GIBBSCAM® 14

Version 14 : September 2020

Mill

CAMBRIO

Proprietary Notice

Contents

Introduction to Mill

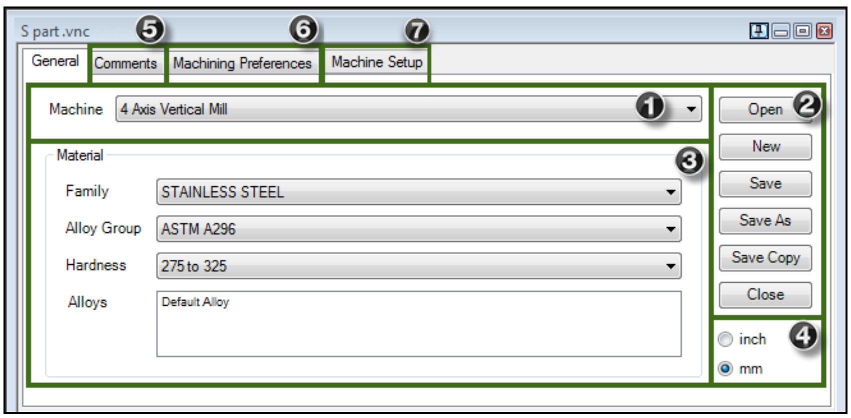
This guide is intended for users of a basic 3-axis mill; however, the lessons learned also apply to more advanced 4-axis and 5-axis milling.

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 Mill 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.

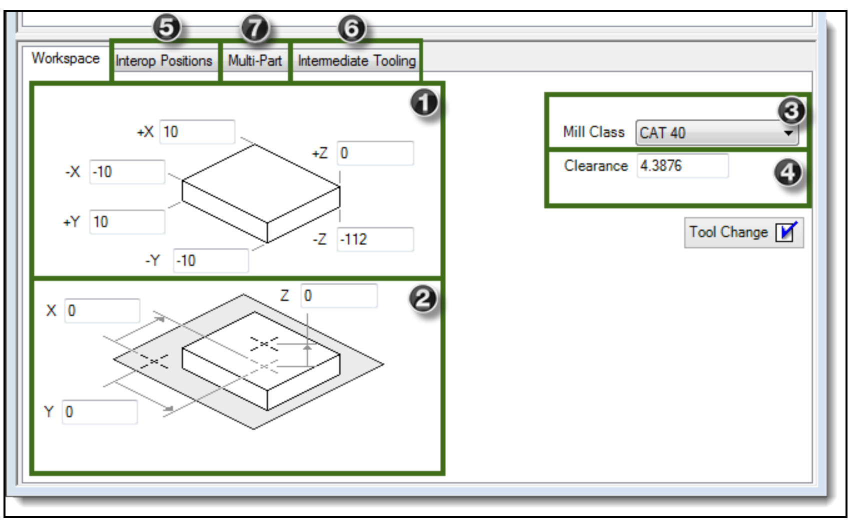
Part Setup - DCD

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. Machining preferences
7. Machine setup

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



1. Workspace stock size
2. Machine part origin
3. Tool holders, current and available
4. Master Z clearance plane
5. Interop settings
6. Intermediate Tooling
7. Multi-Part

Bottom portion of the Document Control dialog. For complete information, see “DCD Tabs: Mill” on page 10.

DCD Tabs: Mill

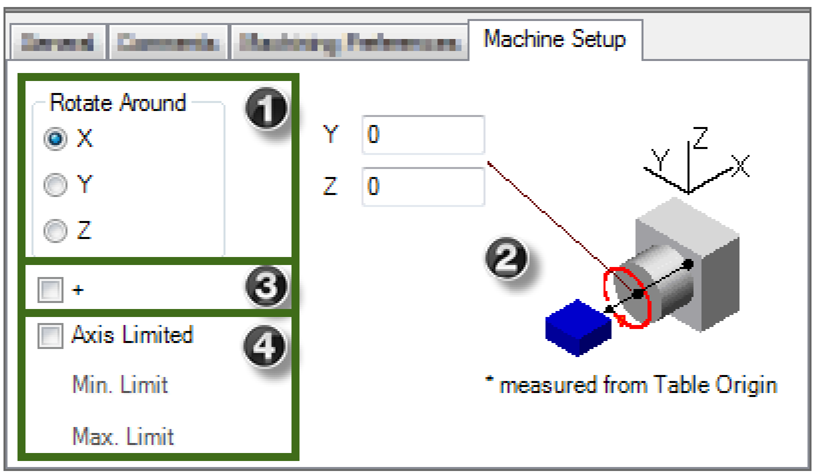
Top half of DCD tab

General tab, Comments Tab and the Machining Preferences Tab are covered in detail in the *Getting Started* guide, under Setting up a part.

Machine Setup Tab

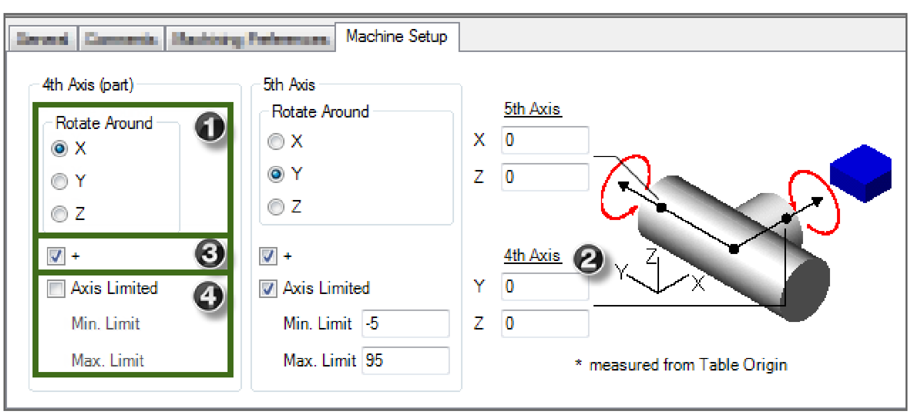
This tab is displayed only for multi-axis machines with generic MDDs – generic 4-axis and 5-axis mills and B-axis mill/turn machines. The tab's contents vary by machine type.

Machine Setup for 4-Axis Generic Mill



1. Rotate Around: { X | Y | Z }
2. Location of Rotary axis
3. Rotation direction ([-] or[+])
4. If axis limited: minimum and maximum

Machine Setup for 5-Axis Generic Mill



1. Rotate Around: { X | Y | Z}
2. Location of Rotary axis
3. Rotation direction ([-] |or [+])
4. If axis limited: minimum and maximum

(for separate 4th and 5th axis setup)

Bottom half of DCD Tab

Workspace tab

Stock and Part-Specific Settings

Stock Size and Part Origin

These values are used to define the size of the workspace stock, or the default stock size. Any positive or negative value is valid, but the +X, +Y and +Z values must always be greater than the -X -Y -Z values.

Stock dimensions are respected when generating toolpath with the Material Only option selected in the process dialog. If custom stock has been created (*custom stock* is stock that is based either on a specified solid body or on workgroup geometry designated Part Stock), then the system will use the custom stock size for toolpath and positioning moves. In that case, the values entered in the DCD will be used only to draw the stock outline and origin marker correctly.

Within GibbsCAM, the part origin is always "`X0 Y0 Z0`" in part space.

Part Offset

Why are these values displayed? The parameters here are for the majority of part programmers, who prefer to set a part origin at some convenient location, usually different from the table origin. (*Table origin* is also called *part station origin*. For simple mills, the table origin is also the machine origin.). For example, many programmers prefer to set the part origin at the top of the stock so that +Z values are above the part and -Z values inside the part.

Who can ignore these parameters?

- If your programming style is always to center the bottom of the part exactly at the table origin, then the part offset values are always `0 0 0` and you can ignore this section of the Workspace tab.
- Or, if you program only on generic 3-axis mills and never use Machine Simulation, then you can ignore these values. However, best practice is to set them correctly in case future programmers might need them on other machines, or for Machine Sim.

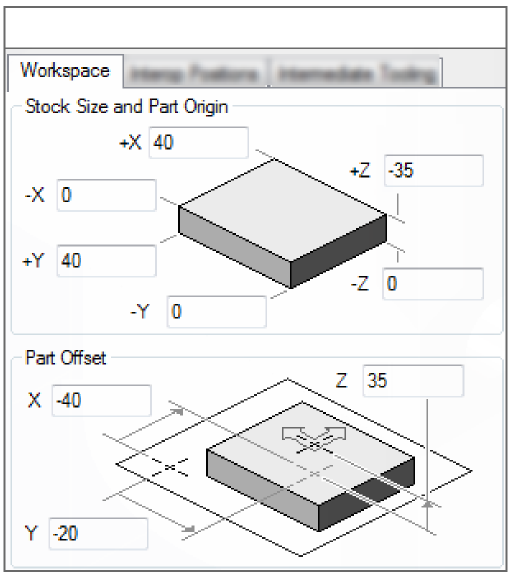
*Who must **not** ignore these parameters?*

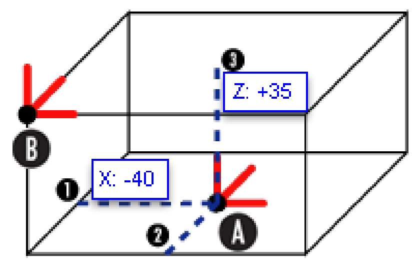
- If you use Machine Simulation, then you must provide Part Offset values for the simulation to be accurate. In releases before v11.0, this was accomplished using either the Set Part Origin plug-in or the **Set Part Origin** dialog box summoned by the Setup button on the **Render Control** palette.
- If your DCD references a generic 4-axis or 5-axis MDD, then the **Machine Setup** tab (explained in "Top half of DCD tab " on page 10) specifies the location of the rotary axis or axes for this machine, measured from the table origin. If nonzero Y or Z values are specified for the fourth axis (and/or if nonzero X or Z values are specified for the fifth axis), then the values for Part Offset are taken relative to the values set up for the machine as a whole.

For 4- and 5-axis mills, why is the reference value tucked away in Machine Setup tab, and why did it change at v11.0?

- Values for Machine Setup are set once per machine. Therefore, its tab in the DCD should be accessed only rarely after initial setup. This allows parts to be more portable from one machine to another.
- Values for Part Offset might vary from part to part on the same machine. Therefore, the **Workspace** tab displays Part Offset values whenever the DCD is re-opened, for immediate at-a-glance reference.

The relationship between Table Origin and Part Origin is illustrated below.





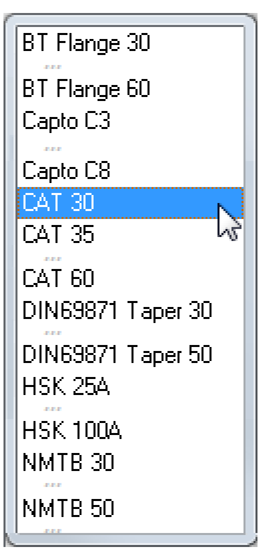
(A) Table Origin at center bottom of this stock.

(B) Part Origin:

1. Negative X Value
2. Negative Y Value
3. Positive Z Value

Part Offset example 1 (generic 3-axis mill): Part Offset places part origin (X0 Y0 Z0) left of, closer than, and above table origin.

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.

Clearance Plane

This position is used as a master clearance plane for the part. This is the Z position the tool will rapid to and from during a tool change. In addition, the tool will retract to this position between holes for drilling operations (if the second Retract to Z option is selected in the Drilling Process dialog). The Clearance Plane is also used for multiple parts in the posted output. For more information on clearance positioning, refer to “ Machining” on page 135 and “Post Processing” on page 169.

Clearance (Δ)

When an MDD specifies Clearance Volume, the DCD for a Mill part presents Clearance (Δ) as an incremental offset from the default stock definition. This allows users of advanced machines to specify that the tool should stay at least Δ away from the part except when cutting. In the textbox, enter the size of the "bubble" to be maintained around the part, within tolerances set within the MDD (usually ±10%).

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.

The clearest example of where Clearance Volume is beneficial is Willemin 508MT and 508MT2 machines where vice and tool can be rotated independently, making it impossible to provide legacy MDD settings for interop moves that are logical and reasonable. Any machine where tool stations and part stations are independently rotatable can be a candidate for Clearance Volume.

Clearance Volume can also be useful for simple machines where more efficient clearances are desirable for interop moves when the tool retracts to accommodate rotation, especially when machining a tall part. This occurs on 5-axis table machines where the fourth axis is distant from the part, and on B-axis mill-turns where the tool goes home between B-axis orientation changes. In cases like these, if you can keep the tool near the part, you often get faster run times.

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.

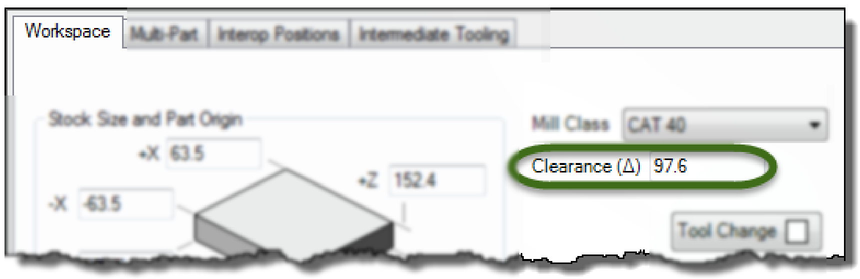
Caveats: Interop moves generated by Clearance Volume contain 5-axis simultaneous moves; thus it is best if the control has TCP capabilities, and it is unsuitable for machines that have indexing rotary axes or rotary axes that must be clamped between moves.

User Interface

In the MDD, Clearance Volume should be implemented by Resellers and/or the Gibbs Post Department. We do not expect end users to exercise Clearance Volume options in the MDD.

When the MDD implements Clearance Volume, a new command is available: Show Clearance Volumes. You can find this command in the Customization dialog and customize the user interface by placing it on a toolbar or menu group.

DCD. When an MDD specifies Clearance Volume, the DCD for a Mill part changes: instead of Clearance for a plane positioned above the part origin, it has Clearance (Δ) as an incremental offset from the default stock definition.



Machine Space and Part Space

Machine space means “absolute; from the standpoint of the machine”; part space is relative to the part, which may be moving with respect to the machine.

Example. When a vinyl record is played on a turntable, consider the path of the needle.

- From the standpoint of the machine, it makes a nearly straight-line traversal from the outside of the disc to the inside.
- From the standpoint of the record, the needle traverses a very tight inward spiral, with occasional small breaks. This follows the spiral tracks in the vinyl.

G-Code

All machines output G-code in machine space; some machines also have a mode that enables part space instead of machine space. Machine Space requires accurate offsets (i.e., tool and part and rotary positions in the MDD), and may be unsafe when inaccurate offsets are entered. Part Space is more forgiving. But: Note that “Turning Enabled” causes Part Space to be ignored.

In most circumstances, the superior output takes advantage of the machine’s interpolation capabilities to create smooth lines and arcs, instead of creating many tiny segments that approximate a curve.

Multi-Part Tab

Depending on the machine, GibbsCAM provides Part Duplication and TMS options on the Multi-Part tab. TMS is covered in detail in the *TMS* guide.A part that previously used the Multi-Part Mode of TMS is automatically converted to Multi-Part when it is opened in this release. However, to take advantage of Multi-Part post improvements requires a post upgrade. (Without such an upgrade, old posts will continue to work, but will use longhand.) To request a post upgrade, contact your Reseller or the Gibbs Post Department.

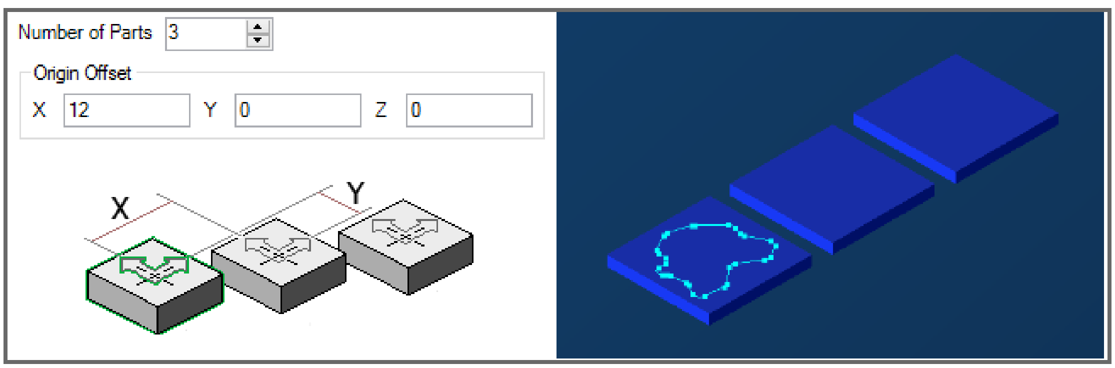
Part duplication

1. In order to view part duplication, you need to select Op Sim Rendering.

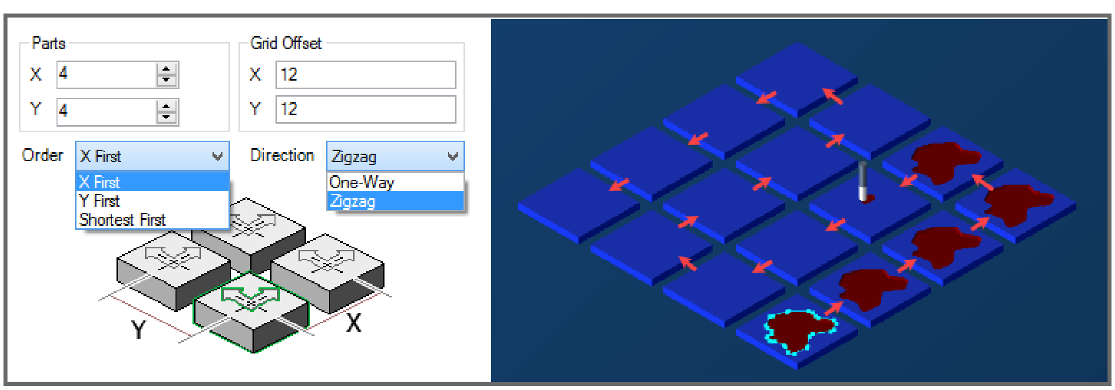
   You can leave the Rendering palette open as you make your selections, rewind and play the simulation as required.

2. You are now ready to specify the duplication pattern - Equally spaced, Grid or Defined.

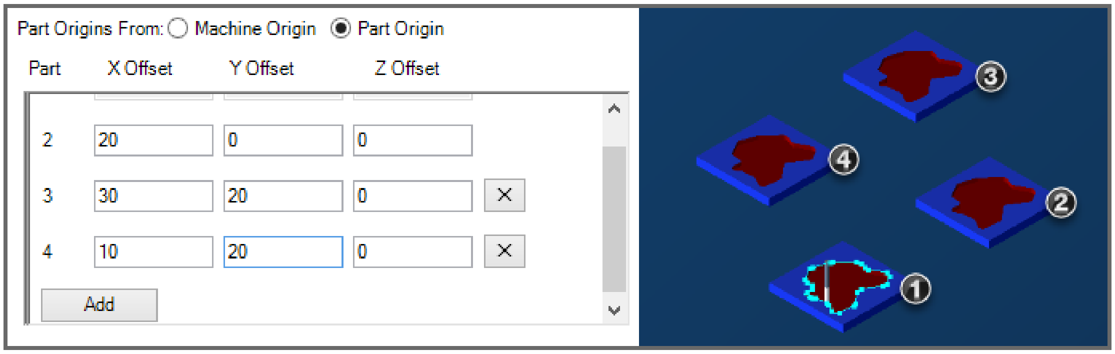
- Equally spaced will duplicate in one line specified by the Origin Offset.



- Grid will duplicate in a grid pattern. You need to specify the number of parts in each direction and then the grid offset. In the example below there will be 4 parts in each direction with an offset of 12 in X and Y. We wish to start in the X direction and follow a zigzag pattern for the operations (as demonstrated by the arrows).



- Define Positions allows you to specify your own grid positions. The first part will be at 0,0 either from the Machine or the Part Origin, depending on which radio button you choose. Subsequent parts can then be added at specific grid locations. Continue adding part locations as required.



If Complete Each Part First is checked all operations will be completed on a part before moving onto the next. If unchecked, operations using the same tool will be completed on each part in turn, then the machine will return to the origin and begin the next operation.

If Back and Forth is checked, the next operation will start at the last part cut, instead of returning to the origin.

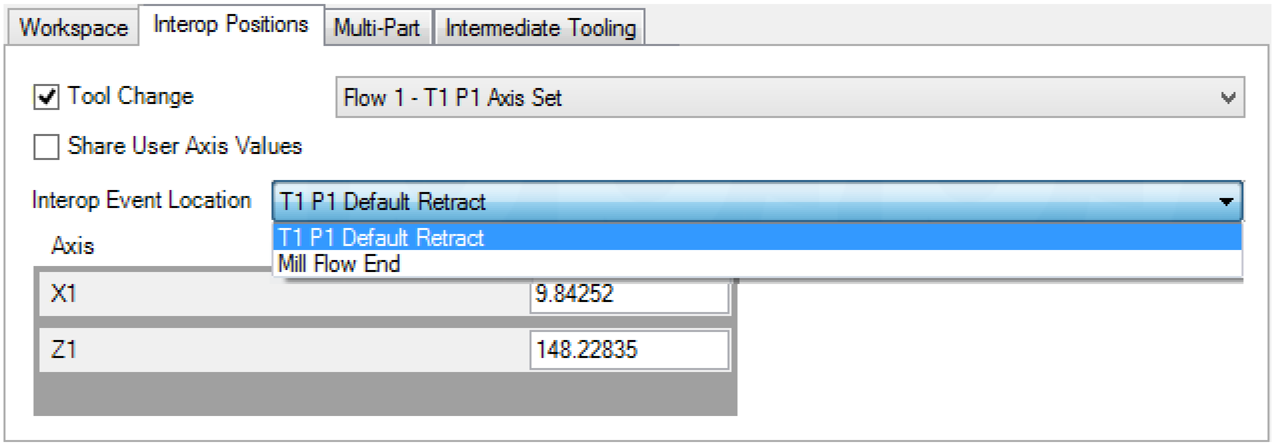
Interop Positions Tab

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 potentially lists all FAS's that contain user axes and whether to share user axis values.

The Tool Change checkbox specifies whether to set tool change positions manually (if the box is selected, further controls are displayed that allow you to specify a position for each FAS) or to allow the system to manage them.

> **Please Note:** Settings in the MDD govern many of the controls displayed in the **Interop Positions** tab. For example, if the checkbox Force Share User Axis Values selected in the Machining Prefs page of MDD's root note, then the DCD's Interop Positions page will not offer the Share User Axis Values checkbox.

When displayed, the Share User Axis Values checkbox is selected by default, so that values are shared across all axes in every FAS. If you clear this checkbox, then a pull-down menu appears that allows you to set user axis values for each interop event location in the FAS. The illustration below shows how you could adjust the default retract values along the X1 and Z1 axes.



Intermediate Tooling Tab

This part of the DCD provides access to the toolblocks and fixtures appropriate to this part setup.

Other Settings

If the MDD has multiple workpiece stations, a dropdown on the Workspace tab allows selection of a workpiece station, and a checkbox and value allow control of the Graphic Part Face Distance. If the MDD has only one workpiece, these elements will be absent.

If the MDD has multiple toolgroups, a dropdown on this tab will allow selection of a toolgroup. If the MDD has only one toolgroup, the dropdown will be hidden, but the toolgroup data will still be shown. A dropdown will be shown with the toolgroup’s Mill Holder Class; it will be read-only if Lock Mill Backend is turned on for the toolgroup. If the toolgroup can access a workpiece station that is “Turning Enabled”, a Shank Size dropdown will be shown; it will be read-only if the MDD is not generic.

The remainder of the tab is configured to collect setup data for a particular type of workpiece station; a given MDD may have multiple workpiece stations of the same type, or of different types.

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.

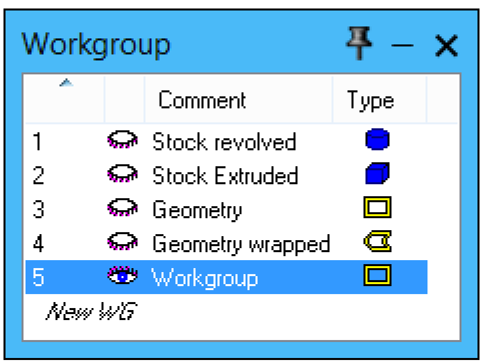
Please note: When deleting any item in the Materials Database (Family, Alloy Group or Material), great care must be taken as the undo function is not available.

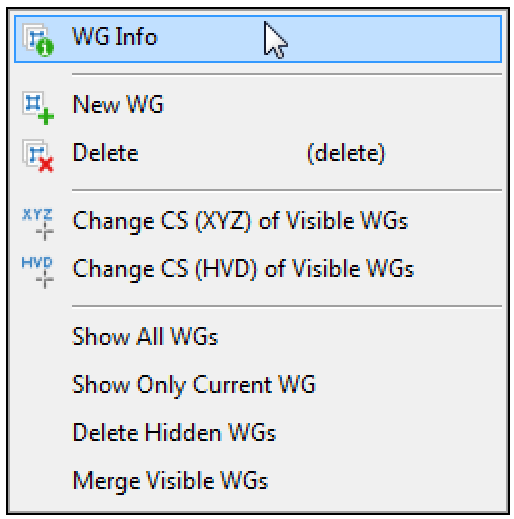
Custom Stock

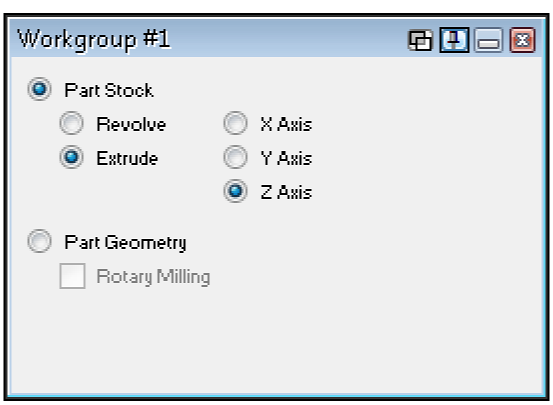
For simple stock – rectangular or cylindrical, with or without a hole – you can use the Stock Wizard plug-in.

Please Note: The Stock Wizard was not designed to work with spinning Part Stations.

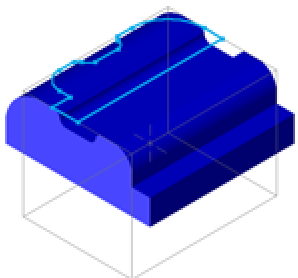
For more complicated stock, the Workgroup context menu `WG Info` choice allows a custom stock shape definition from geometry. This dialog is accessed by right-clicking a workgroup's name in the workgroup list.



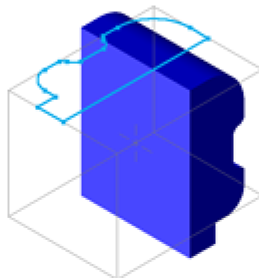




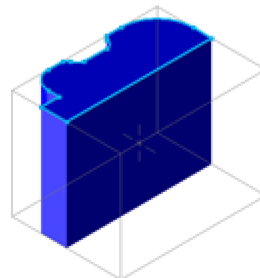
The shape can be extruded or revolved along the X, Y or Z axis. Extrusion geometry can be in any orientation. Extrusions will extend to workspace stock boundary of the axis. The stock shape may be a concave or convex shape with one hole. The hole may be a blind hole or a through hole. Revolved stock should lie along the axis of revolution and must not cross the axis.



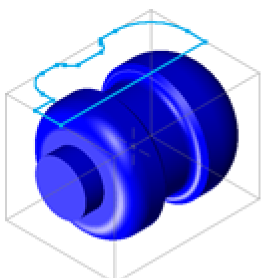
Extrude X Axis



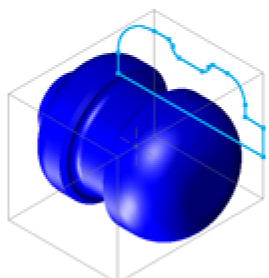
Extrude Y Axis



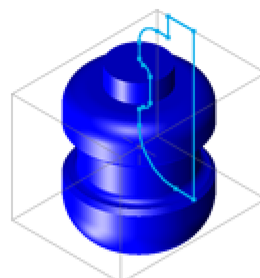
Extrude Z Axis



Revolve X Axis



Revolve Y Axis

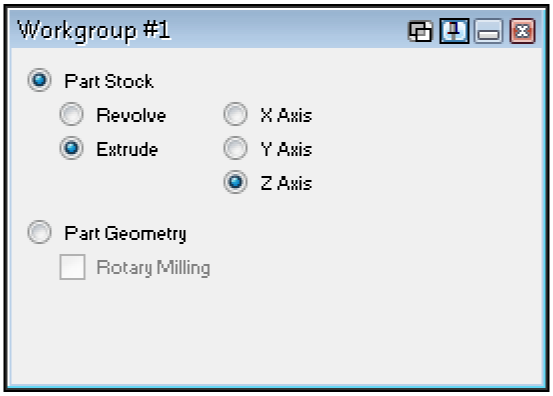


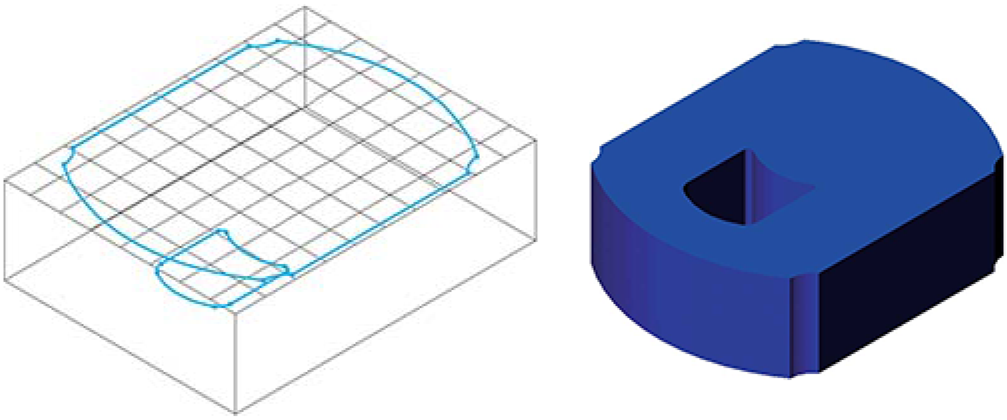
Revolve Z Axis

Custom Stock With A Hole

Custom stock can also have a single hole in it. The hole can be of any closed shape and may be a through hole or a blind hole. For the system to create a custom stock shape with a hole, the stock must be extruded along the Z axis. The system uses the Z values in the Document Control dialog to determine the depth of the stock.

To create a hole in the stock, simply create the hole shape in the stock workgroup. The Z value of the hole shape designates the floor of the hole. If the Z value of the hole is coincident with the bottom of the stock or is deeper than the bottom of the stock, then the system will create a through hole. The closed shape must be contained within the stock shape. If the hole goes beyond the X or Y bounds of the stock shape, the system will not create a hole but will still create the stock shape.





Example of Custom Stock with a blind hole

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 *Getting Started* guide's section on "Tools".

The following material describes tools used specifically for milling.

Defining Tools

You can define tools for machining processes in the following ways:

- Directly from the Tool List
- Using ITA (ISCAR Tool Advisor)

To define a tool directly from the Tool List:

1. In the Tool List, double-click an empty tool tile.

   The Milling Tool definition dialog appears.

2. Type or select the options you want to use and close the dialog.

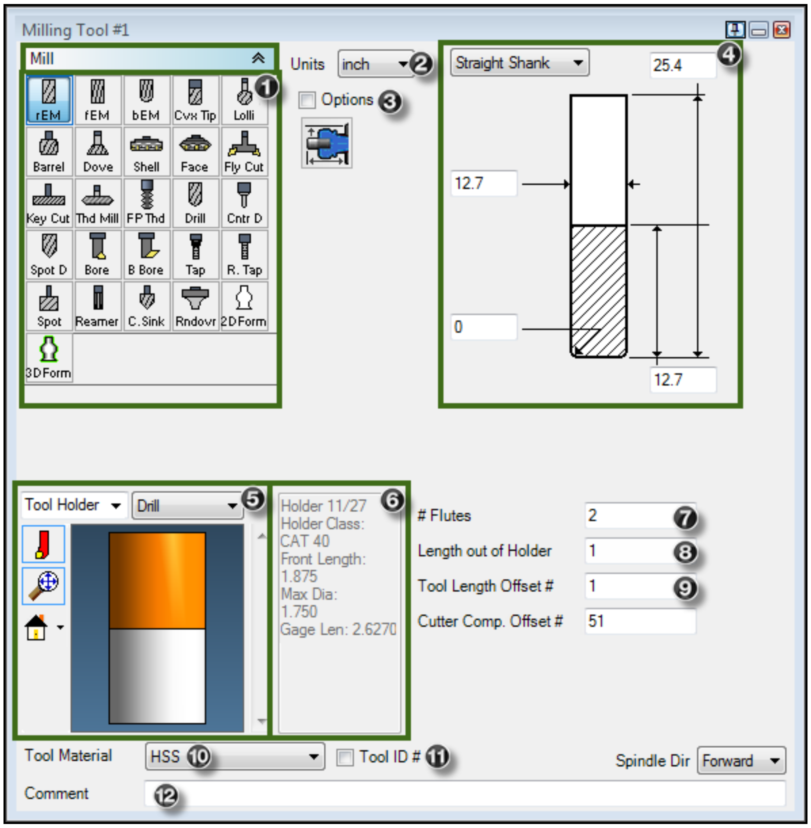
For general information on creating, saving and manipulating the tool lists and reports, see the section on "Tool Creation" in the *Getting Started* guide.

ISCAR Tool Advisor (ITA)

You can use the ITA ITA (Iscar Tool Advisor) pull-down choice on the title bar of the **Select Tool Type** flyout dialog.

Milling Tool Dialog

The Tool dialog defines the specific type, shape and material of a tool as well as how the machine uses and stores data for that tool.



1. Tool Type
2. Unit (Inch/MM)
3. Options:
4. Tool Diagram
5. Tool Holder Definition
6. Tool Holder
7. Length out of Holder
8. Tool Length Offset #
9. Cutter Compensation Offset #
10. Tool Material
11. Tool ID #
12. Tool Comment

Components of the Tool dialog

Tool Type

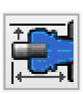
The tool type changes the tool diagram to define various tool shapes. For a description of the parameters for each type, see "Tool Type" on page 24.

Unit (Inch/MM)

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.

Options:

Select this checkbox to create more complex tools. You can enter additional specifications, which are not normally required for the standard tool types, in the Options tool diagram. The Options setting is available only for certain tool types. For more information, see "Tool Options" on page 34.



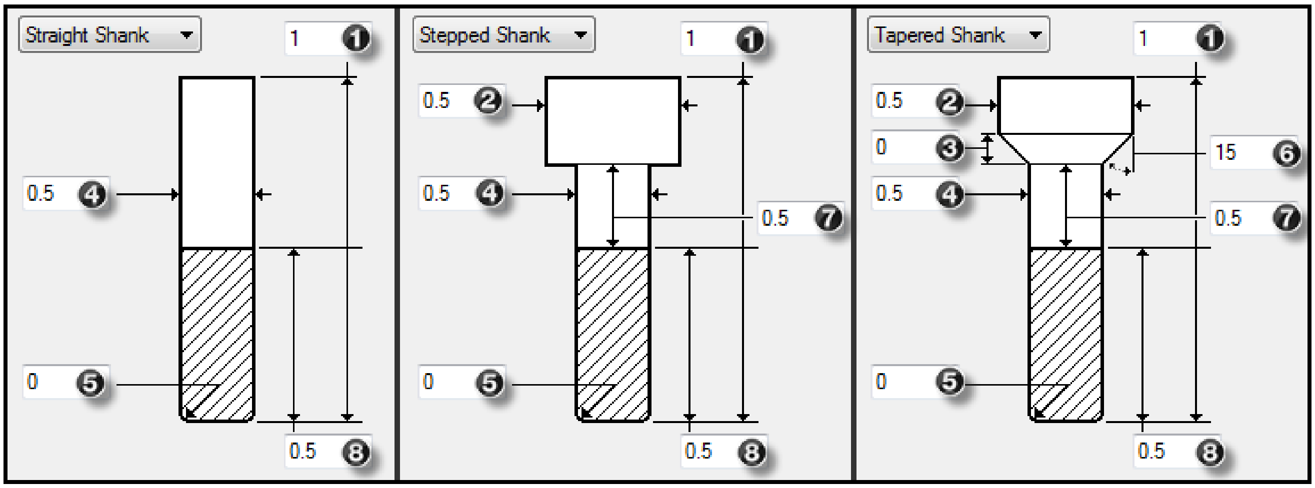
Offsets

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 Mill Tool Offset Data section.

Tool Diagram

The diagram and tool specifications depend on the tool type you select. The shaded sections of a tool diagram illustrate the cutting surfaces of a tool while the white areas are non-cutting surfaces of the tool. If these surfaces come in contact with a part, the system draws this contact area in red during rendering to show interference. The tool types are divided into general groups based on the similarities of the tool diagrams and specifications, see the diagram above.

The dropdown in the Tool Diagram section of the dialog, enables definition of the shank type of the tool. Options are Straight, Stepped and Tapered. Below are the options available for a Rough End Mill tool.



1. Overall Tool Length
2. Tool Shank Diameter
3. Tool Shank Taper Length
4. Tool shank neck diameter
5. Bottom Corner Radius
6. Tool Shank Taper Angle
7. Tool Shank Neck Length
8. Taper/Flute Length

Tool Holder Definition

You can define the front end tool holder assigned to a tool as one of the following: Tool Holder, Custom, or None.

Tool Holder

You can select pre-defined industry standard holders, based on the Tool Holder Class set in the Document Control dialog as well as the tool size. For more information, see Tool Holder Definition and Mill Class.

Custom

You can define your own tool holder. For more information, see Custom.

None

Select this option if you do not want to display the holder when rendering the part.

Length out of Holder

When using a pre-defined tool holder, the distance from the tool tip to the face of the holder must be set. This parameter allows the overall tool length to be the actual length of the tool. The Length out of Holder value must be less than or equal to the overall tool length; if it is greater than the tool length, a gap appears between the tool and holder.

Tool Length Offset
This number designates the numeric location in the machine where the Z offset amount is entered.

Cutter Compensation Offset
This number designates the numeric location in the machine where the XY offset amount for Cutter Radius Compensation is found. For more information on CRC, see Cutter Radius Compensation (CRC).

Tool Material
This menu specifies the material of the tool. The information selected here can be used by the Material Database to determine speeds and feeds. The default material for Mill tools is High Speed Steel. For more information on Tool Material, see "Materials" section in the *Common Reference* guide.

Tool ID
This number indicates the control the location of a tool in a gang or slide. This is used to override the existing tool number. It refers to a carousel location or “POT” number. 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 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

For descriptions of the tool types you can select, see

Milling Tools,

Drilling Tools,

Advanced Tools, and

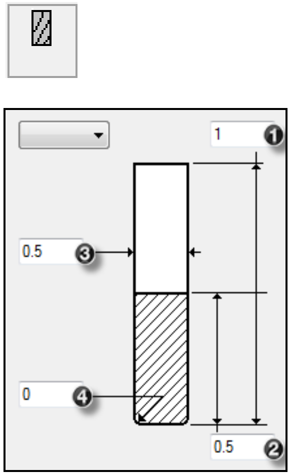
2D Form Tool.

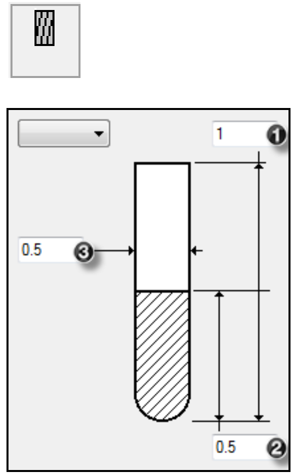
For detailed information about tool specifications and options, see Tool Specs and Tool Options.

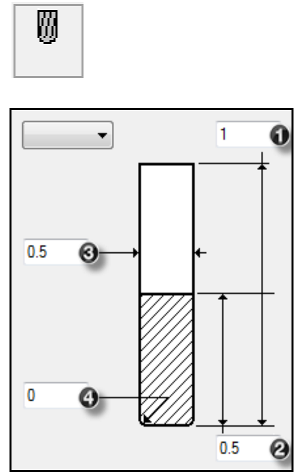
Milling Tools

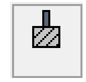
Milling Tools - Type 1

Tools in this category have a single diameter and include Rough and Finish Endmills, Ball Endmills and Spot tools. Ball Endmills do not have a bottom corner radius specification. There is no setting for Bullnose endmills, but Rough and Finish Endmills may be given a bottom corner radius to create a Bullnosed tool. For descriptions of "Overall Tool Length", "Flute Length", "Cutting Diameter", and “Bottom Corner Radius”, see Tool Specs.





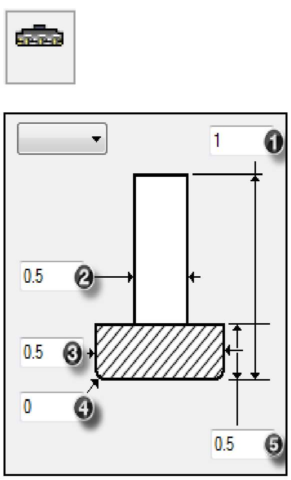




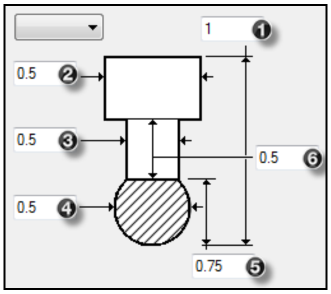
1. Overall Tool Length
2. Flute Length
3. Cutting Diameter
4. Bottom Corner Radius

Milling Tools - Type 2

Tools in this category have a cutting diameter that is greater than the shank. Included in this category are Shell, Face, Fly cutters, Key cutters, and Thread tools. These tools share “Overall Tool Length”, “Shank Diameter”, “Cutting Diameter” and “Flute Length” dimensions. A Keyway Cutter has a “Top Corner Radius” and “Bottom Corner Radius” specification. The Thread mill has a “Tip Angle” instead of a bottom corner radius. For descriptions of these attributes, see Tool Specs.



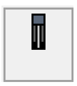
Also in this category are Lollipop tools. Lollipop tool specifications include “Overall Tool Length”, “Shank Diameter”, “Length of Shank Diameter”, “Bottom Shank Diameter”, “Lollipop Diameter”, and “Clearance Length”.

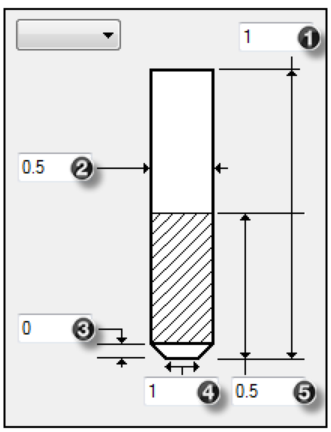


1. Overall Tool Length
2. Tool Shank Diameter
3. Tool Shank Neck Diameter
4. Lollipop Diameter
5. Taper/Flute length
6. Tool Shank Neck Length

Milling Tools - Type 3

The only tool in this category is the Reamer. Reamer specifications include “Overall Tool Length”, “Cutting Diameter”, and “Non-Cutting Tip Height”. For descriptions of these attributes, see Tool Specs.



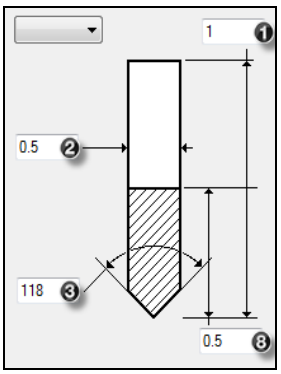


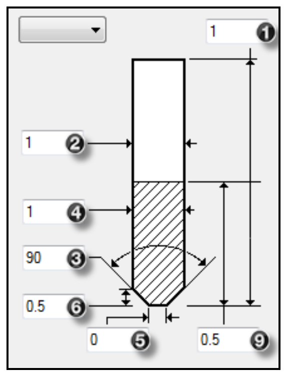
1. Overall Tool Length
2. Tool Shank Neck Diameter
3. Non-Cutting Tip Height
4. Non-Cutting Diameter
5. Taper/Flute Length

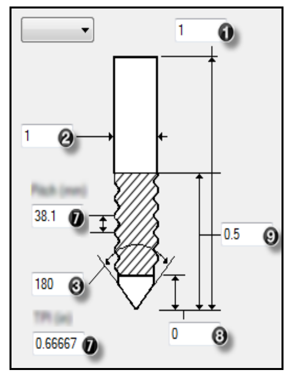
Drilling Tools

Drilling Tools - Type 1

Drills in this category are effectively straight tools that is to say that the system sees the shank as the same size as the cutting diameter. The tools in this category include Drills, Spot Drills, Counter Sinks Taps and Rigid Taps. These tools share the “Overall Tool Length”, “Cutting Diameter” and “Tip Angle” specifications. Countersinks have a “Flat Tip Diameter” and “Chamfer Height” dimension which are interactive with the diameter and tip angle specified. You only need to specify the tip angle and any two of the three specifications for the cutting diameter, flat tip diameter and chamfer height. The third value is automatically calculated. Tapping tools have a “Non-Cutting Tip Height” and a “Pitch” (for metric parts) or “TPI” (Threads Per Inch) specification. The TPI is not shown in the diagram but is entered in a separate text entry box. Descriptions of these attributes can be found in Tool Specs.



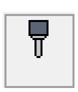


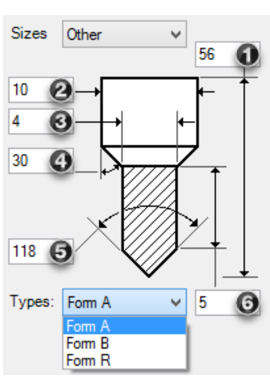


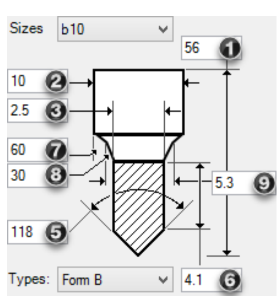
1. Overall Tool Length
2. Tool Shank Neck Diameter
3. Tip Angle
4. Main Tool Diameter
5. Flat Tip Diameter
6. Chamfer Height
7. Pitch (metric parts) or TPI
8. Non-Cutting Tip Height
9. Taper/Flute Length

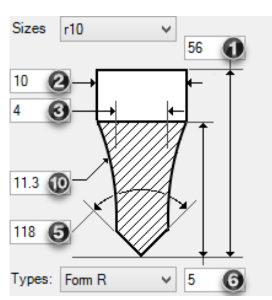
Drilling Tools - Type 2

The only tool in this category is the Center Drill. This item includes a menu of standard tool sizes for both metric and inch parts. Selecting an entry from the menu automatically fills in the specifications for the dimensions of that tool. Any value may be manually changed if it is not exactly the tool you have. A Center Drill's specifications include "Overall Tool Length", "Shank Diameter", "Cutting Diameter" and "Draft Angle", "Tip Angle" and a "Pilot Length". Descriptions of these attributes can be found in Tool Specs. Please note that the Center Drill's Pilot Length does not include the length of the tool's tip or what is sometimes referred to as the Flute length.





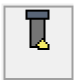


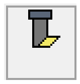


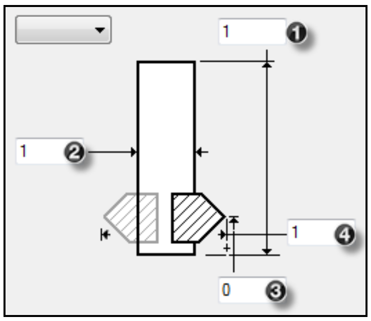
1. Overall Tool Length
2. Tool Shank Diameter
3. Main Tool Diameter
4. Draft Angle
5. Tip Angle
6. Pilot Length

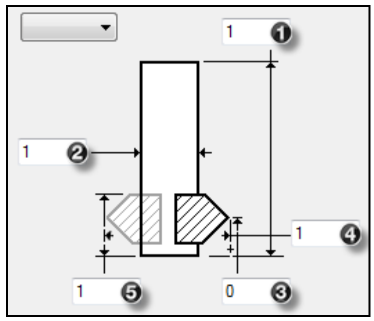
Drilling Tools - Type 3

This category consists of boring tools, i.e. standard Bores and Back Bores. These tools share “Overall Tool Length”, “Cutting Diameter” and “Non-Cutting Tip Length” dimensions. Back Bores have “Shank Diameter” and “Cutting Tip Length” values which are not needed by standard Bores. Descriptions of these attributes can be found in Tool Specs.





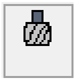


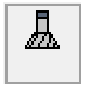


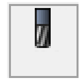
1. Overall Tool Length
2. Tool Shank Diameter
3. Non-Cutting Tip Height
4. Main Tool Diameter
5. Taper/Flute Length

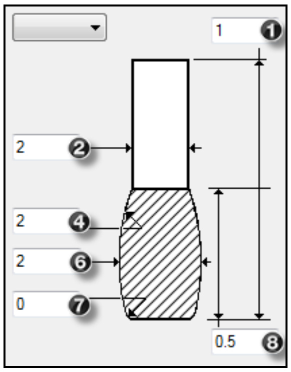
Bore and Back Bore tools use a theoretical insert tip corner as the touch off Z which is shown in the tool diagram. This part of the tool will go to the Z position entered in the drill process dialog (or Hole Wizard) for the hole depth. This is also the Z location of the tool tip in CPR. This position is programmed in the output G-code.

