GIBBSCAM 2027 CAM for Production Machining

Version 2027 : September 2026

VoluMill

GIBBSCAM

Contents

Introduction ... 3
Welcome ... 3
Save a Copy – Warning ... 3
Installation ... 4

Using VoluMill ... 4
Quick Start ... 4

Interface ... 10
Predrilling ... 11
Controls on the VoluMill Page ... 11
VoluMill Feeds and Speeds ... 11
Controls for Side-Mill ... 14
Plus-or-Minus Material Amounts ... 17
Repositioning Parameters ... 17
Wireframe Only ... 17
Depths Diagram ... 18
Z Step ... 18
Flats ... 19
Plunge Controls ... 20
Climb or Conventional; Back and Forth ... 22
Minimum Toolpath Radius / Minimum Feedrate ... 22
Coolant / Pattern / Machining CS ... 23
Stock Options ... 24
Controls on the Mill Feature Page ... 25
Controls on the Solids Page ... 26
Controls on the Rotate Page ... 29

Troubleshooting ... 31

OpenSSL Acknowledgment And License ... 32

Conventions ... 36
Text ... 36
Graphics ... 36

Links To Online Resources ... 37

Introduction

Welcome

This *VoluMill* guide describes and explains the use of VoluMill within the GibbsCAM system. This guide can be used to jump right into using the software, or it can be read cover-to-cover.

- "Quick Start" on page 4
- "Interface" on page 10
- "Troubleshooting" on page 31

VoluMill is an optional product module for GibbsCAM. VoluMill is a high-performance toolpath engine developed by Celeritive Technologies. It is designed to be used in place of traditional pocketing methods when it is a priority to reduce cycle times, extend tool life, and reduce the stress on machine tools.

VoluMill is a roughing technology, and it is assumed that a finish pass will follow.

A VoluMill toolpath is designed to never exceed a defined Material Removal Rate during the entire program. It does this by dynamically adjusting the axial depths of cut and feedrates, where necessary, as it drives the tool along a highly efficient path. See http://www.volumill.com for more information on the engine and its toolpath.

Note: The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer, GibbsCAM Operator and GibbsCAM Student Edition provide a subset of the full functionality.

Celeritive Technologies maintains a permanent presence on the World Wide Web:

The Celeritive Technologies website contains company news, product information, e-mail links, user forums and much more. It is the preferred means of connecting to Celeritive Technologies electronically. The Celeritive Technologies website is located at www.celeritive.com or http://www.volumill.com. The site contains useful files available for downloading.

Save a Copy — Warning

For parts with VoluMill or VoluTurn toolpath at this release:

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

Installation

VoluMill is included in GibbsCAM2027, version 2027.0. The most recent version of GibbsCAM can be obtained from your local authorized GibbsCAM Reseller or downloaded from the Gibbs website at online.GibbsCAM.com (requires username and password).

Using VoluMill

Quick Start

This chapter is a quick start to using VoluMill. It is assumed that you are already familiar with how to use processes in GibbsCAM.

When the VoluMill product option is enabled, a VoluMill process tile is available on the CAM palette:

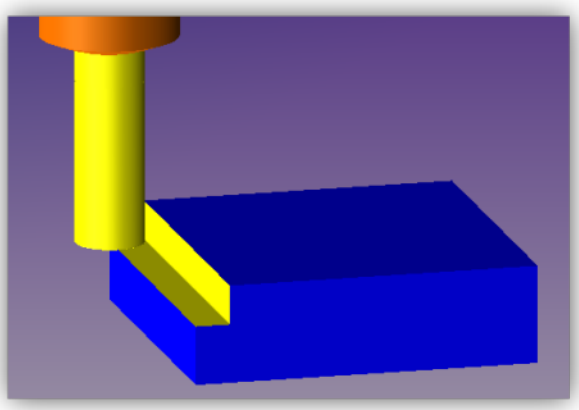
1. Define the appropriate tool for your part. VoluMill supports Type 1 milling tools (Rough/Finish/Ball Endmills).
2. Create a VoluMill process by dragging the tool to the Process List and the VoluMill process tile on top of it.
3. Enter a spindle RPM, Feedrate, desired Z step, and Cut Width for best-case machining conditions.

- How do I know the best-case machining conditions?

  A straight-line cut.

  Traditionally you have programmed the spindle RPM, feedrate, desired Z step, and cut width to account for the worst-case machining conditions, which are when the tool is driven into a corner or moves from one cut to another by slotting at full width.

With VoluMill you do not have to worry about the worst-case machining conditions. Program your spindle RPM, Feedrate, desired Z step, and Cut width for the best machining conditions: a straight-line cut.



Program for a straight-line cut

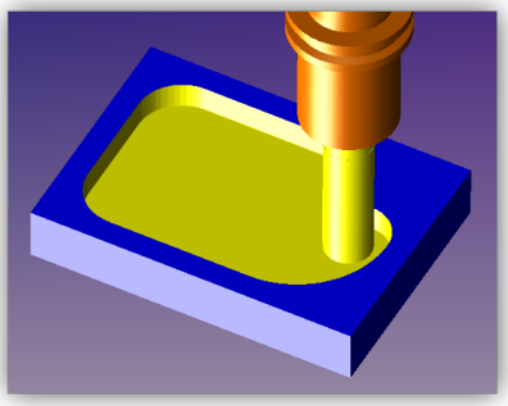
Since VoluMill toolpaths do not contain sharp corners or require full slotting cuts when moving from the end of one cut to the beginning of the next, you can use machining parameters that are suitable to straight-line cuts for the entire toolpath, regardless of the shape of the part.

4. Enter a High Feedrate.

- What is a "High Feedrate"?

The fastest interpolating feedrate (not rapid) that your machine is capable of.

When VoluMill completes a cut, it often must reposition for the next cut by moving the tool through an area that has already been machined. Instead of lifting the tool all the way to the Z Clearance Plane or dragging the tool over an already-machined floor of the part (see Enter a Floor Clearance.), VoluMill keeps the tool down in the machining area. To increase efficiency, this move to the next cut is at the highest feedrate possible.



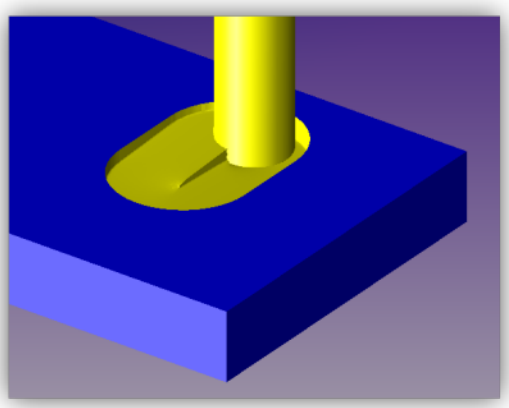
Tool moving to the next cut at the High Feedrate

5. Enter a Ramp Feedrate and Angle.

- I understand what ramping is. What should I know about ramping in VoluMill?

  VoluMill uses the defined Ramp Feedrate and Angle when getting down to the desired depth of cut. It is also important to note that the ramp does not contain any sharp changes in direction.

  You may see VoluMill ramp in other places within the toolpath, but these are done when the tool is not in the cut, and they are at the programmed High Feedrate.



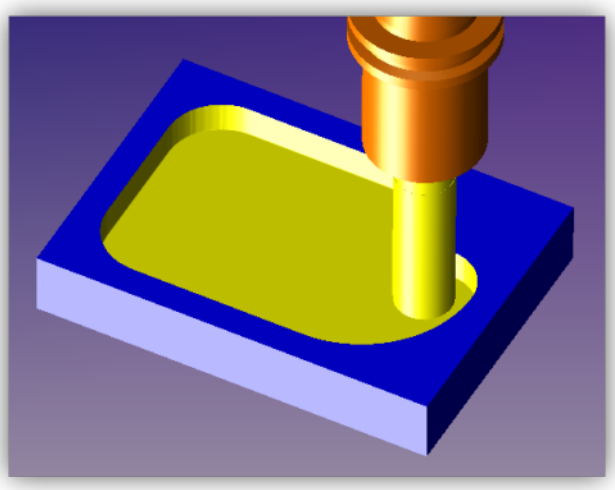
Tool ramping at the Ramp Feedrate and Angle

6. Enter a Floor Clearance.

- What is a "Floor clearance"?

  The amount the tool lifts when repositioning to the next cut.

This parameter is used in combination with the “High Feedrate” described earlier. When VoluMill completes a cut, it often must reposition for the next cut by moving the tool through an area that has already been machined. Instead of lifting the tool all the way to the Clearance Plane, VoluMill keeps the tool down in the machining area. To prevent dragging the tool across the already-machined floor, the tool lifts by the Floor Clearance value along a helix.



Tool moving to the next cut at the Floor Clearance height above the pocket floor

7. Optionally, select zero or more options under Slot/Side Mill Overrides.

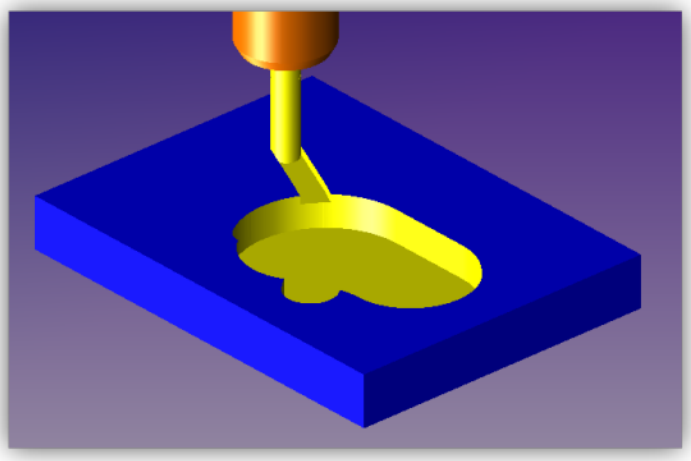
- What are the “Slot/Side Mill Overrides” for?

