable Proxy Server Settings and complete ALL of the following fields accurately. This information would be provided by your network administrator.

Proxy IP
This is the IP Address for your Proxy Server. This information would be provided by your network administrator.

Proxy Port #
Enter the Port Number for your Proxy Server. This information would be provided by your network administrator.

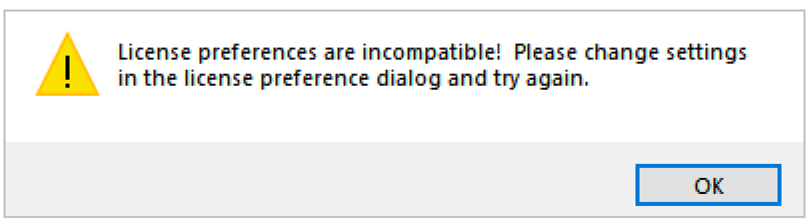
Proxy User
Enter the Proxy Server user name. This information would be provided by your network administrator.

Proxy
Enter your Proxy Server password. This information would be provided by your network administrator.

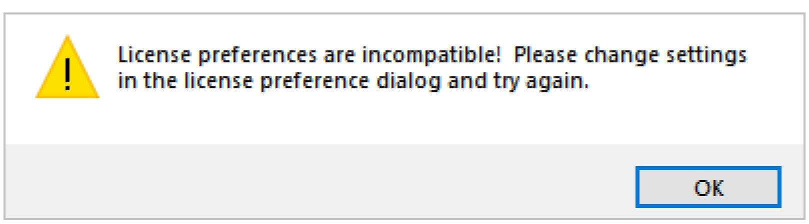
Troubleshooting and Messages

Here are some troubleshooting messages that you may encounter.

If you have node locked license activated and you select Using Lan Daemon, this will display the following message and release your node locked license.



If Using Lan Daemon is checked and you are entering a valid node locked activation code in the license dialog, the following message is displayed. Make sure Using Lan Daemon is unchecked before activating a node-locked license.



Overview of Machining Methods

There are two major classes of machining operations that can be created in the RhinoCAM TURN Module - Turning and Drilling. Turning operations are used to mill material to form shapes. Drilling operations are used to create holes. Both classes are essential in any manufacturing industry.

Turning operations can be categorized as Roughing, Finishing, Grooving, Threading and Parting Off. The various hole machining operations that can be used for hole making are Drilling, Tapping, Boring and Reverse Boring.

Turning

Provides access to Roughing, Finishing, Grooving, Threading and Parting Off operation. The tool moves in X and Z axis. Material is removed in successive passes in Roughing operation.

Holes

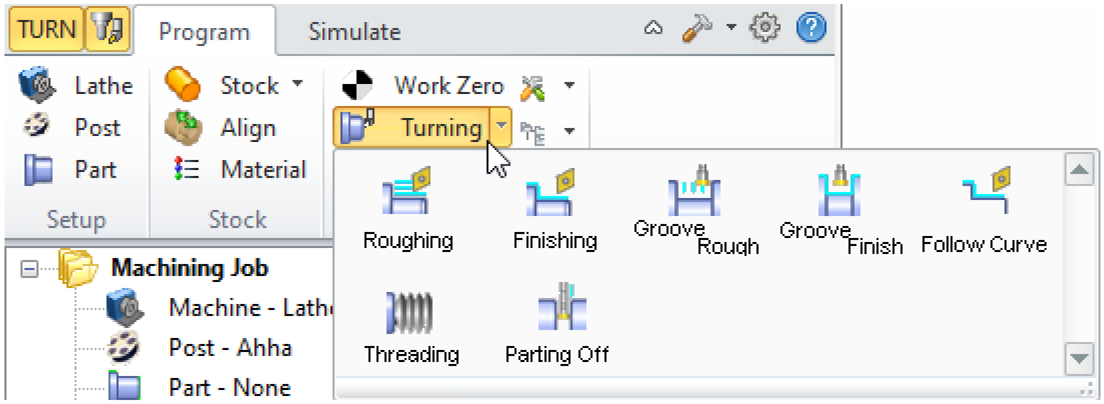
These operations are used to create holes in the part, including drill holes, counter sunk holes and through holes. Tapped and bored holes can also be created The tool is oriented along the Z axis and is at the center of the rotational axis.

These categories, and the available operations, within them are described in the sections to follow.

6.1 Turning Operations

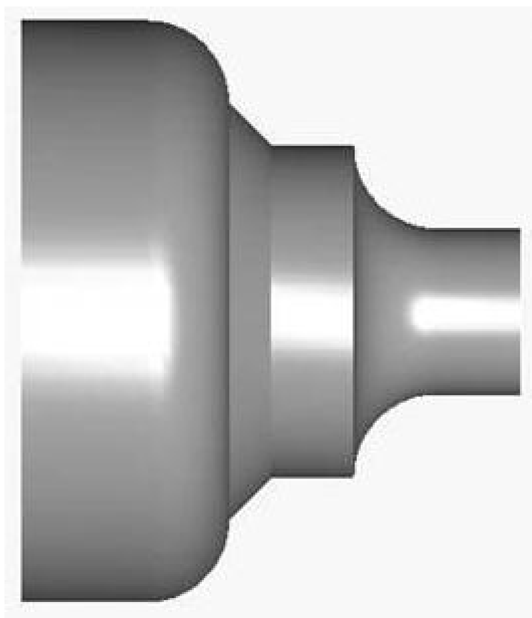
The RhinoCAM TURN Module allows you to choose from a variety of rough, semi-finish and finish machining operations to satisfy various machining conditions and requirements. A list of the available types with a short description for each type is given below. The drop down menu in the icon can be used to access Turning operations and Hole machining operations. The Turning operations under the Turning menu that can be used in TURN Module are Roughing, Finishing, Grooving, Follow Curve, Threading and Parting Off.

Create Turning Operations Menu



Create Turning Operations Menu

Using this class of machining, you can machine parts that are defined by 2D curves or 3D part geometries. A typical machining operation would involve roughing then finishing.

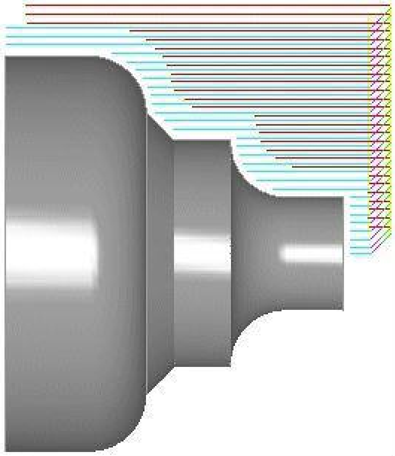

Typical Turned Part

6.1.1 Roughing

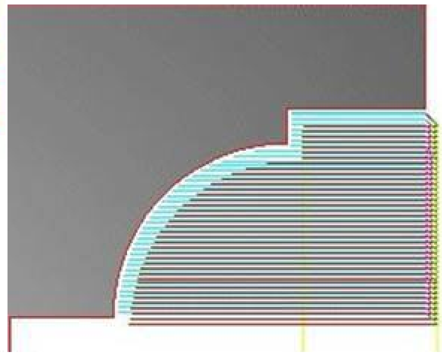
This is the RhinoCAM TURN Module's principal method of roughing, in which the material is roughed out in multiple cuts. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large radius tool. Roughing is typically followed by finishing toolpaths.

Both part and stock geometry are used to determine the regions that can be safely machined. Roughing can be of 3 types: OD Roughing, ID Roughing, and Front Facing (Face Roughing). See Turn Roughing for cut parameters.

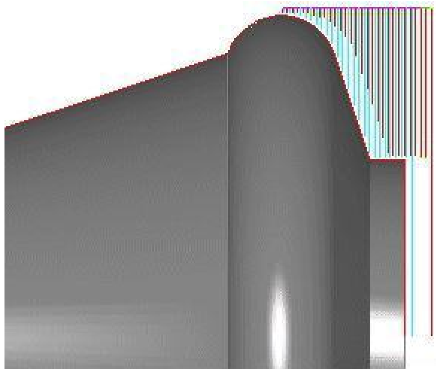
Turn Roughing Examples



OD Roughing Toolpath Example



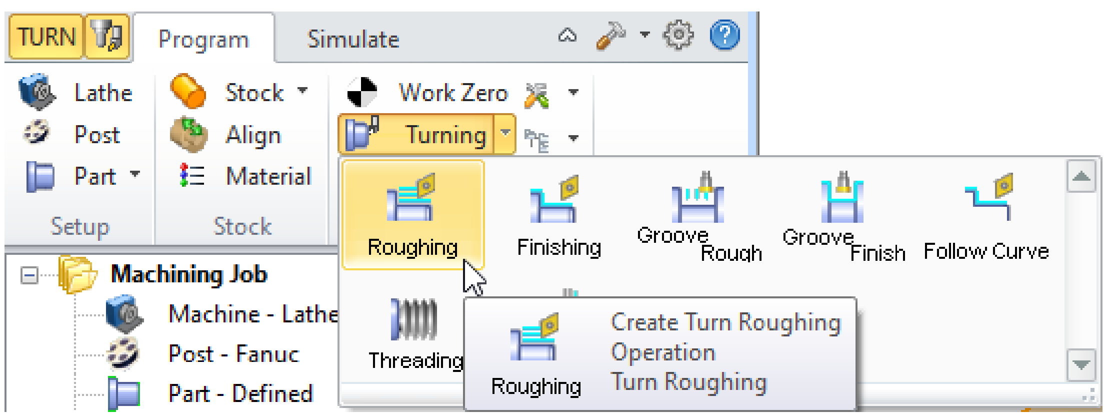
ID Roughing Toolpath Example



Face Roughing Toolpath Example

Turn Roughing Menu Item

The Turn Roughing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Roughing operation.

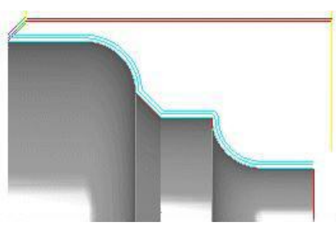


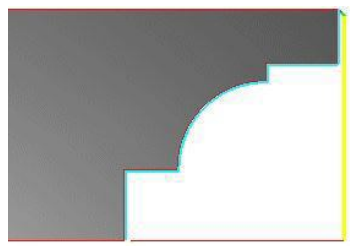
Turn Roughing Menu Item

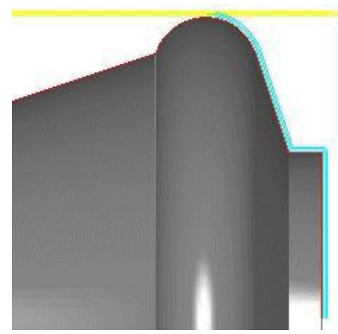
6.1.2 Finishing

This is one of the most commonly used machining processes. The part is rotated while a single point cutting tool is moved parallel to the axis of rotation following the contour of the geometry. Finishing can be done on the external surface of the part (OD, Face) as well as internally (boring). You can define offsets so that the tool makes multiple passes relative to the regions. Finishing is typically done after a Roughing operation, or it can be used alone. See Turn Finishing for cut parameters.

Turn Finishing Examples

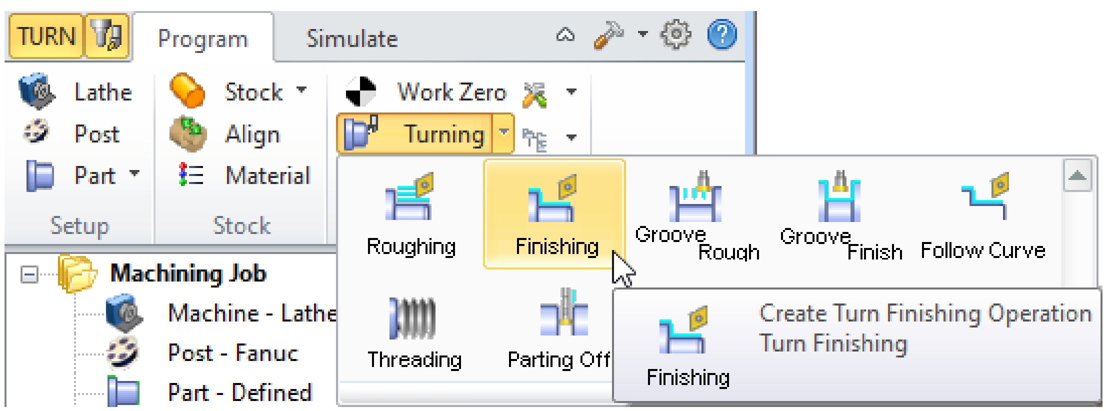

OD Finishing Toolpath Example


ID Finishing Toolpath Example


Face Finishing Toolpath Example

Turn Finishing Menu Item

The Turn Finishing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Finishing operation.



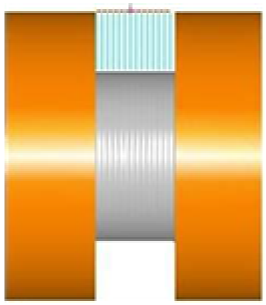
Turn Finishing Menu Item

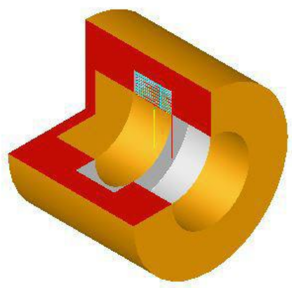
6.1.3 Groove Roughing

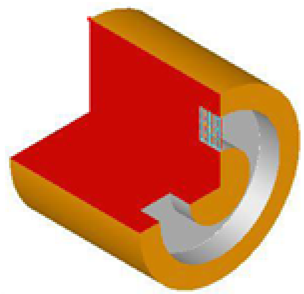
This operation is performed to machine grooves in multiple cuts on the part. This roughing operation provides user the control to set the step down and step over and choose the cut direction.

Both part and stock geometry are used to determine the regions that can be safely machined. Groove Roughing can be of 3 types: OD Groove Roughing, ID Groove Roughing, and Face Groove Roughing. The grooves are typically used to slide/fit one part into another to obtain the required assembly. See Turn Groove Roughing for cut parameters.

Turn Groove Roughing Examples

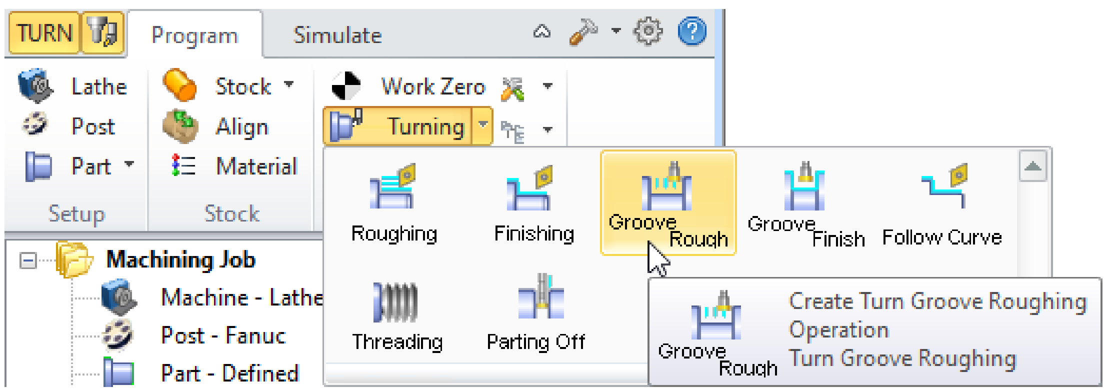

OD Groove


ID Groove


Face Groove

Turn Groove Roughing Menu Item

The Groove Roughing toolpath method is invoked by selecting the Program tab, clicking on the **Turning** button in the Machining Browser and selecting the Groove Roughing operation.



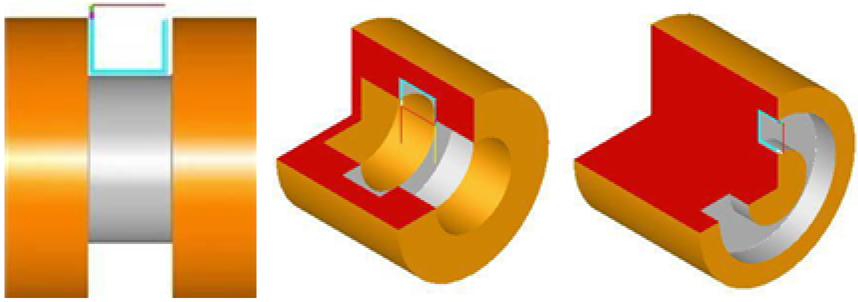
Turn Groove Roughing Menu Item

6.1.4 Groove Finishing

This operation is used to finish the grooves. This operation is performed after the Groove Roughing operation. Groove Finishing can be of 3 types: OD, ID, and Front Facing. See Turn Groove Finishing for cut parameters.

Turning Operations

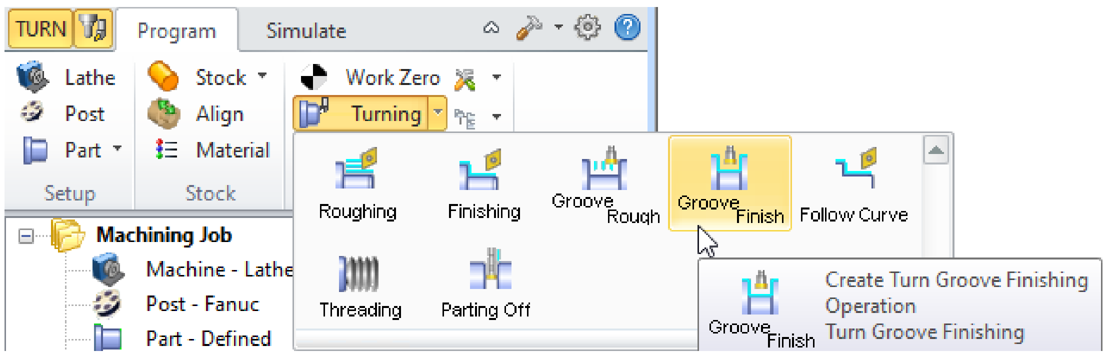
Turn Groove Finishing Examples



OD Groove ID Groove Face Groove

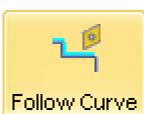
Turn Groove Finishing Menu Item

The Groove Finishing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Groove Finishing operation.



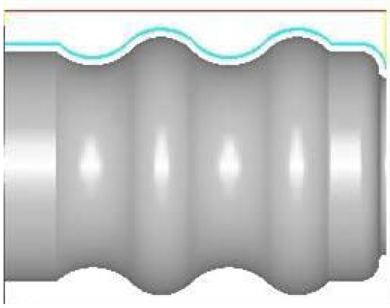
Turn Groove Finishing Menu Item

6.1.5 Follow Curve



This operation is performed after the roughing operation. This is similar to finishing operation where the toolpath follows the selected curve to obtain better surface finish and is characterized by smaller depth of cut to obtain tighter tolerances and better surface finish. This method is similar to engraving in milling where the tool nose radius compensation is not applied to the toolpath. Follow curve can be used in OD, ID, and Front Facing operations. See Turn Follow Curve for cut parameters.

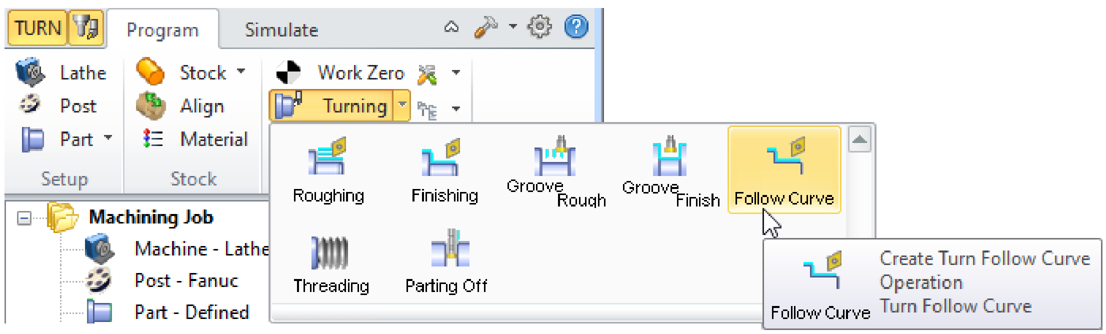
Turn Follow Curve Example



Follow Curve Operation Toolpath Example

Turn Follow Curve Menu Item

The Follow Curve toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Follow Curve operation.

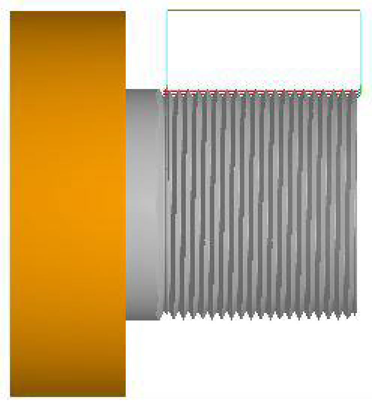


Flow Curve Menu Item

6.1.6 Threading

This operation is performed to machine threads on the part. Threads are used as fasteners for assembly purposes. OD and ID threads can be programmed using this method. See Turn Threading for cut parameters.

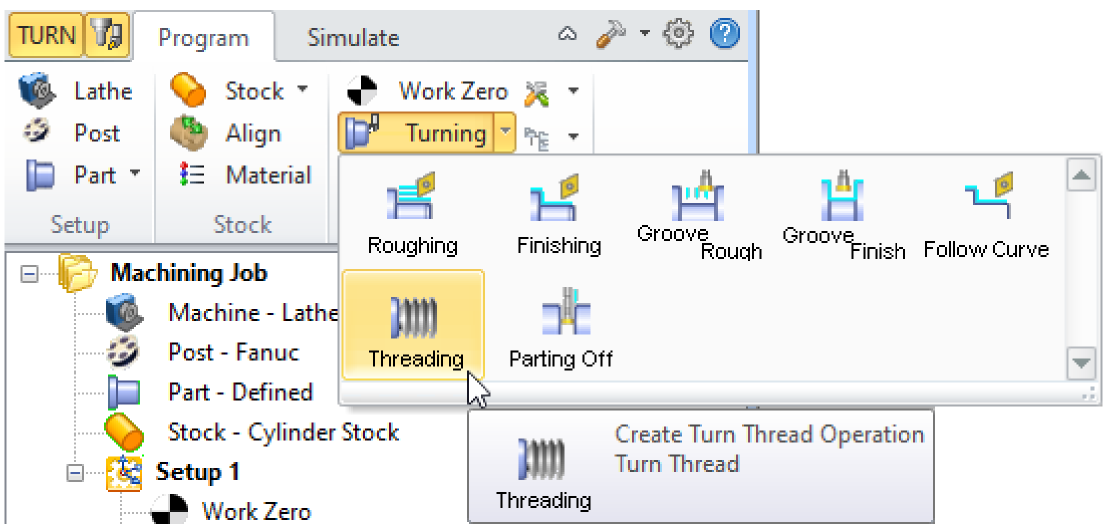
Turn Threading Example



Threading Operation Toolpath Example

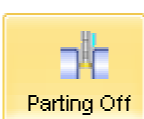
Turn Threading Menu Item

The Threading toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Threading operation.



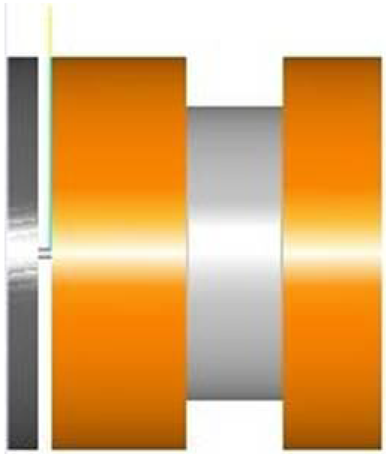
Turn Threading Menu Item

6.1.7 Parting Off



This operation is performed to cut off the finished part from the rest of the bar stock which is typically done as the last operation in OD. See Turn Roughing for cut parameters. See Turn Parting Off for cut parameters.

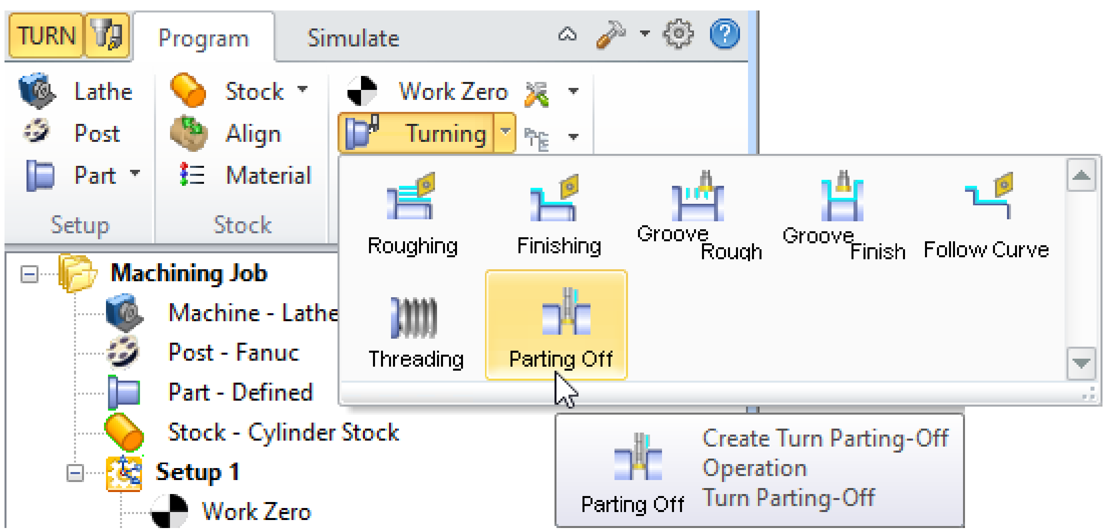
Turn Parting Off Example



Parting Off

Turn Parting Off Menu Item

The Parting Off toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Parting Off operation.

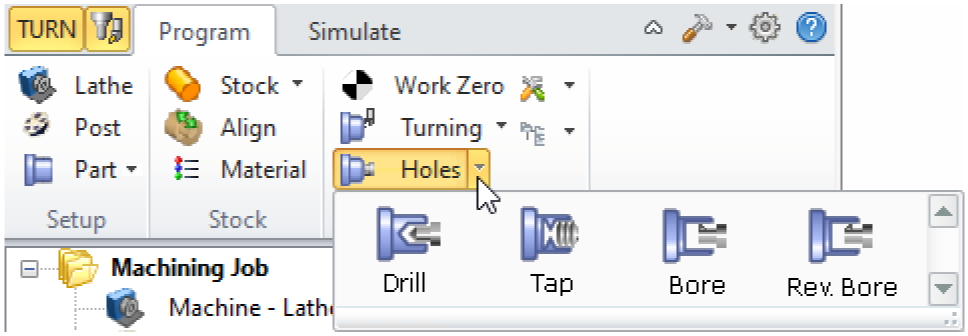


Parting off Menu Item

6.2 Drilling (Hole Making) Operations

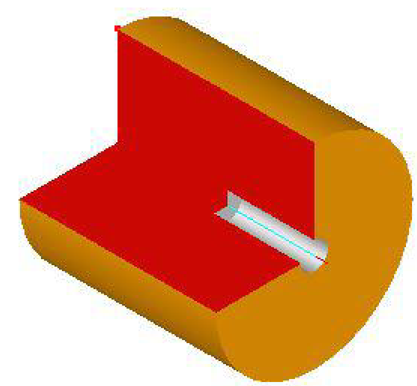
These operations are used to create holes in the part, including drill holes, counter sunk holes and through holes. Tapped and bored holes can also be created. The tool is oriented along the Z axis and is at the center of the rotational axis. The following drilling operations are available:

Turn Drilling (Hole Making) Operations Menu



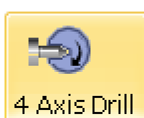
Drilling Menu Item

Using this class of machining, you can machine parts that are defined by 2D curves or 3D part geometries. A typical machining operation would involve roughing then finishing.

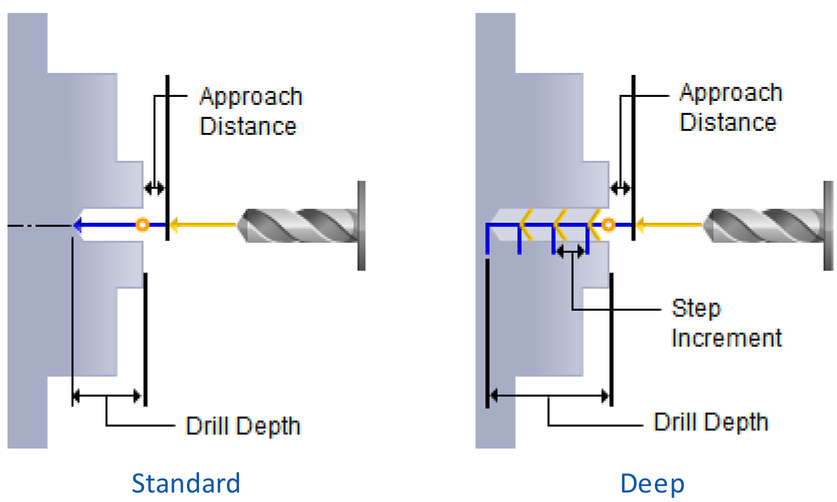


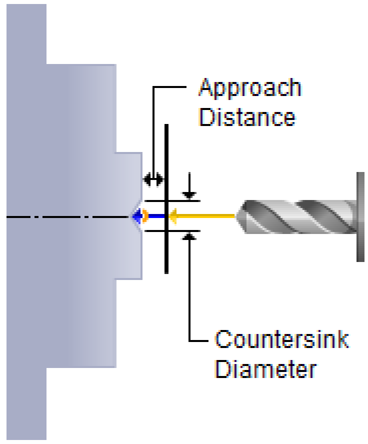
Typical Turned Hole Operation

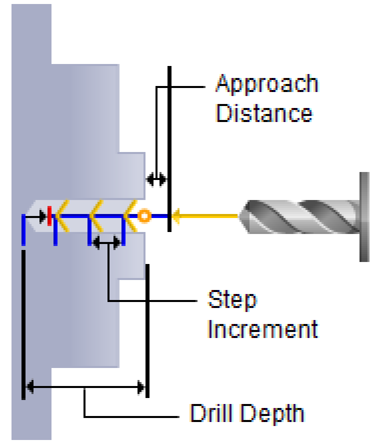
Turn Drill Operation



The following drill cycles are available:





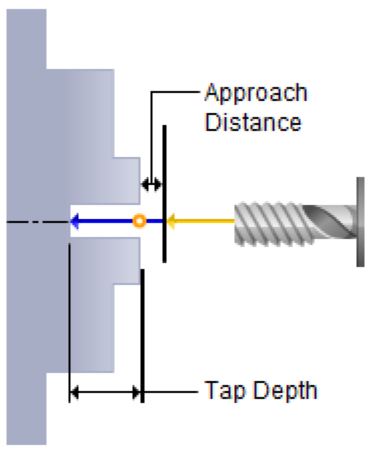


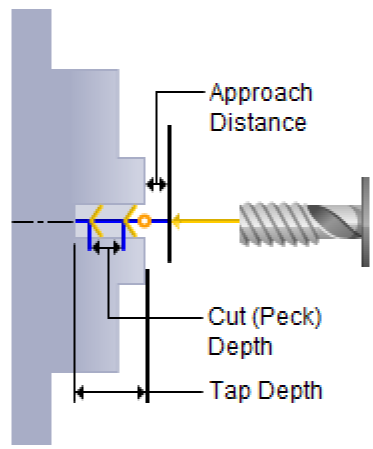
Counter Sink: Break Chip

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Turn Tap Operation

The following tap cycles are available:

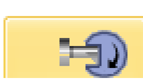




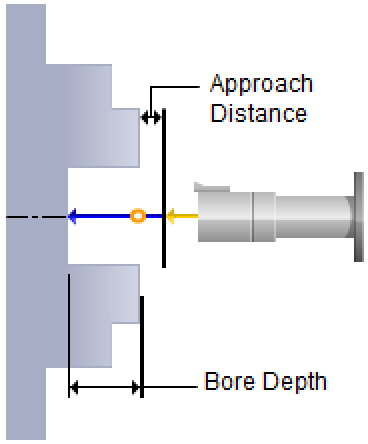
Standard Tap Peck Tap

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Turn Bore Operation



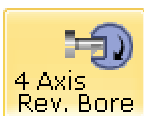
The following bore cycles are available:



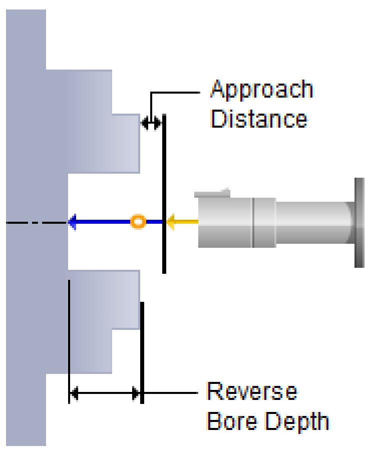
Drag Bore, No Drag Bore, Manual Bore

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Turn Reverse Bore Operation



The following reverse bore cycles are available:



Reverse Bore

User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

Creating Machining Operations

Creating machining operations in the RhinoCAM TURN Module is a very simple process. You load the part, select the part geometry for turning, create stock geometry if necessary, select a tool and then specify the feeds and speeds to be used in the machining operation.

You then pick the type of machining operation required and set the parameters for the operation and then generate the toolpath. You can also specify cut containment to restrict areas to be machined. Generation of the toolpath begins once you click on the "Generate" button in the machining operation parameter dialog. Once the toolpath generation is complete the machining operation will be created and displayed in the Machining Browser (Mops). The following sections describe each of the necessary and optional items that need to be selected or set before creating a machining operation.

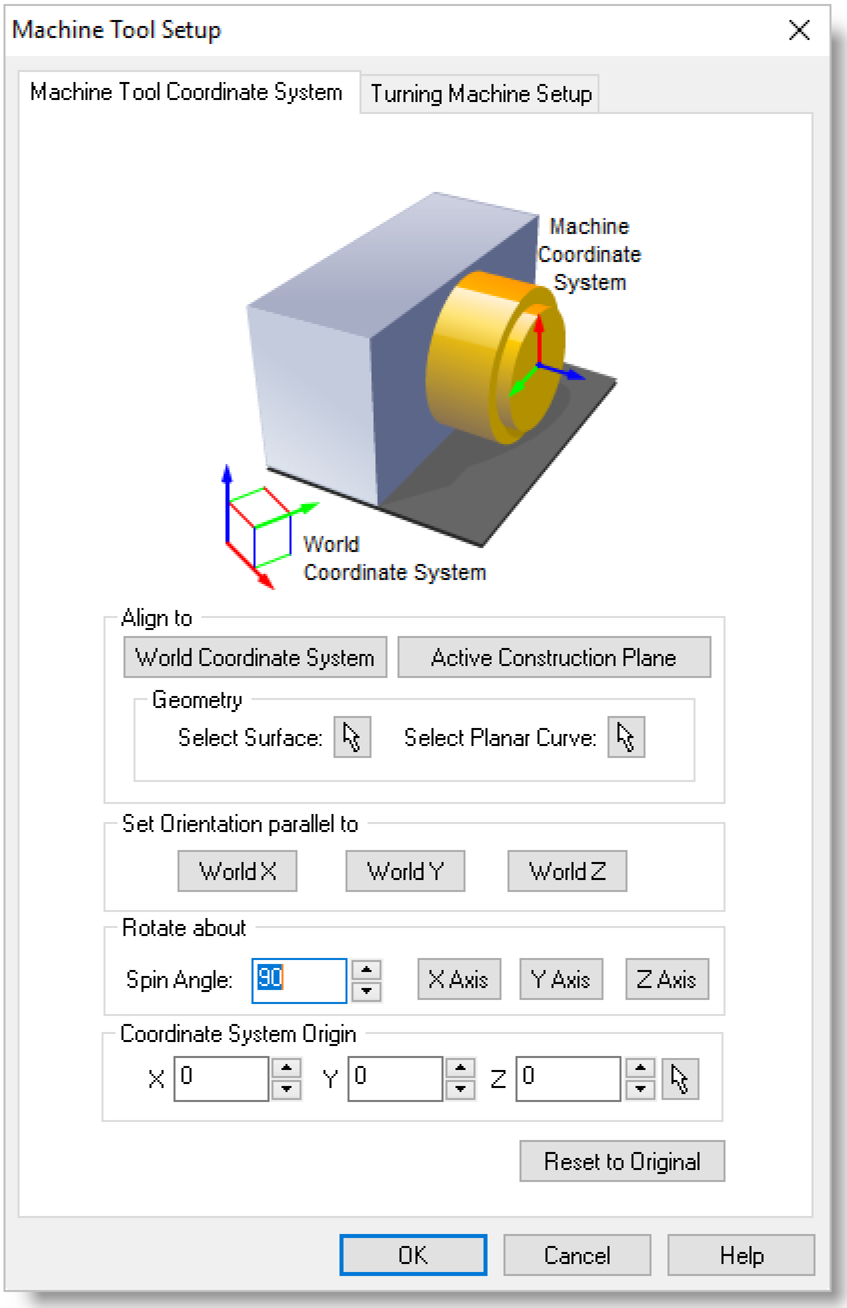
7.1 Lathe

7.1.1 Coordinate System

Once part geometry is loaded, user can set the cutting direction. This orients the Machine Coordinate System for Turning to have the part aligned in the same way as it would be fixtured on the machine tool for cutting. The stock geometry and machine tool definition will be defined based on the orientation of the machine tool coordinate system.

This dialog offers a convenient way of aligning the Machine Coordinate System (MCS).

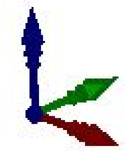
Dialog Box: Machine Tool Setup, Machine Tool Coordinate System tab



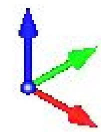
Dialog Box: Machine Tool Setup, Machine Tool Coordinate System tab

The Machine Coordinate System (MCS)

The Machine Coordinate System (MCS) is displayed as a triad with **Blue** line representing the Z-axis, **Red** representing X-axis and **Green** representing the Y-axis. The WCS is displayed the same way as MCS and is located at the origin. The lengths of the WCS arrows are shorter when compared to MCS.



(MCS)



(WCS)

Align to

This aligns the MCS orientation. Select from the following:

World Coordinate System
Setting the MCS to the WCS. In this case MCS and WCS would have the same coordinate location.

Active View Construction Plane
Setting the MCS to the orientation of the active viewport of the construction Plane.

Geometry:

Part Surface
User can select a point on a surface determine the alignment of MCS.

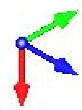
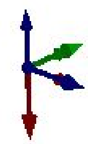
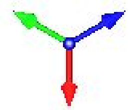
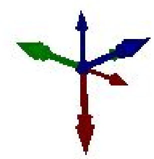
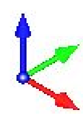
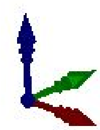
Geometry:

Planar Curve
User can select a planar curve (2D sketch) to determine the alignment of MCS.

Set Orientation Parallel to

Allows you to set the Machine Coordinate System parallel to the World X Y or Z co-ordinate axis.

- World X
  Orients the Z axis of MCS parallel to World X axis.
- World Y
  Orients the Z axis of MCS parallel to World Y axis.
- World Z
  Orients the Z axis of MCS parallel to World Z axis.

Orientation Parallel to	Triad Display		
	MCS Visibility ON WCSVisibility OFF	WCS Visibility ON MCS Visibility OFF	MCS Visibility ON WCS Visibility ON
World X			
World Y			
World Z			

Rotate about

Allows you to rotate the Machine Coordinate System in X Y Z coordinate axis by any angle specified under Spin Angle.

Specify Spin Angle and click the axis to rotate about. Clicking the same coordinate axis button multiple times rotates by the specified angle incrementally. For example if you set the Spin Angle = 90 and click X Axis button 2 times, the MCS is rotated about X coordinate axis by 180 degrees.

Coordinate System Origin

This translates the MCS origin to the desired location. This can be set to any location on the part geometry.

Reset to Original

Resets MCS orientation to current MCS orientation.

7.1.2 Machine Setup

This dialog allows you to setup your Machine Tool Definition. Refer to each section below for more information.

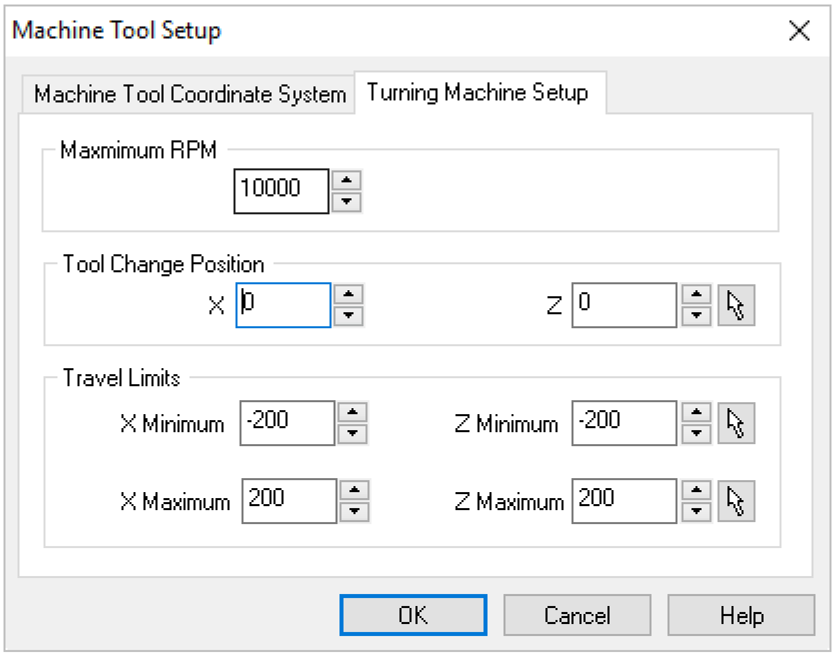
Turning Machine Setup

In the TURN Module, this dialog sets up the machine for 2 axis turning operations. This is done by selecting Lathe (Turning Machine Tool Setup) under Program tab in Machining Browser. This will bring up the following dialog.

Machine Tool Setup, Machine Tool Coordinate System tab

See: Machine Tool Setup, Machine Tool Coordinate System tab

Machine Tool Setup, Turing Machine Setup tab



Dialog Box: Machine Tool Setup, Turing Machine Setup tab

Maximum RPM

Allows you to specify the maximum spindle RPM that can be set for the machine tool.

Tool Change Position

User can specify a coordinate location in X and Z axis. TURN Module will take this coordinate values and output it for every tool change. The tool change variables must be configured in the post processor.

Travel Limits

Allows you to specify the minimum and maximum travel limits for X and Z Coordinate axis of your machine. **Note**: These parameters are not applied and reserved for future use.

7.2 Geometry

This refers to the types of geometry that can be defined and used in TURN Module.

7.2.1 Part Geometry

Part Geometry constitutes the end product of the manufacturing operation. This is also the design model. Design models in various data formats can be imported into Rhino. These design models can either be solid models or surface models or even faceted triangle models.

Refer to the following section for information on defining part geometry for Turning - Part Geometry

7.2.1.1 Part Geometry Types

TURN Module requires Solid, Surfaces, Polygon Meshes, Regions/Curves that define the part geometry. Since all parts that can be created in a 2-Axis turning machine are solids of revolutions, it is enough to describe the profile that needs to be revolved to create this shape. The profile can be created in VisualCAD as a region or curve. TURN Module automatically creates the 2D silhouette of this part region when 3D Solid or Surface geometry is selected as TURN part geometry.

Basic steps

1. Make sure you have either a 2D profile or a 3D surface model visible.
2. From the TURN Program tab, select Part and then pick Select Turn Part.
3. Define the Turn part using the Select Part Geometry for Turning dialog,

The First Quadrant, XY Plane Restriction

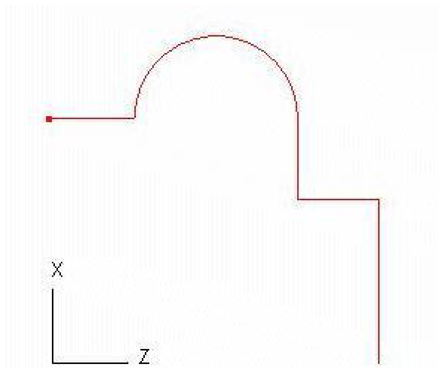
Furthermore, TURN Module places a further restriction that these part geometries need to be constrained to lay ***only*** in the first quadrant of the ZX plane in turn coordinate system. This would essentially be XY plane (Top view) in world coordinate system in VisualCAD.

If Geometry Falls Outside the First Quadrant of the XY Plane

TURN Module will be unable to process a part region that fall outside the first quadrant. If the selected part region is outside the first quadrant, TURN Module will trim this to the first quadrant.

Example of Curves Correctly Positioned

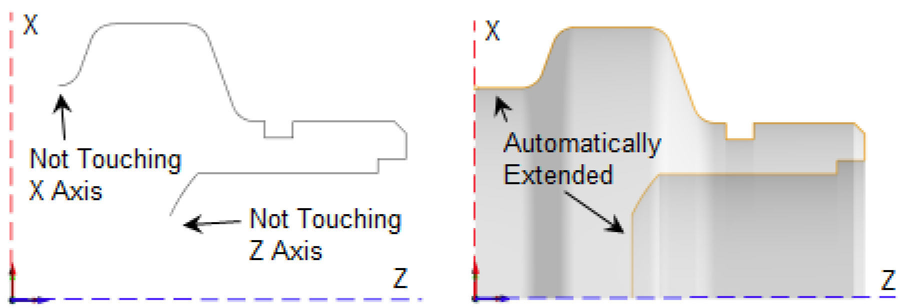
Example below shows curve region correctly positioned in the first ZX quadrant of the turn coordinate system touching both the X and Z axis. This would essentially be XY plane (Top view) in world coordinate system in VisualCAD.



Example of Curves Correctly Positioned within the First Quatrant of the XY Plane

Example of Curves NOT Touching the X or Y Axis

Example below shows region not touching the X axis and/or Z axis



Part Profiles are Automatically Extended to the X and Z axis

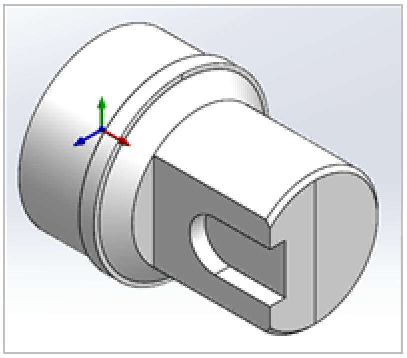
In such cases, TURN Module automatically extends the part regions to the X and Z axis when the regions do not touch the X axis and/or Z axis.

Part regions need to be constrained to the first quadrant of the ZX coordinate system.

Parts can be imported or can be created within VisualCAD using the Geometry creation and editing tools found under the CAD ribbon bar.

TURN Geometry with MILL features

You can select 3D geometry (surfaces, solids or meshes) for defining the TURN part that contains milling features. Such milling features are ignored while the turn part profile is being defined.

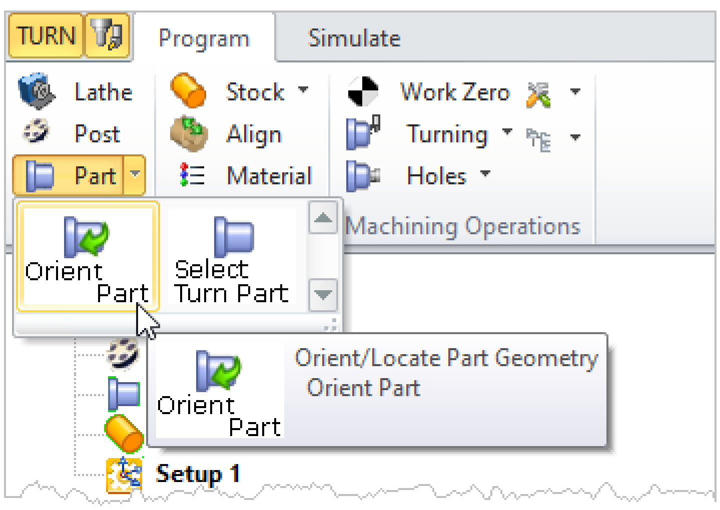


7.2.1.2 Orient Part

Once part geometry is loaded, you can set the cutting direction. This allows you to orient the part to have it aligned properly for machining. This dialog can be invoked by selecting Setup and Orient Part from Program tab under the Machining Browser. This dialog has some very powerful options to help you quickly position your part for machining. refer to the dialog and options listed below.

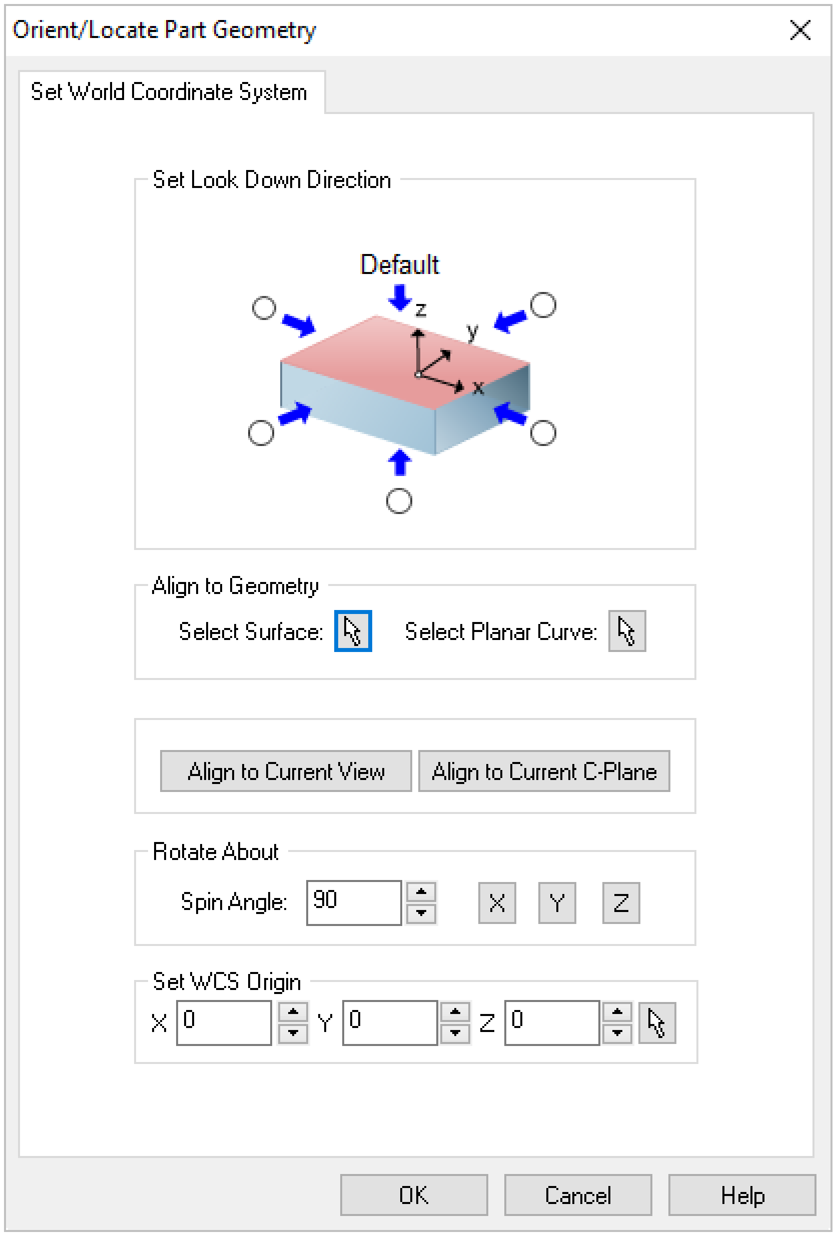
Orient Part icon

Locate the Orient Part Icon



Machining Browser: Setup, Orient Part

Dialog Box: Orient/Locate Part Geometry



Dialog Box: Orient Geometry

Set Look Down Direction

Select the radio button representing the orthographic view of your part that you wish to machine. For example, select the right side radio button (on the positive X side in the dialog image) will rotate your part so that the positive X direction becomes the positive Z direction.

Align to

If your part is not aligned orthographically, you can use these options to align your part to selected geometry or active display element. Select from one of the options that will orient the part that you wish to machine:

Select Surface

Choose the Pick button and then select a surface of your part to orient to. The part will be aligned such that the surface normal direction is aligned with the -Z axis.

Select Planar Curve

Choose the Pick button and then select a Planar Curve of your part to orient to. The part will be aligned such that the curve will be parallel to the XY plane (i.e., normal to the Z axis).

Align to Current View

Pick this button to align the part such as the Current View direction is aligned with the Z axis (i.e., you are looking in the -Z direction).

Align to Current C-Plane

Pick this button to align the part so that the Current C-Plane becomes parallel with the XY plane.

Rotate About

Use this option to Rotate About one of the principal XY or Z axis. Enter an angle and then select the button representing the axis you wish to Rotate About.

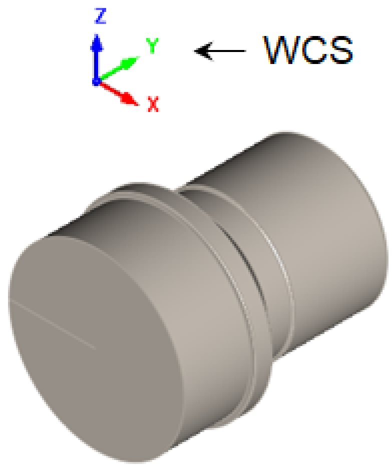
Set WCS Origin

You can also independently set the coordinate location for the WCS. So for example, you can orient the part normal to a surface and then also choose a point on the surface to become the new WCS origin. In this case the part would be oriented to the surface while the point on the surface remains at the WCS.

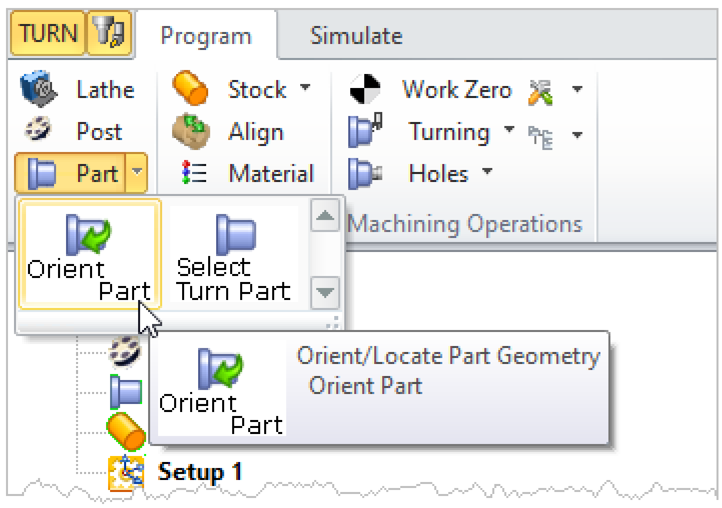
Example: Steps to orient your part for Turning

In TURN, the default rotational axis is along the X Axis of the World Coordinate System or WCS. **Note:** Once you select a direction and pick OK from the dialog, all of the geometry will be moved and/or rotated. In the example part shown below we want to orient the part so that the rotation axis is along the X Axis of the WCS. We also want the WCS origin to be located at the center of the back face of the part. Look at the steps below to see how it's done.

1. From the Program tab select Part and then Orient Part.



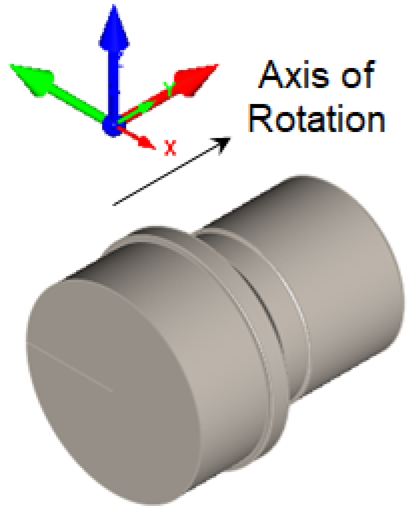
Incorrect Orientation



Machining Browser: Setup, Orient Part

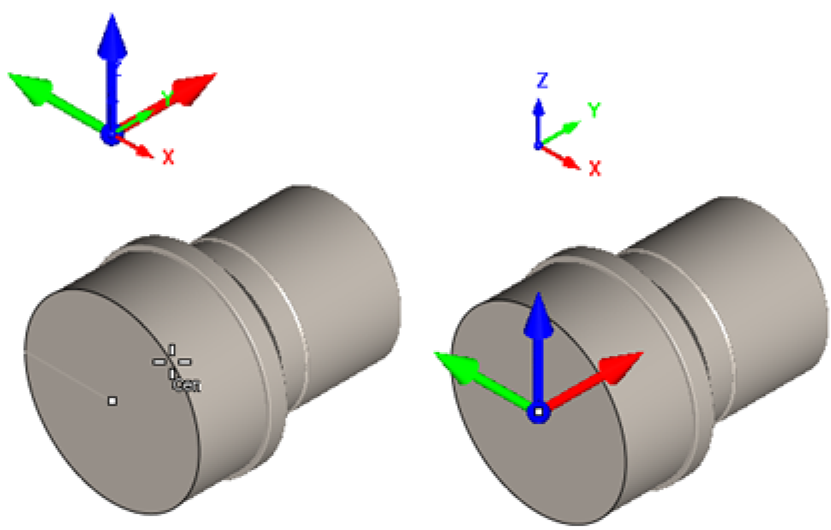
The dialog is displayed and the WCS is also displayed on the screen.

2. Under Rotate About, set the Spin Angle to 90 and then pick the Z button to rotate the WCS displayed on the screen. We want the X Axis of the WCS to point along the rotational axis of the part.



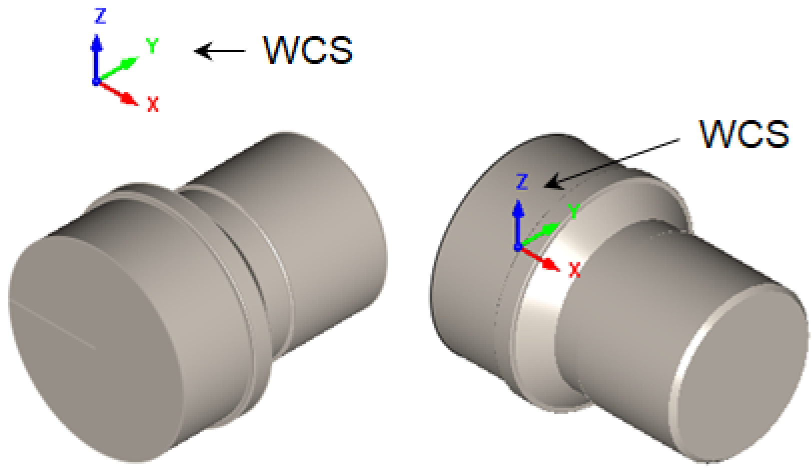
3. Now, let's locate the WCS origin. In the dialog under Set WCS Origin, select the Pick button.

4. Select the center point of the circular face of the back of the part. This is the face that will be mounted on the spindle of the lathe. When you select the point, the WCS triad is move to that point.



Before After

5. Now pick OK from the dialog and the part is oriented as desired.



Before After

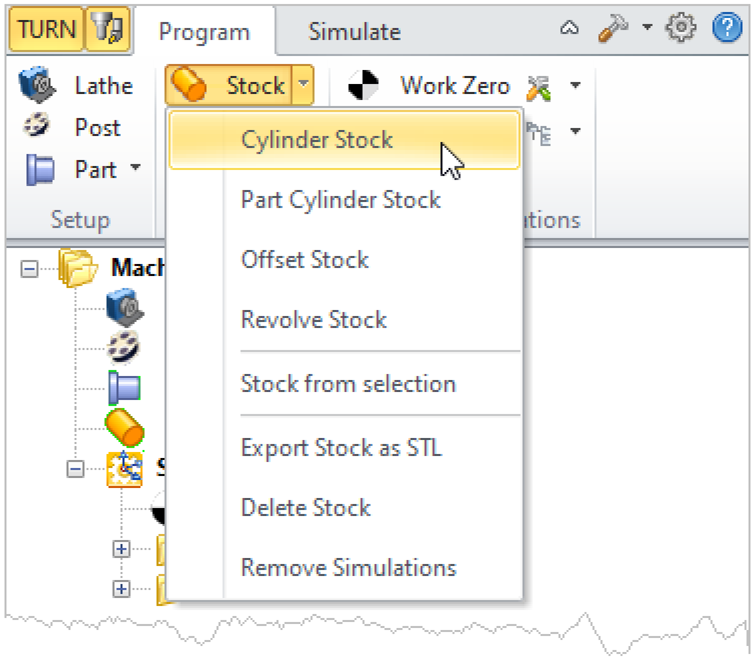
7.2.2 Stock Geometry

Stock Geometry represents the raw stock from which the designed part needs to be manufactured. TURN module allows the definition of various kinds of stock models.

7.2.2.1 Cylinder Stock

You can define the raw stock model as a simple cylinder by selecting the Cylinder Stock option from Create Turn Stock Model under the Program tab in Machining Browser.

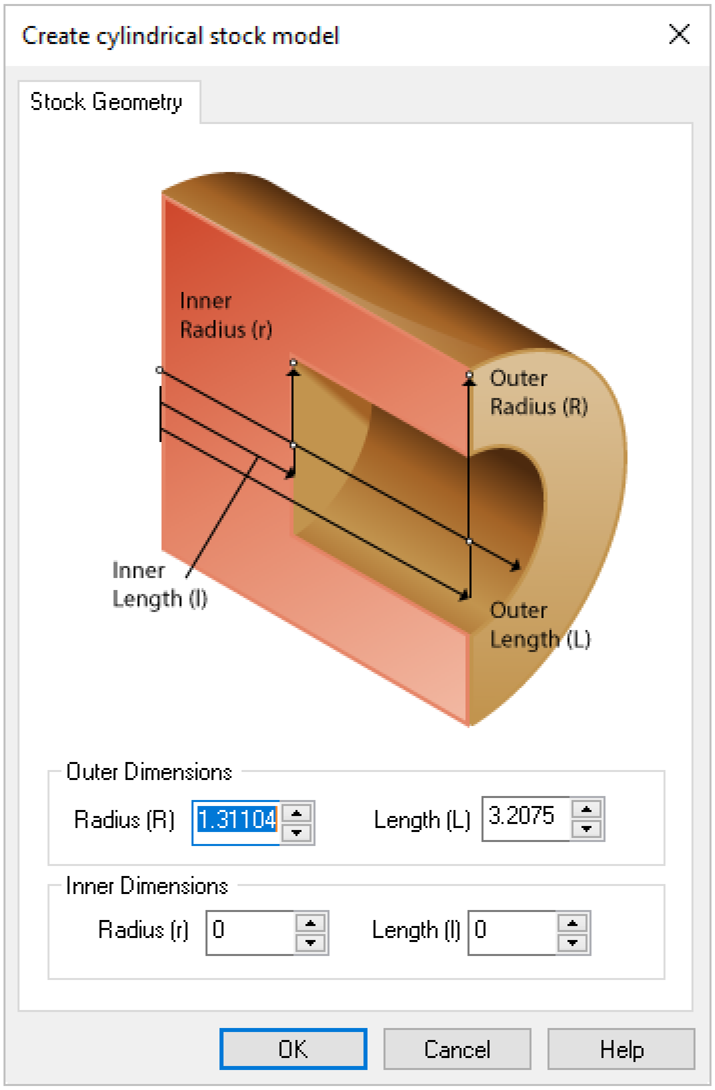
Stock Menu Selection



Cylinder Stock Menu Item

Dialog Box: Create cylindrical stock model

When you select this option, the following dialog will be invoked.

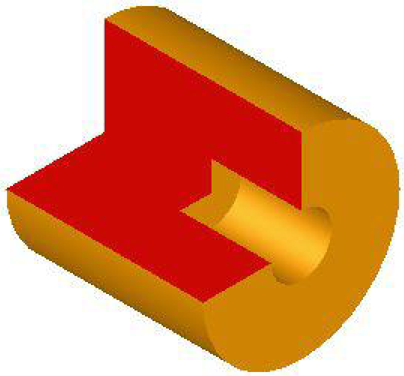


Dialog Box: Create cylindrical stock model

Inner/Outer Radius Dimensions

In this type of Stock model user can specify the Radius (Outer and Inner) and Length (Major and Minor) for the stock. By default TURN Module displays with outer Radius (R) and Major Length (L) values based on the defined part geometry.

The stock is displayed as a cylinder positioned at the reference point of the lathe machine. Its color can be set in the Color Preferences under CAM Preferences.

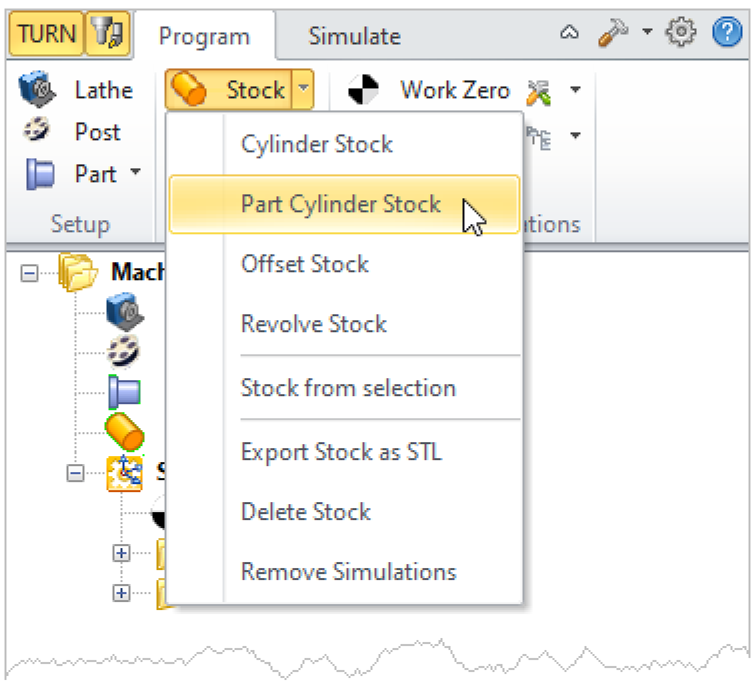


Cylindrical Stock Example

7.2.2.2 Part Cylinder Stock

You can define the raw stock model as a part bounding cylinder by selecting the Part Cylinder Stock option from Create Turn Stock Model under the Program tab in Machining Browser.

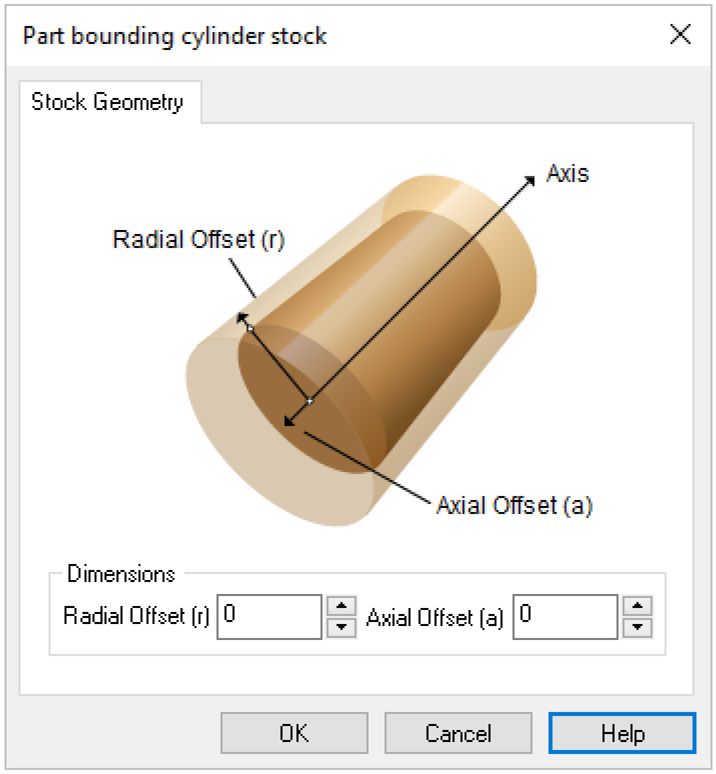
Stock Menu Selection



Part Cylinder Stock Menu Item

Dialog Box: Create cylindrical stock model

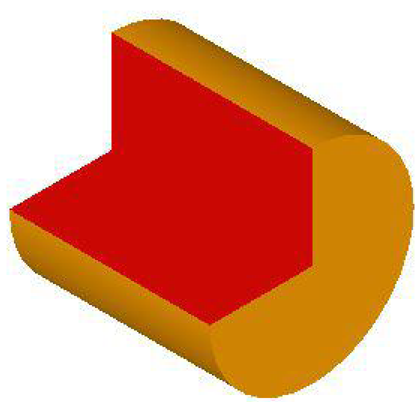
When you select this option, the following dialog will be invoked.



Dialog Box: Part bounding cylinder stock

Radial/Axial Offset Dimensions

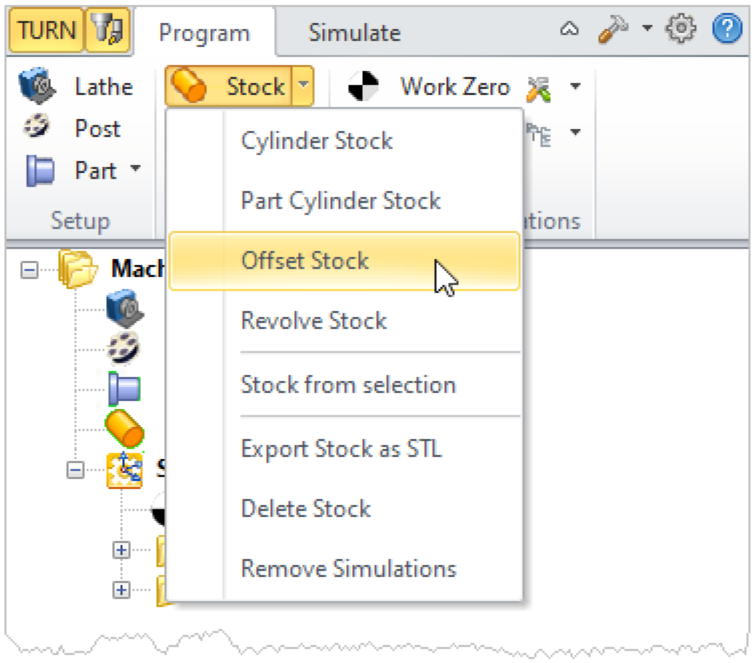
You can define the cylinder by simply specifying the Radial (r) and the Axial (a) offset distances. The bounding cylinder will be calculated based on the defined part geometry and these offset values will be used to expand the cylinder in both the radial and axial direction.


Create cylindrical stock model example

7.2.2.3 Offset Stock

You can define the raw stock model as a uniformly offset part model by selecting the Part Offset option from Create Turn Stock Model under the Program tab in Machining Browser.

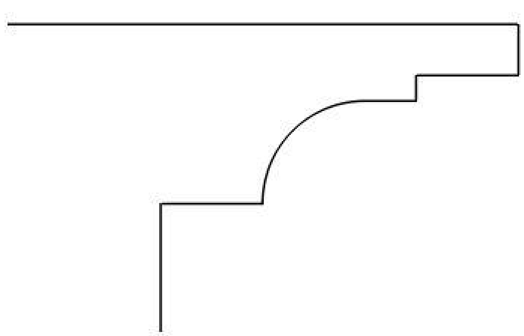
Stock Menu Selection



Offset Stock Menu Item

Select 2D Planar Curve to Offset

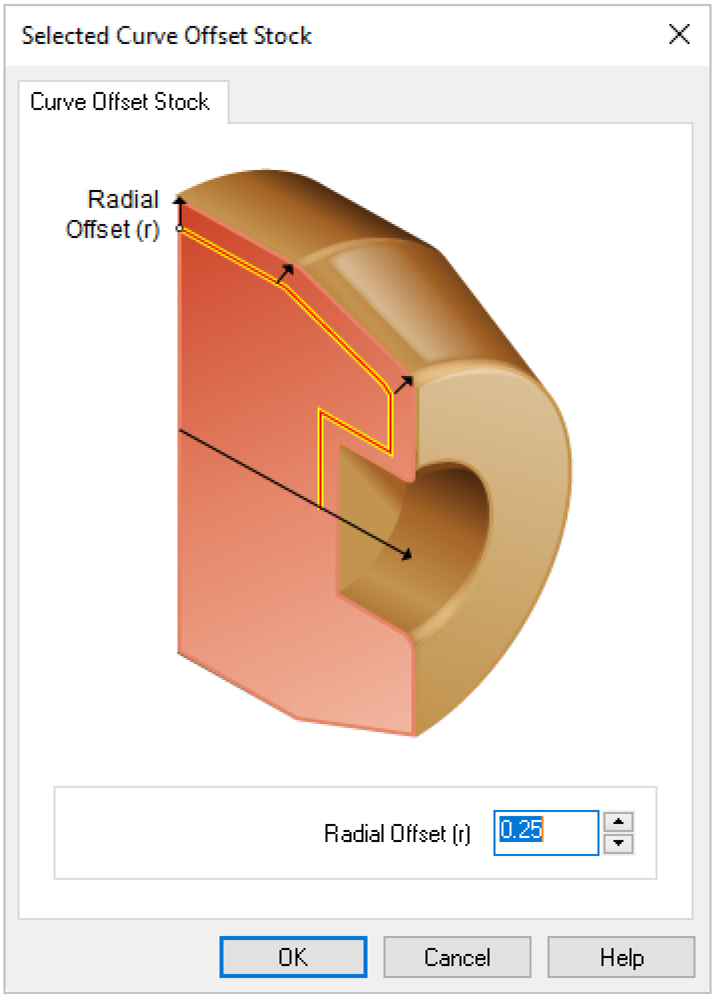
First, you need to select a 2D planar curve on the XZ axis before creating a Curve Offset Stock. User can then specify offset value to create the stock model.



Select 2D Planar Curve to Offset

Dialog Box: Selected Curve Offset Stock

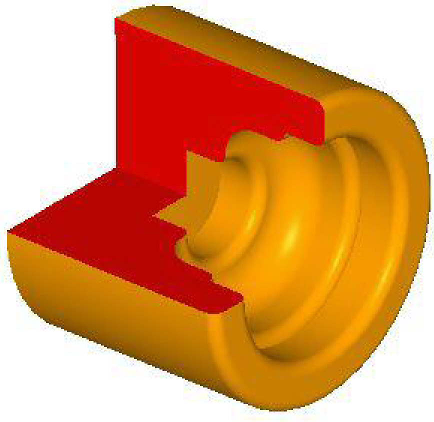
The following dialog will be invoked.



Dialog Box: Selected Curve Offset Stock

Radial Offset (r)

When you click on the OK button, a stock model based on the specified Radial Offset (r) is created. You can switch to the Simulate tab of the browser window to display the stock model that was created.

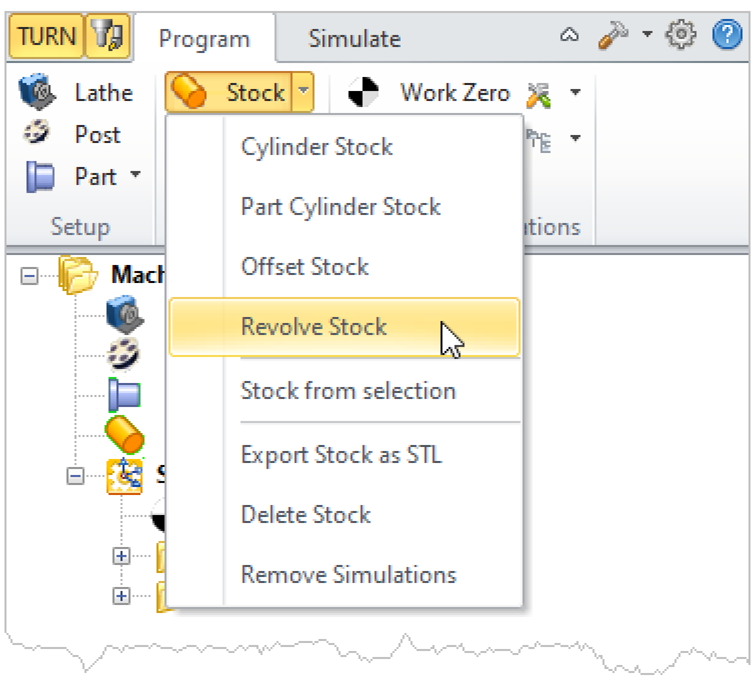


Selected Curve Offset Stock Example

7.2.2.4 Revolve Stock

You can define the raw stock model by revolving selected 2D curve profile as Revolve Stock from Create Turn Stock Model under the Program tab in Machining Browser.

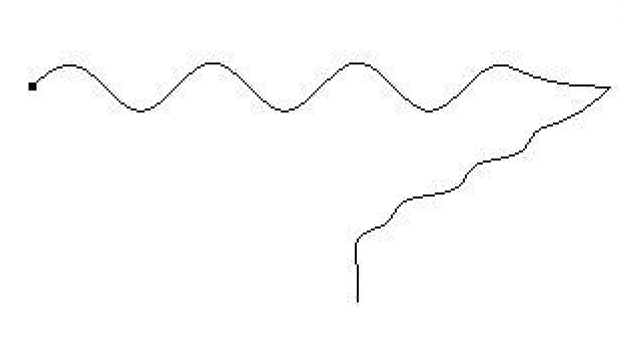
Stock Menu Selection



Revolve Stock Menu Item

Select 2D Planar Curve to Revolve

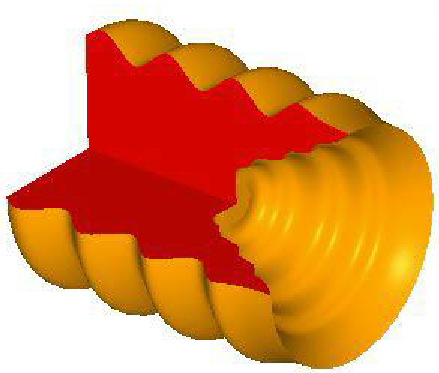
First select a 2D planar curve on the XZ axis before creating Revolve Stock.



Select 2D Planar Curve to Revolve

Revolve Stock Example

The system creates a stock model by revolving the 2D profile curve about the Z axis. You can switch to the Simulate tab of the browser window to display the stock model that was created.



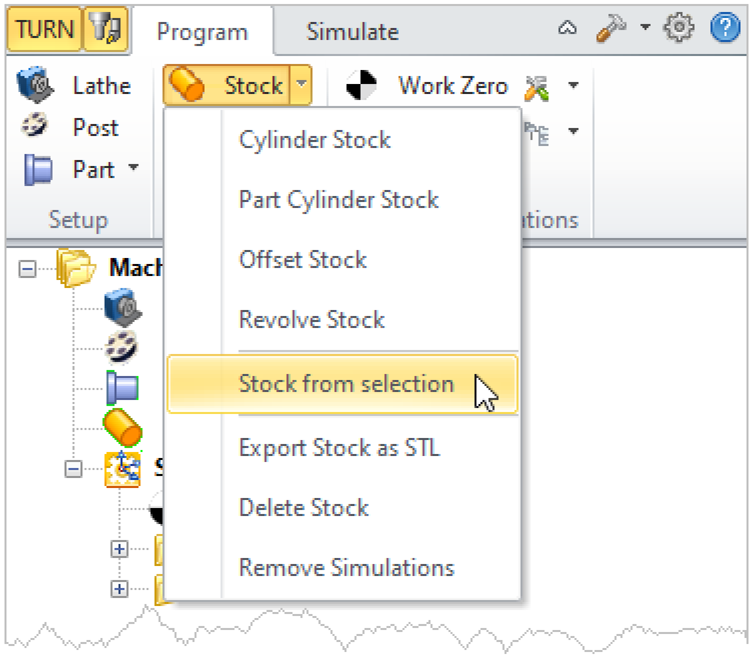
Revolve Stock Example

7.2.2.5 Stock from Selection

You can define the raw stock model from the currently active selected 3-D geometry. You can select 3D surfaces and/or meshes and then selects Stock from Selection option from Create Turn Stock Model under the Program tab in Machining Browser.

No dialog will be invoked but the system will use the selected geometry and create a triangulated stock model. You can switch to the Stock tab of the browser window to display the stock model that was created.

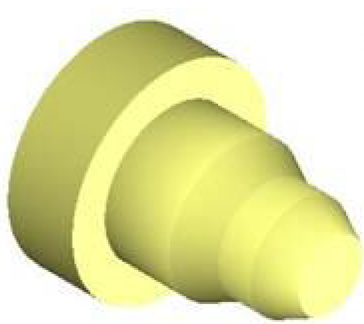
Stock Menu Selection



Stock from Selection Menu Item

Stock from Selection Example

You can switch to the Simulate tab of the browser window to display the stock model that was created.



3D geometry selection for stock

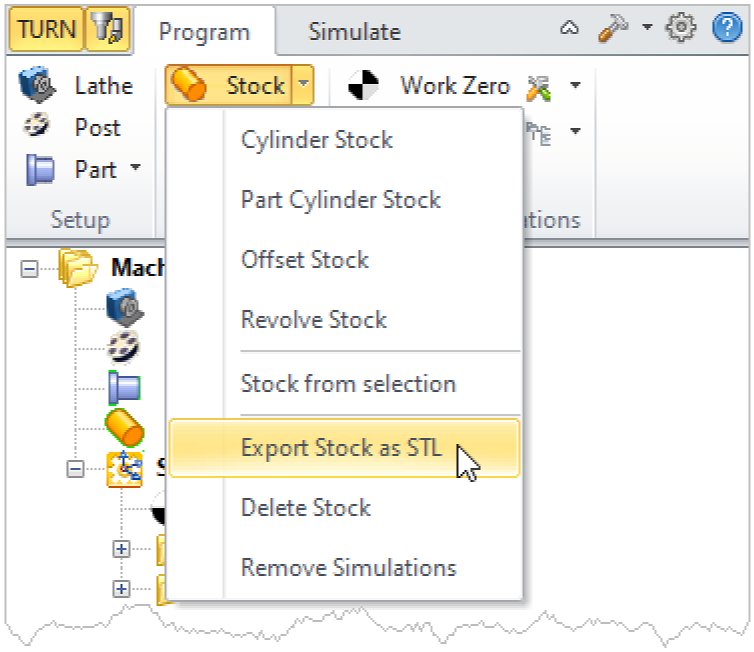
Stock created from 3D geometry

Stock from selection will fail to create stock if the selected 3-D geometry is not a watertight model.

7.2.2.6 Export Stock as STL

User can export the stock geometry by selecting Export Stock as STL from Create Turn Stock Model under the Program tab in Machining Browser.

Stock Menu Selection



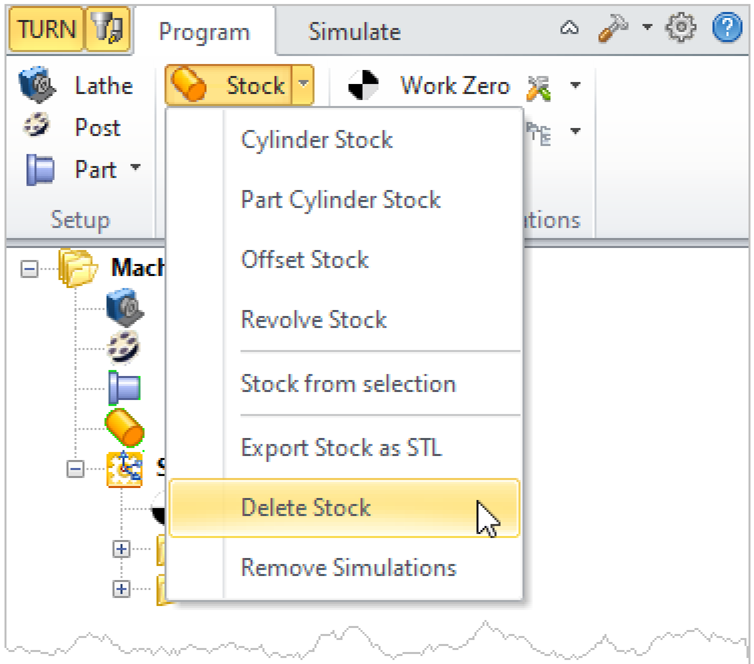
Export Stock as STL Menu Item

A Save As dialog appears that allows you to save the stock model as STL. The stock model can also be exported by selecting the stock entry under Machining Job, right mouse button click and select Export Stock as STL.

7.2.2.7 Delete Stock

User can delete the stock geometry by selecting Delete Stock from Create Turn Stock Model under the Program tab in Machining Browser.

Stock Menu Selection



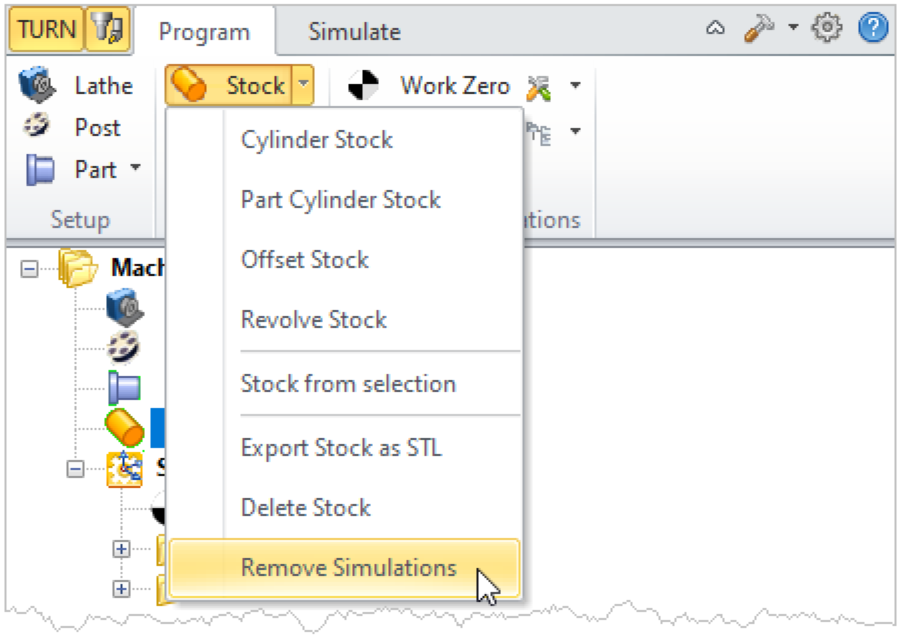
Delete Stock Menu Item

The stock model can also be deleted by selecting the stock entry under Machining Job, right mouse button click and select Delete Stock.

7.2.2.8 Remove Simulations

You can remove the in-process stock model by selecting Remove Simulations from Stock menu under the Program tab in Machining Browser. This is different than the Delete Stock command. The stock is not deleted, only the current simulations are removed.

Machining Browser: Remove Simulations



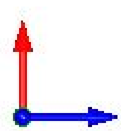
Machining Browser: Remove Simulations stock menu

7.3 Setup

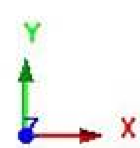
Setup defines the Turn coordinate system. CNC turning centers use the Cartesian coordinate system for programmed coordinates but they are typically different from that used in milling. Turning centers follow the convention that axis of rotation that is aligned with the spindle is designated as the Z axis. Secondly the axis perpendicular to this axis along which the tool travels to cut into the stock is designated the X axis. Thus the part is rotated about the Z-axis of the lathe machine. Moving the tool along the Z-axis provides the direction of feed and moving it along the X-axis provides the depth of cut.

Triad Display States

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

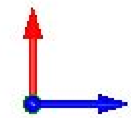
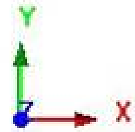
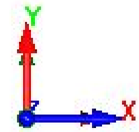


CSYS



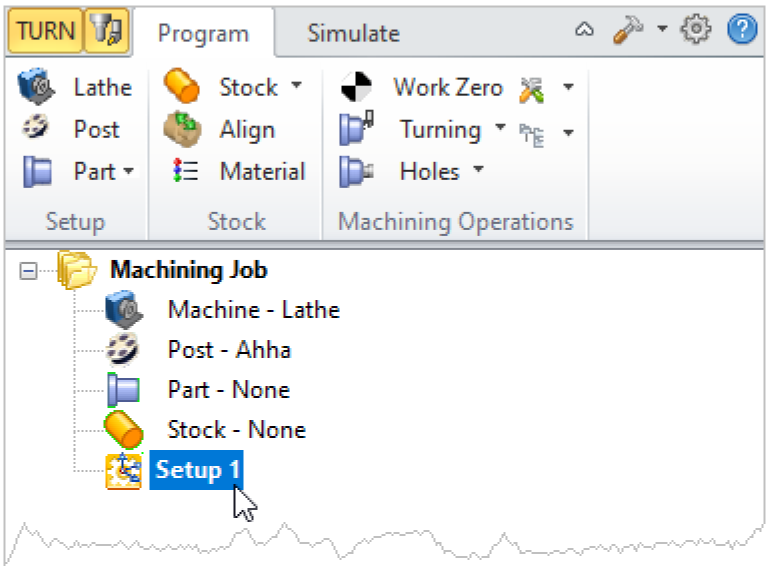
WCS

Orientation Parallel to	Triad Display States		
	CSYS Visibility ON	CSYS Visibility OFF	CSYS Visibility ON

	WCS Visibility OFF	WCS Visibility ON	WCS Visibility ON
Top View			

Setup 1 in the Machining Browser

By default Setup 1 is created when a new part is loaded. The Setup cannot be edited in the Machining browser.

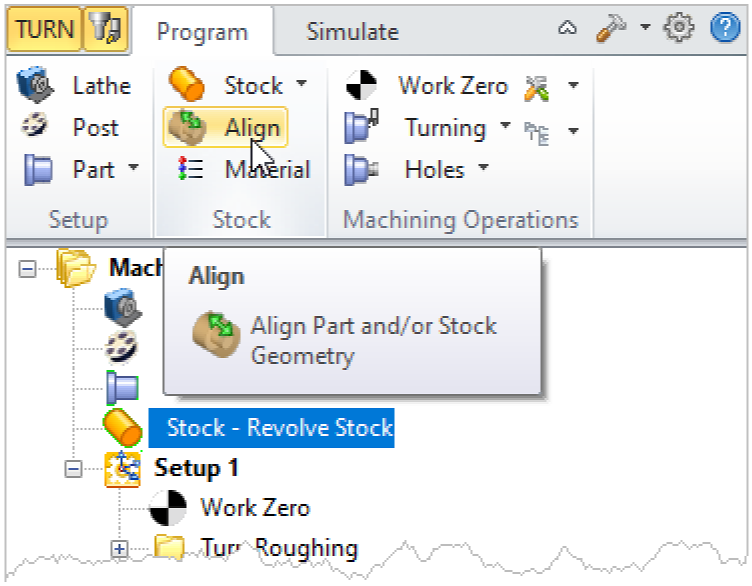


Setup1 in the Machining Browser

7.4 Align Part and Stock Models

It is typical for users to need the ability to position stock geometry in some geometric relationship with the part geometry. This dialog offers a convenient method of relative positioning the stock along the Z axis based on defined part geometry. This dialog can be invoked by selecting Align Part Stock from Program tab under the Machining Browser.

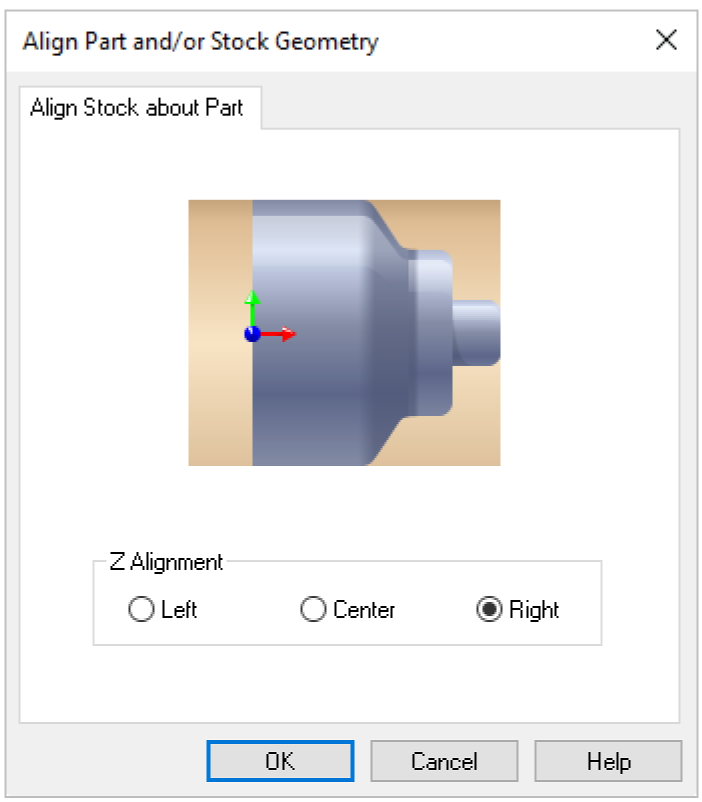
Align (Menu Selection)



Align Part and Stock Models Menu Item

Dialog Box: Align Part and Stock Geometry

The Align Part and Stock Geometry dialog appears as shown below.