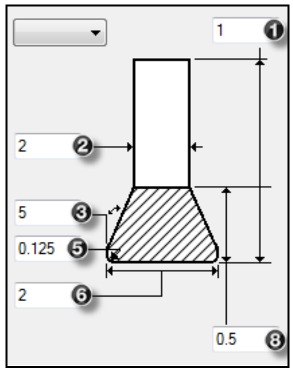
Advanced Tools

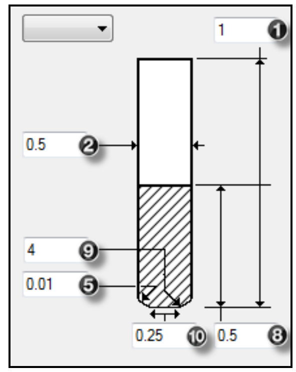




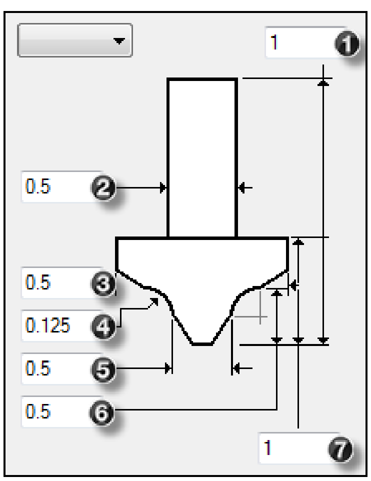






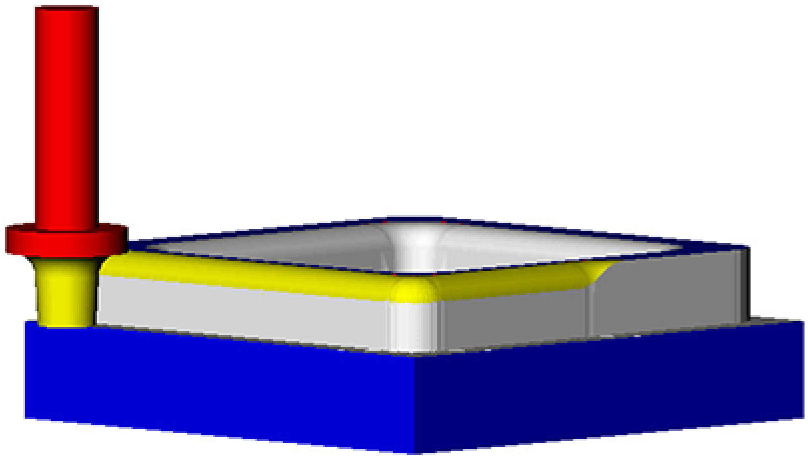


1. Overall Tool Length
2. Tool Shank neck diameter
3. Taper angle of tool
4. Profile radius of tool
5. Bottom corner radius
6. Main tool diameter
7. Bottom and top corner radii
8. Taper/flute length
9. Convex tip radius of tool
10. Flat diameter of tool



1. Overall Tool Length
2. Shank Diameter
3. Body Diameter
4. Corner Radius
5. Pilot Diameter
6. Touch-Off to Top of Radius
7. Body Length

You use Roundover tools with a Contour process to mill rounded edges. Roundover tool specifications include “Overall Tool Length”, “Shank Diameter”, “Body Diameter”, “Top Corner Radius”, “Pilot Diameter”, a “Touch-Off to Top of Radius” value and the “Body Length”. For descriptions of these attributes, see Tool Specs. The standard 3° angles off the top corner radius are a fixed value and are exaggerated in the tool setup dialog.



Example of a Roundover Contour operation.

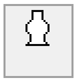
When creating a process using a Roundover tool, set the Top Surface Z value and subtract the tool’s radius from the Top Surface Z value. You should not modify the final depth, that is, the final depth should be the intended depth of the tip of the pilot. This is because the pilot diameter of the tool is used to evaluate the Z level to cut. This allows you to determine which part of the tool to offset.

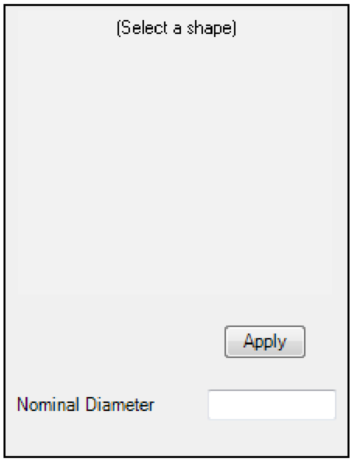
2D Form Tool

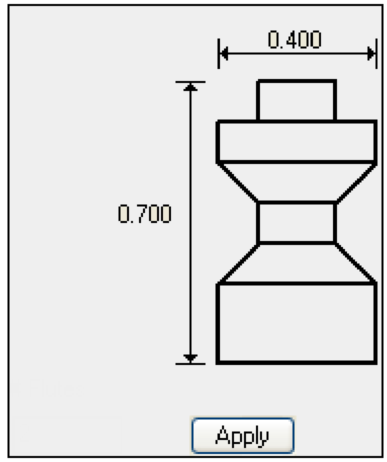
Any tool that cannot be created using the standard tools can be created with the form tool. The Form Tool can be used to create custom tools by drawing the profile around X0. The profile is revolved about X0 to determine the tool shape. For Mill Form tools, the profile must be an open, terminated shape. Only connected geometry will be used for the tool. Select any part of the profile and Apply the profile to define the Form tool.

If you want any portion of the profile geometry to be non-cutting in the 2D Form tool, right-click it and use the context menu to change it from Wall to Air.

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.





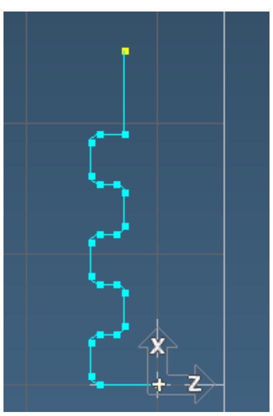


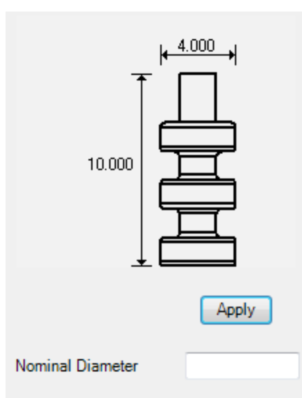
2D milling will offset the form tool from the geometry as if the geometry is at the top Z surface level and the tool is at the final cut depth, similar to the way the system offsets for tapered tools or tools with a bottom corner radius. Form tools are not compatible with 3D milling. For more information about tool offsets refer to Tool Offset. Please note that form tools may slow down cut part rendering, especially as their complexity increases.

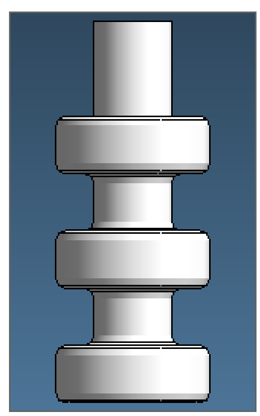
These pictures illustrate the creation path for a sample form tool. The first image is the profile geometry; the second, the form tool diagram with the example tool loaded in the Tool Creation dialog; and the third image, a rendered image of the tool. Remember that for the system to load a shape as a form tool, the shape must be a selected, open, terminated shape drawn around the vertical axis.

Nominal Diameter

If the nominal tool diameter in the form tool specification is less than the calculated maximum radius of the profile provided, the value can be entered here.







Profile Geometry, Tool Diagram and the Rendered Tool

3D Form Tool 

This is created in the same way, only using a solid revolved shape. Select the solid from the workspace or bodybag and Apply. The solid must be centered about the X axis.

Tool Specs

Generic Specs

The following specs can be found in the different tool types. The function names listed here can be found using the balloons feature in the measurement specs of the tools.

Overall Tool Length

This is the total length of the tool to be displayed during rendering. The Tool length is usually used to specify the length a tool sticks out of the tool holder, such as how far a drill sticks out of a drill chuck.

Cutting Diameter

This is the largest diameter width that a tool will cut with. Also referred to as the Main Tool Diameter.

Flutes

This is the number of flutes or cutting edges in the tool.

Bottom Corner Radius

For tools that have a rounded edge on the bottom this should be less than the Main Tool Diameter and greater than or equal to zero.

Flute Length

This is the size of the cutting part of the tool.

Shank Diameter

This is the diameter of the non-cutting part of the top of the tool.

Shank Taper Length / Shank Taper Angle

For tapered shanks, specify either the length of the taper or the taper angle.

Shank Neck Diameter / Shank Neck Length

For tapered shanks, specify either the diameter or the length of the shank neck.

Non-Cutting Tip Length

This for reaming tools that have a bottom that does not cut.

Cutting Tip Length

This is the length of the cutting tip for a Back Bore tools.

Top Corner Radius
For tools that have a rounded edge on the top this should be less than the Main Tool Diameter and greater than or equal to zero.

Taper Length
This is the length of the tapered part of the tool. Same value as the Flute Length and is usually used in Counter Sink tools or option tool definitions.

Tip Angle
This is the angle of the tip of the cutting edge of the tool for drilling and threading tools.

Tip Diameter
For countersink tools this is the diameter of the tip of the tool.

Non-Cutting Tip Height
Commonly referred to as “lead in”. This is the height of an extra non-cutting surface of a tool measured from the bottom of the tool. If a tool has a non-cutting surface, be sure to give the tool clearance at the floor of a pocket. This is used to accurately render the cut part, ensuring that the tool does not contact the stock.

Length of Shank Diameter
This is the height of the top of the non-cutting section of the tool.

Bottom Shank Diameter
This is the width of the bottom of the non-cutting part of the tool.

Lollipop Diameter
This is the width of the cutting section of the tool.

Clearance Length
This is the height of the non-cutting section of the tool.

Drill and Bore Type Specs

Tip Angle
For Drilling tools this is the angle of the bottom tip.

Flat Tip Diameter
This value is the size of a flat tip on counter sink tools. A value of “0” will create a tool with a sharp tip. This value is interactive with the diameter and Chamfer Height.

Chamfer Height
This is the overall height of the chamfer on a counter sink tool. This value in interactive and will modify the tool diameter or flat tip diameter, depending on which last had a value entered.

Sizes
This is a list of standard tool sizes.

Draft Angle
For tools with a built in chamfers such as Center Drills this is the draft angle of the tool.

TPI
For parts created in inches this is the Threads Per Inch ratio.

Pitch
For parts created in metric this is the distance from one thread tip to the next.

of Teeth
For Full Profile Threadmills only: This is the number of teeth in the threadmill profile.

Style
For Full Profile Threadmills only: This is the thread standard to use for this threadmill:

- UN: Unified screw thread, ASME/ANSI B1.1
- UNJ: Unified screw thread, ASME/ANSI B1.15
- ISO: International Standard (metric)
- NPT: National Pipe Thread Taper
- Whitworth 55°: Also called British Standard Whitworth (BSW)
- BSP: British standard pipe thread

Taper
For Full Profile Threadmills only: This is the standard or user-specified taper angle.

Non-Cutting Tip Height
This is the height of the tool's cutting surface from the bottom of the tool for Back Bore.

Cutting Tip Length
This is the height of the tool's cutting surface from the bottom of the boring bar. This is used for accurate cut part rendering, ensuring that the tool does not contact the stock.

Roundover Tool Specs

Body Diameter
This is the overall width of the tool.

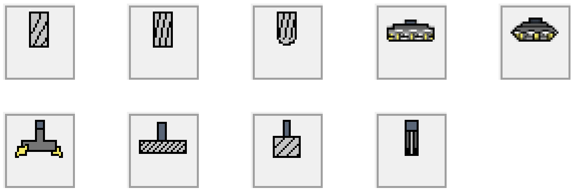
Top Corner Radius
This is the radius of the round left by the tool.

Pilot Diameter
This is the smaller tip diameter below the Top Corner Radius and the smallest space the tool can fit to round two parallel edges.

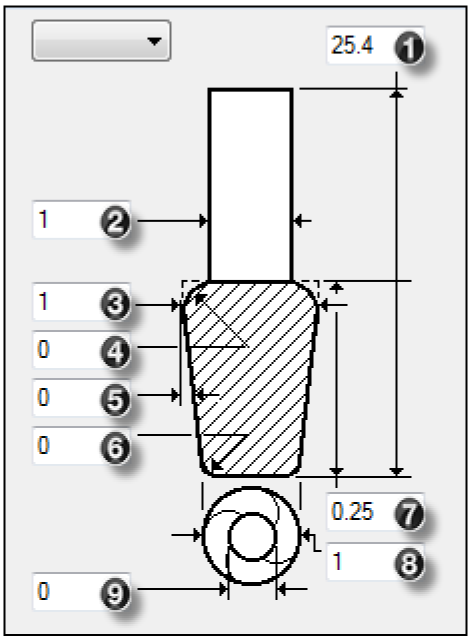
Touch-Off to Top of Radius
This value is the length of the tool from its tip to the top of the tool radius. This is the cutting area of the tool.

Body Length
This is the length of the cutting section of the tool, the 4° taper and the wall section of the tool.

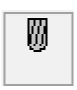
Tool Options
The tools shown can have custom definitions.



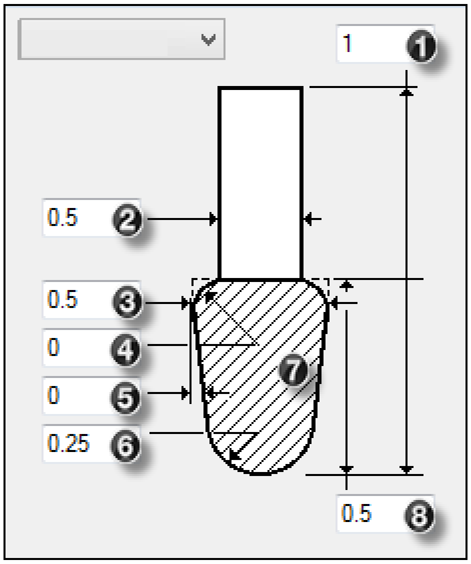
To enter additional specifications for these tools, select the Options box.



1. Tool Length
2. Shank Diameter
3. Cutting Diameter
4. Top Corner Radius
5. Draft Angle
6. Bottom Corner Radius
7. Flute Length
8. Sharp Tip Diameter of Tapered Tool
9. Hollow Tool Diameter



Ball Endmills have a slightly different tool diagram when the Options box is selected. You can define tapered ball endmills by designating a Draft Angle and Tip Radius. The Cutting Diameter, Taper Angle and Flute Length specifications are interactive. For example, if you enter a Draft Angle of 10° and change the Cutting Diameter, the system recalculates the Flute Length to maintain the specified Draft and Diameter.



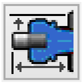
1. Tool Length
2. Shank Diameter
3. Main Diameter
4. Top Corner Radius
5. Taper Angle
6. Bottom Corner Radius
7. Flute Length
8. Sharp Tip Diameter of Tapered Tool

Sharp Tip Diameter
The sharp tip diameter is used for tools with a taper angle. Changing the Draft Angle or Cutting Diameter recalculates the Sharp Tip Diameter or the Flute Length.

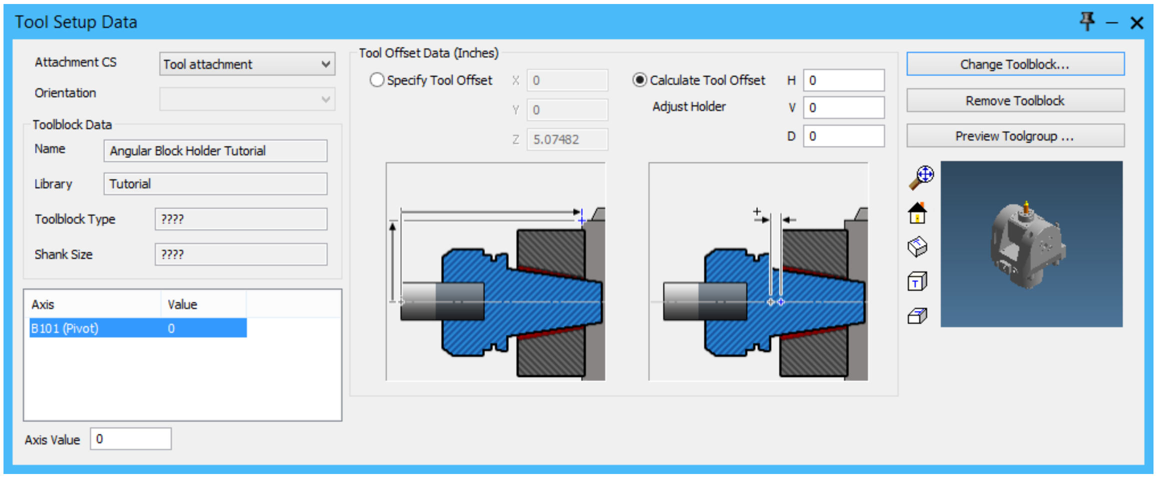
Flute Length
When you select or type a value in the Sharp Tip Diameter box, the flute length is calculated. Entering a value for the Flute Length recalculates the Sharp Tip Diameter.

Hollow Tool Diameter
The Hollow Tool Diameter specifies the center diameter of the non-cutting surface of the tool tip.

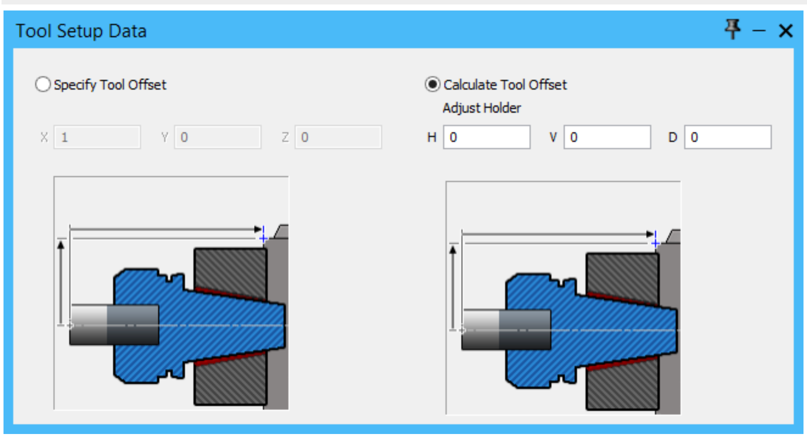
MILL TOOL OFFSET DATA


This button is where you specify offset 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.

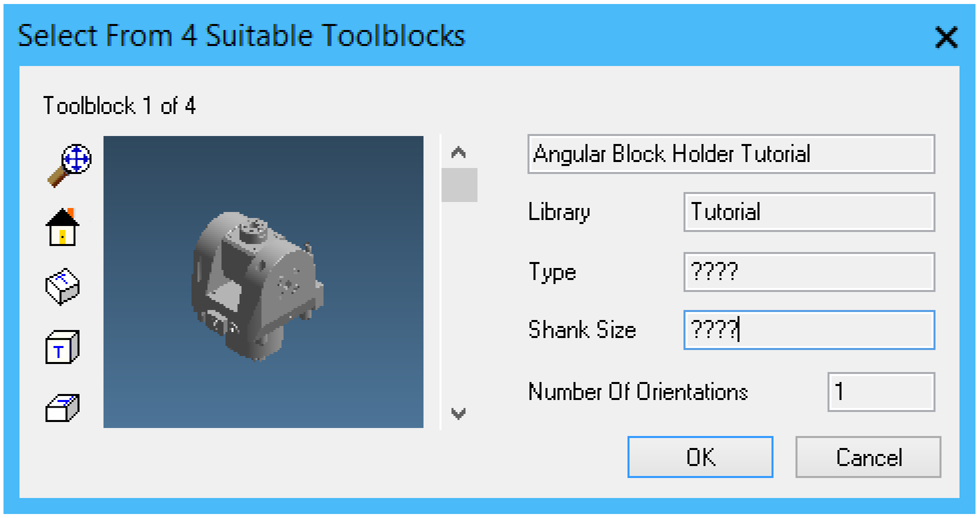
With Toolblocks



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 a suitable 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.

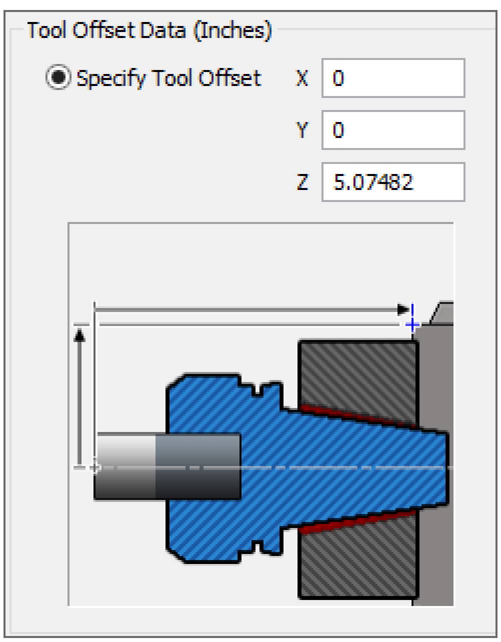
Remove Toolblock

Removes the selected toolblock.

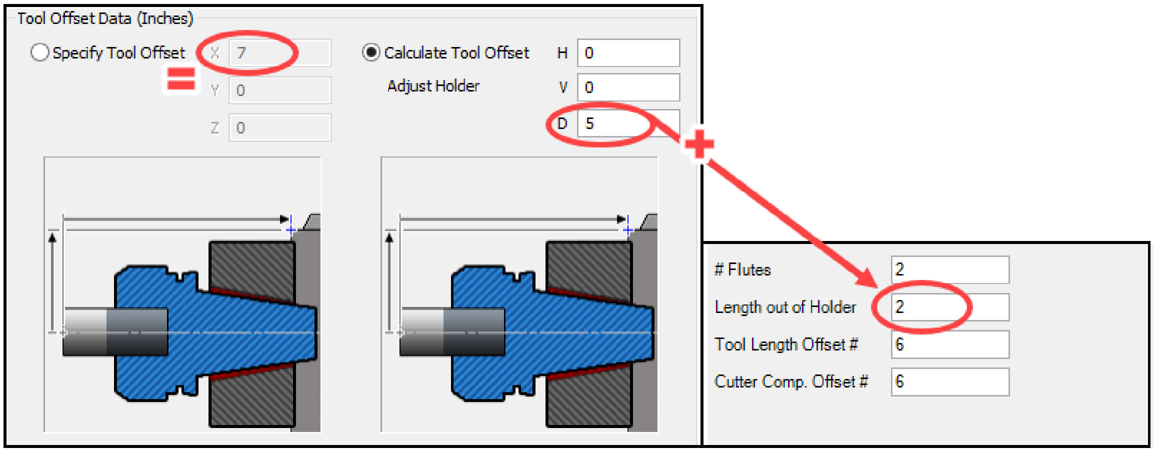
Tool Offset Data

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.



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 length out of holder for a milling tool.

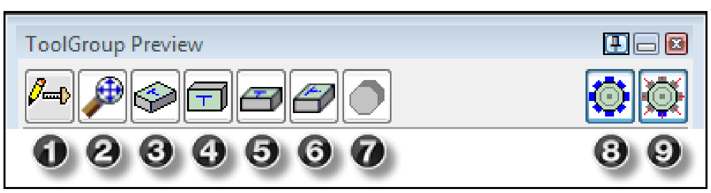


Attachment CS
If different attachment CS's are associated with the Toolblock they are displayed in a dropdown list.

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.

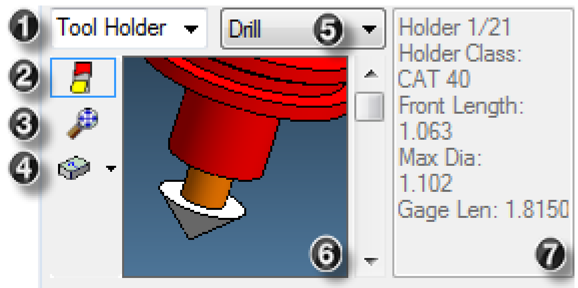
Preview ToolGroup
Checking this option opens a new window which displays an interactive view of the Toolgroup.



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 Holder Definition

Tool Holder Section



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

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

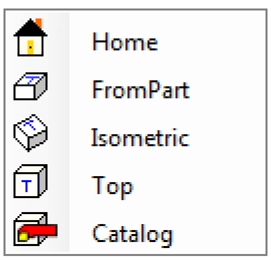
A thin blue line is drawn around the icon if holder is displayed.

Unzoom

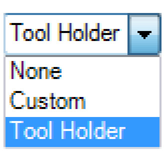
This is useful if you have expanded the tool using the mouse.

View control Dropdown

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



Tool Holder Options Dropdown



Three options are provided to specify Tool Holders. The front end tool holders display during Rendering using predefined or custom holders. Predefined holders are based on the Tool Holder Class (the back end of the holder) selected in the Document Control dialog, see “Mill Class” on page 12.

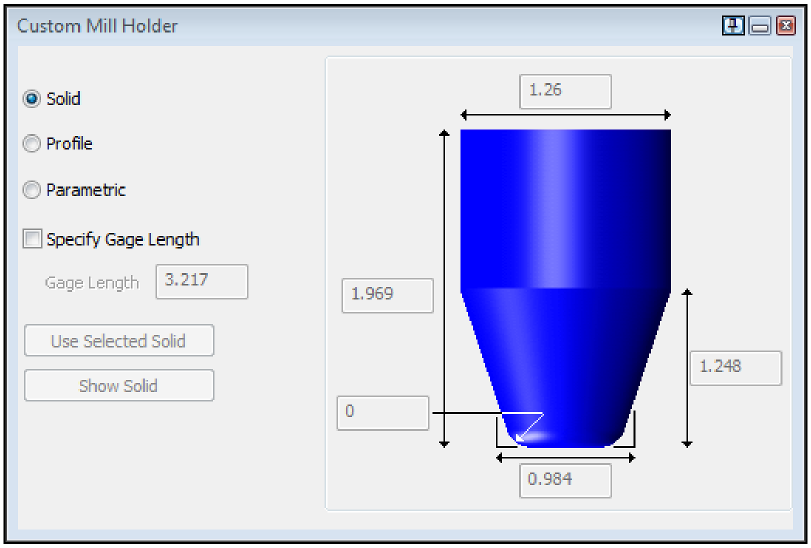
None

By default, the holder is set to None, meaning a holder is not used.

Custom

Choosing this option provides an Edit button. Click this to bring up the Custom Mill Holder dialog.

Use this option if you wish to create a custom holder shape.



You can define a holder using a geometry profile, a solid model of the holder, or numeric values (Parametric). Using a geometry profile is similar to creating a custom tool shape.

Solid

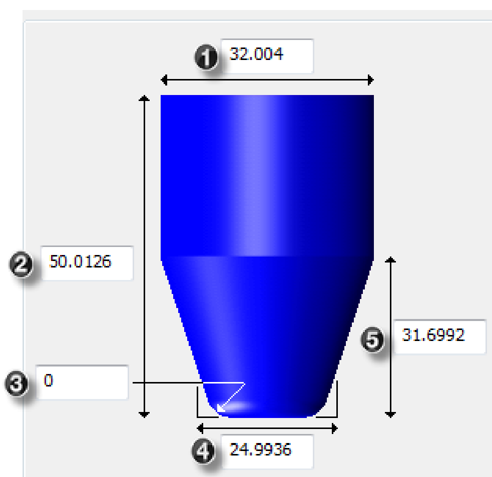
The Solid option allows you to use an existing solid to designate the tool holder. Select the solid and click Use Selected Solid. Clicking Show Solid will display the custom holder associated with the tool.

Profile

The Profile option allows you to utilize existing geometry to designate the tool holder. Select the geometry and click Use Selected Profile.

Parametric

Define a custom holder using numeric values.



1. Diameter of the holder where it meets the spindle face
2. Height of the holder from its bottom to the spindle face.
3. Bottom corner radius value , otherwise "0"
4. Diameter at the bottom of the holder (Or projected diameter, if the holder has a bottom corner radius.)
5. Height from the bottom of the holder to the top of its taper.

Specify Gage Length

Check this box and enter Gage Length if required. Available with Solid and Profile options.

Important information:

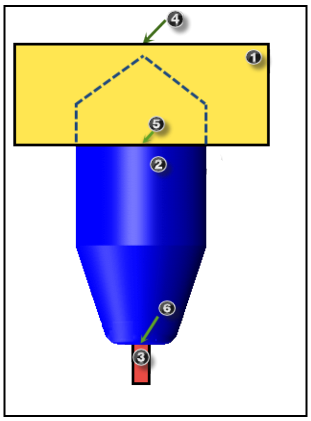
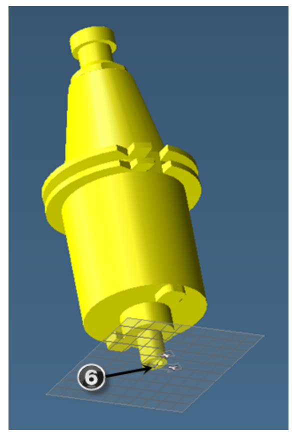
- Orientation

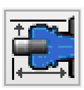
In general, custom holder orientation is based on the orientation of the holder in machine space. As GibbsCAM works primarily in part space, this is not always straightforward.

Mill custom solid tool holders are normally positioned such that the tool spin axis aligns with the first part station's Z axis, regardless of the tool's actual orientation

- **Positioning**

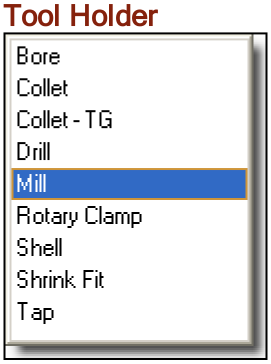
Custom holders are placed relative to the first part station's origin. For mill tools, this means that the tool attachment position (and therefore tool stickout and holder offsets) are calculated from the origin. This is behavior is different from V10.7.

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

- **Offsets** 

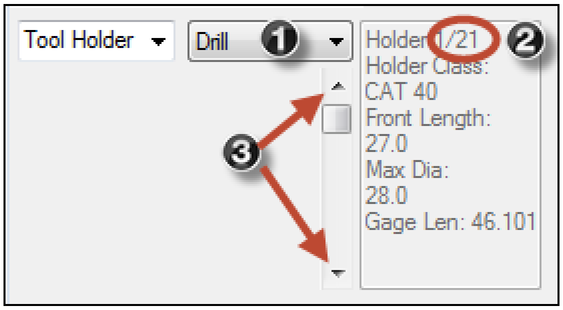
When a custom holder is applied, the system will calculate holder offsets using data from the Toolblock (if used) and the Toolholder. For more information on offsets see the Mill Tool Offset Data section.

Tool Holder



When you select Tool Holder, a dropdown menu for Pre-defined Holder types is available and a rendered image of the tool displays. You can choose from an extensive library of standard mill tool holders. The specific holders available are based on three criteria: Tool Holder Class set, Holder Type, and the size of the tool. You set Tool Holder Class in the Document Control dialog, see "Mill

Class" on page 12. You select the Holder Type from the dropdown menu. The holders are grouped by type, for example, Shrink Fit, Collet, and Rotary Clamp. If multiple holders are available, you can scroll through the preview window to switch between the available holders. The holder specs indicate how many holders are available for the current tool definition.



1. Tool Holder Type
2. Number of available holders and specifications
3. Slider - use this to scroll through options.

Basic specifications of the holder are seen to the right of the tool and holder image. The specifications shown for each are as follows:

Holder 1/(x):
Indicates how many holders are available for the tool within the holder class. Click the up or down arrow to cycle through the list to choose the holder to you want to use.

Holder Class:
Shows the selection made in the Document Control dialog.

Front Length:
Length the holder extends from the flange.

Max Diameter:
Largest diameter of the holder.

Gage Length:
Distance from the face of the spindle to the end of the holder.

Setting the Pre-Defined Tool Holder

First, you must set the tool dimensions. The available holders are based on the tool size. You select the tool, then you select the type of holder for the tool. Depending on your selection, one or more valid tool holders are available. Scroll through the list to find the holder you want to use. If no holders appear, then there are no available holders for the combination of tool size and holder type specified.

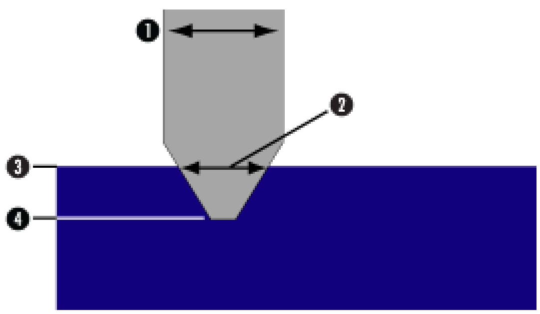
- The overall tool length set in the tool dialog defines the distance from the tool tip to the face of the tool holder.
- Note that if a tool holder is not defined, the overall length of a tool in the tool dialog is the tool's distance out of the spindle.
- Note that holders on vertical mills will need to be re-oriented to lie along the Z axis.

Tool Offset

When pocketing or contouring, the system calculates a tool offset amount based on the radius of the tool. 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.

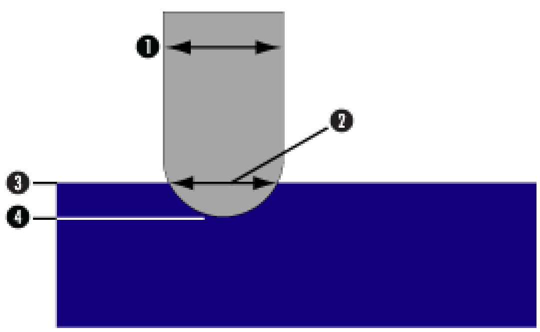
When milling with tapered or pointed tools, the system calculates the tool offset based on the Floor Z compared against the Surface Z. The Surface Z is the top surface of the material. The Floor Z specifies the finished depth of the pocket or contour. These values are entered in the Contouring and Roughing Process dialogs in the Entry/Exit Clearance Diagram.

The tool diameter used to calculate the offset amount is the diameter of the tool at the Surface Z. In order for the system to correctly calculate the tool offset when using these tools, accurate Floor Z and Surface Z positions must be entered in the Process dialogs.



1. Tool Diameter
2. Diameter used to calculate tool offset
3. Surface Z
4. Floor Z

When milling with tools (both tapered and non-tapered) that have a bottom corner radius, the system checks the bottom corner radius at the Floor Z compared against the Surface Z and adjusts the tool offset amount accordingly.



1. Tool Diameter
2. Diameter used to calculate tool offset
3. Surface Z
4. Floor Z

This offset calculation is useful when cutting pockets shallower than the corner radius on the tool. Also, chamfering a pocket is easily accomplished by entering accurate Z positions in the process dialog and entering a negative Stock value equal to the desired chamfer amount. The system will correctly calculate the tool offset from the geometry when creating the toolpath in order to correctly machine the chamfer.

Offset calculation with tapered tools is only made when Tool Center is selected for the Mill CRC Type in the DCD, Machining Preferences tab.

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.

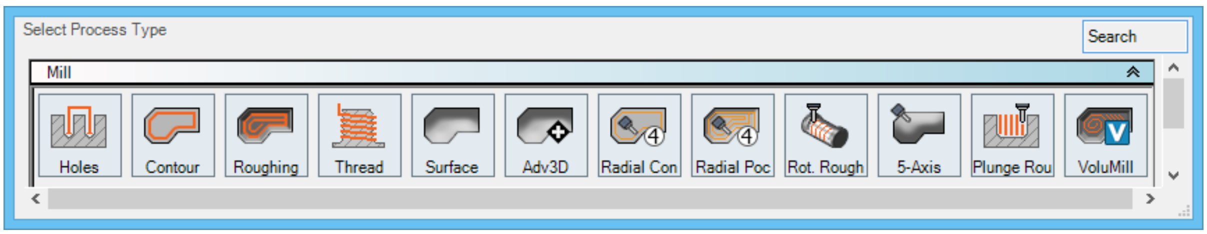
> ***WARNING:*** The system does a much better job offsetting the tool than the majority of controls currently available. Regardless of the setting made in this preference, all toolpath drawing and cut part rendering is calculated and displayed using the system's offsetting mechanism. Therefore, it is 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

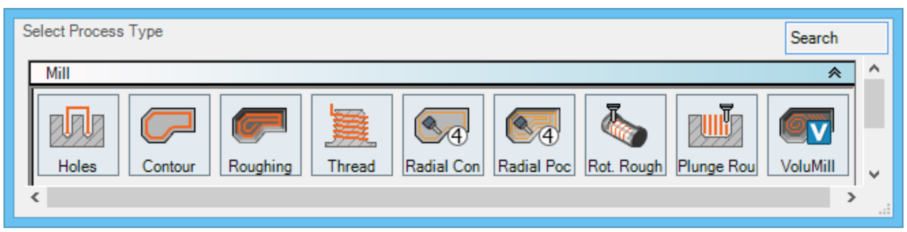
A process is a combination of a machining process and a tool. You apply a process or combination of processes to the part geometry to create an operation.
For more information on processes, see the section on "Processes" in the *Getting Started* guide.

Mill Machining palette

Each tile in the Machining palette for Mill has a specific function.



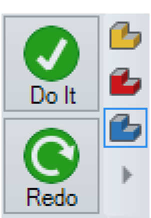
Machining palette (Level 2)



Machining palette (Level 1)

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.
See "Function Tiles Available With Additional Product Options" on page 47.

Buttons: Do It, Redo

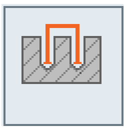
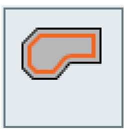
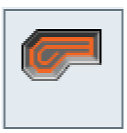
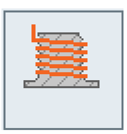


Click the Do It button to create new operations after you complete the Process tiles and select a cut shape.

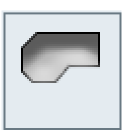
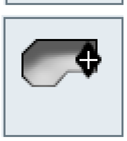
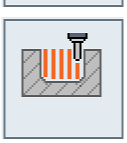
The Redo button is available when existing Operation tiles are selected. Its availability indicates that these operations can be reprocessed.

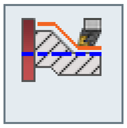
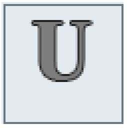
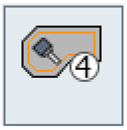
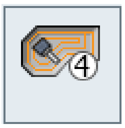
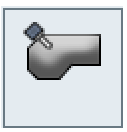
These buttons are documented fully in the *Getting Started* guide.

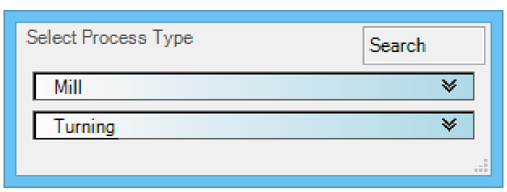
Function Tiles and Controls for Basic Milling Machines

Tile	Process
	You use the **Holes** function to drill and bore holes at defined point or circle locations. Multiple hole operations created from one Process list generate a subprogram of the selected pattern of points or circles in the posted output. You can also use the Holes function in conjunction with the Roughing and Contouring functions to drill entry holes.
	The **Contouring** function takes a single pass along a shape or engraving.
	The **Roughing** function removes material from the inside of a closed shape, or to face mill.
	The **Thread Milling** function creates ID and OD threads at defined point or circle locations.

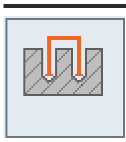
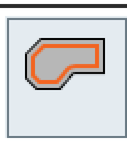
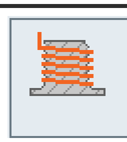
Function Tiles Available With Additional Product Options

Tile	Process
  	The **Surfacing**, **Advanced 3D Machining**, **Plunge Rough** and **Eccentric** and **Elliptical Turning** functions generate 3D toolpath on solids and sheets. For more information on these functions, as well as the Part Body, Local Fixtures, and Local Stock buttons, see the *SolidSurfacer* guide.

Tile	Process
	
	The **Utility operation** function tile is available only if the MDD and VMM support utility operations on multi-task machines. For information on standard utility ops like Move Tool Group (MTG), see the *Multi-Task Machining (MTM)* guide. For information on custom utility ops provided with custom VMMs, refer to the material supplied with your MTM package.
 	The **Radial Contouring** and **Radial Pocketing** functions generate toolpath for machines capable of performing radial milling. For more information, see the *Radial Milling (4-Axis)* guide.
	The **5-Axis** function generates toolpath for machines capable of conventional 5-axis milling (3 linear axes + 2 rotary axes). For more information, see the *5-Axis* guide.
	The **VoluMill** function generates ultra high-performance toolpath (UHPT) in place of traditional roughing methods when the emphasis is on reducing cycle times, extending tool life, and reducing the stress on machine tools. For more information, see the *VoluMill* guide.



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.

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.

- Holes Process, next
- Contour Process

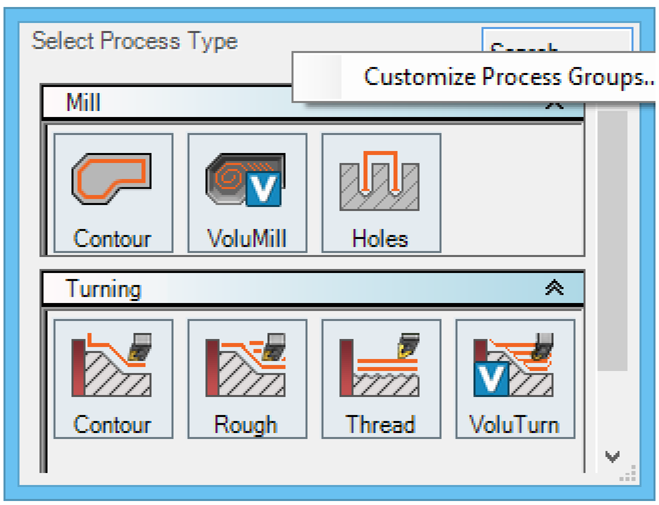
- Roughing Process
- Thread Milling Process

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.

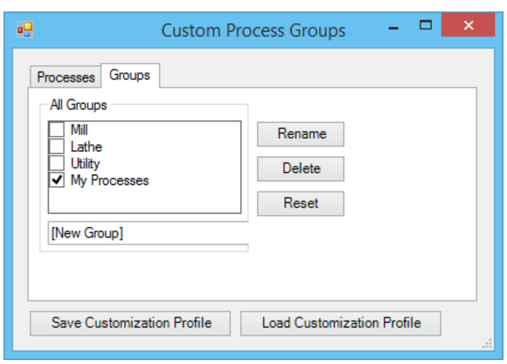
Customizing Process Groups

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.

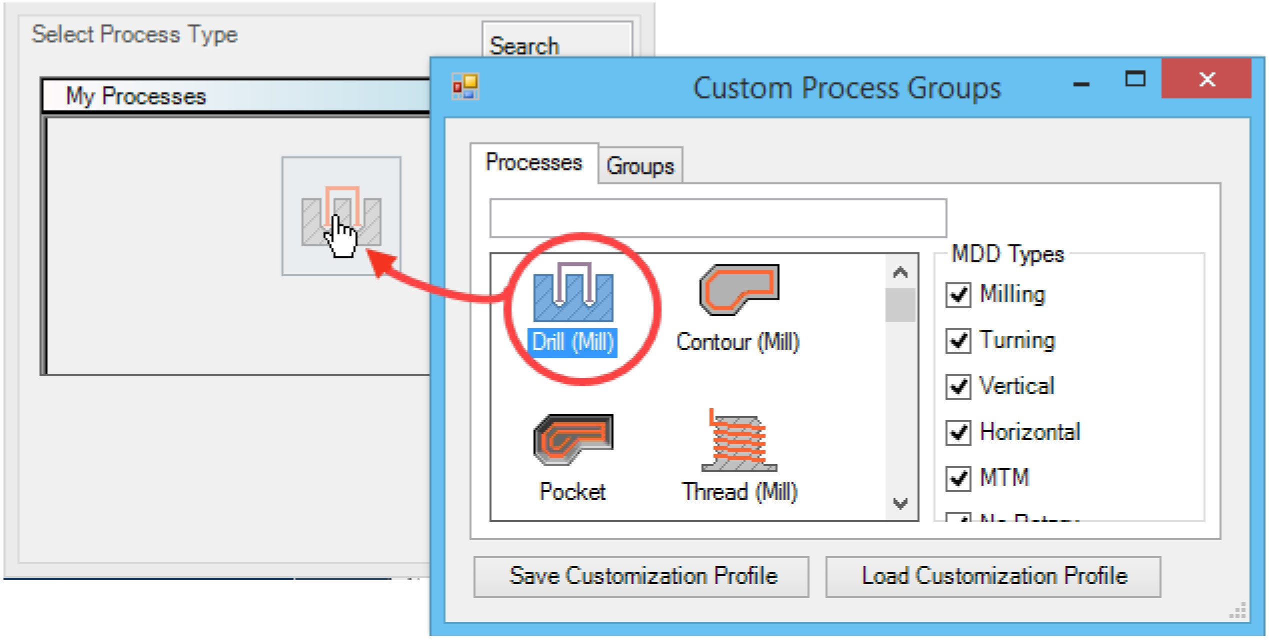
Right-click the title bar of the Select Process Type dialog and choose Customize Process Groups as shown below.



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.



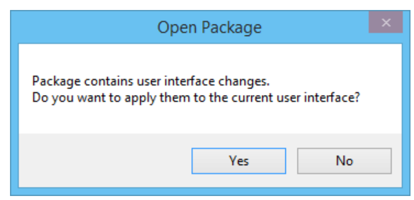
In the Processes Tab you can view available processes and MDD types. To create your own process list simply drag the required processes in or out of the Select Process Type dialog.



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.



Mill Feature Tab

The Mill Feature page offers the following types of controls:

- "Attribute-Driven Controls" below
- "Absolute-Only Controls" on page 51

Attribute-Driven Controls

The attribute-driven controls consist of five pull-down menus. Four of them (Approach Z, Retract Z, Top Surface Z, and Feature Depth Z) let you set depth values. The fifth (Mach. CS) lets you specify the machining CS.

Choices in the pull-down menus include the following:

Absolute

For Approach Z or Retract Z, Absolute specifies that the depth comes directly from the value specified in the depths diagram. (For example, for Approach Z, the value would come from the depth specified for Clearance Plane.)

For Mach. CS, Absolute specifies that the CS comes directly from the value specified in the Mach. CS pull-down menu below depths diagram.

From Attribute
Specifies that the depth comes from picking an attribute associated with the user feature. When this choice is active, another pull-down menu appears immediately below. For depth values, this lets you pick from a list of all Real-type attributes for the user feature. For Mach. CS, this lets you pick from a list of all Integer-type attributes for the user feature.

Automatic
Top Surface Z and Feature Depth Z only. Specifies that the system will retrieve the value directly from the geometry of the user feature.

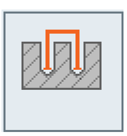
Incremental
Approach Z only. Specifies that the value comes the distance specified for the distance between the Clearance Plane and the Top Surface.

Same as Approach Z
Retract Z only. Specifies that the tool retracts to the same depth as its initial approach.

Reset All to Absolute
Clicking this button affects the settings of all attribute-driven controls – the pull-down menus on the left. Any settings that are dependent on other parameters (such as Incremental or Automatic, or derived from attributes or features), are changed to Absolute.

Absolute-Only Controls

The absolute-only controls consist of two option buttons controlling the depths diagram, the values in the depths diagram itself, and a pull-down menu of choices for Machining CS.



Holes Process

The Holes process is used to drill, tap, or counter-bore selected points, circles or hole features and it can be used to drill entry holes for other processes. When the Drilling Function tile is combined with a Tool tile, the Holes Process dialog will appear on the screen.

The six potential tabs for the Holes process are:

- Drill
- Hole Feature
- Bore
- Pre-Mill

- Mill Feature
- Rotate

The tabs for Hole Feature and Pre-Mill are never both bold at the same time, because their parameters are never operative simultaneously. The following rules govern how a tab's parameters are shown, available, and operative:

1. When a tab's name is **bold black**, its parameters are *operative*: in other words, its settings and values will be used when the toolpath is generated. For example, parameters in the **Drill** tab are always operative, but parameters in the Holes Feature tab are operative only when the Process List consists entirely of Holes processes.
2. When a tab's name is unbolded black, it is *available*, but its parameters are inoperative in the current circumstances. For example, parameters in the Pre-Mill tab are always available, but they become operative only when the Process list contains a non-Hole process. Similarly, parameters in the Mill Feature tab become operative only if **Pre-Mill** is bolded, Bore is not bolded, and a mill feature is selected.
3. When a tab's name is gray, it is *unavailable*, but it can be made available by changing a setting within the dialog. For example, the Bore tab is always shown, but it is available only when the choice for Drill > Entry/Exit Cycle is **Rough Mill Bore** or **Finish Mill Bore**. When a tab is not available, its parameters are inoperative.
4. When a tab is not shown, it can only be made available by changing a setting outside the dialog. For example, the Rotate tab is shown only when the current MDD supports rotation. When a tab is not shown, its parameters are unavailable and inoperative.

Drill tab

Entry/Exit Cycle

The selections made here determine the cycle the drill will use to make its entry and exit moves. The choices include: Feed In - Rapid Out, Feed In - Feed Out, Tap, Rigid Tap, Peck Full Out, Peck Chip Breaker, Rough Mill Bore, Finish Mill Bore, and Helix Bore.

If your site has been configured to use Custom Drill Cycles, a pull-down menu of further choices appears below the main options. For more information on Custom Drill Cycles, see the *Installation* guide and the Macros wiki.

Additionally, if you have a custom Post Processor that supports additional drill cycles, you may use a pop-up menu for Boring options including Bore, Fine Bore and Back Bore. The Rough Mill Bore option works like a Roughing operation in that it will clear out a designated area as defined by the information entered in the Bore tab. The Finish Mill Bore option works similarly to a Contouring operation in that it will only take a finish pass as defined by the information entered in the Bore tab. Please note that output from these three extended cycles (Bore, Fine Bore, and Back Bore) requires a modified post processor; if you try to use one of these cycles with a post that does not support them, you will receive an error message. Post modifications to support these drill cycles are available free of charge.

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
The value entered is the rate of the spindle measured in revolutions per minute. Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition.

Feed
The value entered designates the rate that the tool will be moving when it enters the material, measured in inches per minute or millimeters per minute. Feed is only active when the selected Entry/Exit Cycle is Feed In - Rapid Out, Feed In - Feed Out, Peck Full Out, or Peck Chip Breaker. Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition.

Cut Feed
Cut Feed is only active when the selected Entry/Exit Cycle is Rough Mill Bore or Finish Mill Bore.

Tap%
The value entered here specifies the percentage of the feedrate that will be used on the tapping cycle. This text box appears only if Tap is the selected Entry/Exit Cycle.

Dwell
The value entered in this text box allows the user to specify the amount of time in seconds the drill will pause at the bottom of the hole with the spindle on. The dwell option is available in all drill cycles (excluding Mill Bore cycles).

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

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

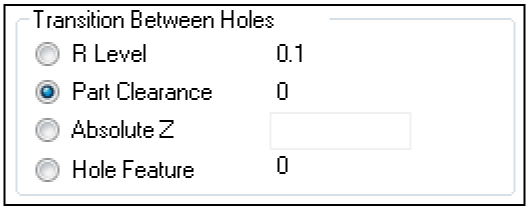
Retract / Pull-Off
The Retract text box is active only if Peck Chip Breaker is the selected Entry/Exit Cycle. The value entered specifies the amount that the tool will retract after each peck.