The Side-Mill Cut Width option enables you to specify a smaller cut width when milling in tighter areas.

The Max. Slot Depth and Slot Feedrate options override the automatically calculated Slot Feedrate and/or Max. Slot Depth values during slotting cuts to maintain the rate of material removal.

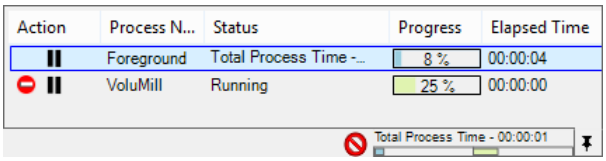
The feedrate, desired Z step, and cut width parameters you specify dictate the rate of material removed in cubic inches/cm per minute. When VoluMill makes a full slotting cut to enter a new area, it will not exceed the rate of material removal dictated by the specified parameters.

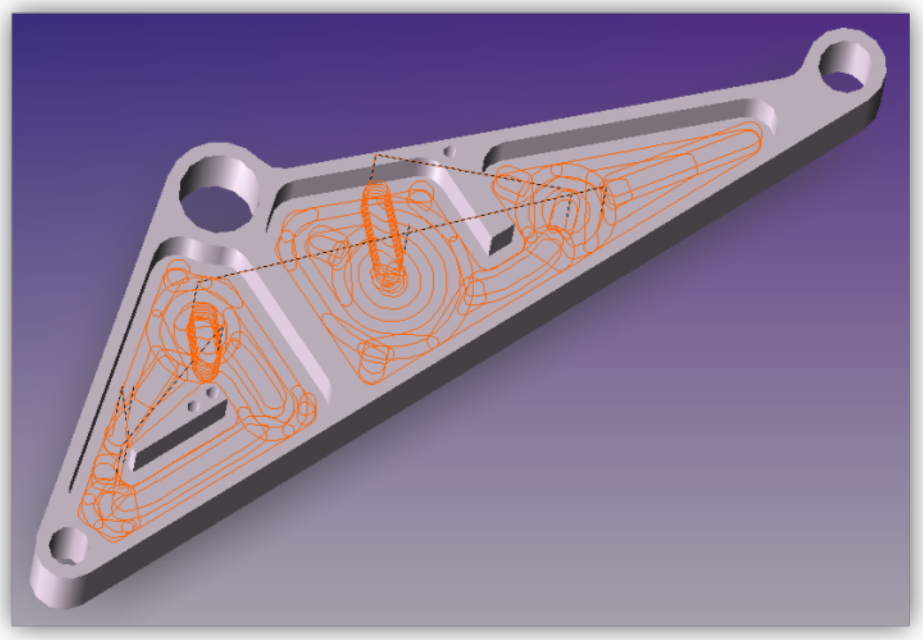
Depending on the specified parameters, VoluMill will reduce the feedrate and/or axial depth of cut to compensate for the increased radial depth of cut and thereby maintain a constant rate of material removal.



Desired Z step and Feedrate automatically modified to not exceed cubic inches/cm of volume removal, moving to the next cut at the Floor Clearance height above the pocket floor

8. Specify the 2D and/or 3D items you want to machine:
   - 2D (geometry). In the workspace, select one or more closed chains of geometry. If multiple chains are selected, the outer chain is used as the pocket and all other shapes are used as islands to be avoided. As with other processes, you can select geometry and/or shapes created by slicing through a solid model using the GibbsCAM Profiler. The VoluMill process also supports the use of Air/Wall features to enable the machining of open-sided pockets.
   - 3D (solids, sheets, or facet bodies). In the workspace, select one or more bodies to be machined. As needed, set further values and options in the "Solids Only" section of the VoluMill process dialog.
9. Press Do It; you will see this VoluMill progress dialog.





That's it! You have created a VoluMill toolpath that should dramatically reduce your roughing cycle times.

Interface

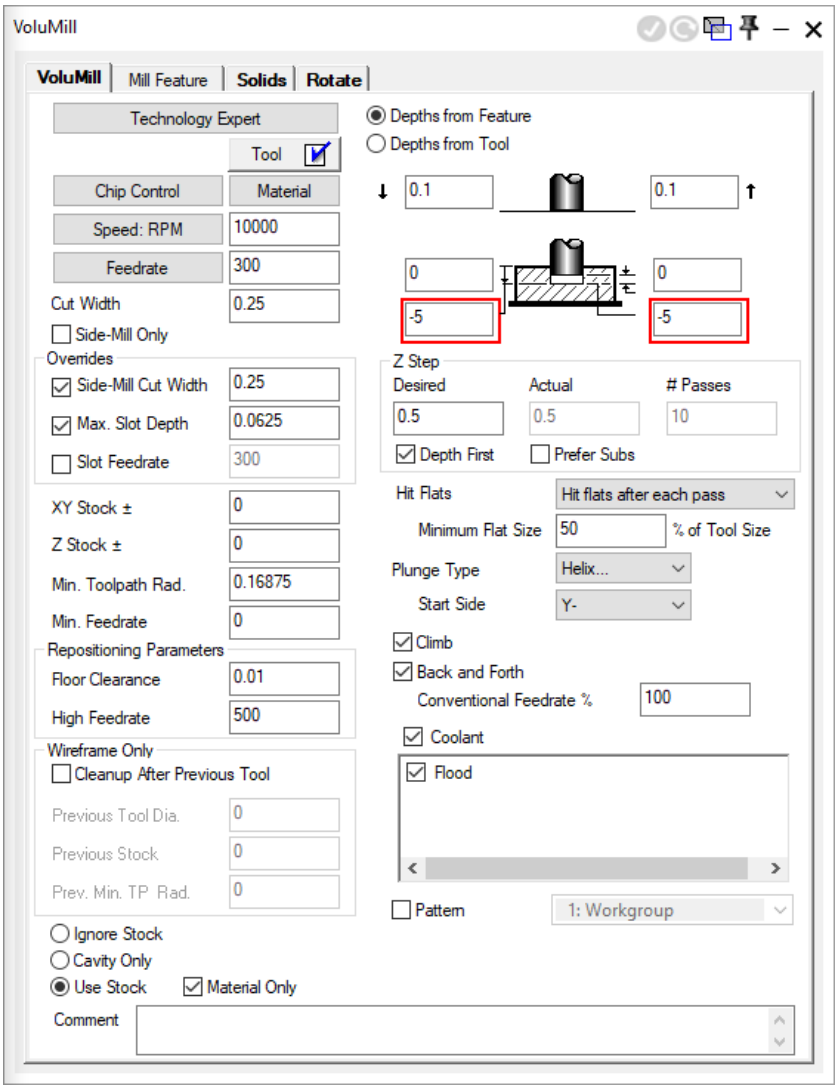
When you create a new VoluMill process or double-click a VoluMill process tile, the system opens a process dialog containing two to four tabs.

VoluMill: See “Controls on the VoluMill Page” on page 11 below.

Mill Feature: See “Controls on the Mill Feature Page” on page 25

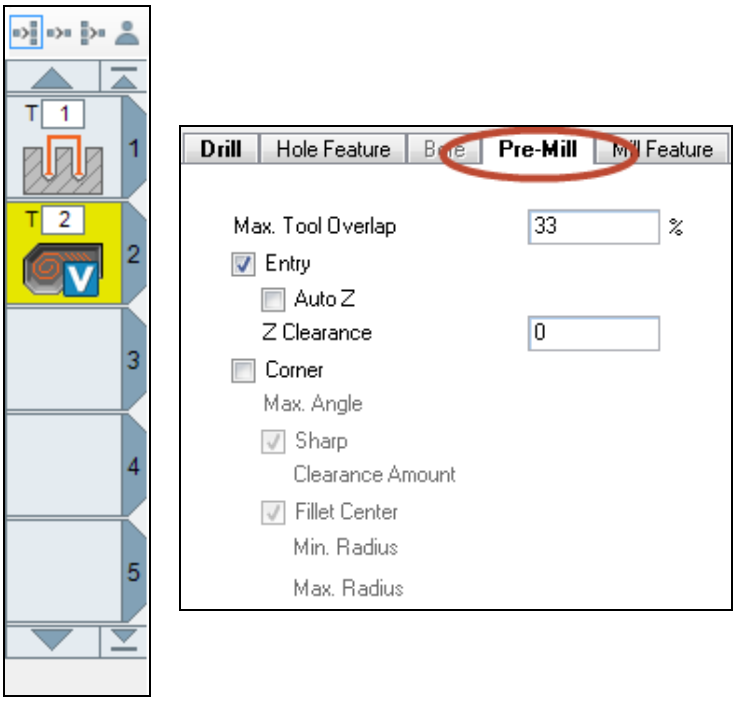
Solids *(available only if you are licensed for **VoluMill Solids**)*: See “Controls on the Solids Page ” on page 26

Rotate *(available only if the current MDD defines a rotary axis)*: See “Controls on the Rotate Page” on page 29



Predrilling

If a Holes process is added before a VoluMill operation, the **Pre-Mill** tab of the Holes process becomes available. This enables predrilling, where VoluMill will determine the location of the start hole(s). For more details, see "Pre-Mill tab" in the *Mill* guide.