Dialog Box: Align Part and Stock Geometry

Z Alignment

> Turn part geometry needs to be defined before aligning stock to part. For information on defining part geometry, refer to Part Geometry.

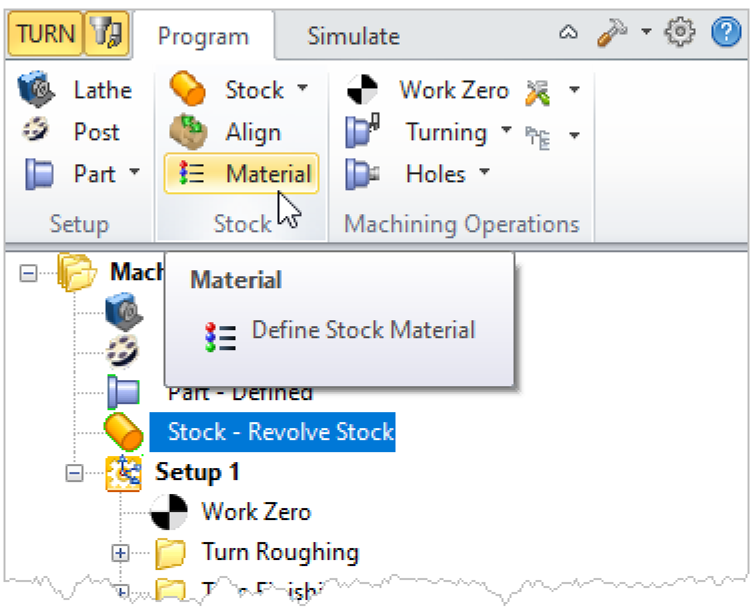
Once both part and stock geometry are defined in TURN Module, use this dialog to perform the relative positioning. Select the necessary Z alignment options using the appropriate radio buttons in this dialog.

7.5 Material

This allows selection of material for Stock geometry. User can select a material from the available list of materials. Selecting a material displays the texture for the material. This texture is applied to stock geometry and can be displayed during simulation.

Machining Browser: Material menu item

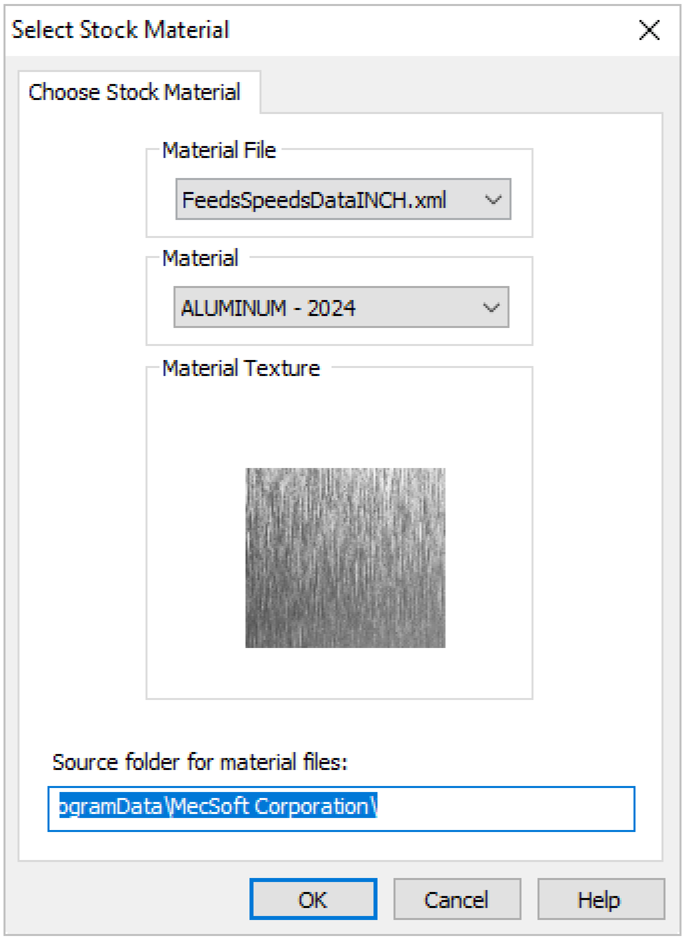
This dialog can be invoked by selecting Material from Program tab under the Machining Browser.



Material Menu Item

Dialog Box: Select Stock Material

Choose Stock Material dialog appears as shown below.



Dialog Box: Select Stock Material

Material File

This points to file where all materials are defined.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in RhinoCAM. (C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Materials).

The Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

If part unit is set to Inches, RhinoCAM automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

The material file is an .xml file format, which can be edited using any text editor to add newer materials.

See Feeds and Speeds for information on the format of the material file and adding new materials.

Material

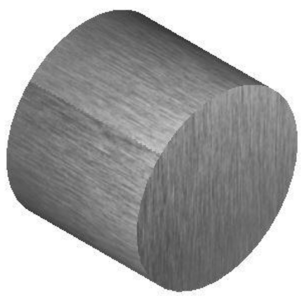
This lists all materials available in the selected Material File. Selecting a Material from the list displays the material name and material texture.

Material Texture

A preview of the Material Texture is displayed for reference.

Material Texture Visibility

Click OK. Once you have defined a Stock geometry, click Material Texture Visibility icon under Program or Simulate tab in Machining Browser to display the texture applied to stock model.

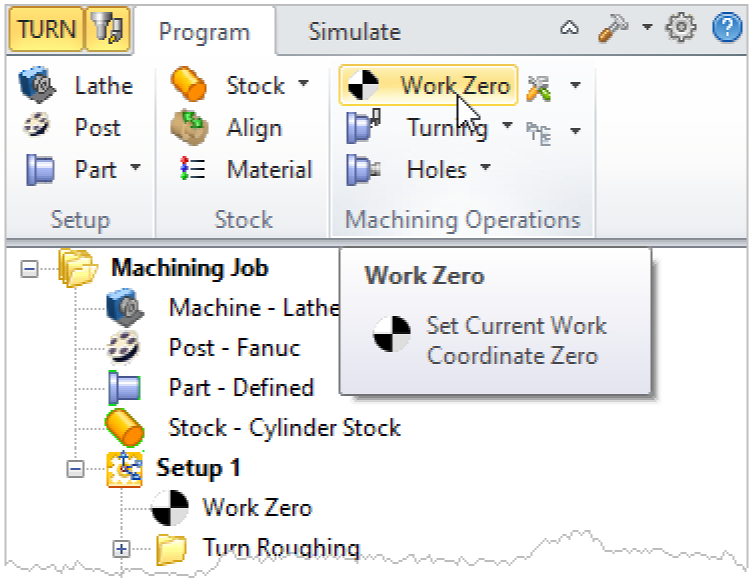


Material Texture (TURN)

7.6 Work Zero

Work Zero defines the work coordinate (part or stock) origin. This is also known as the program zero. Work Zero translates the turn machine coordinate system origin (CSYS) to the desired location on the Z axis. Typically this is set to the face of the part or stock geometry. This dialog can be invoked by selecting Work Zero from Program tab under the Machining Browser.

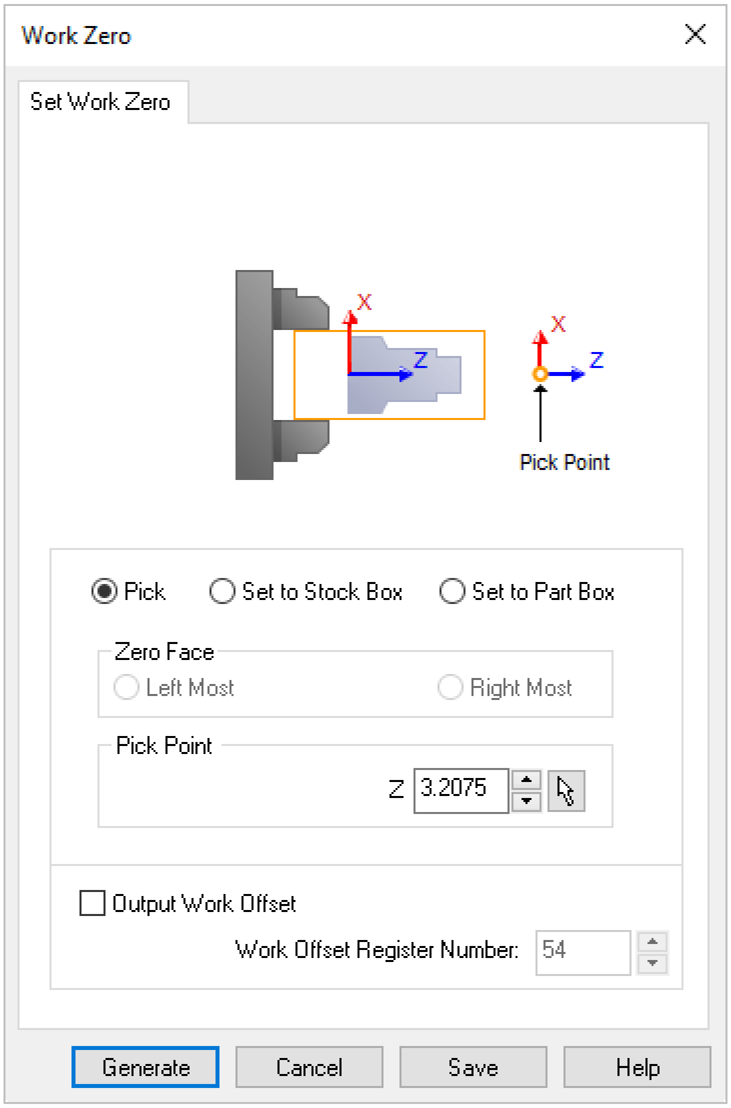
Work Zero (Menu Selection)



Work Zero Menu Item

Dialog Box: Work Zero

Set Work Zero dialog appears as shown below. You can set the origin by explicitly picking a point on the Z axis or can set it with respect to the Part or Stock geometry bounding boxes.



Dialog Box: Work Zero

Pick

If you select the Pick option, the button with the pick cursor will be activated. You can then click on this button to graphically select a point to set Work Zero.

You can use object snaps located in Rhino's status bar to snap to part geometry.

Set to Stock Box

Selecting this item will activate the Zero Face section of the dialog. You can then select the zero face to the Left Most or to the Right Most face of the stock by choosing the appropriate selections in the dialog.

Set to Part Box

Similar to the previous selection, selecting this item will activate the Zero Face section of the dialog. You can then select the zero face to the Left Most or to the Right Most face of the part by choosing the appropriate selections in the dialog.

Output Work Offset

This allows you to specify a Work Coordinate Offset number which is then output in the posted code. This is set under Work Offset Register Number. Work offsets are used to set work piece origin(s) on CNC machines that are assigned to a register number G54, G55 etc... Entering a positive number will make incremental offsets positive (i.e., G54, G55, etc.) Entering a negative number will make decremental offsets (i.e., G54, G53, etc.).

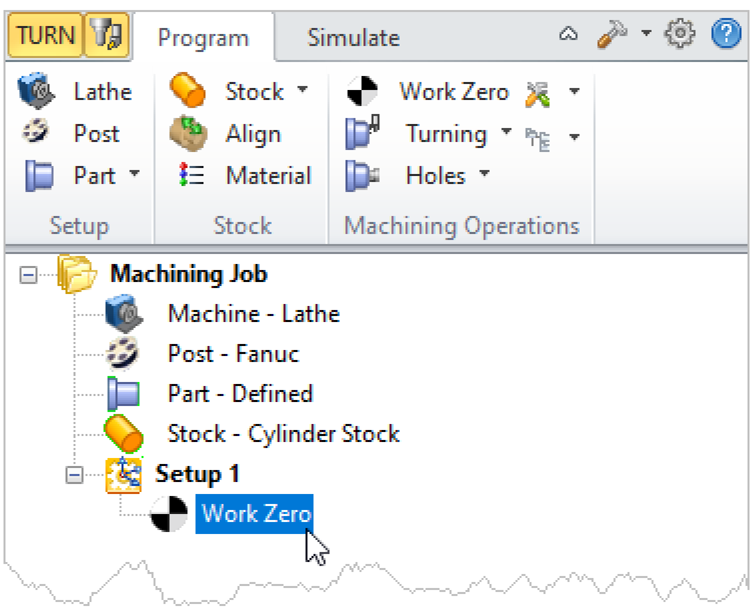
Work Zero - Output Work Offset

For example:

- To output G54, G55, etc., set the Work Offset Register number to 54. To output G54, G53, etc., set the Work Offset Register number to -54 (negative).
- The Work Offset Prefix "G" is set in the post-processor generator.

Generate

Click Generate and Work Zero is now listed under Setup in Machining Browser. The Machine CSYS origin is now translated to the specified location.

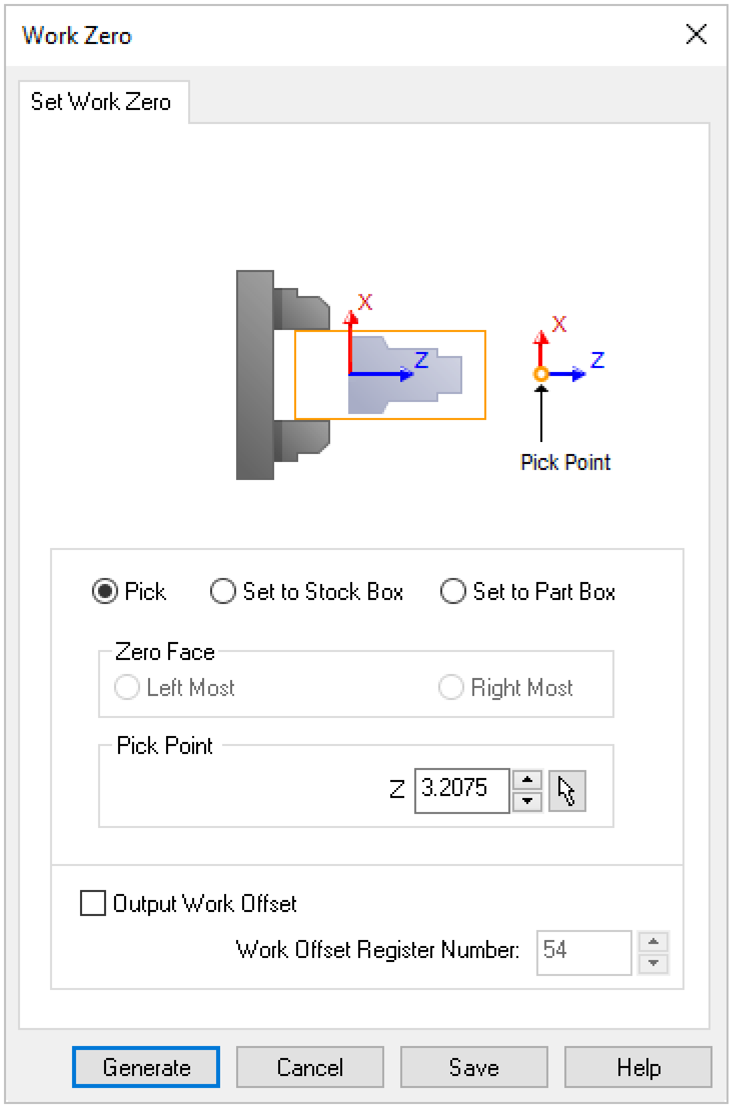


Work Zero displayed in the Machining Browser

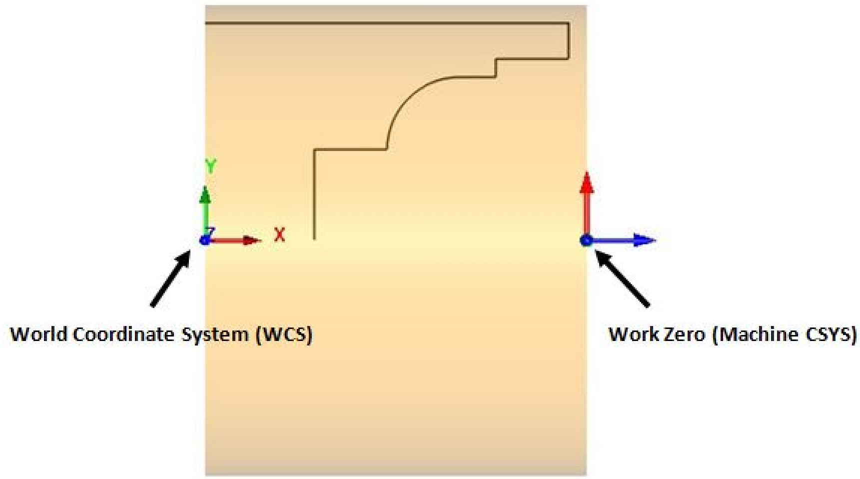
Work Zero Example

In the example shown below Work Zero is set to:

- Set to Stock Box
- Zero Face – Right Most



Dialog Box: Work Zero



7.7 Machining Features

Refer to the following section for additional information - Part Geometry

It should be noted that regions can be created and be present in a part file but if they are not selected in defining turn part geometry they will be ignored during toolpath computation. So creating a region does not make it active; you select it as turn part geometry before creating the toolpath.

7.7.1 Avoid Sections

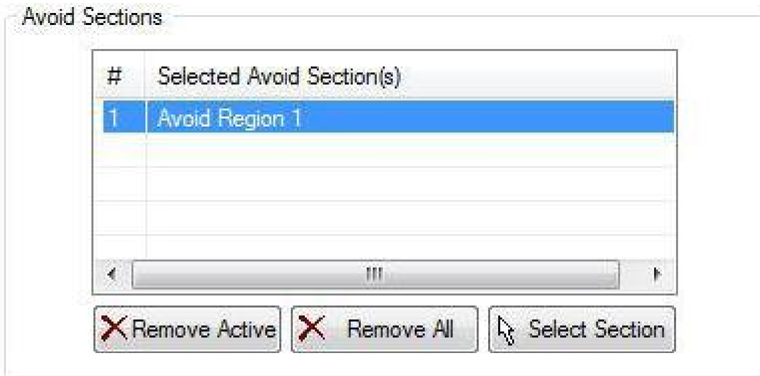
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

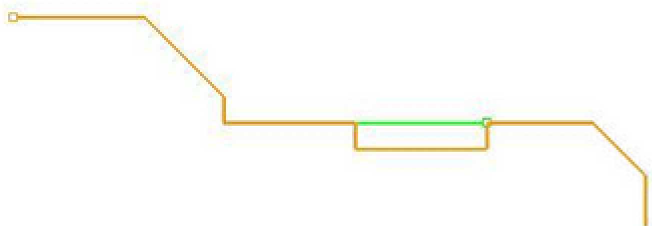
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

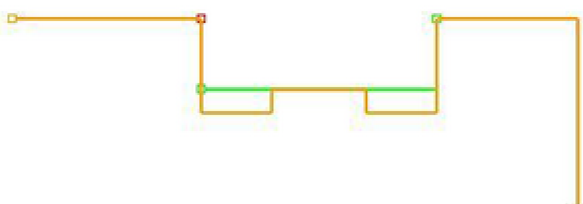


Avoid Sections, Global Parameters tab of Turn Operations dialog box

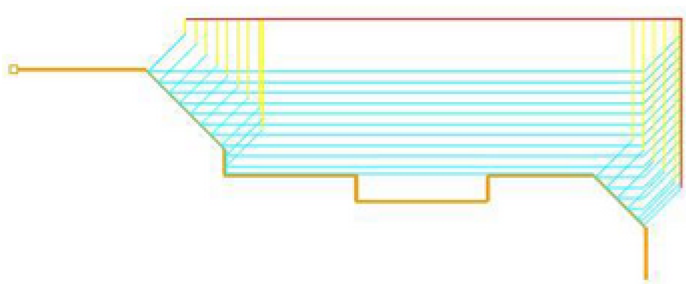
Examples for various Turn Operations



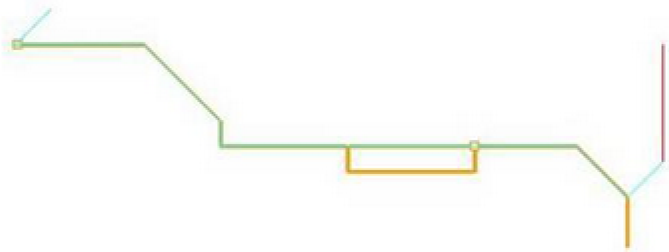
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section



Finishing toolpath with Avoid Section

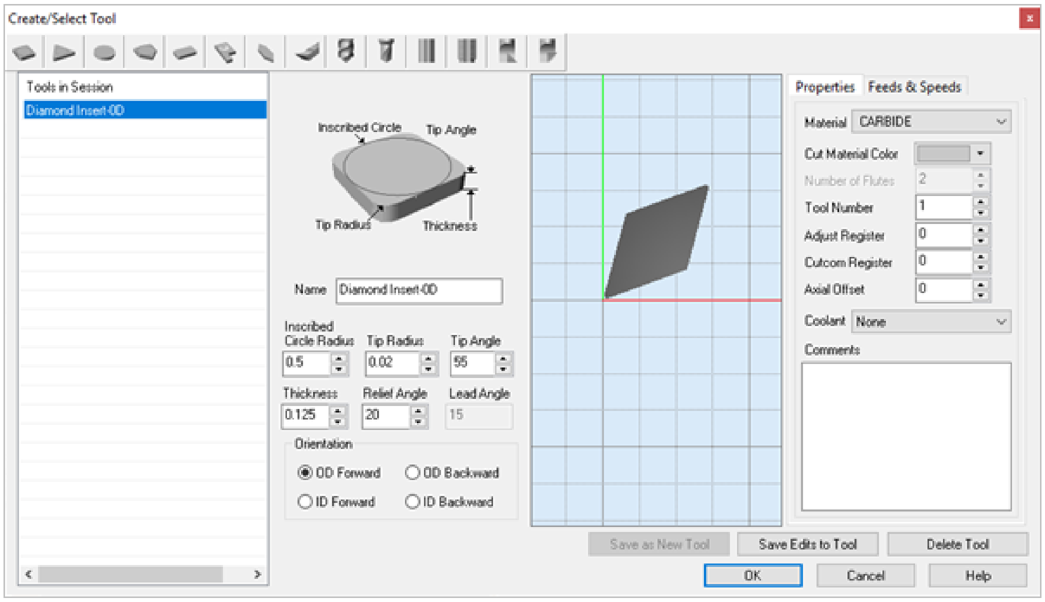
7.8 Tool

TURN Module allows you to define, use and archive various types of turning inserts and drilling tools. The tool types that are currently supported are Diamond, Triangular, Circular, Trigon, Parallelogram, Groove, Threading, Parting off, Drill, Center Drill, Reamer, Tap, Bore and Reverse Bore.

For each tool you can specify standard APT parameters: Tip radius, Relief angle, thickness, width, length, nose angle. All defined tools can be viewed in the Tools tab of the Machining Objects Browser. You can also save a set of tools to a library that can be loaded in other files.

7.8.1 Create/Select Tools

To create a tool, you either selects the Create/Edit Tool option under the Tools tab in Machining Objects Browser or alternatively by selecting the Create/Edit/Select Tool button under the Tool tab in the machining operation. This brings up the following dialog box that you can utilize to create and edit tool definitions.



Dialog Box: Create/Select Tools

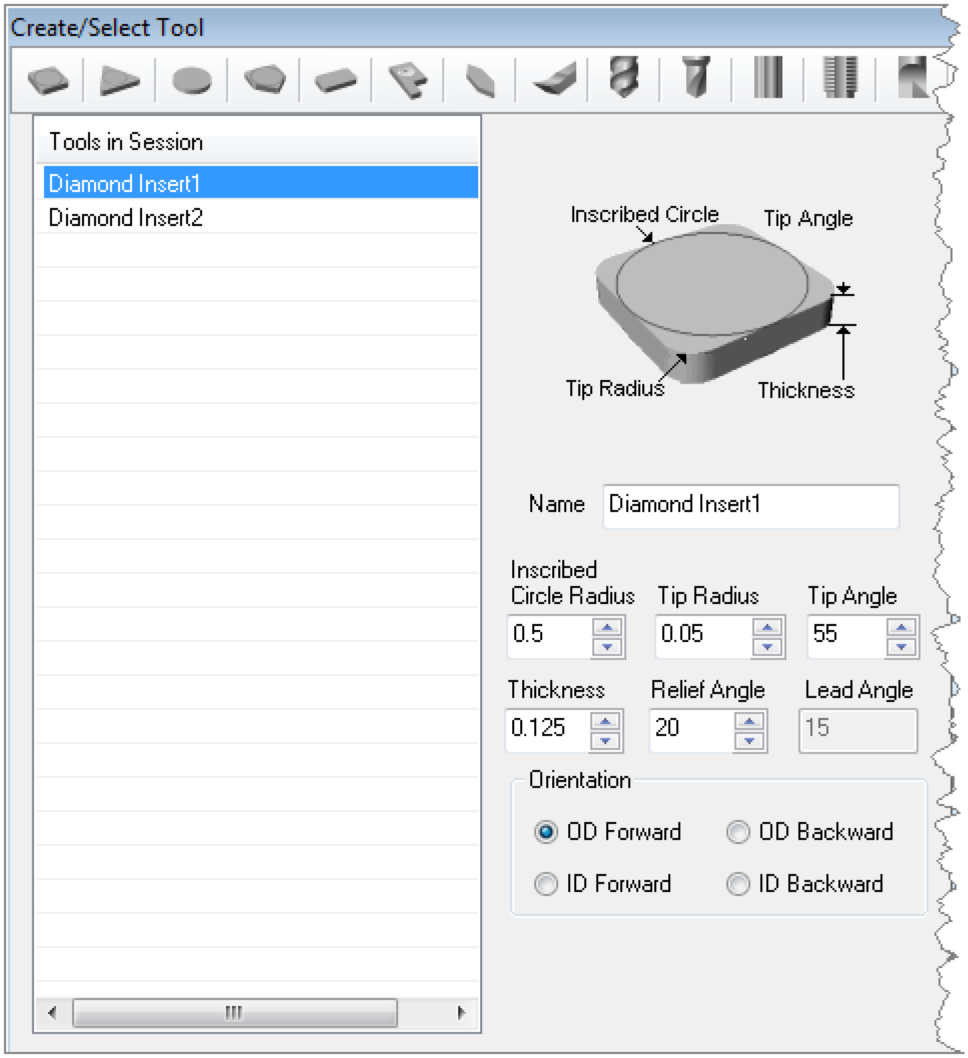
Create Tools Toolbar

The tool toolbar on the top of the dialog displays all various types of tools availabl in TURN Module. Different tool types can be defined by selecting the desired icon the dialog box.

Create Tools Toolbar

Tools in Session

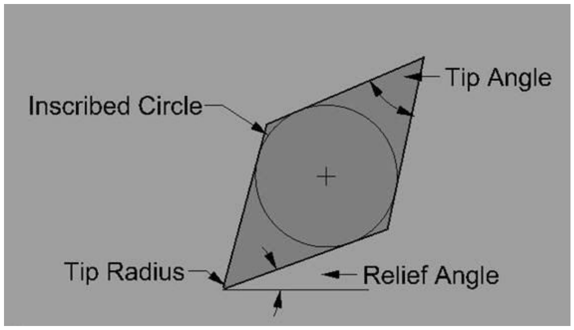
The dialog box shows the tool name of the current selection if there is one selected in the list-box under Tools in Session. If there is no selection then the tool name will be the name used for a new tool definition. The list box itself lists all of the tools of the corresponding type.



Tools in Session

Standard APT Parameters for Tool Definition

The geometry definition of the tool contains edit boxes for the Tip Radius, Relief Angle, Thickness, Width, Length and Nose Angle. These definitions are standard APT parameters for the tool definition. The parameters would be different depending on tool insert type.

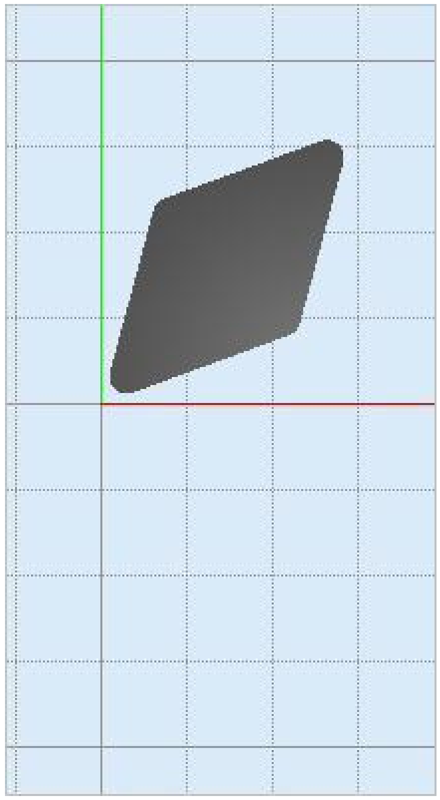


Diamond Tool Insert

Save/Edit/Delete Tool

- Save As New Tool saves a new tool and lists under Tools in Session. If a tool of the same name already exist under Tools in Session, the Save as New Tool button will be grayed out.
- Save Edits to Tool saves edits or changes made to tool parameters
- Delete Tool deletes the selected tool. A tool will not be deleted a tool if is being used in a machining operation.

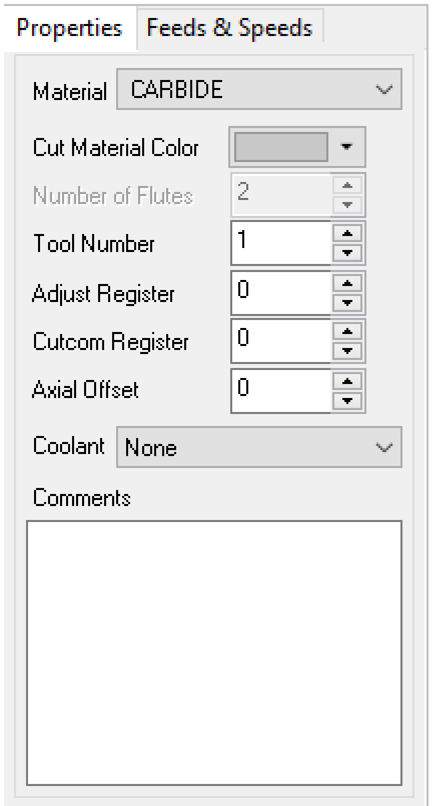
Tool Preview



Tool Preview

The Properties tab

The Properties tab to the right side of the tool preview allows you to set the Tool Material, Cut Material Color, Tool Number, Adjust Register, Cutter Compensation Register, Axial Offset and Coolant Type. The Tool Number and Adjust Register are used when post processing toolpaths.



The Properties tab

Adjust Register

This is used to set the Tool Offset register (an integer). Generally this is set the same as Tool Number. In turning applications, this is typically output along with the tool number. The posted code would output this number that corresponds to the offset value in the controller's tool table. Note the post processor needs to be configured to output the Adjust Register and 2 digit format for Tool numbers and registers.

For example

N20 T0101 M6
Where first set of 01 is the tool number and the next set of 01 points to the offset location in controllers tool table.

Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. The posted code would output D<#> and the # corresponds to the offset value in the controller's tool table.

For example

N30 G41 X 2.0 Y 1.0 D1

Where D1 points to the controllers tool table for diameter compensation.

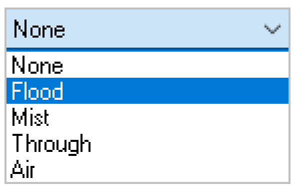
Note the post processor needs to be configured to output the Cutter Compensation. Refer to Cutter Compensation for detailed description.

Axial Offset

This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



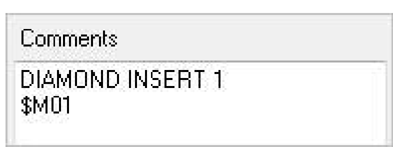
Coolant selections available

Comments

Outputs specified comments in the posted g-code before a tool change.

Comments

Adding $ as prefix would skip the comment start and end characters in the posted output file.



Comments

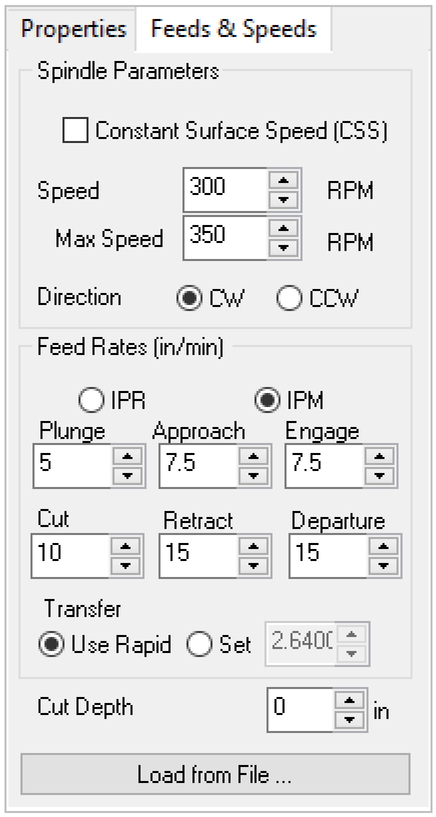
The posted code would include the comments in the output as shown below.

(DIAMOND INSERT 1)
M01

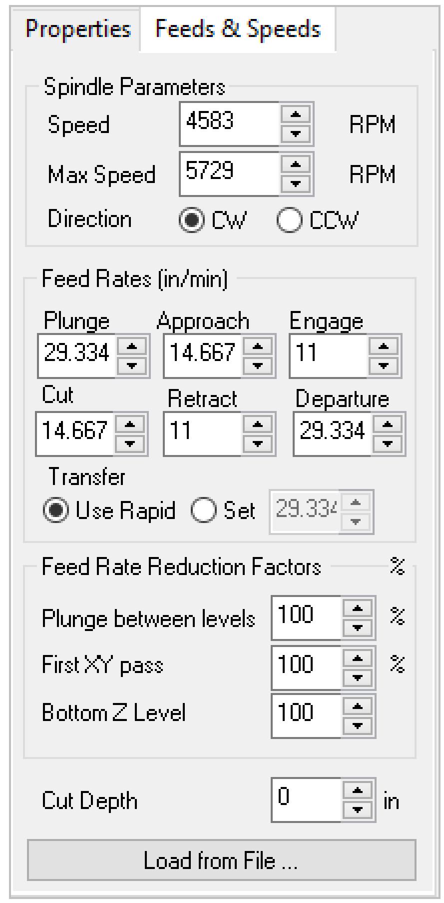
Make sure comments to output is selected in your post processor.

The Feeds & Speeds tab

The Feeds & Speeds tab located next to Properties tab allows you to set feeds and speeds for each tool. Feeds & Speeds parameters for Dill, CenterDrill, Tap, Bore and ReverseBore are slightly different from Turning inserts. Also note that there is Max Speed value for Drill tools in the TURN module.



Feeds & Speeds, Turning Inserts



Speeds & Feeds, Drill Tools

Cut Depth

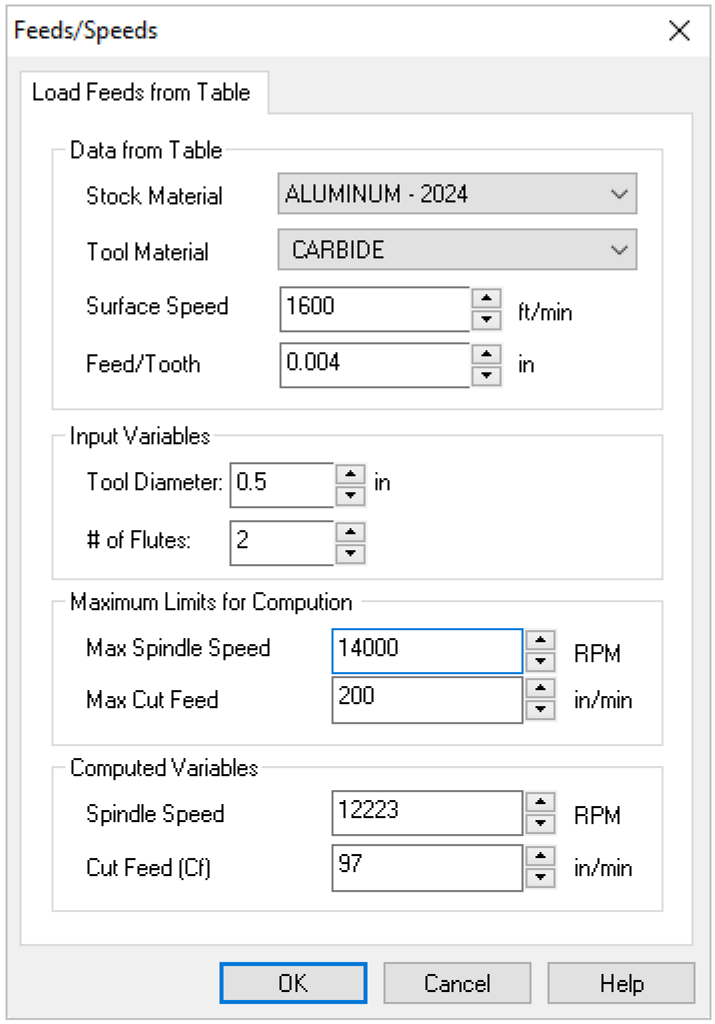
You can set the Cut Depth specific for each tool you create. If you set this value here, you will see a Depth From Tool icon next to the Rough Depth/Cut parameter in the Cut Levels tab of each operation where it applies. Selecting the icon will use this value for Rough Depth/Cut. If Cut Depth is left at 0 in this dialog, the icon will not appear in the Cut Levels tab.

Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

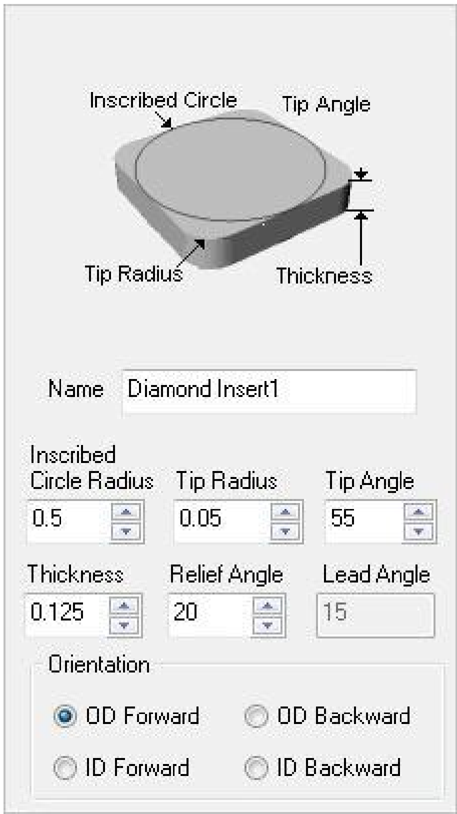
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

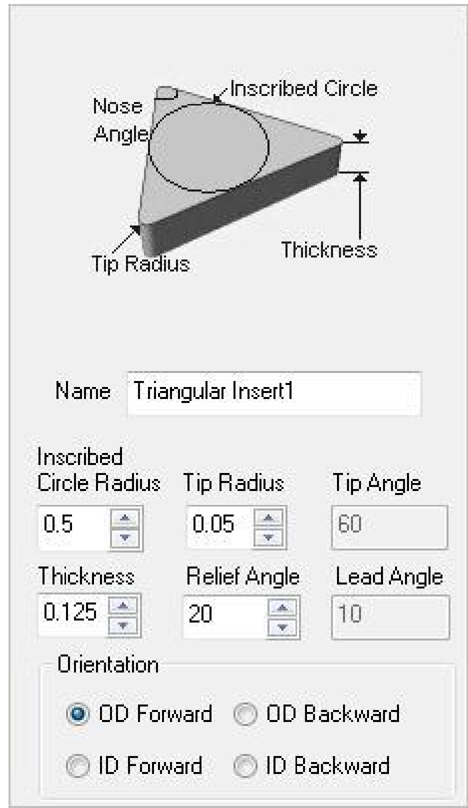
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

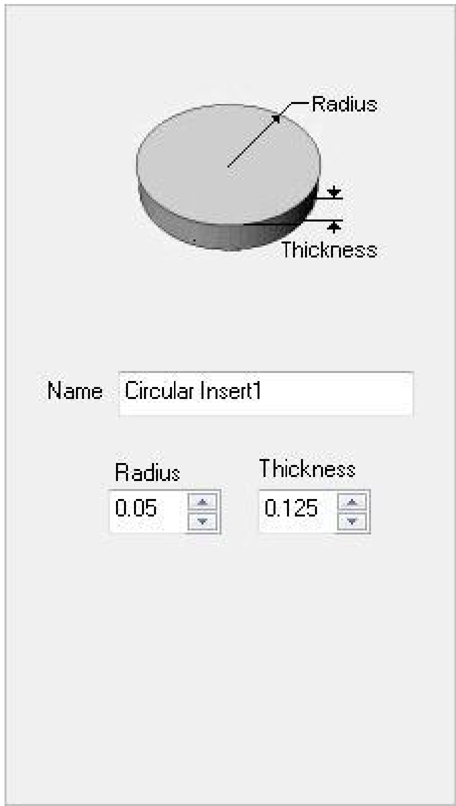
Turning insert types available



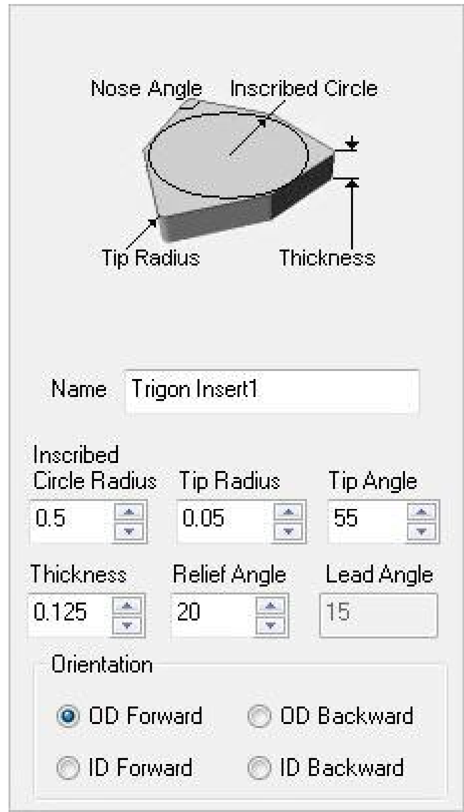
Diamond Insert



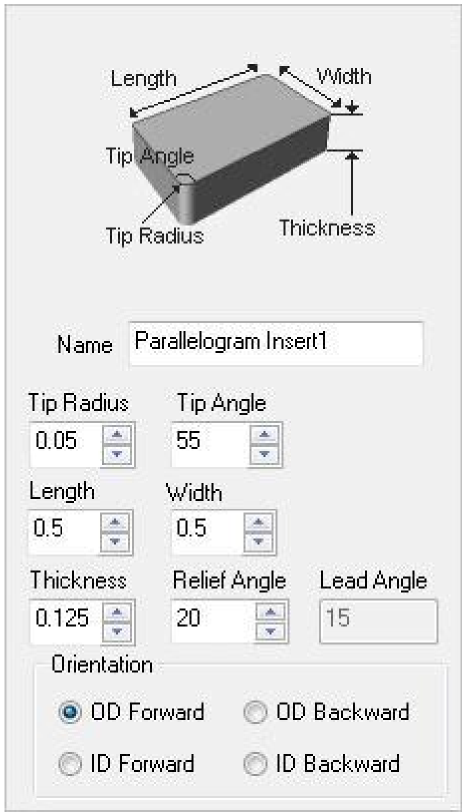
Triangular Insert



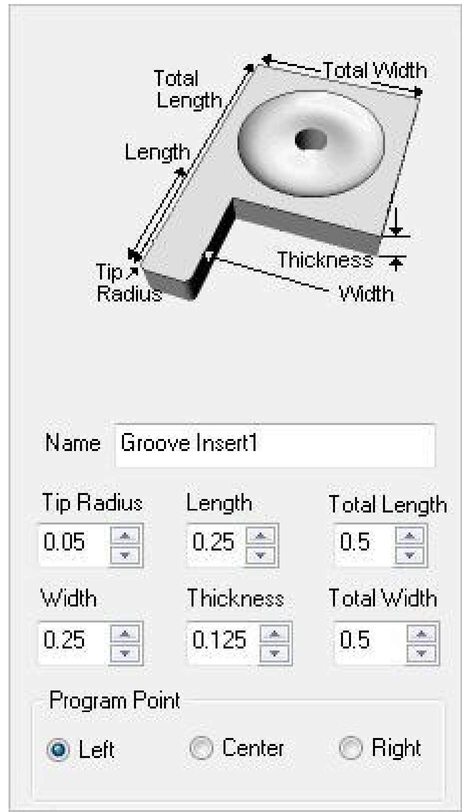
Circular Insert



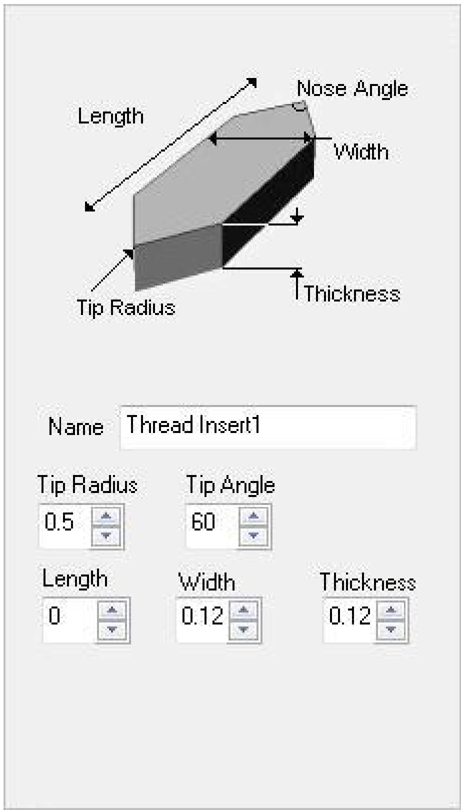
Trigon Insert



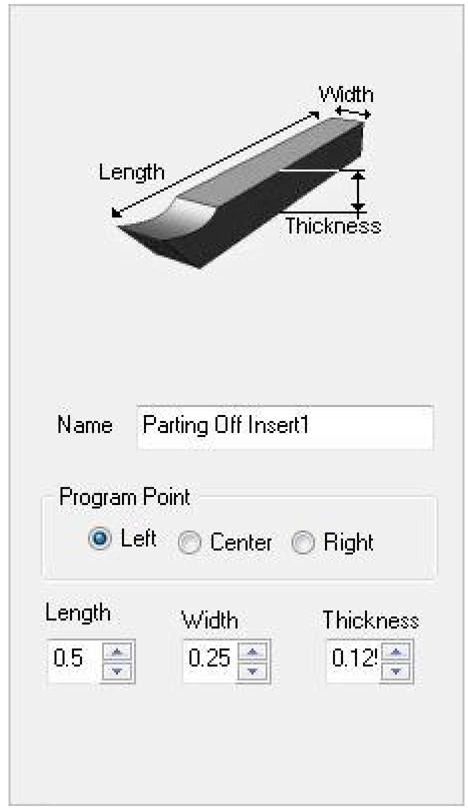
Parallelogram Insert



Groove Insert

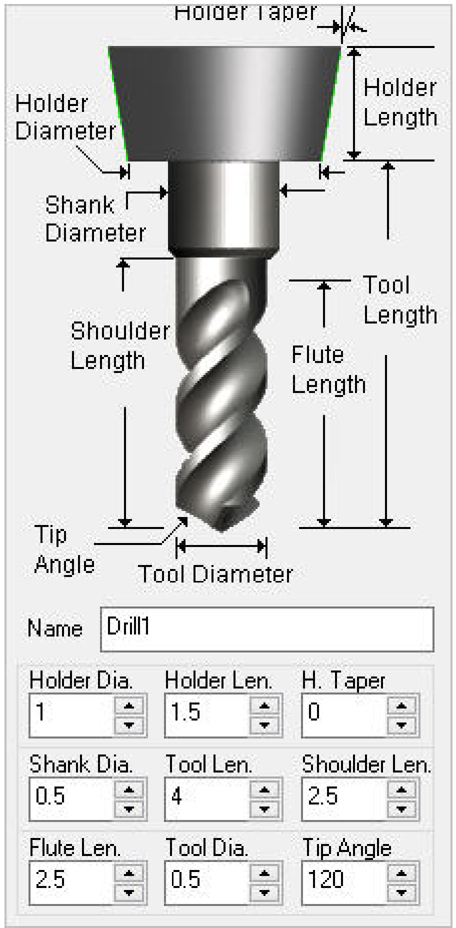


Thread Insert

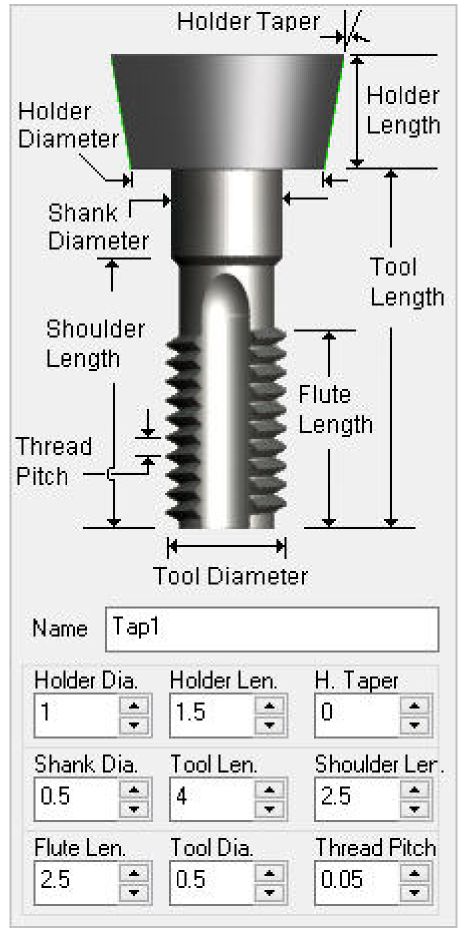


Parting off Insert

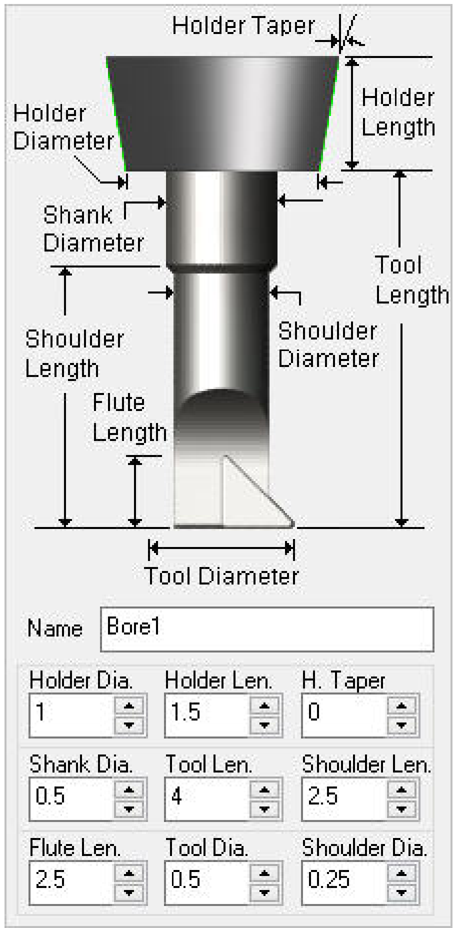
Drilling Tools available in TURN Module



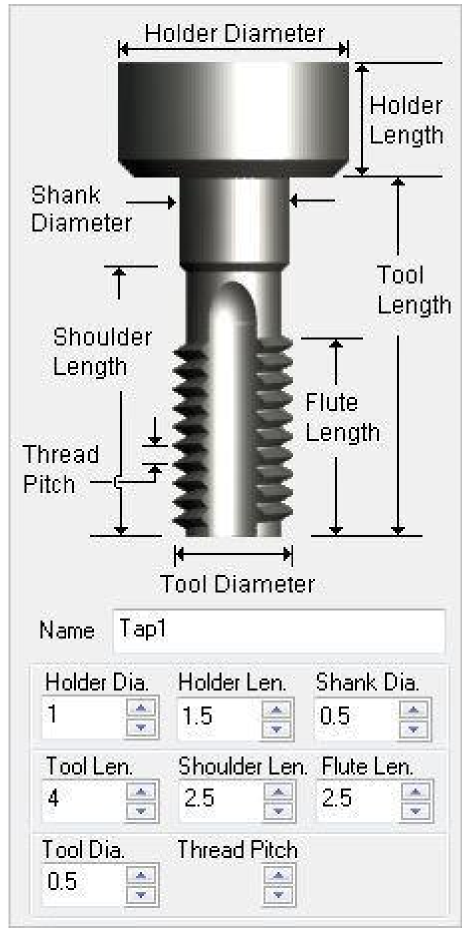
Standard Drill



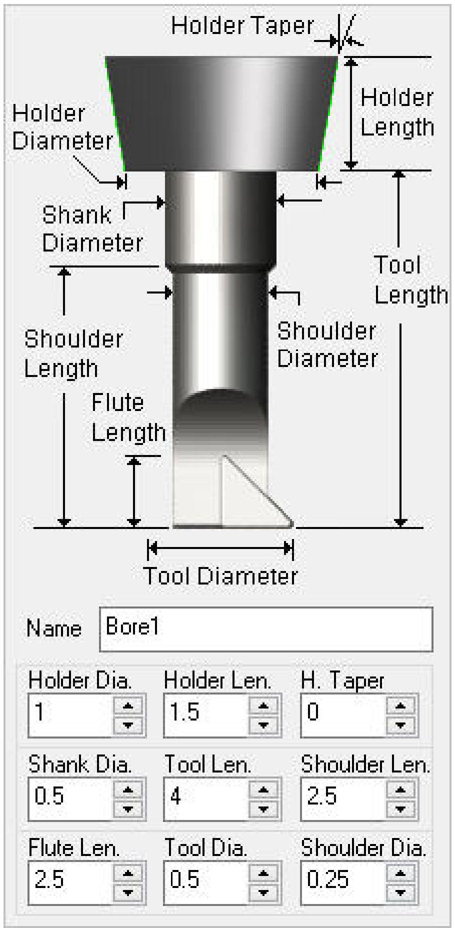
Center Drill



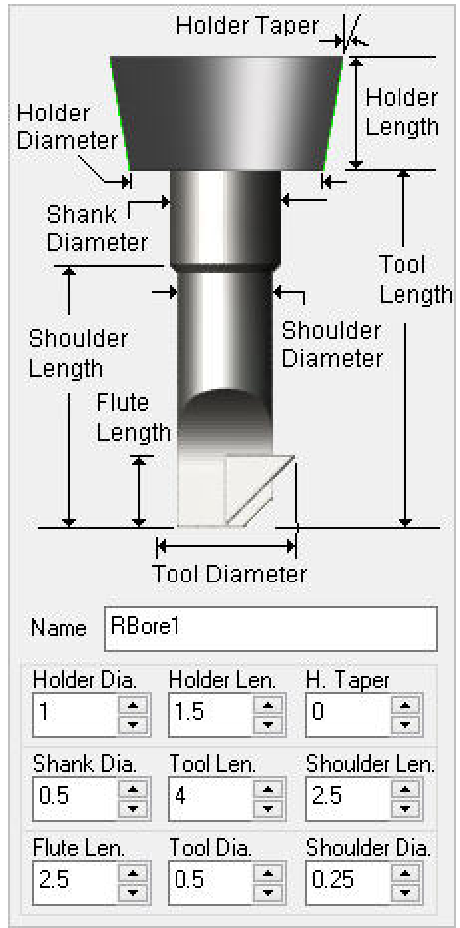
Reamer Drill



Tap Drill



Bore Drill



Reverse Bore Drill

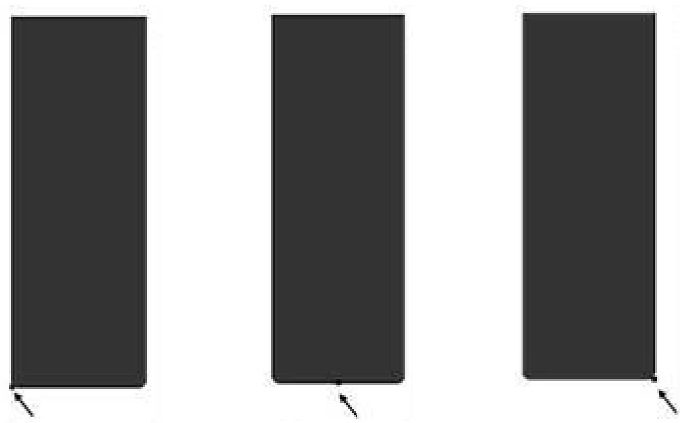
Orientation

Orientation is set for Diamond, Triangular, Trigon, Parallelogram inserts to set the orientation of the tool inserts for OD, ID programming.

- OD Forward is commonly used for both OD and Front Facing operations.
- OD Backward is used for a Back Facing insert. This is used for programming a back relief on a part.
- ID Forward is used for programming ID operations.

Program Point

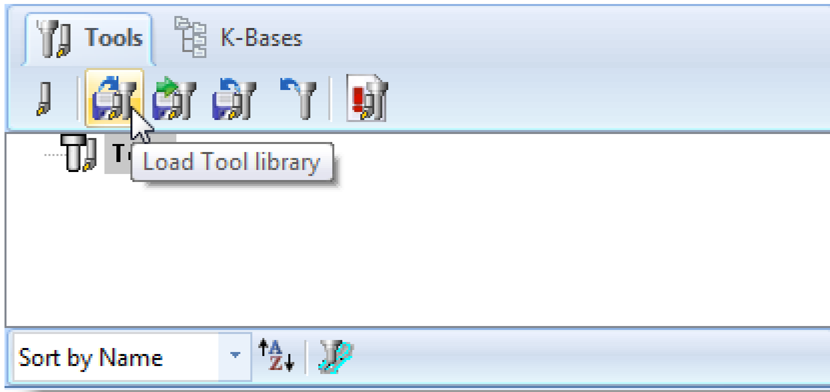
Program Point is set for Grooving and Parting off inserts. This can be set to left, center or right in the insert definition.



7.8.2 Load Tool Library

Select the Load Tool Library button from the Tools tab of the Machining Objects Browser. This enables the loading of a previously saved tool library.

Locating the Load Tool Library button



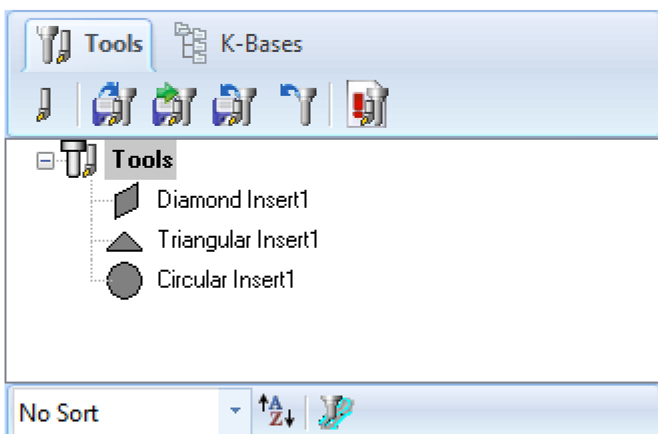
Locating the Load Tool Library button

Select the Tool Library File

Browse to the folder; double click on the desired file (*.vkb or *.csv) to load it into TURN Module.

To load a Tool Library from TURN 1.0 module

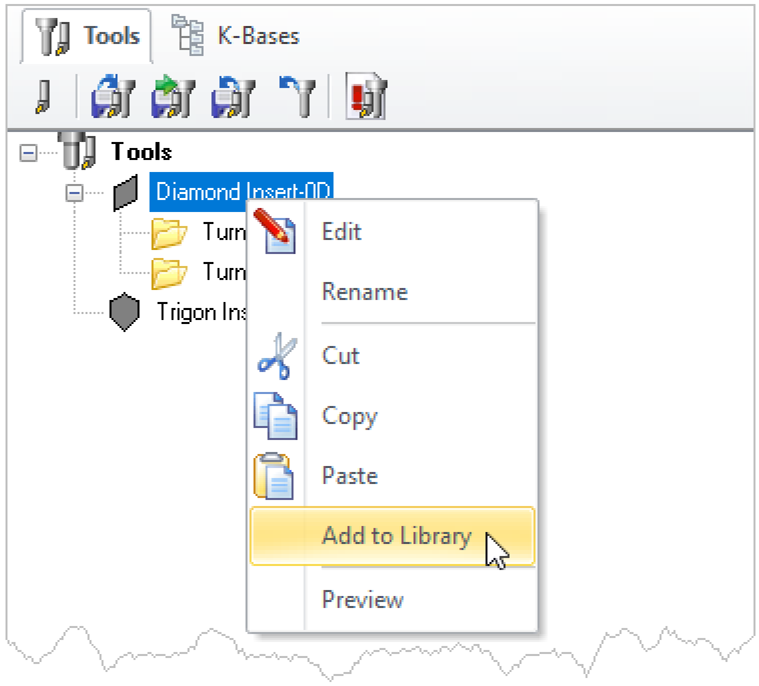
To load a Tool Library from TURN 1.0 module, select Version 1.0 Tool Library Files (**.vtl*) under file type. The loaded tool list will be seen under the tool button in the browser. To perform the Edit, Rename, Cut, Copy or Paste operations on any of these tools, hit the right mouse button while highlighting the desired tool.



The loaded tool list will be seen under the tool button in the browser

7.8.3 Add Tool to Library

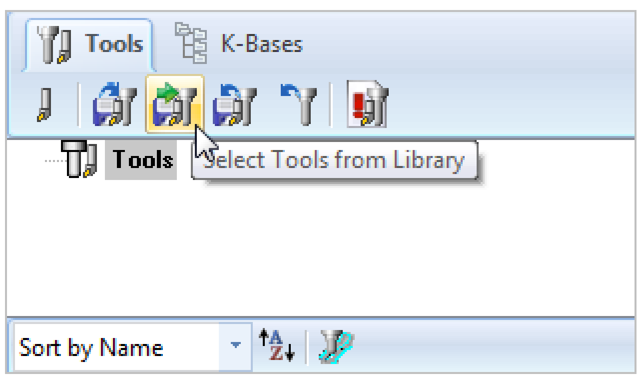
You can right-click on a Tool listed in the Mobs Browser to Add the Tool to an exiting Tool Library *.csv data file.



Add Tool to Library

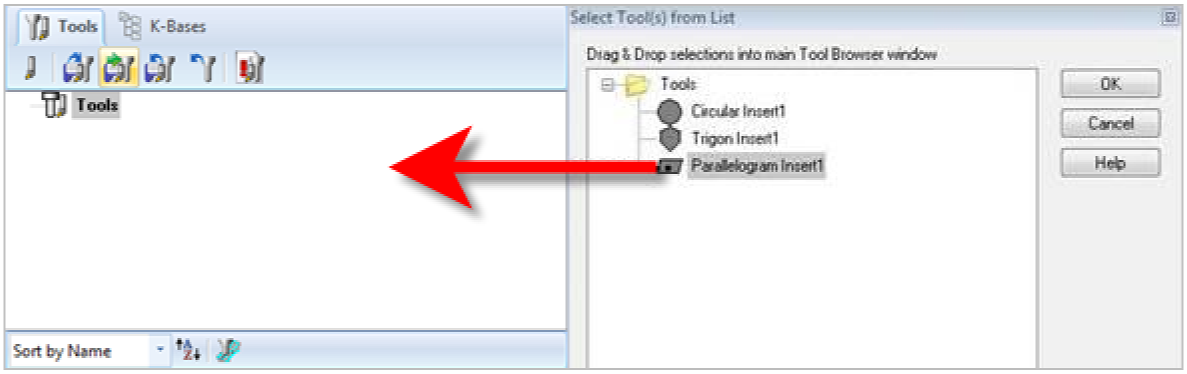
7.8.4 Select Tools from Library

The Select Tools from Library button enables users to select tools from a previously saved Tool Library.

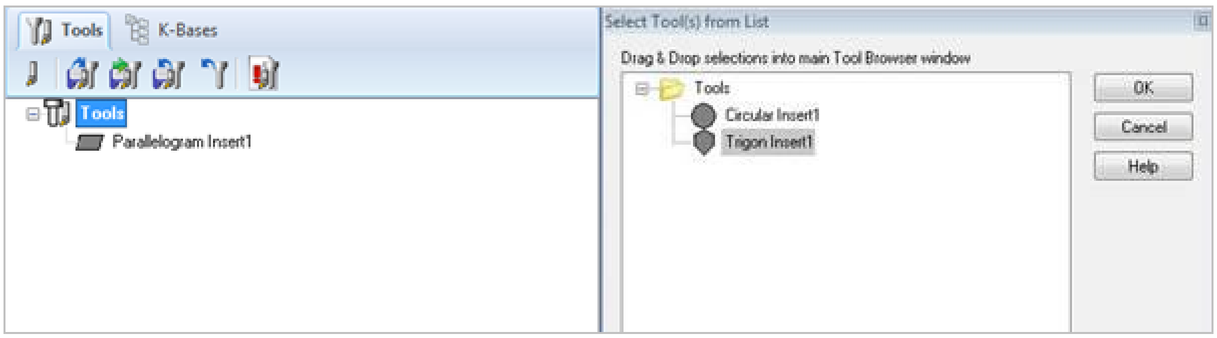


Locating the Load Tool Library button

The list of tools will now be displayed under Select Tools(s) from list dialog and you can drag and drop the tools from the selection list to the cutting tools browser.



Dialog Box: Select Tools(s) from list (Drag & Drop)



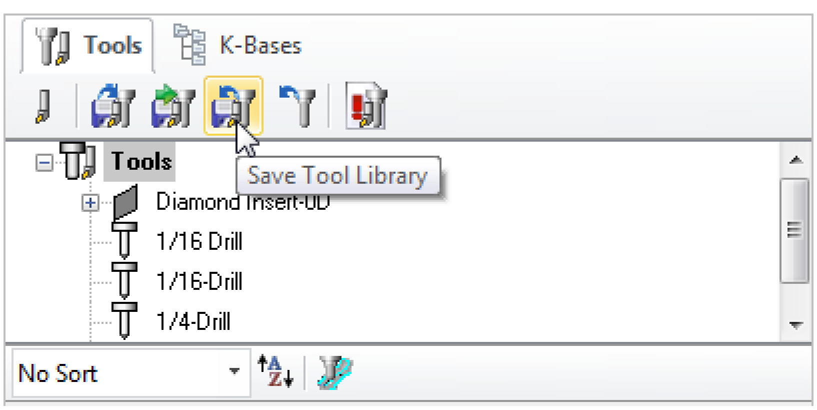
Dialog Box: Select Tools(s) from list (Drag & Drop)

To Edit, Rename, Cut, Copy or Paste on any of these tools, use right mouse button click after selecting the tool under Tools tab.

7.8.5 Save Tool Library

This button enables the created tools to be saved in a Tool Library file. The file can be saved in the desired directory and read in when required.

Locating the Save Tool Library button



Locating the Save Tools Library button

Dialog Box: Save As

Specify a file name and click Save.

File Types Supported

TURN Module supports 2 types of tool library file format *.vkb and *.csv. Both formats save and loads tools with the feeds and speeds assigned for each tool.

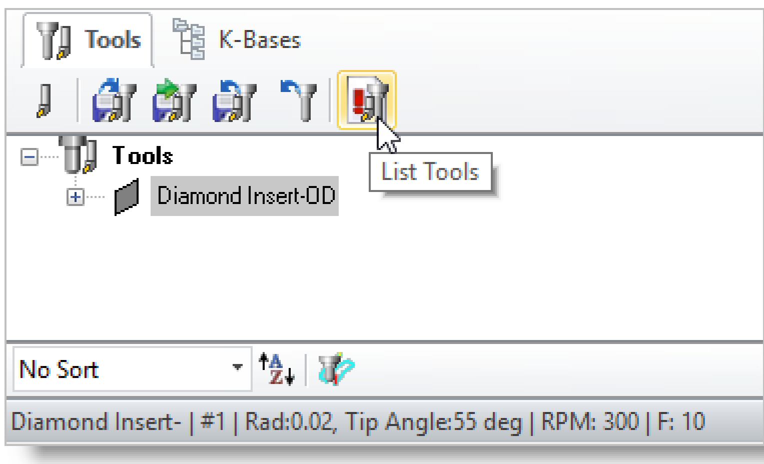
7.8.6 List Tools

This allows you to List and Print your tools.

List Tools (in the TURN Module)

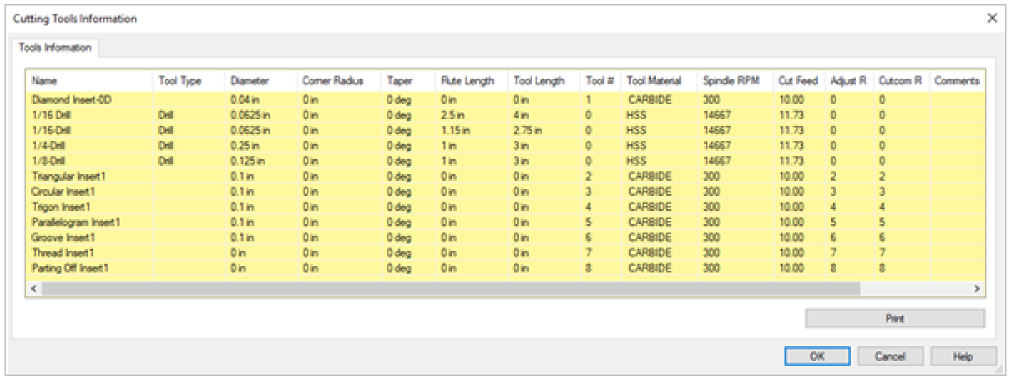
1. From the Tools tab of the Machining Objects Browser, select the List Tools button .

   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



Locating the List Tools button

2. The button brings up all the tool properties associated with the tools currently recorded in the current session. From the Cutting Tools Information dialog box, you can view or Print your Tool List.



Cutting Tools Information

Tools Information

Name	Tool Type	Diameter	Corner Radius	Taper	Flute Length	Tool Length	Tool #	Tool Material	Spindle RPM	Cut Feed	Adjust R	Cutcom R	Comments
Diamond Insert-OD		0.04 in	0 in	0 deg	0 in	0 in	1	CARBIDE	300	10.00	0	0	
1/16 Drill	Drill	0.0625 in	0 in	0 deg	2.5 in	4 in	0	HSS	14667	11.73	0	0	
1/16-Drill	Drill	0.0625 in	0 in	0 deg	1.15 in	2.75 in	0	HSS	14667	11.73	0	0	
1/4-Drill	Drill	0.25 in	0 in	0 deg	1 in	3 in	0	HSS	14667	11.73	0	0	
1/8-Drill	Drill	0.125 in	0 in	0 deg	1 in	3 in	0	HSS	14667	11.73	0	0	
Triangular Insert1		0.1 in	0 in	0 deg	0 in	0 in	2	CARBIDE	300	10.00	2	2	
Circular Insert1		0.1 in	0 in	0 deg	0 in	0 in	3	CARBIDE	300	10.00	3	3	
Trigon Insert1		0.1 in	0 in	0 deg	0 in	0 in	4	CARBIDE	300	10.00	4	4	
Parallelogram Insert1		0.1 in	0 in	0 deg	0 in	0 in	5	CARBIDE	300	10.00	5	5	
Groove Insert1		0.1 in	0 in	0 deg	0 in	0 in	6	CARBIDE	300	10.00	6	6	
Thread Insert1		0 in	0 in	0 deg	0 in	0 in	7	CARBIDE	300	10.00	7	7	
Parting Off Insert1		0 in	0 in	0 deg	0 in	0 in	8	CARBIDE	300	10.00	8	8	

Print

OK Cancel Help

From the Cutting Tools Information dialog box, you can view or Print your Tool List

3. Pick OK to close the dialog.

7.8.7 Cutter Compensation

Cutter compensation is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in TURN Module. For example, if the cutter used in programming in TURN Module has a tool nose radius of 0.02 inches and due to tool wear the actual tool nose radius is only 0.01 inches in size, you can compensate for this in the controller rather than having to re-program the operation in TURN Module again.

Enable Cutter Compensation

To enable Cutter Compensation in your toolpaths:

1. On the Global Parameters tab of the Turn operation, set Compensation to Auto/ON.
2. Set Adjust Register and Cutcom Register from the Parameters tab of the Create/Edit Tools dialog box.
3. Enable Cutter Compensation in your Post Processor (see below).

In the Post Processor

Specify the Cutter Compensation value and the Compensation Register in the controller (the controller needs to be capable of doing this).

1. From the Program tab of the Machining Browser, select 

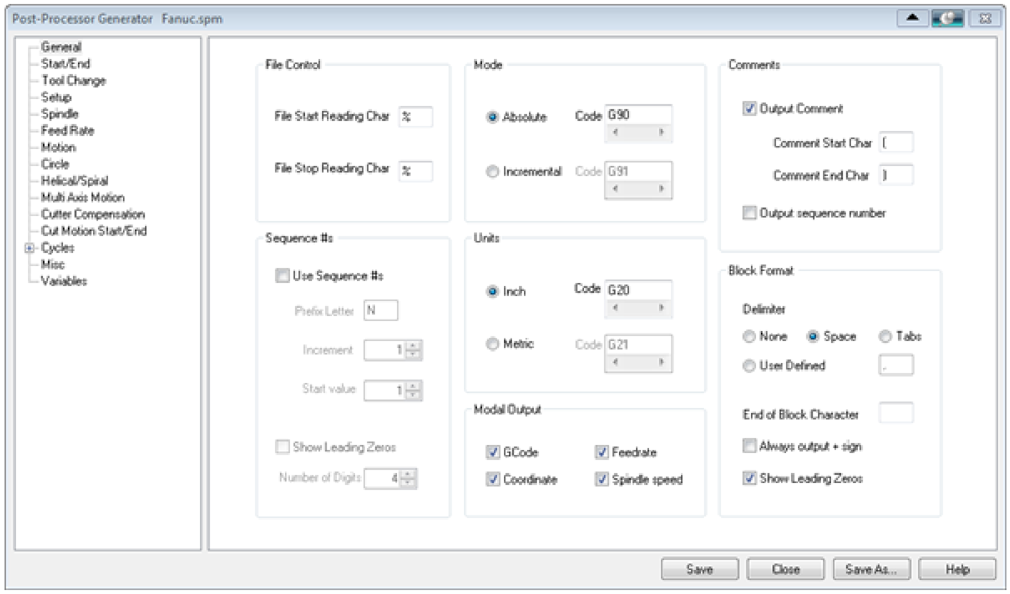
.
2. From the Set Post-Processor dialog box, select the Edit ... button located to the right of the Current Post Processor.
3. Enable Cutter Compensation from the Tool Change tab of the Post processor Generator dialog box.

- or -

1. From the Machining Browser, select  and then select Post processor Generator ...

4. Select the Post Processor and then pick Edit.

5. Enable Cutter Compensation from the Tool Change tab of the Post processor Generator dialog box.



Dialog Box: Post-Process Generator, Tool Change tab

A few things to watch out for:

Cutter Compensation makes sense only in finishing operations. If you are using multiple passes in finishing, the compensation will be turned on only in the final pass.

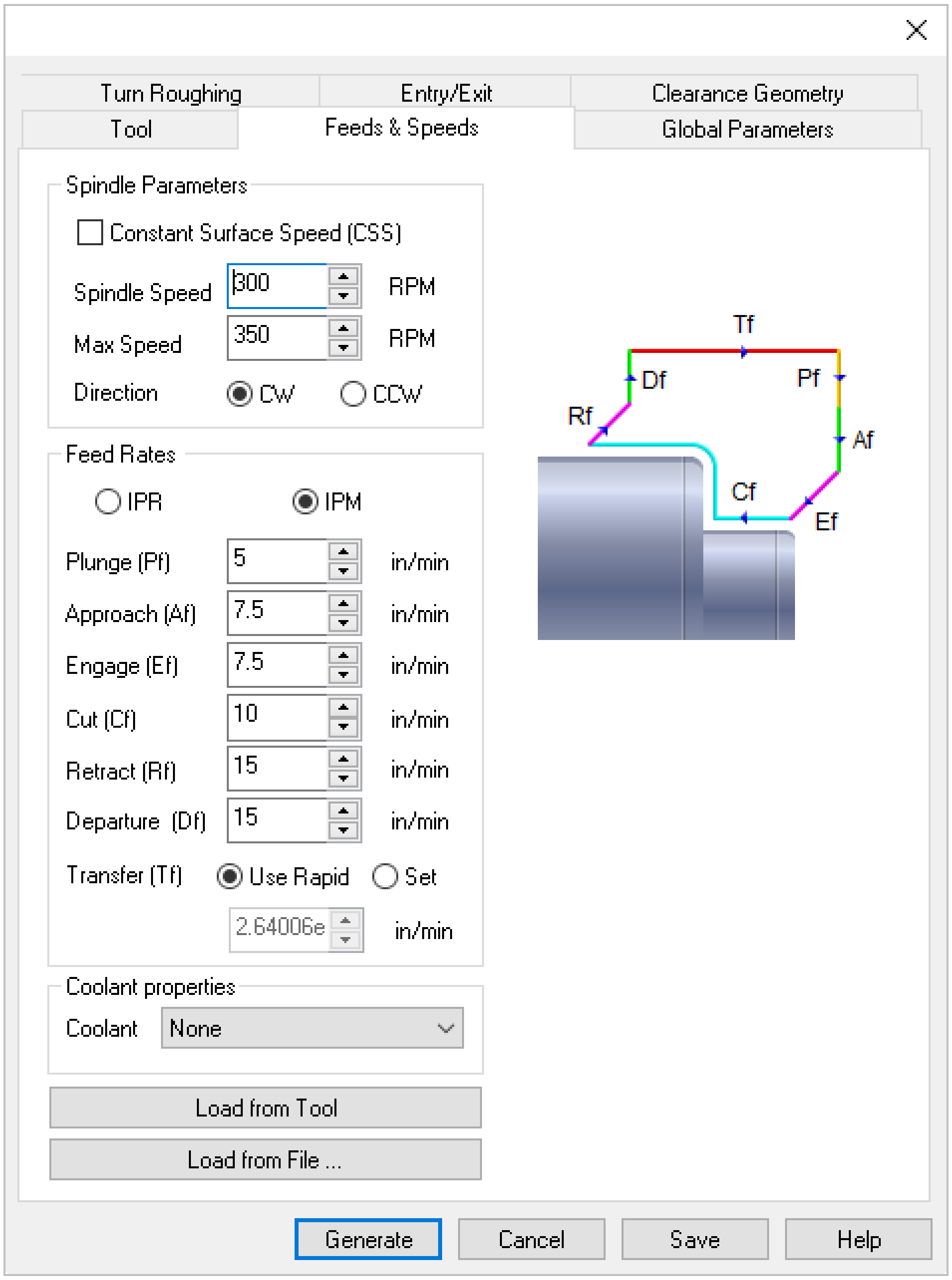
Make sure you are not using Zig-Zag cut traversal in the finishing methods that you want to turn compensation on.

Make sure you have a linear motion for the controller to turn on the compensation value on. Thus, in turn finishing, make sure there is a linear entry motion for the controller to be able to turn Compensation on.

7.9 Feeds & Speeds

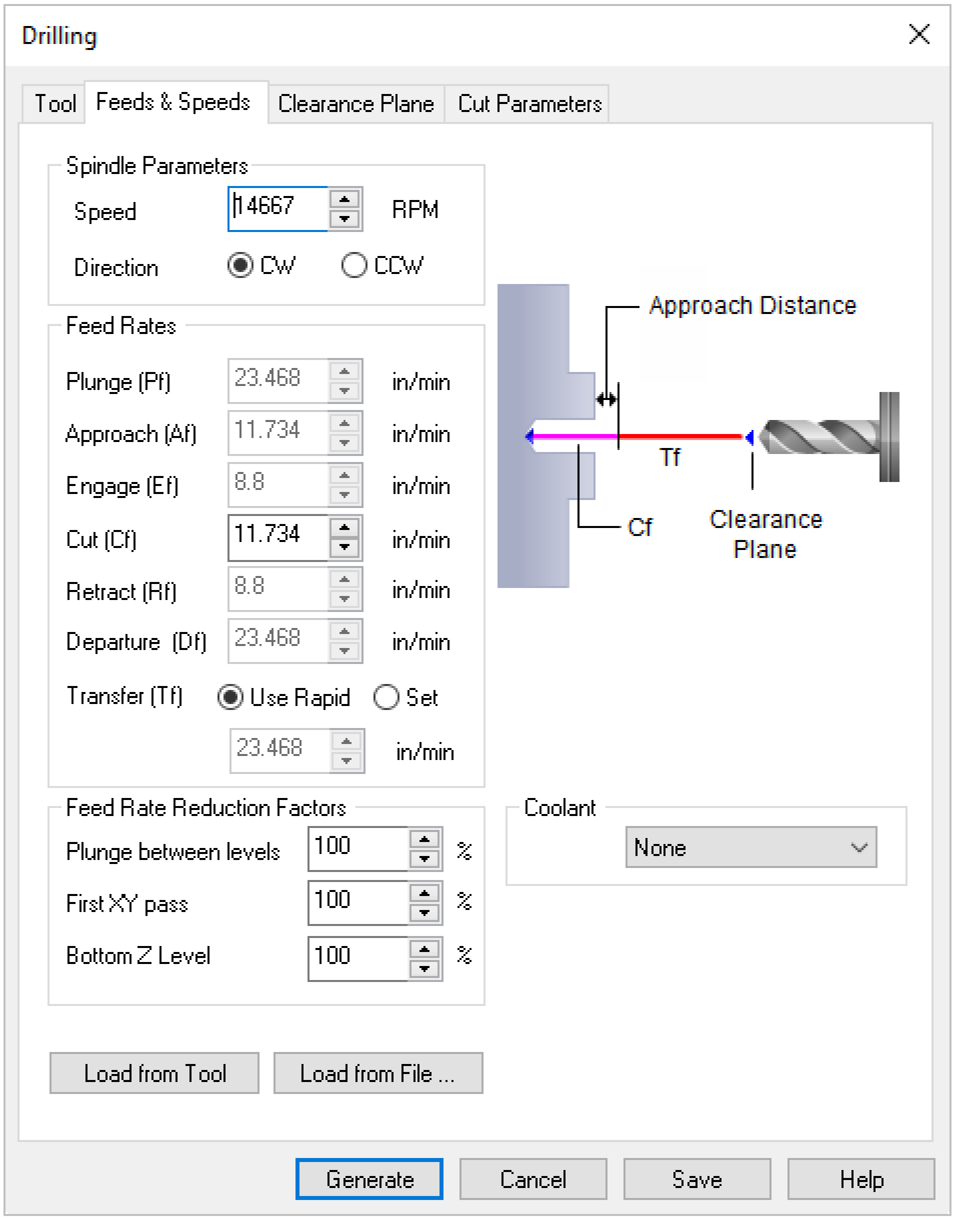
TURN Module allows setting of feeds and speeds to be used in the toolpaths via the Feeds/Speeds dialog. This dialog gives you the ability to set the following parameters:

Dialog Box: Feeds & Speeds for Turning operations



Dialog Box: Feeds & Speeds for Turning operations

Dialog Box: Feeds & Speeds for Hole Machining operations



Dialog Box: Feeds & Speeds for Hole Machining operations

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).

If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed
his is the rotational speed of the spindle expressed in RPM.

Surface Speed
Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed
The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction
This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

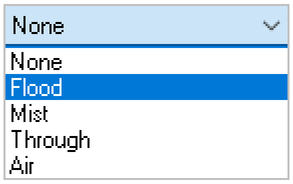
Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Feed Rates Reduction Factors (Hole Operations Only)

This sets Feed Rate Reduction Factors for Plunge Between Levels and the First XY pass.

Load from Tool

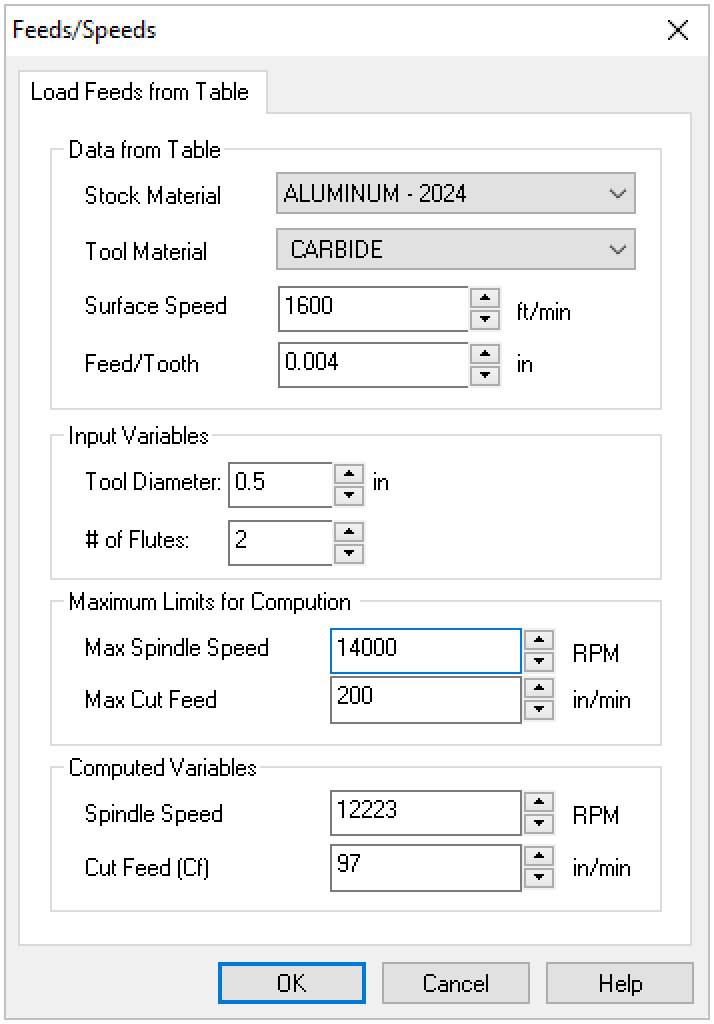
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

Customizing Feeds & Speeds

TURN Module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in RhinoCAM. (C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Materials).

The Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials.

These values can then be recalled at any time to compute the feeds/speeds to be used in the current program. The format for this file is shown below.

The format for this file is shown below:

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
    <Name>Stock Material</Name>
    <TextureFile>Texture Bitmap</TextureFile>
    <FeedsSpeedsRecord>Operation type, Tool Material, Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
  </Material>
</FeedsSpeeds>
```

An example entry is shown below:

```
<Material>
    <Name>ALUMINUM - 2024</Name>
    <TextureFile>ALUMINUM.bmp</TextureFile>
    <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00, 0.0200</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00, 0.0200</FeedsSpeedsRecord>
    <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

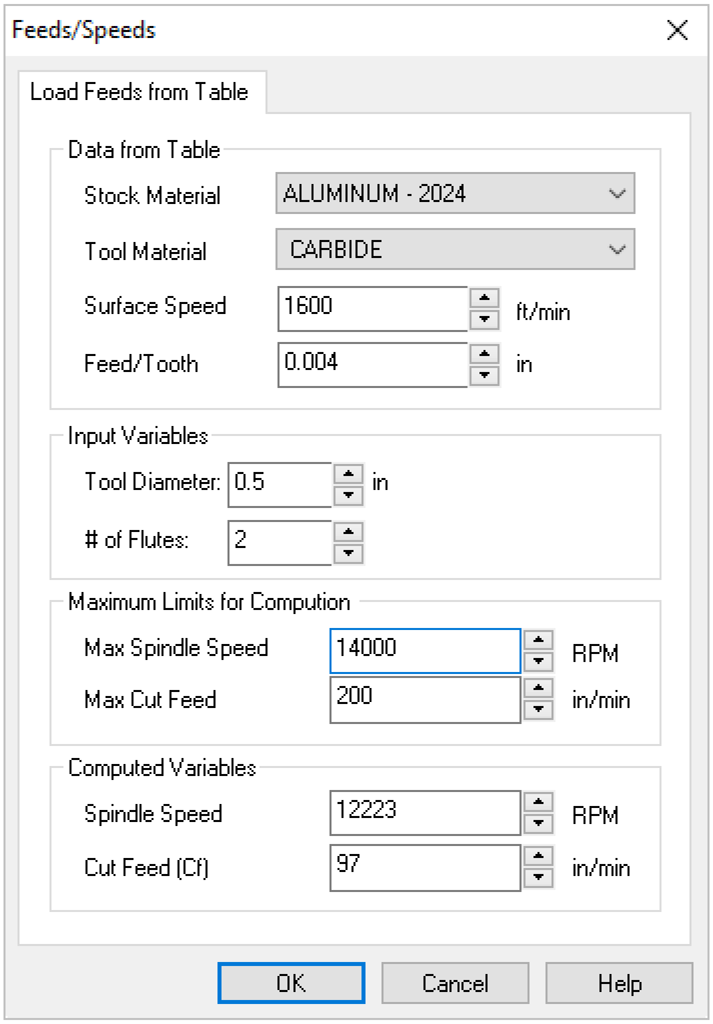
If part unit is set to Inches, TURN Module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

7.9.1 Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

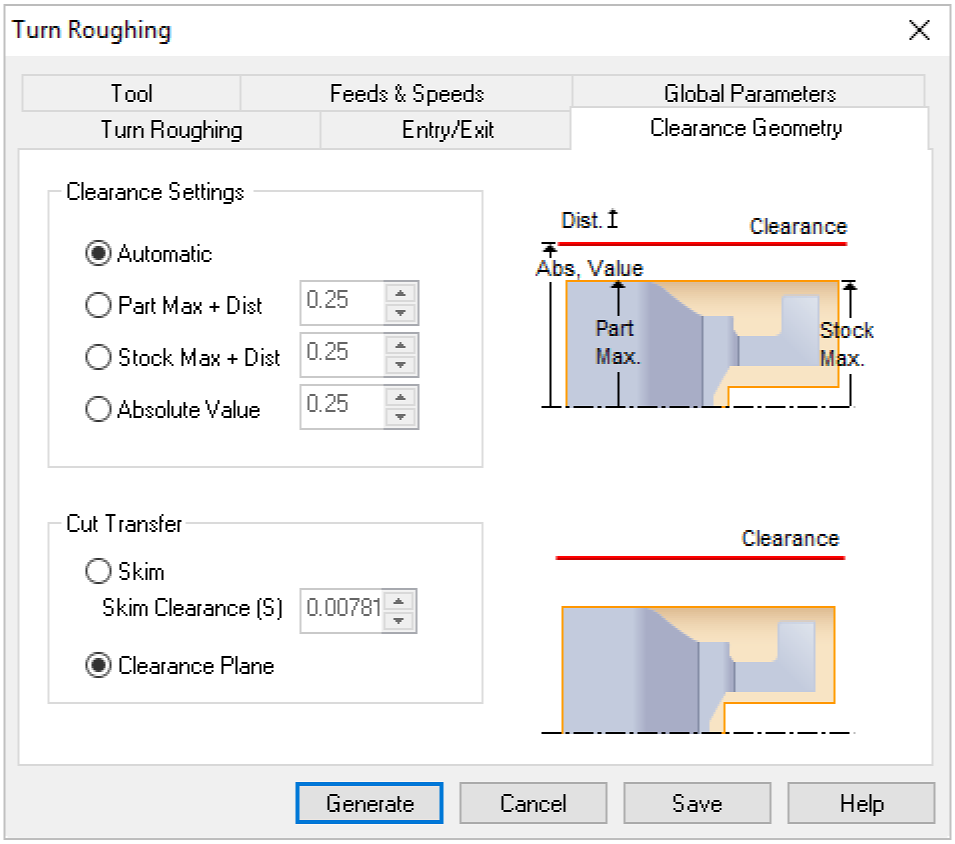
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

7.10 Clearance Plane

The clearance plane is a plane from which the approach motions start and retract motions end. After retracting, the tool moves rapidly along this plane to the position of the next engage. This plane is typically a certain safe distance above the part geometry. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

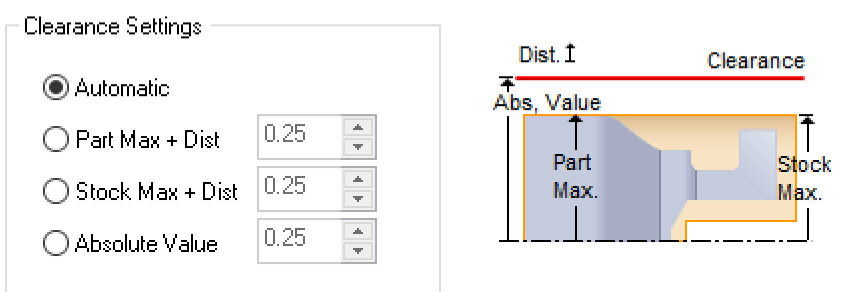
By default (Automatic option), the clearance level is calculated by adding a safety distance to the extreme point (depending whether Outer Diameter, Inner Diameter or Face is machined) found on both part and stock geometry. You can set the clearance level to be a set distance from either the part or stock, or enter the absolute Z level.