The Pull-Off text box is available only for Fine Bore and Back Bore. The value entered specifies the amount that the tool will move in Z+ for pecks or retracts.

1 Direction
Your machine must support this option for it to be effective. When this checkbox is selected, all tools will approach each hole from the same direction (a positive axis move), eliminating backlash from the ball screws of the machine. A custom post processor is required for this function to work.

Drill Depths Diagram
The values set here specify the clearance and cut depth values for the process. The depths diagram will change to one of three appearances depending on the type of tool you have designated for the process. The depths and clearances are fully detailed in Diagram Options .

Transition between Holes



R Level specifies that moves between holes in the operation will be done at the level entered for the Entry Clearance Plane. Clicking the Load H1 D button loads the depth of the first selected point or circle into this box.

Part Clearance specifies that the tool will retract to the operation's clearance plane, then rapid to the Master Clearance Plane also known as as CP1, set in the Document Control dialog (shown as a fixed value), move to the next hole and rapid down to the operation clearance plane before drilling.

Absolute Z with a user-specified number allows for a custom level that the tool will use when travelling between holes. The tool rapids from this level to the Entry Clearance plane, reducing the program time.

Hole Feature will retract the tool to the top of the Hole Feature plus the top clearance amount specified in the hole manager.

Vary Depth With Geometry

This option will cause the drilling depth to be variable, based on the selected geometry. The retracts will all be to the same level but the final Tip Z or Full Diameter Z are relative to the geometry, based on the first selected point. Turning this item off allows a constant Z depth drill process to be defined from geometry at different depths. This could be very useful for constant depth spot drilling. When this option is selected, the Post Processor will not have the option to combine similar holes into subprograms.

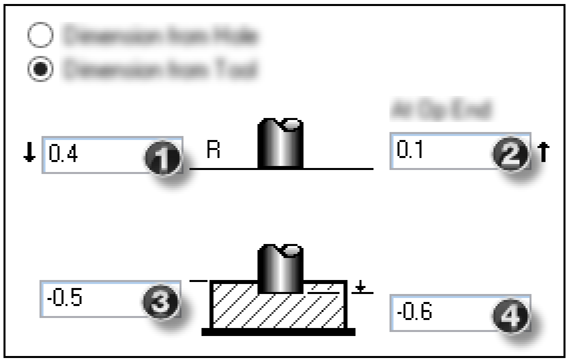
Reverse Order

This option reverses the direction of the hole selection order.

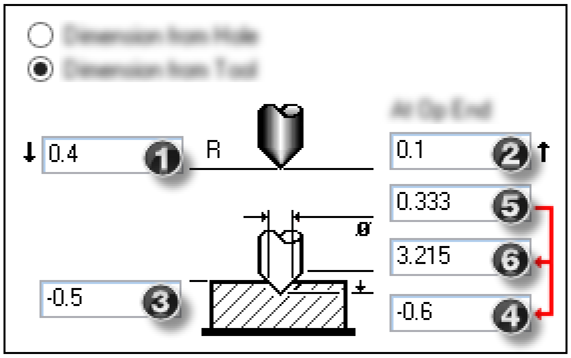
Diagram Options

End mills, shell and face mills, thread mills, keyway cutters, bores, taps, spot face tools, reamers and form tools generate a mill-style depth diagram. Drills, center drills, spot drills, countersinks and roundover tools generate a drill-style depth diagram. Back bores generate a diagram specifically for performing a back bore process. Additionally, the diagram may have several extra depth fields if you have selected the Dimension from Hole option.

Mill style

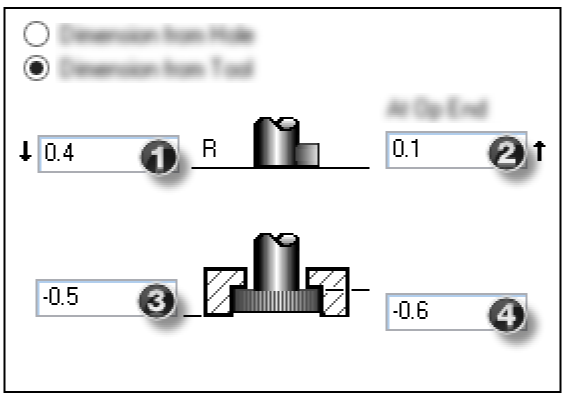


Drill style



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

Back Bore



When setting a drill-style process the Drill Surface Z, Sharp Tip Z, Spot Diameter and Full Diameter Z are interactive and calculated from the tool information as well as values entered. Red arrows are drawn to show you what will change when you modify a value.

Entry Clearance Plane

This item specifies the position the tool will rapid to when approaching the part.

Exit Clearance Plane

This item specifies the position the tool will feed to when retracting from the part.

Drill Surface Z

specifies the Z position of the top surface of the material. When a back bore process is being defined the values for the Drill Surface Z and Floor Z are absolute from the Z axis part origin. Thus, if the part is 50mm deep and the bore hole is 40mm deep, the Surface Z should be -50 and the Floor Z should be -10. The functionality of retract values remains unchanged.

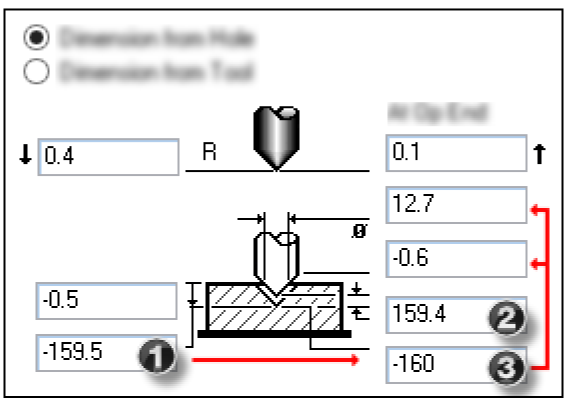
Spot Diameter
specifies the diameter of the hole at the Drill Surface Z. This is useful when counter-sinking.

Full Diameter Z
specifies the lowest Z depth the full diameter of the tool will plunge when drilling.

Sharp Tip Z
When setting a mill-style or back bore process this is the Floor Z value. This item specifies the final Z depth of the tool tip, and is the number that will be used in the posted output of the finished code. If a Full Diameter Z is entered, the Sharp Tip Z will be calculated from the tool diameter and the tip angle, otherwise the user can simply enter the desired Sharp Tip Z.

Dimension from Hole or Tool
This option lets you decide how to define the drilling process. Dimension from Tool functions like older versions of GibbsCAM, you set the Surface Z and Full Diameter Depth or Tip Depth values. Dimension from Hole is commonly used if you want to determine the toolpath not from the tool but from the hole itself, which can be geometry or a solid. Dimension from Hole is also very useful when performing multi-tool machining on a hole, e.g. spot, pre-drill, drill and tap. Selecting Dimension from Hole adds several values you can set to control the tool, based on what the hole should be.

Incremental Depth
This value is an incremental distance (with polarity) from the Top Surface Z value.

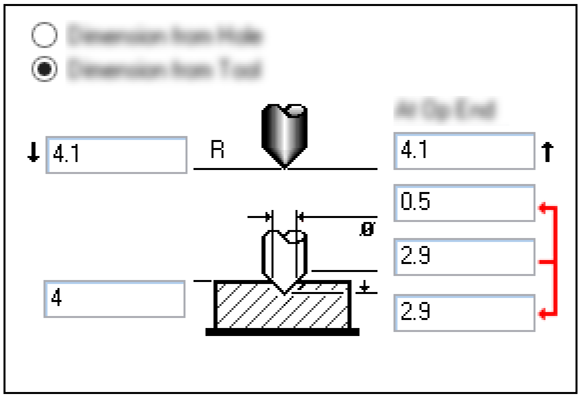


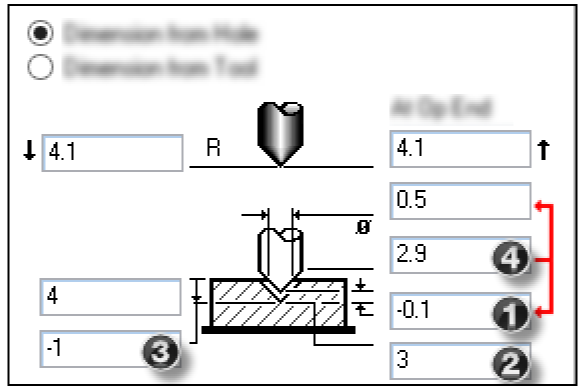
1. Incremental Hole Depth
2. Hole Depth
3. Tip Distance

Hole Depth
The absolute depth position of the hole depth.

Tip Distance
Distance you want the tool to be from the bottom of the hole.

The Dimension From Tool and Dimension From Hole options show the same data just in two different ways. If we do a side-by-side comparison we can see the similarities and differences. From this you can hopefully decide which is best for you in a situation. In the images below we see the same process, toggled between the two options. With the Depth From Tool option we see that the top of the part is at 4 and we are sending the tip to 2.9 resulting in the full diameter falling at approximately 3. The process data knows nothing about the actual hole. In the Depth From Hole item we see that the top of the part is still 4 but we have set some additional data. For example we have specified that the tip of the tool should remain 0.1 off the bottom of the hole (#1). We also stated that the bottom of the hole is at 3 (#2). Setting this value filled in two other values, first it set that the hole is -1 from the top surface (#3) and the full diameter will fall at approximately 3 (#4).





Dimension From Tool	Dimension From Hole

Load H1D

Clicking this button will load the depth of the first selected point into the Tip Z depth box. This is useful when there are many holes and finding the first hole may not be easy, especially when using variable depth geometry.

Other Common Controls

Coolant

The 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.

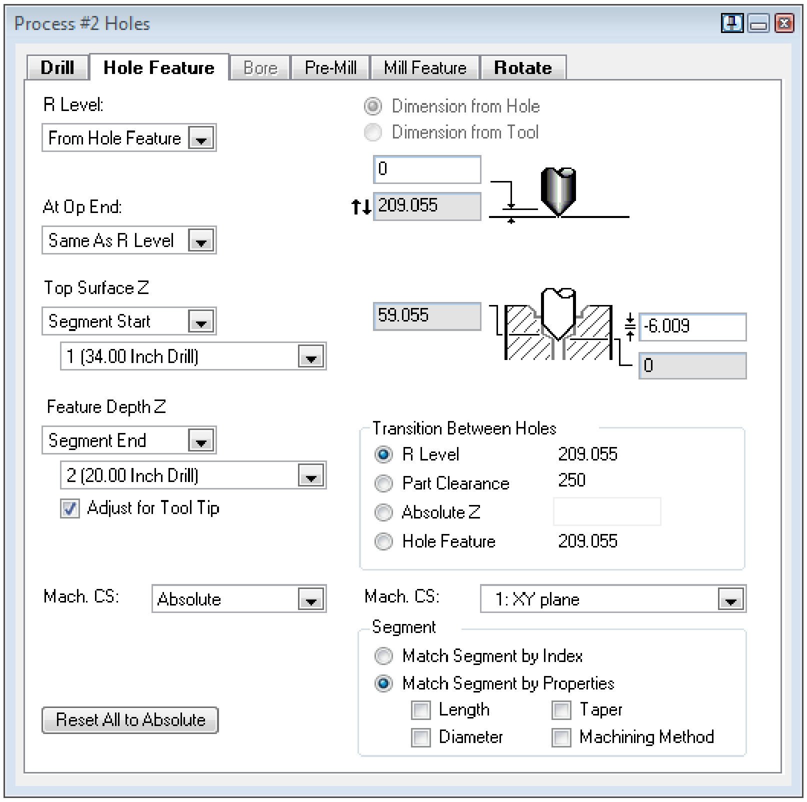
Pattern

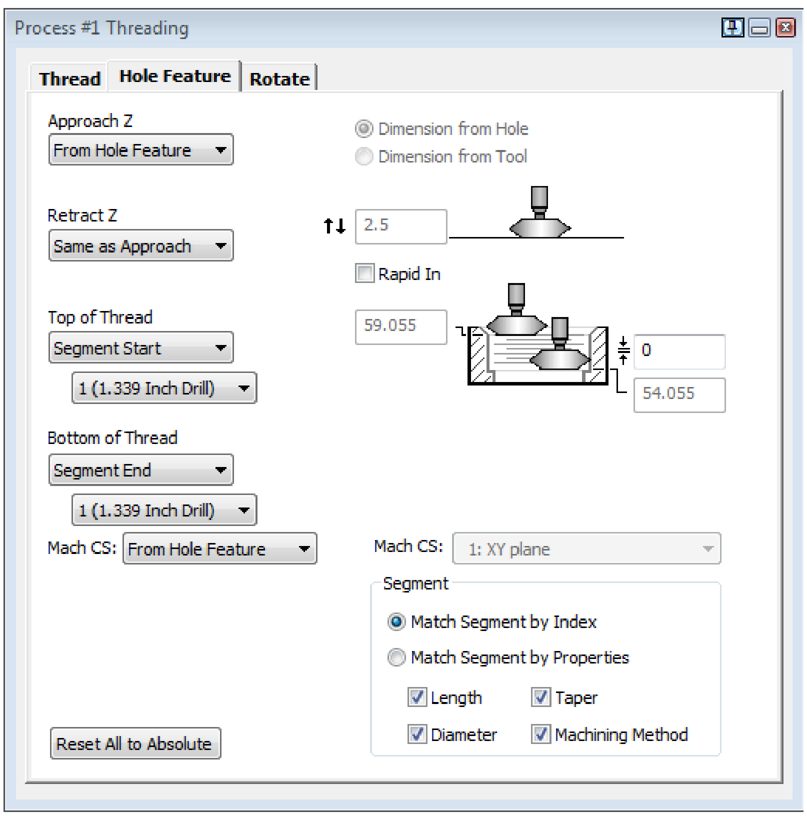
When the Pattern checkbox is selected, the process will create identical toolpaths in different locations on the part. The toolpath generated will be cut once for each point in the selected pattern workgroup. The pattern workgroup, which is selected from the adjacent pop-up menu, contains unconnected, plain points that serve as origin points for the toolpath created by the process. The original toolpath created will NOT be cut unless the origin point for that toolpath is included in the pattern workgroup. Posted output will create one subprogram for the primary toolpath and call that subprogram once for each point in the pattern workgroup. For more information, see Pattern.

Mach. CS

The Mach. CS drop-down list appears on this tab when a 3-axis MDD is active. For more information, see Mach. CS .

Hole Feature Tab

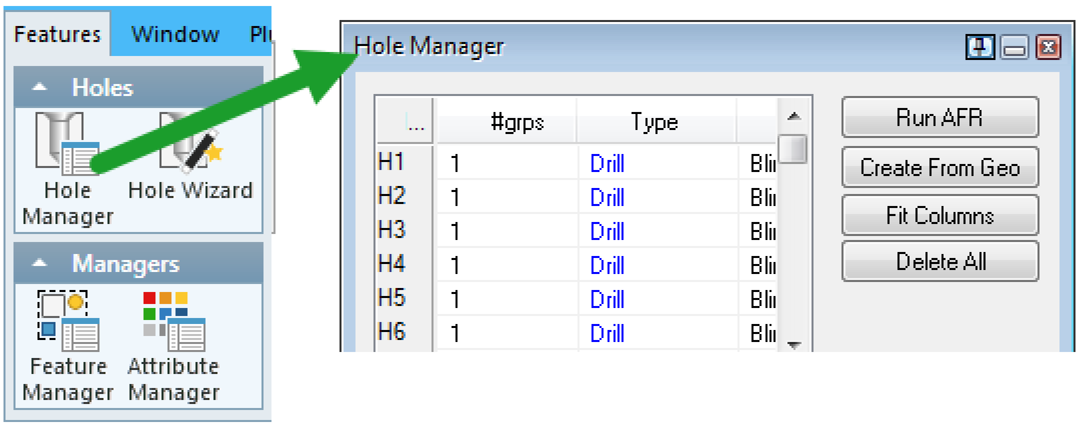




The Hole Feature tab provides a very powerful way to generate drilling and threading processes and must be used in conjunction with the Hole Manager feature. Processes can be applied to the individual data of each of the holes selected within the Hole Manager. Also, if a tool is substituted the values in the Process will automatically adjust without further need to open the process.

Please note: It is not advisable to mix processes where some contain "from attribute" or "from feature" and some do not. When GibbsCAM encounters this, processes set to "from attribute" or "from feature" are always machined first.

The parameters in the tab become operative only if hole features have been created or loaded by Hole Manager. Use Hole Manager to select holes, edit and recognize hole features, sort machining order, and so forth. If no Hole Manager data is applied to holes, then the Hole Feature tab is not bold and its parameters are inoperative – that is, their settings and values have no effect on the generated toolpath. Ensure the Hole Manager dialog is left open.



Please be aware that all settings for segments use data from the FIRST hole selected. We call this the **Reference hole**. It is therefore most important that you have selected the correct hole.

Settings, Options, and Parameters

R Level

For the Threading process, this control is named: Approach Z.

This is the Top of hole ("D") measurement including the Top Clearance value. In the Hole Feature tab, in addition to options for Absolute and Incremental, there is an option for From Hole Feature.

Absolute

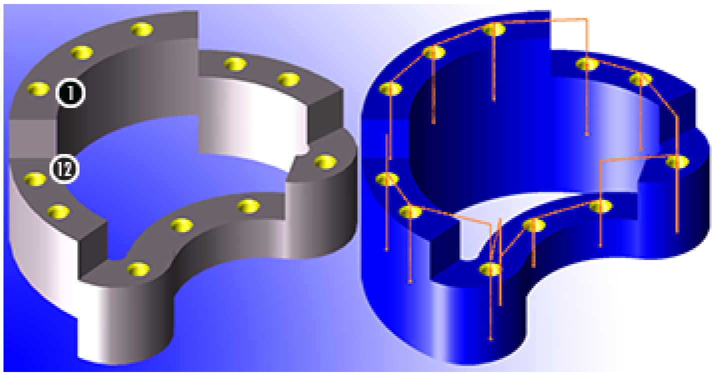
Enter a value for Entry Clearance Plane.

Incremental

Enter the distance between entry plane and surface.

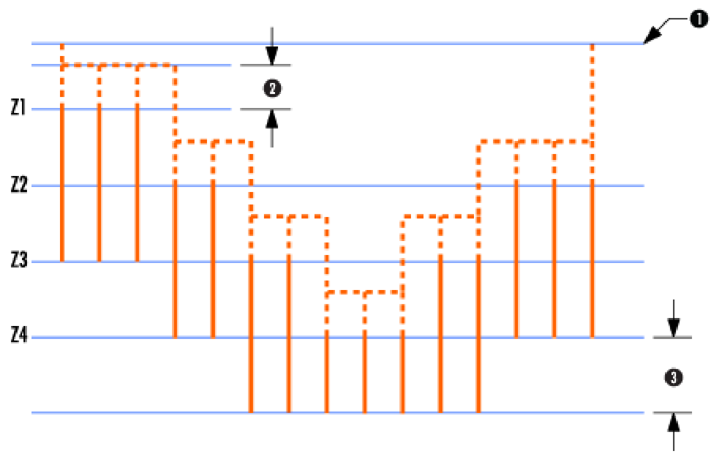
From Hole Feature

This option will cause the retract level to be variable for each hole feature. The retract amount is relative to the settings for the Reference hole (See note above).



The illustration shows an example with twelve drill holes. The Hole Manager was used to create hole features (points with data about the hole dimensions) and a drilling process was applied. We can see the toolpath using different retract levels and even different cut depths on the 8th through 10th holes.

If we look at the toolpath in a linear fashion, we can see more clearly what is happening. The tool drills the holes at the top of the part and moves to the second set of holes. The retracts for the second set of holes are the same amount shifted down in Z. This is repeated for all holes at a given Z depth. The interesting part is the set of three holes at Z4. The toolpath does not go as deep as the other holes. This is the associativity between the Hole Manager and the toolpath generation. The system knows this hole is only 1 inch deep where the other holes are 2 inches.



1. Master Clearance Plane
2. Retract Amount
3. Smaller step due to hole depth

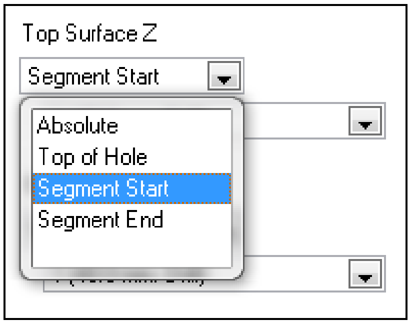
At Op End

For the Threading process, this control is named: Retract Z.

This is the exit clearance plane and can either be Absolute or the Same as R Level.

Top Surface Z

For the Threading process, this control is named: Top of Thread.

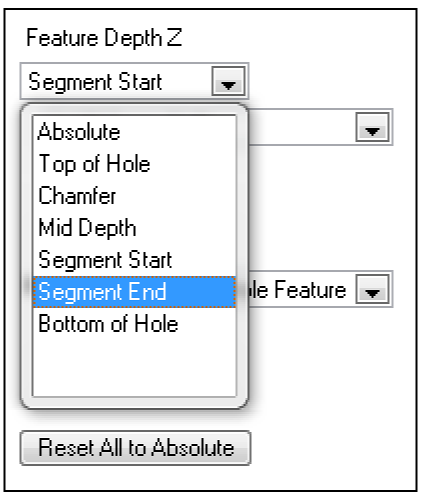


You can choose an Absolute value or use the Top of hole data from the Hole Manager (which will be variable for each hole feature).

You can choose the start depth to be at the Segment Start or Segment End of any one of the segments that are part of the **Reference hole** (See note above). All segments that form part of the Reference Hole are displayed in a dropdown, numerically from top to bottom. The diameter and machining method of each segment are shown in brackets. The top clearance box is greyed out when the Top of Hole or Segment options are chosen, as the value is automatically drawn from the Hole Manager.

Feature Depth Z

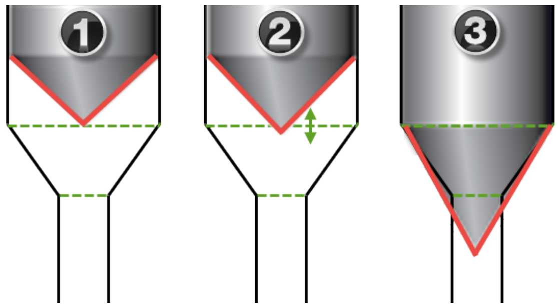
For the Threading process, this control is named: Bottom of Thread.



Feature Depth Z contains the same choices as the previous "Z depth" options located on the Hole Feature tab prior to GibbsCAM Version 10.8. An Absolute value can be entered, or you can specify the Top of hole, the (single) Chamfer depth, or the Mid Depth of the hole.

Bottom of Hole, Segment Start and Segment End options get their values automatically from the Hole Manager, where holes can be compound, with multiple segments and chamfers. Choose the end depth to be either the Segment Start or Segment End of a particular segment. A dropdown will list all segments (with their diameter and machining methods) that are part of the **Reference hole** (as in Start Depth above).

If you choose Bottom of Hole, Segment Start or Segment End, an additional Clearance adjustment value can be entered which is then added to the end depth. There is also an Adjust for Tool Tip checkbox which, when enabled, sends the **shoulder** of the tool to the segment start.



Feature Depth Z options:

1. Segment start
2. Segment start with clearance amount (negative shown)
3. Adjust for tooltip - sends the shoulder of the tool to the Segment start (this may cause gouging depending on the tool)

Machining CS

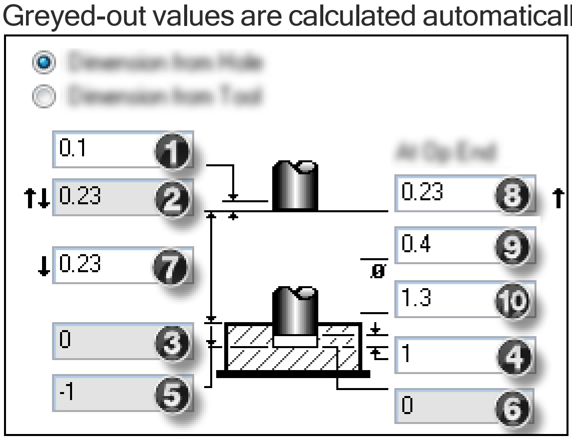
Absolute enables the Mach CS: dropdown on the right of the dialog and enables you to choose in which CS to machine. From Hole Feature will use the value from the Hole Feature, which means that you can machine in all orientations.

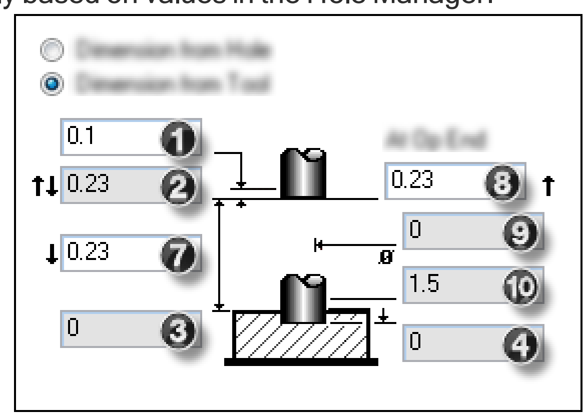
Reset All to Absolute

Clicking this button will reset previously used parameters back to "Absolute".

Depths Diagram

The process parameters for a Hole Feature are different than the standard Drill or Threading parameters. The values are interactive, meaning that changing one can change one or more other values. The change is based on the size of the tool and also on parameters in the process dialog. Greyed-out values are calculated automatically based on values in the Hole Manager.





1. Incremental R Level (for Drilling; or, for Threading, Approach Z)
2. Operation Clearance Plane
3. Top Surface Z
4. Depth of hole to tip of tool
5. Incremental depth of hole
6. Absolute hole depth
7. If R Level (or Approach Z) is set to Incremental: Incremental distance between entry plane and top surface of part
8. If Op End (or, for Threading, Retract Z) is set to Absolute: Exit Clearance Plane
9. Spot diameter
10. Full diameter Z.

Incremental R Level / Approach Z

Incremental value that lets you move the tool tip above R Level (for Drilling; or, for Threading, Approach Z). Default: `0`. The level where the tool will rapid to (before beginning the op) and rapid from (after ending the op). This is in additional amount to the Top Clearance level set in the Hole Manager. On holes subsequent to the first, the tool will retract to the next Top Clearance Plane to give variable retract levels.

Operation Clearance Plane
Z level that the tool will rapid to before beginning the drill cycle.

Top Surface Z
Z position of the top surface of the first hole. Unless the value is Absolute, this is greyed out, as the value is automatically inserted from settings in the Hole Manager.

Depth of hole to tip of Tool
Typing a value in the Dimension from Hole dialog automatically calculates and displays the corresponding spot diameter and full diameter depth.

Incremental Depth
An incremental distance (with polarity) from the Top Surface Z value.

Absolute Hole Depth
Absolute depth position of the hole depth. Can be entered directly or calculated from the Top Surface Z and the Incremental Hole Depth.

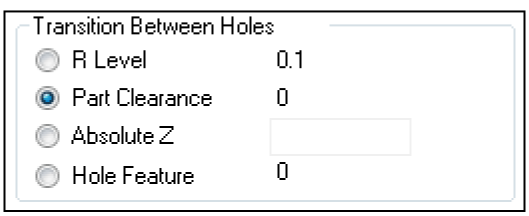
Incremental distance
Distance between the entry plane and the top surface of the part.

Exit Clearance Plane
Position to which the tool will rapid to before performing the next operation.

Spot Diameter
Entering a desired spot diameter will automatically calculate the corresponding tool tip and tool diameter depths. This is useful when counter-sinking. The maximum is the full tool diameter.

Full Diameter Z
The lowest Z depth to which the full diameter of the tool will plunge when drilling. By entering a value, the system will automatically calculate and display the corresponding stop diameter and tip depth.

Transition between Holes



R Level specifies that moves between holes in the operation will be done at the level entered for the Entry Clearance Plane. Clicking the Load H1 D button loads the depth of the first selected point or circle into this box.

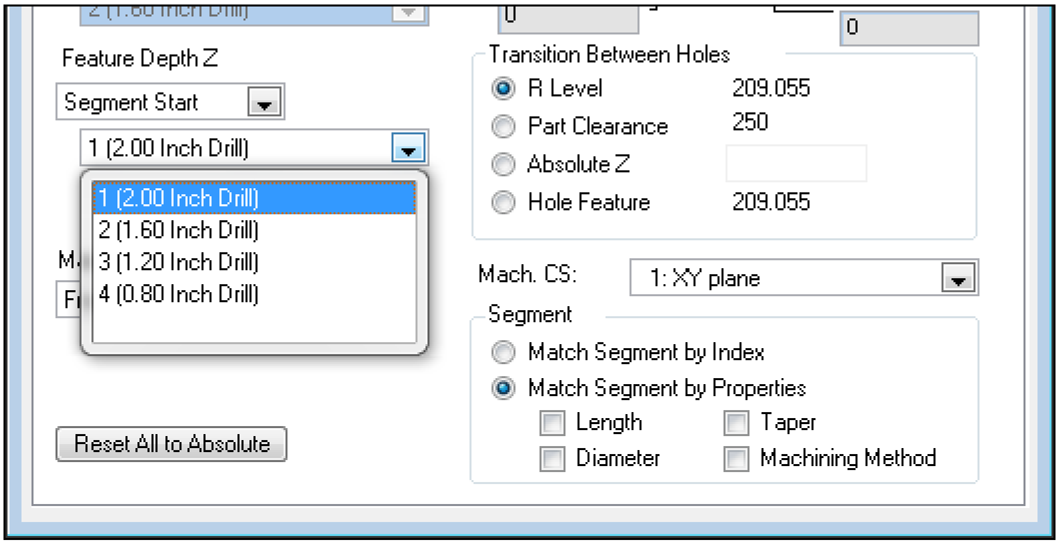
Part Clearance specifies that the tool will retract to the operation’s clearance plane, then rapid to the Master Clearance Plane, also known as as CP1, set in the Document Control dialog (shown as a fixed value), move to the next hole and rapid down to the operation clearance plane before drilling.

Absolute Z with a user-specified number allows for a custom level that the tool will use when travelling between holes. The tool rapids from this level to the Entry Clearance plane, reducing the program time.

Hole Feature will retract the tool to the top of the Hole Feature plus the top clearance amount specified in the Hole Manager.

Segment

If you choose Start Depth or End Depth using Segment options, the Segment matching area of the dialogue becomes active. This is used to match the correct segment of each selected hole, based on the segments of the **Reference Hole** (See note above). If a subsequent hole has a feature not present in the Reference Hole, this will be ignored.



The segment matching option is a powerful function within the Hole Manager. Segment matching uses the Reference Hole and matches the segments within that hole with all other holes situated anywhere on the part. These can then be machined within the same operation.

Match Segment by Index

Selecting this option will machine every hole selected, in the order they were selected, in the same way as the Reference hole.

Match Segment by Properties

For each Hole Feature, you can choose to match the properties of the selected segment from the Reference hole. Only the selected properties will be matched. These can be very powerful, but care must be taken. The following match parameters are available and can be used in combination:

Length

If the length of the segment of a chosen Reference hole is of equal length to any segment within other selected holes, then the first segment found in the hole will be machined. If a second equal-length segment exists within one of the selected holes, then this will not be found.

Diameter

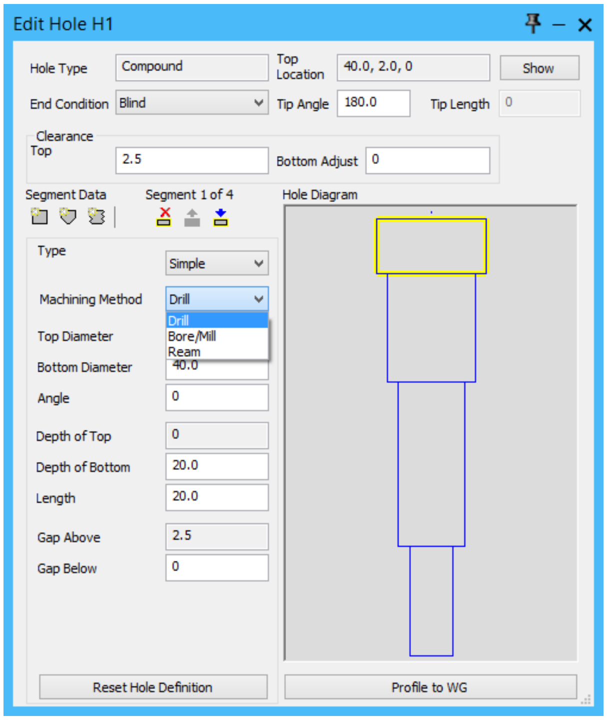
If the diameter in the segment of the chosen Reference Hole matches the diameter in any segment of other selected holes, then these will also be machined.

Taper

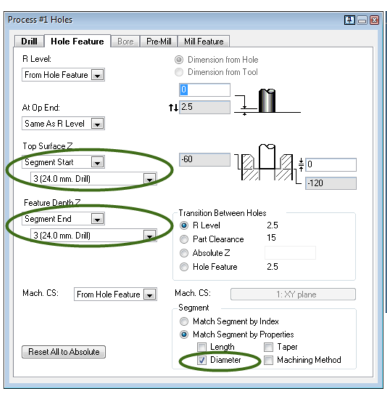
If the taper angle in the segment of the chosen Reference Hole matches the taper angle in any segment of other selected holes, then these will also be machined.

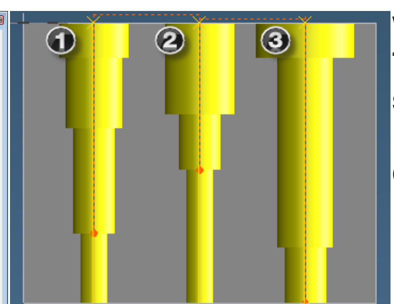
Machining Method

This will match the Machining method defined for the segment in the chosen Reference Hole.



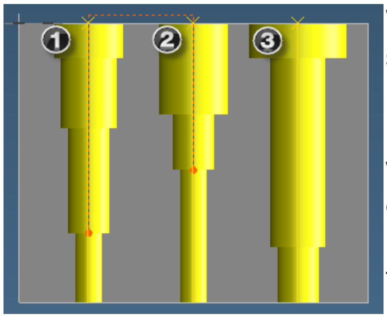
Example of matching by Properties



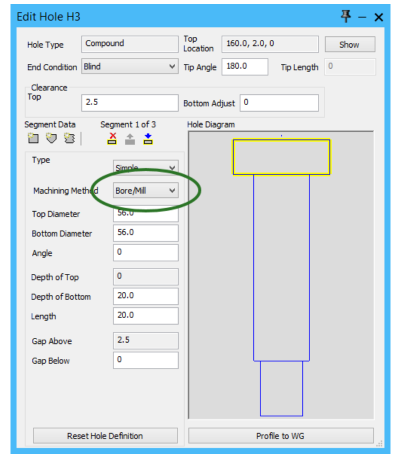


We set the Hole Feature Tab to drill the second segment of the Reference hole (1) and to match the diameter on subsequent holes.

Example of matching by Properties



We again matched the third segment of the Reference hole, but also checked Match Machining Method, which is Drill for all holes except the third hole, which is set to Bore/Mill - this is therefore not drilled.



Bore Tab

When the Rough Mill Bore, Finish Mill Bore, or Helix Bore options are selected, the Bore tab becomes available to define the operation. The options are detailed below.

Bore Diameter

The Bore Diameter setting is the final diameter of the bore as measured to the edge of the tool.

Use Circle Diameter where available

When checked, if at least one arc or circle is selected, the diameter of the helical toolpath will match the selected arc or circle instead of the Bore Diameter value.

Clearance Diameter (Rough Mill Bore or Finish Mill Bore)

The Clearance Diameter specifies the size of the area or entry hole that the tool has available. This value and the Bore Diameter determine how much material must be removed.

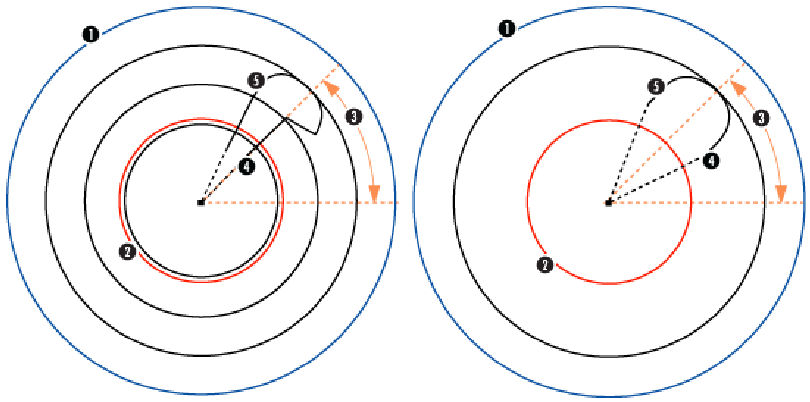
Clearance Amount (Rough Mill Bore or Finish Mill Bore)

The tool will rapid from the bore center to this distance from the Clearance Diameter. The tool feeds from the Clearance Amount to the Clearance Diameter.

Finish Entry/Exit

The values entered in these text boxes add radius and/or line moves at the beginning and end of the last pass of the roughing toolpath. If a radius value is entered, a 90° arc of the specified radius will be added at the beginning and end of the finish pass of the pocket. If a Min. Line value is entered in addition to a radius value, a line of the specified length will be added tangent to the entry/exit radius. If an entry/exit radius is not being used, a line of the specified length will be added perpendicular to the first and last move of the finish pass on the roughing toolpath. Some machines require a Min. Line entry.

The illustration shows toolpath generated by a bore operation. The black lines are toolpath. The blue circles are geometry. An extra circle has been added to show the clearance diameter.



1. Bore Diameter
2. Clearance Diameter
3. Approach Angle
4. Clearance Amount
5. Min Line

Rough Bore and Finish Bore diagrams

Start/Finish at Center
When these checkboxes are selected, the tool will enter and exit at the center of the bore hole.

Z Step Settings (Rough Mill Bore and Finish Mill Bore)
When you specify a value for Desired Z Step, the system uses the Desired Z Step value and the Floor Z value to calculate the Actual Z Step and the number of Passes that will need to be made.

Z Pitch (Helix Bore)
This defines the desired Z pitch per 360 degrees of motion.

Rapid in (Helix Bore)
When this is selected, then the tool will rapid from the Entry Clearance Z level to the Z Start Level. If it is not selected, then the tool will feed in.

Spiral Up (Helix Bore)
Generates a helical toolpath upon retract, where 360 degrees represents one revolution.

Cut Width
The value entered specifies the width that the tool will move out on each pass. This value automatically defaults to half of the tool diameter. If the value is made smaller, then the passes will overlap. If the value is made larger, then areas may be left uncut.

Stock
The value entered for Stock is the amount of material that will be left on the wall of the bore hole. A positive stock amount will leave material on the hole and a negative stock amount will cut into the hole geometry.

Overlap
An Overlap value forces the endpoint of the toolpath to extend past the start point by the specified amount.

Spring Passes
The number entered is the number of extra times the final pass will be made.

Approach Angle
The angle from 0° (standard Cartesian measurement) that the tool will begin to cut the part.

Cutter Radius Compensation On
This indicates whether Cutter Radius Compensation is turned on or off.

Climb / Conventional

This set of option buttons lets you specify the direction the tool will travel, either making a Climb cut or a Conventional cut.

Conventional Climb

Pre-Mill Tab

The Holes process has an option to specifically define entry holes and corner drilling for milling and VoluMill operations. Separate operations will be made for Entry and Corner pre-mill selections. Thus, if both Entry and Corner are both selected in the **Pre-Mill** tab, two operations will be generated. The items found in this tab are used with multi-process operations and will not generate anything unless a milling process is also in the process list. When a drilling process and one or more milling processes are in the process list, the **Pre-Mill** tab is bold.

Max Tool Overlap

This option specifies the maximum percentage of a tool's diameter that tools may overlap when entering the toolpath, useful if there are drill holes near each other. Negative values are valid and will keep tools apart.

Entry

This option will drill an entry hole at any start point for the roughing or contouring toolpath. Please note that if a drill is significantly larger than the tool it is creating an entry for, the drill may gouge the part. Be sure to inspect your toolpath.

Auto Z

This option will override the Z Depth specified in the clearances diagram. The drill will go to the Z Depth of the pocket floor. When this option is off, the pre-mill operation will cut straight to the final Z of the drilling operation. This option should be used with caution.

Z Clearance

This option will modify the drilling depth. A positive value will keep the drill tip above the pocket floor while a negative value will send the drill deeper. Essentially this value will be subtracted from the milling Z Depth. Thus if a milling operation specifies to cut to -0.5" and the Z clearance is 0.1" then the drilling operation will be to -0.4".

Corner

This option will drill a hole at any qualifying corner of the roughing or contouring toolpath. This can minimize the amount of uncut material left for a finishing pass. This results in less tool deflection, longer tool life and a better finish. Please note that if a drill is significantly larger than the tool it is creating an entry for, the drill may gouge the part. Be sure to inspect your toolpath.

Max Angle

The maximum angle between lines that will be used to determine whether two lines constitute a corner or not. All lines that meet within this angle will be considered a corner. Larger angles will not leave as much uncut material as sharper corners do.

Sharp

When this option is selected, any Sharp corners (not rounded or filleted corners) will be drilled. A Clearance Amount may be entered. The hole will be drilled this distance from the edges that make the corner. Larger clearance amounts will leave more uncut material for the finish tool.

Fillet Center

When this option is checked, any corners that have a fillet between the Min. Radius and Max. Radius will have a hole drilled at the centerpoint of the fillet.

The illustration demonstrates corner drilling. Looking counter-clockwise from the top right there is a sharp corner drill with a clearance value, a fillet corner drill at the fillet's centerpoint, and an entry point for the contouring operation. Note that the large fillet and obtuse angle were not drilled. These values were beyond the operation's tolerances.

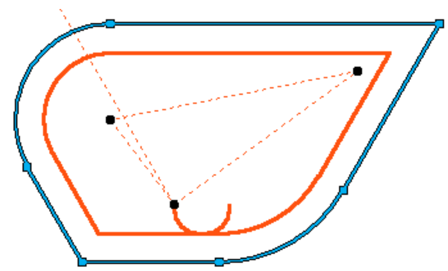


Illustration of Corner Drilling

Mill Feature Tab for Holes

The **Mill Feature** page of the Holes process dialog offers the following types of controls:

- "Attribute-Driven Controls" below
- Absolute-Only Controls

In the context of hole creation, Mill Feature is intended only for pre-drilling. Therefore, its settings are inoperative when the **Bore** tab is bolded (for **Drill** Entry/Exit Cycle choices Rough Mill Bore, Finish Mill Bore and Helix Bore).

Attribute-Driven Controls

The attribute-driven controls, on the left side of the Mill Feature page, consist of five pull-down menus. Four of them (R Level, At Op End, Top Surface Z, and Feature Depth Z) let you set depth values. The fifth (Mach. CS) lets you specify the machining CS. The pull-downs for R Level, At Op End, and (Mach. CS) are also displayed on the Drill page.

Choices offered by the pull-down menus include the following:

Absolute

For R Level or At Op End, Absolute specifies that the depth comes directly from the value specified in the depths diagram. (For example, for R Level, the value would come from the depth specified for Clearance Plane.)

For Mach. CS, Absolute specifies that the CS comes directly from the value specified in the Mach. CS pull-down menu below depths diagram.

From Attribute
Specifies that the depth comes from picking an attribute associated with the user feature. When this choice is active, another pull-down menu appears immediately below. For depth values, this lets you pick from a list of all Real-type attributes for the user feature. For Mach. CS, this lets you pick from a list of all Integer-type attributes for the user feature.

Automatic
Top Surface Z and Feature Depth Z only. Specifies that the system will retrieve the value directly from the geometry of the user feature.

Incremental
R Level only. Specifies that the value comes from the distance specified for the distance between the Clearance Plane and the Top Surface.

Same as R Level
At Op End only. Specifies that the tool retracts to the same depth as its initial approach.

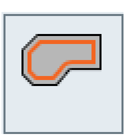
Reset All to Absolute
Click this button to set all controls to "Absolute"

Absolute-Only Controls
The absolute-only controls on the right side of the Mill Feature page consist of two option buttons controlling the depths diagram, the values in the depths diagram itself, three option buttons controlling the retract level, and a pull-down menu of choices for Machining CS.

All of the absolute-only controls are also displayed on the Hole Feature page. For more information, see “Hole Feature Tab” on page 57.

Rotate Tab for Milling Machines

The Rotate tab is available when using a Mill/Turn MDD or a 4-axis or 5-axis MDD. The settings found in this tab allow you to rotate the part or create rotary operations. For more information, see Rotate Tab.



Contour Process
The contouring process is used to make passes along a shape or multiple shapes. The toolpath can be set to either side of the geometry or on center. When multiple shapes are selected the toolpath is automatically on center, which is typically used for engraving.

> When a part file from GibbsCAM 2012 (version 10.1) or earlier is opened in this release, it is searched for Contour operations that use Thread Mill tools. If any such process has values for Surface Z and Floor Z (also called Top Surface Z and Finish Z Depth) that do not take the Thread Mill tool profile into account, then Surface Z is automatically adjusted to a value that will reproduce the existing toolpath. When this occurs, the system displays a message that tells you what was done.
>
> *Important:* No automatic adjustment will occur on **new** processes. Therefore, when you create tapered threads (whether interactively or by using macros, `*.prc` or `*.prc2` process group files, or custom plug-ins), ensure that the values for Surface Z and Floor Z are different by at least half the thickness of the Thread Mill tool. For more information, refer to Knowledge Base article http://kb01.GibbsCAM.com.

Material

Clicking this button will open 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 is the rate of the spindle measured in revolutions per minute. Clicking the button will load a recommended speed from the Material Database based on the part material and tool composition.

Entry Feed

The value entered designates the feedrate, measured in inches per minute or millimeters per minute, for the entry move (from clearance plane to the point where the tool enters the material). Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition. This value is always used for the Z plunge, ramp, or helix.

Note: The lesser of the two feedrates (Entry Feed or Contour Feed) is used as the entry feedrate when the choice for **Entry And Exit**, below, is Line with 90° Radius or (Advanced) Radius Entry or Line Entry.

Contour Feed

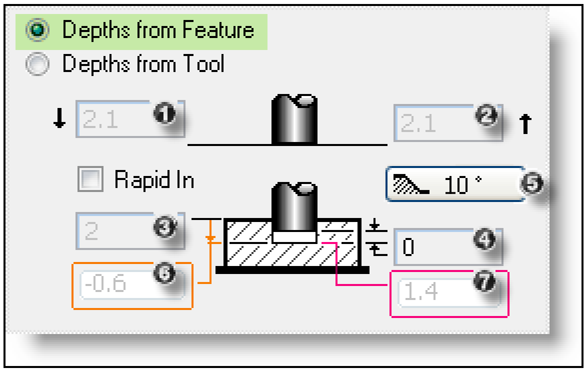
The value entered designates the feedrate, measured in inches per minute or millimeters per minute, used while cutting. Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition.

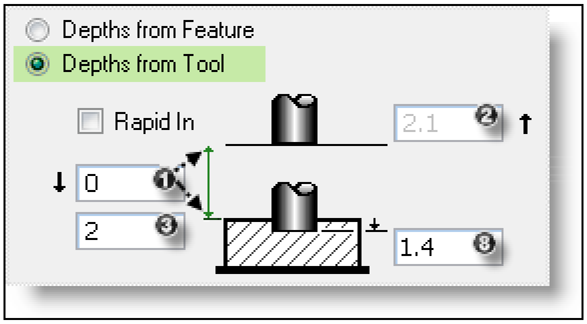
Depths Diagram

The items in this section of the dialog define the clearances and depths for the toolpath. Additionally, the Wall Control option lets you make 2 1/2 axis cuts.

Depths from Feature / Depths from Tool

These values define the clearances and depths for the toolpath for any parameter set to Absolute, including feature-specific depths like Feature Top Surface Z and Feature Depth Z.





1. CP2, Entry Clearance Plane (or* Incremental distance from plane)
2. CP3, Exit Clearance Plane
3. Surface Z
4. Incremental Tip Z
5. Wall Control (if applicable)
6. Incremental Feature Depth
7. Feature Depth Z
8. Floor Z

Items 4, 6, and 7 are displayed only for Depths from Feature. Item 8 is displayed only for Depths from Tool.

[1] When Approach Z is set to Incremental, the diagram changes slightly and you supply a value for the distance between the Exit Clearance Plane and the Top Surface.

[2] When Retract Z is set to Same As Approach, you cannot supply a value for Exit Clearance Plane.

[3] When Top Surface Z is set to Automatic, you cannot supply a value in the corresponding text box.

[5] The Wall Control button, when present (feature-based milling usually uses straight walls only), lets you make 2 1/2 axis cuts.

[6, 7] When Feature Depth Z is set to Automatic, you cannot supply a value in the corresponding text boxes.

Entry Clearance Plane

Entry Clearance Plane (also called CP2) specifies the location the tool will make a rapid move to before feeding to the start point of the toolpath.

Exit Clearance Plane

The Exit Clearance Plane (also called CP3) specifies the location the tool may rapid to after completing the toolpath.

Surface Z

The Surface Z specifies the top level of the material.

Incremental Tip Z

Distance from tool tip to the bottom of the feature.

Incremental Feature Depth

Distance from top to bottom of the machining feature.

Feature Depth Z

Z Value of the lowest depth of the machining feature.

Floor Z

The Floor Z specifies the finished depth of the pocket.