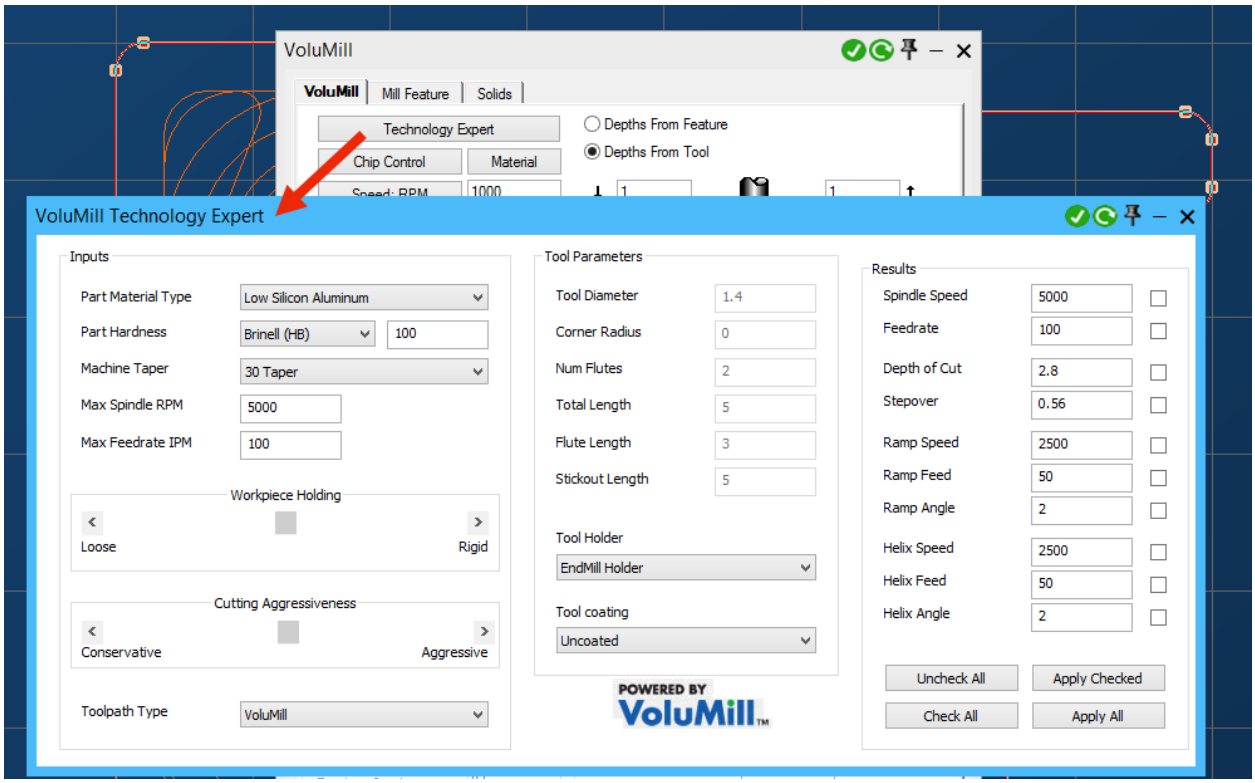
The tool used for the Holes process should be a larger diameter than the tool used for the VoluMill process.

Controls on the VoluMill Page

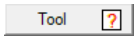
VoluMill Feeds and Speeds

Technology Expert

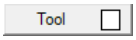
Because VoluMill operations generate a different type of toolpath, values for feeds and speeds and other parameters can be fine-tuned using Technology Expert.

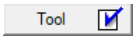


Instead of using the standard Materials dialog to suggest RPM and feedrate, you can calculate recommendations for up to ten parameters — not just spindle speed and feedrate, but also depth of cut, stepover, and speed/feed/angle for ramp or helix plunge types. Recommendations are calculated based on not just material type but also hardness, current tool values, tool coating and holder choice, how well the part is held, the degree of taper. After you choose a machining style (conservative or aggressive) and calculate the recommendations, you can choose to apply some or all of the recommendations to your VoluMill process.



Tool

- 

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

indicates that the tool instance has data attached to it.

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

Material

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

Chip Control

Clicking the Chip Control button displays the Active Chip Thickness Control dialog. Active Chip Thickness Control (ACTC) is a calculator designed specifically for use with VoluMill toolpaths.

It lets you calculate the value of one chip control parameter based on values you supply for the others, and includes the maximum chip thickness (MCT) in the parameter calculations.

Calculate the:

On the dropdown menu, choose which cutting parameter to calculate. The choices are: Speed; Feed; Cut Width; and Chip Thickness. Chip Thickness is the default.

Making a choice here makes some parameters available for you to enter values (which will be used in the calculation) and makes other parameters unavailable (the ones whose values will be calculated).

Speed parameters: RPM and SFM/SMM

RPM is the spindle speed for the toolpath, in revolutions per minute. The default value is inherited from the VoluMill toolpath dialog. A corresponding value for surface speed (SFM for English units, SMM for metric units) is also displayed.

These values mutually calculate: a change in RPM will update the value for SFM/SMM, and vice versa.

Feed parameters: IPM/MMPM and IPT/MMPT

IPM/MMPM is the feedrate for the toolpath. The default value is inherited from the VoluMill toolpath dialog. A corresponding value for feed-per-tooth (IPT for English units, MMPT for metric units) is also displayed.

These values mutually calculate: a change in IPT/MMPT will update the value for FPT/MMPT, and vice versa.

Stepover parameters: Cut Width and % of Tool Dia.

Cut Width is the maximum width of cut for the toolpath (except for plunging motion and in some cases when the Side Mill Only checkbox is unchecked). The default value is inherited from the VoluMill toolpath dialog. A corresponding value expressing the Cut Width as a percentage of tool diameter is also displayed.

These values mutually calculate: a change in Cut Width will update the value for % of Tool Dia., and vice versa.

Chip Thickness

The displayed value is the maximum thickness of the chips that the toolpath will produce, based on the current values for speed (RPM or SFM/SMM), feed (IPM/MMPM or IPT/MMPT), and stepover (Cut Width or % of Tool Dia.), along with the diameter and number of flutes of the current cutting tool. Initially, this value is grayed out because Chip Thickness is the default option from the Calculate the: dropdown list at the top of the dialog.

Calculate

Clicking the Calculate button updates the values for the parameter(s) selected in the Calculate the: dropdown, based on values of the other parameters. Pressing the Tab keys can also trigger the calculation.

Material Removal Rate:

This is an informational value showing the in-cut material removal rate, in cubic inches or cubic centimeters per minute, that will result from the current parameter values.

Apply Parameters

Clicking the Apply Parameters button closes the ACTC dialog and updates the affected parameters in the VoluMill process dialog with the newly calculated values.

Cancel

Clicking the Cancel button closes the ACTC dialog and makes no changes to the parameters in the VoluMill process dialog.

?

Clicking the ? button in the upper right-hand corner opens a new window that provides detailed help for the ACTC functionality, including background information, examples, and recommendations for getting the best results.

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.

Feedrate

This is the feedrate at which the tool will travel while engaged in the material. VoluMill automatically adjusts this feedrate downward in concave arcs in order to maintain a constant rate of material removal. Pushing the button will load a recommended speed from the Material Database based on the part material and tool composition.

The Ramp Feedrate will also be updated to 50% of the calculated feedrate.

The concept of VoluMill is to generate toolpaths that never exceed a given rate of material removal when machining. It does this by dynamically adjusting the feedrates and depths of cut. To achieve reduced cycle times, you should program the Spindle speed and Feedrate for optimal cutting conditions, such as a straight-line cut described above. VoluMill will adjust the feedrate and/or depth of cut to maintain the optimal cutting conditions regardless of the shape of the part.

Cut Width

This input field specifies the width of cut for the toolpath. This is an actual distance, rather than a percentage of the tool diameter. The width of cut will never exceed this value unless you have chosen to allow slot milling (see Max. Slot Depthand "Slot Feedratebelow). With VoluMill, any value that is less than the diameter of the flat portion of the tool can be used without fear of leaving uncut stands of material behind.

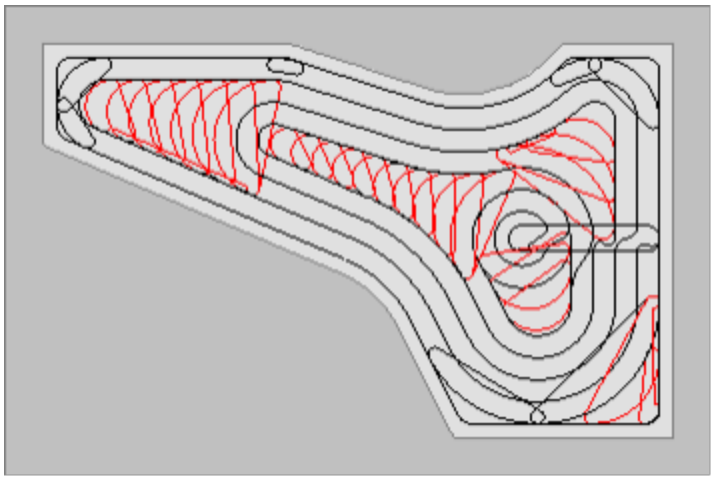
Controls for Side-Mill

Side-Mill Only

VoluMill achieves its superior cutting performance by striving to maintain a constant rate of material removal throughout the toolpath, regardless of the shape of the geometry.

