GIBBSCAM 2025 CAM for Production Machining

Version 2025 : September 2024

Turning

GIBBSCAM

Contents

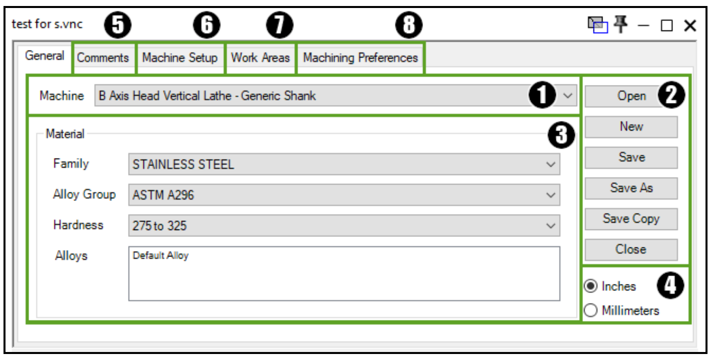
Introduction to Turning

This guide is intended for users of a basic 2-Axis Lathe; however, the lessons learned are applied across more advanced C-Axis and Multi-Task Machines. This guide covers information specific to turning machines; however, most of the interface concepts are similar to other types of machining. After elaborating the concepts of creating geometry, this guide proceeds with information on part set-up, tools, toolpath generation, Posting and communications with a CNC. The most effective way to learn the system is to look through the *Getting Started* guide to become familiar with the system and how it works. You should then complete the *Geometry Creation* tutorial followed by the Lathe tutorial.

For simple explanations of on-screen items and their purpose, use Balloons provided in the Help menu. The *Common Reference* guide will help you with items contained in the various menus and palettes.

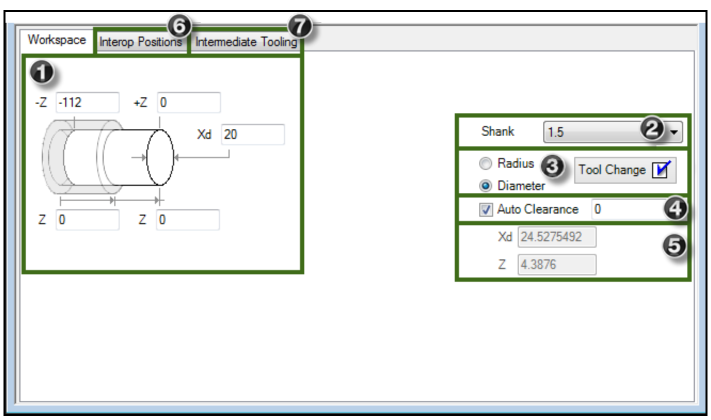
Setup - Document Control Dialog

To display the Document Control dialog (DCD), click the Document button. The top portion of the dialog contains general information about the part, such as the Machine type, Material information, and measurement units. The top portion also provides file management options that you use to control where the file is stored on the computer. For more information about this dialog see the *Getting Started* guide.



1. Machine types, current and available
2. File controls
3. Part material information
4. Measurement units
5. Comments for part and programming
6. Machine Setup
7. Work Areas
8. Machining preferences

Top portion of the Document Control dialog. For complete information, see the *Getting Started* guide.



1. DCD Tabs: Lathe
2. Shank Size
3. X Dimension Style
4. Auto Clearance Option
5. Fixed Clearance Positions
6. Clearance Moves
7. Shank Size

Bottom portion of the Document Control dialog. For complete information, see “DCD Tabs: Lathe” on page 8.

About Clearance Volume

Clearance Volume allows users of advanced machines to say to GibbsCAM, in effect, “Here’s my part; don’t let the tool come too close to it except when cutting. You figure it out so I don’t have to.”

Clearance Volume was devised to address situations where the traditional clearance plane (CP1) is not a good match for machines of more than three axes, especially those with rotary heads or tables, tools with right-angle heads (or any tool that is not Z-aligned), vices that can be held at varying B-axis angles, and the like.

For turning, Clearance Volume is required for eccentric turning, where clearances must be calculated from a CS that is not parallel to the base XZ axis.

Generally: If it is very difficult to calculate the "right" CP1, or if there is no right CP1, then Clearance Volume may offer a better solution.

Caveats:Interop moves generated by Clearance Volume contain 5-axis simultaneous moves; thus it is best if the control has TCP capabilities, and it is unsuitable for machines that have indexing rotary axes or rotary axes that must be clamped between moves.

For complete information on Clearance Volume, see the *Common Reference* guide appendix.

DCD Tabs: Lathe

Workspace

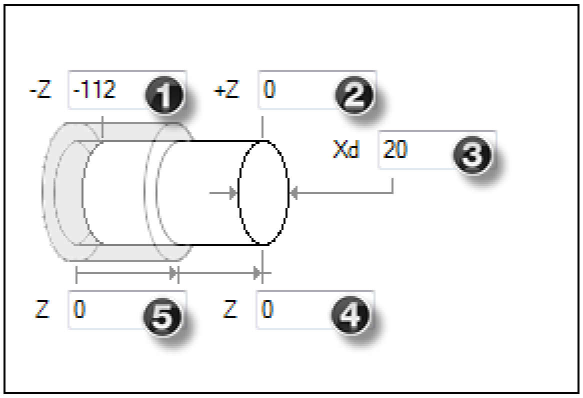
This section of the Document Control dialog is used to specify the starting size of the part stock. The stock size entered here will be used by the system to determine positioning moves when using the Auto Clearance function. The stock dimensions will also be taken into account when generating toolpaths with the Material Only option selected in the Process dialog. If custom stock has been created, the system will use the custom stock size for toolpath and positioning moves. In that case, the values entered here will only be used to draw the stock outline and origin marker correctly.

Stock Settings

Stock diagram. The stock type depends on the current MDD's settings in the Part Station page. Other settings in the MDD, such as Has Guide Bushings, affect stock-related controls presented on the **Workspace** page.

Cylindrical Stock (No Bushing)

If the part station has Turning Enabled selected, and does not have a guide bushing, the stock diagram portion of the **Workspace** page presents the following controls.

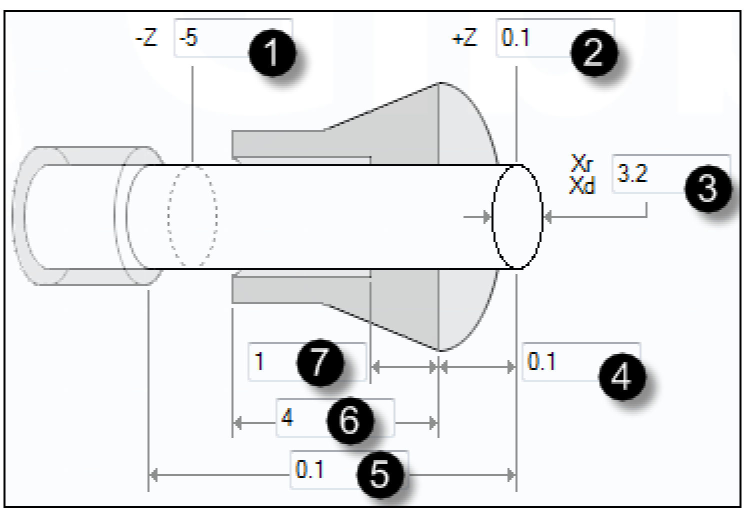


1. Negative depth
2. Positive Depth
3. X Dimension (Radial or Diametral)
4. Distance between Face of stock and Chuck or Spindle
5. Z Thickness of Chuck face.

The text box for the X dimension will be a radius or diameter value depending on which option is selected for the X Dimension Style.

Cylindrical Stock (With Guide Bushing)

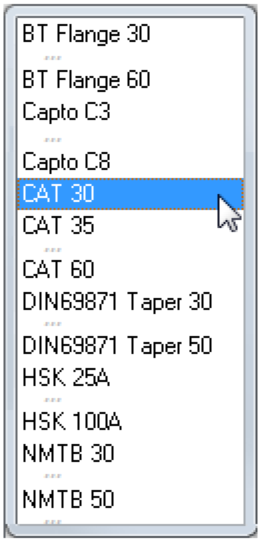
If the part station has Turning Enabled selected, and has a guide bushing, the stock diagram portion of the **Workspace** page presents the following controls.



1. Negative depth
2. Positive Depth
3. X Dimension (Radial or Diametral)
4. Stickout length, measured from the front of the guide bushing to the front face of the part
5. Distance between the face of the stock and the chuck (or part station).
6. Depth of the guide bushing, measured from the guide bushing's back to its front
7. Pullback distance of the guide bushing, measured from the back of the pullback distance to the front of the guide bushing

Other controls:
Outer Diameter of Guide Bushing
Check Guide Bushing

Mill Class



This menu allows you to select the classification of mill tool holders found on the machine this part will be cut on. The six basic holder types on the list include: BT; Capto (Sandvik Capto); CAT (Caterpillar); DIN69871; HSK(type A hollow taper shank holders); and NMTB(National Machine Tool Builder standard).

Each of the types has multiple sizes. The selection of this back end of the holder affects the tool-specific front end holders available in the Tool dialog. The items found in this menu can be modified using File > Preferences, Machining Prefs tab.

Shank Size

This is the shank size of lathe tool holders for the current machine. This setting controls what tool holders are actually available when defining tools.

X Dimension Style

These two radio buttons determine whether the X values for the part are input as radii or diameters. Some text boxes in particular dialogs specify that the value entered is either a radius or a diameter value, regardless of the selection made here.

Auto Clearance Option

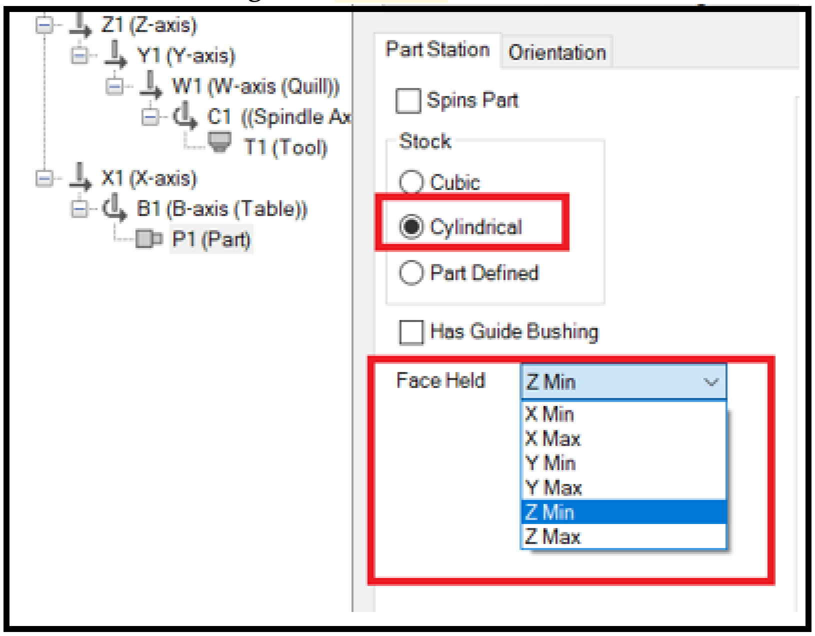
When the Auto Clearance option is turned on, the system will calculate positioning moves between operations. These positions will be dynamically calculated, meaning that they will change as the material conditions of the part change. The value entered is an offset amount from the current part stock that the system will use to maintain adequate clearance from the material. Refer to “Clearance Moves” on page 111 for more information.

Fixed Clearance Positions

Fixed Clearance positions must be entered when the Auto Clearance option is turned off. When the Auto Clearance option is on, the fixed clearance position text boxes will be grayed out. The X and Z values entered specify the location the tool will rapid to and from during a tool change. This position will also be used when moving from one approach type to another. Refer to “Clearance Moves” on page 111 for more information.

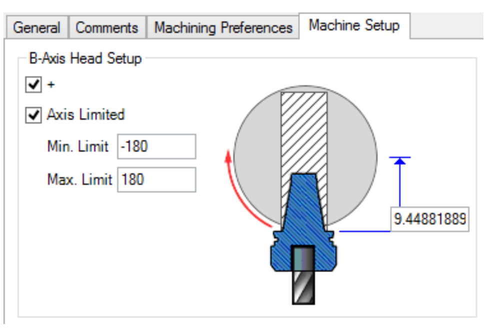
Non-Z Aligned Stock and Clearances

Non-spinning part stations using cylindrical stock and Clearance Volumes enabled can choose a Face Held of X, Y, or Z, minimum or maximum, instead of being restricted to Z minimum. This allows the use of Cylindrical or Part-defined stock types on 4-axis vertical mills while retaining the correct Z orientation. In order to change the Face Held, a custom MDD must be created in



the Machine Manager.

Machine Setup tab for B-Axis Lathe



Head setup for rotary axis:

- Direction ([-] or [+])
- If axis-limited: minimum and maximum.
- Offset from pivot.

Material Database

To open the Materials dialog
Use either of the following methods:

- From the File menu, select Materials.
- From within a process dialog, click the Material Material button.

You use the Materials Database for storing and quickly retrieving feeds and speeds for various types of materials. The Materials Database contains default material information and can also include the CutDATA™ material library, if you have purchased this option. You can also enter custom information into the Materials Database. For more information about the Materials Database, see the *Common Reference* guide.

Please note: When deleting any item in the Materials Database (Family, Alloy Group or Material), great care must be taken as the undo function is not available.

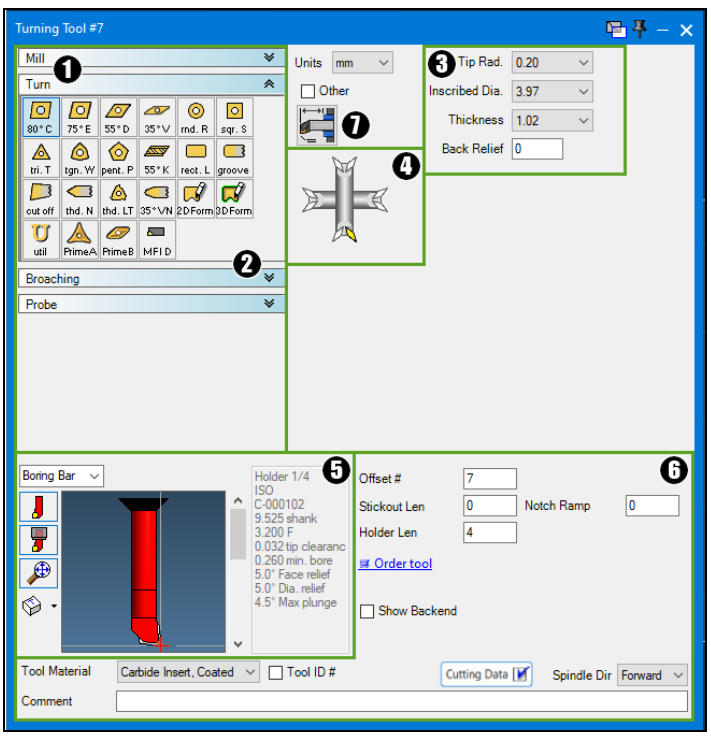
Tools

You select the tools that you want to use for machining processes using tool tiles in the **Tools List**. For more information on the Tools List and the **Tool** dialog, see the "Tools" section in the *Getting Started* guide.

The following material describes tools used specifically for turning.

Lathe Tool dialog

To define lathe tools, you must select a Lathe machine type in the Document dialog. The basic turning tools are created using the Tool Creation dialog shown below. The following section describes each of the dialog items.



1. Tool Type
2. Insert Types
3. Insert Specifications
4. Insert Orientation Diagram
5. Tool Holder Definition
6. Tool Options
7. Lathe Tool Offset Data

Order Tool

For some tools, the lower center portion of Tool dialog might display this link: Order tool

Its presence indicates that the tool was imported from a manufacturer's catalog. Clicking the Order tool link directs you to the catalog of the specific tool library.

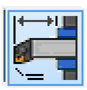
Tool Comment
You can type an optional comment for each tool. The comment is output in the finished code at the beginning of every operation that uses the tool. The comment also displays in the Tooltip for the tool in the Tool list.

Tool Type

Scroll up or down to choose between mill tools and lathe inserts. You should only use mill tools with the face drilling function, unless the Mill/Turn or Multi-Task Machining module is installed. For more information about specific Mill tools, see the *Mill* guide.

Tool Options

Units
Use the Units pull-down menu to set the unit of measurement for the current tool. For each tool you can specify the dimensions in either imperial or metric. Tool units can differ from Part units. Tool unit settings do not affect the units for the lower portion of the same tool dialog, such as Stickout and Holder Length which use the unit specified in the DCD.

 Offsets

When a custom holder is applied, the system will calculate holder offsets using data from the Toolblock (if used) and the Toolholder. For more information on offsets see the Lathe Tool Offset Data section.

Spindle Direction Forward/Reverse
Forward will turn the spindle in the forward or normal direction. Selecting Reverse will reverse the spindle.

Offset
Normally, the offset number of the tool is determined by its location in the Tool List. This box allows the user to override that default with a different number.

Deflection Compensation
If this option is turned on, all contour and rough toolpaths generated with this tool will contain deflection tool offset utility markers at every location in the toolpath where deflection occurs. This lets you fine-tune the deflection compensation that occurs while using this particular groove tool.

Cut
Choose from the dropdown to either cut on the X- or X+ side.

Tool ID
Enter the tool ID you wish to use instead of the tool list position. Note that a Tool ID greater than 999 will display on tiles as ##, because tiles are not big enough to display four-digit tool IDs.

Tool Material
This is a pop-up menu used to specify the material of the tool. The information given here is used by the Material Database as another factor in determining speeds and feeds. The default setting for Lathe parts is Carbide Insert, Coated.

Stickout Length
Distance from the holder to the contact tip.

Notch Ramp
By selecting a notch ramp amount in a Tool dialog, the toolpath will be created by adding the ramp value to alternating strokes: one with, one without. In Roughing Operations, this will reduce the depth of cut on one stroke and increase it on the next. Please ensure that the ramp value is smaller than the depth of cut. (Notch Ramp is not available for Groove, Cut Off or Thread tools.)

Comment
This is a comment associated with the tool. It will be output in the finished code at the beginning of every operation that uses this tool.

Insert Types

You select the type of insert to use with the tool holder. The Insert Specs change depending on the insert you select. Below is a table of Insert Types that includes available specifications for each type. In many cases, checking Other displays different options. Any additional modifications are noted in the table. For a detailed description of each option, see Insert Specifications on page 17.

80° Diamond Insert 75° Diamond Insert 55° Diamond Insert 35° Diamond Insert Square Insert Triangle Insert Trigon Insert Pentagon Insert	Tip Radius on page 20 Inscribed Diameter on page 19 Thickness on page 20 Other on page 20
Round Insert	Tip Radius on page 20 Thickness on page 20 Included Angle on page 19 Other on page 20

55° Parallelogram 35° Profiling Groove Style Insert	Tip Radius on page 20 Tip Width on page 20 Thickness on page 20 Other on page 20 Length on page 20
Rectangle	Tip Radius on page 20 Size on page 20 Thickness on page 20 Other on page 20 Checking Other replaces Size with insert Length (L) and Width(W).
Grooving Insert	Tip Width on page 20 Tip Radius on page 20 Insert Width on page 20 Full Radius on page 19 Face Angle on page 19 Tip Length on page 20 Deflection Compensation on page 14 Other on page 20 Checking Other replaces Full Radius with Length on page 20.
Cut Off Insert	Tip Width on page 20 Tip Radius on page 20 Face Angle on page 19 Other on page 20 Length on page 20
Groove Style Threading Insert	Style on page 20 TPI on page 20 Insert Width on page 20 Insert Type on page 20 Other on page 20

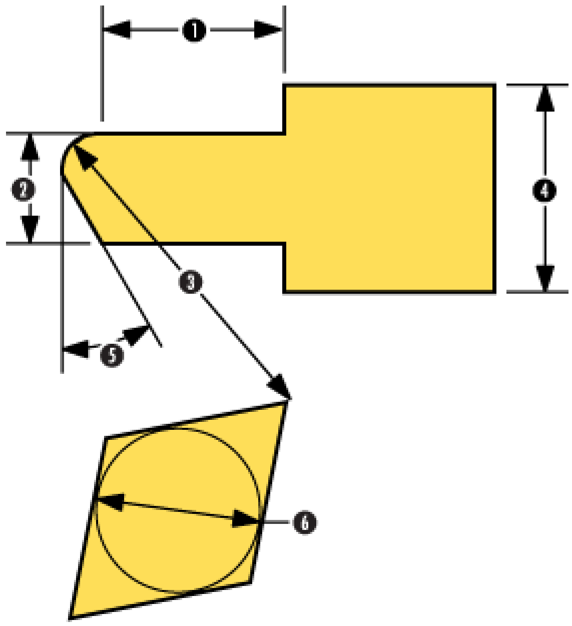
	Checking Other removes the TPI option and replaces Insert Type with Length .
Lay Down Style Threading Insert	Style on page 20 TPI on page 20 Inscribed Diameter on page 19 Other on page 20 Checking Other adds additional tool specification.
2D Form Tool	Thickness on page 20 Entry/Exit Angle on page 19 For additional data, see Form Tool (2D or 3D) on page 20.
3D Form Tool	Thickness on page 20 Entry/Exit Angle on page 19 For additional data, see Form Tool (2D or 3D) on page 20.
Utility Tool	
PrimeA CoroTurn® Prime Type A	For additonal data, see PrimeTurning Process on page 87
PrimeB CoroTurn® Prime Type B	
MFI D Multifunction Indexable Drill Turning and Offset Drilling	For additional data, see Multifunction Tool Definition on page 23.

Insert Specifications

This information will change depending on the currently selected insert type and MDD. Each of the pull-down menus will limit the selections in the pull-down menus that follow it. Choice of tip radius changes the choices presented for inscribed diameter and thickness. Your choice of inscribed diameter changes the choices presented for thickness.

Your choices in the pull down menus limit the available tool holders and boring bars in the holder diagram. If no tool holders or boring bars are available or the other check box is selected, then you can enter any specifications you want.

When the Other checkbox situated next to the tool specifications is selected, you can enter any tool specifications you want.



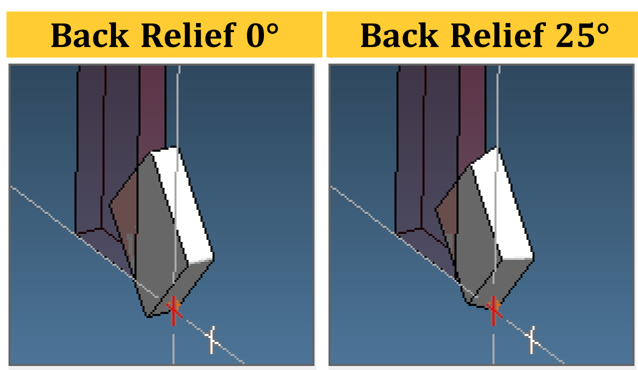
1. Tip Length
2. Tip Width
3. Tip Radius
4. Insert Width
5. Face Angle
6. Inscribed Diameter

Cutting B (Mach Cutting B)
Actual angle used to create toolpath. Mach Cutting B - This is the value actually sent to the machine as set in the MDD.

Please note that if you require to use the same tool at different angles, make copies of the tool with different angle settings.

Back Relief
The back relief of a tool (between zero and 30 degrees) can be entered in this field.

Back Relief is important for modeling tool shape to evaluate for collisions in modeling the entire tool for assessing collisions during interpolated turning processes, such as eccentric, elliptical and U-axis turning.



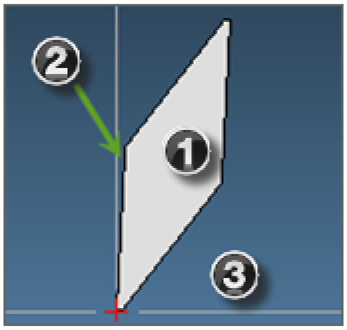
Setup B (Mach Setup B)
Touch-off angle of the tool tip when setting up offsets. Mach Setup B - This is the value actually sent to the machine as set in the MDD.

Diameter Relief
The angle the tip is approaching. Changing this affects the Face Relief, because the sum of the three angles (insert, face relief angle, and diameter relief angle) must be 90 degrees. A Diameter

Relief of 0 degrees places the insert edge on the part face. +(?) degrees moves the edge away from the part face. -(?) degrees moves the edge into the face.

Face Relief

The angle of the insert's approach. Changing this affects the Diameter Relief.



1. 35° Insert
2. 3° Face Relief
3. 52° Diameter Relief

Entry/Exit Angle

The angle used for plunging into and retracting out of material before and after cutting.

Face Angle

The angle of the insert's cutting face.

Full Radius

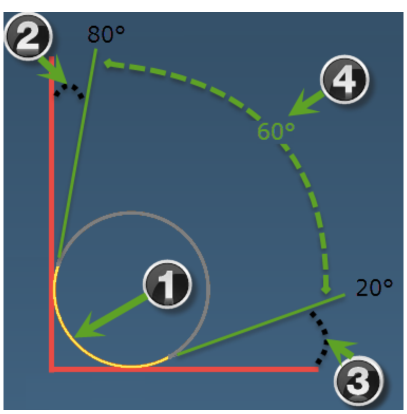
When enabled, this option will limit the grooving inserts available to only those with a full radius tip.

Inscribed Diameter

The inscribed diameter of the insert for triangular thread tools. This is the diameter of a circle that fits exactly within the boundaries of the insert.

Included Angle

Used to define how much of the tool has cutting surface, for Round inserts. The value represents the angle between the two tangents to the ends of the cutting part of the tool (see diagram below). The Face Relief and Diameter Relief settings are calculated automatically from this value.



1. Cutting surface
2. Face relief angle
3. Diameter relief angle
4. Included angle

Insert Face Up

Check this box if the tool has its insert face up, looking from the spindles ZX view.

Insert Width
The width of the insert.

Insert Type
Type of thread the insert will cut.

Length
The length of the insert.

Shank Front Attachment
When this is unchecked, the standard attachment point will be used — in other words, the back face of the shank will be placed against the toolblock. When this is checked, the attachment point is at the front face of the shank.

Other
If this item is on, the insert specs will switch from pop-up menus to text boxes. Any value can be entered in the text boxes. The type of tool holder will automatically be set to None(although there might be tool holder or boring bar selections available).

Size
The IC size of the rectangle. If the Other button is turned on with this type of insert, then the length and width of the insert need to be entered instead of the size.

Sub Pos
This allows input of a turret or slide sub position for the tool.

Thickness
The thickness of the insert.

Tip Length
The length of the tip

Tip Radius
The tip radius of the insert.

Tip Width
The width of the tip of the insert.

TPI
The threads per inch specified by the blueprint.

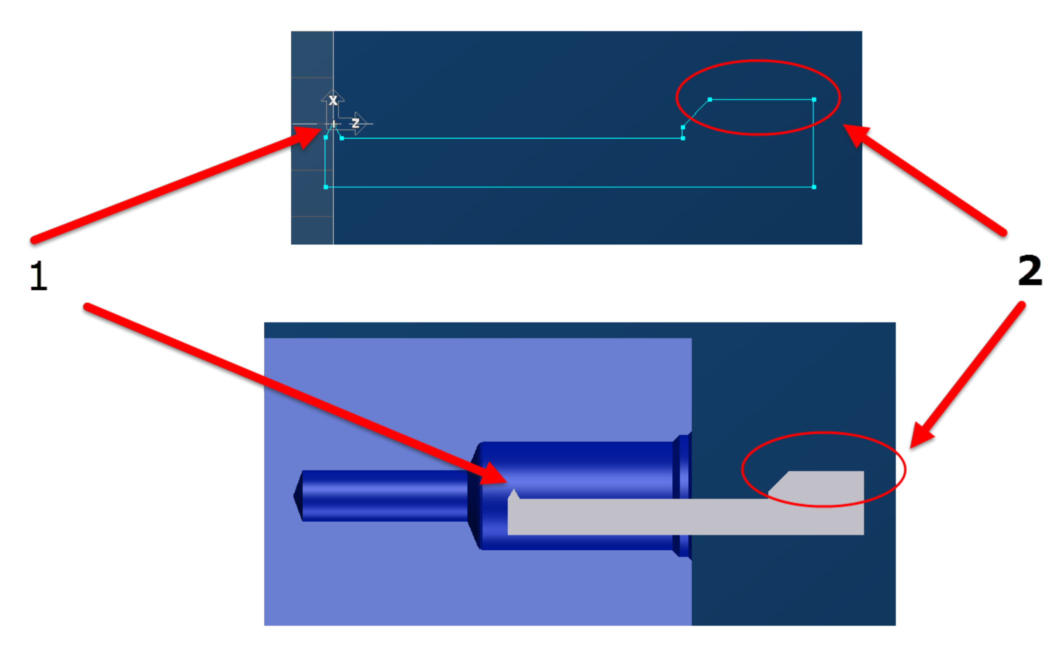
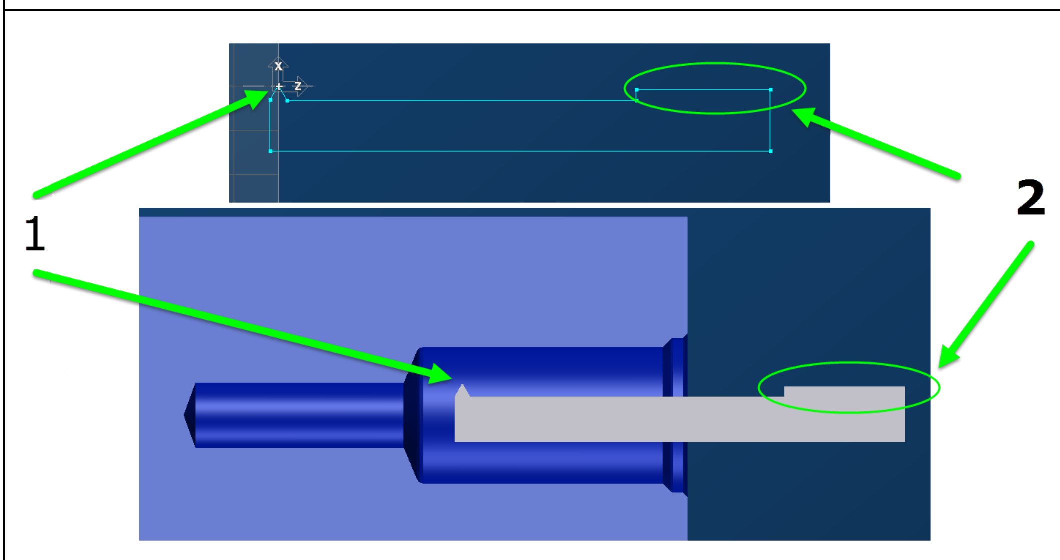
Style
The thread style of the insert.

Tip Width
The width of the insert. This is used when the tip width and the insert width would be the same measurement.

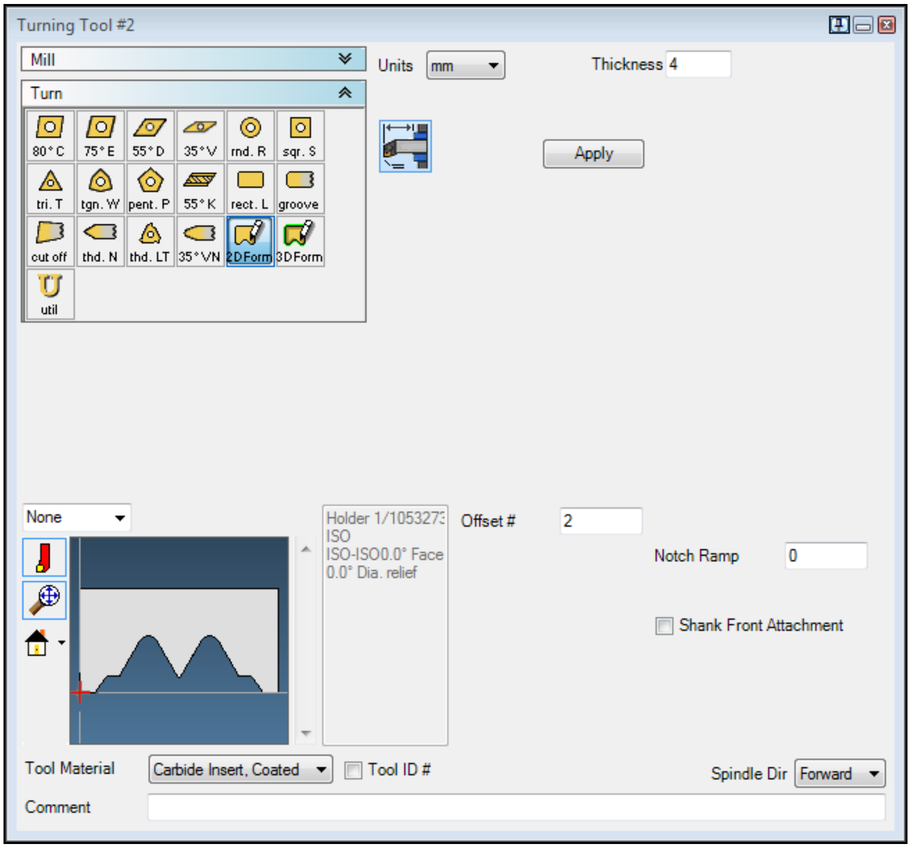
Form Tool (2D or 3D)
The system supports custom form tools for Lathe parts. Unlike Mill parts, Lathe form tools must be a closed shape. Be sure to create the shape with the part origin in mind. The origin is used as the touch-off point for the tool. All posted output with this tool is relative to this point. Avoid concave shapes in your tool geometry, if possible, unless that shape is actually used in the material removal.

A form tool does not have a tip radius, and so tool edge path is unavailable for form tools. The touch-off point is shown as a red cross in the tool diagram. If the toolpath generated from a form tool is undesirable, avoid drawing form tool geometry that is irrelevant to the actual cutting, such as the tool holder or areas of the insert that will not actually be used.

The illustrations below provide examples of correct and incorrect form tool geometry, with its effect on toolpath. In the first example, form tool geometry extends past the touch-off point, resulting in a collision. In the second, form tool geometry is no higher than the touch-off point.

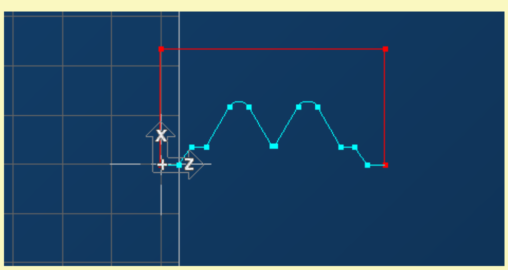
	Bad results: Geometry(2) is higher than touch-off point(1)
	Good results: Geometry(2) is lower than touch-off point(1)

Important: The value for Spline Machining Tolerance is used by 2D Form tools and 3D Form tools, which typically use free-form curves (spline geometry) in their construction. This value is set in the Document Control dialog (DCD), **Machining Preferences** tab.