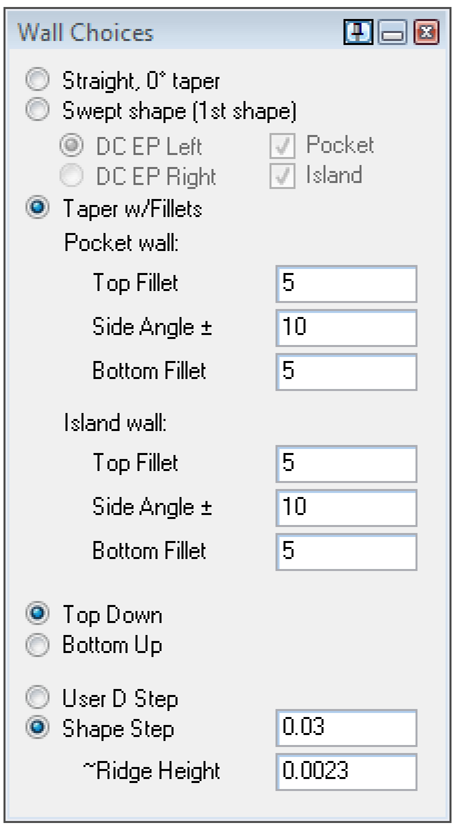
Rapid In

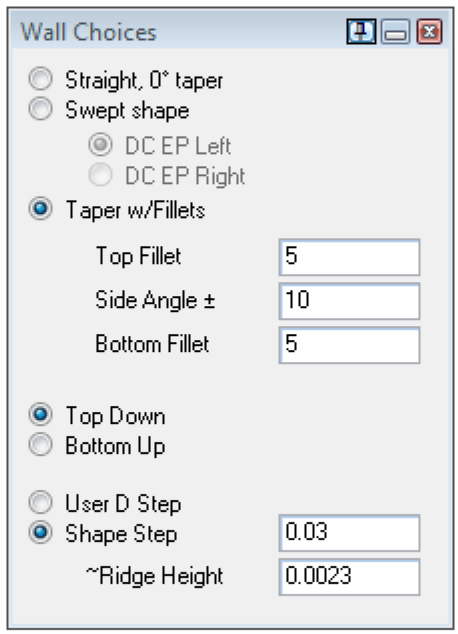
When this checkbox is selected, the move from the Entry Clearance Plane position to the start point of the toolpath will be a rapid move rather than a feed move. The Rapid In option should be used with caution, as it can create rapid moves directly into the part material.

Wall Control

The Wall Control button brings up a dialog which provides for the creation of 2 1⁄2 axis surfaces (tapered or swept wall shapes) on contouring processes. If the wall is tapered, the button will show the angle of the taper. If the wall is a swept shape, the button will say “Swept”.

The three option buttons at the top of the dialog determine the type of wall that will be created by the contouring process. The available choices are Straight, Swept shape, and Tapered w/Fillets. The Straight option is the default, and when it is selected no information needs to be entered in this dialog. The information necessary for tapered and swept walls is described below. Additional information is found in this dialog if the contouring process is combined with a roughing process in the Process list. For an example of how to use Wall Control, see 2 ½ Axis Surfacing.





Example of the Wall Choices dialog when a Roughing process is present, contrasted to its state when it is a single process

Swept Shape

When the Swept shape option is selected, a designated drive curve will be swept around the base curve shape. The drive curve is the shape of the wall. The DC EP Left (Drive Curve End Point) and DC EP Right selections are used to indicate which side of the base curve cut shape the end point of the drive curve will be located on. This depends on the cut direction. The cut direction is determined by the Machining Marker arrows. Visualize looking down the base curve along the cut direction; the drive curve will be attached to the left or right of the base curve. When the Contour process is a part of a multi-process operation you can also specify whether to apply the sweep to the Pocket walls, the Island walls or both.

Tapered Shape

When the Taper w/Fillets option is selected, the walls of the shape will be created with the designated side angle and any radii specified for the top and bottom fillets. When the Contour process is a part of a multi-process operation you can specify different settings for both the Pocket and Island walls.

Top Down / Bottom Up

These selections indicate whether the toolpath will start at the top of the shape and cut down (Top Down) or start at the bottom of the shape and cut up. The Bottom Up selection creates the smoother surface finish.

One Direction / Back & Forth

If One direction is selected, the tool will always cut in the same direction. The tool will make each pass from the start point to the endpoint of the toolpath, moving back to the start point for each additional pass. The move from the end point back to the start point will be a rapid move if Depth First is turned on in the Process dialog. If it is turned off, the move will be a feed move. If Back & Forth is selected, the tool will alternate between climb cutting and conventional cutting. The tool will begin cutting at the start point of the toolpath and cut to the end of the toolpath, then reverse direction and cut from the end point to the start point.

User D Step

This option creates a depth step of a specific value. This is an absolute distance in Z that determines the depth of cut on each pass.

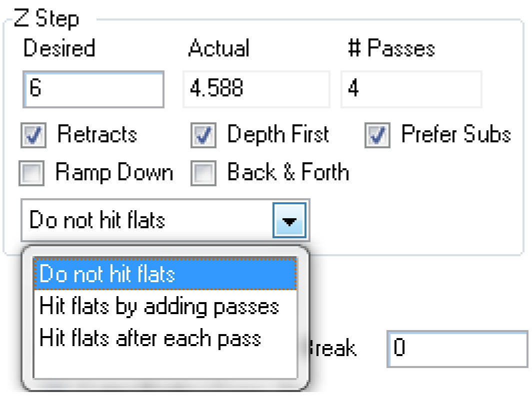
Shape Step

This option generates a parametric step based on the drive curve or taper. This specifies a distance along either the drive curve or taper angle that determines the depth of cut on each pass.

Ridge Height

This parameter is available when creating a tapered wall. The Shape Step and Ridge Height text boxes are interactive; either value can be entered and the other will be calculated. The Ridge Height (or “scallop height”) is an approximate calculation of the material left on the tapered wall between each pass of the tool.

Z Step



The items in this section help you define the toolpath behavior when stepping down in Z and transitioning between shapes.

Desired

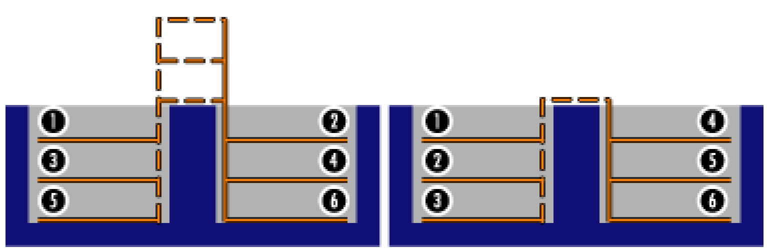
This is used to specify the depth of each pass. The system uses the Desired Z Step value and the Floor Z value to calculate the Actual Z Step and the # Passes that will need to be made.

Retracts

Retracts becomes active when multiple passes are being taken when cutting a given shape and the Depth First option is activated. When it is turned on, the tool will rapid up to CP3 (the exit clearance plane) after each pass, and will then rapid to the start point of the next pass. When Retracts is off, the tool will feed from the end point of one pass to the start point of the next pass without retracting up in the Z axis.

Depth First

This option lets you specify a preference for how multiple contours with more than one Z Step are to be machined. Activating Depth First will cause the toolpath to completely machine the first item to the final Z depth, then move onto the next item. By deselecting Depth First, the user has told the system to first machine all selected items at the first Z step. Once the first level is complete on all selected items, the operation starts over at the first pocket or contour and begins to cut at the second Z step. This will continue until the operation is complete.



Example of machining without Depth First versus with Depth First

Prefer Subs

This checkbox provides the user with the option of using subprograms in the posted code. Activating this item produces shorter G-code output.

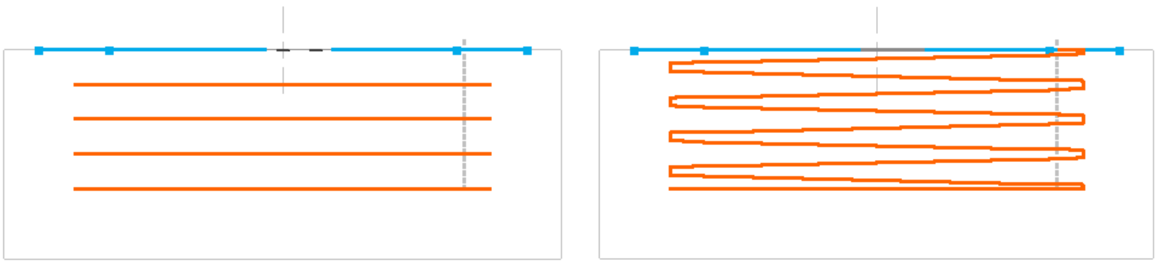
Hit Flats

This option modifies the Z Step so that a contour pass is taken at each flat surface, such as a boss top or the pocket floor. Z Step will be recalculated and the step will vary to hit the flats; therefore the Z Step will not match the value shown under Actual.

Ramp Down

For contours with vertical (not tapered) walls, creates a continuous spiraling toolpath with one finish pass at final depth.

Shown below are two contour toolpaths with 10mm Z steps between each pass. The second shows the effect of selecting Ramp Down: Toolpath consists of two complete spiral loops, each 10mm deep, with one final complete pass.



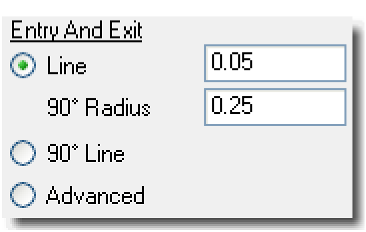
Ramp Down not selected. Ramp Down selected.

Back & Forth

Allows you to specify how the tool moves from the end of one pass to the start of the next: either always cutting in the same direction (unselected), or else alternating between climb cutting and conventional cutting (when Back & Forth is selected).

Conventional Climb

Finish Entry / Exit



The items in the Finish Entry / Exit section let you create additional moves to add to the start and end of the toolpath. Entry/Exit lines are useful when using Cutter Radius Compensation (CRC) because CRC is typically turned on and off on the first and last line moves of the toolpath.

This section offers three options: Line with 90° Radius; 90° Line; and Advanced.

Line and 90° Radius

This option will generate a 90° arc (you specify the radius) to be added at the beginning and end of the toolpath. This arc will be tangent to the start feature at the start point and to the end feature at the end point. If a value is entered in the Line text box, a line of the specified length will be created tangent to the arc as the first and last move in the toolpath.

90° Line

When this option is selected, a line of the specified length will be added to the toolpath. This line will be perpendicular to the start feature at the start point and the end feature at the end point.

Advanced

Use this option to create a custom entry and/or exit move. When this option is selected the Entry/Exit tab is bolded. Define the custom entry and exit in the Entry/Exit tab. Use an advanced move as described in the Entry / Exit tab. For more information, see Entry/Exit Tab.

Controls Specific to Contour Process

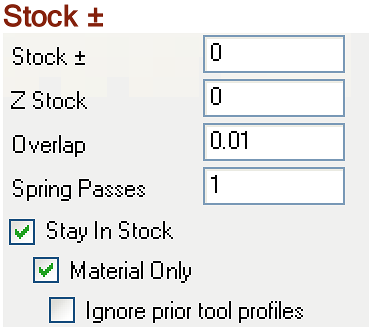
No. of Extra Offsets

You can set a positive number of extra offsets and set a stepover value to generate multiple operations. Each operation's toolpath corresponds to an additional stepover.

Extra Stepover

When No. of Extra Offsets is nonzero, specify the amount of each extra stepover in this field.

Stock ±



The value entered specifies the amount of material left on the part geometry after the completed toolpath. A positive value will offset the tool away from the geometry, leaving material on the wall. A negative value will move the toolpath into the geometry. If you are cutting the geometry on center, this option will have no effect.

Z Stock

This is the amount of stock in the Depth you wish to remain or remove. A negative value will cut deeper into the stock by the amount specified. A positive value will leave material.

Overlap

An Overlap value extends the end point past the start point by the specified amount. This is very useful when using CRC.

Spring Passes

The number entered is the number of extra times the final pass will be made. In operations with multiple Z depth passes, the tool will retract to the entry clearance plane defined in the contouring process.

Stay In Stock

Toolpath generated by the system can be optimized in various ways by using the Stay in Stock, Material Only and Ignore Tool Profile options. These options are hierarchical – one depends on another being active. If Use Stock is active, then Material Only is available and if Material Only is active, then Ignore Tool Profile is available. The Stay In Stock option will confine the toolpath of any Contouring operation that goes beyond the bounds of the stock. The operation will be trimmed to the edge of the stock, causing the tool to retract and rapid to the next entry point.

Material Only

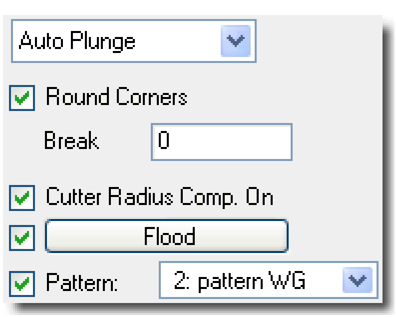
Material Only depends upon Stay In Stock being active. Material Only optimizes toolpath by limiting the toolpath to areas that have material. If a part has already been partially machined, then Material Only will optimize the cut areas and ensure there will be no “cutting air.” For an extended discussion on Material Only, see Material Only.

Ignore Tool Profile

Ignore Tool Profile will cause Material Only operations to ignore the shape of tools in preceding operations. This is useful when re-machining with a tool that has a corner radius equal to or greater than that of prior tools. When Ignore Tool Profile is activated, then Material Only pretends that all mills are sharp endmills. When a part is defined by 2D geometry only, it is recommended that Ignore Tool Profile be activated as material left on 2D walls can be easily visualized by the system.

Deselecting Ignore Tool Profile makes things somewhat more complex. First of all, the remaining material is more accurate, factoring in all tool tapers and corner radii of the tools in prior operations. If you have a roughing tool with a large corner radius and a finishing tool with a smaller corner radius that will be cleaning up material left on the floor by the larger tool, be sure to turn off Ignore Tool Profile. Leaving this option off is also best for machining non-2D parts, such as a pocket in a solid with bottom fillets.

Feed Entry Type



The pull-down menu allows you to select how the tool will feed into the part. By default, the tool plunges using automatic settings (Auto Plunge), but you can set Plunge settings manually or select a Ramp or Helix entry.

Ramp

Selecting this option will let you define a ramping motion when entering the part.

Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the ramp. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the ramp.

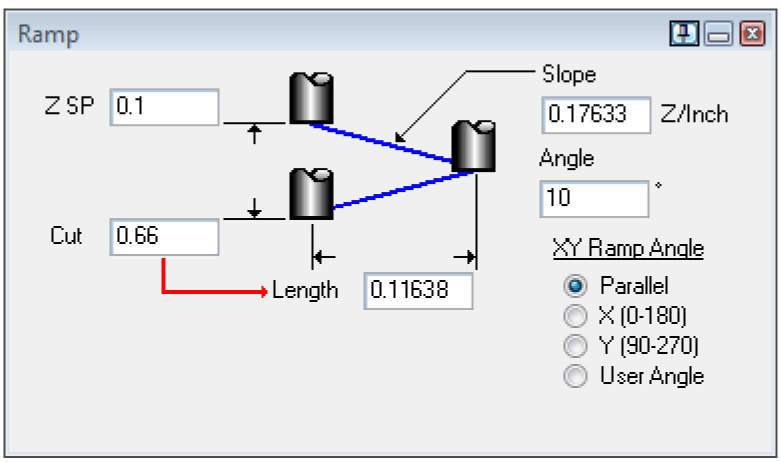
Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Cut

This value is the maximum Z step that the tool can take. The value is equal to twice the Z depth of a single ramping move. In other words, it is the total depth of the zig and the zag in a ramping move. This value controls the Ramp Length based on the current Slope and Ramp Angle.

Slope: Z/Inch or Z/mm

This value specifies the slope of the ramp. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY. Specifying the Slope will calculate the Ramp Angle and Ramp Length values based on the current Cut value.



Ramp Angle

This is the angle of descent for the ramping motion. Specifying this value will calculate the Slope and Ramp Length based on the current Cut value.

Ramp Length

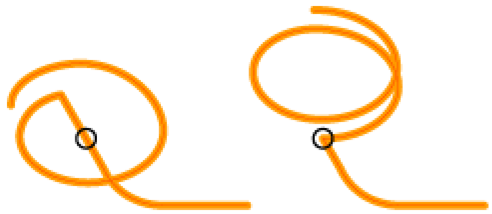
This value specifies how long the ramp is from the Z start to Z end position of a single stroke. This value controls the Cut based on the current Slope and Ramp Angle.

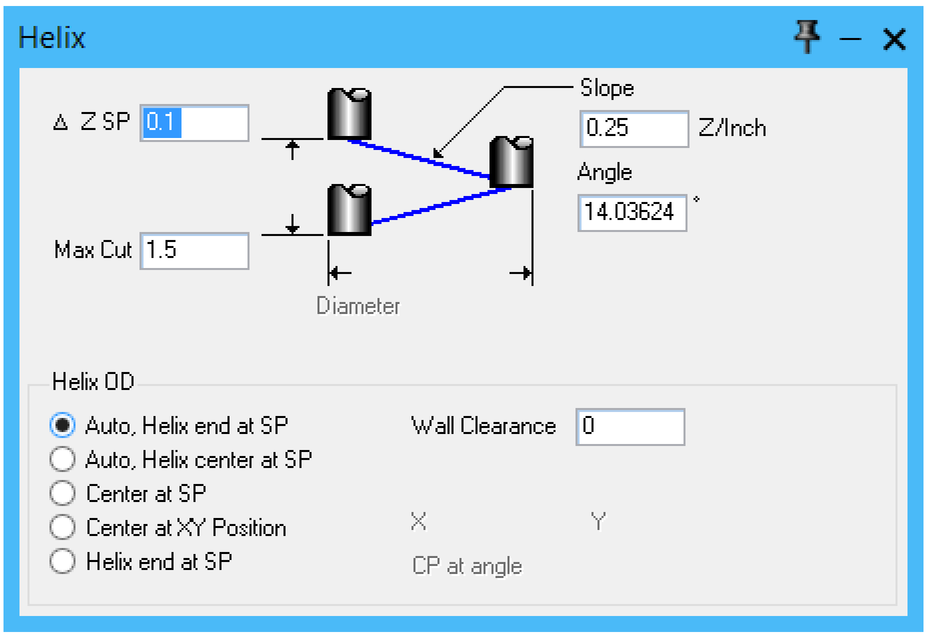
XY Ramp Angle

The ramp angle determines the starting angle for ramping into the part. You can let the system choose, or specify to start along the X or Y axis, or specify a particular angle.

Helix

Selecting this option will let you define a helical motion when entering the part.





Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the helix. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the helix.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Cut

This value is the maximum Z step that the tool can take. The value is equal to the Z depth of a fill 360° helical revolution. This value controls the Diameter based on the current Slope and Angle.

Slope: Z/Inch or Z/mm

This value specifies the slope of the helix. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY. The XY distance is measured along the circumference of the helix. Specifying the Slope will calculate the Angle and Length values based on the current Cut value.

Angle

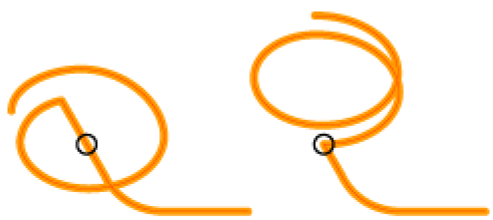
This is the angle of descent for the helical motion. Specifying this value will calculate the Slope and Length based on the current Cut value.

Diameter

This value is the diameter of the helix. This value controls the Cut based on the current Slope and Angle.

Helix Location

This setting specifies where the helix should be situated relative to the tool's entry position. Center at Entry SP creates the helix so its center is at the start point and an additional move from the helix end to the start point will be generated. Helix End at Entry SP generates the helix so its endpoint is at the same position as the start point for the rest of the toolpath. This eliminates the move from the helix center to the start point.



Center at SP and End at SP examples

Round Corners

This checkbox allows the user to designate how the system will handle the external corners of a contour. When the Round Corners option is selected, the system will add a radius move to the toolpath at every external corner of the cut shape. The tool always stays in contact with the finished shape and does not create burrs at the corners. Sharp corners can be created when this option is on by entering a corner Break of zero. When the Round Corners option is off, no radius move will be created.

Break

The value entered in this text box specifies a radius that will be put on every external corner of the selected cut shape. It will only be available only if the Round Corners option is active. Operations that include a corner break value should not be used prior to a Material Only operation. Material Only assumes the part shape is always equal to or smaller than the material at all times. This will be true unless the corner break is used because corner break cuts a radius onto a sharp corner, which can cause inaccurate Material Only calculations.

Cutter Radius Compensation On

A checkbox that indicates whether Cutter Radius Compensation is turned on or off. Most CNC machines require that CRC be turned on for Entry line moves and turned off for Exit line moves.

Other Common Controls

Coolant

The 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.

Pattern

When the Pattern checkbox is selected, the process will create identical toolpaths in different locations on the part. The toolpath generated will be cut once for each point in the selected pattern workgroup. The pattern workgroup, which is selected from the adjacent pop-up menu, contains unconnected, plain points that serve as origin points for the toolpath created by the process. The original toolpath created will NOT be cut unless the origin point for that toolpath is included in the pattern workgroup. Posted output will create one subprogram for the primary toolpath and call that subprogram once for each point in the pattern workgroup. For more information, see Pattern.

Mach. CS

The Mach. CS drop-down list appears on this tab when a 3-axis MDD is active. For more information, see Mach. CS .

Solids Tab

This item is bolded when a solid is selected. The items found on this tab apply only to machining solids. For information on the contents of this tab, see the *SolidSurfacer* guide.

Open Sides Tab

This tab is always available. The settings found here affect the toolpath when there is one or more open sides or “Air” geometry.

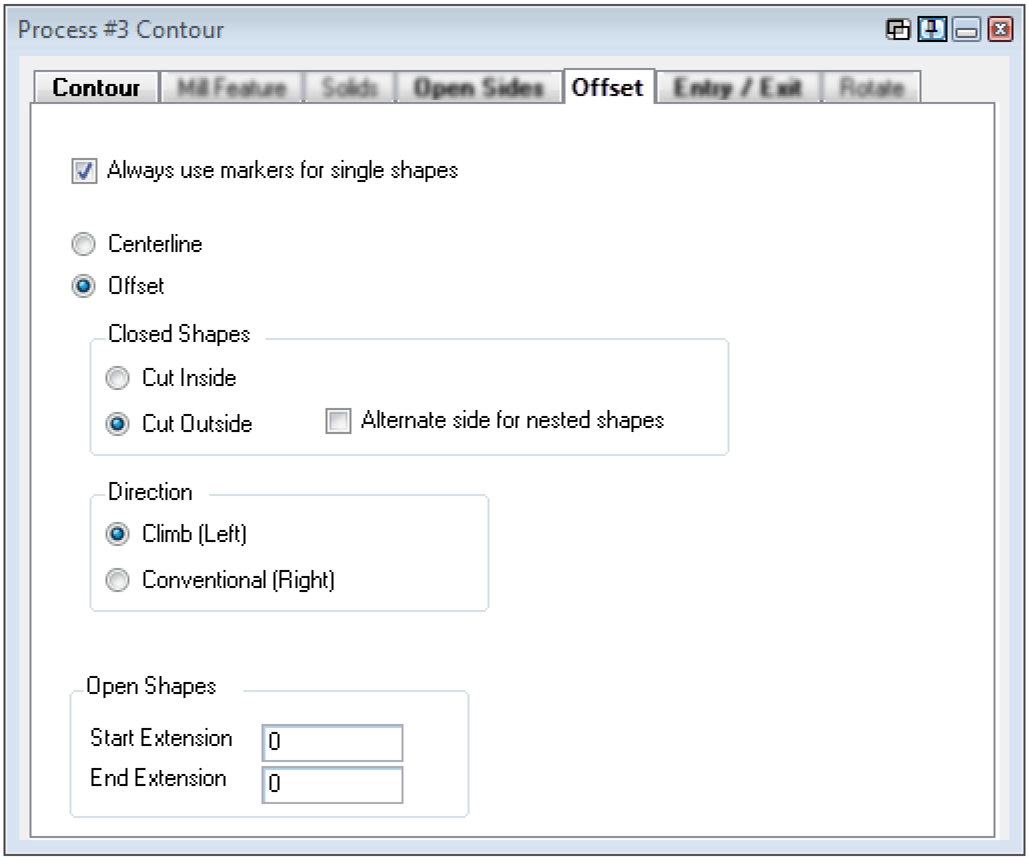
Minimum Cut

This is the smallest amount of material left behind that the system will target for machining. Extra toolpath will be created to cut areas that have this amount of material or more remaining. Areas with this amount of material or less will not be targeted for machining, although they may incidentally be cut due to normal process parameters. A value of 0 would cut all around the part (because everything has at least 0 stock), but a large value, such as the tool diameter, might not cut anything.

When using the Material Only machining option, the Minimum Cut value is very important. A value of “0” will attempt to find all possible Material Only situations, but a value greater than the tool radius is unlikely to find much to cut. This function helps you maximize the efficiency of Material Only so that you can ignore really small bits of material and better focus your Material Only operations.

Offset Tab for Contouring

This tab is always available. It provides support for performing multiple contours on geometry with user-specified cutter side offsets.



Inapplicable situations. Settings in the Offset tab are ignored, and the tab itself is unbolded, in the following circumstances:

- When the item selected for part machining is a body (solid or surface).
- When the GibbsCAM profiler is being used.
- When the Contour process is paired with a Pocket process.

Functionality

Always use markers for single shapes

Selecting this checkbox preserves the behavior of releases prior to GibbsCAM 14. When it is deselected, then machining markers are suppressed.

For information on machining markers, see the topic in the *Mill* guide: "Machining Markers" on page 135.

Centerline / Offset

You can choose Centerline to cut each shape on the tool's center, without offset. Selecting this option disables incompatible controls such as Cut Inside or Climb / Conventional.

Note that the Offset settings provide controls for both Closed Shapes and Open Shapes. This allows you to select closed and open shapes together, knowing that the system will use the correct settings for the corresponding types.

Closed Shapes

- Cut Inside will offset the tool towards the inside of the closed shape. If Alternate Side for Nested Shapes is checked, then the tool will cut inside the outermost shape, outside any shapes nested in that shape, inside shapes nested within those, and so forth, following the same rules as Pocketing does by default.

- Cut Outside will offset the tool towards the outside of the closed shape.

- If Alternate Side for Nested Shapes is checked, the tool will cut outside the outermost shape, inside any shapes nested in that shape, outside shapes nested within those, and so forth, following the same rules as Pocketing does when Outermost Shape as Boss is selected.

Direction:

- Climb (Left) and Conventional (Right) will offset the tool to the left or right of the shape. (If the spindle direction is reversed, then the labels will change accordingly to Climb (Right) and Conventional (Left).) Shape direction is determined from the shape, with one exception: If an open shape has only one end terminator selected, then it is considered the start point regardless of the shape's innate direction.

Open Shapes

- Start Extension and End Extension determine the distance from the start point (or end point) of the shape to the start point (or end point) of the toolpath. If applicable, Start Extension and End Extension values are applied instead of Overlap values that are set the **Contour** tab, so as to retain consistency with legacy behavior where machining markers can disable Overlap values.

Entry/Exit Tab

You can create very customized entry and exit moves for Roughing and Contour operations with the options found on the Entry/Exit tab. By default the options you set will be applied to both the entry and exit moves but you can make the entry and exit moves completely different. With the options found on this tab you can make far more complex entries and exits than the basic options found on the Pocket or Contour tab. You can specify a radius other than 90° and specify a different starting Z value which can result in a ramp and/or helical arc as a part of the entry/exit moves. Additionally you can specify CRC line moves and “off part” line moves. To access these options choose the Advanced option. This tab is bolded when the Advanced option is selected or if you have specified separate entry and exit values. This section will only focus on the options available once you select Advanced.

Rotate Tab

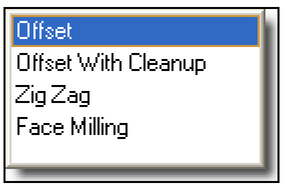
The Rotate tab is available when using a Mill/Turn MDD or a 4-axis or 5-axis MDD. The settings found in this tab allow you to rotate the part or create rotary operations. For more information, see Rotate Tab.

Roughing Process

The Roughing process is used to create pockets and bosses by removing material from the inside of a closed shape or to clear material from the face of a part. Pocketing styles include Offset, Zig Zag, and Face Milling. The following discussion applies to all types of roughing process types. Additional tabs that may or may not apply to the process are discussed elsewhere, because all the basic parameters for roughing a part are covered on the **Pocket** tab.

Pocket milling GibbsCAM in Version 10.*x* and later can produce toolpath that is significantly different from toolpath generated in v9.5 or earlier; the most prominent changes are in the Retracts section. If your parts are several years old, please render the toolpath and check it visually before running in production.

Process Type list



This list menu is where you set the Rough type for the process.

Material

Clicking this button will open the **Materials** dialog, where you can select and edit materials. See the *Common Reference* guide for a full description of the material database.

Speed: RPM

The value entered is the rate of the spindle measured in revolutions per minute. Clicking the button will load a recommended speed from the Material Database based on the part material and tool composition.

Entry Feed

The value entered designates the feedrate, measured in inches per minute or millimeters per minute, for the entry move (from clearance plane to the point where the tool enters the material). Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition. This value is always used for the Z plunge, ramp, or helix.

Note: The lesser of the two feedrates (Entry Feed or Contour Feed) is used as the entry feedrate when the choice for **Entry And Exit**, below, is Line with 90° Radius or (Advanced) Radius Entry or Line Entry.

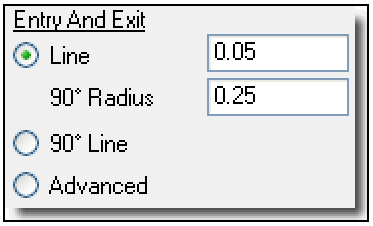
Contour Feed

The value entered designates the feedrate, measured in inches per minute or millimeters per minute, used while cutting. Clicking the button loads a recommended speed from the Material Database based on the part material and tool composition.

Cut Width

The value entered specifies the width the tool will move out on each pass. This value automatically defaults to half of the tool diameter. If the value is made smaller, the passes will overlap. If the value is made larger, areas may be left uncut, especially if the Sharp corners option is selected.

Entry and Exit



The items in the Entry and Exit section let you create additional moves to add to the start and end of the toolpath. There are three options, Line and 90° Radius, 90° Line, and Advanced. Entry/Exit lines are useful when using Cutter Radius Compensation (CRC), because CRC is typically turned on and off on the first and last line moves of the toolpath.

Line and 90° Radius

This option will generate a 90° arc (you specify the radius) will be added at the beginning and end of the toolpath. This arc will be tangent to the start feature at the start point and the end feature at the end point. If a value is entered in the Line text box, a line of the specified length will be created tangent to the arc as the first and last move in the toolpath.

90° Line

When this option is selected a line of the specified length will be added to the toolpath. This line will be perpendicular to the start feature at the start point and the end feature at the end point.

Advanced

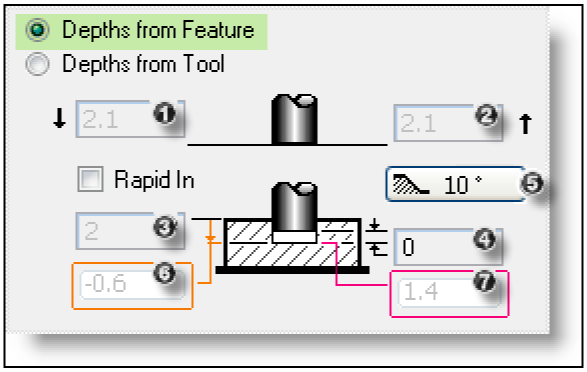
Use this option to create a custom entry and/or exit move. When this option is selected the Entry/Exit tab is bolded. Define the custom entry and exit in the Entry/Exit tab. Use an advanced move as described in the Entry / Exit tab. Refer to Entry/Exit Tab for more information.

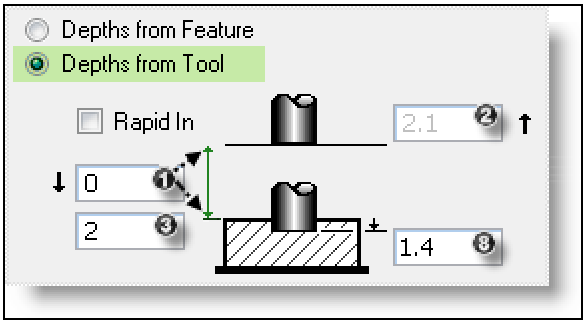
Depths Diagram

The items in this section of the dialog define the clearances and depths for the toolpath. Additionally, the Wall Control option, available on Offset and Zig Zag processes, lets you make 2 1/2 axis cuts.

Depths from Feature / Depths from Tool

These values define the clearances and depths for the toolpath for any parameter set to Absolute, including feature-specific depths like Feature Top Surface Z and Feature Depth Z.





1. CP2, Entry Clearance Plane (or* Incremental distance from plane)
2. CP3, Exit Clearance Plane
3. Surface Z
4. Incremental Tip Z
5. Wall Control (if applicable)
6. Incremental Feature Depth
7. Feature Depth Z
8. Floor Z

Items 4, 6, and 7 are displayed only for Depths from Feature. Item 8 is displayed only for Depths from Tool.

[1] When Approach Z is set to Incremental, the diagram changes slightly and you supply a value for the distance between the Exit Clearance Plane and the Top Surface.

[2] When Retract Z is set to Same As Approach, you cannot supply a value for Exit Clearance Plane.

[3] When Top Surface Z is set to Automatic, you cannot supply a value in the corresponding text box.

[5] The Wall Control button, when present (feature-based milling usually uses straight walls only), lets you make 2 1/2 axis cuts.

[6, 7] When Feature Depth Z is set to Automatic, you cannot supply a value in the corresponding text boxes.

Entry Clearance Plane

Entry Clearance Plane (also called CP2) specifies the location the tool will make a rapid move to before feeding to the start point of the toolpath.

Exit Clearance Plane

The Exit Clearance Plane (also called CP3) specifies the location where the tool might rapid to after completing the toolpath.

Surface Z

The Surface Z specifies the top level of the material.

Incremental Tip Z

Distance from tool tip to the bottom of the feature.

Incremental Feature Depth

Distance from top to bottom of the machining feature.

Feature Depth Z

Z Value of the lowest depth of the machining feature.

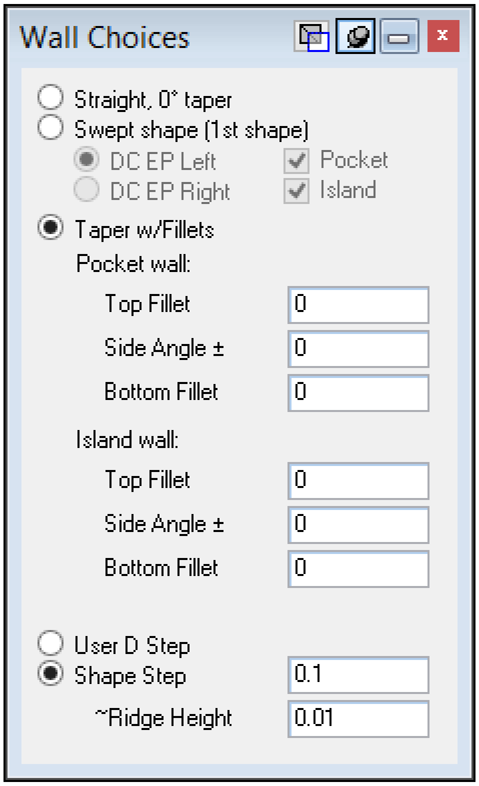
Floor Z

The Floor Z specifies the finished depth of the pocket.

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.

Wall Choices



The Wall Control button brings up a dialog that provides for the creation of 2 1/2 axis surfaces (tapered or swept wall shapes) on pocketing processes. If the wall is tapered, then the button will show the angle of the taper. If the wall is a swept shape, then the button will say “Swept”. The three radio buttons at the top of the dialog determine the type of wall that will be created by the contouring process.

The available choices are Straight, Swept shape, and Taper w/Fillets. The Straight option is the default, and when it is selected no information needs to be entered in this dialog. The information necessary for tapered and swept walls is described below. Additional information is found in this dialog if the contouring process is combined with a roughing process in the Process list. Refer to 2 ½ Axis Surfacing for an example of using this feature.

Swept Shape

When this option is selected, the wall of the pocket will be cut as a swept shape based on the drive curve specified. The DC EP Left (Drive Curve End Point) and DC EP Right selections indicate which side of the base curve cut shape the end point of the drive curve will be located on. This depends on the direction of the cut shape. The cut shape direction is based on whether the tool is making a climb or conventional cut as set with the Machining Markers. The Pocket and Island checkboxes allow the drive curve to be applied to the pocket wall, the island wall, or both.

Tapered Shape

When the Taper w/Fillets option is selected, the walls of the shape will be created with the designated side angle and any radii specified for the top and bottom fillets. You can specify different settings for both the Pocket and Island walls.

User D Step

This option creates a depth step of a specific value. This is an absolute distance in Z that determines the depth of cut on each pass.

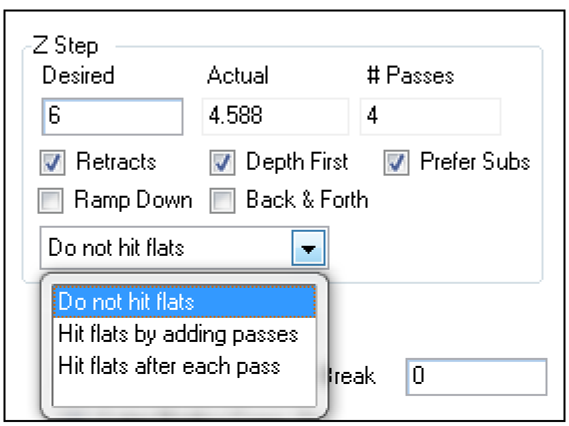
Shape Step

This option generates a parametric step based on the drive curve or taper. This specifies a distance along either the drive curve or taper angle that determines the depth of cut on each pass.

Ridge Height

This parameter is available when creating a tapered wall. In the case where you are applying a Side Angle of the Pocket Wall field, the Shape Step and Ridge Height text boxes are interactive; either value can be entered and the other will be calculated. The Ridge Height (or “scallop height”) is an approximate calculation of the material left on the tapered wall between each pass of the tool. In cases other than a Side Angle for a Pocket Wall, these values will need to be explicitly entered.

Z Step



The items in this section help you define the toolpath behavior when stepping down in Z and transitioning between shapes.

Desired

This is used to specify the depth of each pass. The system uses the Desired Z Step value and the Floor Z value to calculate the Actual Z Step and the # Passes that will need to be made.

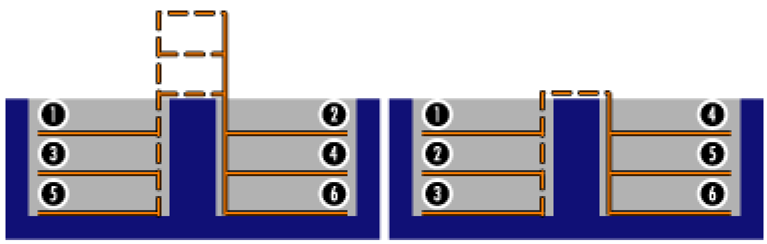
Retracts

Retracts becomes active when multiple passes are being taken when cutting a given shape and the Depth First option is activated. When it is turned on, the tool will rapid up to CP3 (the exit clearance plane) after each pass, and will then rapid to the start point of the next pass. When Retracts is off, the tool will feed from the end point of one pass to the start point of the next pass without retracting up in the Z axis.

With parts imported from previous versions of GibbsCAM, please verify existing Retracts, especially with regard to Air Walls. Retracts may be present that require modified lead moves, and additional Retracts may be needed to prevent gouging.

Depth First

This option lets you specify a preference for machining multiple pockets with more than one Z Step. Activating Depth First will cause the toolpath to completely machine the first item to the final Z depth, and then move onto the next item. By deselecting Depth First, you tell the system to first machine all selected items at the first Z step. Once the first level is complete on all selected items, the operation starts over at the first pocket or contour and begins to cut at the second Z step. This will continue until the operation is complete.



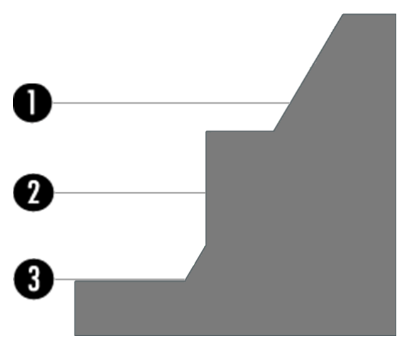
Example of machining without Depth First versus with Depth First

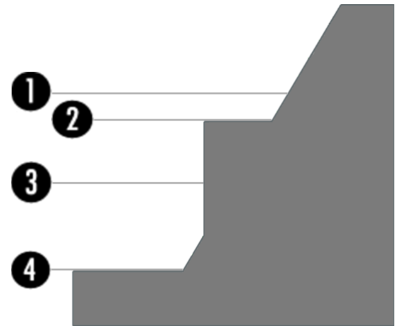
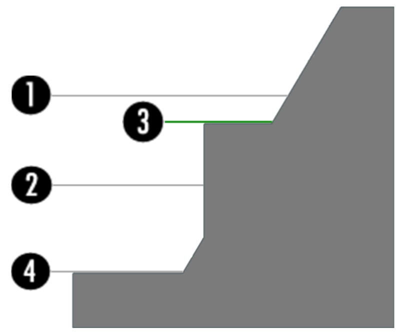
Prefer Subs

This checkbox provides the user with the option of using subprograms in the posted code. Activating this item produces shorter G-code output.

Hit Flats (dropdown)

This option modifies the Z Step so that a pass is taken at each flat surface, such as a boss top or the pocket floor. The Z Step is recalculated for this option, and the step will vary to hit the flats. Therefore the Z step will not match the value shown under Actual.

Do not hit flats	Flat surfaces will be machined only if the Z Step, plus or minus any stock, happens to coincide with the flat surface, as it does in this illustration in step #3.	

Hit flats by adding passes	After performing a regular depth cut (Z Step), the tool will add one or more Z Steps (in this case, for #2) to machine flat surfaces after they are newly exposed by the cut.	
Hit flats after each pass	A Z Step will be added to machine each flat, as illustrated; the flat at #3 is machined after the regular step #2. If you want to hit all flats from the bottom up, simply set your Desired Z Step to produce one pass.	

Other Common Controls

Coolant

The 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.

Pattern

When the Pattern checkbox is selected, the process will create identical toolpaths in different locations on the part. The toolpath generated will be cut once for each point in the selected pattern workgroup. The pattern workgroup, which is selected from the adjacent pop-up menu, contains unconnected, plain points that serve as origin points for the toolpath created by the process. The original toolpath created will NOT be cut unless the origin point for that toolpath is included in the pattern workgroup. Posted output will create one subprogram for the primary toolpath and call that subprogram once for each point in the pattern workgroup. For more information, see Pattern.

Mach. CS

The Mach. CS drop-down list appears on this tab when a 3-axis MDD is active. For more information, see Mach. CS .

Offset and Zig Zag Processes

Entry Styles

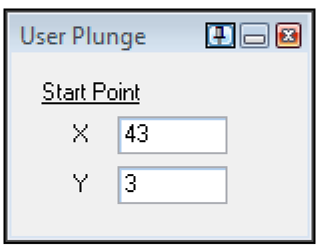
Tool Entry

This pop-up menu allows the user to designate how the tool will enter the material. There are four options, Auto Plunge, Plunge, Ramp and Helix. Auto Plunge is a system-controlled entry while the other items require user input to specify where and how the tool will enter the material. All options are described below.

Auto Plunge

If the Auto Plunge option is selected, the system determines the best location for the tool to plunge into the part based on the toolpath created by the operation. Additionally, if there is a Holes process preceding the Roughing tile in the Process list, Auto Plunge will automatically control where the drill will make an entrance hole.

Plunge



If this option is selected, the tool will plunge into the material at the X and Y coordinates entered. The tool will feed from the Entry Clearance Plane to the specified start point. The tool will then feed to the start point of the toolpath. This option is useful if there are pre-existing holes in the stock where the tool can enter before moving to the start point of the toolpath. It should only be used when only one pocket will be created by the process.

Ramp

A Ramp may be specified instead of using a pre-drill entry. The tool will ramp in from the start point determined at the depth specified in Z SP. The tool will ramp at the angle specified by the XY Ramp Angle and have a slope equal to the value specified for the Slope in the Z/Inch text box. The Angle text box allows the user to specify the angle of the entry cut taken in a Ramp entry operation. The Slope value and the Angle value are interactive. If the Angle is set, the Slope will be automatically calculated and vice versa. The Max Cut is the maximum depth of cut in Z the tool can take and the Wall Clearance specifies the distance the tool must stay clear of the finished wall. The system will verify that the ramping moves do not violate any pocket geometry.

Helix

The Helix dialog allows a helical entry in Z.

Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the helix. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the helix.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Max Cut

This is the total amount of movement in Z that is allowed in a full rotation of the helix.

Slope Z/(inch/mm)

This is the ratio of movement in Z relative to 1 unit of XY movement.

Angle

This value allows an entry angle to be specified instead of the Slope ratio.

Wall Clearance

Distance the Helix should remain from a finish wall in addition to any values specified in the Process.

Auto, Helix end at SP

The helix will end at the start point of the pocket.

Auto, Helix center at SP

The helix will end at the floor Z of the pocket and the tool will move from the end point of the helix to the start point of the pocket. The center line (or center point) of this helical circle is centered on the pocket start point.

Center at SP

The pitch of the helix will be equal to the value in the Max Cut box and the diameter will be determined by the value in the Diameter box. The helix will end at the finish Z of the pocket and the tool will move from the end point of the helix to the start point of the pocket. The movements may violate some pocket geometry.

Center at XY Position

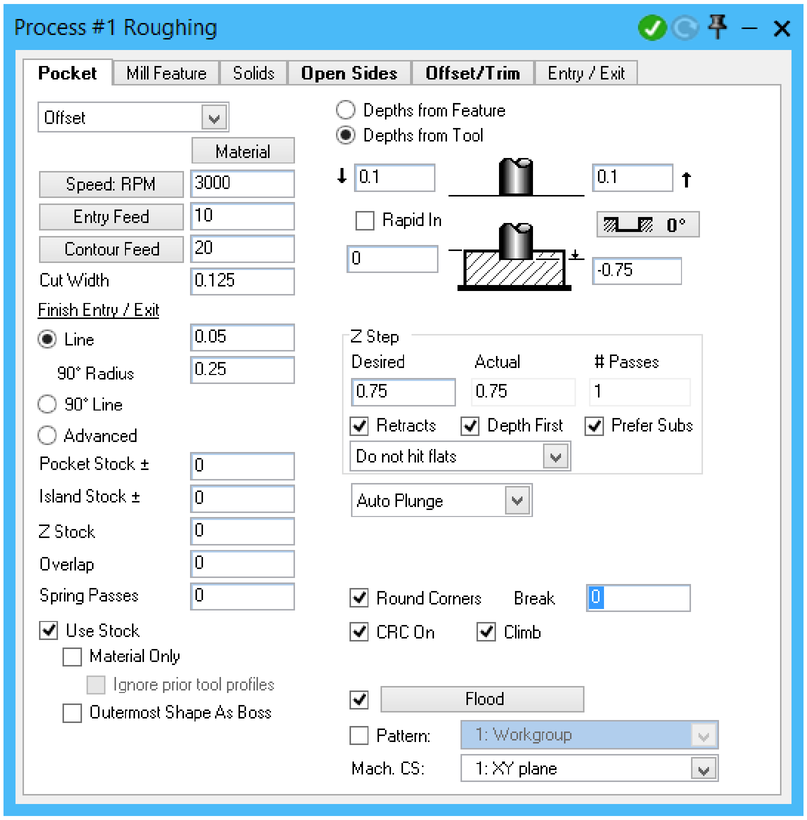
Allows the helix center to be defined by X and Y values. A Diameter must be specified. The movements may violate some pocket geometry.

Helix end at SP

The center of the helix will be determined by the value specified for the CP at angle and Diameter. A Diameter must be specified. The movements may violate some pocket geometry.

Offset and Offset With Cleanup Processes

“Offset” processes are the standard pocketing operations with concentric toolpath. “Offset with Cleanup” processes create an Offset toolpath with the corners extended to clean areas that didn’t receive much of the tool in the first pass. This section details options that are common to the Offset-type processes. The speeds and clearances are covered in Roughing Process .



Finish Entry and Exit

The items in the Entry and Exit section let you create additional moves to add to the start and end of the toolpath. There are three options, Line and 90° Radius, 90° Line and Advanced. Entry/Exit lines are useful when using Cutter Radius Compensation (CRC) because CRC is typically turned on and off on the first and last line moves of the toolpath.

Line and 90° Radius

This option will generate a 90° arc (you specify the radius) will be added at the beginning and end of the toolpath. This arc will be tangent to the start feature at the start point and the end feature at the end point. If a value is entered in the Line text box, a line of the specified length will be created tangent to the arc as the first and last move in the toolpath.

90° Line

When this option is selected a line of the specified length will be added to the toolpath. This line will be perpendicular to the start feature at the start point and the end feature at the end point.

Advanced

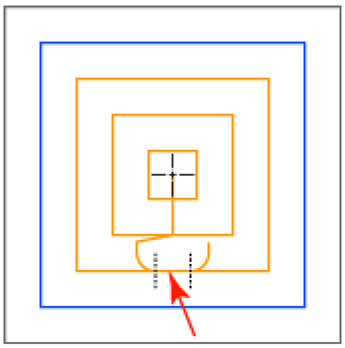
Use this option to create a custom entry and/or exit move. When this option is selected the Entry/Exit tab is bolded. Define the custom entry and exit in the Entry/Exit tab. Use an advanced move as described in the Entry / Exit tab. Refer to Entry/Exit Tab for more information.

Pocket Stock ±
This is the amount of material that will be left on the wall of the pocket (per side). A positive stock amount will leave material on the pocket or island and a negative stock amount will cut into the pocket geometry.

Island Stock ±
This is the amount of material that will be left around any bosses (per side) that are contained in the pocket and are selected as part of the roughing cut shape. A positive stock amount will leave material on the pocket or island and a negative stock amount will cut into the pocket geometry.

Z Stock
This is the amount of stock in the Depth you wish to remain or remove. A negative value will cut deeper into the stock by the amount specified.

Overlap



An Overlap value will force the end point to extend past the start point by the specified amount. This is very useful for Cutter Compensation.

Spring Passes
The number entered is the number of extra times the final pass will be made. In operations with multiple Z depth passes, the tool will retract to the entry clearance plane defined in the Roughing process.

Use Stock
Toolpath generated by the system can be optimized in various ways by using the Use Stock, Material Only and Ignore Tool Profile options. These options are hierarchical–one depends on another being active. If Use Stock is active, then Material Only is available and if Material Only is active, then Ignore Tool Profile is available.

When Use Stock is active, toolpath will be confined to the current stock definition even if the part extends past the stock. The only exception is any value defined in the Open Pocket dialogs, which specifically allow a tool to move beyond the stock.