Dialog Box: Clearance Geometry for Turning operations



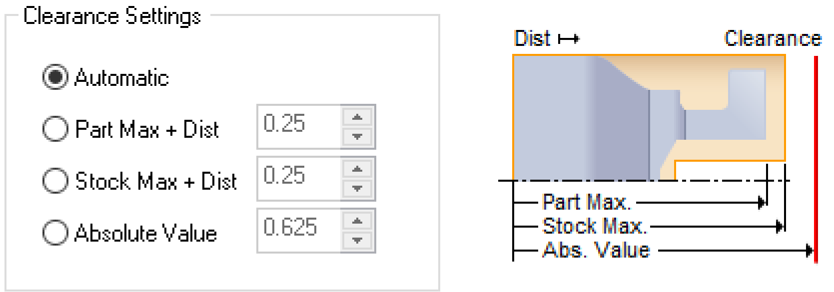
Dialog Box: Clearance Geometry for Turning operations

Dialog Box: Turn OD Clearance



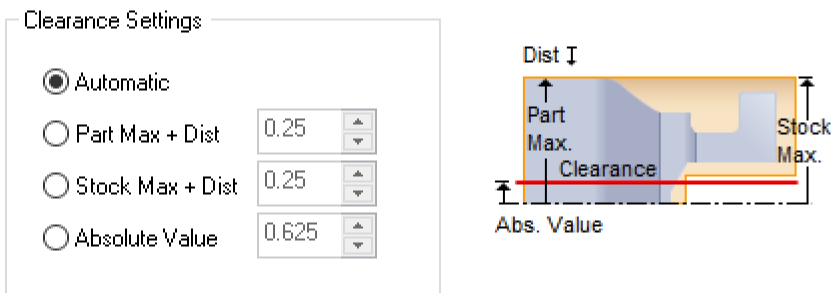
Turn OD Clearance

Dialog Box: Turn Face Clearance



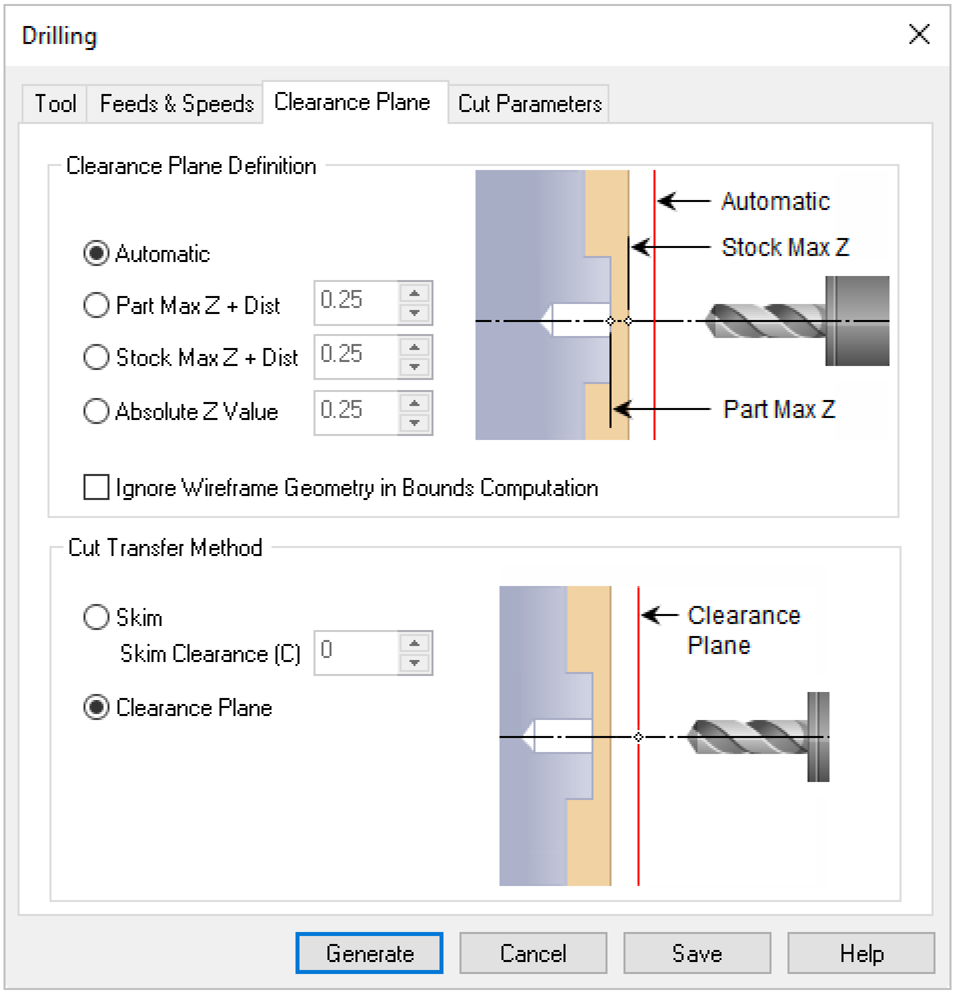
Turn Face Clearance

Dialog Box: Turn ID Clearance



Turn ID Clearance

Dialog Box: for Hole Machining operations



Dialog Box: Clearance Plane tab, Turn Reverse Boring

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

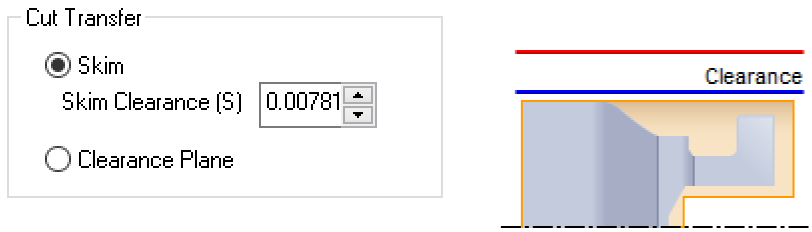
For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

Dialog Box: Cut Transfer Skim



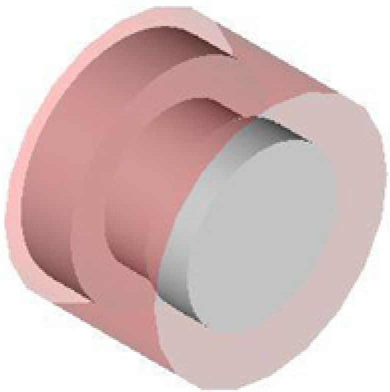
Cut Transfer Skim

Dialog Box: Cut Transfer Clearance Plane

Cut Transfer Clearance Plane

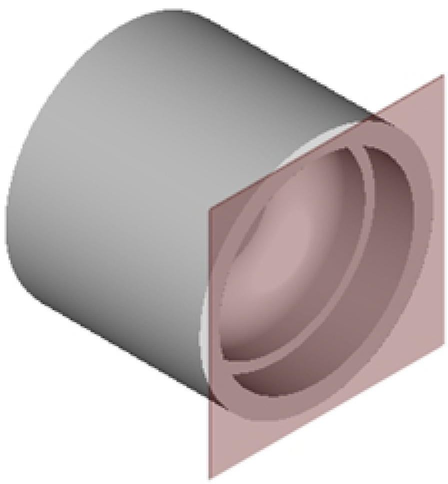
Clearance Plane for Turn OD operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Clearance Plane for Turn OD operations

Clearance Plane for Hole Machining operations



Clearance Plane for Hole Machining operations

7.11 Knowledge Base

The MILL and TURN modules includes powerful Knowledge Base functionality that makes "push button" programming a reality. You can archive an entire machining strategy specific to a certain class of parts in a Knowledge Database (also referred to as a K-Base) and then optionally assign Geometry Selection Rules that are applied automatically when toolpath operations are selected for use from the Knowledge Base.

In a *family of parts* situation, where the same set of machining operations and tools can be applied to machine these parts, it would be most appropriate to archive this processes in a K-base file and then apply it across all of the parts in this family. Another situation where this feature can be used is in shop floor programming. Experienced programmers can determine the sequence of operations to be used to machine a certain class of parts and create a K-base file capturing that knowledge for automation purposes.

More about Knowledge Bases

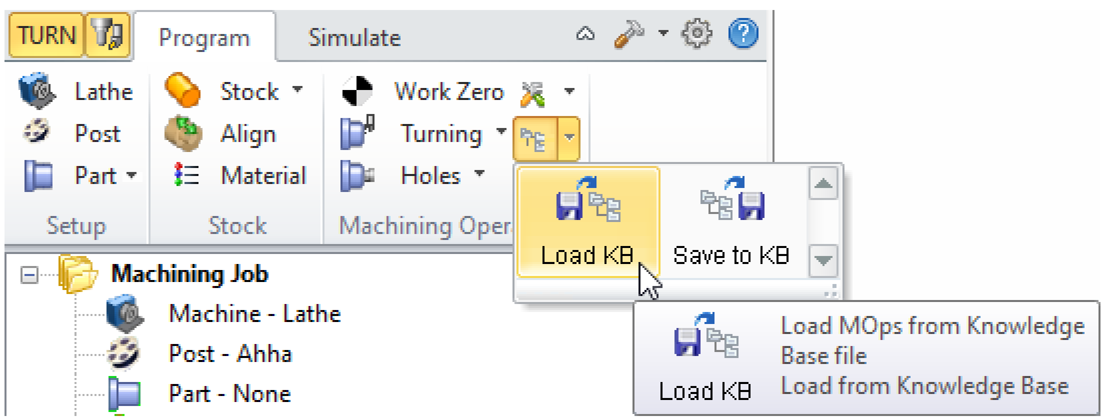
Once these K-base files are thoroughly debugged, operators at the shop floor can then load and generate toolpaths automatically. Doing this not only increases the throughput but also the productivity of the entire manufacturing team, resulting in dramatic cost savings for the enterprise.

The machining strategy can include the sequence of machining processes used, the specific parameters used in each machining processes as well as the Geometry Selection Rules.

7.11.1 Create Knowledge Base

To create a Knowledge Base, start creating machining operations. Once created these machining operations can be re-sequenced if necessary. When completely satisfied with the machining operations used and their sequence, click Knowledge Base and Save to KB from the Program tab or pick the Save to Knowledge Base option by right clicking on Machining Operations and selecting the Save to Knowledge Base option. If no Geometry Selection Rules have been defined you are asked if you wish to define them.

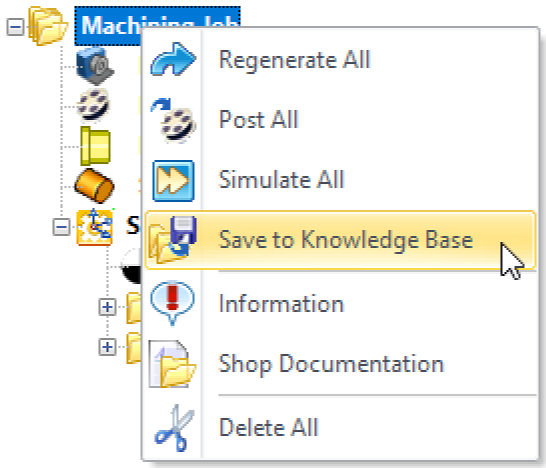
From the Program tab of the Machining Browser



From the Program tab of the Machining Browser

by right click on Machining Operations

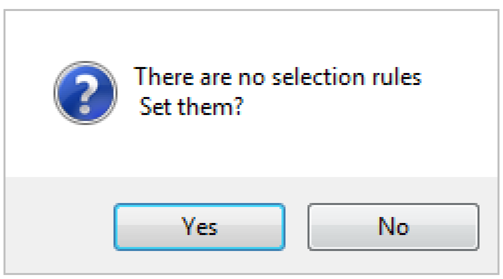
Select the Machining Job or one or more operations, then right-click and select Save to Knowledge Base.



by right click on Machining Operations

Geometry Selection Rules (MILL ONLY)

If there are no Geometry Selection Rules set for the Knowledge Base the following message is displayed. If you wish to set Global selection rules for the Knowledge Base, pick Yes to display the Geometry Selection Rules dialog. You can assign Geometry Selection Rules for each operation in the Knowledge Base after it is loaded into another part. See Knowledge Base Rules for information about setting Selection Rules for a Knowledge Base.



Geometry Selection Rules

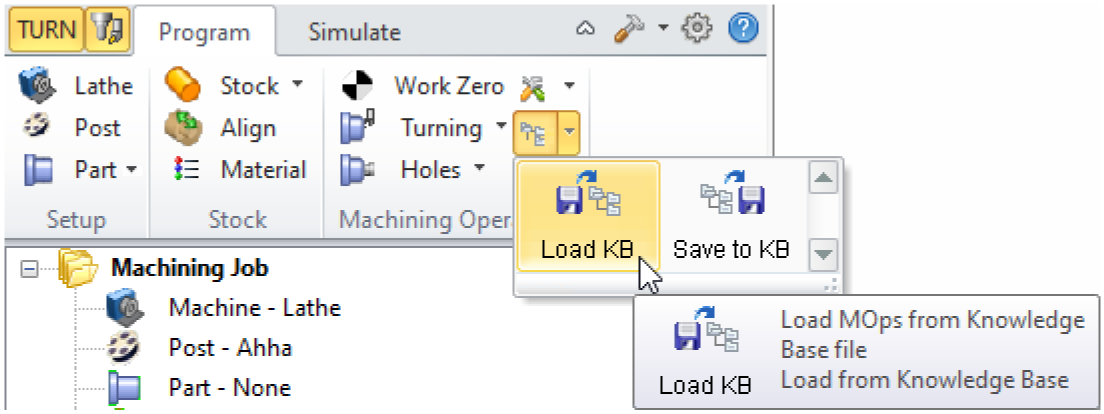
7.11.2 Load Knowledge Base

A Knowledge Base file must be created before you can Load it. See Create a Knowledge Base for information about creating one. Once created, you can Load the Knowledge Base and re-use it in any other part file that may be appropriate. You can Load one or more Knowledge Base files into the currently active part. When a Knowledge Base is loaded, its operations are automatically appended to the existing list of machining operations shown in the K-Bases tab of the Machining Objects Browser.

A Knowledge Base DOES NOT have associated toolpaths. It contains the Knowledge parameters and the Geometry Selection Rules for machining. Once loaded, you then Drag & Drop operations from the Knowledge Base from the K-Bases tab into your Setup in the Machining Browser. If Geometry Selection Rules have been set in the Knowledge Base, they are applied automatically when the toolpath operations in your Setup are Generated.

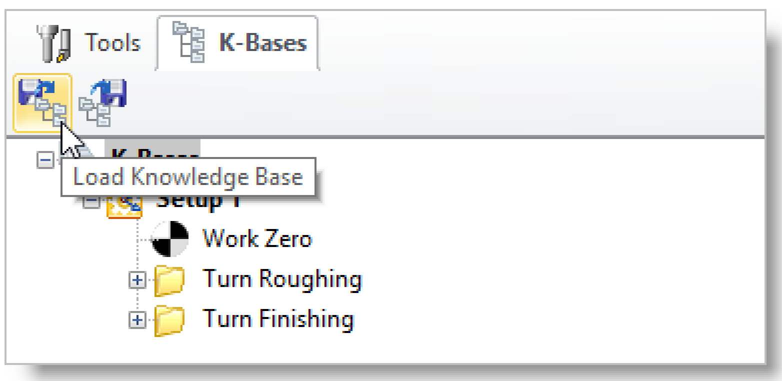
You can load a Knowledge Base either from the Machining Browser or the K-Bases tab of the Machining Objects Browser. Both methods are shown below:

Load a Knowledge Base from the Machining Browser



To Load a Knowledge Base from Machining Browser

Load a Knowledge Base from the K-Bases tab



To Load a Knowledge Base from the K-Bases tab of the Machining Objects Browser

7.11.3 Apply a Knowledge Base

After a Knowledge Base is Loaded into the K-Bases tab of the Machining Objects Browser you can apply its operation definitions and selection rules to any current Setup in the Machining Browser.

Basic Work Flow

1. Open a part file containing operations to use to create your Knowledge Base.
2. Create a Knowledge Base file.
3. Open a part that you want to apply the Knowledge Base to.

4. Load the Knowledge Base from the K-Bases tab of the Machining Objects Browser.

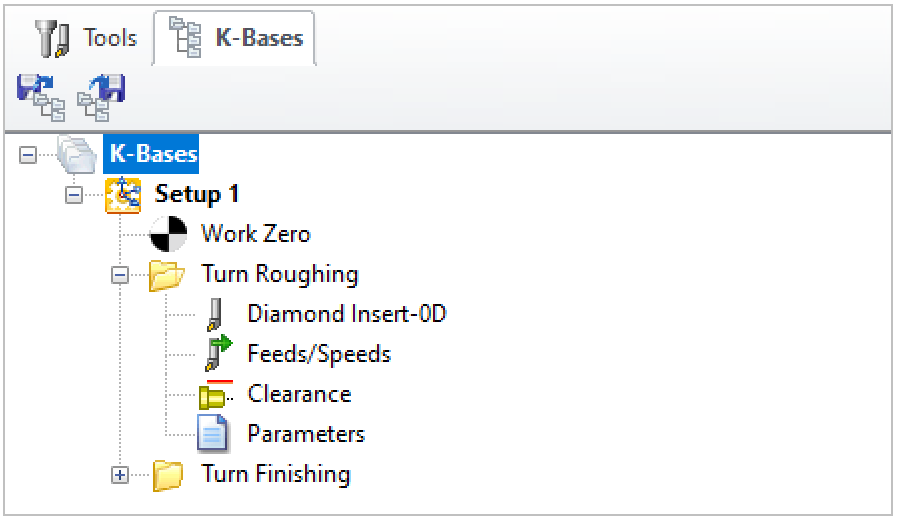
5. When asked if you want to set rules, pick No.

6. If desired, defined selection rules after the Knowledge Base is Loaded into the K-Bases tab.

7. Drag & Drop operations from the K-Bases into your Setup.

8. Open each operation in the Setup, assign Control Geometry and Regenerate.

Knowledge Base is Loaded

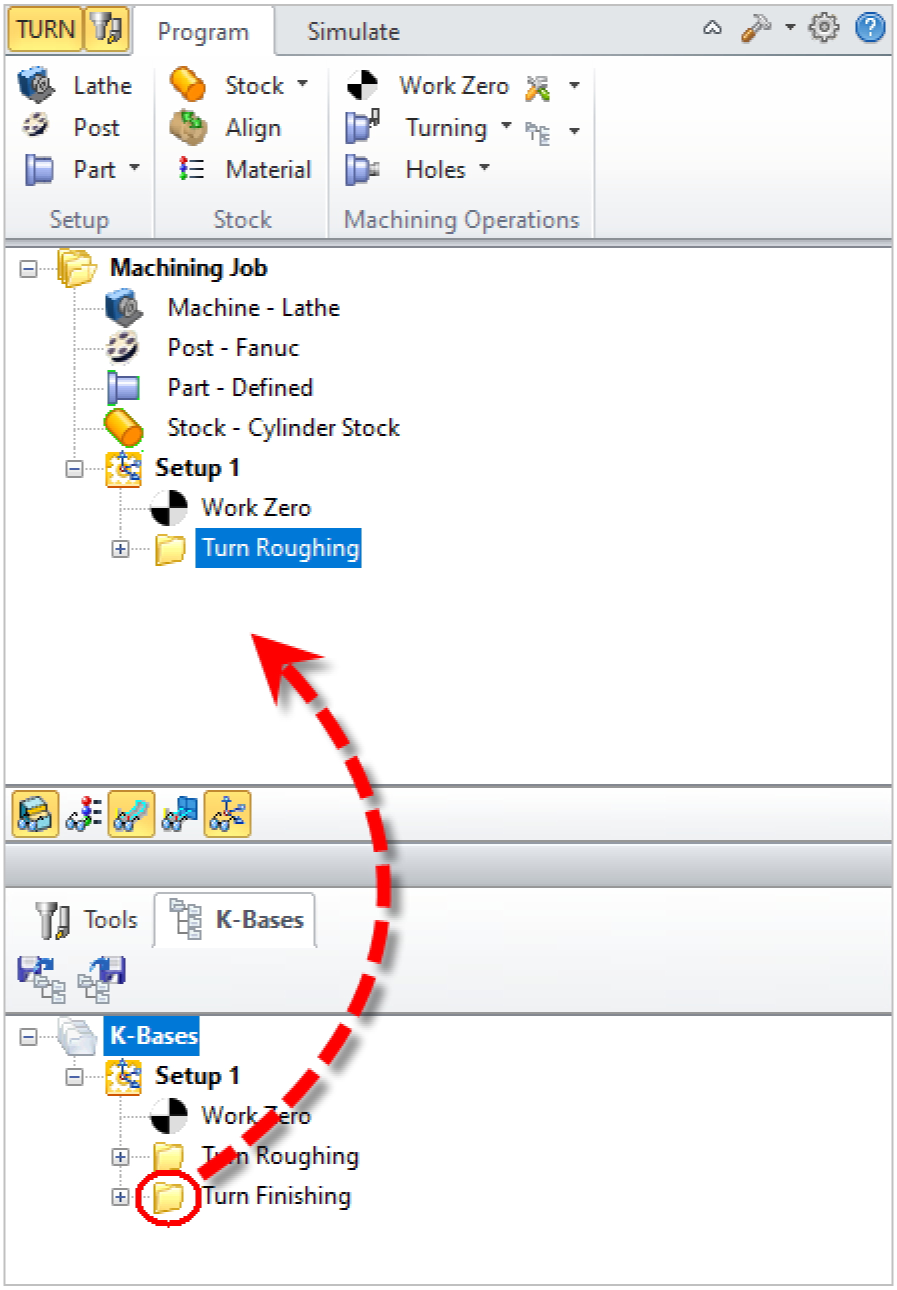


K-Bases tab of the Machining Objects Browser

Drag & Drop Operations from the Knowledge Base

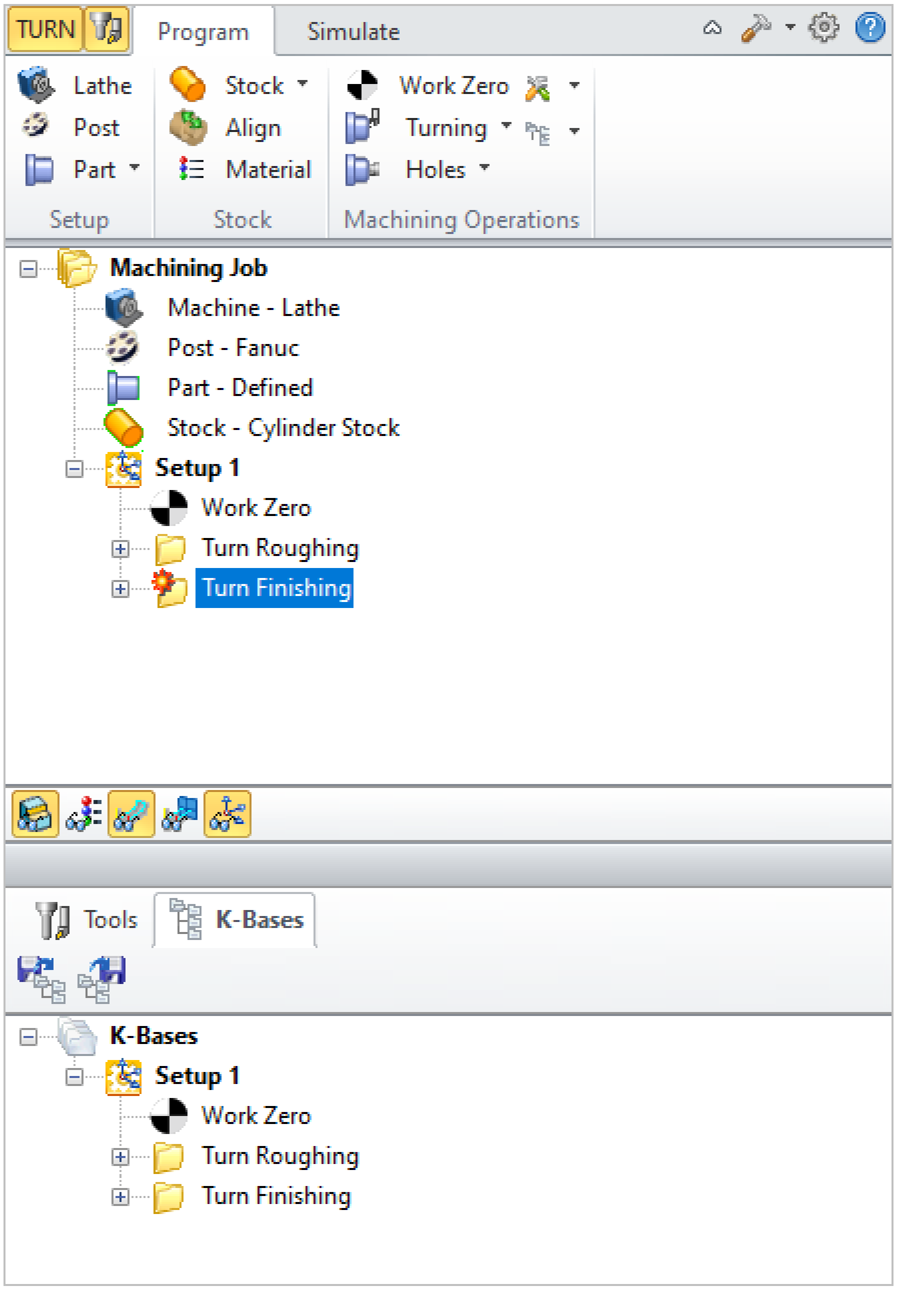
Expanding an operation under the K-Bases tab displays the Selection Rules, Tool, Feeds/Speeds, Clearance and Parameters for that operation type in the Knowledge Base.

You can Drag & Drop an operation type from the K-Bases tab up to your current Setup in the Machining Browser for programming.



K-Bases tab of the Machining Objects Browser

The operation is now available under the Machining Browser. Once you have defined the part geometry, you can edit the operation from the Machining Browser and generate the toolpath.

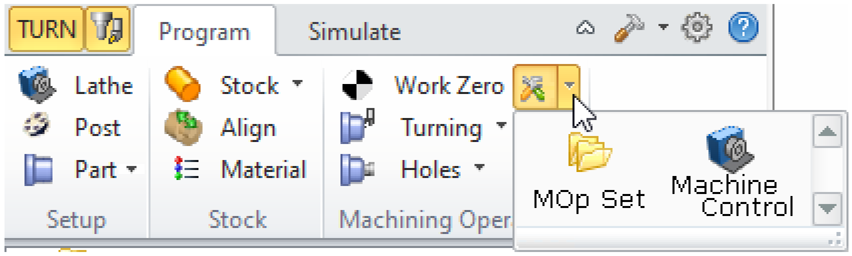


K-Bases tab of the Machining Objects Browser

Drag & Drop the same operation from Machining Objects Browser to the Machining Browser multiple times, creates copies of the same operation.

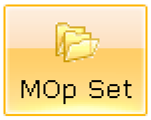
7.12 Create Miscellaneous Operations

Mop Sets and Machine Control operations are grouped under miscellaneous operations. These can be found under Program tab in Machining Browser.



Create Miscellaneous Operations Menu Item

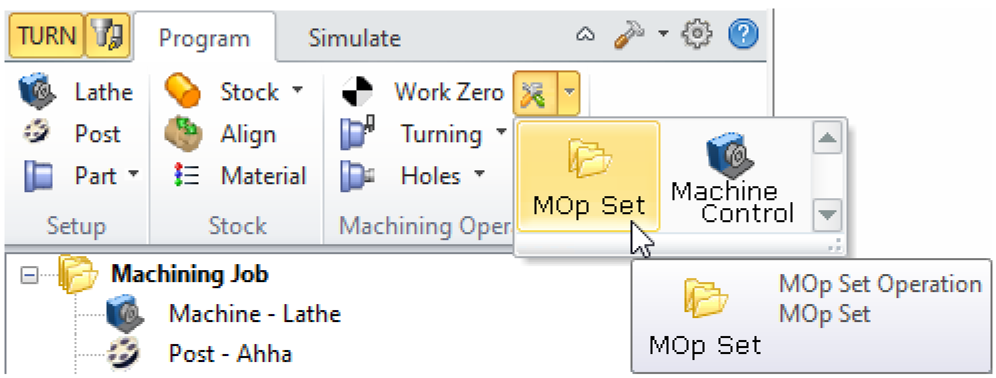
7.12.1 Mop Set



This feature allows you to group operation to a folder called Machining Operation Set. This can be useful where you would like to group operations by type or by tool. You can then post process machining operations by selecting a Mop Set.

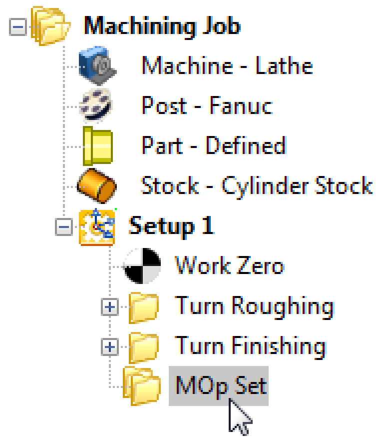
Selecting Mop Set from Create Miscellaneous Operations under Program tab creates a Mop Set folder in the Machining Browser.

Machining Operation Set (Mop Set) Menu Item



Machining Operation Set (MOp Set) Menu Item

Mop Set folder in the Machining Browser

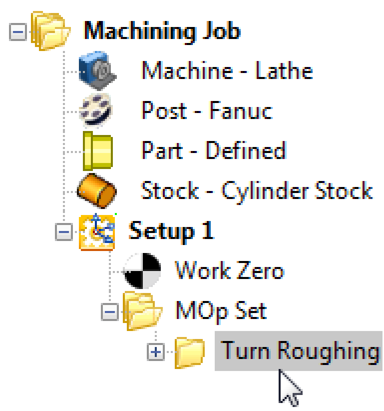


Selecting MOp Set from Create Miscellaneous Operations under Program tab, creates a MOp Set folder in the browser.

Create a new operation under a Mop Set

You can now create machining operations or move existing machining operations under Mop Set folder.

To create a new operation under a Mop Set, select the Mop Set under the Machining Browser and then choose the Turning or Hole Machining operation. The operation would appear below the Mop Set and is a level into the Job tree structure.



create a new operation under a MOp Set

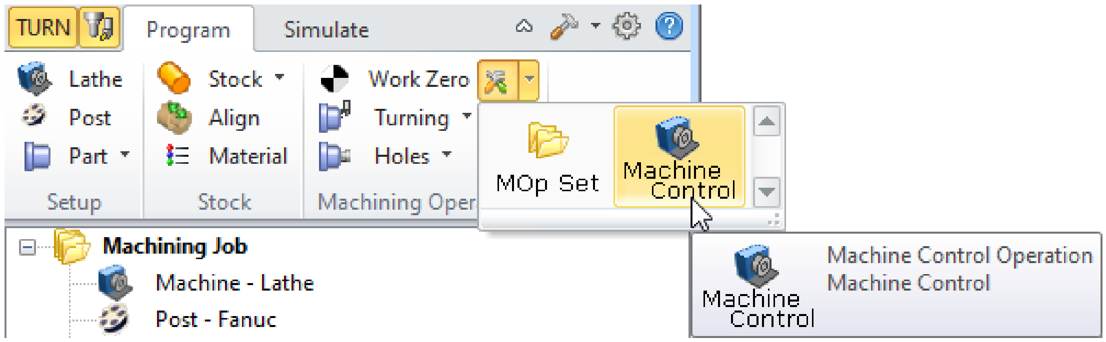
Move existing operations to a Mop Set

To move existing operations, select a machining operation, drag and drop it into the Mop Set so the machining operation appears one level into the Job tree. Multiple Mop Sets can be created and operations can be grouped under each Mop Set.

7.12.2 Machine Control

This operation allows you to output a set of code or instructions for the machine. These can be inserted at the start, end and between machining operations. The instruction set for these operations are defined in the post processor. This eliminates the need for inserting codes manually in the posted code.

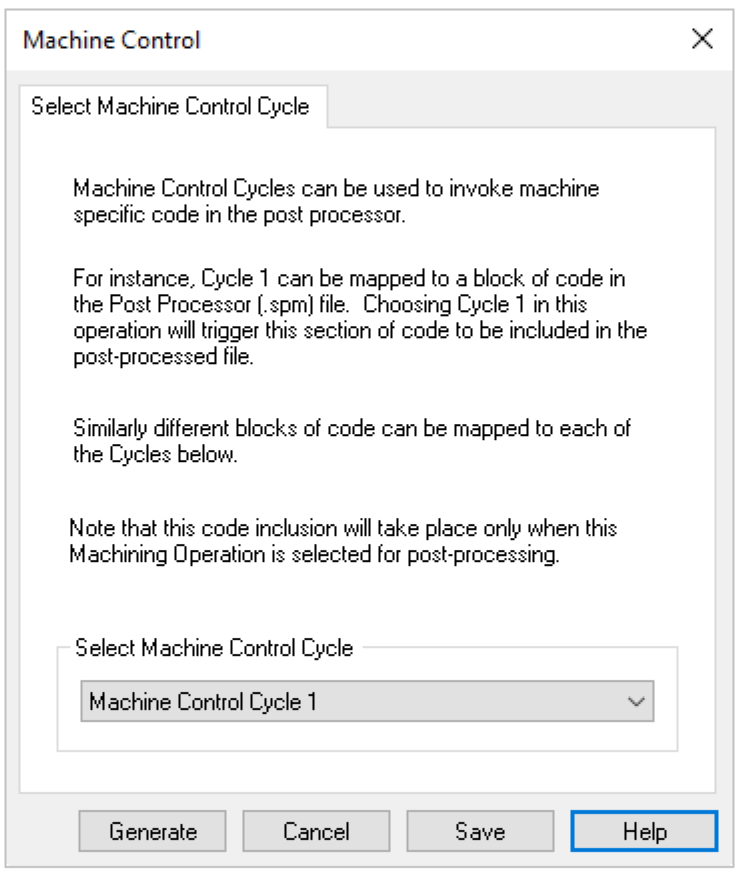
Machining Control Operation Menu Item



Machining Control Operation Menu Item TURN Module)

Dialog Box: Machining Control

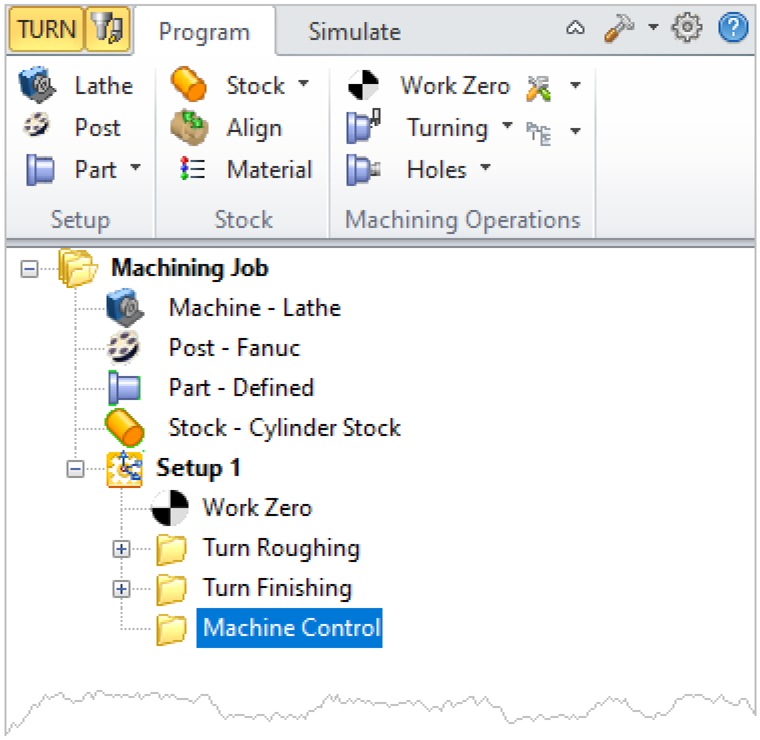
When you select this option, the following dialog will be invoked.



Dialog Box: Machining Control

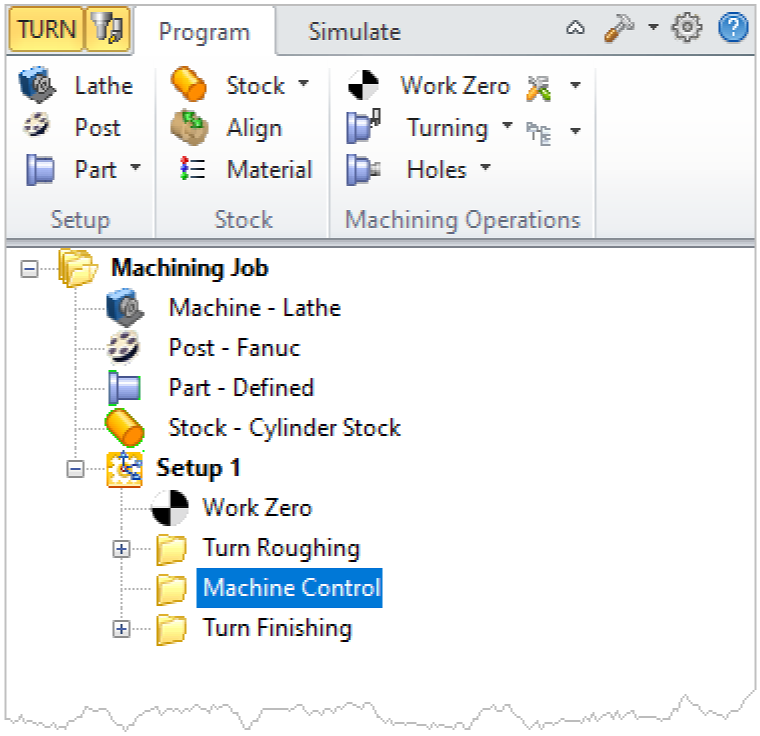
The cycle code for each cycle is specified in the post processor file. Selecting a cycle triggers this section of code to be included in the post processed file.

Select a Cycle and click Generate. A Machine Control operation is now created and listed under Setup.



Select a Cycle and click Generate. A Machine Control operation is now created and listed under Setup

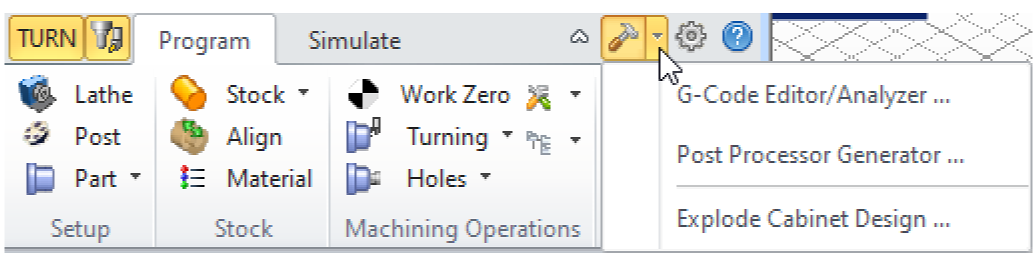
Post processing the operation will output the Machine Control code specified in the post processor file to the posted output file. Here is an example that shows Machine Control cycle inserted between two machining operations.



example that shows Machine Control cycle inserted between two machining operations

7.13 Tools and Utilities

 CAM system Tools and Utilities provides access to G-Code Editor/Analyzer and Post process generator. To access the functions, select the Tools option under the Machining Browser.



CAM System Utilities Menu Item

Loads the NC editor. By default this is set to notepad.

This is specified under Program to send posted file to which can be found in Set Post-Processor Options dialog.

Refer to Set Post Options for additional information.

Post Process Generator - This loads Post Processor Generator utility.

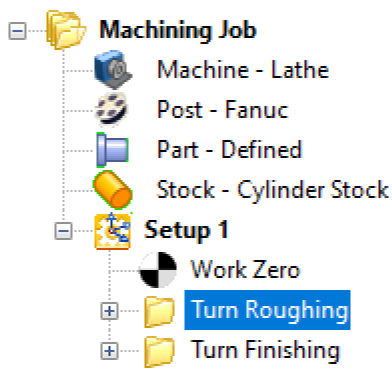
7.13.1 Save As Defaults

Save As Defaults allows you to set default parameters for machining operations. This allows the reuse of the machining parameters without having to enter the same parameters when creating new machining operations on same part or new part files.

To Save As Defaults:

Step 1: Create or Select a Machining Operation

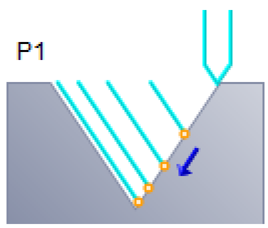
Create or Select a machining operation under the Program tab in Machining Browser.



Step 1: Create or Select a Machining Operation

Step 2: Save As Defaults

Right mouse button click on a machining operation and select Save As Defaults.



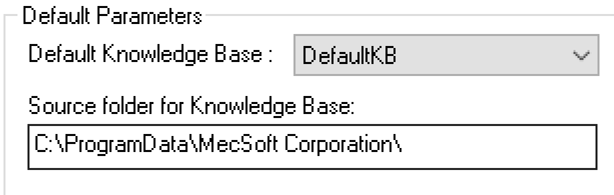
Step 2: Save As Defaults

Step 3: Specify a File Name

This displays a Save As dialog when a default knowledge base is not specified under Set Machining Preferences.

Specify a file name Click Save. This creates a default knowledge base for the profiling operation and is saved to the file name you chose.

The saved knowledge base is then automatically set as the default knowledge base to load under Machining Preferences and the parameters defined in this knowledge base are used when creating new machining operations.



Set Machining Preferences

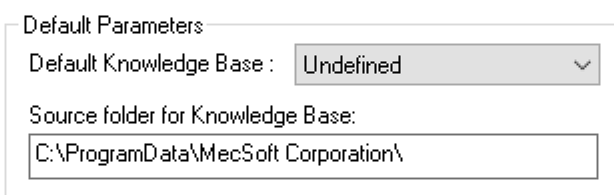
Step 4: Things to Remember

- Save As Defaults can be set for all machining operation types.
- Once a default Knowledge base is specified under Machining Preferences, selecting Save as Defaults for other machining operation types, adds other operation parameters to the same Default knowledge base file.
- If a default for a specific operation type does not exist, the system defaults are loaded.
- Changing the parameters for the above machining operation and saving as defaults overwrites the default parameters with the new one.
- Multiple Default Knowledge bases can be created and saved. This could come in handy when machining different types of materials, which requires different cutting parameters. You could create one for machining Steel, Aluminum, Wood, etc...

Step 5: To Create a NEW Default Knowledge Base

To create a new default knowledge base:

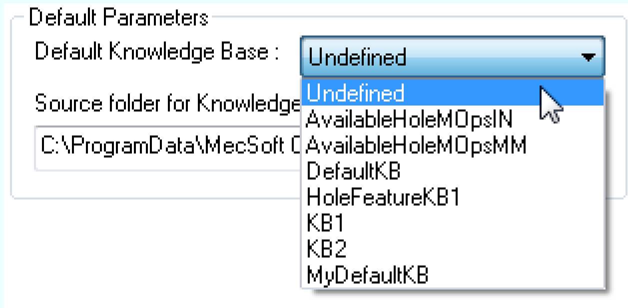
a. Under Machining Preferences, change the Default Knowledge Base to load to Undefined.



b. Select a machining operation under the Mops browser, right mouse button click and select Save As Defaults.

c. Specify a new file name and click Save. The saved knowledge base is now set as the default knowledge base to load under Machining Preferences and the parameters defined in the knowledge base is used when creating new machining operation.

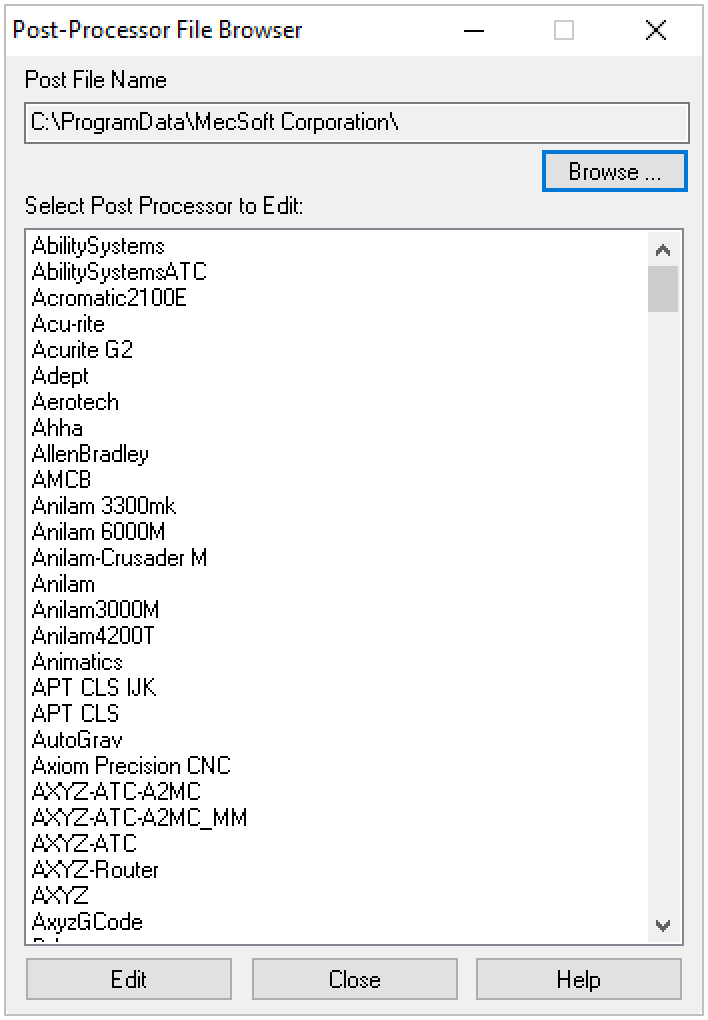
Only one Default Knowledge base can be loaded. You can change the default knowledge base to load before creating a new machining operation.



Refer to the following section for information on selecting default knowledge from Machining Preferences.

7.13.2 Post Process Generator ...

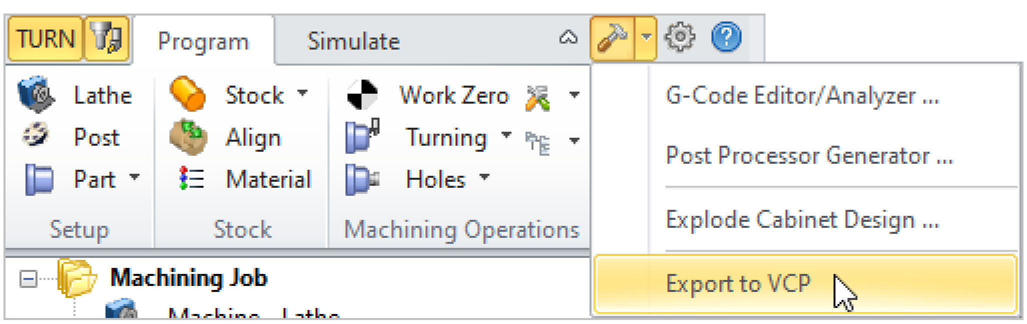
This utility can be used to edit and set up new post-processors to be used in RhinoCAM. The default location of the Post File Names is selected. Pick Browse ... to select a different location. Select a post processor from the list and click Edit to display the Post Processor Generator dialog box.



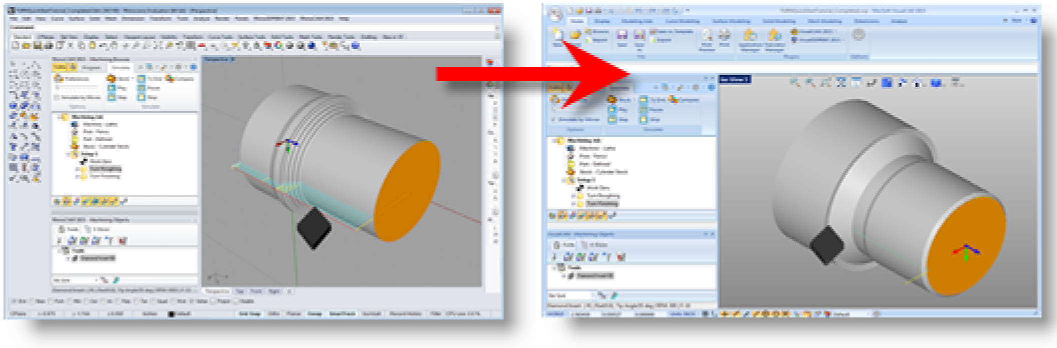
Dialog Box: Post Process Generator

7.13.3 Export to VCP

You can use this utility to export the current RhinoCAM *.3dm part file (*.3dm) to a VisualCADCAM part file (*.vcp) retaining all part geometry and existing toolpath operation definitions.



Export RhinoCAM Parts Files to VisualCADCAM *.VCP Files

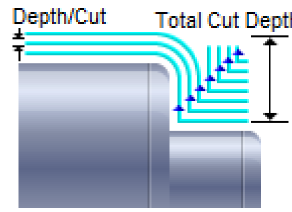


Creating Turning Operations

The section details all of the Turning operation types that can be created in TURN Module.

8.1 Roughing

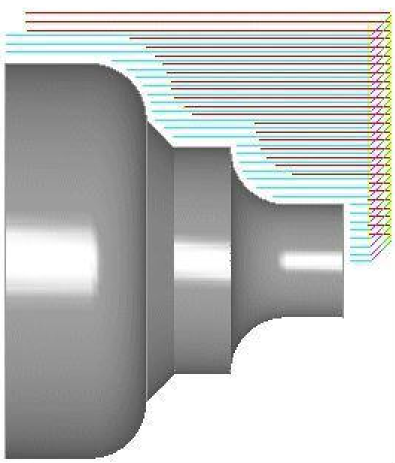
This is the RhinoCAM TURN Module’s principal method of roughing, in which the material is roughed out in multiple cuts. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large radius tool. Roughing is typically followed by finishing toolpaths.



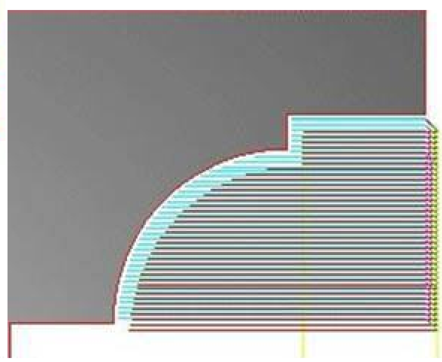
Turn Roughing Operation

Both part and stock geometry are used to determine the regions that can be safely machined. Roughing can be of 3 types: OD Roughing, ID Roughing, and Front Facing (Face Roughing).

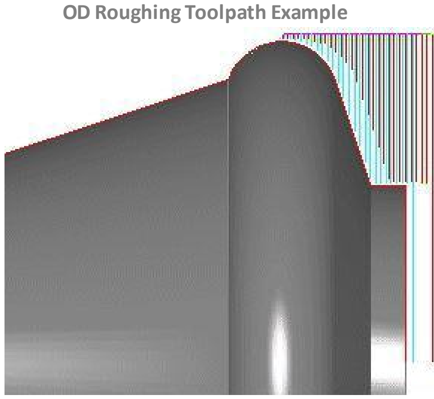
Turn Roughing Examples



OD Roughing Toolpath Example



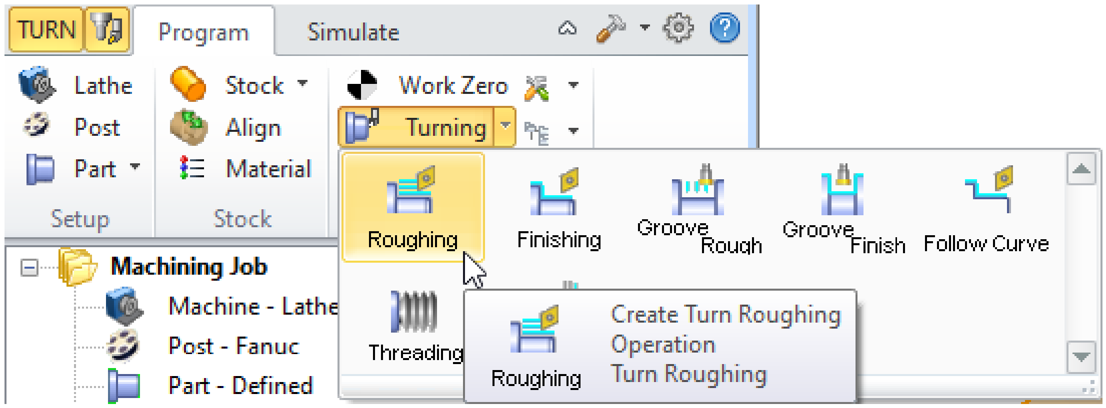
ID Roughing Toolpath Example



Face Roughing Toolpath Example

Turn Roughing Menu Item

The Turn Roughing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Roughing operation.

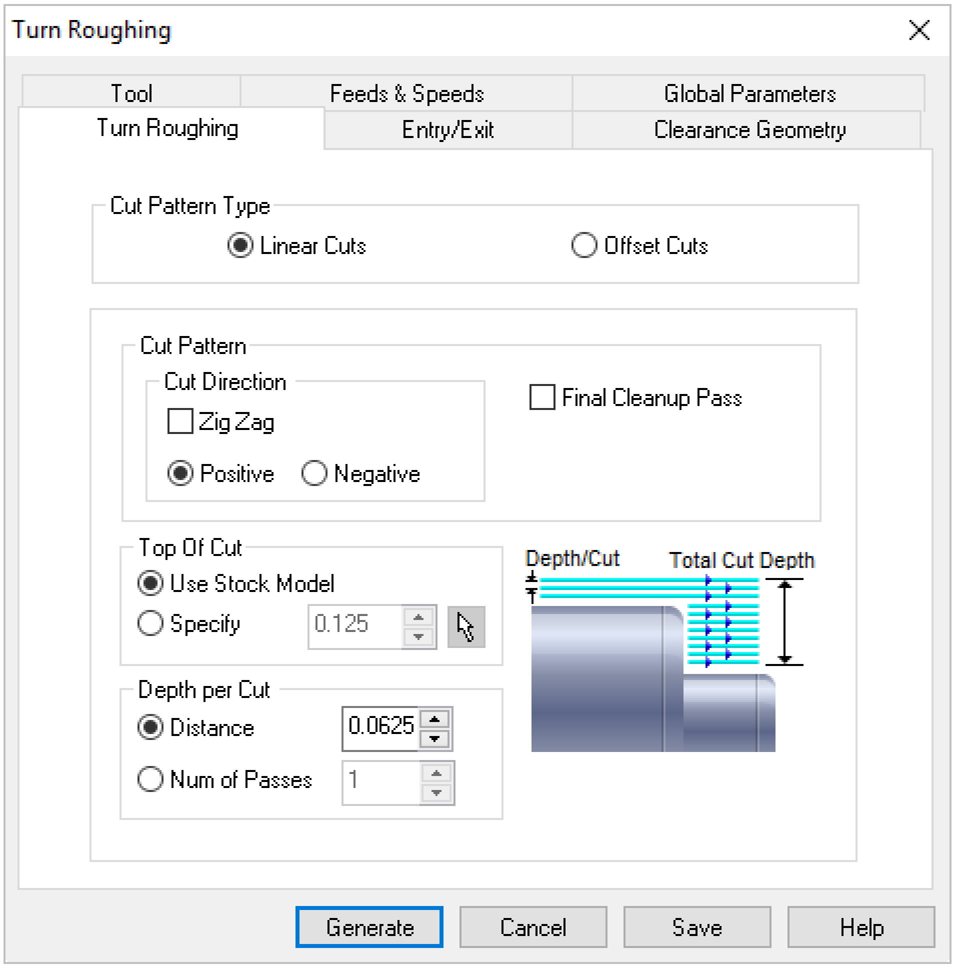


Turn Roughing Menu Item

Dialog Box: Turn Roughing

The toolpath generated depends on user-defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Turn Roughing operation. This dialog box is shown below.

This dialog has six tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

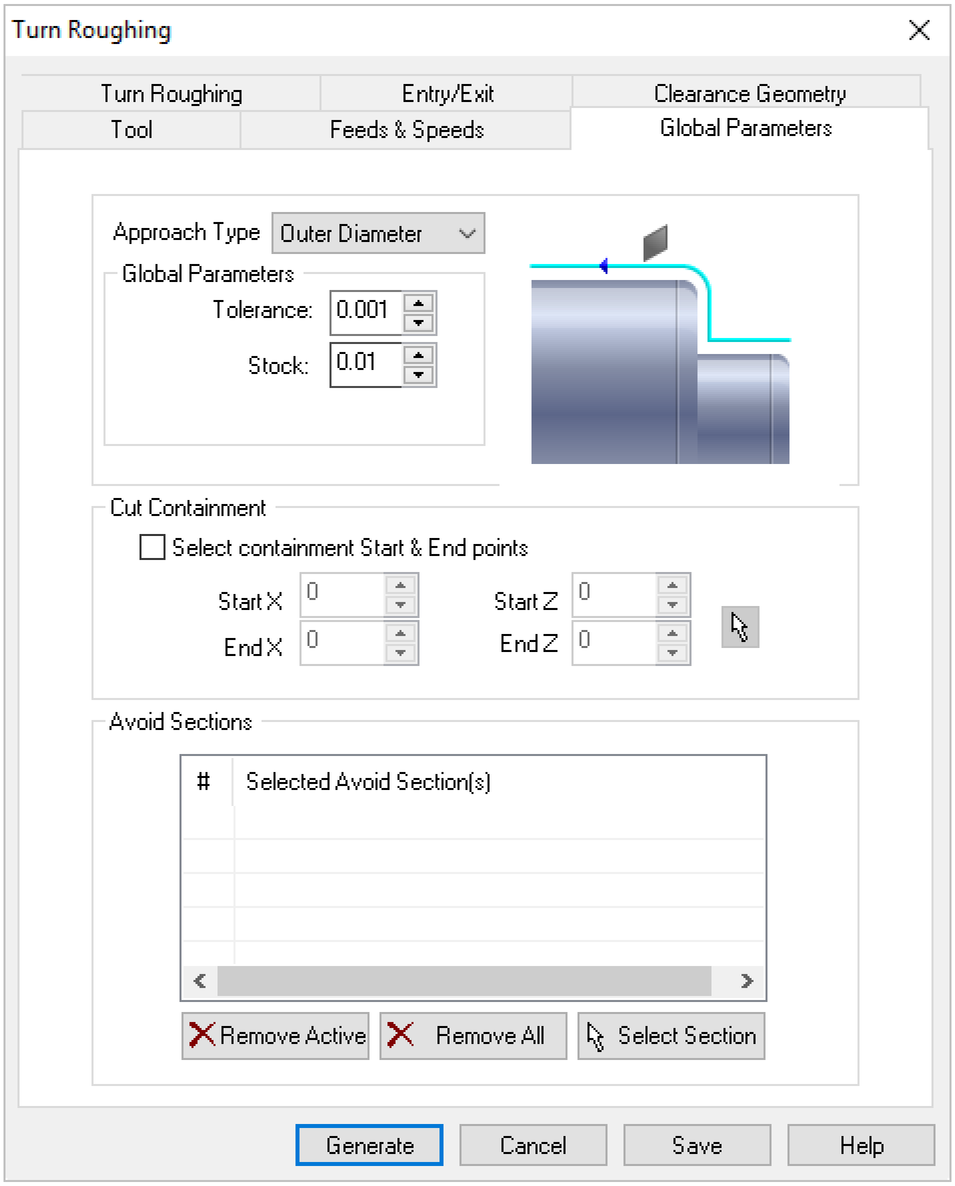


Dialog Box: Turn Roughing

8.1.1 Global Parameters

The following dialog allows you to set the Global Parameters for Turn Roughing operations. You can set the Approach Type, Global Parameters and Cut Containment and Avoid Sections via this property page.

Dialog Box: Global Parameters tab, Turn Roughing



Dialog Box: Global Parameters tab, Turn Roughing

Approach Type

Approach Type

Allows user to choose between Outer Diameter (OD), Front Facing and Inner Diameter (ID). The toolpaths are generated for the selected approach types.

In rouging and finishing operations, for tools with OD orientation, the approach type can be set to Outer Diameter or Front Facing. For tools with ID orientation, the approach type is automatically set to Inner Diameter.

Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

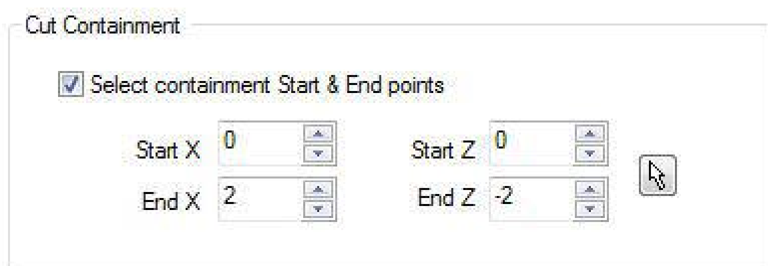
This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Stock (Roughing Operations Only)

This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Cut Containment

This allows you to select an area to contain the toolpath. This is useful in cases where a section of the part needs to be machined. This is done by selecting the check box for Select containment Start & End points.



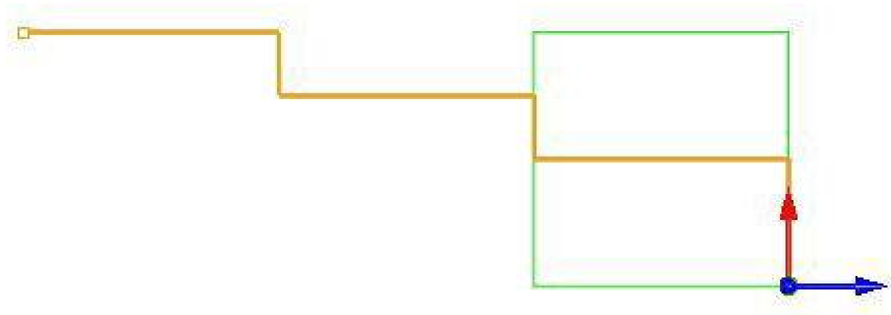
Cut Containment

You can either input the X and Z coordinate values that represent 2 corners of a containment rectangle or use the pick option to graphically select 2 corners of a rectangle for containment.

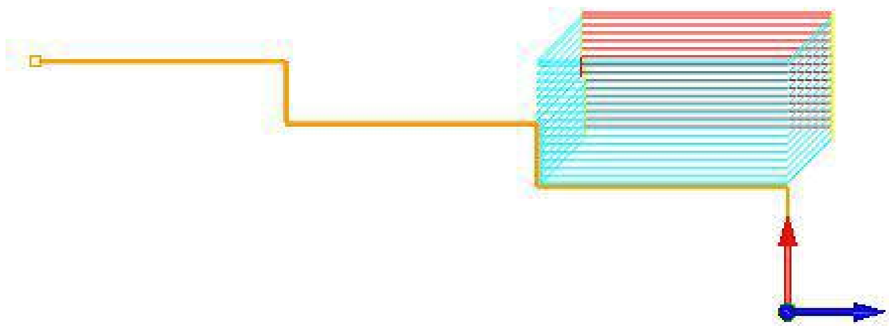
> You can use the object snap tools from the status bar to snap to points on the part geometry.

Cut Containment Examples for Turn Roughing

A containment rectangle is displayed in the viewport.

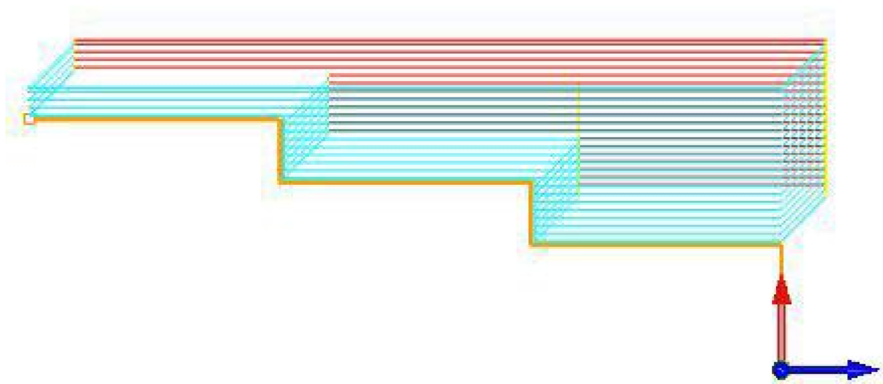


Cut Containment Examples for Turn Roughing



Cut Containment Examples for Turn Roughing

If containment is not specified, the roughing toolpath is generated for the turn part geometry based on the approach type and the defined stock geometry.



Turn Roughing Example without Cut Containment

Avoid Sections

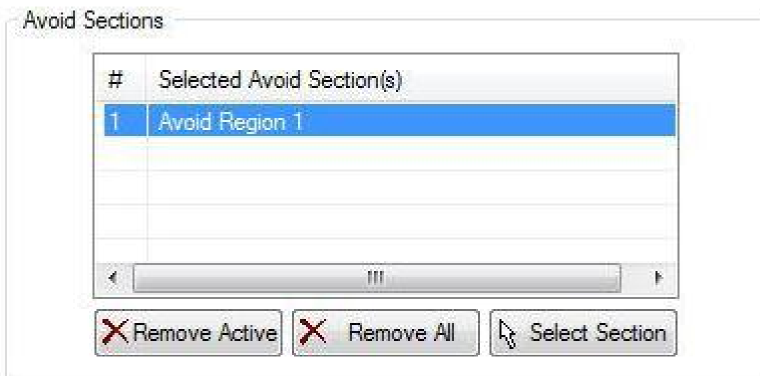
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

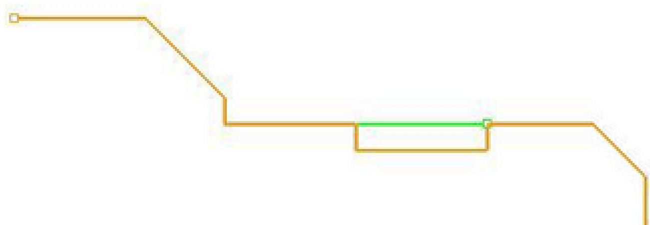
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

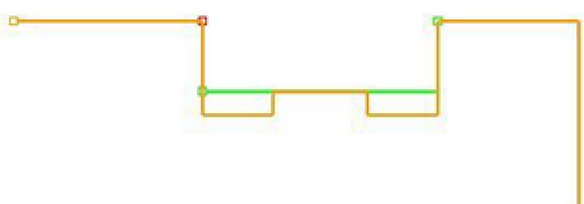


Avoid Sections, Global Parameters tab of Turn Operations dialog box

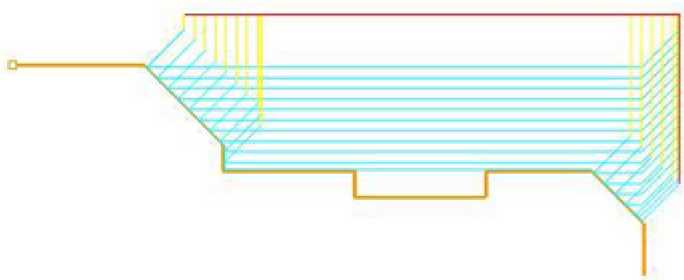
Examples for various Turn Operations



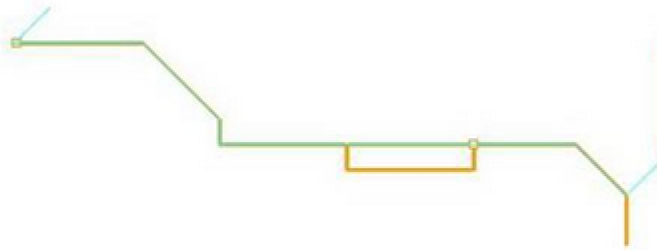
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section

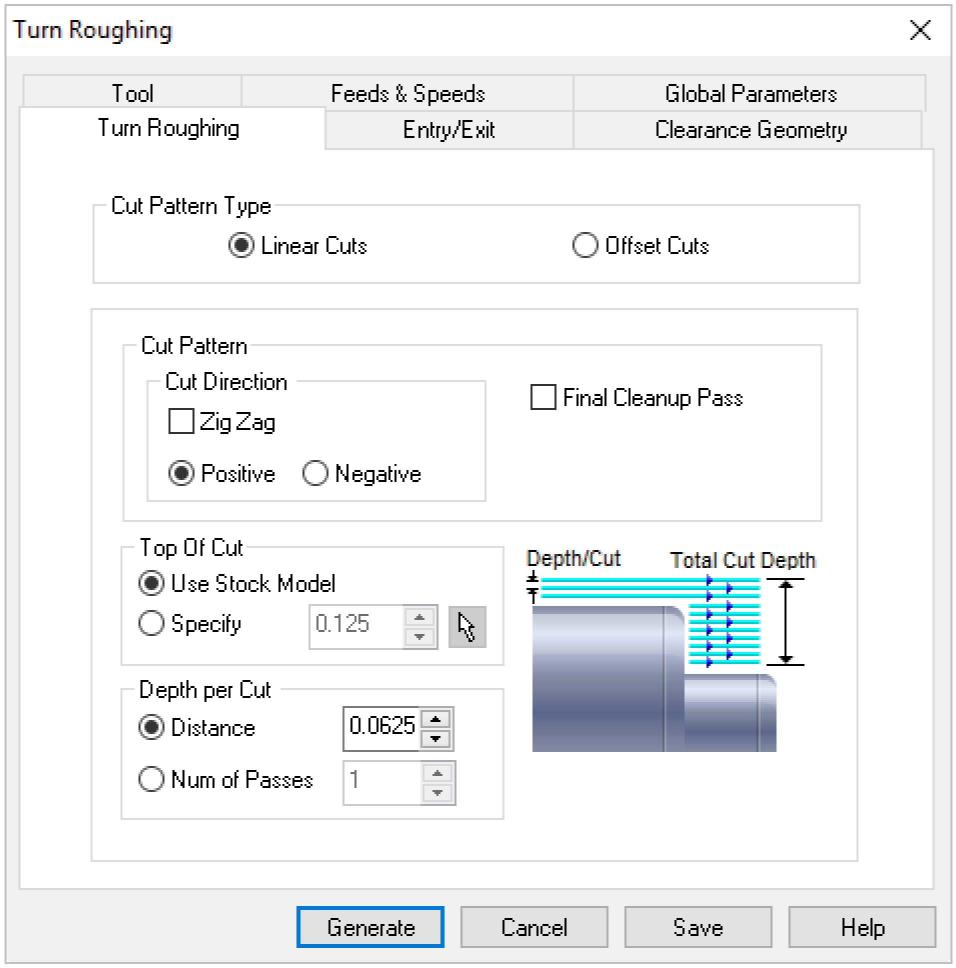


Finishing toolpath with Avoid Section

8.1.2 Turn Roughing Options

The following dialog allows you to set Turn Roughing parameters. In this tab, parameters like the Cut Pattern Type, Cut Pattern Direction and the Depth per cut can be specified.

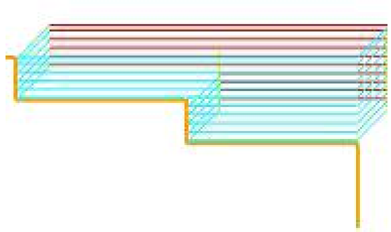
Dialog Box: Turn Roughing tab



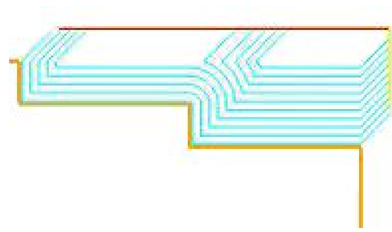
Dialog Box: Turn Roughing tab, Turn Roughing operations

Cut Pattern Type

This defines the type of path that the tool will follow at each cut level. You can choose a linear cut pattern where the tool will always traverse in linear cuts or an Offset cut pattern where the tool will traverse in successive uniform offsets of the part shape.



Linear Cut Pattern



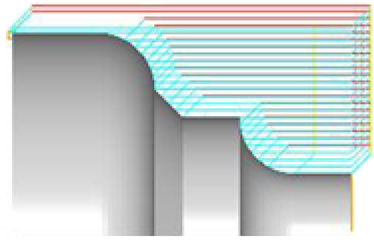
Offset Cut Pattern

Cut Direction

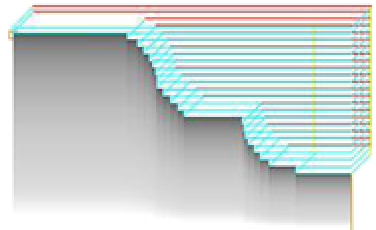
This can be specified as Positive, Negative or ZigZag.

Final Cleanup Pass

This adds a finish pass at the end of the roughing passes.



With Final Cleanup Pass



Without Final Cleanup Pass

Top of Cut

The top value for the cut can be either specified or determined using the internal algorithms of TURN Module.

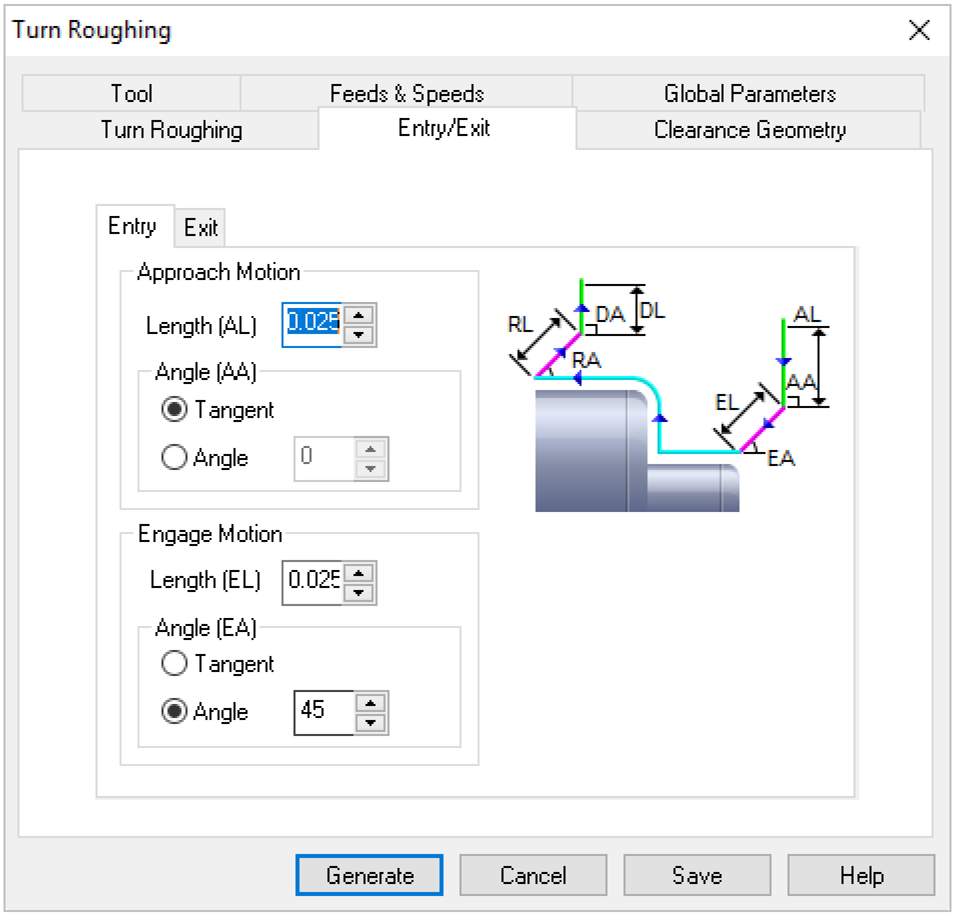
Depth per Cut

The depth per cut can be either specified as the actual depth per cut or as the number of passes required to remove the stock material to manufacture the final part.

8.1.3 Entry/Exit

The following dialog allows you to set Entry/Exit Parameters for Turn Roughing operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Turn Roughing operations

Entry Tab

The Entry tab (shown in the dialog box above) consists of Approach and Engage. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

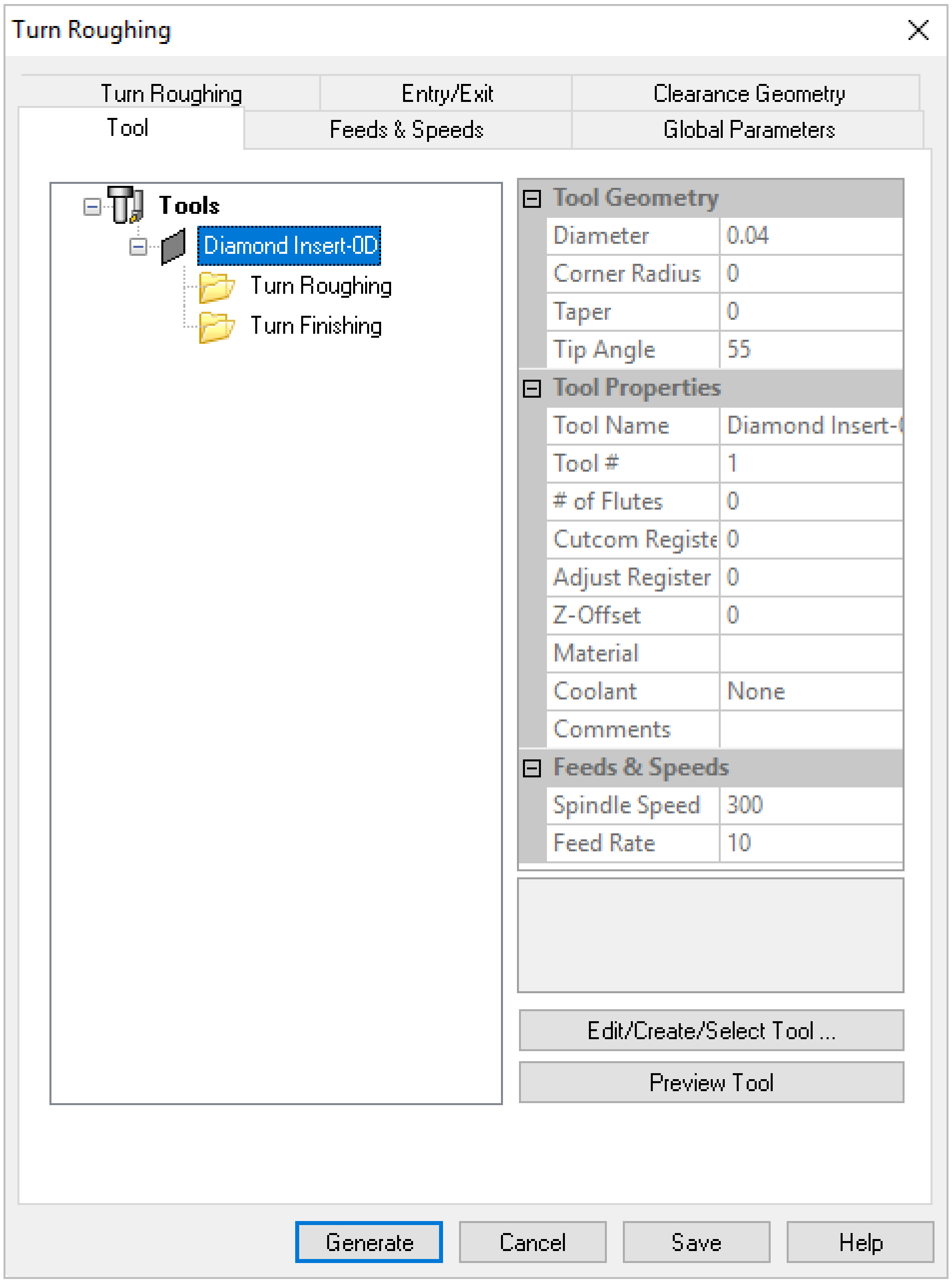
Exit Tab

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

8.1.4 Tool

The following dialog allows you to select the appropriate tool for the Turn Roughing operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Roughing operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to

Select a different tool for the current operation.

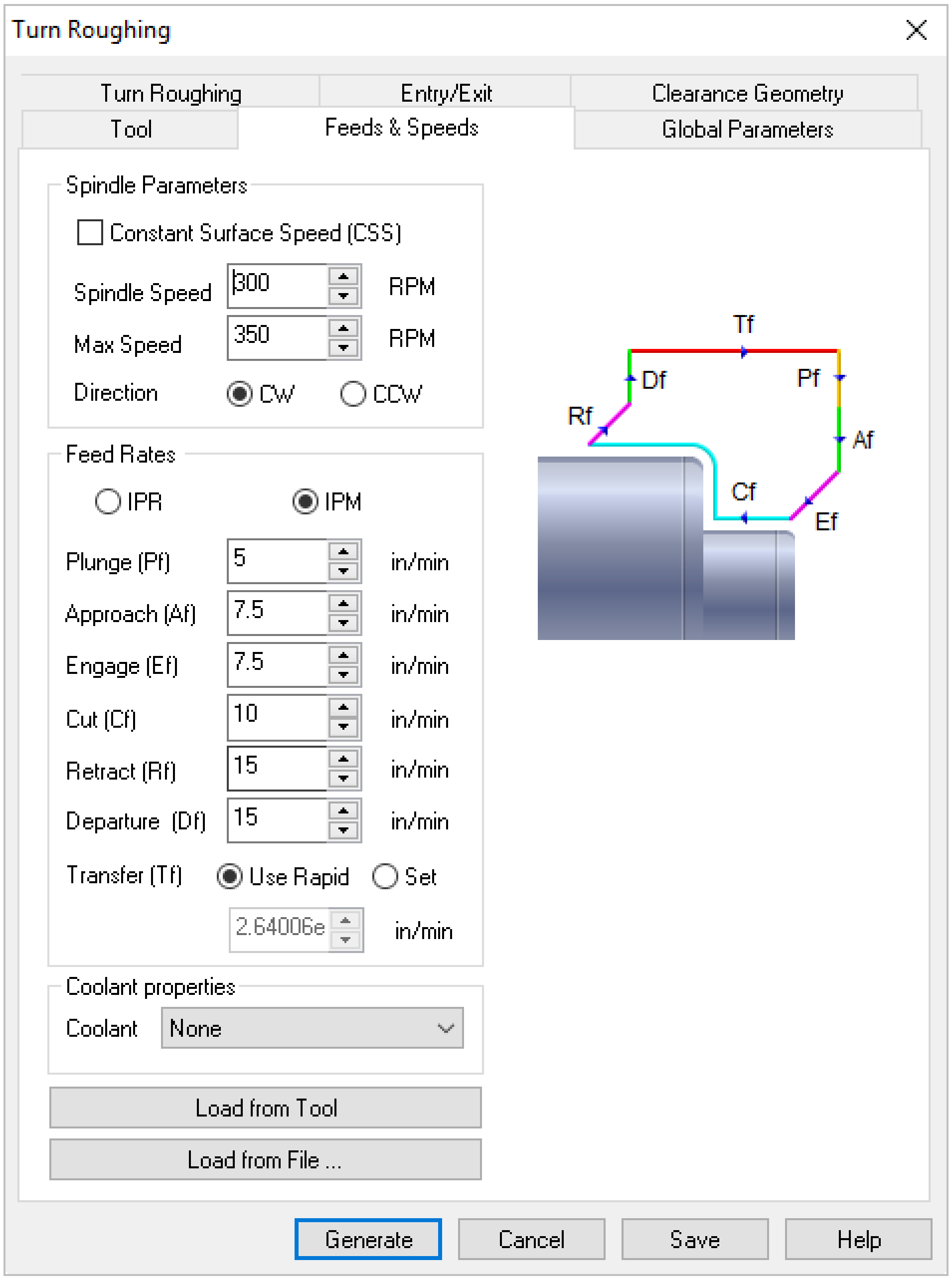
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.1.5 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Roughing operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Roughing

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).

If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your

post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed
his is the rotational speed of the spindle expressed in RPM.

Surface Speed
Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed
The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction
This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

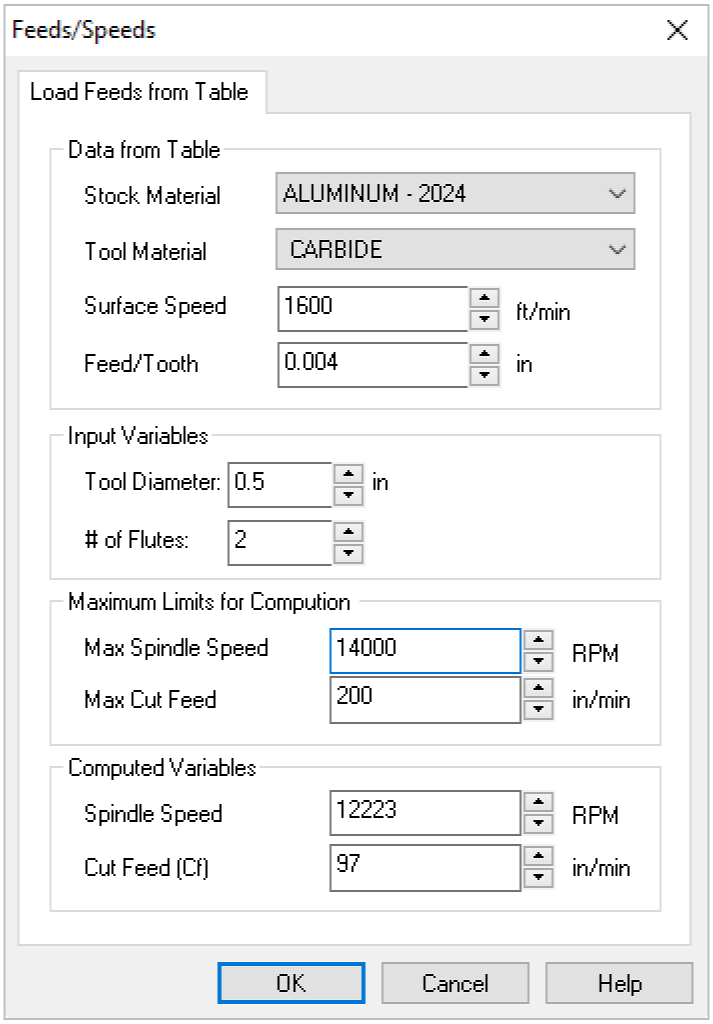
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

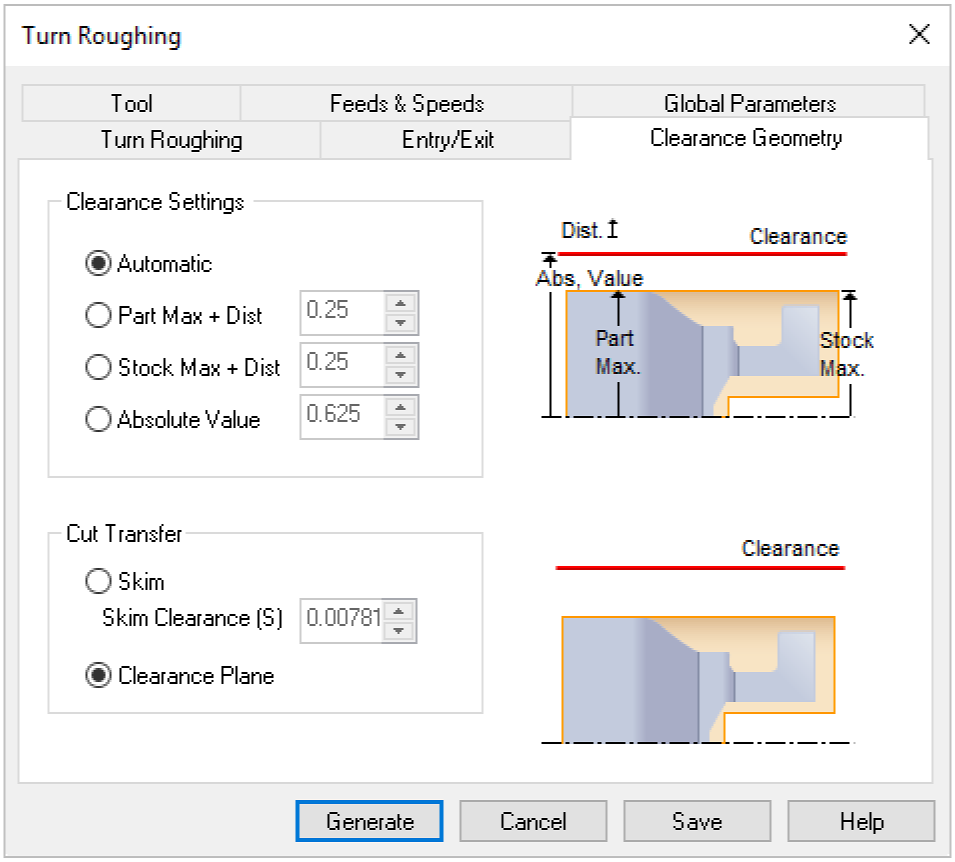
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.1.6 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Roughing operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Roughing

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist
Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist
Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value
Uses the specified distance for clearance height.

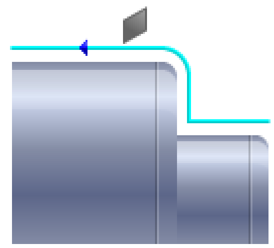
> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.
>
> For Hole Machining operations, the clearance plane is normal to the Z axis.

Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

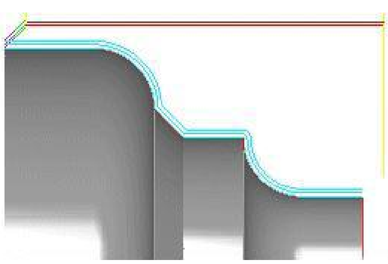
8.2 Finishing

This is one of the most commonly used machining processes. The part is rotated while a single point cutting tool is moved parallel to the axis of rotation following the contour of the geometry. Finishing can be done on the external surface of the part (OD, Face) as well as internally (boring). You can define offsets so that the tool makes multiple passes relative to the regions. Finishing is typically done after a Roughing operation, or it can be used alone.

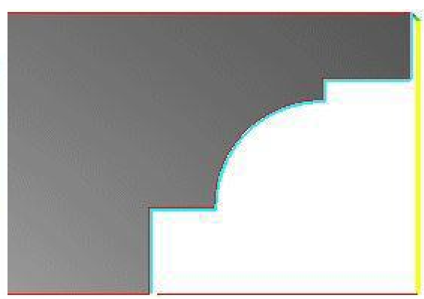


Turn Finishing Operation

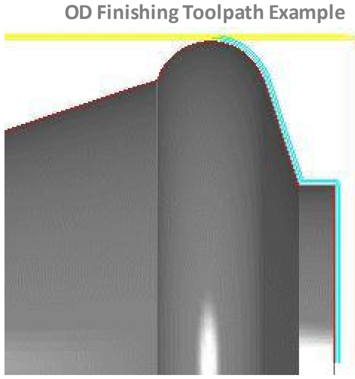
Turn Finishing Examples



OD Finishing Toolpath Example



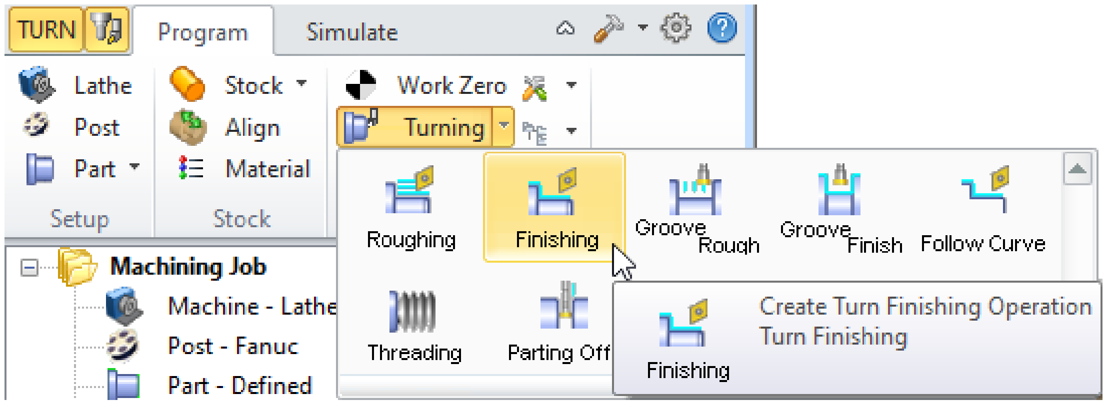
ID Finishing Toolpath Example



Face Finishing Toolpath Example

Turn Finishing Menu Item

The Turn Finishing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Finishing operation.