The Steps To Make a Form Tool:

1. Create profile geometry, taking into account the touch-off point. The system uses the geometrys CS origin as the tool's touch-off point as illustrated below. (In this example we have designated the non-cutting surfaces as "air" geometry.)



2. Select the geometry (double-click it).

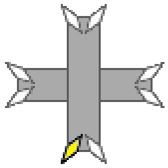
3. Click an empty tool tile and choose the appropriate Form Tool type.

(eg. 

)

4. Click the Apply button.

Insert Orientation Diagram



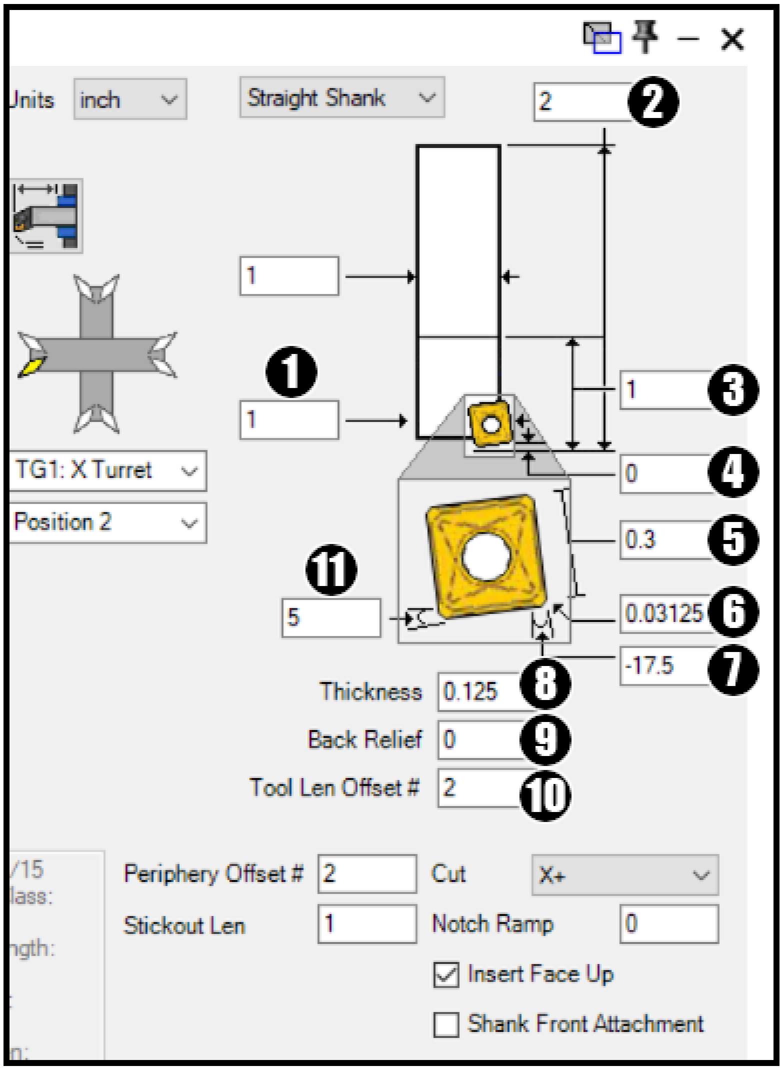
This diagram is used to specify the orientation of the insert in the tool holder or boring bar. Changing the information will not affect the availability of other items in the dialog, but it will change the orientation of the drawing in the Holder diagram.

Multifunction Tool Definition

Multifunction Indexable Drill (MFID) Turning and Offset Drilling

This tool type normally consists of a cylindrical body carrying one or more asymmetrically placed inserts (such as the Sandvik CoroDrill® 880), which produce an approximately flat-bottomed hole when used as a drill. The outermost insert, which produces the final hole diameter, is designated the ”periphery insert”. The periphery insert can also be used for turning operation, most commonly ID boring, but also facing and OD cutting if necessary. Two tool offsets must be defined. The length offset programs the center of tool body (like a conventional drill), while the periphery offset programs the corner of the periphery insert (like a conventional turning tool).

A new option in Lathe Holes allows the use of MFI Drill tools for radial-offset drilling. Enter an offset amount between 0 and the tool radius to create a hole that is slightly larger than your tool. You can also select whether to output toolpath with the length offset at the center of the tool (typical for drilling) or with the periphery offset at the periphery insert corner, which may provide better dimensional control.



1. Cutting Diameter
2. Overall Flute Length
3. Flute Length
4. Point Length
5. Periphery Insert Side Length
6. Periphery Insert Corner Radius
7. Periphery Insert Diameter Relief
8. Periphery Insert Thickness
9. Perophery Insert Back Relief
10. Tool Offset Number
11. Periphery Insert Face Relief

Flute Length
Overall length, and shank (straight/square/taper), as a milling tool. This information will also be used to construct the lathe "toolholder."

Point Length
The difference between the shoulder depth and the deepest depth cut by the drill.

Periphery Insert Side Length
This will determine the cutting edge size when used as a turning tool.

Periphery Insert Corner Radius
The corner or tip radius of the periphery insert.

Periphery Insert Diameter Relief
Negative diameter relief will cut a tapered hole and positive will cut a straight sided hole.

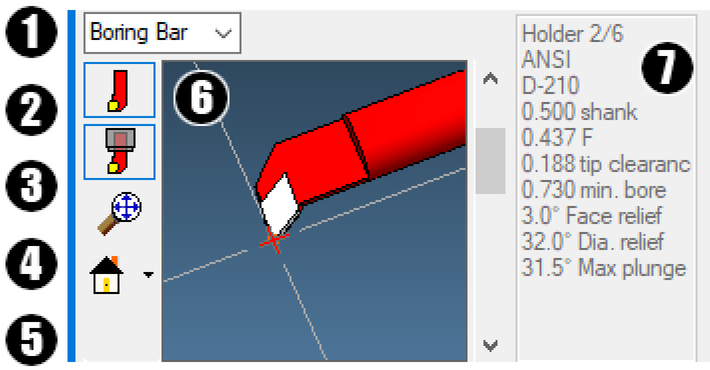
Periphery Insert Back Relief
Taper angle from the front to the back of the insert. Back relief avoids cutting with the back of the tool when doing ID turning or boring.

Tool Offset Number
Tip offset (Z). The distance between the overall (mill-style) tip, and the actual touchoff point of the (lathe-style) periphery insert.

Periphery Insert Face Relief
Positive face relief will cut a cone to the point length. 0 or negative face relief will cut a flat-bottomed hole.

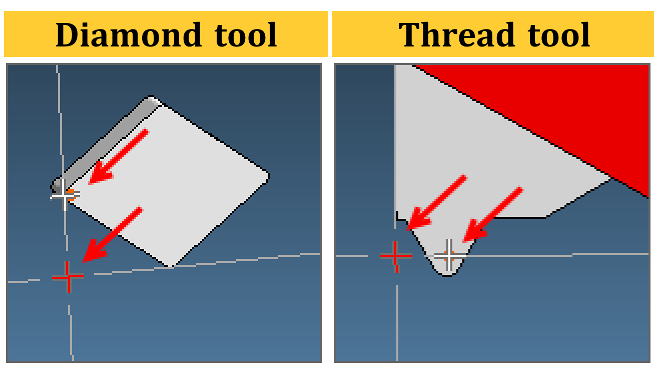
Tool Holder Definition

Tool Holder Display



1. Tool Holder Dropdown
2. Show/Hide Holder display
3. Show/Hide Tool Blocks
4. Unzoom
5. View dropdown
6. Tool/Holder display
7. Holder specification

The tool display provides information about the touch off and type of holder or boring bar that is used for the insert. The red cross-hatch on the insert shows the location of the current Touch-off point. Most Lathe tools have a choice of touch-off points. Simply click the required cross-hatch. Choosing an alternative touch-of point will affect the toolpath and the output code.



View Controls
The Tool/Holder display is mouse-enabled. You can mouse-drag a rectangle to expand an area, turn the mouse wheel to zoom in or out, or hold down the wheel and move the mouse to change the view.

Show/Hide Holder
A thin blue line is drawn around the icon if holder is displayed.

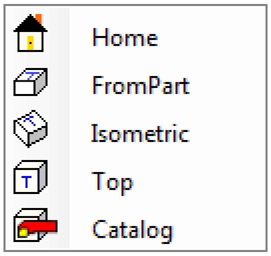
Show/Hide Tool Blocks
A thin blue line is drawn around the icon if Intermediate Tooling toolblocks are displayed.

Unzoom

This is useful if you have expanded the tool using the mouse.

View control Dropdown

This allows you to choose from four preset views. (Catalog is only available for turning tools.)

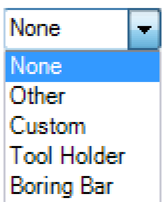


Catalog

This option will display the tool holder as shown in the Other Holder option for Lathe tools.

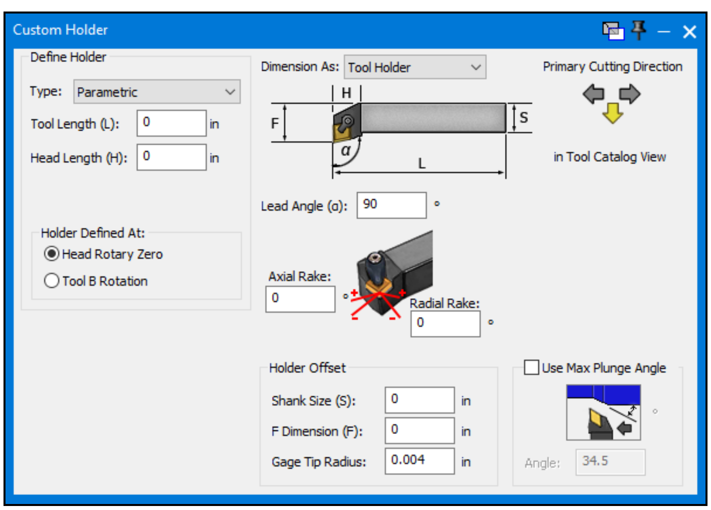
Tool Holder Dropdown

Depending on the tool selected and Machine type used, there are up to five tool holder options.



Custom

This enables you to define a Tool Holder or Boring Bar. You can choose either a Solid or a Profile to define the holder. The new holder will be shown in the tool holder display. Use the Edit button to change any of the settings.



Use this option only if you must create a custom holder shape. You can define a holder using a geometry profile, or a solid model of the holder. Using a geometry profile is similar to creating a custom tool shape.

Define Holder

Solid

The Solid option allows you to use an existing solid to designate the tool holder. Select the solid and click Use Selected Solid. Clicking Show Solid will display the custom holder associated with the tool.

Profile

The Profile option allows you to utilize existing geometry to designate the tool holder. Double click the geometry and click Use Selected Profile. Click Recreate Profile to display the profile used to create the holder.

Holder Defined At:

Specify the orientation of the holder, either at the Head Rotary Zero or at the Tool B Rotation.

Dimension As:

Choose either Tool Holder or Boring Bar and specify the Primary Cutting Direction using the arrow keys, either West, South or East. The default cutting direction is westward for boring bars and southward for toolholders.

Lead Angle (L):

This is the insert lead angle.

Depending on the tool, other checkboxes appear:

The Obtuse angle checkbox applies to 80 degree Diamond inserts and indicates an obtuse angle insert which requires a special toolholder.

The Included Angle (A) checkbox applies to Round inserts. The value represents the angle between the two tangents to the ends of the cutting part of the tool.

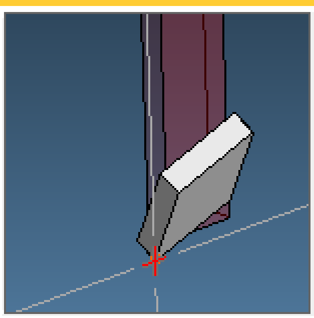
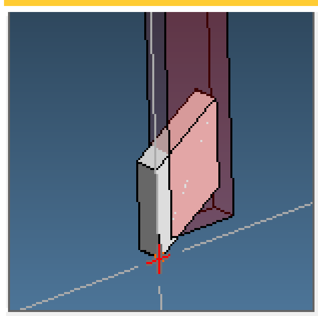
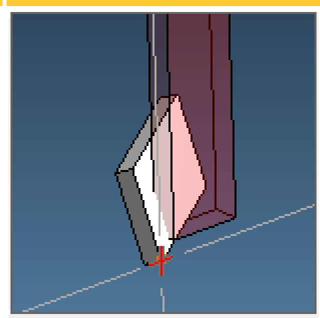
Use Max Plunge Angle checkbox only applies to certain tools. Max plunge angle is usually based on the profile of the side of the holder that is away from the cut.

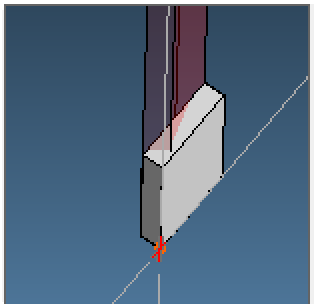
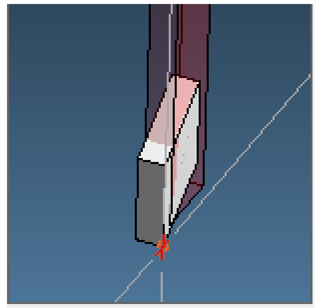
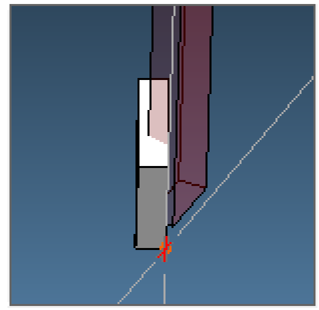
Rake Angle

Rake angle is changed to accurately model the position of the tool in the tool holder to for calculation of collisions.

Changes to the Axial Rake field tilts the tool as it contacts the part.

Changes to the Radial Rake field turns the tool as it contacts the part.

Angle (degrees)	-25	0	25
Axial Rake			

Angle (degrees)	-25	0	25
Radial Rake			

Allow Toolblock

Check this option if this Toolholder uses an adapter block.

Shank Size

The shank size of the Toolholder to fit into the adapter block.

F Dimension

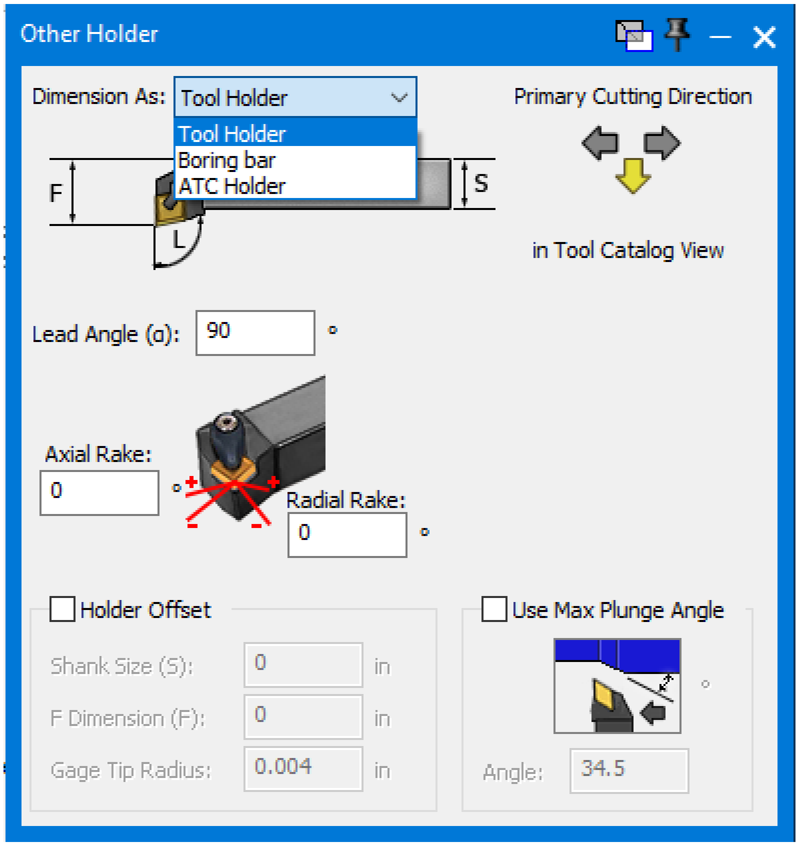
For Toolholders the F dimension is from the tool tip to the back of the holder. With Boring Bars the F dimension is from the tool tip to the center of the Datum or Boring bar.

Gage Tip radius

If an F dimension is used, in order to calculate it correctly, enter the Gage insert tip radius.

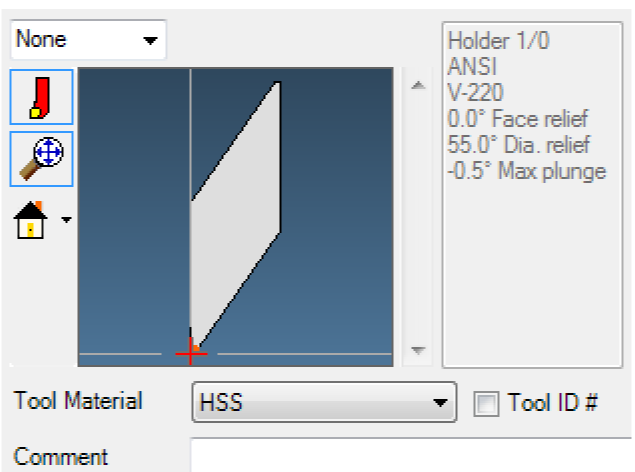
Other

This is the default. Use this option if there are no tool holders or boring bars available in the database for the selected insert and you do not wish to define a custom holder shape. Dimensions of the toolholder can be defined by clicking the Edit button. You must specify if the holder is a Tool Holder or Boring bar, the Primary Cutting Direction and the Lead Angle. Allow Toolblock and Use Max Plunge Angle are optional. In the display window, the insert is shown in the Tool Dialog View Window with an outline of a holder.



None

For certain lathes which have no rotary axes apart from a C axis with no toolgroups attached, the option None is available. In this case Face and Diameter relief fields are provided. If None is selected, no holder will be displayed on the preview pane.



Please note the following with regard to positioning and orientation of Lathe toolholders.

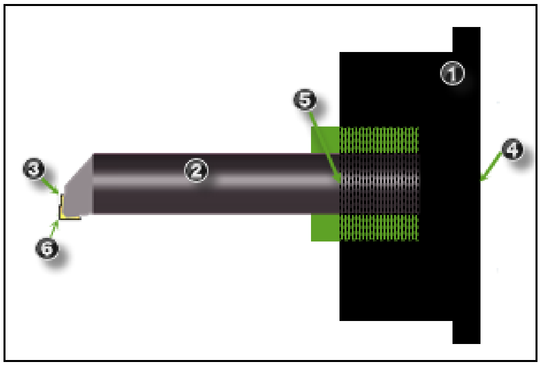
- Orientation

Lathe custom solid tool holders may be positioned at either the cutting orientation specified in the tool dialog, or at machine rotary zero. The orientation of the holder in the first part station's XY CS is preserved when the holder is applied. The only variation is whether the machine is at its zero position, or has the second rotary axis rotated to the tool's setup angle. The insert is oriented in the first part station's ZX CS.

Lathe custom profile holders are positioned much like lathe custom solid holders, but as if they were going to cut in the current CS, rather than in the first part station's ZX CS.

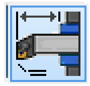
- Positioning

Custom holders are placed relative to the first part station's origin. For lathe tools, this means that the tool touchoff point and holder offsets are calculated from the origin. This is identical to behavior in previous releases.

Positioning of Tool Holders		
	1. Toolblock 2. Tool Holder 3. Tool 4. Toolblock CS 5. Tool attachment CS 6. Toolholder datum	

Tool Holder/Boring Bar

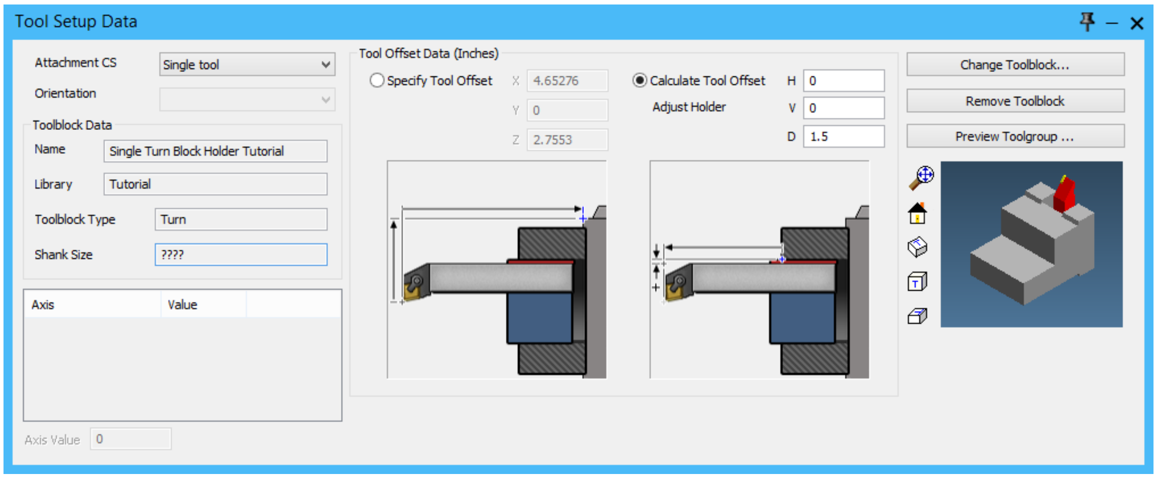
Displays the tool holders and boring bars available for the specified insert type and size, as well as the machine shank size. Use the scroll bar to scroll through the list of available holders. The holder selection is used to determine the diameter relief and face relief angles. The Stickout Length must be entered.



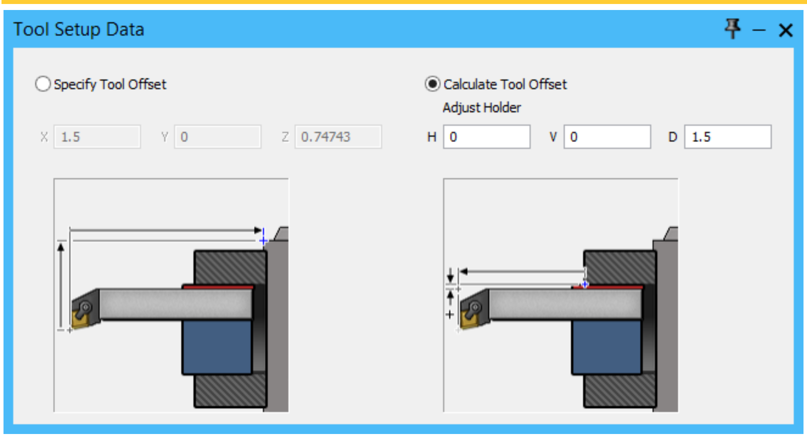
Lathe Tool Offset Data

This button is where you specify Tool Setup data. If Toolblocks have been enabled within Machine Data (located in File>Intermediate Tooling), you can add a Toolblock to the Tool and its Toolholder. The Toolblock and the Toolholder can be fully visualized to double check orientation.

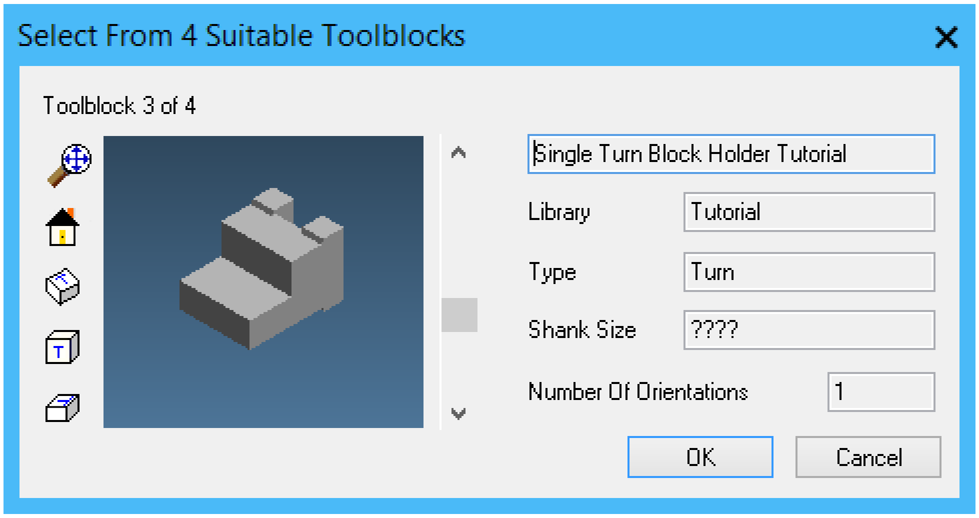
With Toolblocks



Without Toolblocks



Add Toolblock



Clicking this button searches all existing Toolblocks and displays the suitable ones in a dropdown list. Scroll through using the slider bar. When the correct block has been found click OK to accept. Checking Quick View displays a static image of the Toolblock to enable rapid scrolling. When unchecked, the view is fully interactive.

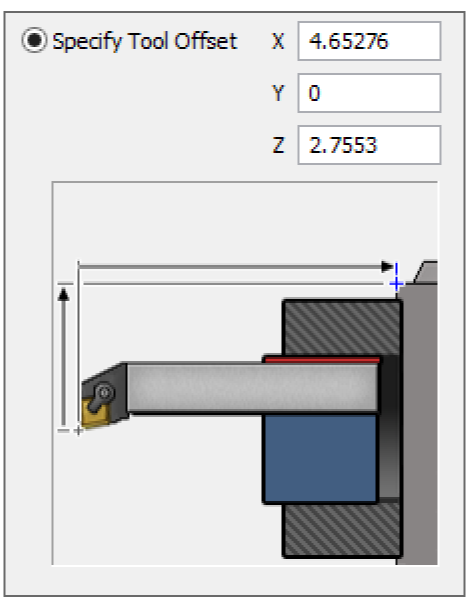
Remove Toolblock

Removes the selected toolblock.

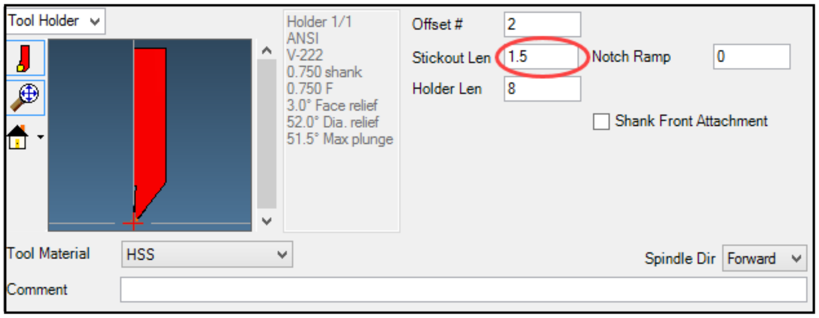
Tool Offset Data

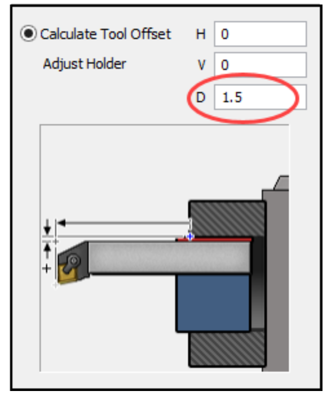
This is used to set distance between the toolgroup (tool attachment position) and the tool tip.

Specify Tool Offset is used to specify the actual distance measured along all 3 axes. Note that specifying values here can affect values in the output code based on tool change positions in the DCD.



Calculate Tool Offset will calculate this distance using the shift applied by a toolblock, plus the shift from the tool holder and the tool shank, plus additional shifts in each axis that you provide here. Note: The shift along the depth axis of the tool is equivalent to the stickout length for a turning tool.



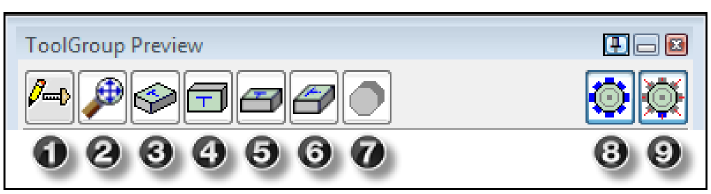


Attachment CS
If different attachment CS's are associated with the Toolblock they are displayed in a dropdown list.

Orientation
If the Toolblock can be mounted in more than one orientation the options are displayed in the dropdown list.

Toolblock Data
Displays the Toolblock data set up in Intermediate Tooling, including Name, Library location (directory name), Type of Toolblock (Turn, Drill, Boring Bar, Cut Off, Right Angle and Live) and Shank Size supported.

Preview ToolGroup
Checking this option opens a new window which displays an interactive view of the Toolgroup.



1. Redraw
2. Unzoom
3. Isometric
4. Top View (Shift-click Bottom View)
5. Front View (Shift-click Back view)
6. Right view (Shift-click Left view)
7. Toggle edges display on/off
8. Toggle Display Current or All Blocks
9. Toggle Display All Tools/Current Tool

Tool Offset

When roughing or contouring, the system calculates a tool offset amount based on the tip radius of the insert. This is the amount the finishing pass of the toolpath (the only pass if contouring) will be offset from the selected part geometry. If a stock amount is entered for the process, that stock amount will be added to the tool radius offset.

Cutter Radius Compensation (CRC)

The Machining Prefs tab of the Preferences dialog contains the Mill CRC Type and Turning CRC Type options to control Cutter Radius Compensation with Contouring and Roughing operations. Tool Center is the recommended option because that is the method used by the system to display the toolpath (orange lines) and cut part rendered images. Regardless of the setting you choose, all toolpath drawing and cut part rendering display as tool center.

To display the Machining Preferences:

1. From the File menu, select Preferences. The Preferences dialog appears.
2. Click the Machining Prefs tab.

Tool Center:
Numbers generated in the posted code are the geometry offset by a tip radius (providing the Stock amount is 0). Tool Center is the recommended selection for this preference. When using Tool Center, the offset in the CRC register at the control should be the difference between the tip radius of the actual tool used and the tip radius of the tool programmed in the system. If the tools are identical, the CRC offset number should be zero. If the actual tool is smaller, you can use a negative value.

Tool Edge:
The offset in the CRC register must be the full tool radius. Toolpath is to the tool edge, including tool geometry. You also need a post processor that supports Tool Edge output. If your post processor is incompatible, a warning message appears. Numbers generated in the posted code are the same as the blueprint numbers. When you select Tool Edge, the toolpath lines still display as tool center. Tool Edge only affects the posted code. Toolpath in Roughing operations is calculated from the tool center, unless in Tool Edge mode, in which case, (because we apply CRC to the last pass only) the last pass will be calculated from the tool edge

When using Tool Edge, you should enter the radius of the actual tool in the CRC register. If you use tapered tools or tools with corner radii, you must calculate the appropriate offset amount based on the taper.

Finish Profile:
The output path is the profile that follows the selected geometry. The CRC register must contain the full tool radius and any desired stock amount.

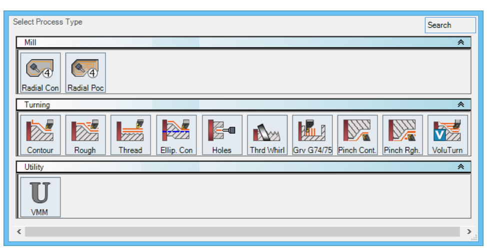
> ⓘ ***WARNING:*** The system does a much better job offsetting the tool than the majority of controls currently available. Regardless of the setting made in this preference, all toolpath drawing and cut part rendering is calculated and displayed using the system's offsetting mechanism. Therefore, it is possible for the cut part rendered image produced by the system to look good while the tool, cutting according to the posted code, will not cut well. If the control's offsetting mechanism is less advanced than the system's, it is possible that when the control produces the offset values, errors and interference will result.

Processes

A process is a combination of a machining process and a tool instance. You apply a process or combination of processes to the part geometry to create an operation.
For more information on processes, see the section on "Processes" in the *Getting Started* guide.

Lathe Machining Palette

Each tile in the Machining palette provides a specific function. The Contouring function takes a single finish pass. The Roughing function takes multiple passes. The Threading function makes various types of threads. The Drilling function drills a hole at X = 0.



Lathe Machining Palette (Level 2)

Note: The processes that appear on the palette depend on which product options are licensed and active. They also vary according to the Machine Definition Document (MDD) associated with the Machine type currently specified in the Document Control dialog.
When the machine is capable of both milling and turning operations, its Machining palette has two dropdown sections. These dropdowns display the Turning and Mill Machining options available for the MDD in use, providing access to both types of machining in one palette.

The order of machining in the finished NC program is the same as in the Operation List. This means that the order of Operation Tiles in the Operation List is very important. Efficient use of multiple process programming may produce operations in a less-than-optimal machining order. The Operation List can be organized as the part is being created or when all operations to cut the part have been completed. Clicking the Sort Ops option on the operation right-click menu reorganizes the operations by tool number and creation order. Operations created in the same Process List will maintain their order to ensure that finishing passes cannot be moved in front of roughing passes, and so forth. The Operation List can also be manually rearranged by moving tiles to different locations in the list.

While the Operation List can be reorganized to create a more optimal machining order, there are some other considerations. When using the Auto Clearance option and/or the Material Only option, the system takes into account the material conditions when it creates the positioning moves and toolpath for each operation. Changing the order of operations has the potential to change the initial material conditions for existing operations. If the order of operations is changed or operations are added or removed from the list, the toolpath and positioning moves should be checked. Rendering the part is a good way to check if changes need to be made to the tool moves due to tool interference or unnecessary incorrect positioning moves. If adjustments need to be made, the operations must be reprocessed. To reprocess all operations in a part file, select Redo All Ops from the operation right-click menu. When the operations are reprocessed, the system recalculates all of the toolpaths and positioning moves based on the new order of operations.

Process Dialogs

Process dialogs appear on the screen when you drag a Function Tile from the Machining palette and a Tool Tile from the Tool List to a Process List tile. The options available with each of these processes are described in this section.

Machines with more than one tool turret also have the option of Pinch Turning processes.