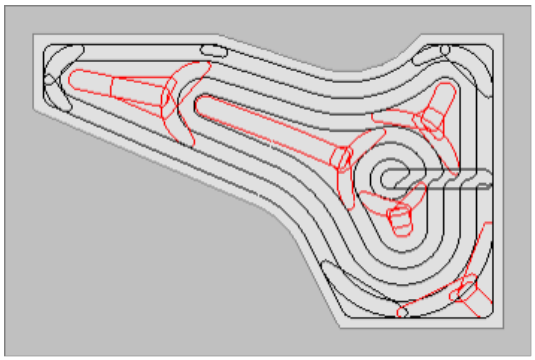
Two strategies are considered when milling in confined areas: Side Milling and Slot Milling. If this checkbox is unchecked, VoluMill will automatically choose the strategy that produces the lowest cycle time using the current feedrate and distance traveled. Depending on the shape of the part, both methods may be used.

In some cases VoluMill will use a progressive milling method wherein the tool is never fully engaged in the material; this is referred to as Side Milling.



Side Milling

In some cases VoluMill will fully engage the cutting tool; this is referred to as Slot Milling. During the slotting cuts, VoluMill will reduce the feedrate and/or use multiple depths-of-cut to maintain the same rate of material removal that is in effect during the rest of the toolpath.



Slot Milling

To force the system to only use the Side Milling strategy, which is especially beneficial in harder materials, check this checkbox.

Overrides

Up to two or four overrides can be specified, depending on which strategy is in effect (Side Milling or Slot Milling). Options are as follows.

Side-Mill Cut Width

When the toolpath Side Mills, it never exceeds the defined Cut Width value. To use a smaller width of cut when side milling in tight areas, check this checkbox and enter the desired distance. The entered value must be less than or equal to the Cut Width value.

Max. Slot Depth

This parameter determines the number and depth of any slotting cuts. VoluMill uses the entered value as a not-to-exceed value, meaning that the slotting depth-of-cut may be adjusted downward from the entered value as needed to ensure that each such cut removes the same amount of material. Since these slotting cuts are invariably at a greater effective stepover (radial depth-of-cut) than the non-slotting cuts of the toolpath, it is necessary to reduce the axial depth-of-cut, which this parameter controls, and/or reduce the slotting feedrate (see the "Slot Feedrate"parameter below) to maintain the material removal rate during these cuts.

If this checkbox is not checked, VoluMill will automatically set this parameter, in conjunction with the Slot Feedrate parameter feedrate to maintain the same material removal rate while slotting that is in effect for the rest of the toolpath. The value that VoluMill calculates will be shown in gray in the input field.

This override is not available if the Side-Mill Only checkbox is checked.

Slot Feedrate

This is the feedrate used for the slotting cuts. Since these slotting cuts are invariably at a greater effective stepover (radial depth-of-cut) than the non-slotting cuts of the toolpath, it is necessary to reduce the slotting feedrate, which this parameter controls, and/or reduce the axial depth-of-cut (see the "Max. Slot Depthparameter above) to maintain the material removal rate during these cuts.

If this checkbox is not checked, VoluMill will automatically set this parameter, in conjunction with the Max slot depth parameter to maintain the same material removal rate while slotting that is in effect for the rest of the toolpath. The value that VoluMill calculates will be shown in gray in the input field.

This override is not available if the Side-Mill Only checkbox is checked.

Plunge Spindle RPM

This is the spindle speed used for the Plunge motion.

If this checkbox is not checked, VoluMill will use the Spindle speed. In softer materials this may be acceptable. However, for harder materials it is recommended that the Plunge spindle speed be reduced to sync with the Ramp Feedrate.

Note: It is important that the post processor be configured to support spindle speed changes not associated with tool changes.

Dwell After Plunge

When Plunge Spindle RPM is in effect, you can also specify a value for Dwell After Plunge to force a dwell after the tool has completed the plunge motion. Some machines will not pause to begin feeding until the programmed spindle speed is reached. For these machines the dwell will give the machine time to do so.

Note: It is important that the post processor be configured to support dwell output with spindle speed changes. The value entered is not interpreted by VoluMill as either a length of time in milliseconds or revolutions. It is up to the post processor to interpret and output the value accordingly.

Plus-or-Minus Material Amounts

Pocket Stock ±
This is the amount of material that the toolpath will leave on the walls of the pocket.

Z Stock ±
This is the amount of material that the toolpath will leave on the floor of the pocket.

Repositioning Parameters

Plunge Clearance
This parameter establishes a distance above the top of the part, in Z, at which the tool stops moving at RAPID and begins to move at the feedrate.

Floor Clearance
The value entered here establishes the Z-component of a helical move that is used when entering or exiting a cut. Only non-negative values are allowed. If a positive value is entered, repositioning moves between cuts will take place above the already-machined floor. If zero is entered, the tool will drag across the already-machined floor during these moves. In this case, set the High Feedrate parameter to be no greater than the cutting feedrate to help ensure more consistent tool marks on the floor.

High Feedrate
This is the feedrate at which repositioning moves within the toolpath occur, other than those that take place at Rapid above the part. Since VoluMill drives the tool in a manner that controls the material removal rate, it is typically necessary to reposition the tool from the end of one cut to the beginning of the next by moving it through an already-machined area. Setting this value higher than the programmed cutting feedrate helps minimize cycle time.

It is recommended that this value be set to the fastest non-Rapid feedrate at which the machine can interpolate linear and circular moves.

Wireframe Only

The controls in the Wireframe Only section apply to 2D machining only.

Cleanup After Previous Tool
This checkbox activates a set of parameters for generating cleanup milling toolpaths (sometimes called rest, reference, or material-only milling). These toolpaths are used to remove material left behind where a larger tool could not fit.

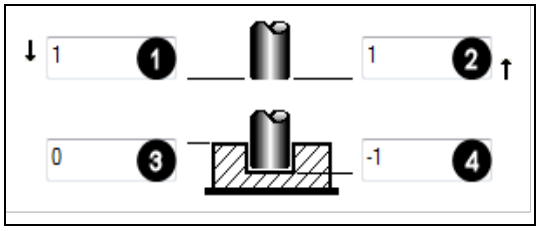
Previous Tool Dia
The diameter of a larger tool used on a previous operation.

Previous XY Stock
The amount of stock that the previous operation left on the boundary. Usually this value will be the same as the XY stock to leave for the current operation.

Prev Min Toolpath Radius
The Min Toolpath Radius used on a previous operation.

Depths Diagram

The items in this section of the dialog define the clearances and depths for the toolpath.



1. Entry Clearance Plane
Entry Clearance Plane specifies the location the tool will make a rapid move to before feeding to the start point of the toolpath.

2. Exit Clearance Plane
The Exit Clearance Plane specifies the location the tool may rapid to after completing the toolpath.

3. Surface Z
The Surface Z specifies the top level of the material.

4. Floor Z
The Floor Z specifies the final depth of the toolpath.

Z Step

Desired / Actual / # Passes
This is the depth of cut per slice of the toolpath. If the entered value is greater than or equal to the total depth, the pocket will be machined to depth in a single pass. If the entered value is less than the total depth, the VoluMill toolpath will consist of multiple passes in Z. (The number of passes appears in the # Passes field.) The value that will be used is adjusted downward to ensure that each pass removes the same amount of material. This adjusted value is displayed in the Actual field.

Depth First
This option specifies how multiple pockets with more than one Z Step are to be machined. Checking Depth First will machine each pocket to the final Z Depth and then move onto the next one. If Depth First is unchecked, all selected pockets are machined at each Z Step. The process is then repeated until all the pockets are machined.

Prefer Subs
This checkbox lets you use subprograms in the posted code. Activating this item produces shorter G-code output.

The lower right half of the **VoluMill** tab presents these categories of controls:

- **Flats**: See Flats, below
- **Plunge**: See “Plunge Controls” on page 20
- **Climb**: See “Climb or Conventional; Back and Forth” on page 22

- **Coolant, Pattern, and CS**: See “Coolant / Pattern / Machining CS” on page 23

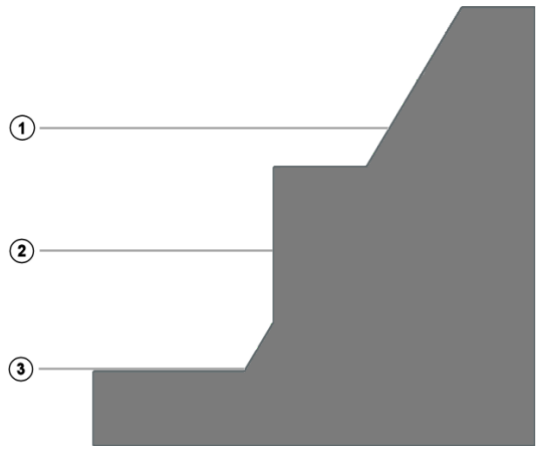
Flats

Hit Flats

This menu offers different strategies for machining flat surfaces, as follows.

Do not hit flats

Flat surfaces will only be machined if the Z Step, +/- any stock, is coincident with the flat surface.

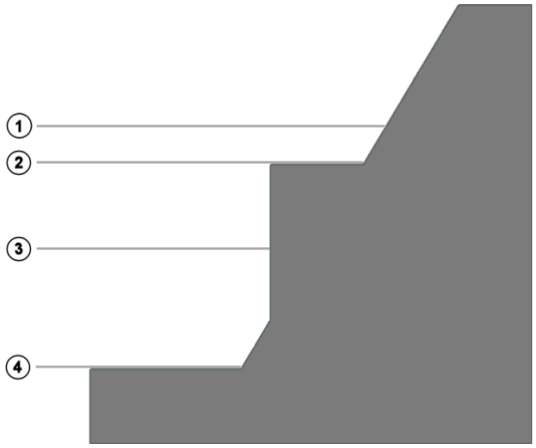


Hit flats by adding passes

After performing a regular depth cut (Z Step), the tool will add one or more Z Steps 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.