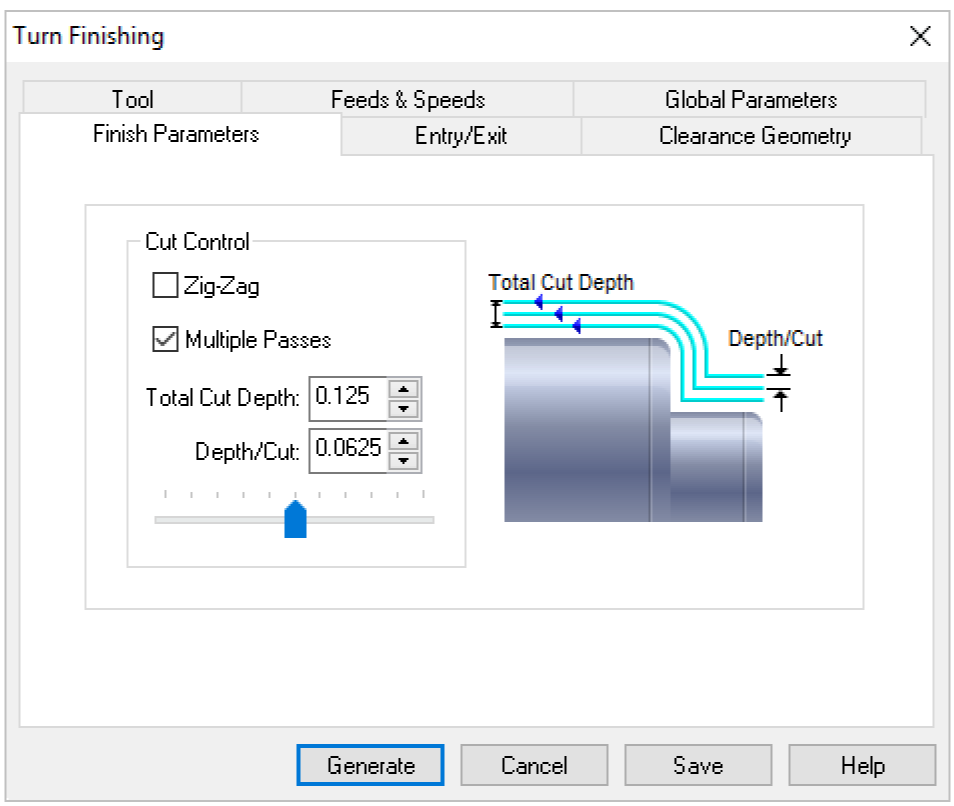


Turn Finishing Menu Item

Dialog Box: Turn Roughing

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has six tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

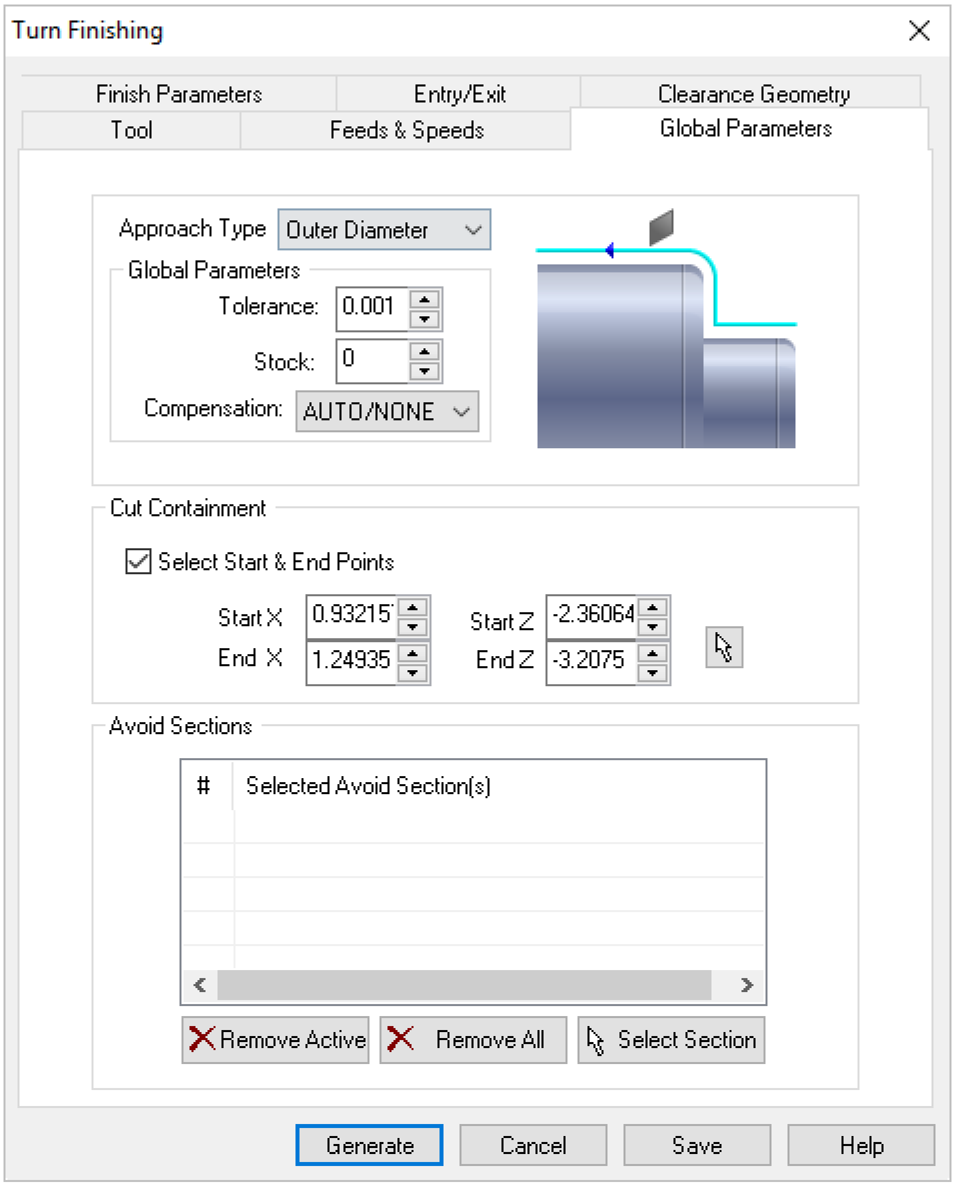


Dialog Box: Turn Finishing

8.2.1 Global Parameters

The following dialog allows you to set Global Parameters for Turn Finishing operations. You can set the Approach Type, Global Parameters and Cut Containment and Avoid Sections via this property page.

Dialog Box: Global Parameters tab



Dialog Box: Global Parameters tab, Turn Finishing

Approach Type

Approach Type

Allows user to choose between Outer Diameter (OD), Front Facing and Inner Diameter

(ID). The toolpaths are generated for the selected approach types.

In rouging and finishing operations, for tools with OD orientation, the approach type can be set to Outer Diameter or Front Facing. For tools with ID orientation, the approach type is automatically set to Inner Diameter.

Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance
This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

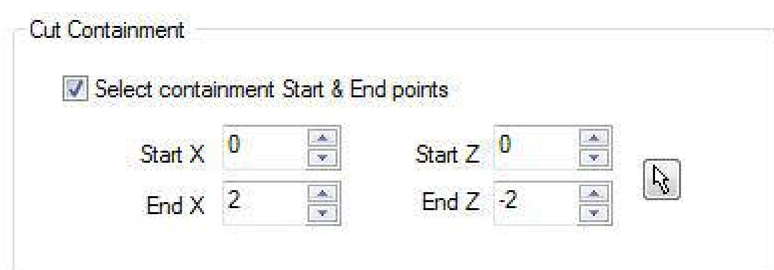
Stock (Roughing Operations Only)
This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Compensation

This stands for cutter compensation. You can turn this on by selecting from the drop down menu. The cutter compensation direction, Left or Right, is determined by the Cut Direction (Climb or Conventional). Refer to the following section for additional information - Cutter Compensation

Cut Containment

This allows you to select an area to contain the toolpath. This is useful in cases where a section of the part needs to be machined. This is done by selecting the check box for Select containment Start & End points.



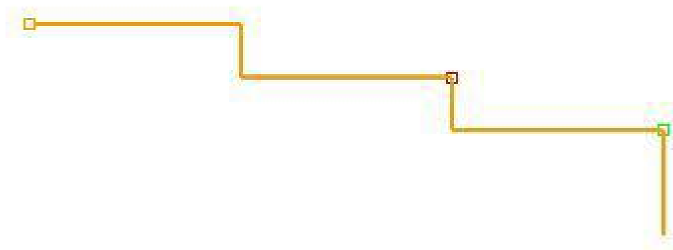
Cut Containment

You can either input the X and Z coordinate values that represent 2 corners of a containment rectangle or use the pick option to graphically select 2 corners of a rectangle for containment.

You can use the object snap tools from the status bar to snap to points on the part geometry.

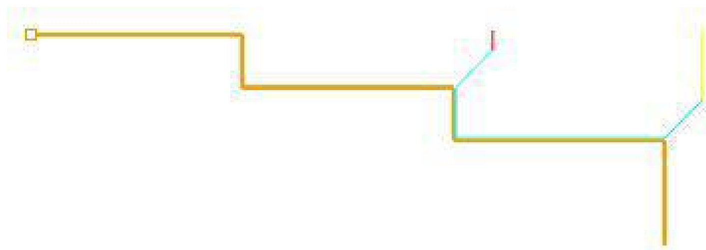
Cut Containment Examples for Turn Roughing

The start and end points are displayed on the part geometry. The start point is represented in **Green** color and end point in **Red**.



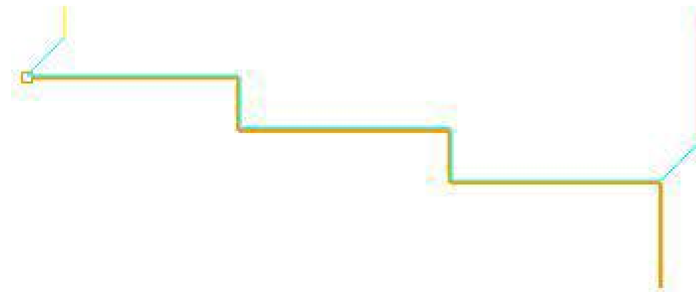
Cut Containment Examples for Turn Finishing

The finishing toolpath is contained between start and end points and cut direction of the toolpath is from the start to end point. The selection of start and end points can also be used to determine the cut direction.



Cut Containment Examples for Turn Finishing

If a containment is not specified, the finishing toolpath is generated for the turn part geometry based on the approach type and the part geometry.



Cut Containment Examples for Turn Finishing

Avoid Sections

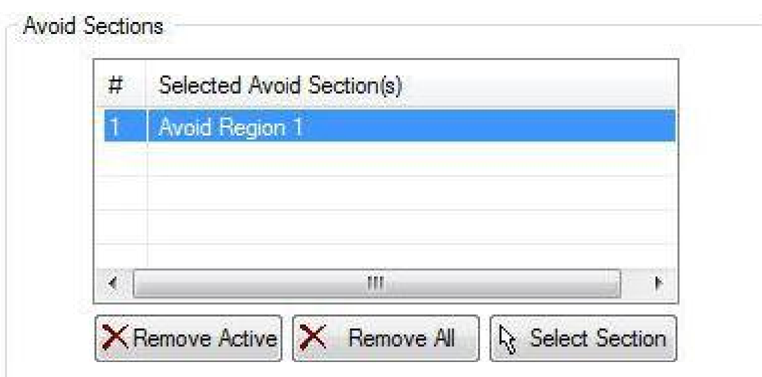
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

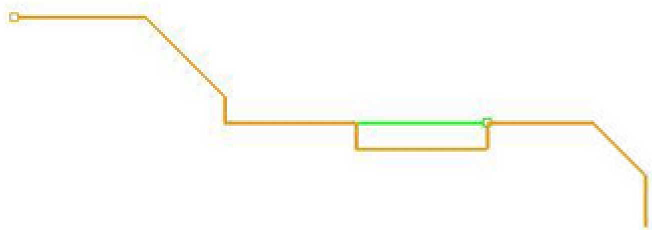
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

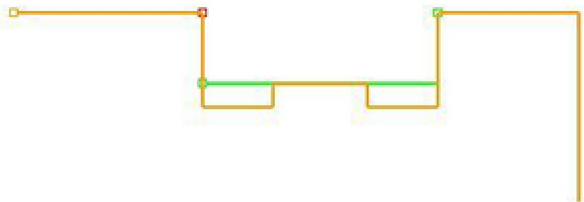


Avoid Sections, Global Parameters tab of Turn Operations dialog box

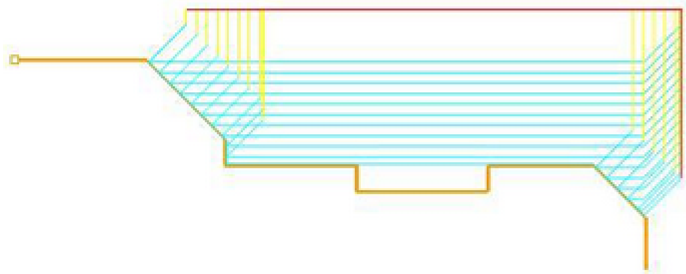
Examples for various Turn Operations



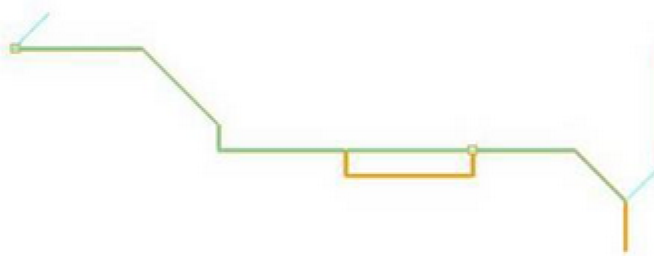
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section

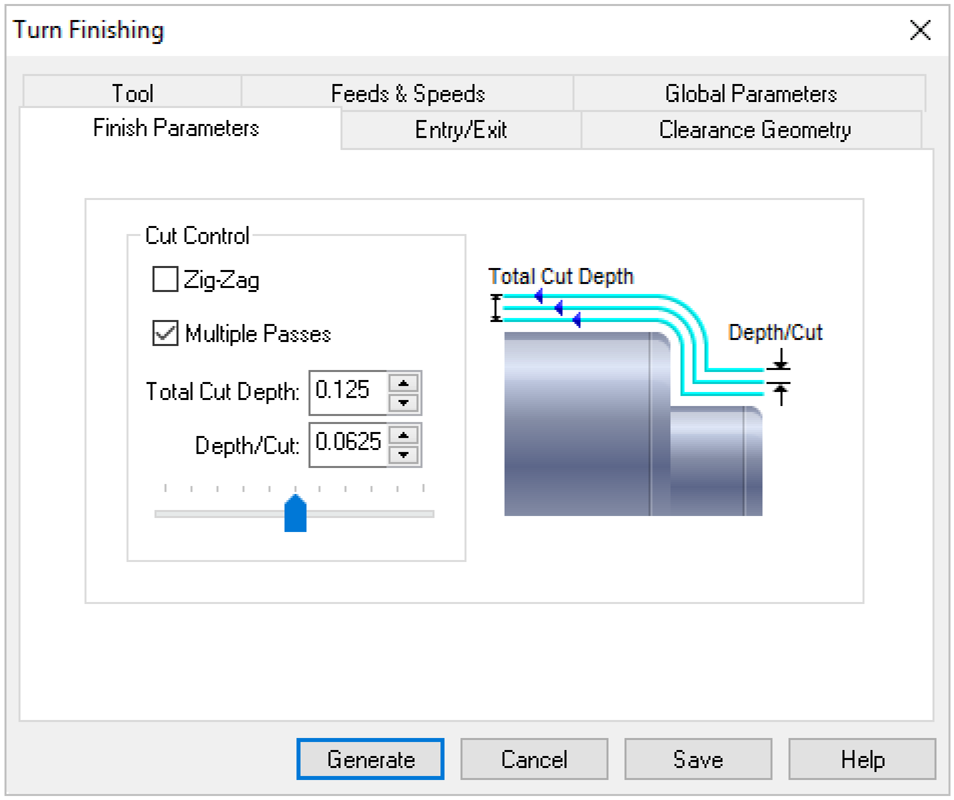


Finishing toolpath with Avoid Section

8.2.2 Finishing Parameters

The following dialog allows you to set Cut Control finishing parameters for Turn Finishing operations.

Dialog Box: Finishing Parameters tab



Dialog Box: Global Parameters tab, Turn Finishing

Cut Control

Final finishing cut can be specified in terms of the total passes of the cutter over the stock.

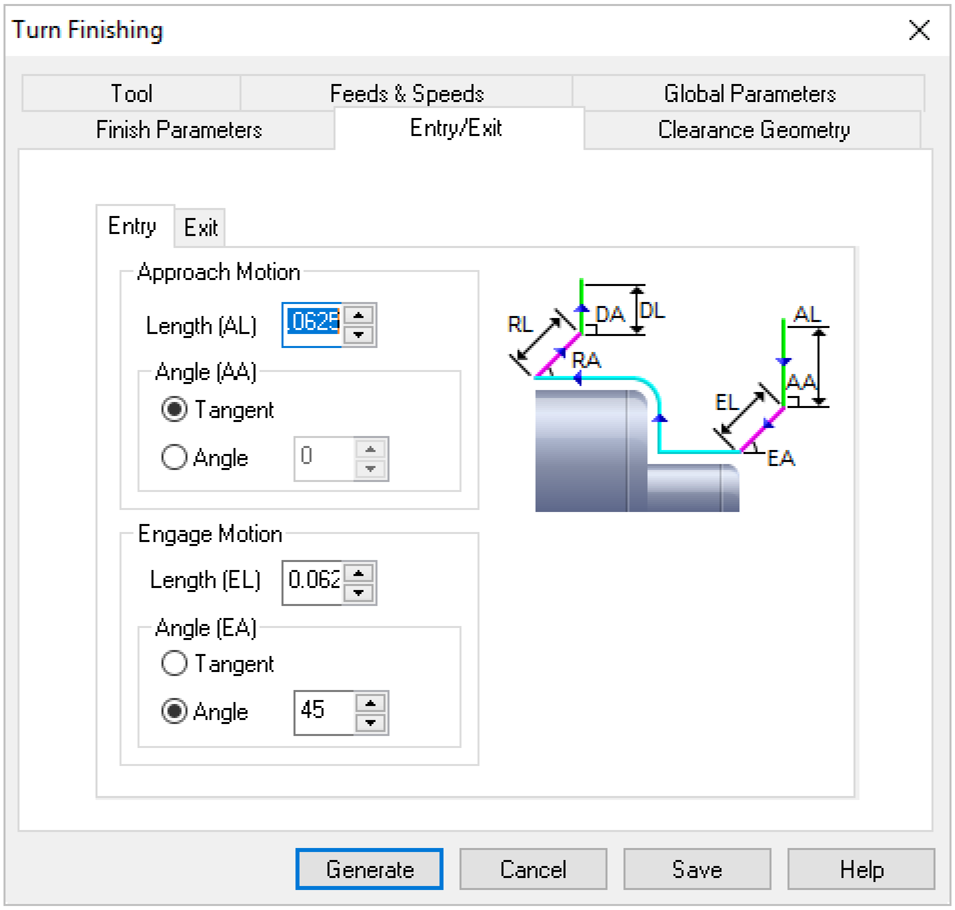
Cut Control - Allows specifying multiple passes for finishing. Selecting Multiple Passes allows user to set the Total Cut Depth and Depth/Cut.

The direction of cut can be either unidirectional or Bi-directional (Zig-Zag)

8.2.3 Entry/Exit

The following dialog allows you to set Entry/Exit Parameters for Turn Finishing operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Turn Roughing operations

Entry Tab

The Entry tab (shown in the dialog box above) consists of Approach and Engage. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

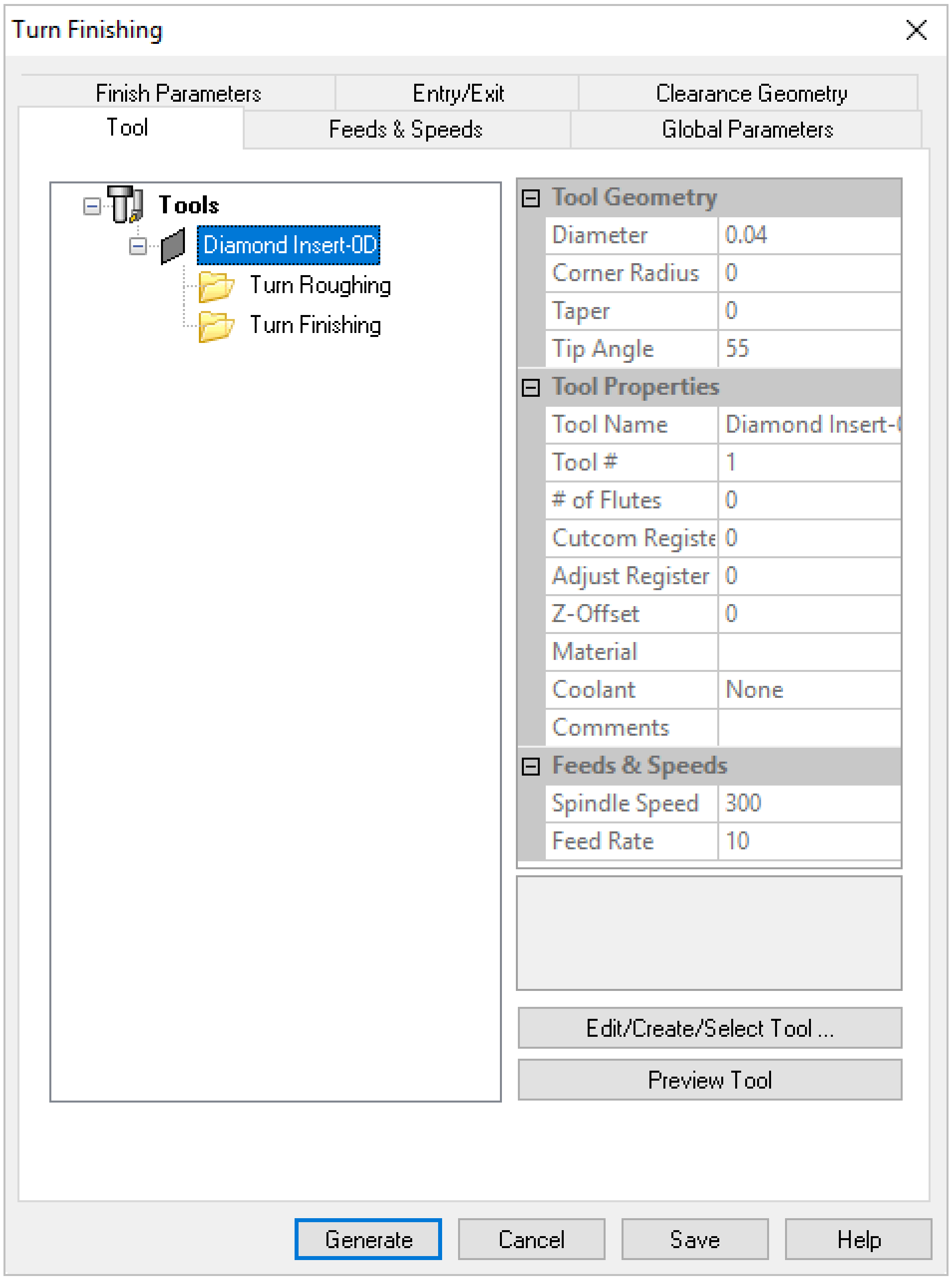
Exit Tab

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

8.2.4 Tool

The following dialog allows you to select the appropriate tool for the Turn Finishing operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Finishing

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to

Select a different tool for the current operation.

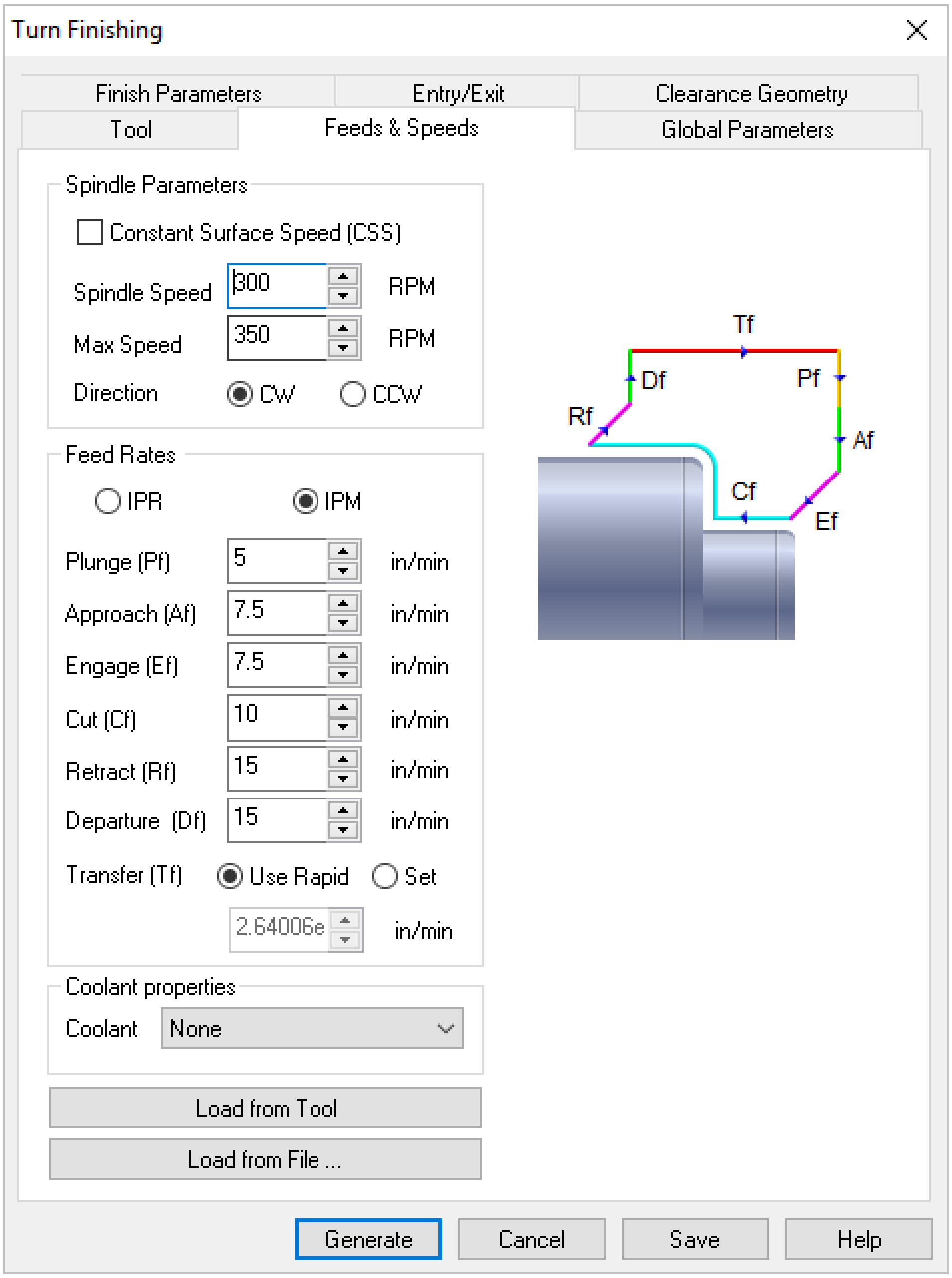
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.2.5 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Finishing operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Finishing

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your

post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed

his is the rotational speed of the spindle expressed in RPM.

Surface Speed

Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed

The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction

This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)

This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)

This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)

This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)

This is the feedrate used when the tool is cutting material

Retract (Rf)

The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)

The feedrate used to retract the tool from the material.

Transfer (Tf)

This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

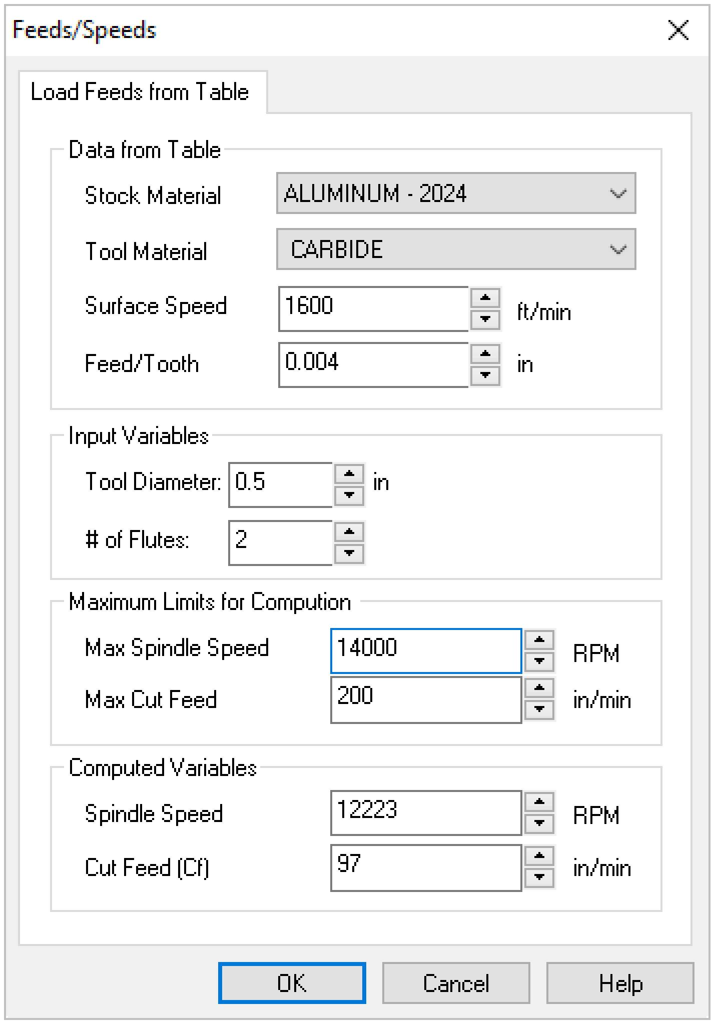
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

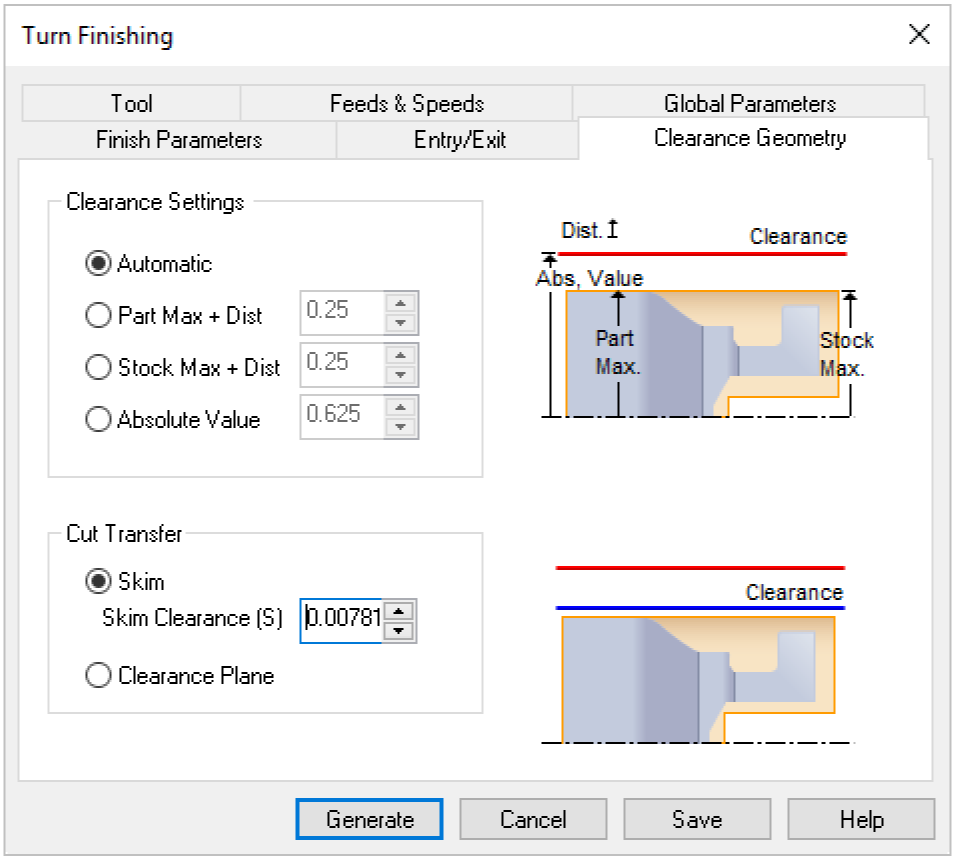
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.2.6 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Finishing operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Finishing

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

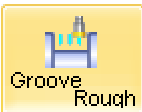
Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.
>
> For Hole Machining operations, the clearance plane is normal to the Z axis.

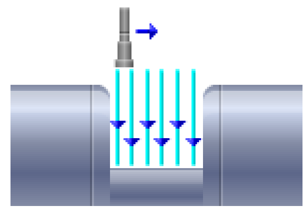
Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

8.3 Groove Roughing

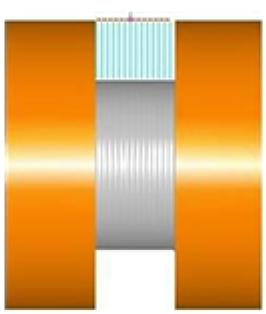


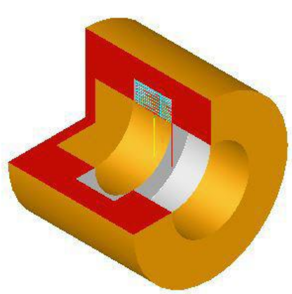
This operation is performed to machine grooves in multiple cuts on the part. This roughing operation provides user the control to set the step down and step over and choose the cut direction.

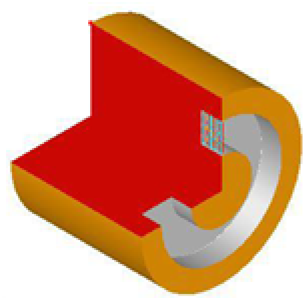

Groove Roughing

Both part and stock geometry are used to determine the regions that can be safely machined. Groove Roughing can be of 3 types: OD Groove Roughing, ID Groove Roughing, and Face Groove Roughing. The grooves are typically used to slide/fit one part into another to obtain the required assembly.

Turn Groove Roughing Examples

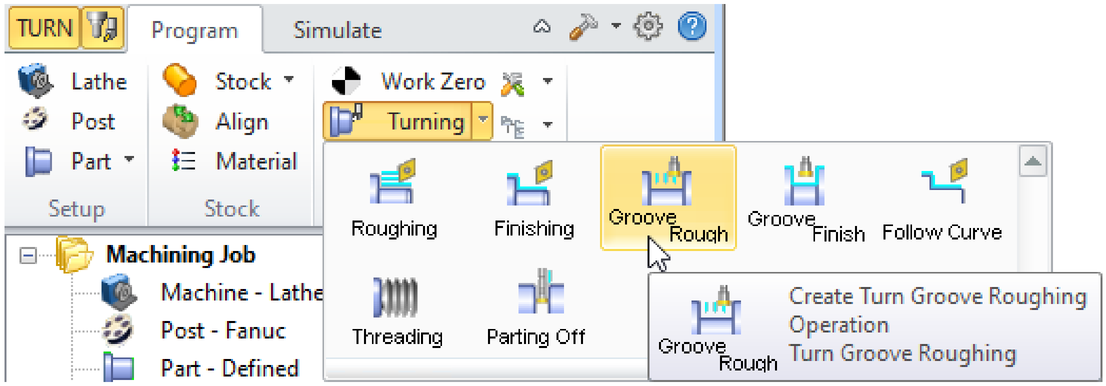

OD Groove


ID Groove


Face Groove

Turn Groove Roughing Menu Item

The Groove Roughing toolpath method is invoked by selecting the Program tab, clicking on the **Turning** button in the Machining Browser and selecting the Groove Roughing operation.

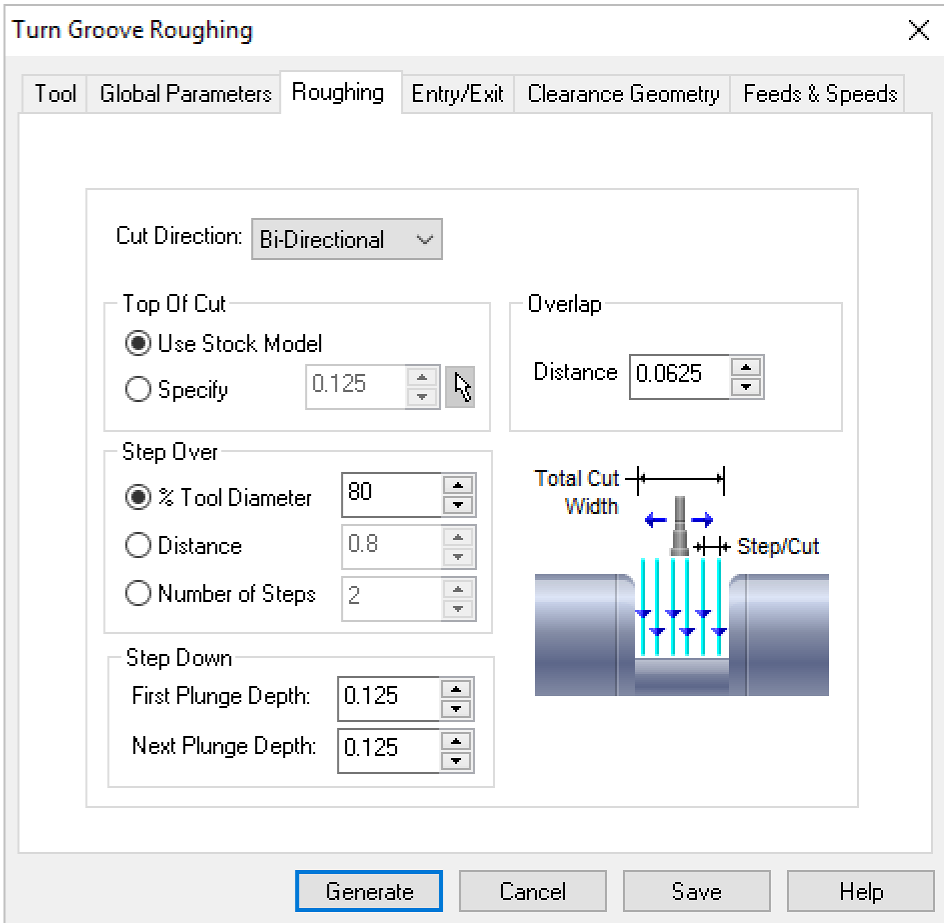


Turn Groove Roughing Menu Item

Dialog Box: Turn Groove Roughing

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has six tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

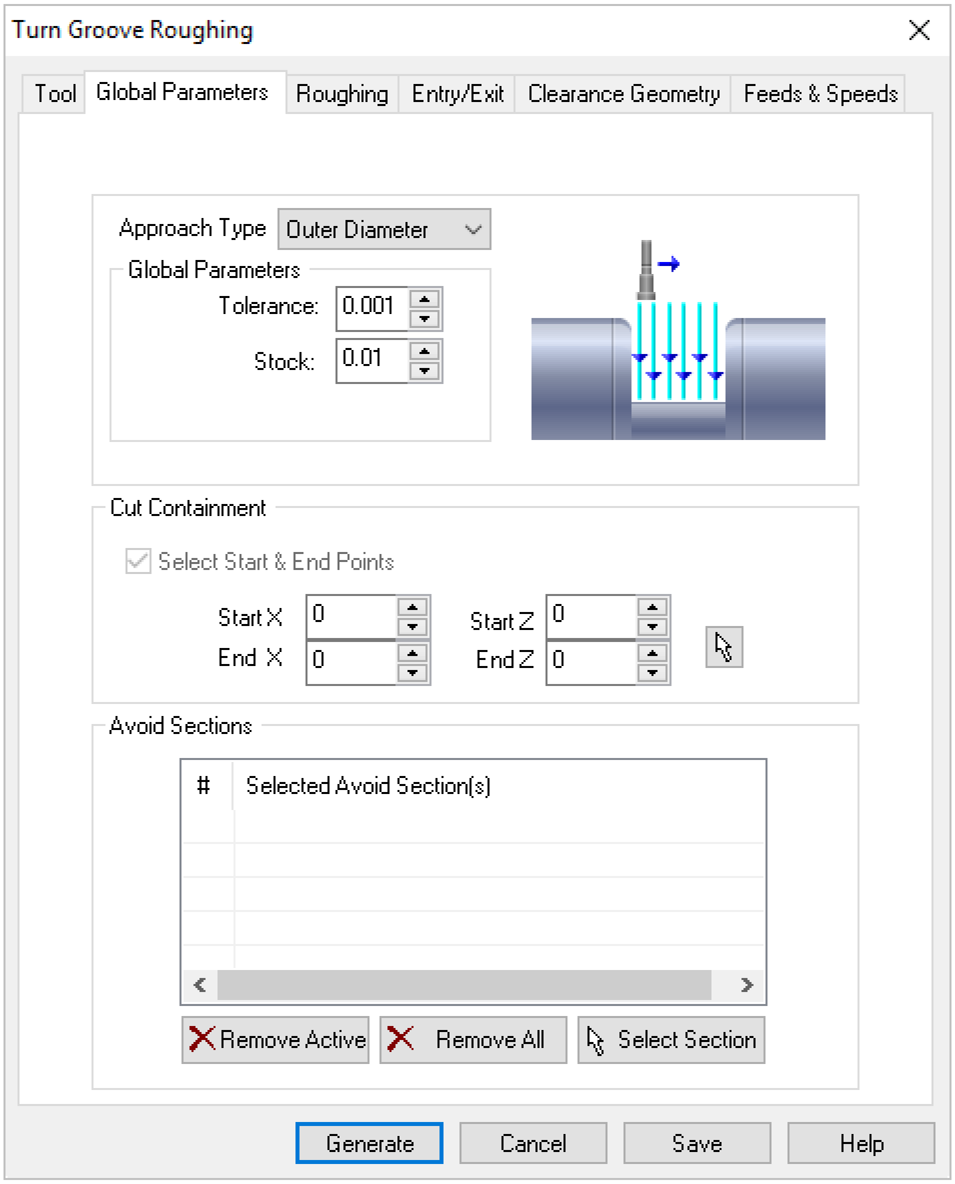


Dialog Box: Turn Groove Roughing

8.3.1 Global Parameters

The following dialog allows you to set Global Parameters for Turn Groove Roughing operations. You can set the Approach Type, Global Parameters and Cut Containment and Avoid Sections via this property page.

Dialog Box: Global Parameters tab



Dialog Box: Global Parameters tab, Turn Groove Roughing

Approach Type

Approach Type

Allows user to choose between Outer Diameter (OD), Front Facing and Inner Diameter (ID). The toolpaths are generated for the selected approach types.

In rouging and finishing operations, for tools with OD orientation, the approach type can be set to Outer Diameter or Front Facing. For tools with ID orientation, the approach type is automatically set to Inner Diameter.

Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

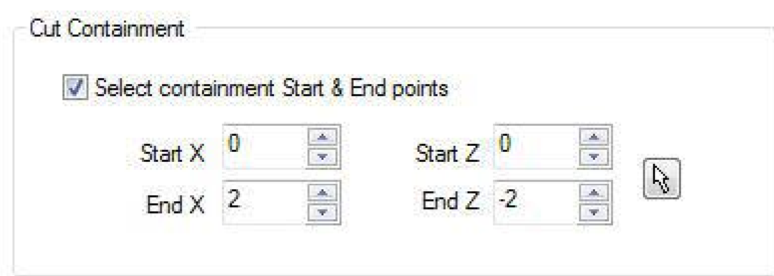
This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Stock (Roughing Operations Only)

This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Cut Containment

This allows you to select an area to contain the toolpath. This is useful in cases where a section of the part needs to be machined. This is done by selecting the check box for Select containment Start & End points.



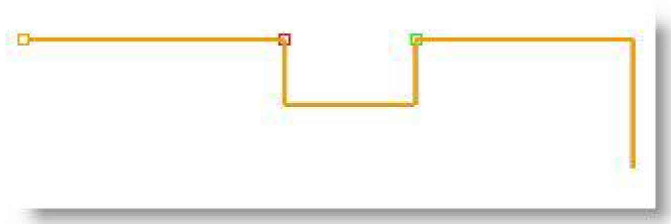
Cut Containment

You can either input the X and Z coordinate values that represent 2 corners of a containment rectangle or use the pick option to graphically select 2 corners of a rectangle for containment.

> You can use the object snap tools from the status bar to snap to points on the part geometry.

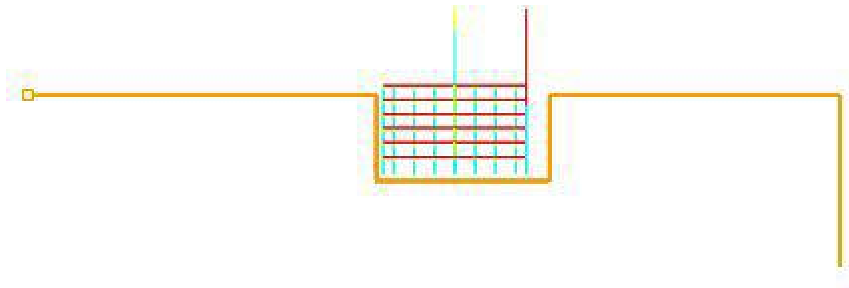
Cut Containment Examples for Turn Groove Roughing

The start and end points are displayed on the part geometry. The start point is represented in **Green** color and end point in **Red**.



Cut Containment Examples for Turn Groove Roughing

The groove roughing toolpath is generated between start and end points.



Cut Containment Examples for Turn Groove Roughing

The start and end points are also used to determine the cut direction when cut direction is set to Uni-Directional under the Roughing tab.

Avoid Sections

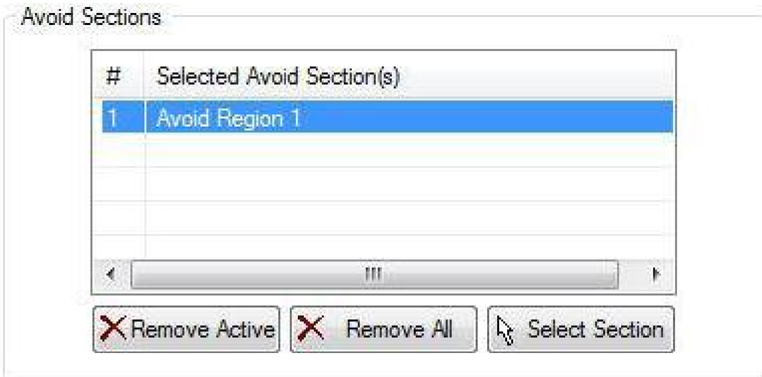
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

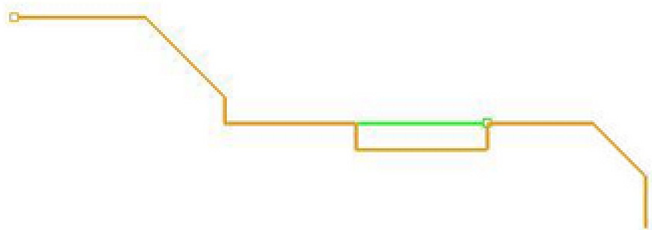
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

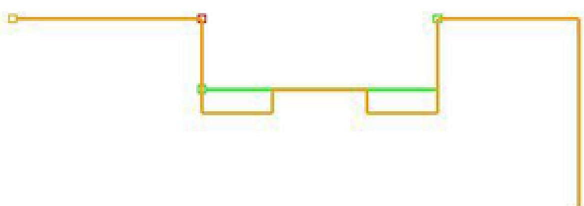


Avoid Sections, Global Parameters tab of Turn Operations dialog box

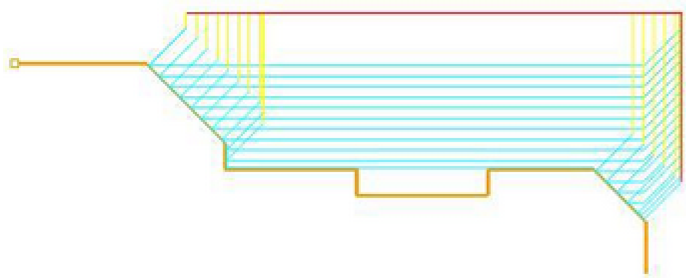
Examples for various Turn Operations



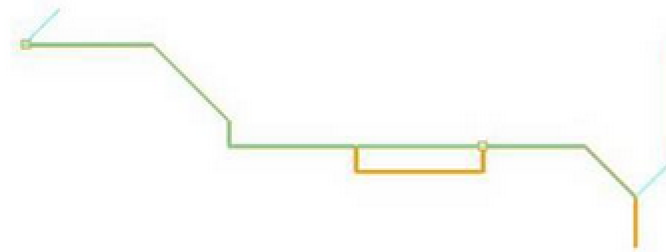
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section

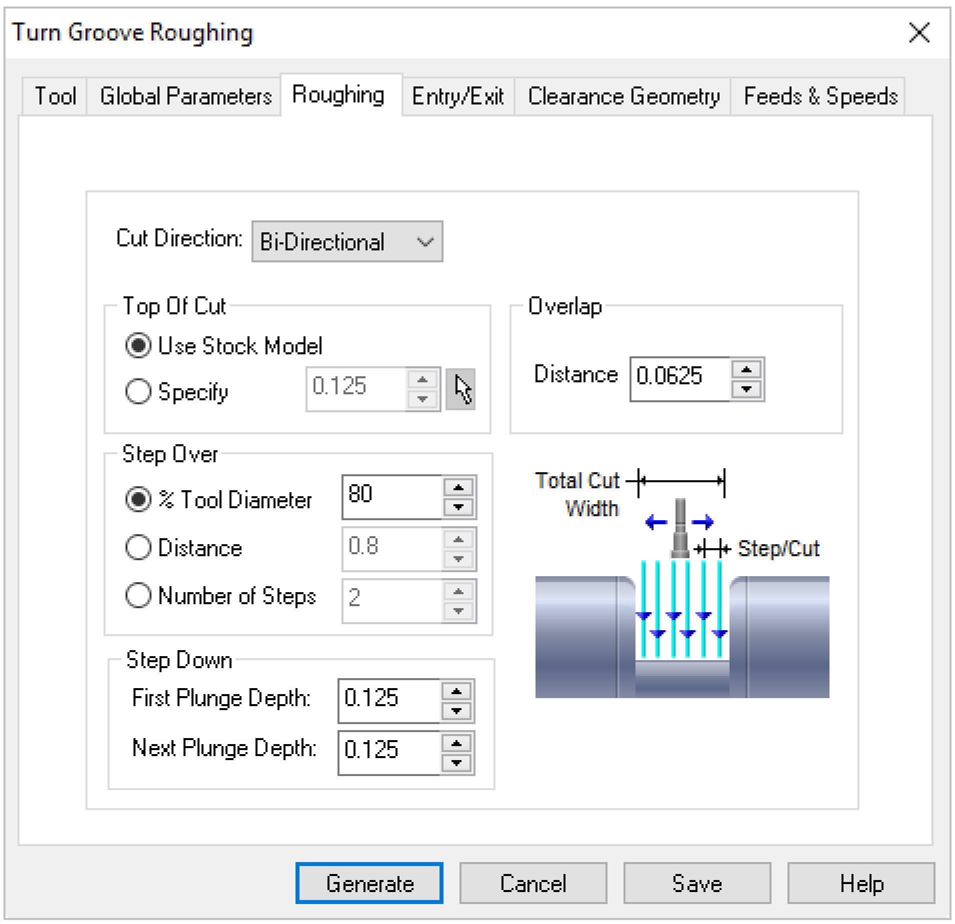


Finishing toolpath with Avoid Section

8.3.2 Roughing Parameters

In this tab, parameters like the Cut Direction, Step Over and Step Down can be specified for Turn Groove Roughing operations.

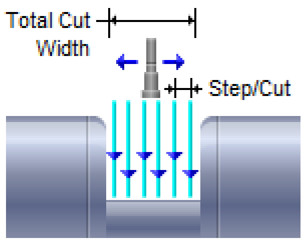
Dialog Box: Roughing tab



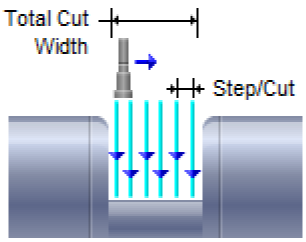
Dialog Box: Roughing tab, Turn Groove Roughing

Cut Direction

This can be either Bi-Direction or Uni-Directional. Bi-Directional cuts start at the center of the groove and works its way in both directions towards the start and end points. Uni-Directional cuts start from the start point and works its way towards the end point of the groove feature. When using Bi-Directional, an Overlap distance parameter can be specified.



Bi-Directional



Uni-Directional

Top of Cut

This tells the system where to begin cutting. Select Use Stock Model to allow the system to determine this location based on the Stock. Select Specify and then enter a value to locate where to begin cutting.

You can select the Pick button and then select a point on your part for the top of cut. It's distance will calculated and entered into the Distance field.

Overlap

With Cut Direction set to Bi-Directional, you can also specify an Overlap Distance. Enter a value in the Distance field.

Step Over

This allows you to specify the spacing between groove cuts. You can set this to one of the following: % Tool Diameter: Enter the percentage of the groove tool to use. Distance: Enter the step over distance. Number of Steps: Enter the total number of steps to use.

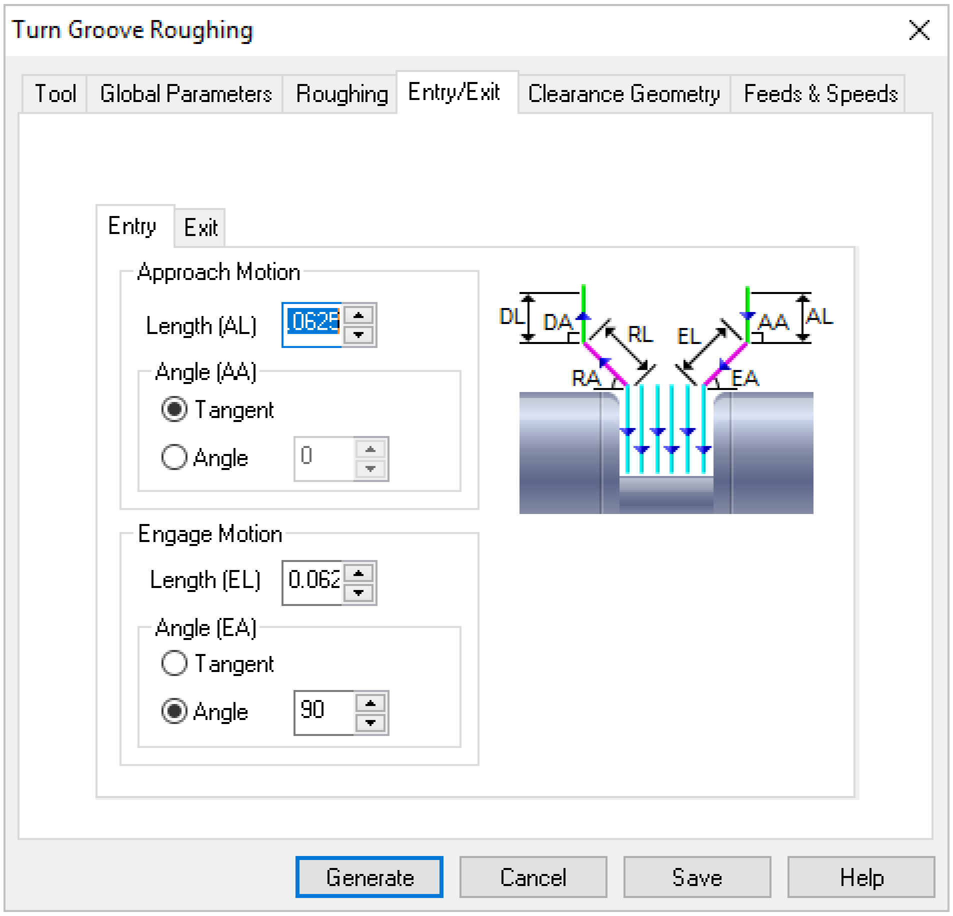
Step Down

This allows you to define the depth between the cut levels for the roughing operation. You can specify the First Plunge Depth and the Next Plunge Depth for subsequent levels. These are set as a specified distance.

8.3.3 Entry/Exit

The following dialog allows you to set Entry/Exit parameters for Groove Roughing operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Groove Roughing

Entry Tab

The Entry tab (shown in the dialog box above) consists of Approach and Engage. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

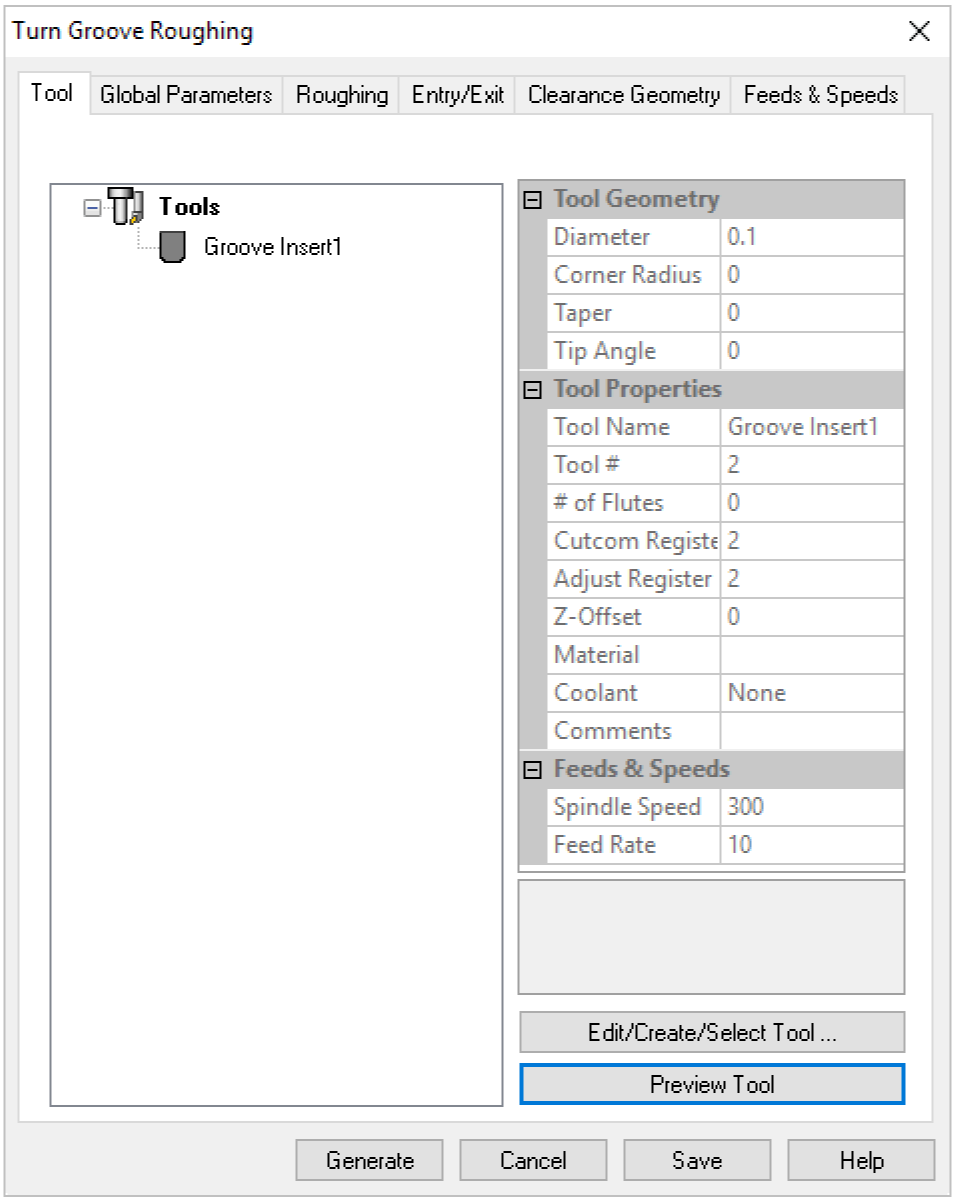
Exit Tab

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

8.3.4 Tool

The following dialog allows you to select the appropriate tool for the Turn Groove Roughing operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Groove Roughing

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

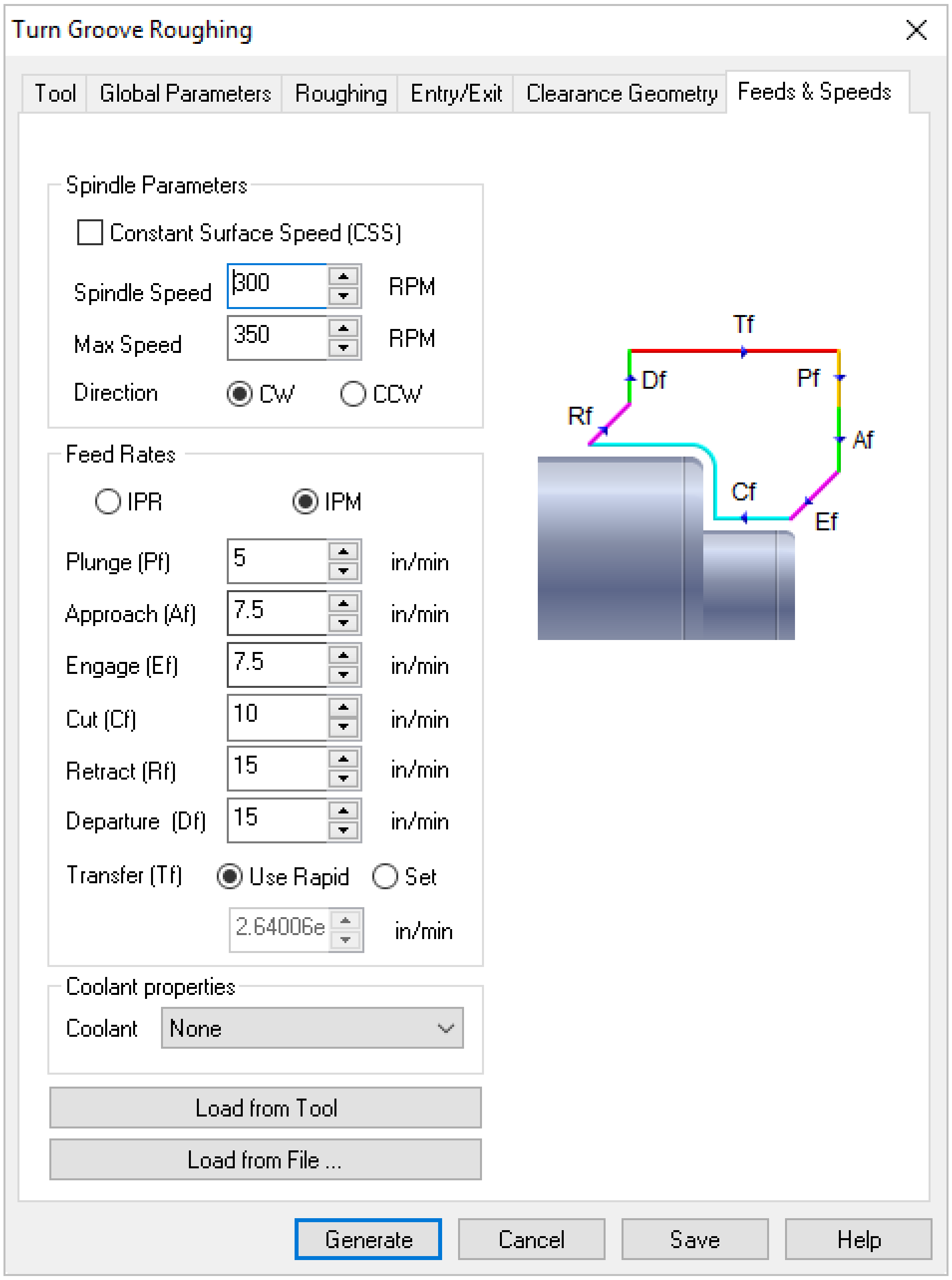
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.3.5 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Groove Roughing operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Groove Roughing

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your

post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed
his is the rotational speed of the spindle expressed in RPM.

Surface Speed
Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed
The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction
This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

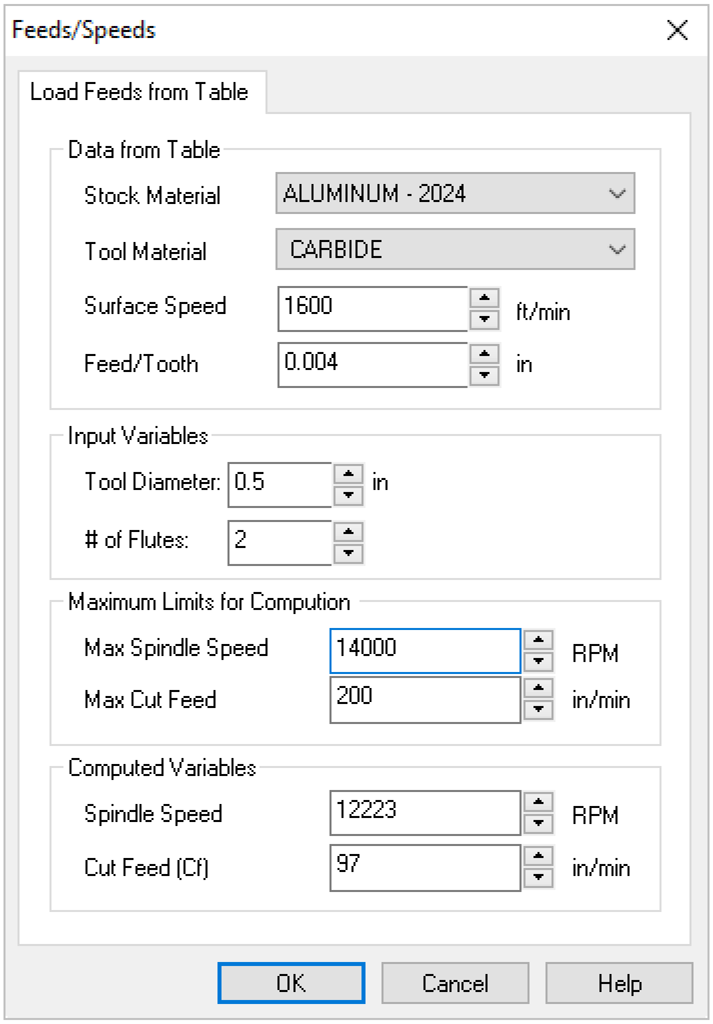
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

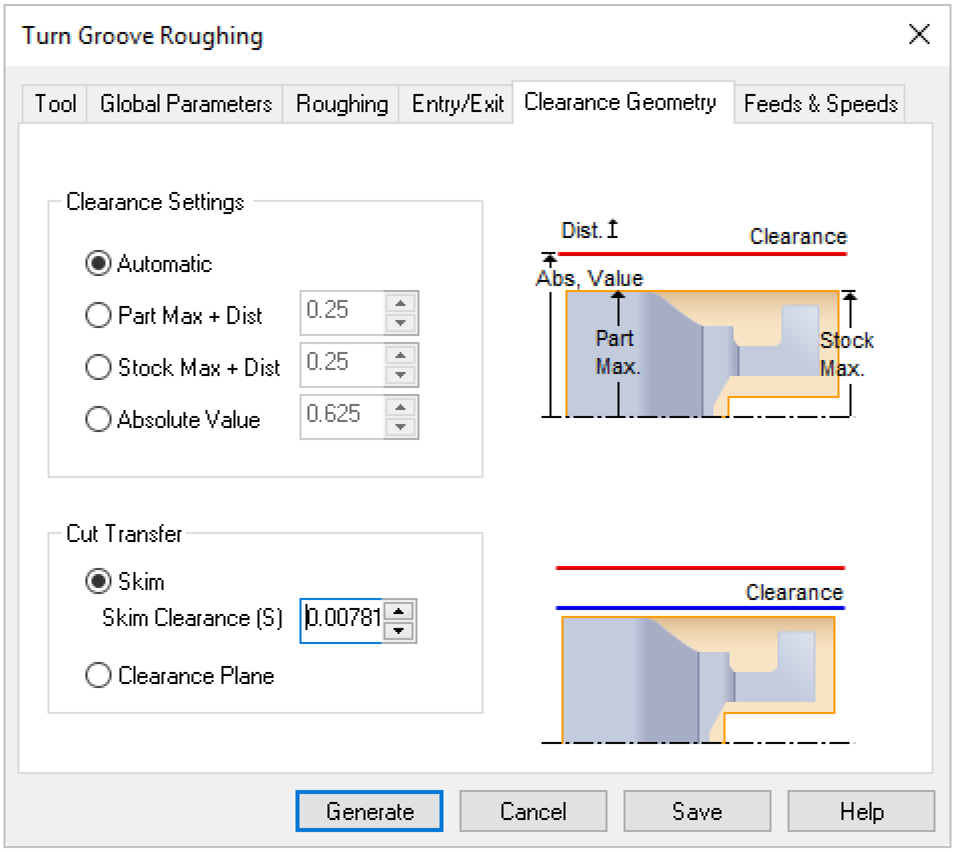
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.3.6 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Groove Roughing operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Groove Roughing

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.
>
> For Hole Machining operations, the clearance plane is normal to the Z axis.

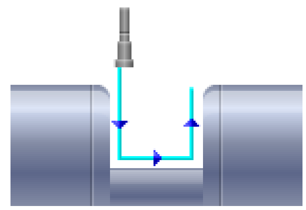
Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

8.4 Groove Finishing

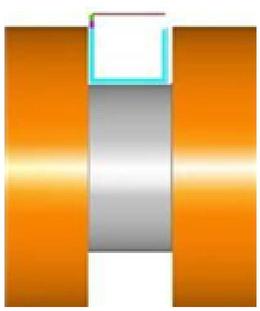
This operation is used to finish the grooves. This operation is performed after the Groove Roughing operation. Groove Finishing can be of 3 types: OD, ID, and Front Facing.

Turning Operations

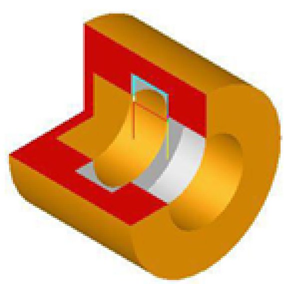


Groove Finishing

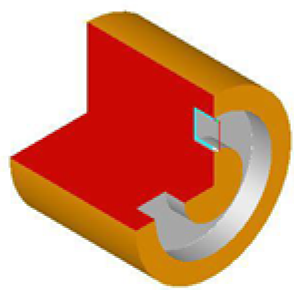
Turn Groove Finishing Examples



OD Groove



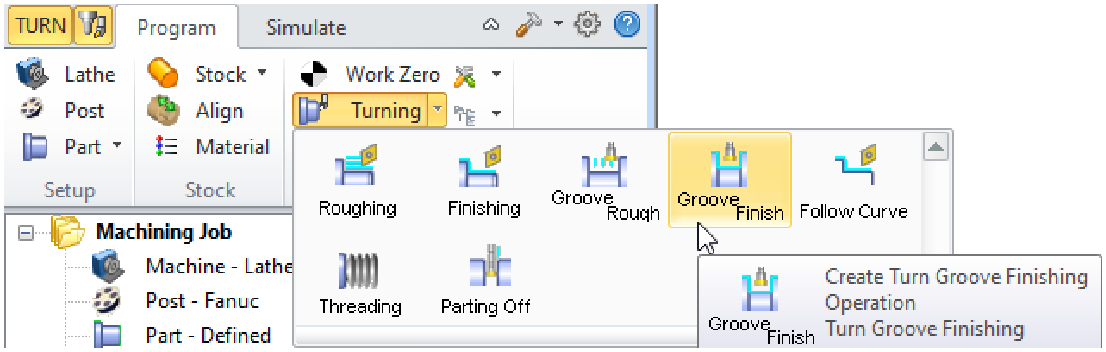
ID Groove



Face Groove

Turn Groove Finishing Menu Item

The Groove Finishing toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Groove Finishing operation.

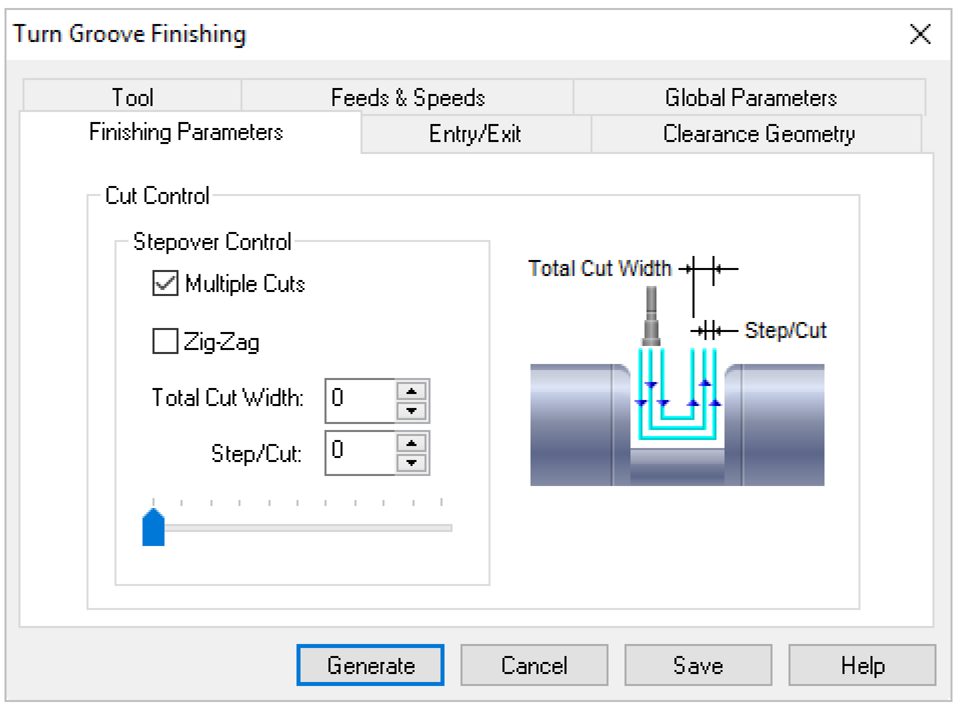


Turn Groove Finishing Menu Item

Dialog Box: Turn Groove Finishing

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has six tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

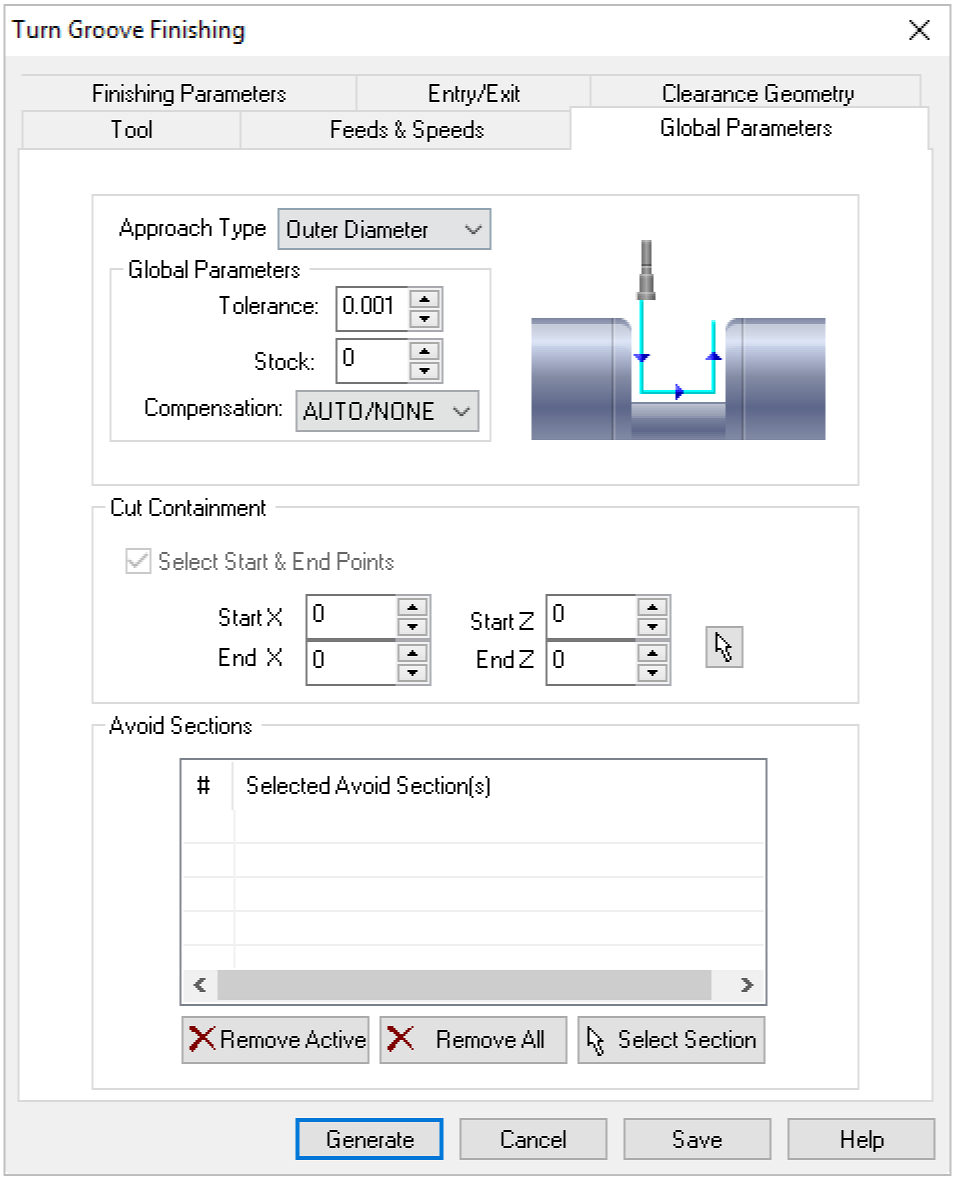


Dialog Box: Turn Groove Finishing

8.4.1 Global Parameters

The following dialog allows you to set Global Parameters for Groove Finishing operations. You can set the Approach Type, Global Parameters and Cut Containment via this property page. For grooving operations, the use needs to specify the start and end points of the groove. This is specified under the Cut Containment.

Dialog Box: Global Parameters tab



Dialog Box: Global Parameters tab, Turn Groove Finishing

Approach Type

Approach Type

Allows user to choose between Outer Diameter (OD), Front Facing and Inner Diameter (ID). The toolpaths are generated for the selected approach types.

In rouging and finishing operations, for tools with OD orientation, the approach type can be set to Outer Diameter or Front Facing. For tools with ID orientation, the approach type is automatically set to Inner Diameter.

Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Stock (Roughing Operations Only)

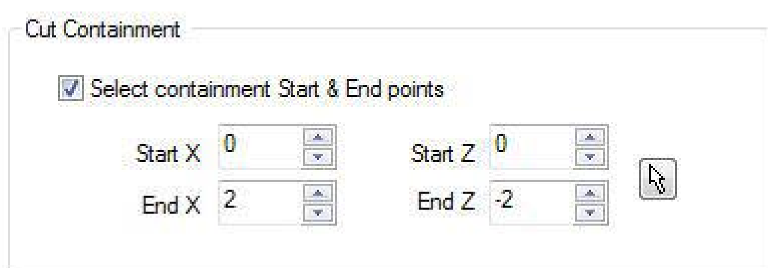
This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Compensation

This stands for cutter compensation. You can turn this on by selecting from the drop down menu. The cutter compensation direction, Left or Right, is determined by the Cut Direction (Climb or Conventional). Refer to the following section for additional information - Cutter Compensation

Cut Containment

This allows you to select an area to contain the toolpath. This is useful in cases where a section of the part needs to be machined. This is done by selecting the check box for Select containment Start & End points.



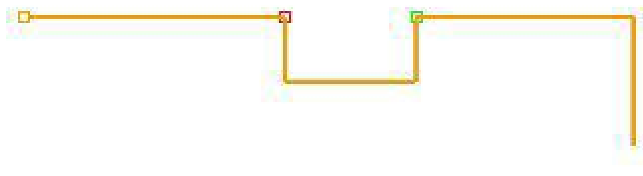
Cut Containment

You can either input the X and Z coordinate values that represent 2 corners of a containment rectangle or use the pick option to graphically select 2 corners of a rectangle for containment.

You can use the object snap tools from the status bar to snap to points on the part geometry.

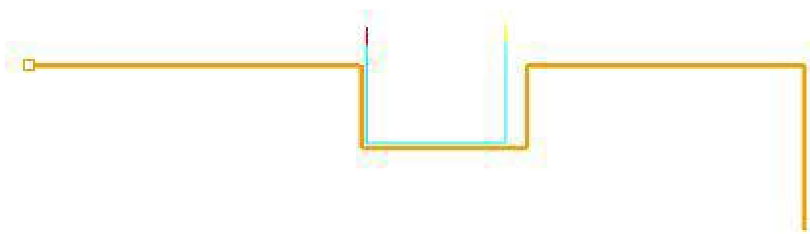
Cut Containment Examples for Turn Groove Finishing

The start and end points are displayed on the part geometry. The start point is represented in **Green** color and end point in **Red**.



Cut Containment Examples for Turn Groove Finishing

The groove finishing toolpath is generated for the specified start and end points.



Cut Containment Examples for Turn Groove Finishing

The start and end points are also used to determine the Cut Direction in Groove Finishing operation.

Avoid Sections

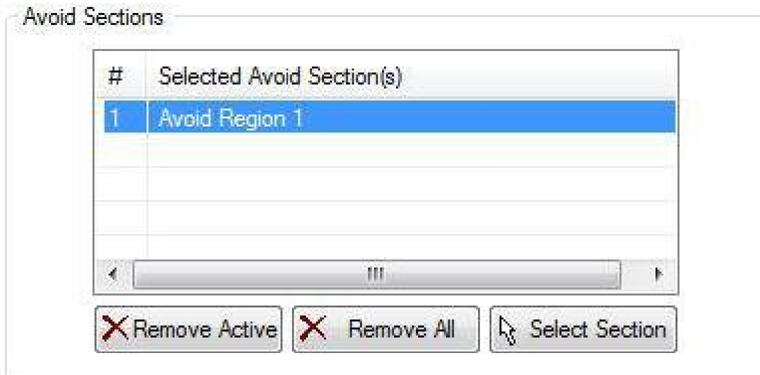
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

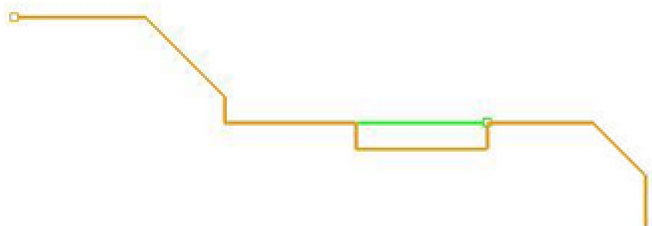
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

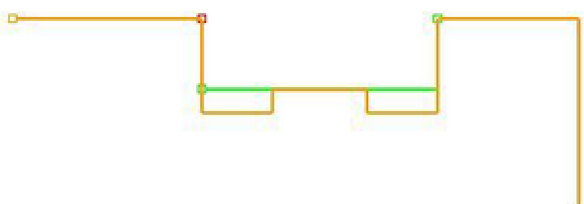


Avoid Sections, Global Parameters tab of Turn Operations dialog box

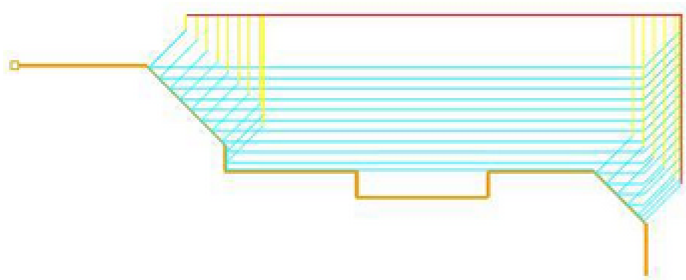
Examples for various Turn Operations



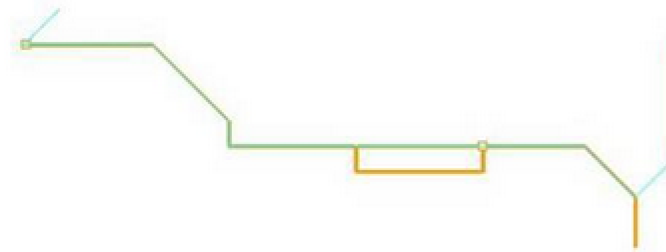
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section

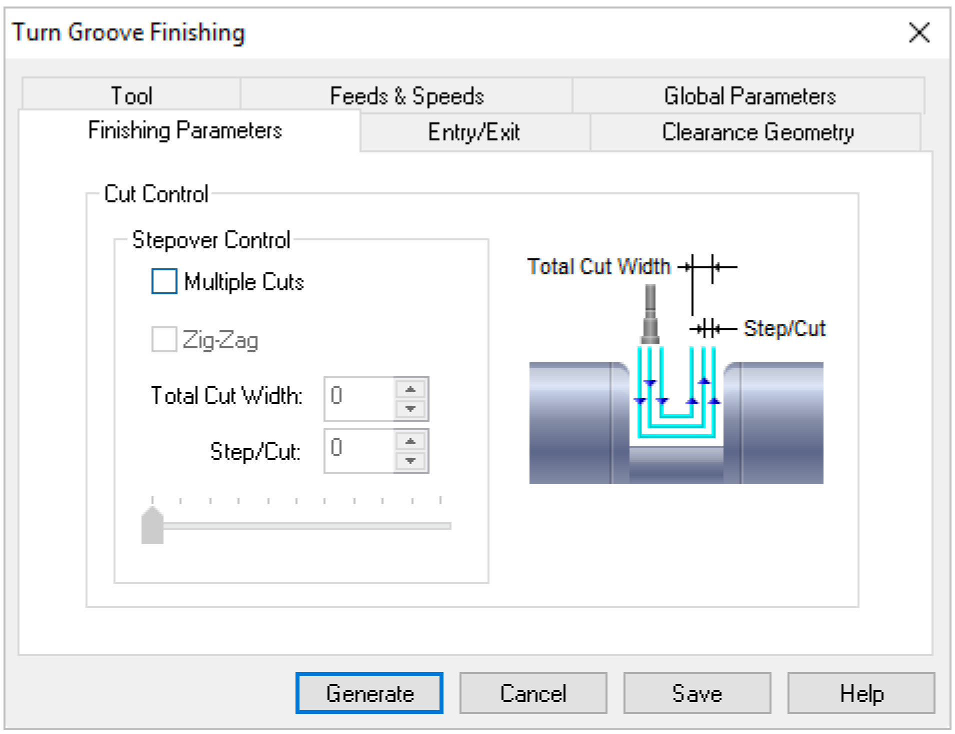


Finishing toolpath with Avoid Section

8.4.2 Finishing Parameters

In this tab, parameters like the Cut Direction, Step Over and Step Down can be specified for Turn Groove Roughing operations.

Dialog Box: Finishing Parameters tab



Dialog Box: Finishing Parameters tab, Turn Groove Finishing

Cut Control

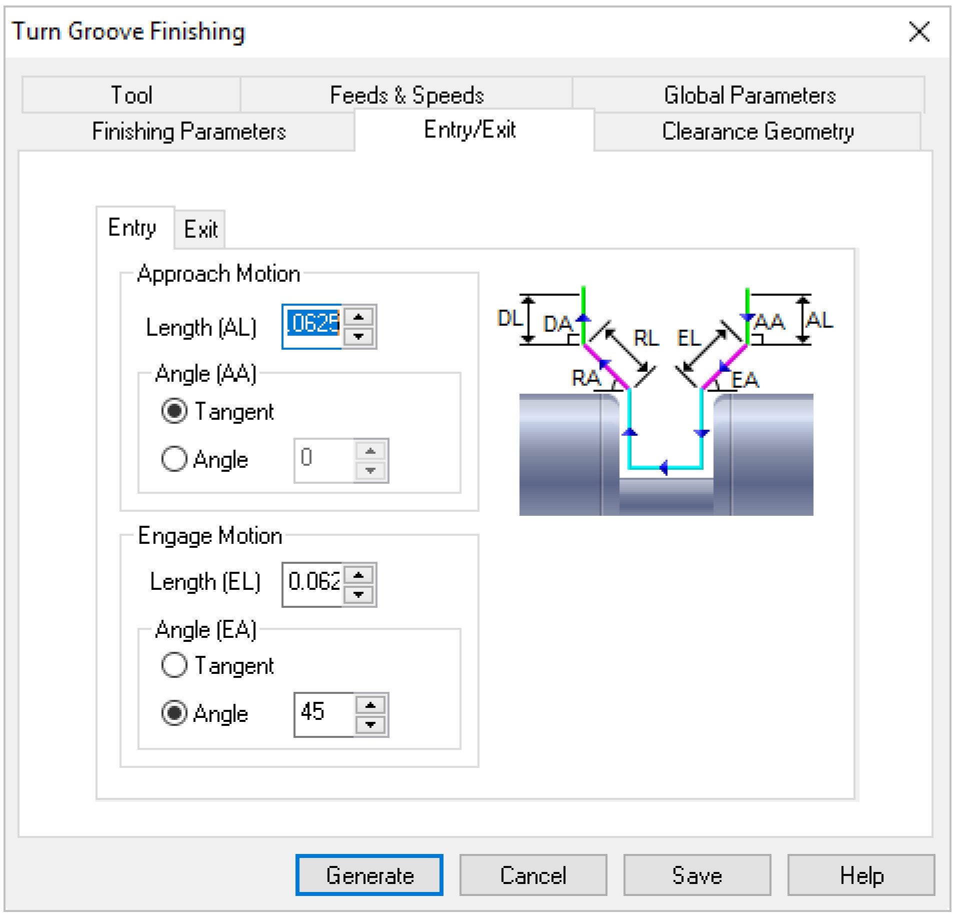
Groove finishing cut can be specified in terms of the total passes of the cutter over the stock.

Stepover Control - Allows specifying multiple passes for groove finishing. Selecting multiple cuts allows user to set the total cut width and step per cut. The direction of cut can be either unidirectional or Bi-directional (Zig-Zag) with Multiple Cuts supported.

8.4.3 Entry/Exit

The following dialog allows you to set Entry/Exit parameters for Groove Finishing operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Groove Finishing

Entry Tab

The Entry tab (shown in the dialog box above) consists of Approach and Engage. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

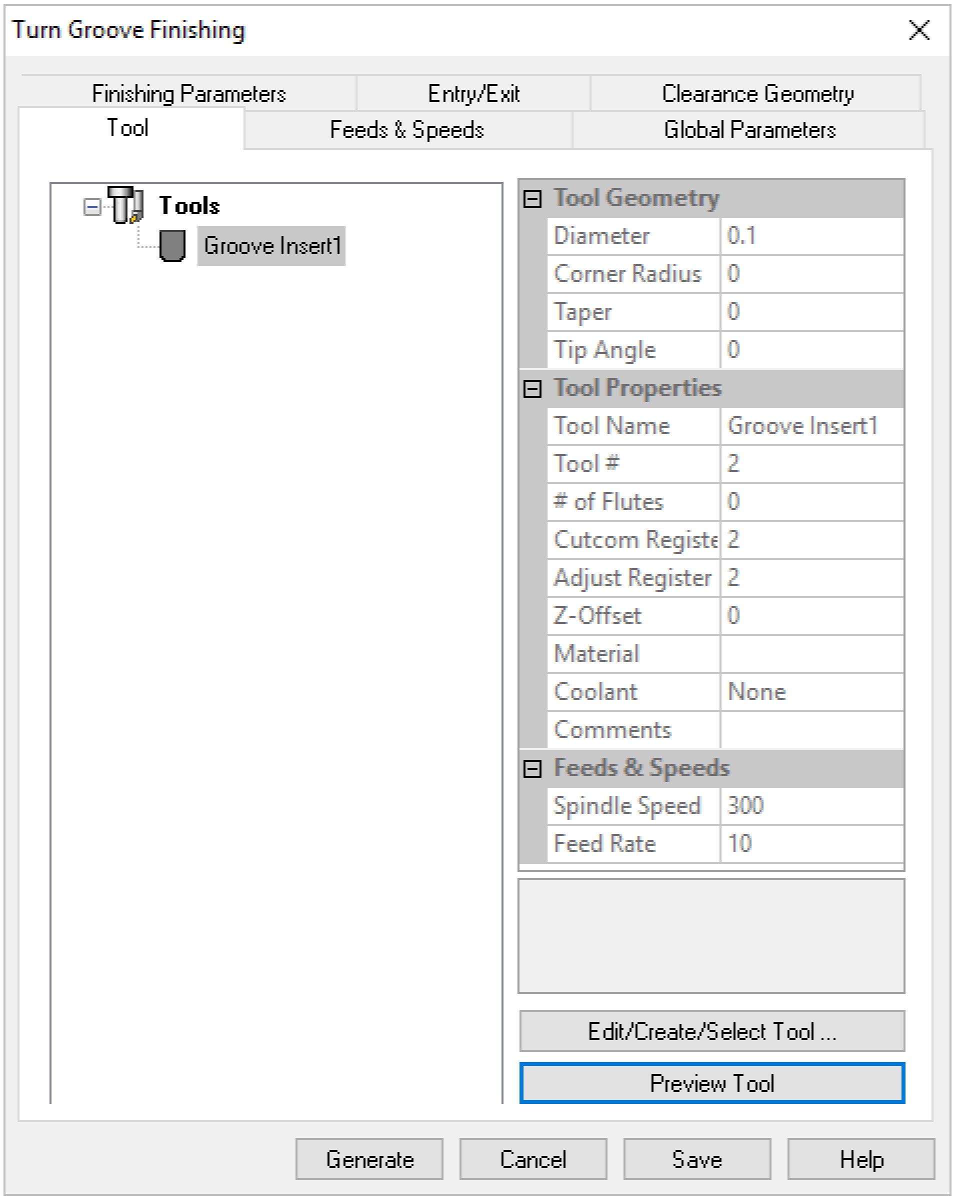
Exit Tab

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

8.4.4 Tool

The following dialog allows you to select the appropriate Groove insert tool for the Turn Groove Roughing operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Groove Finishing

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

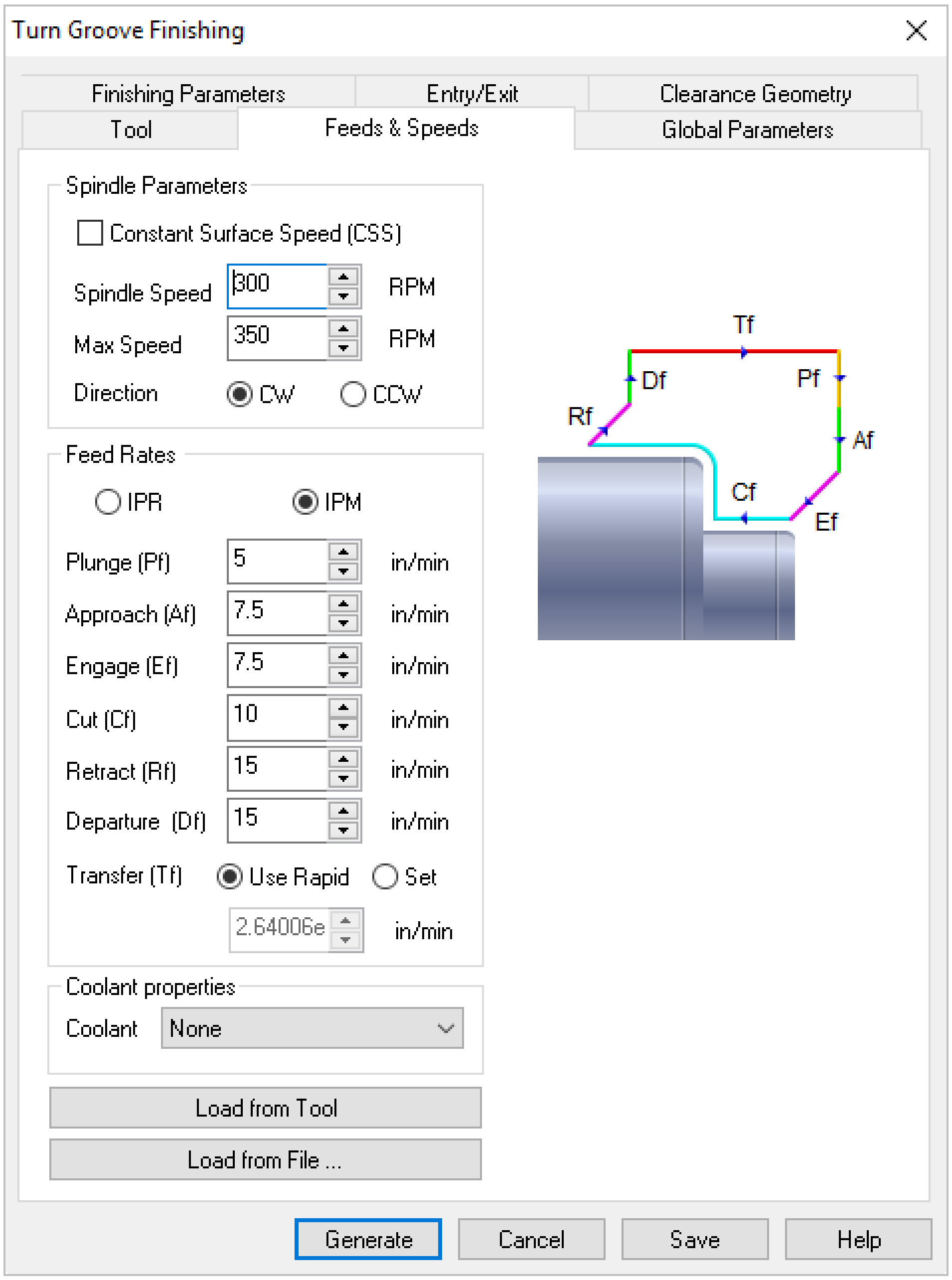
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.4.5 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Groove Finishing operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Groove Finishing

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed

his is the rotational speed of the spindle expressed in RPM.