- Contour Process on page 37
- Elliptical Contour Process on page 48
- VoluTurn Process on page 51
- Rough Process on page 57
- Holes Process on page 69
- Thread Process on page 75
- Thread Whirling on page 90
- Groove Cycle on page 92
- / Pinch Contour/Rough on page 96

Also see Process Groups on page 101

Process dialog tabs have several states to help you set operation parameters. The tabs appear as gray, black (normal), or bold, depending on whether they apply to the current process settings. Gray tabs, as usual, are not available to the current process. Tabs that are bold have a direct effect on the toolpath you are going to generate and the items in the tab must be set. Items in the normal (black) text generally do not have any effect on your toolpath.

Clearance Diagrams for Turning Processes

Process dialogs provide different clearance diagrams for different approaches: OD, Front ID, Front Face, and Back Face.

- Radial approaches (OD and Front ID) are along X for most turning machines.
- Axial approaches (Front Face and Back Face) are along Z for most turning machines.

Clearance diagrams are also affected by settings in the DCD. For example:

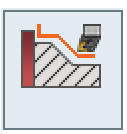
- **Xr** indicates a Radius dimension style; **Xd** indicates a Diameter dimension style.
- Auto Clearance is available in the process diagram only if it is enabled in the DCD.
- A pull-down menu for Part Station (or Spindle) is available only if the DCD has multiple part stations.

Auto Clearance

If the process diagram provides an Auto Clearance checkbox, select it to use the auto clearances specified in the DCD, or deselect it to enter clearances manually.

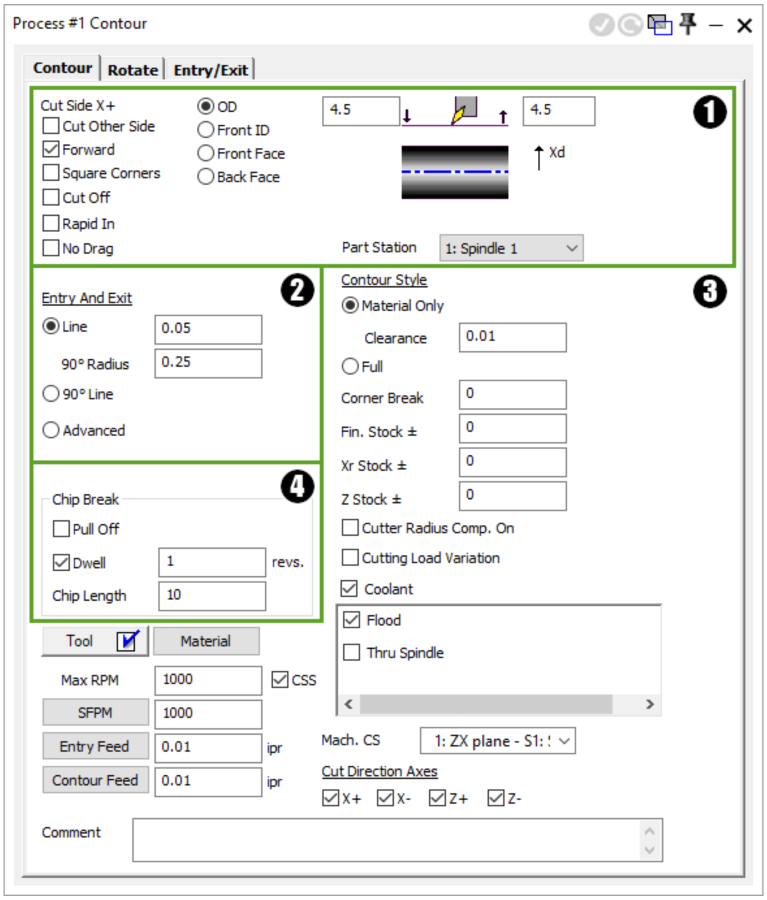
Part Station (Spindle)

If the process diagram provides a pull-down menu for Part Station, choose which spindle you want to program.



Contour Process

The Contour process is used to take a single pass along a shape of circular cross-section. When a Turning Contour process is combined with a tile from the Tool List, the following process dialog appears.



1. Contour Cut Options
2. Contour Entry and Exit
3. Contour Style
4. Chip Break

The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

For information on controls in the **Entry/Exit** tab, see “Contour Entry and Exit ” on page 40.

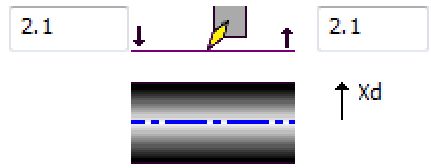
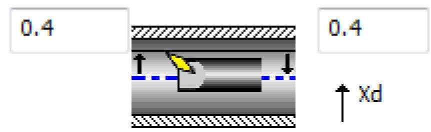
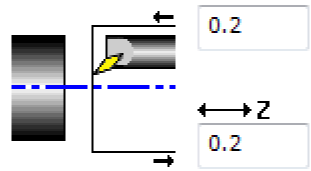
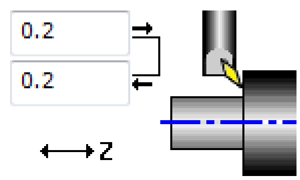
Contour Cut Options

Approach Type

This should be the first selection made in any Process dialog. The Approach Type selection designates the axis along which the tool will approach the part: On most turning machines, radial is X and axial is Z. The OD and Front ID options specify that the tool approach and retract raidally along the X axis, while the Front Face option requires that the tool approach and retract axially along the Z axis. Also, selecting one of these radio buttons will change the Clearance Diagram that appears in the middle of the Process dialog.

Clearance Diagram

This picture will change depending on the Approach Type selection and on the Clearance selection made in the Document Control dialog (DCD). The Approach Type selection will change the axis of approach.

Approach Type	*Clearance Diagram*
OD	
Front ID	
Front Face	
Back Face	

If Auto Clearance is selected in the Document dialog, the diagram will disable the clearance position values because they are calculated based on the Auto Clearance value.

Entry Clearance specifies the diameter or radius location that the tool will make a rapid move to before feeding to the operation start point. The Exit Clearance position specifies the location that the tool may rapid to after completing its toolpath for that operation. Both boxes are labeled with arrows going towards and away from the part, respectively.

Forward
This indicates the direction the tool will move along the designated cut shape. If the Forward option is selected, the tool will move from the start point to the end point of the selected cut shape as designated by the machining markers. Left unchecked, the tool will move from the end point to the start point of the selected cut shape.

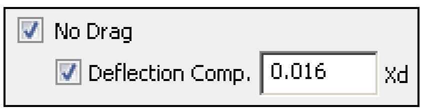
Square Corners
Determines the external corner moves for a cut shape. When this checkbox is selected, the system does not add a radius move at the corners of the cut shape. Instead, the tool makes sharp moves only when going around a corner and will leave contact with the finished shape, possibly creating a burr at the corner. When the checkbox is not selected, the system automatically makes a radius move when rounding a corner, so that the tool always stays in contact with the part.

Cut Off
For use with cut off tools. If the post processor has been appropriately customized, selecting this checkbox will trigger the post processor to output any special codes necessary for removing a part from bar stock.

Rapid In
When this checkbox is selected, the move from the Entry Clearance Plane position to the start point of the toolpath will be a rapid move rather than a feed move. The Rapid In option should be used with caution, as it can create rapid moves directly into the part material. Available only when Use Auto Clearance is not selected. *Note:* To use Rapid In, a post modification is required.

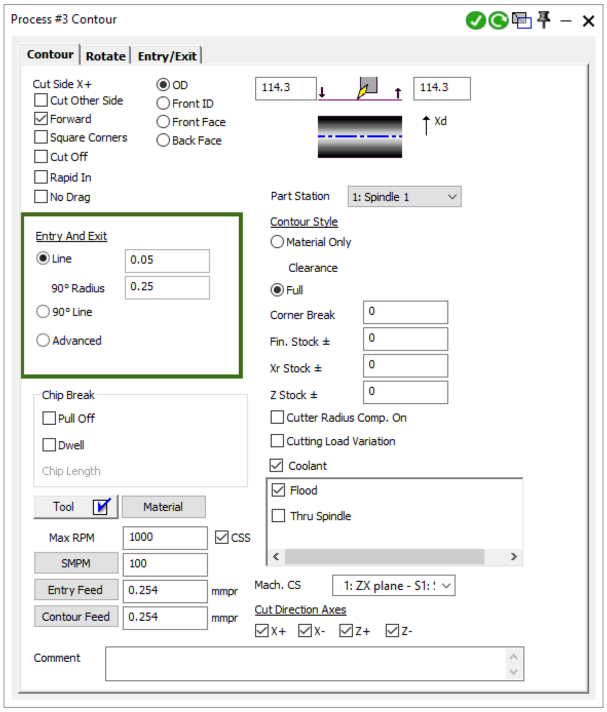
No Drag
Indicates how the contour will be cut. When this checkbox is selected, the chosen cut shape is automatically broken up into segments that will be cut along the positive insert angle direction. All cutting will be "pushing" the insert, not "pulling" it.

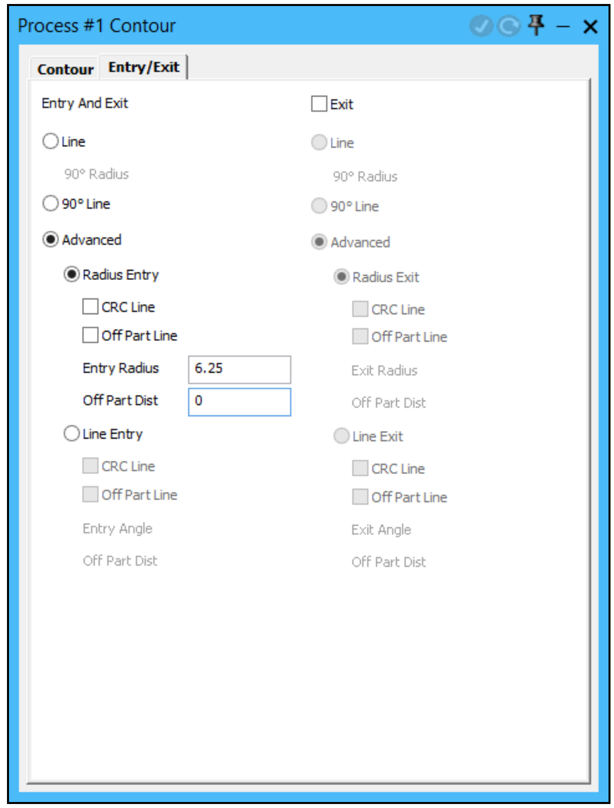
Deflection Compensation (Groove Tool):



By choosing a "No Drag" style toolpath you have the option to specify a deflection compensation amount. The tool motion is modified whenever the deflection compensation is in effect. This will break a contour into (possibly) several toolpaths so that the insert is always cutting in a "forward" direction. This eliminates drag or cutting with the back side of the insert.

Contour Entry and Exit
The Entry and Exit options can create additional movements that will be added to the toolpath. When the first option is selected, a 90° arc of the specified radius value will be added to the toolpath. This arc will be tangent to the start feature at the start point. If a value is entered in the Line text box, a line of the specified length will be created tangent to the arc. Also, if this is selected and the radius value is zero, the line will not be perpendicular but instead will be parallel. When the second option is selected, a line of the specified length will be added to the cut shape. This line will be perpendicular to the start feature at the start point.





Selecting the Advanced option allows you to define custom Entry and Exit Moves using the **Entry/Exit** tab. The options and behavior of these moves are similar to Mill entry/exit except that there are no Z Ramp moves for Lathe. You can define moves for Entry and Exit independently. Checking the Exit option enables the Exit moves, which can be set to the defaults or to advanced options.

Radius Entry/Exit:

Select this option to base your entry/exit move on a radius.

CRC Line:

This generates a line that allows Cutter Radius Compensation to activate. The CRC line can be tangent or perpendicular to the Off Part Line depending on your machining preferences.

Off Part Line:

This is generated after the CRC line, and generates a line that feeds into (or off for exit) the part.

Entry/Exit Radius:

This determines the radius of the entry/exit curve.

Off Part Dist:

This determines how far the system should follow the radius of the entry/exit curve. If this value is equal to the entry/exit radius, a 90-degree curve will result. In the case where the entry or exit angle is either `0` or `180` degrees, the off part distance will be added to the line and no entry/exit move will be created.

Line Entry/Line Exit:
Line entry or exit will be based on a custom angle. The entry/exit axis will be perpendicular to the feature.

Contour Style
The Contour Style selection affects the toolpaths created for the current operation. If the Material Only checkbox is selected, the system takes into account the current stock conditions, including custom stock specifications, when creating the toolpaths for an operation. When Material Only is on, the toolpath will only feed over areas that have not yet been machined in previous operations. The system keeps track of material removed in previous operations and generates the current toolpath based on that information, providing for “no air cutting.”

Because of this, the order of operations directly affects how the part will be cut. If the order of operations is changed or operations are added or removed, all operations should be reprocessed in order to account for the change. The Redo All Ops item in the Edit menu makes reprocessing all operations of a part very easy.

The Clearance value specifies an offset amount from the material that the system uses to calculate where the tool can safely rapid during an operation. If the tool is within the clearance amount, only feed moves will be allowed.

Be careful when using Pinch Turning in conjunction with Material Only. Using Material Only can create strokes that may not sync with the lag applied to the second tool. With some stock conditions, it is possible that the second stroke in a pair of roughing or contouring strokes can start further into the part than the first stroke.

Therefore, with Pinch Turning, always check the rendering. If the second tool has this problem, you will see a gouge.

The Full option gives you more control over toolpath creation. When the Full option is selected, the toolpath generated will feed over the selected cut shape from the start point to the end point as designated by the machining markers.

Corner Break
The value entered in this text box specifies a radius that will be put on every outside sharp corner of the selected cut shape. A value of zero will not break the corner, but will keep the tool in contact with the part as it moves to the next feature. Note that Corner Break is only available when Square Corners is not selected.

Fin. Stock ±
The Fin. Stock value specifies the minimum amount of material that will be left on the cut shape (equally on all faces) after a toolpath is completed.

Xr Stock ±
The Xr Stock ± value lets you specify any additional stock amount for the X axis. The value entered here specifies the amount of material that will be left on the cut shape along the X axis only.

Z Stock ±
The Z Stock ± value lets you specify a stock amount for the Z axis. The Z Stock ± value specifies the amount of material that will be left on the cut shape along the Z axis only.

Cutter Radius Compensation On

A checkbox that indicates whether Cutter Radius Compensation is turned on or off. GibbsCAM has a number of rules for when and where it will generate CRC markers. These rules have been chosen so as to be as safe as possible for the widest range of machines. This means that while a specific machine may be able to handle different CRC rules, we will not generate markers for all cases by default. CRC rules on arcs are the primary example of this.

For new toolpaths, GibbsCAM will do the following:

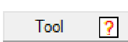
1. CRC will be activated on entry moves, before the entry arc. If there is no move before the entry arc, CRC will be activated on the arc. GibbsCAM has a warning that will tell the end user when they are using CRC without a line move. In general, we consider CRC activation on an arc to be an invalid case, because it does not accurately cut the arc.
2. CRC will be activated on exit moves, after the exit arc. If there is no move after the exit arc, the CRC deactivation will be made on the Depth move. Again, GibbsCAM will warn when a user does not have a line move. In general, we consider CRC deactivation on an arc to be an invalid case, because it does not accurately cut the arc.
3. Some Operations have the option of deferring CRC activation until later in the toolpath (roughing with a finish pass.) Rules 1 and 2 will be applied to the finish pass only.

For old Toolpaths, GibbsCAM will only follow rules 1 and 2. No markers will be added for rapids imbedded in the toolpath.

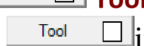
Coolant

A checkbox which indicates whether coolant is turned on in a process. Flood is the standard coolant option. Additional coolant options are available with custom post processors.

Material, Feeds, and Speeds



Tool

- 

indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material

Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

Max RPM

The Max RPM setting is used to set an upper safe limit on the spindle RPM. If CSS is off, the specified RPM value will be used for the spindle speed.

CSS (Constant Surface Speed)

Selecting the CSS item will activate Constant Surface Speed (CSS). CSS will cause the spindle RPM to constantly change based on the diameter the tool is at and the SFPM used.

The SFPM/SMPM and Feed values can be automatically calculated based on the material selected if the CutDATA Materials database is installed. In order for these values to be calculated and entered in the appropriate boxes, the SFPM/SMPM and Feed buttons must be clicked. If no material is selected, or if the CutDATA Materials database is not installed, you might need to manually enter values for the feed and speed.

Entry Feed:

When you click the Entry Feed button, the software will calculate the value based on our materials database. Alternatively, you can manually override the calculated value by inputting your own value. The entry feedrate is written to the toolpath for output in G code.

Contour Feed

This will calculate the best feedrate based on the material type selected using CutDATA.

Cut Direction Axes

The Cut Direction Axes checkboxes allow you to regulate the axes and directions of the cut shape. Deselecting the checkbox for an axis will prevent cut shape moves in that axis direction. The default settings should have all axes selected.

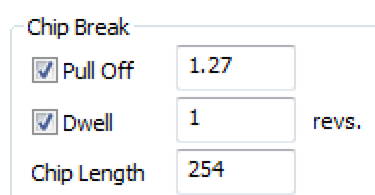
Chip Break

Turning processes **Contour** and **Rough** provide a Chip Break capability, whose controls give you the ability to break off chips according to parameters you set.

What problem does it solve? Especially when machining material that is soft or spongy, chips can sometimes run to great length, interfering with the machining of the part.

> **Please Note:** A post change is required if your existing post does not support the output of Dwell Markers in toolpath. If you are unsure, contact your Reseller or the Gibbs Post Department to verify or request a modification.

The interface offers the following types of settings:



Pull Off

When this is enabled, you can specify how far the tool will retract from the stock.

Dwell

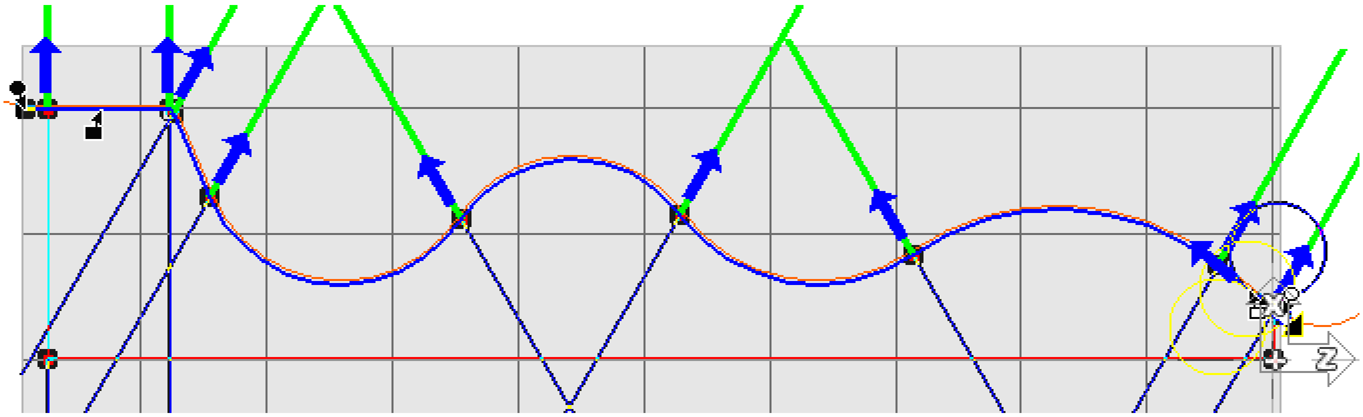
Chip Length

When this is enabled, you can specify how many revolutions the tool wil stay in place before it continues to cut.
Specify the length of chip to tolerate before Pull Off and/or Dwell occur. The length of chips that are removed will remain constant even though the circumference of the stock diminishes (in an OD process).

B-Axis Turning

What is B-Axis Turning?

B-Axis turning machines provide features similar to a 5-Axis mill machine. On a B-Axis capable Lathe, the user may specify vectors on which the tool will rotate, allowing for a part to be cut in a single pass rather than utilizing multiple setups. The two principal benefits of B-Axis turning are fewer tool changes and the ability to cut parts in which there would normally not be enough clearance.



The above image represents one way GibbsCAM depicts a B-Axis path. The green vector lines with blue arrows represent the angle of a tool at a given point. For each vector, depending on the strategy, tool rotation occurs on a preceding feature or features.

B-Axis Tab and Its Controls

In the Lathe Contour process dialog, the B-Axis tab presents the following controls.

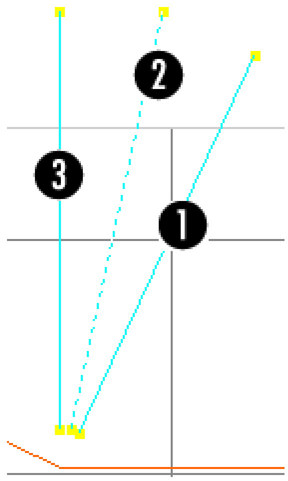
Vary B-Axis

Selecting this checkbox enables B-Axis turning based on one of three choices: Normal to Drive Curve, Guide Curve, or based on Selected Vectors for the start point and end point of the feature.

Normal to Drive Curve

Keeps the tool relative to the normals of the geometry based on the selected drive curve. This strategy must be selected when using the Profiler.

Sharp Corners. The Normal to Drive Curve strategy offers two methods for determining the way the tool transitions at corners.



Smooth Normals

- The tool transitions gradually over the curve, and will not be exactly normal to each feature.

Rotate at Transition

- The tool rotates directly from feature (3) to feature (1). The tool will dwell at the corner while rotating.

Guide Curve

You may also indicate B-Axis rotation by selecting a guide curve. The guide curve must have the same number of geometry elements as the drive curve. Each element in the drive curve will directly correspond to elements at the same position in the guide curve. Your guide curve should be open geometry, otherwise it will attempt to use the whole shape.

Selected Vectors

To select vectors, click the Select button to summon a list of existing transition elements to be used as vectors. For a workspace element not on the list, you can `Ctrl-Click` it to add it to the list. For items on the list, you can `Ctrl-Click` either the workspace element or the list item to remove it. Vectors can be selected in any order.

Candidate elements for Selected Vectors. Vector lines must be one of the following.

- Created through the "line at angle through point" geometry function, where the point selected was on the drive curve.
- Fully terminated with exactly one terminator coincident with a point on the drive curve.

The blue arrow indicates the direction of the vector. Ensure that the arrow points away from the desired approach direction. (If the arrow is pointing in the wrong direction, you will need to recreate that vector.)

Transition Over. The two option buttons in this section apply only to transitions that do not have vectors.

Preceding Feature Only

- The B-Axis will stay in the specified orientation until it comes to the geometry element before the next vector. At this point it will transition over the geometry element only.

Multiple Features

- The B-Axis will transition gradually between vectors.

Minimum Angle / Maximum Angle

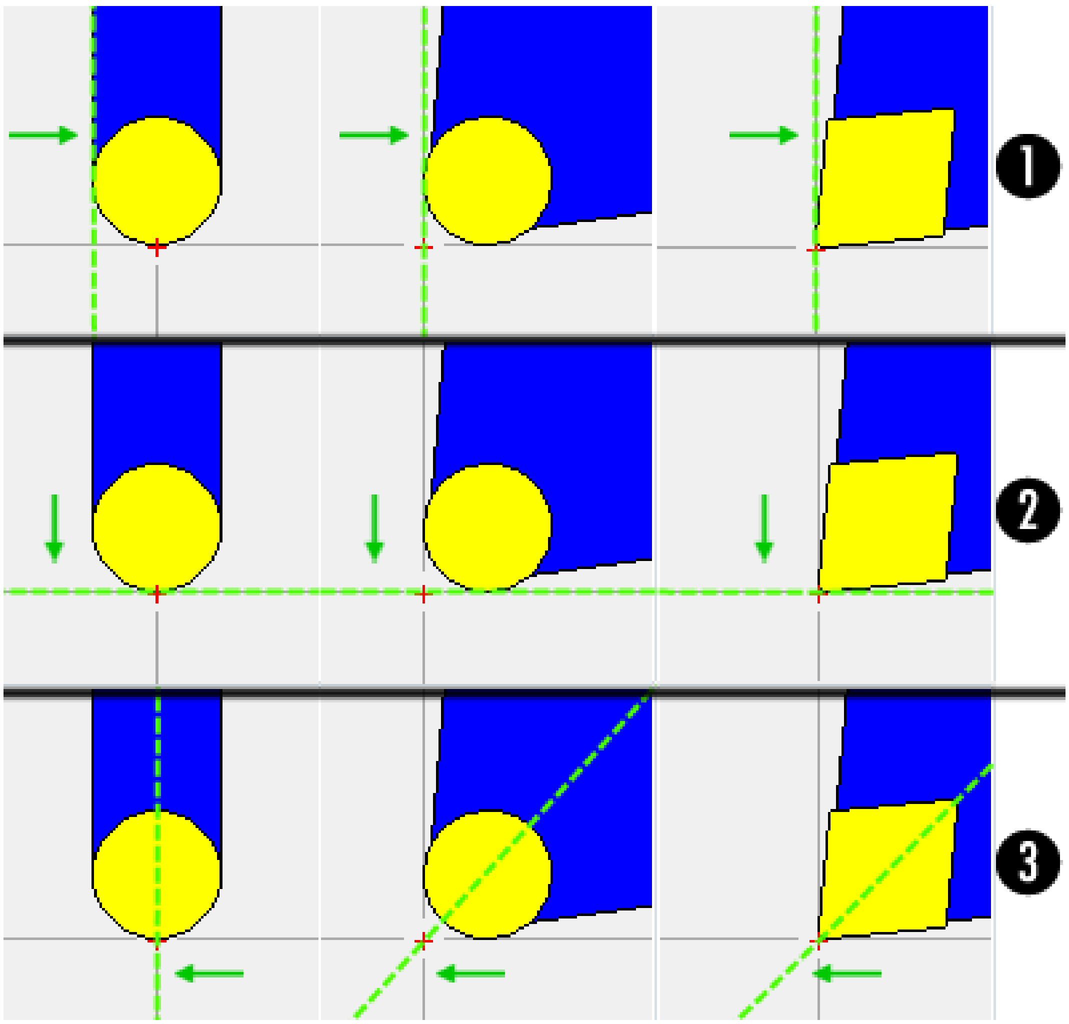
For all strategies, you can specify the minimum and maximum values that the tool axis can assume.

Additional Lead/Lag Angle

For all strategies, you can specify an additional lead (positive) or lag (negative) angle to be added to each vector.

Interpret Vectors As

For all strategies, vectors may be specified as Setup Face, Setup Diameter, or Insert Vector, as illustrated below.



- Setup Face (row 1): The orientation of the tool corresponds to the face of the tool.
- Setup Diameter (row 2): The orientation of the tool corresponds to the diameter of the tool.
- Insert Vector (row 3): The orientation of the tool corresponds to the insert angle.

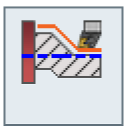
Caveats

Requires `B0` in tool dialog. Tool direction is supported only when the tool dialog for the current tool has a B Rotation value of `0`.

Disables controls in Contour tab of process dialog. Selecting the Vary B-Axis checkbox in the B-Axis tab will limit some of the selectable options in the Contour tab. Please note the following defaults (toggled state in parentheses):

- Cut Off (*off*)
- No Drag (*off*)
- Use Auto Clearance (*off*)
- Contour Style: Full (Full)
- Corner Break: (*suppressed:* `0`)

No automatic collision avoidance. GibbsCAM does not automatically avoid tool and holder collisions. We strongly recommend using FlashCPR or Simulation for collision checking before posting.



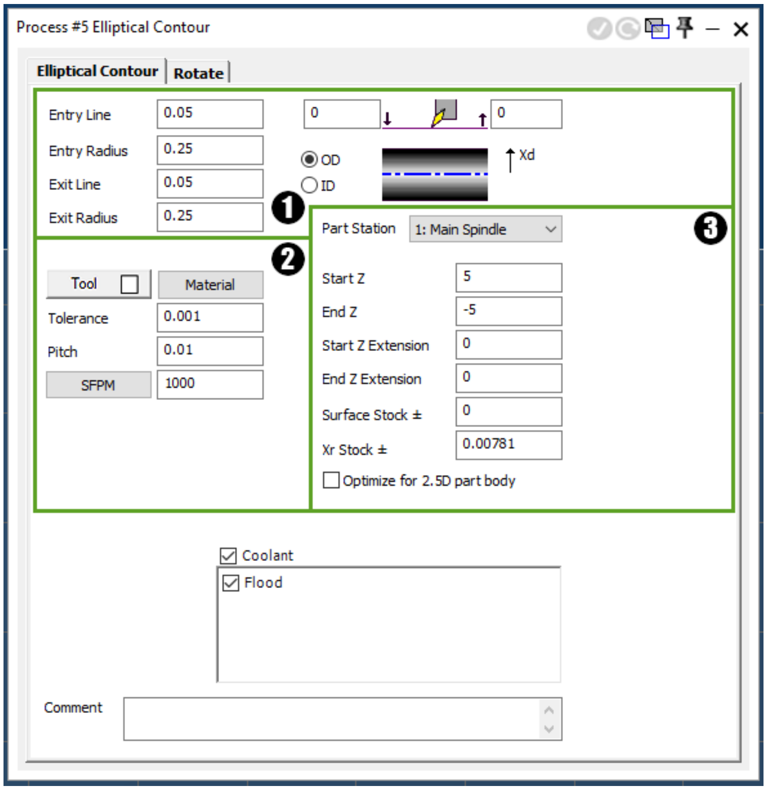
Elliptical Contour Process

The Elliptical Contour process is used to contour a round-but-not-circular shape. When an Elliptical Contour process is combined with a tile from the Tool List, the following process dialog appears.

OD and ID Elliptical turning are available by selecting the radio buttons.

To create OD elliptical toolpath, no reference curve is needed. Only the OD curve is selected.

To create ID elliptical toolpath requires additionally creating and selecting a reference curve or spine curve that goes through the middle of the part. The software needs the reference curve to project the toolpath from.



1. Entry/Exit Parameters and Clearance
2. Material, Feeds, and Speeds
3. Start/End Parameters and Stock Parameters

The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Entry/Exit Parameters and Clearance

Entry Line

Enter a length for the radial line (along X on most turning machines) feeding in to the solid. This is a virtual line that will be expanded into a spiral around the part.

Entry Radius

Enter a value for radius of the entry curve.

Exit Line

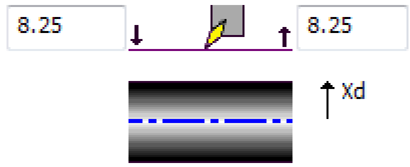
Enter a length for the radial line feeding away from the solid. This is a virtual line that will be expanded into a spiral around the part.

Exit Radius

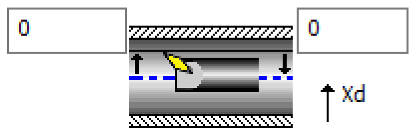
Enter a value for radius of the exit curve.

Clearance Diagram

The picture shows you the orientation (X is radial and Z is axial on most turning machines) and whether the DCD's workspace is measured by radius (Xr) or by diameter (Xd). This diagram is in OD mode as selected by the radio button.



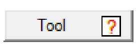
The picture shows you the clearance diagram in the ID mode as selected by the radio button.



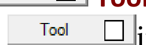
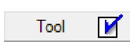
The textboxes just above the clearance diagram let you specify values for:

- Entry clearance — that is, where the tool will rapid to before it begins to feed radially to the operation's start point.
- Exit Clearance — that is, where the tool will rapid to after it has completed cutting at the operation's end point.

Material, Feeds, and Speeds



Tool

-  indicates that the tool instance has no data attached to it.
-  indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material

Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

Tolerance

Enter a value for the maximum distance that the toolpath is allowed to deviate from the selected solid. A small value produces the most accurate toolpath and large amounts of code. A large value takes less time to calculate and produces less code.

Pitch

Enter a value for axial pitch (Z pitch on most turning machines), equivalent to linear feedrate in inches or millimeters per rotation (ipr or mmpr) when cutting a straight diameter section.
A small value produces the most accurate toolpath and longer time to run on the machine.
A large value requires less time to calculate and requires less machining time.

SFPM (SMPM)

Enter a value for surface feedrate. Note that elliptical turning uses variable RPM so as to keep the surface speed constant.

Start/End Parameters and Stock Parameters

Start Z

Enter an axial Z value for where the elliptical turning operation should begin. A start value less than the value specified for End Z reverses the direction of cut.

End Z

Enter an axial Z value for where the elliptical turning operation should end. An end value greater than the value specified for Start Z reverses the direction of cut.

Start Z Extension

Enter an axial Z value specifying how far to extend the first pitch beyond the Start Z point. If solid faces exist under the extension, they will be ignored. This parameter can be used to produce an entry in Z instead of X.

End Z Extension

Enter an axial Z value specifying how far to extend the lastt pitch beyond the End Z point. If solid faces exist under the extension, they will be ignored. This parameter can be used to produce an exit in Z instead of X.

Surface Stock ±

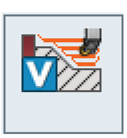
A positive value specifies the amount of material to leave on the surface of the solid; a negative value specifies the amount of overcut; a value of `0` specifies an exact cut.

Xr Stock ±

A positive value specifies the amount of material to leave on the radial (Xr) stock offset, for both ID and OD cutting; a negative value specifies the amount of overcut; a value of `0` specifies an exact cut.

Optimize for 2.5D part body

For parts that are extruded or tapered (that is, where every pitch cuts a similar shape without any twisting or morphing), selecting this checkbox creates a better cut, regardless of tolerance, with less calculation and less code. For any kind of solid other than extruded or tapered geometry, be sure to leave this checkbox unselected.