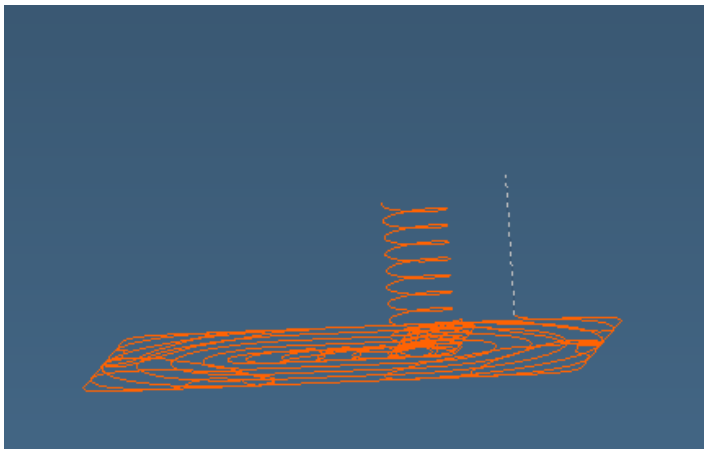
Plunge Controls

Plunge Type

This menu offers three types of entry motion that VoluMill can use to machine to the desired depth of cut. All types let you specify a Start Side: X+, X–, Y+, Y–, or Custom (with a starting value).

Helix

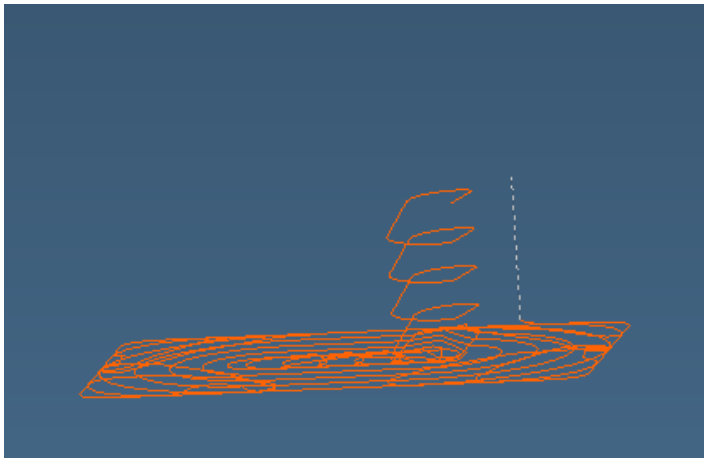
This plunge type uses a helical entry to machine to the desired depth of cut. This is the default plunge type and is recommended for harder materials.



Helical motion to the desired depth of cut

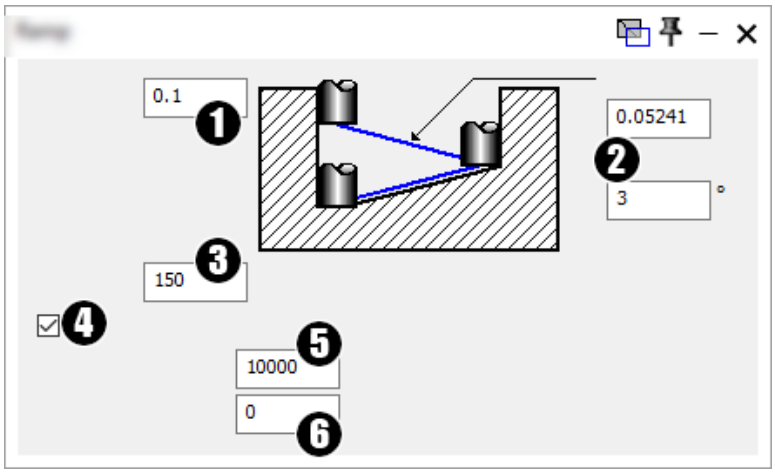
Ramp

This plunge type uses a special ramping motion to machine to the desired depth of cut. VoluMill calculates the optimal position and shape of the ramp to create a transition area. This transition area is then used to connect from the end of one cut to the next while disengaged from the material at the High feedrate. This plunge type is recommended for softer materials.



Ramping motion to the desired depth of cut

If you choose Helix or Ramp, a dialog box opens that lets you fine-tune plunge parameters:



1. Clearance
2. Slope / Angle
3. Feedrate
4. Override Speed:
5. Spindle RPM
6. Dwell After Plunge

Clearance

This is the clearance for beginning the helix or ramp plunge.

Slope / Angle

This pair of parameters (a change to one causes the other to change) establishes the rate of descent at which the tool enters the material from the top, as is required when machining completely enclosed areas (pockets). VoluMill uses the entered value as a not-to-exceed value, meaning that the actual ramp angle may be adjusted downward from the entered value as needed to fill the ramping area. VoluMill automatically calculates the location, length, and orientation of the ramp based on the shape of the selected geometry. The feedrate for the ramp entry is determined by the Feedrate parameter.

Feedrate

This is the feedrate used for the initial ramp entry into the material.

Spindle RPM

This the spindle speed, in revolutions per minute.

Dwell After Plunge

This is the amount of time to wait before retracting from the plunge.

Climb or Conventional; Back and Forth

Climb

Sets the preferred direction. When this checkbox is selected, the tool will start and end its motion in a direction that is climb milling; when the checkbox is cleared, it will start and end in the conventional milling direction.

Back and Forth

This checkbox indicates whether the tool is permitted to use both climb and conventional direction of motion. The system will attempt to minimize cutting time and travel distance, according to these criteria:

- The first and last cut will use the preferred direction.
- Short distances might use a single direction, because each change in direction requires some overhead time to lead out, to change direction, and to lead in. This applies especially when the feedrate for the nonpreferred direction is slower than for the preferred direction.
- The system will look ahead for maximum efficiency. For example, if the nonpreferred direction is most efficient on cut #3, cuts 1 and 2 will both be in the preferred direction.

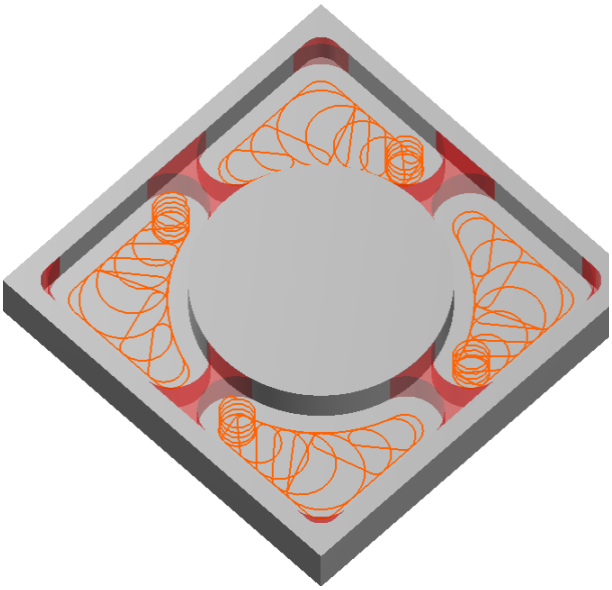
Conventional Feedrate %

When Back and Forth has been chosen, you can specify how much to reduce [or, rarely, to increase] the feedrate when the tool motion direction uses conventional milling direction.

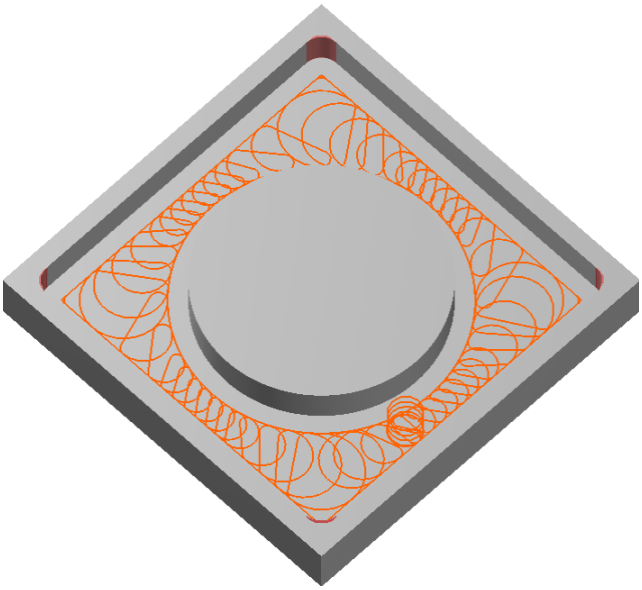
Minimum Toolpath Radius / Minimum Feedrate

Min. Toolpath Radius

This is the minimum radius the tool will traverse when cutting. To machine into sharp corners or tight areas, VoluMill has to make small moves that may be more effectively accomplished using a smaller tool in a clean-up operation. The optimal and default value for Min Toolpath Radius is 45% of the tool diameter. At this value, VoluMill can reach velocities that can dramatically reduce cycle times. However, it may leave areas uncut.



A large Min Toolpath Radius may leave uncut material



A small Min Toolpath Radius may enable the tool to machine tight areas

Using a smaller value may enable the tool to machine more material. This, however, may be less efficient than switching to a smaller tool and using another VoluMill toolpath to machine the remaining material.