Surface Speed

Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed

The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction

This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)

This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)

This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)

This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)

This is the feedrate used when the tool is cutting material

Retract (Rf)

The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)

The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

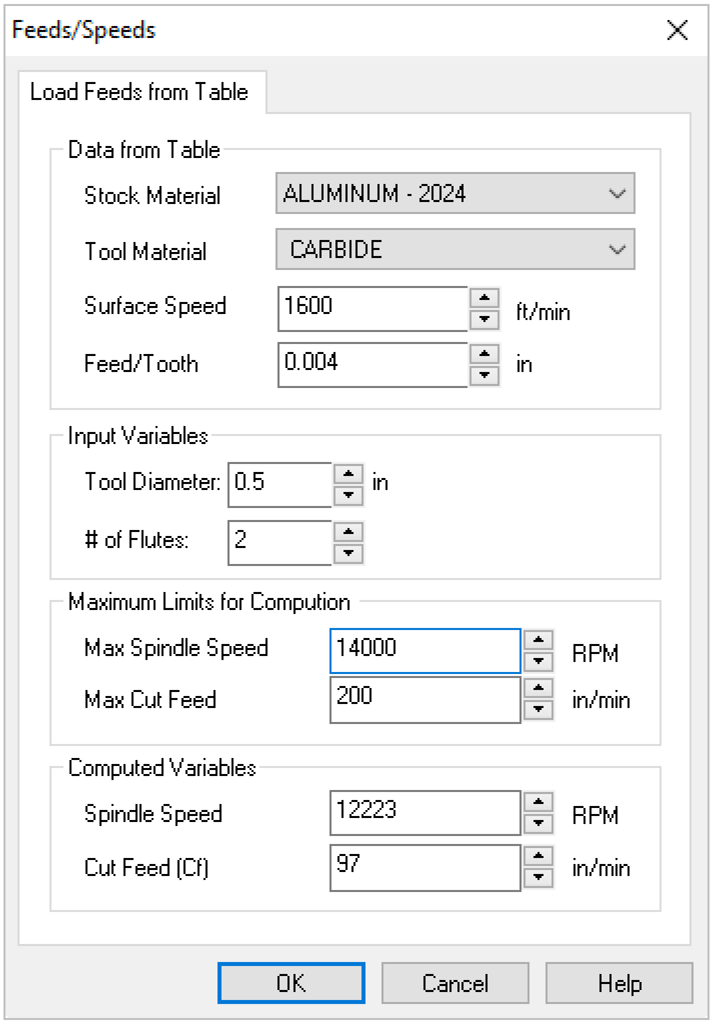
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

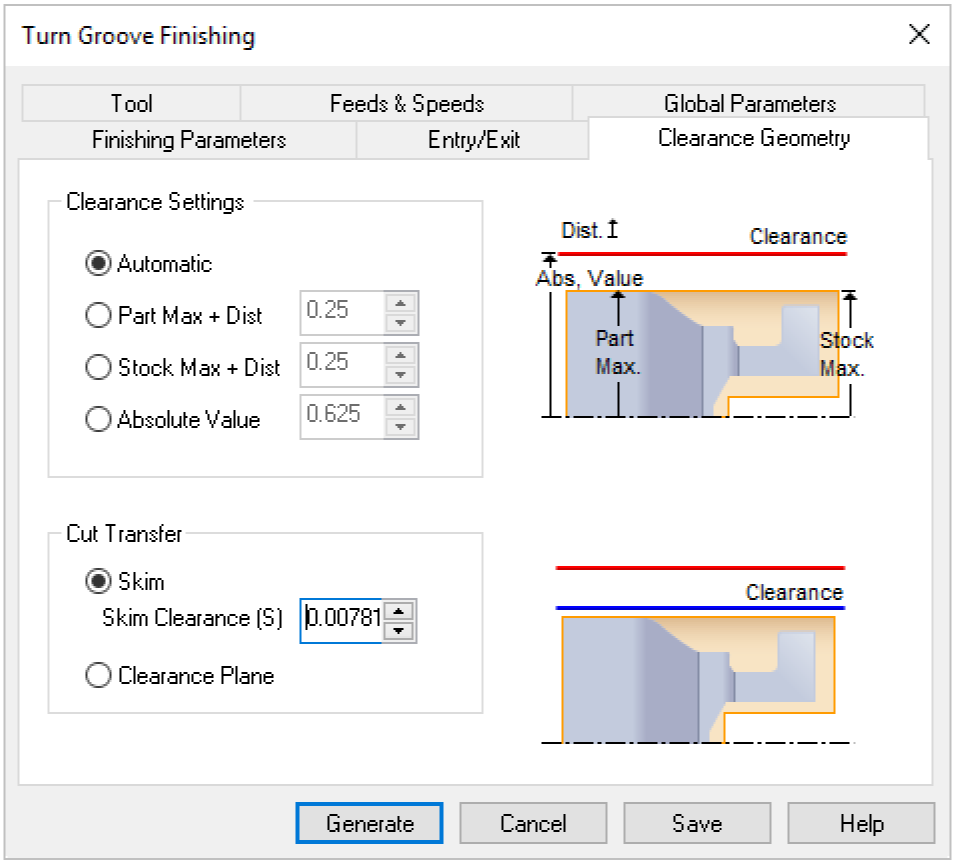
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.4.6 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Groove Finishing operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Groove Finishing

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

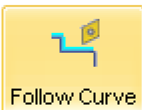
Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.
>
> For Hole Machining operations, the clearance plane is normal to the Z axis.

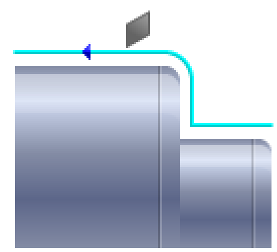
Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

8.5 Follow Curve

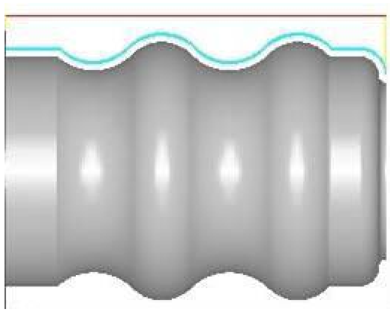


This operation is performed after the roughing operation. This is similar to finishing operation where the toolpath follows the selected curve to obtain better surface finish and is characterized by smaller depth of cut to obtain tighter tolerances and better surface finish. This method is similar to engraving in milling where the tool nose radius compensation is not applied to the toolpath. Follow curve can be used in OD, ID, and Front Facing operations.



Follow Curve

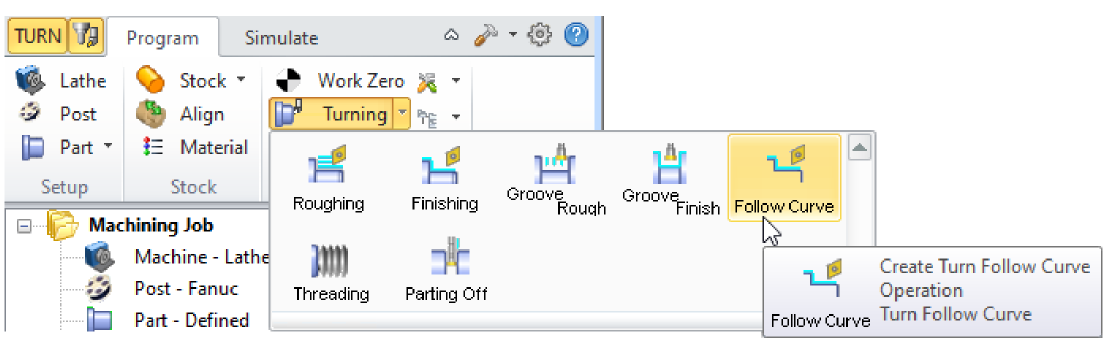
Turn Follow Curve Example



Follow Curve Operation Toolpath Example

Turn Follow Curve Menu Item

The Follow Curve toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Follow Curve operation.

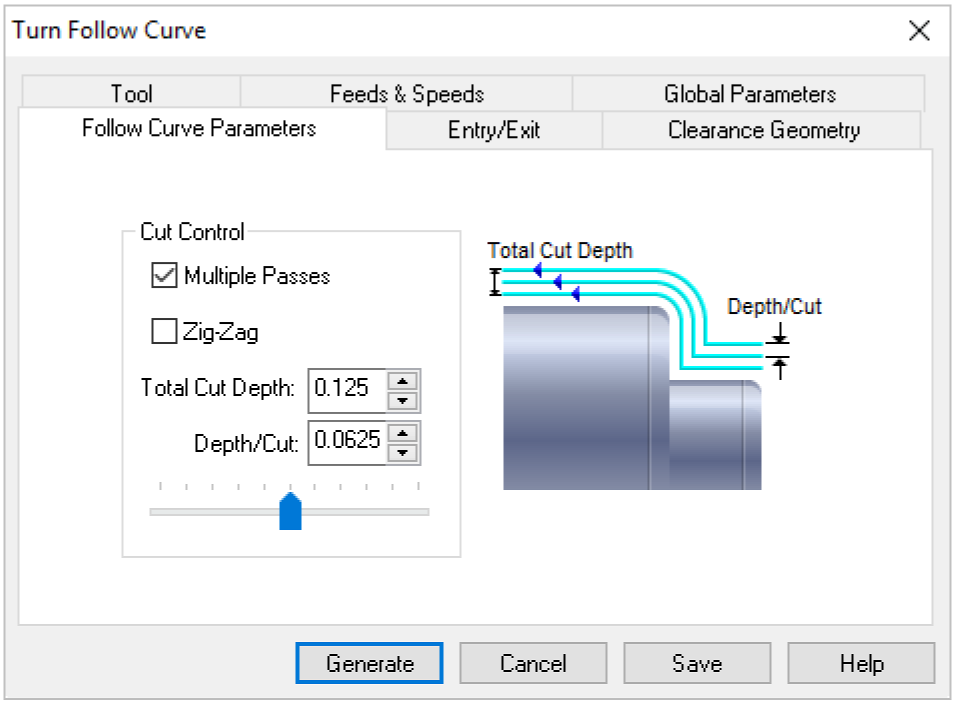


Flow Curve Menu Item

Dialog Box: Turn Follow Curve

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has six tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

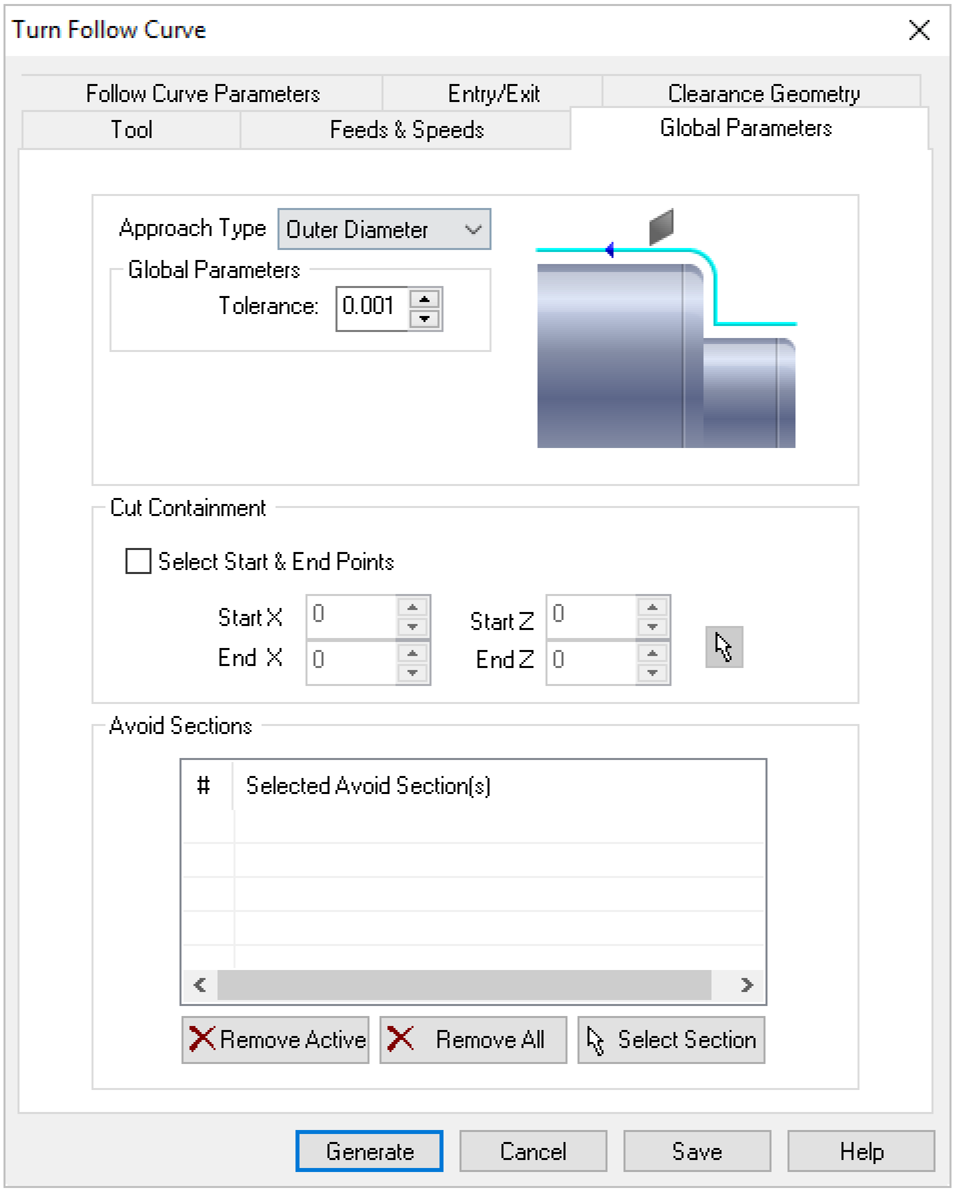


Dialog Box: Turn Follow Curve

8.5.1 Global Parameters

The following dialog allows you to set Global Parameters for Turn Follow Curve operations. You can set the Approach Type, Global Parameters and Cut Containment via this property page.

Dialog Box: Global Parameters tab



Dialog Box: Global Parameters tab, Turn Follow Curve

Approach Type

Approach Type

Allows user to choose between Outer Diameter (OD), Front Facing and Inner Diameter (ID). The toolpaths are generated for the selected approach types.

In rouging and finishing operations, for tools with OD orientation, the approach type can be set to Outer Diameter or Front Facing. For tools with ID orientation, the approach type is automatically set to Inner Diameter.

Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

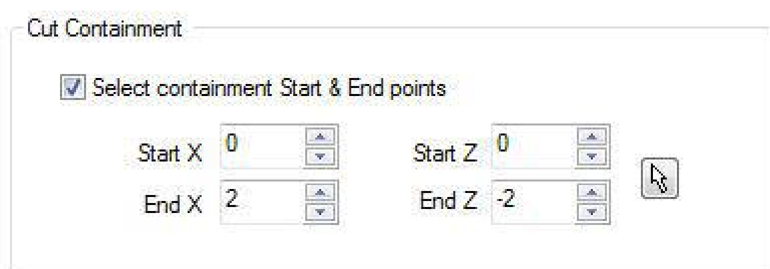
This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Stock (Roughing Operations Only)

This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Cut Containment

This allows you to select an area to contain the toolpath. This is useful in cases where a section of the part needs to be machined. This is done by selecting the check box for Select containment Start & End points.



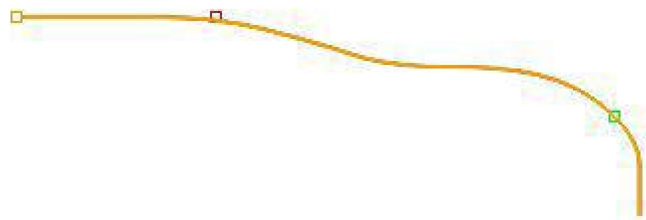
Cut Containment

You can either input the X and Z coordinate values that represent 2 corners of a containment rectangle or use the pick option to graphically select 2 corners of a rectangle for containment.

> You can use the object snap tools from the status bar to snap to points on the part geometry.

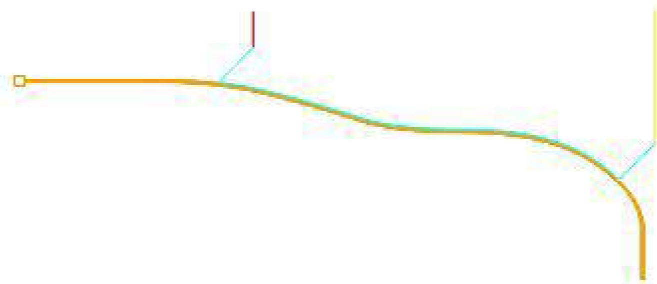
Cut Containment Examples for Turn Follow Curve

The start and end points are displayed on the part geometry. The start point is represented in **Green** color and end point in **Red**.



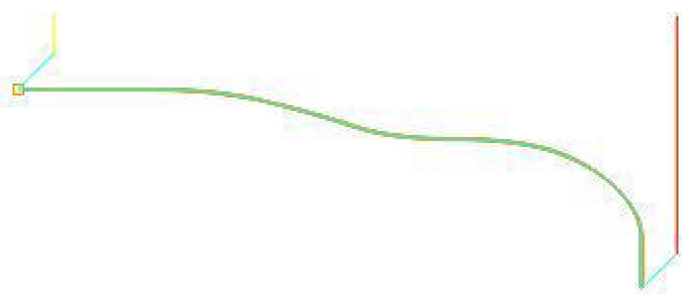
Cut Containment Examples for Turn Follow Curve

The finishing toolpath is contained between start and end points and cut direction of the toolpath is from the start to end point. The selection of start and end points can also be used to determine the cut direction.



Cut Containment Examples for Turn Follow Curve

If a containment is not specified, the follow curve toolpath is generated for the turn part geometry based on the approach type and the part geometry.



Cut Containment Examples for Turn Follow Curve

Avoid Sections

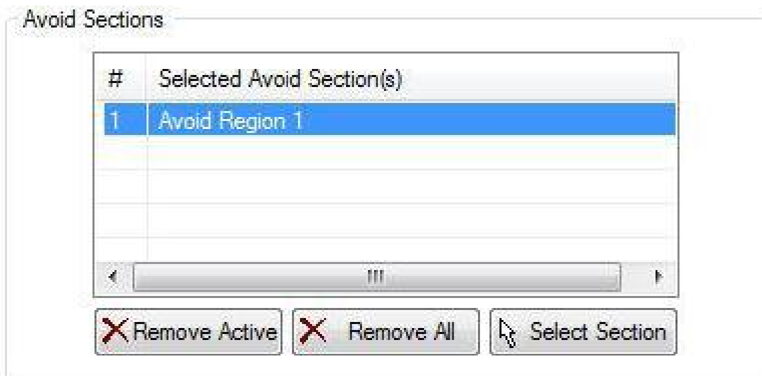
This allows user to select areas to be excluded from the turn part geometry for toolpath computation. This is done by selecting 2 points on the part geometry. A line is inserted between the 2 selected points as avoid region and this now becomes part of your turn part geometry. One or more avoid areas can be selected.

Defining Avoid Sections

To select an area to avoid:

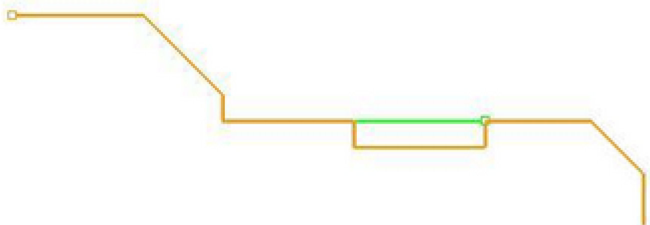
From the Global Parameters tab in the Turn Operations dialog box, click Select Section under Avoid Sections and pick 2 points on the part geometry. The selection is now displayed under avoid selection.

Selecting an Avoid Region from the list highlights it on the part geometry.

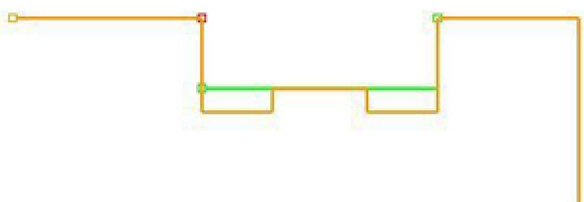


Avoid Sections, Global Parameters tab of Turn Operations dialog box

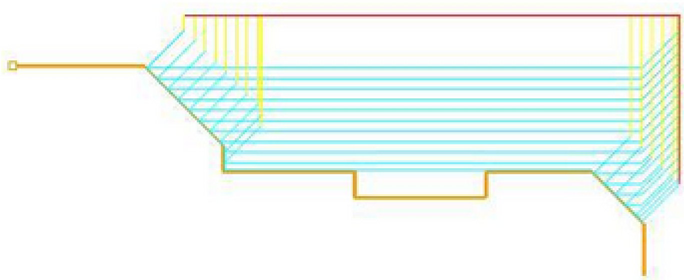
Examples for various Turn Operations



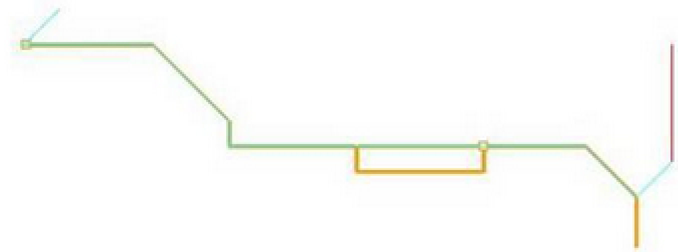
Turn Roughing and Finishing Avoid Section



Groove Roughing and Finishing Avoid Section



Roughing toolpath with Avoid Section

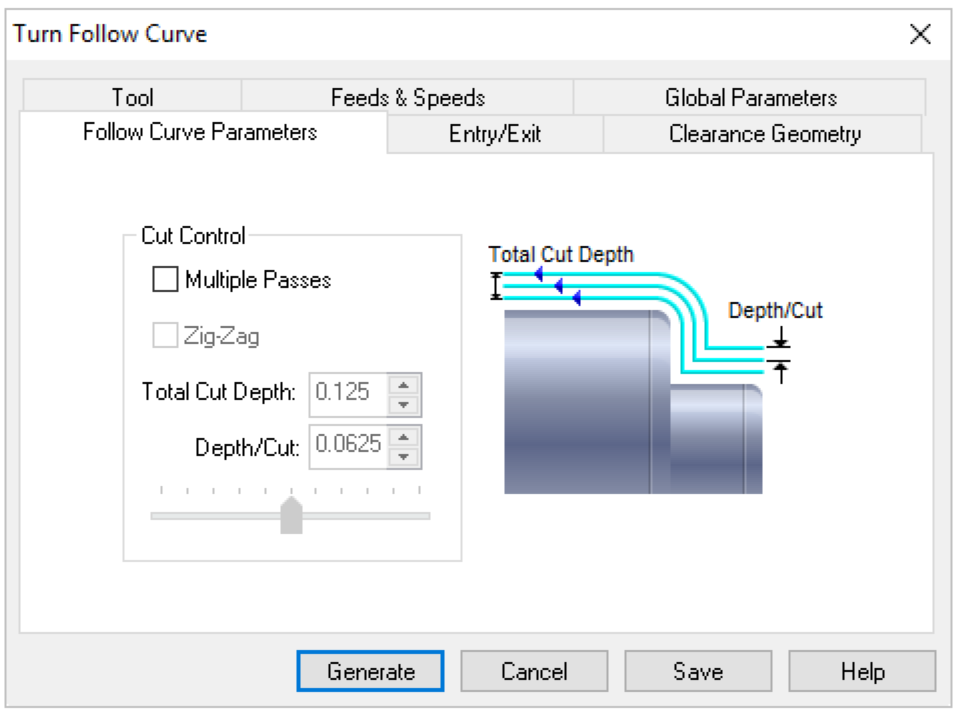


Finishing toolpath with Avoid Section

8.5.2 Follow Curve Parameters

In this tab, Cut Control parameters can be specified for Turn Follow Curve operations.

Dialog Box: Follow Curve Parameters tab



Dialog Box: Follow Curve Parameters tab, Turn Follow Curve

Cut Control

Final finishing cut can be specified in terms of the total passes of the cutter over the stock.

Cut Control
Allows specifying multiple passes for follow curve operation.

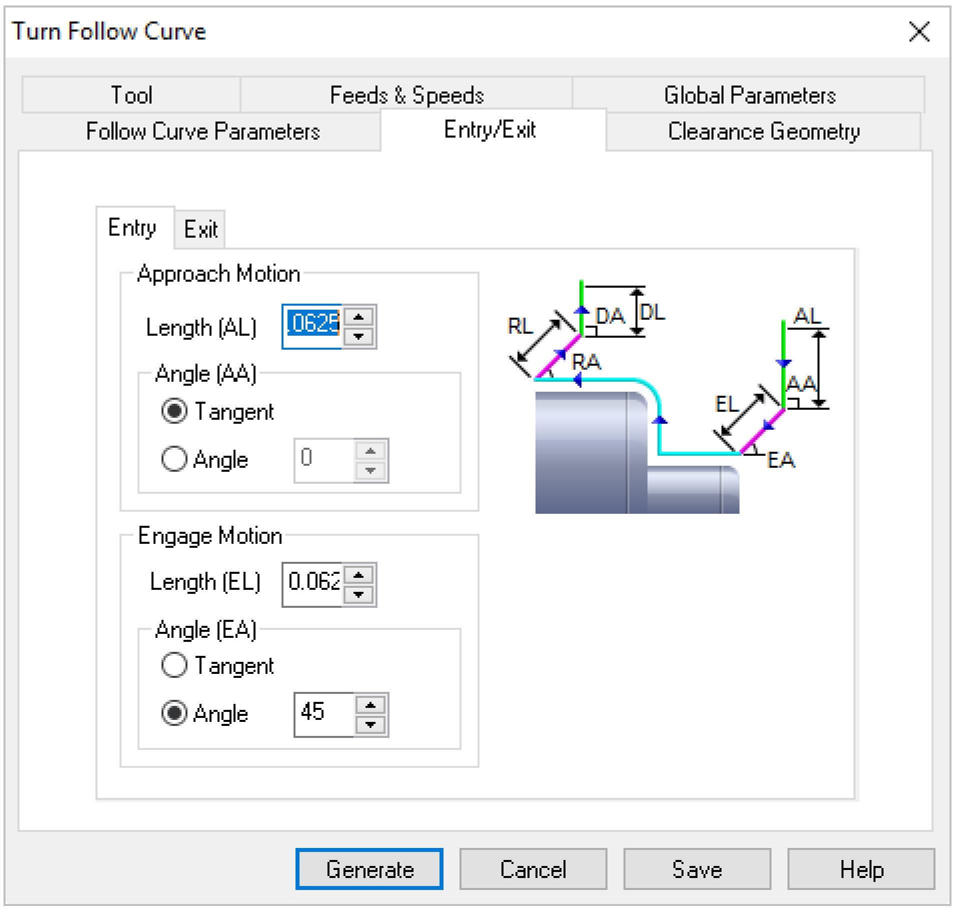
Multiple Passes

Allows user to set the Total Cut Depth and Depth/Cut. The direction of cut can be either unidirectional or Bi-directional (Zig-Zag)

8.5.3 Entry/Exit

The following dialog allows you to set Entry/Exit parameters for Turn Follow Curve operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Turn Follow Curve

Entry Tab

Approach Motion

Entry consists of an Approach and an Engage move. Approach consists of a length and an angle. Enter a value for Length (AL). Then for Angle (AA) you can select Tangent or Angle. Tangent will approach the part tangent to the part surface. Angle will approach at the angle you specify.

Engage Motion
Entry consists of an Approach and an Engage move. Engage consists of a length and an angle. Enter a value for Length (EL). Then for Angle (EA) you can select Tangent or Angle. Tangent will engage tangent to the part surface. Angle will engage at the angle you specify.

Exit Tab

Retract Motion
Exit consists of a Retract Motion and a Departure Motion. Length (RL) is added beginning at the last cut point.

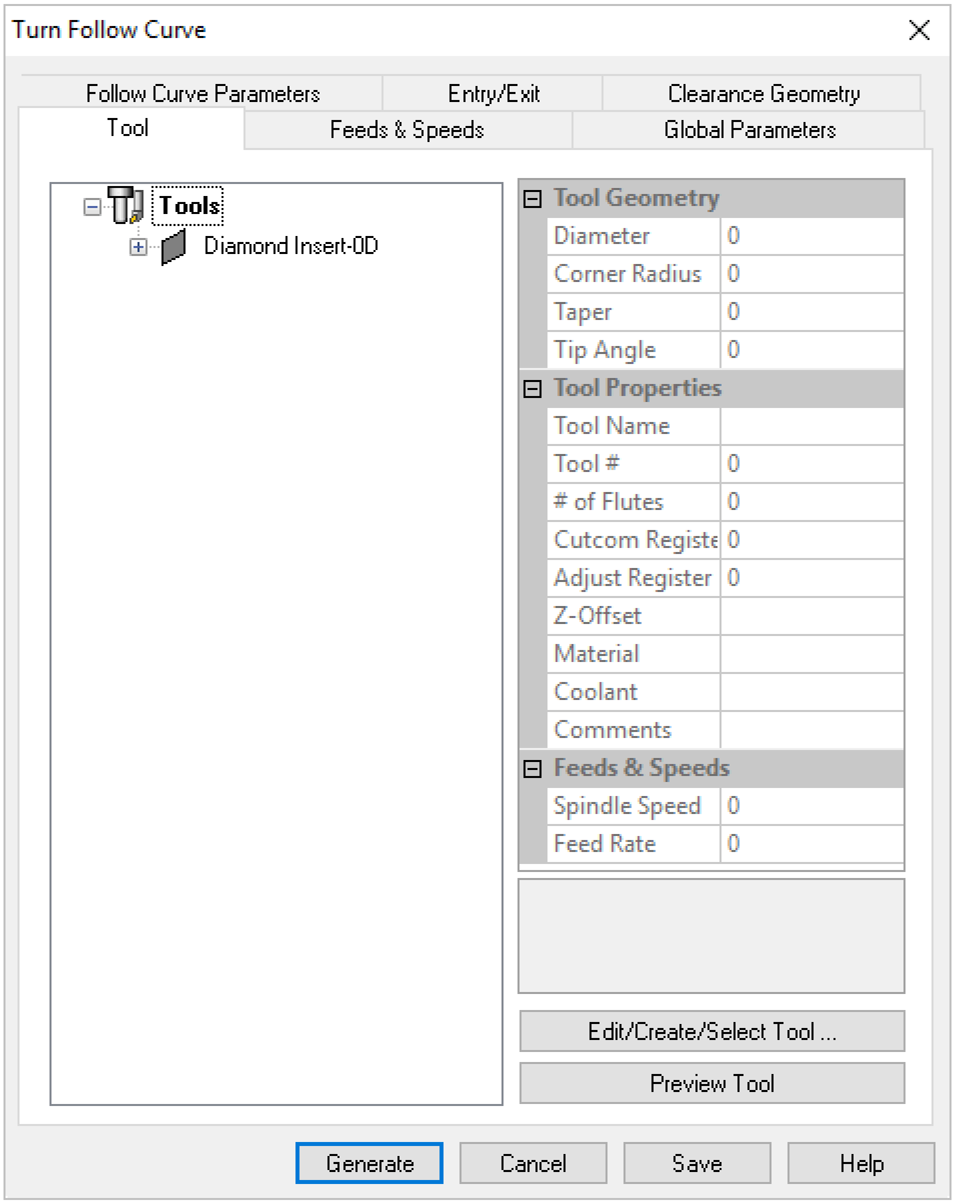
Angle (RA)
This is the angle of the departure from the last cut point. You can select Tangent or Angle.

Depart Motion
Departure consists of a length and an angle. Enter a value for Length (DL). Then for Angle (DA) you can select Tangent or Angle. Tangent will depart tangent to the part surface. Angle will depart at the angle you specify.

8.5.4 Tool

The following dialog allows you to select the appropriate tool for the Turn Follow Curve operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Follow Curve

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

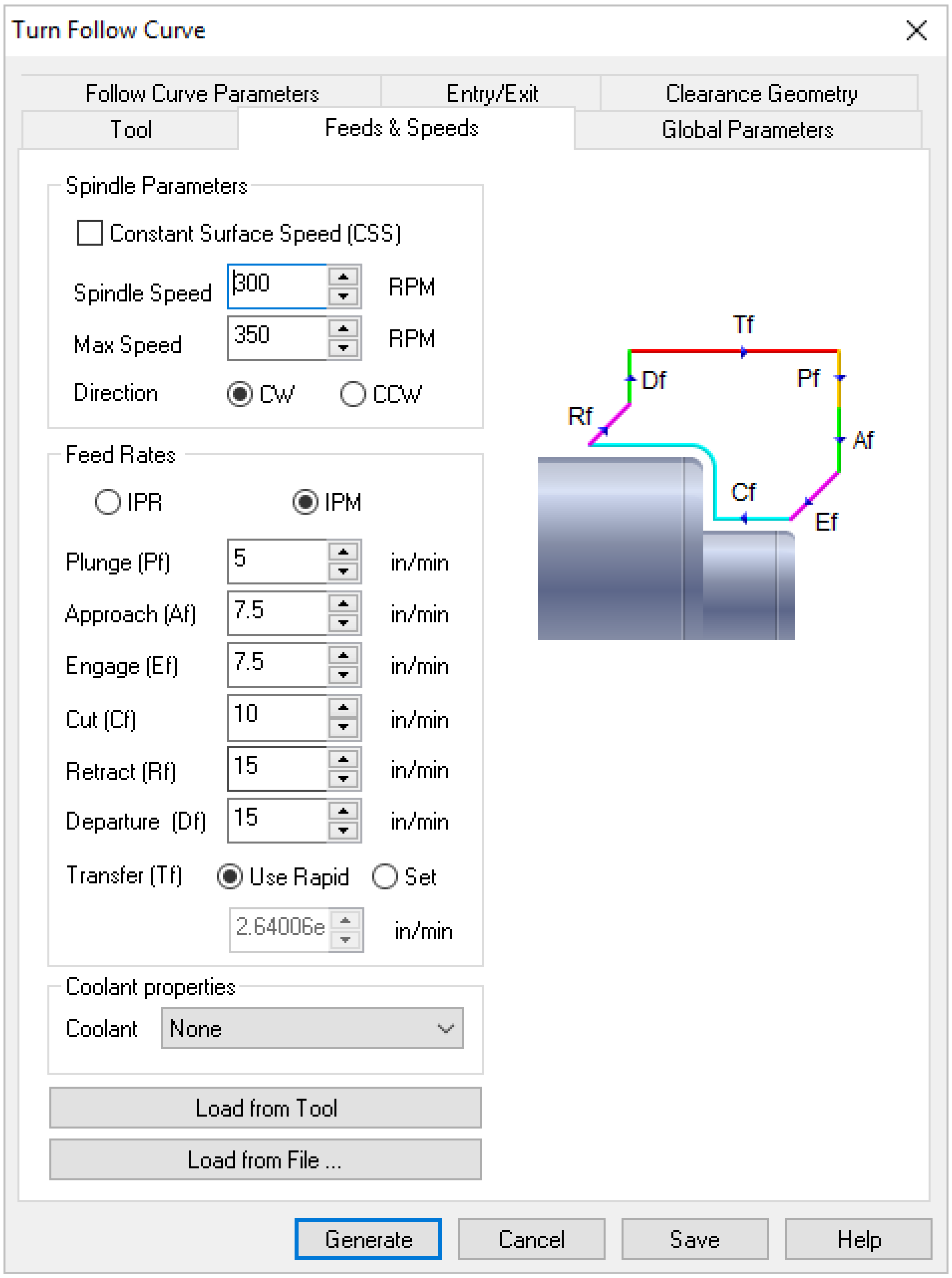
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.5.5 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Follow Curve operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Follow Curve

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed

his is the rotational speed of the spindle expressed in RPM.

Surface Speed

Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed

The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction

This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)

This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)

This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)

This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)

This is the feedrate used when the tool is cutting material

Retract (Rf)

The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)

The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

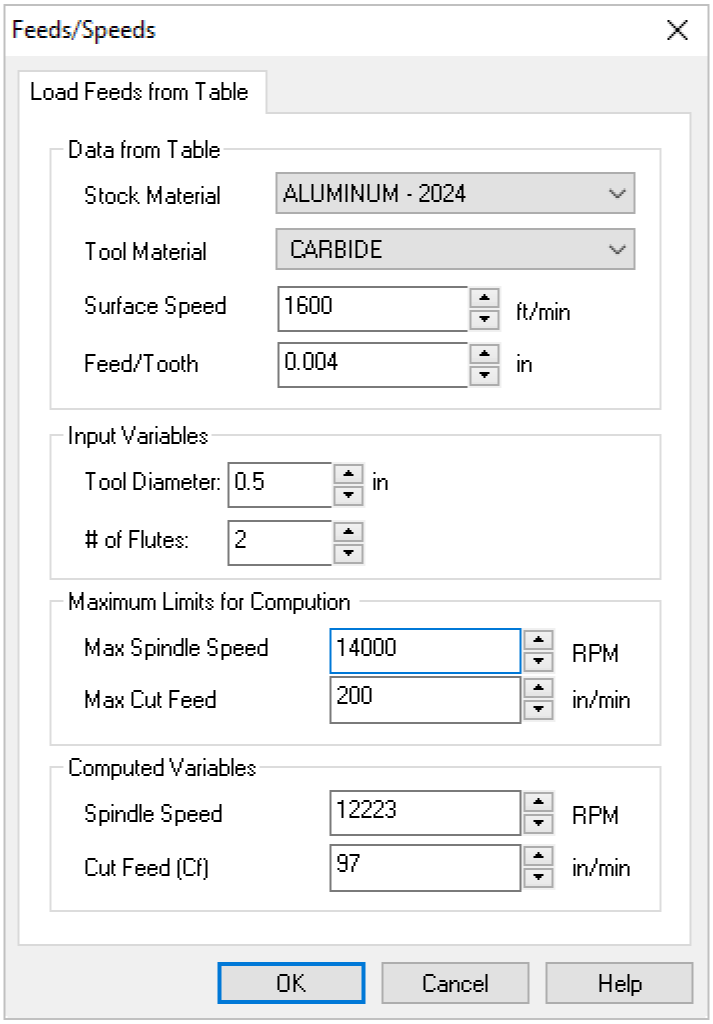
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

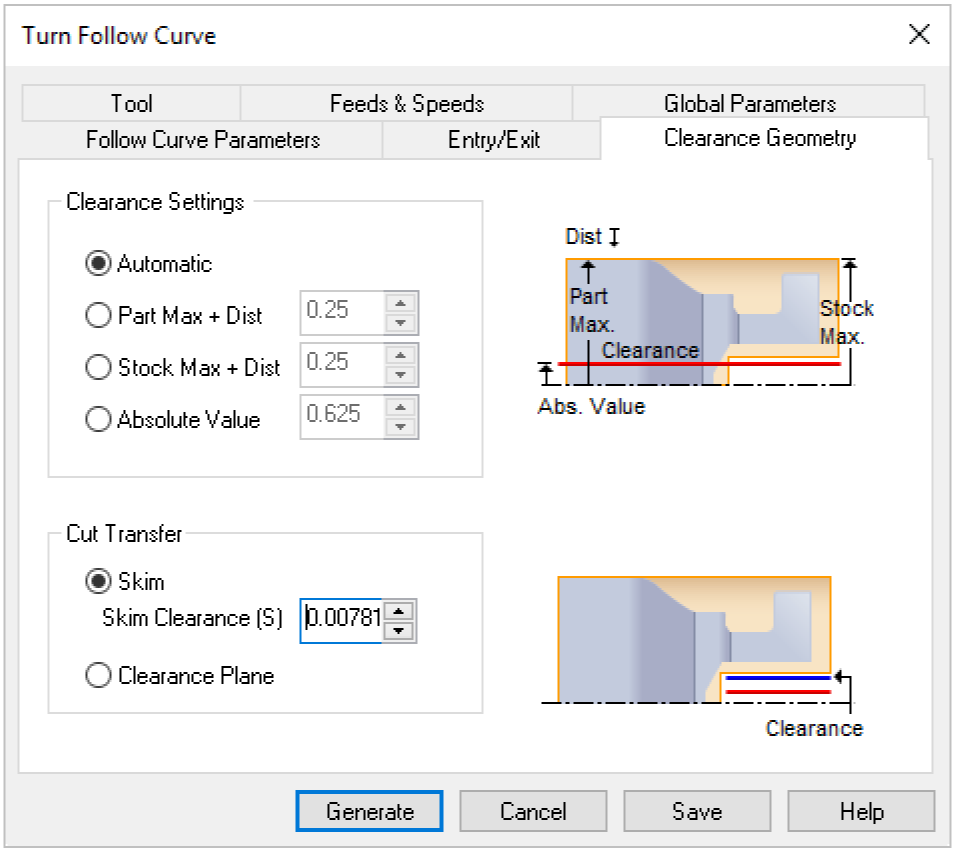
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.5.6 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Follow Curve operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Follow Curve

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

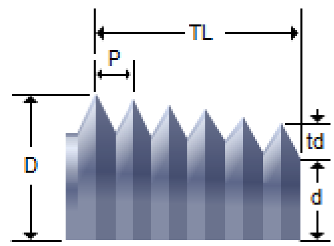
For Hole Machining operations, the clearance plane is normal to the Z axis.

Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

8.6 Threading

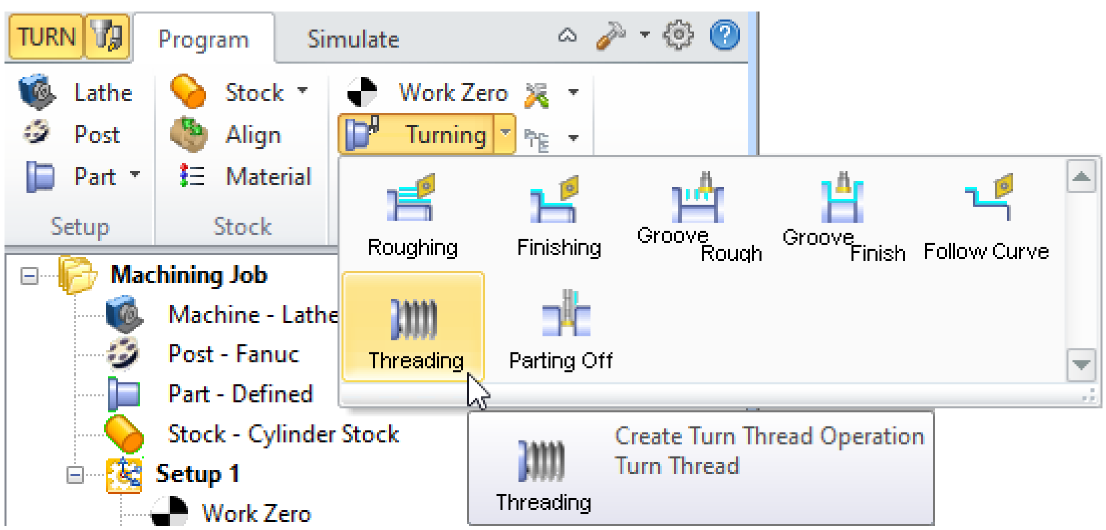
This operation is performed to machine threads on the part. Threads are used as fasteners for assembly purposes. OD and ID threads can be programmed using this method.



Turn Threading

Turn Threading Menu Item

The Threading toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Threading operation.

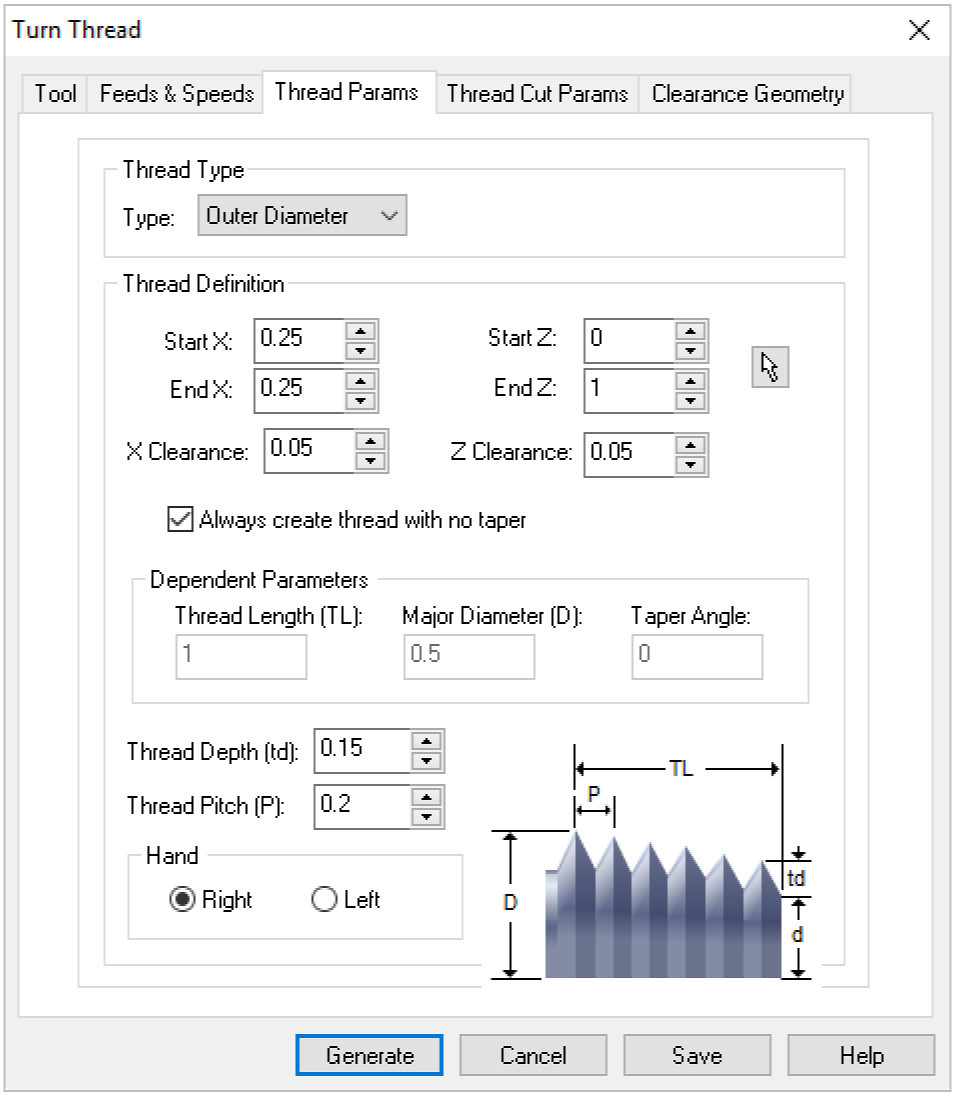


Turn Threading Menu Item

Dialog Box: Turn Thread

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has five tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

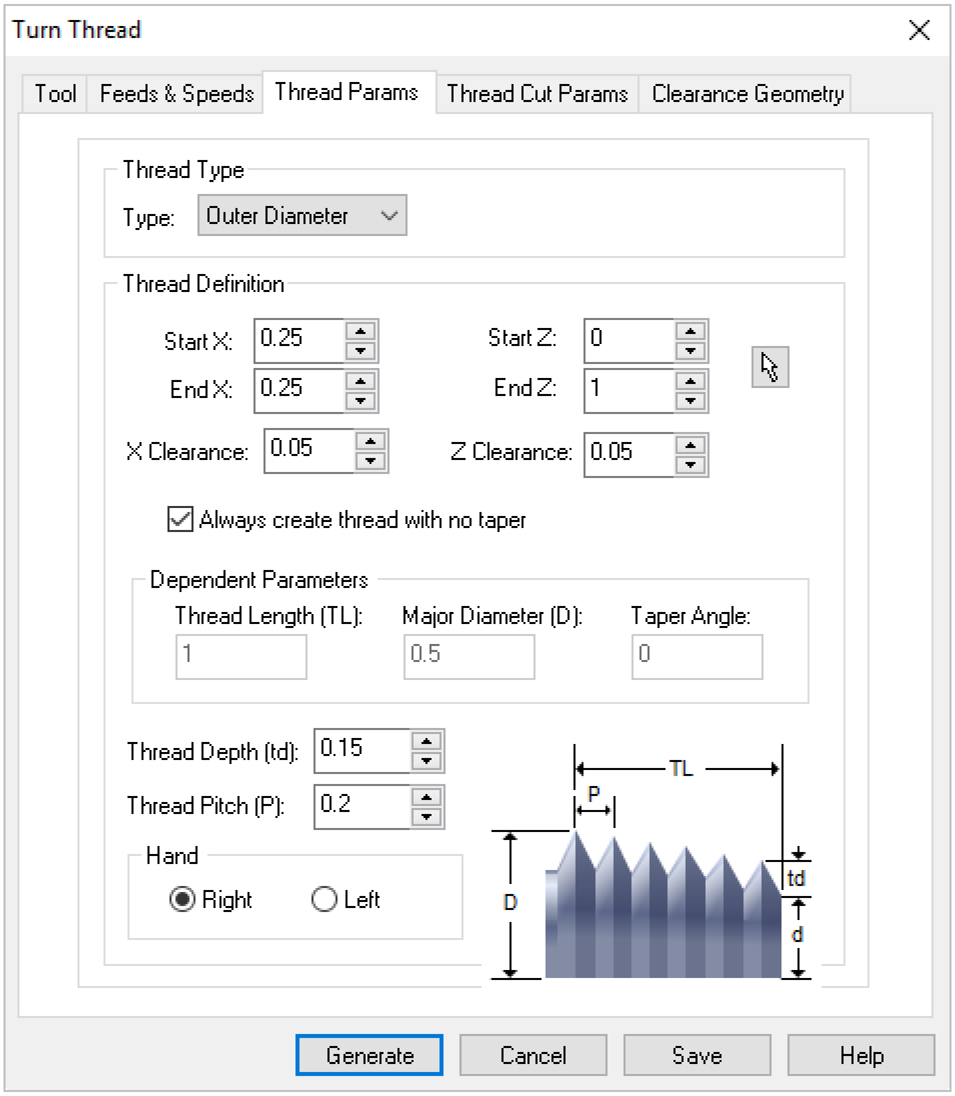


Dialog Box: Thread Parameters tab, Turn Threading

8.6.1 Thread Parameters

The following dialog allows you to set the Thread Parameters for Turn Threading operations. You can set the Thread Type and Thread Definition via this property page.

Dialog Box: Thread Parameters tab



Dialog Box: Thread Parameters tab, Turn Threading

Thread Type

This parameter determines the threading operation type for Inner Diameter or Outer Diameter (i.e., internal or external threads).

Thread Definition

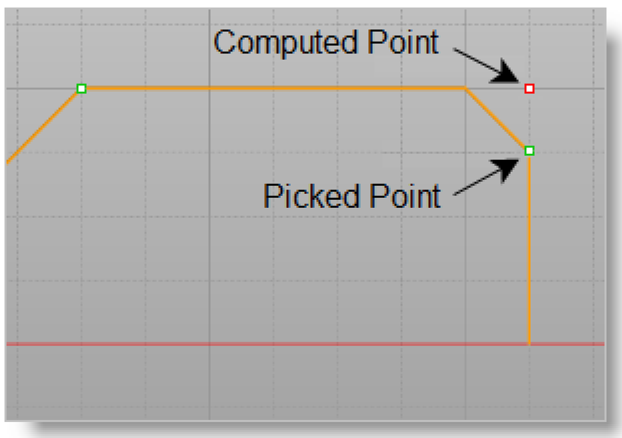
You can define the starting and ending position of the threads either by inputting the X and Z coordinate values for the start and end points or using the pick option to graphically select 2 points.

- **Start XZ / End XZ**
  This defines the Start and End of the Turn Threading operation. You can enter the coordinate values directly use the Pick button and then select

the start and end locations directly from the turn part profile. You can use the object snaps available for selecting points.

- **XZ Clearance**
  These fields allow you to set clearance along the X and Z axis. The Z Clearance is applied axially to the start of the thread. The X Clearance is an arbitrary clearance for the tool to move away from the thread.

- **Always create thread with no taper**
  Checking this box allows you to pick points on a non-straight area of the model and create a straight thread with no additional geometry creation. Refer to the illustration below:



Always create thread with no taper

- **Dependent Parameters**
  You can use the Pick button under the Thread Definition section to select the thread start and end points. Once defined, the system automatically determines the Thread Length (TL), Major Diameter (D) and Taper Angle from the part model.

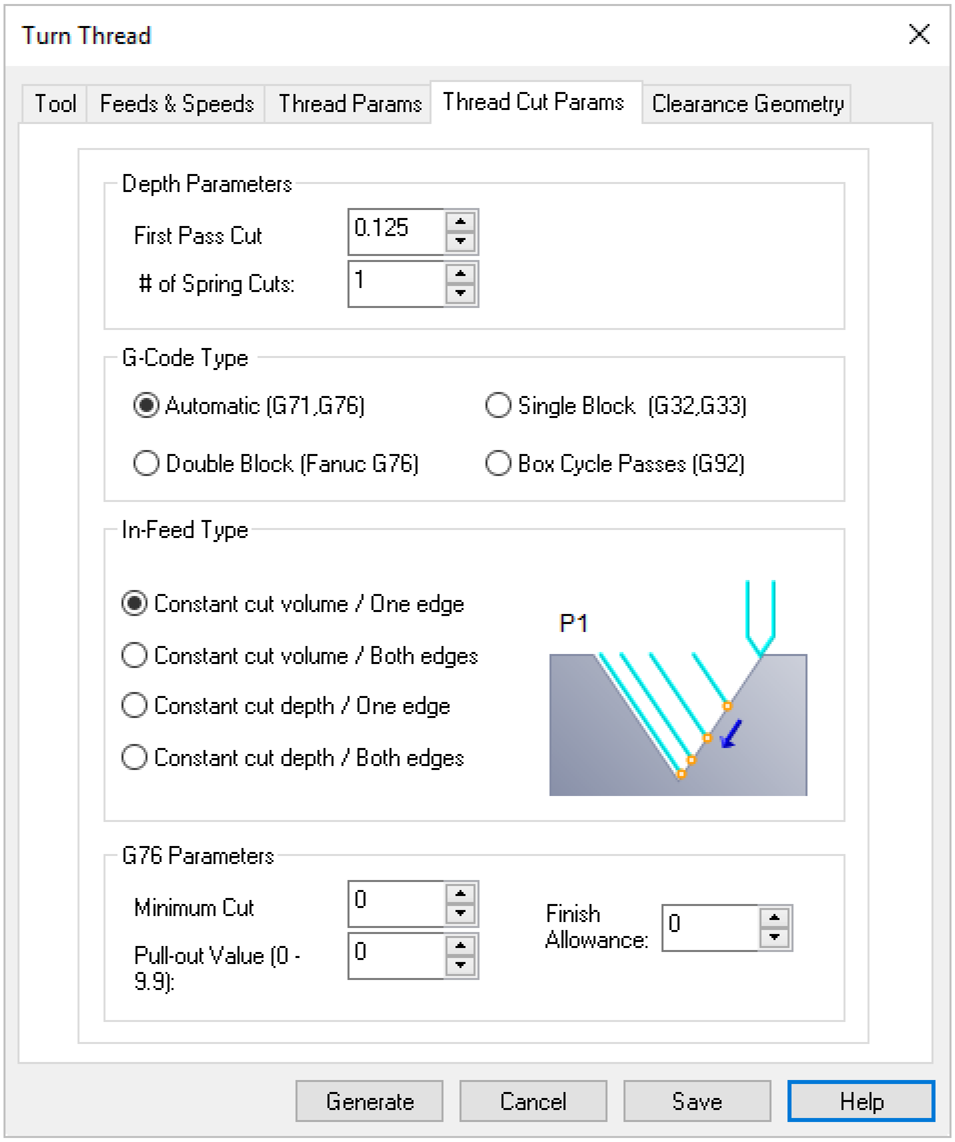
- **Thread Depth (tp) and Thread Pitch (P)**
  These allow you to define the Thread Depth and Pitch manually.

- **Hand**
  The type of threads to be generated (Right Handed / Left Handed)

8.6.2 Thread Cut Parameters

This tab allows user to set the Depth parameters, Thread Cycle type and its parameters.

Dialog Box: Thread Cut Parameters tab



Dialog Box: Thread Cut Parameters tab, Turn Thread

Depth Parameters

First Pass Cut
Enter the thread depth for the First Cut Pass. The tool will make one First Cut Pass along the length of the thread at this depth.
# of Spring Cuts
Enter the total number of Spring Cuts to achieve the total thread depth. The tool will make one complete Spring Cut along the length of the thread and then repeat the Spring Cut the number of times specified. The depth of each Spring Cut is calculated automatically.

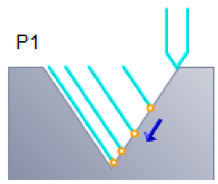
G-Code Type

The following types of threading cycles are supported:

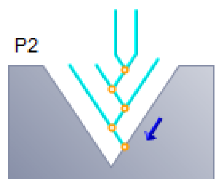
Automatic (G71,G76)
Pick this option to choose the Automatic (G71,G76) g-code type. This is a repetitive threading cycle. With this selection, the In-Feed Type section and the G76 Parameters section of this dialog are activated.
Single Block (G32,G33)
Pick this option to choose the Single Block (G32,G33) g-code type. This is a single pass threading cycle.
Double Block (Fanuc G76)
Pick this option to choose the Double Block (Fanuc G76) g-code type. This is a double pass threading cycle.
Box Cycle Passes (G92)
Pick this option to choose the Box Cycle Passes (G92) g-code type. This is a multiple pass threading cycle.

In-Feed Type

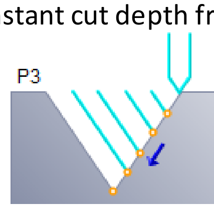
Constant cut volume / One edge - When G-Code Type is set to Automatic (G71,G76), select this option to perform a constant cut volume from one edge as shown.



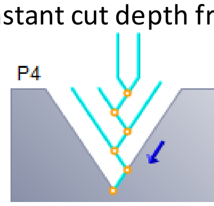
Constant cut volume / Both edges - When G-Code Type is set to Automatic (G71,G76), select this option to perform a constant cut volume from both edges as shown in the dialog image.



Constant cut depth / One edge - When G-Code Type is set to Automatic (G71,G76), select this option to perform a constant cut depth from one edge as shown.

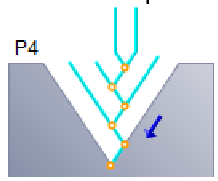


Constant cut depth / Both edges - When G-Code Type is set to Automatic (G71,G76), select this option to perform a constant cut depth from both edges as shown.



G76 Parameters

Constant cut depth / Both edges - When G-Code Type is set to Automatic (G71,G76), select this option to perform a constant cut depth from both edges as shown.



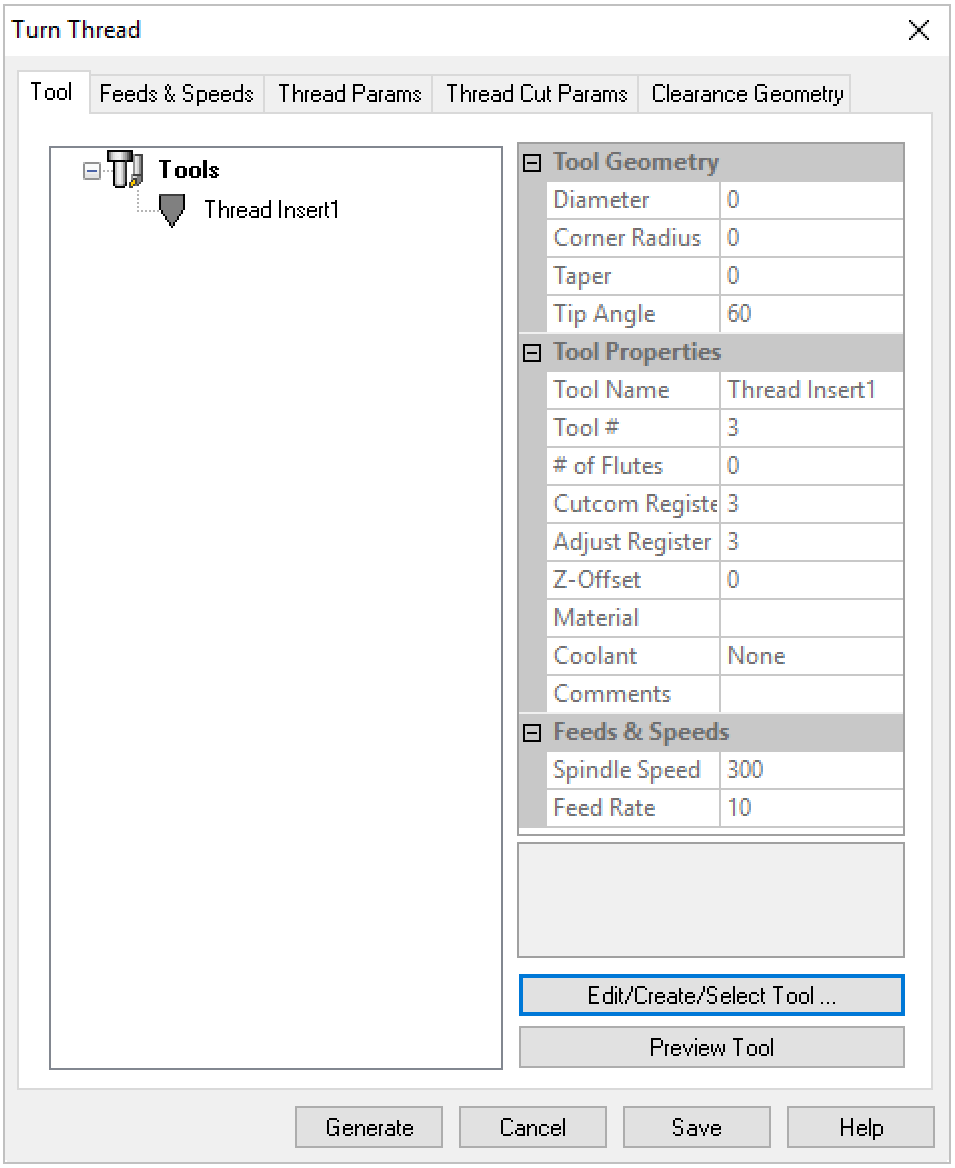
G76 Parameters / Pull-out Value (0-9.0) - The G76 threading cycle allows for a Pull-out Value. This is the lead or pullout when exiting the thread. The value can be set between 0 and 9.9.

G76 Parameters / Finish Allowance - This is the amount of material left on the thread before the final Finish pass.

8.6.3 Tool

The following dialog allows you to select the appropriate Thread Insert tool for the Turn Thread operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Thread

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

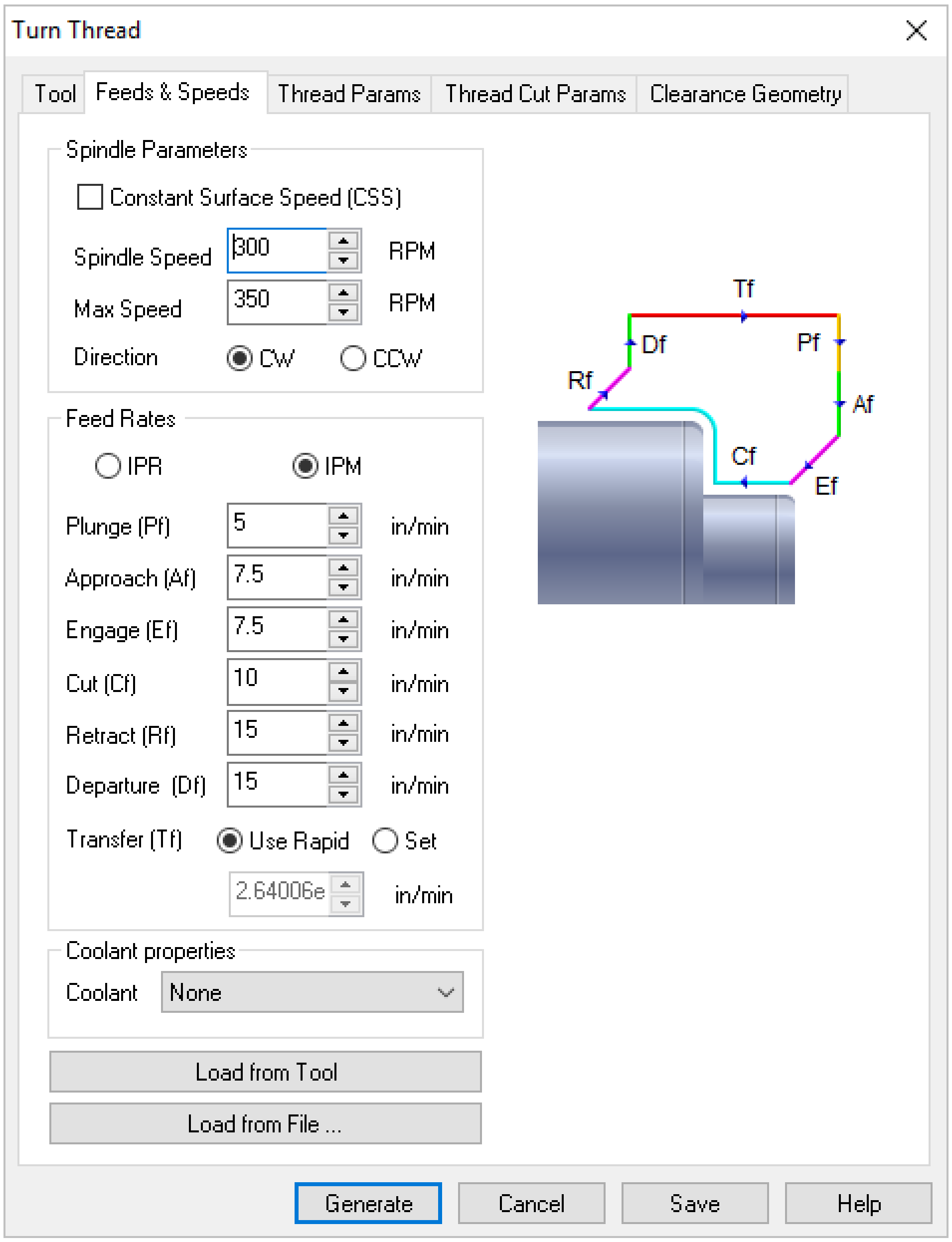
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.6.4 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Thread operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Thread

Spindle Parameters

Constant Surface Speed (CSS)

This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed

his is the rotational speed of the spindle expressed in RPM.

Surface Speed

Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed

The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction

This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)

This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)

This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)

This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)

This is the feedrate used when the tool is cutting material

Retract (Rf)

The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)

The feedrate used to retract the tool from the material.

Transfer (Tf)

This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

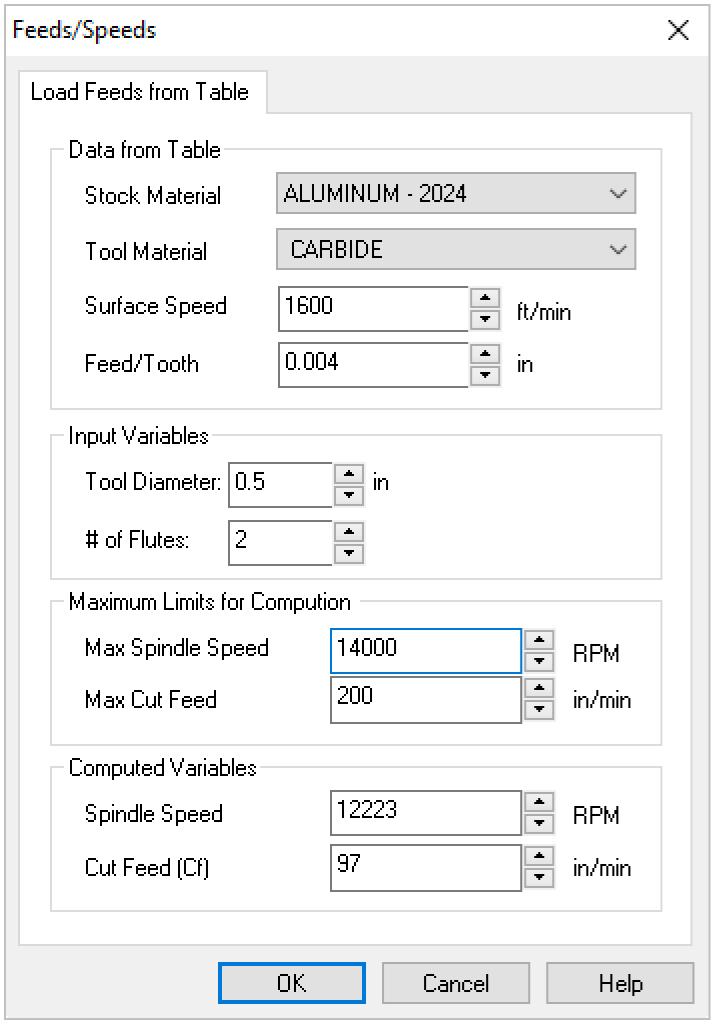
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

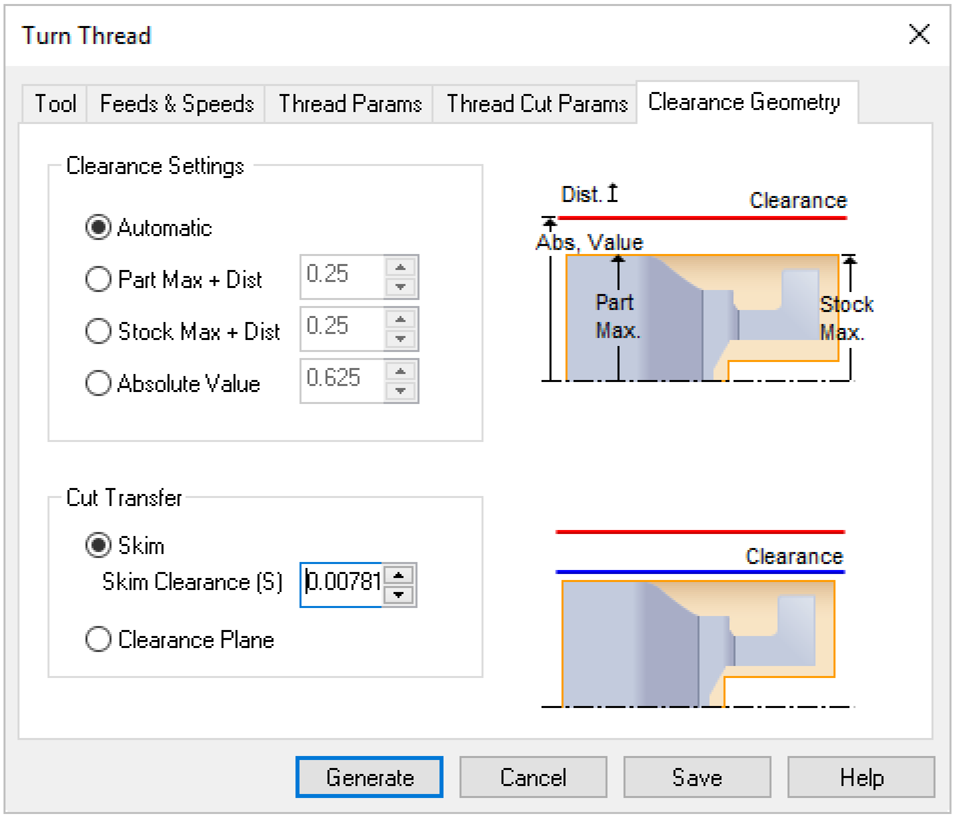
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.6.5 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Thread operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Thread

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.
>
> For Hole Machining operations, the clearance plane is normal to the Z axis.

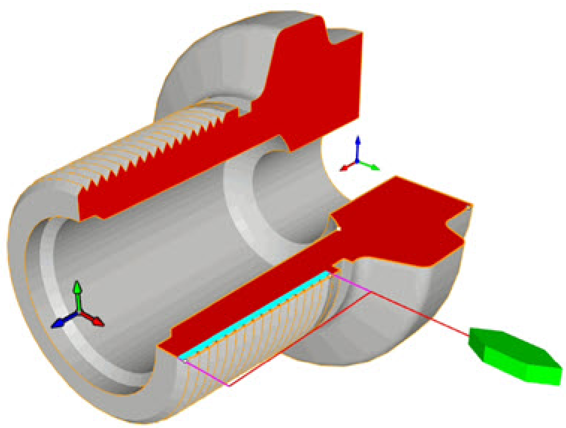
Cut Transfer

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

8.6.6 Threading in Practice

Introduction

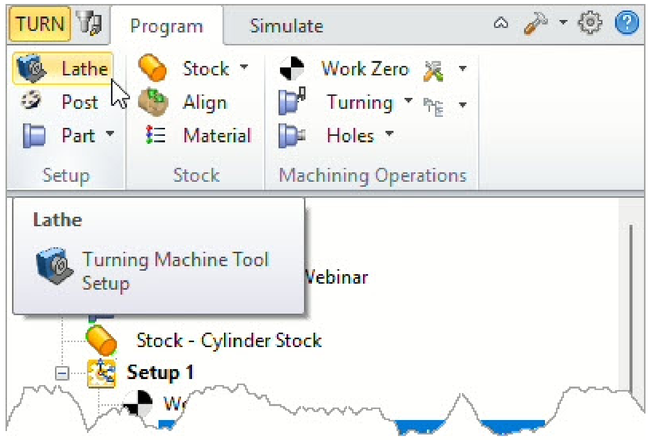
The TURN Module in MecSoft CAM's RhinoCAM and VisualCADCAM supports 2½ Axis Turning Centers and supports the most comment Turning toolpath strategies including Roughing Finishing, Grooving, Threading and Parting-Off operations. In this article we will discuss the Turn Threading operation and its support for the most common canned threading cycles on Fanuc controllers. This discussion covers the procedures needed to CAM program threads in the TURN Module, from control geometry to post-processing.



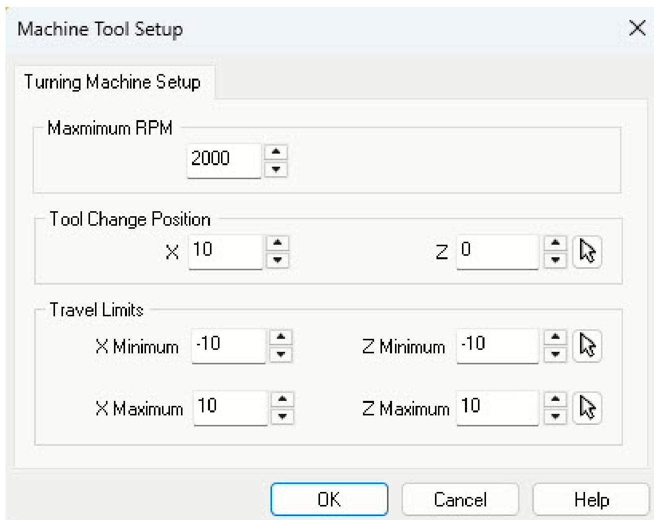
TURN Threading in Practice

The Machine Tool Setup

The Setup procedure begins at the Lathe command located on the Program tab of the TURN Machining Browser as shown below. The Machine Tool Setup dialog provides the basic parameters for the TURN Setup including Maximum RPM, Tool Change Position and Travel Limits. You can see our parameters in the dialog below.



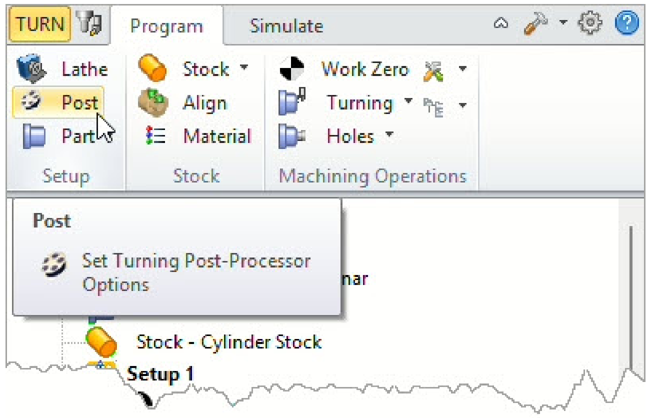
Locating the Machine Tool Setup Command



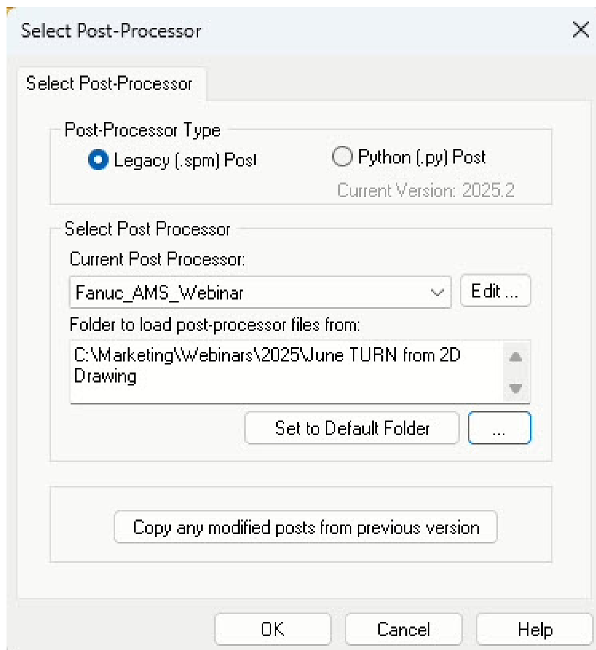
The Machine Tool Setup dialog

The Post-processor

In this step we select a post-processor to match the 2½ Axis Turning Center's Controller software. In this case we are selecting a Fanuc post which is one of the most common controller software for Turning centers. We have customized one of our legacy Fanuc posts for this article. It is listed as the Current Post Processor in the dialog below.



Locating the Post command from the Program tab of the TURN Machining browser



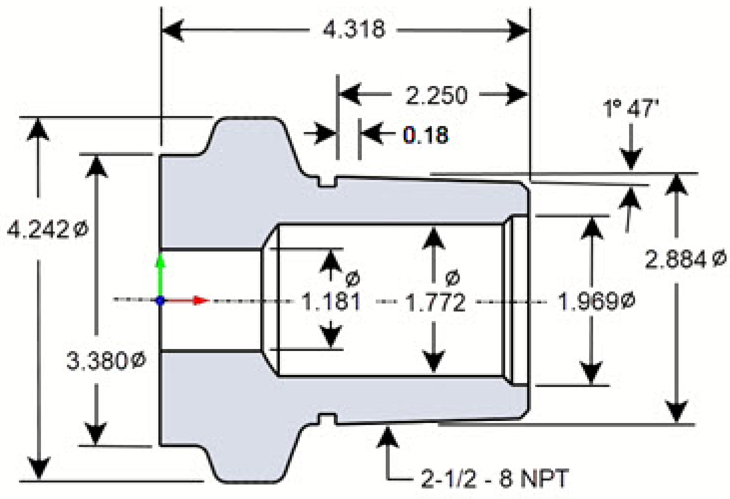
The Select Post-Processor dialog

Control Geometry for Turning

In the TURN Module, part geometry can be either a 2D drawing or a 3D surface model. In the case of a 2D drawing, the geometry define the cross-section of the Turned part. For a 3D model, the Turn Part is defined by extracting the 2D cross-section profile. In both methods, the cross-section profile is considered the Control Geometry for all Turning operations in the Setup. In this exercise we are defining the Turn Part from a 2D drawing.

The 2D Dimensioned Drawing

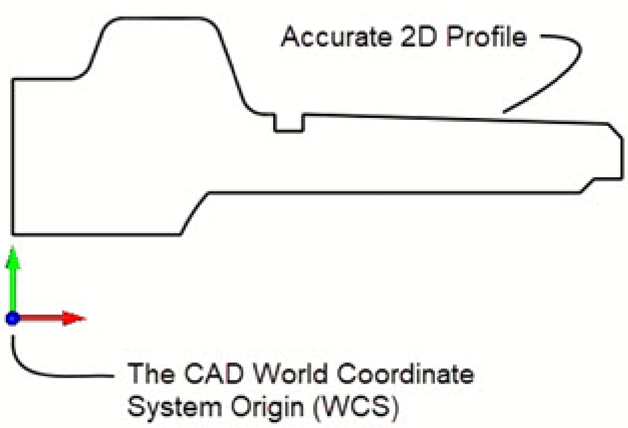
The dimensioned drawing below illustrates a cross-section of the resulting Turned part. When using a 2D drawing for Turning, the geometry representing the part cross-section must be an accurate representation. Prior to drawing or importing the 2D drawing make sure the CAD unit tolerances are set to at least 5 decimal places (0.00001). In RhinoiCAM, the drawing tolerance is set from the Units tab of the Tools > Options dialog. In VisualCADCAM it is set from the Tools & Units tab of the Options dialog. Our 2D drawing is illustrated below.



The 2D cross-section drawing of the Turn Part

The Accurate 2D Cross-section Profile

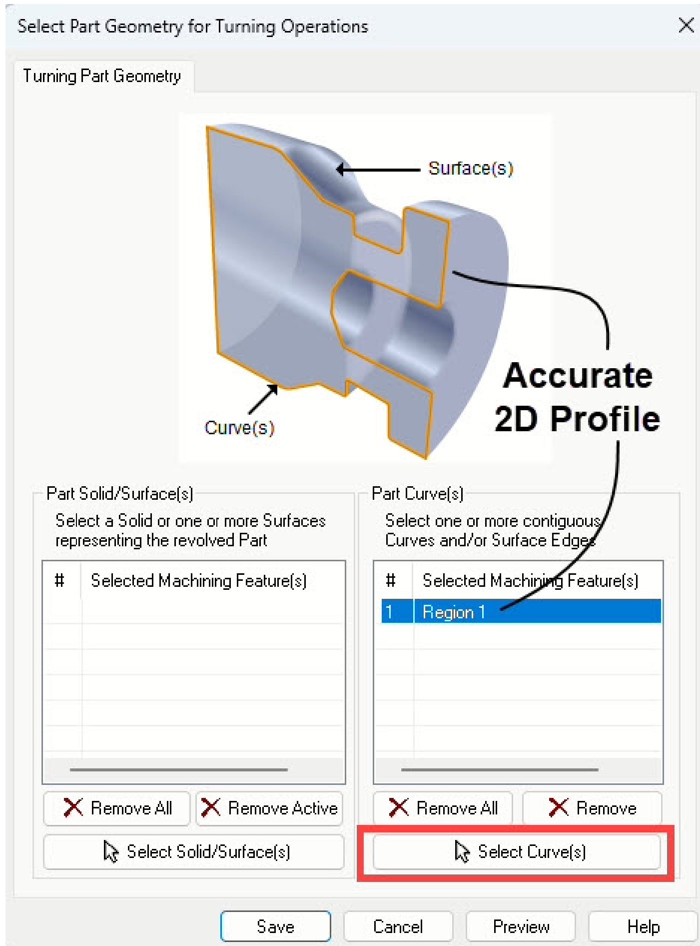
From our 2D Dimensioned drawing we have removed all CAD entities except for the profile geometry. Because our part has internal cutouts that pass completely through the part, we have moved the profile such that the turn part axis is aligned with the CAD World X Axis (in red). We have also moved the right side of the profile so that it lies on the CAD World Y Axis (green). The profile must also lie on the XY plane. Note that we are referring to the CAD WCS (World Coordinate System). This is not to be confused with the MCS (Machine Coordinate System which we will be defining shortly. The 2D profile is shown below.



The 2D Cross-section profile geometry

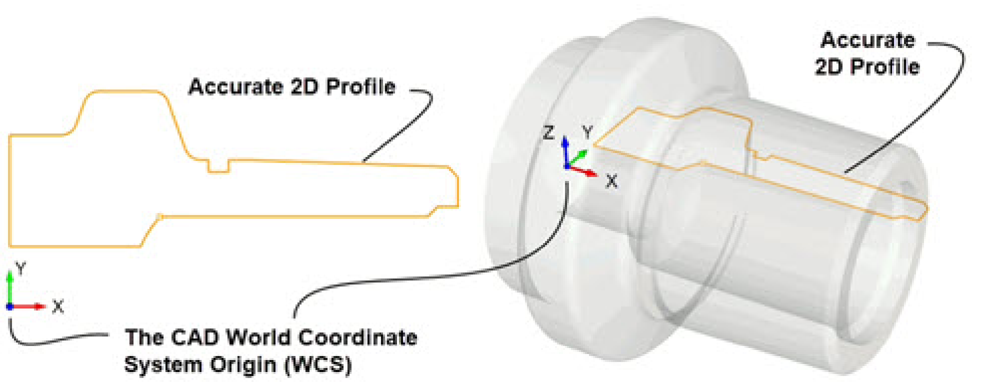
Defining the TURN Part

With our accurate 2D cross-section profile positioned we can now define it as our TURN Part. This is done using the Select Turn Part command on the Program tab of the Turn Machining Browser. This will display the Select Part Geometry for Turning Operations dialog shown below. Notice that the dialog is divided into 2 parts, Part Solid/Surface(s) and Part Curve(s). Since we are using a 2D Profile, we pick the Select Curve(s) button under the Part Curve(s) section of the dialog. As prompted, we select our 2D profile and press <Enter>. The profile is then added to the Select Machining Feature(s) list as Region 1. Refer to the dialog below.



The Select Part Geometry for Turning Operations dialog

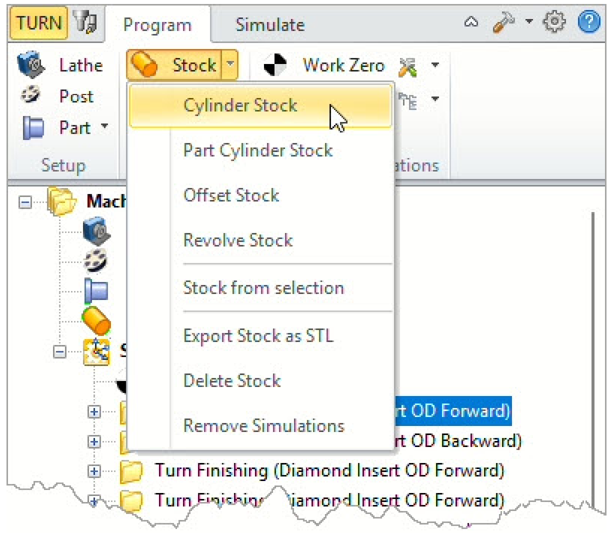
Now we pick Save from the dialog and see that the Turn Part model is now displayed on the screen. You can also see the 2D cross-section profile highlighted in orange. Refer to the illustrations below.



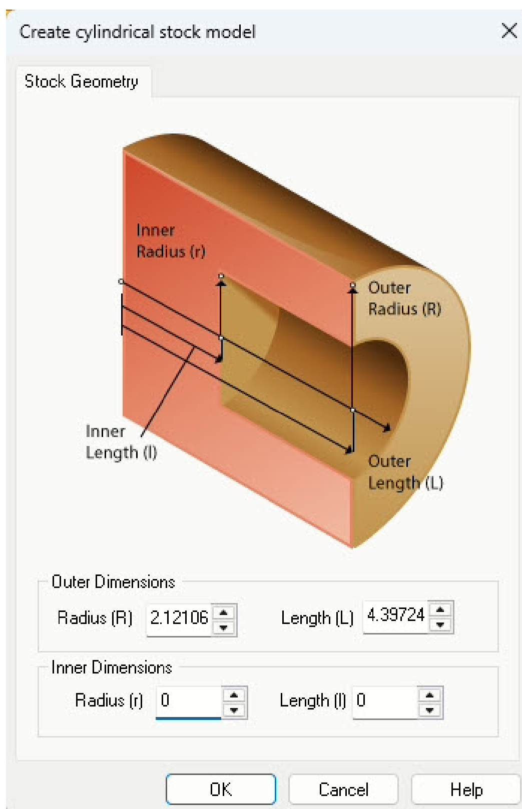
The Turn Part is defined using the cross-section profile and the Select Part Geometry for Turning Operations dialog.

Defining the TURN Stock

The next step is to define the Turn Stock model. For this we go to the Stock menu located on the program tab and we select Cylinder Stock to display the dialog shown below.



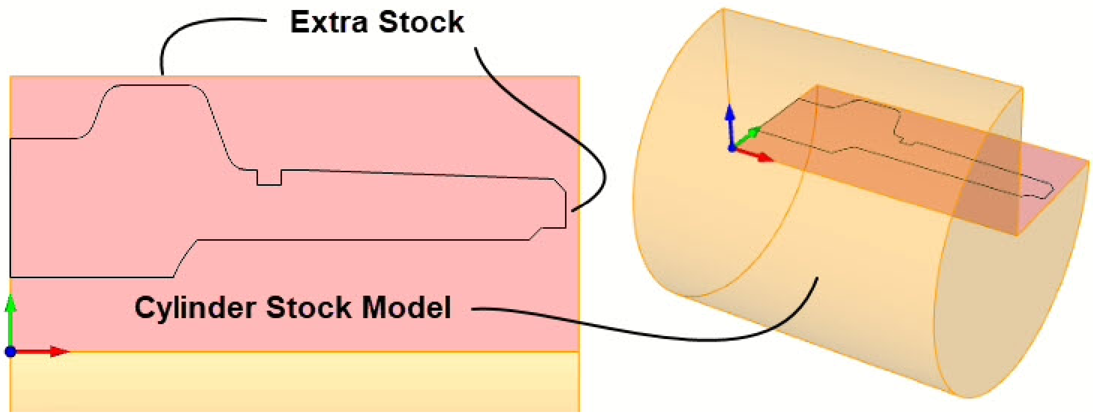
The Stock menu with Cylinder Stock selected



The Create Cylindrical Stock Model dialog

The dialog shows the actual outer radius and length of the part based on the Turn Part that we just defined above. We will add some distance to both of these dimensions. For the Radius (R) we will enter 2.1875 and for Length (L) we will enter 4.500. This will provide some extra material to ensure that the completed part has a good surface finish when completed. Notice the Inner Dimensions are set to zero. This is fine since we machining the Inner Diameter out.