VoluTurn Process

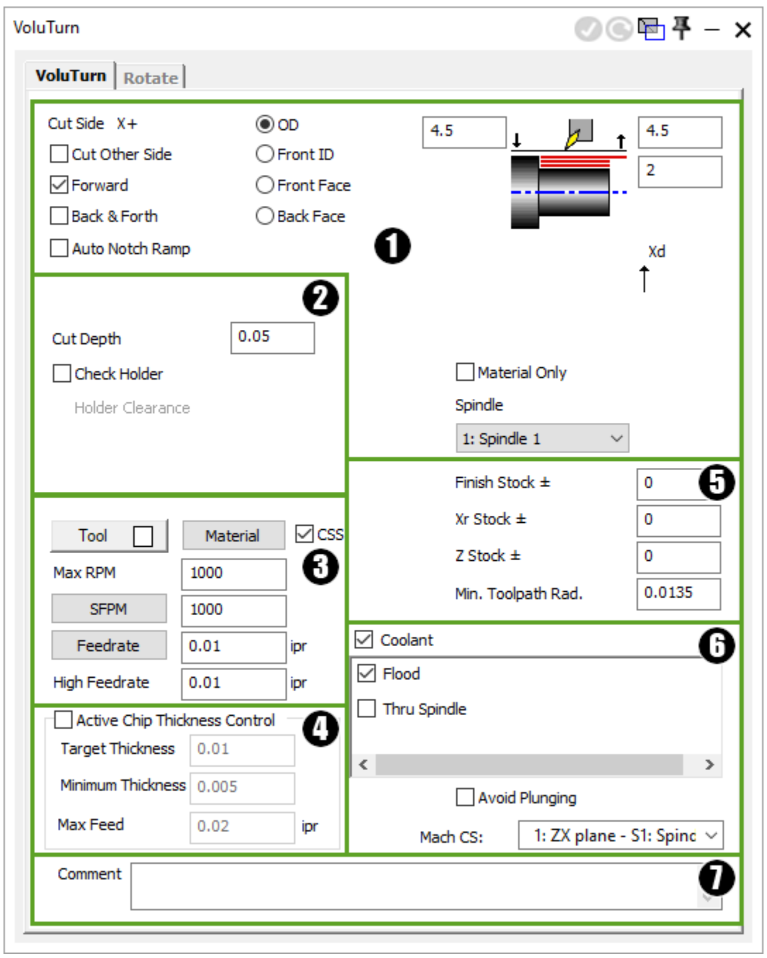
The VoluTurn process provides smooth flowing motion that evenly distributes wear on tool inserts and reduces machining loads by providing smooth circular tangential entry/exit, with efficient repositioning between cuts. VoluTurn is particularly well-suited to machining tough materials like titanium and hardened steels. It uses round inserts such as button inserts.

Save a Copy — Warning

For parts with VoluTurn toolpath at this release:

> ***WARNING:*** Please note that saving a file to an older version may, and in some cases will, cause the part to lose capabilities, functions, tools, and intermediate tooling blocks not available in the older version.

When a VoluTurn process is combined with a tile from the Tool List, the following process dialog appears.



1. VoluTurn Cut Options
2. “VoluTurn Cutting Parameters ” on page 54
3. “VoluTurn Active Chip Thickness Control ” on page 54
4. “VoluTurn Feeds and Speeds ” on page 55
5. “VoluTurn Stock Parameters ” on page 56
6. “VoluTurn Machining Parameters ” on page 56
7. “Comment” on page 57

VoluTurn Stock Parameters

VoluTurn Cut Options

Cut Side
Cut Other Side

Cut Side tells you which side (usually X+ or X-) will be cut. To flip the positioning, select or deselect the Cut Other Side checkbox.

Forward

Indicates the direction the tool will move along the designated cut shape. If the Forward checkbox is selected, then the tool will move from the start point to the end point of the selected cut shape as designated by the machining markers. If it is deselected, then the tool will move from the end point to the start point of the selected cut shape.

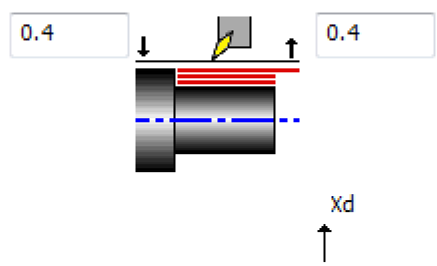
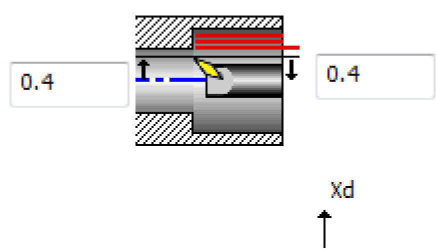
Back & Forth

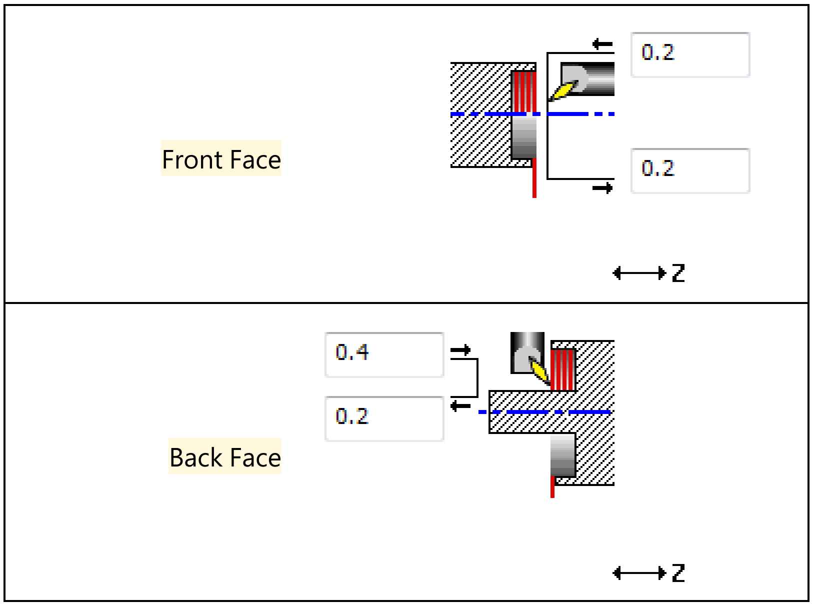
If the Back & Forth checkbox is selected, then the tool will cut in both directions without rapiding to the beginning of the toolpath after each pass.

Auto Notch Ramp

Selecting Auto Notch Ramp overrides the value set in the Tool dialog, and substitutes an automatic calculation to reduce notching to a minimum.

Clearance Diagrams

Approach Type	*Clearance Diagram*
OD	
Front ID	



Use Auto Clearance
If the Auto Clearance checkbox is selected, then the system will calculate the clearance positions automatically. If it is deselected, then it will use the values entered in the Entry and Exit Clearance Positions textboxes.

Material Only
If the Material Only checkbox is selected, then the system keeps track of material that has already been removed. Calculations are performed on the exact shape of material left from the initial stock shape and all prior machining operations. Using Material Only uses more processor power and creates larger part files, but it makes runs more efficient.

Spindle
On machines with multiple spindles (part stations), select the spindle to be used for this operation.

VoluTurn Cutting Parameters

Cut Depth
Enter a value to specify the depth of cut the tool will make on each pass.

Check Holder / Holder Clearance
To enable toolholder collision checking, select the the Check Holder checkbox and specify a clearance value for the holder.

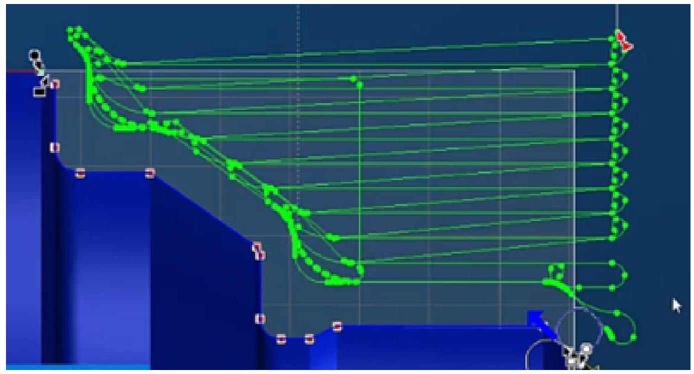
VoluTurn Active Chip Thickness Control

Active Chip Thickness Control
If the Active Chip Thickness Control (ACTC) checkbox is selected, then the system will enable changes to be made in the Target Thickness, Minimum Thickness, and Max Feed text boxes. If it is deselected, then it will ignore the values entered in the textboxes.

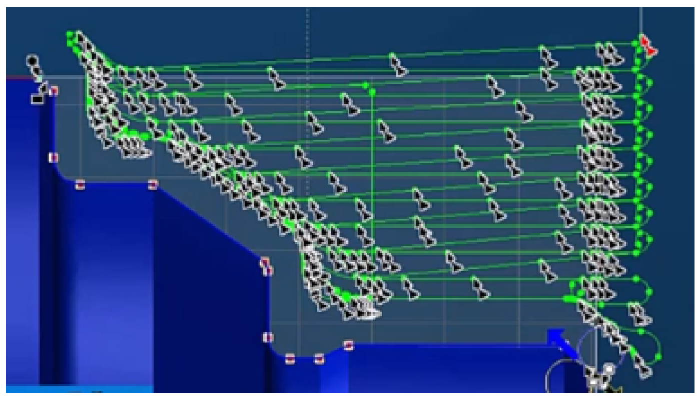
ACTC allows VoluTurn to control the chip thickness during the cut by varying the feedrate as the instantaneous cut depth changes. This option should be used when when the instantaneous cut depth is very small.

The effect on the feedrate can be observed by additional Utility Markers indicating modifications to feedrate.

Voluturn toolpath



Toolpath with ATCTl enabled



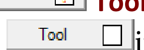
Target Thickness
Specify the target thickness of the chips in the cutting process.

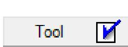
Minimum Thickness
Specify the minimum thickness of the chips in the cutting process.

Max Feed
The value in this box will be used as the suggested max feed in Inches Per Revolution (or Millimeters Per Revolution for metric).

VoluTurn Feeds and Speeds

Tool

- 

indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material

Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

CSS

Selecting the CSS item will activate Constant Surface Speed (CSS). CSS will cause the spindle RPM to constantly change based on the diameter the tool is at and the SFPM / SMPM used.

Max RPM

Specify the maximum revolutions per minute of the spindle.

SFPM (SMPM)

The value in this box will be used as the suggested Surface Feet Per Minute (or Surface Meters Per Minute for metric) when a material is selected.

Feedrate

The value in this box will be used as the suggested Inches Per Revolution (or Millimeters Per Revolution for metric) when a material is selected.

High Feedrate

The suggested feedrate on machines that distinguish high feedrate from regular feedrate.

VoluTurn Stock Parameters

Finish Stock ±

Specify the minimum amount of material that will be left on the cut shape after a toolpath is completed

Xr Stock ± (Xd Stock ±)

Specify any additional stock amount for the X axis. (If your machine's radial approach is not along X, the parameter's name will echo the radial axis.) This is the amount of material that will be left on the cut shape along the radial axis only.

Z Stock ±

Specify any additional stock amount for the Z axis. (If your machine's axial approach is not along Z, the parameter's name will echo the axial axis.) This is the amount of material that will be left on the cut shape along the axial axis only.

VoluTurn Machining Parameters

Minimum Toolpath Radius

VoluTurn cuts use smooth circular motion, reducing jerky motion and wear on tool inserts. Specify a tightest radius you will allow the toolpath to have.

Coolant

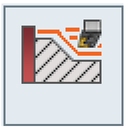
A checkbox indicates whether coolant is turned on in a process. Flood is the standard coolant option. Additional coolant options are available with custom post processors.

Avoid Plunging
You can select this checkbox to specify smooth non-plunge tool entry when the tool first touches the stock.

Machining CS
If your part has multiple CS's select the CS to be used for this operation.

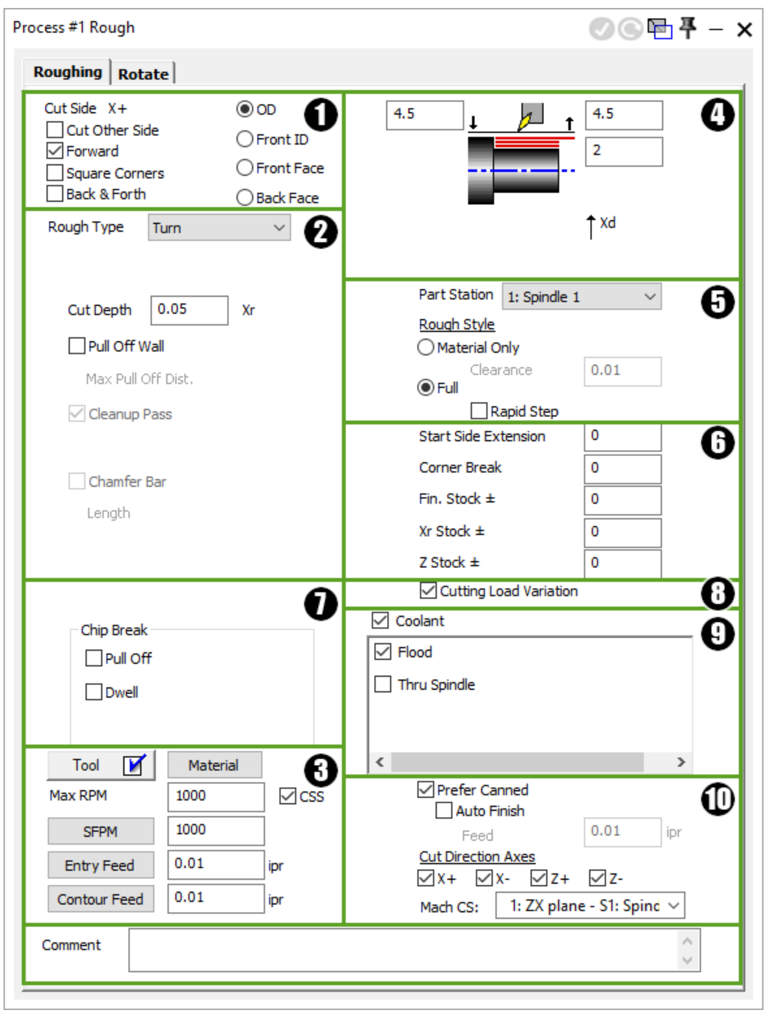
The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Comment
A field where the operator will type process specific information.



Rough Process

Rough processes are used to take multiple passes on a shape. When the Rough function tile is combined with a Tool tile, the following Process dialog will appear.



1. Roughing Cut Options , below
2. “Rough Type” on page 58
3. “Roughing Feeds and Speeds ” on page 67
4. “Clearance Diagram ” on page 64
5. “Rough Style ” on page 65
6. “Stock Options ” on page 66
7. “Chip Break ” on page 67
8. “Cutting Load Variation” on page 66
9. “Coolant ” on page 68
10. “Cut Direction Axes ” on page 68

The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Roughing Cut Options

Approach Type

The Approach Type selection designates the axis (Z or X) along which the tool will approach the part. The OD and Front ID options specify that the tool approach and retract along the X axis, while the Front Face option requires that the tool approach and retract along the Z axis. Also, selecting one of these radio buttons changes the Clearance Diagram that appears in the upper right corner of the process dialog.

Cut Direction

These checkboxes indicate the direction the tool will move along the designated cut shape. If the Forward option is checked, then the tool will move from the start point to the end point of the selected cut shape as designated by the machining markers. Otherwise, the tool will move from the end point to the start point of the selected cut shape. When the Back & Forth option is turned on, the tool will cut in both directions without rapiding to the beginning of the toolpath after each pass.

Square Corners

A radius of 0 (square corner) is supported in Turn Roughing when Rough Type is set to Turn (only).

Start Side Extension

This option allows you to set an extra start distance for each roughing pass. This helps to ensure the tool will have a feed move starting off of the material.

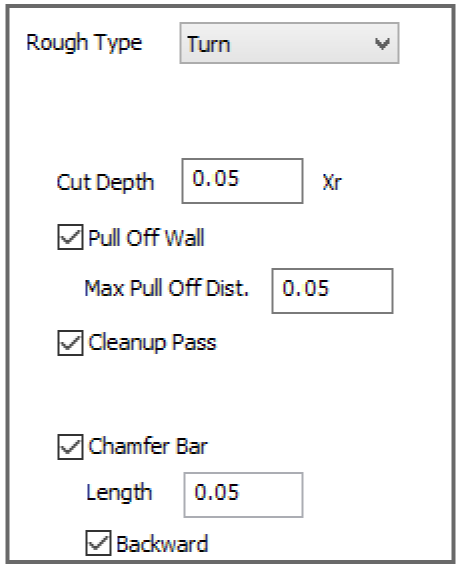
Rough Type

Use the Rough Type pull-down menu to specify the type of roughing cycle to use for the current process:

- Turn, below
- “Plunge” on page 60
- “Pattern Shift” on page 62
- “Offset Contour” on page 63
- “Rib Cut Plunge” on page 63

Choosing a rough type displays the additional information required for each option.

Turn



When the Turn option is selected, a Cut Depthamount must be entered that specifies the depth of cut the tool will make on each roughing pass. Depending on the Approach Type selected, the cut depth will either be an Xr (Xd) value or a Z value. Please note that Notch Ramp (set in the tool dialog) will reduce the depth of cut on one stroke and increase it on the next. Please ensure that the ramp value is smaller than the depth of cut.

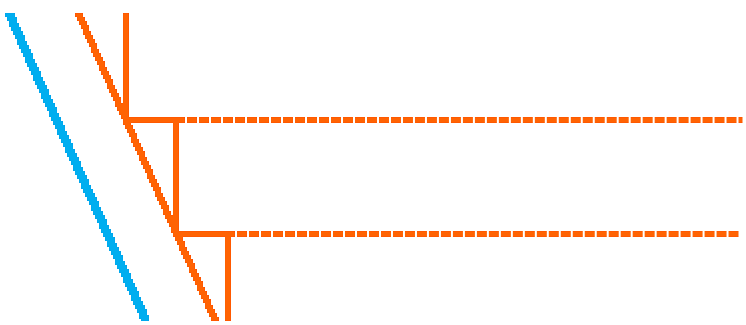
When Turn is selected as **Rough Type**, the Chamfer Bar checkbox appears. It can be selected only when the **Rough Style** selection is Material Only.

Pull Off Wall

Checking this box will cause the tool to pull off the wall instead of machining the wall. This will result in a “stair-step” toolpath as the tool pulls up and retracts. When this checkbox is selected, an additional parameter becomes available: Max Pull Off Distance allows you to specify the maximum distance that the tool will pull off.

Cleanup Pass

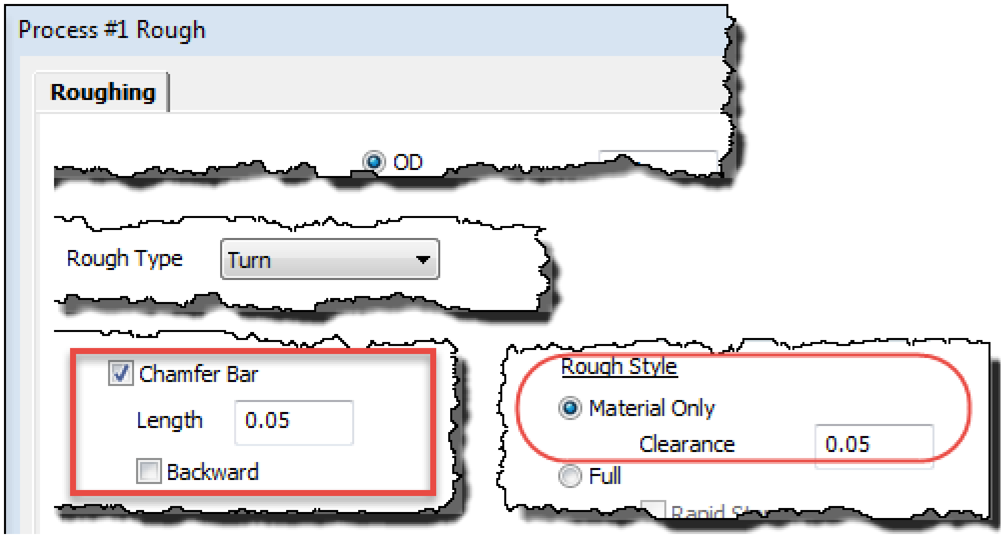
A cleanup pass will go back and remove any material left by Pull Off Wall.



Toolpath generated with Pull Off Wall and a Cleanup Pass.

Chamfer Bar

*What problem does it solve?*When a Swiss-style turning machine runs a roughing cycle, the part retracts into the guide bushing. A common problem is that a pass may leave a burr on the outside of the bar stock that can snag and cause problems for the machine.



To prevent this, it is common practice for the outermost roughing pass to leave a chamfer or fillet on the outside of the bar, removing or weakening the burr.

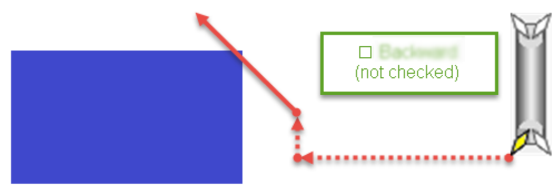
When Rough Type is set to Turn, a new capability is provided when Material Only is selected: Chamfer Bar.

The interface offers the following types of settings:

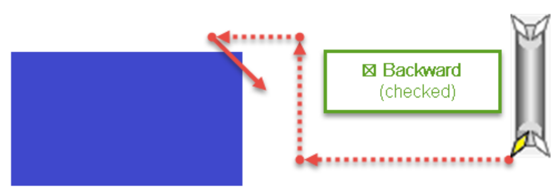
Length
Specify the point-to-point length of the 45-degree chamfer. Any value smaller than the cut depth is valid.

Backward

- If this box is ***not*** checked (the default setting), the tool starts closer to the center of the stock and cuts the chamfer outward.

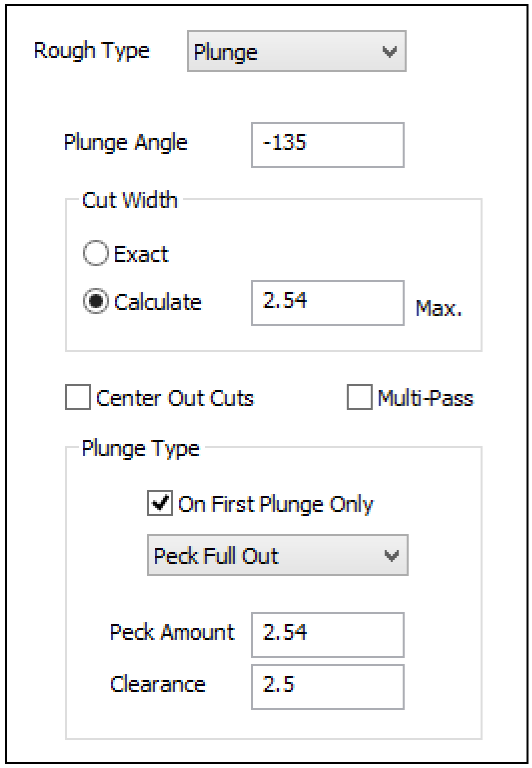


- If this box ***is*** checked, the tool starts closer to the outside of the stock and cuts the chamfer inward.



Plunge

When a Plunge Rough Type is selected, the following options are available.



The Plunge Angle specifies the angle at which the groove tool will plunge into the part. The default value for the Plunge Angle is 270°, which causes the tool to plunge straight down.

There are two options available for the Cut Width. When the Exact option is selected, you enter a distance in Z that the tool will step over on each plunge. The Calculate option will vary the cut width as necessary so that the toolpath hits the endpoints of every feature in the selected cut shape.

When the Center Out Cuts option is selected, the tool will make its first plunge in the center of the groove, and then proceed to rough out each side.

The Multi-Pass option performs "breadth first style plunging." This will take cuts across at the same level X and then drop down by the step amount. The Plunge Type options allow you to select the type of move the tool will make when it first enters the part in a plunge roughing operation. The Plunge Type options include Plunge, Feed, Peck Full Out and Peck Retract and are described in detail below. When moving tool sideways to the wall or down to the floor within clearance tolerances, we will feed rather than rapid. The clearance used is the same for the ISCAR implementation (.02mm or 0.008").

On First Plunge Only

For Peck Full Out and Peck Retract, you can designate these options for the first plunge only.

Peck Full Out

This option designates that the first plunge be a peck. You specify a Peck Amt and a Clearance amount. Because it is a Peck Full Out, after each peck the tool will retract all the way out of the groove to the clearance position. The tool will then reenter the part and begin its peck move a clearance distance away from the remaining material.

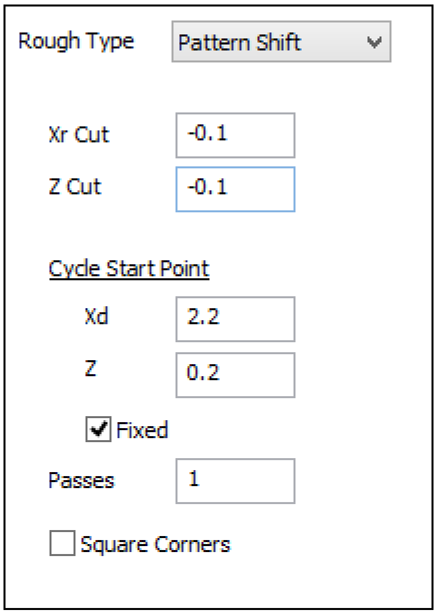
Peck Retract

This option also designates that the first plunge will be a peck. A Peck Amt is again specified. In addition, you provide a Retract amount that specifies how far the tool will come out of the actual cut instead of coming all the way out of the part.

Plunge

This option designates that the first plunge will be a continuous feed move from the clearance position to the bottom of the groove. The First Feed percent value specifies the percentage of the feed rate setting for the Process.

Pattern Shift

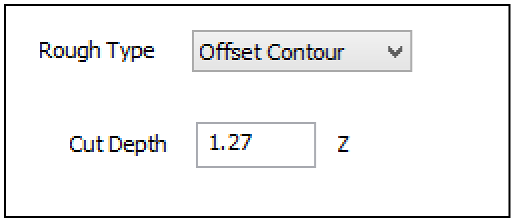


This allows you to input specifications for Pattern Shift roughing cycles. The Xr Cut and Z Cut values specify the amount of material to be removed on each roughing pass. The cut amount in each axis does not need to be the same.

If the Full option is selected for the Rough Style, you must enter a Cycle Start Point and designate the number of passes to be made. The Cycle Start Point specifies the coordinate the tool uses as the beginning point for the Pattern Shift roughing cycle. This point should be clear of the part. The Fixed option, when turned on, designates that the tool will return to the Cycle Start Point after each pass. When this option is not on, the tool will return to the Cycle Start Pointminus the Xr Cut and Z Cut after each pass. The Passes value specifies the number of cuts necessary to remove the desired amount of material in this process.

If theSquare Cornersoption is selected, then the system will not add a radius move at the corners of the cut shape. Instead, the tool will only make sharp moves when going around a corner and will leave contact with the finished shape, possibly creating a burr at the corner. If this option is not selected, the system will always stay in contact with the part when moving around corners.

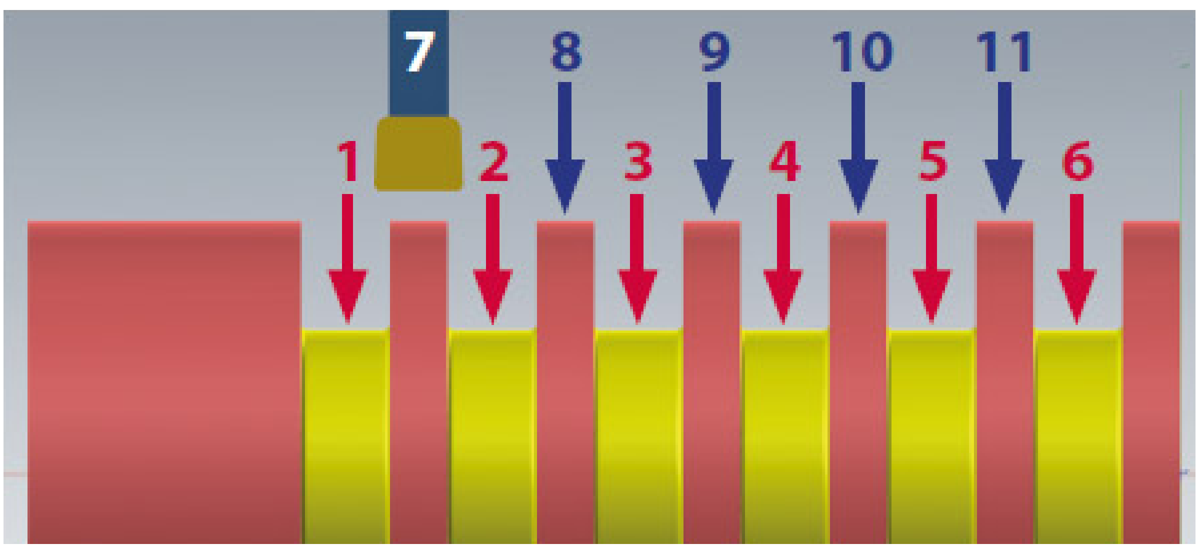
Offset Contour



This type of roughing produces passes that follow the shape of the part profile on every pass, using a decreasing offset for each pass. Offset Contour should be used when it is desirable to follow the shape of the final contour with all roughing passes. This is advantageous in that each pass removes a consistent amount of material for the entire pass, unlike other roughing types that can intersect the part profile at different positions along the pass. This is particularly important for materials that work-harden, because these materials can cause tool wear or breakage if any of the cutting is performed with a thin amount of material being removed.

Rib Cut Plunge

The Rib Cut Plunge strategy consists of a preliminary pass where the tool repeatedly plunges into the part at full engagement to create a series of cuts with each cut more than a tool-width away from the previous one. Then the next pass cuts the remaining “ribs”. Rib cutting avoids tool deflection in both passes, and it offers excellent chip control. The rib cuts can be made safely at high speed to decrease the total amount of cycle time.



Reverse Rib Direction
Checking this box will reverse the direction (left-to-right) in which the ribs are cut.

Max Rib Width
Ticking this checkbox allows you to override the suggested maximum value for rib width.

Rib Cut Feed %
If you want a higher or lower feedrate for the rib cuts, enter a percentage amount here.

Rib Cut Speed %
If you want the rib cuts to occur at a different spindle speed, enter a percentage amount here.

Main Cut Peck
Choose the entry type when the tool first plunges into the part, of Peck Full Out, Peck Retract, and Plunge.

- If you choose Peck Full Out, specify values for Peck Amount(the depth of each peck) and for Clearance(how closely the tool can rapid back in after a peck plunge).
- If you choose Peck Retract, specify values for Peck Amountand for Retract(how far the tool will pull back after a peck plunge).

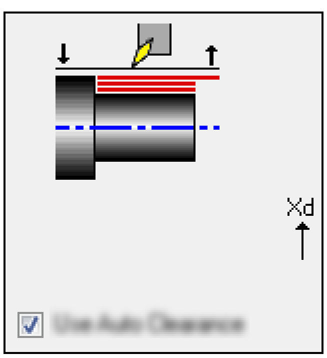
Rib Cut Peck
Choose the entry type when the tool first plunges into a rib, of Peck Full Out, Peck Retract, and Plunge.

- If you choose Peck Full Out, specify values for Peck Amount(the depth of each rib peck) and for Clearance(how closely the tool can rapid back in after a rib peck plunge).
- If you choose Peck Retract, specify values for Peck Amountand for Retract(how far the tool will pull back after a rib peck plunge).

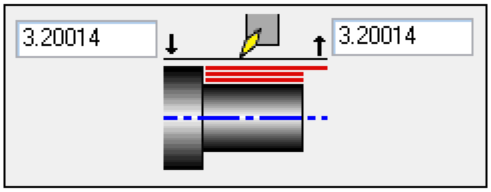
Retract Feed
Ticking this checkbox allows you to enter a feedrate value for main cut pecks. If the checkbox is not ticked, the tool will rapid to approach and retract from main cut pecks.

Clearance Diagram
The picture will change depending on various options such as the Approach Type selected and the Clearance option selected in the Document dialog.

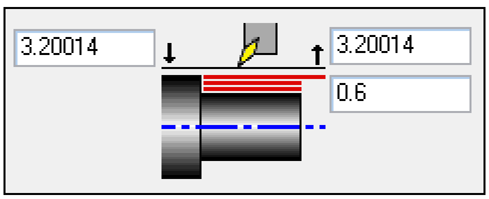
If the Auto Clearance checkbox is selected, then the system will calculate the clearance positions automatically. If it is deselected, then it will use the values entered in the Entry and Exit Clearance Positions textboxe, which vary by approach type. An Auto Clearance value in the Document dialog will disable the Entry and Exit Clearance positions because they are handled universally.



If Material Only is selected as the Rough Style, the diagram will have options for Entryand Exit Clearance Positions as shown in the picture. The Entry Clearance Position specifies the location the tool will make a rapid move to before feeding to the operation start point. The Exit Clearance Position specifies the location the tool may rapid to after completing its toolpath for that operation. Both boxes are labeled with arrows going towards and away from the part, respectively. The use of the values entered for the Entry Clearance Position and Exit Clearance Position changes, depending on the Approach Type selected. Refer to the Clearance Moves section in this chapter for more details.



The X Stock Start Position designates the position the first cut will be calculated from. This position will only need to be specified if the Full option is selected for the Rough Style (instead of Material Only). The move from this position to the first cut will be the amount of the cut depth. It will be a rapid move if the Rapid Step option is turned on under the Full option. Otherwise, it will be a feed move.



The Entry Clearance Position specifies the position the tool will retract to between each pass. The use of this value changes depending on the Approach Type selected. The use of the value entered for the Exit Clearance Position changes depending on the Approach Type selected. Refer to the Clearance Moves section in this chapter for more information.
The X Stock Start Position only needs to be specified when the Full option is selected for the Rough Style. This position will only be used when either Peck Full Outor Peck Retractis chosen for the First Plunge option. When that is the case, the value entered will be used as the point the first peck will be calculated from. The axis will change depending on the Approach Type selected.

Rough Style

The Rough Style selection affects the toolpaths created for the current operation. If the Material Only option is selected, the system takes into account the current stock conditions, including custom stock specifications, when creating the toolpaths for an operation. When Material Only is on, the toolpath will only feed over areas that have not yet been machined in any previous operation. The system keeps track of material removed in previous operations and generates the current toolpath based on that information, providing for “no air cutting.”

Because of this, the order of operations directly affects how the part will be cut. If the order of operations is changed or operations are added or removed, all operations should be reprocessed

Redo All Ops item in the Edit menu makes reprocessing all operations of a part a very easy process.

The Clearance value specifies an offset amount from the part geometry that the system uses to calculate where the tool can safely rapid during an operation. If the tool is within the clearance amount, only feed moves will be allowed. This Clearance amount will be looked at along with the Auto Clearance amount when creating any necessary entry and exit moves.

Be careful when using Pinch Turning in conjunction with Material Only. Using Material Only can create strokes that may not sync with the lag applied to the second tool. With some stock conditions, it is possible that the second stroke in a pair of roughing or contouring strokes can start further into the part than the first stroke.

Therefore, with Pinch Turning, always check the rendering. If the second tool has this problem, you will see a gouge.