The smallest value allowed is 5% of the tool diameter. VoluMill is designed to never make sharp directional changes while in the cut. This means that if your part has a fillet in the corner that is equal to the tool radius, a small amount of material will be left in the corners, even if the Min Toolpath Radius value is set to the minimum allowed.

VoluMill is a roughing technology, and it is assumed that a finish pass will follow.

Min. Feedrate

A feedrate that is too small can damage the tool or the part. To prevent this, you can specify a minimum feedrate.

Coolant / Pattern / Machining CS

Coolant

This 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.

Use Pattern

When Use Pattern is on, 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.

Machining CS

This menu allows you to specify which coordinate system the operation will be created from. By default, the XY plane is selected. All coordinate systems that have been created will be available.

Stock Options

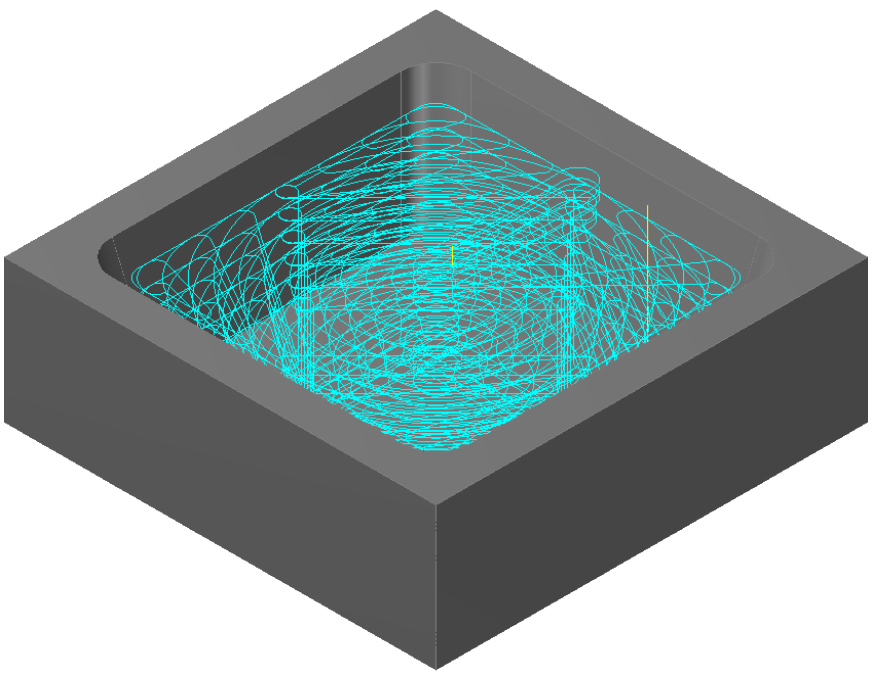
At the bottom left of the VoluMill page is an array of three option button: Ignore Stock, Cavity Only, and Use Stock. Each of these choices is explained below.

Ignore Stock

When Ignore Stock is selected, the stock definition is ignored. Roughing a solid will machine all selected faces. In this case, a pocket can be machined by simply selecting the floor (if the pocket's floor is flat).

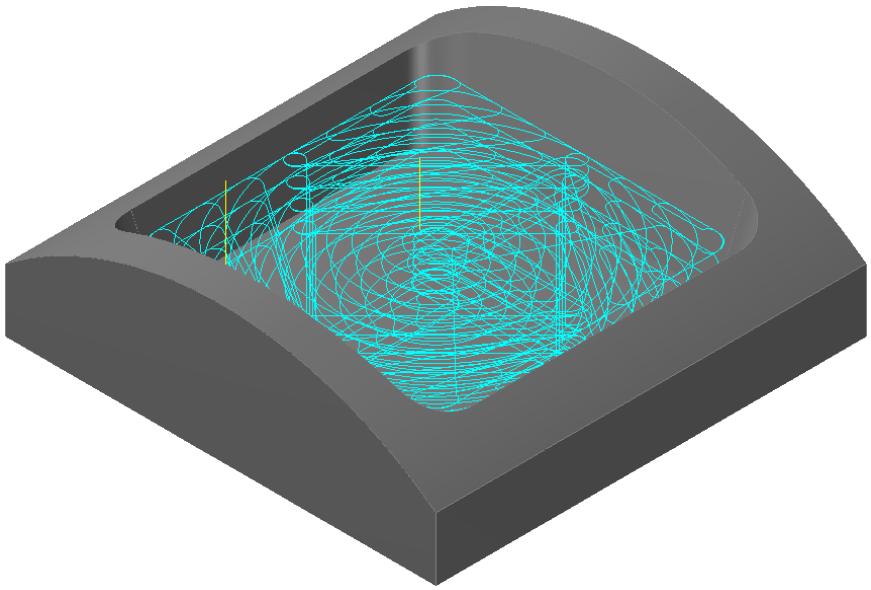
Cavity Only

This option should be used when machining a cavity from material having a flat top surface. The benefit of using this option is that stock need not be defined separately.



Cavity with flat surface

If the Cavity Only machining option is used on a core or a shape that does not have a flat top surface, the toolpath will begin machining at a level where it can create a closed area. This can create a situation where the first depth of cut is too deep. To avoid this, use a stock definition appropriate for the part shape.



Cavity with curved surface

Use Stock

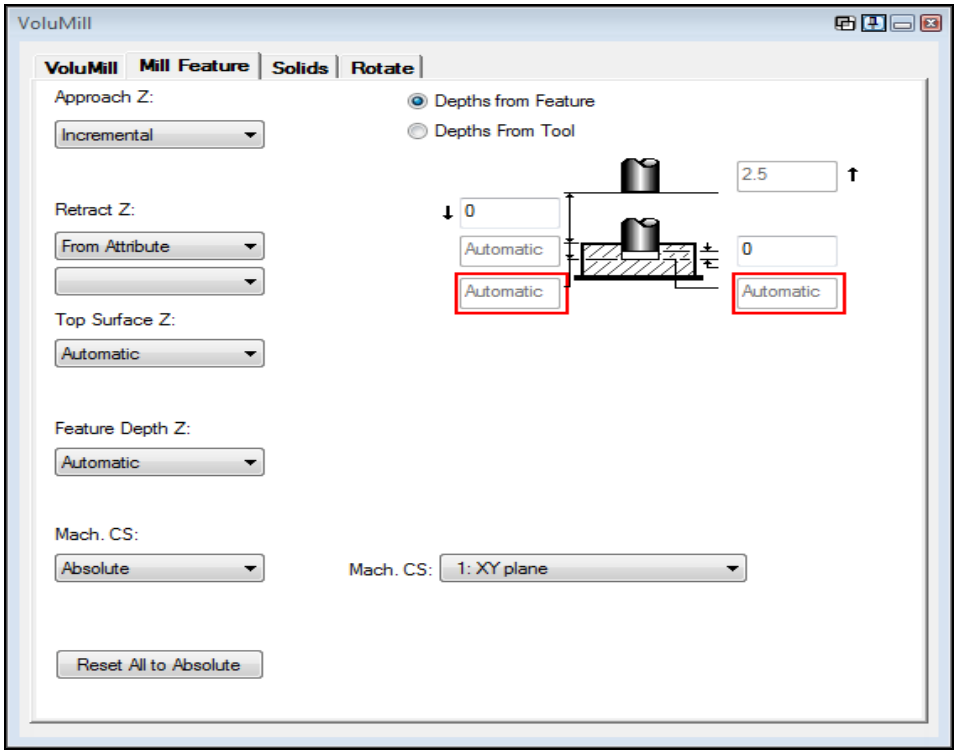
When Use Stock is selected, toolpath will be confined to the current stock definition even if the part extends past the stock.

Material Only

Material Only calculates toolpath for all remaining material left by prior operations only. Remaining material is stored for 2D operations including contouring, roughing and drilling. Remaining material is **not** stored for 3D operations including Lace, Surface Flow, and 2 Curve Flow cuts. Material Only supports custom stock definitions, sharp/bullnose/tapered/ball endmills and most form tools. Undercutting tools are not supported. Material Only can be used as a single operation or as part of a multiple process group.