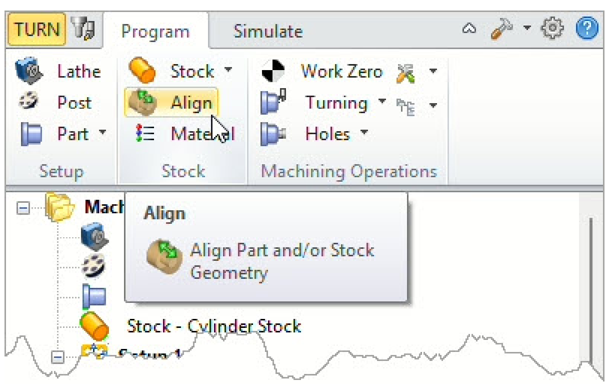
Picking OK from the dialog will create the stock model shown below.



The Turn Stock model is shown here with extra material add to the diameter and the length. Note:The ¾ diameter view of the stock is set from the Simulation tab of the CAM Preferences dialog in the TURN Module.

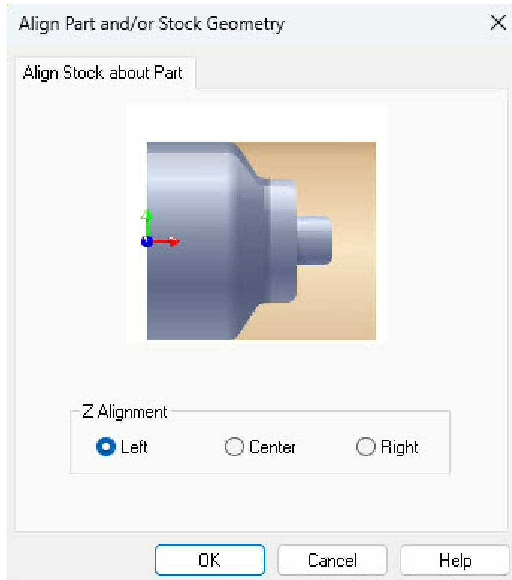
Aligning the Stock with the Part

In the TURN module, it is recommended that you align the Stock model so that it is flush with the left side of the part as it is shown in the illustration above. This is done from the Align command located on the Program tab as shown below.



The Align command on the Program tab of the Turn Machining Browser

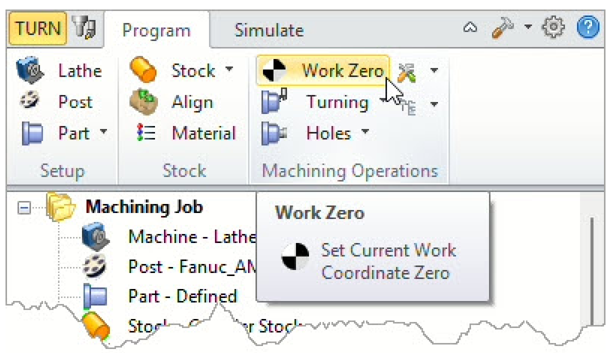
From the Align Part and/or Stock Geometry dialog select Left and then pick OK.



Z Alignment is set to Left in the Align part and/or Stock Geometry dialog

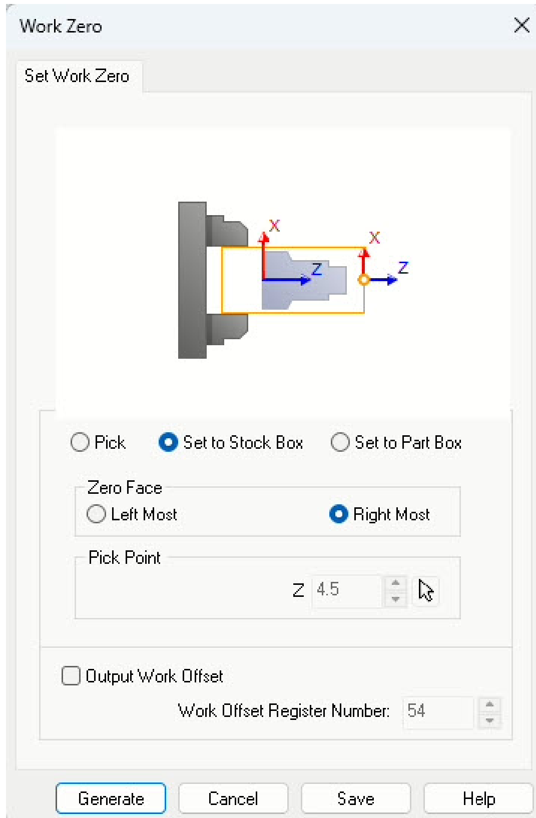
Defining the Work Zero for Turning

The Work Zero in the TURN module is where you plan to zero out the Turning Center. This usually lies on the right-most side of the stock on the rotational axis. Note that the left side of the stock is the spindle chuck.



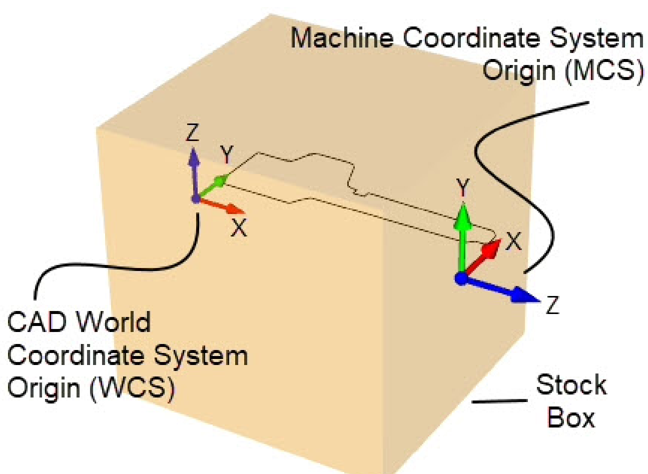
The Work Zero command on the program tan of the TURN Machining Browser

Selecting Work Zero from the TURN Machining Bowser will display the Work Zero dialog shown below. We select Set to Stock Box and and for Zero Face we select Right Most and then pick Generate.



The Work Zero dialog allows you to define where you plan to set the machine zero on the Turning Center.

The illustration below shows where the Work Zero will be located. It is very important that you understand the difference between the CAD WCS (World Coordinate System) triad and the CAM MCS (Machine Coordinate System) triad. Note that the CAM MCS Z Axis is aligned with the CAD WCS X Axis.

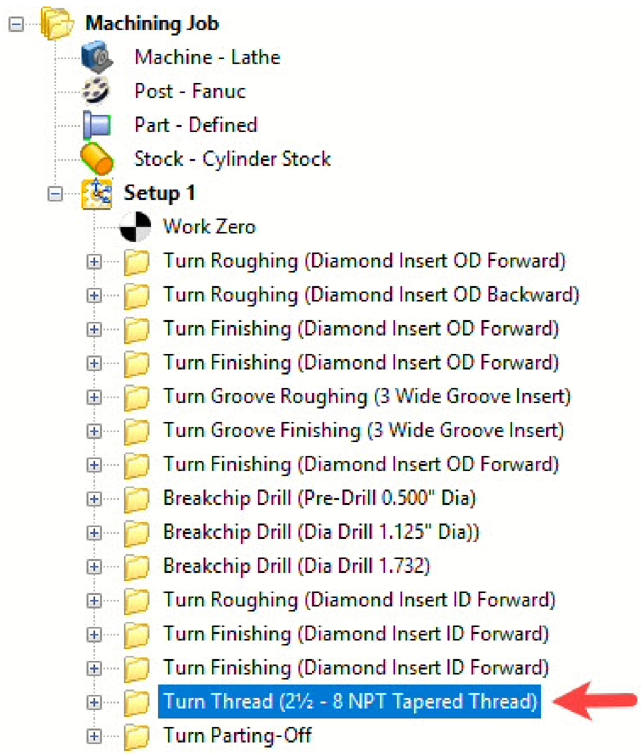


The Work Zero is located on the right side face of the Stock Box

TURN Operations

While this article covers Turn Threading, there are many Turn operations that will precede it in the Machining Job. The reason threading is one of the last operations performed is due to surface finish and quality. The last thing you want is extracted stock material damaging those perfect threads that you just cut. Plus, unlike in Milling, in Turning Centers it is more difficult to channel away the extracted workpiece stock so finishing operations are always performed last.

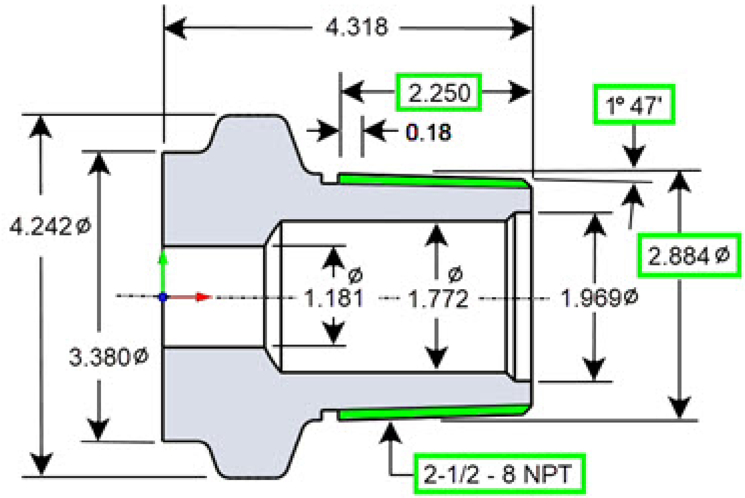
For this part, the following roughing and finishing TURN operations were performed first:



The TURN Threading operation is next to the last operation performed

Thread (2½ - 8 NPT External Tapered Thread)

This part requires a 2½" – 8 NPT tapered external pipe threads. For Turn Threading, the profile geometry must be drawn to the outer diameter of the thread and include the taper. This is critical to get the correct thread after machining. The illustrations below shows where the threads are located on the dimensioned drawing and in the 2D profile used to define the Turn Part.

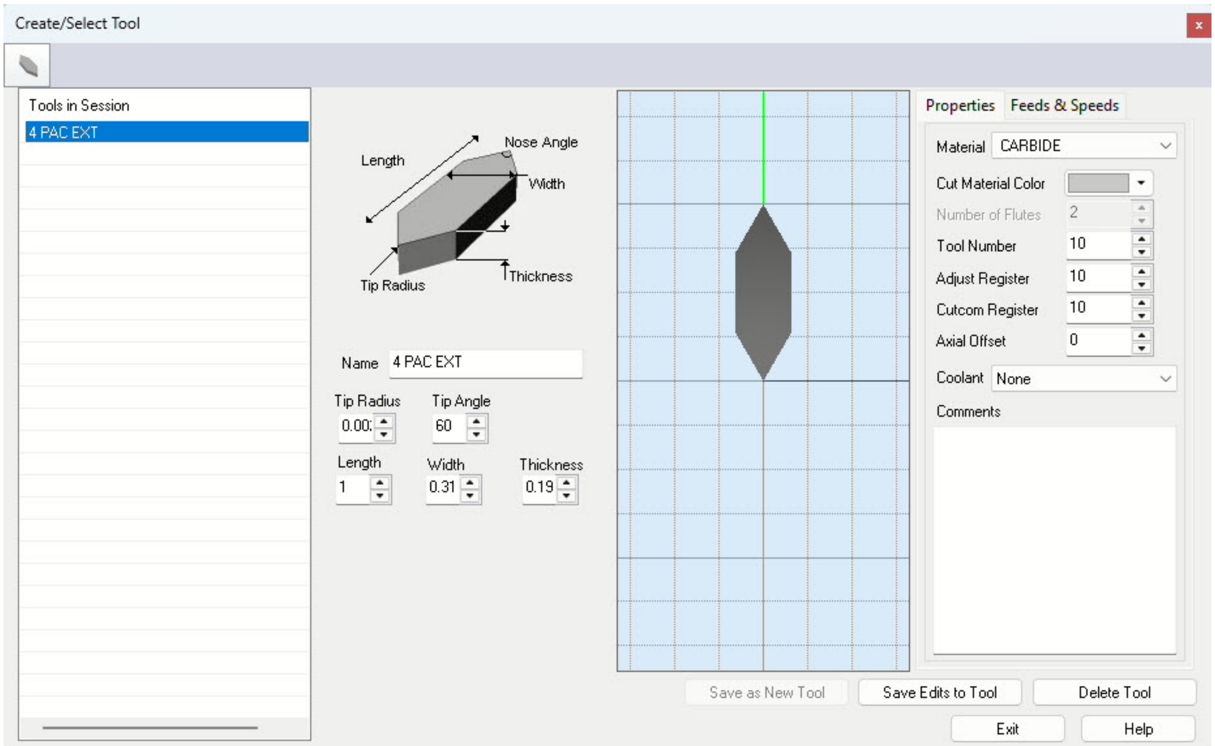


The location and dimensions for the 2½ – 8 NPT external threads

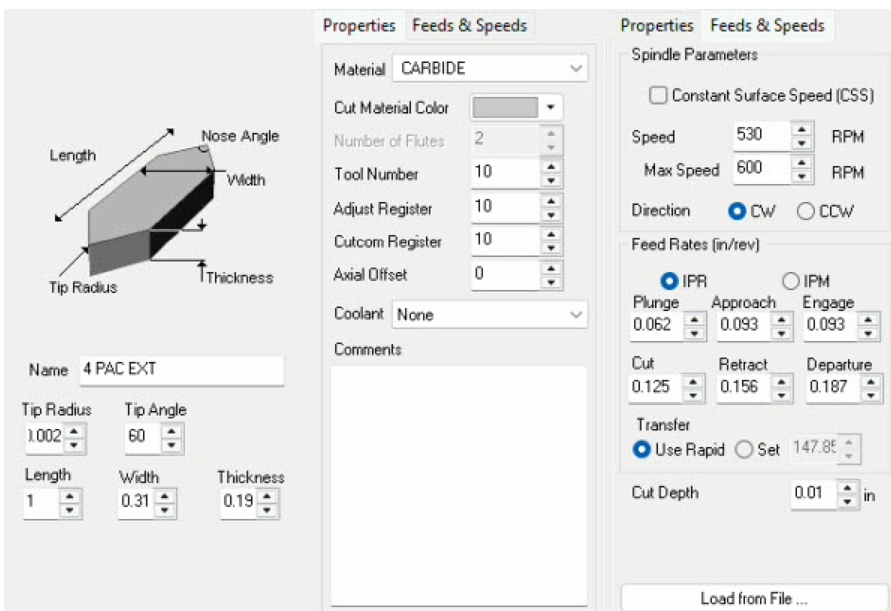
It is critical that the profile geometry match the major diameter of the 2½" – 8 NPT external thread. In this case it is the 2.884 diameter. It is also critical tha the taper angle of 1 degree, 41 seconds be represented in the profile. If these two dimensions are not accurate, your thread will not be cut correctly.

Thread Insert Parameters

We will be using a single point thread insert cutting tool for this guide. The Create/Select Tool dialog and and thread parameters are shown in the dialogs below. It is very critical that you know and understand YOUR requirements, including which threading cycles your Turning Center and perform, the thread size dimensions and cutting parameters. You can look this information up using any of the available AI tools such as Grock (X.com), Gemini (Google) or ChatGPT (ChatGPT.com)



The Create/Select Tool dialog showing tool insert dimensions and properties

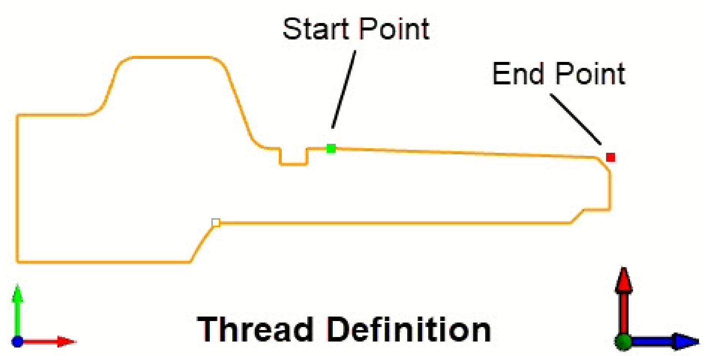


Composite view of the Create/Select Tool dialog showing insert dimensions, Properties and Feeds & Speeds

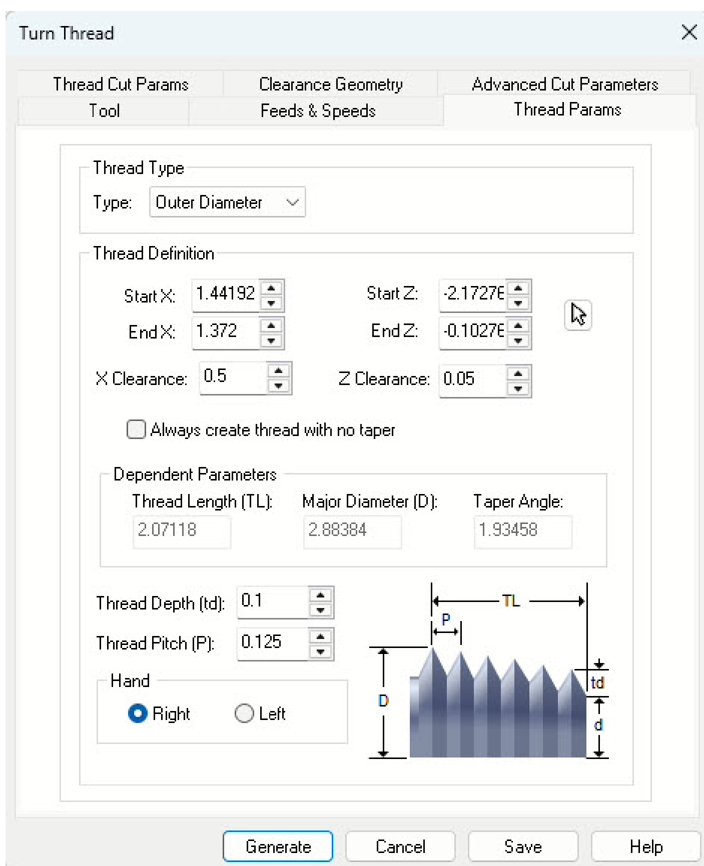
Thread Parameters

There are two tabs in the TURN Thread operation dialog that are key to defining our 2½ – 8 NPT external thread. These are the Thread Params tab that controls the definition, size and constraints of the thread and the Thread Cut Params tab that controls the type of Threading cycle that is posted to the G-Code file.

The Thread Type is set to Outer Diameter. The Thread Definition includes the Start and End point of the thread. We used the Pick button to select these two points. The Start Point is located on the profile to the right of the o-ring groove (shown in **green**) and the End Point is located at the end of the taper. We want the thread to extend out past the chamfer at the tapered end so we have pre-positioned a point at the theoretical end of the taper (shown in **red** below).

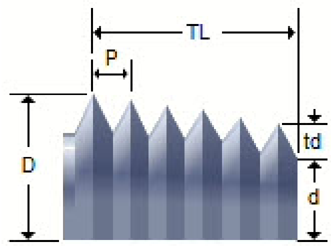


Here we see the Start and the End points of the external thread



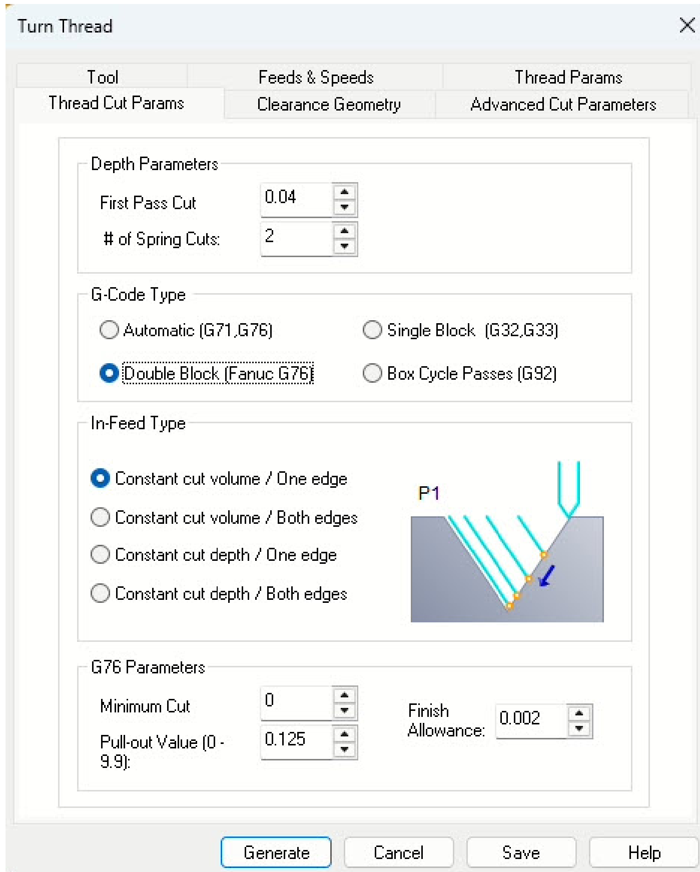
The Thread Params tab of the Turn Thread operation dialog

Two Clearance parameters are provided in the dialog. They allow you to control both the X and Z Clearance. Both are set to 0.5 and 0.05 respectively. Because we are defining a Tapered thread, we make sure the box labeled Always create thread with no taper is not checked. Additionally, the dialog allows us to define the Thread Dept (td) which is 0.1 and the Thread Pitch (P) which is 0.125. These dimensions adhere to the standard 2½ -8 NPT external thread specifications. We are also specifying a Right Hand thread. The dependent parameters Thread length (TL), Major Diameter (D) and Taper Angle are each derived from the Start and End points we picked from the 2D profile.



Thread Cut Parameters

As mentioned previously, the Thread Cut Params tab controls how the G-Code is formatted for threading. The format selected will depend on the capabilities of CNC Turning Center used and operator preferences. The dialog supports multiple canned cycle formats. A Canned Cycle is a piece of G-Code that tells the Turning Center to use one of its predefined cycles for cutting the thread. What follows is a brief description of each of the Thread Cycle types supported by the dialog. What follows is an over-simplified discussion of each of the threading G-Codes supported by the dialog.



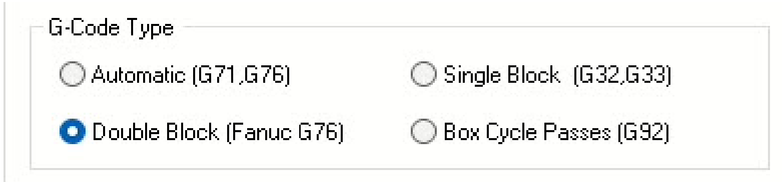
The Thread Params tab of the Turn Thread operation dialog

Depth Parameters

This section of the dialog controls the cutting depth of the thread. First Pass Cut tells the machine how deep to cut on the first spiral pass. # of Sping Cuts tells the machine how to divide up the remainder of the thread depth. You can consider these two parameters as roughing and finishing of the thread.

G-Code Type

In this section of the dialog you are telling the CNC machine which threading commands you want to use when posting the G-Code for cutting the 2½ – 8 NPT external thread. The supported commands include the G76 automatic cycle, the G76 double block cycle (specifically designed for Fanuc controllers), the G32/G33 single block and the G92 box cycle. Each of these are discussed below.



Automatic (G71 / G76)

When you need to cut screw threads on a CNC lathe, whether it's on the inside or outside of a part, the machine uses a special command called G76. Think of G76 as a smart, automated program for threading.

Instead of manually telling the machine to cut a little bit, move back, cut a little more, and so on, G76 handles all of that. You just tell the machine what kind of thread you want – how deep it should be, if it's a straight thread or taped, and how many times the tool should pass over the material to get to the final depth of the thread.

The machine then works its magic by precisely matching the rotation of the part with how fast the cutting tool moves along the part's length. This synchronized movement is what creates the perfect spiral of a thread. It's designed to be very precise and customizable, making complex thread cutting much simpler for the operator.

Double Block (Fanuc G76)

The G76 threading cycle on Fanuc-controlled CNC lathes is a canned cycle that automates multi-pass threading (straight or tapered) using a two-line (double-block) format for precise control over thread cutting.

The First Block is the roughing pass. It specifies pass count, chamfering, tool angle, minimum cut depth, and finishing allowance while the second block is the finishing pass. It defines thread geometry (diameter, length, depth, first pass depth, and pitch).

Single Block (G32 / G33)

G32 and G33 are distinct from G76 (see above) because they are single-pass (i.e., Single Block) threading commands. Instead of automating the entire process, you're essentially guiding the machine for each individual cut the thread insert makes.

With G32 (and similarly G33), the parameters in the dialog tell the machine precisely where to execute one specific cut from the Start point to the End point on the 2D profile. Using Thread Pitch (P), which is how far the thread advances for every full rotation of the workpiece, the CNC machine perfectly synchronizes the spinning of the part with the cutting tool's movement to achieve the exact spiral cut.

G32 and G33 are nearly identical on Fanuc controllers. G33 also cuts the thread in one pass but also supports rigid tapping or variable pitch threads on controllers other than Fanuc.

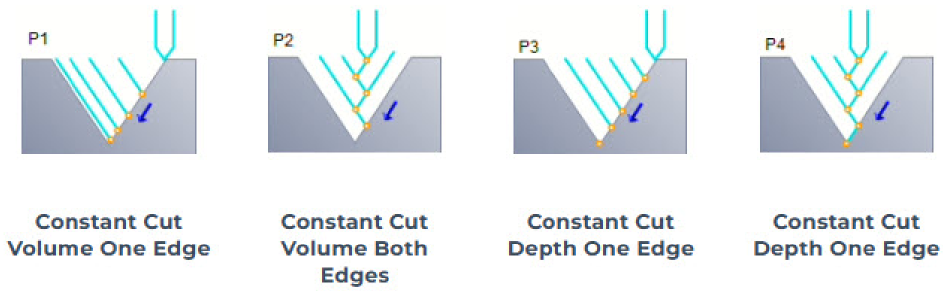
Box Cycle Passes (G92)

The G92 threading cycle is sometimes called a "box cycle" due to its repetitive, box-like toolpath) used for cutting threads. Unlike G76 (a multi-pass canned cycle) or G32/G33 (single-pass threading commands mentioned previously, G92 is a simpler threading cycle that cuts threads in a series of passes at a fixed depth or with incremental depth adjustments. It's often used on older CNC controllers or for straightforward threading tasks.

With the G92 command, the threading tool cuts along the Z-axis (thread length), then retracts radially (in X), and rapidly returns to the start. This "box cycle" repeats, with each pass going to the same or a deeper X-depth, until the thread is complete. All these repetitions are managed by a single G92 command based on the depth settings entered into the dialog.

In-Feed Type

This portion of the dialog allows you to control how the cutting tool is fed into the material to form the finished thread. You can refer to the dialog illustrations below.



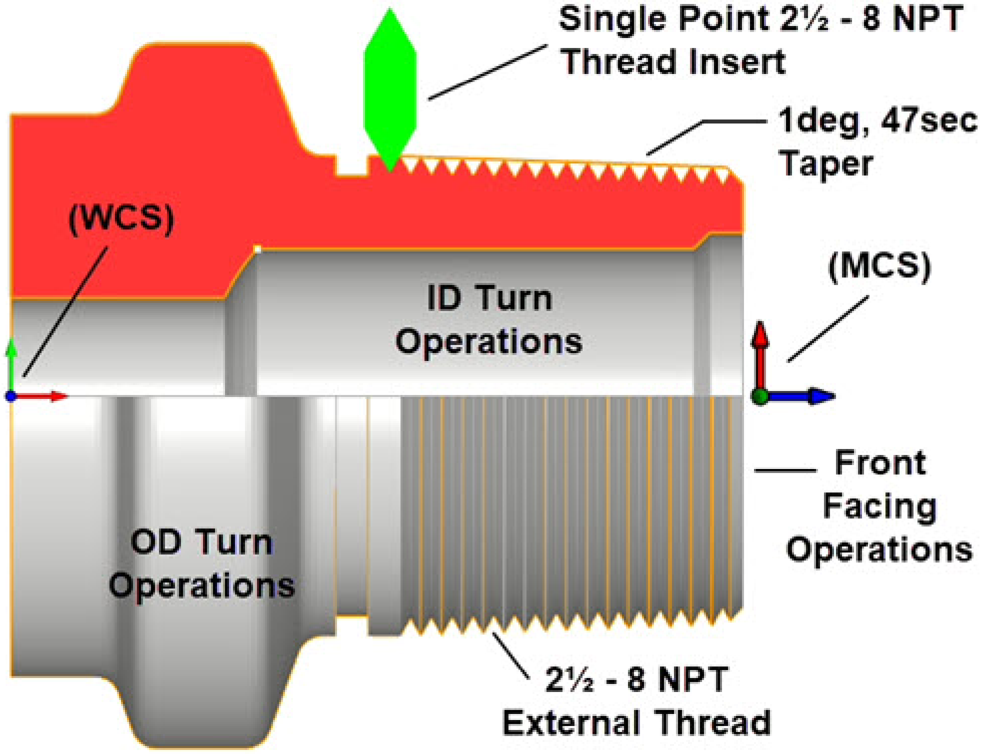
The G-Code Type in the Thread Cut Params tab of the Turn Thread dialog set to Double Block (Fanuc G76)

G76 Parameters

The following parameters in this section of the dialog further specify the Automatic G76 thread cycle. The Minimum Cut controls the minimum amount allowance that controls how deep the thread insert can cut. The Pull-out Value determines the amount of chamfering when the cutter retracts on the end of the thread. The Finish Allowance is amount of material left before the final finishing pass.

Cut Material Simulation

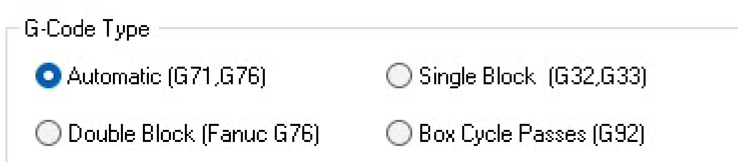
With our threading parameters complete its now time to simulate and see our 2½ – 8 NPT external tapered thread.



Post-processing

With our thread generated and simulated, its time to see what the G-Code looks like for a 2½ – 8 NPT External Thread. Below is the G-Code for each thread type.

Automatic G71 / G76 Posted G-Code Sample

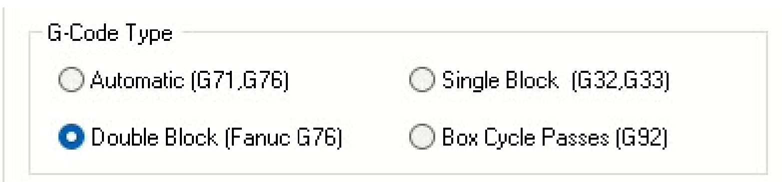


The G-Code Type in the Thread Cut Params tab of the Turn Thread dialog set to Automatic (G71 / G76)

```
%
(Turn Thread (2½ - 8 NPT Tapered Thread))
T10
G54
G50 S600
G97 S530 M03
M08
G00 X1.942 Z-2.223
G76 X2.544 Z-0.103 I2.071 K0.1 D400 F0.125 A60
G00G28W0.
G00G28U0.
M09
M05
M30
%
```

The G-Code created with the TURN Fanuc Post-processor for the 2½ – 8 NPT External Thread

Double Block (Fanuc G76) Posted G-Code Sample

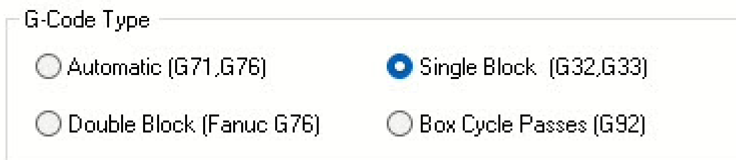


The G-Code Type in the Thread Cut Params tab of the Turn Thread dialog set to Double Block (Fanuc G76)

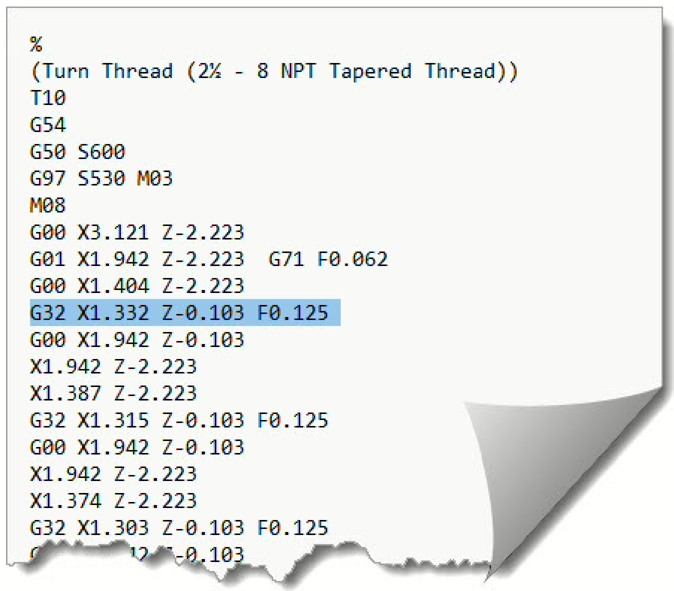
```
%
(Turn Thread (2½ - 8 NPT Tapered Thread))
T10
G54
G50 S600
G97 S530 M03
M08
G00 X3.121 Z-2.223
G01 X1.942 Z-2.223  G71 F0.062
X1.404 Z-2.223  G71 F0.093
X1.332 Z-0.103  G71 F0.125
X1.942 Z-0.103  G71 F0.156
G00 X1.942 Z-2.223
G01 X1.387 Z-2.223  G71 F0.093
X1.315 Z-0.103  G71 F0.125
X1.942 Z-0.103  G71 F0.156
G00 X1.942 Z-2.223
G01 X1.374 Z-2.223  G71 F0.093
X1.303 Z-0.103  G71 F0.125
X1.942 Z-0.103  G71 F0.156
```

Double Block (Fanuc G76) G-Code for the 2½ – 8 NPT External Thread

Single Block (G32 / G33) Posted G-Code Sample



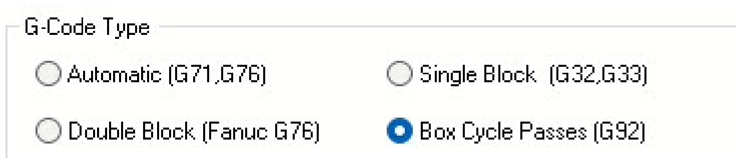
The G-Code Type in the Thread Cut Params tab of the Turn Thread dialog set to Single Block (G32 / G33)



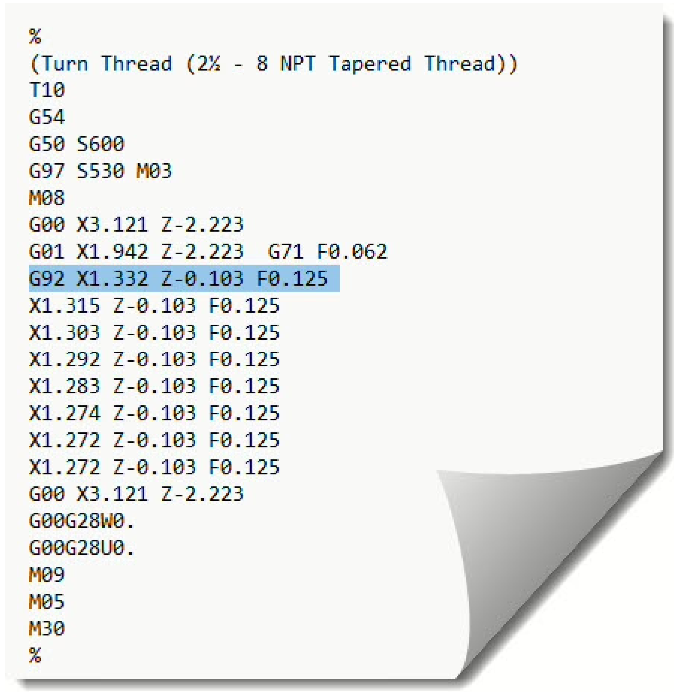
```
%
(Turn Thread (2½ - 8 NPT Tapered Thread))
T10
G54
G50 S600
G97 S530 M03
M08
G00 X3.121 Z-2.223
G01 X1.942 Z-2.223  G71 F0.062
G00 X1.404 Z-2.223
G32 X1.332 Z-0.103 F0.125
G00 X1.942 Z-0.103
X1.942 Z-2.223
X1.387 Z-2.223
G32 X1.315 Z-0.103 F0.125
G00 X1.942 Z-0.103
X1.942 Z-2.223
X1.374 Z-2.223
G32 X1.303 Z-0.103 F0.125
```

Single Block (G32 / G33) G-Code for the 2½ – 8 NPT External Thread

Box Cycle Passes (G92) Posted G-Code Sample



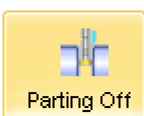
The G-Code Type in the Thread Cut Params tab of the Turn Thread dialog set to Box Cycle Passes (G92)



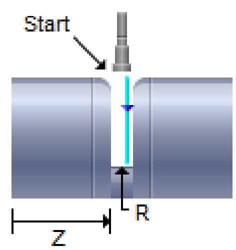
```
%
(Turn Thread (2½ - 8 NPT Tapered Thread))
T10
G54
G50 S600
G97 S530 M03
M08
G00 X3.121 Z-2.223
G01 X1.942 Z-2.223  G71 F0.062
G92 X1.332 Z-0.103 F0.125
X1.315 Z-0.103 F0.125
X1.303 Z-0.103 F0.125
X1.292 Z-0.103 F0.125
X1.283 Z-0.103 F0.125
X1.274 Z-0.103 F0.125
X1.272 Z-0.103 F0.125
X1.272 Z-0.103 F0.125
G00 X3.121 Z-2.223
G00G28W0.
G00G28U0.
M09
M05
M30
%
```

Box Cycle Passes(G92) G-Code for the 2½ – 8 NPT External Thread

8.7 Parting Off

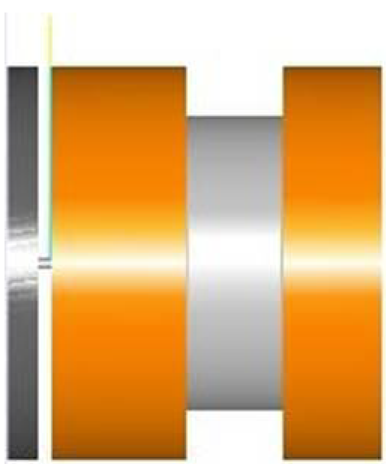


This operation is performed to cut off the finished part from the rest of the bar stock which is typically done as the last operation in OD.



Turn Parting Off

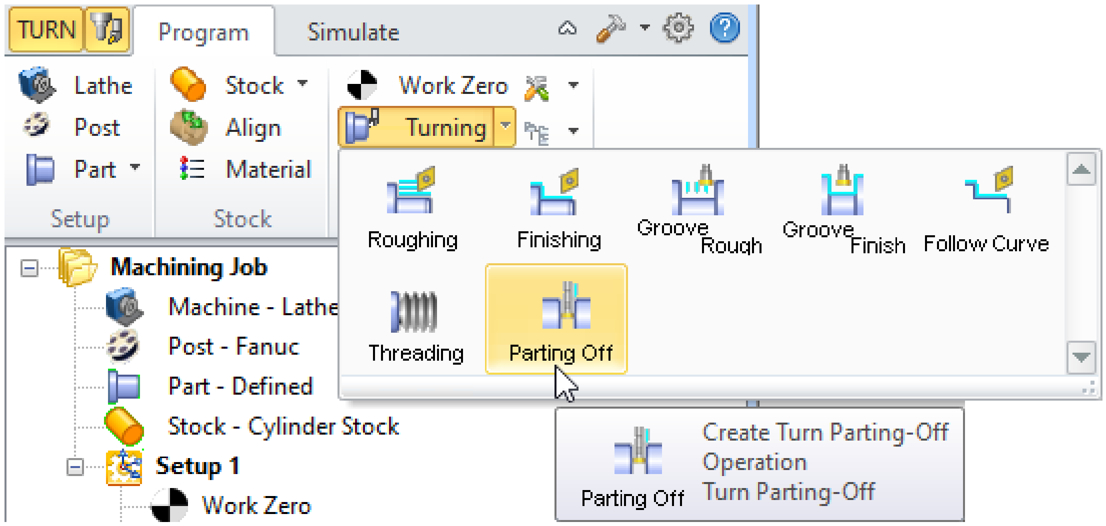
Turn Parting Off Example



Parting Off

Turn Parting Off Menu Item

The Parting Off toolpath method is invoked by selecting the Program tab, clicking on the Turning button in the Machining Browser and selecting the Parting Off operation.

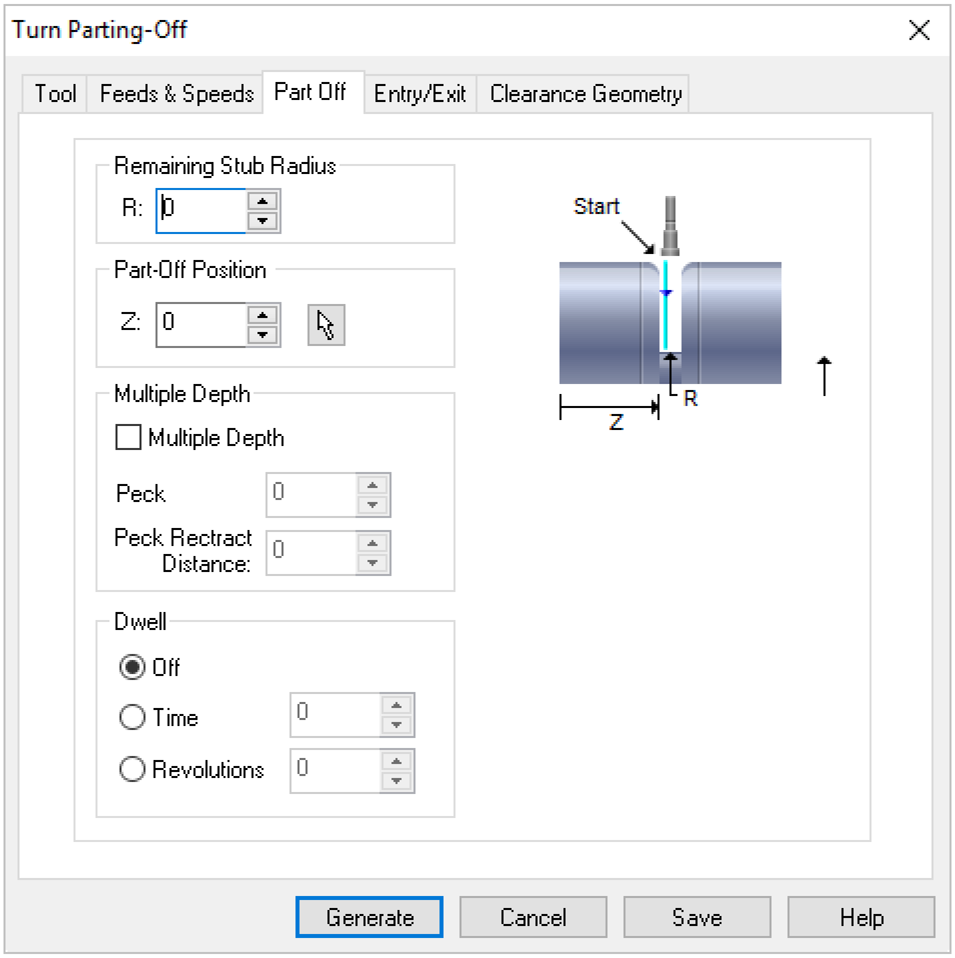


Parting off Menu Item

Dialog Box: Turn Parting-Off

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

This dialog has five tabs. Each tab defines a set of parameters that you can specify. The sections below describe them in detail.

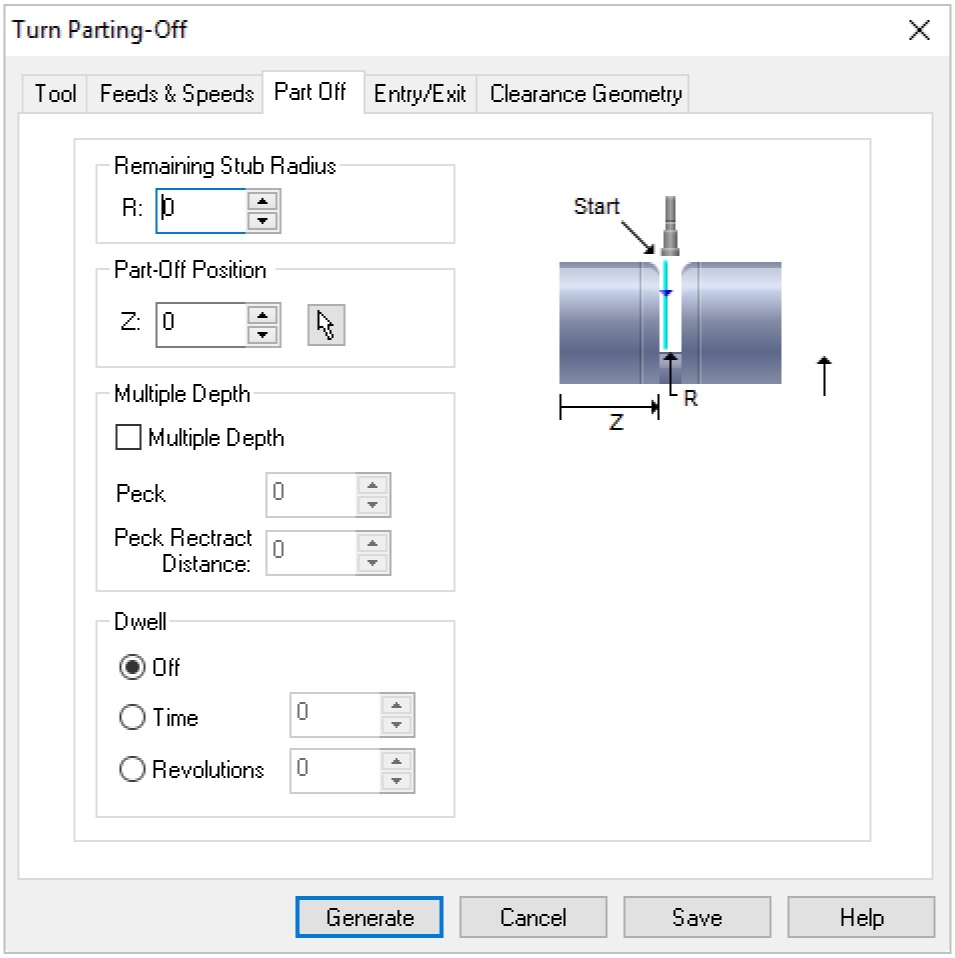


Dialog Box: Turn Parting-Off

8.7.1 Part Off Parameters

The following dialog allows you to set the Part Off Parameters for Turn Part Off operations. You can set the Remaining Stub Radius, PartOff Position, Multiple Depth and Dwell via this property page. All the turning operations as mentioned below, except Part Off Position, can be carried on the Outer Diameter, Inner Diameter or the Front Face of the work-piece.

Dialog Box: Part Off tab



Dialog Box: Turn Parting-Off

Remaining Stub Radius

Remaining Stub Radius (R) - This is the radius of the final part of the stock to be left uncut during the parting operation.

Part Off Position

Part-Off Position - This is the Z coordinate location for positioning the part-off location.

You can enter the coordinate value directly or select the  Pick button to select the point from your part model whose Z coordinate to use.

Multiple Depth

Multiple Depth - Check this box to enable Multiple Depth pecking for the part-off operation. For Peck enter the peck depth. For Peck Retract Distance, enter the distance that you want the tool to retract between pecks.

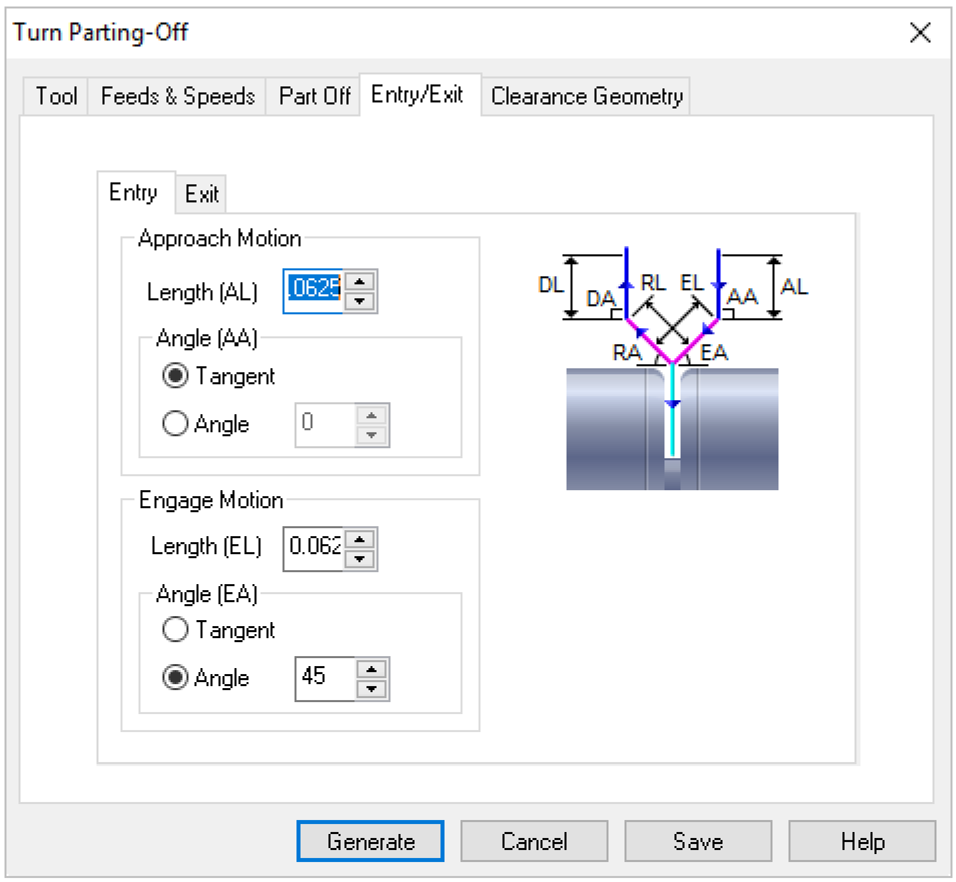
Dwell

Dwell - This is an optional parameter that allows you to set a machine delay (Dwell) during the Part-Off operation. You can set this to Off (no dwell), Time (enter the number of seconds to Dwell) or Revolutions (enter number of revolutions to Dwell).

8.7.2 Entry/Exit

The following dialog allows you to set Entry/Exit parameters for Turn Part Off operations. Entry and Exit determines the way in which tool enters and leaves the part geometry. TURN Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Turn Part Off

Entry Tab

The Entry tab (shown in the dialog box above) consists of Approach and Engage. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

The approach can be either Tangential or at an angle to the Engage motion. This is followed by the engage motion that can be Tangential or at an angle.

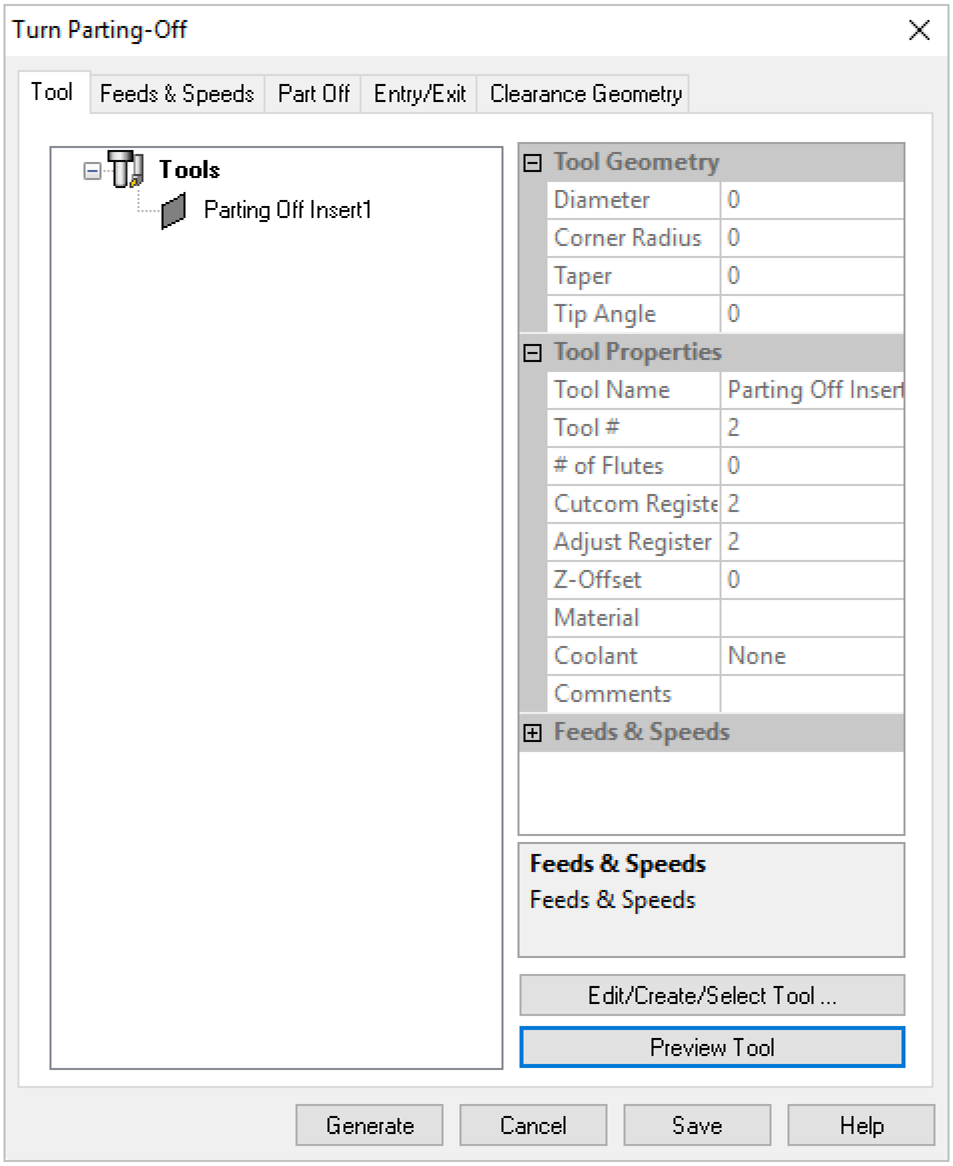
Exit Tab

Similarly the Exit motion consists of a Retract motion followed by a departure motion. The retract motion can be either Tangential or at an angle. The departure motion can be either Tangential or at an angle to the Retract motion.

8.7.3 Tool

The following dialog allows you to select the appropriate insert tool for the Turn Part Off operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Part Off

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

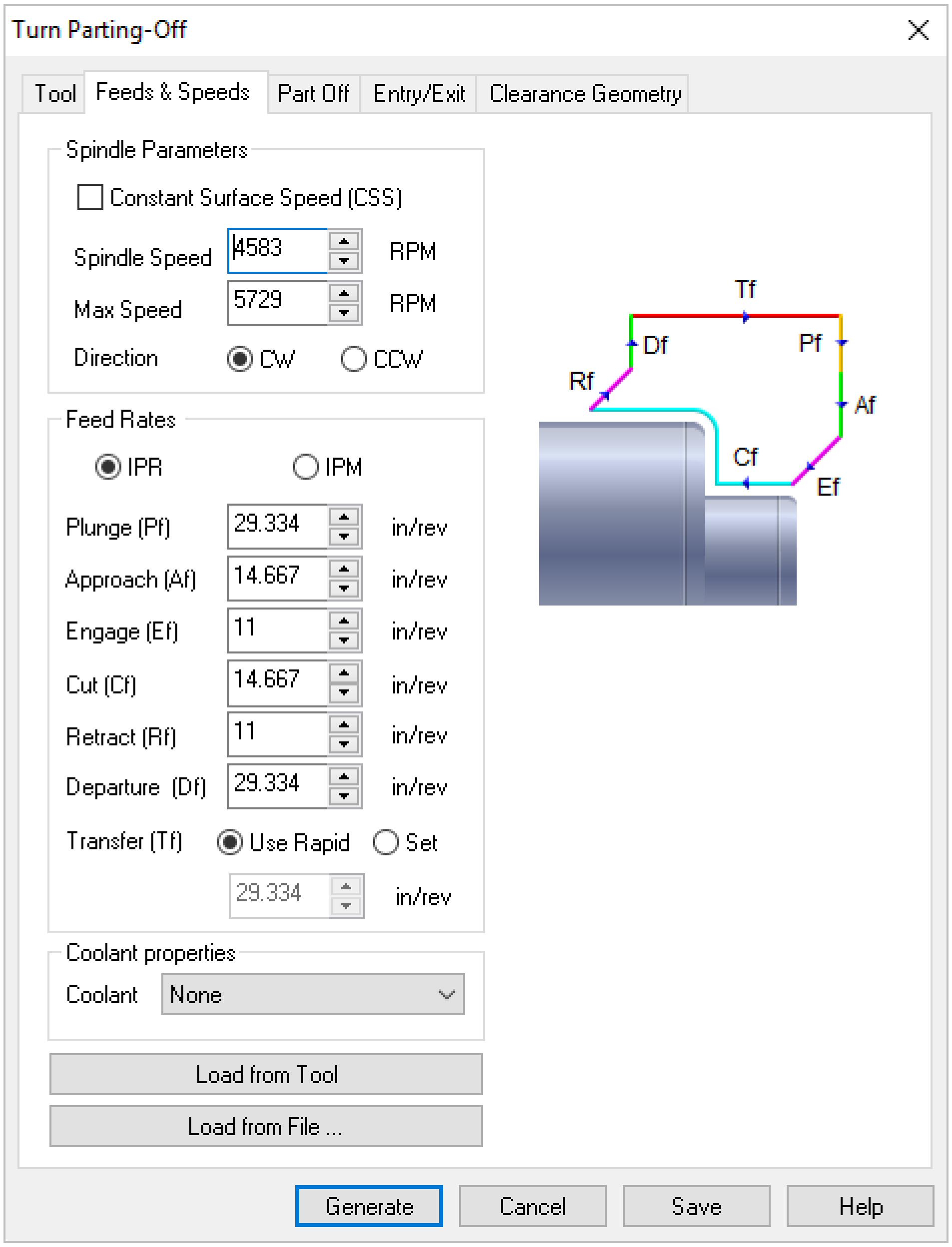
Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.7.4 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Part Off operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Part Off

Spindle Parameters

Constant Surface Speed (CSS)
This is the Spindle Speed Mode. If this box is checked, the mode is set to Constant Surface Speed (CSS). If unchecked, the mode is set to Constant Rotational Speed (CRS).
If the Constant Surface Speed is checked, the controller would automatically calculate and adjust the spindle speed based on the current diameter of the work-piece. If this calculated spindle speed is greater than the maximum spindle speed specified in your post, the spindle speed would be reduced to the maximum speed. Refer to the Spindle section of the Post-Processor Generator to ensure your Spindle Mode is set correctly.

Spindle Speed
his is the rotational speed of the spindle expressed in RPM.

Surface Speed
Surface speed is set in units/min when Constant Surface Speed is selected. This is only applicable for turning inserts.

Max Speed
The maximum rotational speed of the spindle, in RPM. This is only applicable for turning inserts.

Direction
This determines the direction of spindle rotation and can be set to Clockwise or Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)

The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

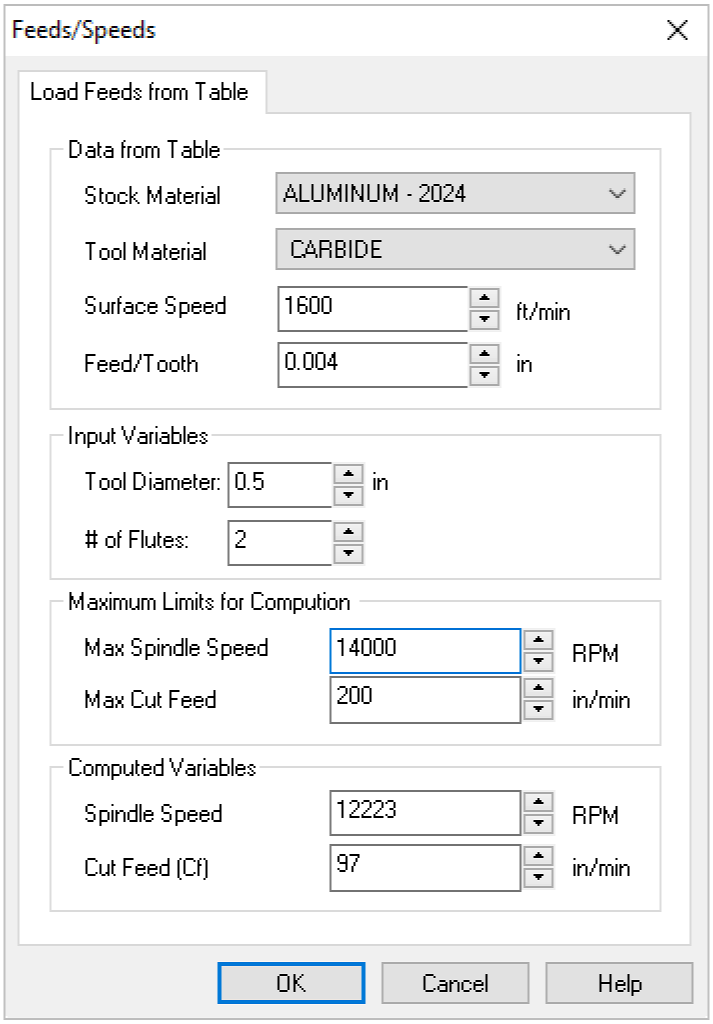
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

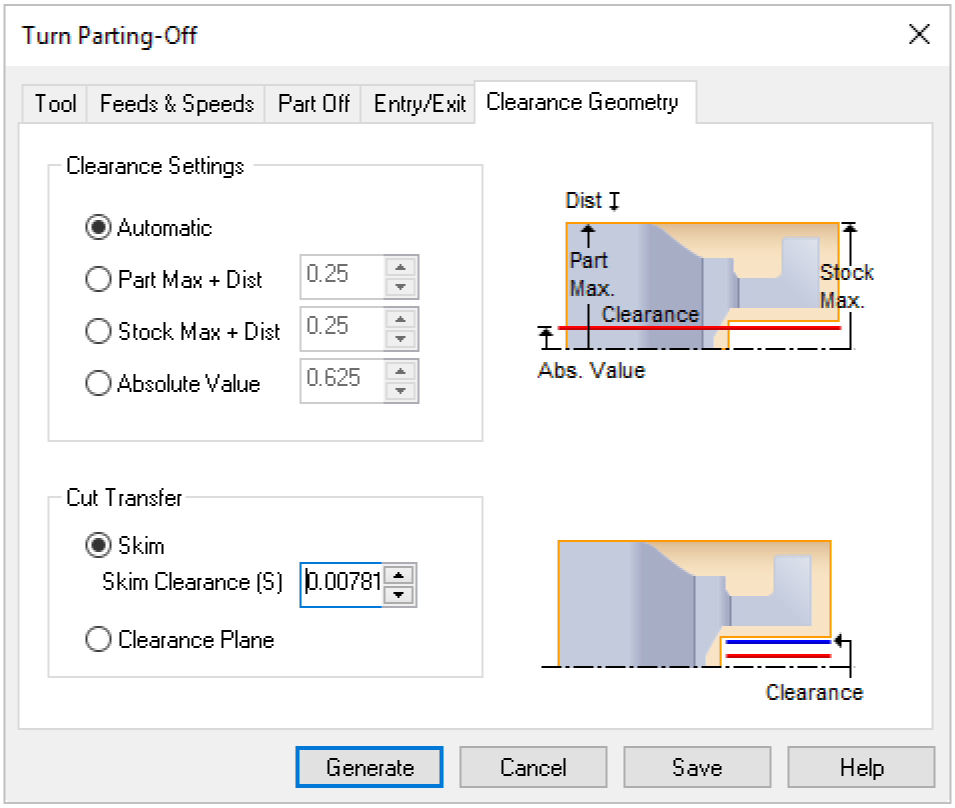
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

8.7.5 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the Turn Part Off operation. In this tab, Clearance Settings and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Geometry tab



Dialog Box: Clearance Geometry tab, Turn Part Off

Clearance Settings

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Cut Transfer

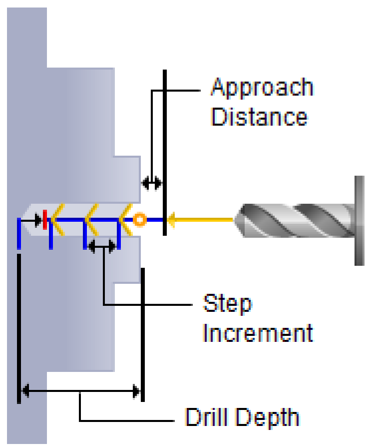
You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

Creating Hole Making Operations

This section details the hole making operation types that can be created.

9.1 Drilling

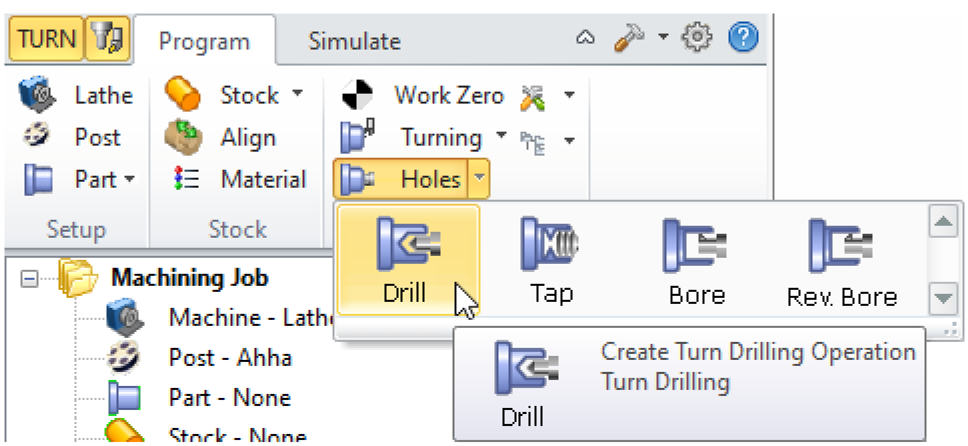
The Drill cycle is used to cut holes in the part. The drilling toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Drill operation.. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



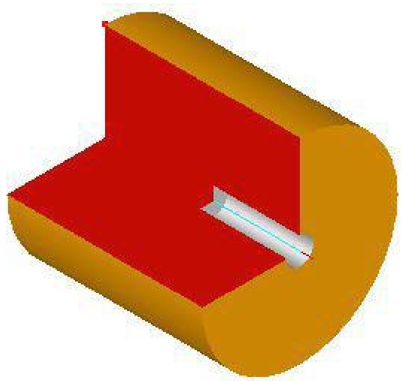
Drilling, Turn

Drill Menu Selection

The Drilling toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Drill operation.



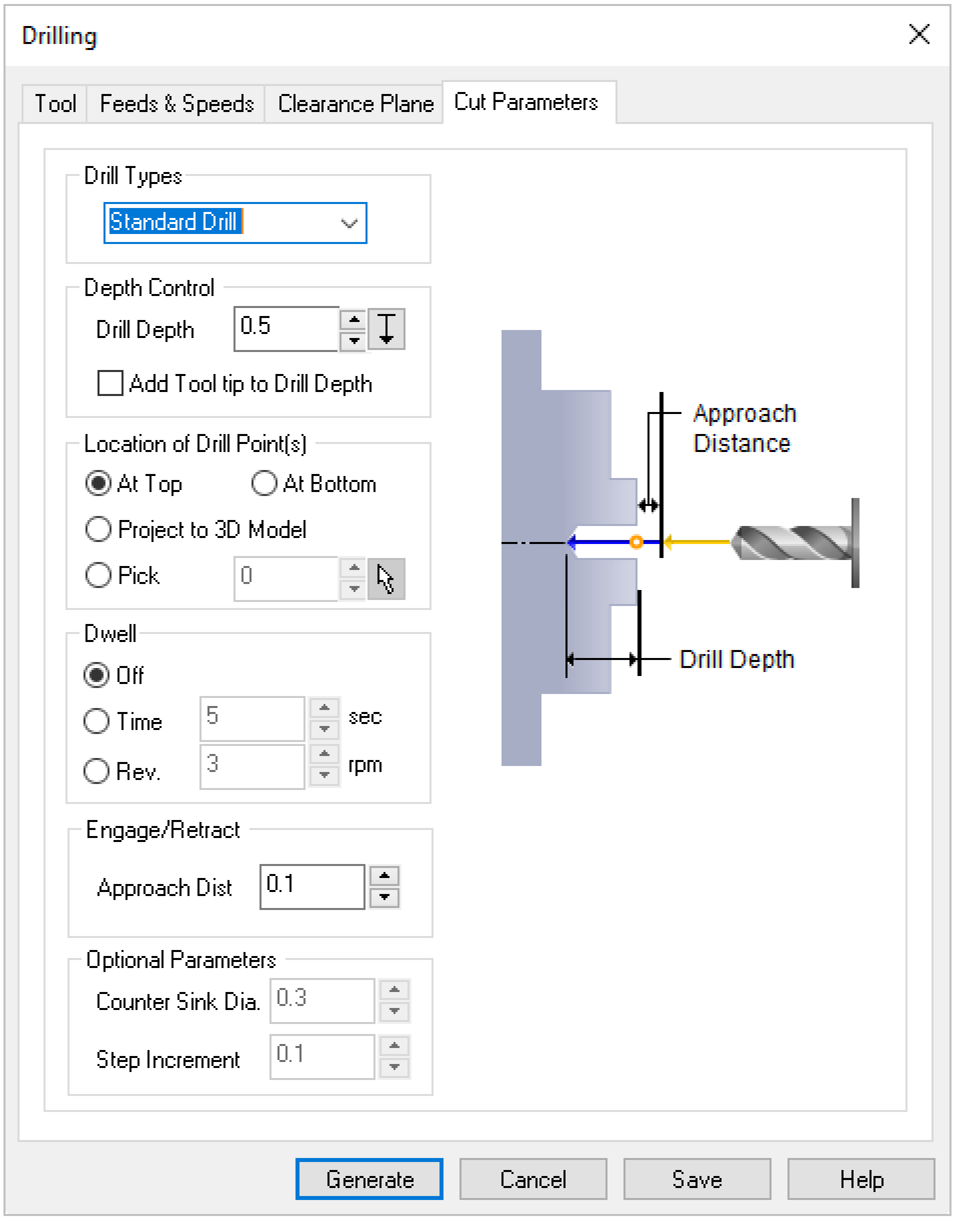
Drilling Menu Item



Drilling Example

Dialog Box: Drill

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Drill operation. This dialog box is shown below.



Dialog Box: Global Parameters tab, Turn Roughing

The following Drill Types are available

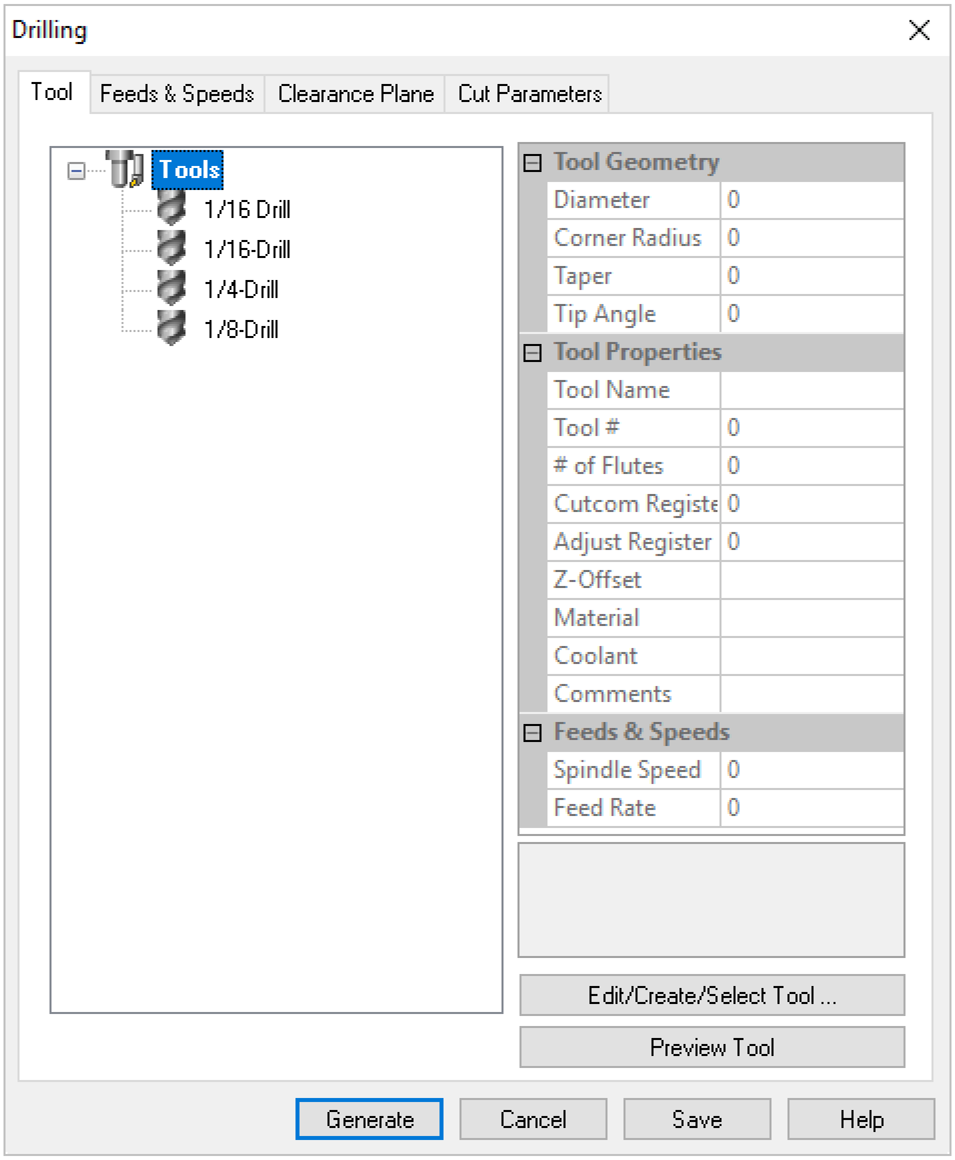
- Standard
  Used for holes whose depth is less than three times the tool diameter.
- Deep
  Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.
- Counter Sink
  Cuts an angular opening at the end of the hole.

- Break Chip
  Similar to Deep drilling, but the tool retracts by a set clearance distance.
- User Defined Drill1 and Drill2
  Allows for the definition of user defined drill types.

9.1.1 Tool

The following dialog allows you to select the appropriate tool (Standard Drill, Center Drill or Reamer) for the Turn Drill operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Drilling

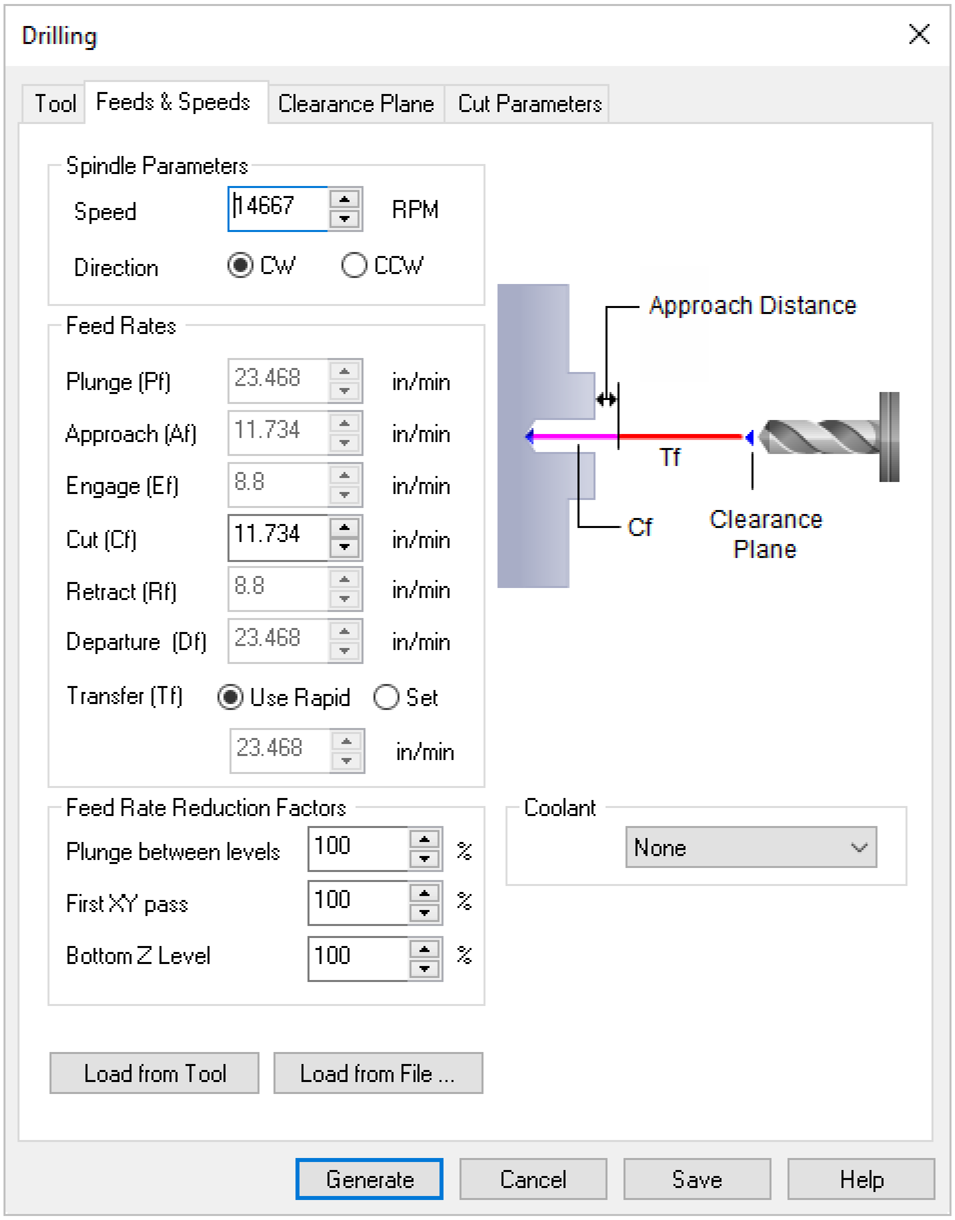
Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.1.2 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Drilling operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds for Hole Machining operations

Spindle Parameters

Speed

This is the rotational speed of the spindle expressed in RPM.

Direction

This determines the direction of spindle rotation and can be set to CW Clockwise or CCW Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Feed Rate Reduction Factors

This sets Feed Rate Reduction Factors for Plunge Between Levels and the First XY pass.

Load from Tool

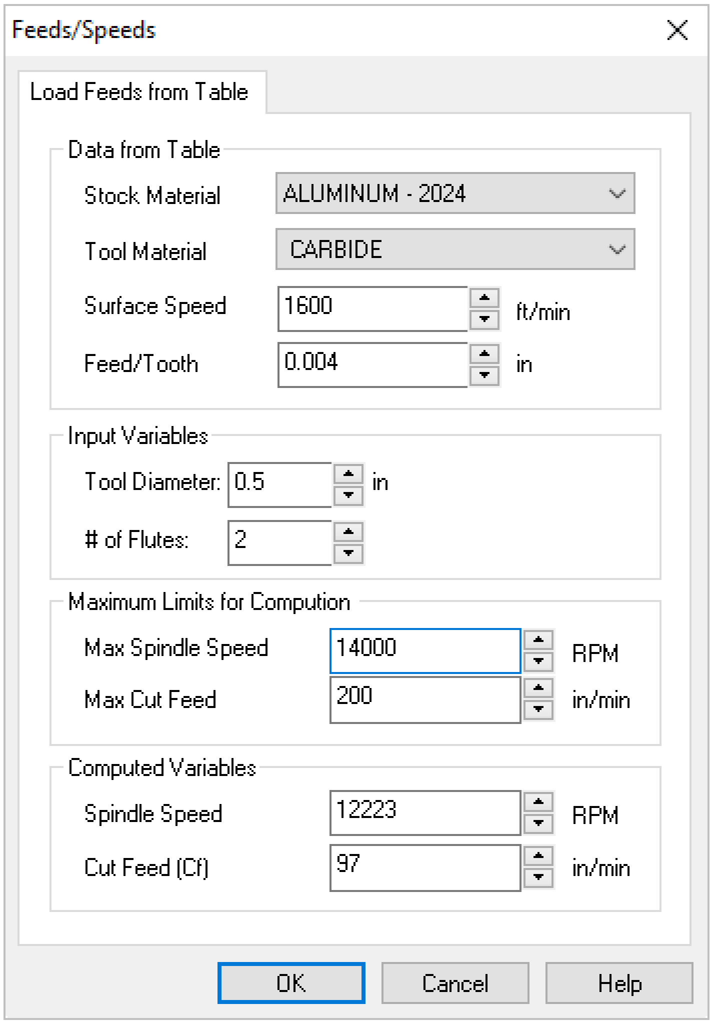
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

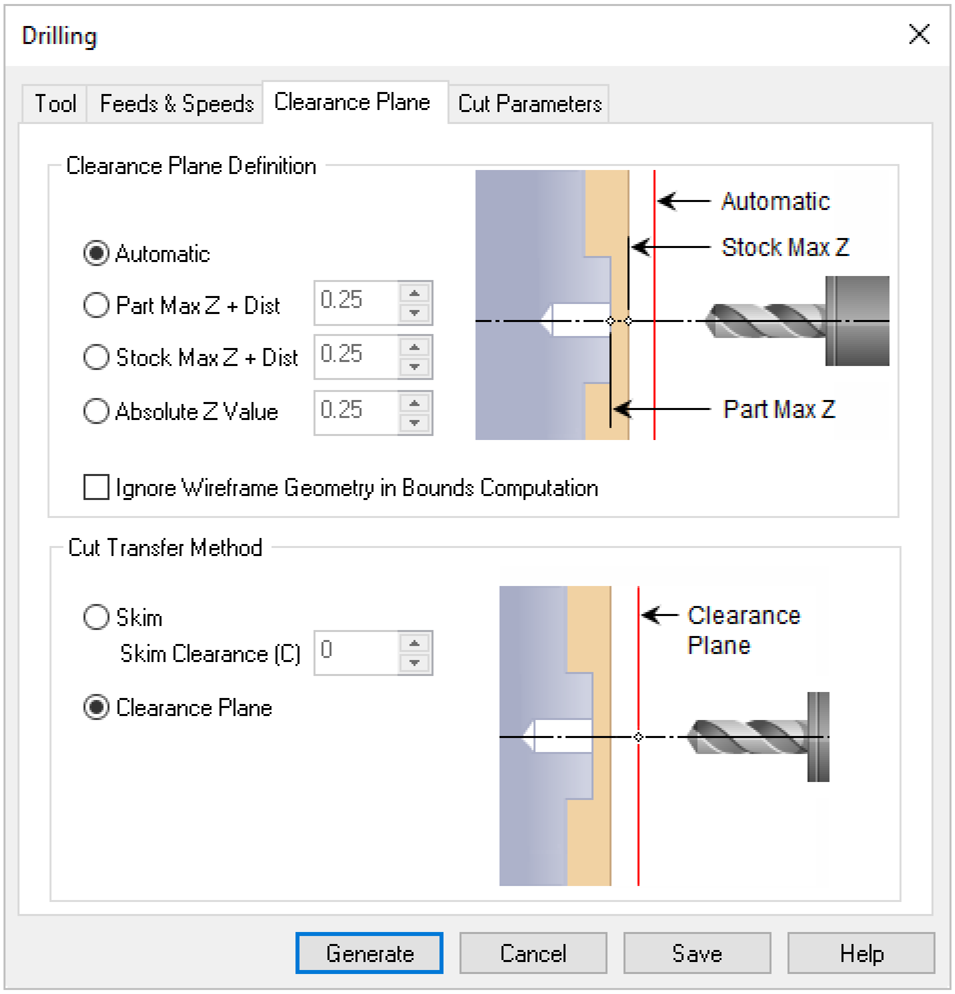
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.1.3 Clearance

The following dialog allows you to select the appropriate Clearance Plane for the Turn Drilling operation. In this tab, Clearance Plane Definition and Cut Transfer Method parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Plane tab



Dialog Box: Clearance Plane tab, Turn Reverse Boring

Clearance Plane Definition

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Ignore Wireframe Geometry in Bounds Computation
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

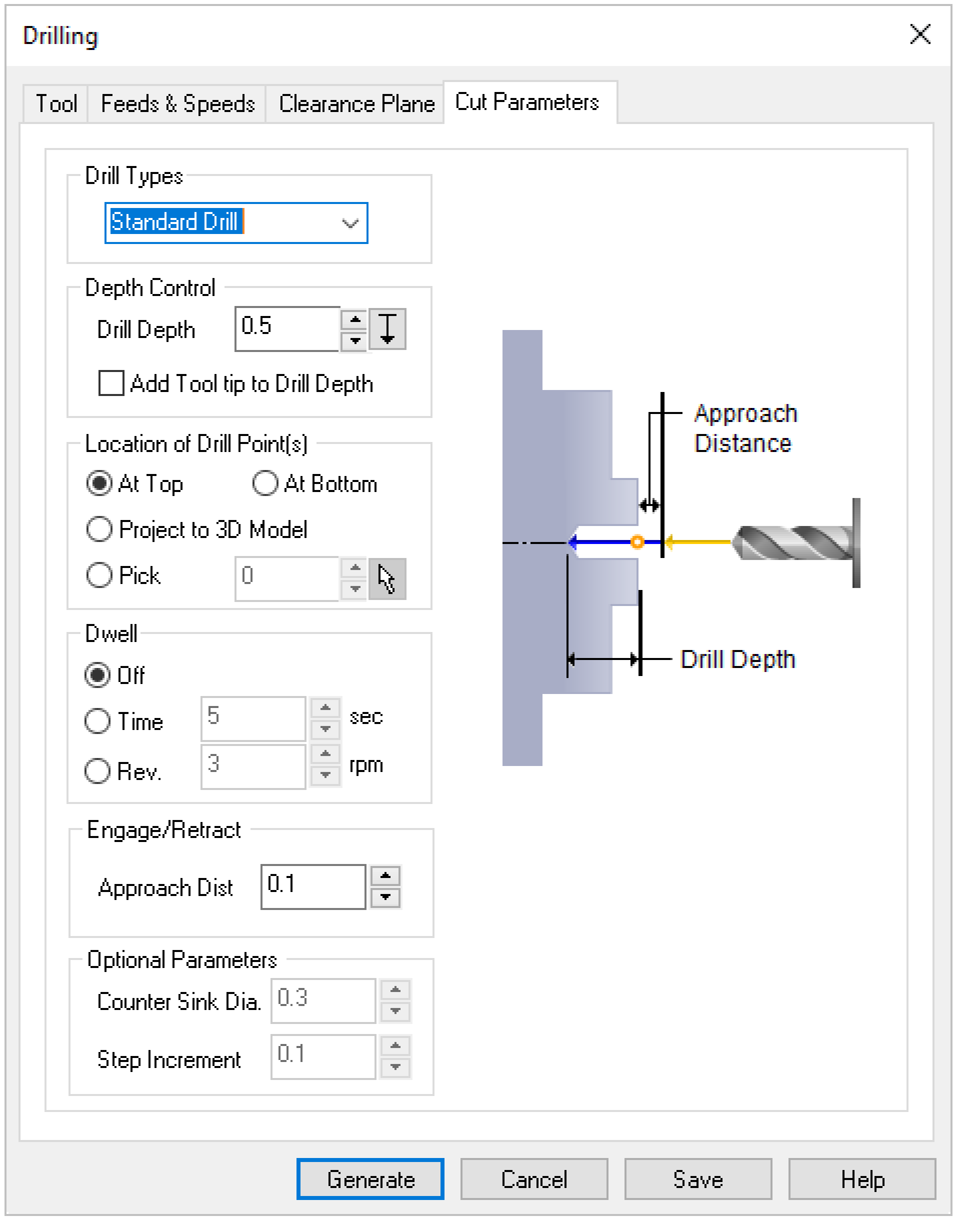
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

9.1.4 Cut Parameters

The following dialog allows you to set the Cut Parameters for Turn Drilling operations. You can set the Drill Type, Depth Control, Location, Dwell and other Optional Parameters via this dialog box.

Dialog Box: Cut Parameters tab



Dialog Box: Global Parameters tab, Turn Roughing

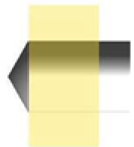
Drill Types

Drill Types - This defines the type of drill cycle to use: Standard Drill (when depth is < than 2x dia.), Deep Drill (when depth is > 2x dia.), Breakchip Drill (with Step Increment), Countersink Drill, (with Counter Sink Dia.) or User Defined Drill1-4 (see Cycles section of the Post Process Generator).

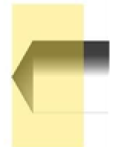
Depth Control

Depth refers to the actual hole depth you wish to achieve.

Add Tool Tip to Drill Depth - Check this box to compute the height of the drill tool taper and add it to the total Drill Depth.



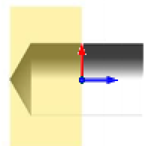
Checked



Unchecked

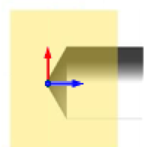
Location of Drill Point(s)

Select this option if drilling should start at your Work Zero. The Drill Depth is then subtracted to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



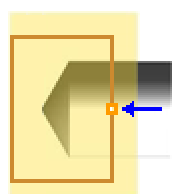
At Top

Select this option if drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



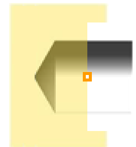
At Bottom

Project to 3D Model
Select this option if you want to project the start point to your 3D model. For example, if your stock is longer than your part but you want the Drill Depth to begin at the face of your part instead of the face of the stock. An actual point is not required. The location is determined automatically by your part model.



Project to 3D Model

Select this option if you want to specify the coordinates of point where the drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location. You can select the Pick button and then select a point on your part. It's coordinate will be calculated automatically.



Pick Top

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

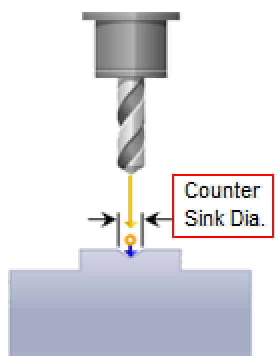
Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in the Z axis to the approach plane and then applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

The following additional parameters are supported:

Under Optional Parameters, Counter Sink Dia. is only required for the Counter Sink Drill Type operation. The system will automatically calculate the drill depth to achieve the required Counter Sink Diameter. Optionally, you can use the pick button to select a circle to define the diameter of the Counter Sink.

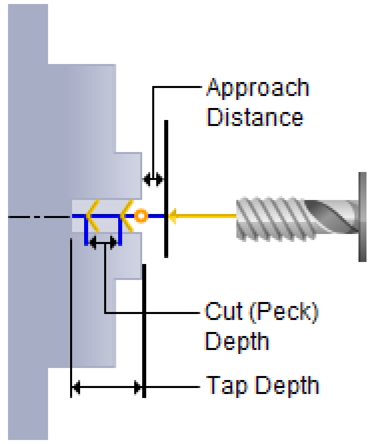


Countersink Diameter

Step Increment - If Drill Type is set to either Deep Drill or Breakchip Drill, enter the Step Increment in the field provided. The tool will retract after each step increment completely to clean out the chips.