The Full option gives you more control over toolpath creation. When the Full option is selected, the toolpath generated will simply feed over the selected cut shape from the start point to the end point as designated by the machining markers. If the Rapid Step option is turned on, the tool will make rapid moves between each pass, otherwise all moves in the toolpath itself will be feed moves.

Stock Options

Corner Break

The value entered in this text box specifies a radius that will be put on every outside sharp corner of the selected cut shape. A value of zero will not break the corner, but will keep the tool in contact with the part as it moves to the next feature. Corner breaks are only calculated with turn and pattern shift roughing cycles.

Fin. Stock ±

The Fin. Stock ± value specifies the minimum amount of material that will be left on the cut shape after a toolpath is completed. The Fin. Stock ± amount affects the cut shape which in turn affects the toolpath created in a canned cycle.

Xr Stock ±

The Xr Stock ± value allows the user to specify an additional stock amount for the X axis. The value entered here specifies the amount of material that will be left on the cut shape along the X axis only. This stock amount is used as a parameter in canned cycles.

Z Stock ±

The Z Stock ± value allows the user to specify a separate stock amount for the Z axis. The Z Stock value specifies the amount of material that will be left on the cut shape along the Z axis only. This stock amount is used as a parameter in canned cycles.

Cutting Load Variation

Cutting Load Variation

The action of the Cutting Load Variation check box depends on your NC control and postprocessor, but it usually involves regular oscillation of either the cutting feedrate or spindle speed to suppress resonance-induced chatter and improve chipbreaking.

- CNC manufacturers that offer options to vary the spindle speed include Haas and Soraluce (Spindle Speed Variation or SSV), Okuma (Harmonic Spindle Speed Control, HSSC, and Variable Spindle Speed Threading, VSST), and DMGMori (Alternating Speed).
- CNC manufacturers that offer options to oscillate the feed axis include Star (High Frequency Turning or HFT), Citizen and Miyano (Low Frequency Vibration or LFV), and Tsugami (Oscillation Cutting).

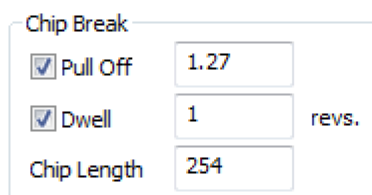
Chip Break

Turning processes **Contour** and **Rough** provide a Chip Break capability, whose controls give you the ability to break off chips according to parameters you set.

What problem does it solve? Especially when machining material that is soft or spongy, chips can sometimes run to great length, interfering with the machining of the part.

Please Note: A post change is required if your existing post does not support the output of Dwell Markers in toolpath. If you are unsure, contact your Reseller or the Gibbs Post Department to verify or request a modification.

The interface offers the following types of settings:



Pull Off

When this is enabled, you can specify how far the tool will retract from the stock.

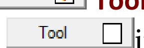
Dwell

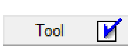
Chip Length

When this is enabled, you can specify how many revolutions the tool wil stay in place before it continues to cut.
Specify the length of chip to tolerate before Pull Off and/or Dwell occur. The length of chips that are removed will remain constant even though the circumference of the stock diminishes (in an OD process).

Roughing Feeds and Speeds

Tool

- 

indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material
Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

CSS (Constant Surface Speed)
Selecting the CSS item will activate Constant Surface Speed (CSS). CSS will cause the spindle RPM to constantly change based on the diameter the tool is at and the SFPM used. The Max RPM setting is used to set an upper safe limit on the spindle RPM. If CSS is off, the specified RPM value will be used for the spindle speed.

The SFPM and Feed values can be automatically calculated based on the material selected if the CutDATA Materials database is installed. In order for these values to be calculated and entered in the appropriate boxes, the SFPM and Feed buttons must be clicked. If no material is selected, or if the CutDATA Material database is not installed, you will need to manually enter values for the feed and speed.

Entry Feed:
When you click the Entry Feed button, the software will calculate the value based on our materials database. Alternatively, you can manually override the calculated value by inputting your own value. The entry feedrate is written to the toolpath for output in G code. This value affects every potential entry move of a toolpath.

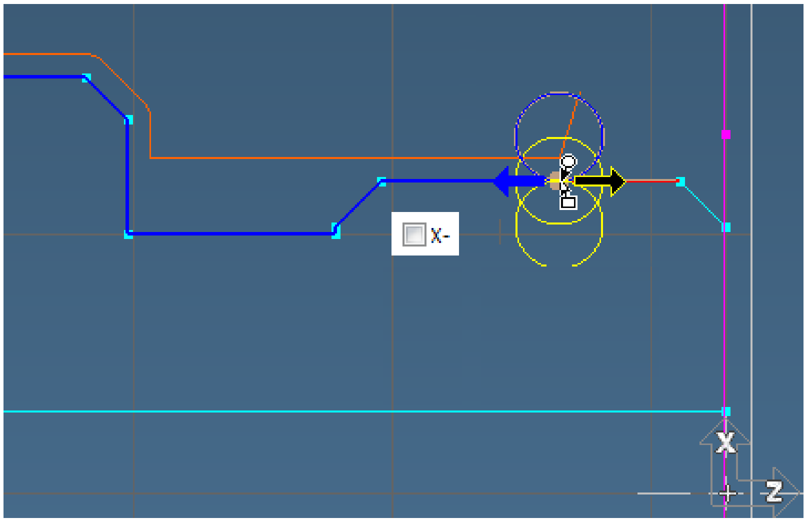
Coolant

Flood
This is the standard coolant option. Additional coolant options are available with custom post processors.

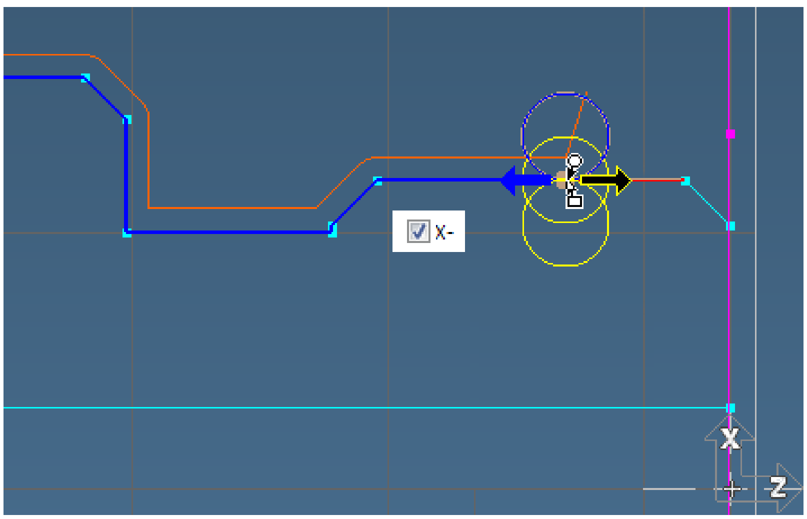
Prefer Canned
A checkbox that will output roughing cycles as canned cycles if the turning machine being programmed is capable of handling canned cycles. If the Auto Finish option is turned on, a canned finishing pass will automatically be added to the post processed code after the roughing canned cycle. The Prefer Canned option is only available when using Fixed Clearance positions (NOT Auto Clearance) and the Full Rough Style (NOT Material Only).

Cut Direction Axes
The Cut Direction Axes checkboxes allow you to limit motion along the cut shape. Deselecting an axis will prevent moves in that axis direction. The default settings should have all axes selected.

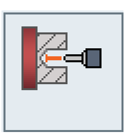
Cut Direction Axis X-



The X- checkbox has **not** been checked. This means the tool will **not** travel in the X- direction.

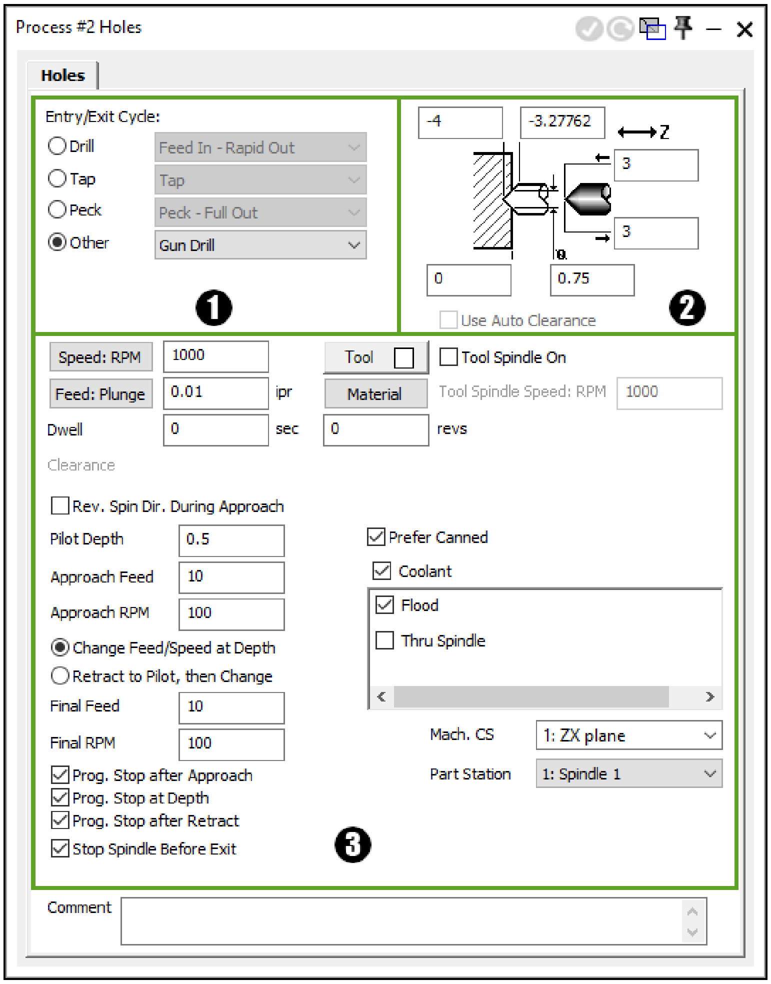


The X- checkbox has been checked. The tool moves in the X- direction.



Holes Process

Holes Processes are used to make holes on center (Xo). When you combine a Drilling Function tile with a Tool tile, the following process dialog will appear.



1. Holes Entry/Exit Cycle
2. Holes Processes are used to make holes on center (X0). When you combine a Drilling Function tile with a Tool tile, the following process dialog will appear.
3. Holes Machining Options

Holes Entry/Exit Cycle

The selections made here determine the cycle that the tool will use to make its hole features. The choices include: Drill, Tap, Peck, and Other.

Drill

Feed In - Rapid Out

Rapid retract to exit clearance plane.

Feed In – Feed Out

Feed back out to Exit Clearance Plane.

Tap

Tap

Tapping for spring-loaded tap holder.

Rigid Tap

Tapping for a rigid solid holder without tension/compression. Spindle rotation and feedrate are synchronized to match a specific thread pitch.

Peck Tap - Full Out

After each peck, reverse spindle direction and retract to the clearance plane.

Peck Tap - Retract

After each peck, reverse spindle direction and retract by the given amount.

Peck:

Peck - Full Out

After each peck, rapid out to the clearance plane, then rapid back into the hole to within the given clearance amount of the previous peck depth before feeding to the next peck depth.

Peck - Chip Break

After each peck, rapid retract by the given amount before feeding to the next peck depth.

Var. Peck - Full Out

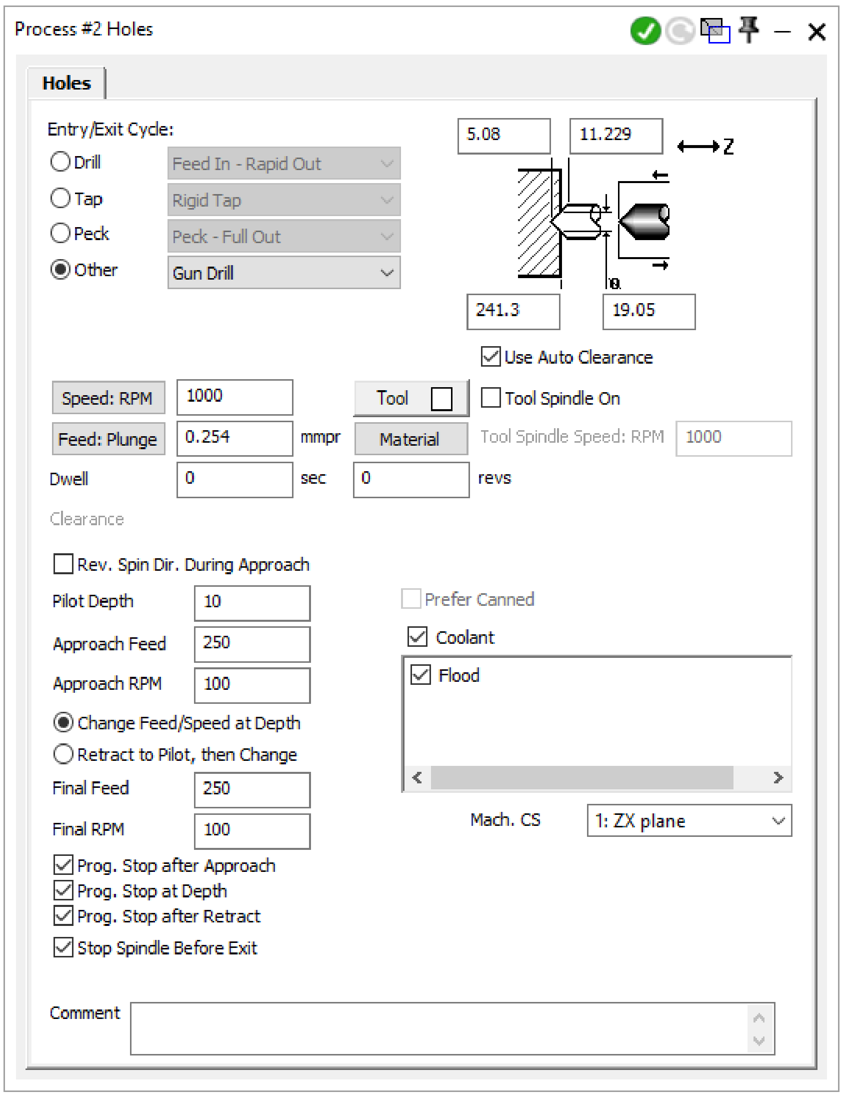
This allows for variable parameters to be input for Peck - Full Out.

Var. Peck - Chip Break

This allows for variable parameters to be input for Peck - Chip Break.

Other:

Holes Process



Gun Drill is a specialized drilling cycle designed for drilling straight and accurate holes with a very high depth-to-diameter ratio (between 10:1 and 100:1 or more). Successful Gun Drilling requires special tooling, high-pressure thru-tool oil-based coolant, several unique process parameters, and a good understanding of the process.

A pilot hole must be drilled first, with a slightly larger diameter than the gun drill's diameter, and to a depth of at least one or two times the diameter.

The Gun Drilling cycle will approach the pilot hole with the drill stopped or spinning slowly at the Approach RPM. If your Approach RPM is too high, you are likely damage the drill. Check Rev. Spin Dir. During Approach to spin the tool backward instead of forward while it is entering the pilot hole; consult your tool manufacturer's recommendations to see if this is necessary. Pilot Depth is the depth that it is safe to move the tool into the pilot hole at the Approach Feed (i.e. not cutting); this should stop slightly short of the bottom of your previously-drilled pilot hole. If you checked Prog. Stop after Approach, the machine will stop here so you can inspect your setup.

Next, the tool will spin up to the specified drilling RPMand Feedrate, and drill the hole. When it reaches the final depth of the hole, it will pause for the specified Dwell amount. If you have checked Prog. Stop at Depth, the machine will also stop here so you can inspect your part.

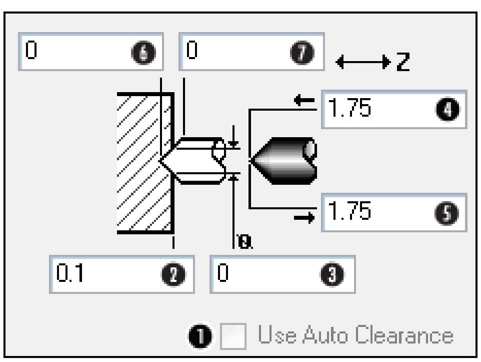
You can feed the drill back out to the pilot hole at either the drilling speed and feed or a different speed and feed. To feed out to the pilot hole at a different speed or feed, select Change Feed/Speed at Depth. The cycle will engage your Final RPM at the hole depth and feed back to the pilot at the Final Feed. If you prefer to feed out at the drilling speed and feed, select Retract to Pilot, then Change. If you checked Prog. Stop after Retract, the machine will stop when the pilot depth is reached again, allowing you to inspect the hole or remove the tool manually. Then, the Final RPM and Final Feedwill be used to exit the pilot hole, unless you have selected Stop Spindle Before Exit, in which case the spindle will be stopped while feeding out of the pilot hole at the Final Feed, until the hole's exit clearance value is reached.

Feedrate and/or RPM can be adjusted at pilot depth and hole depth. Machine stops can be added at each position.

If your site has been set up to use Custom Drill Cycles, then a pull-down menu of further choices appears below the other choices. For more information on Custom Drill Cycles, see the *Installation* guide.

Additional Entry/Exit Cycles are available with custom post processors.

Holes Clearance/Drill Diagram



1. Auto Clearance
2. Drill Surface Z
3. Spot Diameter
4. Entry Clearance Plane
5. Exit Clearance Plane
6. Sharp Tip Z
7. Full Diameter Z

Values for entry and exit clearance positions will need to be entered only if the Auto Clearance option is turned off, in which case these values specify the positions the tool can use when approaching and retracting from the part. The other four values described below are all interactive, automatically calculating the unknown values.

Sharp Tip Z
Specifies the absolute Z depth of the tool tip, and is the number that will be used in the posted output of the finished code.

Drill Surface Z
Specifies the absolute Z value of the surface of the part. This is used in the calculation of Spot Diameter controlled Z depth.

Spot Diameter
Specifies the diameter of the hole at the Surface Z. Controls the hole depth by typing in the desired spot diameter to be left with the tool. This is useful when counter-sinking, for instance.

Full Diameter Z
Specifies the absolute Z depth of the full diameter of the tool.

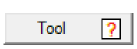
Holes Machining Options

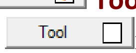
Clearance
This text box is active only if Peck Full Out is the selected Entry/Exit Cycle. The value entered specifies the incremental distance away from the material that the tool will start its next peck from.

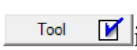
Peck
This text box is active only when either Peck Full Outor Peck Chip Breaker is the selected Entry/Exit Cycle. The value entered here specifies the depth increment the tool will drill on each peck.

Retract
This textbox is active only if Peck Chip Breaker is the selected Entry/Exit Cycle. The value entered here specifies the amount the tool will retract after each peck.

Dwell
The value entered in this text box specifies the length of time that the drill will pause at the hole bottom with the spindle on. The value can either be measured in seconds (entered in the text box labeled sec) or in revolutions per second (entered in the text box labeled revs). Because the two boxes are interactive, a value only needs to be entered in one, and the system will calculate the other.

 Tool

- 

indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material
Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

Speed: RPM

The value entered here is the rate of the spindle measured in revolutions per minute. You can manually enter a value for RPM or else click the button to auto-calculate the value from the Material Database. For information, see the Material Database section in this chapter for details.

Feed: Plunge
The value entered here is the inches per revolution. You can manually enter a value for feedrate or else click the button to auto-calculate the value from the Material Database. For information, see the Material Database section in this chapter for details.

Coolant
A checkbox which indicates whether coolant is turned on in the process. Flood is the standard coolant option. Thru Spindle is used for toolholders that allow coolant to pass through deep holes. Additional coolant selections are available with custom post processors.

Prefer Canned
A checkbox that will output the drilling moves as canned cycles if the lathe being programmed is capable of handling canned cycles.

Tool Spindle On
Tool Spindle On automatically commands the live tooling spindle to be spinning the opposite direction from the lathe spindle.

Tool Spindle Speed RPM
Tool Spindle Speed RPM can be changed by user input. This increases the effective spindle speed and can significantly boost material removal rates, especially when performing simultaneous centerline drilling and OD turning with different toolgroups.

Rev. Spin Dir. During Approach
When selected, approach with spindle direction reversed.

Pilot Depth
Approach distance into pilot hole.

Approach Feed
Feedrate for approach move.

Approach RPM
Spindle speed for approach move.

Reduce Feed/Speed at Depth
Adjust feed / speed after move to depth, then move back to pilot depth\stop spindle and retract out of hole.

Retract to Pilot, then Reduce
Move back to pilot depth, adjust feed/speed and then retract out of hole.

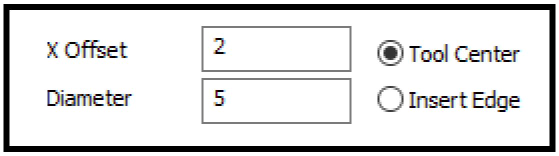
Prog. Stop after Approach
When selected, outputs a program stop after the approach move (pilot depth).

Prog. Stop at Depth
When selected, outputs a program stop after reaching the end depth.

Prog. Stop after Retract
When selected, output a program stop after retracting back to the pilot depth.

Stop Spindle Before Exit
When selected, stops the spindle before exiting the pilot hole.

Multifunction Indexable Drill (MFID)
The following options become available with the MFID tool selected.



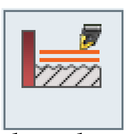
X Offset
Sets location of the center of the drill relative to the center of the part. If Diameter is entered, X Offset will be displayed.

Diameter
Sets the diameter of the hole. If X Offset is entered, Diameter will be displayed.

Tool Center
Sets the programming point at the center of the tool using the Tool Len Offset entered in the tool.

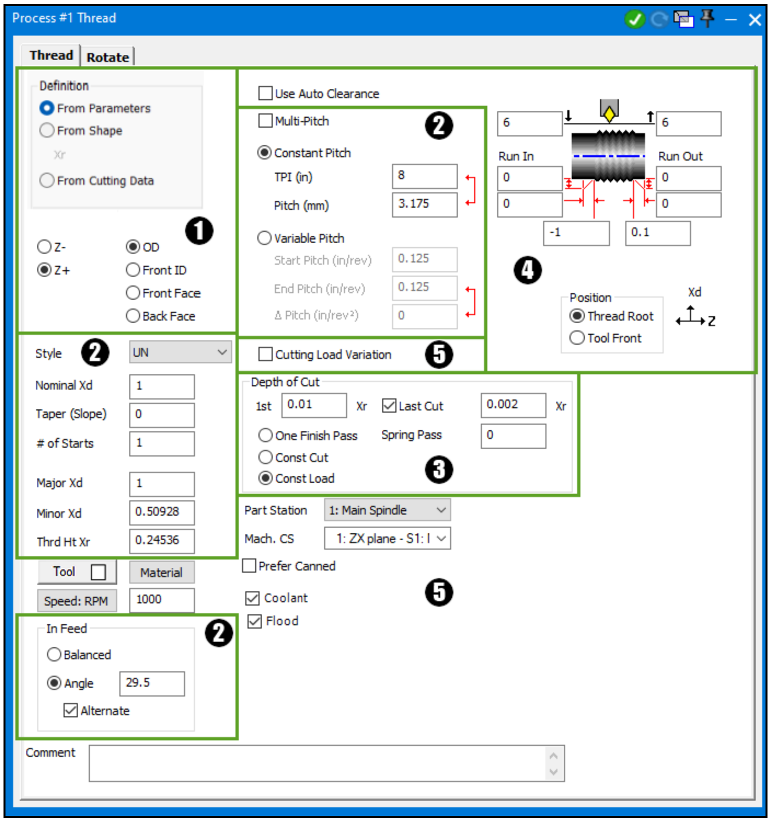
Insert Edge
Sets the programming point on the Periphery Insertusing the Periphery Offset in the tool.

Part Station
Selects the spindle to be used.



Thread Process

Thread processes are used to create ID and OD threads. When the Threading Function tile is combined with a Tool tile, the Process dialog shown below will appear. For more information on thread creation, see Threading.



1. Thread Cut Options
2. Thread Definition
3. Thread Depth of Cut
4. Thread Clearance Diagram
5. Thread Machining Parameters

The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Thread Cut Options

From Parameters / From Shape / From Cutting Data

From Parameters lets you specify values for taper and for Xd (nominal, major, and minor), and it allows you to choose to use canned cycles option if your turning machine supports them.

From Shape is especially useful for bone screws, where you want to create a thread along a general shape that might include several connected lines, arcs, and splines. Xr Shift follows the up/down direction of Xd: for an OD, a negative value shifts inward, and for an ID, a positive value shifts outward.

Cut Direction

The selection made for this option determines the direction the tool will move when creating the thread. If the Z- option is selected, the tool will move towards the spindle. If the Z+ option is selected the tool will move away from the spindle. The Run In and Run Out distances and the actual thread start and end will change positions in the Clearance/Thread Diagram depending on the cut selection.

Approach Type

With OD or Front ID selected in the threading process, the approach is along the X axis. These selections allow the user to determine whether the thread will be located on the OD or the Front ID of the part. Front Face and Back Face allow the user to perform face threading (scroll threading), producing a spiral thread on the face of the part.

Thread Definition

Style

The choices for Style are contained in a pop-up menu and allow the user to specify what type of thread will be cut. The selection made here designates the appropriate thread form for control of calculations.

Nominal Xd

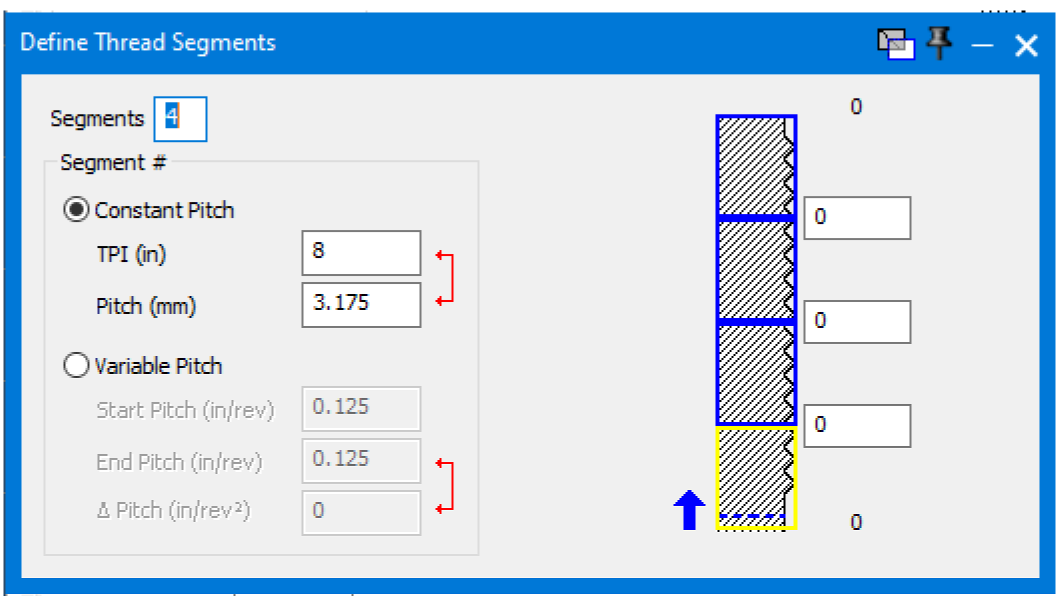
The value entered in this text box is the diameter location of the thread as specified on the part blueprint.

TPI

The value entered in this text box specifies the number of threads per inch.

Multi-Pitch

Check the Multi-Pitch checkbox to activate Multi-Pitch Threading. Click Define Segments to open the Thread Segments dialog where you specify the segments of your thread (in Z for OD/ID threading, or X for Face Threading). Enter the segment count and the boundaries between each segment. Selecting a segment lets you define the pitch for that segment (either constant or variable pitch). Multiple thread features will be output along your thread with the specified pitch changes, to produce a continuous thread with different pitches.

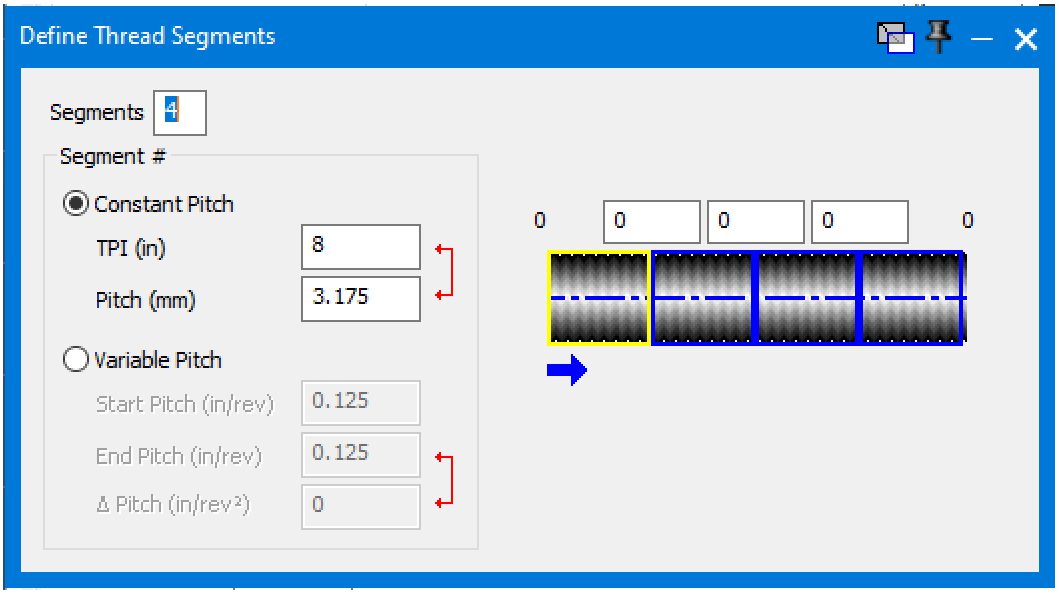


Constant Pitch

TPI and Pitch are interactive inputs. Entering either the TPI or Pitch calculates the value for the other input. For example, entering a TPI value of 1.0 will calculate a Pitch value of 25.4 (and vice versa.)

Variable Pitch

Variable Pitch produces variable pitch threads. Specify a Start Pitch and End Pitch to blend between them, or a Start Pitch and Δ Pitch to increment the pitch by the specified amount every revolution. Variable Pitch Threading typically requires a post modification, as the appropriate cycle must be activated (G34 or similar).



Taper

Taper is a "slope" value, not an angle. A slope is a ratio of vertical/horizontal distances. The equivalent angle is:

angle = tan (vertical/horizontal) or tan (slope)

The NPT specification defines the taper as 1/16, or 1" vertical for 16" horizontal, with the horizontal measured on the diameter. This entry requires a radial slope, or 1/32. You may type in `1/32` or you may type in `.03125`, the decimal equivalent. If your taper is defined as a radial angle, the slope = arctan (angle).

of Starts

The value entered here is the number of starts for the thread. Most standard threads have one start. If a value greater than one is entered here, the process will create a multiple thread start.

Major Xd

The value in this text box automatically defaults to the value entered for the Nominal Xd; however, it can be changed. Cutting begins at this diameter on an OD thread.

Minor Xd

The value in this text box defaults to a calculated value based on the Nominal Xd and the desired pitch. Cutting begins at this diameter on an ID thread.

Thrd Ht Xr

This value is calculated by taking the difference between the Major and Minor diameters and dividing it by two. It represents the Thread Height given as a radius value.

In Feed

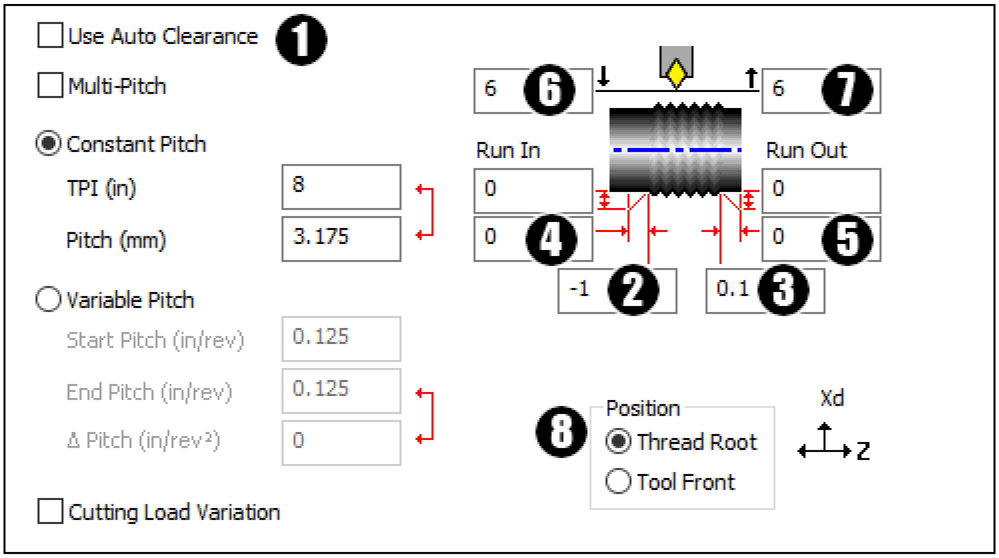
This section allows the user to control how a threading insert will cut. The Balanced option will cut with both sides of the insert equally. For UN thread forms, a Balanced or 0° In Feed takes all cuts at the same Z position. The Thrd Angle selection allows the user to specify the In Feed angle. The value entered is measured in degrees and specifies the single edge In Feed angle for the thread form. The value 29.5° is the default Thrd Angle for all thread types. Each cut starts at a different Z position, always cutting with one edge. The Alternate option is available when Thrd Angle is the In Feed selection. When turned "on," each cut taken at the specified angle will alternate (e.g. 29.5°, -29.5°, 29.5°) Only one edge is used at a time to cut, but it alternates to provide for maximum insert life. This is also known as "using the leading edge & trailing edge alternately".

Thread Depth of Cut

The selections made in this box allow the user to designate the cut depth for each pass of the threading operation. The One Finish Pass option specifies that the tool will make a single pass over the thread. Its primary use is to remove burrs or small excesses of material on an existing thread. Selecting One Finish Pass will grey out the Last Cut dialog, as there is only one pass. The Const Cut selection allows the user to designate theDepth Of Cut that the threading tool will make on each pass. The value is measured as a radius and is entered in the text box labeled 1st. The Const Load selection allows the user to specify the depth of the cut made on the first pass. This value is also measured as a radius and entered in the text box labeled 1st. The amount of material (the load) removed for that depth of cut will be calculated, and on each successive pass the depth of cut will decrease while the tool pressure remains constant.