When Use Stock is active the system can generate toolpath even when there is no selected geometry or body. This is similar to Face Milling. The system will generate a pocket based on the existing stock. This can be the stock as defined in the Document dialog, a workgroup or a solid. The pocketing operation will cut from the Surface Z to the Final Z depth. This function is aware of fixtures if the SolidSurfacer option is installed.

Material Only

Available only when Use Stock is in effect. Material Only optimizes toolpath by limiting the toolpath to areas that have material. If a part has already been partially machined, Material Only will optimize the cut areas and ensure there will be no “cutting air.” More information can be found in Material Only.

Ignore Tool Profile

Ignore Tool Profile will cause Material Only operations to ignore the tool shapes of preceding operations. This is useful when re-machining with a tool that has a corner radius equal to or greater than that of prior tools. When Ignore Tool Profile is activated, Material Only pretends that all mills are sharp endmills. When a part is defined by 2D geometry only, it is recommended that Ignore Tool Profile be activated as material left on 2D walls can be easily visualized by the system.

Deselecting Ignore Tool Profile makes things a bit more complex. First of all, the remaining material is more accurate, factoring in all tool tapers and corner radii of the tools in prior operations. If you have a roughing tool with a large corner radius and a finishing tool with a smaller corner radius that will be cleaning up material left on the floor by the larger tool, be sure to turn off Ignore Tool Profile. Leaving this option off is also best for machining non-2D parts, such as a pocket in a solid with bottom fillets.

Outermost Shape As Boss

Available only when Use Stock is in effect. Applies to 2D geometry only. When this checkbox is selected, the system will regard remaining stock to be machined as a boss (rather than as a pocket with one or more air walls), and will therefore remove material outside the outermost loop. Sample part: `Outermost_Shape_As_Boss.vnc`.

Feed Entry Type

This menu allows you to select how the tool will feed into the part. By default the tool plunges (Auto Plunge) but you may select a user-defined Plunge, Ramp or Helix entry.

Auto Plunge

If the Auto Plunge option is selected, the system determines the best location for the tool to plunge into the part based on the toolpath created by the operation. Additionally, if there is a Holes process preceding the Roughing tile in the Process list, Auto Plunge will automatically control where the drill will make an entrance hole. See Pre-Mill Tab for more information on controlling the drill points.

Plunge

If this option is selected, the tool will plunge into the material at the X and Y coordinates entered. The tool will feed from the Entry Clearance Plane to the specified start point. The tool will then feed to the start point of the toolpath. This option is useful if there are pre-existing holes in the stock where the tool can enter before moving to the start point of the toolpath. It should only be used when only one pocket will be created by the process.

Ramp

Selecting this option will let you define a ramping motion when entering the part.

Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the ramp. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the ramp.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Max Cut

This value is the maximum Z step that the tool can take. The value is equal to twice the Z depth of a single ramping move, i.e. it is the total depth of the zig and the zag in a ramping move. This value controls the Ramp Length based on the current Slope and Ramp Angle.

Slope: Z/Inch or Z/mm

This value specifies the slope of the ramp. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY. Specifying the Slope will calculate the Ramp Angle and Ramp Length values based on the current Cut value.

Ramp Angle

This is the angle of descent for the ramping motion. Specifying this value will calculate the Slope and Ramp Length based on the current Cut value.

Wall Clearance

This value specifies the distance the tool must stay clear of the finished wall. The system will verify that the ramping moves do not violate any pocket geometry.

XY Ramp Angle

The ramp angle determines the starting angle for ramping into the part. You can let the system choose, specify to start along the X or Y axis or specify a particular angle.

Round Corners

This checkbox lets you specify how the system will handle the external corners of a contour. When the Round Corners option is selected, the system will add a radius move to the toolpath at every external corner of the cut shape. The tool always stays in contact with the finished shape and does not create burrs at the corners. Sharp corners can be created when this option is on by entering a corner Break of 0. When the Round Corners option is off, no radius move will be created.

Break

The value entered in this text box specifies a radius that will be put on every external corner of the selected cut shape. It will only be available only if the Round Corners option is active. Operations that include a corner break value should not be used prior to a Material Only operation. Material Only assumes the part shape is always equal to or smaller than the material at all times. This will be true unless the corner break is used because corner break cuts a radius onto a sharp corner, which can cause inaccurate Material Only calculations.

CRC On

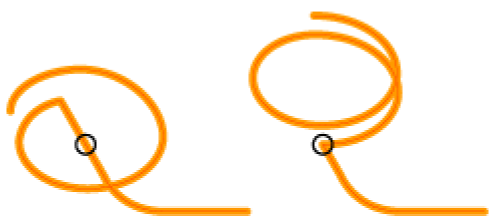
A checkbox that indicates whether Cutter Radius Compensation is turned on or off. Most CNC machines require that CRC be turned on for Entry line moves and turned off for Exit line moves. We recommend not using CRC during Roughing operations when the From Tool Edge option in the Machining Preferences is being used. If From Tool Edge is selected, the toolpath lines are still displayed as “tool center”. Therefore in Roughing operations with the From Tool Edge item has no effect on the output.

Climb

This checkbox lets you specify the direction the tool will travel, either making a climb cut or a conventional cut. When checked, the system will generate climb cuts. When unchecked, the system generates conventional cuts.

Conventional Climb

Helix



Selecting this option will let you define a helical motion when entering the part. This option is available for Contour, Offset and Offset with Cleanup. Not all options are available for each mode.

Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the helix. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the helix.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Cut (Max Cut)

This value is the maximum Z step that the tool can take. The value is equal to the Z depth of a fill 360° helical revolution. This value controls the Diameter based on the current Slope and Angle.

Slope: Z/Inch or Z/mm

This value specifies the slope of the helix. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY The XY distance is measured along the circumference of the helix. Specifying the Slope will calculate the Angle and Length values based on the current Cut value.

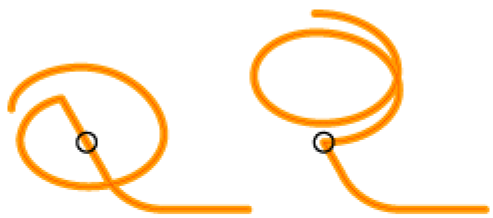
Angle

This is the angle of descent for the helical motion. Specifying this value will calculate the Slope and Length based on the current Cut value.

Diameter

This value is the diameter of the helix. This value controls the Cut based on the current Slope and Angle.

Helix Location



Center at SP and End at SP examples

This setting specifies where the helix should be situated relative to the tool's entry position. Center at Entry SP creates the helix so its center is at the start point and an additional move from the helix end to the start point will be generated. Helix End at Entry SP generates the helix so its endpoint is at the same position as the start point for the rest of the toolpath. This eliminates the move from the helix center to the start point.

Wall Clearance

Distance the Helix should remain from a finish wall in addition to any values specified in the Process. This is only available with either of the Auto Helix options.

Helix OD

These options let you define where the helix will be positioned relative to the start point of the toolpath.

Center at SP

The pitch of the helix will be equal to the value in the Max Cut box and the diameter will be determined by the value in the Diameter box. The helix will end at the finish Z of the pocket and the tool will move from the end point of the helix to the start point of the pocket. The movements may violate some pocket geometry.

Center at XY Position

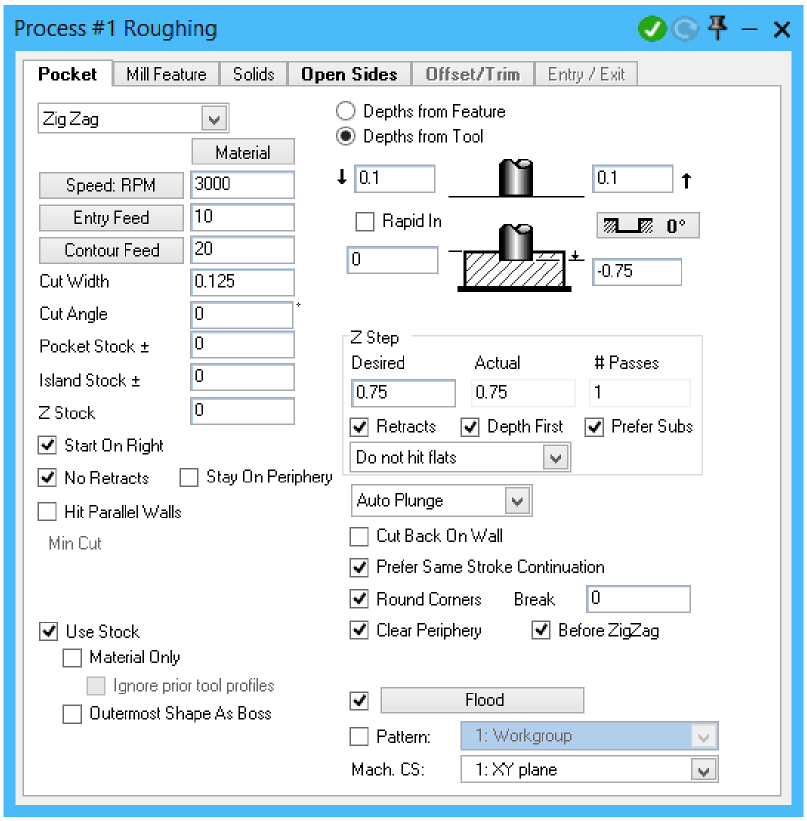
Allows the helix center to be defined by X and Y values. A Diameter must be specified. The movements may violate some pocket geometry.

Helix end at SP

The center of the helix will be determined by the value specified for the CP at angle and Diameter. A Diameter must be specified. The movements may violate some pocket geometry.

Zig Zag

Rough option Zig ZagK creates straight lines at a specified angle and contour moves at the boundaries.



Cut Width

The value entered specifies the width the tool will move out on each pass. This value automatically defaults to half of the tool diameter. If the value is made smaller, the passes will overlap. If the value is made larger areas may be left uncut.

Cut Angle

The value entered specifies the angle of the Zig Zag.

Pocket Stock±

This is the amount of material that will be left on the wall of the pocket (per side). A positive stock amount will leave material on the pocket or island and a negative stock amount will cut into the pocket geometry.

Island Stock±

This is the amount of material that will be left around any bosses (per side) that are contained in the pocket and are selected as part of the roughing cut shape. A positive stock amount will leave material on the pocket or island and a negative stock amount will cut into the pocket geometry.

Z Stock

This is the amount of stock in the Depth you wish to remain or remove. A negative value will cut deeper into the stock by the amount specified.

Start On Right

When active, the first stroke of the Zig Zag toolpath will be on the right side of the part. When inactive, the first stroke will be from the left side of the part. This option is on by default.

No Retracts

When active, the system will generate Zig Zag toolpath that does not retract in order to avoid obstacles in a pocket during a Z Step. Instead it will follow one of two paths set by the Stay On Periphery option. When No Retracts is off, the toolpath generated for each Z step will retract over obstacles between regions of the same pocket. This option is on by default.

When a Zig Zag pocketing operation with No Retracts starts on an Air feature the tool will feed down onto the part without using the Open Pocket Clearance value.

Stay On Periphery

When this option is on, a tool will travel along the periphery of a pocket to connect to the next region to be cut. When this option is off, the system will make a direct connection to the next region to be cut with full gouge protection. This option is on by default.

The following images are examples of the No Retracts option. Image #1 illustrates the default setting, No Retracts is on and Stay On Periphery is off. Note that the toolpath is taking the shortest path possible to the next cut region without gouging the part by wrapping around the boss. Image #2 illustrates toolpath with No Retracts enabled and Stay On Periphery on. Note that the toolpath is travelling around the edge of the part. Image #3 illustrates toolpath with No Retracts off; thus, the tool will rapid up and over the boss and feed down to the starting point of the next region to be cut in this Z step.

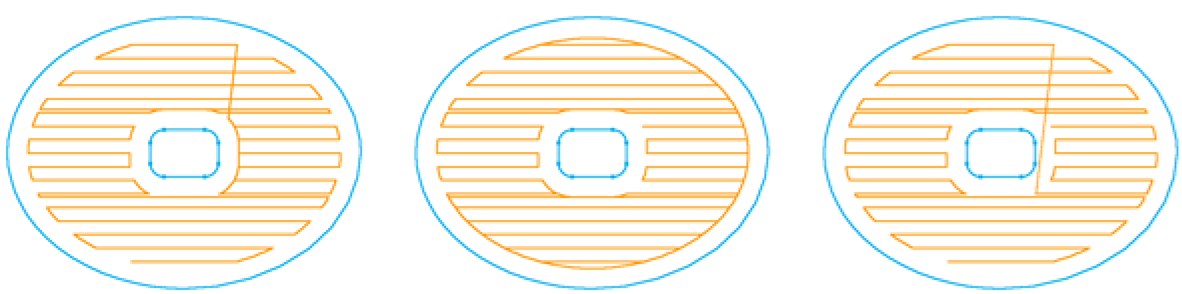


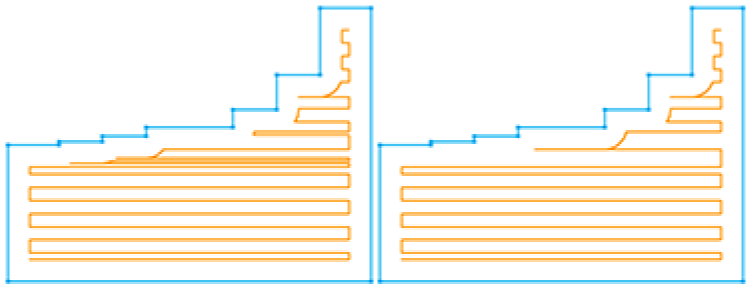
Image #1	Image #2	Image #3

Hit Parallel Walls

When active, the system will machine walls that are parallel to the natural stroke whose distance between each other is greater than the Min Cut. Hit Parallel Walls will adjust the Cut Width on the last pass. This option is off by default.

Min Cut

This value should be less than the Cut Width setting. The wall must be exactly parallel to the Cut Angle. The larger the value, the more the system ignores walls that are close to each other. The images below show a part where the Min Cut value is set low (so more walls are cut) and the same process with a higher setting, creating toolpath that is more optimized.



Examples of low and high Min Cut settings

Hit Parallel Walls Example
A 70mm wide pocket is to be cut with a 30mm tool. In two passes at a cut width of 30mm the tool would cut an area 60mm wide leaving 10mm of material on the pocket. By turning Hit Parallel Walls on and inputting the required Min Cut value of less than 10mm (the remaining amount of material) will force the Zig Zag operation to adjust its cut width for this last pass. By doing this the tool will cut to the wall and make a pocket that is 70mm wide.

Clear Periphery

Activating this option will generate toolpath that includes a single contour pass around the selected shape. This pass may be generated prior to or after each Zig Zag Z step.

Before Zig Zag

Leaving Before Zig Zag off will generate a contour pass after each Zig Zag pocketing Z step. Activating Before Zig Zag will cause the single pass to be made prior to the Zig Zag cut.

Use Stock

Toolpath generated by the system can be optimized in various ways by using the Use Stock, Material Only and Ignore Prior Tool Profile options. These options are hierarchical: one depends on another being active. If Use Stock is active, then Material Only is available (but does not save the cut region), and if Material Only is active, then Ignore Prior Tool Profile is available.

When Use Stock is active, toolpath will be confined to the current stock definition even if the part extends past the stock. The only exception is any value defined in the Open Pocket dialogs, which specifically allow a tool to move beyond the stock.

When Use Stock is active the system can generate toolpath even when there is no selected geometry or body. This is similar to Face Milling. The system will generate a pocket based on the existing stock. This can be the stock as defined in the Document dialog, a workgroup or a solid. The pocketing operation will cut from the Surface Z to the Final Z depth. This function is aware of fixtures if the SolidSurfacer option is installed.

Material Only

This option depends upon Use Stock being active. Material Only optimizes toolpath by limiting the toolpath to areas that have material. If a part has already been partially machined, Material Only will optimize the cut areas and ensure there will be no “cutting air.” More information can be found in Material Only.

Ignore Prior Tool Profile

Ignore Prior Tool Profile will cause Material Only operations to ignore the tool shapes of preceding operations. This is useful when re-machining with a tool that has a corner radius equal to or greater than that of prior tools. When Ignore Profile Tool Profile is in effect, Material Only pretends that all mills are sharp endmills. When a part is defined by 2D geometry only, it is recommended that Ignore Prior Tool Profile be activated, as material left on 2D walls can be easily visualized by the system.

When Ignore Prior Tool Profile is deselected, the result is more complex. For example: The remaining material is more accurate, factoring in all tool tapers and corner radii of the tools in prior operations; therefore, if you have a roughing tool with a large corner radius and a finishing tool with a smaller corner radius that will be cleaning up material left on the floor by the larger tool, be sure to deselect Ignore Prior Tool Profile. Turning this option off is also best for machining non-2D parts, such as a pocket in a solid with bottom fillets.

Feed Entry Type

This menu allows you to select how the tool will feed into the part. By default the tool plunges (Auto Plunge), but you may select a user-defined Ramp or Periphery Ramp entry.

Auto Plunge

If the Auto Plunge option is selected, the system determines the best location for the tool to plunge into the part based on the toolpath created by the operation. Additionally, if there is a Holes process preceding the Roughing tile in the Process list, Auto Plunge will automatically control where the drill will make an entrance hole. See Pre-Mill Tab for more information on controlling the drill points.

Ramp

Selecting this option will let you define a ramping motion when entering the part.

Z SP (Z Start Point)

This is an increment to Surface Z that tells the system where to start the ramp. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the ramp.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Max Cut

This value is the maximum Z step that the tool can take. The value is equal to twice the Z depth of a single ramping move; i.e. it is the total depth of the zig and the zag in a ramping move. This value controls the Ramp Length based on the current Slope and Ramp Angle.

Slope: Z/Inch or Z/mm

This value specifies the slope of the ramp. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY. Specifying the Slope will calculate the Ramp Angle and Ramp Length values based on the current Cut value.

Ramp Angle

This is the angle of descent for the ramping motion. Specifying this value will calculate the Slope and Ramp Length based on the current Cut value.

Wall Clearance

This value specifies the distance the tool must stay clear of the finished wall. The system will verify that the ramping moves do not violate any pocket geometry.

Periphery Ramp

This option generates a continuous ramping motion around the shape's perimeter, similar to a helical entry.

Z Start Point

This is an increment to Surface Z that tells the system where to start the ramp. If this value is negative, the tool will plunge to a position lower than Surface Z before it starts the ramp.

Note: In releases before GibbsCAM 2013 v10.5, this was an absolute value, not incremental. The change at v10.5 and later makes it consistent with other parameters and accommodates Mill Feature. For existing parts, the adjustment from absolute to incremental is made automatically when the part is opened. When Save a Copy is used to save to v10.3 or earlier, the value is converted to absolute.

Slope Z per Inch/MM

This value specifies the slope of the ramp. A value of 1 will move the tool down 1 unit in Z for every unit of movement in XY. A value of 0.25 will generate a slope where the tool will move down 1 unit in Z for every 4 units of movement in XY. Specifying the Slope will calculate the Ramp Angle.

Ramp Angle

This is the angle of descent for the ramping motion. Specifying this value will calculate the Slope and Ramp Length.

Cut Back On Wall

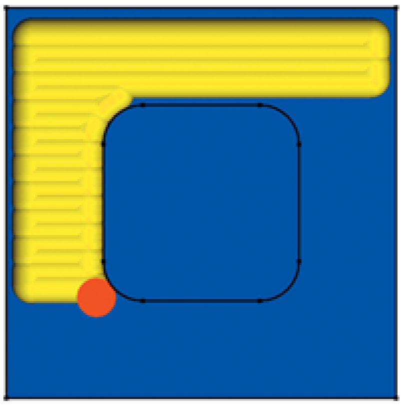
When this option is active, the system will generate toolpath that will cut backwards along the periphery to overlap the previous stroke before making the next stroke. This will clean up scallops left by the toolpath. This is one of two options which can be thought of as a general preference for any Zig Zag operations.

Prefer Same Stroke Continuation

This option applies to pockets that will have several sections or areas to be machined. Such a pocket might have a boss in the middle of it. When the tool encounters the boss, there are two options for continuing the toolpath. This is one of two options which can be thought of as a general preference for any Zig Zag operations.

When this option is active, the toolpath generated will continue the stroke, avoiding obstacles but cutting as much as possible around the obstacles. When the toolpath has covered as much area as it can, it will move to uncut areas.

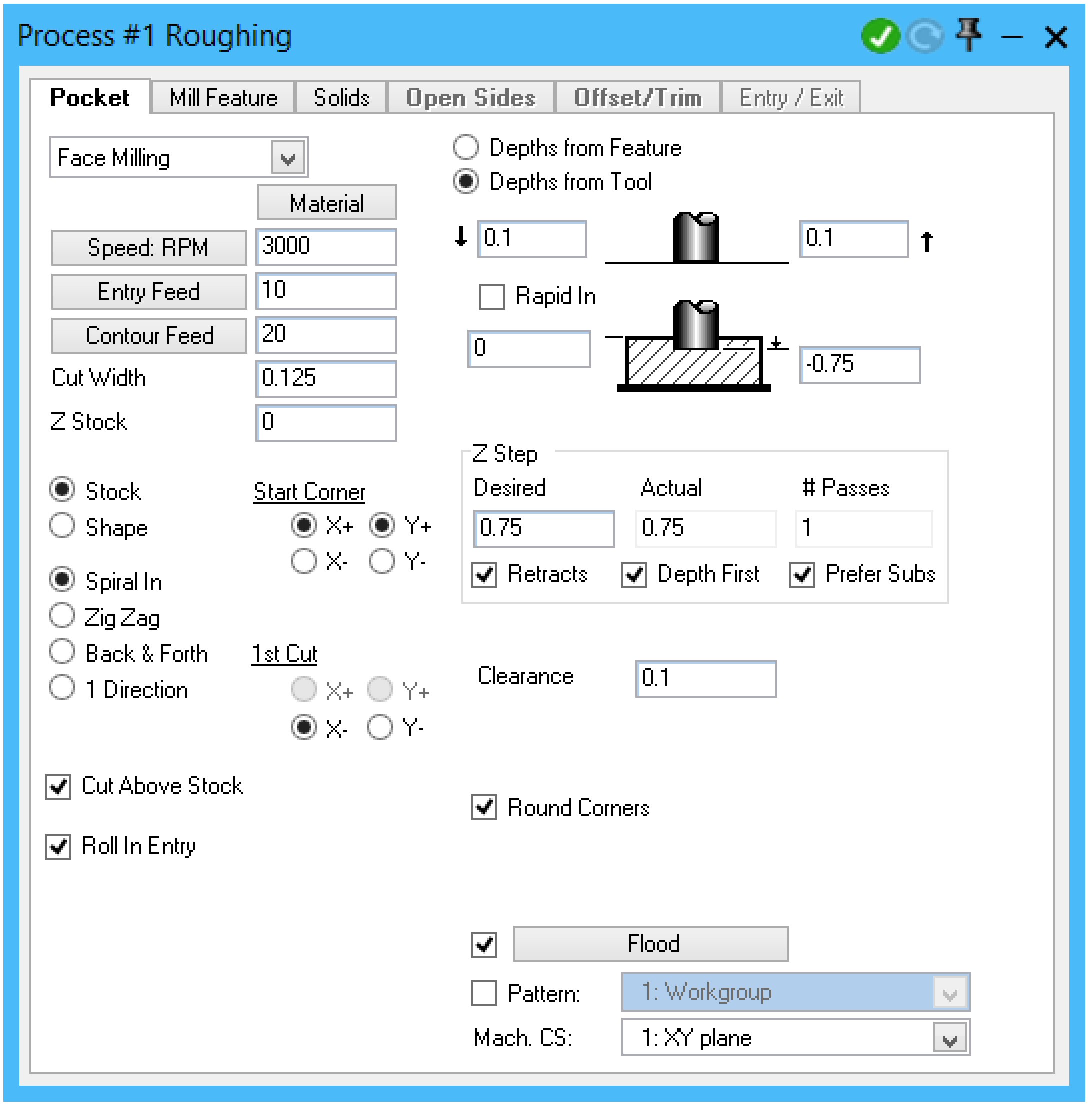
When this option is off, the toolpath will move around or over the obstacle to machine the area blocked by the obstacle and then continue clearing out the pocket. This option is on by default.



The image illustrates a situation where this setting would apply. If Prefer Same Stroke Continuation is on, the tool will continue its stroke below the boss. If this option is off, the tool will move over or around the boss to machine the area that was obstructed by the boss.

Face Milling

The Face Milling item completely automates clearing material from the face of a part. When the Face Milling option is selected, the bottom portion of the Roughing dialog changes as shown. Each of the Face Milling features is described below. The other items contained in the dialog function as they do when performing an offset Roughing operation. Please note that Face Milling operations will not avoid defined fixture bodies.



Cut Selection

If the Stock option is selected, no geometry needs to be selected. The process will face the entire stock shape.

If the Shape option is selected, a closed shape must be selected. The system will face the selected closed shape. The Profiler  can also be used to select a shape.

Multiple nested shapes are handled efficiently: All shapes with overlapping bounding boxes are now machined together in the requested cut order, using Clearance values and Roll In to transition between shapes as needed.

Nesting of pockets and bosses allows for rapiding over large voids in the part, using the “Outermost Shape as Boss” approach.

Shapes that are independent, in the sense that their bounding boxes do not intersect, continue to be machined separately.

Cut Options

These selections indicate how the toolpath will be generated. They determine the finish of the cut and the length of the roughing cycle. Each selection is described below.

Spiral In

This option generates the fastest roughing cycle, but produces a rougher surface finish than some of the other options. The tool starts off the part and ends on the part. The tool spirals into the material and makes a square pattern to remove the material.

Zig Zag

This option also generates a fast roughing cycle, but a rougher finish. The tool starts off the part and ends off the part. The tool zig zags across the material alternating between climb cutting and conventional cutting.

Back & Forth

This option produces a better surface finish because the tool is always climb cutting. The toolpath alternates cutting from both ends of the part.

1 Direction

This option produces the best surface finish but generates a slower roughing cycle. The tool makes one pass across the part, rapids up and back across the part, and then makes each additional pass to clear off the necessary material.

Fixture Avoidance behavior for Cut Options

Unlike other machining methods, face milling is not normally able to discontinue and resume cutting. Normal mechanisms for avoiding fixtures, such as cutting around the fixture, or retracting over it and plunging, are not desirable. To address this, avoidance behavior is based on the selected strategy:

- For Spiral In: the cut region is reduced to avoid the fixture without introducing concavities. Some areas of the part may be left unmachined.
- For Zig Zag: each cut terminates where it intersects a fixture and immediately transitions to the next cut going the opposite direction. Neither cut will machine the part on the other side of the fixture.
- For Back & Forth and 1 Direction: each cut terminates where it intersects the fixture, and the cut will not be resumed. Instead, the system immediately retracts and transitions to the next entry point.

For all strategies, use appropriate fixture clearance. If a part body is selected, collisions with unselected parts of that body should be avoided just like collisions with fixtures, consistent with ordinary solid machining.

Start Corner

These radio buttons determine the start point of the toolpath. The selections indicate in which corner of the shape, whether it be the entire stock shape or a selected closed shape, that the tool will start cutting in. For example, selecting X+, Y+ will begin the toolpath in the upper right hand corner; X+, Y- will begin the toolpath in the lower right hand corner; and so on. The four possible combinations represent the four quadrants.

1st Cut

The available choices change depending on the Start Corner selections. These buttons determine the direction of the first cut from the start corner of the face milling operation. For example, if an operation was set to start in the X+, Y+ quadrant, the tool may make its first move in either the X- or Y- direction. These will be the only choices available.

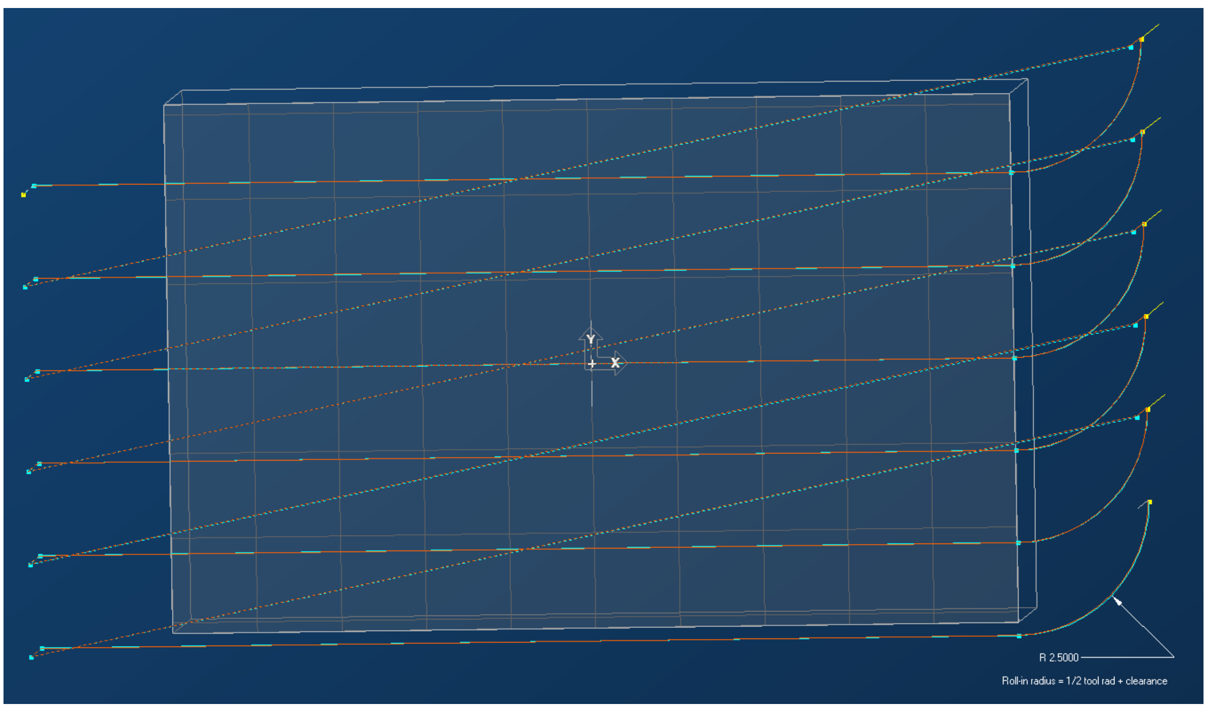
Roll-In Entry

A 90-degree roll-in entry is recommended to shape chips appropriately, to improve tool life, and to reduce chatter. For a newly created part, Roll In Entry is on by default; or it can be chosen for older parts opened in the current release.

When Roll In Entry is chosen, a roll-in arc is attached to each move from off-part if the move has clearance applied. This includes:

- the initial entry for all strategies
- all trimmed passes (so long as the roll-in arc would not plunge within another portion of the profile)
- every normal stroke for choices Back & Forth and 1 Direction

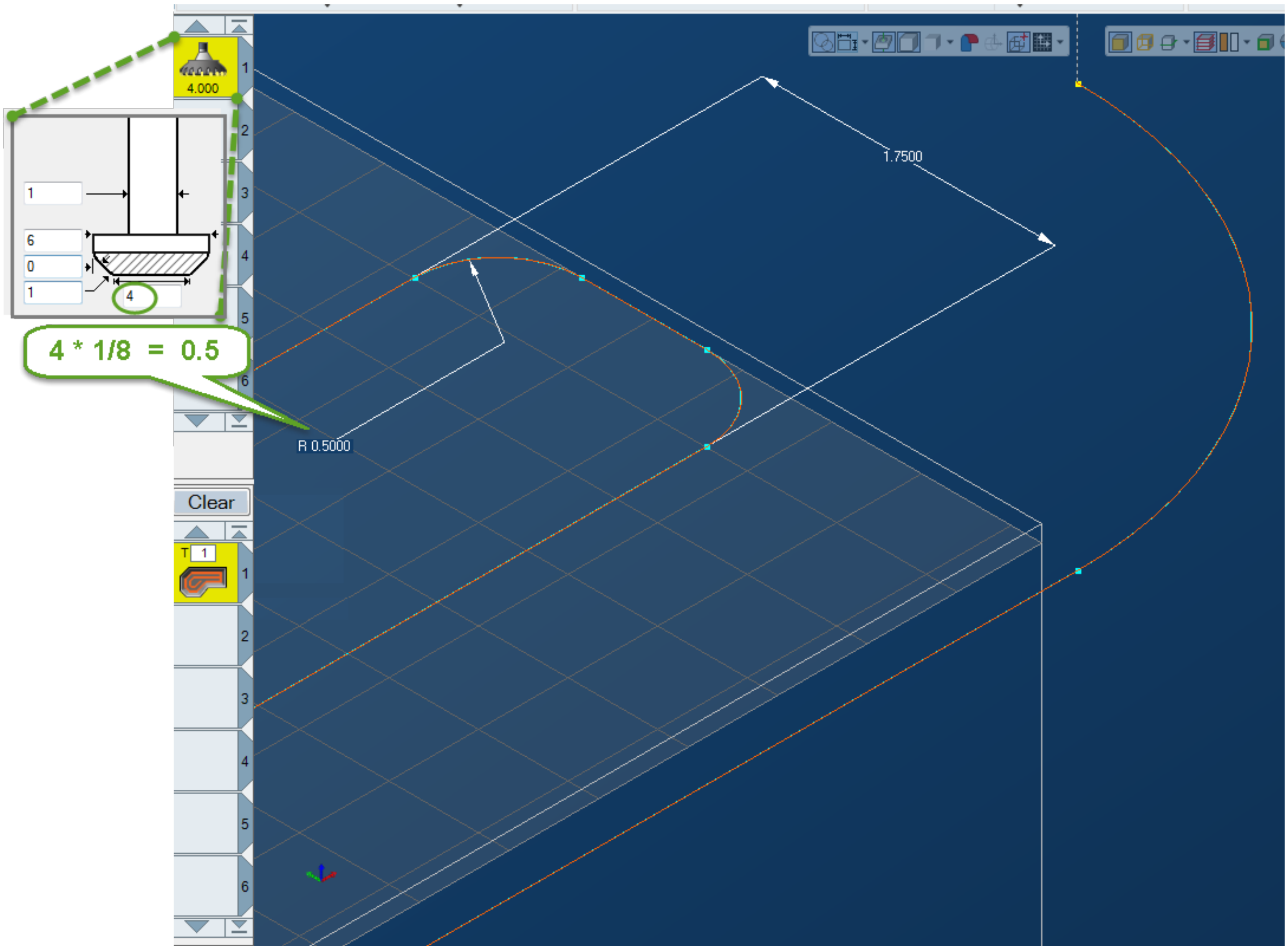
The arc direction produces an inside-cutting corner based on the cutting direction: the arc is clockwise for forward-rotating tools. The radius of the arc is the tool radius plus the clearance value, so that the arc ends with the tool center at the material edge.



Round Corners

Round corners are recommended for face milling while the tool is engaged, such as at every corner for strategies Spiral In or Zig Zag, which change direction while engaged. For a newly created part, Round Corners is on by default for those strategies; or it can be chosen for older parts opened in the current release.

The radius for the corner arc is calculated as 1/8 (12.5%) of the tool diameter. The user must ensure an appropriate stepover size when using this option. In particular, strategy Spiral In might require smaller radii on passes very close to the center, and smaller radii might also be required near trimmed ends that are shorter than the normal round-corner radius. In such cases, use the largest radius that fits.



Cut Above Stock

Common tasks like face milling are often part of saved processes. Such processes can be set up to cut significant amounts of material that might or might not be present on the part, using Material Only to eliminate air cutting. Prior to this release, face milling ignored the Z parameter of the default stock definition (while correctly avoiding air cuts against solid stocks).

Cut Above Stock is turned on by default for both newly created parts and older parts opened in the current release. You can turn it off to generate no cuts in the air above the stock, regardless of the Maximum Z value.

Clearance

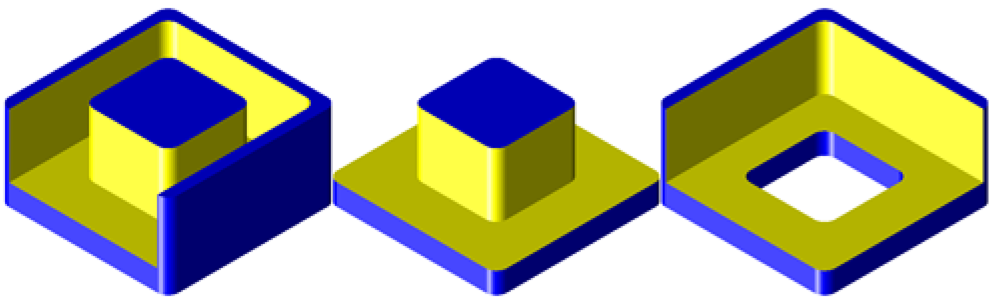
The Clearance amount entered is an XY offset value that is added to the beginning of the toolpath. The toolpath is always offset from the stock shape or selected geometry by a tool radius. The Clearance value is added to the tool radius offset amount.

Solids Tab

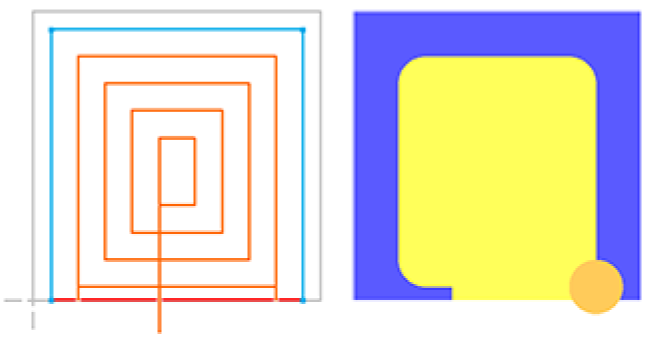
This tab is bolded when a body is selected. The contents of this tab only affect machining solids and sheets. For more information on these items, see the *2.5D Solids* or *SolidSurfacer* guides.

Open Sides Tab

An “open side” is a pocket that does not have a complete set of sides, such as a square pocket with only three sides. The open pocket settings also relate to toolpath behavior when encountering holes in the stock. In general, open pocket settings affect the behavior of toolpath at the edges of a pocket.



Overhang



Open Sides > Overhang governs behavior when the tool moves parallel to the part edge. This parameter specifies the amount by which the tool will overlap an Air feature to clean up edges that might otherwise have a ridge. The value is measured from the tool’s outer edge to the Air wall. If no value is entered, the system will automatically overhang the tool on Air geometry by the tool’s cutting radius.

The recommended value for overhang is the tool radius. The maximum is a value equal to the tool diameter minus a small adjustment (0.001" = 0.0254mm) to ensure that the tool does not cut only air

An automatic Air/Wall corner cleanup capability is implemented when a pocket is defined by a single loop of geometry that contains combination geometry. Combination geometry is regular blue geometry (Wall geometry) combined with red geometry (Air geometry).

Please note that in the case where a complete loop of geometry is designated as Air, the overhang parameter will not be applied. Overhang is only applied to Air/Wall combinations where the toolpath does not start from the outside of the loop and work its way inward. In the case of a complete Air loop, use the Cut Width parameter to control the toolpath.

Clearance

Open Sides > `Clearance` governs behavior when the tool approaches the part. This parameter specifies the distance from the inner edge of the tool to the Air wall (or, more generally, to the edge of the part when the tool enters the pocket). This can be used with Air geometry, Corner Cleanup, Material Only, and open-sided pockets in solid models. If the geometry or solid is a closed pocket (no open sides or Air geometry), then this value is not used.

Minimum Cut

This is the smallest amount of material left behind that the system will target for machining. Extra toolpath will be created to cut areas that have this amount of material or more remaining. Areas with this amount of material or less will not be targeted for machining though they may incidentally be cut due to normal process parameters. A value of `0` would cut all around the part (because everything has at least `0` stock). But a large value, such as the tool diameter, might not cut anything.

When using the `Material Only` machining option, the Minimum Cut value is very important. A value of `0` will attempt to find all possible Material Only situations, whereas a value greater than the tool radius is unlikely to find much to cut. This function helps you maximize the efficiency of Material Only so that you can ignore really small bits of material and better focus your Material Only operations.

Offset/Trim Tab

The `Offset/Trim` tab offers the following settings. Sample parts showing the effects of different settings are provided in the folder `Production\Sample Files\`.

Offset from Part and Material

This setting is useful for keeping the tool engaged at all times. Operationally, any `.vnc` file prior to GibbsCAM v10 uses this style.

`Trimmed finish pass`

This checkbox is available only when the pre-v10 style is active (`Offset from Part and Material`, where all four of Pocket/Boss/Fill/Void are set to Offset). If it is not selected, then finish toolpath will machine both walls and air walls. When it is selected, finish toolpath will machine only walls.

When you use a Pocketing process on combination geometry (mixed shapes containing both Air and Wall geometry), we do not recommend selecting the Cutter Radius Compensation (CRC) checkbox. Instead, for operations on combination geometry where CRC is needed or desirable, go to the **Offset/Trim** tab and select the `Trimmed finish pass` checkbox, which is specifically designed to machine only Walls and not Air walls.

Offset from Part, Trim to Material

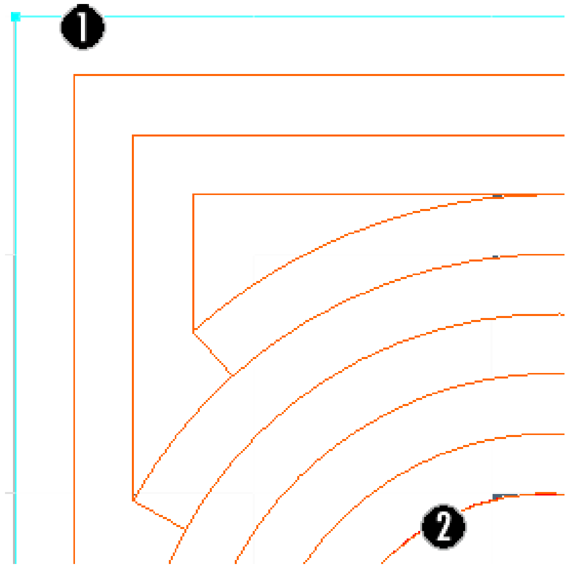
The recommended setting for most parts, especially for those with open pockets. Using this option allows the tool to both trim and rapid in air. As a result, the tool will generally start from the outside-in, reduce full diameter cuts, and stick closer to the programmed step-over amount.

Fewest Offsets from Part, Offset from Material

This option is useful for core parts where there is a large volume of material to remove around a complex boss shape.

Advanced

This option provides the finest level of control over the type of toolpath that GibbsCAM creates, as illustrated here:



Pocket, Fill, Void, Boss

a. Pocket or Fill

b. Void or Boss

When determining where to Offset or Trim your toolpath, if the outside selection (1) is an air wall (red geometry) use Fill, otherwise use Pocket. Similarly, if the inside selection (2) is an air wall use Void, otherwise use Boss.

Depending on your desired toolpath, you may select Offset, Trim or Fewest Offsets for any combination of these features

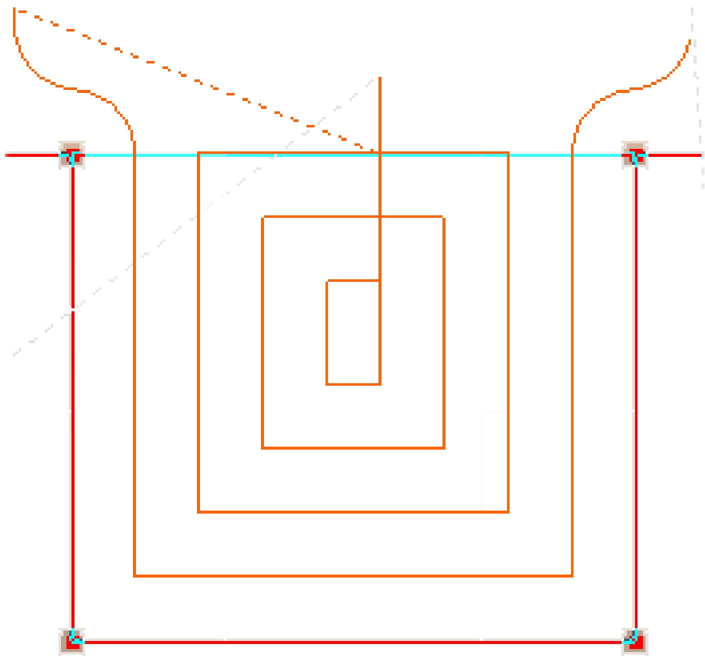
Traverse between segments trimmed by material with:

This option is available if you select "Trim" for either fill or void in the Advanced options or Offset from Part, Trim to Material. The available options are Retract and Rapid, Direct Rapid and Direct Feed. A toolpath that has been trimmed will require linking, via a rapid or a feed motion. Exercise caution when using the "Direct Rapid" choice, as rapid motion is not always in a straight line (depending on the machine.)

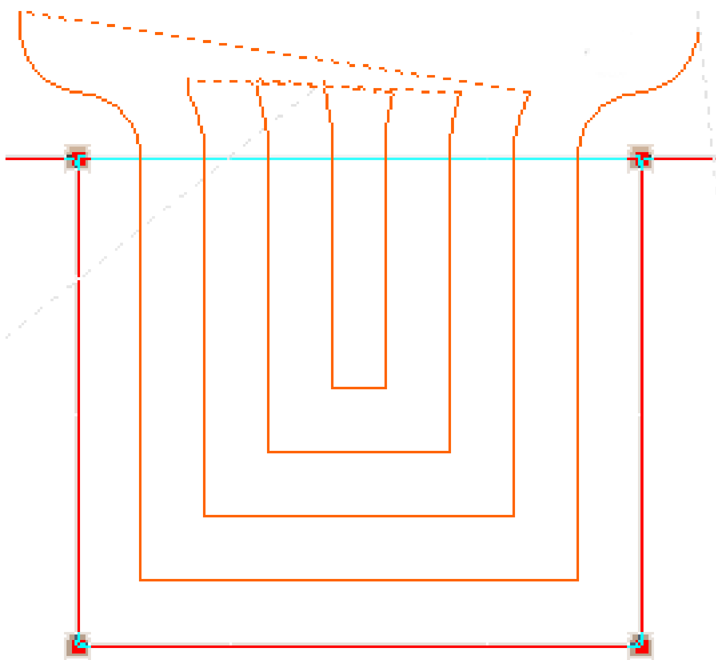
Caveats

a. In the case where you are using the Material Only option on a part that has an existing toolpath, material boundaries are implicitly generated. These boundaries may be either fill or void depending on the configuration of your part.

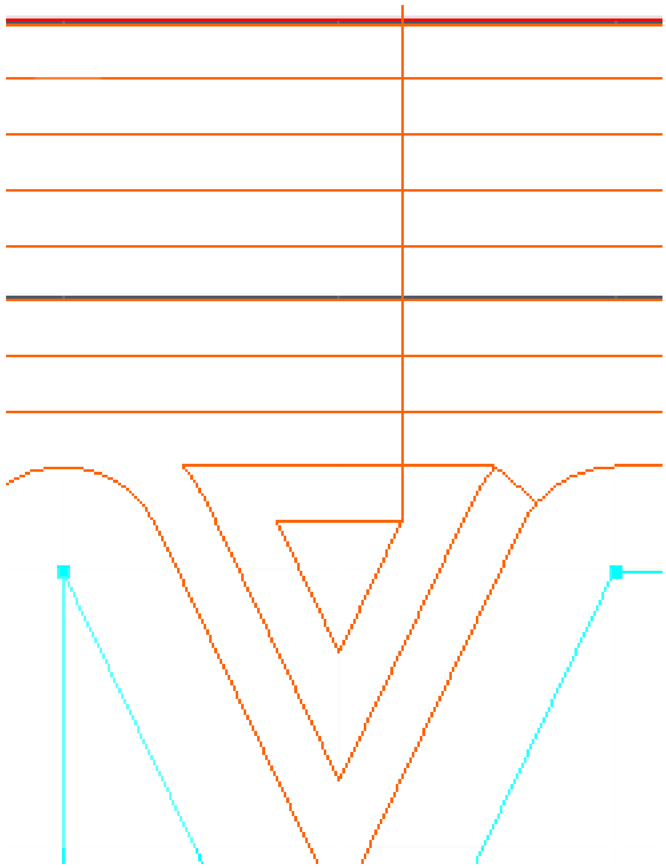
b. Solids with open walls will have a material boundary when Use Stock is enabled. In the case of a solid, extruded or revolved stock we slice the stock which generates fill or void boundaries at each z-level.



Offset
This is the shape that we use to create the offset pocket. The toolpath is generated by an iterative offset from this shape.



Trim
By selecting Trim, GibbsCAM will trim away any portions of the toolpath that take place on the outside of a fill or pocket boundary or the inside of a void or boss boundary. Selecting Trim allows GibbsCAM to create an offset pocket iteration that begins outside of the trimmed shape, creating a more efficient toolpath.



Fewest Offsets
Allows us to offset one shape until the offset would intersect a second shape, at which point we offset both.

Entry / Exit Tab

This tab contains advanced options for Entry and Exit cycles. By default the options you set will be applied to both the entry and exit moves. You can set the entry and exit to use entirely different types of motion, e.g. the entry can be a radius while the exit can simply be a line. To accomplish this, click the Exit option. This tab allows you to generate rather complex moves including ramping. The Entry/Exit tab is bolded when the Advanced Entry/Exit option is selected on the Pocket tab or if the Advanced option is selected on this tab. This section will only focus on the options available once you select Advanced. The Line and 90° Radius and 90° Line options are discussed in Entry and Exit .

Exit
Select this option to make the entry and exit moves different. In the example shown here the entry move is a 1mm line and a 5mm radius while the exit is simply a 1mm line. The functionality of each option is discussed in Entry and Exit .

Radius Entry/Exit
Select this option for your entry and/or exit move to be based on a radius. Using the following options you can define the size of the radius, whether to include line moves and ramp options.

CRC Line

Select this option to generate a line that allows Cutter Radius Compensation to activate. Set the length of the CRC Line you wish to make.

Off Part Line

Select this option to generate a line to feed into or out of the part. This line is perpendicular to the start/end point. This line is generated after the CRC line on entries and before the CRC on exits.

Entry/Exit Radius

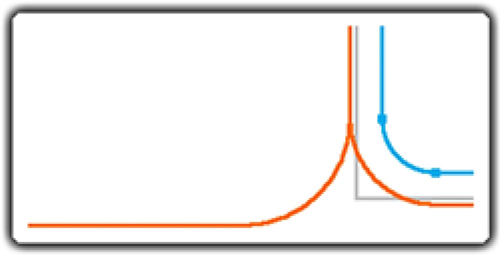
Specify the radius you wish to generate for the entry and/or exit move.