9.2 Tapping

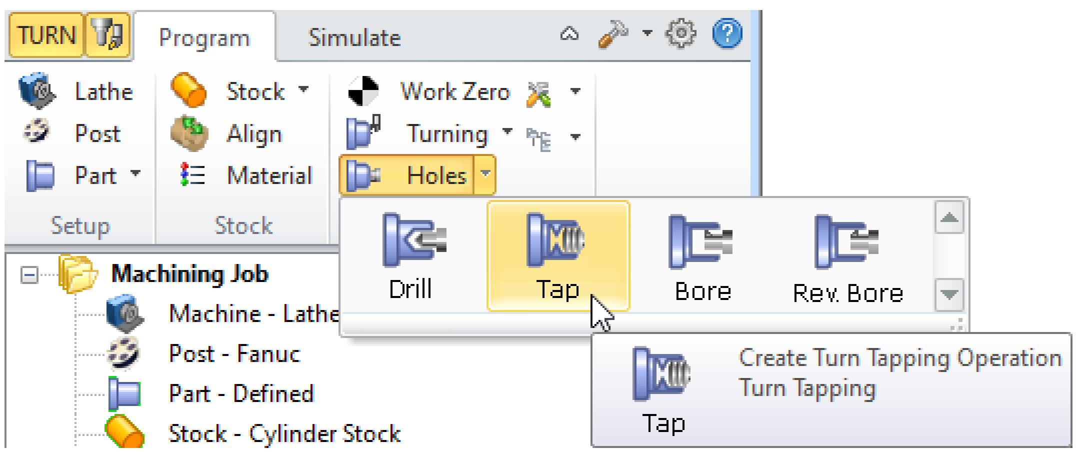
These operations are used to create tapped holes in the part. The Tapping toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Tap operation. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



Tapping, Turn

Tapping Menu Selection (TURN Module)

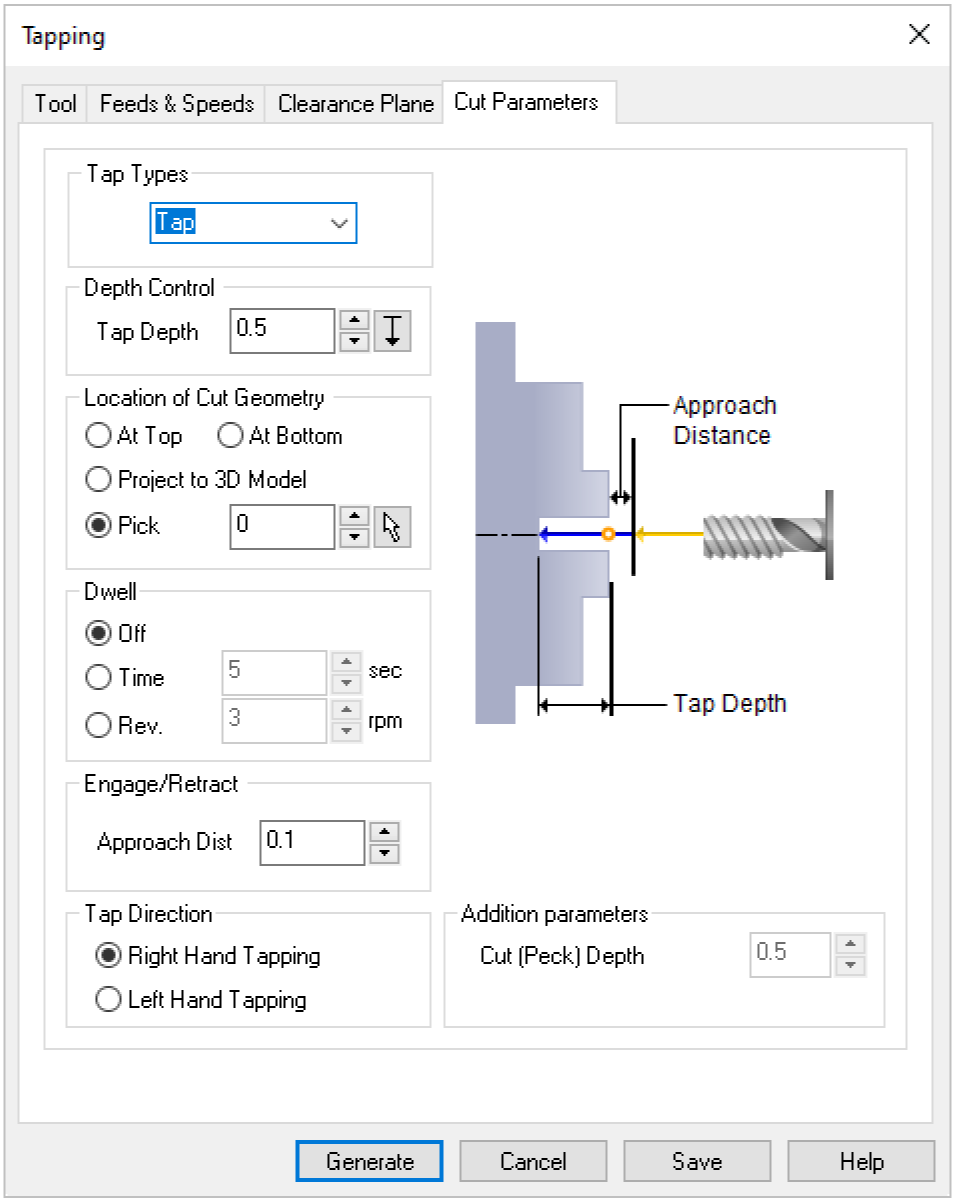
The Tapping toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Tap operation.



Tapping Menu Item

Dialog Box: Tap

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Tap operation. This dialog box is shown below.

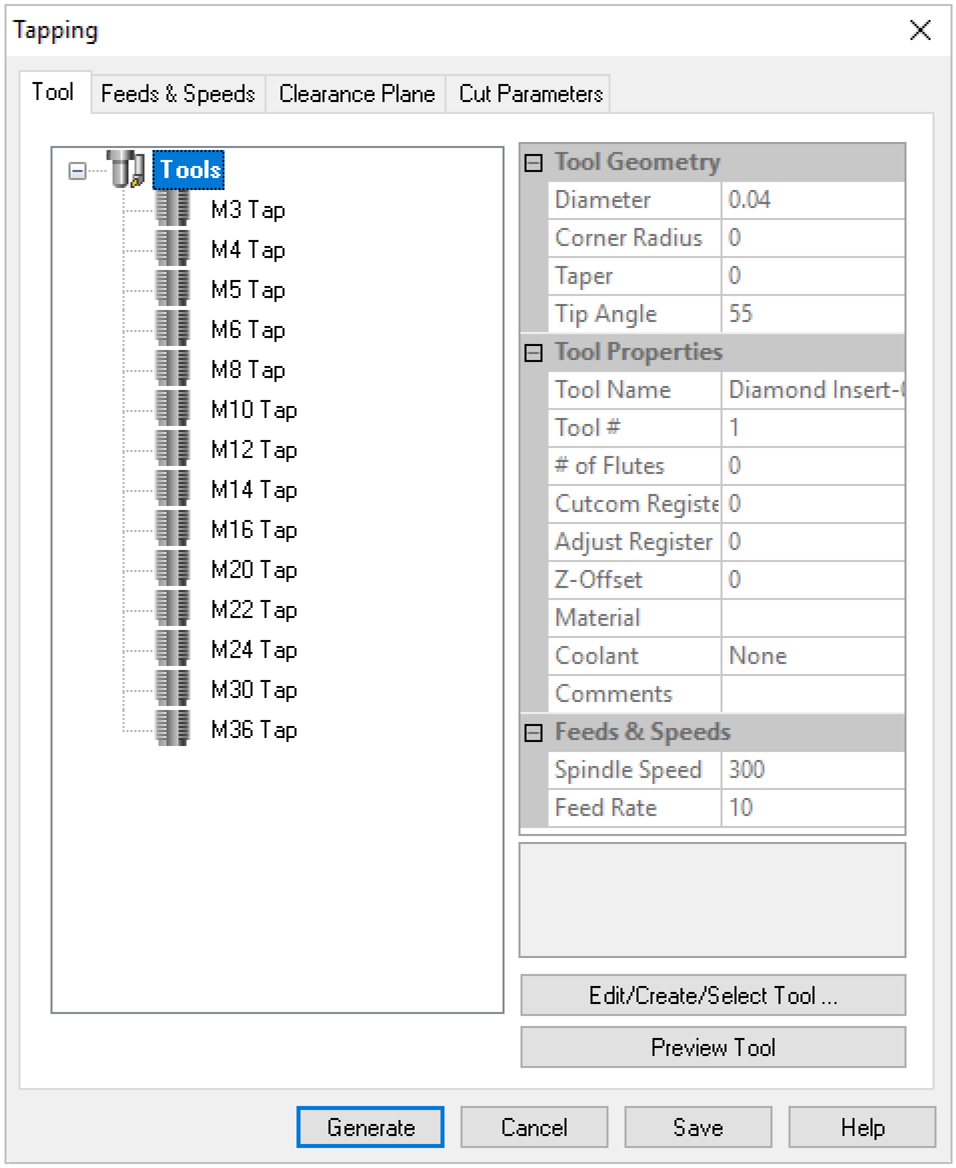


Dialog Box: Cut Parameters tab, Turn Tapping

9.2.1 Tool

The following dialog allows you to select the appropriate Tap tool for the Turn Tap operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Tapping

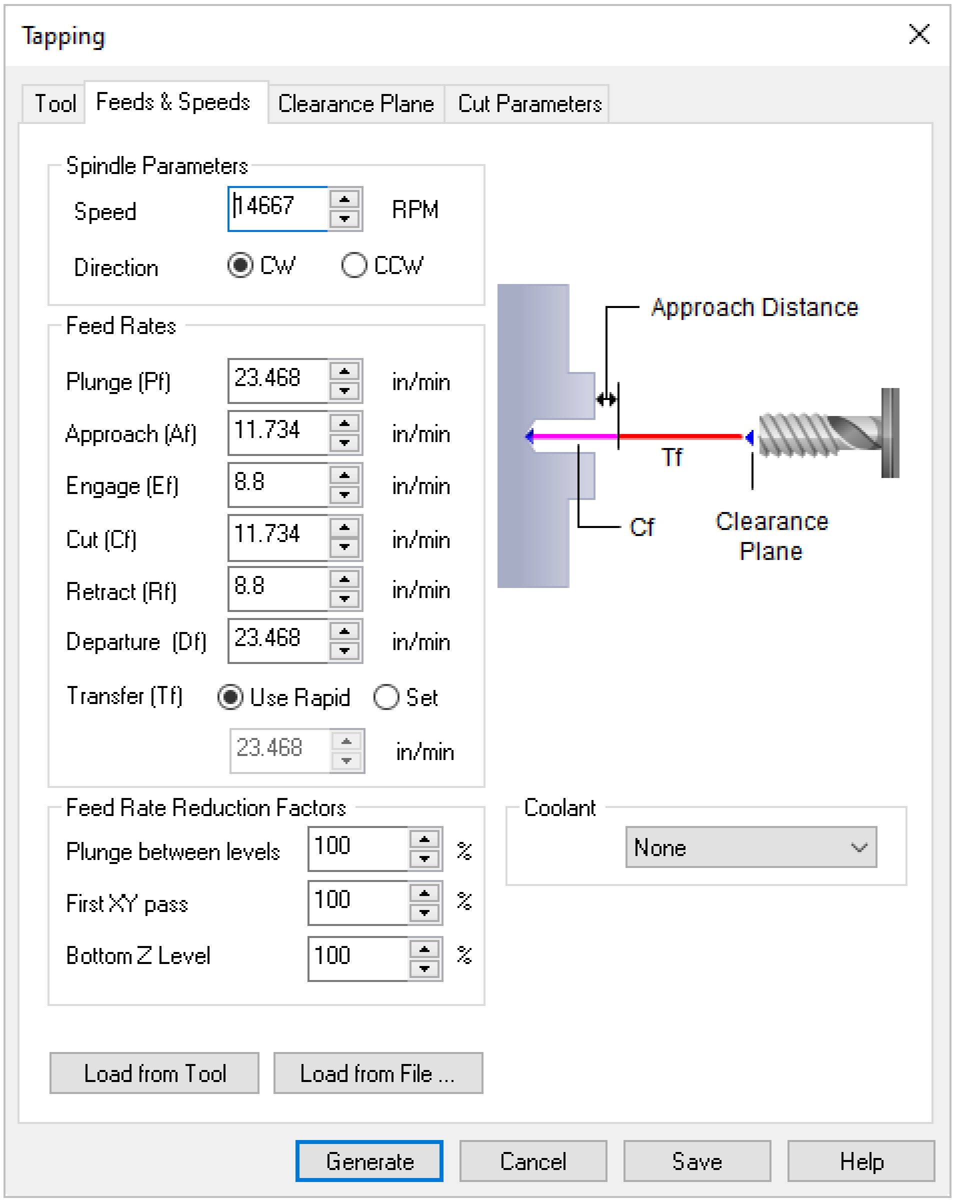
Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.2.2 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Drilling operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Tap

Spindle Parameters

Speed

This is the rotational speed of the spindle expressed in RPM.

Direction

This determines the direction of spindle rotation and can be set to CW Clockwise or CCW Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Additional Feed Rate information for Tapped Holes

For tap operations, the feedrate can be computed in different ways. This depends on what is expected by the controller.

- Spindle Speed x Thread Pitch
- Thread Pitch
- 1/ Thread Pitch
- Cut Feedrate

The post needs to be setup with the appropriate variable to output the feedrate. The Thread Pitch is defined under the Tool tab and Spindle Speed is set under the Feeds & Speed tab shown above.

Use the following macro's in the post to output the feedrate

- [CYCL_IPR] – Spindle Speed x Thread Pitch
- [CYCL_TPI] – Thread Pitch
- [CYCL_1/TPI] – 1/ Thread Pitch

- [CUT_FEED] – Cut Feedrate

Feed Rate Reduction Factors

This sets Feed Rate Reduction Factors for Plunge Between Levels and the First XY pass.

Load from Tool

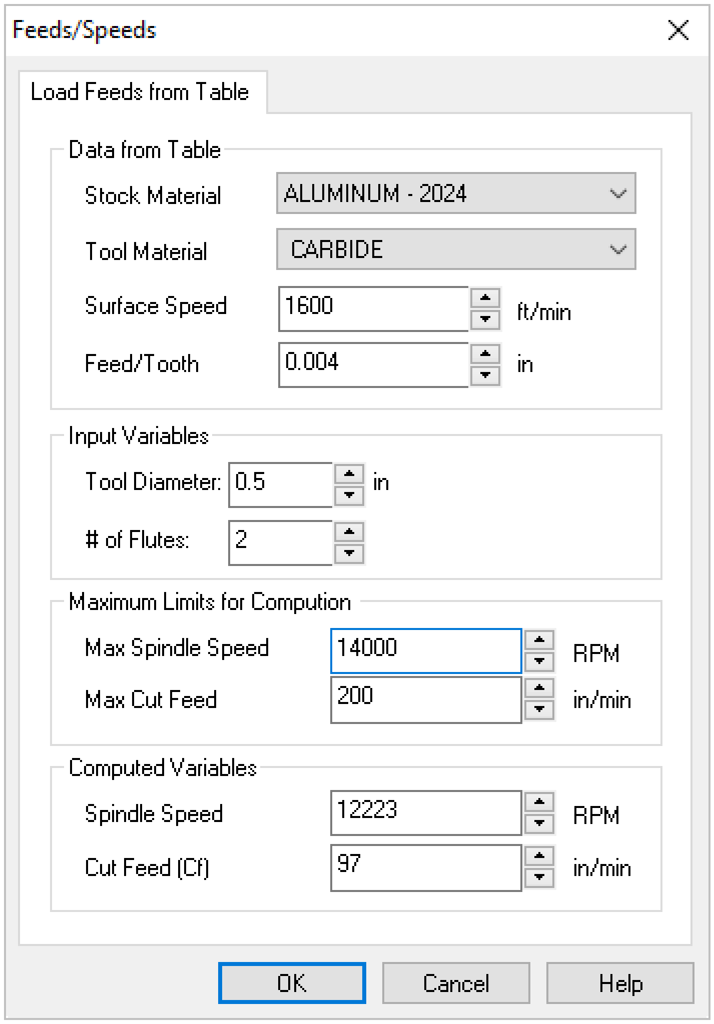
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

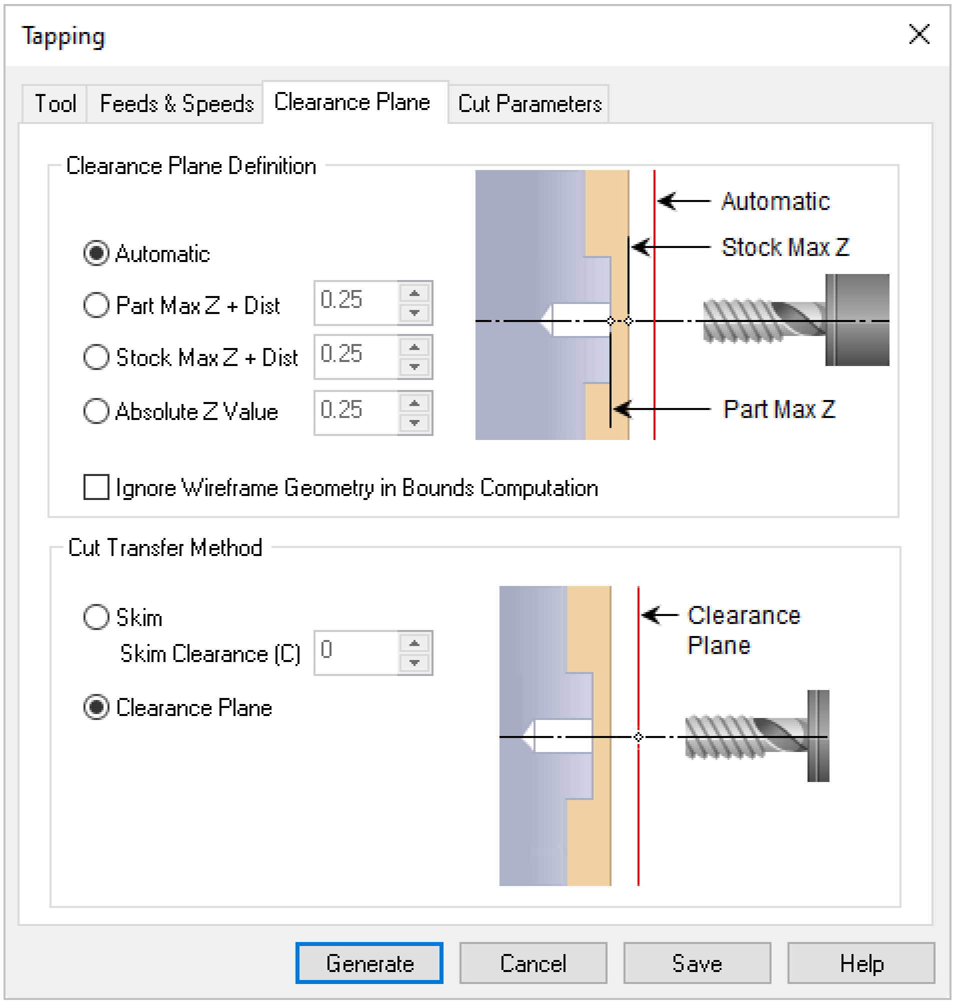
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.2.3 Clearance

The following dialog allows you to select the appropriate Clearance Plane for the Turn Tapping operation. In this tab, Clearance Plane Definition and Cut Transfer Method parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Plane tab



Dialog Box: Clearance Plane tab, Turn Tapping

Clearance Plane Definition

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist
Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist
Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value
Uses the specified distance for clearance height.

> For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Ignore Wireframe Geometry in Bounds Computation
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

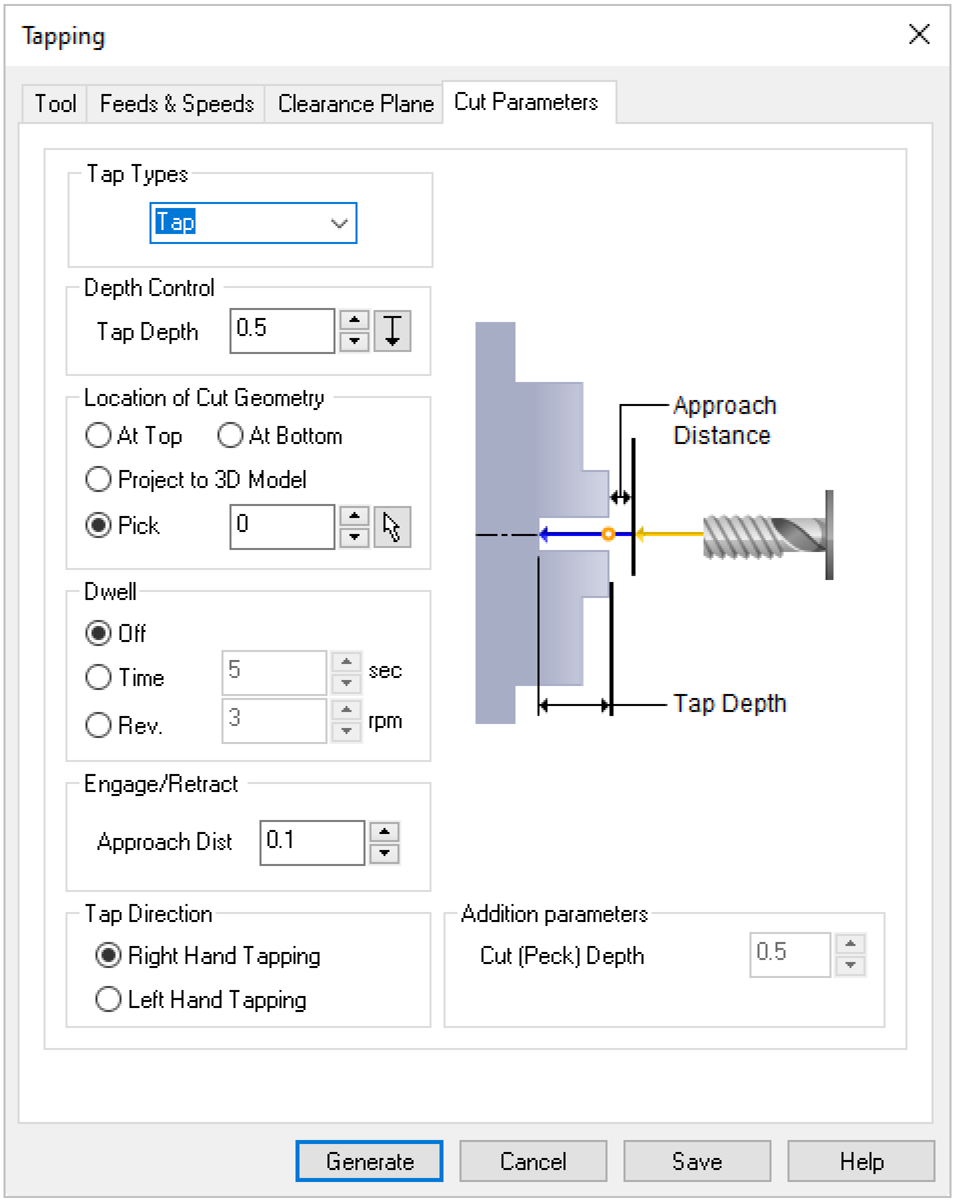
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

9.2.4 Cut Parameters

The following dialog allows you to set the Cut Parameters for Turn Tapping operations. You can set the Tap Type, Depth Control, Location, Dwell and other parameters via this dialog box.

Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Turn Tapping

Tap Types

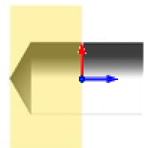
Tap Types - This defines the type of tap cycle to use: Standard Tap (for canned tap cycles) or Peck Tapping (see Cut Peck Depth field below) and Tap 1-4 (see Cycles section of the Post Process Generator for user defined tap cycles 1-4).

Depth Control

Tap Depth - Enter the depth for the current drill operation. You can select the Pick button to pick a point on your part model that defines the drill depth.

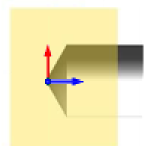
Location of Cut Geometry

Select this option if drilling should start at your Work Zero. The Drill Depth is then subtracted to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



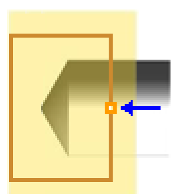
At Top

Select this option if drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



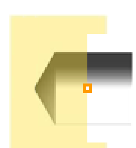
At Bottom

Project to 3D Model
Select this option if you want to project the start point to your 3D model. For example, if your stock is longer than your part but you want the Drill Depth to begin at the face of your part instead of the face of the stock. An actual point is not required. The location is determined automatically by your part model.



Project to 3D Model

Select this option if you want to specify the coordinates of point where the drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location. You can select the Pick button and then select a point on your part. It's coordinate will be calculated automatically.



Pick Top

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in the Z axis to the approach plane and then applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Tap Direction

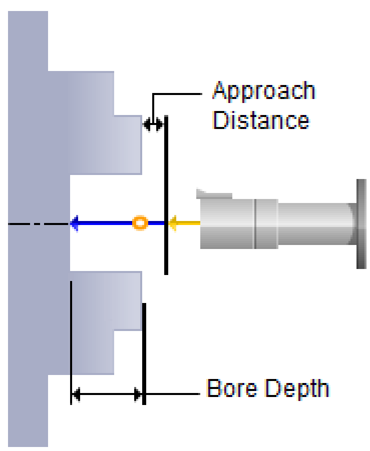
Tap Direction - This allows you to select between clockwise and counter clockwise tap. Right Hand Tapping creates a clockwise tap cycle. Left Hand Tapping creates a counter clockwise tap cycle.

Additional Parameters

Cut Peck Depth
Used to set the peck depth increment. This needs to be specified for Peck Tapping. The tool retracts after each increment completely to clean out all the chips.

9.3 Boring

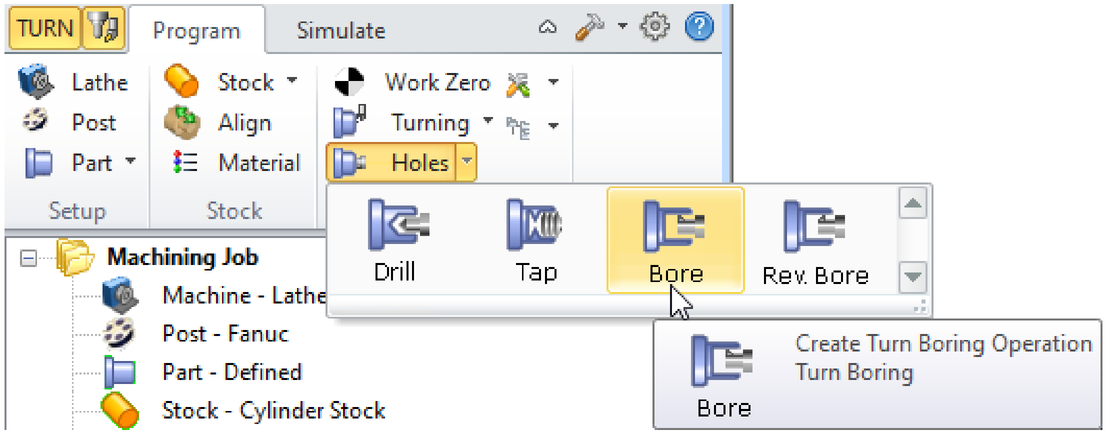
These operations are used to form shapes inside of a hole. Bore Types including Drag, No Drag and Manual are supported. Choose a selection from below for more information. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



Boring, Turn

Reverse Bore Menu Selection (TURN Module)

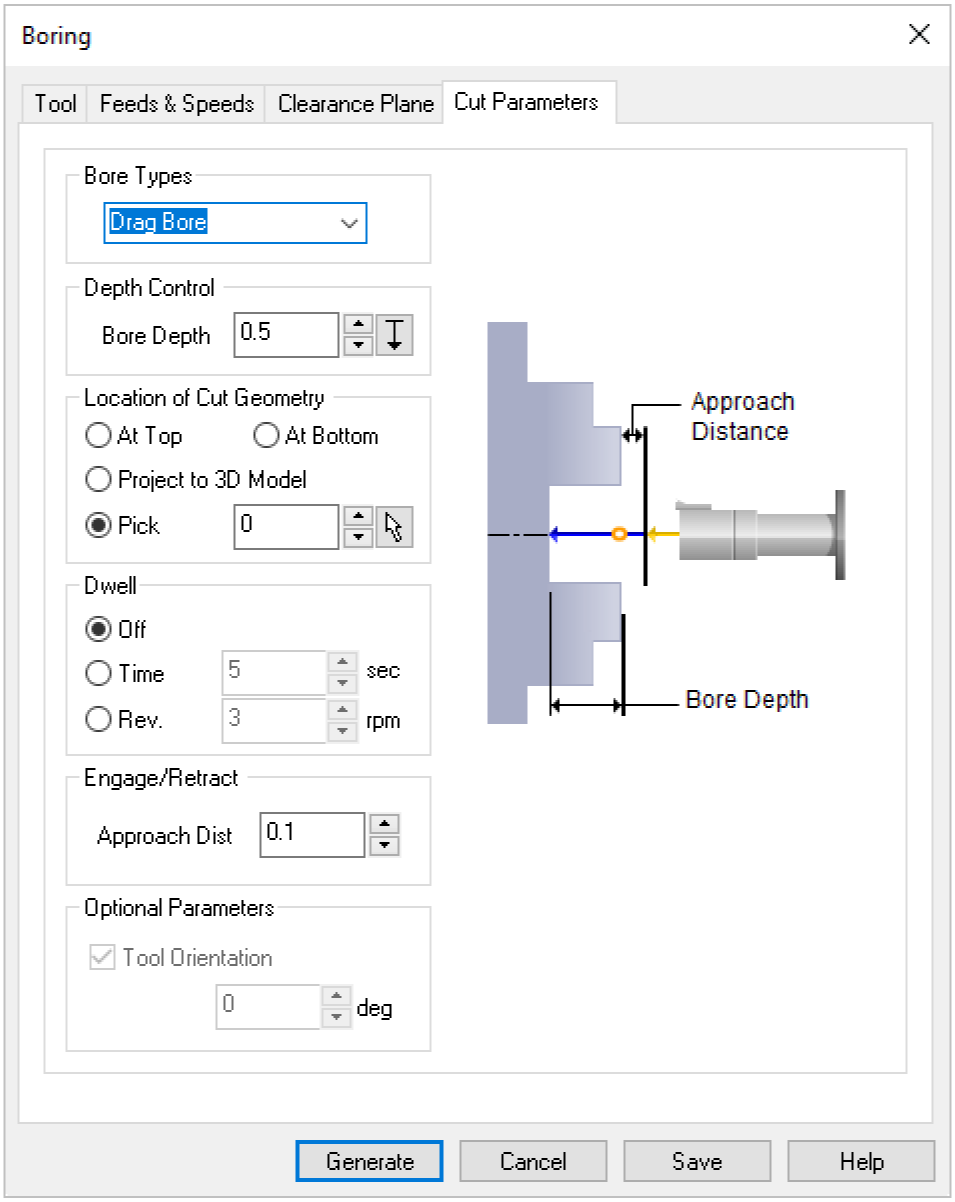
The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Rev. Bore operation.



Turn Boring Menu Item

Dialog Box: Boring

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Bore operation. This dialog box is shown below.



Dialog Box: Cut Parameters tab, Turn Boring

The following Bore Types are available

The following boring cycles are available:

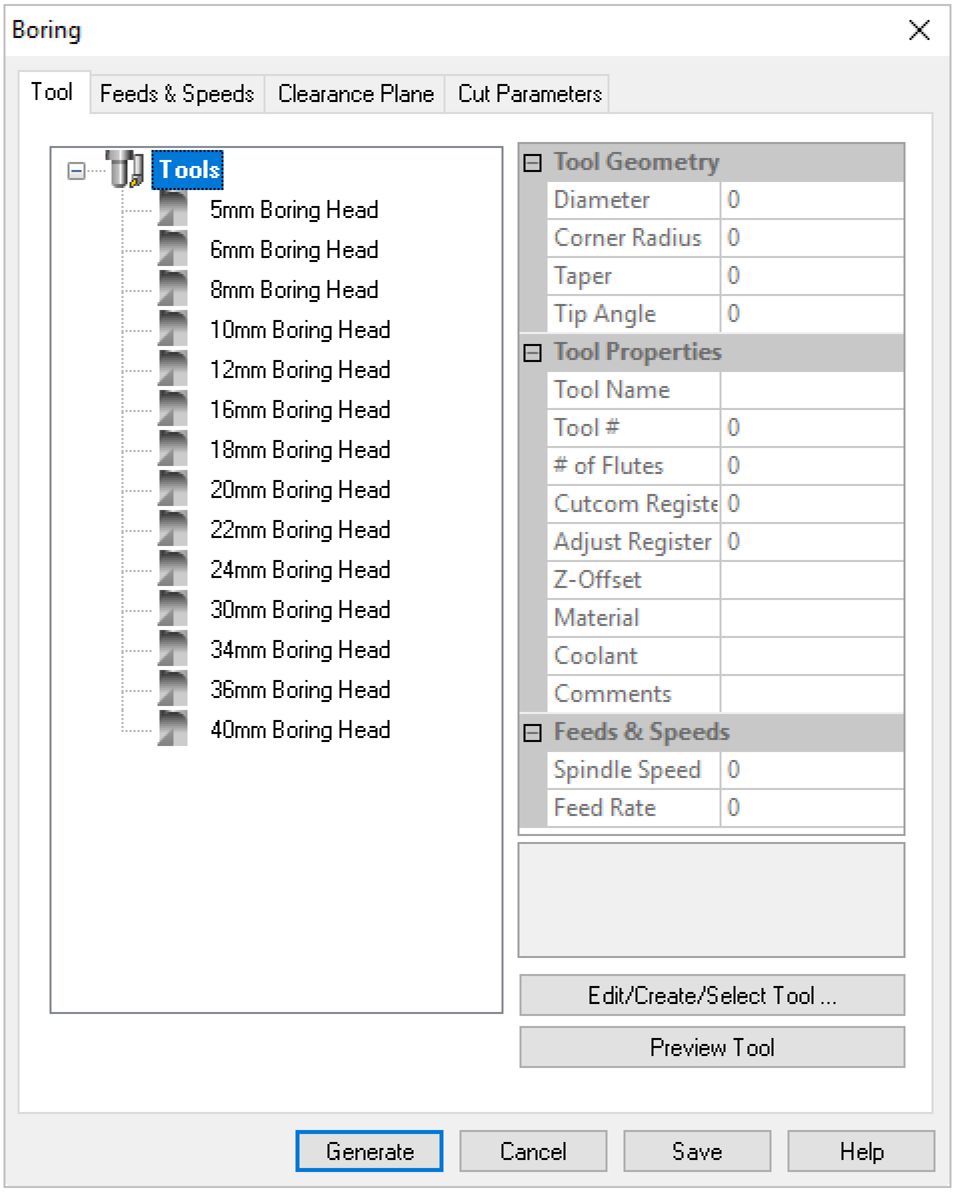
- **Drag**
  The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.
- **No Drag**
  The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

- **Manual**
  The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.
- **User Defined Bore1 and Bore2**
  Allows for the definition of user defined bore types.

9.3.1 Tool

The following dialog allows you to select the appropriate Bore tool for the Turn Bore operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Boring

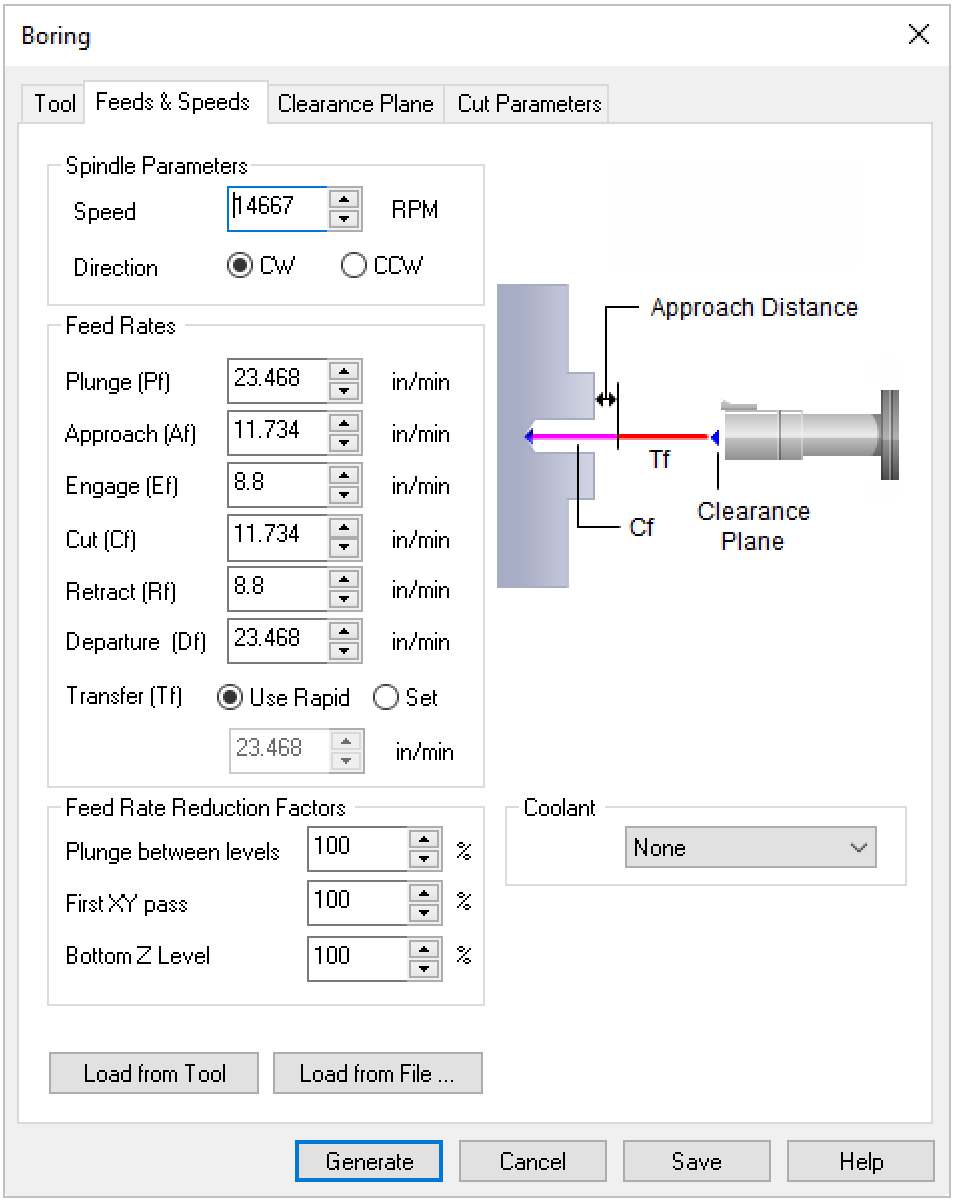
Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.3.2 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Boring operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Boring

Spindle Parameters

Speed

This is the rotational speed of the spindle expressed in RPM.

Direction

This determines the direction of spindle rotation and can be set to CW Clockwise or CCW Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Feed Rate Reduction Factors

This sets Feed Rate Reduction Factors for Plunge Between Levels and the First XY pass.

Load from Tool

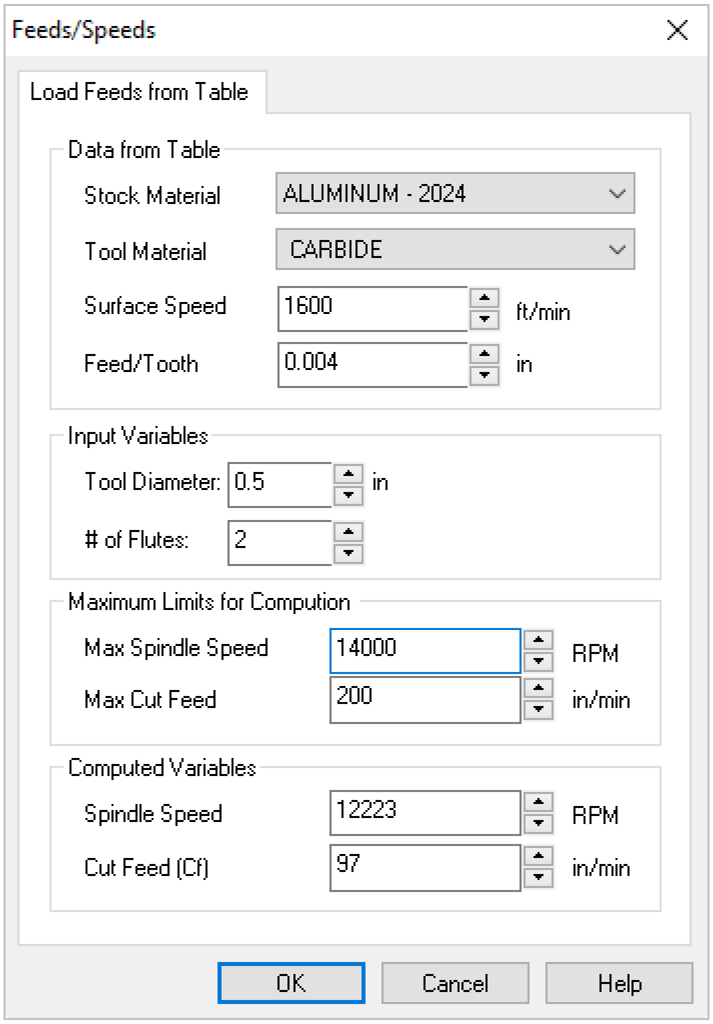
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

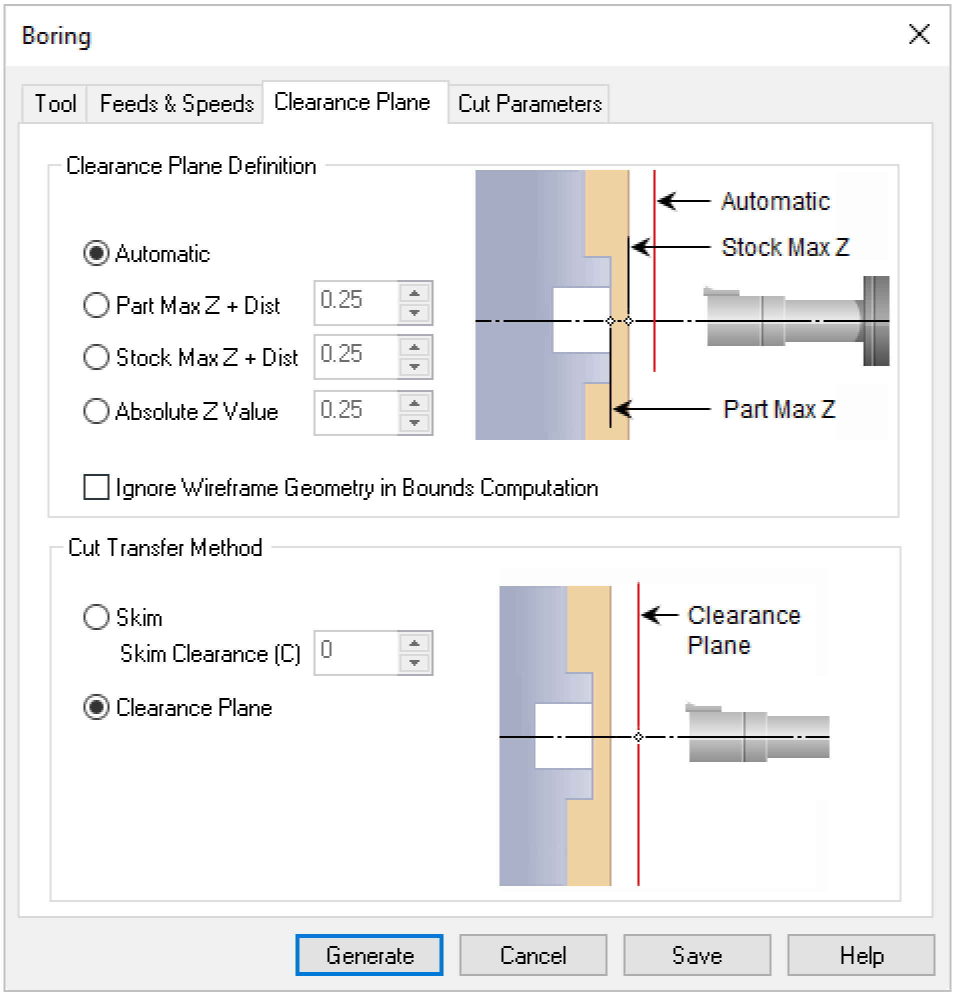
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.3.3 Clearance

The following dialog allows you to select the appropriate Clearance Plane for the Turn Boring operation. In this tab, Clearance Plane Definition and Cut Transfer Method parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Plane tab



Dialog Box: Clearance Plane tab, Turn Boring

Clearance Plane Definition

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value

Uses the specified distance for clearance height.

For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Ignore Wireframe Geometry in Bounds Computation
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

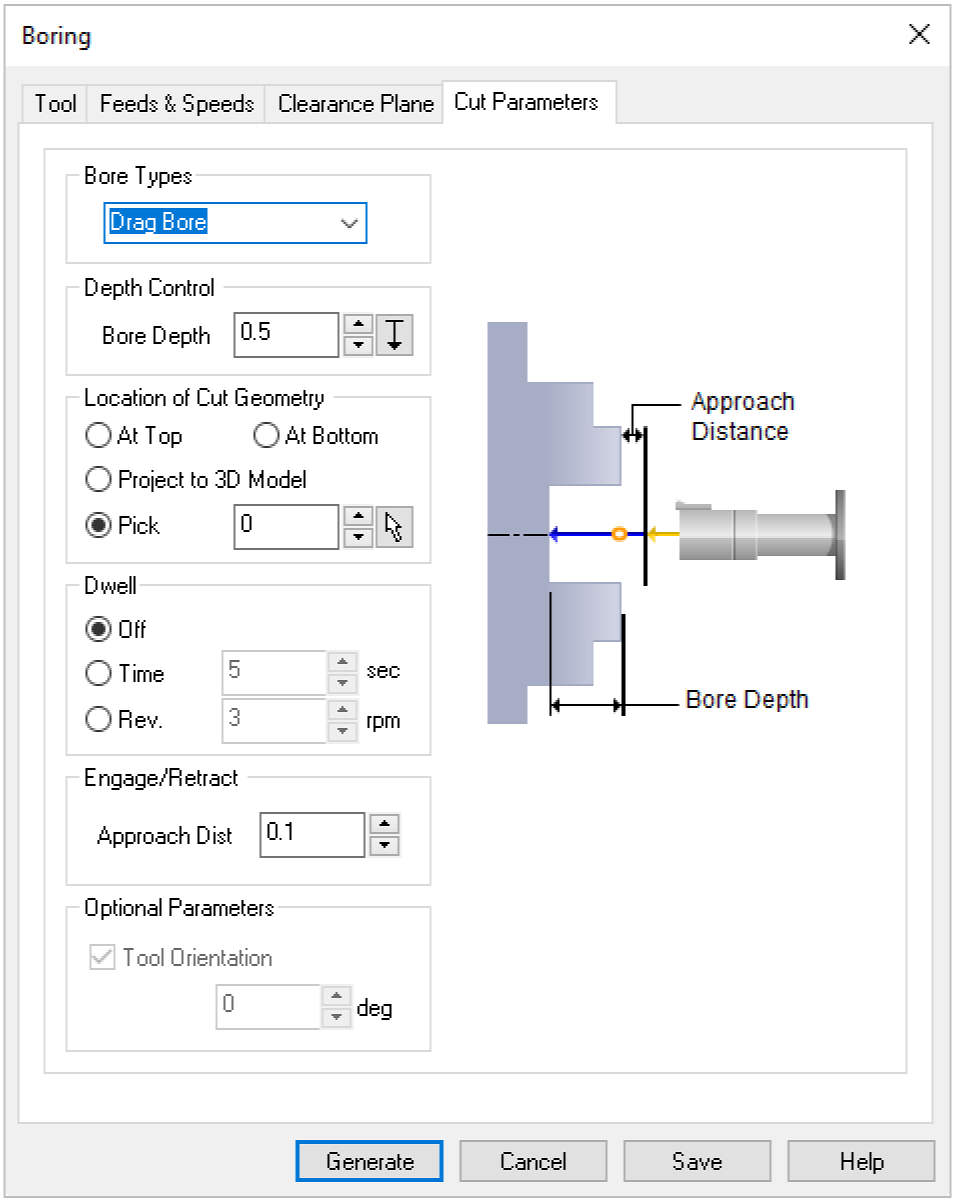
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

9.3.4 Cut Parameters

The following dialog allows you to set the Cut Parameters for Turn Boring operations. You can set the Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box.

Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Turn Boring

Bore Types

The bore types supported are:

Drag Bore
The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed.

No Drag Bore
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Tool Orientation is supported. See Optional Parameters listed below.

Manual Bore

The tool traverses to the programmed point; feeds to the specified depth at the controlled feed rate; and then stops motion for a manual retract.

User Defined Bore1 / User Defined Bore2

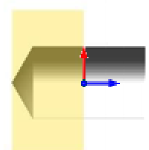
Defined using the Post-Processor Generator for your selected post definition.

Depth Control

Bore Depth - Enter the depth for the current drill operation. You can select the Pick button to pick a point on your part model that defines the drill depth.

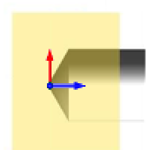
Location of Cut Geometry

Select this option if drilling should start at your Work Zero. The Drill Depth is then subtracted to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



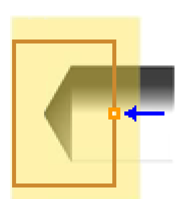
At Top

Select this option if drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



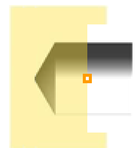
At Bottom

Project to 3D Model
Select this option if you want to project the start point to your 3D model. For example, if your stock is longer than your part but you want the Drill Depth to begin at the face of your part instead of the face of the stock. An actual point is not required. The location is determined automatically by your part model.



Project to 3D Model

Select this option if you want to specify the coordinates of point where the drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location. You can select the Pick button and then select a point on your part. It's coordinate will be calculated automatically.



Pick Top

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

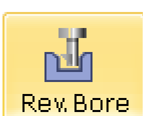
Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in the Z axis to the approach plane and then applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

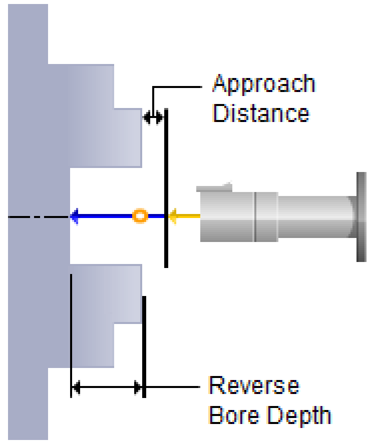
Optional Parameters

Tool Orientation - The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

9.4 Reverse Boring



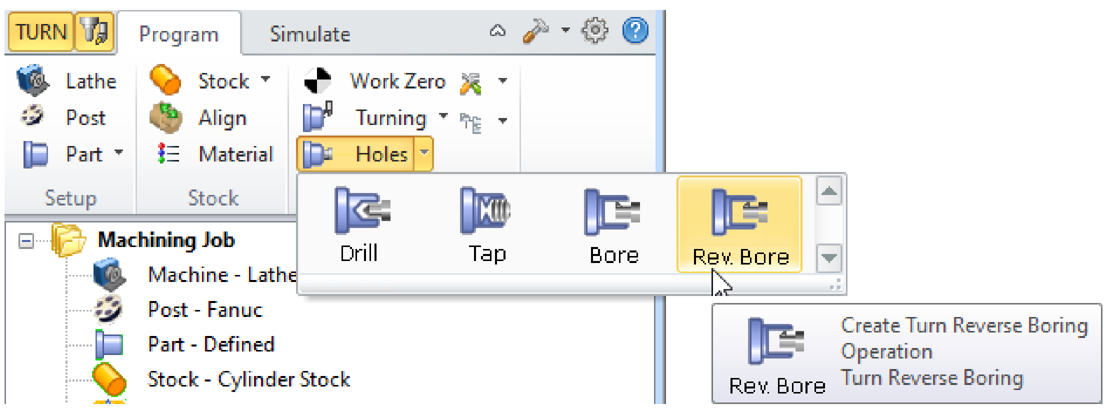
This method is used to reverse-bore a stepped hole that cannot be accessed by regular bores. The Reverse Bore parameter settings are similar to the Drill parameters. Two user defined reverse boring cycles (RBore1 and RBore2) are supported. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



Reverse Boring, Turn

Reverse Bore Menu Selection (TURN Module)

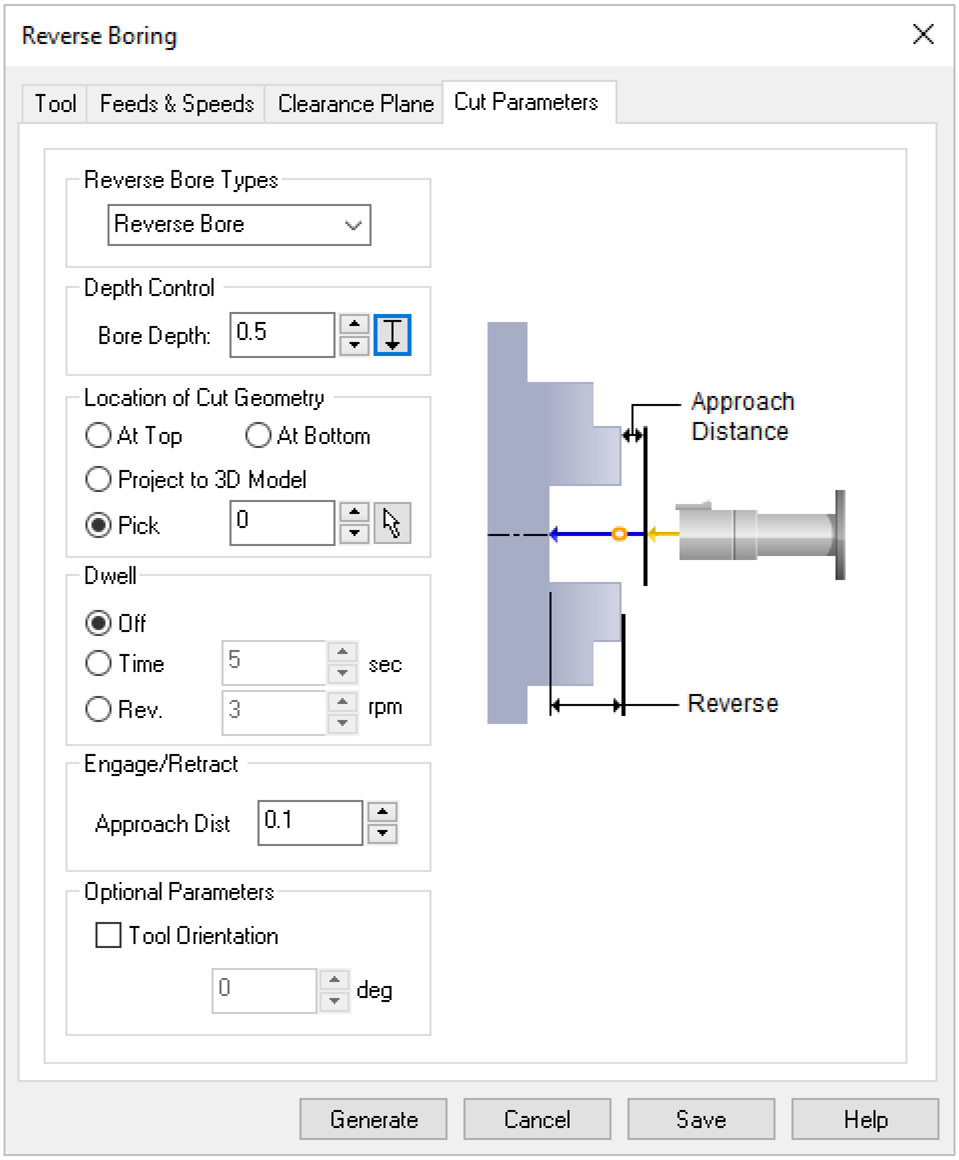
The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Rev. Bore operation.



2½ Axis Hole Making, Reverse Bore Menu Item

Dialog Box: Hole Making, Reverse Bore

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Reverse Bore operation. This dialog box is shown below.



Dialog Box: Cut Parameters tab, Turn Reverse Boring

The following Reverse Bore Types are available

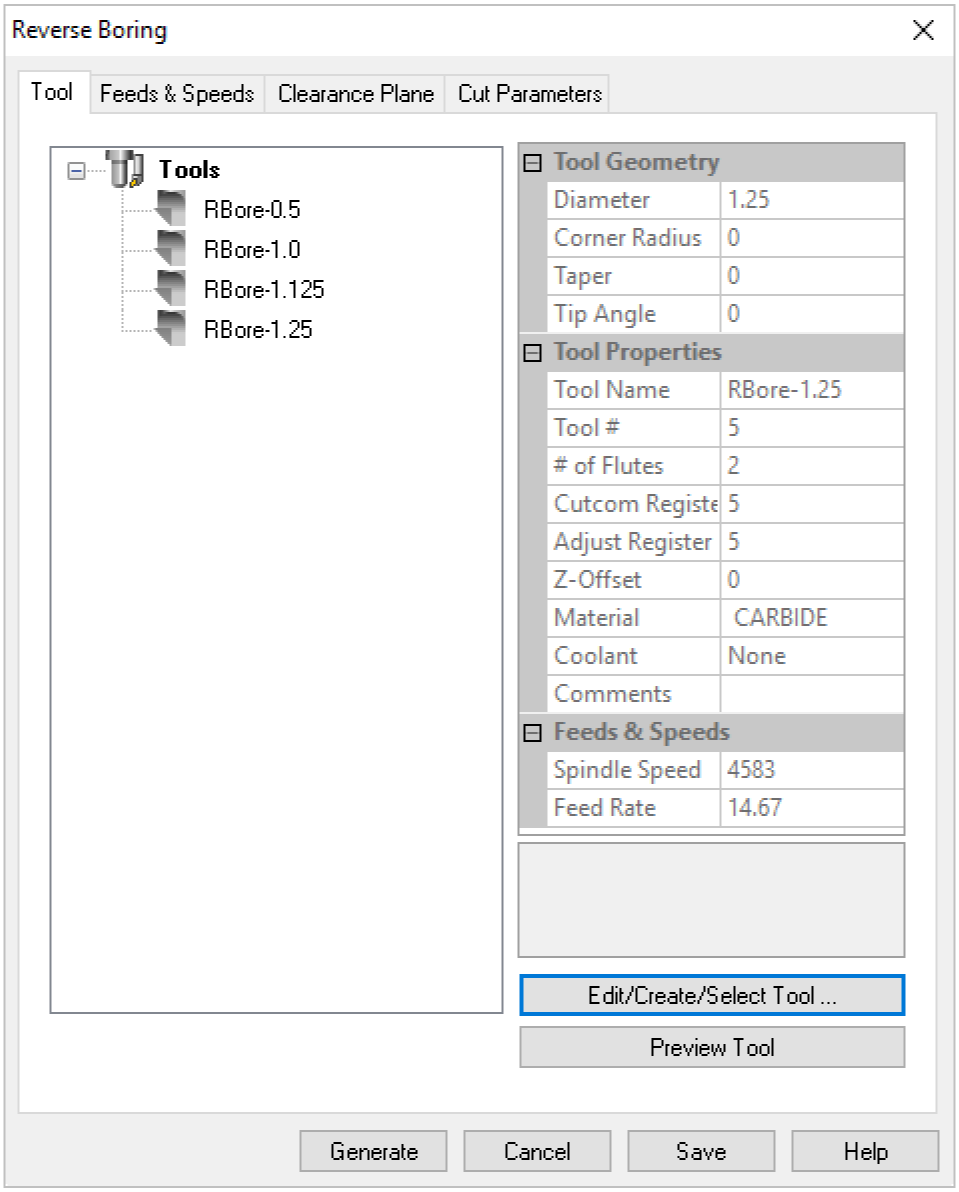
The Reverse Bore Types supported are:

- Reverse Bore
- RBore1: User Defined Reverse Bore
- RBore2: User Defined Reverse Bore

9.4.1 Tool

The following dialog allows you to select the appropriate Bore tool for the Turn Reverse Bore operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, Turn Reverse Boring

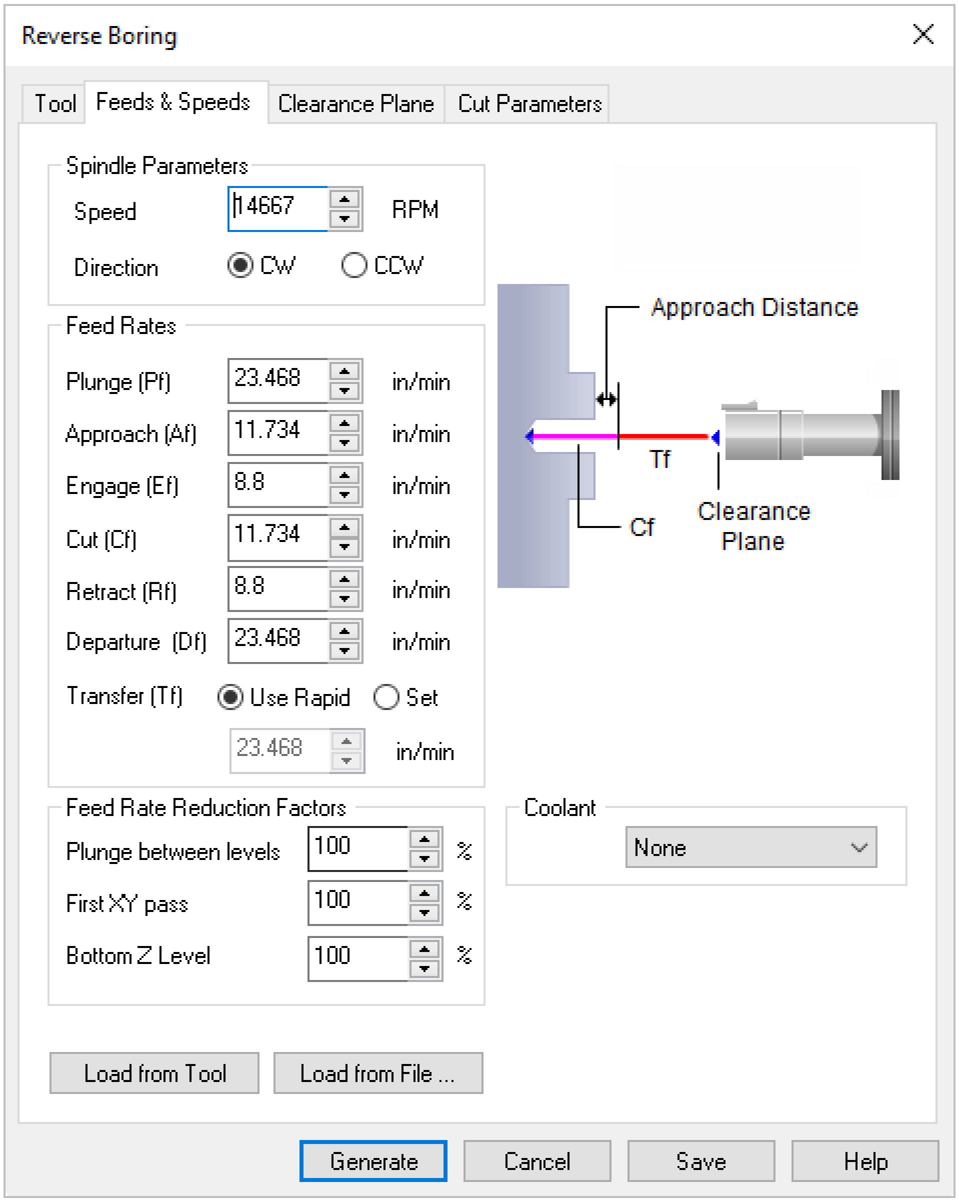
Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.4.2 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the Turn Reverse Boring operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, Turn Reverse Boring

Spindle Parameters

Speed

This is the rotational speed of the spindle expressed in RPM.

Direction

This determines the direction of spindle rotation and can be set to CW Clockwise or CCW Counter Clockwise.

Feed Rates

Feedrate can be set in Units/Min or Units/Revolution for Turning Inserts.

Plunge (Pf)
This rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the feedrate used that prepares the cutter just before it starts engaging into material as it starts cutting. The approach motions are dependent on the method of machining.

Engage (Ef)
This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut (Cf)
This is the feedrate used when the tool is cutting material

Retract (Rf)
The feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to also be 75% of the cutting speed.

Departure (Df)
The feedrate used to retract the tool from the material.

Transfer (Tf)
This is the feedrate (in Units/Min), used for Transfer motions. Select Use Rapid to set this to the Transfer Feed value defined in the Feeds & Speeds section of the CAM Preferences dialog.

Feed Rate Reduction Factors

This sets Feed Rate Reduction Factors for Plunge Between Levels and the First XY pass.

Load from Tool

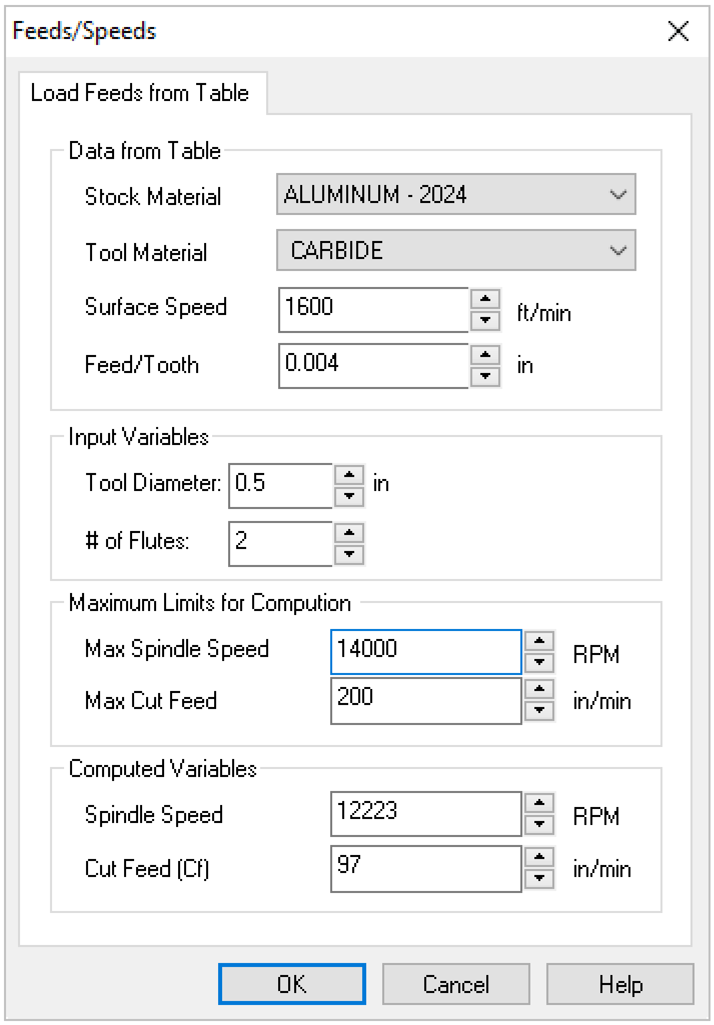
Feeds & Speeds are defined when a tool is created using Create/Edit Tools from the Machining Objects Browser. Selecting this button loads the Feeds & Speeds from the tool that is selected for the current machining operation.

Load from File ...

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

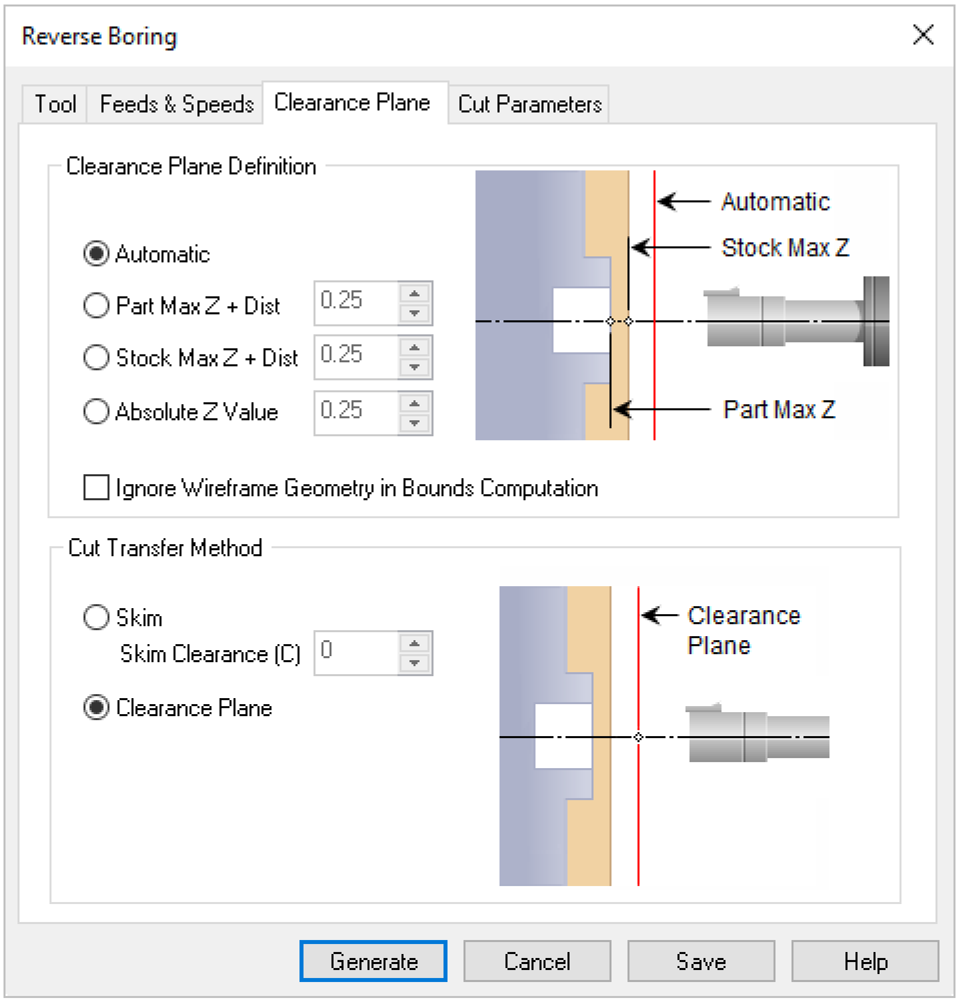
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.4.3 Clearance

The following dialog allows you to select the appropriate Clearance Plane for the Turn Reverse Boring operation. In this tab, Clearance Plane Definition and Cut Transfer Method parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance Plane tab



Dialog Box: Clearance Plane tab, Turn Reverse Boring

Clearance Plane Definition

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max + Dist

Uses Part maximum plus the specified distance for clearance height.

Stock Max + Dist

Uses Stock maximum plus the specified distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute Value
Uses the specified distance for clearance height.

For Turning operations, the User Interface for clearance settings are automatically set for OD, ID or Face depending on the approach type specified under global parameters.

For Hole Machining operations, the clearance plane is normal to the Z axis.

Ignore Wireframe Geometry in Bounds Computation
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

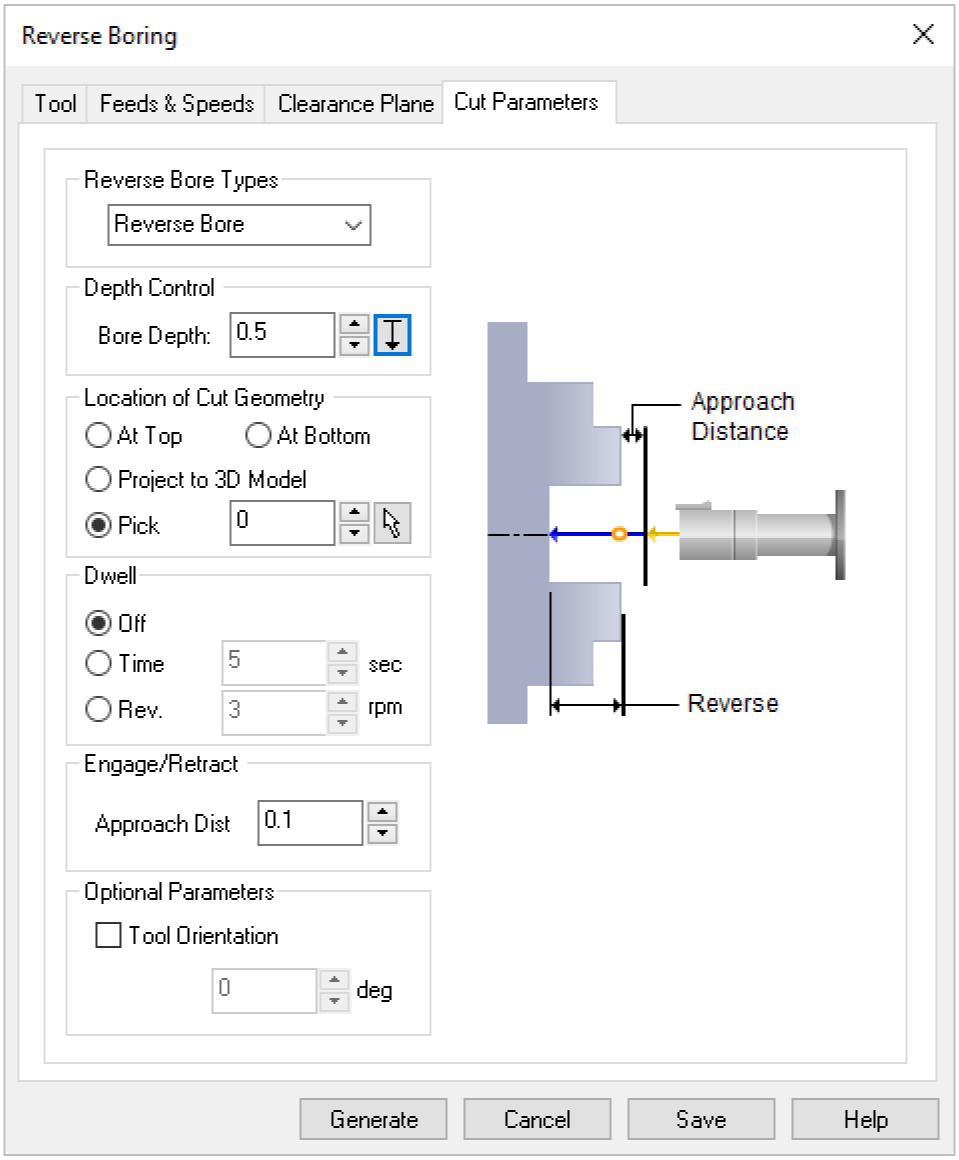
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can either be instructed to move to the clearance plane and then perform the transfer motion to the next cut location or it could do a skim motion. In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part model and using this Skim Clearance (S) value specified as the height to perform the transfer motions.

9.4.4 Cut Parameters

The following dialog allows you to set the Cut Parameters for Turn Reverse Boring operations. You can set the Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box.

Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Turn Reverse Boring

Bore Types

The reverse bore types supported are:

Reverse Bore
The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted.

User Defined Bore1 / User Defined Bore2

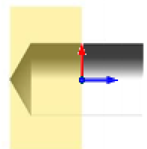
Defined using the Post-Processor Generator for your selected post definition.

Depth Control

RBore Depth - Enter the depth for the current drill operation. You can select the Pick button to pick a point on your part model that defines the drill depth.

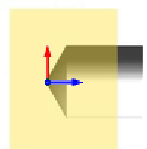
Location of Cut Geometry

Select this option if drilling should start at your Work Zero. The Drill Depth is then subtracted to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



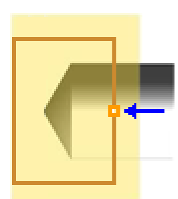
At Top

Select this option if drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location.



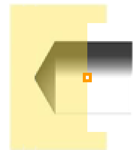
At Bottom

Project to 3D Model
Select this option if you want to project the start point to your 3D model. For example, if your stock is longer than your part but you want the Drill Depth to begin at the face of your part instead of the face of the stock. An actual point is not required. The location is determined automatically by your part model.



Project to 3D Model

Select this option if you want to specify the coordinates of point where the drilling should end at your Work Zero. The Drill Depth is then added to this location. An actual point is not required. The location is determined automatically by your Work Zero location. You can select the Pick button and then select a point on your part. It's coordinate will be calculated automatically.



Pick Top

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in the Z axis to the approach plane and then applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

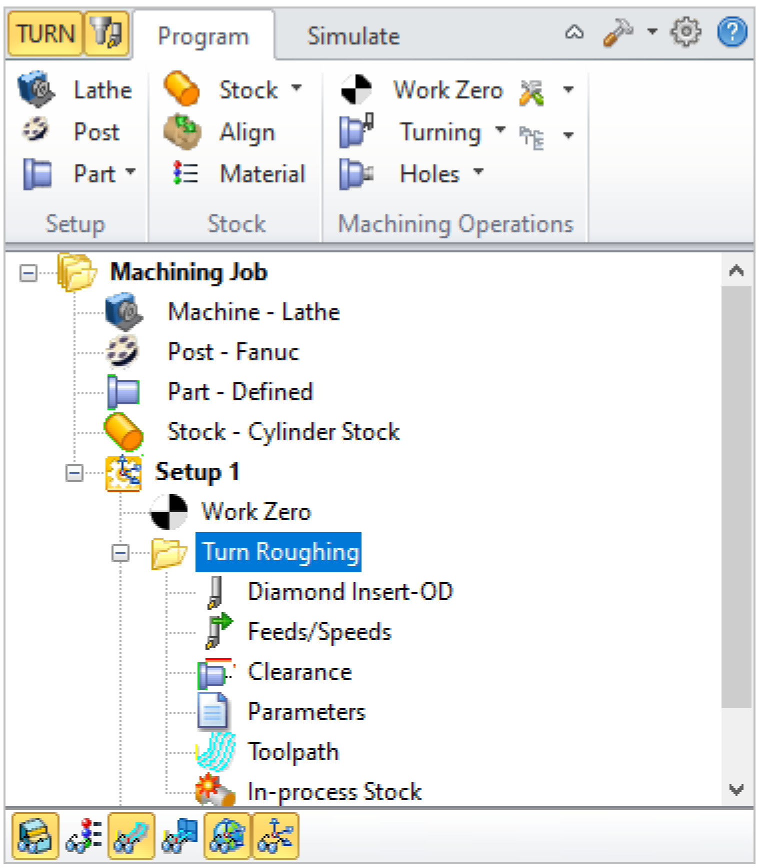
Optional Parameters

Using the Reverse Bore types (see above), the tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

Editing Machining Operations

Once a Machining operation is created it is listed under the Machining Browser. By default all the operations are created under Setup 1. A Setup can hold several machining operations. The operations can be edited in a couple of ways. Changes can be made to any of the objects that make up the operation such as the Tool, Feeds/Speeds, Clearance Geometry and Machining Parameters. This type of editing is called associative editing. This is because the edits made to the operation are saved with the operation and upon regeneration the changes would be effected.

Machining Browser

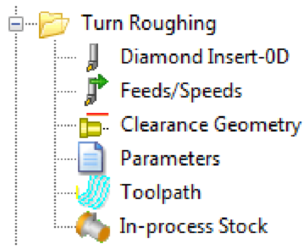


Machining Browser

10.1 Editing Operations Associatively

Machining Operations can be edited by using the Machining Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, five icons representing five different objects that make up the operation are displayed. These are:

Machining Browser, Turn Operation Tree



Machining Browser Toolpath Operation Icons

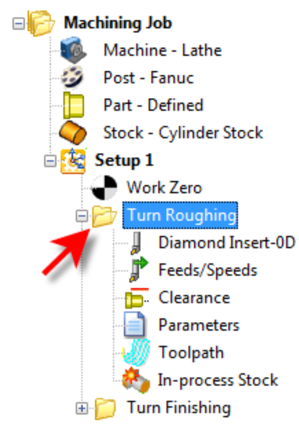
- Tool
- Feeds/Speeds
- Clearance Geometry
- Parameters
- Toolpath
- In-process Stock

Double-click Editing an Operation Folder

Double clicking on Tool, Feeds/Speeds, Clearance Geometry or Parameters icons gives you an opportunity to edit the object.

- Tool
- Feeds/Speeds
- Clearance Geometry
- Parameters

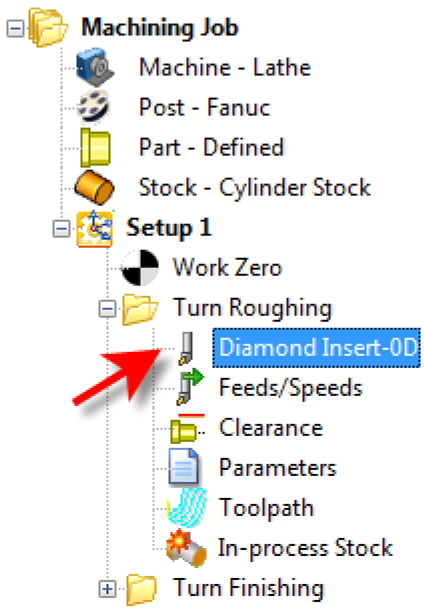
For the selected operation.



Double-click Editing an Operation from the Machining Browser

Right-Mouse click or Double-click on an Operation Icon

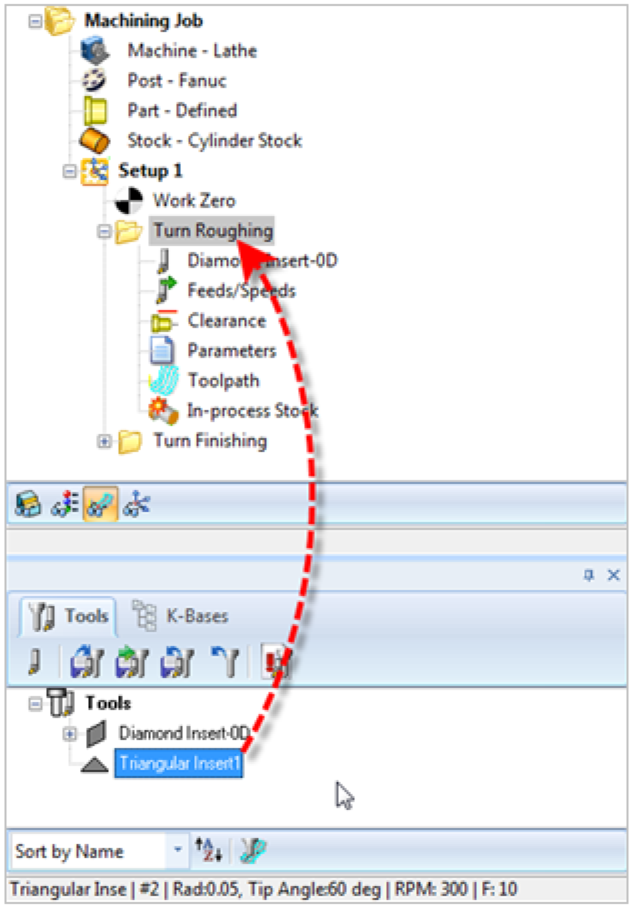
Right mouse click or double clicking a specific icon for example the Tool icon would bring up the Create/Edit Tool dialog, upon which you can substitute the current tool with another or edit the parameters of the current tool.



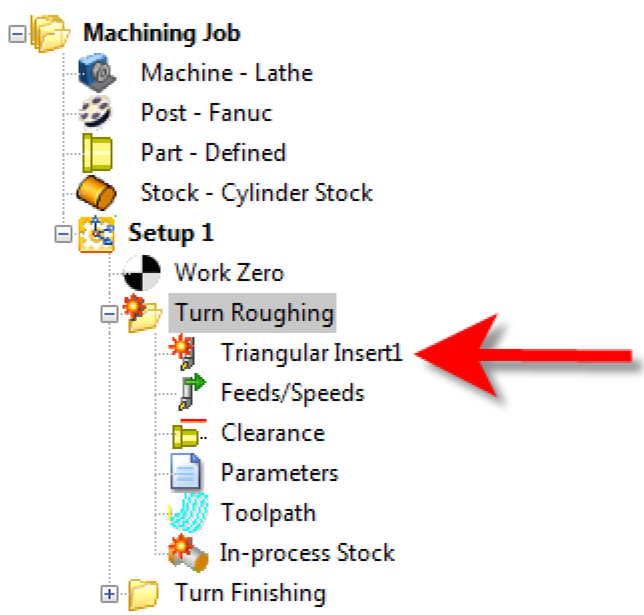
Right-Mouse click or Double-click on an Operation Icon from the Machining Browser

Drag & Drop New Tools into an Operation

The tool can also be edited by dragging and dropping it from the Tools tab to of the Machining Objects Browser to the operation folder in the Machining Browser.



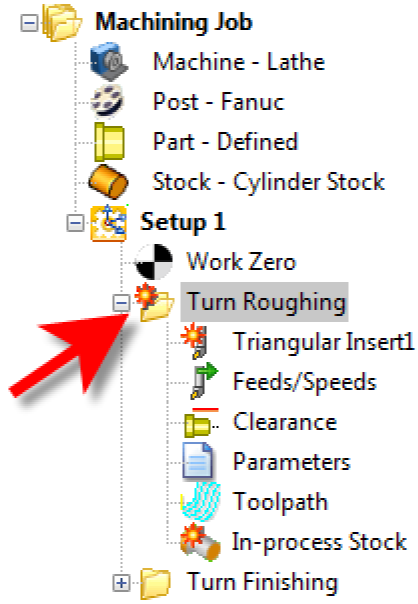
Drag & Drop Editing an Operation from the Machining Browser



Drag & Drop Editing an Operation from the Machining Browser

Indicator that an Operation needs to be Regenerated

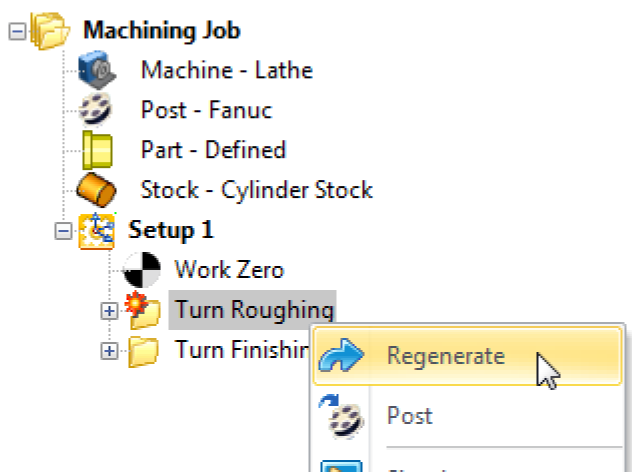
If any of the objects that make up the operation were to be edited after the toolpath was generated, the operation will be flagged dirty as needing re-computation. TURN Module indicates such a condition by adding a red flag to the operation folder (i.e.,). The object that necessitated this re-computation is also displayed with a red flag (i.e.,). An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.



Indicator that an Operation Needs to be Regenerated

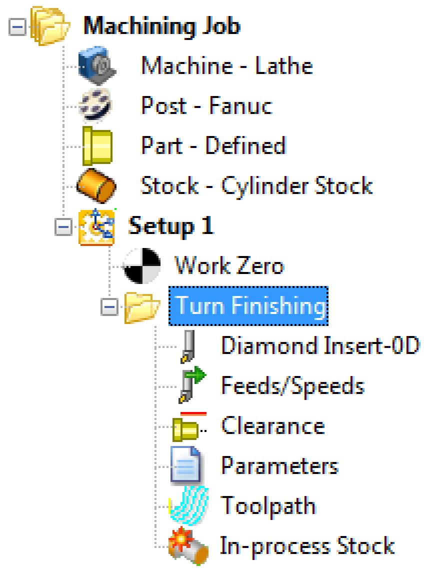
Regenerating an Operation from the Machining Browser

To regenerate an operation that is flagged dirty, first select the operation folder, right click and select Regenerate as shown in the example below.



Regenerating an Operation from the Machining Browser

The toolpath is now regenerated with the modified settings.

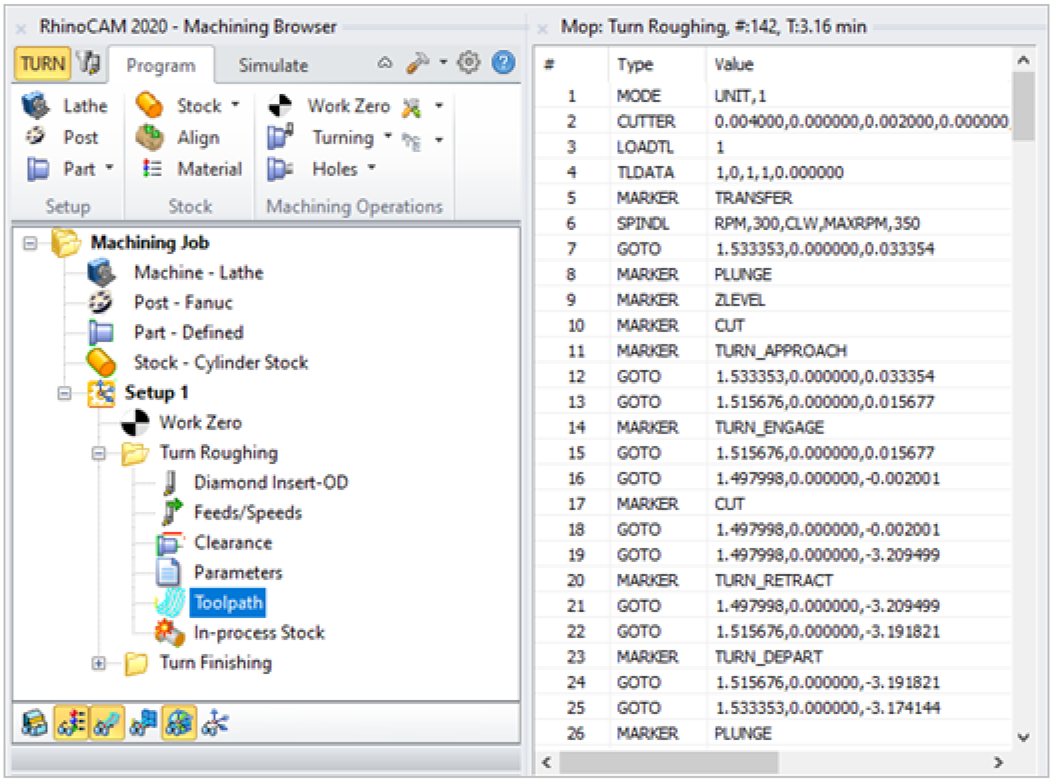


The toolpath is now regenerated with the modified settings

10.2 Toolpath Viewer

Once a machining operation is created, you can step through the toolpath motions using the Toolpath Viewer. To display the viewer, double-click on the toolpath icon of the operation in the Machining Browser. The Toolpath Viewer is a dockable dialog bar that will be initially docked next to the Machining Browser.

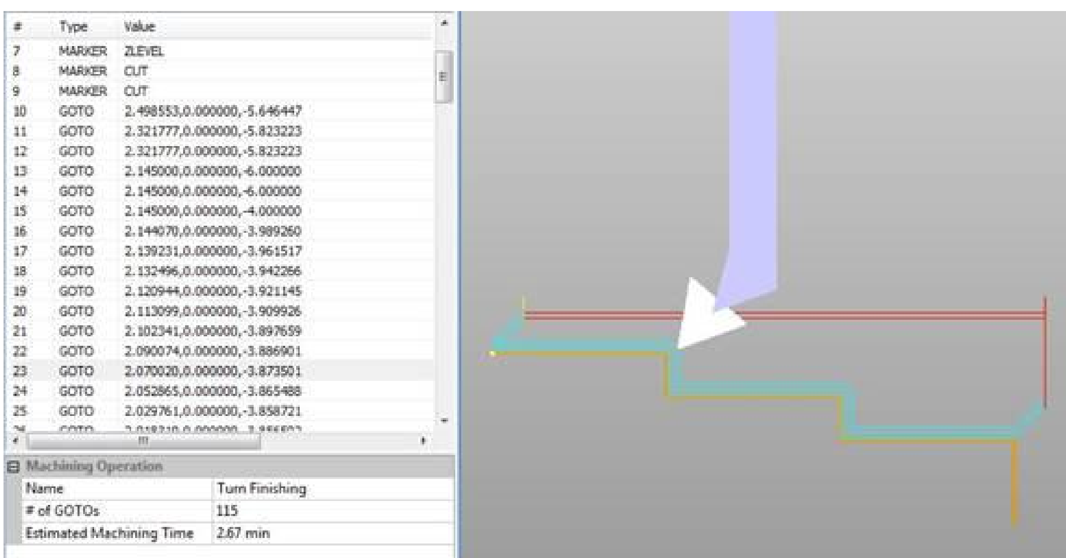
The Toolpath Viewer Displayed



The Toolpath Viewer Displayed

Viewing Toolpath Motions

Select a GOTO motion in the toolpath viewer to view the tool motion for the generated toolpath. Make sure to turn on Toolpath Visibility.