The Last Cut option is selected to prevent any cut from removing less than a given amount of material on the last pass. The value entered is measured as a radius value and specifies the minimum cut for the constant load to diminish to. The Spring Pass option can be used in conjunction with any of the depth of cut selections. It will create additional passes equal to the number entered after the thread has been cut.

Thread Clearance Diagram



1. Auto Clearance
2. Thread End
3. Thread Start
4. Z Run Out
5. Z Run In
6. Entry Clearance
7. Exit Clearance

If the Auto Clearance checkbox is selected, then no Entry or Exit Clearance Positions need to be entered. If Auto Clearance is off, then Entry and Exit Clearance Positions must be entered to specify where the tool will move to when approaching and retracting from the part.

Run In values are used if the threading tool needs to begin a certain distance away from the actual thread start in order to accelerate to the proper feedrate. The Z Run In distance allows the user to designate a distance along the Z axis to begin the threading pass. The X Run In distance can be used in conjunction with the Z Run In distance to start the thread at an angle. The Run Out values allow you to designate a distance and angle for the threading tool to come off the thread and function the same as the Run In values.

The Run In and Run Out labels and values will change positions in the diagram depending on whether the tool is cutting towards the spindle or away from the spindles, which is determined by the selection made for cut direction (Z+ or Z-).

The Actual Thread Start and Actual Thread End values specify where along the Z axis the thread will begin and end. Any Run In or Run Out values will be added on the actual length of the thread.

Position: Tool Front and Thread Root

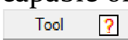
Position allows the selection of either Thread Root or Tool Front to define the values in the thread diagram. Thread Root means that the numbers define the thread itself; the Start Z value, for example, is where the thread starts on the part. Tool Front means that the numbers define the position of the front of the tool, so Start Z will be where the tool is when the cutting starts.

This setting can be useful cutting threads up to a square shoulder, when the precise thread length is less important than the shoulder location. With a Laydown-style (LT) thread insert, selecting Position Tool Front and also selecting the alternate touchoff point at the front of the tool in the tool dialog will typically result in the exact start/end numbers that you type in this dialog being output in the G-code.

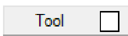
Thread Machining Parameters

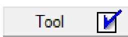
Coolant
This checkbox indicates whether coolant is on. Flood is the standard coolant option. Additional coolant options are available with custom post processors.

Prefer Canned
A checkbox that will output threading passes as canned cycles if the lathe being programmed is capable of handling canned cycles.



Tool

- 

indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Material
Clicking this button opens the **Materials** dialog, where you can select and edit materials. For a full description of the Material database, see the *Common Reference* guide.

RPM
Speed:
The value entered here is the rate of the spindle measured in revolutions per minute. You can manually enter a value for RPM, or you can click the button to auto-calculate the RPM based on the Material database.

Cutting Load Variation
The action of the Cutting Load Variation check box depends on your NC control and postprocessor, but it usually involves regular oscillation of either the cutting feedrate or spindle speed to suppress resonance-induced chatter and improve chipbreaking.

- CNC manufacturers that offer options to vary the spindle speed include Haas and Soraluce (Spindle Speed Variation or SSV), Okuma (Harmonic Spindle Speed Control, HSSC, and Variable Spindle Speed Threading, VSST), and DMGMori (Alternating Speed).
- CNC manufacturers that offer options to oscillate the feed axis include Star (High Frequency Turning or HFT), Citizen and Miyano (Low Frequency Vibration or LFV), and Tsugami (Oscillation Cutting).

Threading

This section is intended to assist in calculating the correct parameters for cutting both straight threads and standard NPT pipe threads using the system. First, an overview of general thread cutting using the system will be outlined. There are three things the user must define in order to properly cut a thread using the system: what kind of thread to cut, how to cut the thread, and where to cut the thread.

Thread Dimensions - Defining the kind of thread to cut

Style
This pop-up menu is used to select the thread style, such as UNF, NPT, etc.

Nominal Xd
This is the nominal thread diameter.

TPI
This is the number of threads per inch, (per millimeter for metric parts).

Taper (Slope)
This is the decimal slope of the thread taper, measured radially. For straight threads, this value should be zero. For standard NPT pipe threads, this value should be 1/32 or 0.03125 (the slope of NPT threads is 1/16 of an inch per inch on diameter, which is 1/32 of an inch per inch radially). If you are creating a tapered thread with Run In, Canned Cycles should not be used. This is because most machines cannot handle this situation.

of starts
This is the number of thread starts. For multiple start threads, enter the number of starts here. Otherwise, this value should be one.

Major Xd & Minor Xd
These values will default to the theoretical major and minor diameters based on a perfect sharp thread. The value as calculated is primarily for reference; this value can be changed as required for the particular thread class and fit desired. For OD threads, the minor diameter is critical as this will be the diameter that the tool will cut on the finish pass. On ID threads, the opposite is true. The major diameter is critical as this will be the diameter that the tool will cut on the finish pass of an ID thread.

Thrd Ht Xr (Thread Height Xr)
This value is the actual thread height as a radius dimension. This value is calculated as the radial difference between the Major Xd and the Minor Xd and can be changed as required.

Cut Information - Defining how to cut the thread

Cuts (Z-, Z+)
This is used to specify the direction of the thread cut; Z- will cut toward the spindle and Z+ will cut away from the spindle. The Z- choice is the default as most threads will be cut toward the spindle; only in rare cases is the Z+ option used.

OD, Front ID (Approach Type)
This is used to specify whether the user is cutting an external or internal thread; the type of thread will affect the approach moves to the thread cutting cycle. It is also correct to think of this as the Thread Type.

In Feed - Balanced
This choice will feed the thread tool straight in for each pass resulting in both edges of the thread tool cutting equally.

> The Balanced In feed is often used when cutting tough stainless steels that are easily work hardened, as the equal metal removal method helps prevent work hardening during the cutting cycle. This method usually does not work well on softer materials that tend to load up on the insert; for these materials it is usually best to use the Thread Angle In feed.

In Feed - Thrd Angle (Thread Angle)

This choice will cause the positioning move at the start of each pass to feed the thread tool in at the angle specified, resulting in the leading edge of the tool doing most or all of the cutting. It is common to set the in feed angle slightly steeper than the thread angle so that the trailing edge of the tool takes a 'light' cut to ensure that the back side of the thread cleans up.

> This option is often used to improve the chip flow on soft or gummy materials that tend to tear during the cutting cycle because of material load up on the tool.

Alternate

This option is only available when the Thrd Angle is selected for the In feed. It will alternate the in feed, resulting in the tool first cutting with the leading edge, then alternating to the trailing edge, and then back to the leading edge, etc. This provides even tool wear, in turn providing maximum tool life.

Depth Of Cut

The values and options in this section of the Thread dialog are used to control the number of cuts as well as depths of cuts, minimum cut depth, and spring passes.

The following controls are available when the option chosen for Definition is From Parameters or From Shape.

1st Xr

This value is the stock amount to remove on the first rough pass. This value also controls the entire roughing cycle as described below for Constant Cut and Constant Load.

One Finish Pass

This option specifies that the tool only take one cut at the finish thread depth. This would normally be used to re-cut a thread as part of a de-burring process.

Const Cut (Constant Cut)

The Const Cut option will cause the roughing cycle to step in the amount specified in 1st Xr on each subsequent pass until the tool reaches the Last Cut amount. A larger 1st Xr will result in fewer passes, while a smaller 1stþXr will result in more passes.

Const Load (Constant Load)

The Const Load option is the most commonly used type of thread roughing cycle. This cycle will take a constant volume of material on each pass, resulting in a smaller depth of cut on each subsequent pass until the tool reaches the Last Cut amount. The volume removed on each pass is calculated based on the depth of cut specified in the 1st Xr field. This can also be considered a constant amount of tool pressure.

Last Cut

When selected, this option will prevent the roughing cycle from taking any rough passes at less than the value specified. In addition, the rough cycle will always leave exactly this amount for the last pass.

Spring Pass

This value is used to specify whether to take one or more spring passes at the finish depth.

Thread Location - Defining where to cut the thread

Thread Start Z

This value is used to specify where the actual thread begins in Z. Note that this is not the Z start of the thread cycle.

Thread End Z

This value is used to specify where the thread ends in Z.

Run In / Run Out. A run-in acceleration distance is typically used to allow the spindle to come up to speed before threading occurs.

Z Run In

Specifies the acceleration distance in Z, incrementally. For example, if the thread cycle is to start 300/1000" before the actual thread start, simply enter `0.3` for Z Run In.

X Run In

Specifies the acceleration distance inX, incrementally, if necessary. The value is normally zero, and almost never larger than the value for Z Run In.

Z Run Out

This value will extend the thread by the amount entered. If the threading tool needs to pull out from the thread on an angle, enter a value for both Z Run Out and X Run Out. Typically, you would enter `0`.

X Run Out

When used with Z Run Out, will cause the tool to pull out of the thread on an angle.

Example. To specify a thread pull out of 100/1000" at 45 degrees, enter `0.707` for X Run Out and `0.707` for Z Run Out; this would cause a pull-out move at 45 degrees for a distance of 0.100 to be added to the thread cycle.

> ⓘ If the value for X Run Out is less than for Z Run Out, a pull out move of less than 45 degrees will occur; or, if the value for X Run Out is larger than for Z Run Out, a pull out move greater than 45 degrees will occur.

Cutting standard NPT Pipe Threads

The primary problem that most people encounter when trying to cut pipe threads is determining the correct Major or Minor diameter, which is necessary in order to program the tool path. Unfortunately, the Machinery's Handbook does not supply these numbers. It provides the pitch diameter, and the major or minor diameters must be calculated accordingly. This becomes tricky due to the fact that all of these diameters are at an angle; therefore, these values will change depending upon the horizontal Z value.

Step by step instructions will be provided for programming both a 2.5"-8 NPT external and a 2.5"-8 NPT internal thread to show the actual process required to determine the minor and major diameters.

First, a given horizontal value must be established to act as a gauge point. Since the Machinery Handbook supplies the pitch diameter at the start of the thread, the horizontal value most commonly used is Z0 (the face of the part). The system also assumes this value for the major and minor diameters, and will calculate the major and minor diameters at the start and end of the toolpath based on this assumption. The advantage of this is that only one value needs to be calculated; in the case of external pipe threads, only the minor diameter at the face of the part is needed, and with internal pipe threads only the major diameter at the face of the part is needed.

2.5" - 8 NPT External Pipe Thread

1. Find the Pitch Diameter at Beginning of External Thread (E0) from Machinery Handbook: American Pipe Threads: Table 3 (Basic Dimensions, American National Standard Taper Pipe Threads). For a 2.5" - 8 NPT external thread this value is 2.71953
2. Find the nominal truncated Height of Pipe Thread (h) from Machinery Handbook: American Pipe Threads: Table 1 (Limits on Crest and Root of American National Standard Taper Pipe Threads). This value is given as a max/min dimension; add the minimum and maximum height and divide by two to obtain the nominal thread height. For a 2.5" - 8 NPT external thread this would be (.1000+.09275)/2 or 0.096375
3. Find the Minor diameter at the start of the thread. To calculate this value, simply subtract the nominal thread height from the Pitch diameter (E0). For a 2.5" - 8 NPT external thread this would be 2.71953 - 0.096375 or 2.623155

2.5" - 8 NPT Internal Pipe Thread

1. Find the Pitch Diameter at Beginning of External Thread (E1) from Machinery Handbook: American Pipe Threads: Table 3 (Basic Dimensions, American National Standard Taper Pipe Threads). For a 2.5" - 8 NPT internal thread this value is 2.76216
2. Find the nominal truncated Height of Pipe Thread. This value does not change for external and internal threads and is the same as the 2.5" - 8þNPT external thread above (0.096375)

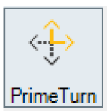
3. Find the Major diameter at the start of the thread. To calculate this value, simply add the nominal thread height to the Pitch diameter (E1). For a 2.5" - 8 NPT internal thread this would be 2.76216 + 0.096375, or 2.858535

American National Standard Taper Pipe Thread (NPT) Chart

This is a simple chart containing the values for the Standard NPT Pipe Thread sizes. For an external thread, enter the Minor diameter as given on the chart, and for an internal thread, enter the Major diameter as given on the chart.

PIPE SIZE		EXTERNAL THREADS		INTERNAL THREADS	
Nominal Pipe Size	TPI	Minor	Major	Minor	Major
1/16”	27	0.2439	0.2985	0.2539	0.3085
1/8”	27	0.3362	0.3908	0.3463	0.4009
1/4”	18	0.4360	0.5188	0.4502	0.5330
3/8”	18	0.5706	0.6534	0.5856	0.6684
1/2”	14	0.7045	0.8124	0.7245	0.8324
3/4”	14	0.9138	1.0216	0.9349	1.0428
1"	11 1/2	1.1475	1.2797	1.1725	1.3047
1 1/4”	11 1/2	1.4910	1.6232	1.5173	1.6495
1 1/2”	11 1/2	1.7300	1.8622	1.7563	1.8884
2"	11 1/2	2.2029	2.3351	2.2302	2.3624
2 1/2”	8	2.6232	2.8159	2.6658	2.8585
3"	8	3.2442	3.4370	3.2921	3.4849
3 1/2”	8	3.7411	3.9339	3.7924	3.9852
4"	8	4.2380	4.4308	4.2908	4.4835
5"	8	5.2944	5.4871	5.3529	5.5457
6"	8	6.3497	6.5425	6.4096	6.6023
8"	8	8.3372	8.5300	8.4037	8.5964
10"	8	10.4489	10.6417	10.5246	10.7173

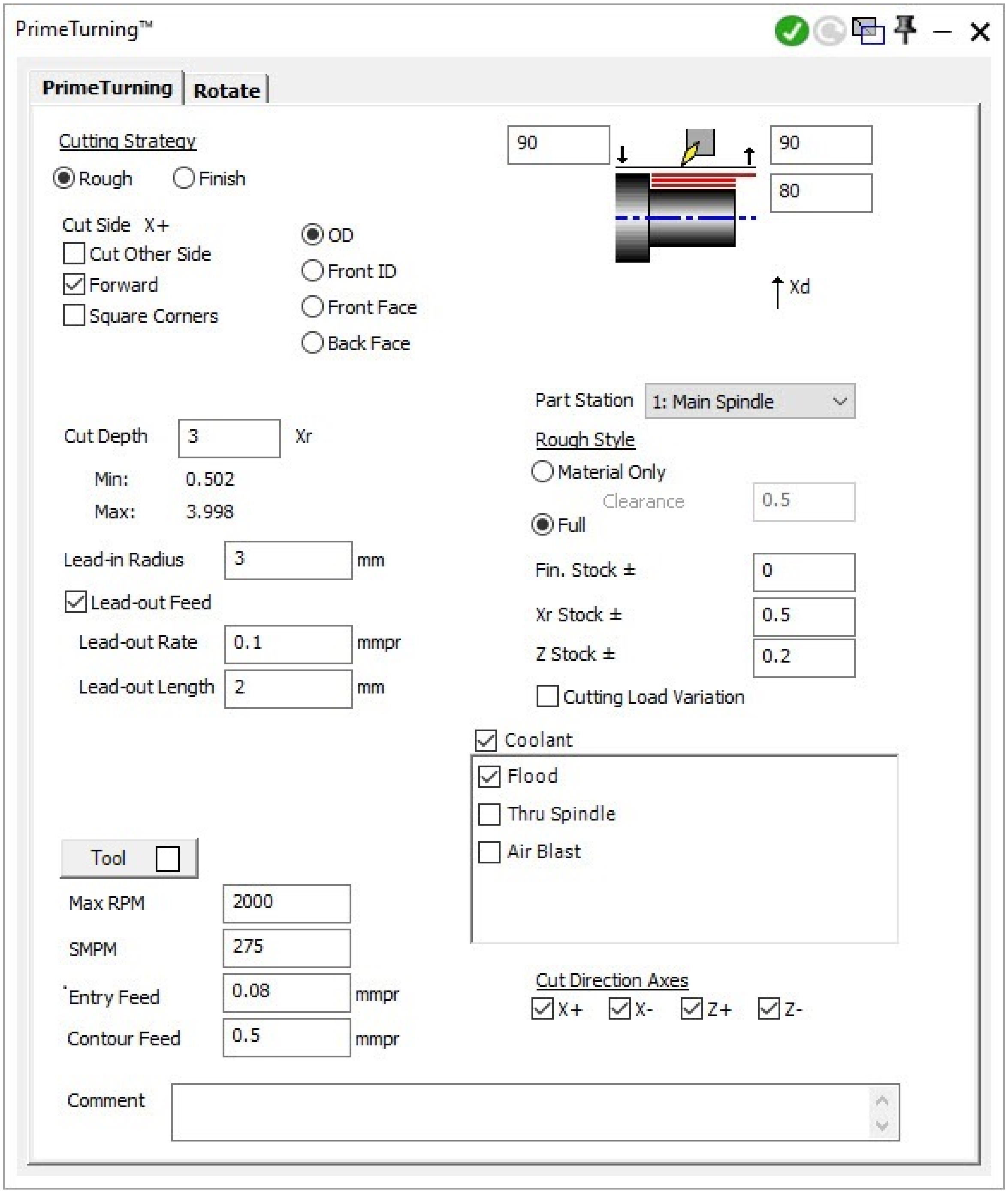
12"	8	12.4364	12.7286	12.6208	12.7142
14" OD	8	13.6786	13.8714	13.7763	13.9690
16" OD	8	15.6661	15.8589	15.7794	15.9721
18" OD	8	17.6536	17.8464	17.7786	17.9714
20" OD	8	19.6411	19.8339	19.7739	19.9667
24" OD	8	23.6161	23.8089	23.7646	23.9573



PrimeTurning Process

PrimeTurning is a high-performance turning strategy from Sandvik Coromant that promises increased material removal rates. PrimeTurning can only be used with CoroTurn Prime Type A and Type B inserts. Roughing and Finishing are both supported through PrimeTurning process. This strategy enters the part gently, and can cut either direction with the tool, automatically adjusting the feedrate as appropriate to maintain correct chip thickness.

PrimeTurning works with all the usual turning features, including Auto Clearance and Material Only. PrimeTurning requires CSS, so no controls are provided to disable it or to specify RPM and per revolution feedrates. Cut Depth is restricted according to Sandvik Coromant recommendations for each insert; these restrictions are displayed under the Cut Depth field as Min and Max values. Entry is always performed with a lead in arc. The radius of this arc may not be less than 1/3 the Cut Depth; we recommend using exactly the Cut Depth as the arc radius for best results.

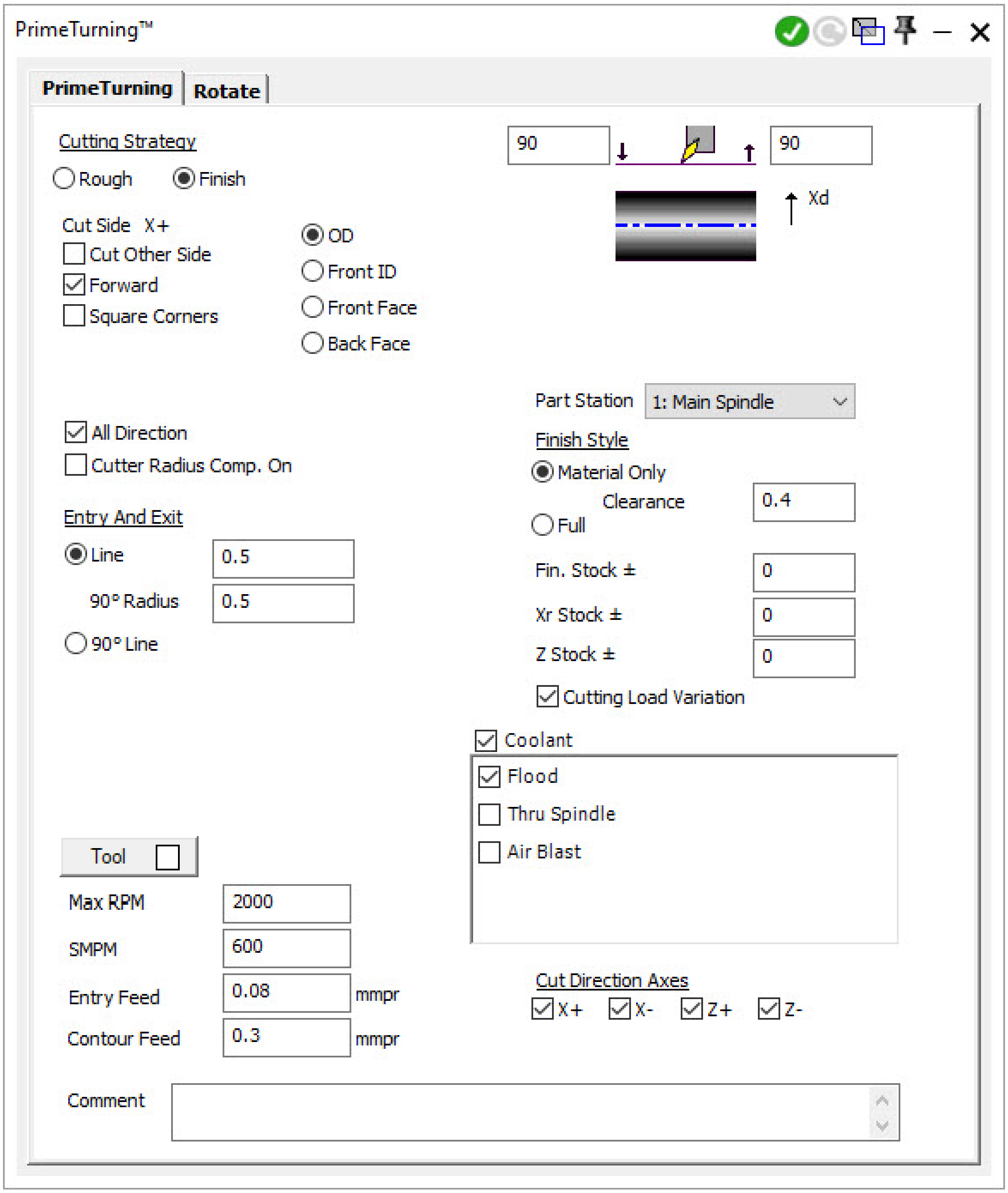


The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Rough strategy

When using the Rough strategy, the you might request a reduced lead out feedrate. Three parameters are available to control this behavior: a Lead out Feed checkbox and corresponding feedrate entry field, and a Lead out Length entry field. The specified feedrate will be applied for the specified length before leaving material during a cut, and from the material boundry to the material clearance (or for the specified length at the end of each cut, if Material Only is not enabled).

The Minimum Cut Depth and Maximum Cut Depth for your selected CoroTurn Prime insert are shown on the process dialog for convenience. Select your desired Cut Depth, Lead-in Radius (recommended value is the Cut Depth), and how much to slow the tool during the Lead-out. See your local Sandvik Coromant representative for additional advice about how to optimally configure this process.

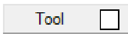


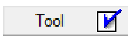
The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Finish strategy

When using the Finish strategy, the All Direction checkbox will machine each segment of the toolpath in the optimal direction for PrimeTurning. This is similar to the way "No Drag" works to machine each segment of a conventional Turn Contour process in the optimal direction for ISO inserts (but, since the optimal direction for CoroTurn Prime inserts in normally the dragging direction, it will usually cut each segment in the opposite direction from what "No Drag" would do).

Tool

- 

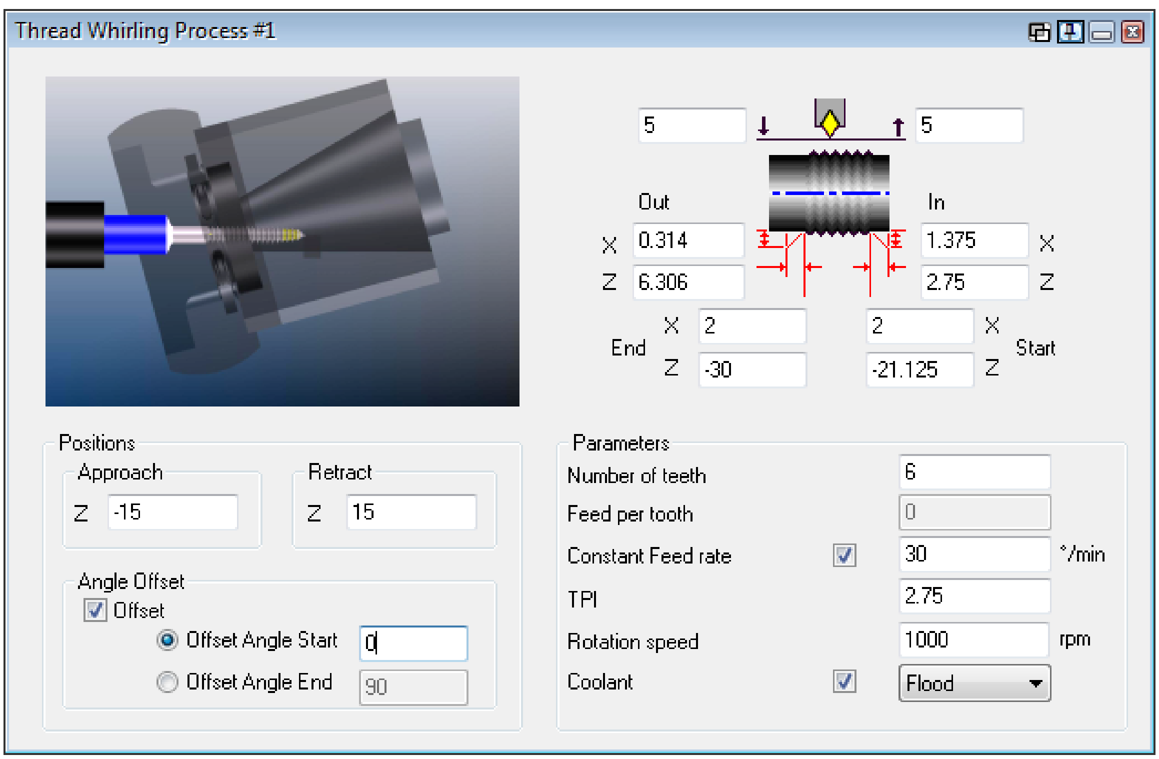
indicates that the tool instance has no data attached to it.
- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Thread Whirling

Thread Whirling is a machining process where the cutters are mounted on the inside of a cutting ring or cutting holder rather than the outside of a milling tool.



Depths Diagram

Entry Clearance Diameter/Radius

The tool will rapid to this diameter/radius before beginning the threading cuts. The tool will also return to this value for each new cutting pass.

Exit Clearance Diameter/Radius

The tool will rapid to this value after completing the threading process. The tool will also move to the next operation at this X value.

In X and In Z

For X - This represents the Xr component of the Run In move. There are several specific behaviors available. If this is equal to the Z Run In, the entry will be 45 degrees from the taper slope. A value of zero will be a straight Run In and will continue the taper.

For Z - This is the incremental distance to position to the right of the true thread start. A value of `0` will start the tool exactly at the thread start. Please note that the Z axis value is not measured along the taper and only positive values are valid.

Out X and Out Z

For X - This represents the Xr component of the Run Out move. There are several specific behaviors available. If this is equal to the Z Run In, the entry will be 45 degrees from the taper slope. A value of zero will be a straight Run Out and will continue the taper.

For Z - This is the incremental distance to position for the tool to over-travel at the left of the true thread end. A value of `0` will stop the tool exactly at the thread end. Please note that the Z axis value is not measured along the taper and only positive values are valid.

Start X and Start Z

These values represent the absolute position of X and Z at the start of the thread.

End X and End Z

These values represent the absolute position of X and Z at the end of the thread.

Positions

Approach

This is the Z approach position. The Whirling tool will rapid to this point in Z before rapiding to the start position of the toolpath.

Retract

This is the Z retract position. The Whirling tool will rapid to this point in Z after completing the Thread Whirling process.

Angle Offset

Offset

This checkbox enables Start/End Angle Offset values. This will allow you to set the rotary axis for a part that needs the start or end of the thread to be oriented to a specific angular value. This will be output in G-Code, but will not render.

Offset Angle Start

Rotary angle at which to start the process.

Offset Angle End

Rotary angle at which to end the process.

Parameters

Number of Teeth

Number of teeth for the Thread Whirling tool.

Feed per Tooth

Allows the Rotary Axis feed rate to be calculated per tooth. Toggling Constant Feed rate will override this value.

Constant Feed rate

This will override the Feed per Tooth with a desired feed rate in degrees per minute.

TPI/Pitch

Pitch represents the distance measured in millimeters from one thread to the next. TPI is the number of threads per inch.

Rotation Speed

This sets the tool spindle speed. The part spindle speed will be controlled by the rotary axis feed rate.

Coolant

Toggle coolant on or off and a drop-down box with coolant choices. Flood is standard.

Groove Cycle

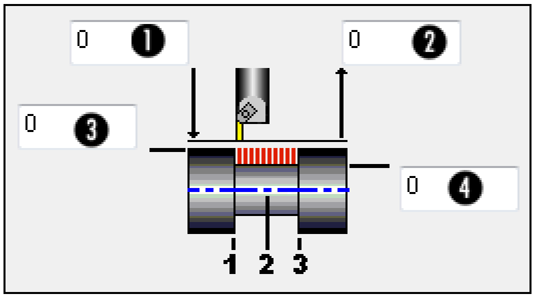
Designed around the Fanuc-style canned cycles for G74 and G75 output, Groove Cycle allows you to cut geometry-independent rectangular grooves.

Note: The user interface you see might display more controls, or fewer, or different ones. The items that appear depend on which product options are licensed and active. They also vary according to the Machine Definition Document (MDD) associated with the Machine type currently specified in the Document Control dialog.

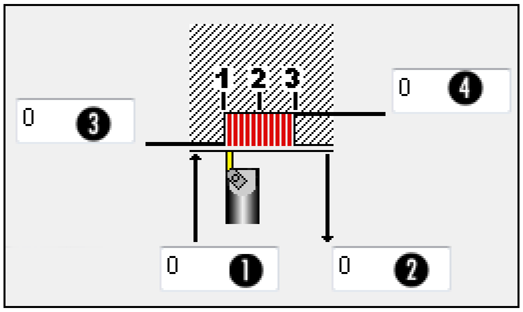
Cut Other Side

Specify whether to use the non-primary side of the tool. For example, an X+ tool for the outer diameter (OD) might use the X- side to cut on the inner diameter (ID).

OD

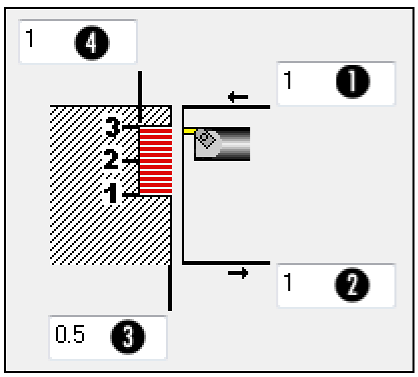


ID

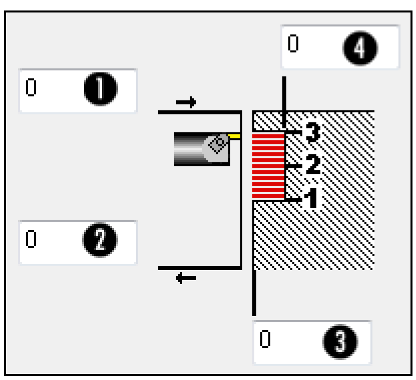


1. Entry Clearance Value
2. Exit Clearance Value
3. Start Cutting
4. Bottom of the Groove

Front Face



Back Face
(MTM only)



1. Entry Clearance Value
2. Exit Clearance Value
3. Start Cutting
4. End of the Groove

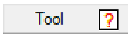
Cutoff
Specify whether the postprocessor should consider this a cutoff process.

Spindle
(MTM only) Choose a spindle from the drop-down list.

X Position/Z Position
Select one of the three radial buttons to specify the dimension of that position in the corresponding dialog.

Groove Width
Enter the groove width.

CSS (Constant Surface Speed)
Selecting the CSS item will activate Constant Surface Speed (CSS). CSS will cause the spindle RPM to constantly change based on the diameter the tool is at and the SFPM used.



Tool

- 

indicates that the tool instance has no data attached to it.

- 

indicates that the tool instance has data attached to it.

Clicking this button opens the **Feeds and Speeds Table** for the tool instance in the current part. This dialog lets you view, add, or delete entries for this tool instance. When an entry is selected, clicking **Calc Speed** copies the entry's Speed value into the process dialog, and clicking **Calc Feed** copies the entry's Feed value into the process dialog. For a full description of the **Feeds and Speeds Table**, see the *Common Reference* guide.

Max RPM
The Max RPM setting is used to set an upper safe limit on the spindle RPM. If CSS is off, the specified RPM value will be used for the spindle speed.

The SFPM and Feed values can be automatically calculated based on the material selected if the CutDATA Material database is installed. In order for these values to be calculated and entered in the appropriate boxes, the SFPM and Feed buttons must be clicked. If no material is selected or the CutDATA Material database is not installed, the user will need to manually enter values for the feed and speed.

Use Auto Clearance
Select this checkbox if you want to use the system defaults for clearances.

Prefer Canned
Select this checkbox to generate canned cycles in the posted code. Checking this box will disable Equalize Depths and Equalize Stepovers.

Coolant
A checkbox indicates whether coolant is turned on in a process. Flood is the standard coolant option. Additional coolant options are available with custom post processors.

Start at Position
Toggle to start at either position 1 or position 3.

Depth of Cut
Define the depth of cut, or the peck distance.

Retract Amount
Distance to retract at the end of each peck.

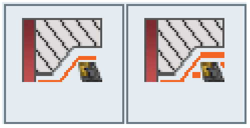
Equalize Depths
When checked, the peck depth will be recalculated to ensure that each peck is the same depth.

Stepover
Define the stepover amount between each peck. A value of `0` will cause a cutoff.

Equalize Stepovers
When checked, the stepover is recalculated to ensure that each stepover move is the same amount.

Relief Amount
Distance to move across at the end of the last peck before retracting out of the groove.

Add Relief to First Cut
When checked, the relief amount will be applied after the first amount. You should only check this option when there is no material on the front wall of the groove.



Pinch Contour/Rough

A Pinch Turning process lets you rough a part on a twin turret lathe using two tools simultaneously. This can reduce cycle times and can provide support for a long part away from the chuck. Both tools begin with each stroke together, with an optional lag distance between turrets. The second cut can finish sooner than the first, depending on the length of cuts.