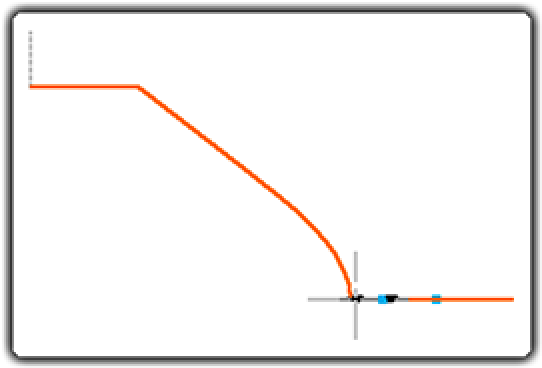
Off Part Distance

This option can limit the radius. If the value is less than the radius the arc will be "chopped off" at the specified distance from the cut shape. The arc will still have the same radius but it will not be a full 90° arc. Entering a value of `0` or something equal to or greater than the radius will have no effect on the toolpath, resulting in a 90° radius.

Z Ramp

Select this option to ramp down onto the part. An Off Part Line is required for this option unless you select the `Include Radius?` option. This is a Z value, so if you enter 5mm, the tool will begin 5mm above the Surface Z and will ramp down the length of the Off Part Line.





Top View	Side View

Include Radius?

This option will include the Entry/Exit Radius in the Z Ramp value, resulting in a helical move (up to 90° only). An example of a Z Ramp including the radius is seen here. You can see how the Off Part line and CRC line are flat and perpendicular while the rest of the entry/exit moves are ramping.

Line Entry/Exit

Select this option for your entry and/or exit move to be based on a line. If you wish to use a line and a radius, choose the Radius Entry/Exit option. Using the following options you can define the size and angle of the line as well as ramp options.

CRC Line

Select this option to generate a line that allows Cutter Radius Compensation to activate. Set the length of the CRC Line you wish to make.

Off Part Line

Select this option to generate a line to feed into the part. This line is perpendicular to the start/end point. This line is generated after the CRC line but before the Off Part Distance line.

Entry/Exit Angle

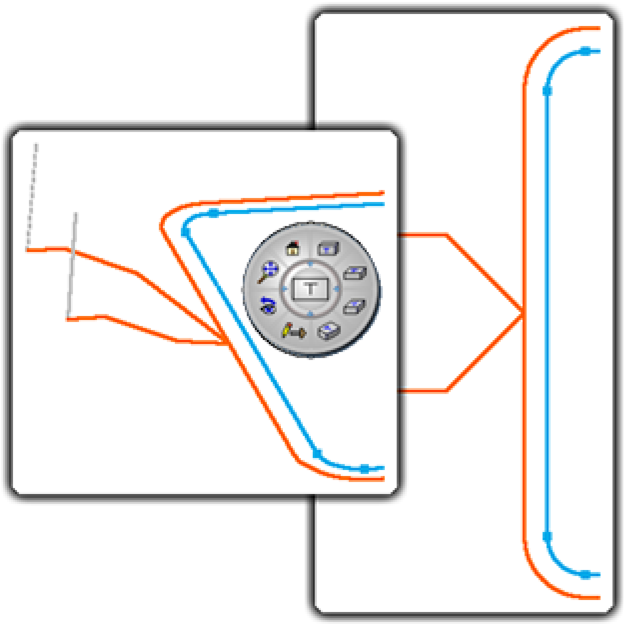
This option allows you to specify the angle of the entry/exit line. Valid entries are 0-180° with 90 being a perpendicular line.

Off Part Distance

This option specified the length of the entry/exit line.

Z Ramp

Select this option to ramp down onto the part. An Off Part Line is required for this option unless you select the Include Line Entry/Exit? option. This is a Z value so if you enter 5mm the tool will begin 5mm above the Surface Z and will ramp down the length of the Off Part Line.

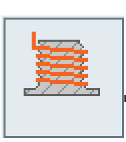


Include Line Entry/Exit?

This option will include the Entry/Exit Line in the Z Ramp value, which will result in a ramp in/out at an angle. An example of a Z Ramp including a line at an angle is seen here. You can see how the Off Part line and CRC line are flat and perpendicular while the rest of the entry/exit moves are ramping.

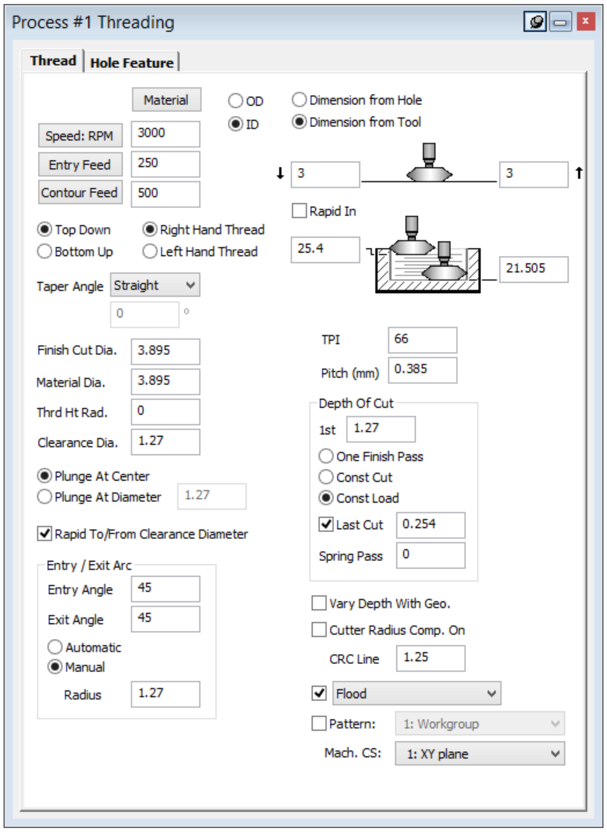
Rotate Tab

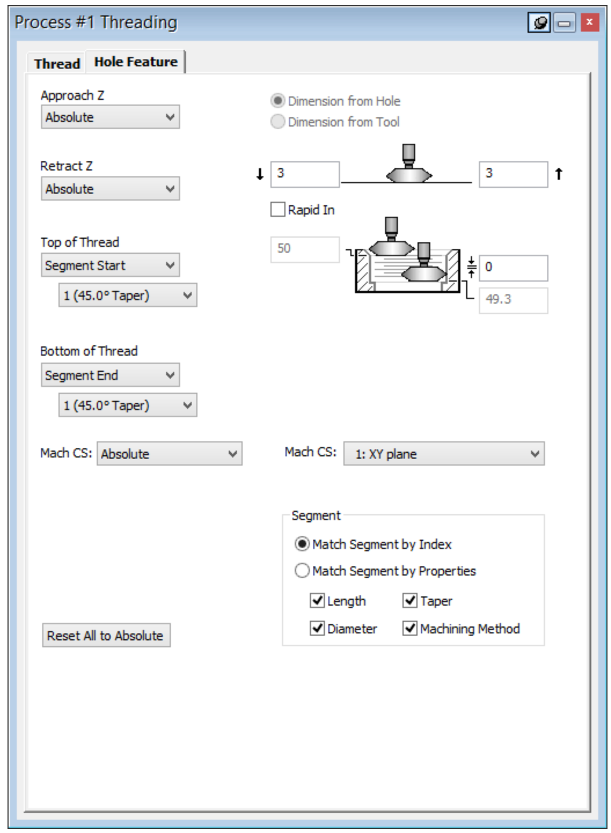
The Rotate tab is available when using a Mill/Turn MDD or a 4-axis or 5-axis MDD. The settings found in this tab allow you to rotate the part or create rotary operations. For more information, see Rotate Tab.



Thread Milling Process

This function, in conjunction with a thread milling tool, lets you mill threads on the outer diameter (OD) or inner diameter (ID), clockwise (CW) or counterclockwise (CCW). The thread milling process is similar to the drilling process in that it requires that either points or circles be selected for the process. As in the Drilling process, Hole Features can be used in conjunction with Hole Manager to apply processes to individual data of selected holes. This process can be particularly useful when used with Full profile or multi-tooth thread milling tools. When used for ID threads, the Dimension from Hole/Dimension from Tool radio buttons become available.





Hole Feature Tab

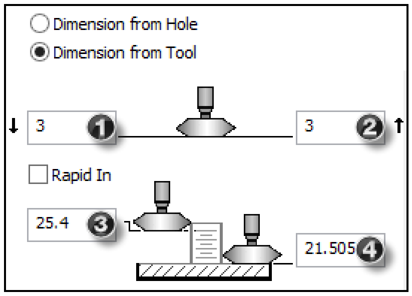
This works in much the same way as Hole Feature Tab within Hole processes. For more information, see Hole Feature Tab.

Thread Tab

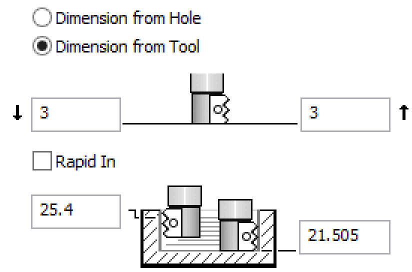
Thread Mill Entry/Exit Diagram

The Entry and Exit Clearance Planes function as they do in the other Process dialogs. The Thread Start Point and Thread End Point positions entered in the diagram specify the actual start and end of the thread specified on the blueprint. The system will add a 45° helix to the start and end of the threading toolpath to provide for a smooth transition into the thread. The start and end points of this helical entry move will be higher and lower in Z than the actual Thread Start and End Points entered by the user in the dialog. The tool can thread up or down, meaning that the start point can be at a higher or lower Z position than the end point.

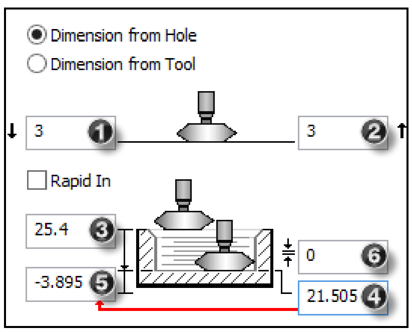
OD

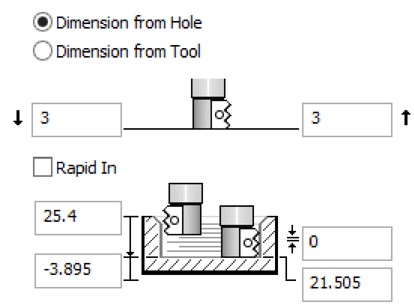


1. Entry Clearance Plane
2. Exit Clearance Plane
3. Thread Start Point
4. Absolute depth of thread
5. Incremental depth of thread
6. Depth of the thread



ID





Thread milling tool	Full profile or multi-tooth thread milling tool

Thread Type

This selection determines whether an OD or ID thread will be created. The Thread Mill Entry/Exit Diagram will change depending on the selection made as shown above.

Top Down / Bottom Up

These selections indicate whether the toolpath will start at the top of the shape and cut down (Top Down) or start at the bottom of the shape and cut up. The Bottom Up selection creates a smoother surface finish.

Thread Direction

This selection determines whether the thread will be Right-Handed (clockwise) or Left-Handed (counter-clockwise) direction.

Taper Angle

The dropdown options are Straight (no taper) BSPT/NPT (Standard pipe thread taper - 1.78991 degrees or 1 in 16 slope) or Other (Enter required angle). Not available with Full profile or multi-tooth thread mill tools.

Finish Cut Diameter

Diameter that the edge of the tool cuts while threading. On an OD thread, the tool cuts in to the minor diameter. On an ID thread, the tool cuts out to the major diameter.

Material Diameter

This parameter is important for multi-pass thread roughing. Specify the diameter from the thread axis at which the tool should engage with material.

Thread Height Radius

Value specifies the difference between the Finish Cut Diameter and the Material Diameter expressed as a radius. Defines the amount of material that will be removed between the material and the last pass.

Clearance Diameter

On an OD thread, the tool will retract to the Clearance Dia after completing the thread before going to the exit clearance position. On an ID thread, the Clearance Dia should be equal to or less than the size of the hole that is being threaded. For ID threads, the tool diameter is less than the clearance diameter which is less than the cut diameter. For OD threads, the cut diameter is less than the clearance diameter.

Plunge at Center/Diameter

Available for ID threads only. Plunge at Center rapids to the center of the hole then moves to the Clearance Diameter. Plunge at Diameter allows you to specify a diameter to plunge to before moving to the Clearance Diameter.

Rapid To/From Clearance Diameter

Available for ID threads only. When checked, uses an XY rapid from the plunge point to the clearance diameter, instead of an entry feed line.

Entry/Exit Arc

Allows you to specify the arc angle for the entry/exit moves from Clearance Diameter to Cut Depth. Check Automatic to calculate the arc radius automatically based on the relationship between the clearance and cut diameters (recommended). Check Manual if you want to specify your own radius. This must fit outside the material diameter.

TPI (Threads Per Inch)

This value specifies the number of threads per inch. The threads per inch and pitch text boxes are interactive. When one value is entered the system calculates the other. This allows the user to enter whichever specification is given on the blueprint, be it the pitch or TPI. When creating a metric part, this value specifies threads per millimeter. Not available with Full profile or multi-tooth thread mill tools.

Pitch (mm)

Entering a value here specifies the pitch of the thread in mm which is calculated by taking the inverse of the TPI. Not available with Full profile or multi-tooth thread mill tools.

Depth of cut

1st

Specifies the depth of the first cut, then choose from the following radio buttons:

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.

Constant Cut

Tool will cut the thread in a series of passes, each of approximately the same depth as the 1st cut.

Constant Load

The tool 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 field.

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. When using a multi-tooth threadmill, spring passes occur after all main passes are complete.

Vary Depth With Geometry

This option will cause the drilling depth to be variable, based on the selected geometry. The retracts will all be to the same level but the final Tip Z or Full Diameter Z are relative to the geometry, based on the first selected point. Turning this item off allows a constant Z depth drill process to be defined from geometry at different depths. This could be very useful for constant depth spot drilling. When this option is selected, the Post Processor will not have the option to combine similar holes into subprograms.

Cutter Radius Compensation

Check the box if you wish to use CRC, and specifiy the length of the line used to turn CRC on and off. For more information see Cutter Radius Compensation (CRC).

Other Common Controls

Coolant

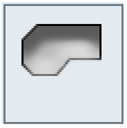
The checkbox indicates whether coolant is turned on in a process. Flood is the standard coolant option. Additional coolant options are available with custom post processors.

Pattern

When the Pattern checkbox is selected, the process will create identical toolpaths in different locations on the part. The toolpath generated will be cut once for each point in the selected pattern workgroup. The pattern workgroup, which is selected from the adjacent pop-up menu, contains unconnected, plain points that serve as origin points for the toolpath created by the process. The original toolpath created will NOT be cut unless the origin point for that toolpath is included in the pattern workgroup. Posted output will create one subprogram for the primary toolpath and call that subprogram once for each point in the pattern workgroup. For more information, see Pattern.

Mach. CS

The Mach. CS drop-down list appears on this tab when a 3-axis MDD is active. For more information, see Mach. CS .



Surfacing Process

The Surfacing Process allows you to follow model surfaces to create toolpath. For more information on this Machining Process type, see the *SolidSurfacer* guide.

Material Only

Enabling the Material Only machining option allows the system to have an awareness of material that has already been removed. A "material only" operation provides for "no air cutting" and can calculate the exact shape of material left from the initial stock shape and all prior machining operations with the exception of Surfacing operations. Material Only utilizes this information to create open pocket shapes to clean up by "slicing" the remaining material model at the Z depth of the current operation being created. Material Only will function on 2D geometry or solids that have been roughed out. The Material Only functionality is only available from the Level 2 interface.

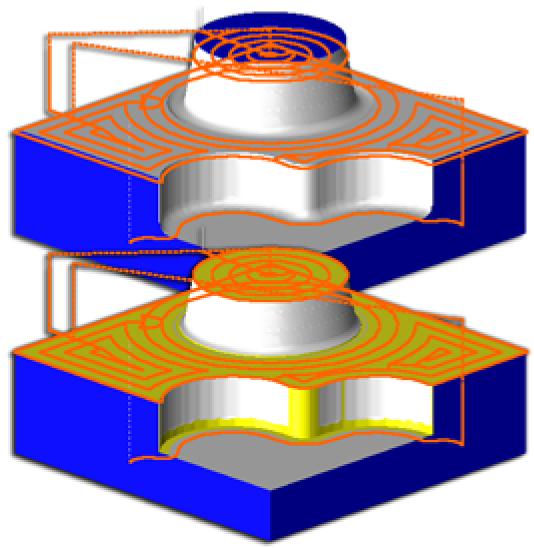
> ⓘ It is recommended that the preference be deselected if it will not be used. This will save processor power and will help to minimize the size of the part file.

Material Only Definition

Material Only calculates toolpath for all remaining material left on walls by prior operations. Remaining material is stored for all contour, pocketing, and drilling operations (2D operations). Remaining material is NOT stored for Lace, Surface Flow, and 2 Curve Flow cuts (3D operations). Material Only supports custom stock definitions, sharp/bullnose/tapered/ball endmills, and most form tools (but no undercutting tools). Material Only may be used as a single operation or as part of a multiple process group for pocketing.

Material Only Description

When the Material Only option is active, the system takes into account the current material conditions in terms of what has already been cut in previous processes and operations, including custom stock specifications. During subsequent operations the system will generate toolpath to remove only the material within these shapes if Material Only is selected, thereby providing for "no air cutting." Toolpath generated in these areas is based upon an open-sided pocket configuration. See Material Only Relating to Closed Pockets and Open Pockets for more information on machining open pockets.



Example of the use of Material Only in a roughing operation

The figure illustrates the use of a Material Only pocketing operation. Several operations roughed the part and the tapered boss was finished. Now a smaller flat endmill is used to remove remaining material. There are three areas with material – the top of the boss is uncut, the large floor has a little bit of Z Stock and the small open pocket has material left by the previous tool's bottom corner radius. All of these conditions are taken care of with a single material only operation. The image shows the toolpath overlaid on the before and after part condition. Note how the toolpath follows the edge of the floor at the top of the open pocket.

Material Only Limitations

It is recommended that any tools used in Material Only operations be of a constant or decreasing radius value. This is because the system does not recognize undercuts. The system does not recognize undercuts from the original stock condition or as a result of machining. Thus, a mushroom-shaped part, an increasing radius, or an undulating form tool is not recommended. The results could range from cutting air to the tool trying to rapid into stock it does not recognize.

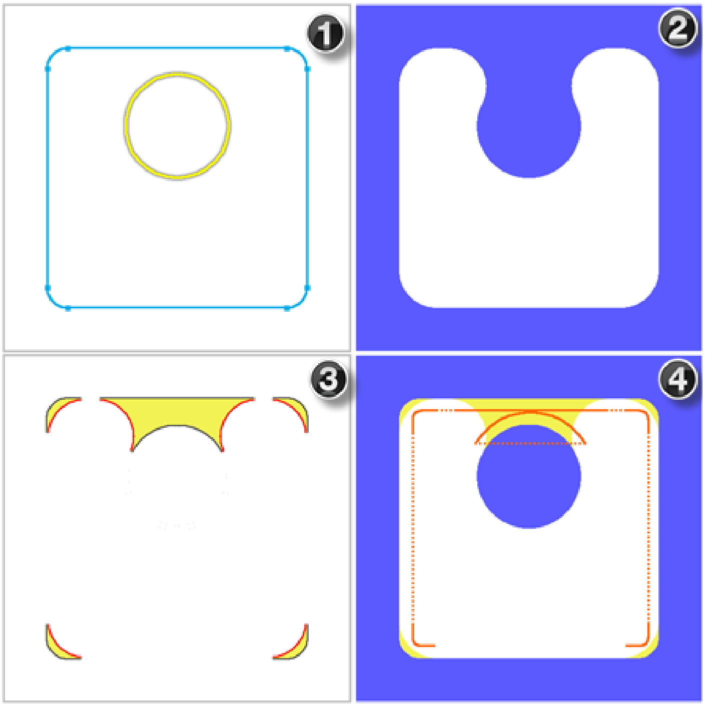
For Roughing (Pocketing) operations that use Zig Zag, the cut region is not saved; selecting or deselecting the Material Only checkbox has no effect on the generated toolpath.

Material Only Relating to Closed Pockets and Open Pockets

The following are four examples that will help to illustrate how Material Only operations work under different circumstances. The examples provide tips on how to best use this powerful feature.

Closed Pockets and Material Only

A closed pocket is defined as a closed shape composed entirely of Wall features. The illustration shows closed pockets and Material Only. A closed pocket with an island is located close to the pocket wall. The initial roughing leaves material in five areas – all four corners as well as between the wall and the island. The system calculates the remaining material and defines those areas with combination geometry, shapes that are made of both Air and Wall features. The second operation, the Material Only operation, machines only those areas. In this example, the recommended values for Past Stock and Overhang are used.



1. Closed pocket with 6mm radius corner and an island close to the pocket wall.
2. Initial Roughing operation with 22mm diameter endmill.
3. The system creates five regions where material has been left. A closed shape consisting of combination geometry - both Air and Wall features - defines each area.
4. Material Only roughing operation with 7mm diameter endmill.

Closed Pockets and Material Only

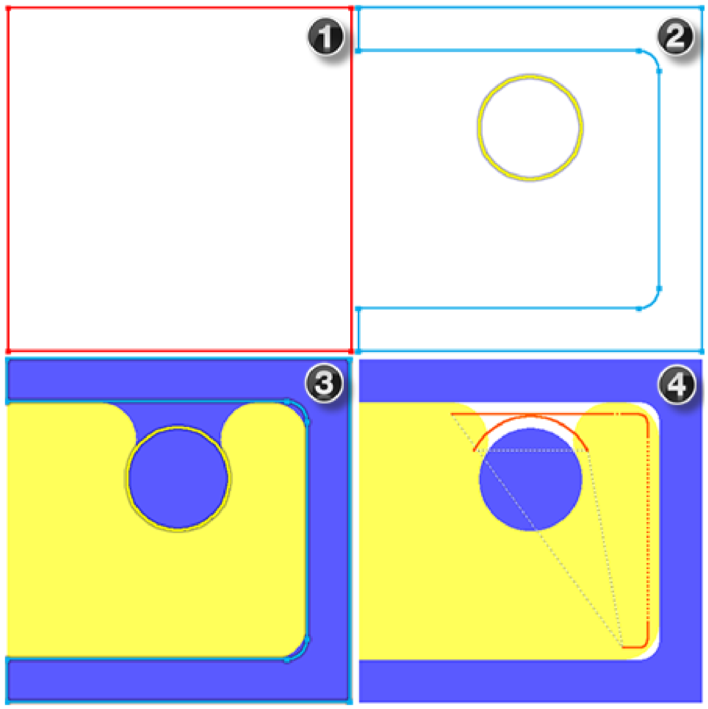
Open Pockets and Material Only

An open pocket is defined as a closed shape composed either of all Air features or a combination of Air and Wall features. This combination of features is referred to as combination geometry. Two methods are recommended for working with an open pocket when generating Material Only cuts. Each method is described in the following text.

When you use a Pocketing process on combination geometry (mixed shapes containing both Air and Wall geometry), we do not recommend selecting the Cutter Radius Compensation (CRC) checkbox. Instead, for operations on combination geometry where CRC is needed or desirable, go to the **Offset/Trim** tab and select the Trimmed finish pass checkbox, which is specifically designed to machine only Walls and not Air walls.

Multiple Shape Method

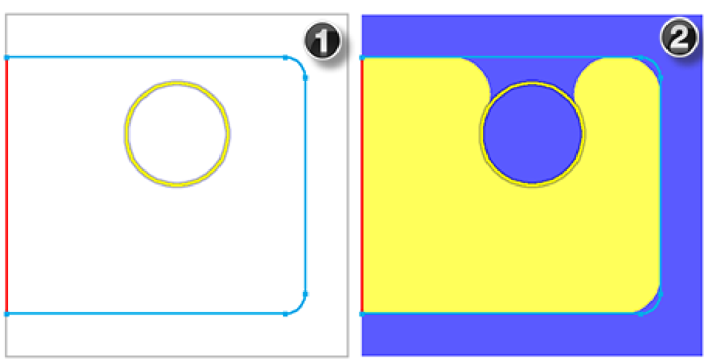
This is the recommended method for assuring the best toolpath when working with an open pocket and generating Material Only cuts. This method requires at least two geometry shape sets. The first shape set is composed entirely of Air features, which represent the stock. The second shape set is made of Wall features. Using this method, the system treats the part as an island inside the stock. This is the method used when creating Material Only operations with a solid body. For more information on solids and Material Only, see the *SolidSurfacer* guide. The figure illustrates open pockets and Material Only using the multiple shape method. The first shape is a single Air shape that represents the stock. The second shape set consists of all Wall features and represents the part as an island within the stock. The initial roughing operation leaves material behind in the two corners as well as between the two islands, as shown in the illustration. In this example, the recommended values for Past Stock and Overhang are used.



1. An all Air shape representation of the stock.
2. Wall shapes representing the part as an island within the stock.
3. Initial roughing op that leaves material behind. Both the Air and Wall shapes were selected.
4. Material Only roughing operation selecting both shape sets.

Combination Geometry Method

This method uses combination geometry (one or more shapes composed of both Air and Wall features) to define the open pocket. This method is the fastest from the standpoint of geometry creation, but can produce undesirable toolpath when a complex open pocket part is processed. The figure illustrates open pockets and Material Only using the combination geometry method. The recommended values are used for Past Stock and Overhang. The outcome is the same as in the illustration. The combination geometry method is faster (less geometry is created) and less complicated, but it requires more visualization on the user's part.

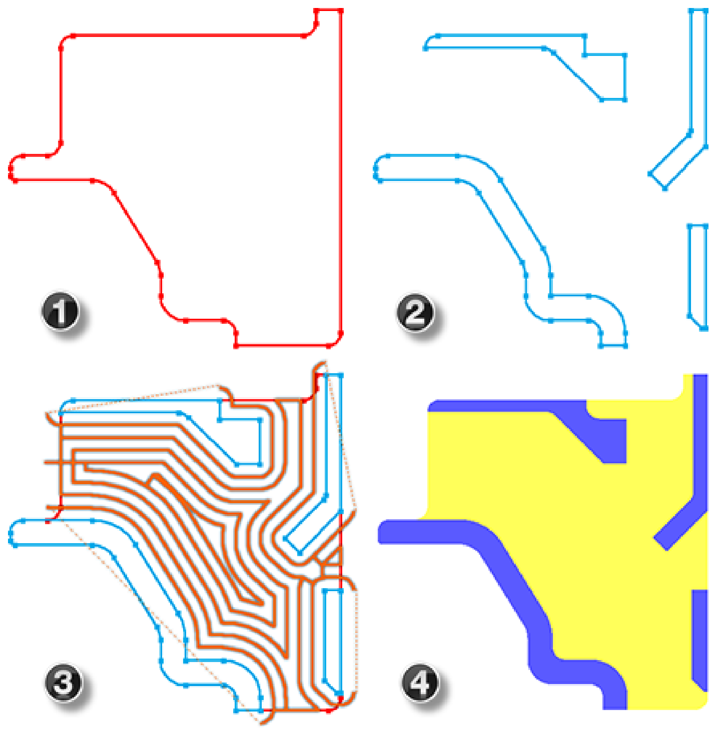


1. The open pocket is drawn as a single mixed shape.
2. The initial roughing operation generates the same three regions as the multiple shape method.

Material Only combination geometry method

Custom Stock

The figure illustrates open pockets and Material Only using the multiple shape method. This example uses a custom stock defined by extruded geometry. (Note that revolved shapes are also supported.) The recommended values for Past Stock and Overhang are used in this example.

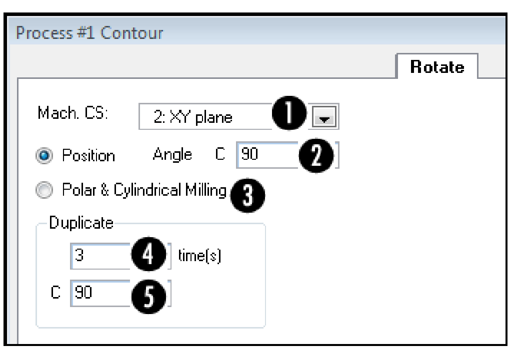


1. An all Air shape representation of the stock.
2. Wall shapes representing the part as a series of islands within stock.
3. The roughing operation toolpath.
4. The rendered part using the multiple shape method.

Material Only multiple shape method

Rotate Tab

The Rotate tab, found in process dialogs when a Mill/Turn, 4-axis, or 5-axis MDD is being used, provides access to 4th- and 5th-axis machining functions. The items found on this tab allow you to create toolpath that is rotated into a position and duplicated (set a number of times and angle to repeat) or create rotary toolpath. When the operation is generated the toolpath will be duplicated in a direction as set by the input angle (positive or negative). The functions found in the Rotate tab are available when working in the Level 2 interface and the Mill, Mill/Turn, Advanced CS, Broaching, or Multi-Task Machining (MTM) module is enabled. Additionally, an A-, B- or C-axis capable MDD must be selected for the current part.



1. Coordinate System from which to generate the process
2. Enable Positioning and Start Angle of rotation
3. Enable Polar & Cylindrical Milling
4. Number of additional times to repeat
5. Incremental Angle for the next repeat

Mach. CS

This drop-down list lets you choose the coordinate system the operation will be created from. By default, the XY plane is selected, but all coordinate systems that have been created will be available. 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.

Position

Select this option to perform a simple rotary positioning move from the selected Machining CS.

Angle

Available with any 4-axis or 5-axis MDD. This determines the position of the angle of the first pass relative to A0, the normal top view of the XY plane. The range of angles may be negative. In the above graphic, a value of 45 is set for this drilling process. Therefore the part will be rotated 45 degrees before drilling the holes.

Polar & Cylindrical Milling

Only available with the Polar & Cylindrical Milling option. This is explained in Polar & Cylindrical Milling .

Duplicate

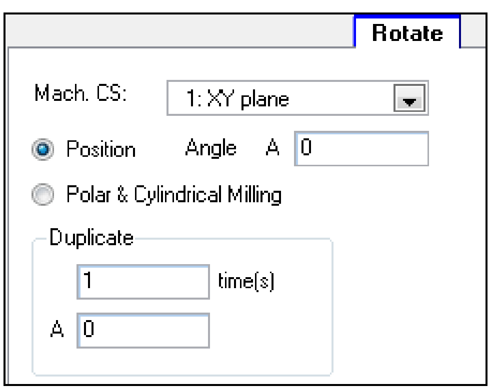
If your positioning or rotary toolpath is to be duplicated you can set the parameters here.

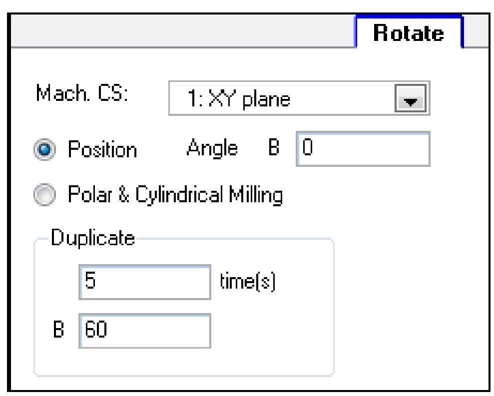
of Times to Repeat

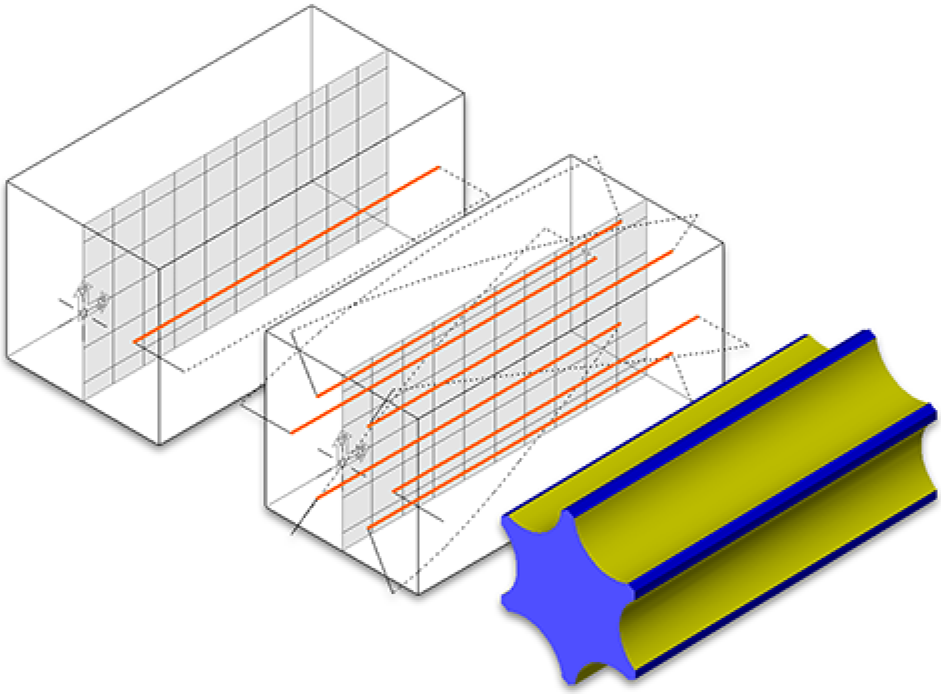
The number of additional toolpaths to generate. By entering a value of `3` as shown above, this toolpath will be made a total of 4 times: the original, plus 3 repetitions. If you are simply setting a machining coordinate system, not positioning the toolpath, be sure to enter the value `0` to avoid duplicating the toolpath. If the value is `1` and an incremental value is set, the toolpath will be generated at that angle.

Incremental Angle

Each additional repetition of the toolpath will be set at this angle value from the last toolpath. If you are simply setting a machining coordinate system, not positioning the toolpath, be sure to enter the value `0` to prevent setting the toolpath at an angle.







Example of the same operation with and without positioning

Rotary Part Clearance Planes

Defining the proper clearance plane for a 4th or 5th axis rotation is very important. First, be sure the master clearance plane, as defined in the Document Control Dialog, is well beyond the part when it rotates. The same is true of clearance planes defined in operations.

For operations involving a tool change, the clearance will not be a problem as the tool retracts full-up. However with operations that involve rotary positioning or that do not require a tool change the tool only retracts to the exit clearance plane. Therefore if your exit clearance plane is not beyond any edge of the part as it rotates the tool will crash. For optimal G-Code, set the Entry Clearance as you would for a non-rotary operation and your Exit Clearance at a greater value.

> One method to determine how high the clearance needs to be on a Vertical machine is to square the Y+ and Z+ values, add them together and determine the square root of that number. Then round the number up as needed. On a Horizontal machine replace the Y value with the X value.
>
> sqrt(Y^2 + Z^2) = distance from center to edge of a part

Entry / Exit Tab

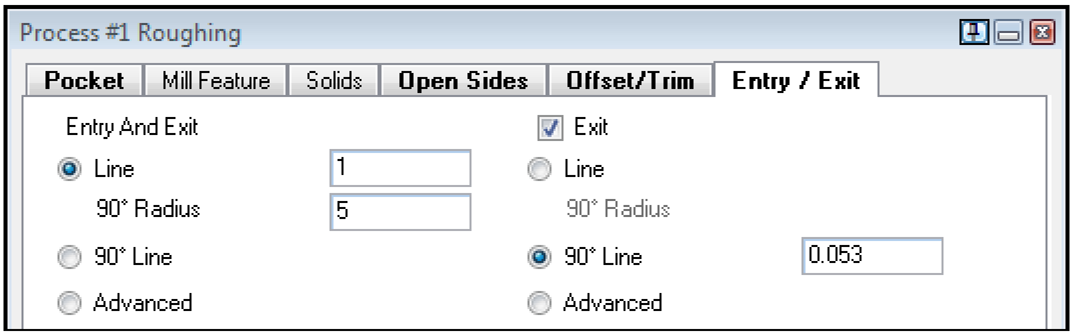
This tab contains advanced movement for entry and exit cycles. It is available when Advanced is selected as the Finish Entry/Exit style. Refer to Entry/Exit Tab for more information.

Same Entry and Exit

To use the same type of move, i.e. 90° Line or Line and 90° Radius, for both entry and exit moves you do not need to use the settings in this tab. The Line and 90° Radius and 90° Line options are discussed in Line and 90° Radius (contouring) and Entry and Exit (pocketing).

Different Entry and Exit

To make your entry and exit moves different, such as different values or one is just a line and the other is a line and a radius, select the Exit option then specify the types and values for the different entry and exit. In the example shown here the entry move is a 1mm line and a 5mm radius while the exit is simply a 3mm line. The Line and 90° Radius and 90° Line options are discussed in Line and 90° Radius (contouring) and Entry and Exit (pocketing).



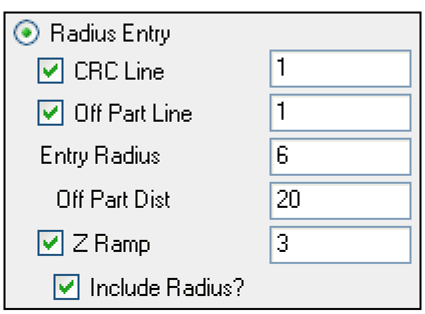
Example where the tool will enter with a line and radius and the exit is only a line

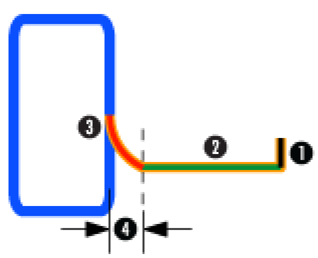
Advanced Moves

The Advanced entry/exit options allow you to create a wide variety of move types. If you do not choose the Exit option at the top of the dialog both the entry and exit moves will be identical.

Radius Entry/Exit

Select this option for your entry and/or exit move to be based on a radius. Using the following options you can define the size of the radius and whether to include line moves and ramp options, even a 3D helical move into or off of the part.





1. CRC Line
2. Off Part Line
3. Entry/Exit Radius
4. Off Part Distance

CRC Line

Select this option to generate a line that allows Cutter Radius Compensation to activate or deactivate. The line is generated before the entry or after the exit. Enter a 2D length, measured in HV on the machining CS. The CRC Line can be tangent or perpendicular to the Off Part Line depending on your Machining preferences. This linear move is always a 2D move regardless what the Z Ramp status is.

Off Part Line

Select this option to generate a line to feed into or out of the part. This line is generated after the CRC line on entries and before the CRC on exits. Enter a 2D length, measured in HV on the machining CS. This line is created perpendicular to the first/last feature of the toolpath. The Off Part Line can be a 2D or 3D move depending on the status and value of the Z Ramp.

Entry Radius

Specify the radius you wish to generate for the entry and/or exit move. The arc will be tangent to the first feature of the toolpath, with the exception that it can ramp in Z if you select that option. The radius can be a 2D arc or a 3D helical move depending on the status and value of the Z Ramp option and the Include Radius check box.

Off Part Distance

This option can limit the radius. This value defines the distance from the first toolpath feature at which the entry/exit arc will be trimmed. The idea here is that the material is located by the Off Part Distance value from the actual part to be machined. Entering a value of `0` or something equal to or greater than the radius will have no effect on the toolpath, resulting in a 90° radius.

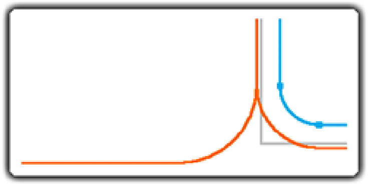
Z Ramp

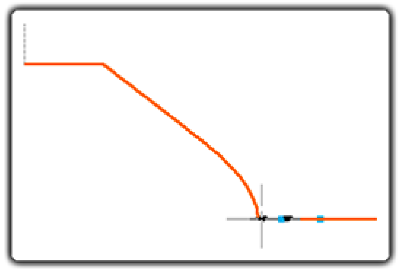
Select this option to ramp down onto the part. This is an incremental Z height above the Z level of the actual Toolpath. If you enter `5mm` the tool will begin 5mm above the Surface Z and will ramp down the length of the Off Part Line. An Off Part Line is required for this option unless you select the Include Radius? option.

For the Entry value the Z Ramp will start at the beginning of the Off Part Line and will end at the end of the Off Part Line or Entry Radius depending on whether the Include Radius option is selected. For the Exit value the Z Ramp will start at the beginning of the Exit Radius or beginning of the Off Part Line depending on the Include Radius option. The Z Ramp will end at the end of Off Part Line. The CRC Line is excluded for both the entry and exit because it is a 2D move.

Include Radius

This option will include the Entry/Exit Radius in the Z Ramp value, converting the arc to 3D helical moves (up to 90° only). Unchecking Include Radius will create a flat arc at the Z depth of the toolpath. This option enables machines that do not allow helical interpretation to arc at the fixed Z. An example of a Z Ramp including the radius is seen here. You can see how the Off Part line and CRC line are flat and perpendicular while the rest of the entry/exit moves are ramping.

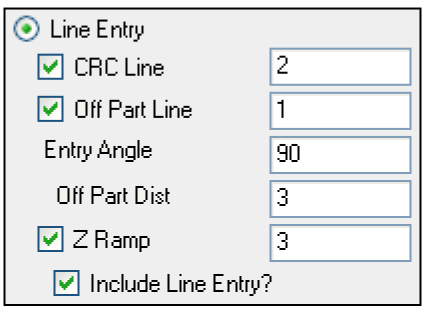


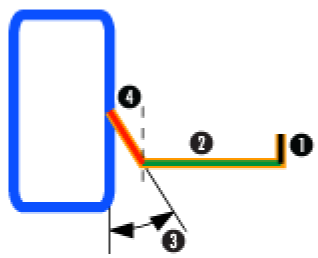


Top View Side View

Line Entry/Exit

Select this option for your entry and/or exit move to be based on a line at a custom angle. If you wish to use a line and a radius, choose the Radius Entry/Exit option. There are numerous options for the line definition including a 3D ramping.





1. CRC Line
2. Off Part Line
3. Entry/Exit Radius
4. Off Part Distance

CRC Line

Select this option to generate a line that allows Cutter Radius Compensation to activate or deactivate. The line is generated before the entry or after the exit. Enter a 2D length, measured in HV on the machining CS. The CRC Line can be tangent or perpendicular to the Off Part Line depending on your Machining preferences. This linear move is always a 2D move regardless what the Z Ramp status is.

Off Part Line

Select this option to generate an additional line to feed into the part perpendicular to the first feature of the toolpath. This line is generated after the CRC line on entries and before the CRC on exits. Enter a 2D length, measured in HV on the machining CS. This line is created perpendicular to the first/last feature of the toolpath. The Off Part Line can be a 2D or 3D move depending on the status and value of the Z Ramp.

Entry/Exit Angle

This is the angle at which the entry/exit move will approach the actual toolpath at the user defined Z level. Valid entries are 0-180° with 90 being a perpendicular line.

Off Part Distance

This value defines the distance from the first toolpath feature at which the entry/exit arc will be trimmed. The idea here is that the material is located by the Off Part Distance value from the actual part to be machined. 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.

Z Ramp

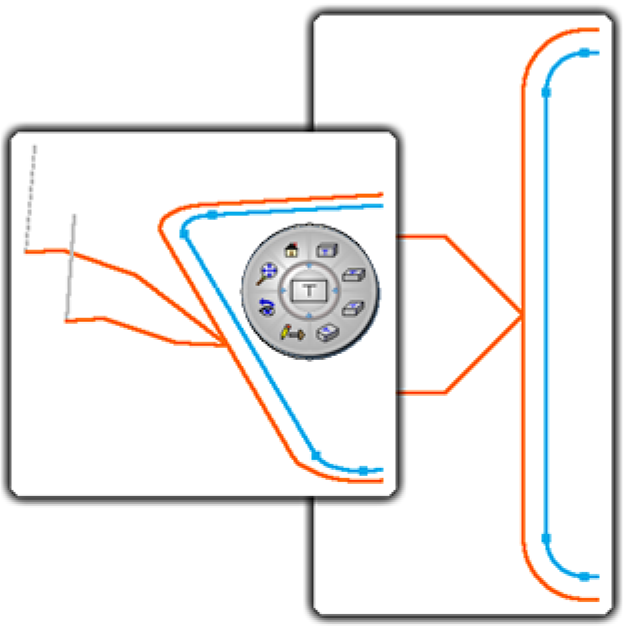
Select this option to ramp down onto the part. An Off Part Line is required for this option unless you select the Include Line Entry/Exit? option. This is an incremental Z height above the Z level of the actual Toolpath. If you enter 5mm the tool will begin 5mm above the Surface Z and will ramp down the length of the Off Part Line.

For the Entry value the Z Ramp will start at the beginning of the Off Part Line and will end at the end of the Off Part Line or the end of the angular line depending on whether the Include Line Entry option is selected. For the Exit value the Z Ramp will start at the beginning of the angular line or beginning of the Off Part Line depending on the Include Line Entry option. The Z Ramp will end at the end of Off Part Line. The CRC Line is excluded for both the entry and exit because it is a 2D move.

Include Line Entry?

This option will include the Entry/Exit Line in the Z Ramp value, which will result in a ramp in/out at an angle, converting the whole entry/exit move to 3D moves. Unchecking Include Line Entry will create a flat line at the Z depth of the toolpath.

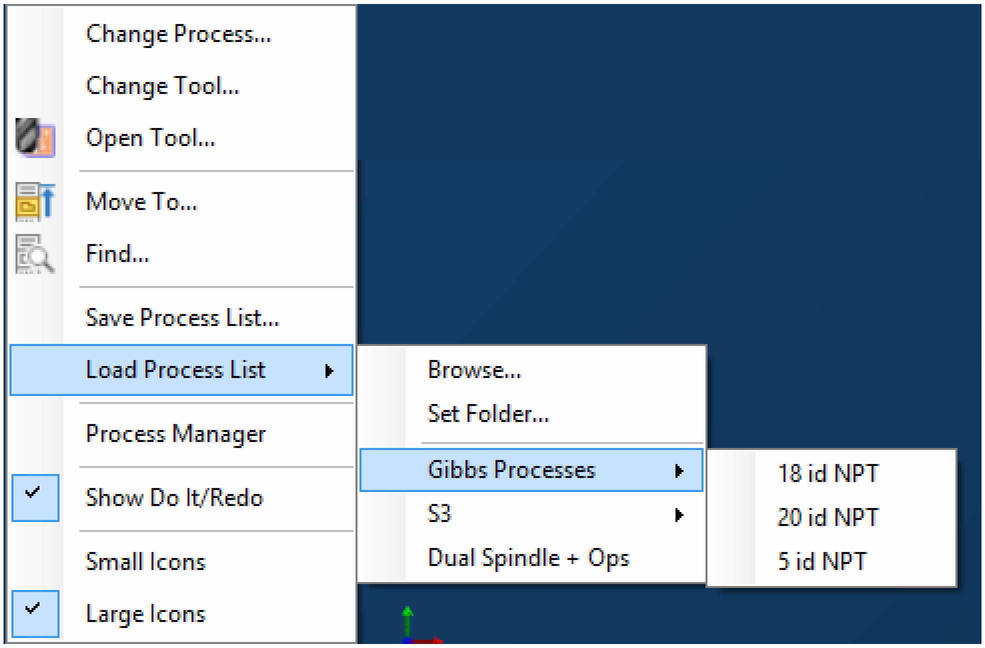
An example of a Z Ramp including a line at an angle is seen here. You can see how the CRC line is flat, the off part line is perpendicular and changing in Z and the rest of the entry/exit moves are ramping at an angle.



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.

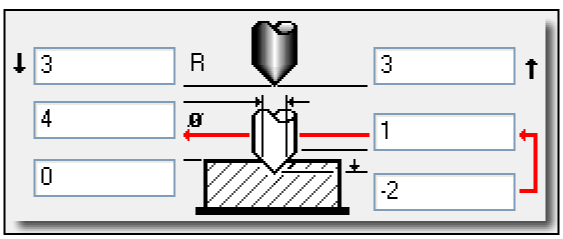
If the Tool list already contains Tool tiles, those tools are deselected but not removed from the list. The system searches the existing Tool list to find the necessary tools for the loaded Process Group. First, the system searches for an exact tool match. If an exact match is not found, the system searches 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.

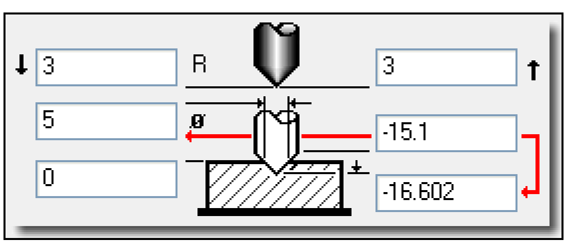
Pre-Defined Process Groups Exercise

In this exercise, we will create a Process Group for tapping 15mm deep ISO M6x1.0 holes with a 9mm diameter chamfer that can be used on any number of different parts.

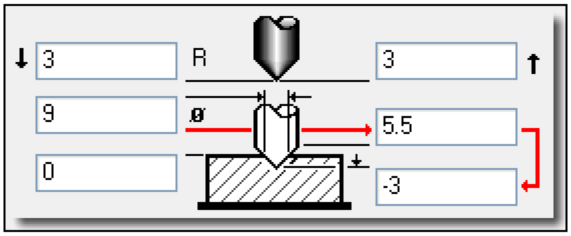
- Create a new part.
- Create the following tools:
  - Spot Drill with a 6mm diameter, 25mm tool length and a 90° tip angle.
  - Drill with a 5mm diameter, 25mm tool length and a 118° tip angle.
  - Countersink with a 20mm diameter, 25mm tool length, 90° tip angle, and 0 for the sharp tip diameter (which makes the taper/flute length 6.009).
  - Tap with a 6mm diameter, 25mm tool length, 1mm Pitch and a 180° tip angle.
- Create the following Process tiles:
  - Drilling process using the Spot Drill. Select the Feed In - Rapid Out Entry/Exit Cycle and enter the following information in the Drill Clearance Diagram.



  - Drilling process using the Drill. Select the Feed In - Feed Out Entry/Exit Cycle and enter the following information in the Drill Clearance Diagram.

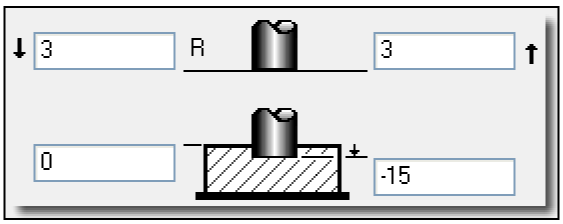


  - Drilling process using the Countersink. Select the Feed In - Rapid Out Entry/Exit Cycle and enter the following information in the Drill Clearance Diagram.