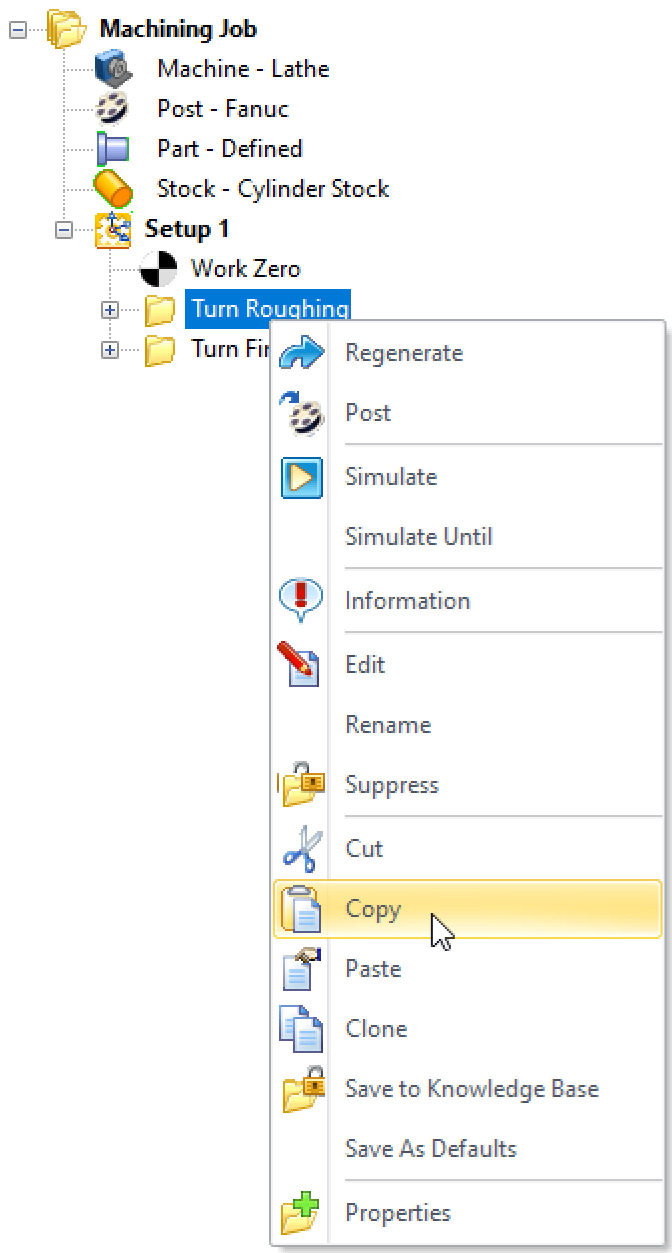


10.3 Copy & Paste Operations

You can copy and paste machining operations in TURN Module.

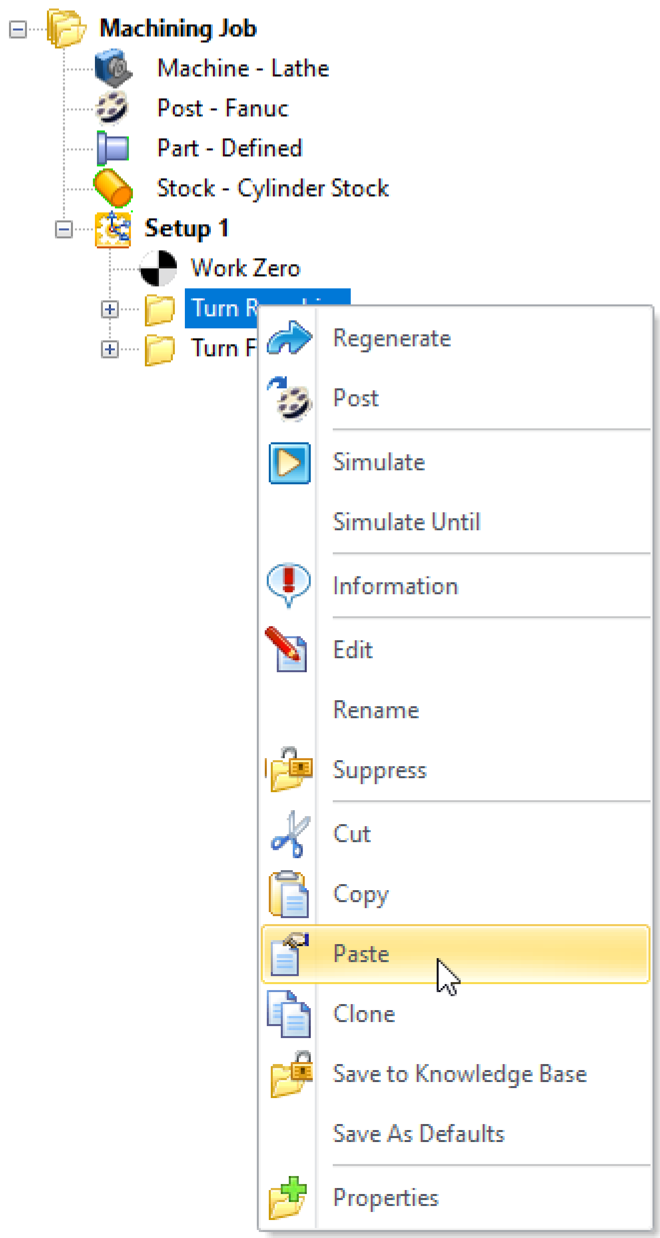
Copy & Paste an Operation from the Machining Browser

1. Right-click on the operation folder and select Copy.



Right-click on the operation folder and select Copy.

2. Right-click on the operation folder again and select Paste. The new operation is located directly below the selected operation.



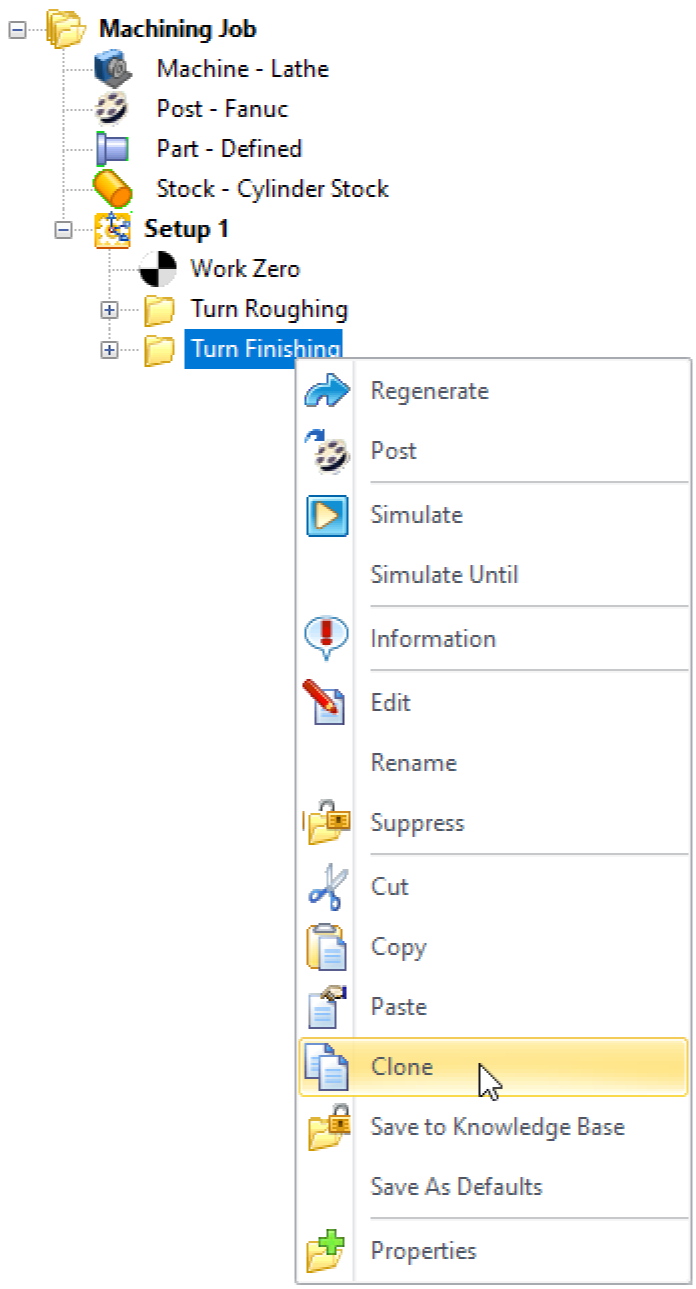
Right-click on the operation folder again and select Paste

3. This creates a copy of the operation. You can then edit the operation and regenerate toolpath.

10.4 Clone Operation

You can Clone machining operations in Machining Browser. To Clone an operation, select the operation under the Machining Browser, right-click and select Clone. This performs a Copy/Paste on the selected operation.

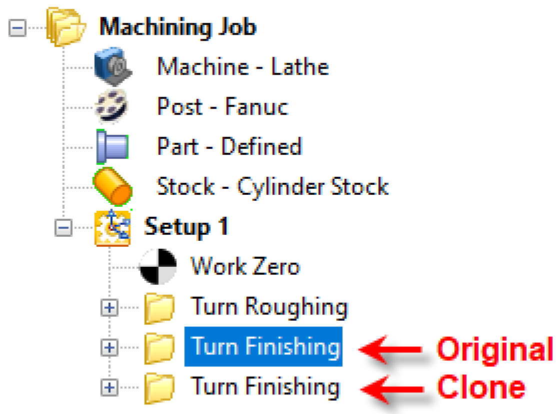
Clone the Operation



Machining Browser: Clone an Operation

Edit or Regenerate the Operation

This creates a copy of the operation located under the currently selected operation.



Cloned Operation

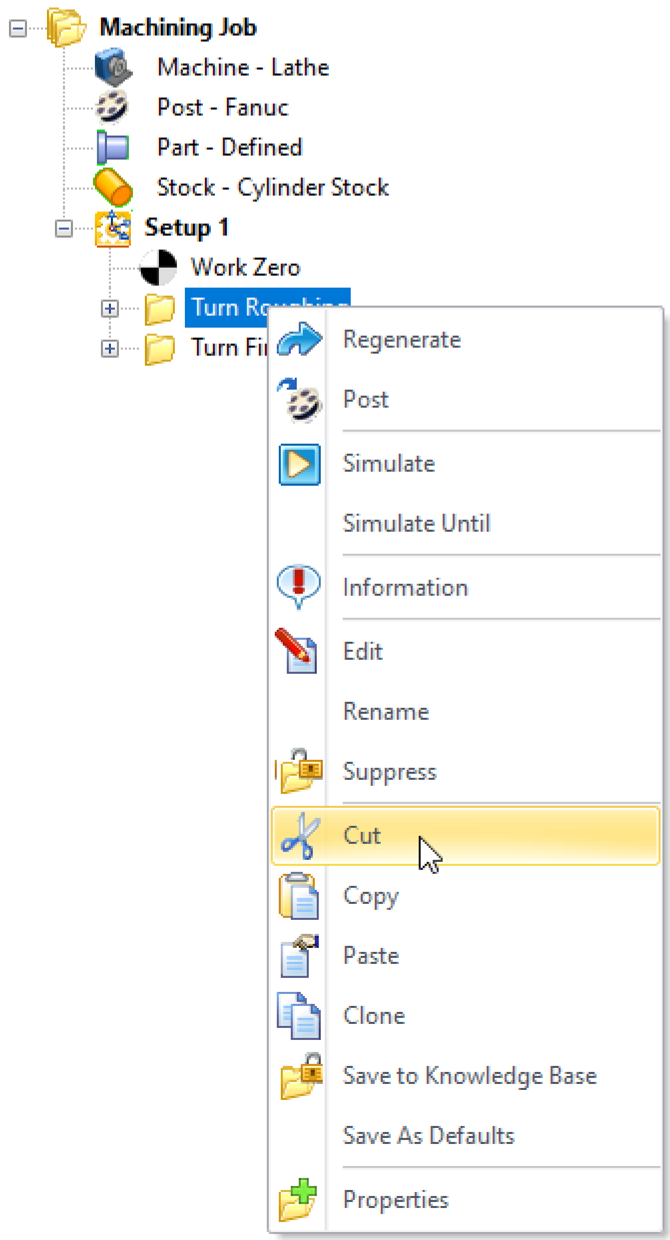
You can then edit the operation and regenerate toolpath.

10.5 Delete Operations

To Delete a machining operation, select the operation from the Machining Browser, right-click and select Cut.

Cut an Operation from the Machining Browser

1. Right-click on the operation folder and select Cut.



Cut an Operation from the Machining Browser

2. The operation is removed and added to the clipboard. If desired, you can right-click and Paste the operation back into the Machining Browser.

Alternate Methods to Delete an Operation

Alternatively you can delete a machining operation by:

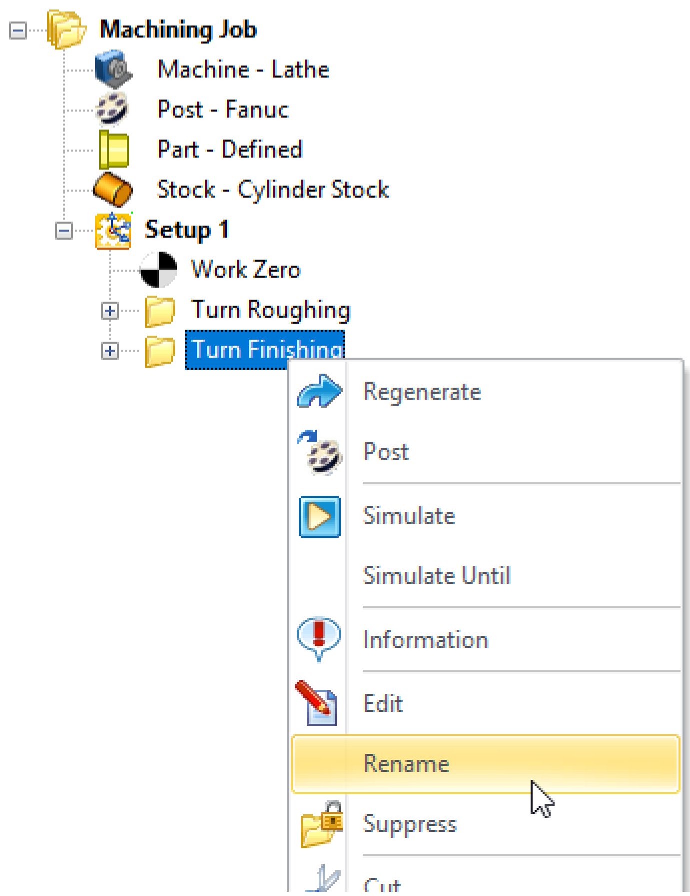
1. Selecting the operation and using the Delete key on your keyboard.
2. Selecting the operation and dragging the operation out of the mops browser to the viewport area.

10.6 Rename Operations

To Rename a machining operation, select the operation from the Machining Browser, right-click and select Rename.

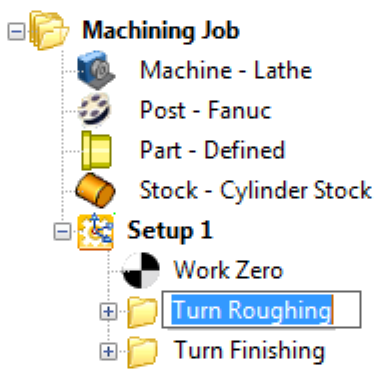
Rename an Operation from the Machining Browser

1. Right-click on the operation folder and select Rename.



Right-click on the operation folder and select Rename

2. Enter a new name for the operation.



Enter a new name for the operation

Conventions for Renaming Operations

Do not use any of these common illegal characters/symbols in your Mop Names:

DO NOT USE these Characters when renaming Mops			
#	pound	?	question mark
%	percent	/	forward slash
&	ampersand	$	dollar sign
{	left bracket	!	exclamation point
}	right bracket	'	single quotes
\	back slash	"	double quotes
<	left angle bracket	:	colon
>	right angle bracket	@	at sign
*	asterisk		

Also, keep these rules in mind:

- Do not start or end your Mop Names with a space or period
- Keep your file names to a reasonable length and be sure they are under 31 characters.

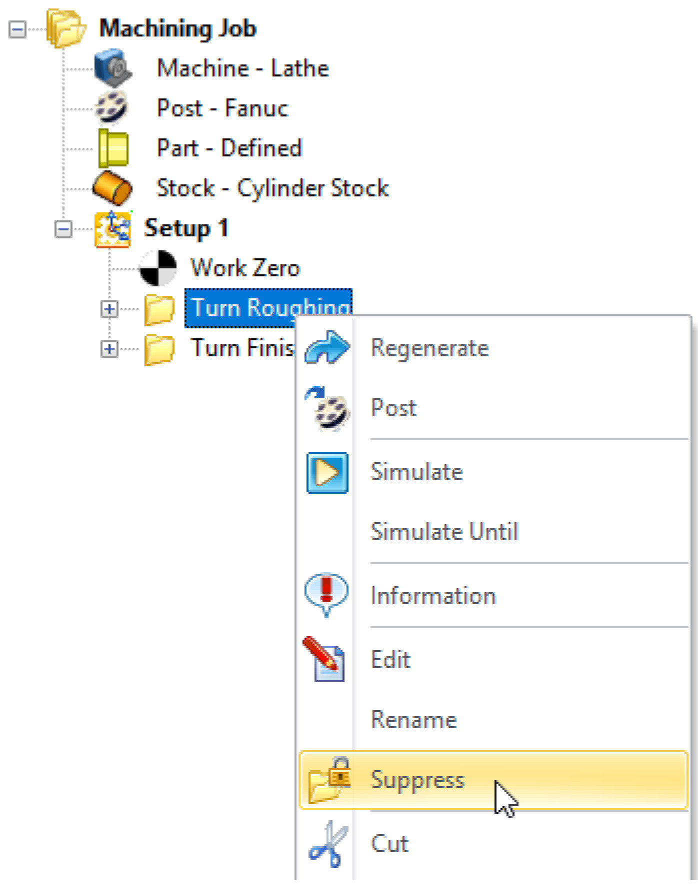
Alternate Methods to Rename an Operation

Alternatively you can double-left-click on the operation name in the Machining Browser and then enter a new name.

10.7 Suppress Operations

You can Suppress a machining operation or a Setup in the Machining Browser by selecting it, right click and select Suppress from the context menu. Suppressed operations will not be displayed, posted or simulated. You can also right-click and Unsuppress an operation. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

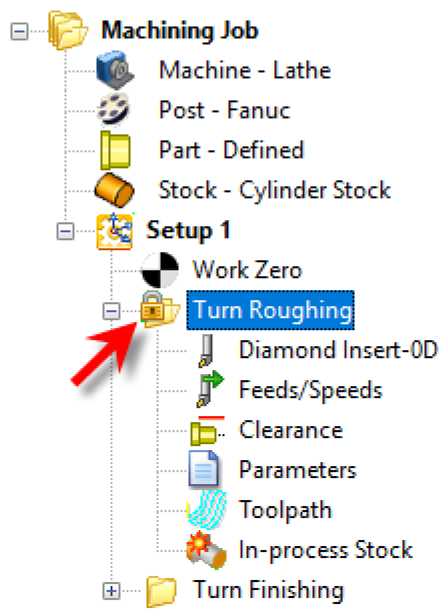
To Suppress an Operation



To Suppress an Operation

The Suppressed Operation Icon

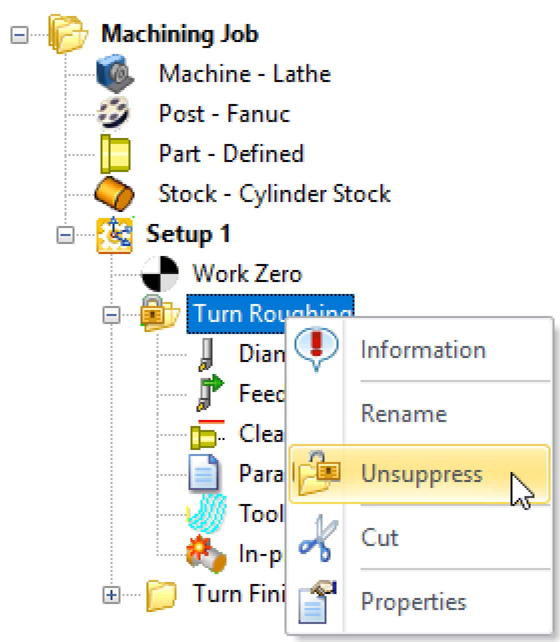
A Suppress operation will display in the Machining Job with the following icon:



Suppressed Operation Icon

To Unsuppress an Operation

To Unsuppress an operation, right-click on it and select Unsuppress.



To Unsuppress an Operation

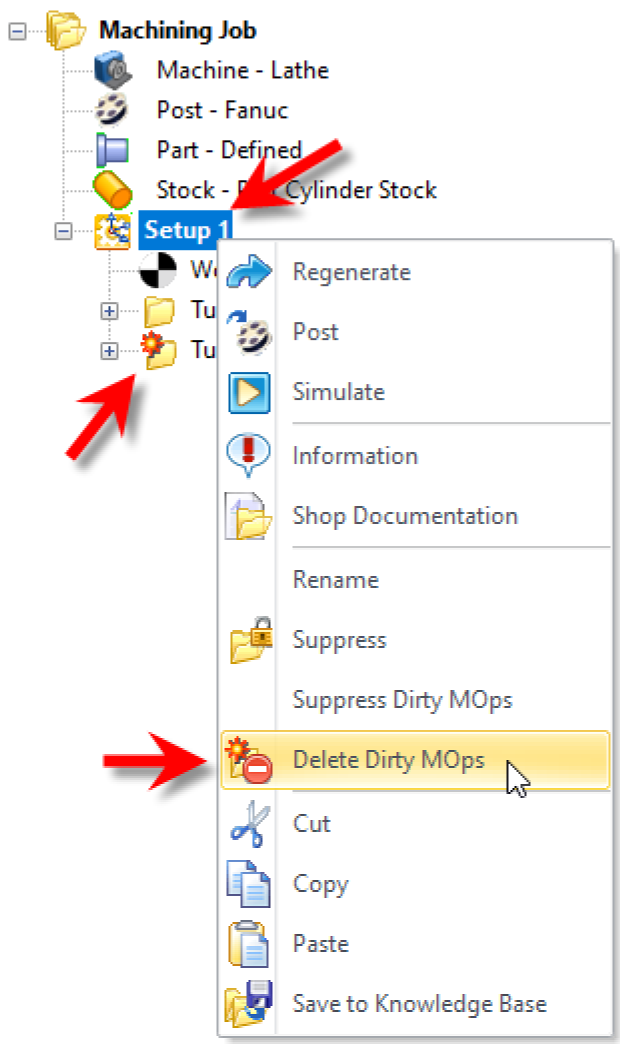
10.8 Delete Dirty Mops

You can Delete all machining operations in a Setup that are flagged as dirty . This command can also be executed from right-clicking on the Machining Job to delete ALL dirty Mops in all

Setups. Warning This a permanent operation and cannot be undone. Select the Setup, right click and select Delete Dirty Mops from the context menu.

Delete All Dirty Mops (BEFORE)

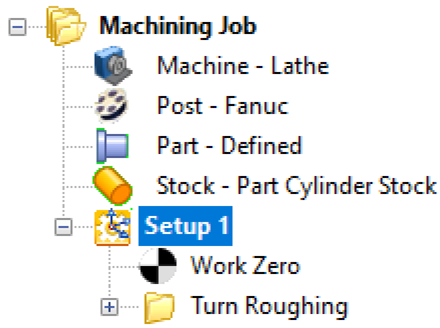
> **Warning!** This operation is permanent and cannot be undone!



To Delete Dirty MOps

Delete All Dirty Mops (AFTER)

A Suppress operation will display in the Machining Job with the following icon:

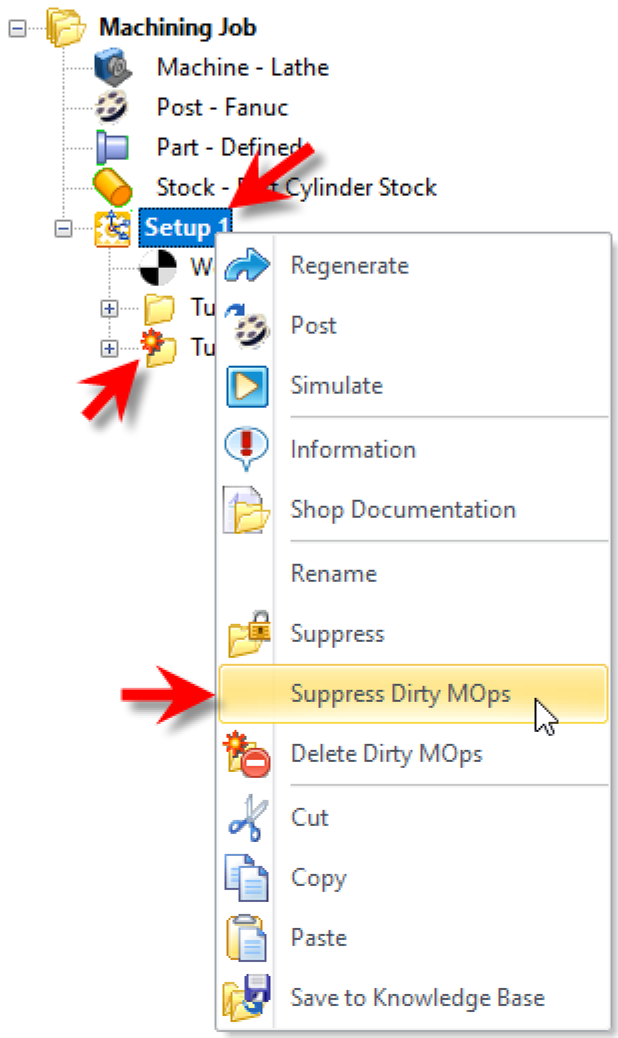


All Dirty MOps are deleted
MILL Module shown, Similar for TURN and Profile-NEST

10.9 Suppress Dirty Mops

You can Suppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Suppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to suppress ALL dirty Mops in all Setups. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

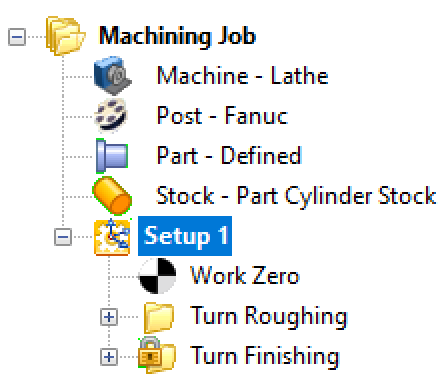
Suppress All Dirty Mops (BEFORE)



To Suppress all Dirty MOps

Suppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:

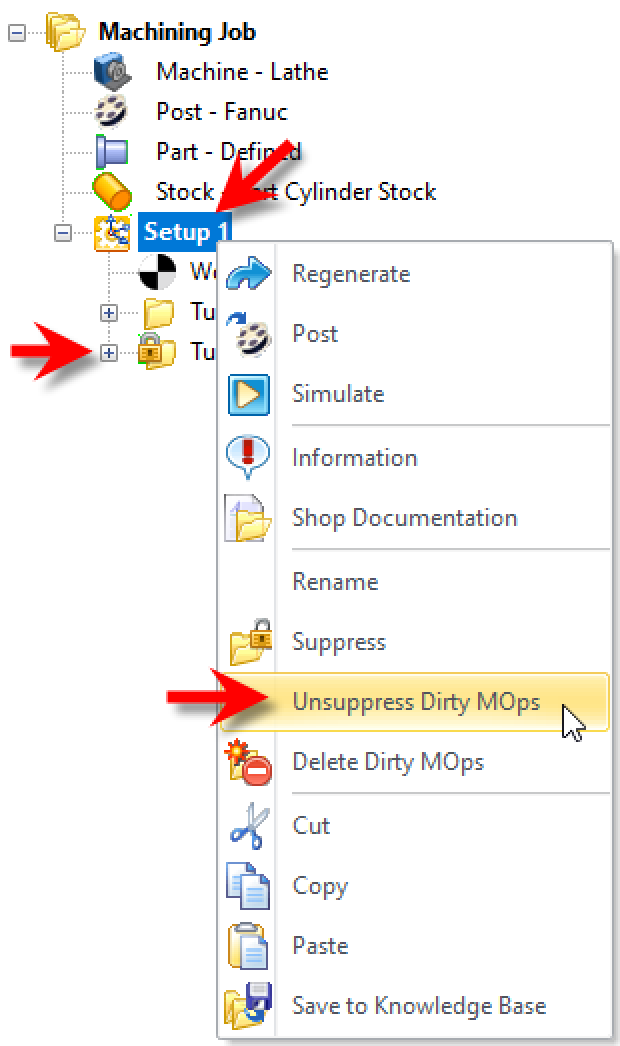


All Dirty MOps are suppressed

10.10 Unsuppress Dirty Mops

You can Unsuppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Unsuppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to unsuppress ALL dirty Mops in all Setups. **Note:** You can customize how suppressed operations are managed using the [Machining Preferences](#) dialog.

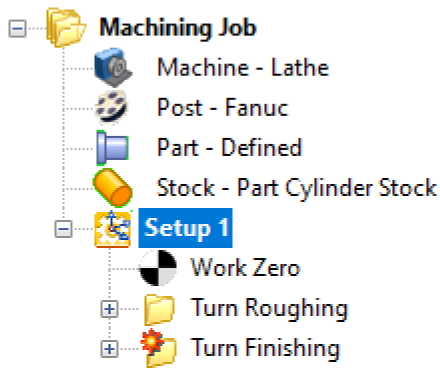
Unsuppress All Dirty Mops (BEFORE)



To Unsuppress all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Unsuppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:



All Dirty MOps are unsuppressed
MILL Module shown, Similar for TURN and Profile-NEST

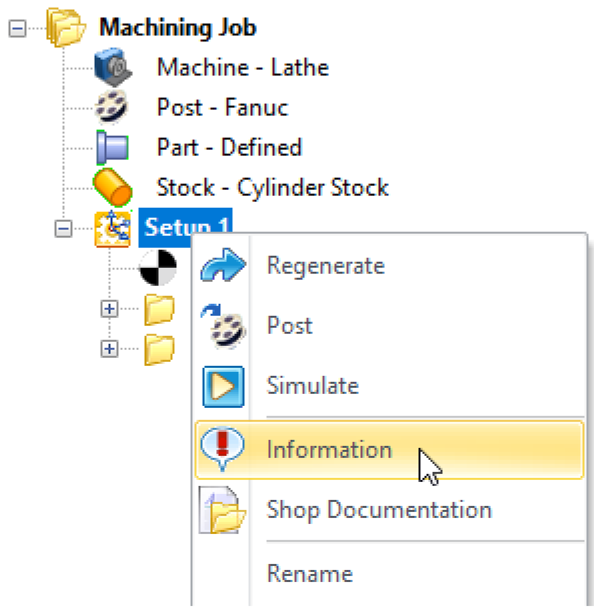
10.11 Machining Information

This displays a dialog box with the following information about the selected Operation, the Setup or the entire Machining Job:

- Operation Name
- Status
- Tool Name & Tool Number
- Cut Feed Rate
- Spindle Speed
- # of GOTOs
- Machine Time

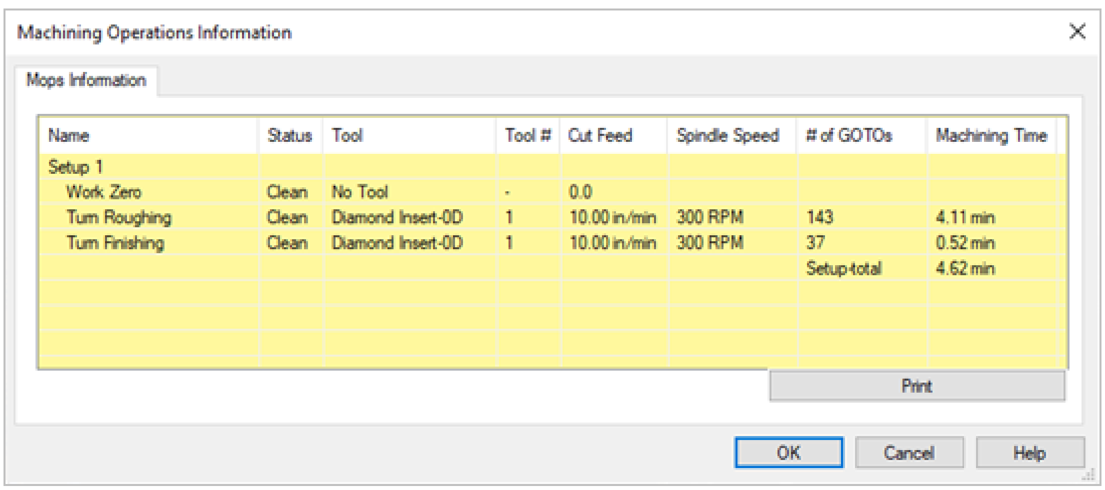
Select Information from the Right-click Menu

Machining Operations Information can be viewed by selecting a Setup, right mouse button click and left click on Information.



Select Information from the Right-click Menu

Dialog Box: Machining Operations Information (TURN Module)



Name	Status	Tool	Tool #	Cut Feed	Spindle Speed	# of GOTOs	Machining Time
Setup 1							
Work Zero	Clean	No Tool	-	0.0			
Turn Roughing	Clean	Diamond Insert-OD	1	10.00 in/min	300 RPM	143	4.11 min
Turn Finishing	Clean	Diamond Insert-OD	1	10.00 in/min	300 RPM	37	0.52 min
						Setup total	4.62 min

Dialog Box: Machining Operations Information

10.12 Shop Documentation

Shop documentation can be generated for any Setup listed under the Machining Job in the Machining Browser. Right-click on Setup and select Shop Documentation. This will create a setup sheet for the programmed part which includes the following information. It can be printed and handed over to the operator in preparation for the part to be machined on the CNC.

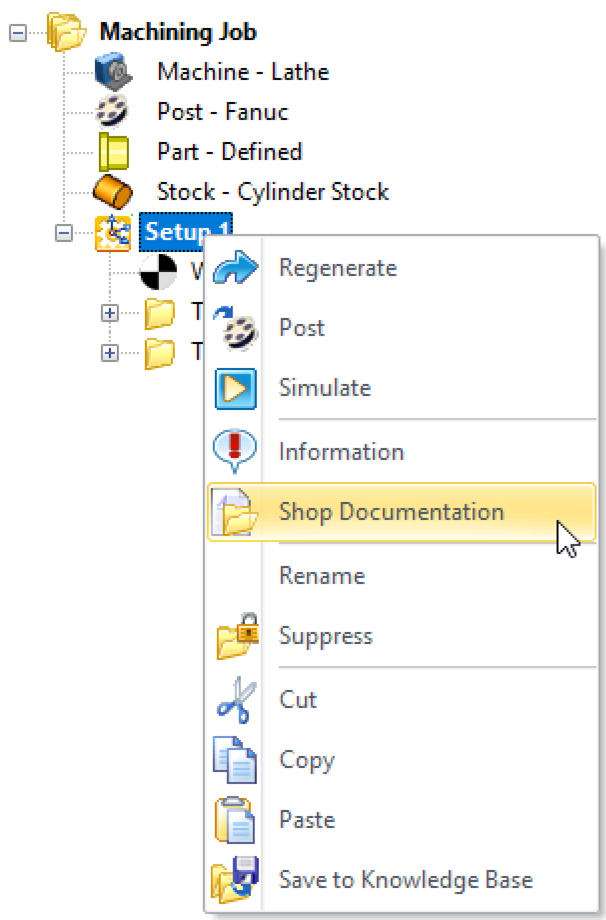
Information in the Shop Documentation Setup Sheet

- Screen capture of the part from the active viewport

- Total Machining time
- Tool list
- Machining Operations List

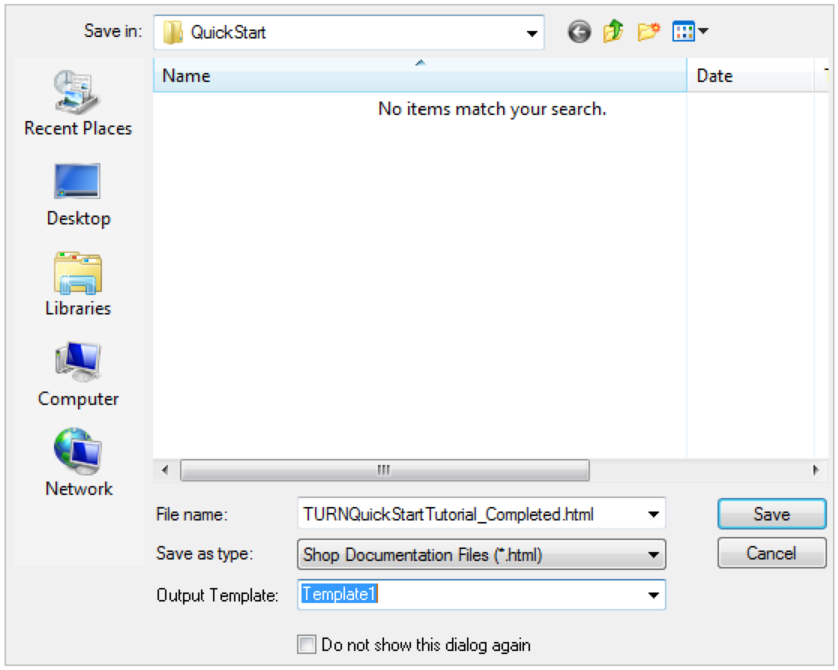
Displaying Shop Documentation for a Setup

1. Right-click on the Setup and select Shop Documentation to display the setup sheet.



Displaying Shop Documentation for a Setup

2. In the Save Shop Documentation File dialog box, select an Output Template and then pick Save. By default, the HTML file is saved to the folder where the current file is located.



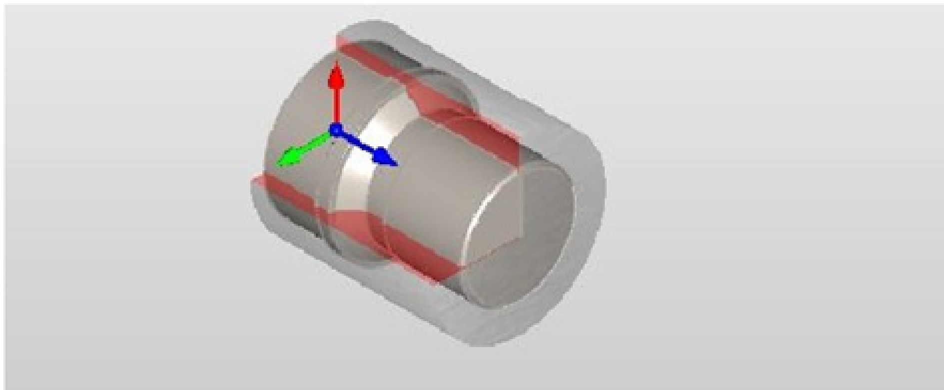
Dialog Box: Save Shop Documentation File

3. The Setup Sheet is displayed in your default Internet Browser.

Example: Shop Documentation Setup Sheet

SETUP SHEET		
File Name:		
Date:	**Units:** inch	**Total Machining Time:** 3 mins 44 secs
Stock Dimensions: Radius: 1.500; Length: 3.207		
No. of Ops: 3	**Stock Material:**	**Number of Tools:** 1

PART SETUP



TOOL LIST

No.	Name	Number	Type	Dimensions					Comments
				Radius	C-Radius	Taper	Length	Coolant	
1	Diamond Insert-OD	1	Undefined	0.020	0.000	0.000	0.000	None	

MACHINE OPERATIONS LIST

Operation Number: 1

Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Work Zero	0	0.000	0.000	Zero (0)	

Operation Number: 2

Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Turn Roughing	1	10.000 in/min	300.000	3 mins 16 secs	

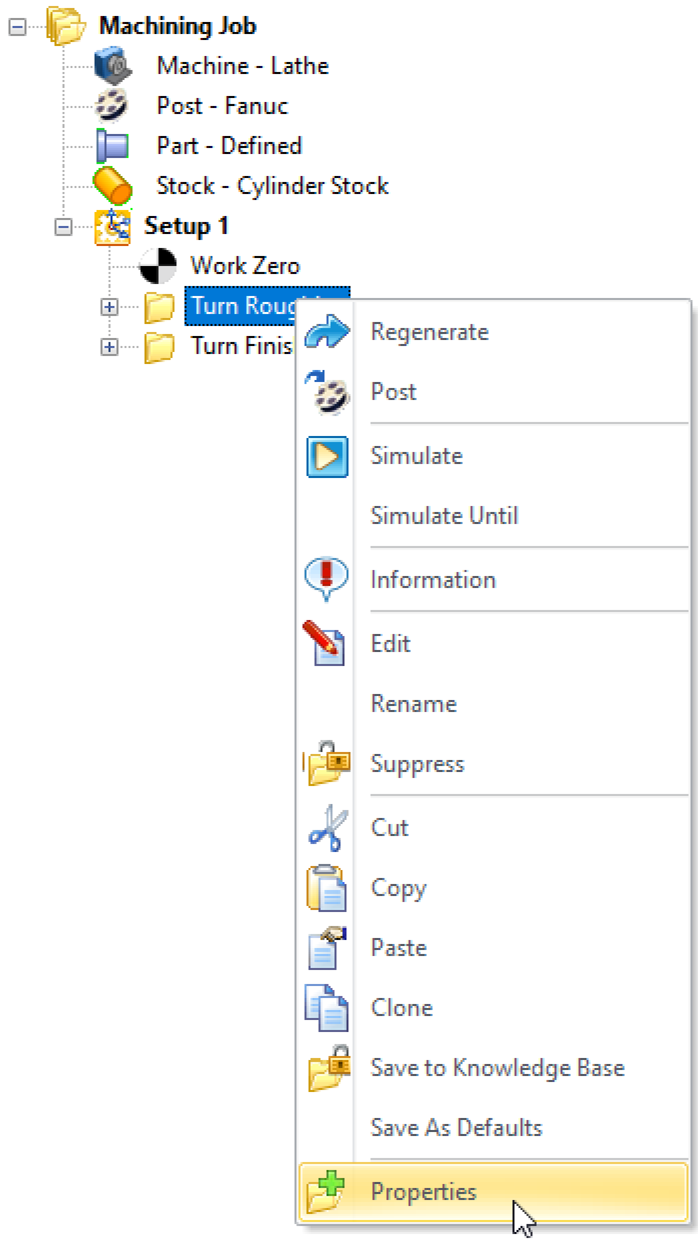
Operation Number: 3

Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Turn Finishing	1	10.000 in/min	300.000	28 secs	

10.13 Machining Operation Properties

You can set the properties of a Operation by selecting it in the Machining Browser window, clicking on the right mouse button and selecting the Properties menu item.

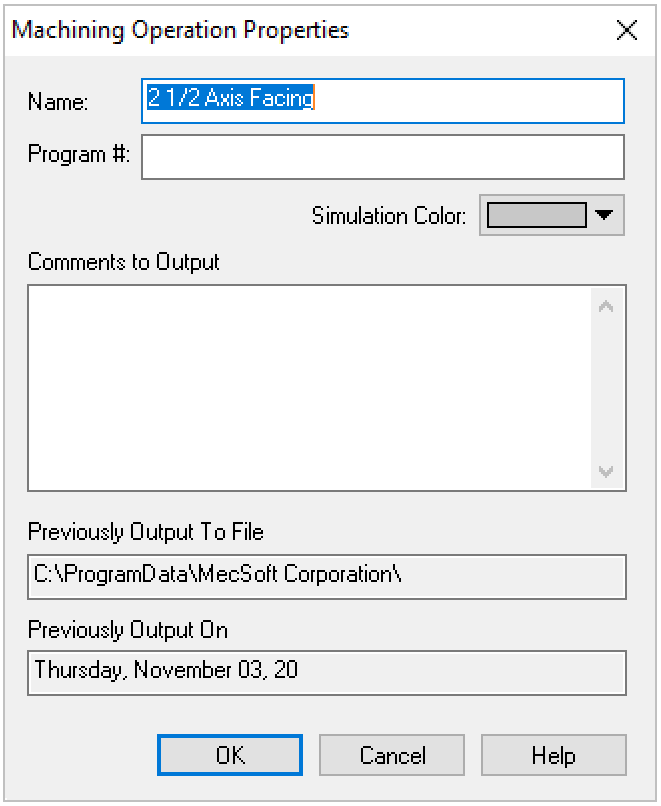
Select Properties from the Right-click Menu



To set Machining Operation Properties
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.



Dialog Box: Machining Operation Properties

Name

Change the Name of the Machining Operation.

Program

Specify Program # for the operation. This program number will be output during post processing of the operation.

Simulation Color

This allows you to specify a unique color for this operation during Simulation display. Refer to the Simulate tab Status Bar for setting the simulation to display by Mop (i.e., machining operation type).

Comments to Output

You can also include commands that will be saved with the operation. These comments will also be output during post-processing of the operation. This might be a good place to put in comments or instructions for the machine tool operator.

This can be used to put in add comments or instructions for the machine tool operator!

Previously Output To File

This refers to the name of the external post-processed file that this particular operation was output to.

Previously Output On

This refers to the last time the operation was post-processed and the time the post-processing was performed.

Simulating Machining Operations

TURN Module offers powerful cut material simulation functionality to allow users to simulate actual machining of the generated toolpaths. The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated.

The 3D cut model can be visually compared with the part model to show areas of uncut material (i.e., undercut) and/or areas of over-cut material (i.e., gouging) using this component. The simulation features allow the early detection and correction of programming errors. The following section describes the material removal simulation functionality available in TURN module.

In the TURN module, rotate the view using the right-mouse button to a suitable orientation that is convenient for viewing the simulation.

Types of Simulations Available

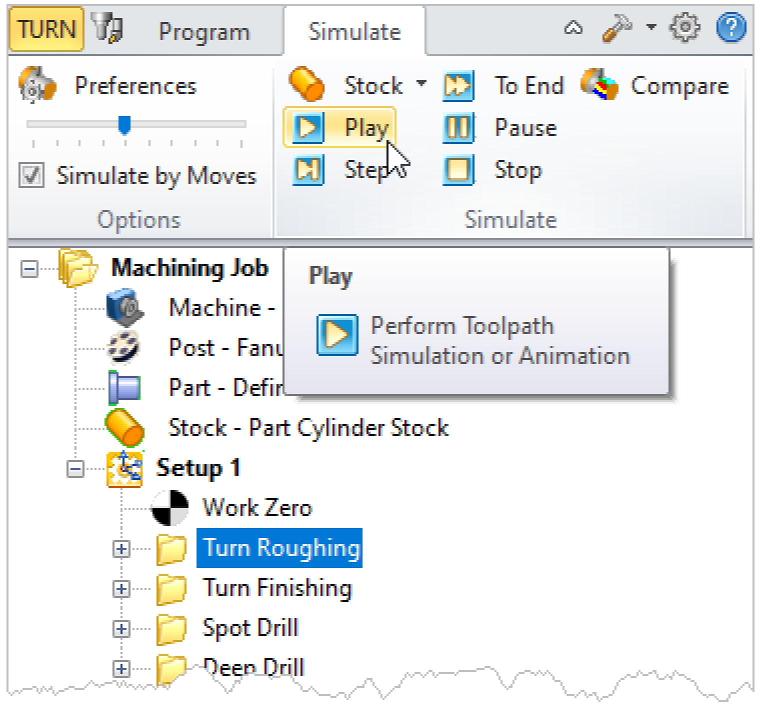
There are three kinds of toolpath simulation available in TURN module. These are:

- Tool Animation
- Cut Material Simulation
- Machine Tool Simulation

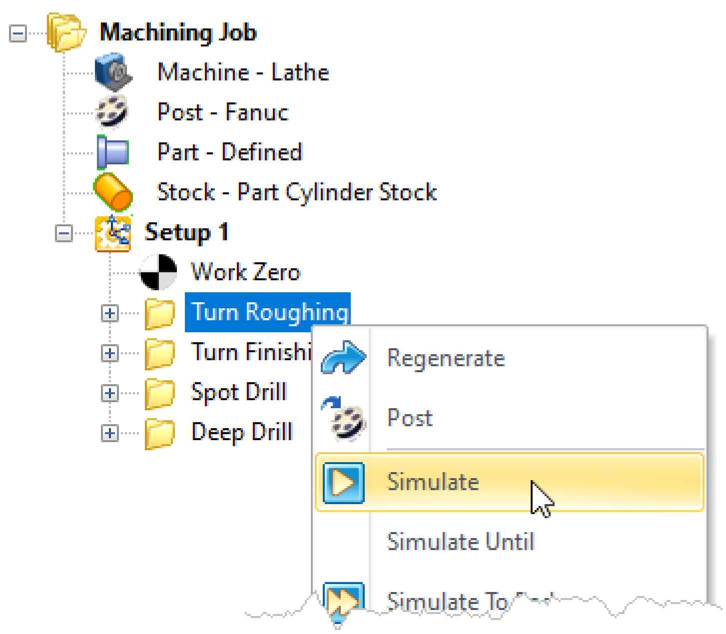
To Simulate an Operation

The simulation can be performed either on the currently active machining operation or on multiple operations. The active operation is the one that is selected and shown highlighted in the Machining Browser. Typically, this would be the last toolpath that was generated.

1. To simulate any operation, select it from the Machining Browser.
2. Select the Simulate tab.
3. Select the Play button or right-click on the operation and pick Simulate.



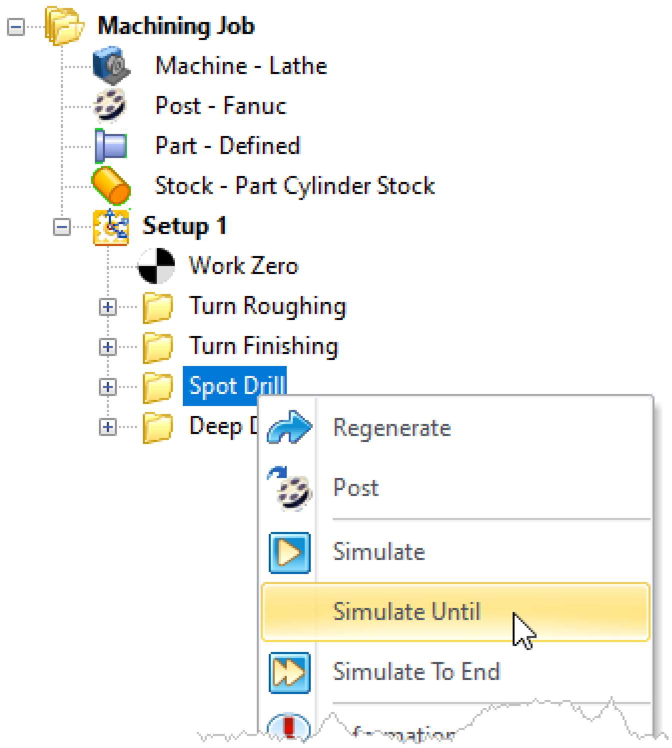
Select an operation and then pick Play from the Simulate tab



Select an operation, right-click and pick Simulate

Simulate Until

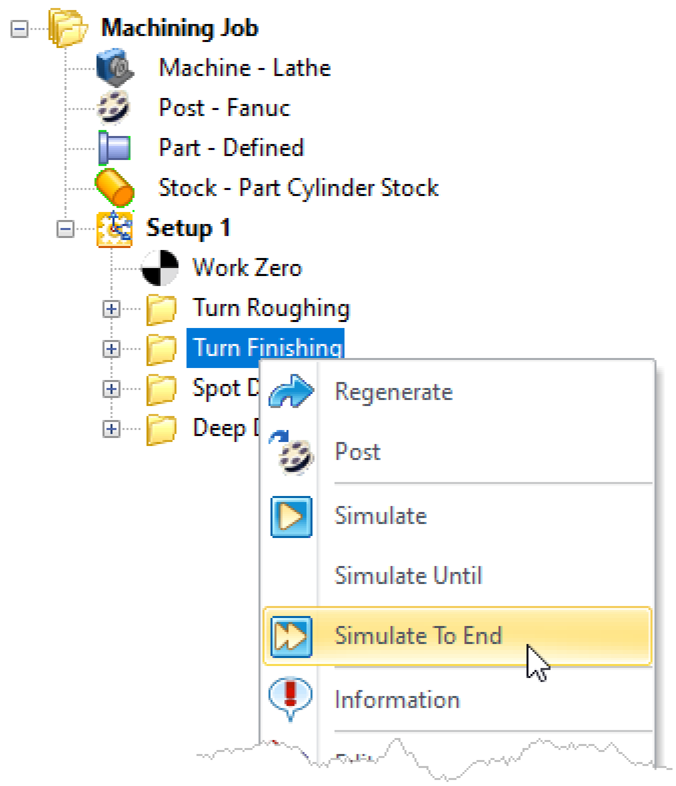
To perform simulations on multiple operations select the last operation, right click and choose Simulate Until. You can also select multiple operations by holding down the Ctrl key.



Simulate Until the Active Operation

Simulate to End

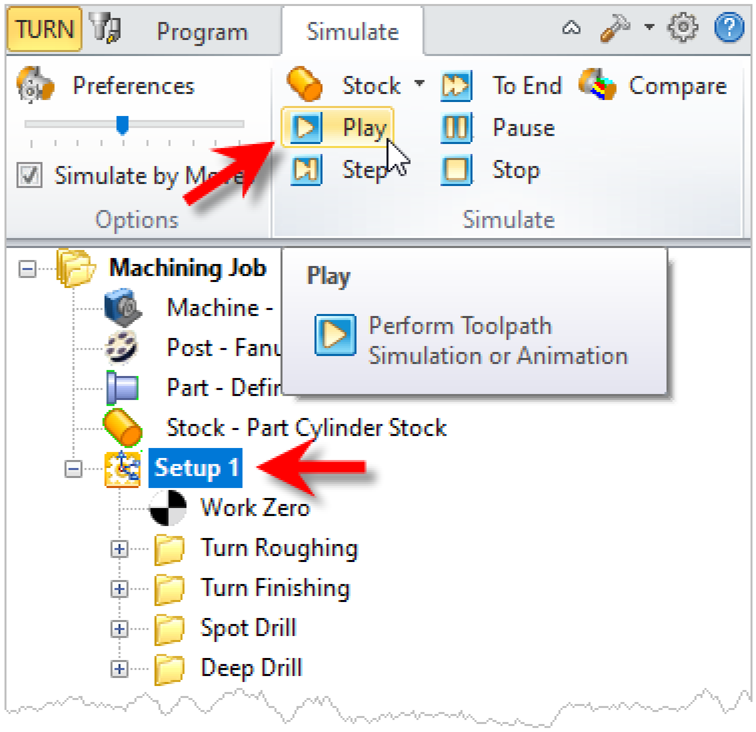
To simulate directly to the end of the Machining Job, right click on any operation and select Simulate to End.



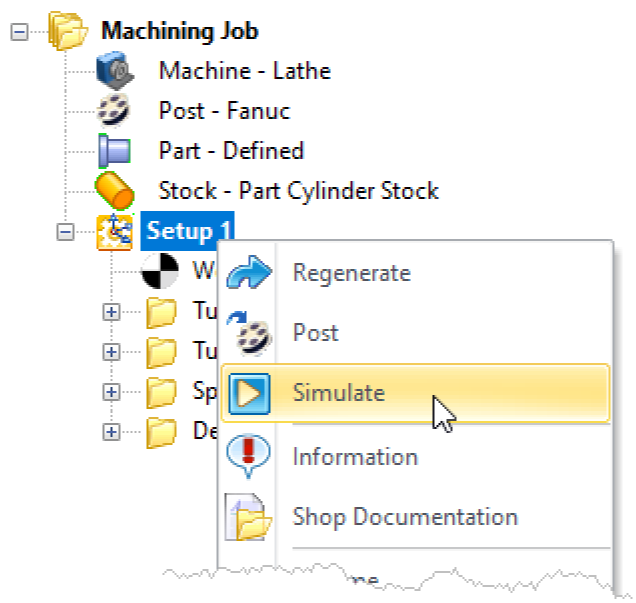
Simulate to End

Simulate all Operations within Active Setup

Alternatively you can select a Setup and select Simulate to simulate all the operations within a Setup.



Simulate all Operations within Active Setup



Simulate all Operations within Active Setup

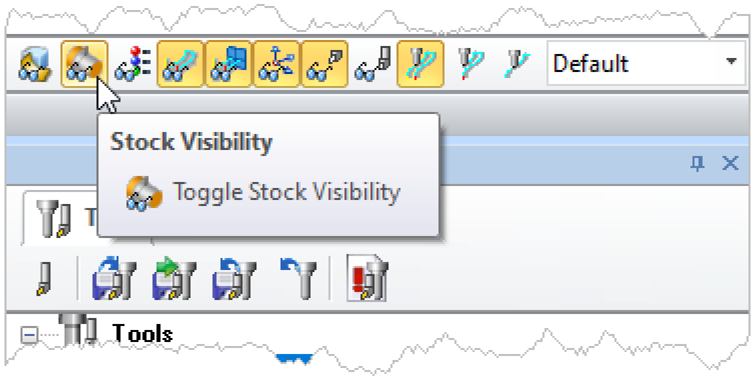
11.1 Tool Animation

Simple tool animation can be carried out in TURN module by using the controls on the Simulate tab.

To Simulate Tool Motions by Move

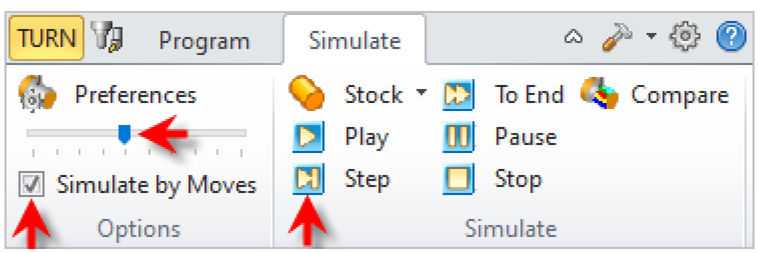
If there is no stock loaded or if the stock is loaded and the Stock Visibility is turned OFF then the tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the Simulate button on the Simulate tab of the browser or by selecting an operation and choosing right click to simulate.

1. Select the Turn Operation from the Machining Browser.
2. If there is stock, turn Stock Visibility OFF.

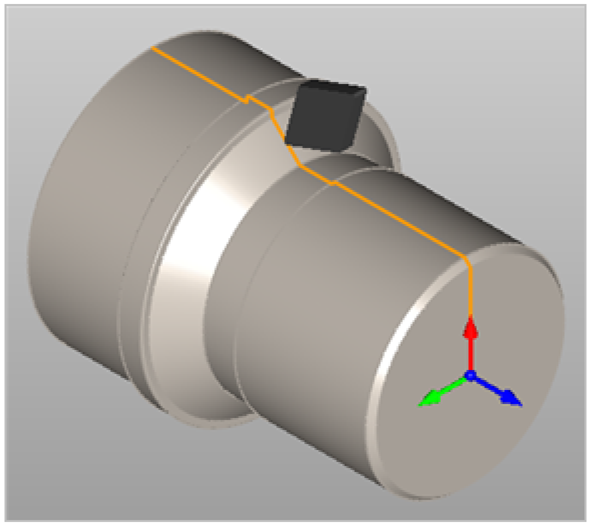


If there is stock, turn Stock Visibility OFF

3. Select the Simulate tab.
4. Check the box next to Simulate by Moves.
5. Adjust the Slider to the far left (i.e., 1 move at a time).
6. Pick the Step button to see one tool motion.
7. Continue to pick the Step button to "step-through" all tool motions.



Simulate Tool Motions by Move

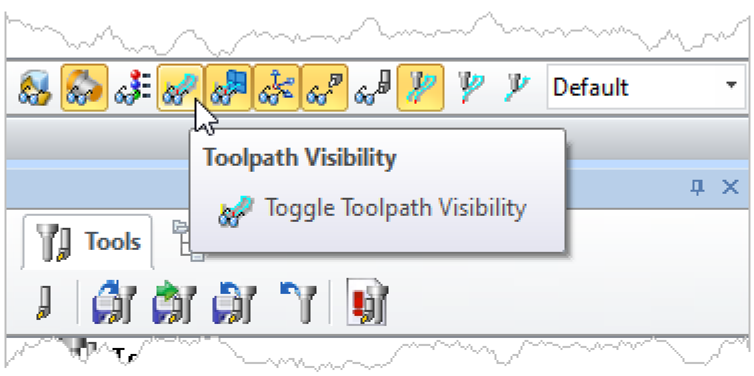


Simulate Tool Motions by Move

To Simulate Tool Motions & Toolpaths by Move

You can also choose to display the toolpath as the tool is being animated. This is a powerful function that allows you to actually watch the toolpath being displayed on the screen incrementally. To do this make sure Toolpath Visibility is turned ON before starting the tool animation along the toolpath.

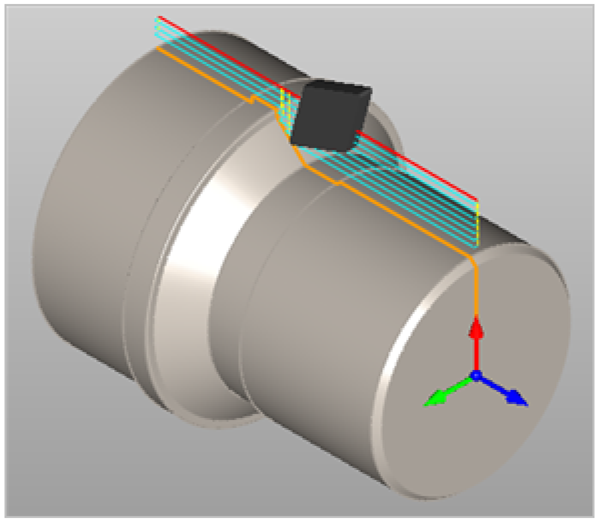
1. Select the Turn Operation from the Machining Browser.
2. If there is stock, turn Stock Visibility OFF.
3. Turn Toolpath Visibility ON.



Turn Toolpath Visibility ON

1. Select the Simulate tab.
2. Check the box next to Simulate by Moves.
3. Adjust the Slider to the far left (i.e., 1 move at a time).
4. Pick the Step button to see one tool motion.

5. Continue to pick the Step button to "step-through" all tool motions.



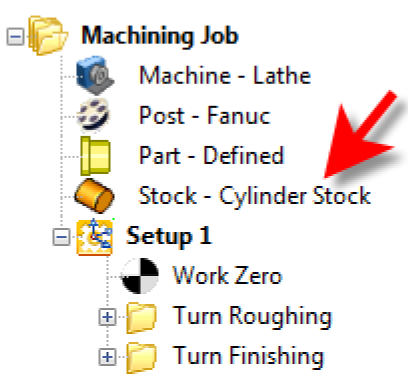
11.2 Cut Material Simulation

TURN Module offers very powerful cut material simulation functionality to allow users to simulate actual machining of the generated toolpaths. To perform cutting simulation, a stock model must be loaded and displayed and a machining operation must be active.

To Simulate Cut Material by Moves

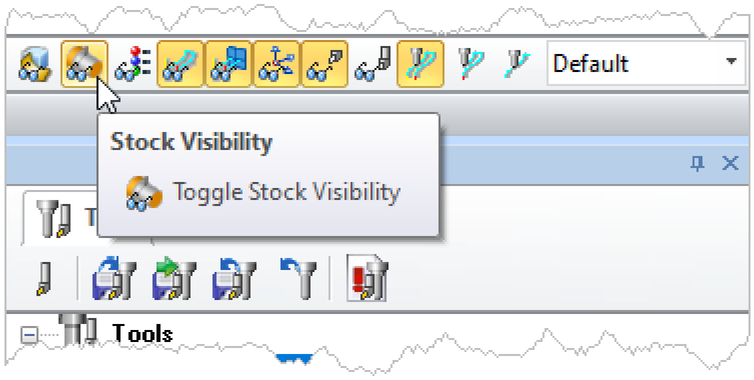
There must be Stock defined and the Stock Visibility turned ON.

1. Make sure Stock is defined for your Machining Job.



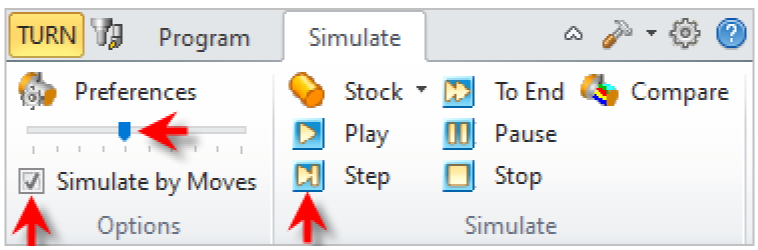
Stock is defined for your Machining Job

2. Select the Turn Operation from the Machining Browser.
3. Turn Stock Visibility ON.

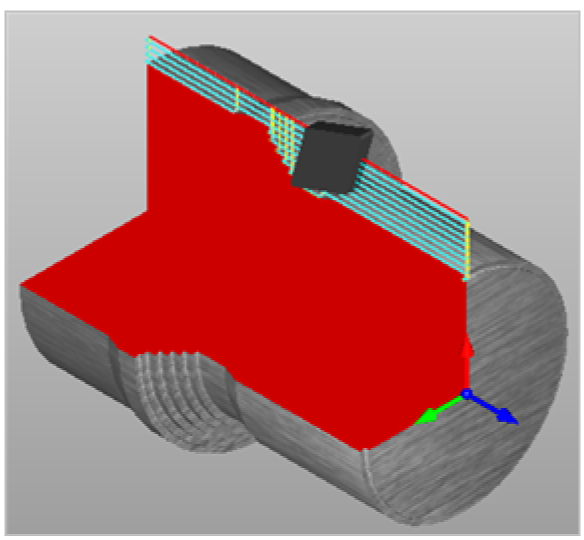


Turn Stock Visibility ON

4. Select the Simulate tab.
5. Check the box next to Simulate by Moves.
6. Adjust the Slider to the far left (i.e., 1 move at a time).
7. Pick the Step button to see one tool motion.
8. Continue to pick the Step button to "step-through" all tool motions.



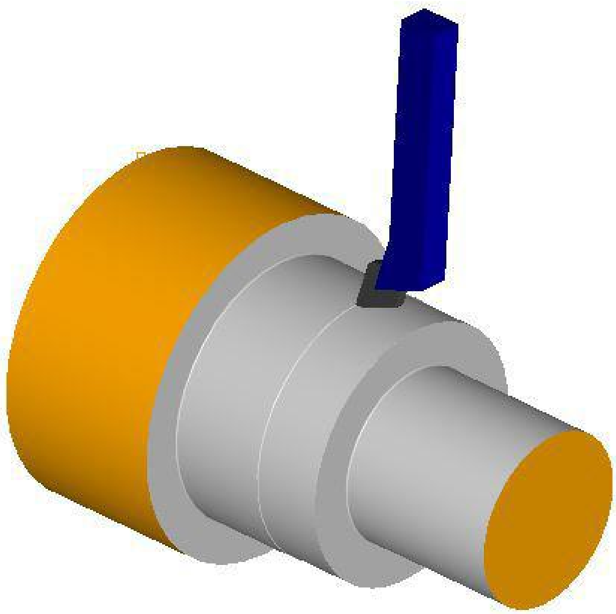
Simulate Tool Motions by Move



Simulating Cut Material Removal

The Simulation is a True 3D Cut Model

The output of the Cut Material Simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated. The cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. An example of cut material simulation is shown below.



The Simulation is a True 3D Cut Model

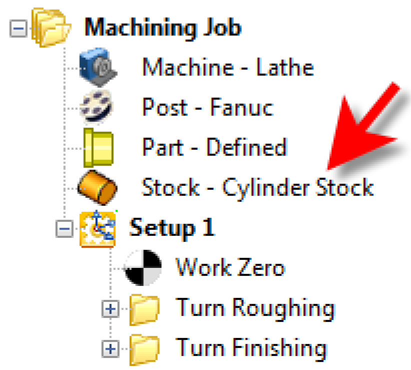
11.3 Material Texture in Simulation

Material texture can be applied to the Cut Material Simulation. This functionality allows users to simulate actual machining of the generated toolpaths with material texture.

To Simulate Cut Material Removal with Material Texture

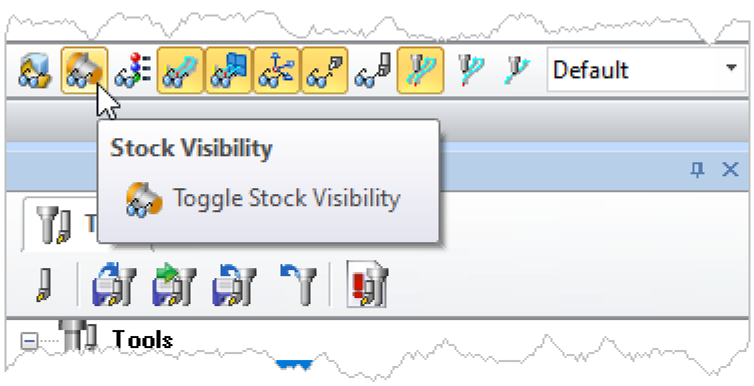
There must be Stock defined and the Stock Visibility turned ON.

1. Make sure Stock is defined for your Machining Job.



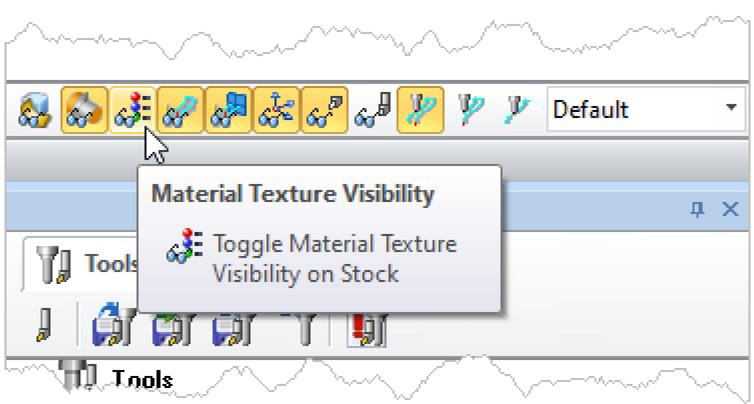
Stock is defined for your Machining Job

2. From the Program tab, select Material Define Stock Material and select a material from the Stock Material dialog box.
3. Select the Turn Operation from the Machining Browser.
4. Turn Stock Visibility ON.



Turn Stock Visibility ON

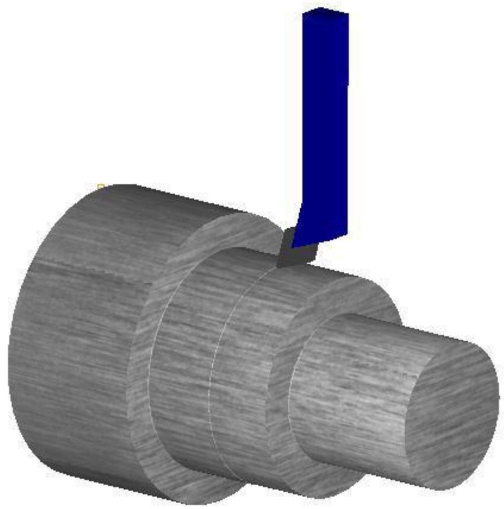
5. Turn Material Texture Visibility ON.



Turn Material Texture Visibility ON

6. Select the Simulate tab.

7. Check the box next to Simulate by Moves.
8. Adjust the Slider to the far left (i.e., 1 move at a time).
9. Pick the Step button to see one tool motion.
10. Continue to pick the Step button to "step-through" all tool motions.
    An example of Cut Material Simulation with Material Texture is shown below. The material is set to Aluminum-6061 under Choose Stock Material of the Stock Material dialog box.

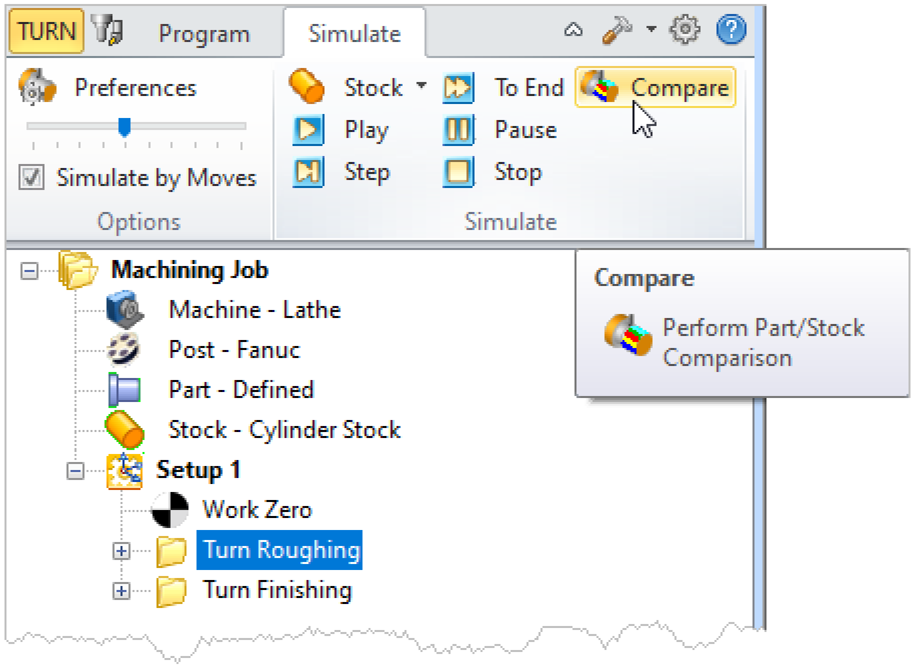


11.4 Part/Stock Compare

This command graphically compares the differences between the part and the stock geometry and displays the results graphically. The part geometry must contain solid/surface/mesh geometry. You can use the Part/Stock Comparison dialog that displays to adjust the tolerance band and continue the comparison. This feature is available in Expert, Professional and Premium configurations.

To Run the Comparison Simulation

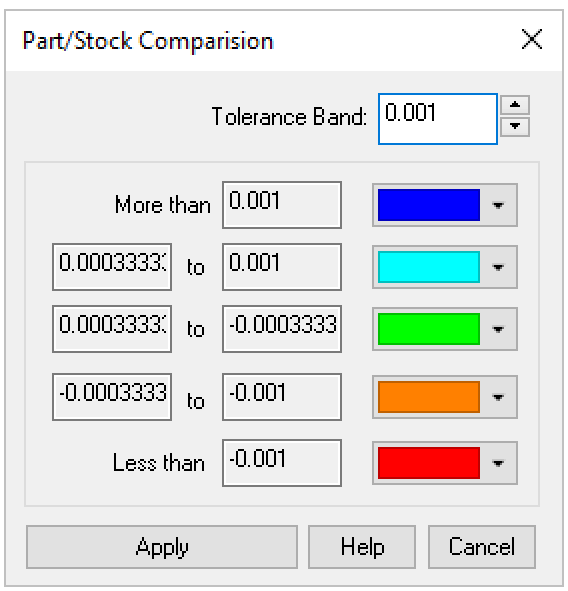
Run the simulation from the Simulate tab and select Perform Part/Stock Comparison.



Location of the Perform Part/Stock Comparison Icon

Dialog Box: Part/Stock Comparison

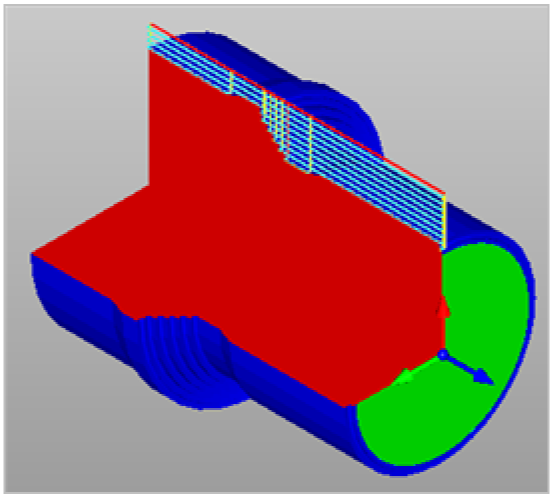
The following dialog is displayed and you can specify the tolerance for part/stock comparison. You can manipulate the part in the graphics window (i.e., pan, zoom, rotate) while this dialog is displayed.



Part/Stock Comparison dialog box

A Visual Stock/Part Model Comparison is Displayed

A visual comparison of the stock model against the part model is displayed. You can color-code areas based on the amount of material remaining or over-cut.



Part/Stock Comparison Model TURN Module)

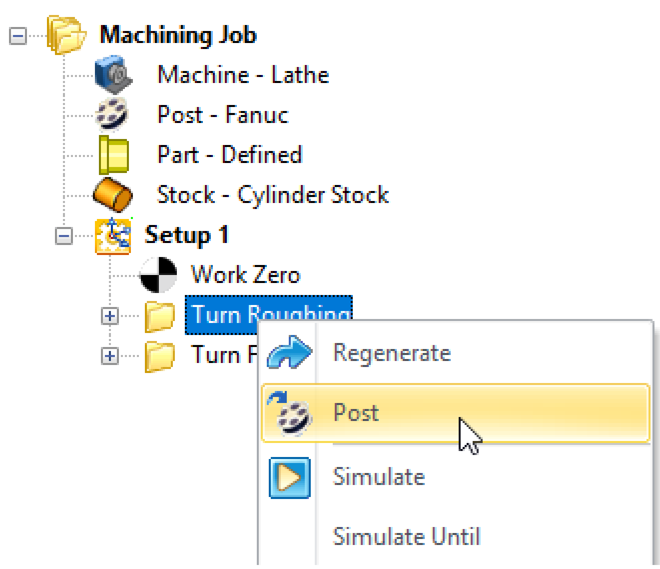
Post Processing Machining Operations

Once machining operations are created they can be post processed to a specific machine controller. To post process a machining operation, select the operation in the Machining Browser, right click and select Post.

To Post Process from the Machining Browser

There must be a Post defined for the Machining Job.

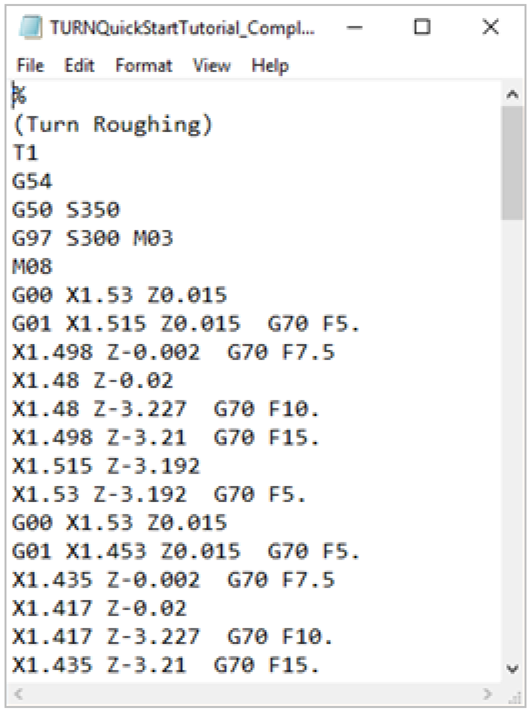
1. If no Post is defined, pick Post Post and then select a post processor from the Set Post Processor Options dialog box.
2. Select the Turn Operation from the Machining Browser.
3. Right-click and select Post.



To Post Process a Turn Operation from the Machining Browser

4. Enter a file name for the posted file and pick Post from the Post & Save As dialog box.

5. The posted file is displayed in Notepad.




```
%
(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
X1.417 Z-3.227  G70 F10.
X1.435 Z-3.21  G70 F15.
```

The posted file is displayed in Notepad

To Post Process Multiple Operations

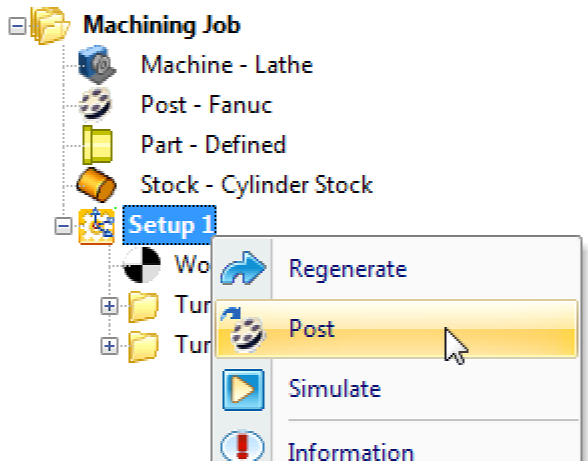
You have the ability to select multiple Turn Operations or the entire set of machining operations and post process all of them with a single button click. To do this:

There must be a Post defined for the Machining Job.

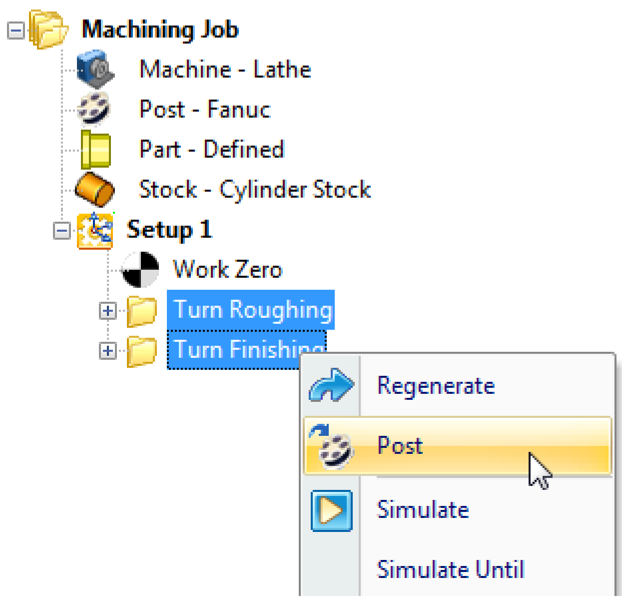
1. If no Post is defined, pick 

Post and then select a post processor from the Set Post Processor Options dialog box.
2. Select the Setup from the Machining Browser. You can also select multiple operations by holding down the <Ctrl> key.

3. Right-click and select Post.



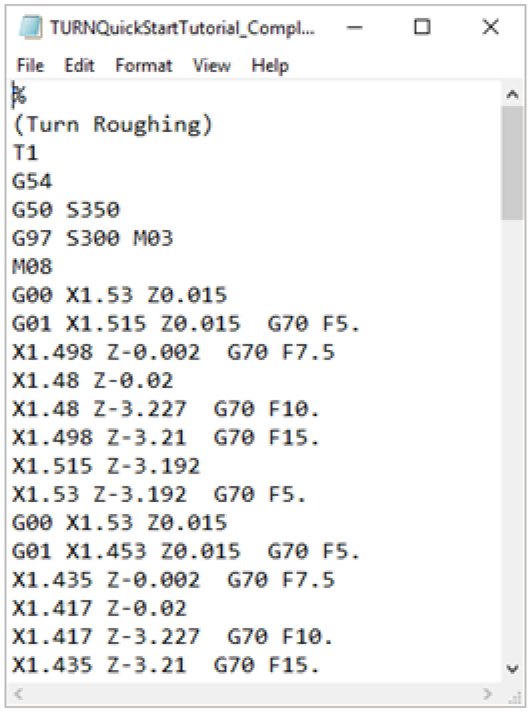
Right-click on a Setup and select Post to post all Turn Operations in the Setup



Example shows selecting multiple operations by holding down the <Ctrl> key for posting.

7. Enter a file name for the posted file and pick Post from the Post & Save As dialog box.

8. The posted file is displayed in Notepad.



```
%
(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
X1.417 Z-3.227  G70 F10.
X1.435 Z-3.21  G70 F15.
```

The posted file is displayed in Notepad

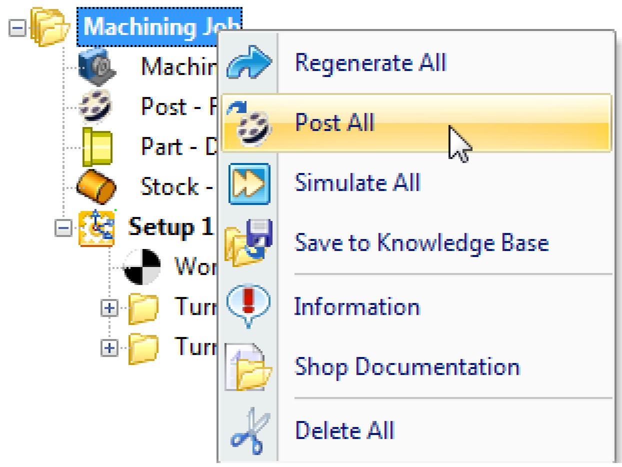
To Post Process All Operations

Alternatively you can select the Machining Job at the root level under the Machining Browser, right click and select Post All.

There must be a Post defined for the Machining Job.

1. If no Post is defined, pick  Post and then select a post processor from the Set Post Processor Options dialog box.
2. Select the Machining Job from the Machining Browser.

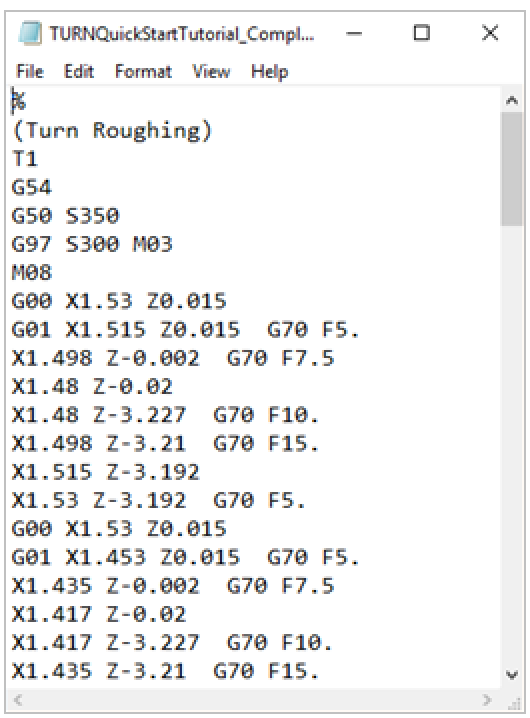
3. Right-click and select Post.



Select the Machining Job from the Machining Browser, right-click and pick Poast All

7. Enter a file name for the posted file and pick Post from the Post & Save As dialog box.

8. The posted file is displayed in Notepad.



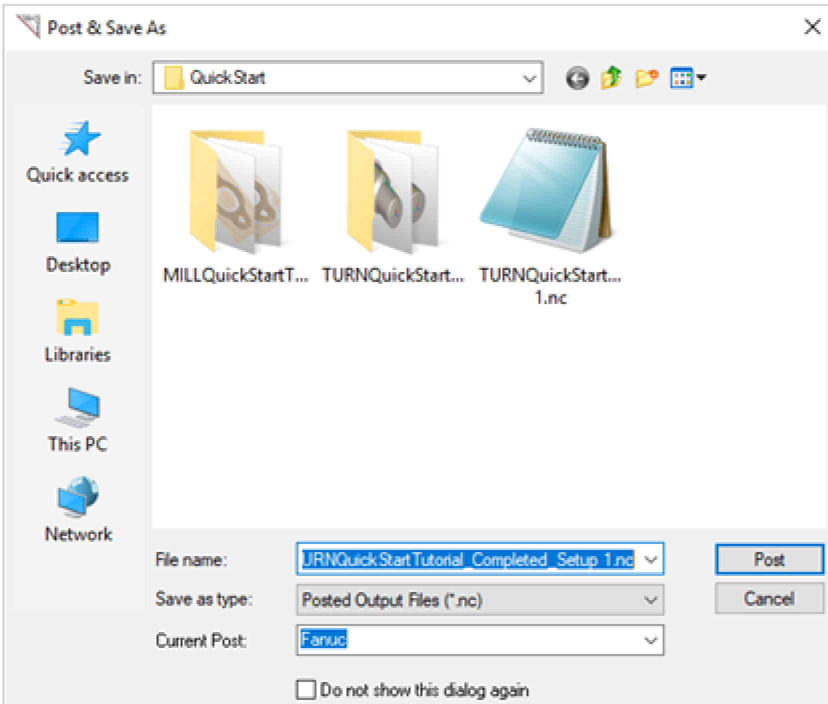
The posted file is displayed in Notepad

About the Post & Save As dialog box

Post-processing can be done from Program and Simulate tabs under the Machining Browser. Selecting Post will display the Post and Save As Dialog. The following are the default settings when the Post and Save As dialog box is displayed:

- Post and Save As dialog points to the folder location where the part geometry is located.
- Save as type – refers to post file extension. This information is obtained from the set post options dialog.
- Current Post - refers to the controller/post processor to post process the toolpath. This information is also obtained from the set post options dialog.

You can override the default settings under the Post & Save As dialog box. Once you click on the Post button in the dialog, post processing will begin and the posted file is located under the specified folder and then displayed in Notepad by default.



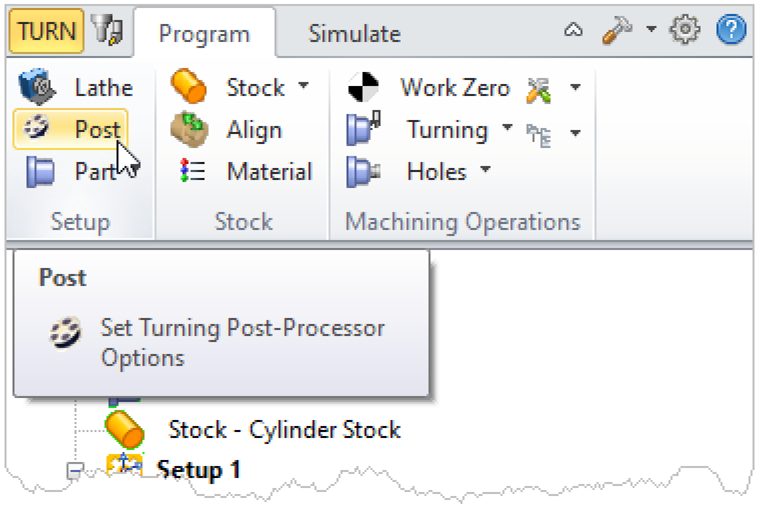
Enter a file name for the posted file and pick Post from the Post & Save As dialog box

12.1 Use Legacy Post

You can specify certain post-processor options and rules for post processing. This is done by selecting "Set Post Options" from the Program tab under the Machining Browser.

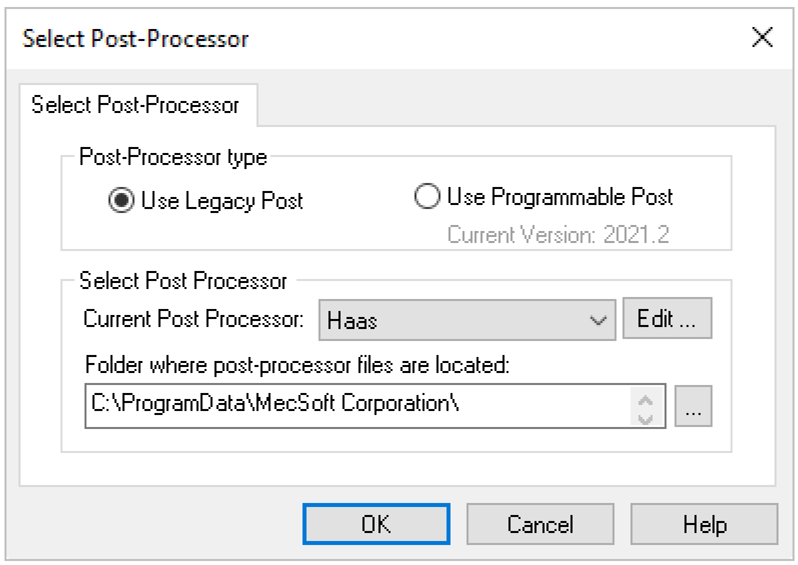
This will bring up the following dialog.

Dialog Box: Set Post-Processor Options



Set Post-Processor Options

See Use Programmable Post to learn more about using the Programmable Post option.



Dialog Box: Set Post-Processor Options

Post Processor Type

Use these options to define the post type to use when posting toolpath operations.

Use Legacy Post

Select this option to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to version 2021.

Use Programmable Post

Select this option to use your *"Programmable Post"* when posting toolpath operations. *"Programmable Post"* refers to your programmable post created with our version 2021 and newer CAM plugins. See Use Programmable Post for documentation.

Current Post Processor

User can change the default post processor by selecting a post from the list of available post processors under Current Post Processor.

Folder where post-processor files are located

MILL module uses macro files with a *.spm* extension to handle post-processing to different controllers. These files are typically located in the "Posts" directory under the RhinoCAM installation folder(*C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Posts\Mill*).

MILL module by default looks in this directory to build the list of available post-processors shown under the Current Post. User can change the post processor by selecting a post from the list of available post processors under Current Post.

To change the post processor file location, you can specify the folder to find the Post-Processor macro files by selecting the "Browser for Folder" button in the dialog.

12.2 Use Programmable Post

The Programmable Post Processor allows you to use Python script files for G-Code generation during toolpath processing by each MecSoft CAM plugin. You can create new post scripts or modify existing ones. Using the Python language for post scripts allows you to more efficiently generate the needed G-Code output allowing the use of programming elements, such as condition processing, loops, logical operations, and more.

You can fine documentation and sample files on our programmable Post here:

C:\Program Files\Rhino x\Plug-ins\RhinoCAM 20xx for Rx\SDK

Archiving Machining Operations

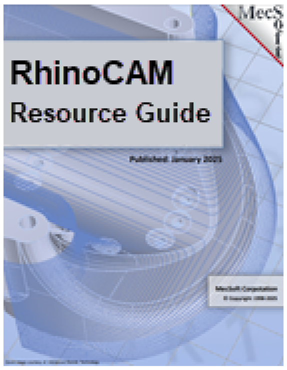
Once machining operations are created they can be archived along with the Rhino part (.3dm) file. This can be accomplished by simply saving the part file. When the part file is retrieved, all archived operations will be loaded along with the part file.

Another powerful way of using archived machining operations is to create operations and save them in a Knowledge Base. You then can load this Knowledge Base file to any part geometry. These machining operations can then be utilized to generate toolpaths to machine the newly loaded geometry. This is a powerful feature that allows you to use existing machining operations without having to recreate these operations in every new session of TURN module.

Find More Resources

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM 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!

What's New | Quick Start Play List

Index

- A -

- B -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- K -

- L -

- M -

- S -

- T -

- U -

- V -

- W -