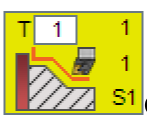
Be careful when using Pinch Turning in conjunction with Material Only. Using Material Only can create strokes that may not sync with the lag applied to the second tool. With some stock conditions, it is possible that the second stroke in a pair of roughing or contouring strokes can start further into the part than the first stroke.

Therefore, with Pinch Turning, always check the rendering. If the second tool has this problem, you will see a gouge.

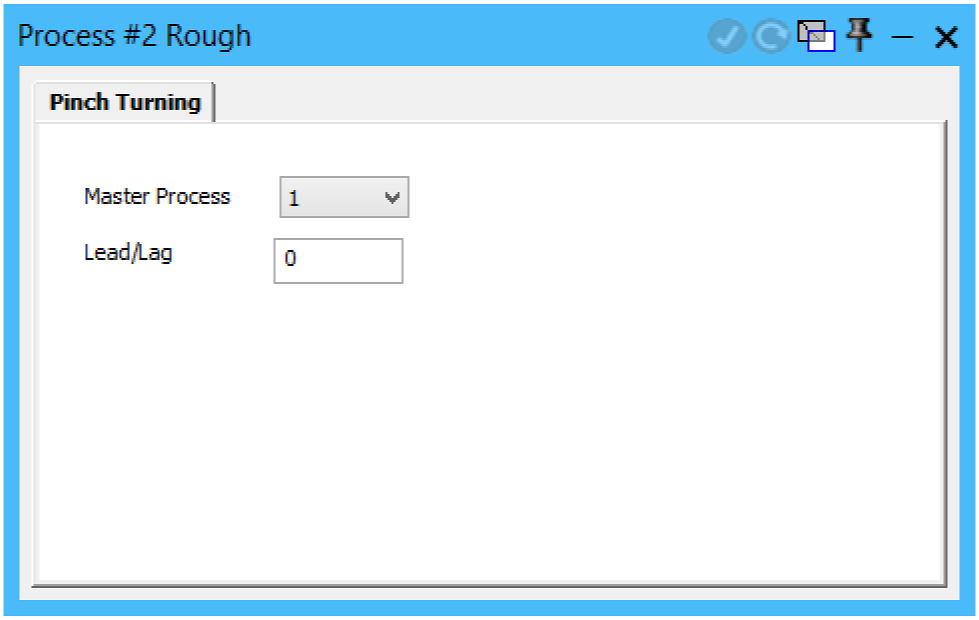
Conditions

- The process is only for ops where each cut depth is a single cut.
- The shape cannot decrease in X (it must be monotonically increasing in X). That is, the shape cannot have grooves of any size.
- The current MDD must be a twin-turret lathe-type machine.
- You must specify an identical tool on the opposite turret.

Steps

1. 

Create a new Roughing process (or double-click an existing roughing operation in order to replace it).
2. Select a tool tile situated on the lower turret. It must be of the same type and size as the upper-turret tool, but must point in the opposite direction.
3. Drop this tool onto an empty process tile. You will notice that Pinch Roughing process is now available. Select the Pinch Roughing Process tile.
4. The Pinch turning dialog appears.



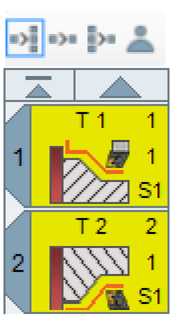
Master Process

If there are more than one roughing processes in the process list utilizing the same tool, you are able to choose the Master Process from the dropdown list. The number on the dropdown corresponds to the tile number of the process.

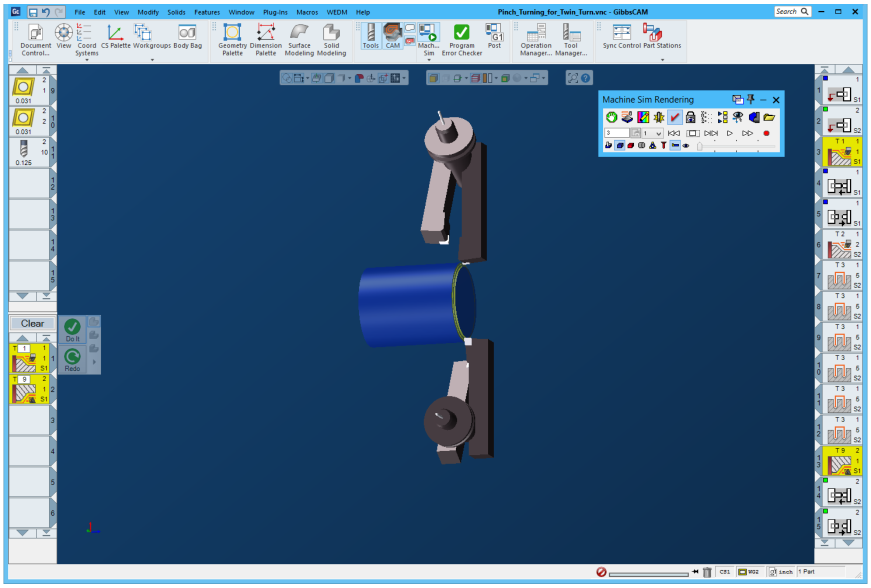
Lead-Lag

Specifies how far the lower turret will lag behind the upper turret, using the current units of measure. A lag distance of `0` (recommended) will result in a lag of 1/2 revolution, because the lower turret is cutting 180 degrees around the bar from the upper turret.

Click Do it(or Redo) to generate two operations, (replacing the previous Roughing operation).



1. Upper Operation tile, using tool on upper turret
2. Lower Operation tile, using tool on lower turret

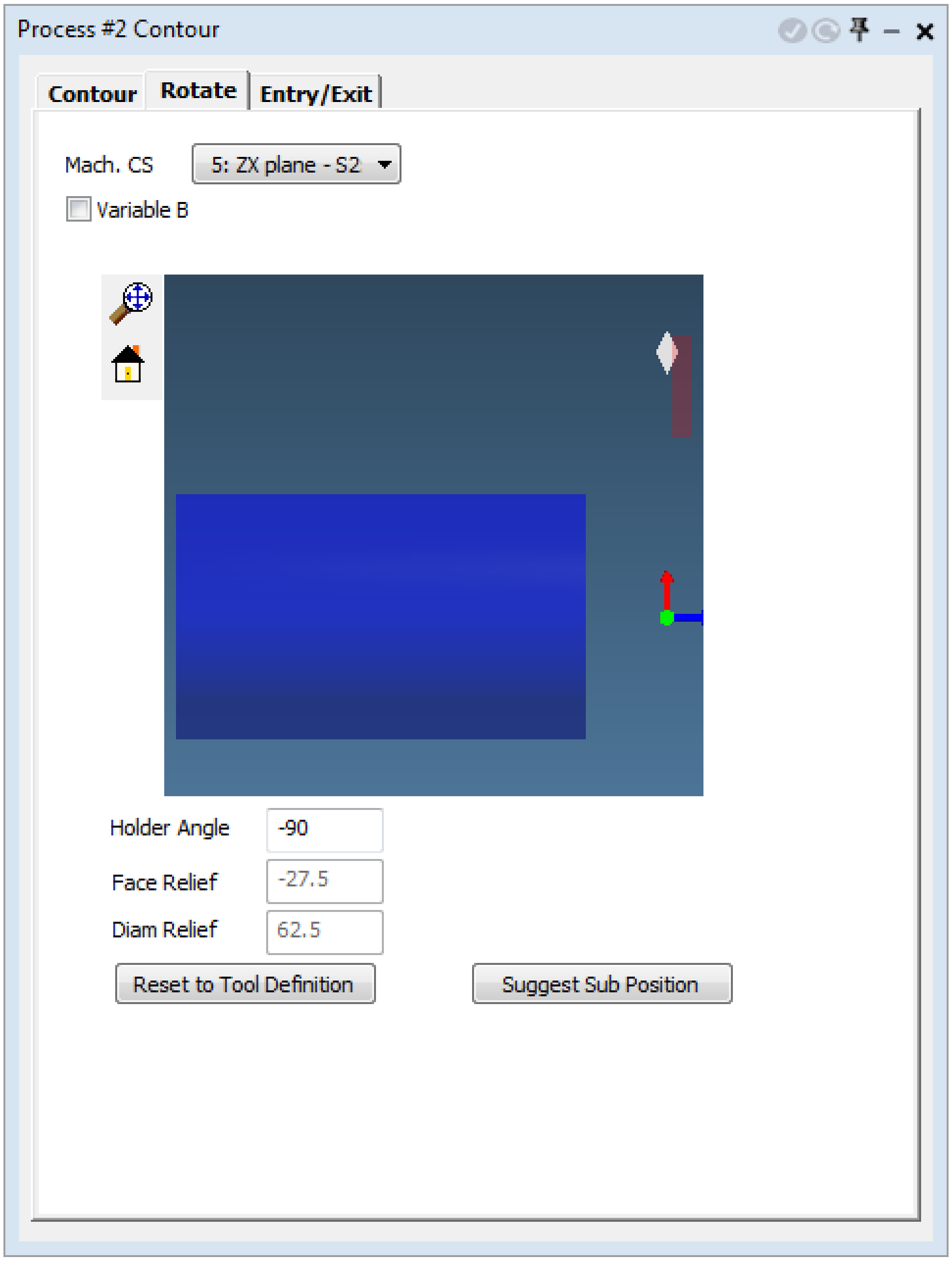


Rotate Tab for Turning Machines

The **Rotate** tab is available for certain Turning processes when your MDD supports rotation. For information on controls offered in this tab, see Rotate Tab Controls .

Rotate Tab Controls

The Rotate tab, found in certain Turning process dialogs when an advanced MDD is being used, provides access to special machining functions.



Mach. CS

This drop-down list lets you choose the coordinate system the operation will be created from. The system will output the appropriate rotation moves to correctly position the part to cut the selected Machining CS. The tool always approaches the part and cuts along the positive depth axis of the selected machining coordinate system.

Variable B

If this checkbox is offered, you can specify many parameters to control the variable B-Axis. When the checkbox is not offered or not selected, a view window shows the relative positioning of toolholder, tool, and stock, and you can specify only three parameters:

Holder Angle

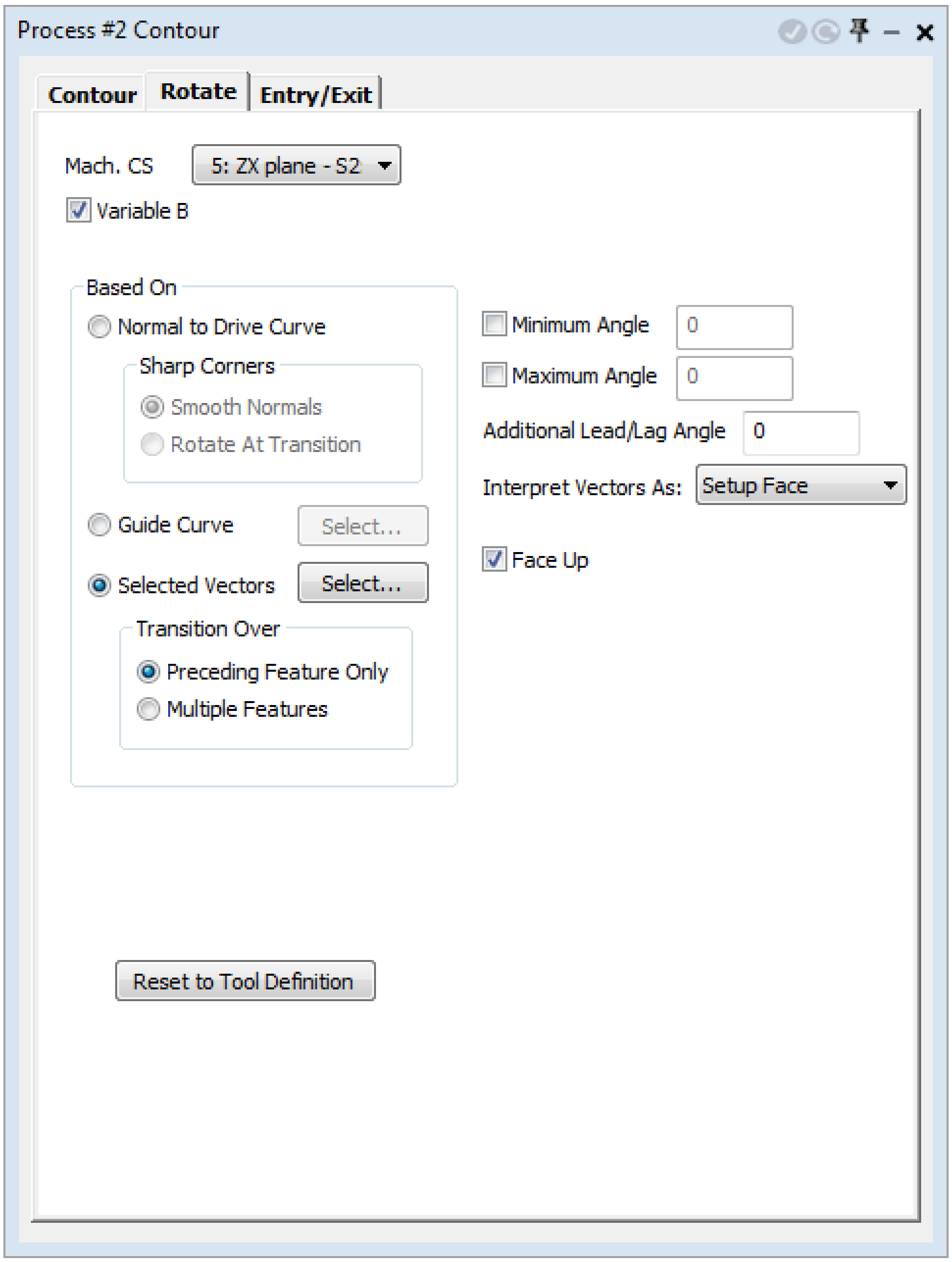
Enter the angle of the toolholder relative to the stock.

Face Relief
Diameter Relief
These values sum to (90° minus the angle of the insert).

Reset to Tool Definition
Restores all values to the defaults calculated for the current tool.

Suggest Subposition
When available, click to accept the system's recommendation for subposition.

Parameters Available for Variable B



Based On: Normal to Drive Curve

Sharp Corners

Choose Smooth Normals to create a smooth transition from one normal (perpendicular) to another. Choose Rotate at Transition to allow the tool to rotate at a sharp normal.

Based On: Guide Curve

Click the Select button to choose a curve.

Based On: Selected Vectors

Click the Select button to choose a vector. For Transition Over:
Choose Preceding Feature Only to apply the vector only to the preceding feature.
Choose Multiple Vectors to apply the same vector to several features.

Minimum Angle

Maximum Angle

If you want to specify a minimum and/or maximum angle, select the checkbox and enter a value.

Additional Lead/Lag Angle

You can specify a nonzero angle for leading or lagging the cut.

Interpret Vectors As

Make a choice from:
Setup Face
Setup Diameter
Insert Vector

Face Up

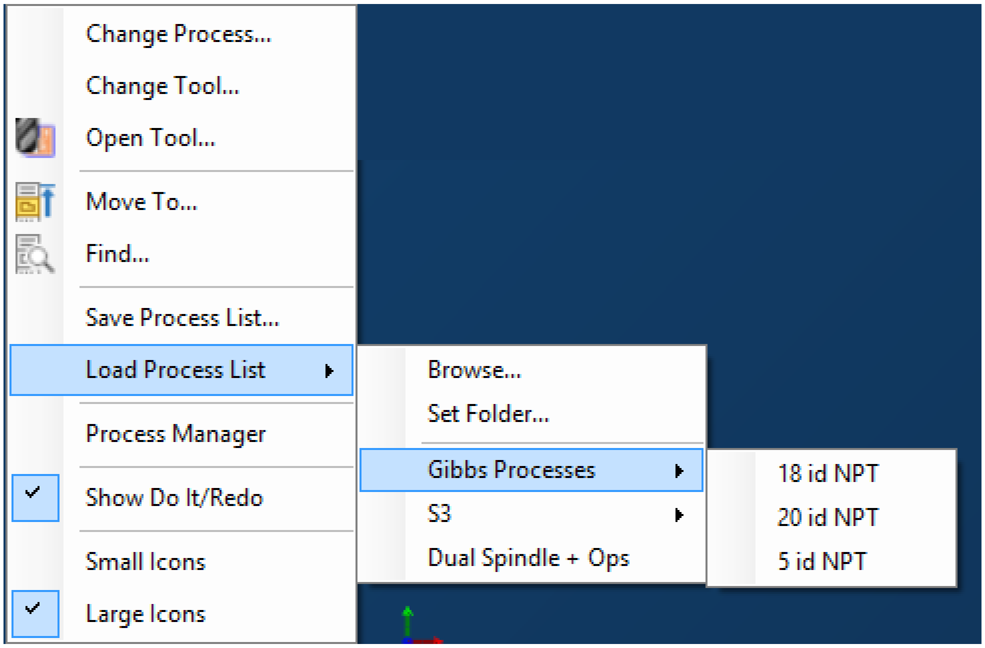
Select this checkbox for face-up cutting. Deselect for face-down cutting.

Process Groups

Pre-Defined Process Groups

All machining operations are created from the information contained in the Process list. You create processes by double-clicking a Process tile, choosing a Process Type and Tool and then entering the necessary information in the Process dialog. A Process Group is the collection of Process tiles contained in the Process List at any one time. A Process Group contains all of the tooling and machining information to generate a specific set of operations, such as creating a group that drills, roughs, and contours a shape.

You can save a Process Group as an external file that you can load into other part files. You can access and reuse common machining and tool data for multiple part files without having to recreate tools and processes. For example, if you regularly drill and tap the same size holes, a Process Group is a great solution for saving time.



You can save Process Groups by selecting Save Process list from the Process List Right-click menu when your Process list contains the completed Process tiles that will compose the group. A prompt appears for a file name and a location to save the file. After you save a Process Group file, it can be loaded into any part file by selecting Load Process List. You can also load process groups by choosing a directory that contains Process Group files. To choose a directory, select Set Folder from the menu. When a directory is set, all the Process Group files contained in that directory appear in the menu.

When a Process Group is loaded into a part file, any Process tiles currently in the Process list that are highlighted are removed and replaced by the loaded Process Group. If this removes Process tiles that were needed, select  Undo from the quick access toolbar. Unselected processes are not replaced.

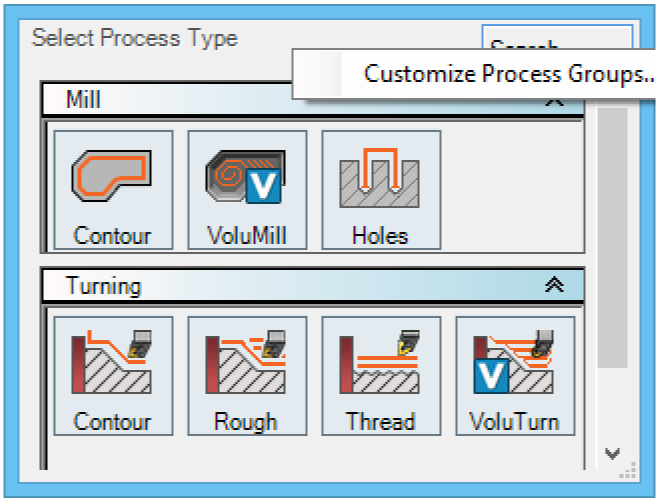
If the Tool list already contains Tool tiles, those tools are deselected but not removed from the list. The system searches the existing Tool list to find the necessary tools for the loaded Process Group. First, the system searches for an exact tool match. If an exact match is not found, the system searches for a close match, such as a tool with a longer tool or flute length. A tool identified as a close match is used. If the system cannot to find an exact match or a close match, the necessary tools for the loaded Process Group are created and added to the Tool list in the first available positions. Added tools are highlighted.

After the Process Group is loaded into the Process list, select the appropriate geometry to act as the cut shape and click the Do It button to create the operation and toolpath.

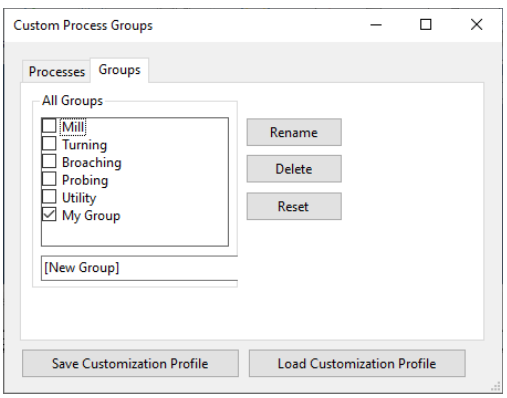
Customizing Process Groups

The Select Process Type dialog can be customized. You can choose which processes are displayed and also create custom profiles based on your MDD type and Processes available.

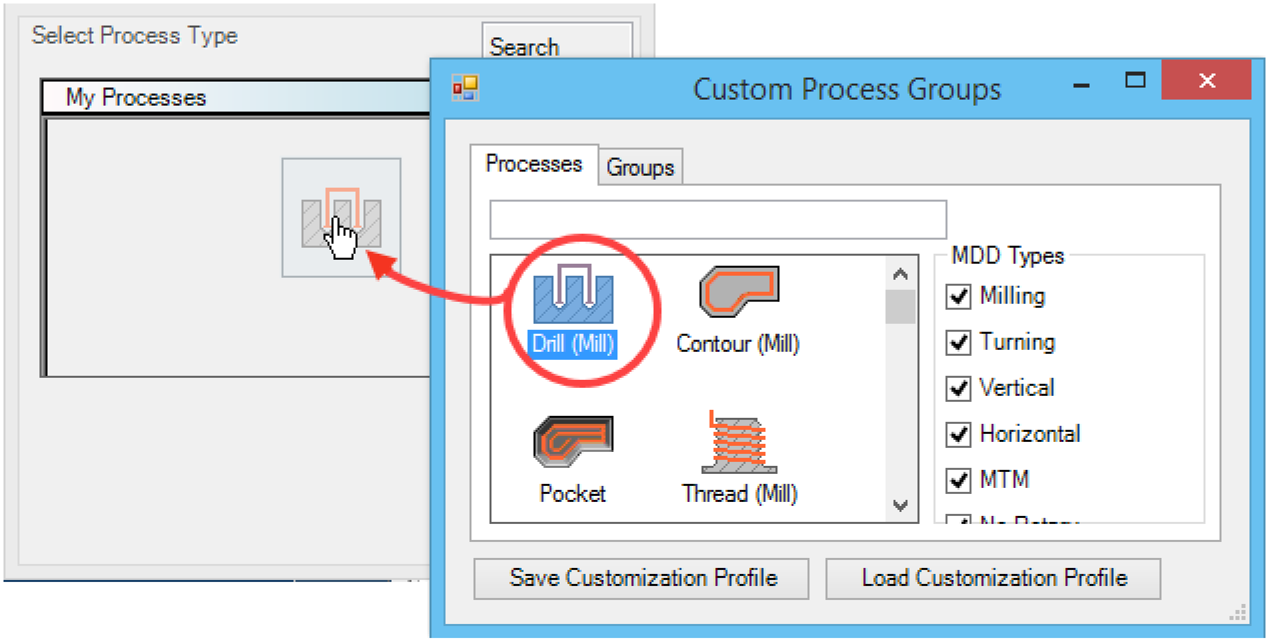
Right-click the title bar of the Select Process Type dialog and choose Customize Process Groups as shown below.



You can now edit an existing group or create your own using the Group Tab. Checking/unchecking the Groups will turn on/off the display of existing process groups.



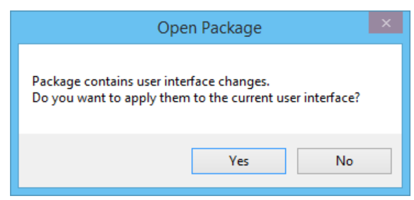
In the Processes Tab you can view available processes and MDD types. To create your own process list simply drag the required processes in or out of the Select Process Type dialog.



Saving and Loading Customization Profiles

The bottom of the dialog has two buttons: Save lets you save the current customization settings in a `*.cus` file for later reuse; Load opens a dialog that lets you find and use a previously saved `*.cus` file.

Note: When you load a package file (`*.gcpkg`) that contains user interface customizations, the system offers you the option of applying or rejecting the customizations before loading the package contents.



Machining

Once a Process has been created it needs to be applied to the geometry on your model. To do this you select the geometry and position machining markers.

What is a Cut Shape?

A cut shape is used to generate a toolpath. It is not drawn on the screen, but can be visualized as the finished shape left after the removal of material by the toolpath. A cut shape (not the original geometry) is used to create a toolpath because programming the toolpath to the geometry as it is defined on the blueprint will usually gouge the part. The software automatically generates the cut shape. Various specifications and limitations are taken into consideration in the creation of the cut shape.

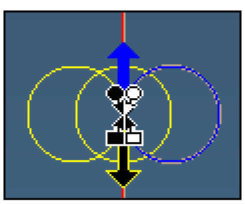
The machining markers (the start and end point and start and end feature markers) allow the user to specify the portion of geometry (or the entire shape) that will act as the initial outline of the cut shape. The system then takes into account the physical attributes of the tool being used in the process, such as insert type, tool holder, relief specs, etc. in order to prevent possible tool interference when applying the tool to the cut shape being machined. The cut shape is further governed by information entered in the Process dialog, such as Entry/Exit Radius, Stock Shape, Axes, etc. The system employs the concept of a cut shape so that it is not necessary to create different geometry for different operations in order to avoid gouging the part.

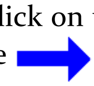
For drilling and threading functions, geometry is not required to create an operation

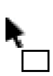
Machining Markers

You use Machining Markers to specify the start and end feature and start and end point of the cut shape, the cut direction, and the offset position of the tool. These markers appear when you select geometry as the cut shape for Contouring and Roughing processes. The exception is when more than one set of geometry is selected. In this case, the system assumes that the cutting is on center or engraving. The D-pointer appears when swept walls are created for roughing and/or contouring processes.

Cutter Side and Direction:



The circles represent the offset position of the tool in relation to the cut shape: on the outside of the geometry, on the inside of the geometry, or on the centerline. The arrows indicate the direction of tool travel, indicating whether a climb or conventional cut is made. Click on the circle and direction arrow you want to use. The arrow for the tool direction is blue  and the cutter side is bold .

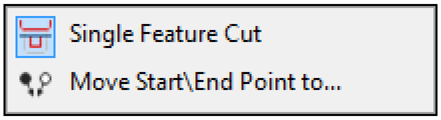
 Start Feature:
The geometry feature, such as a line or circle, on which the tool starts cutting.

 Start Point:
The point on the start feature where the tool starts cutting.

 End Feature:
The geometry feature on which the tool stops cutting.

 End Point:
The point on the end feature where the tool stops cutting.

Move Start\End Point to:



The right-click context menu for a start or end point includes the Move Start\End Point to option. By selecting this option, you will be prompted by a dialog to enter a new value (+ or -) by which to extend or trim the point from the beginning or end of the last feature. The options that appear depend on whether you select a Start Point or an End Point machining marker.

 D-Pointer:
The drive curve used when creating swept surfaces. Must be an open, terminated shape.

How Machining Markers Work

Machining Markers appear on selected geometry for contouring and roughing processes only. To move a marker, place the cursor over the marker and click and hold down the mouse button. The cursor changes to the marker. This is called “picking up a marker.” You can then move the marker to the desired location and drop it by releasing the mouse button.

Note: When positioning or placing a marker, place the tip of the marker arrowhead onto the line, circle, or point.

When the Start Feature marker is moved to a new feature on the geometry, the Start Point marker will “follow” it and snap to the same point as the Start Feature. This is also true for the End Feature marker. To make the Start Point and End Points the same: drag the Start Feature to the desired feature, and drag the Start Point to the desired location, drag the End Feature to the same feature- the End Point automatically snaps to the Start Point.

ⓘ For precise control over the Start and End Point marker locations, create a geometry point at the desired location. Dragging a Start or End Point marker close to the point will cause the marker to snap to the point and use its exact XZ values.

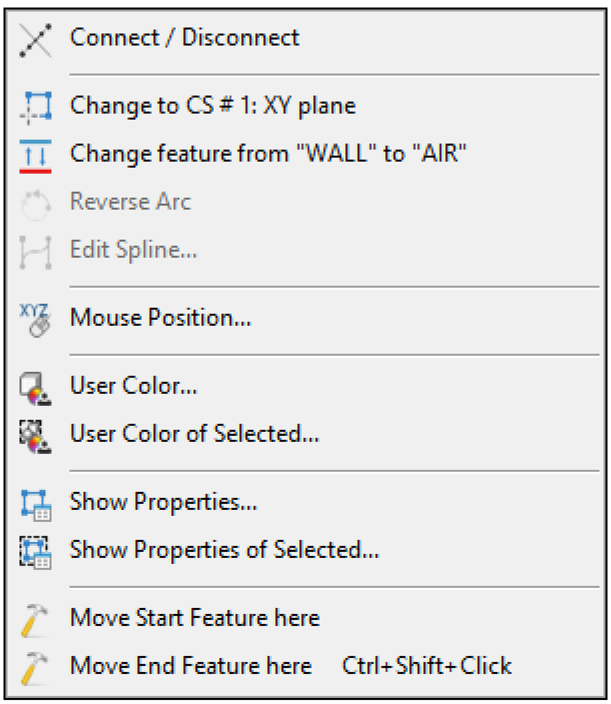
ⓘ After placing a contour machining marker in wireframe mode, you can `Ctrl-Click` off the geometry to move the marker to the midpoint of an element.

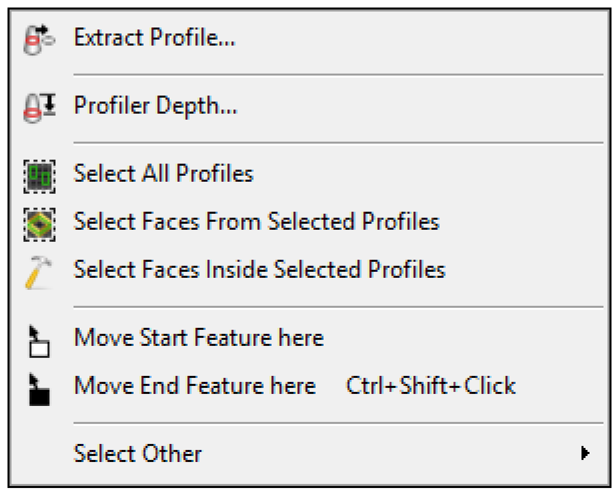
Start and End Points

The Start and End Points do not necessarily have to be on the part geometry. You may want the tool to start or end its toolpath off the part. You can do this by moving the markers. A geometry feature, such as a line or circle, is trimmed between two connectors. When the Start Point marker is dragged off the part, it automatically snaps to the nearest extension of the Start Feature. The nearest section of the Start Feature may be a section that was trimmed away, so the Start Point snaps to an extension of the Start Feature. This is also true for the End Feature. Press `Ctrl+Shift click` to set end feature markers. When you press `Ctrl+Shift click` , the end point markers snap to the location you click.

Move Machining Marker Options

Machining Marker Start and End Features positions can be set with a right mouse click. This works with Turning Roughing, Turning Contour, and Mill Contour processes on geometry or a Profiler shape. Simply right-click where you wish to place the Start Feature or End Feature marker and make a selection from the menu. The Start Feature and Point or End Feature and Point markers will be placed exactly where you clicked on the geometry or profile.





Geometry Right Mouse Menu | Profiler Right Mouse Menu

Selected Geometry

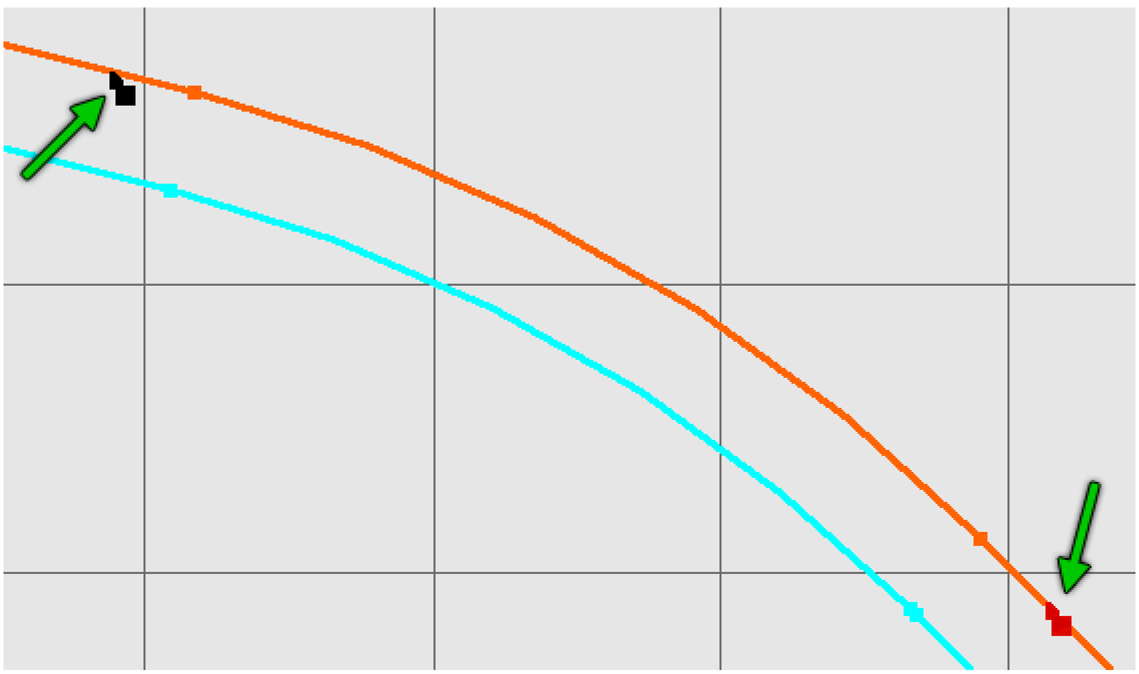
You use the machining markers to specify the portions of geometry to act as the outline for the cut shape. When markers are present on geometry, the cut shape is indicated by a dark blue color. When the cut shape is not the entire contour, the geometry not included as part of the cut shape is drawn in light blue.

You can also use the profiler to create the geometry for lathe processes from a solid. See the guides for *SolidSurfacer* or *2.5D Solids*.