Controls on the Mill Feature Page

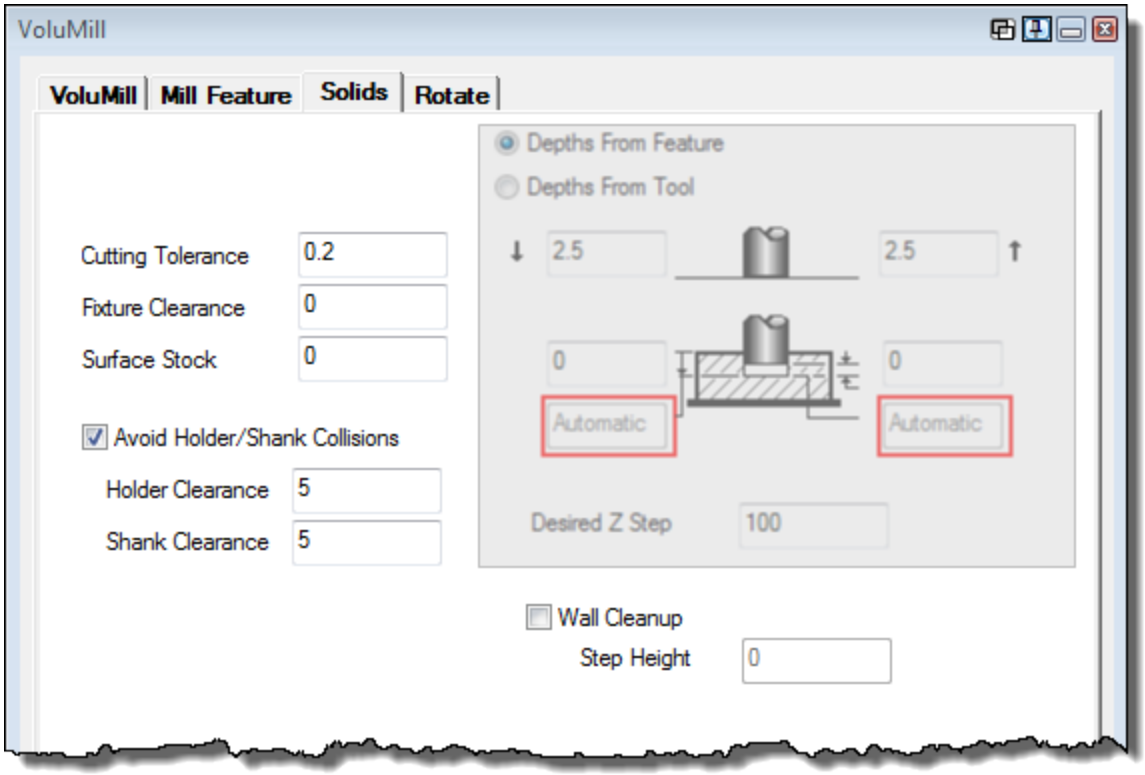
The **Mill Feature** tab provides access to items in the Depths Diagram that can be controlled through attribute values.



For more information see the *Mill* guide.

Controls on the Solids Page

The **Solids** tab, found in process dialogs if you are licensed for VoluMill Solids, provides controls for machining solid bodies.



Depths From Feature / Depths From Tool / Desired Z Step

See “Depths Diagram” on page 18.

Cutting Tolerance

This is the tolerance of the toolpath, also referred to as the chordal deviation. This is the accuracy of the toolpath to the surface to be cut. Small values create more accurate toolpath but can take longer to calculate. Larger values can be used where a rough pass that leaves stock is being performed.

Fixture Clearance

This value is the additional distance the toolpath will be offset from the fixture.

Surface Stock

The Surface Stock value specifies the amount of material that will be left by the toolpath on any sheet or solid machined by the process. The toolpath will be offset by the Surface Stock amount in X, Y and Z. Contrast this with the Stock ± amount, which adds stock only in the cutting plane (machining CS X,Y).

Avoid Holder/Shank Collisions

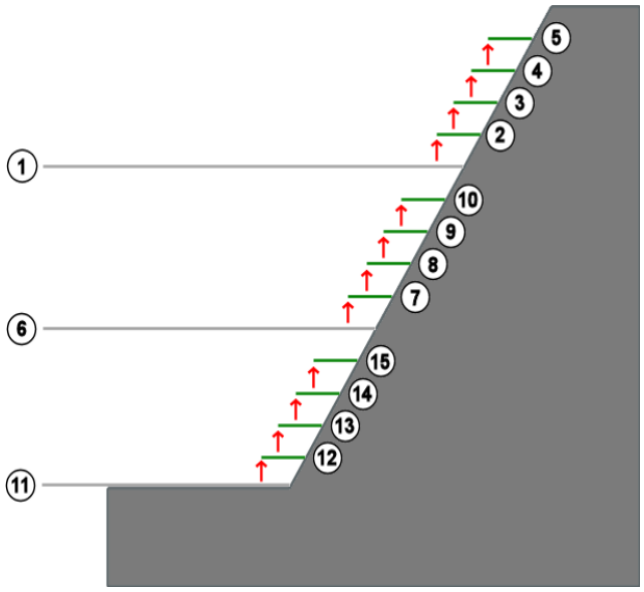
When this checkbox is cleared, then checking is not done for near-collisions with the tool holder or tool shank, thus reducing the processing time for calculating toolpath.

When this checkbox is selected, you can specify values for Holder Clearance and Shank Clearance. The toolpath must clear the tool holder, tool shank, or both.

Wall Cleanup / Step Height

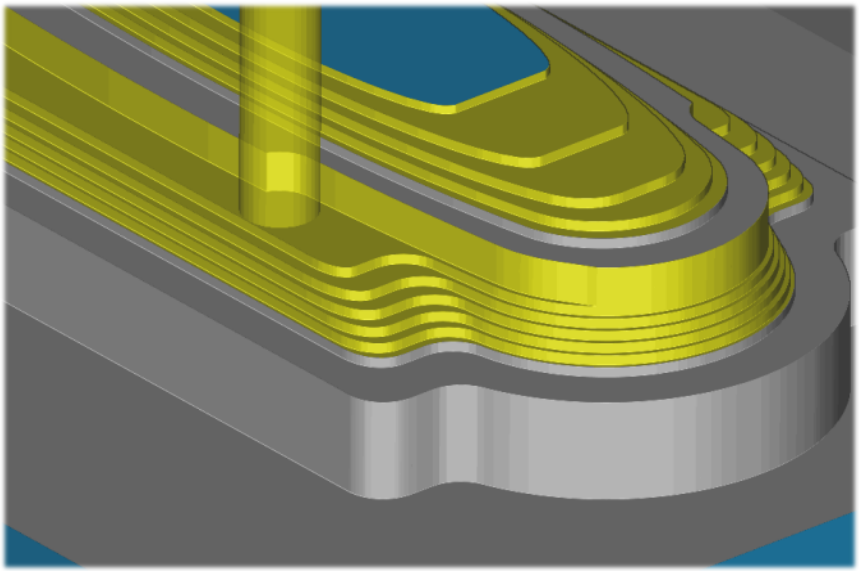
Use this option to control the height of the steps that will remain. If the checkbox is cleared, the tool will use a shallow depth of cut across the entire part in order to leave smaller steps. If

the checkbox is selected, the tool will first take a large step, using up to the full flute-length, and then automatically work its way back up to leave smaller steps using the value specified for Step Height. Using this option, the tool can remove the bulk of material most efficiently and still leave smaller steps for a semi-finish or finish toolpath.



When Wall Cleanup is selected, large primary depth cuts are refined by smaller steps

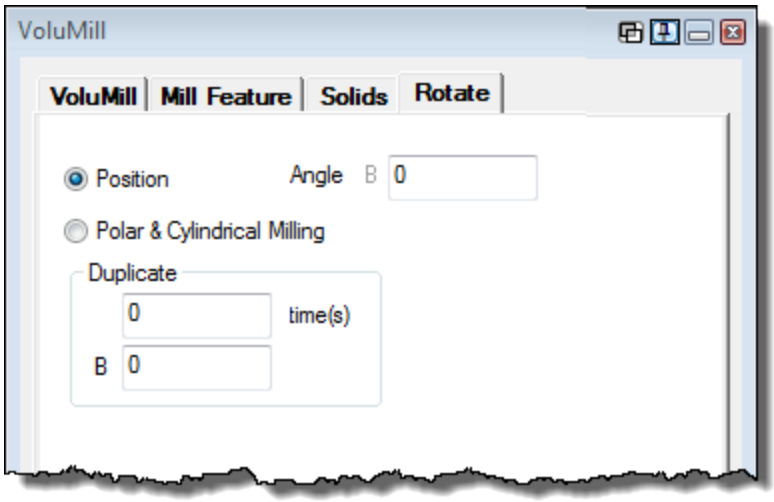
The tool begins by machining the part using the Desired Z Step value. After each primary depth is machined, the tool reduces the step using the Step Height value.



Rendered part cut with Wall Cleanup selected

Controls on the Rotate Page

The Rotate tab, found in process dialogs when a 4-axis or 5-axis MDD is being used, provides access to machining functions that require a 4th or 5th axis.



The items found on this allow you to create toolpath that is rotated into a position, rotated into 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 either the Mill, Mill/Turn, Advanced CS, or Multi-task Machining (MTM) module is enabled. Additionally, the MDD currently in effect must define at least one A-axis, B-axis, or C-axis.

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.

Polar & Cylindrical Milling

The Polar & Cylindrical Milling option is an add-on option that enhances the Mill functionality. The Polar & Cylindrical Milling option allows for continuous A- or B-axis rotation when programming milling operations. This is often referred to as wrapping. The more general term rotary milling is also sometimes used.

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.

Duplicate

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

Times

The number of additional toolpaths to generate. By entering a value of `3`, a toolpath will be made a total of 4 times: the original and 3 repetitions. If you are simply setting a machining coordinate system, not positioning the toolpath, make sure to set this value to "`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.

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, make sure to set this value to "`0`" to avoid setting the toolpath at an angle.

Troubleshooting

- Corners not completely machined? To avoid sharp directional changes and maintain smooth motion, VoluMill, by default, will not generate the sharp moves required to machine corners that are less than or equal to the tool radius. Reduce the Min Toolpath Radius to make the tool machine farther into corners.
- Once the VoluMill progress dialog is shown, it should take a few seconds to create the toolpath. Complex shapes may take a little longer. It is possible that the specified tool cannot fit into the selected geometry within VoluMill's motion requirements, which results in no toolpath being generated.
- Narrow areas uncut? To avoid sharp directional changes and maintain smooth motion, VoluMill uses a dynamically calculated minimum radius in the toolpath. This can result in material not being machined in areas where the tool can fit. Reduce the Min Toolpath Radius to make the tool machine further into narrow areas. Please note, however, that if a smaller tool will subsequently be used, it may be more efficient to leave the Min Toolpath Radius at a higher value and let the smaller tool remove the uncut material.
- Depth First may not function as desired when the pockets are open and the distance between the open edges are equal or less than the tool radius. This is a design limitation.