- Drilling process using the Tap. Select the Tap Entry/Exit Cycle and enter the following information in the Drill Clearance Diagram.

There should be four completed Process tiles in the Process list. Because this is an exercise in saving and loading Process Groups, we will not apply these processes to geometry in this file.



- Select Save under the Processes menu. Save the file in a location that can be easily be accessed.
- Create a new file with the following stock specifications: X+ = 100, X- = -100, Y+ = 75, Y- = -75, Z+ = 0, Z- = -12.
- Open the Tool list and the Machining palette. Select Load from the Processes menu and point to the file you just saved. Click Open.

The Process list and Tool list will contain the Process Group information that we created in the previous file.

- Create and select a point or group of points. Apply the loaded process. The operations to drill, tap and countersink the holes will be created.

This Process Group can now be applied to any group of points in any file. For more information on the Process menu, see the *Common Reference* guide.

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.

Machining Markers

Machining Markers appear on selected geometry for contouring processes in order to designate the cut shape. 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 Feature Markers are moved, the Point Markers “follow” them and snap to the same position. To place the Start Point and End Point Markers in the exact same location, place the Start Feature Marker in the correct location and drag the Start Point Marker to the desired position. Then drag the End Feature Marker to the same location as the Start Feature Marker. The End Point Marker will automatically snap to the same position as the Start Point Marker.

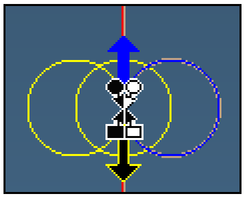
For precise control over the Start and End Point Marker positions, create a point at the correct position. Dragging a Start or End Point Marker close to the point will cause the marker to snap to the point and use its exact coordinates.

The geometry between the start and end points will be dark blue to indicate that it is the cut shape that will be machined by the process. If the start and end features are the same, double-clicking on one of the markers will allow the toolpath to pass over the end point once. This will create an overlap in the toolpath.

To quickly place the end feature and end point markers, hold down `Shift+Ctrl` and click the desired end feature. Then place the end point marker. This is the easiest method, particularly if the end points need to be adjusted, and it eliminates any possible mouse movement errors that might occur when using the drag method.

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 .

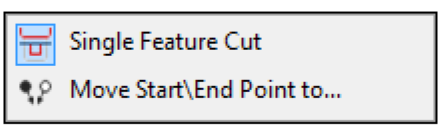
Start Feature:
The geometry feature, such as a line or circle, on which the tool starts cutting.

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.

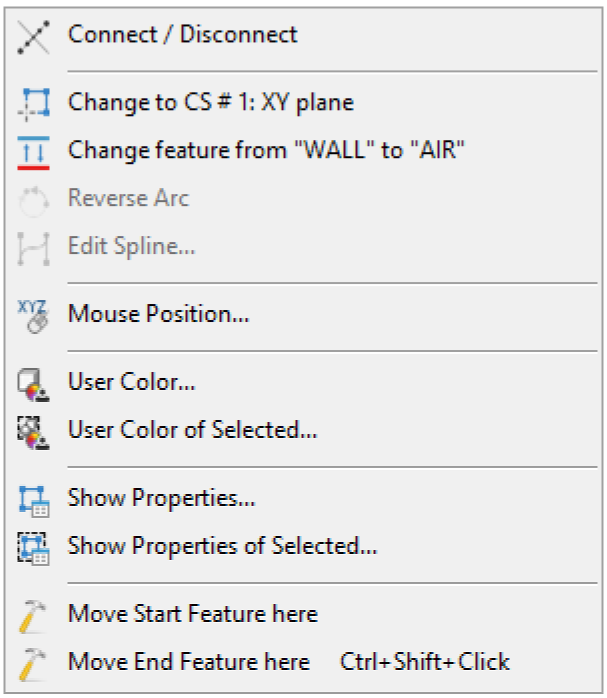
D-Pointer:
The drive curve used when creating swept surfaces. Must be an open, terminated shape.

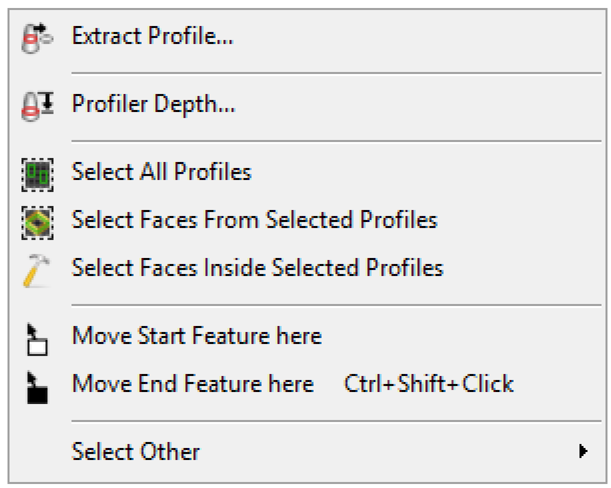
Start and End Points

The start and end points do not necessarily have to be on the part geometry. There may be times when it is desirable to have the tool start or end its toolpath off the part. This can be done by moving the markers off the part geometry. A geometry feature (e.g. 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 extension of the start feature may be a section that was trimmed away, so the start point will snap to an extension of the start feature off the part. This is also true for the end feature and end point.

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 want 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

D-Pointer

The D-pointer Marker only comes up when creating swept surfaces on contour and pocket walls. It designates the drive curve shape. When the cut shape is selected, the Cutter Side and Direction and Start/End Point/Feature Markers will appear on the shape selected for the base curve. If there is an open, terminated shape in the same workgroup as the base curve, the D-pointer will snap to one of the terminated ends. If not, it will come up with the other markers. The D-pointer can be dragged like the other markers; however, it can only be placed on a terminated point.

Operations

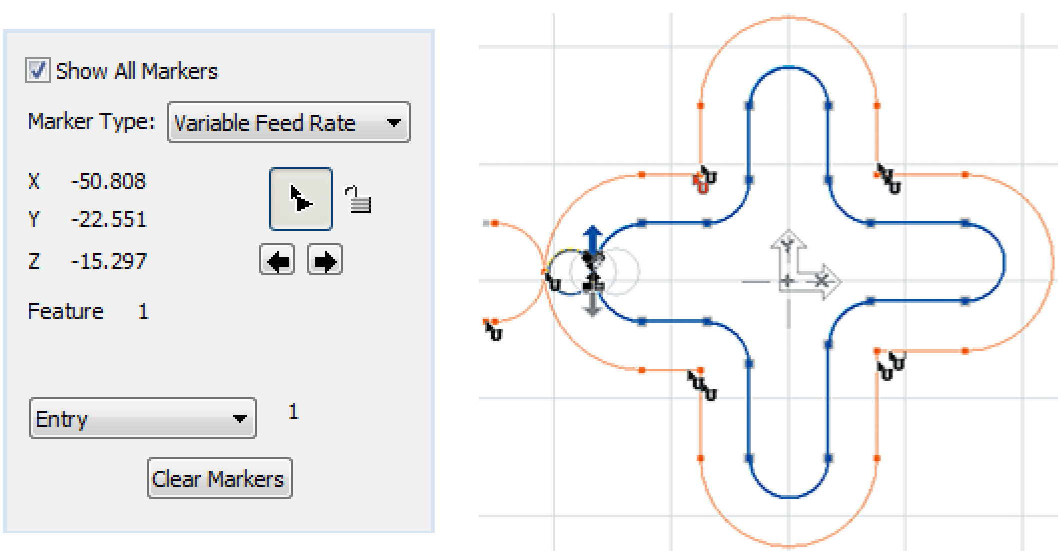
Operations contain finished toolpath. The toolpath consists of the actual moves the tool will make to cut a part, a visualization of the G-code to be output. For more information see the section on "Operations" in the *Getting Started* guide.

- Boss Top Machining
- Machining Air Geometry
- Clearance Moves
- 2 ½ Axis Surfacing
- Pattern
- Engraving
- Printing the Toolpath

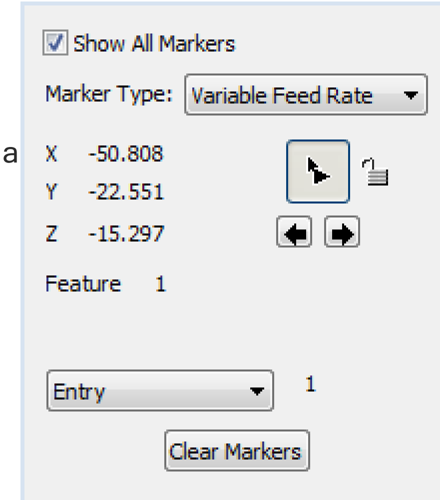
Utility Markers

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, and Program Stop.

This image shows the use of Utility Markers. In this example, utility markers are being used to slow the tool down as it enters a filleted corner and return to the base speed once out of the corner.



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:

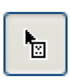
You can set the feedrate for the elements of the toolpath following the marker. The five sub-options for this type of marker are User, Entry, Contour, Percent, and Max. User allows you to explicitly set the feedrate. Entry sets to feedrate to the defined entry feedrate for the operation. Contour sets 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.

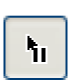

Text:

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 45.


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`).

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

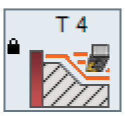
Previous Marker:
Highlights the previous marker in the toolpath and displays the marker information.

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.

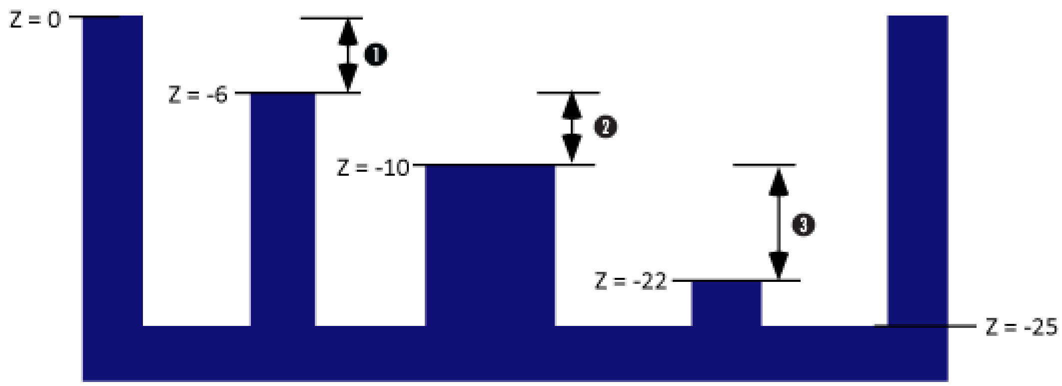
2. Drag the marker to the position you want on the toolpath.

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

Boss Top Machining

When creating pocketing operations with islands, the system will analyze the selected island geometry and create the toolpath so that it cuts to the tops of the islands based on the Z depth of the island geometry. Therefore, the Z depth of island geometry is important. In order for the system to machine the tops of islands, the island geometry must be created at the appropriate Z depth. The selected pocket geometry should also be drawn at the appropriate Z depth in order to facilitate the correct Z steps. The pockets must not be intersecting, nor can multiple pockets be contained within each other. When geometry is contained within a pocket, it is always treated as an island rather than a contained pocket. If geometry is contained within an island, it will be treated as a pocket.

Typically, the system makes constant steps in Z based on the Actual Z step value displayed in the Roughing Process dialog. In the case of islands at varying Z depths, the system will create separate pocketing operations at varying Z depths to ensure that the tops of islands are cut. The system accomplishes this by analyzing the selected pocket and island geometry. Each range from the surface Z to the island geometry is treated as a separate depth to be pocketed. The surface Z is based on the previous range. The system will take the Desired Z step entered in the Roughing Process dialog and apply that Z step to each range. If the ratio between the top surface Z and the floor Z of the range for that pocket is less than 1.5, the system will take one pass. If the ratio is greater than 1.5, the system will cut the range with as many passes as are required with each step being equal.

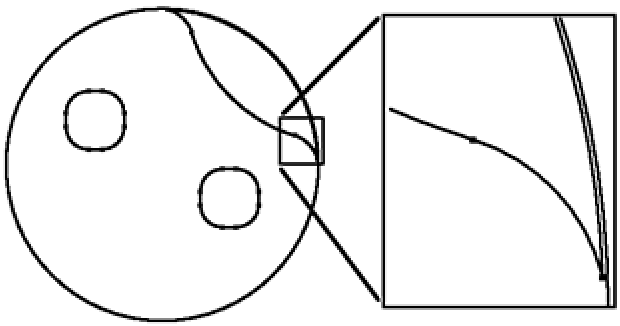


Example of pocketing with multi-level islands

The above figure illustrates a pocketing operation with three islands at varying Z depths, with a Desired Z step equal to 5mm. Four separate pocketing operations will be created from one pocketing process in order to cut to the tops of each of the islands. The first pocketing operation will cut Range 1, with a starting surface Z at 0 and final floor Z of -6mm (top of 1st boss). The second pocketing operation will cut Range 2, with a starting surface Z at -6mm and final floor Z at -10mm (top of 2nd boss). It will cut Range 2 in one Z step because the ratio between the total depth of cut and the desired Z step is less than 1.5. The third pocketing operation will cut Range 3 with a starting surface at -10mm and a final floor Z at -22mm. This pocketing operation will be cut in two steps, each 6mm because the ratio of the total depth of cut and the desired Z step is 1.5 or greater. The final pocketing operation will cut to the final floor Z of -25mm with a Z step of 3mm. The ranges are determined by the Z depths of the selected pocket and island geometry.

Pocket and island stock specified in the Roughing Process dialog is XY stock only and does not leave any stock in Z. In order to leave stock on the top of an island (in Z), the island geometry should be offset in Z to account for the desired stock amount.

As stated previously, pockets must not be intersecting. In other words, the geometry must not be touching or coincident. A solution to this situation is to offset the coincident geometry by the smallest amount possible. The following image shows a pocket with an island that is coincident. The circles were originally of the same radius. Once the geometry was made, the island's large circle was offset by 0.2mm and reconnected. A pocketing operation will now work perfectly.



Example of a solution to coincident geometry

Machining Air Geometry

You can designate geometry as "Air" by two methods: either by right-clicking the geometry and selecting Show Properties (or Show Properties of Selected when multiple elements are selected), or by selecting Toggle Wall/Air from the Modify menu. When geometry is designated as "Air" it is changed from its normal color of blue to red. This red or "Air" geometry acts as a constraint similar to regular geometry except that the toolpath will overhang this area by the amount specified in the machining dialog. The default setting is to have the tool overhang the geometry by its cutting radius.

Additionally, geometry that is designated as "Air" will affect any Roughing or Contouring operations applied to this geometry. That means there are four general possibilities on how a tool will enter and machine a pocket.

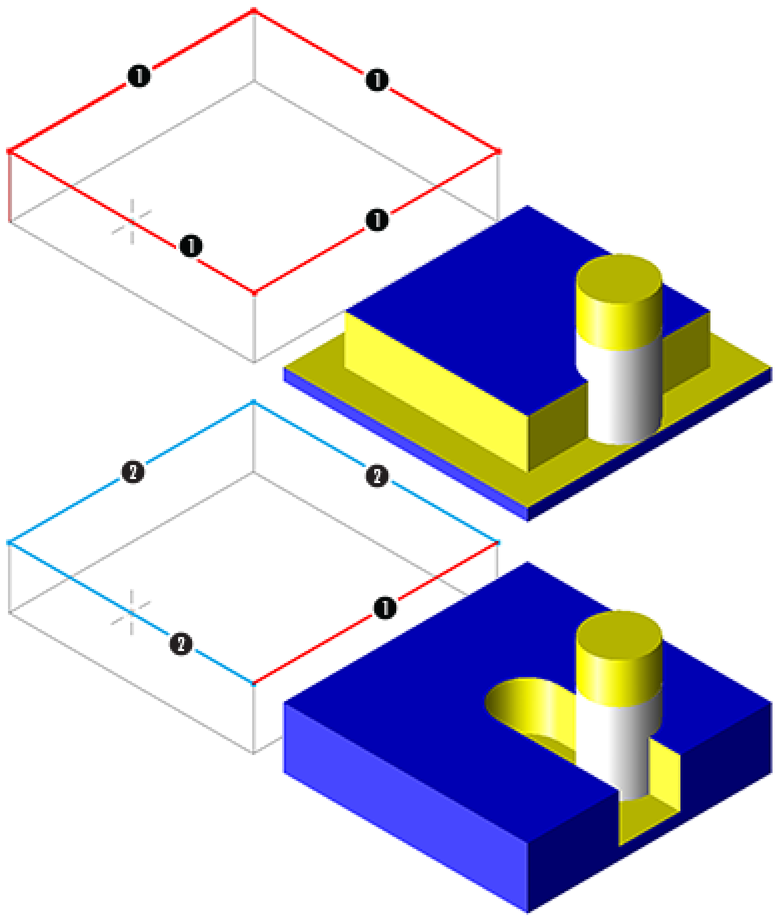
- Plunge in the center and spiral out. This is the standard method the system uses to machine a pocket or pocket with an island. See Example 2 .
- Start at the outside of the geometry and spiral in. This will occur when a complete loop of closed geometry is designated as "Air". The operation will begin a tool-on-center cut on the "Air" geometry. See Example 1 .
- Start at the inside of the geometry and spiral out. This will occur when "Air" geometry is interior to regular, "Wall" geometry. The operation will begin a tool-on-center cut of the "Air" geometry.
- Start at the outside, dig to the center, and machine outward. This occurs on "Combination" geometry (a closed loop of geometry that consists of both "Air" and "Wall" geometry). The tool will begin at a corner of the geometry, dig its way to the center and spiral out.

The **Offset/Trim** tab offers additional controls that can alter how air geometry is machined. See Offset/Trim Tab for more details.

When you use a Pocketing process on combination geometry (mixed shapes containing both Air and Wall geometry), we do not recommend selecting the Cutter Radius Compensation (CRC) checkbox. Instead, for operations on combination geometry where CRC is needed or desirable, go to the **Offset/Trim** tab and select the Trimmed finish pass checkbox, which is specifically designed to machine only Walls and not Air walls.

Example 1

This example illustrates the differences in entries between an entirely “Air” loop and a loop that is a combination of “Air” and “Wall” geometry. Note how the tool machines inward from where the “Air” geometry is coincident with the stock on the combination geometry example.



1. “Air” Geometry
2. “Wall” Geometry

Illustration of “Air” geometry toolpath compared to “Combination” geometry toolpath

Example 2

This example uses “combination” geometry to machine an open-sided pocket as well as machining around an existing pocket. Image 1 shows the existing pocket. The darker, bold lines in Image 2 are designated as “Air”. The toolpath is generated in image 3. Image 4 shows the tool machining to the center of the part so it may machine outward. Image 5 shows the tool overhanging the existing pocket. Image 6 is the completed open-sided pocket.

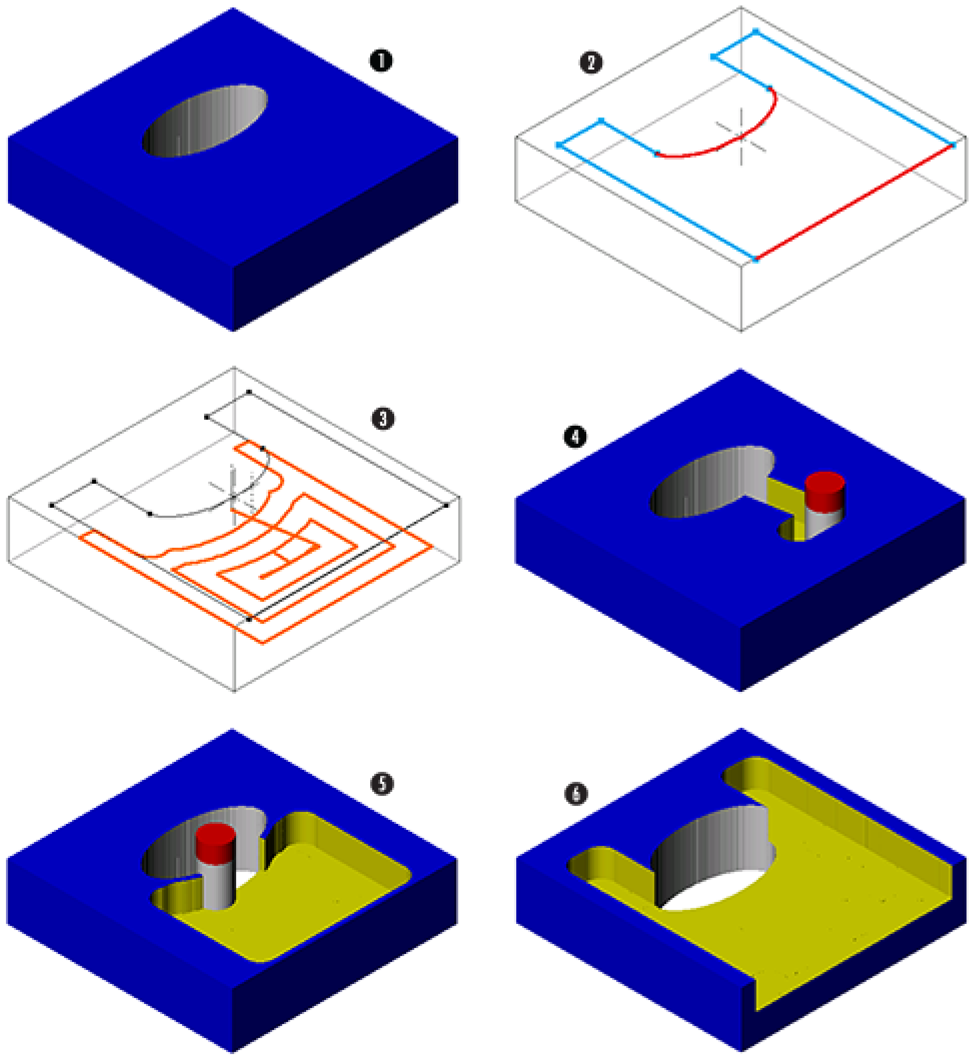


Illustration of machining combination geometry for an open-sided pocket and overhanging an existing pocket

Example 3:

This example illustrates how a user could stretch an operation's toolpath around a boss to clean up the part. The boss geometry is offset from the stock by 2mm. The outer geometry is designated as "Air". We will use a 13mm endmill to cut the part. To clean up the boss a value smaller than the 2mm offset is entered in the Minimum Cut box. This ensures that open sides with material greater than the Minimum Cut will be machined. If this value was 2mm or more then the small space would be uncut.

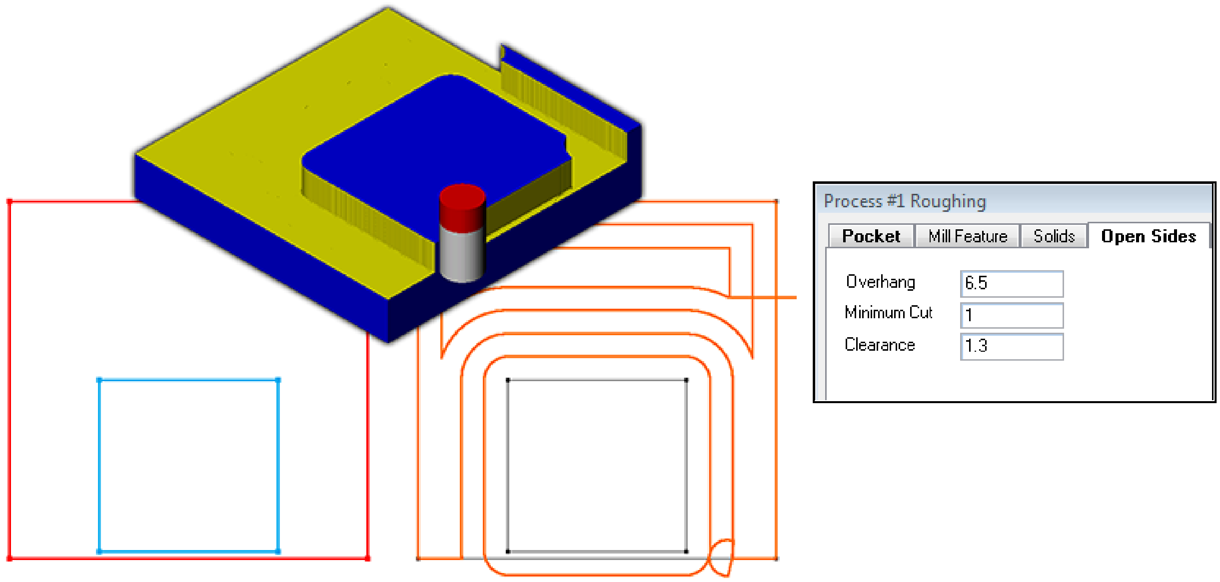


Illustration of stretching toolpath around a boss

Clearance Moves

This section details how the system handles tool moves between operations and between holes in drilling cycles. The following conventions are used in the pictures shown below.

Dashed Arrow
Rapid Move

Solid Arrow
Feed Move

CP
Clearance Plane

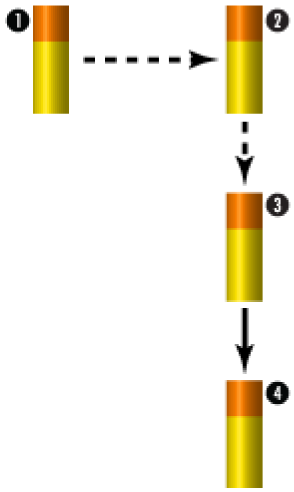
SP
Start Point; the first move of the operation but not necessarily the location of the Start Point Marker.

EP
End Point; the last move of the operation but not necessarily the location of the End Point Marker.

Op1
Operation 1; the first series of cuts made on the part.

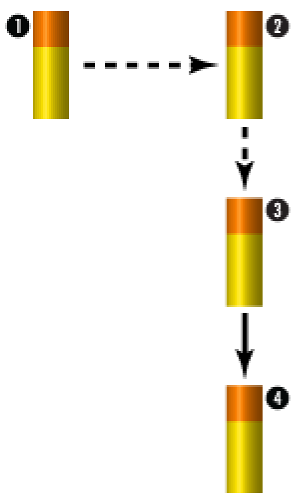
Op2
Operation 2; the second series of cuts made on the part.

Entry Move: Same Tool



1. Exit CPZ, XY End Point of Op 1
2. Exit CPZ, XY Start Point of Op 2
3. Entry CPZ, XY Start Point
4. Cut Depth Z, XY Start Point

Entry Move: Tool Change



1. Tool Change Position
2. Tool Change Z, XY Start Point
3. Entry CPZ, XY Start Point
4. Cut Depth Z, XY Start Point

Intra-Operation Moves

All operations, except for a few exceptions noted below, will follow the same rule for clearance planes through multiple passes. As a clarification of terms used, the following is a definition of each clearance plane as used by the system.

CP1
The master Clearance Plane defined in the Document Control dialog.

CP2
Entry Clearance Plane defined in the Process dialog

CP3
Exit Clearance Plane defined in the Process dialog

Multiple Passes

Rather than always retracting to a fixed Clearance Plane above the Surface Z, successive Z steps will have an incremental CP2. This clearance plane will always be offset down by the CP2 value plus the designated cut depth. Between operations, the tool will Rapid to CP3 and traverse at that clearance plane.

If illustrated using three passes, the toolpath would read as the following:

First Step

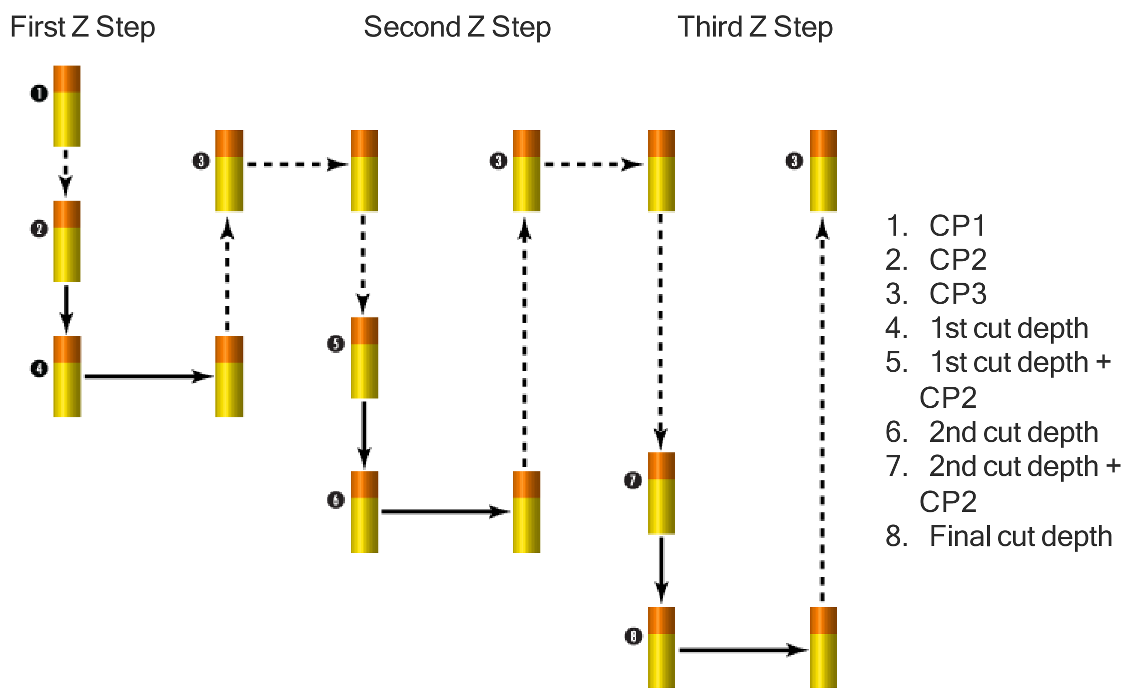
The tool will rapid down from *CP1* to the *CP2* value designated in the Process dialog. The tool will then Feed to the first cut depth, cut the part, rapid up to *CP3* and rapid over (traverse) to the start position of the next pass.

Second Step

The tool will rapid down to *CP2's* amount above the previous cut depth (this depth is now considered *CP2*) then feed to the new cut depth, cut the part, rapid up to *CP3* and traverse to the start position.

Third Step

The tool will rapid down to the former cut depth (the new *CP2*) then feed to the new cut depth, cut the part, rapid up to *CP3* and traverse to the next pocket or the tool change position.

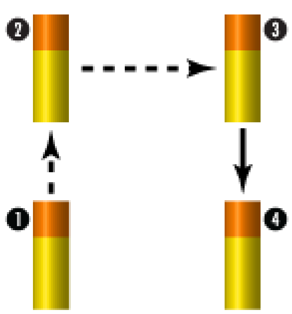


Not all processes and options will follow this rule. Items such as drilling will still provide the option of which clearance plane to use on retracts. Processes that have exceptions to the standard clearance plane rules follow.

Process Type or Settings	Clearance Plane Used
Drilling and Mill Bore	The option of CP1 or CP2
Patterns and Thread Milling	Always retract to CP2 without stepping CP2 down (between Z reps)

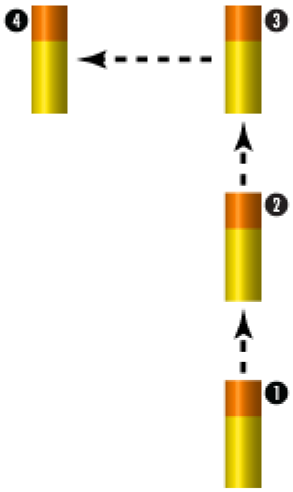
Process Type or Settings	Clearance Plane Used
Prefer Subs is OFF	All retracts will be made to CP3
Retracts (Entry/Exit Connect) are OFF	There are no in-operation retracts. All between op retracts will be made to CP3
Depth First is OFF	Final pass is at CP3, all prior passes are at CP3 plus the step amount(s).
Engraving Multiple Contours	All retracts will be made to CP3, stepping down with successive passes
Open Pockets	All retracts will be made to CP3, stepping down with successive passes
Material Only Pocketing of Separate Areas	All retracts will be made to CP3, stepping down with successive passes
Spring Pass on an Open Contour	All retracts will be made to CP3, stepping down with successive passes
Rotary Repeats (Mill/Turn and Rotary Mill)	Always retract to CP3 between positions
Surfacing Processes	All retracts will be made to CP3
Multi-Parts	The option of CP3 or Full Up
Rotary Positioning (Advanced CS)	Retract Full Up

Exit Move: Same Tool



1. XYZ End Point of Op1
2. Exit CPZ, XY End Point of Op1
3. Entry CPZ, XY Start Point of Op2
4. Cut Depth Z, XY Start Point

Exit Move: Tool Change



1. Cut Depth Z, XYZ End Point
2. Exit CPZ, XY End Point
3. Tool Change Z, XY End Point
4. Tool Change Position

During a drilling cycle, the tool can retract to one of two different Z clearance planes when it is making inter-hole moves. There are two retract values in the Drilling Process dialog. The top retract value is the Entry Clearance Plane entered for the process. The bottom retract value is the Clearance Plane Z specified in the Document Control dialog.

2 ½ Axis Surfacing

The system provides users with the ability to create tapered walls and swept surfaces on roughing and contouring operations using the 2 1⁄2-axis surfacing capability of the system. The name is derived from simple surfaces that are machined as a series of 2-axis toolpaths, utilizing the CNC machine's circle interpolation capability (G2/G3). This produces the smoothest-looking part and smallest program. These additional features are integrated into the existing roughing and contouring processes. Users can specify the wall of any contour or pocket as straight (90°), tapered with optional top and bottom fillets or swept with a specified drive curve shape.

When using the 2 1⁄2-axis capabilities of the system, accurate depth positioning of geometry is important. With swept surfaces, Surface Z and Floor Z values are not entered in the Entry/Exit Clearance Diagram in Roughing or Contouring Process dialogs. The drive curve determines the depth of the toolpath. The system creates the toolpath by attaching the drive curve to the base curve geometry at the Z depth of the base curve. Therefore, if a pocket or contour with a swept wall does not start at Z = 0, the base curve geometry must be created at the appropriate Z depth. When using the Taper w/Fillets selection, Surface Z and Floor Z values must be entered in the Entry/Exit Clearance Diagram to determine the overall depth of cut. Accurate depth positioning of geometry is also important when creating walls with tapers because the system calculates the taper from the Z level of the geometry, NOT the Surface Z level entered in the Entry/Exit Clearance Diagram.

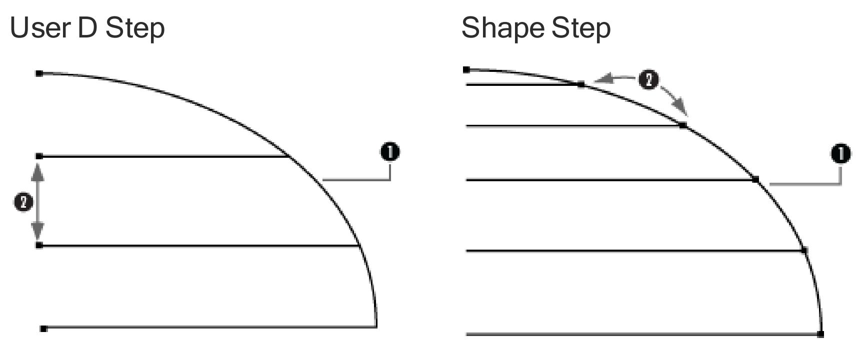
There is a button in the Entry/Exit Clearance Diagram in Roughing and Contouring Process dialogs that accesses the Wall Choices dialog. The Wall Choices dialog allows the user to specify whether the pocket or contour will be cut with a straight (90°) wall, a tapered wall, or a swept curve. Refer to the Contouring and Roughing Process sections for additional information on the Wall Choices dialog.

Swept Shapes

When creating swept shapes, the user designates a base curve shape and a drive curve shape. The base curve is the geometry selected for the cut shape. The drive curve is the shape of the wall that will be swept around the base curve to create the surface.

The drive curve must be an open, terminated shape. The drive curve must also be a one-to-one function, meaning that if a horizontal or vertical line is drawn through the curve, it will only intersect the curve in one place. The drive curve is designated by the D-pointer Machining Marker, which comes up on the screen when the cut shape (base curve) is selected. If there is an open, terminated shape in the same workgroup as the selected base curve, the D-pointer will appear on one of the terminated ends of the open shape. The D-pointer can be dragged to a different location, but it can only be placed on a terminated point. The drive curve will be attached to the base curve at the selected start point of the base curve using the end point indicated by the D-pointer. Base Curve depth axis must be aligned with the Drive Curve vertical axis.

The overall depth of cut for swept shapes is calculated from the drive curve. The step move for each pass depends on whether User D Step orShape Step is selected in the Wall Choices dialog. The pictures below illustrate how the step is created based on the selection made. Shape Step will provide a smoother finish, while User D Step, in most cases, will provide for faster material removal. Shape Step is a good choice for finishing operations and User D Step works well when roughing.



1. Drive Curve
2. Step in Z = 2

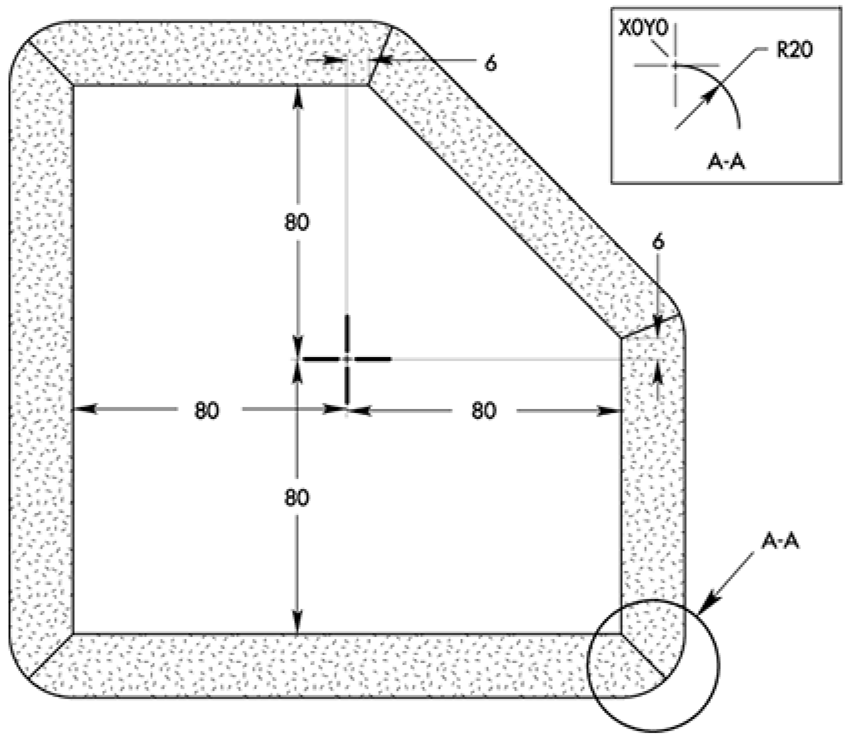
Comparison of User-Defined steps and Shape steps

Swept Shape Example

If you are not familiar with the basics for creating geometry and toolpaths using the system, refer to the exercises in the *Geometry Creation* guide and the Mill tutorial. The instructions in those sections are much more detailed than the ones found in the following examples. In this example, we will machine a very simple swept surface using a contouring process.

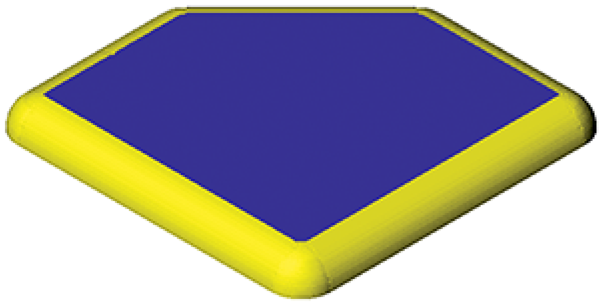
1. Create a new part with the following stock specifications: X+ = 105, X- = -105, Y+ = 105, Y- = -105, Z+ = 0, Z -= -20.

2. Draw the 5-sided polygon shown. The radii at the corners will be added when we create the machining process. This will be the base curve shape.



3. In the same workgroup, not connected to the polygon, draw a 90° arc with a 20mm radius, as shown in view A-A. This will be the drive curve shape.
4. The exact position of the drive curve is not important, but the shape must be an open, fully terminated shape. Terminator points are created by selecting the feature, in this case the arc, and the point at which it should be terminated and clicking on the Connect-Disconnect button. Both ends must be terminated. The drive curve must be drawn in the same workgroup as the base curve.
5. Create a ball endmill (Ball EM) with a 25mm diameter.
6. Create a contouring process using the ball endmill. Click the Wall Control button to access the Wall Choices dialog. Select the Swept shape choice and choose the DC EP Left selection. Select the Top Down and One direction items. Select Shape Step and enter 2 for the step amount. Close the Wall Control dialog and the Contouring Process dialog.
7. Click the shortest horizontal line at the top of the polygon to select the cut shape. Make sure that the right arrow and outside circle are selected on the machining markers. The D-pointer should snap to one of the terminated ends of the arc. If necessary, move the D-pointer to the top end point of the arc. Click the Do It button.

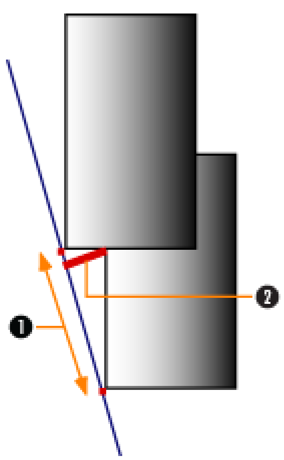
The cut part rendered image of the part, shown in the isometric view, should look like this picture. You will have additional stock on the angled side of the shape that can easily be roughed away.



Tapers with Fillets

The Taper w/Fillets option allows the user to specify a taper for the wall of a pocket or contour and also automatically add top and bottom fillets. For roughing processes, tapers and fillets can be added to both pocket and island walls. If a contouring process is in the same Process list as a roughing process, a taper and fillets can also be specified for islands in the contouring process.

When Shape Step is selected, an additional parameter called theRidge Height becomes active. The ridge height is an approximate calculation based on the wall angle and the tool specifications as to how much material will be left on each pass of the tool. The Shape Step and Ridge Height text boxes are interactive in that either value can be entered and the other will be calculated. The smaller the ridge height, the better the finish on the wall.

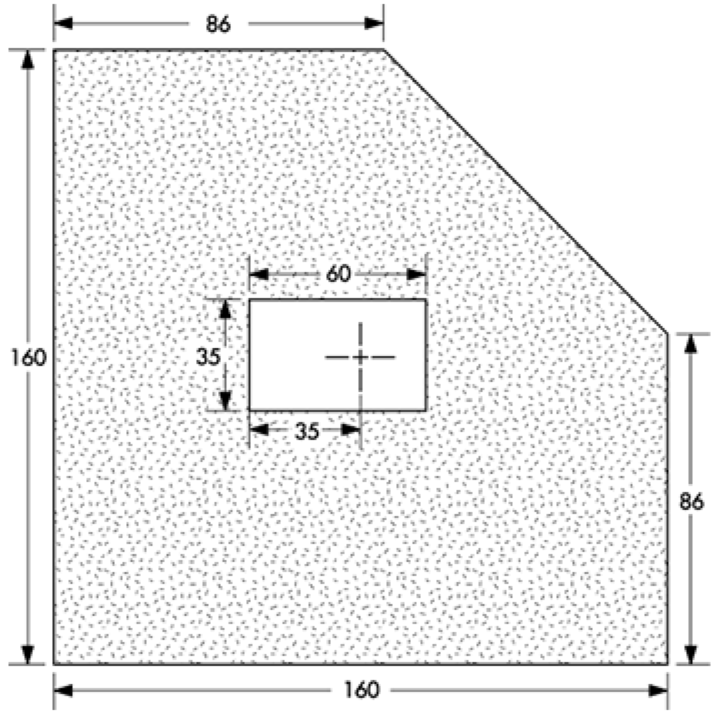


1. Shape Step
2. Ridge Height

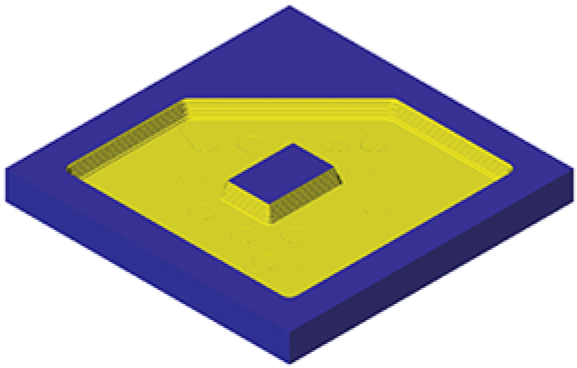
Tapered Wall Example

We will machine the same shape as the swept shape example. You can either recreate the geometry in a separate file, duplicate the swept surface file, or reopen the swept surface example and make changes to that file.

1. Create a simple shape like a circle or square within the polygon to act as a boss. Your part geometry should look like this picture.



2. Create a rough endmill (rEM) with a 20mm diameter and a 1mm bottom corner radius. Create a finish endmill (fEM) with a 12mm diameter and a 1mm bottom corner radius.
3. Create a roughing process using the 20mm rEM. Enter Pocket Stock± and Island Stock± values of 0.5mm. Click the Wall Control button to access the Wall Choices dialog. Select the Taper w/ Fillets option and enter the following values for the Pocket wall: Top Fillet 2, Side Angle 20, Bottom Fillet 2. Enter a Side Angle of 20 for the Island wall. Select User D Step and enter a value of 2. Surface Z and floor Z values must be entered in the entry/exit clearance diagram when using the Taper w/Fillets option. Enter 0 for the surface Z and -12 for the floor Z.
4. Create a contouring process using the 12mm fEM. Click the Wall Control button. Select the Taper w/Fillets option. Because the contouring process is in the same Process list as the pocketing process, there are Island wall specifications. The values for the pocket and island walls should default to the values entered in the roughing process. Select Shape Step and enter a value of 1.
5. Select the outside pocket wall and the boss; you will need to hold down the `Ctrl` key to select the island. Click the Do It button to create the toolpath.



Before rendering the part, change to one of the side views to better see the taper and fillets created by the toolpath. The cut part rendered image of the part should look like this picture.

Pattern

The Pattern checkbox is available for all 2D and 2½D milling processes and for Advanced 3D. It allows the toolpath generated by the process to be duplicated in different locations on the part. This is accomplished by creating a template workgroup and a pattern workgroup. Any points within the pattern workgroup that are unconnected will produce the template when Pattern is selected in the process dialog and the corresponding workgroup is selected from the drop-down list.

Each of the points in the pattern workgroup acts as an origin point. If the template is not drawn at X0,Y0 then the toolpath will be offset relative to the template. Only the origins created in the pattern workgroup selected will produce toolpath. To create toolpath for the template as well, it must also have an origin defined in the pattern workgroup.

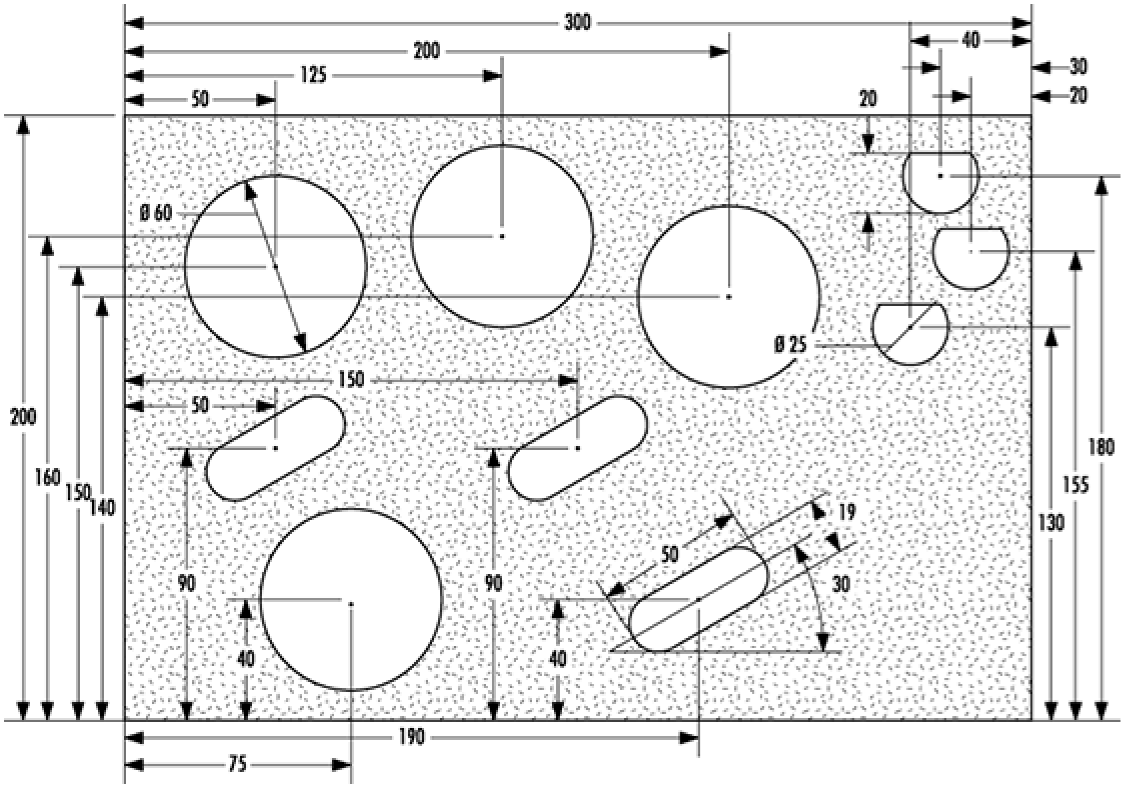
It is recommended that each template toolpath and Pattern of points be contained in separate workgroups. For more information on using multiple workgroups, refer to the Geometry Creation guide.

Using Pattern will create subprograms in the post file to make the code more efficient.

Pattern Example

The following example of using a Pattern will machine the base plate shown below. Six workgroups will be created–three template shapes, and three patterns for the templates. Each of the template shapes will be created around the origin point (X0Y0) of the workgroup so that the points in the pattern workgroup can be created in the same location as they appear on the blueprint.