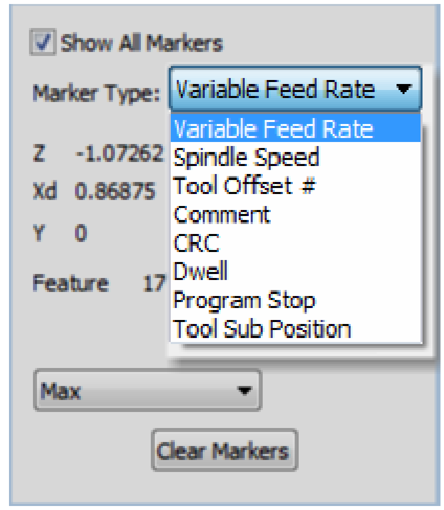
Utility Markers

You use the Utility Markers dialog to edit various position-dependent toolpath data. For each operation, you can select a variety of utility marker types, many of which have additional sub-options. Utility marker types include Variable Feed Rate, Spindle Speed, Tool Offset #, Text, CRC, Dwell, Program Stop, and Tool Sub Position.

This image shows Utility Markers placed on toolpath. The markers are placed by an arc, modifying the speed going into and coming out of the arc.



Utility Markers dialog options:
Show All Markers:



Display the icons for all utility marker types on the toolpath. When you step through the markers, the dialog updates so you can see the details of each utility marker. Each marker displays a unique icon.

Marker Type:
Except for Variable Feed Rate and CRC, markers require specific post processor support. Please contact the Tech Support Department for more information.

 Variable Feed Rate:

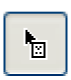
You can set the feedrate for the elements of the toolpath following the marker. The five sub-options for this type of marker are User, Entry, Contour, Percent, and Max. User allows you to explicitly set the feedrate. Entry sets to feedrate to the defined entry feedrate for the operation. Contour sets the feedrate to the defined contour feed rate for the operation. Percent sets the feedrate as a percentage that you specify of the last fixed feedrate marker. Max sets the feedrate to the maximum feedrate defined by the post processor.

 Spindle Speed:

For turning operations, this marker sets the spindle speed to the value defined in the SMPM (Surface Meters Per Minute) or SFPM (Surface Feet Per Minute) field.

 Tool Offset #:

This marker sets the tool offset. Three options are available: Tl Offset, Deflect Tl Offset, and Explicit Offset. Tl Offset sets the offset to the Offset # defined by the tool. Deflect Tl Offset sets the offset to the Deflection Compensation Offset # defined by the tool. Explicit sets the offset to a value you define.

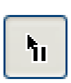
 Text:

You use this marker to insert a comment into the posted output.

 CRC:

You use this marker to turn CRC on or off during an operation. Three options are available: On, Off, and Reverse.

For more information, see “Cutter Radius Compensation (CRC)” on page 34.

 Dwell:

This marker causes the program to pause (dwell) for the specified time. This marker has two options: Seconds or Revolutions. The Revolutions option uses the current spindle speed to compute the time.

Program Stop:

This marker causes the post to output a program stop (M0). If Optional Program Stop is selected, the post outputs an optional stop (M1).

Tool Sub Position:

This option is only available for turning machines that allow tool sub positions. You can use this marker to set the tool sub position.

Next Marker:
Highlights the next marker in the toolpath and displays the marker information.

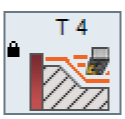
Previous Marker:
Highlights the previous marker in the toolpath and displays the marker information.

RPM:
For the Spindle Speed marker, type a number for revolutions per minute.

Edit Text:
For the Text marker, type the text you want to add.

Clear Markers:
Removes all markers from the toolpath.

Lock Button:
Locked items ( ) retain the values entered in this dialog even if the operation is reprocessed. Unlocked items ( ) return to their original values if the operation is reprocessed. Changes that affect the toolpath appear in the toolpath drawing and the rendered image. The information in



the process tile that created the operation is modified to reflect the changes made in this dialog. If an operation contains one or more locked values, a small lock symbol appears on the Operation Tile.

To lock or unlock a value:
Click the graphical button next to the right of the control to toggle its state between "locked"( ) or "unlocked"( ).

To display the Utility Markers dialog and the toolpath for an operation:
In the Operation List, right-click an operation tile and select Utility Markers.

To add a marker to a toolpath:

1. From Marker Type, select the type of marker you want to add.

   The icon changes to the type of marker you select.

2. Drag the marker to the position you want on the toolpath.

   The currently selected marker properties display in the dialog, and the currently selected marker displays in red.

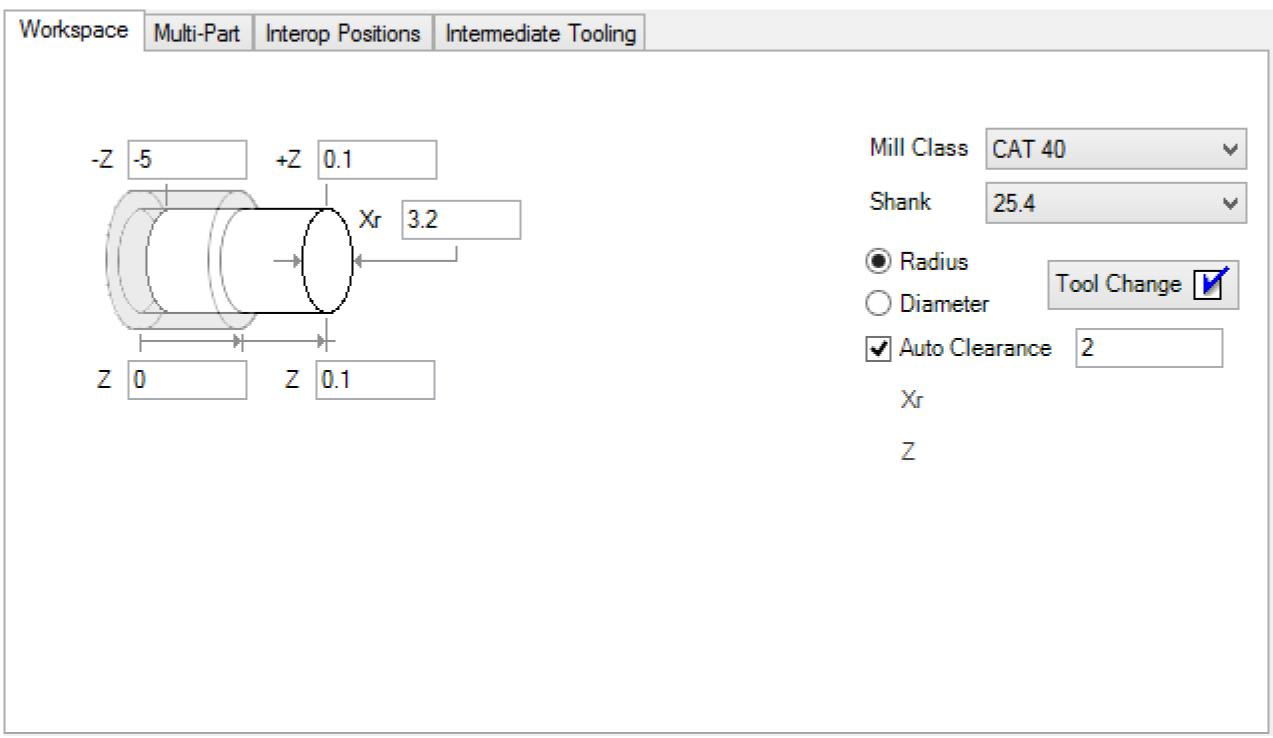
Operations

Operations contain finished toolpath. The toolpath consists of the actual moves the tool will make to cut a part, a visualization of the G-code to be output. For more information see the section on "Operations" in the *Getting Started* guide.

- Clearance Moves
- Canned Cycles
- Touch-Off Point Information

Clearance Moves

This section contains information and diagrams on rapiding and feeding around lathe parts. It is very important when working with lathes to avoid tool interference with the part, the spindle, etc., while at the same time quickly and efficiently maneuvering around the part. Clearance positioning is the term used for various positions the tool will move to when not actually cutting the part.



The primary tool change position is specified in the Document dialog. This position can be overridden on a tool by tool basis using the Tool Offset Data button in the Tool Creation dialog.

For more information on Tool Offset Data, refer to the Tool Creation chapter. If Tool Change is not enabled, it is assumed that the finished code will be manually edited to handle the tool change. Otherwise, the tool will start at the Tool Change position entered in the Interop Positions tab.

DCD/Setup Tab: Interop Positions

For any generic MDD, or for any custom MDD that specifies a Flow Axis Set (FAS) with an Interop Event Location whose axes are set to User, the **Interop Positions** page lets you decide whether or not to specify tool change positions for parkable axes.

The default setting is the Tool Change checkbox is unselected by default. When selected, then a pull-down menu appears that allows you to set user axis values for each FAS interop event.

- Partdefines the toolchange location relative to the part origin. This is useful in many cases, such as close turret rotation on a typical lathe setup.
- Part Station defines toolchange location relative to the part station origin without the part offset.
- Machine (not available for Generic MDDs) defines toolchange location relative to the machine root, which lets toolchange occur in the same machine location regardless of the part setup.
- TG Home defines the toolchange location relative to the toolgroup home location. Note that selecting an alternate origin changes the meaning of the inputcoordinates but does not affect output by default. The postprocessor may also choose to change the output mode based on your origin selection; this will require a post modification.

In addition to specifying the position of the turret when tools are changed, the Document dialog provides the user with two options for handling part clearance, Auto Clearance or Fixed Clearance. The selection made will determine how the system will calculate positioning moves between operations.

Auto Clearance

The Auto Clearance option performs several functions when it is turned on. It will calculate the part clearances in both Z and X that are used to position the tool between each operation. These positioning moves will be dynamically calculated for each operation. This means that as the stock conditions of the part change as material is removed, the clearance positions will adjust accordingly. When Auto Clearance is on, the system will also take into account where the tool needs to be to begin the next operations' toolpath when calculating the positioning moves. Additionally, the Auto Clearance function may add entry and/or exit moves to the toolpath in order to safely maneuver around the part. The Auto Clearance function generates the most efficient positioning moves around a part. However, canned cycles cannot be used in conjunction with Auto Clearance. In order to use canned cycles, which are turned on in Process dialogs by selecting the Prefer Canned option, Fixed Clearance positions must be used.

The Auto Clearance option requires the user to enter an offset amount from the part stock that the system uses to calculate the clearance positioning moves between operations. Because the stock conditions are constantly changing as material is removed from the part, in order to optimize the toolpaths, an offset amount is used for positioning rather than absolute positions. Fixed clearance, which is used when Auto Clearance is turned off, uses absolute positions.

Fixed Clearance

When the Auto Clearance option is turned off, fixed clearance positions are used by the system to calculate clearance moves. The user must enter an overall part clearance in the Document dialog, as well as Entry and Exit Clearance Positions in the Process dialogs for each operation. When using canned cycles, fixed clearance positioning should be used.

The overall part clearance is entered in the Document dialog in the X and Z text boxes that become active when Auto Clearance is turned off. They designate the position the tool will rapid to and from during a tool change. This position will also be used when moving from one approach type to another between operations that use the same tool. The absolute positions specified in the X and Z text boxes are locations the tool can rapid to when moving around the part. One or both of these fixed positions are used whenever a tool is moving to the start point of the toolpath or exiting from the toolpath. Where the tool moves when approaching and retracting from the part depends on the Approach Type selected and the positions specified in the Clearance Diagrams in the Process dialog.

The Approach Type selections are located in the upper left corner of the Process dialog. The tool can approach the part along two different axes—either X or Z. The tool will approach the part along the Z axis if Front Face is selected. The tool will approach the part in X if OD or Front ID is selected. When a Drilling Process is selected, the Approach Type is automatically set to Front Face. Only one selection can be made for each process.

Once the Approach Type is selected, the corresponding Clearance Diagram appears in the Process dialog. The boxes with the arrows next to them represent the Entry and Exit Clearance Positions that the tool may use when approaching and retracting from the part. The Entry and Exit Clearance Positions are only required when Auto Clearance is turned off.

When a Turn roughing cycle is selected, an additional move will be added between the Entry Clearance Position and the X Stock Start Position. When a Pattern Shift roughing cycle is selected, an additional move will be added between the Entry Clearance Position and the contour start point.

Clearance Diagrams

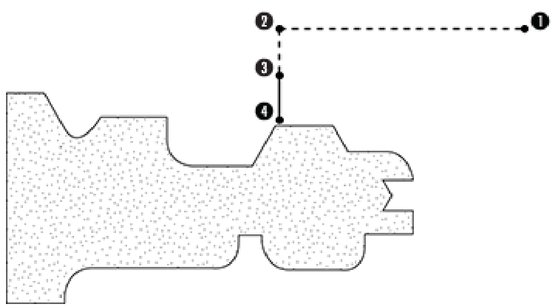
The tool will use some or all of the clearance positions depending on which Approach Type is selected. When Auto Clearance is selected, the tool will still move to the positions indicated in the diagrams shown below. However, the system will calculate these positions and they will change as the material conditions of the part change. Also, when Auto Clearance is on, the system may add additional entry and exit moves as necessary to prevent tool interference. The following conventions are used in the clearance diagrams.

Black Dot	Absolute coordinate the tool will move to; each Black Dot has an X and Z coordinate
Dashed Line	Rapid Move
Solid Line	Feed Move
SP - Start Point	The first move of the operation. Not necessarily the Start Point Machining Marker.
EP - End Point	The last move of the operation. Not necessarily the End Point Machining Marker.
OP1	Operation 1 (the first series of cuts made on the part)
OP2	Operation 2 (the second series of cuts made on the part)

Approaches from Tool Change Position

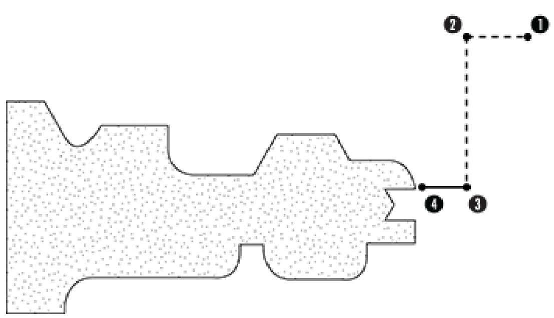
The tool can approach the part in three different ways from the tool change position.

OD Approach From Tool Change



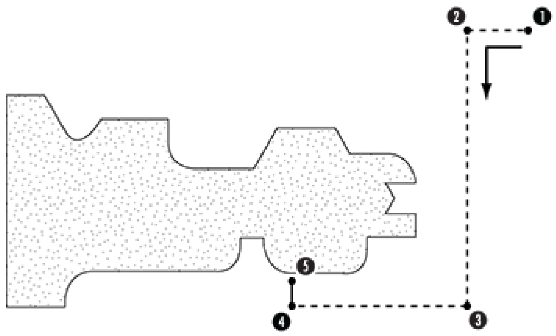
1. Tool Change
2. SP Z, Part Clearance Xd
3. SP Z, Entry Clearance Xd
4. SP Xd

Face Approach From Tool Change



1. Tool Change
2. SP Z, Part Clearance Xd
3. Entry Clearance Z, SP Xd
4. SP Z

ID Approach From Tool Change

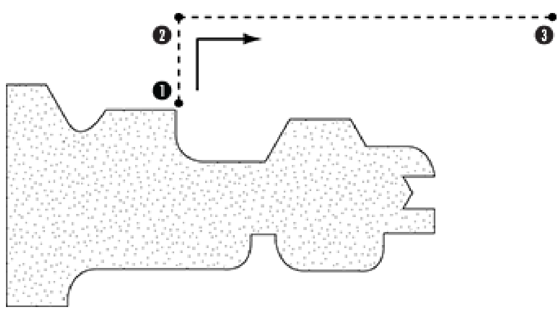


1. Tool Change
2. Part Clearance Z, Part Clearance Xd
3. Part Clearance Z, SP Xd
4. SP Z, Entry Clearance Xd
5. SP Xd

Exits To Tool Change Position

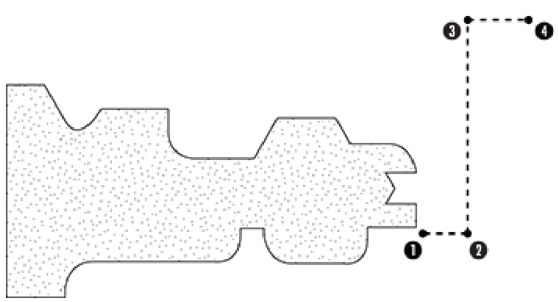
The tool can exit from the cut shape to the tool position in three different ways.

OD Exit To Tool Change



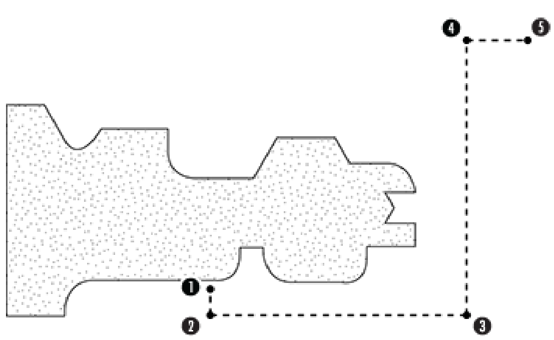
1. EP Xd
2. EP Z, Part Clearance Xd
3. Tool Change

Face Exit To Tool Change



1. EP Z
2. Part Clearance Z, EP Xd
3. Part Clearance Z, Part Clearance Xd
4. Tool Change

ID Exit To Tool Change

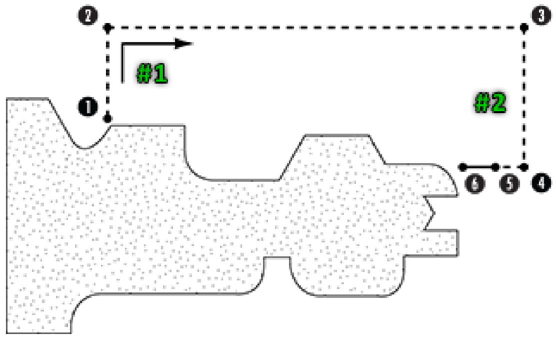


1. EP Z
2. EP Z, Part Clearance Xd
3. Part Clearance Z, Exit Clearance Xd
4. Part Clearance Z, Part Clearance Xd
5. Tool Change

Same Tool Positions

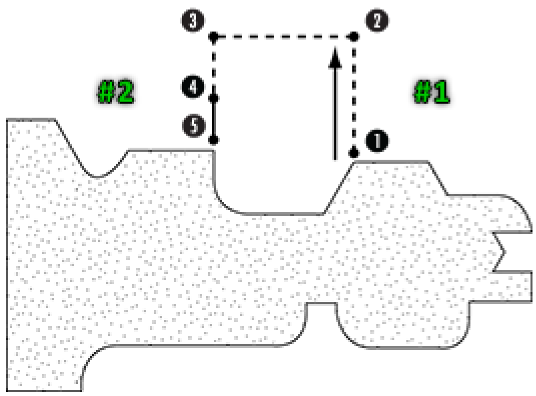
If the next operation uses the same tool, there are seven different methods the tool could use to get from the first operation to the start point of the next operation.

OD To Face



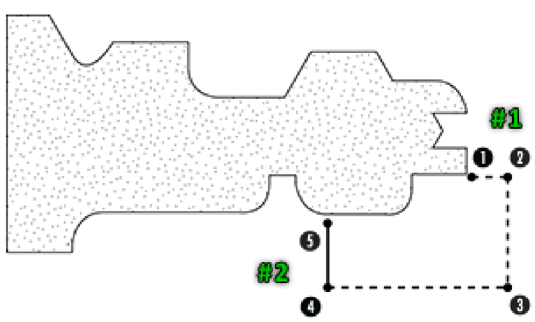
1. EP Z
2. EP Z, Part Clearance Xd
3. Part Clearance Z, Part Clearance Xd
4. Part Clearance Z, SP Xd
5. Entry Clearance Z, SP Xd
6. SP Z

OD To OD



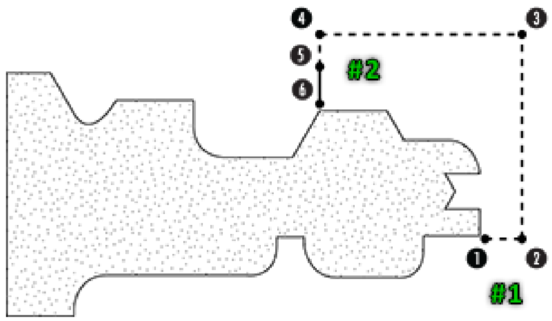
1. EP Xd
2. EP Z, Exit Clearance Xd
3. SP Z, Exit Clearance Xd
4. SP Z, Entry Clearance Xd
5. SP Xd

Face To ID



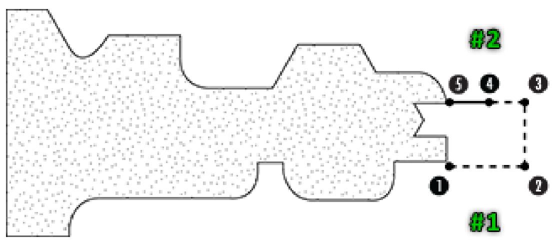
1. EP Z
2. Part Clearance Z, EP Xd
3. Part Clearance Z, Entry Clearance Xd
4. SP Z, Entry Clearance Xd
5. SP Xd

Face To OD



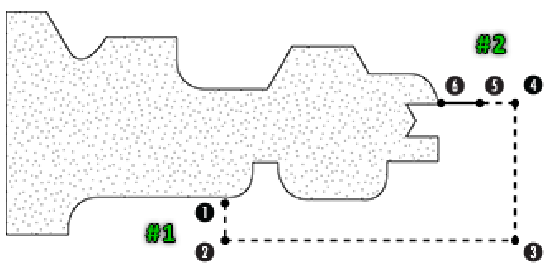
1. EP Z
2. EP Z, Part Clearance Xd
3. Part Clearance Z, Part Clearance Xd
4. SP Z, Part Clearance Xd
5. SP Z, Entry Clearance Xd
6. SP Xd

Face To Face



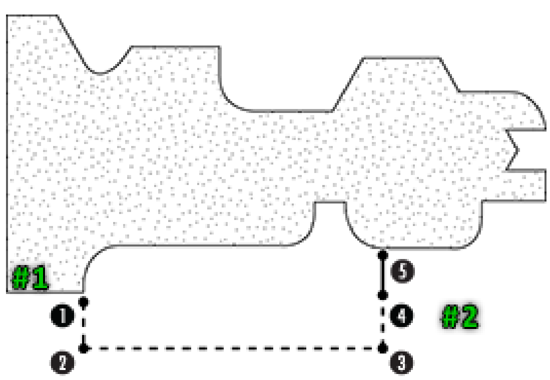
1. EP Z
2. EP Z, Part Clearance Xd
3. Part Clearance Z, Part Clearance Xd
4. Entry Clearance Z, SP Xd
5. SP Z

ID To Face



1. EP Xd
2. EP Z, Exit Clearance Xd
3. Part Clearance Z, Exit Clearance Xd
4. Part Clearance Z, SP Xd
5. Entry Clearance Z, SP Xd
6. SP Z

ID To ID



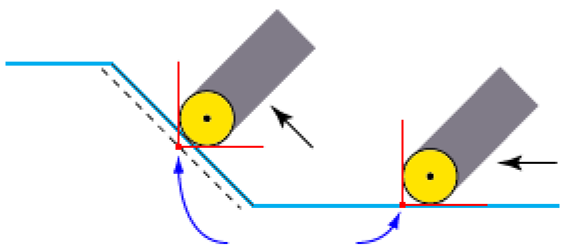
1. EP Xd
2. EP Z, Exit Clearance Xd
3. SP Z, Exit Clearance Xd
4. SP Z, Entry Clearance Xd
5. SP Xd

Canned Cycles

The Auto Clearance and Material Only functions of the system calculate more efficient toolpaths than canned cycles. Auto Clearance is activated in the Document dialog and designates that the system dynamically calculate clearance positioning moves for the part. The Material Only option is located in Process dialogs and designates that toolpath calculation for an individual process take into consideration the material conditions of the part to provide for no "air cutting." If either of these options are being used, the Prefer Canned option found in the Process dialogs will not be available.

Using canned cycles will output shorter processed code, but the Auto Clearance and Material Only functions will produce more efficient toolpaths in general. To generate canned cycles in the posted code, turn off Auto Clearance and enter fixed X and Z clearance positions in the Document dialog, and select the Full Rough Style in the Rough Process dialog.

Touch-Off Point Information



All post data is output to the theoretical tool tip. If the tool is machining parallel to the Z-axis, the X values are output to blueprint dimensions. If the tool is machining parallel to the X-axis, the Z values are output to blueprint dimensions. So, the theoretical tool tip only aligns with blueprint dimensions on faces and diameters.

When the tool is machining at an angle, the X and Z-axis values will not match the blueprint dimensions. This is because the theoretical tool tip is not always a blueprint dimension. So, in order for the system to get the surface of the tool in position to cut the part, the theoretical tool tip is calculated closer to the part, and in some cases inside the part.

Printing the Toolpath

After an operation has been created, the resulting toolpath can be printed. There is an option to print black and white, full color or color on a white background. When the desired toolpath is on the screen, choose Drawing from the Print sub-menu in the File menu. To change the printing style go to the Display tab in the Preferences. The Printing Preferences specifies how the system will handle the background color and contrast of lines.

Cut Part Rendering

Cut Part Rendering is the process of running a visual inspection of the operations you've created. Rendering steps through each operation, displaying the movement made by each tool as it cuts the part. Once you have one or more operations you can render the part for a visual inspection. This can be very helpful in catching any errors in the toolpath. Rendering is accessed by clicking the Sim button in the Command Toolbar. For more information on CPR see the section on Rendering in the *Common Reference* guide.

Post Processing

Once the operations to machine the part have been created, the file needs to be post processed. Post processing converts a part file (VNC file) which contains the machining operations (toolpaths) into a text file (NC program) that can be transferred to the machine control. Post Processors specific to individual machine controls are used to convert the VNC file into a text file. For general information on Post Processing see the section on "Post processing" in the *Getting Started* guide.

Lathe Post Label Definitions and Code Issues

Lathe post names use letters to signify their capabilities. The designation may be a single letter or multiple letters to specify the post's capability. Following the letter designation is a unique number for this post.

The general format of a post can be described as:

```
<control name><machine name>[client initials]<letter>###.##
```

Note that a metric post will end with an “m”.

Following is a description of how Lathe Posts are named and what they do. Also included are brief explanations of code issues that might be encountered in Lathe Posts.

2-Axis Lathe

Label Definitions

`L` This designates a regular 2-axis turning post. A Lathe post has 2 linear axes (X and Z) that can position and cut simultaneously.

Example: Fanuc 16T [VG] L800.18.pst

Code Issues

- Tool Tip

a. The system draws the toolpath to the center of the tool tip radius. The X and Z-axis values are output to the theoretical tool tip if the system is able to calculate a touch-off point in both axes. X or Z-axis values are output to the center of the tool tip radius when the software is not able to calculate the touch-off point in that particular axis.

b. If the tool is machining parallel to the Z-axis, the X values are output to blueprint dimensions. If the tool is machining parallel to the X-axis, the Z values are output to blueprint dimensions. So, the theoretical tool tip only aligns with blueprint dimensions on faces and diameters.

c. When the tool is machining at an angle, the X and Z-axis values will not match the blueprint dimensions. This is because the theoretical tool tip is not always a blueprint dimension. So, in order for the system to get the surface of the tool in position to cut the part, the theoretical tool tip is calculated closer to the part, and in some cases inside the part.

d. Most Lathe Posts output X and Z values to the theoretical tool tip. Posts can be modified to output X and Z values to the center of the tool tip radius.

- Canned Cycles

a. Lathe canned cycles are output when the Prefer Canned checkbox is checked. This checkbox will only be available if Auto Clearance and Material only are not selected. If Auto Clearance and/or Material Only are selected, the system will not output Canned Cycles.

3-Axis and 4-Axis Mill/Turn

A Mill/Turn post supports both milling and turning operations in the same part. A 2-axis lathe post is no longer needed if a Mill/Turn post is available.

Label Definitions

`ML` This designates a Mill/Turn post.

`S` This designates a Mill/Turn post that segments rotary arcs into linear moves.

Example: Fanuc 16T [VG] SML800.19.pst

`I` This designates a Mill/Turn post that supports Polar and Cylindrical Interpolation. A Polar and Cylindrical Interpolation Mill/Turn post will output a G2 or G3 with rotary moves.

Example: Fanuc 16T [VG] IML800.19.pst

`Y` This designation is for a 4-axis Mill/Turn machine that has a linear Y-axis.

Example:
Fanuc 16T [VG] YIML800.19.pst
Fanuc 16T [VG] YSML800.19.pst

P This designates a C-axis positioning post. A Mill/Turn positioning post will rotate the part and then move in X and Z. It will not rotate and cut simultaneously.

Example: Fanuc 16T [VG] PML800.19.pst

N This designates a Mill/Turn post that does not use subprograms. This is known as a "Long Hand post". Subprograms are frequently used for multi-process drilling, C-repeat drilling, Z-repeat milling, C-repeat milling, Patterns (OD only), and so forth.

Example: Fanuc 16T [VG] NSML800.19.pst

Fanuc 16T [VG] NIML800.19.pst

B This designates a B-axis rotation post. This supports the creation of coordinate systems that has the tool rotate about the B-axis.

Example: Super Hicell 250 HS [JMC] BSML1082.19.7.pst

Code Issues

Tool Orientation

- When using a mill tool on the Face or OD, it is important to define the orientation of that tool correctly. When Milling or Drilling on the face, make sure the orientation of the tool is perpendicular to the face. When Milling or Drilling on the OD, make sure the orientation of the tool is perpendicular the OD. If the tool is not oriented properly, the output will not be correct.

C-Axis And Y-Axis Output

- In the Rotate tab, the option buttons Position and Polar & Cylindrical Milling determine whether C-axis moves or Y-axis moves are output during Polar & Cylindrical Milling operations. If the Position radio button is selected, the system calculates Y-axis moves. If the Polar & Cylindrical Milling option button is selected, the system calculates C-axis moves.
- If your machine does not have a Y-axis, then you need to select the Polar & Cylindrical Milling option button.
- If your machine has a Y-axis, this capability can be added to any Mill/Turn post.

Rotary Feedrates

a. Most rotary feedrates are calculated in Degrees Per Minute per rotary segment based on its length. Since the length of each segment is variable, the system outputs a different feedrate for each segment. The resulting rotary feedrate can be a large value based on the Degrees Per Minute calculation.

b. Certain CNCs, such as Haas and Mazak, calculate rotary feedrates using Inverse Time. Any Mill/Turn post can be modified to use Inverse Time for feedrates.

c. Polar Interpolation posts use inches per minute for rotary feedrate calculations. Any Mill/Turn post can be modified to use Polar Interpolation with inches per minute feedrates.

Communications

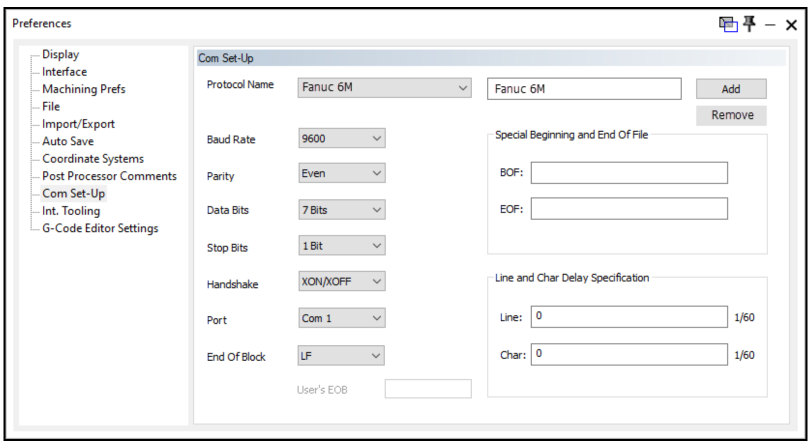
The system contains integrated communications. Third party communications packages can also be used to communicate with CNC machines. Before data can be sent to the CNC machine, the communication parameters need to be set up. To access the Com Set-Up tab choose File > Preferences. This dialog is used to set up communication protocols needed for sending a file to a control or receiving a file from the control. Different controls have different protocols (parameters). Refer to the machine control manual for the necessary protocol specifications.

For detailed information about Communication, see the *Getting Started* guide.

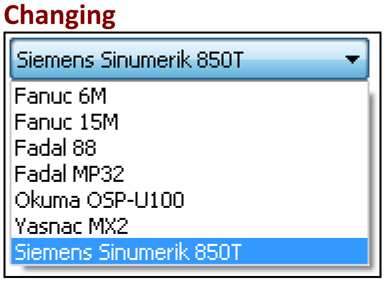
Protocols

Adding

To add a new protocol, type a new name and change the settings for the machine. `Click` the Add button. The name will appear in the list.



Changing



To change a protocol, select it from the protocol list and modify the information. The changes are automatically saved.

Removing

To remove a protocol, select the protocol from the list and `click` the Remove button.

Conventions

GibbsCAM documentation uses two special fonts to represent screen text and `keystrokes or mouse actions`. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

Text

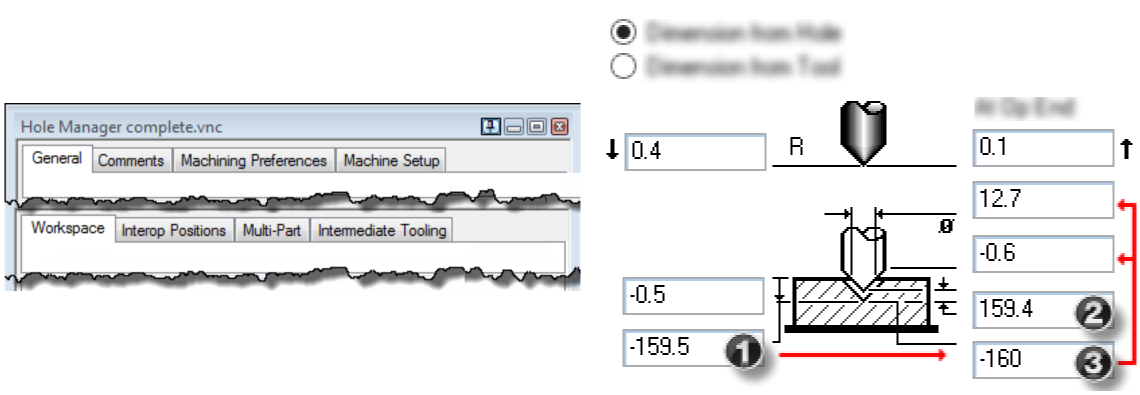
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

`Code`. Text with `this appearance` indicates computer code, such as lines in a macro or a block of G-code.

Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A “torn” edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:



Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

Links to Online Resources

Please contact your reseller for support.

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens Gibbs Online page to download GibbsCAM and all supported material.