OpenSSL Acknowledgment and License

```
OPENSSL LICENSE
---------------
/* ====================================================================
* COPYRIGHT (C) 1998-2007 THE OPENSSL PROJECT.  ALL RIGHTS RESERVED.
*
* REDISTRIBUTION AND USE IN SOURCE AND BINARY FORMS, WITH OR WITHOUT
* MODIFICATION, ARE PERMITTED PROVIDED THAT THE FOLLOWING CONDITIONS
* ARE MET:
*
* 1. REDISTRIBUTIONS OF SOURCE CODE MUST RETAIN THE ABOVE COPYRIGHT
*    NOTICE, THIS LIST OF CONDITIONS AND THE FOLLOWING DISCLAIMER.
*
* 2. REDISTRIBUTIONS IN BINARY FORM MUST REPRODUCE THE ABOVE COPYRIGHT
*    NOTICE, THIS LIST OF CONDITIONS AND THE FOLLOWING DISCLAIMER IN
*    THE DOCUMENTATION AND/OR OTHER MATERIALS PROVIDED WITH THE
*    DISTRIBUTION.
*
* 3. ALL ADVERTISING MATERIALS MENTIONING FEATURES OR USE OF THIS
*    SOFTWARE MUST DISPLAY THE FOLLOWING ACKNOWLEDGMENT:
*    "THIS PRODUCT INCLUDES SOFTWARE DEVELOPED BY THE OPENSSL PROJECT
*    FOR USE IN THE OPENSSL TOOLKIT. (HTTP://WWW.OPENSSL.ORG/)"
*
* 4. THE NAMES "OPENSSL TOOLKIT" AND "OPENSSL PROJECT" MUST NOT BE USED TO
*    ENDORSE OR PROMOTE PRODUCTS DERIVED FROM THIS SOFTWARE WITHOUT
*    PRIOR WRITTEN PERMISSION. FOR WRITTEN PERMISSION, PLEASE CONTACT
*    OPENSSL-CORE@OPENSSL.ORG.
*
* 5. PRODUCTS DERIVED FROM THIS SOFTWARE MAY NOT BE CALLED "OPENSSL"
```

```
*    NOR MAY "OPENSSL" APPEAR IN THEIR NAMES WITHOUT PRIOR WRITTEN
*    PERMISSION OF THE OPENSSL PROJECT.
*
* 6. REDISTRIBUTIONS OF ANY FORM WHATSOEVER MUST RETAIN THE FOLLOWING
*    ACKNOWLEDGMENT:
*    "THIS PRODUCT INCLUDES SOFTWARE DEVELOPED BY THE OPENSSL PROJECT
*    FOR USE IN THE OPENSSL TOOLKIT (HTTP://WWW.OPENSSL.ORG/)"
*
* THIS SOFTWARE IS PROVIDED BY THE OPENSSL PROJECT ``AS IS'' AND ANY
* EXPRESSED OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE
* IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR
* PURPOSE ARE DISCLAIMED.  IN NO EVENT SHALL THE OPENSSL PROJECT OR
* ITS CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL,
* SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT
* NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES;
* LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION)
* HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT,
* STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE)
* ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED
* OF THE POSSIBILITY OF SUCH DAMAGE.
* ====================================================================
*
* THIS PRODUCT INCLUDES CRYPTOGRAPHIC SOFTWARE WRITTEN BY ERIC YOUNG
* (EAY@CRYPTSOFT.COM).  THIS PRODUCT INCLUDES SOFTWARE WRITTEN BY TIM
* HUDSON (TJH@CRYPTSOFT.COM).
*
*/
ORIGINAL SSLEAY LICENSE
-----------------------
/* COPYRIGHT (C) 1995-1998 ERIC YOUNG (EAY@CRYPTSOFT.COM)
* ALL RIGHTS RESERVED.
*
```

```
* THIS PACKAGE IS AN SSL IMPLEMENTATION WRITTEN
* BY ERIC YOUNG (EAY@CRYPTSOFT.COM).
* THE IMPLEMENTATION WAS WRITTEN SO AS TO CONFORM WITH NETSCAPES SSL.
*
* THIS LIBRARY IS FREE FOR COMMERCIAL AND NON-COMMERCIAL USE AS LONG AS
* THE FOLLOWING CONDITIONS ARE AHEARED TO.  THE FOLLOWING CONDITIONS
* APPLY TO ALL CODE FOUND IN THIS DISTRIBUTION, BE IT THE RC4, RSA,
* LHASH, DES, ETC., CODE; NOT JUST THE SSL CODE.  THE SSL DOCUMENTATION
* INCLUDED WITH THIS DISTRIBUTION IS COVERED BY THE SAME COPYRIGHT TERMS
* EXCEPT THAT THE HOLDER IS TIM HUDSON (TJH@CRYPTSOFT.COM).
*
* COPYRIGHT REMAINS ERIC YOUNG'S, AND AS SUCH ANY COPYRIGHT NOTICES IN
* THE CODE ARE NOT TO BE REMOVED.
* IF THIS PACKAGE IS USED IN A PRODUCT, ERIC YOUNG SHOULD BE GIVEN
ATTRIBUTION
* AS THE AUTHOR OF THE PARTS OF THE LIBRARY USED.
* THIS CAN BE IN THE FORM OF A TEXTUAL MESSAGE AT PROGRAM STARTUP OR
* IN DOCUMENTATION (ONLINE OR TEXTUAL) PROVIDED WITH THE PACKAGE.
*
* REDISTRIBUTION AND USE IN SOURCE AND BINARY FORMS, WITH OR WITHOUT
* MODIFICATION, ARE PERMITTED PROVIDED THAT THE FOLLOWING CONDITIONS
* ARE MET:
* 1. REDISTRIBUTIONS OF SOURCE CODE MUST RETAIN THE COPYRIGHT
*    NOTICE, THIS LIST OF CONDITIONS AND THE FOLLOWING DISCLAIMER.
* 2. REDISTRIBUTIONS IN BINARY FORM MUST REPRODUCE THE ABOVE COPYRIGHT
*    NOTICE, THIS LIST OF CONDITIONS AND THE FOLLOWING DISCLAIMER IN THE
*    DOCUMENTATION AND/OR OTHER MATERIALS PROVIDED WITH THE DISTRIBUTION.
* 3. ALL ADVERTISING MATERIALS MENTIONING FEATURES OR USE OF THIS SOFTWARE
*    MUST DISPLAY THE FOLLOWING ACKNOWLEDGEMENT:
*    "THIS PRODUCT INCLUDES CRYPTOGRAPHIC SOFTWARE WRITTEN BY
*     ERIC YOUNG (EAY@CRYPTSOFT.COM)"
*    THE WORD 'CRYPTOGRAPHIC' CAN BE LEFT OUT IF THE ROUINES FROM THE
LIBRARY
```

```
*    BEING USED ARE NOT CRYPTOGRAPHIC RELATED :-).

* 4. IF YOU INCLUDE ANY WINDOWS SPECIFIC CODE (OR A DERIVATIVE THEREOF) FROM

*    THE APPS DIRECTORY (APPLICATION CODE) YOU MUST INCLUDE AN
ACKNOWLEDGEMENT:

*    "THIS PRODUCT INCLUDES SOFTWARE WRITTEN BY TIM HUDSON
(TJH@CRYPTSOFT.COM)"

*

* THIS SOFTWARE IS PROVIDED BY ERIC YOUNG ``AS IS'' AND

* ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE

* IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE

* ARE DISCLAIMED.  IN NO EVENT SHALL THE AUTHOR OR CONTRIBUTORS BE LIABLE

* FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL

* DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS

* OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION)

* HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT

* LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY

* OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF

* SUCH DAMAGE.

*

* THE LICENCE AND DISTRIBUTION TERMS FOR ANY PUBLICALLY AVAILABLE VERSION OR

* DERIVATIVE OF THIS CODE CANNOT BE CHANGED.  I.E. THIS CODE CANNOT SIMPLY
BE

* COPIED AND PUT UNDER ANOTHER DISTRIBUTION LICENCE

* [INCLUDING THE GNU PUBLIC LICENCE.]

*/
```

Conventions

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

Text

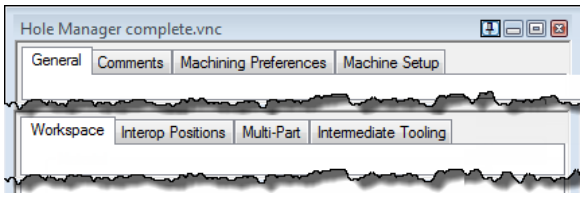
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

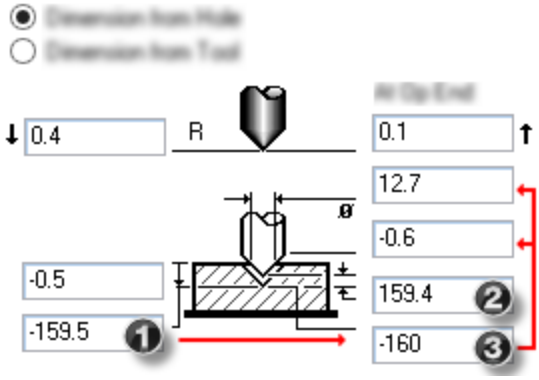
`Keystroke/Mouse`. Text with `this appearance` indicates a keystroke or mouse action, such as `Ctrl+C` or `right-click`.

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

Graphics

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





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

Links to Online Resources

Please contact your reseller for support.

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