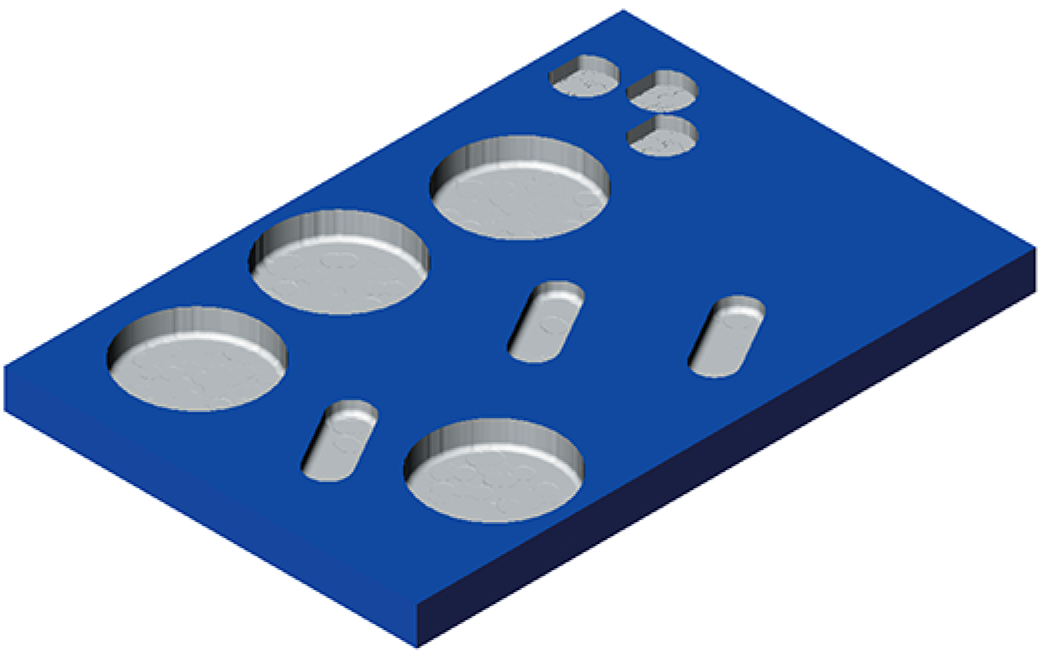
We will create the stock outline slightly larger than indicated on the print so that the template shapes can easily be created around the origin point. Once the machining operations have been created, we can change the stock to a more appropriate size. Instead of adjusting the stock size, the template shapes could be created and then translated about the origin point using the Modify menu. Another option (using Advanced CS) would be to create another coordinate system with an origin point more towards the center of the stock shape.



1. Create the stock shape with the following specifications: X+ = 300mm, X- = -25mm, Y+ = 200mm, Y- = -25mm, Z+ = 0mm, Z- = -20mm. We will adjust the stock size when we are done programming the part.
2. The first workgroup will contain the pattern of points for machining the 60mm diameter circles. Create points at X = 50, Y = 150; X = 125, Y = 160; X = 200, Y = 140; X = 75, Y = 40. These are the center points of the circles.
3. Create a second workgroup. Create a circle with a 60mm diameter with X0Y0 as the center point.
4. Create a 15mm finish endmill (fEM) with a 2mm bottom corner radius. Create a roughing process with this tool. Cut Width = 7.5mm; Finish Entry/Exit 90° Radius = 1.5mm; Surface Z = 0; Floor Z = -15mm; Desired Z Step = 15mm. Click the Pattern checkbox and select Workgroup #1 in the pattern pop-up menu. If you are creating this part in Level 2 make sure that Use Stock is unchecked, since the circle extends outside of the stock boundary. Select the circle and click the Do It button to create the operation. Notice that toolpaths are only created at the points contained in the pattern workgroup and not where we initially drew the circle.
5. Create a third workgroup. This will contain the pattern of points for the 2" slots. Create points at X = 50, Y = 90; X = 150, Y = 90; X = 190, Y = 40.
6. Create a fourth workgroup. Create the geometry for the slot around X0Y0.
7. Using the same process as the Circle pattern create a second operation using the slot pattern. Select the slot pattern Workgroup #3 in the Pattern pop-up menu. Change the Depth of the cut to 6mm, select the slot geometry and create the operation.

8. Create a fifth workgroup. This will contain the pattern of points for the D-Holes. Create points at X = 260, Y = 130; X = 270, Y = 180; X = 280, Y = 155.
9. Create the last workgroup. Create the geometry for the D-Holes around X0Y0.
10. Create a 6mm finish endmill (fEM) with no bottom corner radius. Create a roughing process with this tool. Select Workgroup #5 in the Pattern pop-up menu. Select the D-Hole geometry and create the operation.
11. Change the stock size to match the blueprint: X+ = 300, X- =0, Y+ = 200, Y- =0, Z+ = 0, Z- = -20.

You should have a total of six workgroups and three operations which machine the base plate. If you encounter any problems, check to make sure that the correct pattern of points is selected in the Pattern pop-up menu for each of the operations. Also, make sure that each of the template shapes was drawn around the origin point, X0Y0. The cut part rendered image of the base plate should look like the picture shown here.



Engraving

The system has the capability to contour multiple shapes with the tool on center. Coupled with the text creation function, which generates geometry from any TrueType font, the user is able to engrave text. With the ability to create, import and machine splines, this feature also allows the user to machine artwork. The contouring function is used to perform engraving.

Usually, when creating a contouring process, only one continuous shape can be defined as the cut shape. When engraving, multiple shapes can be selected to be machined on the centerline of the tool. To engrave, first select all of the shapes to be machined. Then create the contouring process.

When more than one shape is selected prior to creating a contouring process, most of the fields in the Contouring Process dialog are grayed out. The information in the Entry/Exit Clearance Diagram and the speeds and feeds must be entered. The Patterns function is also available to create the toolpath in multiple locations on the part (if the contouring process is created before geometry is selected the other information in the process dialog can be entered, although it will not be used to create the operation if more than one continuous shape is selected for the cut shape).

The system will generate one operation that will contour every selected shape along the centerline. The connective moves between the noncontinuous shapes to be machined are calculated by the system and incorporated into the operation's toolpath. The system calculates these connective moves based on the Entry Clearance Plane entered in the Process dialog. The tool will use the Entry Clearance Plane value as the retract level for the last Z pass of the toolpath when the tool is cutting at the final Z depth for the operation. As a result, earlier passes will retract to a higher Z level than the Entry Clearance Plane.

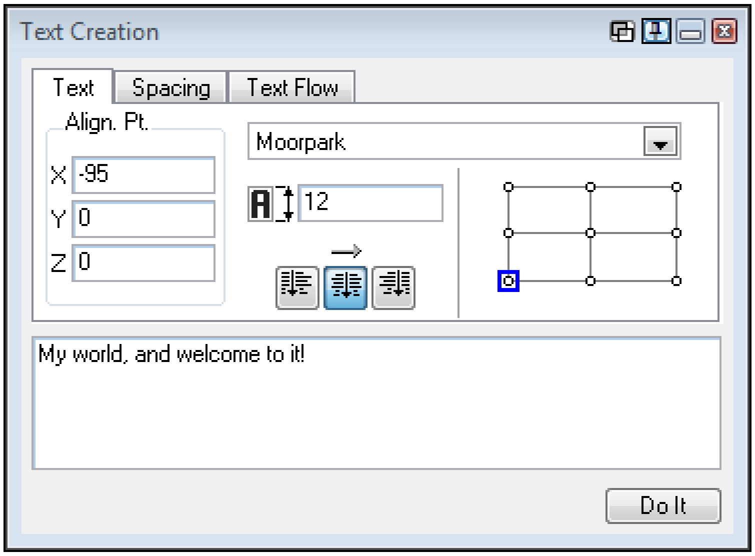
Normally, when geometry is selected as the cut shape for a contouring process, Machining Markers appear on the selected shape and are positioned to cut either the entire connected shape or a portion of the shape. If more than one continuous shape is selected prior to creating the contouring process, the Machining Markers do not come up on the screen. If the cut shape geometry is selected after the contouring process is created, the Machining Markers will appear on the first shape selected. Upon selection of another shape, the markers will disappear and automatically machine all selected shapes along the centerline. The `Ctrl` key must be held down in order to select multiple shapes.

Engraving Text Exercise

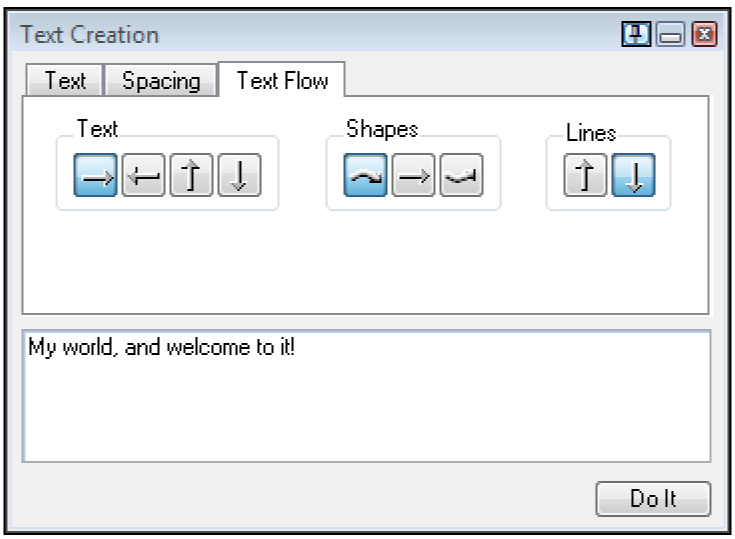
1. Create a new part with the following stock specifications: X+ = 100, X- = -100, Y+ = 75, Y- = -75, Z = 0, Z- = -25.

2. Open the Geometry Creation palette and click the Text-Auto Shape button. Click the Text Creation button (button with an 'A'). This will bring up the Text Creation dialog. Detailed descriptions of the items in the Text Creation dialog can be found in the *Geometry Creation* guide.

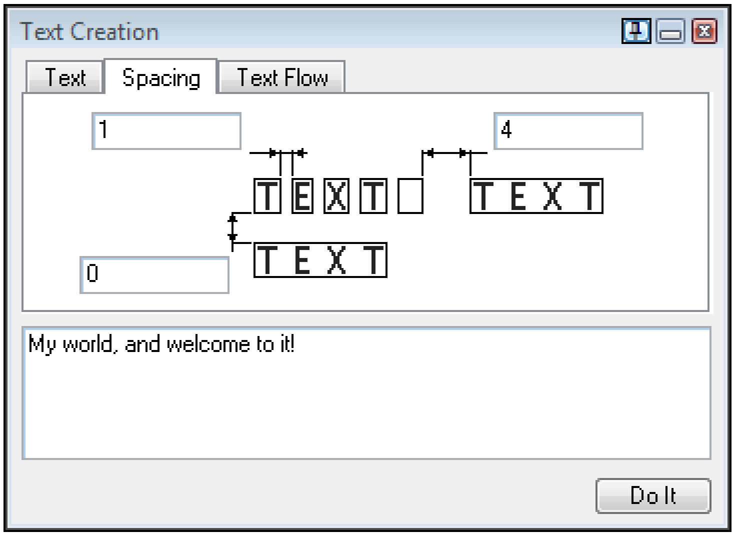
3. Enter the information shown below in the Text Creation dialog (for now we will not worry about spacing or text flow). The font pop-up menu should contain all TrueType fonts available on the system. Click the Do It button at the bottom of the dialog. The text should appear across the center of the stock. Turn off Draw Points from the View menu. This will make the text clearer.



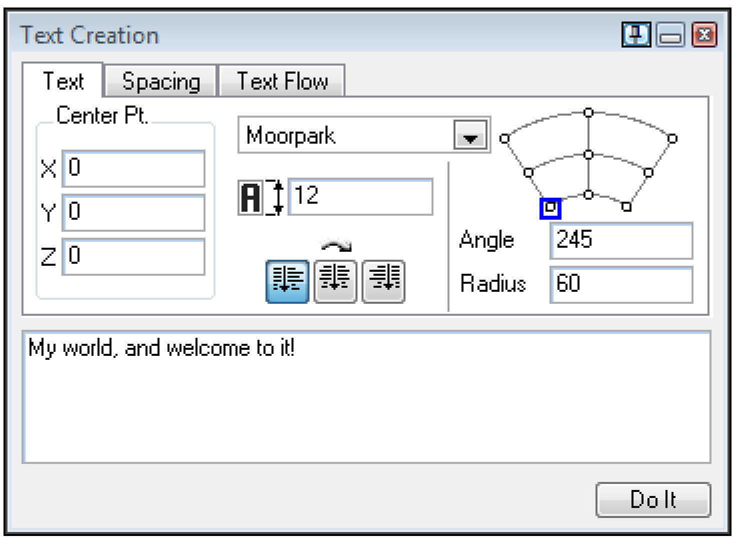
4. Select Undo from the Edit menu to erase the text. Now we will create text on an arc. Click the Text Flow tab. Click the Clockwise Arc button (first button) in the Shapes box as shown.



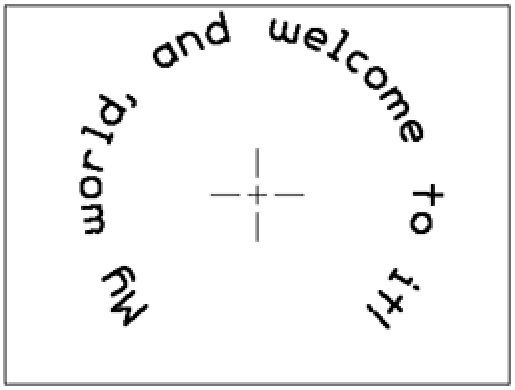
5. Click the Spacing tab. Enter 1 for the tracking amount (space) between letters and 4 for the tracking amount (space) between words as shown.



6. Click the Text tab. Note that the Alignment Diagram is different and a Radius and Angle value must be entered. The Radius specifies the size of the arc and the Angle specifies the position on the arc that the text will begin. Also, a centerpoint for the arc must be entered rather than an alignment point. Enter the values shown below.

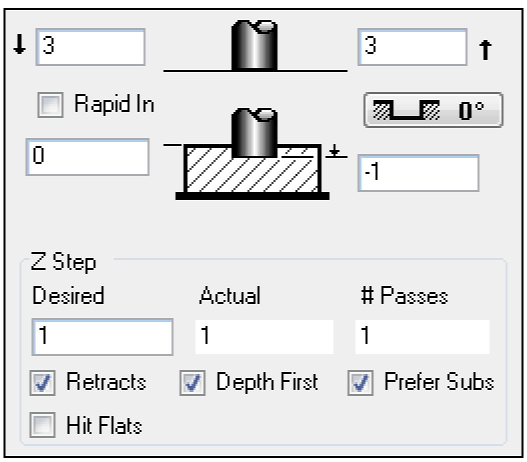


7. Click Do It to create the text. Close the Text Creation dialog. Close the Geometry Palette. Your screen should look like the picture shown below.

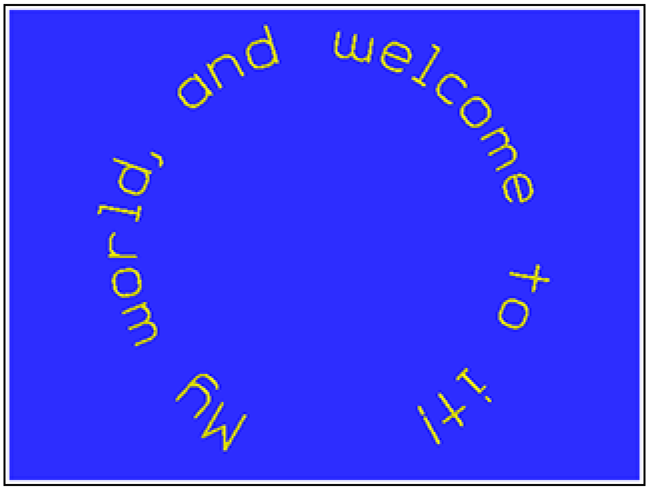


8. Click anywhere on the screen. Choose Select All (`Ctrl+A`) under the View menu. This will select all of the text.

9. Open the Tool list and create a Spot Drill with a 1mm diameter. Open the Machining palette. Create a Contouring Process tile using the Spot Drill. Note that most of the items in the Contouring Process dialog are grayed out. When more than one continuous shape is selected prior to creating a Contouring Process tile, the system will machine all the selected shapes with the tool on center (engraving). Many of the items in the Contouring dialog will be grayed out. Enter the information shown below in the Entry/Exit Clearance Diagram.



10. Click the Do It button to create the operation. Note that only one operation is created and the toolpath machines all the shapes and includes the connective moves between shapes. The Entry Clearance Plane value is used calculate the connective moves between shapes. Render the part. The cut part rendered image should look like the picture shown below.



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.

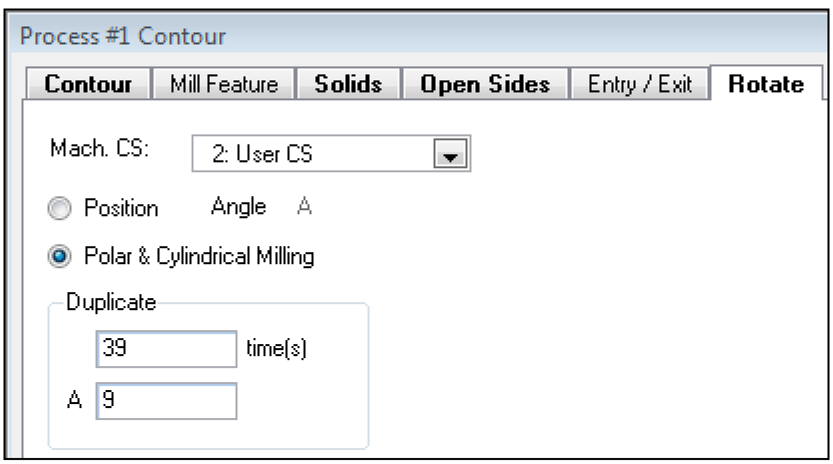
Polar & Cylindrical Milling

This information applies strictly to interaction with the Mill module. Polar & Cylindrical Milling in conjunction with Turning-specific Mill/Turn functionality can be found in the *Mill/Turn* guide.

The Polar & Cylindrical Milling option is an add-on option that enhances the Mill functionality. It allows for continuous A-axis or B-axis rotation when programming milling operations. This is often referred to as wrapping. This section describes functions that are specific to the system when the Polar & Cylindrical Milling option is installed. This section assumes a familiarity with the standard Mill functionality described elsewhere. The term "A-axis" is used as a general term for the A-axis or B-axis except where noted.

When a 4-axis Vertical Mill machine is chosen in the Document Control dialog, the system will allow for A-axis rotation. When a 4-axis Horizontal machine is chosen the system will program for B-axis rotation. This is discussed in Top half of DCD tab , in the section detailing Part Setup.

Polar & Cylindrical Milling and Rotary Interpolation



The term rotary is used to signify the continuous or simultaneous movement of a rotary axis. In the case of Mill/Turn parts, the rotary axis is referred to as the C-axis. The Polar & Cylindrical Milling option allows for the wrapping of toolpaths about the A-axis by rotary interpolation of the A-axis during a milling operation. When the Polar & Cylindrical Milling option is installed, the Rotate tab for milling processes contains two rotation options: Position and Polar & Cylindrical Milling. The operation can either be programmed as a simple position move (Position, described in Rotate Tab) or as a wrapped toolpath with continuous A-axis motion (Polar & Cylindrical Milling).

Flat vs. Radial geometry

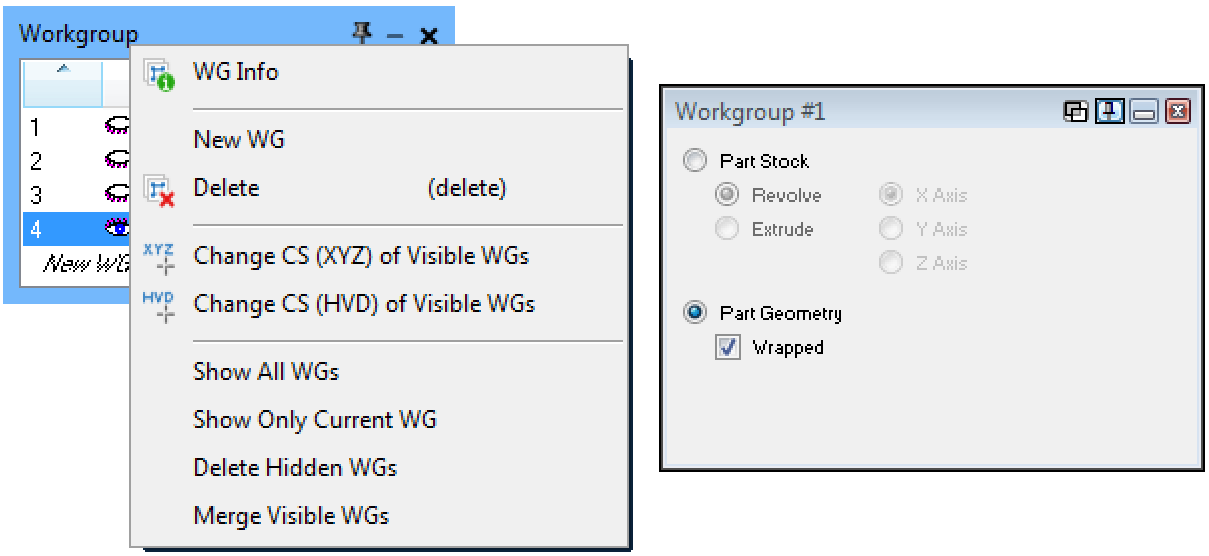
Geometry can either be created as *flat geometry* or *radial geometry*:

- Flat geometry is defined using XYZ values.

- Radial geometry is defined either using XAR values (where R designates the radius or diameter and A designates an angle of rotation for the A-axis) or, when working with B-axis rotation, using BYR values (where R designates the radius or diameter and B designates an angle of rotation for the B-axis).

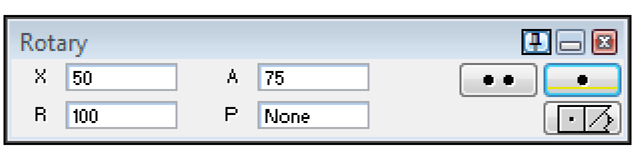
Geometry does not need to be viewed as wrapped in order to be machined using the Polar & Cylindrical Milling function. The toolpath that results with the Polar & Cylindrical Milling option checked will be the same whether the geometry selected for the cut shape is viewed as flat or wrapped.

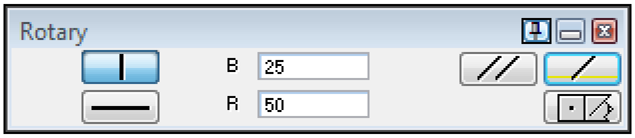
Two interface items must be used in order to create and view radial geometry. First, in the Workgroup Info dialog, the Wrapped checkbox must be selected. (To open the Workgroup Info dialog, either double-click the workgroup name or else choose WG Info from the context menu summoned by right-clicking the title bar of the Workgroup list dialog.)



WG list context menu and WG Info dialog

In addition to the Wrapped checkbox, the Wrap Geometry button in Modify>Wrap or the Wrap WGs button in the floating toolbar must be selected in order to view geometry radially on the screen. When both of these items are appropriately selected, the system will be in *radial mode*. When you are working in radial mode, geometry dialogs that require coordinate input will contain specifications for an A value (for angle of rotation) and a radius value. For example, when you create points by entering coordinates, the text boxes will not be labeled X, Y, and Z, but rather X, A, and R (or Y, B, and R).



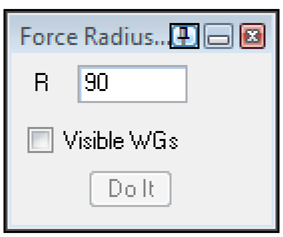


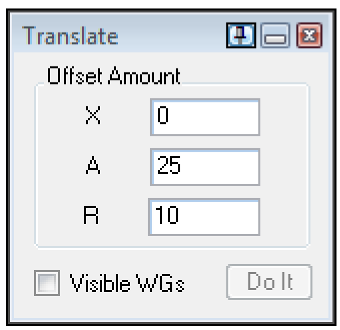
Geometry dialogs in radial mode

Modify Menu Items

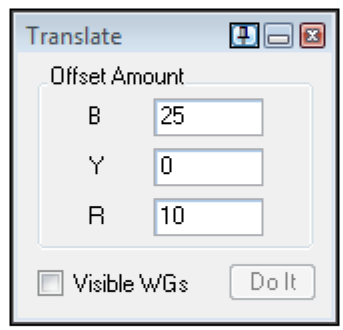
When you work in radial mode, certain Modify functions are enhanced to provide for radial value input. The two primary functions are the Force Depth item and the Translate item. The Force Depth item is a Force Radius item when in radial mode. Users can enter an absolute radius value and the selected geometry will be changed to that radius. The dialog Translate does not change, it still modifies geometry in relative values. When the radius value has been changed the geometry will move to that depth and the overall size of the geometry will change so that the angle of the arc remains the same. Thus the same toolpath will be generated.

The following images show dialogs in radial mode.



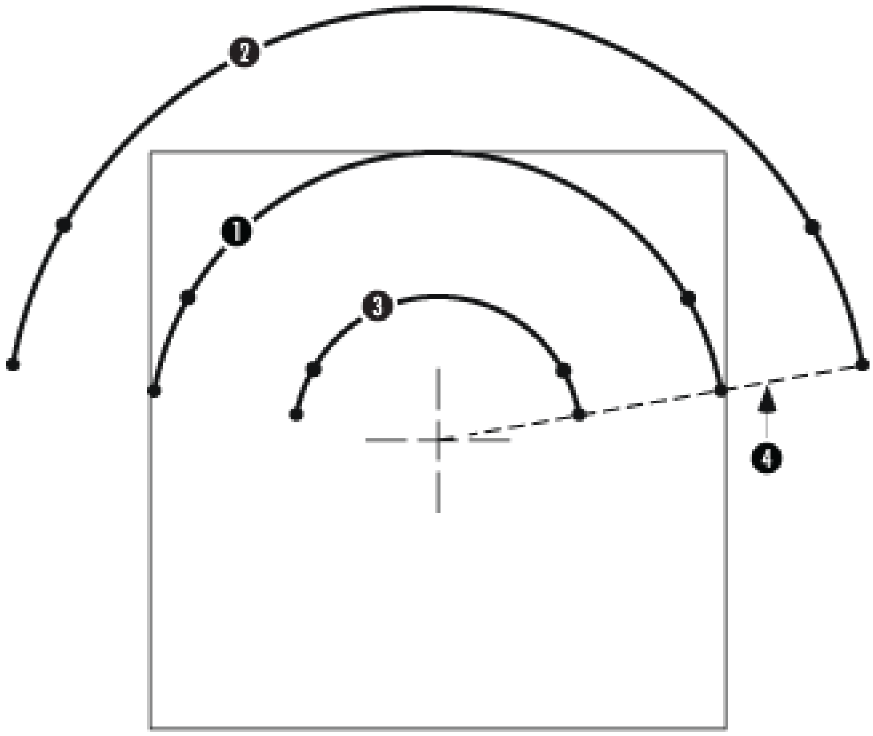


5-Axis Vertical Mill



4-Axis Horizontal Mill

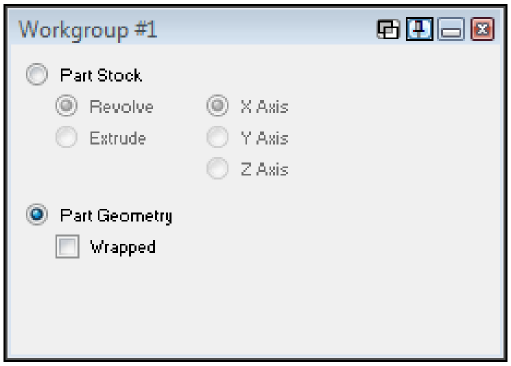
The following figure illustrates geometry that has been modified in radial mode.

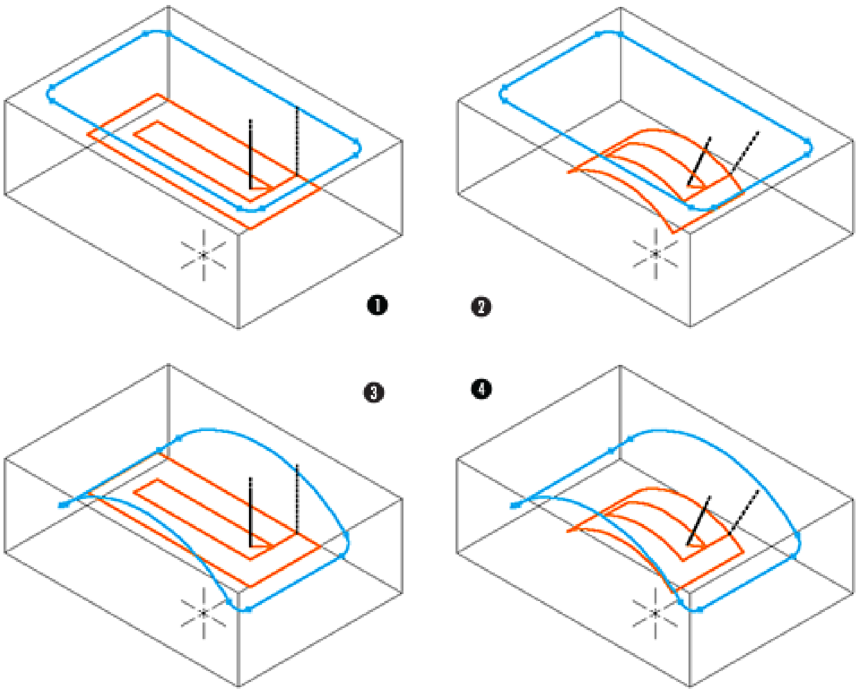


1. Original geometry at 150mm
2. Geometry modified to 225mm
3. Geometry modified to 75mm
4. Note the constant angle of the geometry

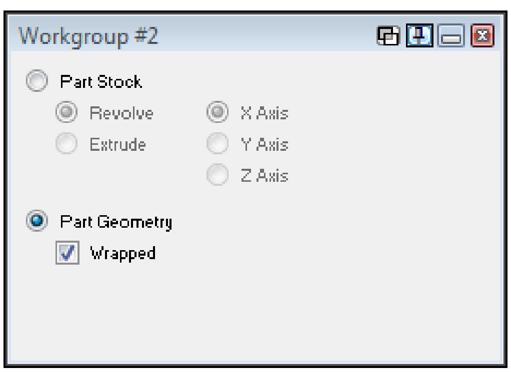
An example of geometry whose depth has been changed in radial mode

The following figure illustrates the different results you will get in geometry and toolpath when the Wrap Geometry or Wrap WGs button is active.





1. Designated as Position in the process dialog
2. Designated as Polar & Cylindrical Milling in the process dialog
3. Designated as Position in the process dialog
4. Designated as Polar & Cylindrical Milling in the process dialog



Geometry and toolpath variations with the Wrap Geometry or Wrap WGs button active

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.

Rendering Polar and Cylindrical Milling

The quality of the rendered image in Op Sim and Machine Sim is determined by the speed of the rendering. The angular tolerance of the rendered image is very tight when the rendering is slow, while the tolerance is loose when rendering at higher speeds. This directly affects the quality of the rendered image. The output is unaffected.

Low Angular Tolerance High Angular Tolerance

A comparison of low and high angular tolerances in Op Simulation without Multi Axis Rendering

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.

Mill Post Label Definitions and Code Issues

Mill 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 Mill posts are named and what they do. Also included are brief explanations of code issues that might be encountered in Mill posts.

3-Axis Mill

Label Definitions

M	This designates a regular 2-axis or 3-axis mill post. A 3-axis mill post has 3 linear axes (X, Y, and Z) that can position and cut simultaneously. Example: Fanuc 6M [VG] M001.19
N	This designates a mill post that does not use subprograms. This is known as a “Long Hand post”. Subprograms are frequently used for multi-process drilling, Z-repeat milling, patterns, thread milling, rough and finish mill bore, multiple parts, and so forth. Any mill post can be modified into a Long Hand post. Example: Fanuc 6M [VG] NM001.19
U	This designates a Mill post that supports Spline Interpolation (also known as NURBS). Example: Fanuc 15M [VG] UM001.19

Code Issues

- Cutter Radius Compensation (CRC)
  - Cutter Radius Compensation options include Tool Center and Tool Edge. These are found in the Preferences dialog, Machining Prefs. tab, Mill CRC Type.
  - The Tool Center option outputs code to the center of the tool in contouring and roughing operations.
  - The Tool Edge option outputs code to the edge of the tool in contouring operations. However, it outputs to the center of the tool in roughing operations.
  - Many CNCs need CRC turned on (e.g. G41/G42) on the entry line move. This is a move immediately prior to the first cutting move. This line move can be programmed by specifying an Entry Line move in the Contour Process window.
  - Many CNCs need CRC turned off (e.g. G40) on the exit line move. This is a move immediately following the last cutting move. This line move can be programmed by specifying an Exit Line move in the Contour Process window.
  - If the Tool Center option is selected, the value entered into the CNC control's offset register should be 0. The system has already compensated the values in the output by the tool radius.
  - If the Tool Edge option is selected, the value entered into the CNC control's offset register should be the tool radius. The values in the output are to the edge of the tool.
  - The method that CRC is output in the posted code can be changed through a post modification.
- Subprograms vs. Long Hand
  - The Prefer Subs checkbox in process dialogs only toggles between subprograms and long hand output for multiple Z steps in contouring and roughing operations.
  - The Prefer Subs checkbox is not available for drilling operations. If multiple processes are used for multiple holes, drilling subprograms will be created.
  - Patterns, multiple parts, and rotary repeats will always output subprograms.
  - If a Long Hand post is used, no subprograms will be output.
- Absolute Subs vs. Incremental Subs
  - The system will only output incremental subprograms during Pattern, Mill Bore, and Thread Milling operations. All other operations that create subprograms will do so in absolute.
  - However, a subprogram that uses ramping or helical milling for entry moves output these moves in incremental. After the entry moves are complete, the subprogram switches back to absolute for all remaining moves.
  - If incremental output is selected in the Post window, all moves are incremental.

Feature Drilling

The Feature Drill capability allows for multiple R levels in a drill cycle. When used with existing posts these multi-R level operations will be split into a separate operation for each R level. The machined results are exactly the same, just the appearance of your posted output will be different. It is highly recommended that you take a close look at the posted output of your first feature drill part in this release. If you request it, and your machine supports it, your post can be upgraded to output multiple R levels within a single drill cycle.

Advanced CS

Advanced CS is an option in GibbsCAM. An Advanced CS post is needed when coordinate systems are defined in any part. An Advanced CS post has the same capability as a 3-axis post. A 3-axis post is no longer needed if an Advanced CS post is available.

Label Definitions

There are three different letter designations for Advanced CS Posts. Most customers use either a “B” or “C” style post. Both the “B” and “C” style posts fall back to “D” style output if they exceed the maximum number of work fixture offsets available for a particular CNC machine.

B	This post style is useful for multiple setups of the same part, tombstone work and machines without automatic rotation capability. The “B” style post uses a Work Fixture Offset for any machining coordinate system. All of the X-, Y-, Z-, A- and B-axis offsets must be stored in the control's Work Fixture Offsets. The output of the rotary axes will always be zero (A0 and/or B0). The X-, Y-, Z-, A- and B-axis offsets are output in the operation comments. Example: `Fanuc 6M [FW] B001.16.pst`
C	This post style is useful if you have a 4th and/or 5th axis rotary table. The “C” Style post also use Work Fixture Offsets for any machining coordinate system. Only the X-, Y- and Z-axis offsets must be stored in the control's Work Fixture Offsets. The A- and B-axis rotations are output in the G-code. The X-, Y- and Z-axis offsets are output in the operation comments. Example: `Fanuc 6M [PW] C001.16.pst`
D	This post style is useful for 4th and/or 5th axis parts and you do not want to use Work Fixture Offsets. It is also useful if you do not like having to input data into the control's Work Fixture Offsets. The “D” Style post uses one Work Fixture Offset for the entire part. This means that the X-, Y- and Z-axis values in the G-code are offset based on the machining coordinate system. The A- and B-axis rotations are output in the G-code. Example: `Fanuc 6M [NW] D001.16.pst`

N	Any Advanced CS post can be modified into a Long Hand post. Examples:
	`Fanuc 6M [FW] NB299.16.pst`
	`Fanuc 6M [PW] NC299.16.pst`
	`Fanuc 6M [NW] ND299.16.pst`

Code Issues

- Advanced CS vs. Simple Positioning and/or Polar & Cylindrical Milling
  - An Advanced CS post is incompatible with a Simple Positioning post or a post that supports Polar & Cylindrical Milling. If you use coordinate systems to specify rotations, you need to use an Advanced CS post.
- Master Clearance Plane
  - The value entered into the Z clearance plane in the Document Control dialog is a fixed point in space. This position or location is not relative to the current coordinate system. In other words, this value is always local to the home coordinate system.
  - This value is output at the beginning of each new tool operation and at the beginning of a same tool operation if there is a new coordinate system specified.
  - If this value is not entered correctly, it is very possible that the system will produce unexpected negative Z rapid moves. Therefore, It is essential to make sure this value is clear of all machining coordinate system rotations.
- Rotate to Shortest Distance
  - The system calculates the shortest distance to rotate from one coordinate system to another. For example, the system will output a positive move in the clockwise direction to get from 270° to 0° degrees. The system will output a negative move in the counterclockwise direction to get from 90° to 0°. The system will output either a clockwise or a counterclockwise move to get from 180° to 0°.

4-Axis Simple Positioning

Rotation information entered in the `Rotate` tab for the process is output in a *Simple Positioning* post. A Simple Positioning post uses **either** the A-axis **or** the B-axis to rotate the part into position. A Simple Positioning post has the same capabilities as a 3-axis post. A 3-axis post is no longer needed if a Simple Positioning post is available.

Label Definitions

P	This designates a 4th axis positioning post. A Simple Positioning post will output an A-axis move in the G-code. No Work Fixture Offsets will be used in the rotation of the part. Example:

`Fanuc 6M [VG] PM001.19.pst`

`Y` This designates a Simple Positioning post which will output a B-axis move in the G-code. No Work Fixture Offsets will be used in the rotation of the part. Example:

`Fanuc 6M [VG] YPM001.19.pst`

Any Simple Positioning post can be modified into a Long Hand post. Examples:

`N` `Fanuc 6M [VG] NPM299.19.pst`

`Fanuc 6M [VG] NYPM299.19.pst`

Code Issues

- Simple Positioning vs. Advanced CS
  - A Simple Positioning post is incompatible with an Advanced CS post. If you use coordinate systems to specify rotations, you need to use an Advanced CS post.
- Origin of Rotation
  - In Simple Positioning, the origin of rotation of the X-, Y- and Z-axes must be `0`.

Posts That Support Rotary and Cylindrical Milling

If you program Wrapped Geometry, or choose the Polar & Cylindrical Milling option button in the Rotate tab, you will need a post that supports Polar & Cylindrical Milling. A post of this sort uses either the A OR B-axis to rotate and machine the part simultaneously, and otherwise has the same capabilities as a 3-axis post or a Simple Positioning post. You do not need a 3-axis postor Simple Positioning Post if a post that supports Polar & Cylindrical Milling is available.

Label Definitions

`R` This designates a 4th axis post that supports Polar & Cylindrical Milling. The post will output an A-axis move in the G-code. Cutting of wrapped arcs will be broken into linear segments. No Work Fixture Offsets will be used in the rotation of the part. Example:

`Fanuc 6M [VG] RM001.19.pst`

`Y` This designates a 4th axis post that supports Polar & Cylindrical Milling. The post will output a B-axis move in the G-code. Cutting of wrapped arcs will be broken into linear segments. No Work Fixture Offsets will be used in the rotation of the part. Example:

`Fanuc 6M [VG] YRM001.19.pst`

`I` This designates a post that supports supports Cylindrical Interpolation. The post will output a G2 or G3 with rotary moves. Examples:

`Fanuc 6M [VG] IRM001.19.pst`

`Fanuc 6M [VG] YIRM001.19.pst`

Any post that supports Polar & Cylindrical Milling can be modified into a Long Hand post. Examples:

N
```
Fanuc 6M [VG] NRM299.19.pst

Fanuc 6M [VG] NYRM299.19.pst

Fanuc 6M [VG] NYIRM299.19.pst
```

Code Issues

- Polar & Cylindrical Milling vs. Advanced CS
  - A post that supports Polar & Cylindrical Milling is incompatible with an Advanced CS post. If you use coordinate systems to specify rotations, you need to use an Advanced CS post.
- Origin of Rotation
  - In Polar & Cylindrical Milling, the origin of rotation of the X-, Y- and Z-axes must be 0.
- Rotary Feedrates
  - 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.
  - Certain CNCs, such as Haas and Mazak, calculate rotary feedrates using Inverse Time. Any post that supports Polar & Cylindrical Milling can be modified to use Inverse Time for feedrates.

Communications

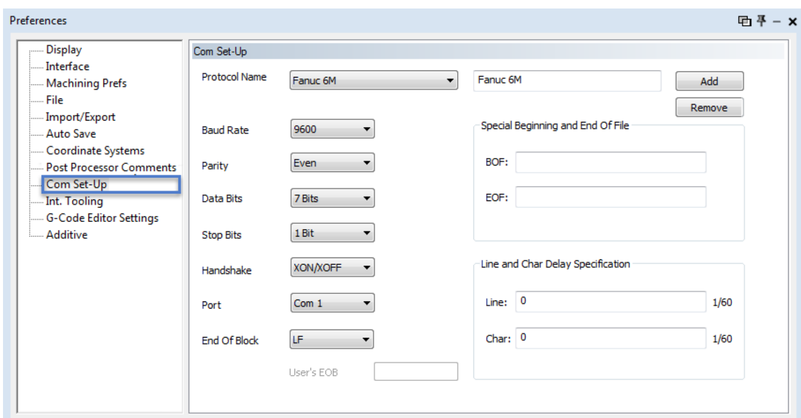
The system contains integrated communications. Third party communications packages can also be used to communicate with CNC machines. Before data can be sent to the CNC machine, the communication parameters need to be set up. To access the Com Set-Up tab choose File > Preferences. This dialog is used 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 refer to the section in 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.



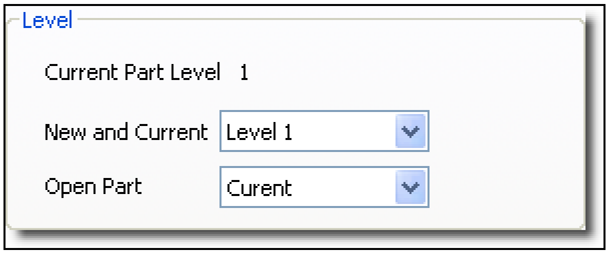
Changing

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.

Appendix



The Interface preference contains an option for two interface levels, Level 1 and Level 2. Level 2 is the default and provides a more complete, feature-rich environment. Level 1 is a simpler interface that some users may prefer if they do not need all the options or flexibility that Level 2 offers. You may think of Level 1 as a training interface that hides the more complicated features. This section details the different interface options found in Level 1.

Not Included In Interface Level 1

Level 1 is exclusive to a 3-axis milling MDD. All other milling MDDs require Level 2. For most operations that require a simple milling process and geometry, the Level 1 interface is the best way to learn the most important basics of milling. There are however several things that cannot be done in the Level 1 interface.

- Any surfaces or solids manipulation as described in any of the solids manuals which includes
  - Global Tolerance settings
  - Surface Machining

  Solids will not be visible or selectable until switching to Level 2.
- Rotated Coordinates as used in Advanced CS, Mill/Turn or Multi-Task Machining

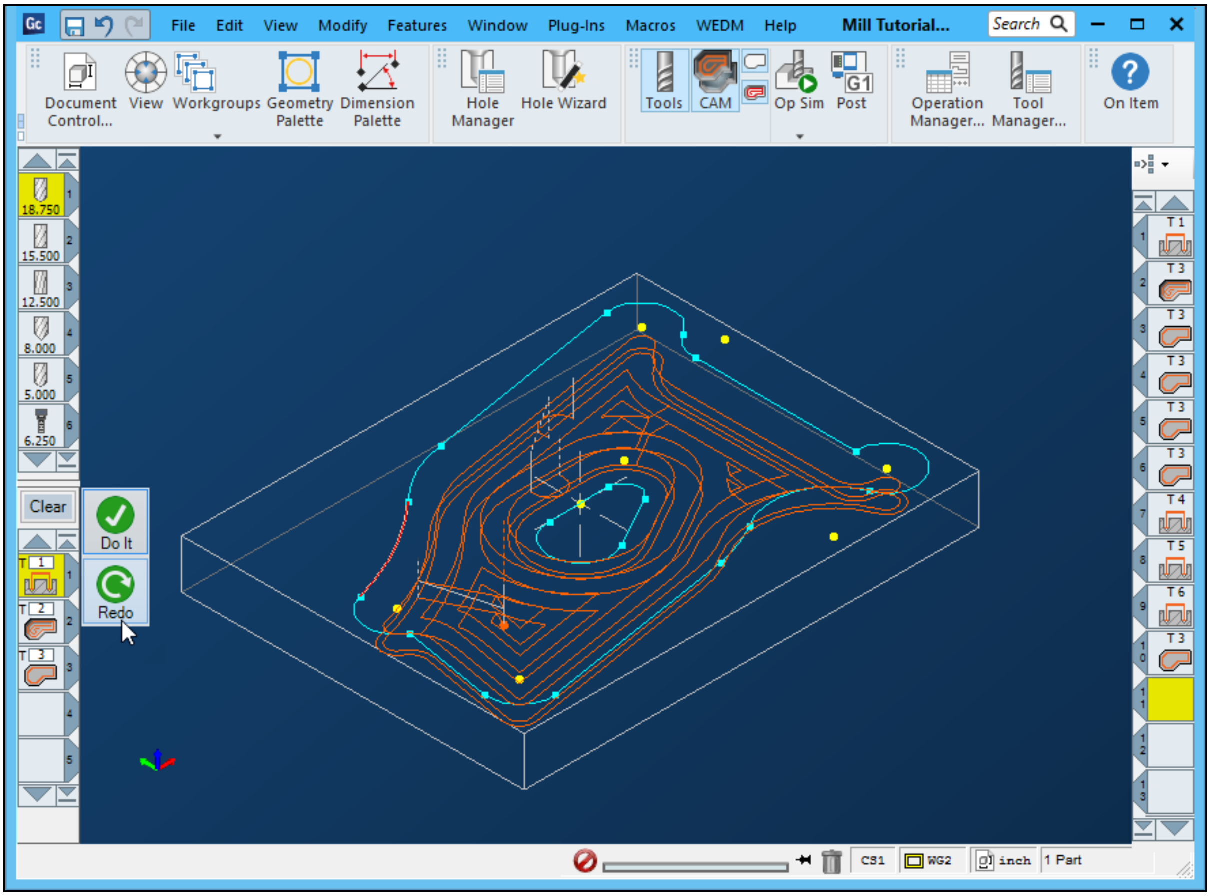
  All Coordinate system options are hidden in Level 1, including the grid, lists and palette.
- Advanced Contour and Roughing options
  - Stay In Stock
  - Material Only
  - Advanced Entry And Exit
  - Hit Flats
  - Open Sides - limited to fixed parameters based on tool size
- Access to some workspace context menus is disabled

Workgroups

To access different workgroups in Level 1, the Workgroup list and information dialogs are located within the geometry palette in addition to on the command palette.

Interface

The interface is different in Level 1. The Floating Toolbar is not present, the Commands Palette is simpler, and the Process selection palette has fewer options.



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

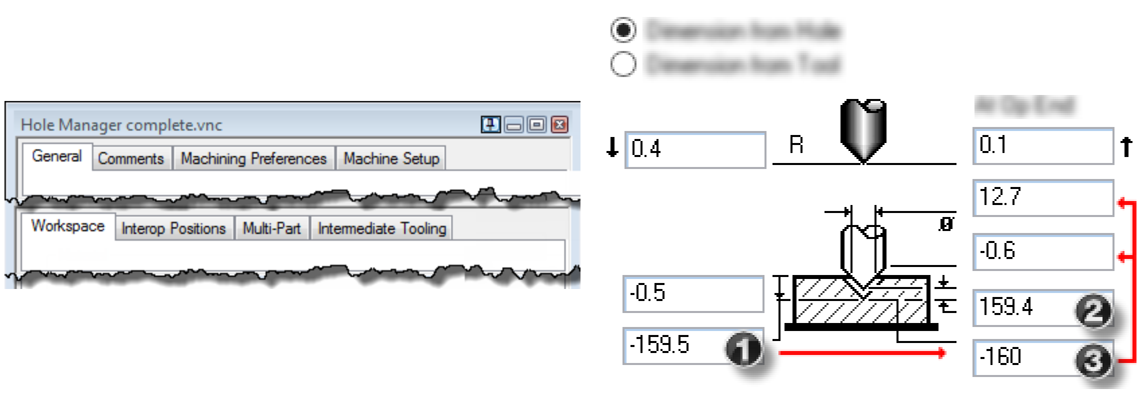
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

`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.

Faint green borders that outline areas within a graphic usually signify an image map. In online help or a PDF viewer, you can click a green-bordered area to follow the link.

Links to Online Resources

Link	URL	Action / Description
Go	http://www.GibbsCAM.com	Opens the main website for GibbsCAM.
Go	https://online.gibbscam.com	Opens a restricted website containing materials available for download. Requires a GibbsCAM Online Services account; to set up an account, contact GibbsCAM Support.
Go	https://store.GibbsCAM.com	Opens the website for the GibbsCAM Student Store.
Go	https://macros.gibbscam.com	Opens a wiki containing documentation and examples of GibbsCAM macros. Requires a GibbsCAM account.
Go	http://kb01.GibbsCAM.com	Opens a Knowledge Base article, **Contour Operations Using Thread Mill Tools**, that explains in detail the correct way to program Contour processes using Thread Mill tools.
Go	mailto:Support@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Technical Support department for GibbsCAM.
Go	mailto:Registration@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Registration department for GibbsCAM.
Go	mailto:Sales@gibbscam.com	Runs your email client to create a new message addressed to the CAMBRIO Sales department for GibbsCAM.
Go	http://www.autodesk.com/inventor	Opens an external website that provides more information on Autodesk Inventor products.
Go	http://www.celeritive.com	Opens an external website that provides more information on VoluMill Ultra High-Performance Toolpath (UHPT) from Celeritive Technologies.
Go	http://www.predator-software.com	Opens an external website that provides more information on a CNC editor and a virtual CNC viewer from Predator Software, Inc.

Index

O

P

R

S

U

V

W

X

Z